

Tekla Structural Designer 2024

Product Guides

March 2024

©2024 Trimble Inc. and affiliates

Contents

1 Get familiar with Tekla Structural Designer....29

1.1 Introducing Tekla Structural Designer....29

Philosophy....30

Tekla Structural Designer way of working....30

1.2 Tekla Structural Designer configurations....31

1.3 Start Tekla Structural Designer....34

Choose the country for your settings....34

Check or change your settings....35

Open a project....36

Start a new project....36

Use templates in new projects....38

Work with autosave and backups....39

The difference between autosave and backup....39

Enable autosave....40

Enable backups....40

Automatically restore an autosave or backup following a Program error....40

Manually restore a backup....42

1.4 Introduction to the Tekla Structural Designer user interface....42

Overview of Quick Access Toolbar, Ribbon, Sign in, and Help....44

Overview of the project workspace....46

View and modify model properties in the Project Workspace....48

Manage groups in the Project Workspace....54

View load status in the Project Workspace....57

View and modify wind properties in the Project Workspace....58

View model status in the Project Workspace....59

Overview of the Properties window and Properties dialog box....61

Overview of scene views....63

Open and close scene views....64

How to control scene content....66

Work with view configurations....67

Overview of scene view interface components....69

Create and modify view tab groups....73

Overview of view regimes....73

Overview of Tabular Data views....75

Overview of Report views....77

Overview of the Load Analysis view....78

Overview of the Loading List and Status bar....78

Overview of Side panes....81

Overview of the Process window....83

How to use the Quick Access window....84

Open Quick Access....84

Quick Access Window layout....84

To find and run an action....84

Example actions....85

How to hide, re-display and move windows....87

Auto-hide a window..... 87
Close a window..... 87
Re-display a closed window..... 87
Move a window..... 88
Dock a window as a tabbed page in another window..... 88
Open a tabbed page in another window..... 88
Dock a window using the docking control..... 88
Keyboard shortcuts..... 88
General..... 89
2D and 3D Views..... 89
Interface window/scene view switcher..... 90
Project Workspace..... 91
Properties window..... 91
Scene Content window..... 92
Quick Access window..... 92
Ribbon commands..... 93
1.5 Contact Tekla Structural Designer support (Support tool)..... 94
2 Create models..... 97
2.1 Get to know Tekla Structural Designer basic working methods..... 97
Zoom, pan, rotate and walk through the model..... 98
Zoom in and out, or zoom extents..... 98
Pan the view..... 98
Rotate the view with the ViewCube..... 98
Rotate the view manually..... 100
Walk through the model in a 3D view..... 100
Display a 2D view in 3D..... 100
Select entities..... 101
Select single entities..... 101
Select multiple entities using area selection..... 102
Select using Find..... 104
Select from the Project Workspace..... 105
Select nodes..... 105
Modify the selection..... 107
Use Ghost Unselected to focus on the selection..... 108
Select a section in the Select Section dialog box..... 110
Edit entity properties..... 111
Edit properties using the Properties window..... 111
Edit properties using the Properties dialog box..... 111
Edit properties of multiple entities..... 112
Re-position entities by moving nodes or edges..... 112
Modify one end of a grid or construction line..... 112
Move a grid or construction line..... 113
Modify one end of a member..... 114
Add a new node into a beam..... 114
Modify slab items and panels by moving a node..... 116
Modify slab items by moving an edge..... 117
Modify walls by moving a node..... 118
Tips for basic tasks..... 118
Use the tooltip for input in a command..... 119
Undo a command..... 119
Cancel a command or go back to the previous prompt..... 119
2.2 Create levels, grids, members, slabs, panels and supports..... 120
Create and manage construction levels..... 121

Open the Construction Levels dialog 121
Insert a single construction level 121
Insert multiple construction levels 121
Make a level an identical copy of another level 122
Make a level an independent copy of another level 122
Modify the properties of a construction level 123
Delete construction levels 123
Create and manage grid lines and architectural grids 123
Overview 124
Create a single grid line 125
Create parallel grid lines 126
Create perpendicular grid lines 128
Create a rectangular grid line system 128
Create a radial grid line system 131
Create a grid arc 133
Import grids or a shadow from a DXF file 134
Extend, move, or rotate grid lines and arcs 136
Number and renumber grids 137
Choose the levels at which to show/hide grid lines and construction lines 138
Apply an architectural grid to a specific level 139
Create and manage construction lines 139
Create a single construction line 139
Create parallel construction lines 140
Create perpendicular construction lines 141
Create a rectangular construction line system 142
Create a radial construction line system 143
Create construction arcs 144
Extend, move, or rotate construction lines and arcs 144
Create frames and slopes 146
Create a frame 146
Create a slope 147
Show frame or slope references 148
Create beams, columns and braces 148
Create columns 149
Create beams 167
Create braces 190
Member global offsets 194
Create walls, cores and bearing walls 197
Create concrete walls 198
Create concrete cores 205
Create bearing walls 208
Create shear only walls 211
Create general walls 213
Create slabs and decks 216
Overview of the slab model 217
Create slab items 221
Create slab or mat openings 223
Add overhangs to existing slab or mat edges 225
Apply curved edges to existing slab items 227
Create column drops 227
Specify the material for general slab types 228
Split and join slabs and mats 229
Modify slab/panel span direction 231
Create trusses and joists 231
Create trusses 232

Create steel joists.... 234
Create portal frames.... 236
Create a single or multi-span portal frame.... 236
Modify the properties of an existing portal frame.... 237
Add copy or mirror spans in an existing portal frame.... 238
Portal frame haunch geometry.... 238
Create members with cold rolled sections.... 239
Create a cold rolled section member.... 239
Modify the position of a cold rolled section member.... 240
Create wall and roof panels.... 240
Create rectangular wall panels.... 240
Create non-rectangular wall panels.... 241
Create wall panels with parapets.... 241
Modify the properties of a wall panel.... 241
Create roof panels.... 242
Modify the properties of roof panels.... 242
Create supports.... 242
Create a single support.... 243
Create a rotated or inclined support using 3 grid points.... 243
Create spring supports.... 243
Create nominally pinned or nominally fixed supports.... 244
Modify support properties.... 244
Rotated and inclined supports.... 245
Partial fixity of column bases.... 252
Create analysis elements.... 253
Create analysis elements.... 253
Create analysis element springs.... 253
Modify the position of analysis elements.... 254
Element types.... 254
Dimensions and measurements.... 257
Create a dimension between existing construction points.... 257
Create a freehand dimension between any two points in a plane.... 258
Measure a distance.... 259
Measure an angle.... 259
2.3 Create isolated foundations.... 260
Create pad bases.... 261
Create a pad base under a specific column.... 261
Create multiple pad bases.... 261
Override the loaded area to be used in the pad base punching check.... 261
Create strip bases.... 262
Create a strip base under a specific wall.... 262
Create multiple strip bases.... 262
Create a pile type catalogue.... 262
Create pile caps.... 263
Create pile cap under a specific column.... 263
Create multiple pile caps.... 263
Create a user-defined pile arrangement.... 264
Override the loaded area to be used in the pile cap punching check.... 265
2.4 Create mat foundations.... 265
Create mats.... 266
Create a minimum area or rectangular mat.... 266
Create a strip mat.... 266
Create an area mat.... 267
Create a mat within bays.... 267
Place piles and pile arrays in mats.... 267

Specify if a piled mat is ground bearing..... 268
Place an individual pile in a mat..... 268
Place a pile array in a mat..... 268
Specify the pile direction of an inclined pile..... 269
2.5 Edit the model..... 270
Copy and rotate objects..... 271
Move and rotate objects..... 271
Mirror objects to new locations..... 272
Copy loads..... 274
Copy all member loads from one span to another..... 274
Only copy one member load to another span..... 275
Copy panel area, level, and slab loads..... 275
Copy panel point, line, and patch loads..... 276
Copy structure loads..... 276
Copy loads to another loadcase..... 277
Delete entities..... 277
Join and split members..... 278
Join members..... 278
Split members..... 279
Automatically join all concrete beams..... 279
Reverse member axes and panel faces..... 280
Reverse the local axis of a beam..... 280
Reverse the outward face of a wind panel..... 281
Create cutting planes..... 281
Position a cutting plane..... 281
Deactivate a cutting plane..... 287
Activate and deactivate all cutting planes..... 291
Align all cutting planes..... 293
Reset all cutting planes..... 296
Move the model or the DXF shadow..... 297
Move the model..... 297
Move the DXF shadow..... 298
Rationalize the model..... 298
Delete unused sloped planes, frames, grids, and construction lines..... 298
Update grid and construction line length..... 299
Create infill members..... 299
Define the infill properties and pattern..... 299
Place the pattern in a single bay..... 300
Place the pattern in multiple bays..... 300
Merge planes..... 301
Create and manage free points..... 301
Create a free point..... 301
Adding, moving or deleting free points from the Edit tab..... 301
..... 302
Planarize..... 302
Overview..... 302
Move nodes into the plane of a grid line..... 307
Move nodes into the plane of a frame..... 309
Planarize properties..... 311
Planarize Example..... 312
2.6 Validate the model..... 323
Run model validation..... 323
Adjust the conditions considered in model validation..... 323

3 Apply loading.....324

3.1 Create and manage loadcases, groups, combinations, envelopes and patterns..... 324

Create and manage loadcases.....325
Create loadcases..... 325
Activate reductions in live or imposed loadcases..... 325
Set up minimum lateral loadcases.....328
Set up global imperfections..... 328
Renumber and/or reorder all loadcases..... 329
Create load groups..... 329
Overview of load groups..... 329
Create load groups.....330
Inclusive and exclusive load groups example.....331
Create and manage load combinations.....338
Load combination classes..... 339
Generate load combinations automatically.....340
Create load combinations manually..... 341
Nominate a gravity combination to be used for minimum lateral load calculations 342
Create modal mass combinations..... 343
Import loadcases and combinations from a spreadsheet..... 343
Renumber and/or reorder all load combinations.....347
Create envelopes..... 348
Create envelopes.....348
Create and manage load patterns.....348
Overview of load patterns.....349
Apply patterning to live/imposed loadcases.....350
Apply patterning to load combinations.....351
Update load patterns.....351

3.2 Apply panel, member, and structure loads..... 351

Apply panel loads..... 352
Create point loads..... 352
Create line loads.....353
Create patch loads..... 353
Create polygonal loads..... 354
Create perimeter loads.....355
Create variable patch loads..... 355
Create area loads..... 356
Create variable area loads..... 357
Create slab loads.....357
Create level loads..... 357
Apply member loads..... 358
Create full-length UDLs.....358
Create partial-length UDLs or VDLs..... 358
Create trapezoidal loads.....359
Create point loads and moment loads.....360
Create full-length torsional UDLs..... 360
Create partial-length torsional UDLs and VDLS.....361
Apply structure loads 361
Diaphragm loads and diaphragm load tables..... 362
Create nodal loads..... 369
Create temperature loads..... 369
Create settlement loads..... 369

Modify panel, member, and structure loads....370
Delete panel, member, and structure loads....370
Decompose panel loads....370
Decompose panel loads for an individual construction level....370
Decompose panel loads to all required levels....371
View decomposed loads graphically....371
View applied and decomposed member loads in a table....372
3.3 Apply wind loading....372
Wind wizard pressure loading....373
Wind wizard workflow....374
Wind model loadcases....377
Wind model load decomposition....378
Instructions for creating, editing and reviewing the wind model....387
Wind wizard workflow for open structures....392
Instructions for applying loads to open structures using the wind wizard....395
Simple wind pressure loading....396
Instructions for applying simple wind loads to diaphragms....398
Simple wind load example (diaphragms)....400
Limitations of wind decomposition to diaphragms....402
Instructions for applying simple wind loads to wall/roof panels and open structure members....411
Simple wind load example (panels)....413
Simple wind load workflow for open structures....416
Open structure wind loads....417
Manual wind loads....423
Wind tunnel testing overview....424
Exporting wind tunnel data workflow....424
Using imported wind tunnel information....426
3.4 Apply snow loading....426
Roof panel types....427
Run the snow wizard....429
Run the snow wizard (ASCE 7)....429
Run the snow wizard (Eurocode)....430
Run the snow wizard (AS, BS, or IS)....430
Apply loads to loadcases on completion of the snow wizard....431
Apply uniform snow loads....431
Apply a valley snow load....432
Apply a local drift snow load....432
Override snow loads....433
Update snow loads....433
Delete the snow model....434
3.5 Apply seismic loading....434
Select the seismic loading code....434
Create seismic loads in the seismic wizard....435
Display the horizontal design spectrum....436
Delete seismic loads....436
The seismic wizard....436
Using the ASCE 7 seismic wizard....436
Using the UBC 1997 seismic wizard....445
Using the Eurocode EN1998-1:2004 seismic wizard....452
Using the IS1893 seismic wizard....460
Code spectra and site specific spectra....466
Seismic loadcases....478

4 Analyze models 482

4.1 Analysis types 482

1st order linear 483
1st order non-linear 483
1st order modal 484
2nd order linear 485
2nd order non-linear 486
2nd order buckling 486
FE chasedown 487
Grillage chasedown 489
Analyze All (Static) 491
3D only (Static) 492
1st order RSA seismic 493
2nd order RSA seismic 493
Analyze All (RSA) 494
Rigorous slab deflection analysis 494
Linear, non-linear and buckling analyses 494
Run a 1st order linear or non-linear analysis 495
Run a 2nd order linear or non-linear analysis 495
Run a 2nd order buckling analysis 496
Run Analyze All (Static) 497
Run 3D only (Static) 498
Run FE chasedown or grillage chasedown analysis 499
Why chasedown analyses are included in Analyze All (Static) 499
Features of 3D analysis and chasedown analysis 506
Modal analysis 508
What is modal analysis? 508
Active mass and mass participation 510
Modal mass 513
Run a 1st order modal analysis 514
RSA seismic analysis 515
Run a 1st order RSA seismic analysis 515
Run a 2nd order RSA seismic analysis 515
Run Analyze All (RSA) 515
Analysis limitations and assumptions 516
Linear analysis of structures containing material nonlinearity 516
Analysis of structures containing geometric nonlinearity 517
Analysis of structures containing curved beams 517
Analysis of compound (plated) steel beams and columns 517
Member Deflections 517
Torsion load analysis - relative angle of twist 518
Unstable Structures 519
Analysis verification examples 519
1st order linear - Simple cantilever 520
1st order linear - Simply supported square slab 520
1st order linear - Simply supported orthotropic square slab 522
1st order linear - 3D truss 524
1st order linear - Thermal load on simply supported beam 525
1st order nonlinear - Simple cantilever 525
1st order nonlinear - Nonlinear supports 526
1st order nonlinear - Displacement loading of a plane frame 527
2nd order linear - simple cantilever 528
2nd order linear - Simply supported beam 529
2nd order nonlinear - Tension only cross brace 531

2nd order nonlinear - Compression only element 532
1st order modal - Simply supported beam..... 533
1st order modal - Bathe and Wilson eignenvalue problem..... 533
2nd order buckling - Euler strut buckling..... 534
2nd order buckling - Plane frame..... 535
4.2 Solver models..... 537
What is a solver model?..... 537
Solver model types..... 540
Working Solver Model..... 541
Solver Model used for 1st Order Linear and 2nd Order Linear..... 541
Solver Model used for 1st Order Non Linear and 2nd Order Non Linear..... 543
Solver Model used for 1st Order Modal..... 543
Solver Model used for 2nd Order Buckling..... 544
Solver Model used for Grillage Chasedown..... 544
Solver Model used for FE Chasedown..... 546
Solver Model used for Load Decomposition..... 548
Refresh Solver Model..... 548
Open a Solver View..... 549
Open a 3D Solver View..... 549
Open a 2D Solver View..... 549
Change an existing view to a Solver View..... 550
Select the solver model type..... 550
Solver element (1D) types..... 551
Beam..... 551
Truss..... 551
Tension only..... 552
Compression only..... 552
Linear axial spring..... 552
Linear torsional spring..... 552
Non-linear axial spring..... 552
Non-linear torsional spring..... 553
Solver element (2D) types..... 553
Shell..... 553
SemiRigid..... 553
Wall..... 553
How meshed walls are represented in solver models..... 554
How mid-pier walls are represented in solver models..... 559
How shear only walls are represented in solver models..... 561
Background..... 561
Solver model in Tekla Structural Designer..... 564
How bearing walls are represented in solver models..... 565
4.3 Analysis model choices..... 568
Beam end releases and partial fixity..... 568
Set Partially fixed release type..... 569
Set Moment (pin My) release type..... 570
Set releases in a Review View..... 571
Column releases 571
Rigid offsets and rigid zones in concrete members..... 572
Rigid offsets..... 572
Rigid zones..... 573
Rigid offsets examples..... 574
Rigid zones examples..... 578
FE meshed slabs..... 584
Define whether two-way slabs are meshed for 3D building analysis and grillage chasedown analysis..... 585

Adjust global slab mesh properties..... 586
Apply different mesh properties at different levels..... 586
Review the slab mesh before the analysis..... 587
How slab properties and features impact on meshing..... 587
Slab meshing controls..... 592
Orthotropic slabs..... 601
Sub models..... 616
Create sub models..... 618
Open a 3D view of a sub model..... 618
Delete sub models..... 618
Diaphragm action in roof panels and slabs..... 619
Overview of diaphragm action in roof panels and slabs..... 619
Managing diaphragm action in roof panels and slabs..... 623
One-way and two-way load decomposition..... 626
Potential load decomposition methods..... 626
Controlling decomposition via the 'Mesh slabs in 3D Analysis' level property..... 630
Decomposition rules for the different types of panel and overlapping panels..... 632
How to view the solver model used for load decomposition..... 634
FE meshed walls..... 635
Mesh type and size guidelines for FE walls..... 635
Adjust the wall mesh parameters specified in structure properties..... 639
Apply different mesh parameters to individual walls..... 640
Review the wall mesh before the analysis..... 640
Meshed wall openings analysis model..... 641
Orthotropic walls..... 644
Cracked, partially cracked, and uncracked concrete members..... 650
Usage of cracked, partially cracked, and uncracked..... 651
Partially cracked modification factor..... 652
Determining wall cracked properties..... 652
Workflow for reviewing wall cracked properties..... 653
The Direct Analysis Method..... 655
DAM implementation in Tekla Structural Designer..... 656
Choice of analysis type..... 657
Analysis model adjustments..... 658
Suggested workflow..... 660
4.4 Analysis results..... 662
Display analysis results in a Results View..... 662
Set the analysis type and loading for viewing analysis results..... 664
Display reactions..... 665
Display 1D results..... 667
Display 1D deflections..... 667
Animate 1D and 2D deflections..... 668
Display sway, drift, and story shear..... 668
Display notional forces, minimum lateral loads and seismic equivalent lateral forces..... 684
Display forces in 2D elements..... 685
Display stresses in 2D elements..... 687
Display 2D deflections..... 691
Display AsReq contours..... 692
Display wall lines..... 692
Display core lines..... 693
Manage and display result strips..... 693
Manage, display and design result lines..... 698
Display mode shapes..... 701
RSA seismic results..... 701

Customize the display of 2D contours....706
Change result diagram scale settings....706
Display 2D view in isometric projection....707
Display tabular analysis results in a Solver Model Data view....707
Nodes....708
Elements....709
Nodal forces....709
Nodal deflections....710
Element end forces....710
Wall lines....711
2D result lines....711
Core lines....712
Shear only walls....713
FE results by node....713
FE results by shell....714
FE slab results by panel....714
FE wall results by panel....715
1D member forces....716
Foundation reactions....717
Pile forces....718
Story forces....718
Story shears....719
Wall/Braced Bay Story shears....719
Modal frequencies....720
Summed mass....720
Dynamic masses....721
Active masses by node....721
Total masses by node....721
Total masses by level....722
Mode shape by node....722
Mode shape by level....723
Buckling factors....723
Working with member forces table options....723
Working with model filters....725
Working with foundation reaction axis systems....727
Display member analysis results in a Load Analysis View....748
Open a Load Analysis View....749
Load Analysis View properties for columns....750
Load Analysis View properties for beams....753
RSA Seismic Results in a Load Analysis View....757
Display loading summaries in the Project Workspace....758
View the load status of a loadcase or combination....759
View the magnitude of minimum lateral loads....760
View the magnitude of notional forces....760
4.5 Analysis settings....761
1st order non-linear settings....761
2nd order non-linear settings....762
1st order modal settings....763
2nd order buckling settings....766
1st order seismic settings....767
Iterative cracked section analysis settings....771
Modification factors....772
Meshing settings....773
Composite steel beams settings....773

5 Design models....776

5.1 Design steel members and cast-in-place concrete beams, columns and walls....777

Apply and modify design settings....777
Modify design settings in the current project....778
Modify design settings defaults for future projects....778
Autodesign versus check design....778
Overview of combined analysis and member design....779
Gravity Design....780
Static Design....782
RSA Design....784
Choice of Design Steel, Design Concrete, or Design All....784
Design force sets used for steel and concrete member design....785
Run combined analysis and member design....786
Run Design Steel (Gravity)....786
Run Design Steel (Static)....786
Run Design Steel (RSA)....787
Run Design Concrete (Gravity)....787
Run Design Concrete (Static)....787
Run Design Concrete (RSA)....788
Run Design All (Gravity)....788
Run Design All (Static)....789
Run Design All (RSA)....789
Check selected members and walls....790
Check an individual member, wall, or core....790
Check selected members and walls....790
Check all members in a level, slope, or frame....790
Check all members....791
Check all walls....791
Check all members and walls....791
Check all members of a particular section or type....791
Check all members in a group....792
Check all members and walls in a sub structure....792
Design selected members and walls....792
Design an individual member, wall, or core....792
Design selected members and walls....793
Interactively design a concrete member....793
Design all members in a level, slope, or frame....794
Design all members....794
Design all walls....795
Design all members and walls....795
Design all members of a particular section or type....795
Design all members in a group....796
Design all members and walls in a sub structure....796
Design individual members for gravity only....796
Apply user defined utilization ratios....797
Overview of user defined U/R....797
Apply user defined U/R for autodesign only....798
Apply user defined U/R for autodesign and check....798
Validate the model for design issues....799
Run design validation....799
Adjust the conditions considered in design validation....800

5.2 Design slabs and mats, run punching shear checks....800

Create and modify patches..... 800
Create column patches.....800
Create beam patches.....801
Create wall patches.....802
Create panel patches.....804
Modify patch properties.....805
Resize patches..... 806
Design and check slabs.....806
Check an individual slab item..... 806
Check all slab items.....807
Check all slab items on a single floor.....807
Check all slab items in a sub structure..... 807
Design an individual slab item.....808
Design all slab items..... 808
Design all slab items on a single floor..... 808
Design all slab items in a sub structure.....809
Design and check mat foundations.....809
Design or check all mats in the model.....809
Check all mats in a single floor..... 809
Design all mats in a single floor.....809
Check an individual mat..... 810
Design an individual mat.....810
Design and check patches..... 810
Check an individual patch..... 810
Check all patches in the model.....811
Check all patches on a single floor.....811
Design an individual patch.....811
Design or check all patches in the model.....811
Design all patches on a single floor..... 811
Create punching checks.....812
Create punching checks in a flat slab..... 812
Create punching checks in a foundation mat.....812
Specify the axis orientation for punching checks applied to point loads 813
Override the punching check loaded area.....813
Confirm punching check positions.....814
Specify stud rail reinforcement..... 814
Design and check punching shear.....815
Design an individual punching check item.....815
Design all punching check items on a floor..... 816
Design all punching check items..... 816
Check an individual punching check item.....817
Check all punching check items on a floor.....818
Check all punching check items.....818
5.3 Design isolated foundations..... 818
Design or check all pad bases and strip bases..... 818
Design or check all pile caps..... 819
Check an individual isolated foundation..... 819
Design an individual isolated foundation.....819
5.4 Check floor vibration..... 819
Create floor vibration check items..... 820
Create floor vibration checks that consider two or three adjoining spans..... 821
Modify the properties of existing floor vibration check items.....822
Check vibration for all floor vibration check items.....822
Check floor vibration for an individual floor vibration check item.....822

5.5 Check drift, sway, seismic drift, wind drift, and overall displacements.822
Drift check....823
Run the check.... 824
Review the check status and details.... 824
Review the check graphically.... 826
Switching off inappropriate checks and merging stack lengths....827
Printing calculations....828
Sway check.... 828
Run the check.... 828
Review the check status and details.... 828
Review the check graphically.... 830
Switching off inappropriate checks and merging stack lengths....831
Printing calculations....832
Seismic drift check....832
Configure the check and set the limit....832
Run the check.... 832
Review the check status and details.... 832
Review the check graphically.... 834
Switching off inappropriate checks and merging stack lengths....835
Printing calculations....836
Wind drift check....836
Configure the check and set the limit....836
Run the check.... 836
Review the check status and details.... 837
Review the check graphically.... 838
Switching off inappropriate checks and merging stack lengths....839
Printing calculations....840
Overall wind drift check.... 840
Configure the check and set the limit....841
Run the check.... 841
Review the check status and details.... 841
Check overall displacement 842
Common tasks for sway and drift checks....842
Automatically merge short stacks....842
Set the wind drift limit.... 843
Choose resultant or directional wind drift checks.... 843
Consider wind cases only for the wind drift check.... 844
Switch off sway checks for selected columns/walls....844
Switch off drift checks for selected columns/walls.... 844
Switch off seismic drift checks for selected columns/walls.... 845
Switch off wind drift checks for selected columns/walls....845
Switch off tabular results for an entire level 846
Override the wind drift limit for selected columns/walls....846
Adjust column stack or wall panel check lengths....846
Considerations for non-linear models with Tension Only bracing 847
Perform checks....847
Review tabular results in a Data Table....847
Review results graphically in a Show / Alter State view 848
Create a report.... 849
5.6 Design timber members, precast members, and masonry walls using Tekla Tedds.... 849
Design and check timber members and precast members 849
Design and check masonry walls....851
Design using Tekla Tedds....852

Where to find each design option 852
Understanding each of the design options..... 852
To design all timber and precast members..... 855
To design a timber or precast member..... 856
To design a masonry wall..... 857
To design a timber or precast member group..... 857
To design selected timber or precast members or masonry walls..... 858
To design timber or precast members or masonry walls in a sub structure..... 859
Check using Tekla Tedds..... 860
Where to find each check option..... 860
Understanding each of the check options..... 861
To check all timber or precast members..... 862
To check a timber or precast member..... 863
To check a masonry wall..... 863
To check a timber or precast member group..... 863
To check selected timber or precast members or masonry walls..... 864
To check timber or precast members or masonry walls in a sub structure..... 864
Clear Tekla Tedds Data..... 865
5.7 Design settings..... 866
Design Settings > Analysis..... 866
United States (ACI/AISC)..... 866
All Eurocode regions..... 869
All British Standards regions..... 870
India..... 872
Australia..... 873
Design Settings > General..... 874
Design Settings > Steel > General..... 875
Design Settings > Steel > Composite Beams..... 876
Design Settings > Steel > Steel Joists..... 877
Design Settings > Concrete (Cast-in-place)..... 878
General..... 878
Beam..... 879
Column..... 889
Wall..... 893
Slab..... 897
Punching Shear..... 899
Foundations > General..... 900
Foundations > Isolated Foundations..... 901
Foundations > Mat Foundations..... 905
Design Settings > Concrete (Precast - Eurocode only)..... 907
Beam..... 907
Column..... 909
Design Settings > Design/Reporting Forces..... 910
Ignore forces for member design below..... 910
Ignore deflections for reporting..... 911
Concrete beams (Eurocode only)..... 911
Minimum brace force for reporting and connection design..... 911
Minimum beam end force for reporting and connection design..... 912
Brace/beam end rounding increments..... 913
Minimum foundation reactions for reporting and connection design..... 914
Minimum design forces..... 914
How Minimum Design Forces are used..... 915
Design Settings > Design Groups..... 927
Design Settings > Autodesign..... 927
Design Settings > Design Warnings (AISC/ASC only)..... 932

Design Settings > Design Checks (IS only)..... 936
Design Settings > Drift Checks/Sway & Drift Checks..... 938
Sway & drift check settings..... 938
Design Settings > Fire check (Eurocode only)..... 939
..... 939
Design Settings > Timber..... 940
Timber (US regional code)..... 940
Timber (Eurocode)..... 941

6 Review models..... 942

6.1 Review designs..... 943
Review member design..... 943
Review member design status..... 943
Review member design ratios..... 944
Review member depth ratios..... 944
Review foundation and pile design..... 945
Review foundation or pile status..... 945
Review foundation or pile ratios..... 946
Review slab and mat design..... 946
Review slab and mat design status..... 946
Review slab and mat design ratios..... 948
Review punching check status..... 950
Review punching check ratio..... 950
Design review filters..... 951
Working with the Status filter..... 952
Working with the Utilization ratio filter..... 954
Working with the Entity type filter..... 956

6.2 Review model properties (show/alter state)..... 958
Modify autodesign settings..... 961
Review and modify diaphragm settings..... 962
Review the diaphragm settings..... 962
Modify the diaphragm settings of slab items or roofs..... 962
Include or remove solver nodes from the diaphragm..... 963
Modify end fixity..... 963
Modify BIM status..... 964
Copy or modify slab and foundation reinforcement..... 964
Copy reinforcement..... 965
Modify reinforcement..... 965
Copy section sizes..... 966
Copy material grades..... 967
Copy properties..... 967
Review and modify member filters..... 968
Review sub structures..... 968
Review concrete beam flanges..... 969
Review and modify column splice positions..... 969
Review and apply property sets..... 969
Copy or modify user-defined attributes..... 970
Show/alter state..... 970
Modify active / inactive settings..... 971
Modify assumed cracked settings..... 972
Modify slenderness settings..... 976
Review and set camber..... 976
Apply cantilever ends..... 978
Review carbon factors..... 979

Review and copy deflection limits..... 980
Review and modify drift checks..... 981
Apply fire proofing..... 981
Modify gravity only settings..... 983
Review and set imposed load reduction..... 984
Review and set live load reduction..... 986
Override effective width..... 988
Modify punching shear check position..... 988
Copy quick connector layout..... 989
Review and modify restraints..... 990
Apply rotational stiffness to a beam end..... 997
Review and modify seismic drift..... 998
Review and modify SFRS settings..... 998
Copy shear connectors..... 999
Modify SidePlates..... 999
Review and copy size constraints..... 1000
Review and modify Staged Construction stages..... 1001
Modify stud auto layout..... 1004
Review and modify sway checks..... 1005
Copy transverse reinforcement..... 1005
Review and modify user defined U/R..... 1006
Review utilization and embodied carbon..... 1008
Copy web openings..... 1009
Copy Westok openings..... 1009
Review and modify wind drift checks..... 1010
Modify wind loading..... 1011

6.3 Tabular review data..... 1011

Review design summary tabular results..... 1012
Review sway check tabular results..... 1013
Review sway check tabular results from the Project Workspace Status Tree..... 1014
Review sway check tabular results from a Review View..... 1015
Locate check in a 3D view..... 1016
Understanding the Critical Sway Summary Only tabular results..... 1017
Review amplification factor Kamp..... 1017
Review kamp tabular results from a Review View..... 1018
Locate check in a 3D view..... 1019
Review drift check tabular results..... 1019
Review drift check tabular results from the Project Workspace Status Tree..... 1020
Review drift check tabular results from a Review View..... 1020
Locate check in a 3D view..... 1022
Review seismic drift check tabular results..... 1022
Review seismic drift check tabular results from the Project Workspace Status Tree..... 1023
Review seismic drift check tabular results from a Review View..... 1024
Locate check in a 3D view..... 1025
Review wind drift check tabular results..... 1026
Review wind drift check tabular results from the Project Workspace Status Tree 1026
Review wind drift check tabular results from a Review View..... 1027
Locate check in a 3D view..... 1029
Review material list tabular results..... 1029
Create material list tabular results..... 1030
Locate material list rows in a 3D view..... 1033
Export material list to Excel..... 1033
Material lists for steel..... 1034
Material lists for concrete..... 1040

Material lists for timber....1050
Material lists for cold formed....1051
Material lists for general materials....1052
Review embodied carbon detail....1054
Create embodied carbon detail tabular results....1055
Locate tabular data in a 3D view....1055
Review embodied carbon overview....1056
Create embodied carbon overview tabular results....1056
Locate tabular data in a 3D view....1057
Review floored area tabular results....1057
Review portal frame deflection....1058
Sort and filter tabular review data....1058
Sort tabular review data....1058
Create and apply filters....1058
Edit filters....1059
Export tabular review data to Excel....1059
6.4 Charts....1059
Display embodied carbon by level chart....1059
Display embodied carbon by construction type chart....1060
Display embodied carbon by material chart....1060
Adding charts into reports....1060
7 Create reports and drawings....1062
7.1 Create and modify reports....1062
Report terminology....1063
Model reports....1063
Member reports....1064
Active model report....1064
Active member report....1065
Active and inactive chapters....1065
Report filters....1065
Available styles....1065
Create reports....1066
Configure a model report....1066
Display a model report....1067
Configure a member report style....1067
Display a member report....1068
Filter reports....1069
Create filters....1069
Apply filters....1070
Format reports....1070
Enter company details in the header....1071
Enter project-specific details in the header....1071
Create a new header or footer layout....1072
Adjust and apply report settings....1073
Navigate reports....1073
Navigation using the Report Index....1074
Navigation buttons on the Report tab....1074
Export reports....1075
Export a report to PDF....1075
Export a report to Microsoft Word....1075
Export a report to Excel....1075
Export a report to Tekla Tedds....1075
Print reports....1076

Example reports ... 1076
Beam End Forces report ... 1076
Bracing Forces report ... 1078
Building Analysis & Drift Checks report ... 1078
Building Design report ... 1079
Building Loading report ... 1079
Connection Resistance report ... 1080
Embodied Carbon report ... 1080
Foundation Reactions report ... 1080
Industrial Structure Loading report ... 1081
Material Listing report ... 1082
Member Design Calcs report ... 1084
Open Structure Wind Load report ... 1084
Seismic Design report ... 1085
Solver Model Data report ... 1085
Report settings ... 1086
7.2 Create drawings ... 1090
Drawing categories ... 1091
Adjust and apply drawing settings ... 1093
Adjust drawing settings in the current project ... 1093
Adjust drawing settings in future projects ... 1094
Create drawing scales ... 1094
Create, modify, or delete layer configurations ... 1094
Create, modify, or delete layer styles ... 1096
Create planar drawings ... 1097
Create general arrangement drawings ... 1098
Create beam end force drawings ... 1099
Create column splice load drawings ... 1100
Create foundation reaction drawings ... 1101
Create loading plan drawings ... 1102
Create member detail drawings ... 1103
Create concrete beam detail drawing ... 1103
Create concrete column detail drawing ... 1104
Create concrete wall detail drawing ... 1104
Create non-concrete beam detail drawing ... 1105
Create non-concrete column detail drawing ... 1105
Create base plate detail drawing ... 1106
Create slab and mat drawings ... 1107
Create slab or mat layout drawings ... 1107
Create punching shear check detail drawings ... 1108
Create foundation drawings ... 1108
Create isolated foundation detail drawings ... 1108
Create foundation layout drawings ... 1109
Create concrete member schedule drawings ... 1110
Create concrete beam schedule drawings ... 1110
Create concrete column schedule drawings ... 1111
Create concrete wall schedule drawings ... 1112
Manage drawings in batches ... 1112
Create or generate drawings in batches ... 1113
Specify the drawing layout ... 1114
Specify the loading for load-dependent drawings ... 1114
Reset reinforcement marks in concrete detail drawings ... 1115
View drawings ... 1115
Review drawings ... 1116
View the revision history of drawings ... 1116

Manage schedule drawings in batches....1117
Create new schedule drawings....1117
Specify the schedule drawing layout....1118
View schedule drawings....1118
Reset reinforcement marks in schedule drawings....1118
Review schedule drawings....1119
View the revision history of schedule drawings....1119
Drawing settings....1119
Export Preferences....1119
General....1120
Layer Configurations....1121
Layer Styles....1122
Options > Planar Drawings....1123
Options > Member Details....1132
Options > Member Schedules....1140
Options > Slabs and Mats > Layout....1141
Options > Slabs and Mats > Punching Check Detail....1147
Options > Foundations > Layout....1148
Options > Isolated Foundations > Detail....1153

8 BIM integration....1156

8.1 Import model data....1156

Import a project from a Structural BIM Import file....1157
Import a project from a TEL file....1158
Restrictions....1158
Instructions....1164
Import data from a 3D DXF file....1164
Restrictions....1164
Instructions....1165
Import 3D reference models....1166
Import a TrimBIM file and use it as a reference model....1166
Use a Trimble Connect TrimBIM file or IFC file as a reference model....1166
Edit the offsets used, or layers displayed in a reference model....1167
Remove a reference model....1167

8.2 Working collaboratively with Trimble Connect....1167

Launch Trimble Connect Project Explorer....1168
Link or unlink a project....1169
Create folders, rename folders, rename files....1170
Upload an IFC file of a model....1170
Upload a multi-member drawing....1171
Upload a single member drawing....1171
Upload a model report....1172
Upload a member report....1173
Link a drawing or report to an existing IFC....1173
Check linking progress in the Process Window....1174
Import an uploaded IFC as a 3D reference model....1174
Open a model view of an uploaded IFC in Trimble Connect....1175
Open Trimble Connect....1175

8.3 Export to Trimble applications....1176

Export a model to Tekla Structures....1176
Export to Tekla Connection Designer....1177
To export a single connection....1177
To export multiple connections....1178
To return connection data from Tekla Connection Designer....1178

Export to Tekla Portal Frame Designer....1178
Export to Tekla Portal Frame Designer workflow....1178
How loading, restraints and supports are handled in the export....1179
To export a single frame....1182
To export multiple frames....1182
To return revised sections from Tekla Portal Frame Designer....1183
Export to Tekla Tedds....1183
Understanding each of the export options....1183
To export all timber members, precast members and masonry walls....1184
To export a single member....1185
To export a single wall....1185
To export multiple members or walls....1185
To export a group....1186
To export a substructure....1186
8.4 Export to and import from other applications....1186
Export to and import from Autodesk Revit....1187
Guidance documentation....1187
Export a model to Autodesk Revit....1187
Import a model from Autodesk Revit....1188
Export a model to IFC....1190
Export to and import from Westok Cellbeam....1191
Limitations....1191
Export to Cellbeam....1191
Import from Cellbeam....1192
Export to and import from FBEAM....1192
Overview....1192
Limitations....1193
Export to FBEAM....1196
Import from FBEAM....1197
Review the imported beams....1197
Export a model to ADAPT....1198
Limitations....1198
Instructions....1201
Export a model to STAAD....1202
Export a model to Autodesk Robot Structural Analysis....1202
Export a model to the cloud....1203
Export to One Click LCA....1204
Overview....1204
Show report....1206
Show online results....1206
Export to IDEA StatiCa Connection Design....1207
Export to IDEA StatiCa Connection Design (version 22 and later)....1207
Export to IDEA StatiCa Connection Design (version 21 and earlier)....1217
9 Manage models....1219
9.1 Manage global and model settings....1219
Manage settings sets....1220
Import a settings set for a specific region....1220
Edit the content of a settings set....1220
Make a duplicate copy of an existing settings set....1221
Restore a settings set to 'factory defaults'....1221
Change the active settings set....1222
Delete a settings set....1222
Load settings from the active settings set to the current project....1222

Save settings from the current project to the active settings set....1223
Copy a settings set from one computer to another.... 1223
Define and modify regional codes and design codes....1224
Define default design codes for new projects....1224
Change design codes in an existing project....1225
Define and modify units....1225
Change units and unit precision in an existing project.... 1226
Define the default units and unit precision for new projects....1227
Manage object references....1228
Basics of object reference formats.... 1228
Modify reference formats and texts in an existing project....1230
Modify the reference format syntax of an object type....1230
Change the text used for the materials and characteristics in the reference format 1231
Renumber members.... 1231
Renumber slabs.... 1232
Adjust the default references to be applied to new projects.... 1232
Global settings.... 1232
Settings set settings....1234
General settings.... 1234
Results Viewer settings....1236
Structure default settings....1237
Section default settings....1238
Section order default settings.... 1238
Solver settings.... 1238
Scene settings....1239
Performance settings.... 1242
Model settings.... 1243
Design code settings....1243
Unit settings....1245
References settings....1245
Loading settings.... 1248
Minimum lateral loads settings....1249
Global imperfections settings....1250
Live/imposed load reduction settings.... 1251
Sway/drift settings.... 1252
Grouping settings....1253
Material list settings....1253
Beam lines settings....1253
Analysis Model settings....1255
Validation settings....1256
User-defined attribute settings 1257
Graphics view settings....1259
Structural BIM settings.... 1259
Embodied carbon settings....1262
Apply and manage model specific settings....1262
Apply and manage model settings.... 1263
Apply and manage analysis settings....1263
Apply and manage design settings....1264
Apply and manage drawing settings.... 1264
Apply and manage slab deflection settings....1264
9.2 Manage materials 1265
Add user-defined materials....1266
Add user-defined sections.... 1266
Add user-defined materials.... 1267

Add user-defined masonry using Tedds ... 1267
Add user-defined reinforcement ... 1268
Add user-defined decking ... 1268
Add user-defined precast concrete planks ... 1270
Add user-defined shear connectors ... 1270
Add user-defined shear connectors ... 1271
Add user-defined bolts ... 1271
Add user-defined welds ... 1272
Manage design section orders ... 1272
View the list of sections in a design section order ... 1272
Specify that a section in the list should not be considered for design ... 1273
Sort the listed sections by a different property ... 1273
Specify that a section is non-preferred ... 1273
Reset a design section order back to the original default ... 1274
Change the default design section order ... 1274
Create a new design section order ... 1275
Import a new design section order ... 1275
Add simple connection resistances ... 1276
Pre-defined connection types and resistances ... 1276
Add user-defined connection types ... 1277
Edit user-defined connection types ... 1279
Add user-defined connection resistances ... 1279
Related video ... 1282
Upgrade material databases ... 1282
Synchronize user-defined materials in models and databases ... 1282
Add to Database ... 1283
Update from Database ... 1284
9.3 Manage properties and property sets ... 1285
Save properties to and recall properties from property sets ... 1285
Save properties from the Properties window to a new property set ... 1285
Save the properties of an existing entity to a named property set ... 1286
Recall a previously saved property set to the Properties Window ... 1286
Apply property sets to existing entities ... 1286
Apply a property set to an individual entity in a Structural View ... 1286
Apply a property set to multiple members in a Structural View ... 1287
Apply a property set in a Review View ... 1287
Review where property sets have been applied ... 1287
Transfer property sets between models ... 1288
Export property sets ... 1288
Import property sets ... 1288
Delete property sets ... 1289
9.4 Manage project revisions and view metrics ... 1289
View and modify project summary ... 1289
Record revisions ... 1290
View revisions ... 1290
Apply revision ID as an attribute for each modified element ... 1290
View sessions ... 1291
View changes ... 1291
View metrics ... 1291
View and modify model notes, assumptions etc. ... 1291
9.5 Create and manage user-defined attributes ... 1292
Create attribute definitions ... 1293
Create attribute definitions in the current model ... 1293
Create attribute definitions for new models ... 1293

Attach UDA values to members and panels.... 1294
Attach a UDA value using the Properties Window.... 1294
Attach an existing UDA value in the Review View.... 1294
Graphically review the attached UDA values.... 1295
Open a file that has been attached as a UDA.... 1295
Apply attribute filters to material lists and reports.... 1295
Apply an attribute filter to material list review data.... 1296
Apply an attribute filter to a report.... 1296
9.6 Create and manage sub structures.... 1296
Sub structure characteristics.... 1297
Create a sub structure.... 1297
Edit a sub structure.... 1298
Delete a sub structure.... 1299
Rename a sub structure.... 1299
Review sub structures.... 1299
Create a sub structure group.... 1300
Open a 3D view of a sub structure.... 1300
Use Ghosted to see the view in the context of the whole model.... 1300
9.7 Working with large models 1303
10 Reference 1306
10.1 Sign conventions and coordinate systems.... 1306
Axis systems.... 1306
Diagram conventions.... 1307
Global coordinate system.... 1308
Building directions 1 and 2.... 1308
User coordinate system.... 1309
1D member local coordinate system (general case).... 1310
1D member local coordinate system (vertical members).... 1315
Mid-pier wall coordinate system.... 1317
2D member local coordinate system.... 1320
2D member forces sign convention.... 1323
Result line coordinate system.... 1324
Result strip coordinate system.... 1325
Foundation reaction coordinate system.... 1326
Punching shear check axis system.... 1327
10.2 Properties reference.... 1329
Structure geometric and analytical properties.... 1331
Structure Properties.... 1331
Level Properties.... 1333
Frame Properties.... 1335
Slope Properties.... 1336
Sub Model Properties 1337
Member properties.... 1338
Beam properties.... 1338
Column properties.... 1374
Brace properties.... 1402
Analysis Element properties.... 1419
Base plate properties.... 1422
Member characteristic, construction and fabrication properties.... 1427
Wall and core properties.... 1431
Bearing wall properties.... 1432
Concrete meshed and mid-pier wall properties.... 1434
Masonry wall properties.... 1443

Shear only wall properties.... 1450
General wall properties.... 1452
Concrete core properties.... 1457
Slab and mat properties.... 1457
Slab item properties.... 1458
Slab/Mat overhang properties.... 1479
Patch properties.... 1479
Foundation mat properties.... 1482
Punching check properties.... 1486
Result strip properties.... 1491
Portal frame properties.... 1492
Pad base strip base and pile cap properties.... 1496
Loading element properties.... 1503
Wall Panel Properties 1504
Roof Panel Properties 1506
Line ancillary properties.... 1507
Area ancillary properties.... 1510
Equipment properties.... 1512
Solver model and support properties.... 1514
Support properties 1514
Solver node properties.... 1517
Solver element properties.... 1518
Solver element 2D properties.... 1519
Floor Analysis Model (FAM) properties.... 1520
10.3 Dialog boxes reference.... 1529
Analysis Settings dialog box.... 1530
Checkbot Control Panel.... 1531
Connection Resistance dialog box.... 1533
1. Title Bar.... 1534
2. Filters.... 1535
3. Connection Types.... 1535
4. Info box.... 1535
5. Resistances.... 1535
6. OK and Cancel.... 1536
Construction Levels dialog box.... 1536
Design Settings dialog box.... 1538
Drawing Settings dialog box.... 1539
Edit stage data dialog box.... 1540
Embodied Carbon Factors dialog box.... 1542
1. Category filter.... 1543
2. Entity filter.... 1544
3. Active flag 1544
4. Filtered list of embodied carbon factors 1544
5. Embodied carbon factor item.... 1545
6. Number of entities the factor currently applies to.... 1545
7. Defined factors applied to possible entities message.... 1545
8. Add factor.... 1546
9. Edit factor.... 1547
10. Remove factor.... 1549
11. Export.... 1549
12. Load and Save.... 1549
13. View options.... 1550
14. OK and Cancel.... 1550
Load Event Sequences dialog box.... 1550
1. Event sequences and submodels pane.... 1551

2a. Event sequence parameters table (Eurocode)..... 1554
2b. Event sequence parameters table (ACI)..... 1555
3. Update custom event sequences..... 1556
4. Buttons..... 1556
Loading dialog box..... 1557
1. Loadcases..... 1557
2. Load Groups..... 1559
3. Combinations..... 1560
6. Envelopes..... 1563
Materials dialog box..... 1564
Sections settings..... 1565
Material settings..... 1565
Reinforcement settings..... 1566
Decking settings..... 1567
Shear Connectors settings..... 1567
Bolts (Rods in US) settings..... 1568
Welds settings..... 1569
Model settings..... 1570
Model Settings dialog box..... 1570
Pile catalogue..... 1571
1. Pile data..... 1571
2. Add/Edit..... 1573
3. Delete..... 1575
4. OK/Cancel..... 1575
Punching shear reinforcement properties dialog box..... 1575
1. Use reinforcement..... 1576
2. Preview graphic..... 1576
3. Auto-design..... 1579
4. Select reinforcement parameters..... 1579
5. Studs parameters..... 1580
6. Buttons..... 1581
Sections dialog box..... 1581
Settings dialog box..... 1583
Simple Wind Loading dialog box..... 1583
1. Apply to (droplist)..... 1585
2. Load table..... 1585
3. Load profile 1585
4. OK and Cancel..... 1585
5. Insert Above/Below..... 1586
6. Quick Above/Below..... 1586
7. Delete..... 1586
8. Paste New..... 1587
9. Ignore diaphragms on lowest level option..... 1587
10. Pressure (droplist)..... 1587
11. Extend wind to top of building option..... 1589
12. Extend wind over full building width option..... 1591
Slab Deflection Check Catalogue..... 1592
Slab Deflection Settings dialog box..... 1593
Sub Models dialog box..... 1593

1 Get familiar with Tekla Structural Designer

Once you have Tekla Structural Designer installed, you can start getting familiar with the software and its possibilities.

The recommended learning path for new users is to run the First steps with Tekla Structural Designer interactive eLearning course. The course consists of a series of modules showing you the essentials for working with Tekla Structural Designer.

In addition to the above, to start familiarizing with Tekla Structural Designer:

- Take a moment to review Introducing (page 29).
- Check which features are available in your configuration, see Tekla Structural Designer configurations (page 31).
- Learn how to start Tekla Structural Designer for the first time and begin a project, see Start Tekla Structural Designer (page 34) and Start a new project (page 36).
- Have a tour around Tekla Structural Designer user interface and check the different ways to explore the tools in the user interface, see Introduction to the Tekla Structural Designer user interface (page 42).
- If you need support, you can contact Tekla Structural Designer support directly by using the Support tool (page 94).

1.1 Introducing Tekla Structural Designer

Tekla Structural Designer is an integrated model-based 3D tool for analysis and design (of both concrete and steel members) in multi-material structures.

Features include interactive modeling, automated structural analysis and design, drawing and report creation.

Multiple design codes are supported:

- ACI/AISC
- Eurocodes
- British Standards
- Indian Standards
- Australian Standards

Philosophy

The aim is to allow you to rapidly build your model, apply loads, and design the model for an appropriate set of design forces.

On a day to day basis, you do not need to be involved with the underlying analysis models to achieve this. Instead, you can focus on the design results.

To make this possible, Tekla Structural Designer automatically creates and analyzes multiple solver models, each one being based on a different but widely accepted approach.

By designing for the forces from all the solver models, you can be confident that each scenario has been catered for.

Tekla Structural Designer way of working

The traditional modeling, analysis and design process can be summarized in the following phases:

1. Provide a way to input or describe the model
2. Analyze the model
3. Design the model
4. Produce calculations
5. Produce drawings

Tekla Structural Designer differs slightly from the above because the analysis and design phases are merged into a single process. As a result, the Tekla Structural Designer workflow is as follows:

1. Input the geometry and loads

   A key requirement these days is BIM integration, or the ability to be able to transfer data from one application to another. Tekla Structural Designer has tools to automatically import model data from Neutral Files and from 3D DXF to facilitate BIM integration.

Of course, you can also build the model directly.

Once the physical model has been created, the next step is loading it. Tekla Structural Designer allows you to apply a wide range of loads in a flexible system of loadcases. The system contains, among other things, a wind load generator available to automatically create wind loadcases. You can also generate load combinations automatically.

You should also consider pattern loading. You can create patterned beam loads automatically, and create patterned slab loads manually for design of slabs.

2. Analyze and design the model

   Tekla Structural Designer automatically performs the analyses required to enable member design to proceed. This means that analysis and design are combined into a single automated process. The only exception to this rule is slab design.

3. Produce reports

   You can create a wide range of calculations. You can also tailor the calculations extensively to meet your requirements.

4. Produce drawings

   You can produce beam and column detail drawings, and member schedules.

1.2 Tekla Structural Designer configurations

The Tekla Structural Designer configurations that you have access to will depend on your organization's online licenses as follows:

- A subscription to Tekla Structural Designer unlocks the **Tekla Structural Designer Full** configuration which can be used for modeling, analysis and design.
- A subscription to Tedds unlocks the **Tekla Structural Designer LT** configuration. This configuration can also be used for modeling, analysis and design, but is limited to 200 members and 8000 shells.
- For students, a special student configuration is also available.

Our documentation covers the content of the **Tekla Structural Designer Full** configuration, so you may not have access to all the described features. If you have more than one configuration available you are prompted to select the one you require when you start Tekla Structural Designer.

Feature map for license configurations

	LT	Full
Modeling		

	LT	Full
Opening and viewing models	✔	✔
Modeling of 1D elements	200 members (1)	unlimited
Modeling of 2D elements	8000 shells	unlimited
Analysis		
1st order linear	✔	✔
1st order non-linear	✔	✔
1st order modal	✔	✔
2nd order linear	✔	✔
2nd order non-linear	✔	✔
2nd order buckling	✔	✔
FE chasedown	✔	✔
Grillage chasedown	✔	✔
Slab deflection analysis	✔	✔
Design (2)		
Steel member design	✔	✔
Composite steel beam design	✔	✔
Cast in place concrete member design	✔	✔
Cast in place concrete wall design	✔	✔
Cast in place slab design	✔	✔
Pad base design	✔	✔
Pile cap design	✔	✔
Mat foundation design	✔	✔
Piled mat foundation design	✔	✔
Simple connection resistance checks	✔	✔
Column base plate design	✔	✔
Drawings and reports		
Create planar drawings	✔(3)	✔(3)
Create member detail drawings	✔(4)	✔(4)
Concrete member schedules	✔	✔
Interoperability		
BIM	✔(5)	✔(5)
Open API capabilities	✔	✔

(1) - For this check the members count is taken as: beam *spans* + columns + braces

(2) - Design capabilities may vary according to the regional code being worked to.

(3) - General arrangement, beam end force, column splice load,foundation reaction, loading plan, slab/mat detailing, foundation layout

(4) - Concrete member and wall detail showing reinforcement in elevation and section, non-concrete member detail, base plate detail.

(5) - Autodesk Revit, Tekla Structures, Trimble Connect, STAAD, Robot, Tekla Portal Frame Designer, Tekla Connection Designer, IDEA Statica Connection Design, Cellbeam, FBEAM, ADAPT

How do I view license configurations available to me?

License details can be viewed from the **My licences** page of your Tekla Online Profile.

How do I switch between available configurations?

If your organization has licenses for different configurations, you can select between them when you start Tekla Structural Designer.

NOTE If you have deactivated the Select Product License dialog you won't be able to select between licenses until the dialog is reactivated. This can be done as follows:

1. Start Tekla Structural Designer
2. On the **Home** ribbon tab, select **License Manager**
3. Select **Settings**
4. Select **User Settings**
5. Select the product from the list
6. From the **Auto select** droplist, select **No (always ask me to choose)**
    a. If you want to prevent the dialog from being deactivated in future, unselect **Show "Don't ask me again" option in the Select Product License dialog.**
7. Select **Apply**, then **OK** then **Close**
8. Close and then restart Tekla Structural Designer.

How do I auto select a license?

1. Start Tekla Structural Designer
2. On the **Home** ribbon tab, select **License Manager**
3. Select **Settings**
4. Select **User Settings**
5. Select the product from the list.

6. Select the **Auto select** option you require:

Choice	Result
Yes (if only one license is available)	The Select Product License dialog is not displayed.
Default (use the default license)	The Select Product License dialog is not displayed. (The **default** license is the one that you last used.)
No (always as me to choose)	The Select Product License dialog is displayed allowing you to choose the license configuration you require.

How do I deactivate the Select Product License dialog?

1. Start Tekla Structural Designer
2. On the **Home** ribbon tab, select **License Manager**
3. Select **Settings**
4. Select **User Settings**
5. Select the product from the list.
6. From the **Auto select** droplist, select Default (use the default license)
7. Select **Show "Don't ask me again" option in the Select Product License dialog.**
8. Select **Apply**, then **OK**, then **Close**
9. Close and then restart **Tekla Structural Designer**.

How do I set the default license type?

The default license is the one that you last used.

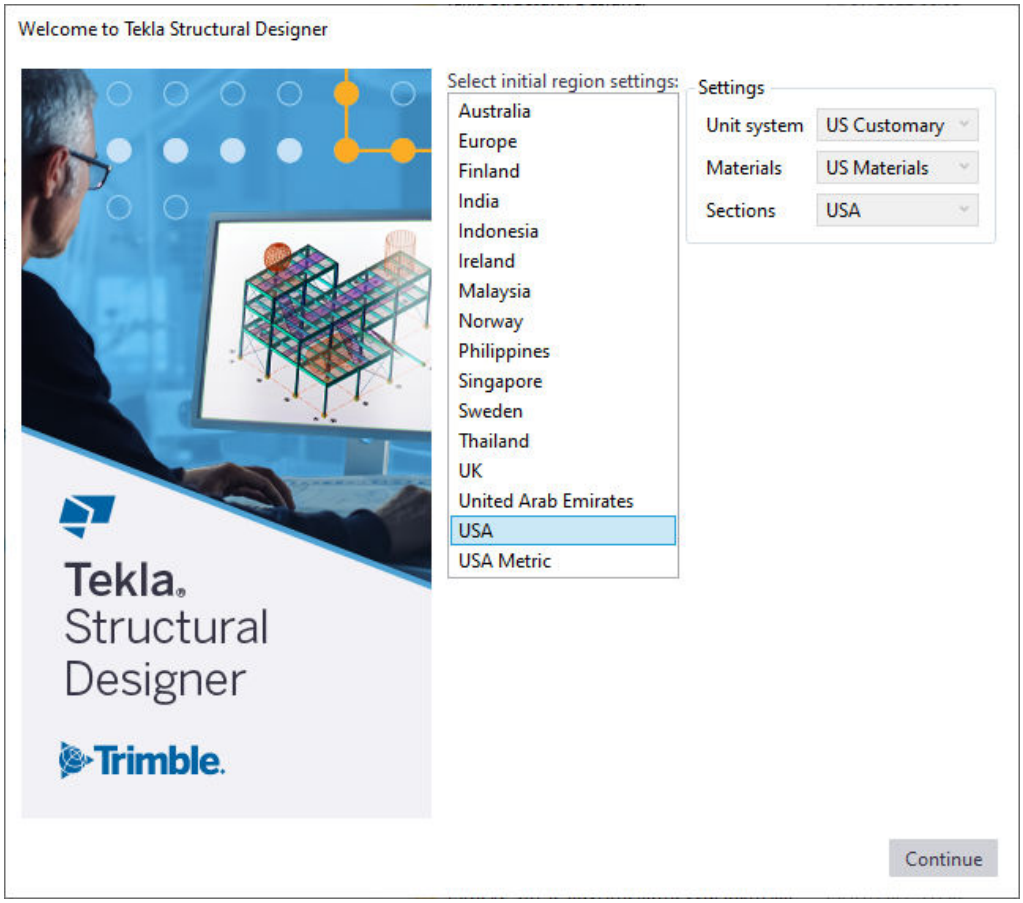
1.3 Start Tekla Structural Designer

When you start Tekla Structural Designer for the very first time, you are asked to choose a country or region. Based on your selection various region specific settings such as design codes, units and default section sizes are configured. These settings, which can be changed to meet your company requirements, are applied when you start each new project.

Choose the country for your settings

1. Start Tekla Structural Designer by selecting it from the Windows Start menu or by double-clicking the desktop icon.

If this is the very first time the program has been started, a dialog box will appear requiring you to select your region.



2. Select the country that best applies for the majority of your projects.
3. Click **OK**

   The Tekla Structural Designer interface appears, with settings configured for the selected country.

Check or change your settings

You can check or change the settings that get applied to new projects at any time.

1. On the **Home** ribbon, click **Settings** The Settings dialog opens, displaying the settings that have been configured for the selected settings set.
2. Change the settings as needed.

For more information about changing settings, see: Manage settings sets (page 1220).

NOTE Any changes that you make only apply to *new* projects, *existing* projects are unaffected.

See also

Open a project (page 36)

Start a new project (page 36)

Open a project

You can have one project open at a time. If you open a project and already have one open, Tekla Structural Designer prompts you to save the first project.

NOTE You can quickly open a recently used project, simply be selecting it from the 'Recent documents' list that is displayed on the **File** menu.

1. On the **Home** tab, click **Open**
2. Browse to the required folder, then select a project in the list.
3. To open the selected project, click **Open** or double-click the project.

   Your project opens and you will see:

   - The scene views (page 62) and (page 61) as they were displayed the last time the project was saved.
   - The (page 46) as it was displayed the last time the project was saved.

Start a new project

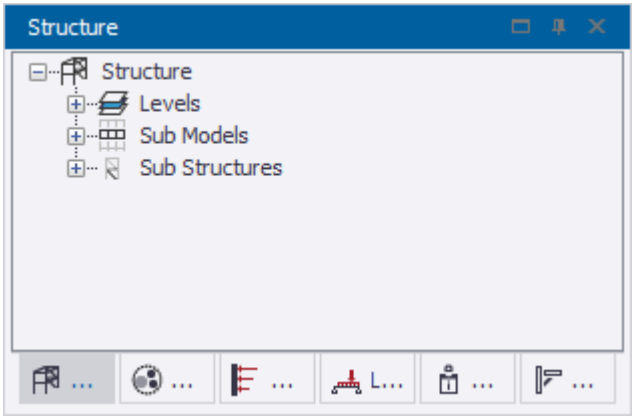
1. On the **Home** tab, click  **New**

   Your new project opens and you will see:

   - Two scene views (page 62): **Structure 3D** and **St. Base (Base) 2D**
   - A tabbed (page 46)
   - A (page 61)

2. Click the  **Structure** tab under the **Project Workspace**

A **Structure** tree is displayed in the **Project Workspace**.



3. In the **Structure** tree, click **Structure**

   The **Properties** window displays the **Structure** properties.

4. In the **Structure** tree, click **Levels**

   The **Properties** window changes to display **Levels** properties - at this point there is only one level in the model.

NOTE The units in which the levels are displayed, (along with other settings), have been copied into the model from the settings set that was active when the project was started. These are known as 'model settings'. Model settings apply to the current project only and can be edited to suit the model as required.

NOTE Once familiar with the program, some users might prefer to set up a template (page 37) to use as the basis for new projects - particularly when there is a lot of commonality between projects.

What to do next:

- Try setting out some grid lines (page 123) in the **St. Base (Base) 2D** view.
- Next, double-click **Levels** in the **Structure** tree to create some levels (page 121).
- With grids in place and levels established, you can then open views of the other levels (page 64).
- Before continuing, take a moment to review the default Model, Analysis and Design Settings etc. that have been applied to the model from the 'active' global settings set (page 1219). These can be modified if necessary as the model is developed.
- You are now ready to begin creating the model (page 120).

Use templates in new projects

If you are creating projects that share a common start point, you can use templates to avoid having to repeat inputting information.

In order to use a template, you must first create one.

A template can contain as much or as little information on the model as you consider applicable to serve as the start point for subsequent models.

For example, you can create a template that only contains a simple grid and the height of the first floor construction level.

See also

Create a new template (page 38)

Create a new project based on a template (page 38)

Create a new template

You can create a new template to simplify creating new models that have the same starting point. For more information, see the instructions below.

1. Start a new project (page 36)
2. Create the model data that you want to include in the template.
3. On the **Home** tab, click  **Save As**.

   The **Save As** dialog opens.
4. Change **Save as type** to **Tekla Structural Designer template file (*.tsmdt)**.
5. Name the template and ensure that it is saved in the right location.
6. Click **Save**.

See also

Create a new project based on a template (page 38)

Create a new project based on a template

You can use an existing template as the base of your project. For more information, see the following paragraphs.

1. On the **Home** tab, click 

   An **Open Template** option is displayed, above which any recently used templates are also listed.
2. Either select a recently used template, or open a specific template.

The template opens.

To continue, the template should now be saved as a project.

3. On the **Home** tab, click  **Save As**

   The **Save As** dialog opens.

4. Change the **Save as type** to **Tekla Structural Designer project files (*.tsmd;*.cscbd)**.
5. Name the file and ensure that it is saved in the right location.
6. Click **Save**.
7. Add the necessary building objects and loads in order to complete the project.

See also

Create a new template (page 38)

Work with autosave and backups

Backups ensure that whatever happens there can be previous versions of the model that can be reverted to.

An added benefit of backups is that they provide a limited (per user setting) history of the model.

Following a program error, an autosave provides a single shot at reverting back to a working model.

The difference between autosave and backup

By having both features the user can be confident that they always has a file with the most recent changes and a backup if that fails.

Autosave	Backup
The auto save file is a single temporary file constantly overwritten after a specified interval.	Multiple backup files are created which are only overwritten when a specified maximum number is reached. Each backup is only created if, after the specified interval, there have been changes since the last backup.
Provides a single shot at reverting back to a working model following an error.	Can be opened as and when required.

Autosave	Backup
Once the model becomes corrupt the autosave file won't help.	Backups can be used to restore versions of the model prior to corruption.
The autosave file is a core save of the model.	Similar to autosave, backups are also a core save of the model.
Typically saved at shorter intervals than backup.	Typically saved at longer intervals than autosave.

Enable autosave

1. On the **Home** tab, click **Settings**
2. Click **General** --> **Autorecover**
3. Under **Autosave**, check the **Enabled** box and set the **Interval** as required.
4. Click **OK**.

Enable backups

1. On the **Home** tab, click **Settings**
2. Click **General** --> **Autorecover**
3. Under **Backups**, check the **Enabled** box and set the **Interval** as required.
4. Accept the default location, or browse to another location if required.

   Both local and network locations are allowed.
5. Click **OK**.

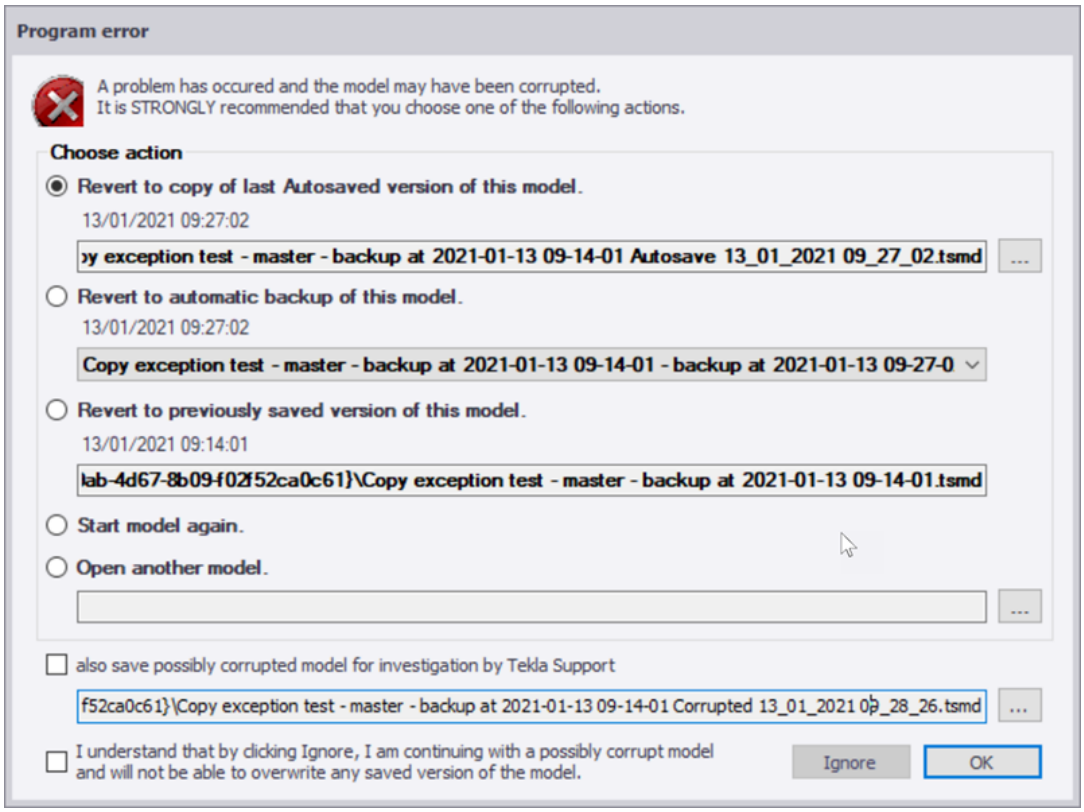
NOTE Backup files are saved with read-only permission.

NOTE If required you can adjust the limit on the number of backups. Once the limit is reached the older backups start to be automatically deleted.

Automatically restore an autosave or backup following a Program error

To minimize the potential for lost work and possible model data corruption, a 'Program error' dialog can be displayed when an error is detected. As shown

below, when an error is detected you are presented with a number of action options which cater for the potential situations of the model.



We strongly recommend that you **do not click Cancel**, as you would then be working with a potentially corrupted file. Instead we advise that you always choose one of the action options and then **OK**.

- Where an autosave file exists, the top option will be available and is typically the recommended choice, (although depending on the intervals you have set, if both an autosave and a backup exist, it is possible the backup could be the most recent version of the model, in which case the second option should be used).
- If a previously saved version of the file exists, the third option will be available (whether or not autosave or backup files exist) listing the last previous save of the model. This is the recommended choice if no autosave or backup file exists.
- If there is a preferred previous file version the engineer wishes to revert to, this can be selected via the option “Open another model”.
- If the model has only just been started, then none of the above scenarios apply and so the remaining option “Start model again” is the recommended choice.
- There is a final option to save the possibly corrupted model for submission to your local Support Team for investigation. We would be most grateful if engineers could use this option to help us with continuing to improve the

program. When submitting such a file, please try and include as much information as possible about the steps undertaken in the program just prior to the error.

Manually restore a backup

Provided you have the backup facility enabled you will have previous versions of the project available to revert to if required. Backups work by periodically saving the model after a specified interval. Each backup is saved to a new name up to the point when the user definable maximum number of backups is reached, after which the older backups are overwritten.

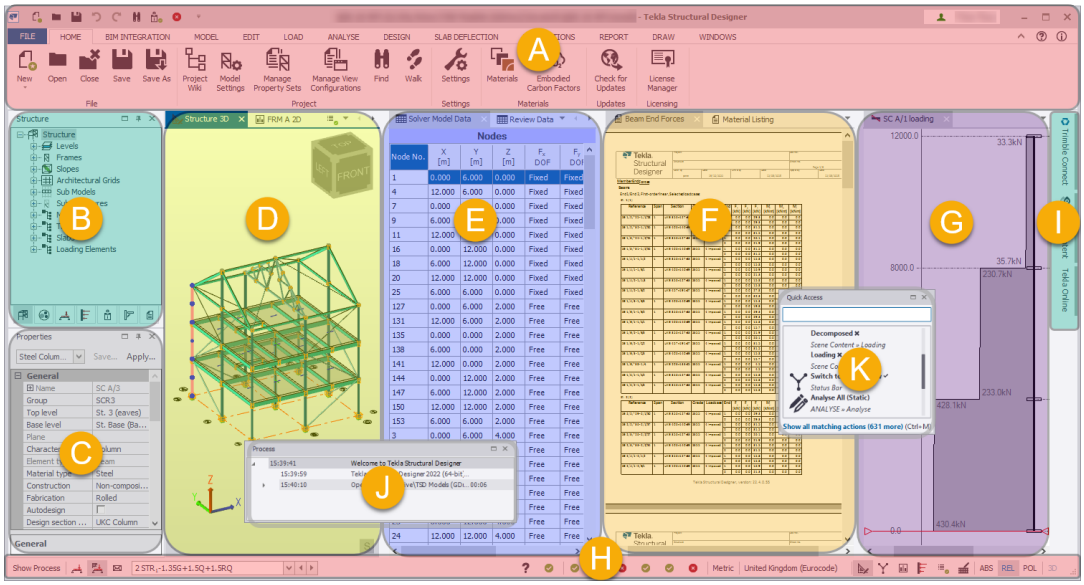
1. On the **File** menu, click **Model Backups** The available backups for this Model ID are listed.

   NOTE The list is for information only, backups cannot be restored directly by selecting them from the list.

2. Click on the Backup path: to open the folder containing these backups.
3. Double-click on the required backup file to open it in another session of Tekla Structural Designer (you may need to exit from the previous session at this point if you only have a single license.) The backup model opens, but note that this is read-only.
4. Click **Save As** from the **File** menu to save the backup to a new name.

1.4 Introduction to the Tekla Structural Designer user interface

Components of the user interface are labeled on the below key-plan.



Click the links to find out more:

(A) Quick Access Toolbar, Ribbon, Sign in, and Help (page 43)

(B) Project Workspace windows (page 46)

(C) Properties window (page 61)

(D) Scene view (page 62)

(E) Tabular data view (page 75)

(F) Report View (page 77)

(G) Load Analysis View (page 77)

(H) Loading List and Status Bar (page 78)

(I) Side panes (page 81)

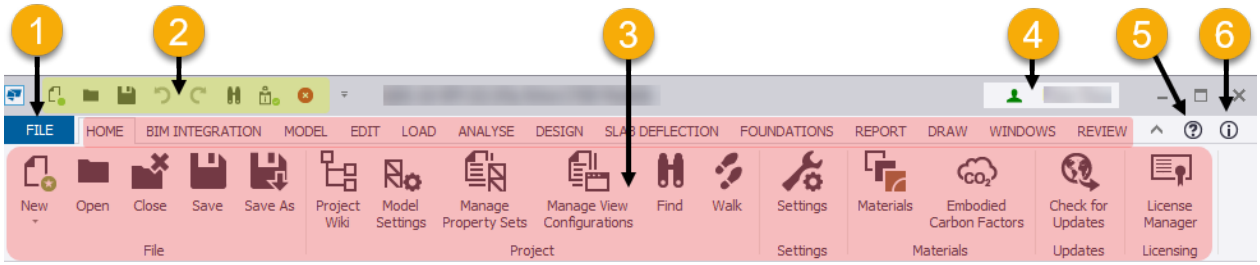
(J) Process window (page 83)

(K) Quick Access window (page 84)

See also

Keyboard shortcuts (page 88)

Overview of Quick Access Toolbar, Ribbon, Sign in, and Help



1. File menu

The **File** menu contains those commands that you can use to perform file-related operations.

Button	Description
New	Create a new blank project from scratch.
Open	Open an existing project.
Save	Save the currently open project.
Save As	Save the currently open project with a new name, or as a template.
Save Model Only	Save smaller model-only files without the analysis results. The model-only files can easily be shared amongst the project team.
Close	Close the currently open project.
Print	Print the active view.
Start Revision	Record changes to this revision of the project.
Send As Email	Create a new email with the project attached.

Button	Description
Contact Support	Automatically upload your model file and send it, together with your issue description and key information (needed by Support to assist you) about your installation and system, to your Local Support Team as defined in your Tekla Online Profile.
Exit	Prompts you to save any open project, and then closes Tekla Structural Designer.

2. Quick Access Toolbar

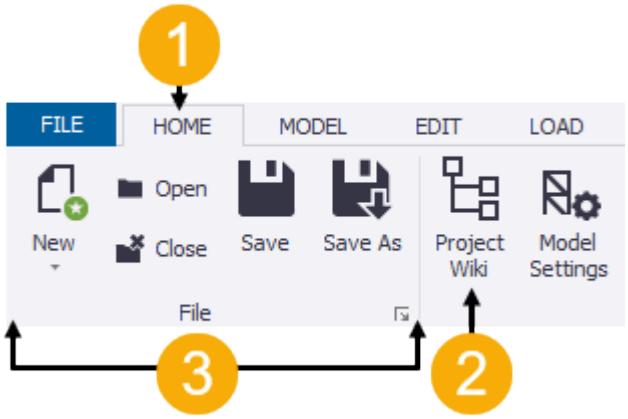


The **Quick Access Toolbar** provides access to the following commonly used commands:

Button	Description
New	Create a new blank project from scratch.
Open	Open an existing project.
Save	Save the currently open project.
Undo	Undo the last action.
Redo	Redo the last undone action.
	Find and then select objects in the model by typing a part of their name.
	Validate the structure for physical model and design issues.
Delete	Delete the selected item.

3. Ribbon

The ribbon contains tabbed toolbars which provide access to related commands organized into logical groups.



1. Ribbon tab
2. Ribbon command
3. Ribbon group

4. Sign in

Signing in to your Trimble Identity gives you access to a greater number of online resourses.

You can sign in to one service and then browse to another online service without a need to log in again. Find the services landing page here: http://www.tekla.com/services

5. Help

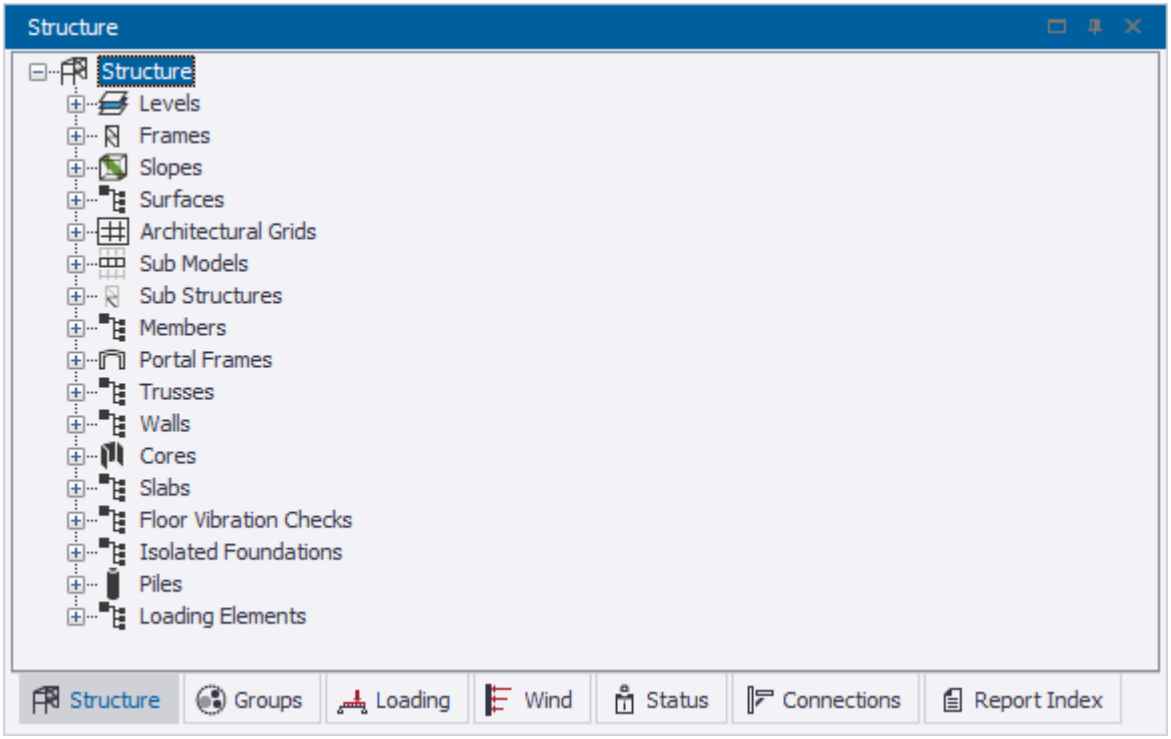
Opens the online version of this product guide in Tekla User Assistance. You can also press F1 to access the product guide. When a dialog box is active you can press F1 to quickly access the correct help page in the product guide.

6. About

Displays the program version number, your customer number and provides access to the Trimble privacy policy.

Overview of the project workspace

The **Project Workspace** consists of several tabbed windows.



Structure

The Structure window contains a tree for opening scene views, viewing and modifying model properties, and for controlling grids and sub models.

- See also: View and modify model properties in the Project Workspace (page 48)

Groups

The Group window contains a tree for organizing the members in your model into design groups. It also allows you to manage design groups according to your needs.

- See also: Manage groups in the Project Workspace (page 54)

Loading

The Loading window contains a tree which lists the individual loads that have been applied in each loadcase. After analysis it sums the applied loads in each loadcase and combination and checks these against the sum of reactions.

- See also: View load status in the Project Workspace (page 57)



Wind

The Wind window contains a tree which is used after completing the wind modeling process to display wind direction views, modify wind directions, and modify wind loadcases.

- See also: View and modify wind properties in the Project Workspace (page 58)



Status

The Status window contains the a tree for reviewing the validation messages and other model status indicators.

- See also: View model status in the Project Workspace (page 59)



Connections

The Connection window contains a tree which organizes the model's steel connections in a hierarchical way, by Type (i.e. Type/Reference), and by Group (i.e. Type/Section size/Reference).

- See also:



Report Index

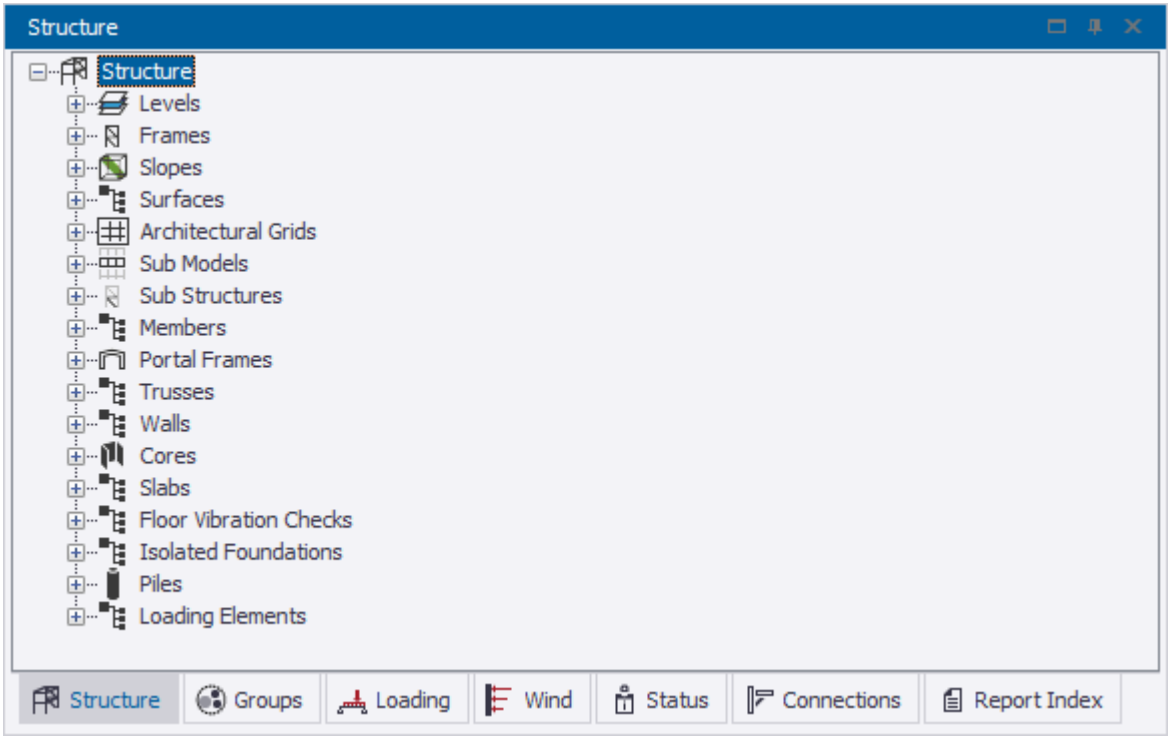
The Report Index window contains a tree with bookmarks that can be used to move around the report when a report view is displayed.

- See also: Navigate reports (page 1073)

TIP If you click **X** in the top right corner to close of any of the above windows, you can re-open them from the **Windows** tab of the ribbon. For more information, see: How to hide, re-display and move windows (page 87)

View and modify model properties in the Project Workspace

The **Structure** tree organizes the model geometry in a hierarchical way. It also allows you to view and modify model properties.



When you open a new model, the **Structure** tree contains two sub-branches: **Levels** and **Sub Models**. As the model geometry develops new branches are added, as shown above.

When a branch or sub-branch is selected, the common properties of the selected branch are viewed in the **Properties** window. You can then modify the common properties in the **Properties** window.

View and modify structure properties

Structure properties, or building parameters, control the principal direction and default meshing properties of the building.

1. In the **Structure** tree, click  **Structure**.

   The structure properties are viewed in the **Properties** window.
2. Modify the structure properties according to your needs.

View and modify construction level properties

- In the **Structure** tree, do one of the following:

To	Do this

View and modify the properties common to all construction levels	1. Click **Levels**. The construction level properties common to all levels are viewed in the **Properties** window. 2. Modify the properties according to your needs.
View and modify the parameters of a single construction level	1. Click the **+** sign next to **Levels**. All construction levels in the model are viewed. 2. Click the desired construction level. The level properties are viewed in the **Properties** window. 3. Modify the level parameters according to your needs.

View and modify frame properties

The **Frames** branch is added in the **Structure** tree when you create the first frame in the model.

1. In the **Structure** tree, click the **+** sign next to **Frames**.
   All frames in your model are listed.
2. Click the desired frame.
   The frame properties are viewed in the **Properties** window.
3. Modify the frame properties according to your needs.

View and modify the slope properties

The **Slopes** branch is added in the **Structure** tree when you create the first frame in the model.

1. In the **Structure** tree, click the **+** sign next to **Slopes**.
   All slopes in your model are listed.
2. Click the desired slope.
   The slope properties are viewed in the **Properties** window.
3. Modify the slope properties according to your needs.

View surfaces

The **Surfaces** branch is added in the **Structure** tree when you create a horizontally curved truss.

1. In the **Structure** tree, click the **+** sign next to **Surfaces**.

   All surfaces in your model are listed.

2. Double click the desired surface to open a view of the surface.

View and modify grid properties

- In the **Structure** tree, do one of the following:

To	Do this
Modify the color, name, or visibility of a grid	1. Click the **+** sign next to  **Architectural Grids**. All architectural grids in your model are viewed. 2. Click the desired grid. The grid properties are viewed in the **Properties** window. 3. Modify the color, name and visibility of the grid according to your needs.
Renumber all grids	1. Right-click  **Architectural Grids**. 2. In the context menu, click **Renumber**.

View and modify sub model properties

1. In the **Structure** tree, click the **+** sign next to  **Sub Models**.

   The sub models in your model are listed.

2. Click the desired sub model.

   The sub model properties are viewed in the **Properties** window.

3. Modify the properties according to your needs.

TIP To open the **Sub Models** dialog dialog, double-click  **Sub Models**.

View and modify sub structure properties

See: Create and manage sub structures (page 1296)

View and modify member properties

Tekla Structural Designer classifies members by material and type, and further by fabrication and by shape. Members are classified by material and type, then further classified by fabrication and then by shape.

- In the **Structure** tree, do one of the following:

To	Do this
View and modify common properties for members of a particular type and fabrication	1. Click the **+** sign next to  **Members**. The existing member types are viewed. 2. Click the **+** sign next to a member type. The existing fabrication types are listed. 3. Click a fabrication type. Common properties of all members of the fabrication type are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View and modify common properties for members of a particular type, fabrication and shape	1. Expand the  **Members** branch and its sub branches by clicking them. 2. Click the **+** sign next to the desired fabrication type. The shapes of the fabrication type are listed. 3. Click a shape. The common properties of all members of the selected shape are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View and modify the properties of an individual member	1. Expand the  **Members** branch and its sub branches by clicking them. 2. Click the **+** sign next to the shape type. The member references of the shape are listed. 3. Click a member reference. The properties of the member are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View a member in a new window, select it in visible views, delete it, or modify it in the **Properties** dialog box	1. Expand the  **Members** branch and its sub branches by clicking them. 2. Right-click a member reference.

	3. In the context menu, select the desired option.

View and modify slab properties

- In the **Structure** tree, do one of the following:

To	Do this
View and modify properties common to all slabs	1. Click **Slabs**. The common properties to all slabs are viewed in the **Properties** window. 2. Modify the properties according to your needs.
View and modify the properties of a parent slab	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Click the desired parent slab. The properties of the parent slab are viewed in the **Properties** window. 3. Modify the properties according to your needs.
Modify the properties of a parent slab, or delete it	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Right-click a parent slab. 3. In the context menu, select if you want to modify the parent slab in the, or delete it.
View and modify the properties of a slab item	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Click the **+** sign next to a parent slab. All the slab items within the parent slab are viewed. 3. Click a slab item. The properties of the slab item are viewed in the **Properties** window. 4. Modify the properties according to your needs.

View and modify wall or roof panel properties

1. In the **Structure** tree, click the **+** sign next to **Walls** or **Roofs**.

   The currently defined walls and roof panels are listed.

2. Click the desired wall or roof panel.

   The properties of the selected wall or roof panel are viewed in the **Properties** window.

3. Modify the properties according to your needs.

View and modify result strip properties

1. In the **Structure** tree, click the **+** sign next to **Result Strips**.

   All the currently exiting result strips are listed.

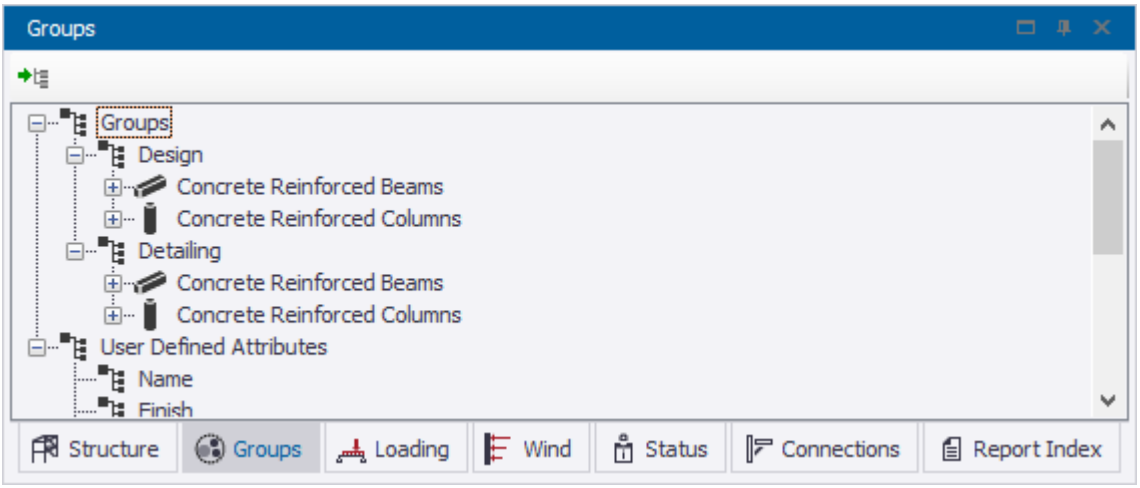
2. Click the desired result strip.

   The properties of the selected result strip are viewed in the **Properties** window.

3. Modify the properties according to your needs.

Manage groups in the Project Workspace

The **Groups** tree organizes the members in your model into design groups. It also allows you to manage design groups according to your needs.



If you have defined concrete members, they are also organized into further groups for detailing purposes. The application of grouping is most useful when handling concrete structures.

User defined attributes (UDAs) are also listed, allowing you to see which members have been assigned specific UDAs.

NOTE You can right-click on a group name (or UDA) in order to select all members in the group, (or with that UDA), simultaneously in the visible views.

Regroup a specific member type

1. In the **Groups** tree, right-click the member type you want to re-group.
2. In the context menu, select **Regroup Members**.

   The selected member type is regrouped.

Re-group all member types

WARNING Re-grouping all member types undoes any manual grouping you may have done.

- Click the **Re-group ALL Model Members** button on the top left corner of the **Groups** tree.

Add group

1. In the **Groups** tree, select the appropriate branch within which you want to manually create a new group.
2. In the context menu, select **Add Group**.

   A new group empty group is created.

Set as default group

When a new empty group is set as default, if you create a new member it will be placed in that group in preference to another empty group.

1. In the **Groups** tree, select an empty group.
2. In the context menu, select **Set As Default Group**.

   An asterix is placed next to the group name to indicate that it is the default group for new members.

Split a member group into smaller groups

1. In the **Groups** tree, right-click the name of the group that you want to split.
2. In the context menu, select **Split Group**.

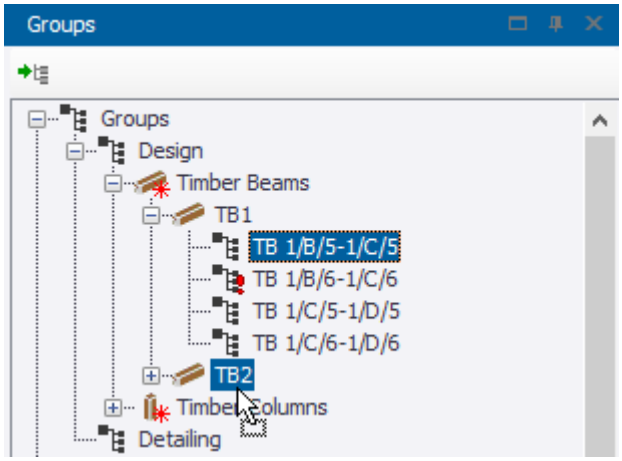
   All members of the group are ungrouped and placed into individual groups.

Move members from one group to another

1. In the **Groups** tree, open the group that contains the member(s) to be moved.
2. Click the first member name to select it, then if you want to add further members to the selection, hold down **Ctrl** while clicking on their names.

3. Drag the selection over the group name where you want to move it.

   Provided that the selection meets the geometric criteria to belong to the group, a small rectangle will be displayed alongside the cursor.



NOTE You must drag over the **group name** as shown above, rather than over the group content for the rectangle to appear.

4. Release the mouse button.

TIP You can also change the group to which members belong by selecting them and then updating the group name in the **Properties** window.

Rename a member group

1. In the **Groups** tree, right-click the group that you want to rename.
2. In the context menu, select **Rename Group**.
3. Type the new name of the group and press Enter.

Modify the group name defaults for future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**, and select the settings set that you want to edit.
3. Go to the **Grouping** page.
4. Review and modify the default group names according to your needs.
5. To save the changes, click **OK**.

Check a member group

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Check Group**.

   The selected member group is checked.

Check a member group using Tekla Tedds

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Check Group**.

   The selected member group is checked using Tekla Tedds.

Design a member group

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Design Group**.

   The selected member group is designed.

Remove a member group

1. In the **Groups** tree, right-click the name of the group you want to remove.
2. In the context menu, select **Remove Group**.

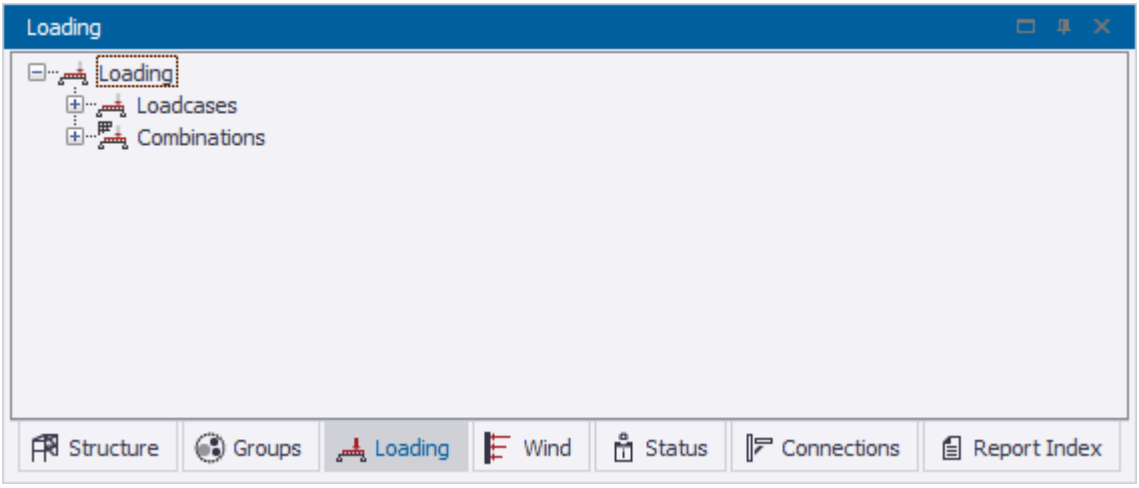
   All members are ungrouped and placed into individual groups.

See also

Create and manage user-defined attributes (page 1292)

View load status in the Project Workspace

The **Loading** tab lists the individual loads that have been applied in each loadcase. After analysis it sums the applied loads in each combination and checks these against the sum of reactions.



See the status options in the following table:

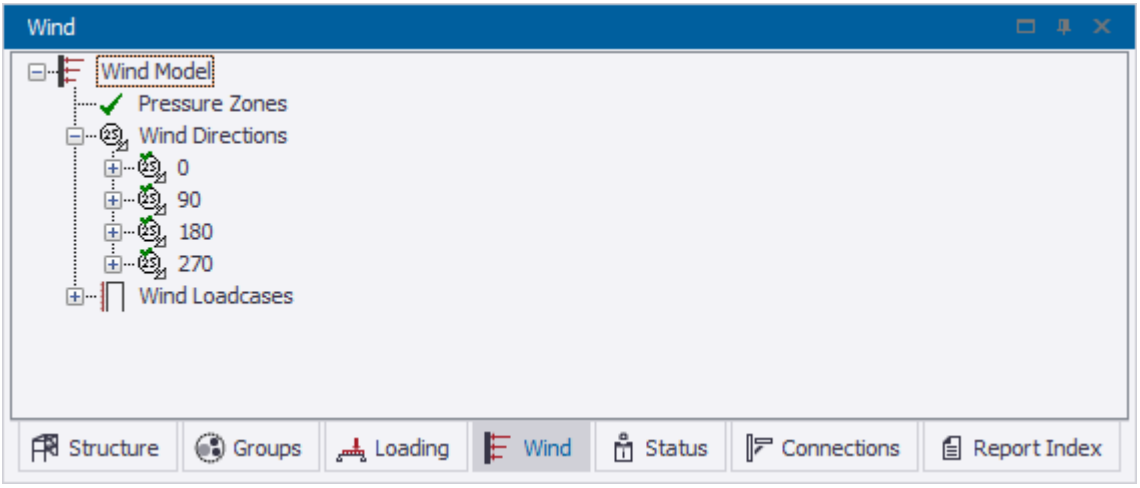
Icon	Meaning
✓	**Total Reaction** is in equilibrium with the **Total Load on Structure**.
✖	**Total Reaction** is not in equilibrium with the **Total Load on Structure**.
?	**Total Reaction** is unavailable.

1. In the **Project Workspace**, go to the **Loading** tab.
2. Click the desired loadcase.

   The properties of the loadcase are viewed in the **Properties** window.

View and modify wind properties in the Project Workspace

The **Wind Model** tree is used after completing the wind modeling process to display wind direction views, modify wind directions, and modify wind loadcases.



View a wind direction view

1. In the **Wind Model** tree, click **Wind Directions**.
2. Right-click the desired wind direction.
3. In the context menu, select **Open View**.

Modify wind direction properties

1. In the **Wind Model** tree, click **Wind Directions**.
2. Click the desired wind direction.

   The properties of the wind direction are viewed in the **Properties** window.
3. Modify the properties according to your needs.

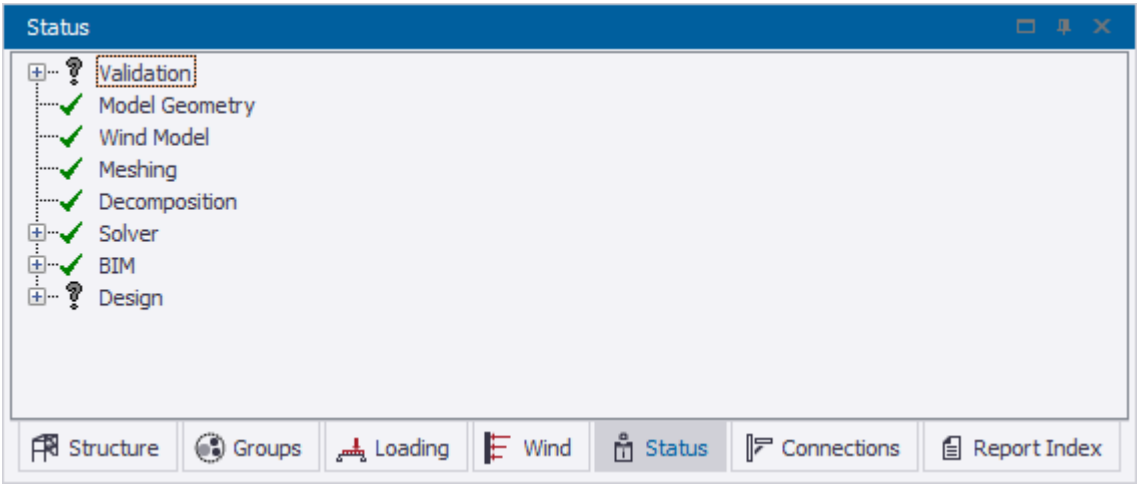
View and modify wind loadcases

1. In the **Wind Model** tree, click **Wind Loadcases**.
2. Click the desired wind loadcase.

   The properties of the wind direction are viewed in the **Properties** window.
3. Modify the properties according to your needs.

View model status in the Project Workspace

The **Status** tree is used to review the validation messages and other model status indicators.



You can review:

- The validation status for the model

  For more information, see: Validate the model (page 323)
- The validation status for the analysis

  The analysis model validation is performed automatically when you analyze the model.
- The decomposition status

  The **Decomposition** branch shows if load decomposition has been successfully completed.
- The solver status

  The **Solver** branch lists the analyses that have been performed.
- The BIM validation

  If the model has been imported or exported, the **BIM** branch lists any warnings and errors that relate to the import or export process.
- The status of drift, sway, wind drift, seismic drift checks and displacements

  Each of these are reported automatically when the appropriate analysis has been performed. For more information, see: Check drift, sway, seismic drift, wind drift, and overall displacements (page 822)

NOTE

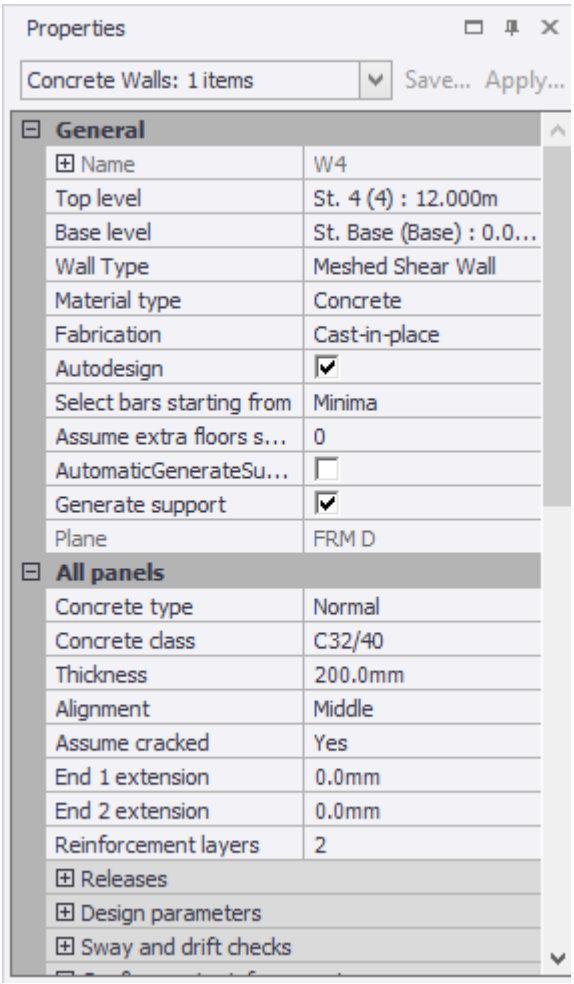
- ✓ indicates that the analysis or process has been successfully completed.
- ✖ indicates that the analysis or process has resulted in an error.
- ⚠ indicates that the analysis or process has resulted in a warning.

Overview of the Properties window and Properties dialog box

Properties window

The **Properties** window facilitates the input, review and editing of model properties.

By default it is docked at the bottom left of the main window, but you can reposition it if required.



TIP If you click **x** in the top right corner to close the window, you can re-open it from the **Windows** tab of the ribbon.

It can be used to:

- Specify properties when running a command from the ribbon.
- Review or edit existing properties (page 111) when individual or multiple items are selected in a scene view.

- Save properties to and recall properties from property sets (page 1285).

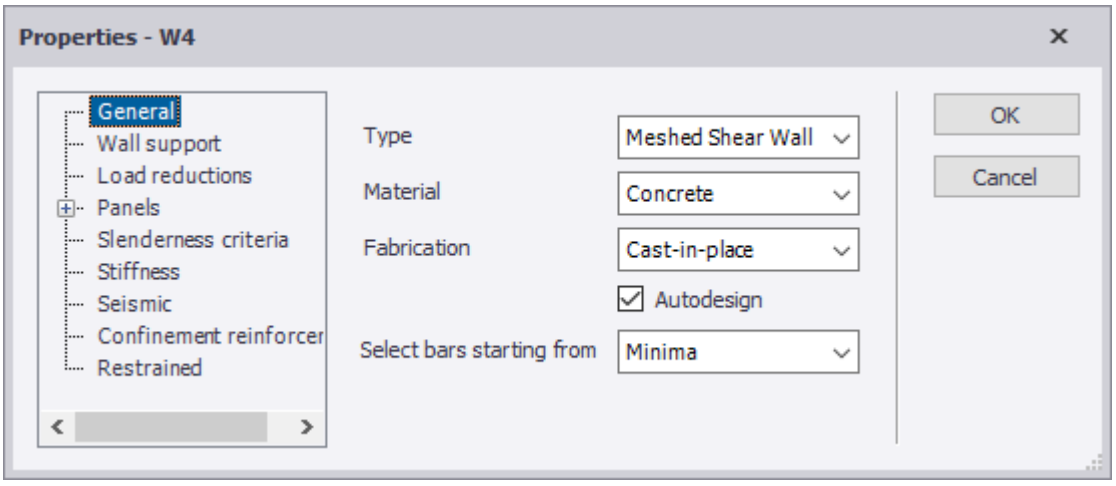
When it is being used to review or edit existing properties:

- If an individual item is selected, all of its properties are displayed.
- If multiple items of the different types are selected, the list at the top of the **Properties** window enables you to choose the item type for which properties are to be displayed.
- When multiple items of the same type are selected, only those property values that have the same value for each item are displayed, property values that differ are left blank.
- If a property is displayed but its value is greyed out, this indicates that it can not be changed.

NOTE Some item properties are not suited for display in **Properties** window and can only be accessed via the **Properties** dialog box. In steel members, web opening properties are one such example.

Properties dialog box

The **Properties** dialog box provides an alternative method for editing an individual item. It can be opened by selecting an individual item in a Scene View and then selecting **Edit** from the right-click context-sensitive menu.



See also

Manage properties and property sets (page 1284)

Properties reference (page 1329)

Overview of scene views

A **scene view** displays the model or a part of the model in a tabbed 3D or 2D window.

Various interface components (page 68) are displayed within each scene view, depending on whether it is in 3D or 2D.

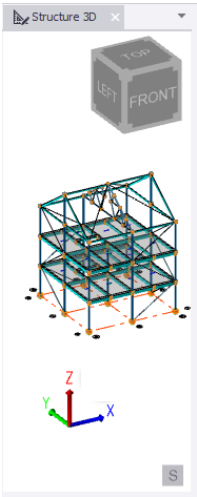
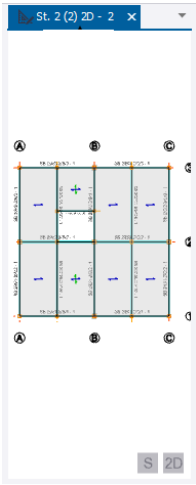
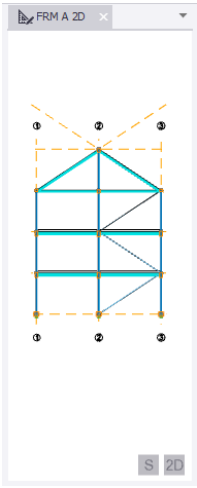
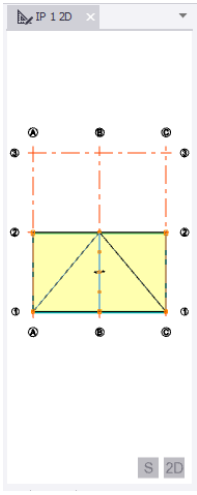
The functionality of a scene view depends on the view regime (page 73) that has been selected.

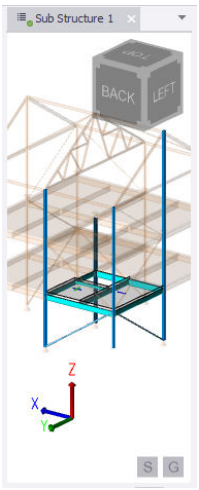
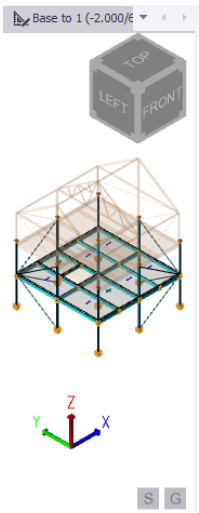
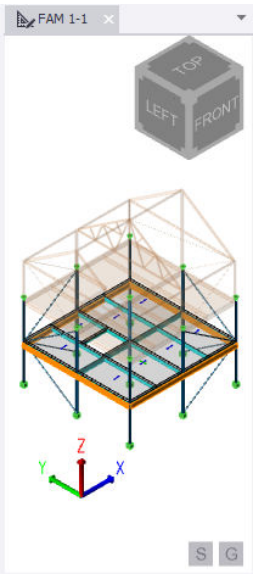
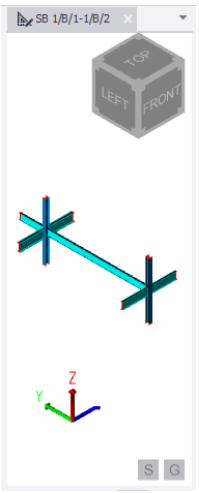
The level of content displayed in a scene view is controlled via the scene content window (page 66).

Once the level of content has been set as required you can choose to save it as a view configuration (page 67).

If multiple scene views are open you can arrange them in tab groups (page 73).

Example scene view categories are illustrated below:

Structure	Level	Frame	Sloped Plane
			
A 3D view of the entire model	A 2D horizontal view of a model level. See: Create and manage construction levels (page 121)	A 2D vertical view of a model frame, defined by grid lines. See: Create a frame (page 146)	A 2D view of a user-defined inclined plane. See: Create a slope (page 147)

Sub Structure	Sub Model	FAM	Member
			
A 3D view of a user-defined sub-set of the model which you can edit, design, or report in isolation. See: Create and manage sub structures (page 1296)	A 3D view of a smaller analysis model generated during chasedown analysis. See: Sub models (page 616)	A 3D view of a floor analysis model generated during footfall assessment. See:	A 3D view of a Member. See: Open a Member 3D view (page 66)

See also

Open and close scene views (page 64)

How to control scene content (page 66)

Overview of scene view interface components (page 68)

Work with view configurations (page 67)

Create and modify view tab groups (page 73)

Open and close scene views

Open a Structure 3D view

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, double-click  **Structure**. (Alternatively, right-click and select **Open view** from the context menu.)

A Structure 3D view is displayed.

Open a Level 2D view

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, click the **+** sign next to  **Levels**. The existing levels are listed.
3. Double-click on the required Level. (Alternatively, right-click and select **Open view** from the context menu.)

A Level 2D view is displayed.

Open a Frame 2D view

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, click the **+** sign next to  **Frames**. The existing Frames are listed.
3. Double-click on the required Frame. (Alternatively, right-click and select **Open view** from the context menu.)

A Frame 2D view is displayed.

Open a Sloped Plane 2D view

NOTE Before you can view a 2D view of a sloped plane, you must create a sloped plane in your model.

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, click the **+** sign next to  **Slopes**. The existing Slopes are listed.
3. Double-click the required Slope. (Alternatively, you could right-click and select **Open view** from the context menu.)

A Slope 2D view is displayed.

Open a Sub structure 3D view

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, click the **+** sign next to **Sub Structures**.
3. The existing sub structures are listed.
4. Double-click the required sub structure. (Alternatively, you could right-click and select **Open view** from the context menu.)

A Sub structure 3D view is displayed.

Open a Sub model 3D view

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, click the **+** sign next to  **Sub Models**.
3. The existing sub models are listed.
4. Double-click the required sub model. (Alternatively, you could right-click and select **Open view** from the context menu.)

A Sub model 3D view is displayed.

Open a FAM 3D view

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, click the **+** sign next to **Floor Analysis Models**.
3. The existing FAMs at each level are listed.
4. Right-click the required FAM, and from the context menu select **Edit**.

The FAM 3D view is displayed.

Open a Member 3D view

1. In a Scene View, hover the mouse over the required member to highlight it.
2. Right-click to open the context menu.
3. Select **Open Member View** from the context menu.

A Member view is displayed.

Close views

- Click **x** on the top right corner of the view tab.

How to control scene content

Different entity types have different levels of information associated with them. You can select how much of this information is displayed in each of the different scene views and view regimes. Scene content selections are saved independently with each scene view.

For example, in a **Solver View** regime, it is generally sufficient to represent beams by their insertion lines. However, in a **Structural View** regime, you are likely to also include their geometric outlines. In either of the views, you may also choose to display their direction arrows and possibly their reference texts also.

Open the Scene Content window

If **Scene Content** is set to **Auto Hide** the **Scene Content** tab will be docked on the edge of the main window:

- Click the tab to expand the **Scene Content** window.

Or, if the **Scene Content** window has been closed:

- You can re-open it from the **Windows** tab of the ribbon.

Select items in the Scene Content window

- To view the sub categories of an entity category:
  - Click the arrow sign on the left side of the category name.
- To view a category in the model view:
  - Select the checkbox on the left side of the category name.
- To hide a category in the model view:
  - Clear the checkbox on the left side of the category name.

NOTE If you hide an entity category, you can no longer perform some commands that affect that entity category.

For example, if you hide slab items, you cannot define slab or area loads on a floor because there are no slab panels to select.

- To adjust the information viewed in the model view:
  1. Click the cell on the right side of the category name.

     A list of possible details opens.
  2. Select the details you want to view.
  3. Click **OK**.

Reinstate the default Scene Content selections

You can discard your current selections and reinstate the default **Scene Content** selections at any time. This is done as follows:

1. Close the current view.
2. Re-open the view using the **Structure** tree.

See also

Open, close and save scene views (page 64)

Work with view configurations

Once a view has been set up as required you can save a view configuration.

View configurations can be used in two ways:

- They can be included as views in model reports.

  In this case, the view configuration retains the scene content settings that were in place when it was saved.
- They can be re-opened in new views.

  In this case, the view configuration adopts the scene content settings that are currently in place in the new view.

Save a view configuration

1. Right-click anywhere in the view.
2. In the context menu, select **Save View Configuration...** The **View name** dialog box opens.
3. Name the view.
4. Click **OK**.

Open a saved view configuration

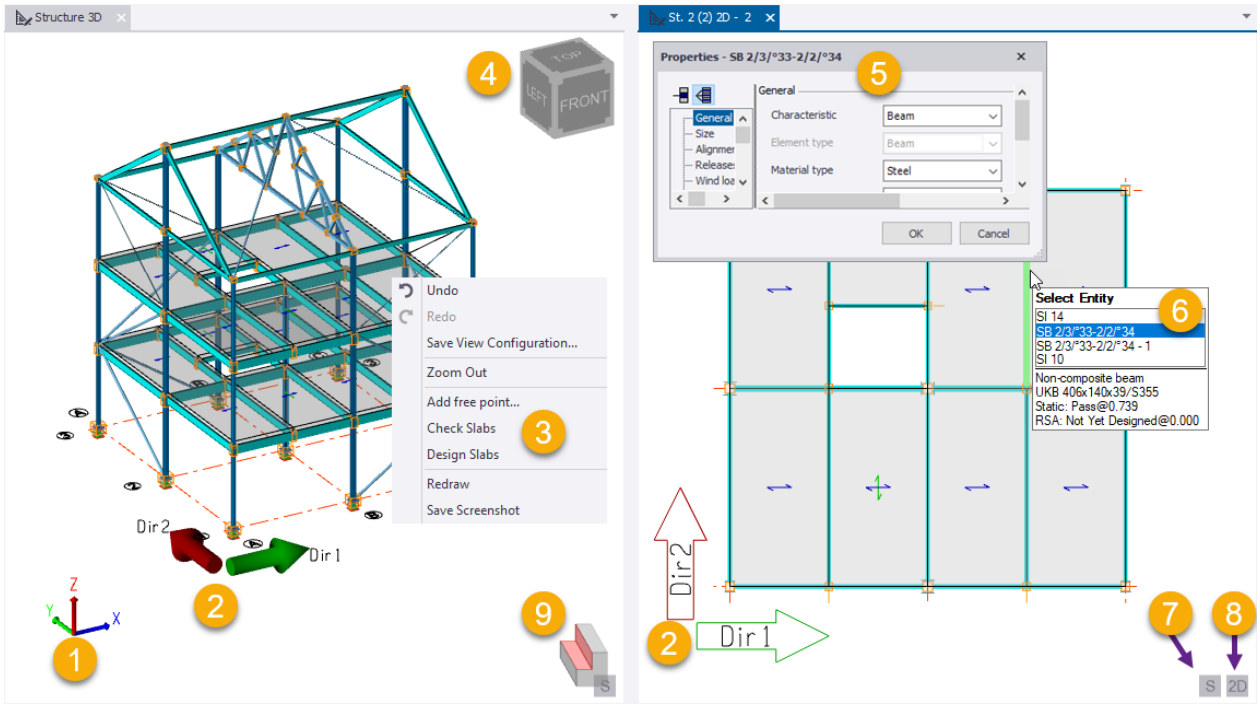
1. On the **Home** tab, click  **Manage View Configurations**.
2. Select the desired view configuration.
3. Click **Open View**.
4. Click **OK**.

Delete a view configuration

1. On the **Home** tab, click  **Manage View Configurations**.
2. Select the desired view configuration.
3. Click **Delete**.
4. Click **OK**.

Overview of scene view interface components

The main interface components are labeled in the 3D scene view (left) and 2D scene view (right) shown below:



(1) The **Global XYZ axis system** within which all other systems exist.

(2) The **Building Directions** axis system indicates the principle axes of the building and can be rotated about global Z if required. These can be switched on/off (and their labeling can be controlled) via Structure Properties (page 1331).

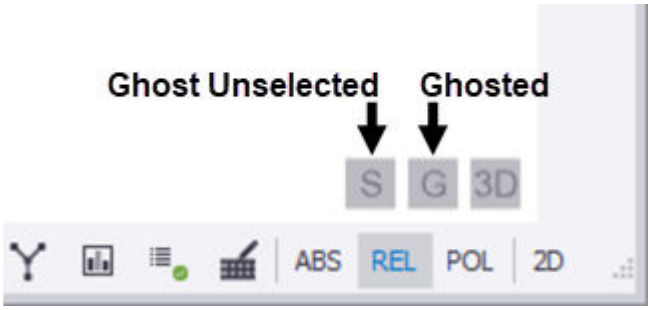
(3) Right-click anywhere within a 2D or 3D view to display a menu that is context-sensitive to the item that is currently highlighted.

(4) In 3D views, you can click a vertex, edge, or face of the **ViewCube** to rotate the model to a preset view.

(5) The **Properties** dialog box can be opened by selecting **Edit** from the right-click context-sensitive menu. It is used as an alternative to the **Properties** window for viewing and editing parameters associated with an individual model object.

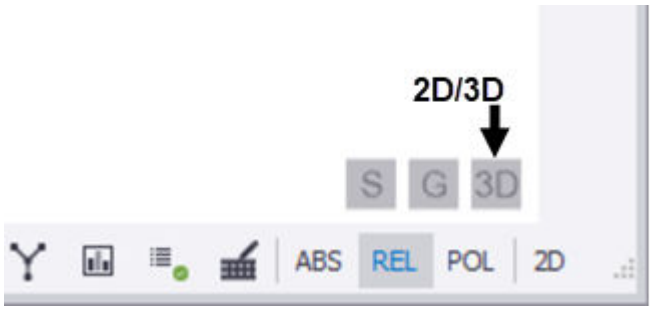
(6) The program is automatically in 'select mode' when no other commands are being performed. In this mode you can hover the cursor over an entity and its name will be displayed in the **Select Entity** tooltip. When the correct entity is displayed, click the entity to select it. If several entities are displayed, you select the one required by using the Tab key or Up/Down arrow keys.

(7) The **Ghost Unselected** S button is displayed in all 2D and 3D Views. It is used to toggle the display of selected and unselected objects, making it easier to focus on a particular subset of objects within the model. See: Use Ghost Unselected to focus on the selection (page 108)

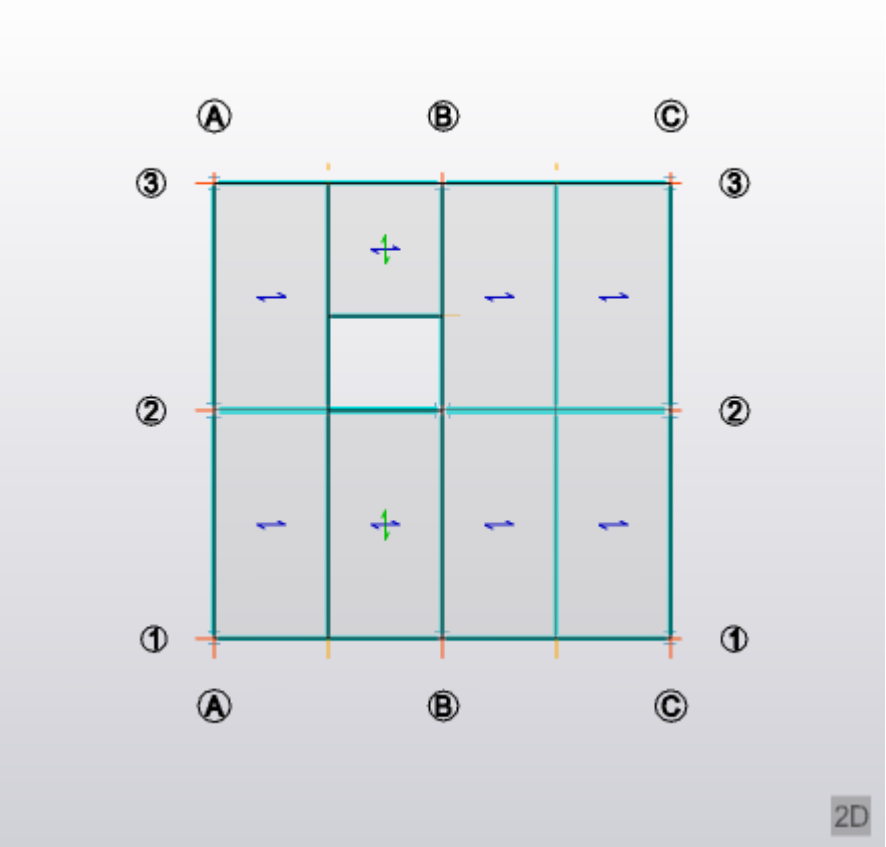


Similarly, the **Ghosted** G button is displayed in Sub Structure and Sub-model Views. It is also displayed in Level, Frame, and Slope Views when they have been toggled into 3D (via the 2D/3D toggle button). It is used to toggle the display of a ghosted view allowing you to see the current view in the context of the whole structure. For more information, see: Use Ghosted to see the view in the context of the whole model (page 1300) and this video Ghosted Structure view.

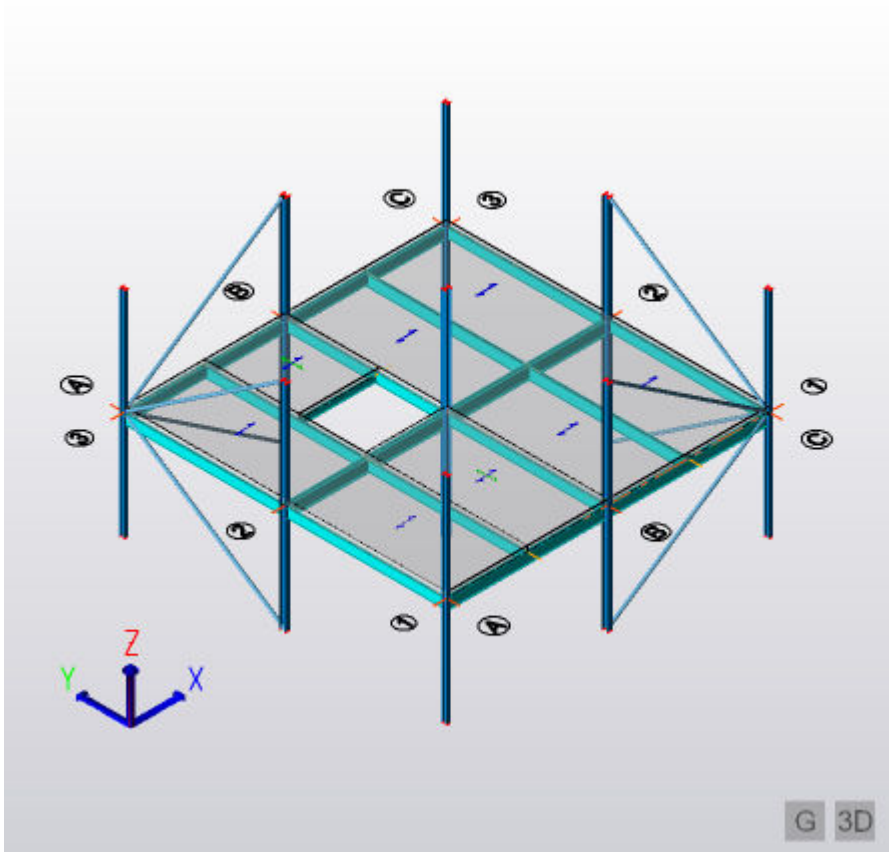
(8) The **2D/3D** 3D button is displayed in 2D Scene Views only. It is used to toggle the view between 2D and a 3D isometric projection.



Example of a level view in 2D:



The same level view when toggled to 3D:



(9) The **Cutting Planes** symbol is displayed in the bottom right corner of the scene view when any of the cutting planes are currently active.



(10) The (page 73) buttons in the status bar can be used to switch the active view between different regimes to suit the task being performed.

(11) You select how much of this information is displayed in each scene view via (page 66)

Multiple scene views can be displayed simultaneously, either horizontally, or vertically, by dragging a view tab and docking it to create a new **tab group**.

SEE ALSO

- Open and close scene views (page 64)
- Set the view regime
- How to control scene content (page 66)
- Create and modify view tab groups (page 73)

Create and modify view tab groups

When you have created multiple scene views, only the active view is visible by default. However, it is often useful to display views side by side or one below another. You can do this by creating tab groups.

Create a new tab group from an existing view tab

1. Right-click an existing view tab.
2. In the context menu, select **New Horizontal Tab Group** or **New Vertical Tab Group** according to your needs.

Create a new tab group using the docking control

1. Start dragging a view tab.

   A docking control appears in the middle of the view.



2. Do one of the following:

To	Do this
Create a new vertical tab group	• Drag the view over the left or right button of the docking control and release the mouse.
Create a new horizontal tab group	• Drag the view over the top or bottom button of the docking control and release the mouse.

Move a view from one tab group to another

1. Right-click the view tab.
2. In the context menu, select **Move to Next Tab Group**.

Overview of view regimes

A **view regime** determines the functionality of the active scene view.

There are 7 view regimes are as follows:

I View regime **c** **o** **n**	**Used for**
Structural View	Creating the structure geometry and applying loads

I View regime c o n	Used for
Solver View	Displaying the solver models for each of the analyses that have been performed. For more information, see: Solver models (page 537)
Results View	Viewing analysis results. For more information, see: Display analysis results in a Results View (page 662)
Wind View	Viewing the wind model. For more information, see: Modify wind zones and wind zone loads (page 389)
Review View	Examining the design status or reviewing specific model properties. For more information, see: Review models (page 942)
Slab Deflections View	Viewing slab deflection analysis results. For more information, see:
Footfall Analysis View	Creating new, and reviewing existing footfall floor analysis models. For more information, see:

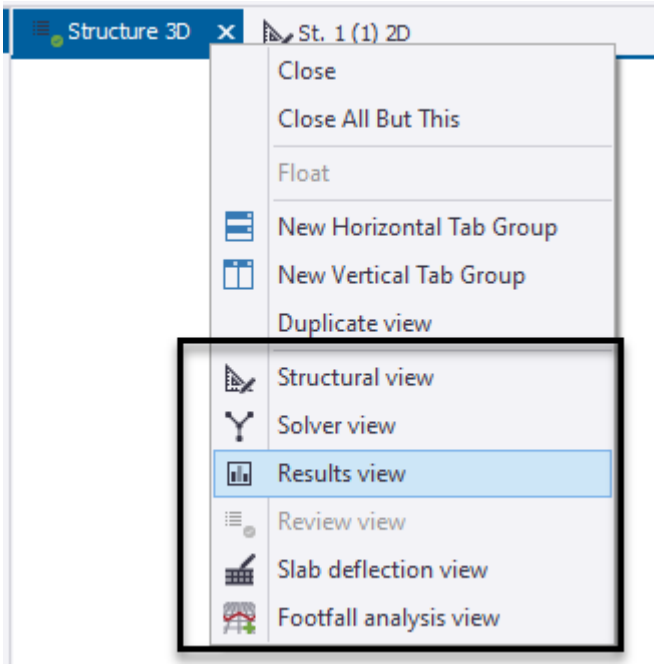
Set the view regime

There are two different ways to set the view regime, either:

- In the **Status bar** at the bottom of the main window, click the desired view regime button.



- Or, right-click on the view tab and then from the context menu, select the desired view regime.



See also

Open and close scene views (page 64)

How to control scene content (page 66)

Work with view configurations (page 67)

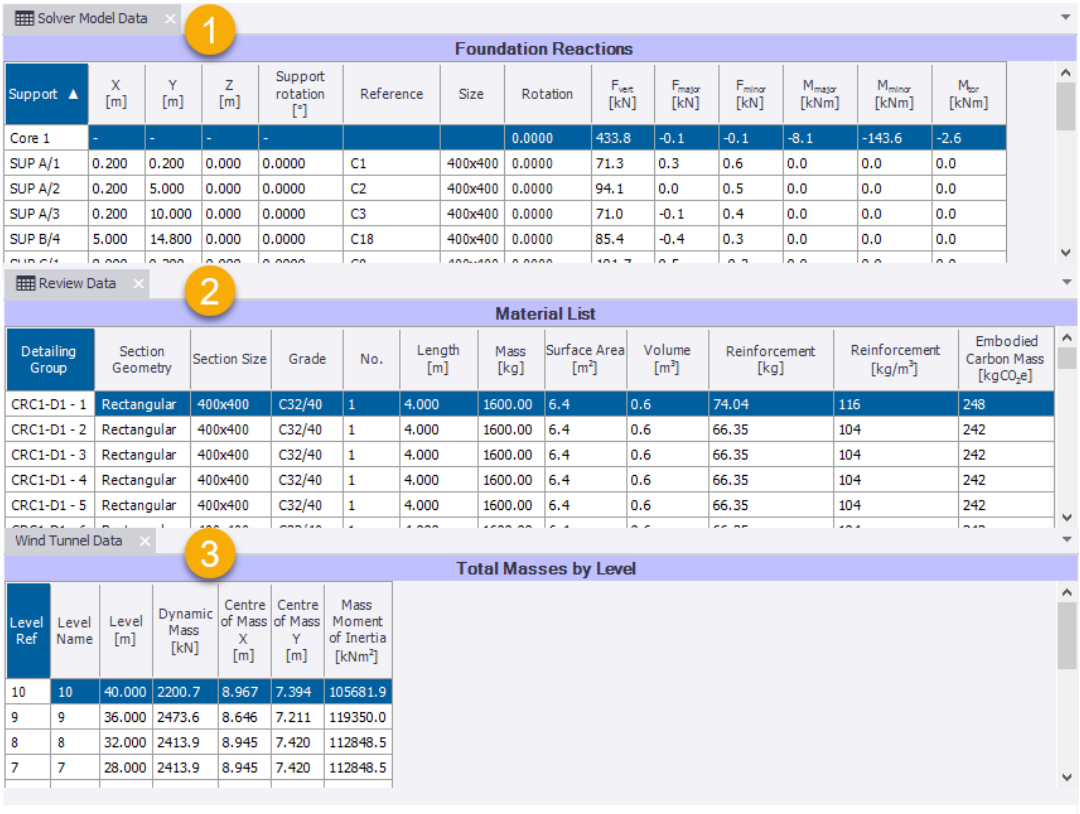
Overview of scene view interface components (page 68)

Create and modify view tab groups (page 73)

Overview of Tabular Data views

A **Tabular Data View** displays model and results data of a selected type within a table.

There are three main categories of Tabular Data View:



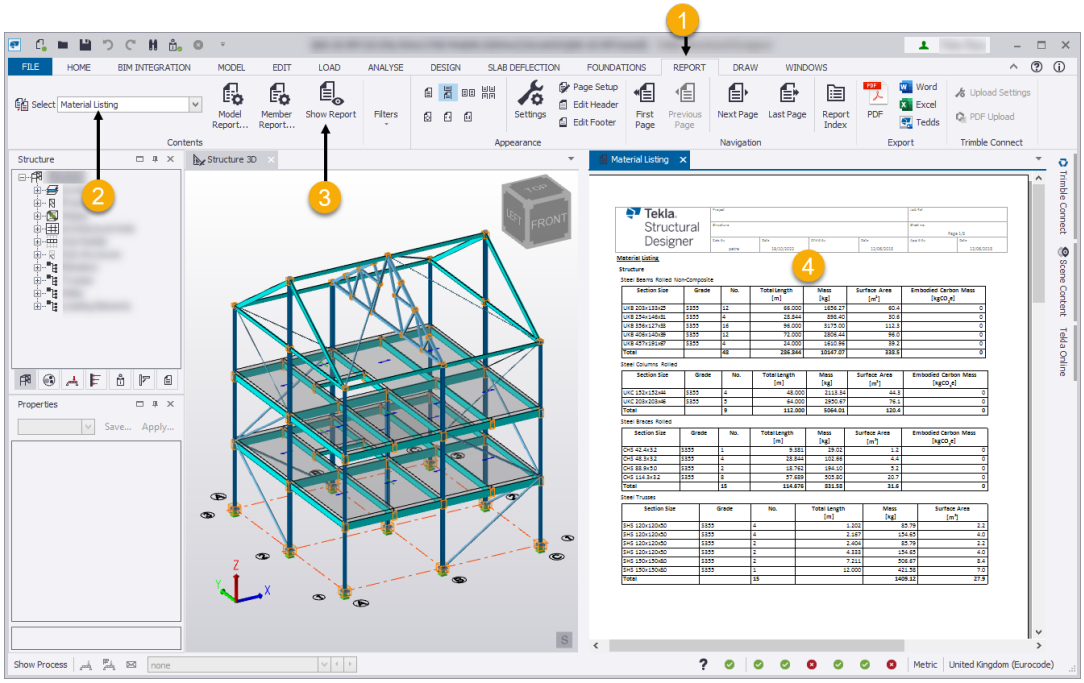
Scene View	Description
Solver Model Data (1)	Accessed from the **Analyze** ribbon , this view is used for displaying tabular model and results data. For more details, see: Display tabular analysis results in a Solver Model Data view (page 707)
Review Data (2)	Accessed from the **Review** ribbon, and configured from the **Review Data** ribbon, this view is displays different types of review data. For more details, see: Tabular review data (page 1011)
Wind Tunnel Data (3)	Accessed from the **Analyze** ribbon, and configured from the **Wind Tunnel Data** ribbon, this view is displays modal analysis results in a form that can be exported to wind specialists. For more details, see: Exporting wind tunnel data workflow (page 424)

Multiple tabular views can be displayed simultaneously, either vertically (as above), or horizontally by dragging a view tab and docking it to create a new **tab group**.

SEE ALSO • Create and modify view tab groups (page 73)

Overview of Report views

Report views can be set up for printing and export to others applications. They can be configured as required to contain tabular data (such as model input, loading and results), pictures, views and charts. Report filters and styles can be applied to control the level of output. Several example reports are provided, but you can also create your own customized reports.



To open a report:

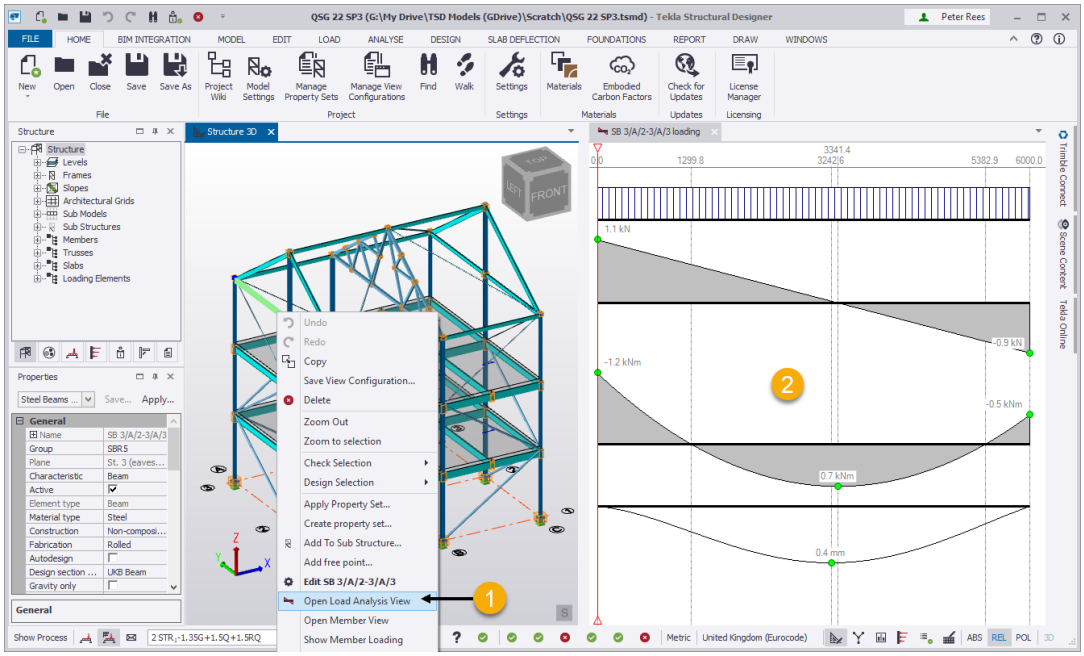
- On the Report ribbon **(1)**, use the **Select droplist (2)** to choose the report to display.
- Select **Show Report (3)**
- The selected report is displayed in a **Report View (4)**.

See also

Create and modify reports (page 1062)

Overview of the Load Analysis view

Analysis results for individual members and walls can be displayed in a **Load Analysis View**.



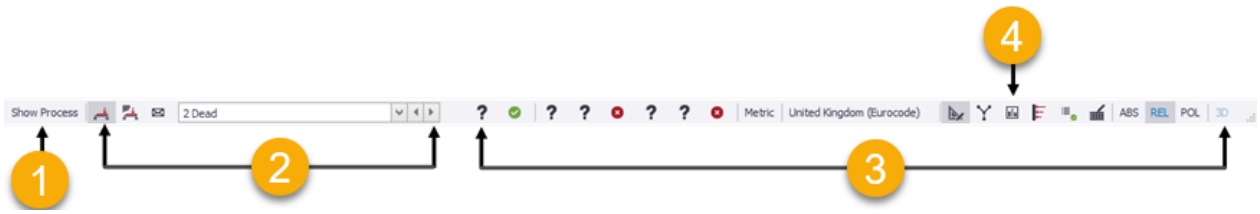
To open a **Load Analysis View**:

- Right click on a member or wall in a Scene View.
- From the context menu select **Open Load Analysis View (1)**.
- The analysis results are displayed in a **Load Analysis View (2)**.

See also

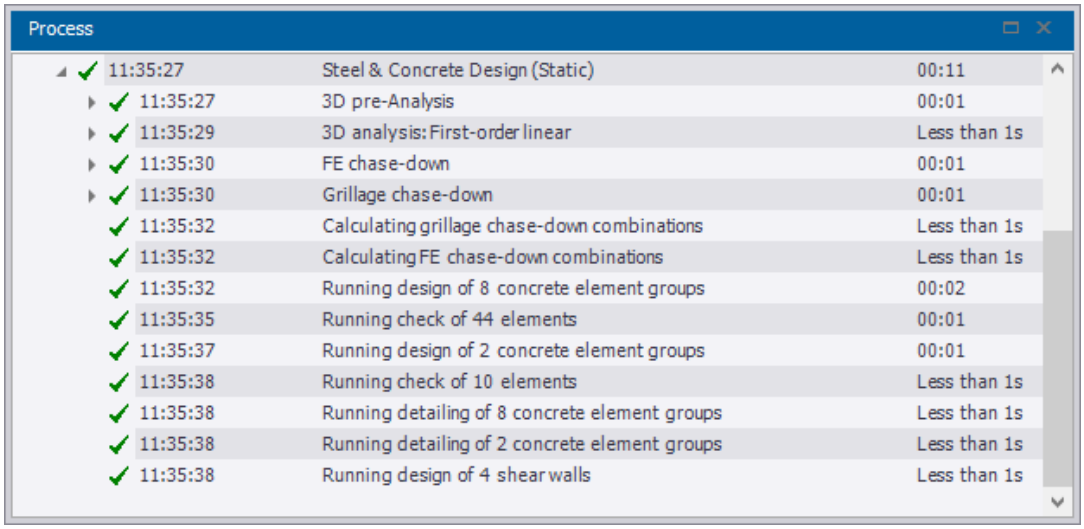
Display member analysis results in a Load Analysis View (page 748)

Overview of the Loading List and Status bar

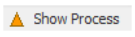


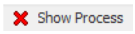
1. Show Process button

Click the **Show Process** button, located at the left end of the **Status bar** to display the **Process Window**.



When you analyze or design the model, each step of the process (including any associated warnings or errors) is logged and displayed in the **Process Window**. When the **Process Window** is closed, the overall status of the last process performed is indicated on the **Show Process** button as below:

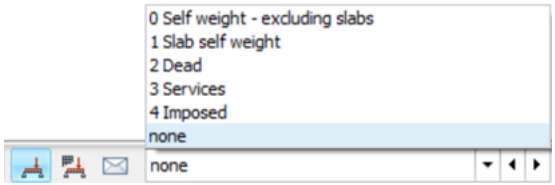
- 

- 

Such warnings and errors should always be fully investigated, as they may have an adverse affect the design.

2. Loading List

The **Loading** list is permanently docked at the bottom edge of the main window.



The **Loading** list has two main functions:

- Selecting a specific loadcase to add loads into
- Selecting a specific loadcase, combination, or envelope to view the results for.

When viewing results, first click the **Loadcase**, **Combination**, or **Envelope** button according to your needs. The, choose the specific loadcase, combination, or envelope name in the list.

3. Status bar

The **Status bar** is permanently docked at the bottom edge of the main window. The **Status bar** performs a number of different functions.



The **Status bar** provides feedback by indicating:

1. The analysis and design validity
2. The units system
3. The design code
4. The coordinates of the cursor relative to the global origin in 2D views
5. The view regime buttons that allow you to switch the view regime applied to the active scene view, see **View Regime buttons** below.
6. Coordinate display that allows you to set the display method for coordinate tooltips:
   - **ABS** (Absolute)
   - **REL** (Relative)
   - **POL** (Polar)
7. **2D**/**3D** toggle button
   - **2D**: the content of the 2D view is displayed in plan.
   - **3D**: the content of the 2D view is displayed in isometric.

4. View regime buttons

The view regime buttons allow you to switch the view regime applied to the active scene view:

- **Structural View** shows the geometry and loading of the structure.
- **Solver View** shows the analysis model.
- **Results View** shows the analysis results.
- **Wind View** shows the wind model.
- **Review View** graphically examines the model properties or status.
- **Slab Deflections View** shows the slab deflection analysis results.

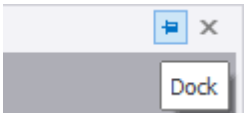
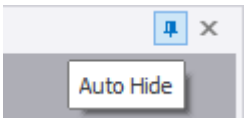
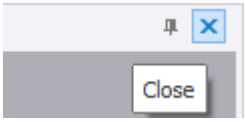
-  **Footfall Analysis View** shows footfall analysis models (FAMs).

For more information, see: Overview of view regimes (page 73)

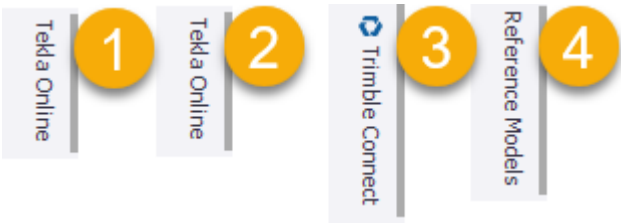
Overview of Side panes

By default most windows are initially docked at a specific position, however some are displayed as **side panes** instead, (a side pane being what a docked window reduces to when it is auto-hidden).

Clicking a side pane opens the associated window. The buttons at the top right of the window then operate as follows:

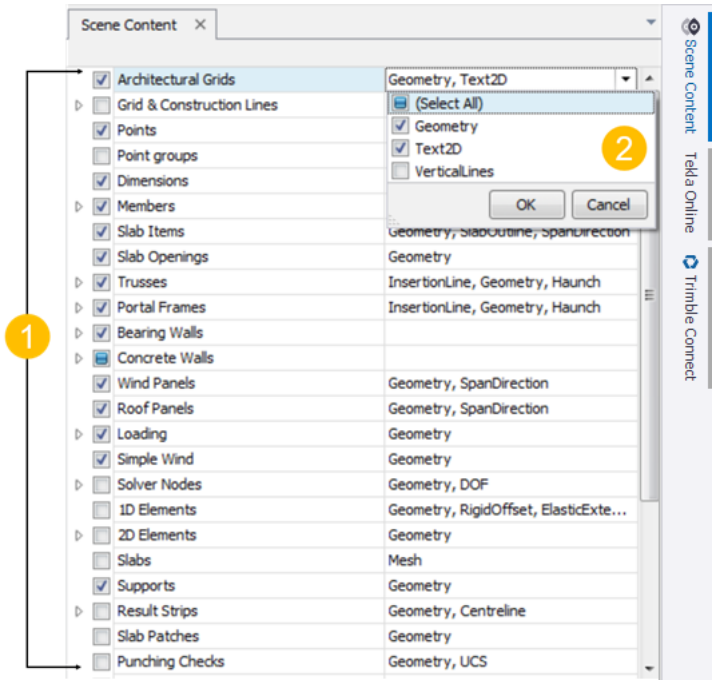
Button	Function
	Docks the window to the edge of the interface and prevents it from auto-hiding.
	For a docked window, clicking **Auto Hide** reduces it to a side pane at the same edge when it is not active.
	Clicking **Close** closes the window and removes the side pane. A closed window can be reopened from the **Windows** ribbon.

Typically the following windows are initially displayed on the right of the interface as side panes:



1. Scene Content side pane

The **Scene Content** side pane is used to control the displayed content in the 2D and 3D Scene Views.



The Scene Content window contains:

1. Entity categories (with check boxes)

   Some categories have arrow symbols to their side, indicating sub categories. Click the arrow symbol in order to see the sub categories.

   The checkbox controls whether the entity category and its associated information is displayed. You can check the entities that you want to display and clear the ones that you do not.

2. Entity information controls

   Entity information controls list the information in each category that will be displayed. When clicked, they expand to lists that allow you to select the information you want to display in the model view.

SEE ALSO • How to control scene content (page 66)

2. Tekla Online side pane

The Tekla Online side pane is used to access Tekla Online services provided for users of Tekla Structural Designer, for example Tekla User Assistance and Tekla Discussion Forum.

3. Trimble Connect side pane

The Trimble Connect side pane is used to access the Trimble Connect project collaboration tool.

SEE ALSO • Working collaboratively with Trimble Connect (page 1167)

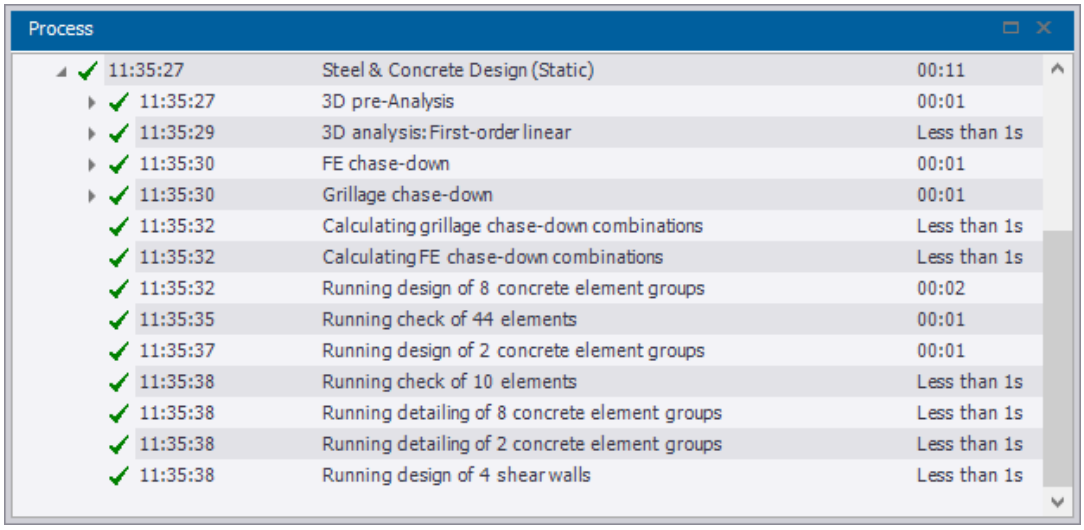
4. Reference Models side pane

The Reference Models side pane is used to control the visibility of imported 3D reference models.

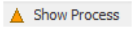
SEE ALSO • Import 3D reference models (page 1165)

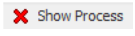
Overview of the Process window

Initially, the **Process Window** is closed. It can be displayed by clicking the **Show Process** button, located at the left end of the **Status bar**.



When you analyze or design the model, each step of the process (including any associated warnings or errors) is logged and displayed with timings in the **Process Window**. When the **Process Window** is closed, the overall status of the last process performed is indicated on the **Show Process** button as below:

- 

- 

Such warnings and errors should always be fully investigated, as they may have an adverse affect the design.

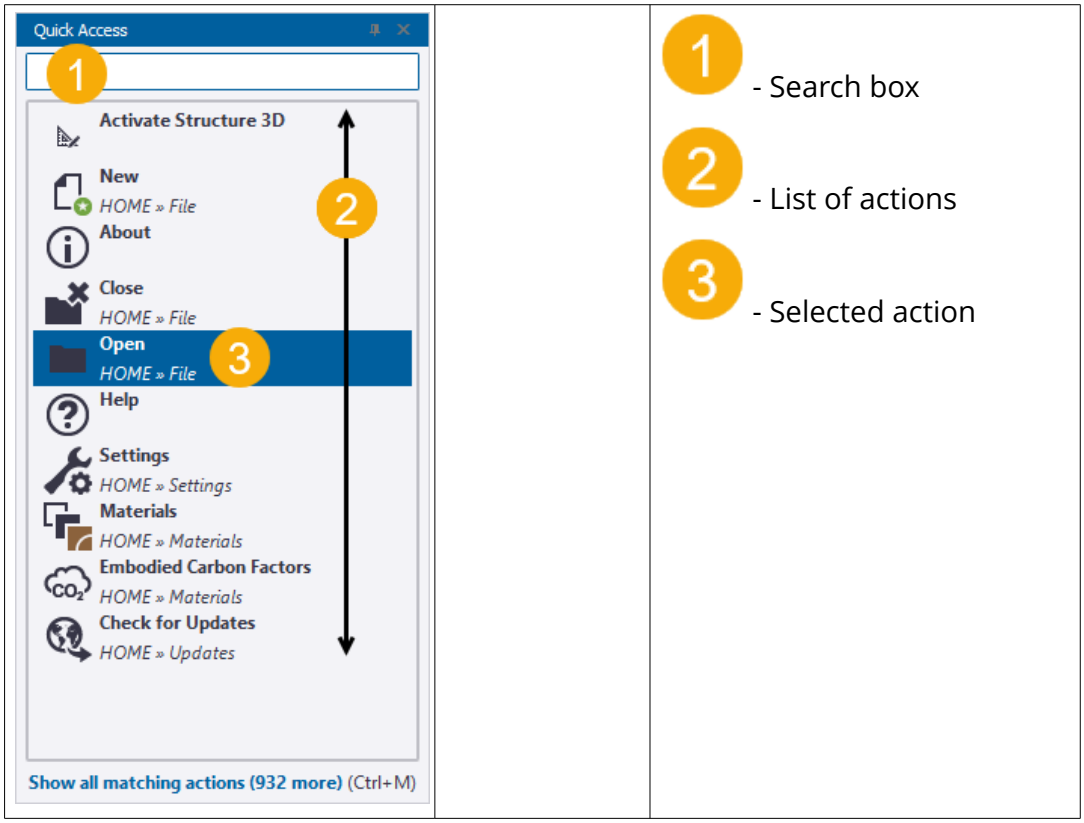
How to use the Quick Access window

You can use the Quick Access window to perform most actions directly from the keyboard.

Open Quick Access

- Use the keyboard shortcut **Ctrl+Q** to open the Quick Access window.

Quick Access Window layout



To find and run an action

1. Start typing in the search box at the top of the window. As you type, the top 10 matching actions are listed.
2. Use the keyboard Up/ Down keys to navigate through the list, then press **Enter** to select and activate your chosen action/ command.

TIP a. Click “Show all matching actions” at the bottom of the window - or press the keyboard **Ctrl+M** keys - to see all matches.

b. When the search box is empty, the last 10 *available* actions are listed giving rapid and convenient access to the actions/ commands you have been using recently.

Note however that an action will not be listed if it is active. (For example the "Activate Stucture 3D" action will not be listed if the Stucture 3D view is the currently active view.)

Example actions

If no model is open	Example actions
Only actions not greyed out on the ribbon can be listed and run. **NOTE** Global settings can only be listed when no model is open.	• **New** *HOME > File* • **Open** *HOME > File* • **Settings** *HOME > Settings* • **Materials** *HOME > Materials* • **Embodied Carbon Factors** *HOME > Materials*

Once a model is open	Example actions
All available File menu and ribbon commands:	• **Construction Levels** *MODEL > Levels* • **Rectangular Wizard** *MODEL > Grid & Construction Lines* • **Column** *MODEL > Steel* • **Beam** *MODEL > Concrete* • **plus many more...**
Actions to open new views:	• **Open St.1 (1) 2D** • **Open FRM 1 (1) 2D** • Etc.

Once a model is open	Example actions
Actions to switch from one view to another:	• **Activate Structure 3D** • **Activate St.1 (1) 2D** • **Activate FRM 1 (1) 2D** • **etc.**
Actions to change the current view regime:	• **Switch to Structural View** • **Switch to Solver View** • **Switch to Results View** • **Switch to Review View** • **Switch to Slab Deflections View**
Actions to locate and review individual Model Settings:	• **Units** *Model Settings > Units* • **Use load patterning for steel beams** *Model Settings > Loading > General > Load Patterns* • **plus many more...**
Actions to open Analysis Settings, Design Settings, Draw Settings dialogs etc.	• **Settings** *ANALYZE > Settings* • **Settings** *DESIGN > Settings* • **etc.**
Actions to define the selected entity's properties:	• **Rotation** *Properties > General* • **Grade** *Properties > All stacks* • **plus many more...**
Actions to rotate the viewcube:	• **Set FRONT view mode** • **Set TOP-LEFT view mode** • **etc.**
Scene content settings:	• **Steel Beam X** (switches off the setting - the entity will be hidden) *Scene Content > Members*

Once a model is open	Example actions
	• **Steel Column** ✓ (switches on the setting - the entity will be displayed) *Scene Content* > *Members* • **plus many more...**

How to hide, re-display and move windows

The active view (2D view, 3D view, tabular data view or report view) is always maximised. If required a second view can be displayed at the same time by opening a vertical or horizontal tab group.

All the remaining windows can be resized and repositioned to float over the views, or docked to an edge of another window - see the instructions below for details.

Auto-hide a window

To increase the area available for graphical display, you can choose to auto-hide any of the windows.

- In the desired window, click **Auto Hide**.

  The window immediately contracts to a tab.
- Click the window tab to expand it.

  You can only expand a window if it is docked against an edge of the main window.

Close a window

- Click **Close** at the top right corner of the window.

Re-display a closed window

1. Click the **Windows** tab.
2. In the **View** group, click the window name.

   If the window is already displayed, its control is highlighted in the **View** group.

Move a window

- Select the title bar of the window, and drag the window to its new location.

  If you place the selected window over an edge of the main window, or over a divider within the main window, the window docks to that edge or divider.

Dock a window as a tabbed page in another window

- Select the title bar of the window, and drag the window over the title bar or the tab group of the other window.

Open a tabbed page in another window

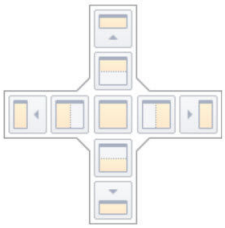
- Select the tabbed page, and drag the page to its new location.

  The tabbed page opens in a new window.

Dock a window using the docking control

1. Select the title bar of the window, and start dragging the window.

   The window docking control appears in the middle of the main window.



2. Drag the window over the docking control button that indicates the desired location.

Dock a window to a tabbed page of the current window

1. Select the title bar of the window, and drag the window over the central button of the docking control.
2. Release the left mouse button.

Keyboard shortcuts

The keyboard shortcuts in Tekla Structural Designer are described below.

If you want a printed copy for reference, a summary list of the shortcuts can also be downloaded from here.

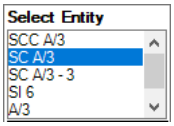
Learn some keyboard shortcuts to improve your productivity! (video)

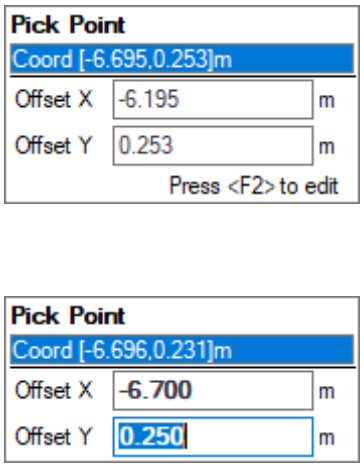
How to use the Quick Access window (page 84)

General

Function	Key
Displays the Tekla Structural Designer Help.	**F1**
Holding down **CRTL** while selecting entities adds the new entities to the current selection.	**CRTL**
Cancels the current command.	**Esc**
Runs the model validation process - on completion the Project Workspace switches to the Status page if not already active.	**CRTL+D**
Opens the Find dialog	**CRTL+F**
Open a file	**CRTL+O**
Open/close the Process Window	**CRTL+P**
Starts a new file	**CRTL+N**

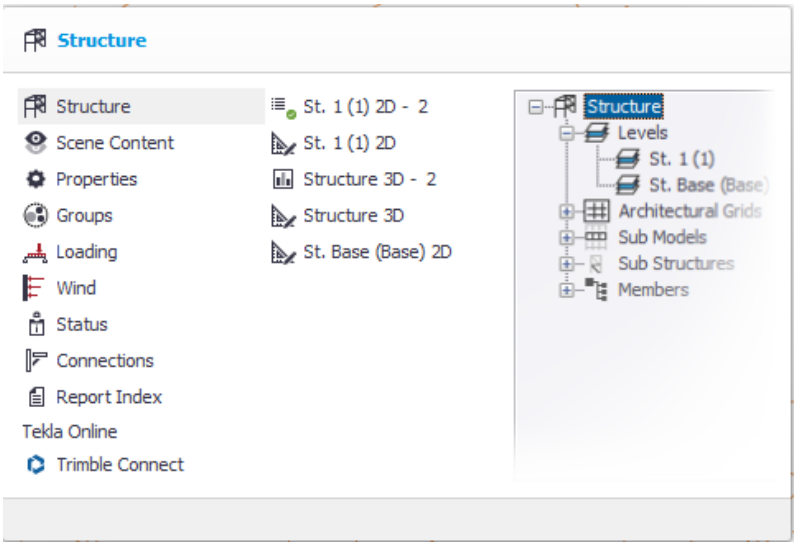
2D and 3D Views

Function	Key
Zoom to the extents of the model	**ZA**
Zoom to the selected entities	**ZS**
Zoom to 1m (3ft) around the cursor location	**Z1**
Zoom to 2m (6ft) around the cursor location	**Z2**
Zoom to 3m (9ft) around the cursor location	**Z3**
Move between entities in the **Select Entity** tooltip when multiple entities are detected 	**UP/DOWN**
Hold **CRTL** while selecting entities adds the new entities to the current selection	**CRTL**
Select entities then use the shortcut to start the copy operation	**CRTL+C**
Switch to a different View Regime. To change the view regime press the **CRTL+R** followed by a number key from 1 to 6 to activate the associated view	**CRTL+R,1..6**

Function	Key
regime as follows (note this only functions with the number keys above keyboard letter keys, not the numpad): **1** - Structural **2** - Solver **3** - Results **4** - Wind **5** - Review **6** - Slab Deflection	
Toggle a 2D view to 3D	**CRTL+T**
Edit the values in the Pick Point tooltip  	**F2**
Toggle the co-ordinate system between Absolute, Polar, and Relative	**F3**
Show/Hide the **ViewCube** in 3D views	**F8**

Interface window/scene view switcher

The switcher provides a quick way to navigate between open windows and scene views.

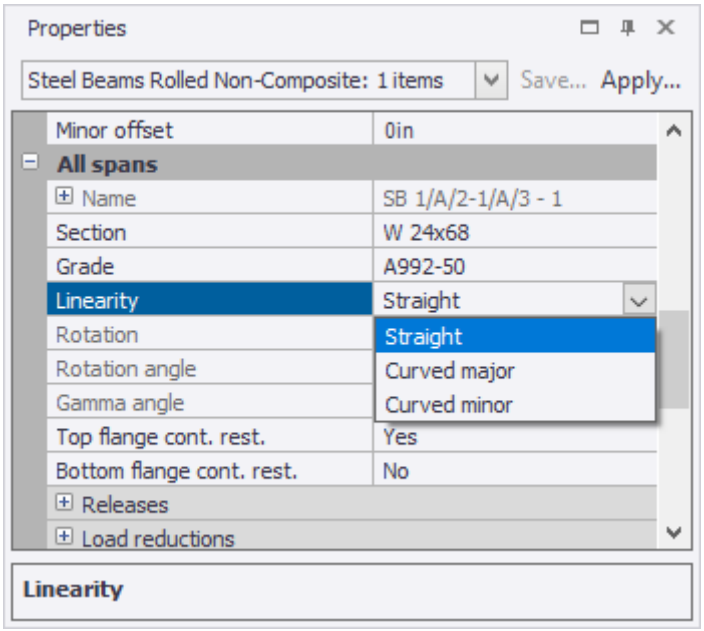
Function	Key
Open the Switcher  *The Switcher is only visible while **CTRL** is held down*	**CRTL+TAB**
With **CTRL** held down, select the required item using the arrow keys and then release the **CTRL** key to switch to the chosen item	**UP/DOWN/ LEFT/RIGHT/ TAB**

Project Workspace

Function	Key
Move between different items	**UP/DOWN**
Expand a heading to reveal sub-items	**RIGHT ARROW**
Return to a heading, or collapse an expanded heading	**LEFT ARROW**

Properties window

Function	Key
Move between different properties and navigate within lists	**UP/DOWN**
Enter/Leave (and save) property editing mode	**TAB**
Space - Toggle check-boxes and expand lists	**SPACE**

Function	Key
	
Select and save property editing	**ENTER**
While a Show/Alter state view is active:	
Cycle through the [A]ttribute drop list options	**A**
Cycle through the [M]ode drop list options	**M**

Scene Content window

Function	Key
Move up and down the list	**UP/DOWN**
Expand a heading to reveal sub-items	**CTRL+ ARROW**
Collapse an expanded heading	**CTRL+ ARROW**
Toggle the item on or off	**SPACE**

Quick Access window

Function	Key
Open the Quick access window	**CTRL+Q**
Type your keyword	
Switch between “Most relevant matching actions” and “All matching actions”	**CTRL+M**
Highlight the required command	**UP/DOWN**

Function	Key
Select the highlighted command	**ENTER**
Useful keywords	
Open new views	**Open**
Switch from one view to another	**Activate**
Rotate the ViewCube actions	**Set**
Change the current view regime	**Switch**

Ribbon commands

All commands on the ribbon toolbars can be accessed from the keyboard after pressing the **ALT** key.

1. Press **ALT** then the appropriate letters (summarized in the table below) to open the required ribbon toolbar.



2. Press the appropriate letter(s) for the command you want to run.



Example: Press **ALT HFN** to open the **Home** toolbar and run the **Find** command shown above.

Toolbar	Shortcut
File	Press **ALT AP** then the letter(s) shown against the required command.
Home	Press **ALT H** then the letter(s) shown against the required command.
BIM Integration	Press **ALT B** then the letter(s) shown against the required command.
Model	Press **ALT M** then the letter(s) shown against the required command. .
Edit	Press **ALT E** then the letter(s) shown against the required command.

Toolbar	Shortcut
Load	Press **ALT LO** then the letter(s) shown against the required command.
Analyze	Press **ALT AN** then the letter(s) shown against the required command.
Design	Press **ALT DE** then the letter(s) shown against the required command.
Slab Deflection	Press **ALT SL** then the letter(s) shown against the required command.
Foundations	Press **ALT FO** then the letter(s) shown against the required command.
Report	Press **ALT RE** then the letter(s) shown against the required command.
Draw	Press **ALT DR** then the letter(s) shown against the required command.
Windows	Press **ALT W** then the letter(s) shown against the required command.
Loading Analysis	Press **ALT LA** then the letter(s) shown against the required command.
Results	Press **ALT RS** then the letter(s) shown against the required command.
Review	Press **ALT RV** then the letter(s) shown against the required command.
Review Data	Press **ALT RV** then the letter(s) shown against the required command.

1.5 Contact Tekla Structural Designer support (Support tool)

The Support tool allows you to contact Tekla Structural Designer support directly. With this tool you can collect the model, related files, and other necessary information in one support request, and safely upload your request to Tekla Structural Designer support.

The Support tool:

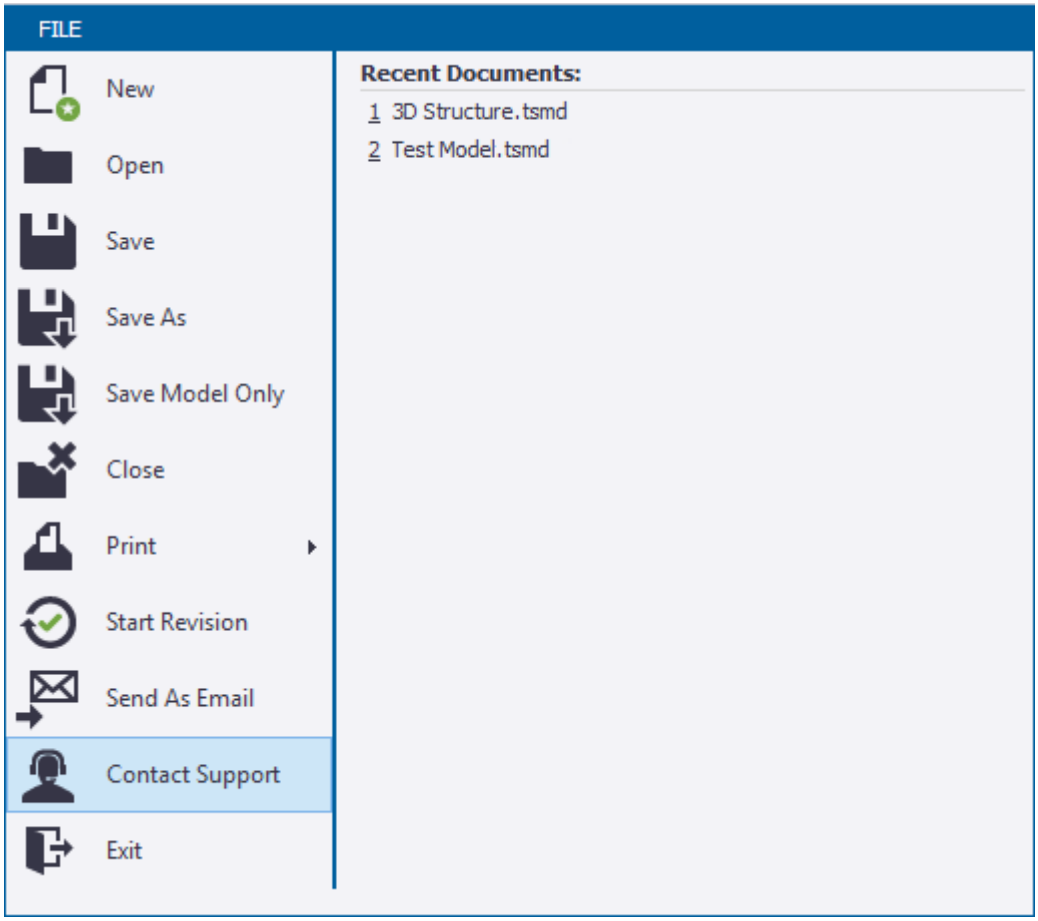
- Automatically identifies the open active model and attaches it to your request.
- Automatically gathers application and system information.
- Uploads the problem description, attached model, attached files, and all other gathered information to Tekla Structural Designer support.

NOTE Confidentiality information

All files you upload are treated as confidential. Only the recipient can access the files.

Create a support request

1. On the **File** menu, click **Contact Support**



2. Log in using your Trimble Identity.

   The Support tool opens and automatically fills in the application version and user information. Support tool reads your name, email address, company name and support email address from your Trimble Identity profile.
3. Select a category from the list of predefined categories.
4. Enter the problem description, steps to reproduce and question.
5. Click **Next**.

6. Select what you want to attach. The file name, file group, file size, and file location are mentioned for each file.
   - By default the model file is selected.
   - Select the **All** check box, or select specific files from the **Select the files** list.
   - If you want to send some other attachments than shown in the **Select the files** list, click the **Add extra files** button and browse for the files.
7. Click **Next**

   The Support tool creates the package and shows the total attachment size. You can also check application information and operating system information before finalizing the support case creation.
8. Click **Create case** to upload your case to Tekla Structural Designer support.

When the upload is complete, you will receive a notification at your email address. After a successful upload, an automatic confirmation message will be sent to you, and then Tekla Structural Designer support will start solving your case.

For a list of offices and resellers together with their contact information, see Offices and resellers.

2 Create models

To get started with modeling in Tekla Structural Designer see:

- Get to know Tekla Structural Designer basic working methods (page 97), for a few first principles
- Create the model (page 120), to learn how to create and modify the main structure
- Create isolated foundations (page 260) and Create mat foundations (page 265), to define the foundations

Once you are comfortable creating the model, see:

- Edit the model (page 270), to get familiar with the model editing commands
- Validate the model (page 323), to learn how to detect and correct validation issues that can arise in the model and analysis.
- BIM integration (page 1156), to learn how to exchange model data between applications

For modeling aspects specific to industrial structures, see:

-

See also

BIM integration (page 1156)

2.1 Get to know Tekla Structural Designer basic working methods

Before creating real models, get to know some basic methods and techniques that will help you to work efficiently.

Firstly, we recommend you familiarize yourself with setting up the workspace:

- review model specific settings (page 1262)
- create an architectural grid (page 123)

- define building construction levels (page 121)

We then recommend you practice how to:

- create (page 120), select (page 100) and edit the properties of (page 111) entities
- re-position entities (page 112) by moving nodes or edges
- copy, move and mirror (page 270) part of, or all of the model
- zoom and rotate (page 98) the model

You should also familiarize yourself with the coordinate systems. (page 1306)

See also

Create levels, grids, members, slabs, panels and supports (page 120)

Zoom, pan, rotate and walk through the model

You can use the mouse to manually zoom, pan, or rotate the model. Additionally, you can rotate the model by using the **ViewCube**, and walk through your model in 3D views.

Zoom in and out, or zoom extents

- According to your needs, do one of the following:

To	Do this
Zoom in or out	• Scroll in or out with the middle mouse button.
Zoom to the center of the visible objects	• Right-click anywhere within the view. • In the context menu, select **Zoom Out**. **NOTE** You can also use the keyboard shorcut **ZA** to zoom extents.

Pan the view

- Hold down the middle mouse button and drag the view according to your needs.

Rotate the view with the ViewCube

TIP Press **F8** to show/hide the **ViewCube**

1. Move the mouse pointer over the **ViewCube**.
2. According to your needs, do one of the following:

To	Do this
Display one of the eight isometric views	• Click the required vertex on the **ViewCube**.  **TIP** If the required vertex is not visible, click one of the other vertices to spin the **ViewCube** until you can see the required vertex.
Display one of the twelve edge views	• Click the required edge on the **ViewCube**.  **TIP** If the required edge is not visible, spin the **ViewCube** by clicking the vertex that is adjacent to the required edge.
Display one of the six face views	• Click the required face on the **ViewCube**.  **TIP** If the required face is not visible, spin the **ViewCube** by clicking the vertex adjacent to the required face. **NOTE** When a face view is displayed, the additional **ViewCube** controls appear:



- If the required face is not visible in a face view, roll the **ViewCube** by clicking one of the triangular controls.

Rotate the view manually

If none of the standard **ViewCube** rotations are appropriate, you can also rotate the model manually.

- Hold down the right mouse button and drag the view according to your needs.

Walk through the model in a 3D view

- On the **Home** tab, click **Walk**.

NOTE In the **Walk** mode:

- To move forward, backward, left or right, use the arrow keys.
- To move up and down, use the **Q** and **Z** keys.
- To rotate the view, hold down the right mouse button and drag the view.
- To exit **Walk** mode, press **Esc**.

Display a 2D view in 3D

1. In a 2D view, click the **2D**/**3D** toggle button, located on the bottom right corner of the window.

Select entities

Many Tekla Structural Designer commands require you to make selections. You can make single and area selections from any 2D or 3D view. You can also make selections from the **Project Workspace**.

TIP • If planar objects, such as slabs, roofs and area loads, obscure what you want to select, you can hide them using the window.

• Use **Ghost Unselected** to display everything *not* in the selection as ghosted, this makes it easier to focus on the selection.

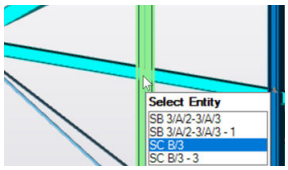
Use the different methods listed below to select the entities.

Select single entities

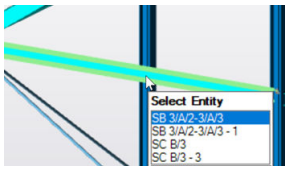
1. In a 2D or 3D view, move the mouse pointer over the desired entity.

   If the entity is the only one at the location, it becomes highlighted and its name is shown in the **Select Entity** tooltip.

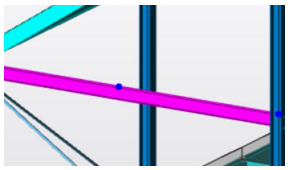
   If several entities are located at the same location, their names are all listed in the **Select Entity** tooltip.



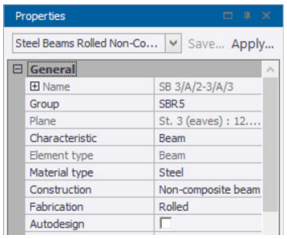
   Use the up and down keys to scroll through the list to reach the desired entity.



2. When the desired entity is highlighted in the **Select Entity** tooltip, press **Enter** to select it.



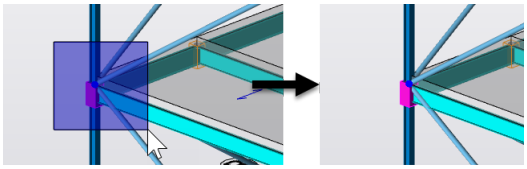
The selected entity is then displayed in the *active selection color*, the nodes for positioning it are exposed, and the entity properties are displayed in the **Properties** window.



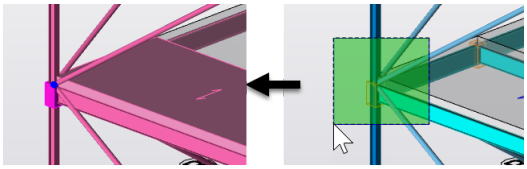
Select multiple entities using area selection

1. You can select multiple objects using area selection. The dragging direction affects the selection of objects.

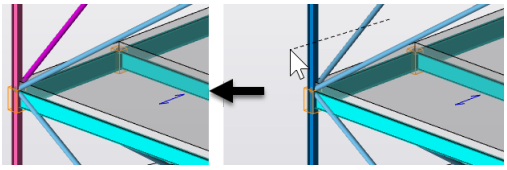
   a. To select all entities that are completely within a rectangular area, hold down the left mouse button and drag the mouse from **left to right**.



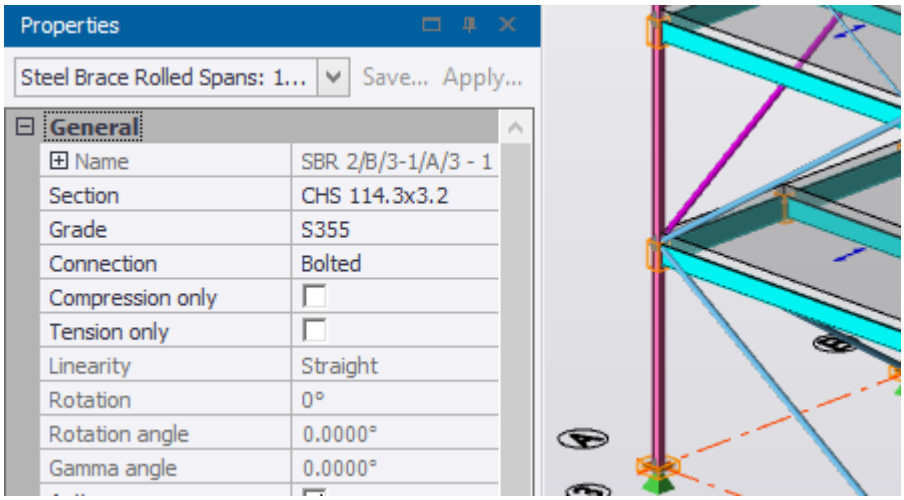
   b. To select all entities that are at least partly within a rectangular area, hold down the left mouse button and drag the mouse from **right to left**.



   c. To select all objects that are at least partly crossed by a straight line, hold down the **shift key**, then hold down the left mouse button and drag the mouse to draw the line.



After selection, one entity type is displayed in the *active selection color* and has its properties shown in the **Properties** window.

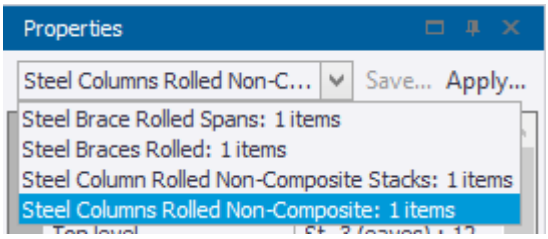


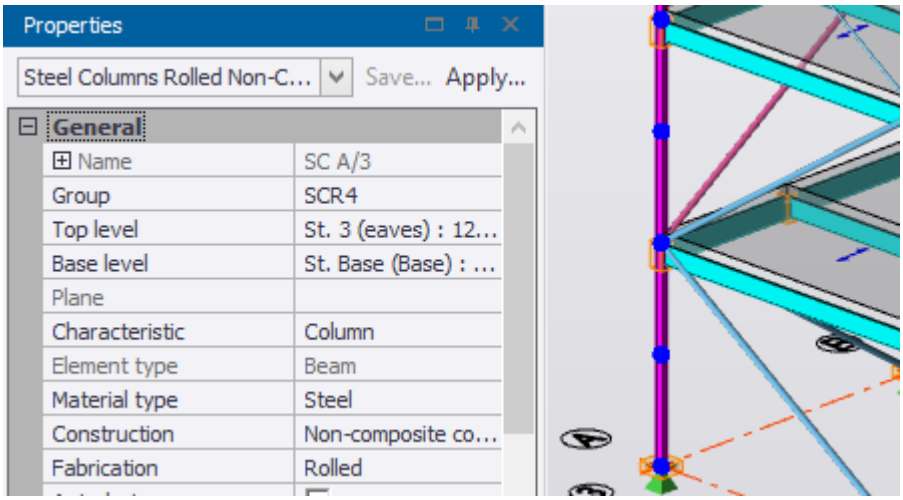
Any other entity types in the selection are displayed in the *local selection color*.

NOTE You can change the *active* and *local* selection colors by clicking **Home > Settings > Scene > Colors > Selection**.

---

2. To display the properties of another entity type in the selection, choose it from the droplist at the top of the **Properties** window.





NOTE If there is only a single item in the active selection, all its properties are shown; if there are multiple items in the active selection, only common properties are shown.

Select using Find

1. Click **Find** on either the **Home** tab, or the **Quick Access Toolbar**.

   The **Find** dialog opens.

2. In the **Find** box type any of the following, in whole, or in part:

   a. An object 'Type'

   b. An object 'Name'

   c. An object 'Index'

   As you type, any objects that match the criteria are listed in the table below.

3. When the object(s) that you require are shown, click to highlight them as follows:

   a. Single click to highlight a single object

   b. Ctrl+click to select multiple non-sequential rows

   c. Shift+click to select multiple sequential rows Once the object(s) are highlighted, **Select** becomes available.

4. Click **Select** to locate the object in the active view, zoom in, and display its properties in the **Properties** window.

NOTE If the object doesn't exist in the active view a new view containing the object will become active.

TIP Sometimes when clicking **Select** the object being located is obscured from view by other objects. By leaving **Ghost Unselected** checked this is avoided, as all objects other than those being found are made semi-transparent.

NOTE If not using Ghost Unselected, you can change the color used to highlight the found entity by clicking **Home > Settings > Scene > Colors > Selection** and changing the **Global - User** color.

Select from the Project Workspace

1. Do one of the following:

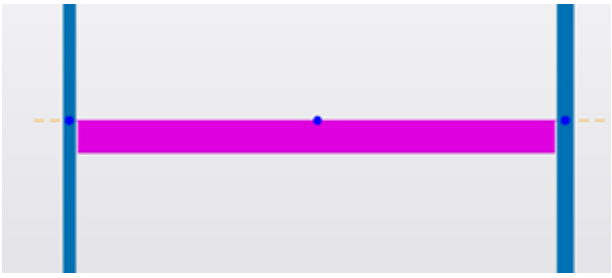
To	Do this
Select individual members from the **Structure** tree	a. Expand the **Members** branch and any appropriate sub branches until you can click the required member. b. Right-click the member name. c. In the context menu, select **Select in visible views**. The member's properties are viewed in the **Properties** window.

To	Do this
Select member groups or individual members from the **Groups** tree	a. Expand the appropriate branches until you can click the required group or member name. b. Right-click the member or group name. c. In the context menu, select **Select in visible views**. The member's or group's properties are viewed in the **Properties** window.

Select nodes

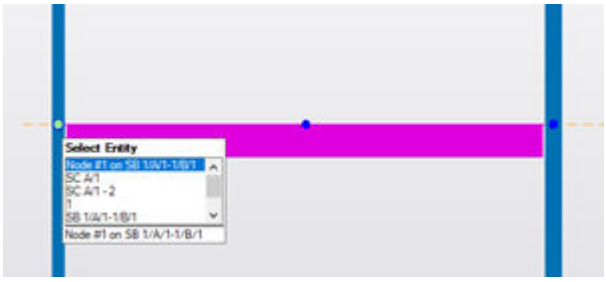
Sometimes you need to select only the nodes of an entity, for example when moving a beam.

1. In a 2D or 3D view, select the entity.



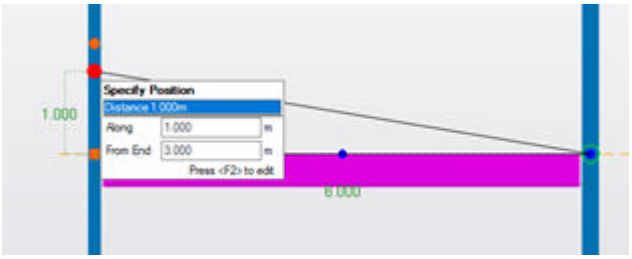
The entity is selected and any nodes associated with the entity are displayed.

2. Move the mouse pointer over the desired node.



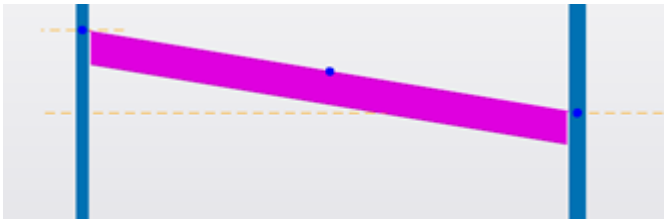
The node becomes highlighted and appears in the **Select Entity** tooltip.

3. Press **Enter**.



You will now be prompted to pick a point to move the node to.

4. Click at a new location to reposition the node.



The entity is updated accordingly.

See also: Re-position entities by moving nodes or edges (page 112)

Modify the selection

You can add or remove entities from the current selection.

1. Do one of the following:

To	Do this
Add an entity to the current selection	a. Hold down the **Ctrl** key. b. Click the entity that you want to add to the selection. The properties of the newly selected entity are added to the **Properties** window.

To	Do this
Remove an entity from the current selection	• Click the entity you want to de-select. The entity's properties are removed from the **Properties** window.

To	Do this
Remove multiple entities	a. Do one of the following: • To de-select only the entities that are encompassed by the box, move the mouse pointer to the **left** corner of the imaginary box that will encompass the entities. • To de-select the entities that are encompassed by the box and that it crosses, move the mouse pointer to the **right** corner of the imaginary box that will encompass the entities. b. Hold down the **Ctrl** key and drag the mouse pointer to the opposite corner of the box. A rectangle on the screen shows the area that you are selecting. c. Release the mouse button and the **Ctrl** key. The entities are removed from the **Properties** window.

To	Do this

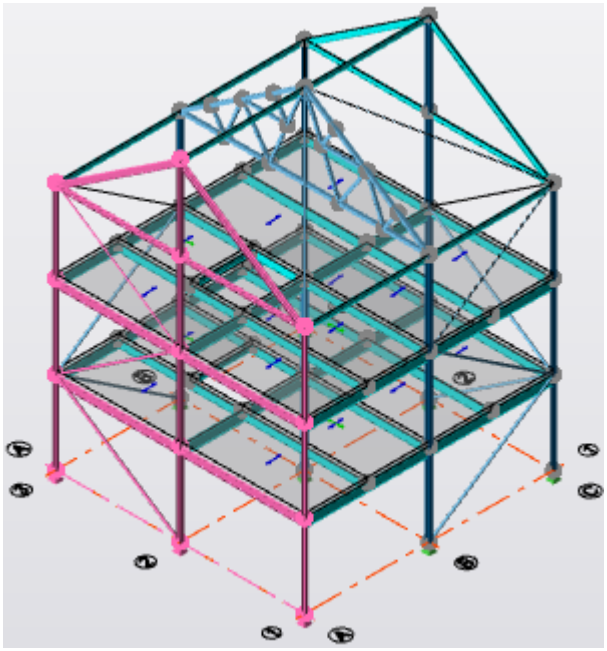
Remove a beam/column/wall and simultaneously remove its constituent spans/stacks/panels	If a continuous beam and its constituent spans are selected, they can all be deselected with a single mouse-click while holding down the **Ctrl** key. The same method can be applied to a continuous column and its constituent stacks, or a wall and its constituent panels.

To	Do this
Remove all entities	• Press **Esc**. The information displayed in the **Properties** window is cleared.

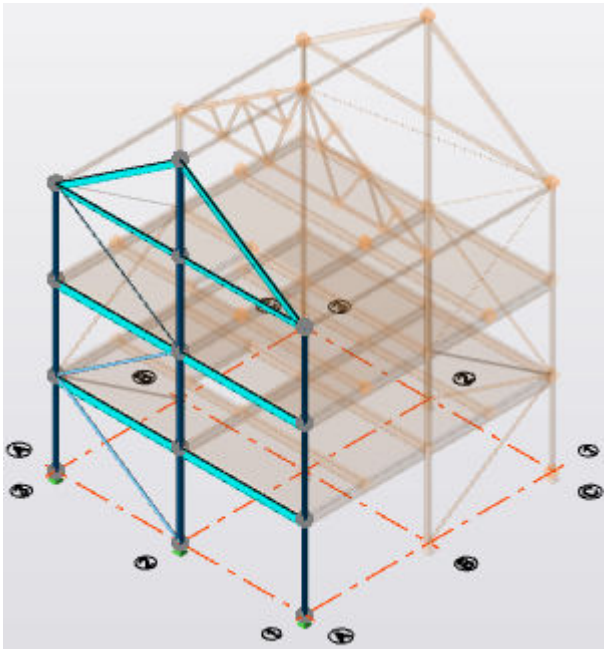
Use Ghost Unselected to focus on the selection

You can toggle **Ghost Unselected** on and off using the S button in the bottom corner of the view.

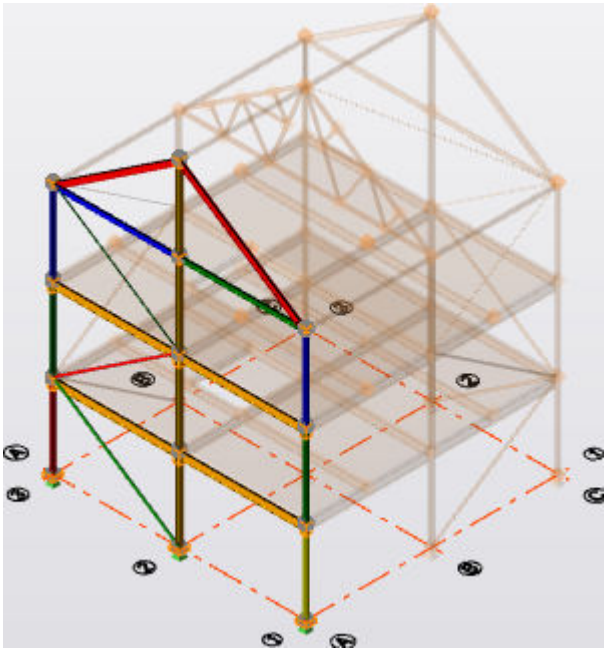
- **Ghost Unselected off**: selected entities are displayed in the *active selection color*, this can make it hard to distinguish between individual entities in the selection.



- **Ghost Unselected on**: selected entities are displayed in their nomal colors (appropriate to the Scene View that is active), unselected objects are semi-transparent. If nothing is selected, everything is displayed in its normal color.
  - In a Structure View, the selection would be displayed as:



- In a Review View showing Design Status, the selection would be displayed as:



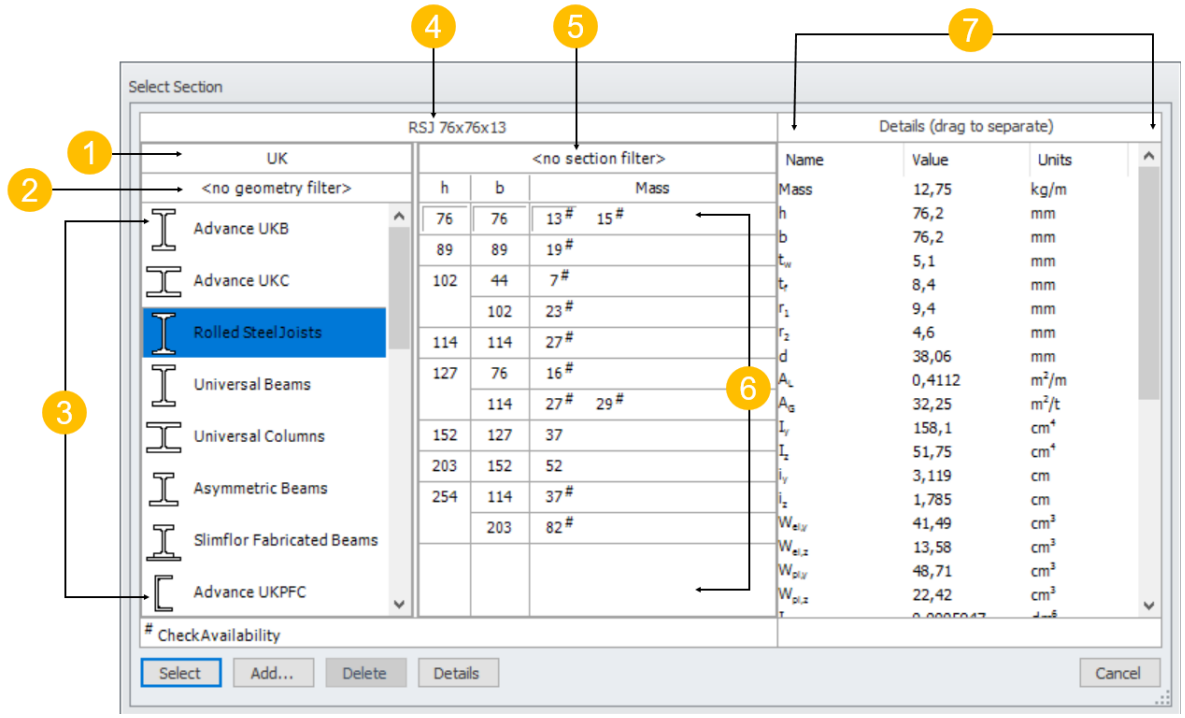
Select a section in the Select Section dialog box

Select Section dialog box allows you to select sections to use them in your model. You can also add and delete user-defined sections in the database by using the dialog, if necessary.

To access the **Select Section** dialog box:

1. Select the member you want to apply a section to.
2. In the **Properties** window, click the arrow on the right side of the **Section** field.
3. Click **<New\Edit...>**.

   The (page 1581) opens.



1. **Country** list
2. **Geometry filter**
3. **Page** pane
4. Selected section
5. **Section filter**
6. **Item** pane
7. **Details** pane

1. In the **Country** list, select the desired country.
2. In the **Page** pane on the left, select the desired geometry.

NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.

3. In the **Item** pane on the right, select the desired section size.
4. Click **Select**.

See also

Add user-defined sections (page 1266)

Edit entity properties

Any selection of one or more entities can be edited using the **Properties** window. Individual **model objects** can also have their properties edited using the **Properties** dialog box.

Edit properties using the Properties window

All entities can have their properties edited using the **Properties** window

1. Select the entity either in the graphical display or in the **Structure** tree.

   For more information, see Selection methods (page 100).
2. In the **Properties** window, modify the properties according to your needs.

Edit properties using the Properties dialog box

Individual **model objects** can have their properties edited using the **Properties** dialog box

1. Hover the mouse pointer over the object that you want to edit.
2. In the **Select Entity** tooltip, select the object.

   For more information, see Selection methods (page 100).
3. Right-click the object.
4. In the context menu, select **Edit**.

   The **Properties** dialog box opens.
5. Modify the properties according to your needs.
6. Click **OK**.

Edit properties of multiple entities

NOTE You can view and modify the common properties of multiple entities in the **Properties** window.

Note that:

- If the value of a certain item is not identical for all the selected entities, the cell appears blank in the **Properties** window.
- If you modify the blank cell, Tekla Structural Designer applies the new setting to all the selected entities.
- If you have selected multiple entities of different types, the property information is displayed separately for each type. Use the list at the top of the **Properties** window for moving between types.

1. Select the entities in the graphical display.

   For more information, see Selection methods (page 100).
2. If you have selected entities of different types, select the desired type in the list on top of the **Properties** window.
3. In the **Properties** window, modify the properties according to your needs.
4. Repeat steps 2–3 for entities of different types.

Re-position entities by moving nodes or edges

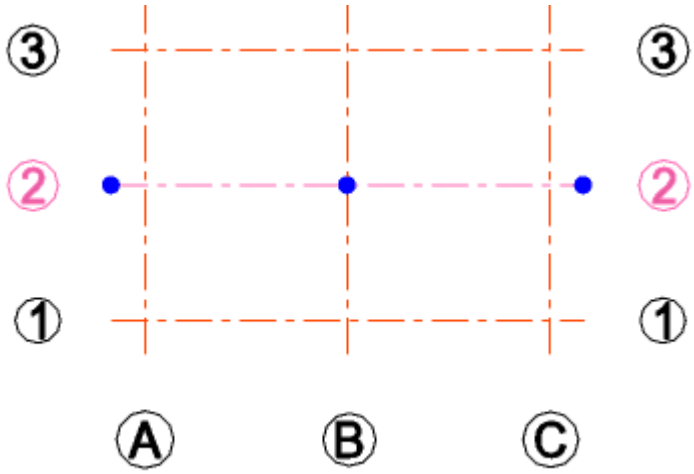
After selecting an entity you can re-position it by moving nodes or edges.

Modify one end of a grid or construction line

Grid and construction lines can only be modified in 2D views.

1. Select the grid or construction line.

The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click where you want to move the end node.

   Tekla Structural Designer redraws the line, moving the end node to the selected point.

Move a grid or construction line

Grid and construction lines can only be moved in 2D views.

1. Select the grid or construction line.

   The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click where you want to move the end node.

   Tekla Structural Designer redraws the line, moving the end node to the selected point.

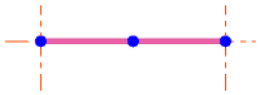
Modify one end of a member

To move an entire member, see: Move objects (page 271).

To move just one end of a member, proceed as follows:

1. Select the member in a 2D or 3D view.

   The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

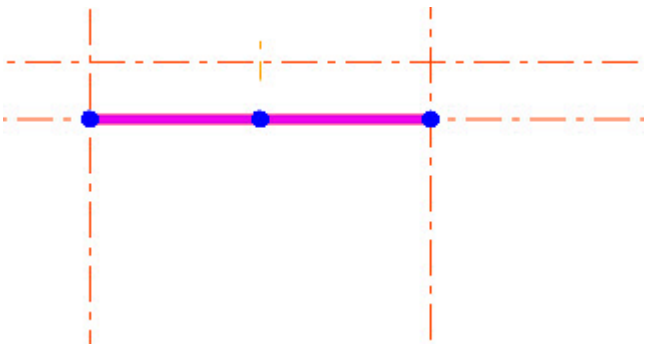
3. Click a grid or construction point where you want to move the end node.

   Tekla Structural Designer redraws the member, moving the end node to the selected point.

Add a new node into a beam

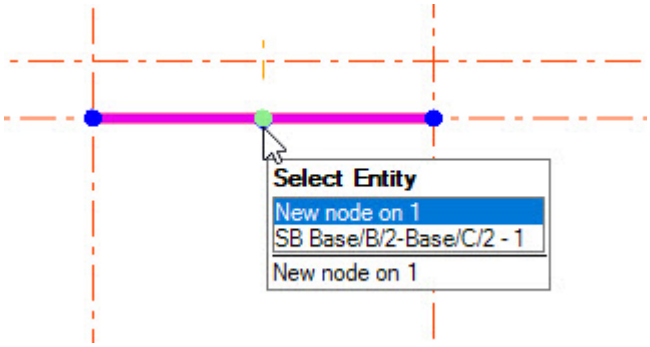
1. Select the beam in a 2D or 3D view.

   The nodes at each end and at the mid-point become visible.



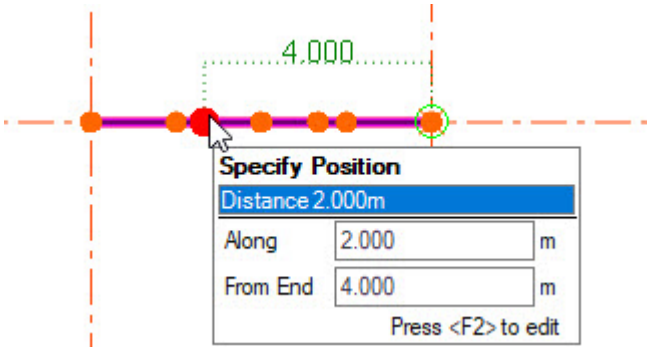
2. Select the node at the mid-point.

In the **Select Entity** tooltip this is described as a 'New node on ...'

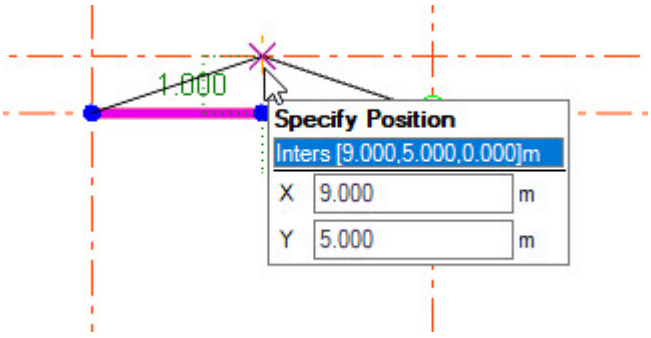


3. Click a point where you want to the new node to be created.

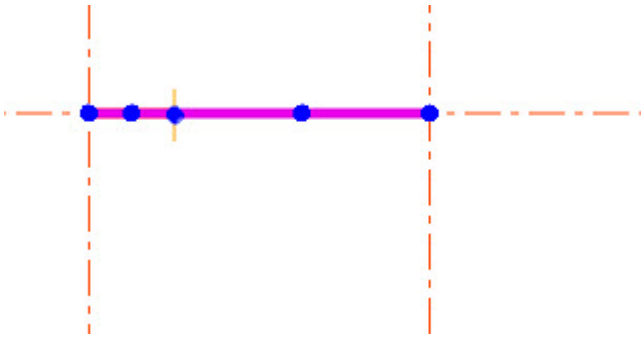
   This can either be a valid point within the span



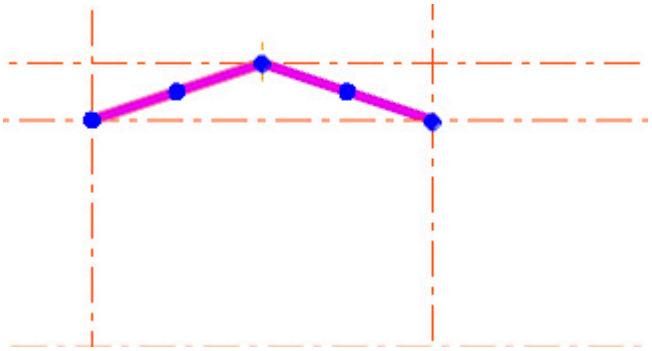
or, a a grid or construction point offset from the member centerline



The new node is introduced into the member and the original span is split according to the position specified, into two inline spans,



or two angled spans.



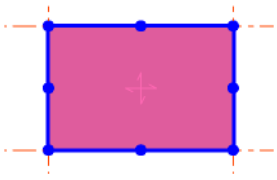
Modify slab items and panels by moving a node

To move entire slab items and panels, see: Move objects (page 271).

To move a slab/panel node, proceed as follows:

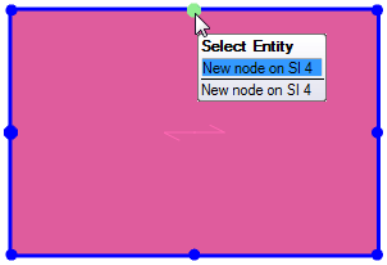
1. Select the slab item or panel that you want to modify in a 2D or 3D view.

   The nodes at each vertex and at the mid-point of each edge become visible.



2. Click a node to select it.

You can select either corner nodes, or nodes at the mid-points of the edge.



3. Move the mouse pointer over an existing construction point or intersection.
4. Click the point to reposition the node.

NOTE When you reposition nodes, note that:

- The new node position must be in the same plane as the slab item or panel.
- When you move a node at the mid-point of an edge, it becomes a new corner node. Tekla Structural Designer automatically creates two new mid-point nodes.
- When you move a corner node directly over an adjacent corner node, Tekla Structural Designer deletes the original corner node.

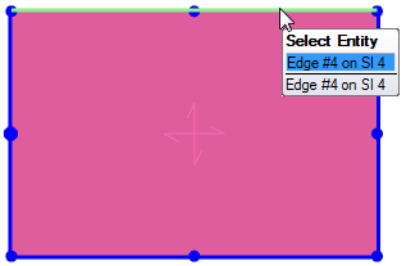
Modify slab items by moving an edge

To move entire slab items and panels, see: Move objects (page 271).

To move a slab/panel edge, proceed as follows:

1. Select the slab item that you want to modify in a 2D or 3D view.
2. Click an edge to select it.

   Ensure that you have selected an edge, and not a node.



3. Move the mouse pointer over an existing construction point or intersection.
4. Click to reposition the edge so that it passes through the selected point.

NOTE The new edge position must be in the same plane as the slab item.

Modify walls by moving a node

To move an entire wall, see: Move objects (page 271).

To move a wall node, proceed as follows:

1. Select the wall that you want to modify in a 2D or 3D view.

   The nodes at each vertex become visible.



2. Click a node to select it. If you want the result of moving the node to be a rectangular wall, the **Rectangular** property in **Properties** window should be selected, otherwise it should be unselected.
3. Move the mouse pointer over an existing construction point or intersection.
4. Click the point to reposition the node.

NOTE The new node position must be in the same plane as the wall.

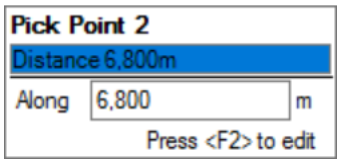
If the **Rectangular** property was checked, both the selected node and one other will be moved as required to form a rectangular wall.

Tips for basic tasks

Here we provide useful hints and tips that help you use the Tekla Structural Designer user interface and its basic features more efficiently.

Use the tooltip for input in a command

When you need to specify a distance or pick a point, a tooltip appears, displaying the current value.



1. Do one the following:
   - Move the mouse pointer to the required point and click the point to use it.
   - Press **F2** to type an exact number, and press **Enter**.

Undo a command

1. In the **Quick Access Toolbar**, click **Undo**.

Cancel a command or go back to the previous prompt

Yellow command prompts are viewed on the top of the active view to guide you through each command. To react to command prompts, see the following instructions.

- Do one of the following:

To	Do this
Cancel a command	• Press **Esc** at the first command prompt.
Go back to the previous command prompt	• Press **Esc** at a subsequent command prompt.

TIP Selecting another command also cancels the current command.

Example

The parallel grid line command consists of four steps. Hence, four consecutive command prompts are displayed.

1. Add Parallel Grid/Construction Line: Select Reference Line <press ESC to cancel>
2. Add Parallel Grid/Construction Line: Specify Distance **<press ESC to cancel>**
3. Add Parallel Grid/Construction Line: Pick Point 1 **<press ESC to cancel>**
4. Add Parallel Grid/Construction Line: Pick Point 2 **<press ESC to cancel>**

The grid line is created after the fourth prompt. After that, prompt 2 is automatically redisplayed.

At this point, you can either continue to create additional lines parallel to the original, or go back to the first prompt by pressing **Esc**.

Then, you can either select a new reference line and continue again, or cancel the command by pressing **Esc** again.

2.2 Create levels, grids, members, slabs, panels and supports

When you model in Tekla Structural Designer, you create and work with different types of model object. In most cases, a model object represents a member type that will exist in the real building or structure, or that will be closely related to it. You will also work with modeling aids such as grids and construction lines, that represent information that is only relevant when you are creating the model.

You can use the following modeling aids:

- Construction levels (page 121)
- Grids (page 123), construction lines (page 139) and free points
- Frames and slopes
- DXF reference drawings (page 134)
- Dimensions and measurements (page 256)

You can create the following physical member/object types:

- Beams, columns and braces (page 148)
- Walls and cores (page 197)
- Slabs and decks (page 216)
- Trusses and steel joists
- Portal frames
- Cold rolled members

The following panels/objects are useful for applying loads to the model:

- Wall panels and roof panels
-
-
-

You can also use the following general members/objects when required:

- Supports (page 242)
- Analysis elements (page 253)

Create and manage construction levels

In the **Construction Levels** dialog box, you can define and modify the levels that you need to construct your model. The levels can be floors, roofs, or intermediate levels required to define specific items.

Open the Construction Levels dialog

- Do one of the following:
  - On the **Model** tab, click **Construction Levels**.
  - In the **Structure** tree, double-click **Levels**.

Insert a single construction level

1. In the **Construction Levels** dialog box, select an existing level.
2. According to your needs, click either **Insert Above** or **Insert Below**.
3. If necessary, change the level type:
   - If you are creating a steel structure, set each new level as **T.O.S.** (top of steel).
   - If you are creating either a concrete or mixed material structure, set each new level as **S.S.L.** (structural slab level).
4. If necessary, change the level name.
5. For the new level, do one of the following:
   - Specify the height above the base in the **Level** field.

     The inter-story spacing is calculated automatically.
   - Specify the inter-story spacing in the **Spacing** field.

     The level is calculated automatically.

   NOTE A default height is calculated for the new level based on the spacings of any existing levels immediately above or below it. Either accept the height, or adjust it as required.
6. If you want Tekla Structural Designer to treat the level as a floor, select the **Floor** option.

Insert multiple construction levels

Multiple levels can be inserted, either above the current top level, or below the current bottom level.

1. In the **Construction Levels** dialog box, select an existing level.
2. According to your needs, click **New on Top** or **New at the Bottom**
3. For each construction level, type the level spacing and click **OK**.

TIP If you have new levels at 12' 6" , 18' 6" , 27' 6" , 36' 6" and 27" above the current level, you can specify the levels as 12' 6" , 18' 6" , 3*9'.

---

4. If necessary, change the level names.
5. If necessary, change the level type:
   - If you are creating a steel structure, set each new level as **T.O.S.** (top of steel).
   - If you are creating either a concrete or mixed material structure, set each new level as **S.S.L.** (structural slab level).

Make a level an identical copy of another level

If needed, you can determine a level as an identical copy of another level. In this case, all changes made to either the source level or the identical level are automatically applied to both levels. To make a level an identical copy of another level, see the following instructions.

1. Open the **Construction Levels** dialog box.
2. At the level you want to be a copy, click the arrow in the **Source** column.
3. Select the level that you want the current level to be identical to.
4. Click **OK**.

Make a level an independent copy of another level

If you determine a level as an independent copy of another level, any changes made to each level only apply to that level. To make a level an independent copy of another level, see the following instructions.

1. Open the **Construction Levels** dialog box.
2. At the level you want to be a copy, click the arrow in the **Source** column.
3. Select the level that you want the current level to be identical to.
4. Click **OK**.
5. Reopen the **Construction Levels** dialog box.
6. At the same level, click the arrow in the **Source** column again.
7. Set the level as **-unique-**.
8. Click **OK**.

Modify the properties of a construction level

You can modify some properties of construction levels in the **Construction Levels** dialog box dialogs, but other parameters can only be edited in the **Properties** window. For more information, see the following instructions.

1. In the **Structure** tree, click the **+** sign next to  **Levels**.
2. Click the name of the construction level that you want to modify.

   The properties of the level are viewed in the **Properties** window.
3. In the **Properties** window, modify the level properties according to your needs.

Delete construction levels

To delete construction levels, see the following instructions.

WARNING Deleting a construction level completely deletes the entire level and all its associated details, such as beams, members, or slab. You cannot recover the level once you have deleted it.

---

1. In the **Construction Levels** dialog box, select the level that you want to remove.
2. Click **Delete**.
3. Click **OK**.

NOTE You can also delete construction levels using the **Structure** tree. Note that :

a. Click the **+** sign next to  **Levels**.

b. Right-click the level that you want to remove.

c. In the context menu, select **Delete**.

Create and manage grid lines and architectural grids

Tekla Structural Designer allows you to create grid lines and arcs according to your needs. For detailed instructions, see the following paragraphs.

Overview

A collection of grid lines is referred to as an architectural grid. (The architectural grid which an individual line/ arc belongs being specified when the line/arc is created.)

Architectural grids

Multiple architectural grids can be used in the same model, each having the following characteristics:

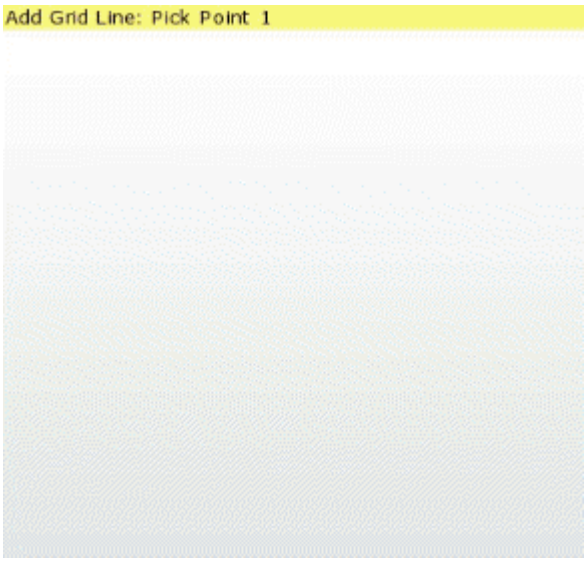
- They are a collection of grid lines that lie in a horizontal plane.
- Their properties can be accessed from the **Structure** tab of the **Project Workspace**, the following properties being available:
  - Color
  - Name
  - Grid line name visible
  - Vertical line visible
- Architectural grids can only displayed in a 3D scene view at the lowest level in the structure.
  - They are displayed by selecting **Architectural Grids** in **Scene Content**.
- Vertical lines can be projected from each grid intersection of each architectural grid. The vertical lines extend to the topmost level at which the architectural grid has been applied.
  - These are displayed by selecting **Architectural Grids** --> **VerticalLines** in .

Grid lines

- Grid lines (and arcs) can be created individually, or in an array. For detailed instructions, see the creation topics listed below.
- They can also be imported from a DXF file (page 134).
- Once created, they can be moved, rotated or extended (page 136).
- They can be numbered/renumbered (page 137) as required.
- They can be shown/hidden at specific levels (page 138) as required.
- Grid lines associated with a specific architectural grid are available at all levels and all sloped planes at which the architectural grid has been applied (page 139).

Create a single grid line

NOTE If you want to create a series of grid lines which form a regular or irregular, rectangular or radial grid line system, the **Rectangular Wizard** and **Sector Wizard** are the fastest options.



1. Ensure that the 2D view containing the construction level on which you want to create your grid line is active.
2. Go to the **Model** tab.
3. In the **Grid Line** list, select **Grid Line**.
4. In the **Properties** window, review the default next vertical/horizontal grid name and edit if required.
5. Select the point where you want the grid line to start.

   TIP The tooltip views the mouse pointer's exact coordinates. If the mouse pointer has not snapped to the required point, press **F2** to type the exact coordinates of the required point.

6. Select the point where you want the grid line to end.

   TIP The tooltip views either the absolute, relative, or polar coordinates of the end point, depending on whether the **ABS**, **REL**, or **POL** button is selected in the .

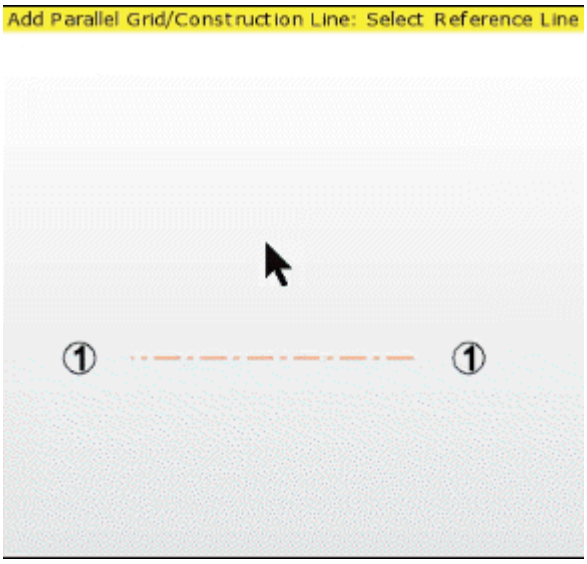
   - To switch the display, click one of the other buttons.

NOTE The grid line does not extend to infinity. Do ensure that the grid line is sufficiently long to meet your needs.

Tekla Structural Designer creates the grid line between the selected points.

Create parallel grid lines

This option creates a grid line parallel to an existing one, but of a different length. To use this option, you must have at least one existing grid line in the current 2D view.



1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Parallel**.
3. In the **Properties** window, type a name for the grid line.
4. Select the grid line to which your new grid lines will be parallel.
5. Move the mouse pointer to the distance where you want to place the parallel grid line.

   TIP The tooltip views the distance of the dotted line from the initial grid line you selected in step 3. Press **F2** to type the exact distance.

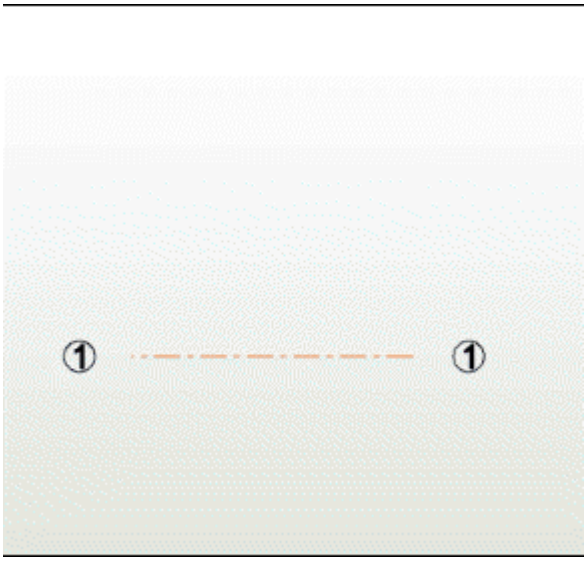
   For more accuracy, you can zoom in.

6. Click to locate the new grid line.

7. Click the points where you want the new grid line to start and end.

   The construction line is now created.

8. Move the mouse pointer and place other grid lines with respect to the line you selected in step 3, or press **Esc** to end grid placement.

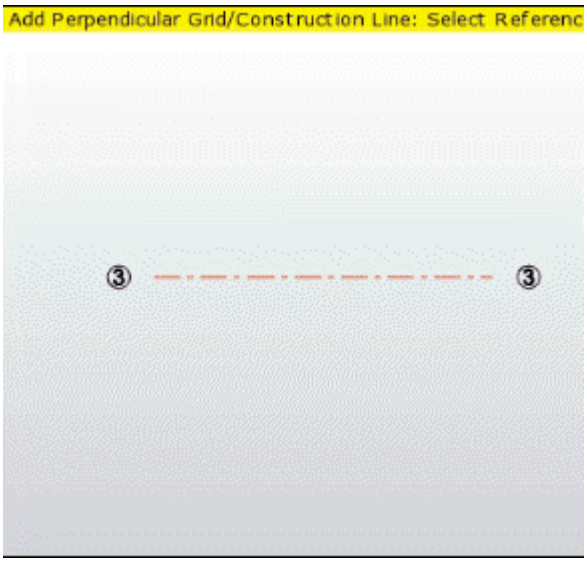


TIP To create parallel grid lines of the same length as the selected gridline, select the **Parallel (quick)** command.

When you create quick parallel grid lines, you can press **F2** to define the distance between successive pairs of grid lines, separating the numbers with commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

For example, you can specify bay centers of 18', 27' , 18' 6" , 18' 6" , 18' 6" and 27" as `18' , 27' , 3x18' 6" , 27"`.

Create perpendicular grid lines



To use this option, you must have at least one existing grid line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Perpendicular**.
3. In the **Properties** window, type a name for the grid line.
4. Select the grid line to which your new grid line will be perpendicular.
5. Move the new grid line to the desired position and click to locate it.

   TIP The tooltip views the perpendicular distance of the dotted line from the middle of the grid line you selected. Press **F2** to type the exact distance.

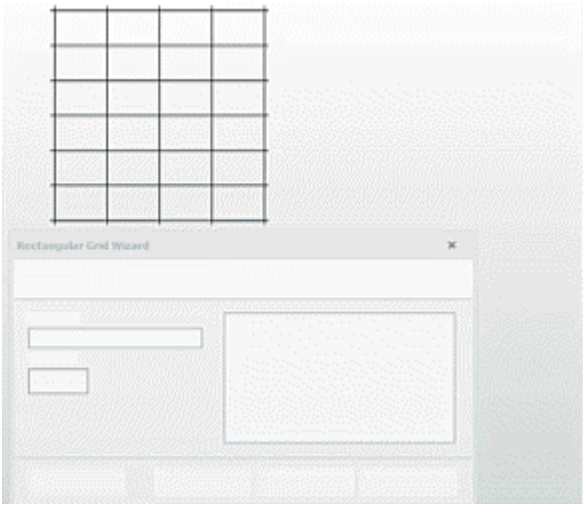
6. Click the points where you want the new grid line to start and end.
7. Move the mouse pointer and place other grid lines with respect to the line you selected in step 4, or press **Esc** to end grid placement.

Create a rectangular grid line system

The **Rectangular Wizard** can be used to define a grid line array defined relative to the global coodinate system X and Y axes.

NOTE The X direction lines can be rotated to an angle from the global X axis. While the Y direction lines are perpendicular to the X direction lines by

default, this can be changed by specifying a different angle between the axes.



1. Ensure that the 2D view representing the construction level on which you want to create your rectangular grid line system is active.
2. On the **Model** tab, in the **Grid Line** list, select **Rectangular Wizard**.

   The **Rectangular Grid Wizard** opens.
3. Type a name for the grid and select a color for the grid lines.
4. Ensure that each construction level on which you want to create the grid layout is selected.
5. Click **Next**.
6. Define the origin of the grid line system either by clicking it in the 2D view, or by entering its coordinates in the **Rectangular Grid Wizard**.
7. Click **Next**.
8. Select if you want to create grid lines in the X direction only, in the Y direction only, or in both directions.
9. Select the style that you want to use for the grid lines.
10. Click **Next**.
11. On the following pages, define the layout of grids for the bays in the X and Y directions.

NOTE You can define either a regular or an irregular grid layout:

- For a regular grid, define the number of bays you want to create and the bay centers.
- For an irregular grid, define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x18'6"`).

You can also specify the reference of the first grid line, and by how much you want to increment that reference to give the references of the other lines that you create.

- For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M, and so on.

---

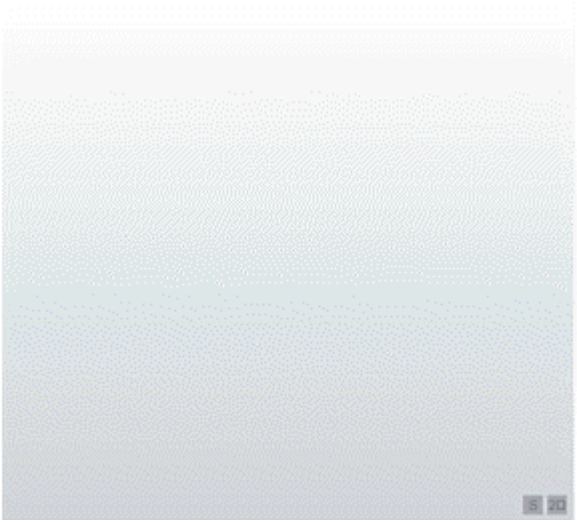
12. Click **Next**.
13. Define the rotation of the grid either by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**.

TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the grid line system's local x or local y direction. This is helpful if you are creating a grid line system that is not orthogonal.

---

14. Click **Next**.
15. Specify the angle between the axes with respect to either the X or Y axis system.
16. Click **Finish**.

Create a radial grid line system



1. Ensure that the 2D view representing the construction level on which you want to create your radial grid line system is active.
2. On the **Model** tab, in the **Grid Line** list, select **Radial Grid Wizard**.

   The **Radial Grid Wizard** opens.
3. Type a name for the grid and select a color for the grid lines.
4. Ensure that each construction level on which you want to create the grid layout is selected.
5. Click **Next**.
6. Define the origin of the grid line system either by clicking it in the 2D view, or by entering its coordinates in the **Radial Grid Wizard**.
7. Click **Next**.
8. Select whether you want to create the radial lines only, the arcs only, or both of these.
9. Select the style that you want to use for the grid lines.
10. Click **Next**.
11. Define the layout of the arcs that you will create.

NOTE You can define either a regular or an irregular grid layout:

- For a regular grid, define the number of bays you want to create and the bay centers.

- For an irregular grid, define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x10'6"`).

You can also specify the reference of the first grid line, and by how much you want to increment that reference to give the references of the other lines that you create.

- For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M, and so on.

You can also select whether the arc grid lines are represented as curves, or as a series of straight lines between the points where the arc intersects the other grid lines created as part of this process.

---

12. Click **Next**.
13. Define the layout of radial grid segments that you want to achieve.

    Again, you can specify the reference of the first grid line, and how you want to increment the reference to give the references of the other radial lines that you create.
14. Click **Next**.
15. Define the rotation of the grid either graphically by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**

TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the grid line system's local x or local y direction.

---

16. Click **Finish**.

Create a grid arc



1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Arc**.
3. In the **Properties** window, type a name for the grid line.
4. Select the point which lies at the center of the grid arc which you want to create.

   The tooltip views the mouse pointer's exact coordinates.
5. Click the points where you want the grid arc to start and end.
6. Move the mouse pointer and place other grid arcs with respect to the line you selected in step 4, or press **Esc** to end grid placement.

Related information

See also

Import grids or a shadow from a DXF file (page 134)

Number and renumber grids (page 137)

Choose the levels at which to show/hide grid lines and construction lines (page 138)

Apply an architectural grid to a specific level (page 139)

Video: Construction lines, grid lines and architectural grids (US customary units)

Video: Construction lines, grid lines and architectural grids (metric units)

Video: Grid and Construction Inference Lines

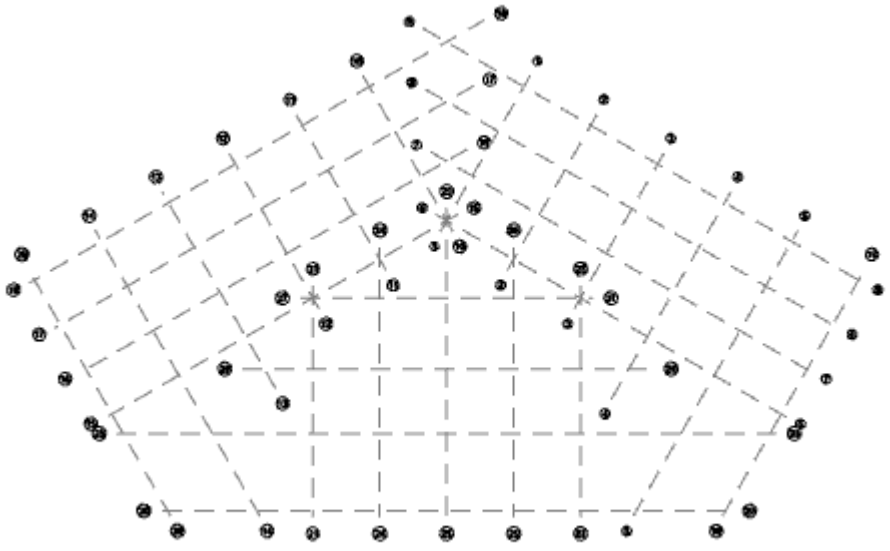
Import grids or a shadow from a DXF file

You can import a DXF file either to create grid lines in specified levels in your Tekla Structural Designer model, or to use the dxf as a shadow to assist in the placement of entities in 2D scene views

1. Open a 2D view of a construction level, and go to the **Model** tab on the ribbon.
2. In the **Grid Line** list, select **Import DXF**.

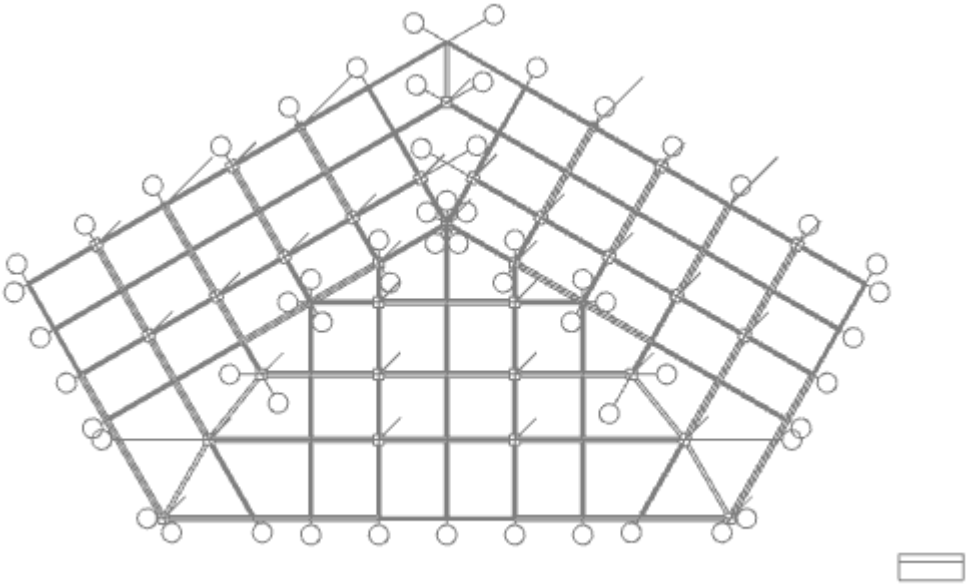
   An **Open** dialog opens.
3. Browse to the file that you want to import, select the file, and click **Open**.

   The **DXF Import Wizard** opens.
4. On the first page, manage the layers and colors that you want to import.
5. Select if you want to import the architectural grids from the file, or import the file as a shadow.
   - If you import the architectural grids, all elements in the selected layers of the DXF file are mapped to the Tekla Structural Designer grid lines. You must therefore ensure that you switch off all the layers in the DXF file, apart from the layers in which the grids have been defined.



   - If you import the DXF file as a shadow, Tekla Structural Designer imports the .dxf file but does not create any Tekla Structural Designer objects. You can use the intersection points and other elements as the

source on which to add the Tekla Structural Designer objects you require.



6. If necessary, adjust the scale and offsets for the DXF file.
7. If you have selected the **Architectural Grid(s)** option, click **Next**, otherwise if you have selected the **Shadow** option, please proceed to step 10.
8. Select how the grids are created:
   - To create separate named grids for each layer, select **By layer**.
   - To create separate named grids for each color in the DXF file, select **By color**.
   - To create a single merged grid containing all layers and colors, select **Merged**.
9. Ensure that each level to which you want to import the grid layout is selected.
10. Click **Finish**.

NOTE If you import the dxf as a shadow at a level, you can choose to have the shadow visible or hidden by adjusting the **Shadow visible** setting in the Level properties.

See also

Create and manage grid lines and architectural grids (page 123)

Extend, move, or rotate grid lines and arcs

You can modify grid lines and arcs in different ways according to your needs. You can stretch, shorten, or rotate grid lines, or move them in a perpendicular direction. As for grid arcs, you can stretch and shorten them, adjust their radius, or move them in any direction. For detailed instructions, see the following paragraphs.

RESTRICTION	You can only move grid lines and arcs in 2D views.

Stretch, shorten, or rotate a grid line

1. In a 2D view, select the grid line that you want to modify.

   The end nodes and the center node of the grid line become visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The node moves to its new position.

Move a grid line to a perpendicular direction

1. In a 2D view, select the grid line that you want to modify.

   The end nodes and the center node of the grid line become visible.
2. Click the center node.
3. Click the location where you want to move the node.

   The grid line moves to its new position.

Stretch or shorten a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The arc stretches or shortens accordingly.

Adjust the radius of a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end and center nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc perimeter.
3. Click the location where you want to move the node.

   The arc radius adjust accordingly.

Move a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc.
3. Click the location where you want to move the node.

   The arc moves to its new location.

See also

Create and manage grid lines and architectural grids (page 123)

Number and renumber grids

Each grid that you create is automatically named. The name is enclosed in a bubble when text width is less than 1.5*text height otherwise a rectangle is used.

Set the initial number or letter used for grids

1. On the **Home** tab, click  **Model Settings**.
2. Go to **References** --> **General**.
3. In the **Grid Line Naming** section, set the initial values and the naming style.
4. Click **OK**.

Renumber all grids

1. In the **Structure** tree, right-click  **Architectural Grids**.
2. In the context menu, select **Renumber**.

   A dialog box is displayed which allows you to renumber the grid line based on

   - the type (i.e. numeric, or alphanumeric) from direction (default ON) or
   - choose grid name type and direction
3. Select the parameters as required, then click OK.

   All grids in the model are renumbered in the chosen sequence.

Change the name of a grid line or grid arc

To change the name of a single grid line or grid arc, see the following instructions.

1. Select the grid line or grid arc that you want to rename.

2. In the **Properties** window, expand the **Name** property so that **User name** is displayed.
3. Enter the new name as the **User name**.

Change the name or color of an architectural grid
To modify the properties of an existing architectural grid, see the following instructions.

1. In the **Structure** tree, click the **+** sign next to  **Architectural Grids**.
2. Select the architectural grid whose name you want to change.
3. In the **Properties** window, modify the architectural grid's properties according to your needs.

Choose the levels at which to show/hide grid lines and construction lines

1. In the **Project Workspace**, go to the **Structure** tab.
2. Open **Levels**.
3. In the **Properties** window:
   - **Show grids in plane view** - select/unselect to show/hide grid lines in every level scene view
   - **Show grids in 3D view** - select/unselect to be show/hide grid lines at every level in 3D scene views
4. To show/hide grid lines and construction lines at a specific levels, select the level in the **Project Workspace**
5. In the **Properties** window:
   - **Show grids in plane view** - select/unselect to show/hide grid lines at the selected level in 2D scene views
   - **Show grids in 3D view** - select/unselect to be show/hide grid lines at the selected level in 3D scene views

   When the above **Show grids** options have been selected, you can further control which of grid line components (i.e. lines, text labels) get displayed in individual scene views by selecting appropriate **Grid & Construction Lines** options in **Scene Content**.

NOTE You can choose the sloped planes at which to show/hide grid lines and construction lines in a similar way.

NOTE If grid lines are not visible in the 3D view at levels where you require them to be, ensure that grids are applied at the levels in question. For

further information, see Apply an architectural grid to a specific level (page 139).

Apply an architectural grid to a specific level

To view an existing architectural grid at a specific level of your model, you must apply the grid to the level.

1. In the **Project Workspace**, go to the **Structure** tab.
2. Open  **Architectural Grids**.
3. Right-click on the architectural grid name you want to apply.
4. In the context menu, select **Edit <grid name>**
5. Ensure that the levels where you want the architectural grid to be applied are selected.

See also

Create and manage grid lines and architectural grids (page 123)

Create and manage construction lines

Construction lines serve a similar purpose to grid lines in that they facilitate the placement of entities.

- They can be created individually, or in an array. For detailed instructions, see the creation topics listed below.
- Once created they can be moved, rotated or extended (page 144).
- They can also be shown/hidden at specific levels (page 138) as required.

Construction lines differ from grid lines as follows:

- Construction lines are not associated with a specific architectural grid and therefore do not automatically repeat over multiple construction levels.
- Construction line names are not displayed in balloons at the ends of the lines.

Create a single construction line

NOTE If you want to create a series of construction lines which form a regular or irregular, rectangular or radial construction line system, the **Rectangular Wizard** and **Sector Wizard** are the fastest options.

1. Ensure that the 2D view representing the construction level on which you want to create your construction line is active.
2. Go to the **Model** tab.
3. In the  **Construction Line** list, select  **Construction Line**.
4. Select the point where you want the construction line to start.

   TIP The tooltip views the mouse pointer's exact coordinates. If the mouse pointer has not snapped to the required point, press **F2** to type the exact coordinates of the required point.

5. Select the point where you want the construction line to end.

   TIP The tooltip views either the absolute, relative, or polar coordinates of the end point, depending on whether the **ABS**, **REL**, or **POL** button is selected in the .

   - To switch the display, click one of the other buttons.

   NOTE The construction line does not extend to infinity. Do ensure that the construction line is sufficiently long to meet your needs.

   Tekla Structural Designer creates the construction line between the selected points.

Create parallel construction lines

This option creates a construction line parallel to an existing one, but of a different length. To use this option, you must have at least one existing construction line in the current 2D view.

1. Go to the **Model** tab.
2. In the  **Construction Line** list, select  **Parallel**.
3. Select the construction line to which your new construction lines will be parallel.
4. Move the mouse pointer to the distance where you want to place the parallel construction line.

   TIP The tooltip views the distance of the dotted line from the initial construction line you selected in step 3. Press **F2** to type the exact distance.

   For more accuracy, you can zoom in.

5. Click to locate the new construction line.
6. Click the points where you want the new construction line to start and end.

   The construction line is now created.
7. Move the mouse pointer and place other construction lines with respect to the line you selected in step 3, or press **Esc** to end construction placement.

TIP To create parallel construction lines of the same length as the selected construction line, select the  **Parallel (quick)** command.

When you create quick parallel construction lines, you can press **F2** to define the distance between successive pairs of construction lines, separating the numbers with commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

For example, you can specify bay centers of 18', 27' , 18' 6" , 18' 6" , 18' 6" and 27" as `18' , 27' , 3x18' 6" , 27"`.

Create perpendicular construction lines

To use this option, you must have at least one existing grid or construction line in the current 2D view.

1. Go to the **Model** tab.
2. In the  **Construction Line** list, select  **Perpendicular**.
3. Select the construction line to which your new construction line will be perpendicular.
4. Move the new construction line to the desired position and click to locate it.

TIP The tooltip views the perpendicular distance of the dotted line from the middle of the construction line you selected. Press **F2** to type the exact distance.

---

5. Click the points where you want the new construction line to start and end.
6. Move the mouse pointer and place other construction lines with respect to the line you selected in step 4, or press **Esc** to end construction placement.

Create a rectangular construction line system

1. Ensure that the 2D view representing the construction level on which you want to create your rectangular construction line system is active.
2. On the **Model** tab, in the **Construction Line** list, select **Rectangular Wizard**.

   The **Rectangular Grid Wizard** opens.
3. Define the origin of the construction line system either by clicking it in the 2D view, or by entering its coordinates in the **Rectangular Grid Wizard**.
4. Click **Next**.
5. Select if you want to create construction lines in the X direction only, in the Y direction only, or in both directions.
6. Select the style that you want to use for the construction lines.
7. Click **Next**.
8. On the following pages, define the layout of constructions for the bays in the X and Y directions.

   NOTE You can define either a regular or an irregular construction layout:
   - For a regular construction, define the number of bays you want to create and the bay centers.
   - For an irregular construction, define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x18'6"`).
9. Click **Next**.
10. Define the rotation of the construction either by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**.

    TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the construction line system's local x or local y direction. This is helpful if you are creating a construction line system that is not orthogonal.
11. Click **Next**.
12. Specify the angle between the axes with respect to either the X or Y axis system.
13. Click **Finish**.

Create a radial construction line system

1. Ensure that the 2D view representing the construction level on which you want to create your radial construction line system is active.
2. On the **Model** tab, in the **Construction Line** list, select **Radial Grid Wizard**.

   The **Radial Grid Wizard** opens.
3. Define the origin of the construction line system either by clicking it in the 2D view, or by entering its coordinates in the **Radial Grid Wizard**.
4. Click **Next**.
5. Select whether you want to create the radial lines only, the arcs only, or both of these.
6. Select the style that you want to use for the construction lines.
7. Click **Next**.
8. Define the layout of the arcs that you will create.

   NOTE You can define either a regular or an irregular construction layout:

   - For a regular construction, define the number of bays you want to create and the bay centers.
   - For an irregular construction, define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x10'6"`).

   You can also select whether the arc construction lines are represented as curves, or as a series of straight lines between the points where the arc intersects the other construction lines created as part of this process.

   ---

9. Click **Next**.
10. Define the layout of radial construction segments that you want to achieve.

    Again, you can specify the reference of the first construction line, and how you want to increment the reference to give the references of the other radial lines that you create.
11. Click **Next**.
12. Define the rotation of the construction either graphically by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**

TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the construction line system's local x or local y direction.

13. Click **Finish**.

Create construction arcs

1. Go to the **Model** tab.
2. In the  **Construction Line** list, select  **Arc**.
3. Select the point which lies at the center of the construction arc which you want to create.

   The tooltip views the mouse pointer's exact coordinates.
4. Click the points where you want the construction arc to start and end.
5. Move the mouse pointer and place other construction arcs with respect to the line you selected in step 3, or press **Esc** to end construction placement.

Related information

See also

Video: Construction lines, grid lines and architectural grids (US customary units)

Video: Construction lines, grid lines and architectural grids (metric units)

Video: Grid and Construction Inference Lines

Extend, move, or rotate construction lines and arcs

You can modify construction lines and arcs in different ways according to your needs. You can stretch, shorten, or rotate construction lines, or move them in a perpendicular direction. As for construction arcs, you can stretch and shorten them, adjust their radius, or move them in any direction. For detailed instructions, see the following paragraphs.

RESTRICTION You can only move construction lines and arcs in 2D views.

Stretch, shorten, or rotate a construction line

1. In a 2D view, select the construction line that you want to modify.

   The end nodes and the center node of the construction line become visible.
2. Click one of the end nodes.

3. Click the location where you want to move the node.

   The node moves to its new position.

Move a construction line to a perpendicular direction

1. In a 2D view, select the construction line that you want to modify.

   The end nodes and the center node of the construction line become visible.
2. Click the center node.
3. Click the location where you want to move the node.

   The construction line moves to its new position.

Stretch or shorten a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The arc stretches or shortens accordingly.

Adjust the radius of a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end and center nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc perimeter.
3. Click the location where you want to move the node.

   The arc radius adjust accordingly.

Move a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc.
3. Click the location where you want to move the node.

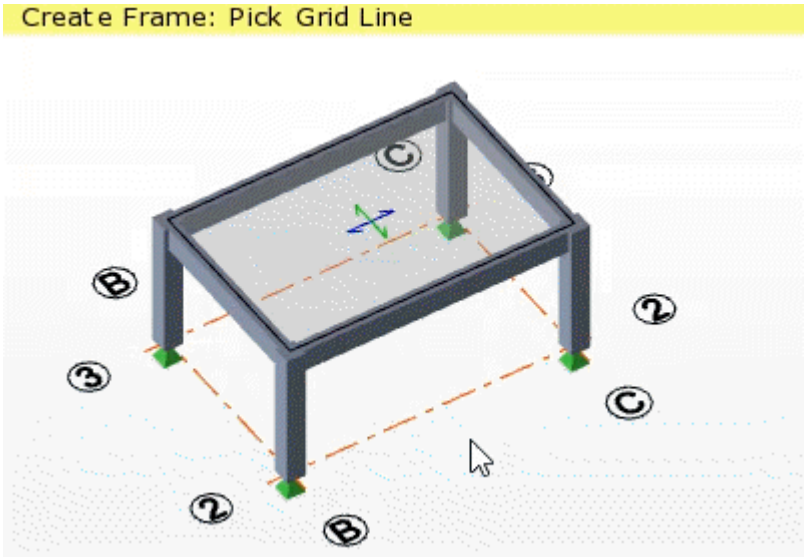
   The arc moves to its new location.

Create frames and slopes

A **frame** is a 2D view of the model, created in a vertical plane defined by an existing grid line. Since only the members that lie within the plane of the frame are displayed, a frame view can be particularly useful for defining bracing.

A **slope** is a 2D view of the model, created in an inclined plane. You can define a slope by selecting 3 existing grid points. Since only those members that lie within the plane of the slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps.

Create a frame



1. Obtain a 3D view of your model where you can see the base grid line associated with the frame that you want to create.
2. On the **Model** tab, click **Frame**.
3. Position the mouse pointer over the grid line for the frame that you want to create.
4. Click to create the frame.

   A frame for the selected grid line is added to the **Frames** branch of the **Structure** tree and a rectangle is displayed in the 3D view to illustrate the frame extents.

TIP To open a 2D view of the frame:

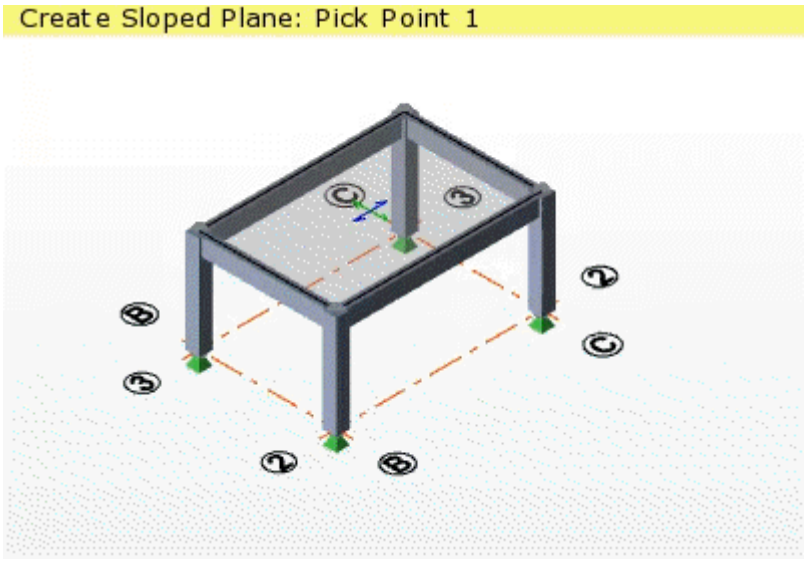
a. Click the **+** sign on the left side of **Frames** in the **Structure** tree.

b. Double-click the name of the frame view you want to open.

Create a slope

NOTE Before creating a slope, note that:

- In order to create a slope, you need to be able to click three existing (not co-linear) grid points that lie in the plane of the slope.
- Grid points are formed at grid line intersections on construction levels. Therefore, if the points required to define the slope do not currently exist, you may need to insert new construction levels and grid lines to form them.
- A sloped plane must be entirely contained within a single sub model because the sub model determines the mesh parameters to be applied.
- In a sloped plane, positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.



1. Obtain a 3D view of your model where you can see three grip points that define the sloped plane.

TIP If you cannot see the grid line intersections on a particular construction level in the 3D view:

a. In the **Structure** tree, select the level.

b. In the **Properties** window, select the **Show grids in the 3D view** option.

If you wish, you can clear the **Show grids in the 3D view** option for other levels to simplify the display.

---

2. On the **Model** tab, click  **Sloped Plane**.
3. Click the three points which define the sloped plane.

A sloped plane is added to the **Slopes** branch of the **Structure** tree and a polygon is displayed in the 3D view to illustrate the slope.

TIP To open a 2D view of the slope:

a. Click the **+** sign on the left side of  **Slopes** in the **Structure** tree.

b. Double-click the name of the slope view you want to open.

Show frame or slope references

The **Show references** command can be used to list all entities referenced by a selected frame or slope.

1. Open the **Frames** or **Slopes** branch in the **Structure** tree.
2. Right-click on the required frame or slope name.
3. From the context menu, select **Show References...**

All entities that are referenced by the selected frame or slope are listed in a dialog box. If entities are selected from the list they are highlighted in the 3D Structural view and their properties are displayed in the **Properties** window.

Create beams, columns and braces

These topics introduce you to the methods of creating beams columns and braces (in any material).

We recommend you familiarize yourself with how to:

- Create beams (page 167)
- Create columns (page 149)
- Create braces (page 190)

Create columns

This section focuses on the operations required to create columns (in any material).

- Specify the column type and section size (page 149)
- Create a single column or series of columns (page 152)
- Create inclined columns and cranked columns (page 154)
- Create gable posts or parapet posts (page 155)
- Align a column to a specific angle or an angled... (page 156)
- Modify the position of columns and column stacks (page 156)

The following topics are relevant to steel columns only:

- Setting out steel and cold formed columns (page 157)
- Create plated or compound section steel columns (page 158)
- Specify a column splice (page 163)
- Add a base plate to a steel column (page 163)
- Create web openings in steel members (page 186)
- Manage design section orders (page 1272)

Concrete columns specifically have an automatic alignment facility:

- Specify concrete column alignment relative to the grid (page 164)
- Modify the column alignment or specify offsets (page 165)

Related information

See also

Column properties (page 1374)

Video: Creating and editing columns (US customary units)

Video: Creating and editing columns (metric units)

Specify the column type and section size

Before you can place a column you must first specify the column type and an initial section size.

Specify the type of column

- On the **Model** tab, do one of the following:

To	Do this
Specify a steel column	1. Click the arrow under  **Steel Column**.

	2. In the list, select the **Steel Column** type.
Specify a plated or compound steel column	1. Click the arrow under  **Steel Column**. 2. In the list, select the **Plated** column type.
Specify a concrete filled steel column	1. Click the arrow under  **Steel Column**. 2. In the list, select the **Concrete Filled** column type.
Specify a concrete encased steel column	1. Click the arrow under  **Steel Column**. 2. In the list, select the **Concrete Encased** column type. 3. In the**Properties** window, click **Encasing section**. 4. Click the arrow on the right side of **Encasing section**. 5. In the list that appears, select **<New\Edit...>** 6. Enter the size of encasing concrete section.
Specify a cast-in-place, or precast concrete column	• 1. Click the arrow under  **Concrete Column**. 2. In the list that appears, select **Concrete Column** for cast-in-place, or **Precast**.
Specify a timber column	Click  **Timber Column**.
Specify a cold formed column	In the **Cold Formed** group, click  **Column**.

Specify the size of steel, cold formed and timber columns

1. Specify the type of column, (see the instructions above for how to do this).
2. In the **Properties** window, click **Section**.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, select **<New\Edit...>**

   The (page 109) opens.
5. Select the desired section size, and click **Select**.

   TIP To define a custom section, click **Add...**
6. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of plated or compound steel columns

1. Specify the type of column, (see the instructions above for how to do this).
2. In the **Properties** window, click **Section**.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, select **<New\Edit...>**

   The (page 109) opens.
5. In the (page 109), choose the required compound section type from the left hand pane.
6. Select the desired section size, and click **Select**.

TIP If the desired section is not listed, you can add it as follows:

a. Click **Add...**

b. Type the sections, plate dimensions, and gaps according to your needs.

c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

d. Select the section in the **Select Section** dialog box.

---

7. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of cast-in-place, or precast concrete columns

1. Specify the type of column, (see the instructions above for how to do this).
2. In the **Properties** window, click **Section**.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, select **<New\Edit...>**

   The **Section** dialog box opens.
5. Select the column shape.
6. Define the dimensions of the column.

NOTE Only click **Add** if you want to create a hollow column.

Then, to define the void:

a. In the tabular part of the dialog, select the shape and dimensions of the void.

b. Leave the minor and major offsets as 0.0 to position the void centrally in the column, or adjust as necessary to create an offset.

7. Click **OK**
8. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Related information

See also

Video: Creating and editing columns (US customary units)

Video: Creating and editing columns (metric units)

Create a single column or series of columns

With Tekla Structural Designer, you can create various kinds of column. You can model columns in 2D views, frame views, and structure views.

Create columns in a level view

1. Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie.
2. Select the column type and size (page 149).
3. In the **Properties** window , adjust the base level and top level of the column, and other column properties, if necessary.
4. Do one of the following:

To	Do this
Create a single column	a. Click the point where you want to place the column. b. Press **Esc** to finish creating columns.
Create a series of columns	a. Move the mouse pointer to one corner of an imaginary box that will encompass the grid intersection points where you want to create columns. b. Hold down the left mouse button. c. Drag the mouse pointer to the diametrically opposite corner of the box. d. Release the mouse button.

Create a single column in a frame or structure view

NOTE In order to define a column in a frame view or a structure view, you must have already created the construction levels between which the column will run, and the grid points between which it will lie.

1. Select the column type and size (page 149).

2. Click the start point of the column.
3. Click the end point of the column.

   Tekla Structural Designer creates the column between the selected points.

Create plated or compound section steel columns

1. On the **Model** tab, click the arrow under  **Steel Column**.
2. In the list that appears, select  **Plated**.
3. In the **Properties** window, click the arrow next to the **Section** property.
4. In the list that appears, select **<New\Edit...>**

   The **Select Section** dialog box opens.
5. Select the desired section type and section size.

TIP If the desired section is not listed, you can add it as follows:

a. Click **Add...**

b. Type the sections, plate dimensions, and gaps according to your needs.

c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

d. Select the section in the **Select Section** dialog box.

---

6. In the **Properties** window, adjust the remaining properties according to your needs.
7. To proceed, see **Create columns in a 2D view** and **Create a single column in a frame view or a structure view**.

TIP In order to apply a plated or compound section an existing steel column, do the following:

a. In the **Properties** window, set **Fabrication** to **Plated**.

b. Click the arrow next to the **Section** property.

c. In the list, either select a compound section that has already been applied in the model, or select **<New\Edit...>** to add a new one.

Create concrete filled or encased concrete columns

1. On the **Model** tab, click the arrow under  **Steel Column**.
2. In the list that appears, select either **Concrete Filled** or **Concrete Encased**.

   The column adopts the properties that are currently displayed in the **Properties** window for creating steel columns.
3. In the **Properties** window, adjust the properties as necessary.
4. To proceed, see **Create columns in a 2D view** and **Create a single column in a frame view or a structure view**.

NOTE You can now modify the **Section** property and specify a different section size above the space position.

Related information

See also

Create gable posts or parapet posts (page 155)

Video: Creating and editing columns (US customary units)

Video: Creating and editing columns (metric units)

Create inclined columns and cranked columns

Tekla Structural Designer allows you to create inclined and cranked steel and concrete columns.

Create an inclined column

An inclined column is any column that is not truly vertical. In order to define an inclined column, you must have defined the construction levels between which the column will run, and the grid points between which it will lie.

NOTE An inclined column can only be created in a frame view or a structure view.

1. Select the column type and size. (page 149)
2. In a frame or structure view, Click the start point of the column.
3. Click the end point of the column.

   Tekla Structural Designer creates the column between the selected points.

Create a cranked column

In order to define a cranked column, you must have created the construction levels between which the column will run and the grid points between which it will lie.

NOTE An inclined column can only be created in a frame view or a structure view.

1. Select the column type and size. (page 149)
2. Click the start point of the column.
3. Hold down the **Ctrl** key and click the next node of the column.
4. Repeat step 3 until you have selected the second to last node of the column.
5. Release the **Ctrl** key and click the end point of the column.

   Tekla Structural Designer creates the column.

   TIP Instead of holding down the **Ctrl** key while clicking the column nodes, you can simply ensure the **Continuous** property is selected in the **Properties** window to achieve the same result.

Related information

See also

Video: Creating and editing columns (US customary units)

Video: Creating and editing columns (metric units)

Create gable posts or parapet posts

Steel parapet posts are single span members with fixed end connections. Their specific purpose is to act as a means to transfer load from wind wall panels into columns: the decomposed load from the panel is applied as a point load, and the moment at the node connects the parapet post to the column. To create parapet or gable posts posts in Tekla Structural Designer, see the following instructions.

NOTE Parapet posts are not designed in Tekla Structural Designer.

1. On the **Model** tab, click any element type (for example,  **Steel Column**).
2. In the **Properties** window, set the **Characteristic** to **Gable post** or **Parapet post**.

   The properties in the **Properties** window are updated to a type that is appropriate for the chosen post.

3. In the **Properties** window, adjust the properties according to your needs.

NOTE For gable posts:

- If an axial load release is required at the top, consider carefully which wind load deflection parameters are required.

4. Click the start point of the post.

NOTE If you are using a point along a member, do the following:

a. Click the member to see its points.

b. Either click the point that you want to use, or type the distance to the point from the start of the member.

5. Click the end point of the post.

   Tekla Structural Designer creates the post.

Align a column to a specific angle or an angled grid line

You can align a column to a specific angle or a grid line before creating the column. Once created an existing column can be realigned by editing the angle in the **Properties** window.

1. Do one of the following:
   - To create a new column, on the **Model** tab, select the desired column type.
   - To align an existing column to a specific angle, in the model, click the desired column.
2. In the **Properties** window, click the **Rotation** property.
3. In the list, select one of the following options:
   - The **0°**, **90°**, **180°**, and **-90°** options align the column to the global axes.
   - The **Angle** option aligns the column to the exact rotation angle you specify.
   - The **Define** option aligns the column to the angle of any grid line you select. (Applies to new columns only).

Related information

Related video

Align columns to grids

Modify the position of columns and column stacks

You may sometimes need to modify the position of columns and column stacks in your model. To do so, see the following instructions.

Move a column

See: Move objects (page 271).

Modify the position of a single column stack

You can change the position of a column stack in either a frame view or a structure view.

1. In , ensure that the **Grid & Construction Lines** options is selected.
2. Select the column stack that you want to move.
3. Click one end node for the column stack to be moved.

   Ensure the node is highlighted in the **Select Entity** tooltip when selecting.
4. Click a grid or construction point where you want to move the selected end node.

   The entire column is redrawn, with the selected node moved to the new position.
5. Select the next end node.
6. Click a grid or construction point where you want to move the selected end node.

   The column is redrawn once more, with the selected node moved to the new position.

Related information

See also

Video: Creating and editing columns (US customary units)

Video: Creating and editing columns (metric units)

Setting out steel and cold formed columns

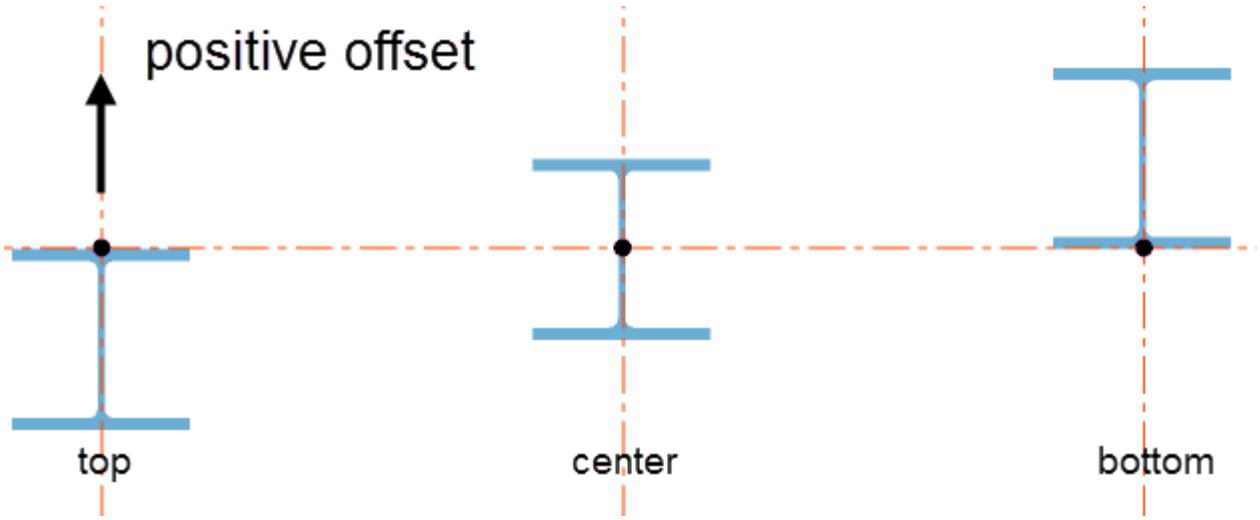
With Tekla Structural Designer, you can create various different types of steel and cold formed columns, including series of columns, inclined and cranked columns, gable or parapet posts, and plated, concrete filled or concrete encased columns. In addition, Tekla Structural Designer allows you to specify a column splice, align a column to a specific angle, or modify the position of existing columns.

Steel column alignment

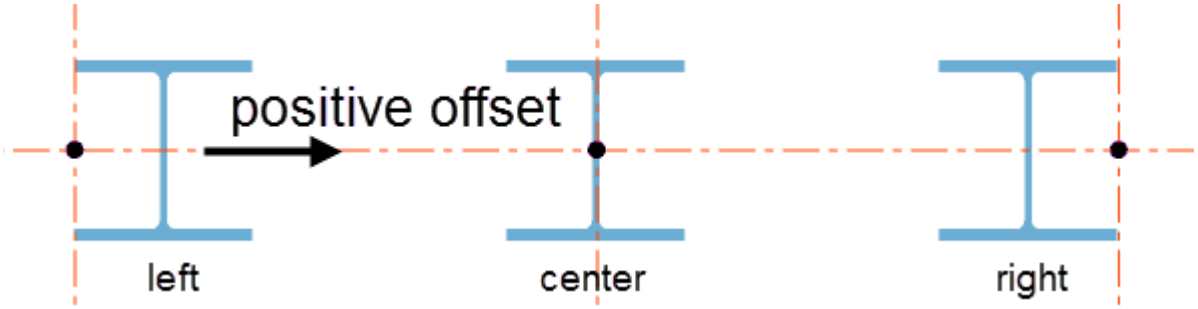
You can create a single steel column over several story heights. These kind of columns are referred to as column stacks. The columns can start and finish at any level. Different column sections can be defined in each stack, provided that a splice is defined at the change point.

Each steel column stack is placed on an insertion line between points, but its geometry is drawn to reflect the major and minor snap points and any offsets specified in the column properties.

Major offset:



Minor offset:



NOTE Steel column alignment snap points and offsets have no effect on the positioning of 1D solver elements in the solver models. The solver elements are always inserted along the steel column insertion lines.

This is different to the approach adopted for concrete column solver elements, in which the alignment snap points and offsets are structurally significant.

Create plated or compound section steel columns

In addition to **rolled**, **concrete filled**, and **concrete encased** steel columns, Tekla Structural Designer also allows you to create **plated** and **compound section** steel columns.

In Tekla Structural Designer plated section and compound sections are distinguished from each other as follows:

- Fabrication-wise **plated sections** are made up exclusively from separate structural steel plates assembled together (rather than rolled as a single cross-section).
- **Compound sections** differ in that they are made up from one or more rolled sections and/or plates.

While both plated section and compound section columns can be modelled and analysed, in Tekla Structural Designer they cannot be designed.

Plated section types

There are two plated section types:

-  Plated Beam
-  Plated Column

Compound section types

There are eighteen compound section types:

Section		Notes
	Rolled I-sections with Bottom Plate	Composed of: Standard parallel flange I-section & Bottom plate
	Rolled I-sections with Plates	Composed of: Standard parallel flange I-section; Top plate; Bottom plate Limitation: Top & Bottom plates must be equal and greater than standard section breadth.
	Rolled I-sections with Top Plate	Composed of: Standard parallel flange I-section & Top plate
	Rolled Steel Joists with Plates	Composed of: Standard tapered flange I-section; Top plate; Bottom plate Limitations: • Top & Bottom plates must be equal and greater than standard section breadth. • For **Indian code - India country** this compound

Section		Notes
		section is not included since it exists already as one of the rolled section types • For **selected country - South Africa**, the steel joist source section is not available. In order to define this compound section a different country would need to be chosen.
	Plated I-sections with Plates	Composed of: Top & Bottom Plates; Plated I-section Limitation: Top & Bottom plates must be greater that plated section breadth.
	Star Sections with Plates	Composed of: 2 Plated sections
	Double Rolled I-sections	Composed of: 2 Rolled standard parallel flange I-sections
	Double Rolled I-section with Plates	Composed of: 2 Rolled standard parallel flange I-sections; Top and Bottom plates Limitation: Top & Bottom plates must be greater that the total breadth of the 2 rolled I-sections (including the gap between them).
	Welded Sections with Plates	Composed of: Plates
	Rolled Angles in Star Arrangement	Composed of: 2 Rolled angle sections
	Rolled Channels (toe to toe)	Composed of: 2 Parallel flange channel sections Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.

Section		Notes
	Rolled Channels (toe to toe) with Horizontal Plates	Composed of: 2 Parallel flange channel sections; 2 horizontal plates Limitations: • Top & Bottom plates must be equal and less than the total breadth of the 2 rolled channels and the gap. • For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (toe to toe) with Vertical Plates	Composed of: 2 Parallel flange channel sections; 2 vertical plates Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (back to back)	Composed of: 2 Parallel flange channel sections Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (back to back) with Plates	Composed of : 2 Parallel flange channel sections; 1 vertical plate; 2 horizontal plates Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Box with Plates	Composed of: Plates
	Rolled Steel Joists in Star Arrangement	Composed of: 2 Tapered flange I-sections Limitation: For **selected country - South Africa**, the steel joist source section is not available. In order to define this compound section a different country would need to be chosen.

Section		Notes
	Rolled Steel Joists with Plates in Star Arrangement	Composed of : 1 Tapered flange I-section; 1 plated I-section Limitation: For **selected country - South Africa**, the steel joist source section is not available. In order to define this compound section a different country would need to be chosen.

Create plated or compound section columns

1. On the **Model** tab, click the arrow under  **Steel Column**.
2. In the list that appears, select  **Plated**.
3. Click the arrow on the right side of the **Section** property.
4. In the list that appears, select **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the section type and size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

   c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

   d. Select the section in the **Select Section** dialog box.

   ---
6. In the **Properties** window, adjust the remaining properties according to your needs.
7. Click the start point of the column.
8. Click the end point of the column.

   Tekla Structural Designer creates the column.
9. Do one of the following:
   - To create more similar column, click the start point of the new column.
   - To stop creating columns, press **Esc**.

Apply a plated or compound section to an existing steel column

1. In the **Properties** window, go to **Fabrication**.
2. Set **Fabrication** to **Plated**.
3. Go to **Section** and do one of the following:
   - Select an existing compound section.
   - Click **<New\Edit...>** to select a new section.

See also

Add user-defined sections (page 1266)

Specify a column splice

You can add splices at the base of each column stack (except for stack 1) as required. The splice offset is used to locate each splice at a practical distance above the floor level.

1. Select the desired columns.

   The properties of the columns are displayed in the **Properties** window.
2. Expand the properties of the column stack within which you want to create the splice.
3. Under **Release**, select the **Splice** option.
4. If necessary, modify the splice offset.

NOTE The **Section** property is now editable, so that you can specify a different section size above the splice position, if necessary.

See also

Specify the column type and section size (page 149)

Add a base plate to a steel column

While base plates are automatically added to **steel** columns when they are created, you might also be required to manually add a base plate to an existing column in some situations (for example if the old base plate has been deleted, or a supporting member has been removed, or the column material has been changed).

To add a base plate:

1. On the ribbon, go to the **Design** tab.
2. Click  **Base Plate**.
3. In the **Properties** window:

a. Leave **Autosize** selected, or clear it to manually define the plate size.

b. Edit the bolt properties (Eurocode) or Rod properties (AISC) as required.

c. Edit the anchor plate properties (Eurocode) or Welded plate washer properties (AISC) as required.

d. Edit the weld properties as required.

e. Edit the concrete base properties as required.

4. Click on a steel column, or drag a box around multiple steel columns to apply. The base plate is applied to those steel columns in the selection which have a support at their base and didn't previously have a base plate attached.

See also

Create a single column or series of columns (page 152)

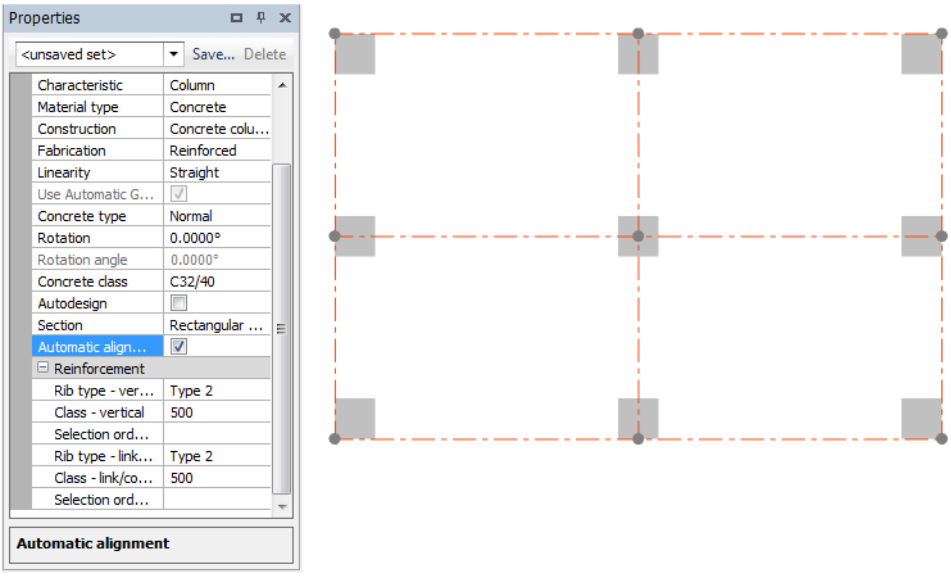
Specify concrete column alignment relative to the grid

The initial placement of each column relative to the grid depends on whether the **Automatic alignment** option is selected in the concrete column properties.

NOTE Columns that have already been placed do not have an **Automatic alignment** option, instead they can be re-aligned by adjusting their major/minor snap levels, and/or offsets (page 165).

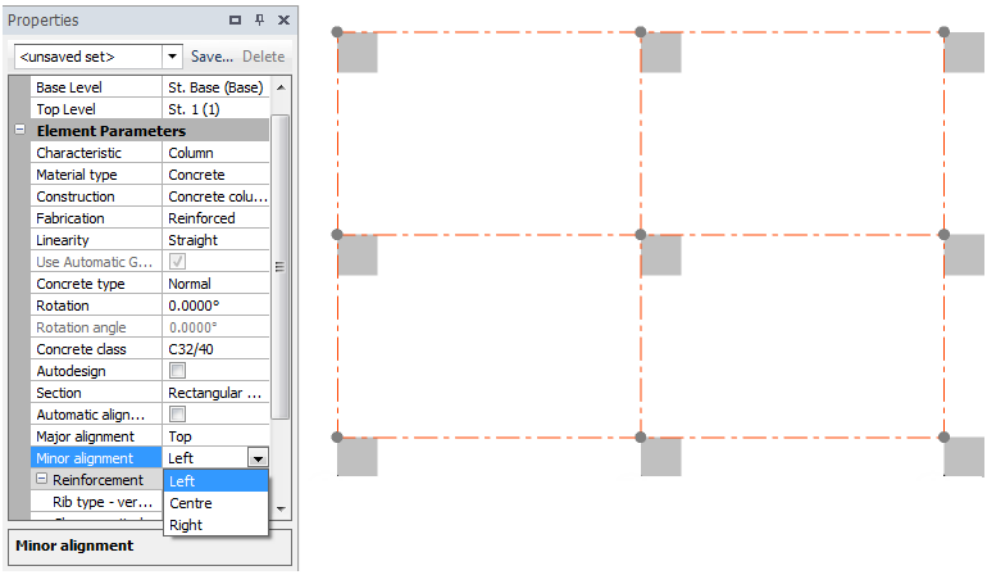
If **Automatic alignment** is selected:

Tekla Structural Designer aligns the columns on the perimeter of the grid with their faces flush to the perimeter, and the internal columns centrally on the grid.



If **Automatic alignment** is not selected:

Tekla Structural Designer aligns the columns according to the **Major alignment** and **Minor alignment** options in the concrete column properties.



- To switch the alignment option, in the **Properties** window, select or clear **Automatic alignment**.

See also

Align a column to a specific angle or an angled grid line (page 156)

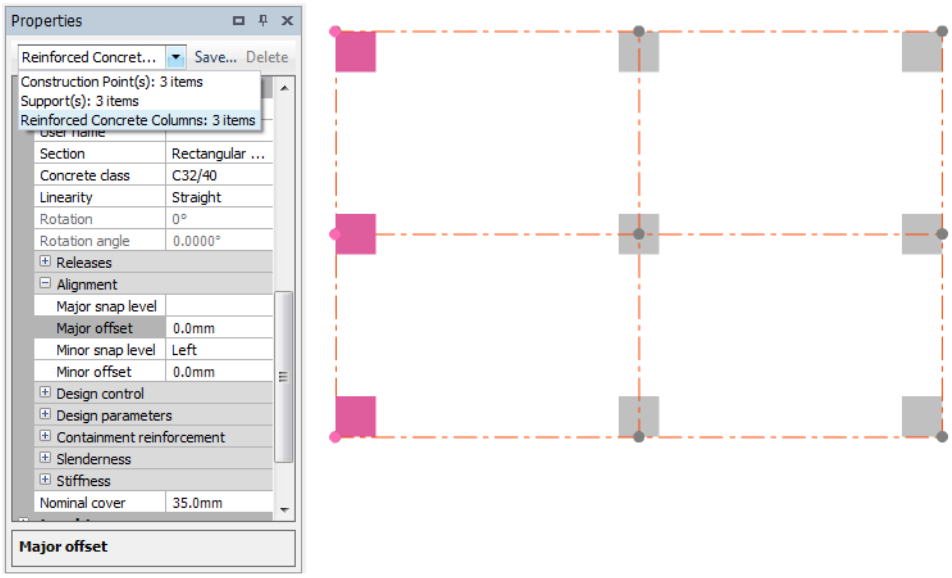
Modify concrete column alignment or specify offsets (page 165)

Modify concrete column alignment or specify offsets

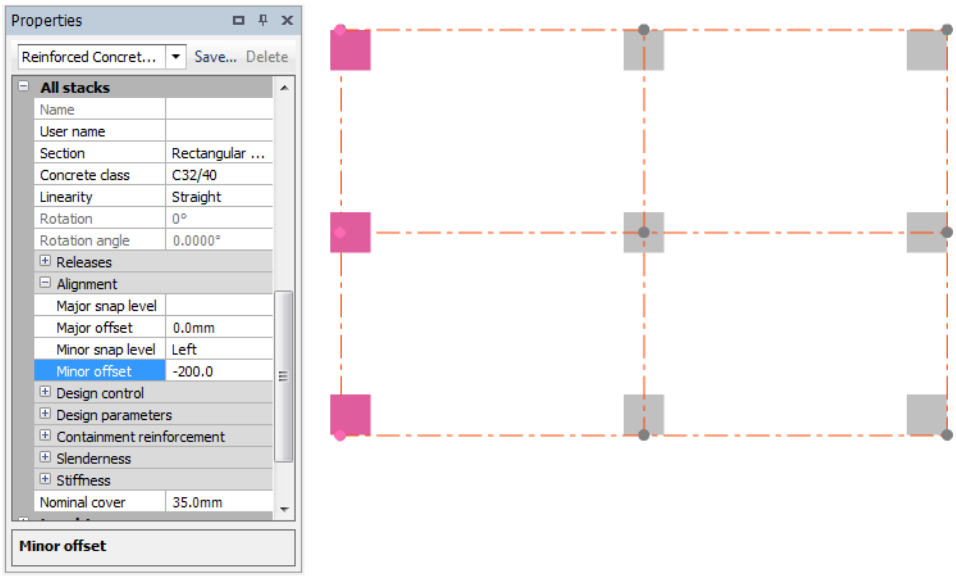
Once you have created columns, you can adjust their alignments and specify their offsets, if necessary. You can adjust the alignment of a single column in the **Properties** window or the **Properties** dialog box, whereas multiple columns can only be realigned in the **Properties** window.

The following example illustrates editing the alignment in the **Properties** window.

1. Select the columns that you want to offset.
2. Ensure that the column properties are viewed in the **Properties** window.



3. Modify the **Major offset** and **Minor offset** values according to your needs.



Tekla Structural Designer moves the columns relative to the major and minor snap levels by the value that you specified.

See also

Specify concrete column alignment relative to the grid (page 164)

Create beams

This section focuses on the operations required to create beams (in any material).

- Specify the beam type and section size (page 168)
- Create single-span beams (page 170)
- Create continuous beams (page 171)
- Create curved beams (page 172)
- Modify the position of beams (page 173)
- Add a span into an existing beam (page 174)

The following topics are relevant to steel beams only:

- Setting out steel and cold formed beams (page 176)
- Create plated or compound section steel beams (page 177)
- Create Westok cellular or Westok plated beams (page 182)
- Create FABSEC® beams (page 182)
- DELTABEAM® (page 183)
- Create web openings (page 186)
- Add haunches to steel beams (page 190)

- Manage design section orders (page 1272)

Related information

See also

Beam properties (page 1338)

Member global offsets (page 194)

Video: Creating and editing beams (US customary units)

Video: Creating and editing beams (metric units)

Specify the beam type and section size

Before you can place a beam you must first specify the beam type and an initial section size.

Specify the type of beam

- On the **Model** tab, do one of the following:

To	Do this
Specify a steel beam	1. Click the arrow under  **Steel Beam**. 2. In the list that appears, select the desired beam type.
Specify a plated, Westok, FABSEC®, or DELTABEAM®	1. Click the arrow under  **Steel Beam**. 2. In the list, select the required beam type.
Specify a cast-in-place, or precast concrete beam	1. Click the arrow under  **Concrete Beam**. 2. In the list that appears, select **Beam** for cast-in-place, or **Precast**.
Specify a timber beam	Click  **Timber Beam**.
Specify a cold formed beam	In the **Cold Formed** group, click  **Beam**.

Specify the size of steel, cold formed and timber beams

1. Specify the type of beam, (see the instructions above for how to do this).
2. In the **Properties** window, click **Section**.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, select **<New\Edit...>**

   The (page 109) opens.
5. Select the desired section size, and click **Select**.

TIP To define a custom section, click **Add...**

6. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of plated or compound steel beams

1. Specify the type of beam, (see the instructions above for how to do this).
2. In the **Properties** window, click **Section**.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, select **<New\Edit...>**

   The (page 109) opens.
5. In the (page 109), choose the required compound section type from the left hand pane.
6. Select the desired section size, and click **Select**.

TIP If the desired section is not listed, you can add it as follows:

a. Click **Add...**

b. Type the sections, plate dimensions, and gaps according to your needs.

c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

d. Select the section in the **Select Section** dialog box.

7. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of cast-in-place, or precast concrete beams

1. Specify the type of beam, (see the instructions above for how to do this).
2. In the **Properties** window, click **Section**.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, select **<New\Edit...>**

   The **Section** dialog box opens.
5. Select the beam shape.
6. Define the dimensions of the beam.
7. Click **OK**

8. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Related information

See also

Video: Creating and editing beams (US customary units)

Video: Creating and editing beams (metric units)

Create single-span beams

Tekla Structural Designer allows you to create concrete, steel, and timber single-span beams. For detailed instructions, see the following paragraphs.

Create a single-span beam

1. Specify the beam type and section size (page 168)
2. In the **Properties** window, adjust the beam properties according to your needs.
3. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

4. Click the end point of the beam.

   NOTE If Tekla Structural Designer prompts you to pick another point, and you do not want to create a continuous beam, do one of the following:

   - Click again the end point again.
   - Press **Enter**.

Create a series of single-span beams

RESTRICTION In order to a series of single-span beams using this method, the floor or construction level must already contain the columns between which the beams will run. You must also use a 2D view of the floor or construction level.

1. Select the beam type and size (page 168)
2. In the **Properties** window, adjust the beam properties according to your needs.

3. Move the mouse pointer to one corner of an imaginary box that will encompass the columns between which you want to create beams.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the diametrically opposite corner of the box.

   Ensure that the box encompasses all of the columns between which you want to create beams.
6. Release the mouse button.

   Tekla Structural Designer creates the beams between each adjacent pair of columns within the area you selected.

Related information

See also

Create continuous beams (page 171)

Create curved beams (page 172)

Video: Creating and editing beams (US customary units)

Video: Creating and editing beams (metric units)

Create continuous beams and cranked beams

You can create continuous beams and cranked beams in your model by selecting the **Continuous** option before creating the beam. For more information, see the following instructions.

RESTRICTION You cannot create continuous beams that are curved either horizontally or vertically.

---

1. Specify the beam type and section size (page 168)

   You can create steel, concrete, timber, or cold rolled beams.
2. In the **Properties** window, ensure that the **Continuous** option is selected.
3. In the **Properties** window, adjust the remaining properties according to your needs.
4. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

   ---
5. Click the next node of the beam.
6. Repeat step 5 until you have clicked the second to last node of the beam.

7. Click the end point of the beam.

   Tekla Structural Designer prompts you to pick another point.

8. To create the beam, do one of the following:
   - Click the end point again.
   - Press **Enter**.

   Tekla Structural Designer creates the continuous beam.

Related information

See also

Create single span beams (page 170)

Create curved beams (page 172)

Video: Creating and editing beams (US customary units)

Video: Creating and editing beams (metric units)

Create curved beams

Tekla Structural Designer allows you to create curved steel, concrete, and cold formed beams in your models, if necessary. To model curved beams, see the following instructions.

1. Select the beam type and size. (page 168)
2. In the **Properties** window, ensure that **Linearity** is set to one of the following:
   - **Curved major**, if you want the beam to curve vertically.
   - **Curved minor**, if you want the beam to curve horizontally.

   TIP You can control the direction in which horizontally curved beams curve.

   When you place the beam, you select its start point and its end point. The curve of the beam always lies on the left side of the line from the start point to the end point of the beam.

3. Still in the **Properties** window, specify an appropriate chord height value to define the curve.

   TIP You can reverse the curve direction of a vertically curved beam by using a negative chord height value.

   NOTE Horizontally curved beams always use the chord height defined in the property set. Therefore, they do not curve automatically to fit on any curved grid line that you may have defined.

4. In the **Properties** window, adjust the remaining properties according to your needs.
5. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

6. Click the end point of the beam.

   Tekla Structural Designer creates the beam.

   NOTE If you are using a 2D view when defining beams that curve vertically, you cannot see the beam on the graphical display.

   Change to a 3D view to see the beams.

See also

Create single span beams (page 170)

Create continuous beams and cranked beams (page 171)

Modify the position of beams

If necessary, you can modify the position of an individual beam in your model by moving one of its end nodes. You can modify the position of a beam in both 2D and 3D views.

To move an entire beam, see: Move objects (page 271).

1. Select the desired beam.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

Related information

See also

Video: Creating and editing beams (US customary units)

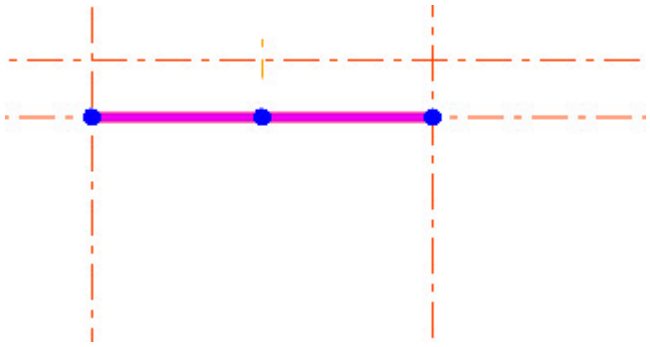
Video: Creating and editing beams (metric units)

Add a span into an existing beam

You can quickly add a span into an existing beam in both 2D and 3D views by introducing a new node.

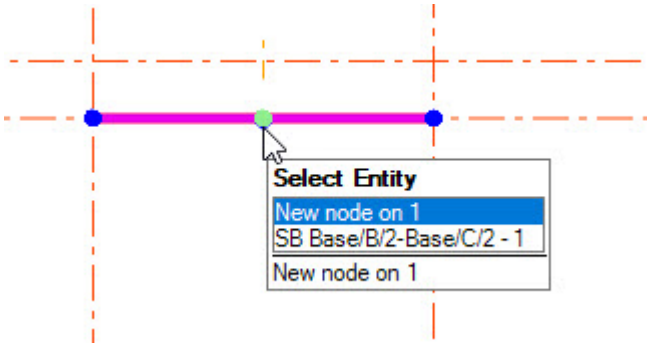
1. Select the beam in a 2D or 3D view.

   The nodes at each end and at the mid-point become visible.



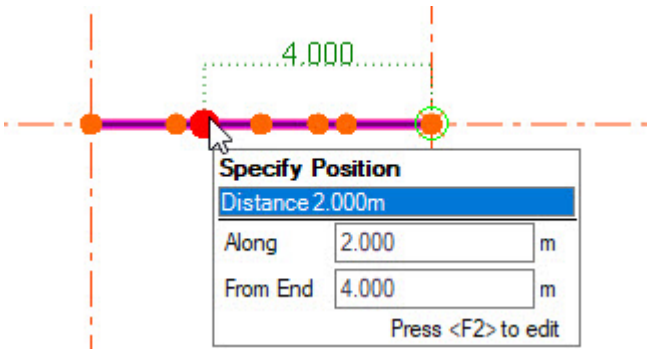
2. Select the node at the mid-point.

   In the **Select Entity** tooltip this is described as a 'New node on ...'

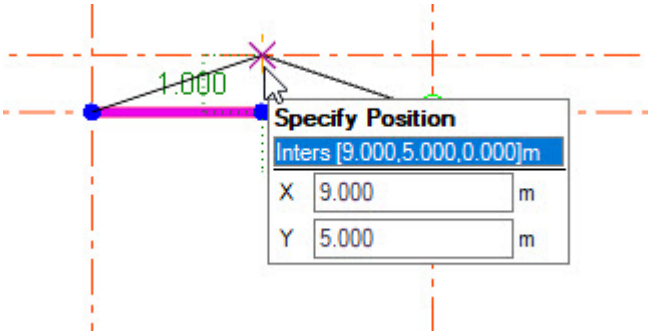


3. Click a point where you want to the new node to be created.

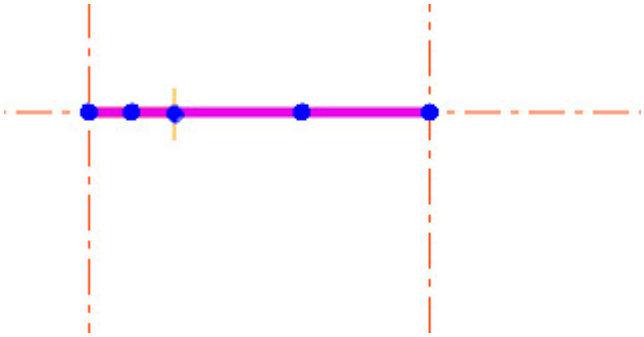
This can either be a valid point within the span



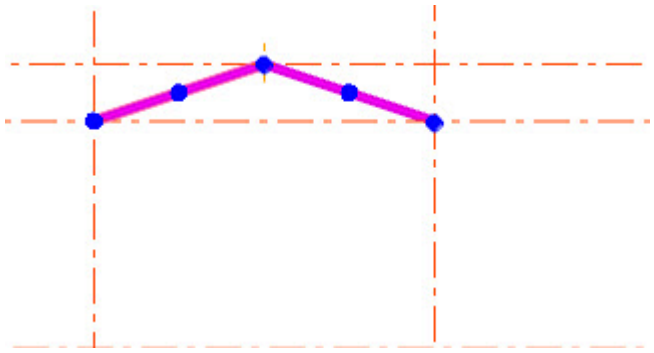
or, a a grid or construction point offset from the beam centerline



The new node is introduced into the beam and the original span is split according to the position specified, into two inline spans,



or two angled spans.

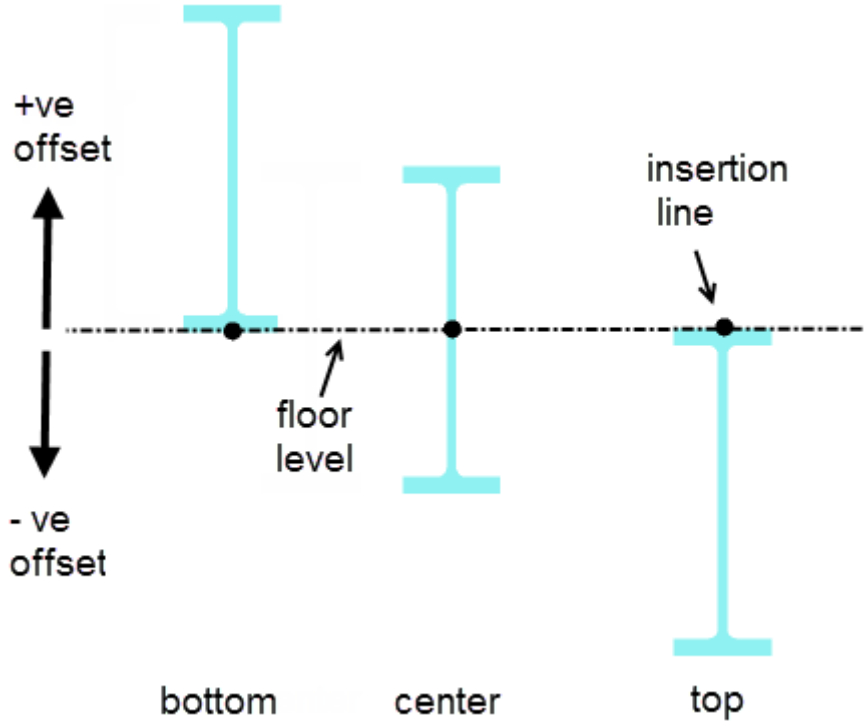


Setting out steel and cold formed beams
You can define steel beams as single span, or continuous over multiple spans. Even if you create a continuous beam, you can still define different beam sections in each span, if necessary.

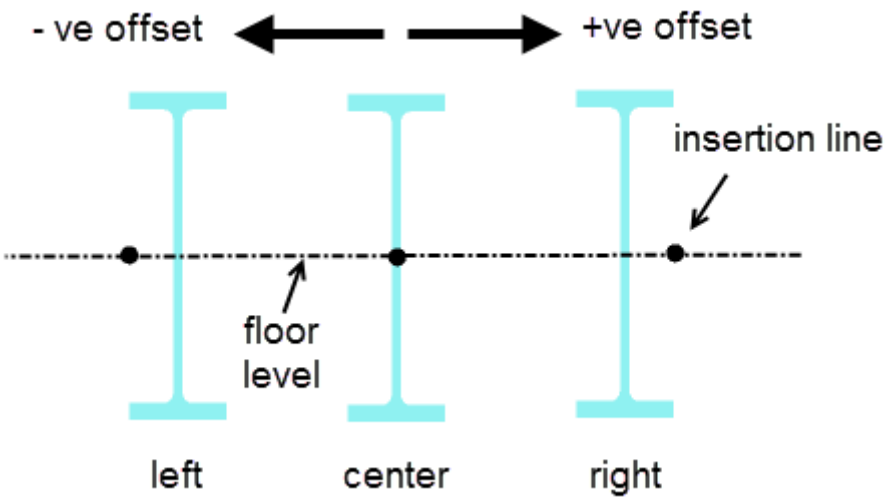
Each steel beam span is placed on an insertion line between points, but its geometry is drawn to reflect:

- the major and minor snap points and any offsets specified in the beam properties

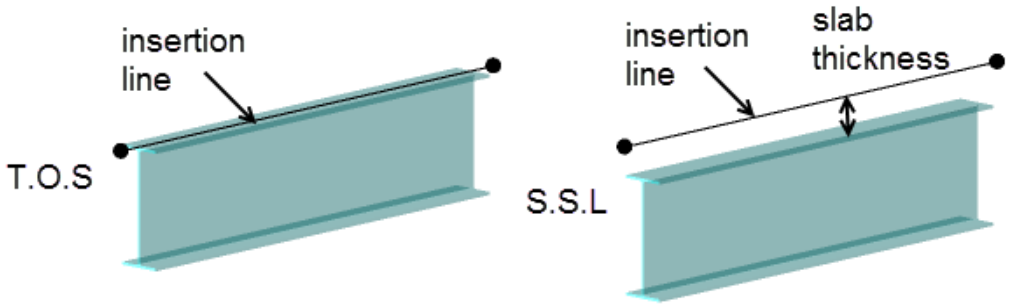
  Major snap and offset:



  Minor snap and offset:



- the level type specified in the **Construction Levels** dialog box dialog:



  - When the type is set to **T.O.S.** (top of steel), each beam is displayed according to the alignment snap points and offsets specified.
  - When the level type is set to **S.S.L.** (structural slab level), each beam is also lowered by the slab thickness specified in the **Construction Levels** dialog box.

NOTE Steel beam alignment snap points and offsets and the construction level type have no effect on the positioning of 1D solver elements in solver models. This is a different approach to that adopted for concrete beam solver elements.

Create plated or compound section steel beams

In addition to **rolled** steel beams, Tekla Structural Designer also allows you to create **plated section** and **compound section** beams.

Plated section and compound section beams are distinguished from each other as follows:

- Fabrication-wise **plated sections** are made up exclusively from separate structural steel plates assembled together (rather than rolled as a single

cross-section). Plated section beams **are designable** in Tekla Structural Designer.

- **Compound sections** differ in that they are made up from one or more rolled sections and/or plates. Compound section beams are provided for modelling and analysis purposes, but **they are not designable** in Tekla Structural Designer.

Plated section types

There are two plated section types:

-  Plated Beam
-  Plated Column

Compound section types

There are eighteen compound section types:

Section		Notes
	Rolled I-sections with Bottom Plate	Composed of: Standard parallel flange I-section & Bottom plate
	Rolled I-sections with Plates	Composed of: Standard parallel flange I-section; Top plate; Bottom plate Limitation: Top & Bottom plates must be equal and greater than standard section breadth.
	Rolled I-sections with Top Plate	Composed of: Standard parallel flange I-section & Top plate
	Rolled Steel Joists with Plates	Composed of: Standard tapered flange I-section; Top plate; Bottom plate Limitations: • Top & Bottom plates must be equal and greater than standard section breadth. • For **Indian code - India country** this compound section is not included since it exists already as one of the rolled section types

Section		Notes
		• For **selected country - South Africa**, the steel joist source section is not available. In order to define this compound section a different country would need to be chosen.
	Plated I-sections with Plates	Composed of: Top & Bottom Plates; Plated I-section Limitation: Top & Bottom plates must be greater that plated section breadth.
	Star Sections with Plates	Composed of: 2 Plated sections
	Double Rolled I-sections	Composed of: 2 Rolled standard parallel flange I-sections
	Double Rolled I-section with Plates	Composed of: 2 Rolled standard parallel flange I-sections; Top and Bottom plates Limitation: Top & Bottom plates must be greater that the total breadth of the 2 rolled I-sections (including the gap between them).
	Welded Sections with Plates	Composed of: Plates
	Rolled Angles in Star Arrangement	Composed of: 2 Rolled angle sections
	Rolled Channels (toe to toe)	Composed of: 2 Parallel flange channel sections Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (toe to toe) with Horizontal Plates	Composed of: 2 Parallel flange channel sections; 2 horizontal plates Limitations:

Section		Notes
		• Top & Bottom plates must be equal and less than the total breadth of the 2 rolled channels and the gap. • For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (toe to toe) with Vertical Plates	Composed of: 2 Parallel flange channel sections; 2 vertical plates Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (back to back)	Composed of: 2 Parallel flange channel sections Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Rolled Channels (back to back) with Plates	Composed of : 2 Parallel flange channel sections; 1 vertical plate; 2 horizontal plates Limitation: For **US code - USA country** or **Canada** parallel flange channels are not available.
	Box with Plates	Composed of: Plates
	Rolled Steel Joists in Star Arrangement	Composed of: 2 Tapered flange I-sections Limitation: For **selected country - South Africa**, the steel joist source section is not available. In order to define this compound section a different country would need to be chosen.
	Rolled Steel Joists with Plates in Star Arrangement	Composed of : 1 Tapered flange I-section; 1 plated I-section Limitation: For **selected country - South Africa**, the steel joist source section is not

Section		Notes
		available. In order to define this compound section a different country would need to be chosen.

Create plated or compound section beams

1. On the **Model** tab, click the arrow under **Steel Beam**.
2. In the list that appears, select **Plated**.
3. Click the arrow on the right side of the **Section** property.
4. In the list that appears, select **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the section type and size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

   c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

   d. Select the section in the **Select Section** dialog box.
6. In the **Properties** window, adjust the remaining properties according to your needs.
7. Click the start point of the beam.
8. Click the end point of the beam.

   Tekla Structural Designer creates the beam.
9. Do one of the following:
   - To create more similar beams, click the start point of the new beam.
   - To stop creating beams, press **Esc**.

Apply a plated or compound section to an existing steel beam

1. In the **Properties** window, go to **Fabrication**.
2. Set **Fabrication** to **Plated**.

3. Go to **Section** and do one of the following:
   - Select an existing compound section.
   - Click **<New\Edit...>** to select a new section.

See also

Add user-defined sections (page 1266)

Create Westok cellular or Westok plated beams

.

You can create Westok Cellular and Westok Plated beams which can be designed directly in Tekla Structural Designer. Once created, Westok Cellular beams can also be exported to and then re-imported from Westok's proprietary Cellbeam software (page 1191).

To create a Westok Cellular or Westok Plated beam:

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select the Westok Cellular or Westok Plated beam type as required.

   The beam adopts the properties that are currently displayed in the **Properties** window.
3. In the **Properties** window adjust the properties according to your needs.
4. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:
   a. Click the member to view its points.
   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
5. Click the end point of the beam.

   TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:
   - Click the end point again.
   - Press **Esc**.

   Tekla Structural Designer creates the beam.

See also

Export to and import from Westok Cellbeam (page 1191)

Create FABSEC® beams

Tekla Structural Designer also allows you to create FABSEC® beams which after analysis can be exported to Fabsec's proprietary FBEAM software for design (page 1192). The optimized beams can then be re-imported to Tekla Structural Designer.

To create a FABSEC® beam:

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select the FABSEC® beam type.

   The beam adopts the properties that are currently displayed in the **Properties** window.
3. In the **Properties** window adjust the properties according to your needs.
4. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the beam.

   TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:

   - Click the end point again.
   - Press **Esc**.

   Tekla Structural Designer creates the beam.

See also

Export to and import from FBEAM (page 1192)

DELTABEAM®

DELTABEAM® is a proprietary slim floor system. While DELTABEAM® sections can be analysed in Tekla Structural Designer they are not designed.

NOTE DELTABEAMS® are manufactured by Peikko in Finland: www.peikko.com.

The Peikko Designer download, (English language version) is available from:

https://www.peikko.com/design-tools/

Workflow in Tekla Structural Designer

- Initial DELTABEAM® section sizes are selected from the database and applied in the model. Peikko Designer can be used if required to better determine suitable initial sizes.
- The model is analysed, and a member forces report is created. This report is then sent to Peiko.
- Peiko provide the detailed design and final analysis properties.
- The Tekla Structural Designer model is updated as required.
- If the properties have been revised, the self weight could be affected and the model should be reanalysed.
- A further iteration may or may not be required.

Create DELTABEAMS®

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select DELTABEAM®

   The beam will adopt the properties displayed in the **Properties** window.
3. In the **Properties** window select the section required.

   NOTE D Series internal beams and DR Series edge beams can be selected provided the database country is set to Finland, Norway, Sweden, or Europe. You can add user defined sections to the database for the same countries if required.
4. Select the construction as either composite, or non-composite, as required.
5. Adjust the remaining properties according to your needs.
6. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
7. Click the end point of the beam.

   TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:

   - Click the end point again.
   - Press **Esc**.

Tekla Structural Designer creates the beam.

NOTE If you are placing a DR series edge beam, the flange will be drawn to the *left* of the line from start point to end point (easily verified in a 3D scene view). Once the beam has been placed, if you want to switch it to the other side simply select **Reverse** on the **Edit** tab and then click on the beam.

See also:

- Create single-span beams (page 170)
- Create continuous beams and cranked beams (page 171)

Create a member forces report

If you haven't already got a suitable report style configured you will need to create one as follows:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. To add a report, click **Add** and type a name for the report in the **Active Style** field.
3. To add the member forces chapter to the report, expand the Analysis heading in 'Chapters and Options' and drag the Member Forces chapter to the Report Structure pane on the right.
4. Review the report structure and modify it according to your needs.
5. To limit the output to selected levels, frames, planes, or sub structures, apply a model filter (page 1069).

   TIP You can further limit the output of **Loadcases** and **Combinations** sub chapters by applying a loading filter (page 1069).
6. Click **OK**.
7. Click **Show Report** to review the report and check that it is displaying the required level of data.

Once the report is configured to your needs, it can then be exported to Excel, PDF, Word, or Tedds.

See also:

- Configure a model report (page 1066)

Add DELTABEAM® properties to the database

Having submitted your member forces report to Peiko, you might be provided with a new set of analysis properties that are not in the Tekla Structural Designer database. In this situation you would need to add them as follows:

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**,
3. Select the unit system.
4. In the regional code list, select the region.
5. In the material list, select either steel, timber, coid rolled, or cold formed as required.
6. Click **Manage Sections**.

   The dialog box (page 1581) opens.
7. In the country list, select the database country.
8. In the **Page** pane on the left, select the appropriate DELTABEAM Series.
9. Click **Add...**
10. Enter values for each of the requested variables as supplied by Peiko.
11. Click **OK**.

The new section size is now displayed in the **Item** pane.

Related video

Easy modeling of Peikko Deltabeams

Create web openings in steel members

You can add square, rectangular, and circular web openings to beams and columns either by specified them manually, or by using a **Quick layout** option. FABSEC® beams can also have elongated openings specified.

RESTRICTION

1. Although web openings can be added to steel beams and columns, they are only considered in the design of:
   - Non-composite beams designed to US codes, BS codes, or Eurocodes
   - Composite beams designed to US codes, BS codes, or Eurocodes
2. Web openings are only valid in check design. If the auto design flag is switched on the openings will be removed.

Web opening creation methods

NOTE You can only specify web openings from the **Web openings** page of the , you cannot specify them directly from the .

The **Quick layout** method adds web openings to meet the geometric and proximity recommendations published by the SCI. With this method, Tekla Structural Designer creates web openings at the maximum depth and spaced at the minimum centers recommended for the section size.

With the **Quick layout** option cleared you can also define web openings manually using one of the following two methods:

- Clicking the **Add** button adds a new line to the web openings grid and allows you to define the geometric properties of the opening.
- Clicking the **Add...** button opens the **Web opening** dialog box where you can get more help and guidance when defining the opening.

Both manual methods use  and  signs to indicate faulty data when defining the opening parameters.

For more assistance, hover the mouse pointer over a  or  sign.

The Web opening dialog box

In the **Web opening** dialog box, you can use the following buttons to automatically define the position of the web opening:

- The **Center** button positions the opening on the center of the member.
- The **Auto** button positions the opening to meet the spacing recommendations by the SCI.

As web openings are defined, they are immediately visible in the diagram in the **Web opening** dialog box.

Tekla Structural Designer also performs design checks when you are defining web openings in the **Web opening** dialog box. The checked areas are end posts, web posts, web opening dimensions and tee dimensions.

The  and  signs in the **Web opening** dialog box help you to decide whether to make any adjustments to the opening parameters before Tekla Structural Designer checks the design.

NOTE The design checks carried out at the current stage are geometric checks only. Compliance with recommended limits is no guarantee that the opening will pass the subsequent engineering design checks.

Add web openings using the Quick layout option

Using the **Quick layout** method allows you to create maximum depth openings spaced at the minimum centers, appropriate to the section size.

1. Click the member to which you want to add web openings.
2. In the **Properties** window, ensure that the **Autodesign** option is cleared.

3. Right-click the member, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
4. Go to **Web openings**.
5. Select the **Quick layout** option.
6. In the **Label openings from** list, select where you want to start the setting out from.
7. In the **Type** column, select the web opening type.

   Data for the first web opening is automatically created as follows:

   - **l_O**: the length of the opening (applies to rectangular openings only)
   - **d_O**: the depth of the opening
   - **L_{CR}**: the distance from the setting out point to the center of the opening
   - **L_C**: the distance from the end 1 on the member to the center of the opening
   - **d_C**: the distance from the top of the member to the center of the opening
   - **L_{CR} relative to**: indicates the setting out point from which L_{CR} is measured.
   - **Nr. rel. to**: specifies an existing opening number that you want to use as the setting out point for the new opening.

     Nr. rel. to only applies if you selected one of the **Opening ->** options in the **L_{CR} relative to** column.
8. If necessary, in the **Stiffening** column, select the location of stiffening.

   a. Type the details of the stiffeners manually as follows:

      - **d_S**: depth of the stiffener
      - **t_S**: thickness of the stiffener
      - **L_S**: length of the stiffener
      - **e_S**: the distance from the edge of the opening to the center of the stiffener
9. To create further openings from the selected setting out points, click **Add** and select the opening type.

   TIP You can use the **Label openings from** list to switch to a new setting out point for the next opening at any point.
10. When you have created the desired openings, click **OK**.

Add web openings manually

1. Click the member to which you want to add web openings.
2. In the **Properties** window, ensure that the **Autodesign** option is cleared.
3. Right-click the member, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
4. Go to **Web openings**.
5. Ensure that the **Quick layout** option is not selected.
6. In the **Label openings from** list, select where you want to start the setting out from.
7. Click **Add...**
8. In the **Type** column, select the web opening type.
9. Specify the following details for the opening:
   - **l_O**: the length of the opening (applies to rectangular openings only)
   - **d_O**: the depth of the opening
   - **L_{CR}**: the distance from the setting out point to the center of the opening
   - **L_C**: the distance from the end 1 on the member to the center of the opening
   - **d_C**: the distance from the top of the member to the center of the opening

     TIP Click Center for dC to be automatically calculated to position the opening centrally in the section depth.

   - **L_{CR} relative to**: indicates the setting out point from which L_{CR} is measured.
   - **Nr. rel. to**: specifies an existing opening number that you want to use as the setting out point for the new opening.

     Nr. rel. to only applies if you selected one of the **Opening ->** options in the **L_{CR} relative to** column.
10. If necessary, in the **Stiffening** column, select the location of stiffening.
    a. Type the details of the stiffeners manually as follows:
       - **d_S**: depth of the stiffener
       - **t_S**: thickness of the stiffener
       - **L_S**: length of the stiffener

- **e_S**: the distance from the edge of the opening to the center of the stiffener

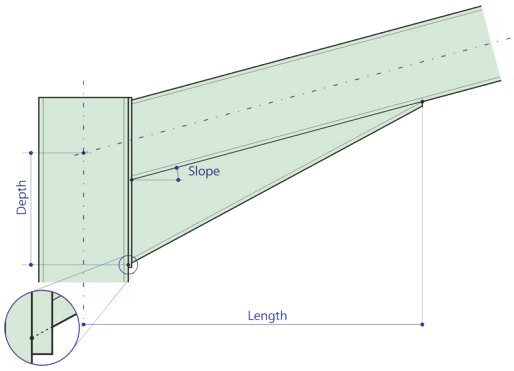
11. To create the opening, click **OK**.
12. To create further openings, do one of the following:
    - To create multiple copies of a selected opening, click **Copy...**
    - To create a single opening of a different size or spacing, click **Add...**

Add haunches to steel beams

1. Right-click the steel beam to which you want to add a haunch, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
2. Go to **Haunches**.
3. Select the appropriate **Create** option for the required haunch location.
4. In the **Haunch** dialog, define the haunch parameters as required.

   The haunch depth is measured from the beam centerline and length is measured horizontally from the sharp end of the haunch to the beam insertion point. In the case of an inclined beam the haunch length and depth would therefore be measured as shown below.



5. When you have defined the haunch parameters, click **OK**.

See also

Create single span beams (page 170)

Create braces

Tekla Structural Designer allows you to create single braces, or pairs of braces (in an X, K, V, or A configuration).

Braces are single span members with pinned end connections that are only able to resist axial compression and tension. Their purpose is to provide lateral stability to your structure.

TIP You can specify rigid frames to achieve the same effect as by using braces. If necessary, you can use both rigid frames and braces within a single structure.

Element loads cannot be applied directly to a brace itself and moments due to self weight loading are ignored.

Setting out braces

Each steel brace is placed on an insertion line between points, with its geometry being drawn to reflect the major and minor snap points and any offsets specified in the brace properties.

NOTE The alignment snap points and offsets have no effect on the positioning of the solver elements created to represent the braces in the solver models, as these will be created directly between the insertion points.

Specify the brace material, pattern and section size

Before creating a brace, you need to specify its material, pattern and section size. For more information, see the following instructions.

1. Do one of the following:

To	Do this
Specify a steel single brace or pattern	a. On the **Model** tab, click the arrow under  **Steel Brace**. b. In the list, select **Brace** for a single brace; or choose an **X**, **K**, **V**, or **A** pattern.

To	Do this
Specify a cold formed single brace or pattern	a. On the **Model** tab, click  **Cold Formed**. b. In the list, click the arrow next to  **Brace**. c. In the list, select **Brace** for a single brace; or choose an **X**, **K**, **V**, or **A** pattern.

To	Do this
Specify a timber single brace or pattern	a. On the **Model** tab, click the arrow on the right side of  **Timber Brace**.

	b. In the list, select **Brace** for a single brace; or choose an **X**, **K**, **V**, or **A** pattern.

2. In the **Properties** window, go to the **Section** property.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, click **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the desired section, and click **Select**.
6. In the **Properties** window, adjust the remaining properties according to your needs.

Create a single brace

NOTE Although it is possible to model A or V brace configurations using individual brace members instead of a brace pair, Tekla Structural Designer is then not able to calculate the notional loads or EHFs (Equivalent Horizontal Forces) correctly. In this case, elements of the vertical loads that are supported by the bracing system are lost, and are not included in the notional load or EHF calculations.

1. Specify the brace material and section size.
2. Click the start point of the brace.
3. Click the end point of the brace.

   Tekla Structural Designer creates the brace.

   TIP If you are using a point along an existing member as either the start or end point, do the following:

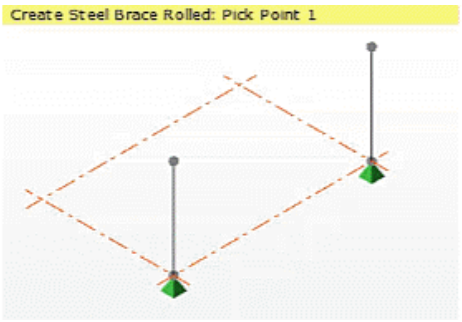
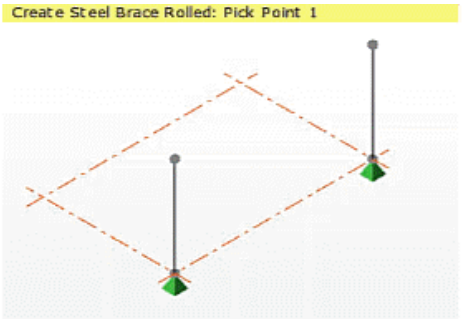
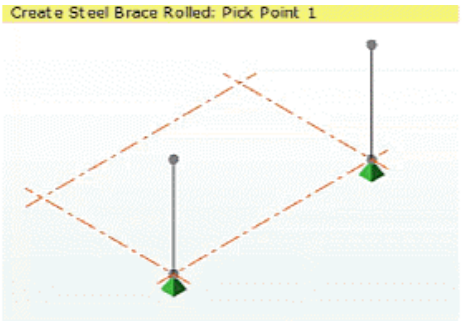
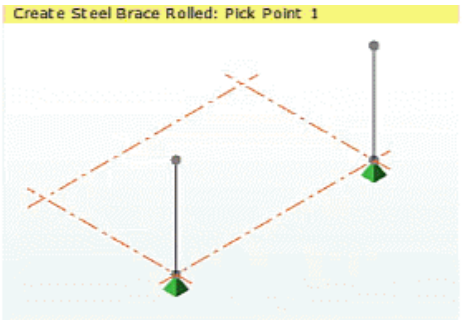
   a. Click the existing member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

Create an X, K, V or A brace

1. Specify the brace material, pattern and section size.
2. Click to identify a bottom corner of the bay that you want to brace.
3. Click to identify the opposite bottom corner of the bay.
4. Click to identify a top corner of the bay.
5. Click to identify the opposite top corner of the bay.

Tekla Structural Designer creates the brace pattern within the selected area.

X pattern	
K pattern	
V pattern	
A pattern	

Specify a brace as tension only or compression only

You can move the end nodes of individual steel, concrete, cold formed, or timber braces in both 2D and 3D views. For more information, see the following instructions.

Once a brace has been created, you can specify it as Tension only or Compression only in the **Properties** window.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed, they will be treated as linear elements.

---

1. Select the brace.
2. In the **Properties** window select either **Compression only**, or **Tension only** as required.

Modify the position of a brace

You can move the end nodes of individual steel, concrete, cold formed, or timber braces in both 2D and 3D views. For more information, see the following instructions.

To move an entire brace, see: Move objects (page 271).

1. Select the brace.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

   ---

3. Click a grid or construction point where you want to move the end node.

   Tekla Structural Designer redraws the brace, moving the end node to the selected point.

Related information

See also

Brace properties (page 1402)

Member global offsets (page 194)

Video: Creating and editing braces (US customary units)

Video: Creating and editing braces (metric units)

Member global offsets

In certain situations you may have a requirement to model different global (X,Y,Z) physical offsets at each end of a member.

In Tekla Structural Designer global offsets can be applied to steel, cold formed, cold rolled and timber member types, but with the following exceptions:

- columns (all materials),
- concrete beams
- analysis elements
- members with haunches
- curved members

NOTE Global offsets have no affect on the positioning of the 1D solver elements in the solver models; they have no impact on loading, analysis or design. Global offsets only affect the graphics and BIM integration.

Global offsets are applied to the member ends **before** any major/minor local offsets, the local offsets thus being applied to the whole length of the member in relation to the new line.

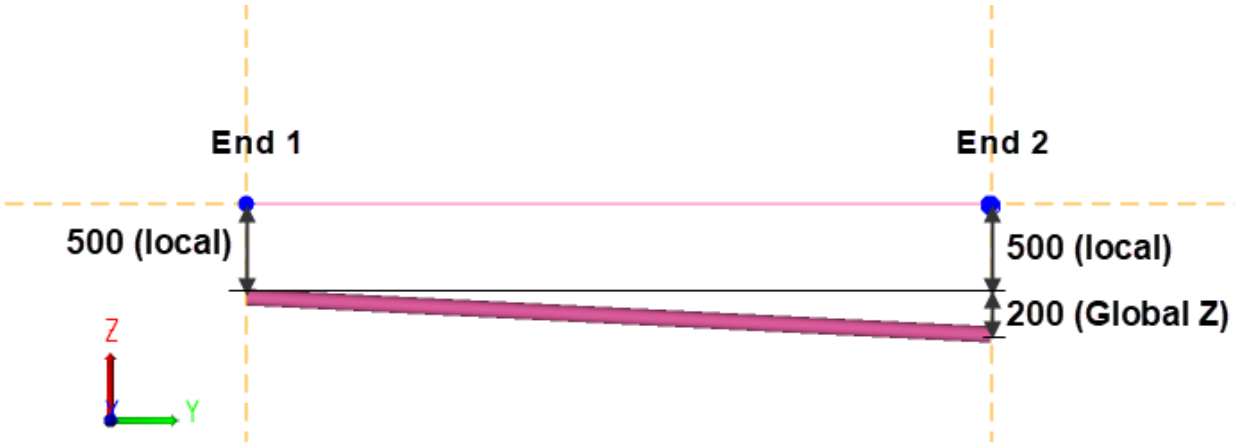
NOTE Global offsets are included in the BIM import/export to/from Tekla Structures

Global offset combined with major offset example

A horizontal brace member is specified with a Global End 2 offset of -200mm and a major (local) offset of -500mm, so that its alignment properties are as follows:

Alignment	
⊞ Global offset end 1	[0.0, 0.0, 0.0] mm
⊟ Global offset end 2	[0.0, 0.0, -200.0] mm
X	0.0mm
Y	0.0mm
Z	-200.0mm
Major snap level	Centre
Major offset	-500.0mm
Minor snap level	Centre
Minor offset	0.0mm

While the offsets do not affect the insertion line (denoting the 1D solver element in the analysis model), the brace itself is offset in the graphics as shown below:

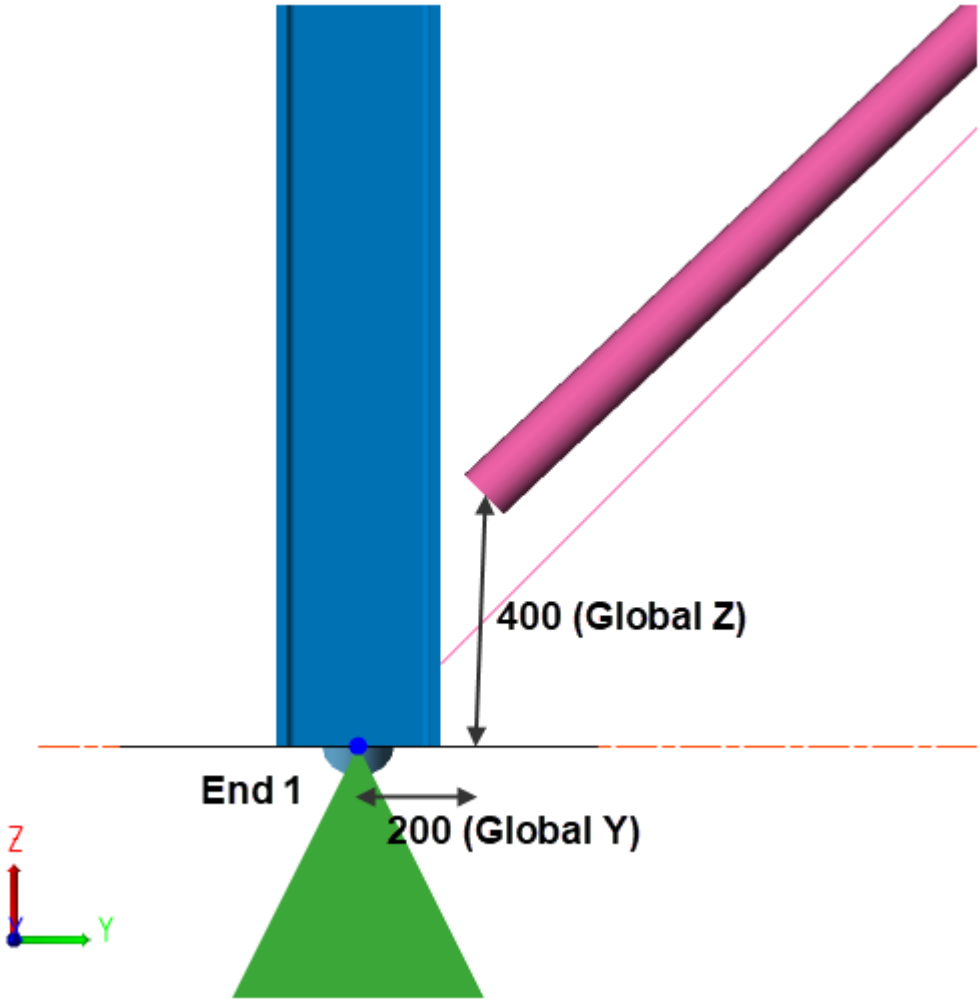


Global offset only example

A diagonal brace member is specified with a Global End 1 offsets of 200mm in Z and 400mm in Y, the Major and Minor (local) offsets are both 0mm, so that its alignment properties are as follows:

Alignment	
Global offset end 1	[0.0, 200.0, 400.0] mm
X	0.0mm
Y	200.0mm
Z	400.0mm
Global offset end 2	[0.0, 0.0, 0.0] mm
Major snap level	Centre
Major offset	0.0mm
Minor snap level	Centre
Minor offset	0.0mm

While the offsets do not affect the insertion line (denoting the 1D solver element in the analysis model), the brace itself is offset in the graphics as shown below:



Global offsets

Create walls, cores and bearing walls

These topics introduce you to the methods of creating walls, cores, and bearing walls.

We recommend you familiarize yourself with how to:

- Create concrete walls (page 198)
- Create concrete cores (page 205) (from existing concrete walls, columns and beams)
- Create bearing walls (page 208)
- Create shear only walls (page 211)
- Create general walls (page 213)

Create concrete walls

This section focuses on the operations required to create concrete walls.

- Overview of the concrete wall model (page 198)
- Create concrete walls (page 200)
- Specify extensions and releases (page 202)
- Create concrete cores (page 205)
- Create and modify wall supports (page 203)
- Create door or window openings (page 204)

Overview of the concrete wall model

Tekla Structural Designer allows you to create both meshed and mid-pier concrete walls.

Geometric rules for meshed concrete walls

- Meshed walls are defined as quadrilaterals in a single plane that can be vertical or sloping, (unlike mid-pier concrete walls which must be rectangular in a vertical plane).
- Openings (page 204) are permitted in meshed walls.
- The alignment and offsets in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- By default, meshed walls use the mesh parameters that are defined in **Structure Properties**. However, you can override these for an individual wall by checking **Override model's** in the Meshing section of the wall's properties. You are then able to apply a user-defined mesh to the wall.
- Concrete meshed walls can be included in cores, General meshed walls cannot.

Sub-division of meshed walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Geometric rules for mid-pier concrete walls

- Mid-pier walls must be rectangular in a vertical plane.
- Wall openings are ignored.
- The alignment and offsets specified in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- Mid-pier walls can be included in cores.

Sub-division of mid-pier walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Continuous concrete walls

Tekla Structural Designer allows you to create walls that are several stories high and that can start and finish at any level.

Although Tekla Structural Designer only creates one wall in this case, you can define different thicknesses for each wall panel. The panels are then set back on one or both faces, depending on the alignment that you have specified.

Creating a continuous wall this way, instead of defining a new wall on each floor, does not have any significance for analysis or design purposes. However, it is ultimately important for detailing purposes.

Analysis of concrete walls

The points that you use to place a concrete wall define the exact size and position of the wall's analysis model. The alignment and extension properties of the wall have no effect on the analysis model.

Tekla Structural Designer can adopt either an FE meshed or mid-pier wall analysis on a wall by wall basis.

For more information, see:

- FE meshed walls (page 635)
- How meshed walls are represented in solver models (page 554)
- How mid-pier walls are represented in solver models (page 559)

Support

Provided that the **AutomaticGenerateSupport** option is selected, if no slab or other member exists beneath the wall when it is first created, Tekla Structural Designer automatically places a support underneath the wall.

Material Type

For walls with the **Material type** set to concrete (as opposed to general), there are then two types of **Fabrication** available:

- Cast-in-place
- Precast

For each fabrication type there are two **Concrete types** available:

- Normal
- Lightweight

The **Grade** lists all the available grades in the Materials database under the current regional code for the selected Concrete type.

Releases

You can apply minor axis releases at the top and the bottom of each panel in order to model pinned connections to incoming slabs and members.

Door and window openings

You can only define door and window openings in meshed walls. Tekla Structural Designer does not allow you to create openings in mid-pier walls. For more information openings, and the alternative ways to model them, see: Concrete wall openings analysis model (page 641)

Purpose of concrete walls

Both meshed and mid-pier concrete walls introduce structural strength and stiffness to your model. However, they do not perform the same function as wall panels. This means that wind loads calculated by the Wind Wizard cannot be applied to your structure if the model does not contain wall panels.

In order to apply wind loads, you must create additional wall panels in the same physical locations as the concrete walls.

Create meshed or midpier concrete walls

You can create meshed or mid-pier concrete walls both in 2D views and frame or structure views. Mid-pier walls are always rectangular; meshed walls can either be rectangular or quadrilateral.

NOTE In order to define a wall you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

Create rectangular walls in a level view

1. On the **Model** tab, click the arrow below  **Concrete Wall**.
2. In the list that appears, select the desired wall type.
   - Mid-pier Wall
   - Meshed Wall
3. Go to the **Properties** window.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. In the model, click the start point of the wall.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

7. Click the end point of the wall.

   Tekla Structural Designer creates the wall.

Create rectangular walls in a frame or structure view

1. On the **Model** tab, click the arrow below  **Concrete Wall**.
2. In the list that appears, select the desired wall type.
   - Mid-pier Wall
   - Meshed Wall
3. Go to the **Properties** window.
4. Ensure the **Rectangular** property is selected.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click a point to define a corner of the wall.
7. Click a point to define the opposite corner. Tekla Structural Designer creates the wall between the selected two points.

Create quadrilateral walls in a frame or structure view

1. On the **Model** tab, click the arrow below  **Concrete Wall**.
2. In the list that appears, select the **Meshed Wall**.
3. Go to the **Properties** window.
4. Ensure the **Rectangular** property is unselected.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click the point where the base of the wall should start.
7. Click the point where the base of the wall should end.
8. Click the point where the top of the wall should start.
9. Click the point where the top of the wall should end. Tekla Structural Designer creates the wall between the selected four points.

See also

Concrete meshed and mid-pier wall properties (page 1434)

How meshed walls are represented in solver models (page 554)

How mid-pier walls are represented in solver models (page 559)

Create door or window openings (page 204)

Specify extensions and releases
Tekla Structural Designer allows you to trim new walls automatically, and trim or extend existing walls manually. In addition, you can specify minor axis releases to the top or bottom part of panels. For more information, see the following instructions.

See also

Create concrete walls (page 200)

Create door and window openings (page 204)

Specify extensions

- Do one of the following:

To	Do this
Automatically trim a new wall back to the face of existing columns and walls	1. In the **Properties** window, select the **AutomaticExtension** option. 2. Create the wall.
Manually trim or extend existing walls	1. Select the wall that you want to trim or extend. 2. Go to the **Properties** window. 3. In the **End 1 extension** or **End 2 extension** field, define the desired extension. **NOTE** A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from its insertion point.

Specify releases

1. Select the wall to be released.
2. In the **Properties** window, go to **Releases**.
3. In the **Minor Top** or **Minor Bottom** list, select the desired release:
   - **Fixed**
   - **Pinned**

- **Continuous (incoming members pinned)**: only available for FE meshed walls

> **TIP** To specify a pinned connection to a supported slab, use an FE meshed wall and select the **Continuous (incoming members pinned)** option.
>
> We recommend this option because the **Pinned** option also releases the wall panel above from the wall panel below. This may result in a mechanism during the analysis.

Create and modify wall supports

If needed, you can set Tekla Structural Designer to automatically generate support for walls. In addition, you can modify the degree of freedom of the wall support, and remove an unnecessary wall support. For more information, see the following paragraphs.

Automatic support generation

The **AutomaticGenerateSupport** option in wall properties controls whether a support is automatically created at the wall base level.

When the **AutomaticGenerateSupport** option is selected:

- If there are members or slabs underneath the wall capable of providing support, no support is generated.
- If there are no members or slabs underneath the wall capable of providing support, a support is generated.

When the **AutomaticGenerateSupport** option is cleared:

- If the **Generate support** option is selected, a support is generated.
- If the **Generate support** option is not selected, no support is generated.

When a support is required, Tekla Structural Designer forms it as follows:

- Under a meshed wall, Tekla Structural Designer creates a line support.
- Under a mid-pier wall, Tekla Structural Designer creates a point support.
- Under a bearing wall, Tekla Structural Designer creates a series of point supports.

When a support is required, its degrees of freedom are as specified in wall properties, under **Wall support**.

See also

Create concrete walls (page 200)

Modify wall support fixity

NOTE Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

1. In the **Properties** window, expand **Wall support**.
2. Specify the degrees of freedom according to your needs.

Remove a wall support

NOTE Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

1. In the **Properties** window, clear the **Generate support** option.

   NOTE The **Generate support** option is only available for editing if the **AutomaticGenerateSupport** option is inactive. When **AutomaticGenerateSupport** is selected, **Generate support** is automatically cleared if members are created underneath a wall to support it. Similarly, **Generate support** is automatically reselected, if the supporting members are deleted.

Create door or window openings

Tekla Structural Designer allows you to create openings for both doors and windows in existing walls. For detailed instructions, see the following paragraphs.

RESTRICTION Openings are only active in meshed walls.

1. Open a frame view of the frame that contains the desired meshed wall.
2. On the **Model** tab, click  **Wall Opening**.
3. Do one of the following:
   - To define the opening relative to the bottom left corner of the wall, click the outline of an existing wall panel.
   - To define the opening relative to the selected node, click a specific node in the model.
4. Click the first corner of the opening or press **F2** to define its exact position.
5. Drag the mouse pointer to the opposite corner of the opening.
6. Click the opposite corner of the opening or press **F2** to define its exact position.

   Tekla Structural Designer creates the opening.

See also

Create meshed or midpier concrete walls (page 200)

Meshed wall openings analysis model (page 641)

Manage, display and design result lines (page 698)

Create concrete cores

Tekla Structural Designer allows you to combine concrete wall panels, columns, and beams to form cores.

Overview of concrete cores

Concrete wall panels, columns, and beams can be combined to form cores for which overall analysis results are automatically calculated and available for review and output.

Such cores might then be used in a number of ways, for example:

- The engineer may wish to see/ output core forces to; better understand the distribution of lateral forces within the structure; for additional checks they wish to undertake; for overall core foundation forces or exclusive core reactions for cores on foundation mats.
- Cores provide a way of checking lateral forces per object type or SFRS type. This might be used when assessing buildings for some codes which require frames in dual systems to resist a minimum amount of seismic load.
- Adding all elements of the lateral system to a single core will give overall building vertical and lateral load, overturning moments or accumulated seismic torsion, in any direction.

Any number of walls (meshed and mid-pier), columns, and beams can be added to a core.

- Constituent objects of a core do not need to be physically attached.
- Once defined the core is listed in the structure tree and can be graphically selected as a single entity. When selected, its properties are displayed in the Property Window and Delete, Move, Copy and Mirror operations can be applied to it.

The directions of each core can be defined as any of; Dir 1/2 - Main Building Directions; Principal 1/2 - Major and minor local axis; Angle - (w.r.t.) Global Coordinate System.

The following core properties are automatically calculated/ displayed; Core section for each level; Core centroid and its coordinates above and below each level; a Core line with Local coordinate system (LCS) assigned and displayed at the centroid location; a Core support (for reporting purposes not analysis).

After analysis the following core results are available:

- **Core Line Forces** - 2D Integrated Results termed “Core Line” results for cores giving the following overall core forces; Axial Force, Major and Minor axis Moment and Shear, Torsion.
  - In the Results View via 2D Integrated Results > Core Lines, force diagrams of these results are displayed along the core line w.r.t. its LCS, with numerical values displayed at the top and bottom of each core stack and in the Tooltip when the result diagram is cursored over.
  - Results are available for all of; All Static Analyses including chase-down; RSA Seismic both for individual modes and Modal combination; Imposed/Live load reductions.
  - Tabular Results - Core results are also available in Tabular form via Analyze > Tabular Data and the associated Report option Analysis > “Core Line Forces”.

    NOTE Settings controls in the Member Forces and Wall Line Forces Report items allow you to optionally exclude columns/walls assigned to cores.

- **Core Reactions** - overall Reaction results are calculated for the core and can be viewed in the Results view via Reactions > Cores and output.

Create concrete core (assisted mode)

Cores are either created in **Assisted** or **Unassisted** mode. In assisted mode (which is the default), all potential members of a core are automatically highlighted when the cursor is moved over any of the constituent members, giving a preview of the core that can be created - a single click then adds all the highlighted walls to a new core.

1. On the **Model** tab, click  **Cores**.

1. **Assisted Mode**
2. In the **Properties** window, select **Assisted**.
3. Select an appropriate wall panel or column and all highlighted members are added to the core.

   The mode switches to **Modify** to enable you to edit the core if required.
4. If no modifications are required, press Esc to finish.

NOTE Assisted Mode detects overlapping concrete walls, columns and coupling beams* that may potentially form a core.

*Beams for which the Construction property is set = “Coupling Beam”. Note that no additional design checks are performed for this construction type so currently it serves only to mark beams;

for consideration by the assisted mode core function; for which additional design checks beyond those currently made in the program may be required.

Create concrete core (unassisted mode)

1. On the **Model** tab, click **Cores**.
2. In the **Properties** window, unselect **Assisted**.
3. Select either a wall panel or column as the first member of the core.
4. Select the next wall panel, column, or beam to be part of the core.

   NOTE You can drag the mouse pointer to select multiple members.

5. Continue selecting until all the members of the core have been chosen, and then press Esc to finish.

   Tekla Structural Designer combines the selected members into a single core.

Set the core axis direction

Analysis results are displayed with respect to the axis diection of each core. This can be aligned to the main building directions, the major and minor local axis, or to a specified angle.

1. Select the core.
2. In the **Properties** window, choose the rotation angle as required.

Modify concrete core

1. On the **Model** tab, click **Cores**.
2. In the **Properties** window,
   a. Set **Mode** to **Modify**
   b. Select the **Active Core** as required.
   c. Set **Area Selection Mode** to **Set On** to add members to the core, or **Set Off** to remove members from the core.
3. Select a member to add or remove from the core.
4. Continue selecting or press Esc to finish.

   Tekla Structural Designer combines the selected members into a single core.

Dissociate concrete core

Existing cores can be disconnected if required.

1. Hover the mouse pointer over the core.
2. If a different entity is highlighted, press the <down arrow> cursor key until the required core reference is shown in the Select Entity tooltip.
3. Right click and select Dissociate Core.

Related video

Core modeling

Create bearing walls

Bearing walls provide resistance to vertical compressive loads (but not lateral loads) and support certain other member types. Unreinforced masonry walls, for example, can be modeled as bearing walls.

You can model bearing walls over several story heights. In these cases, Tekla Structural Designer creates a single wall with a uniform thickness between the base and top level.

Tekla Structural Designer determines the location of the wall from the alignment specified in the bearing wall properties, and the selected insertion points.

NOTE Bearing walls do not perform the same function as wall panels. In other words, bearing walls do not allow you to apply loads calculated by the **Wind Wizard** to your structure.

Therefore, in order to apply wind loads, you must create additional wall panels in the same locations as the bearing walls.

Material type

Three **Material types** are available for bearing walls:

- Concrete
- Timber
- General

Concrete Bearing Walls

There are two **Concrete types** available:

- Normal
- Lightweight

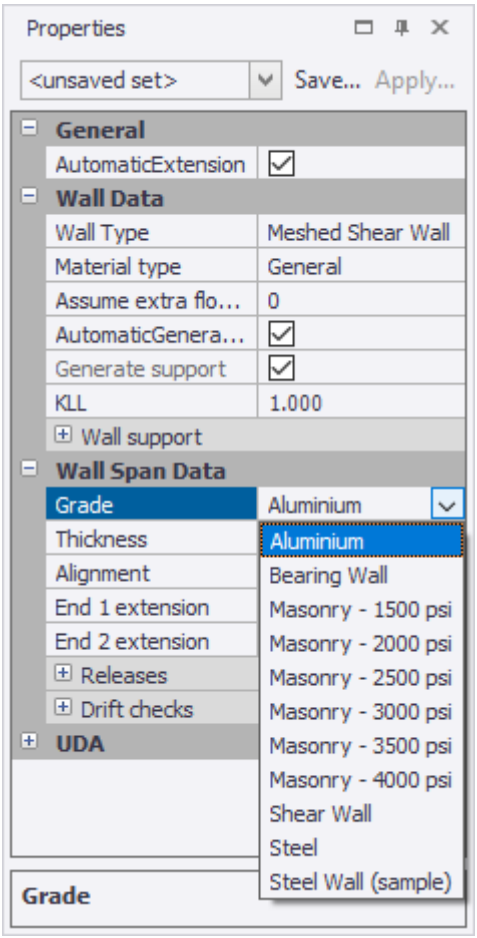
The **Grade** lists all the available grades in the Materials database under the current regional code for the selected Concrete type.

Timber Bearing Walls

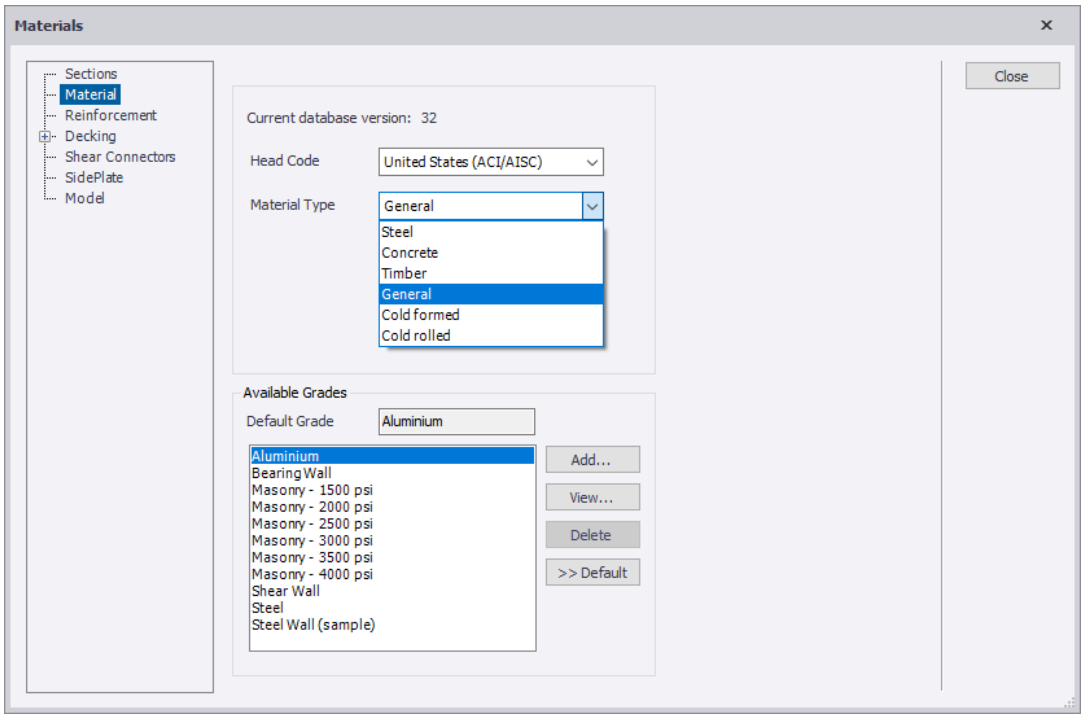
The **Grade** lists all the available grades in the Materials database under the current regional code for Timber.

General Bearing Walls

The **Grade** lists all the available **General** materials in the Materials database under the current head code.



If the grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 1267), taking care to first select the General material type as shown below.



Geometric rules

- Bearing walls can only be created as rectangular in a vertical plane.
- Wall openings are ignored.
- Bearing walls cannot be included in cores.

Create bearing walls in a level view

NOTE Ensure that you have defined the construction levels between which the wall will run, and the grid points between which the wall will lie.

1. On the **Model** tab, click  **Bearing Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. Go to the **Properties** window.
3. If necessary, adjust that the base level and top level of the wall.
4. Adjust thickness and other wall properties according to your needs.
5. In the model, click the start point of the wall.
6. Click the end point of the wall.

   Tekla Structural Designer creates the wall between the selected start and end points.

Create bearing walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run and the grid points between which it will lie.

1. On the **Model** tab, click  **Bearing Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. In the **Properties** window, adjust the thickness and other wall properties according to your needs.
3. In the model, click the first corner of the wall.
4. Click the opposite corner of the wall.

   Tekla Structural Designer creates the wall between the selected points.

See also:

Bearing wall properties (page 1432)

How bearing walls are represented in solver models (page 565)

Create shear only walls

Shear only walls provide resistance to in-plane lateral loads only.

You can model shear only walls over several story heights. In these cases, Tekla Structural Designer creates a single wall with a uniform thickness between the base and top level.

Tekla Structural Designer determines the location of the wall from the alignment specified in the shear only wall properties, and the selected insertion points.

NOTE Requirements for these walls are that they must be; within a single bay (i.e. do not overlap one or more columns); strictly rectangular; vertical and surrounded by columns and beams (other than at the bottom edge).

NOTE Shear only walls do not perform the same function as wall panels. In other words, shear only walls do not allow you to apply loads calculated by the **Wind Wizard** to your structure.

Therefore, in order to apply wind loads, you must create additional wall panels in the same locations as the shear only walls.

Material type

The chosen material will only affect the weight of the wall. Whilst Shear only walls are most suited to masonry, three **Material types** are available:

- Concrete

- Timber
- General

Geometric rules

- Shear only walls can only be created as rectangular in a vertical plane.
- They must be totally enclosed by members (concrete or steel), except at the base.
- Wall openings are ignored.
- Shear only walls cannot be included in cores.

Create shear only walls in a level view

NOTE Ensure that you have defined the construction levels between which the wall will run, and the grid points between which the wall will lie.

1. On the **Model** tab, click **Shear Only Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. Go to the **Properties** window.
3. If necessary, adjust the base level and top level of the wall.
4. Adjust stiffness, thickness, and other wall properties according to your needs.
5. In the model, click the start point of the wall.
6. Click the end point of the wall.

   Tekla Structural Designer creates the wall between the selected start and end points.

Create shear only walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run and the grid points between which it will lie.

1. On the **Model** tab, click **Shear Only Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. In the **Properties** window, adjust the stiffness, thickness, and other wall properties according to your needs.
3. In the model, click the first corner of the wall.
4. Click the opposite corner of the wall.

   Tekla Structural Designer creates the wall between the selected points.

See also:

Shear only wall properties (page 1450)

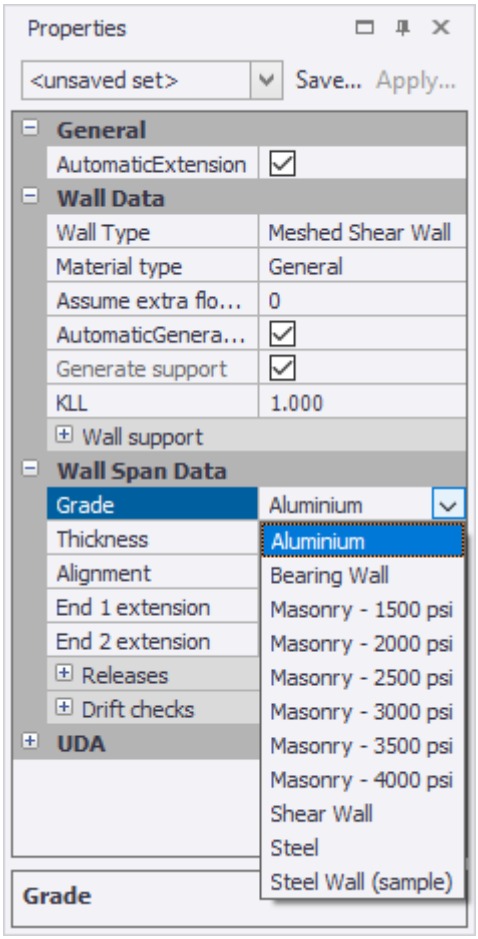
How shear only walls are represented in solver models (page 561)

Create general walls

You can create meshed general walls in level, frame, or structure views. For detailed information on creating general walls, see the following instructions.

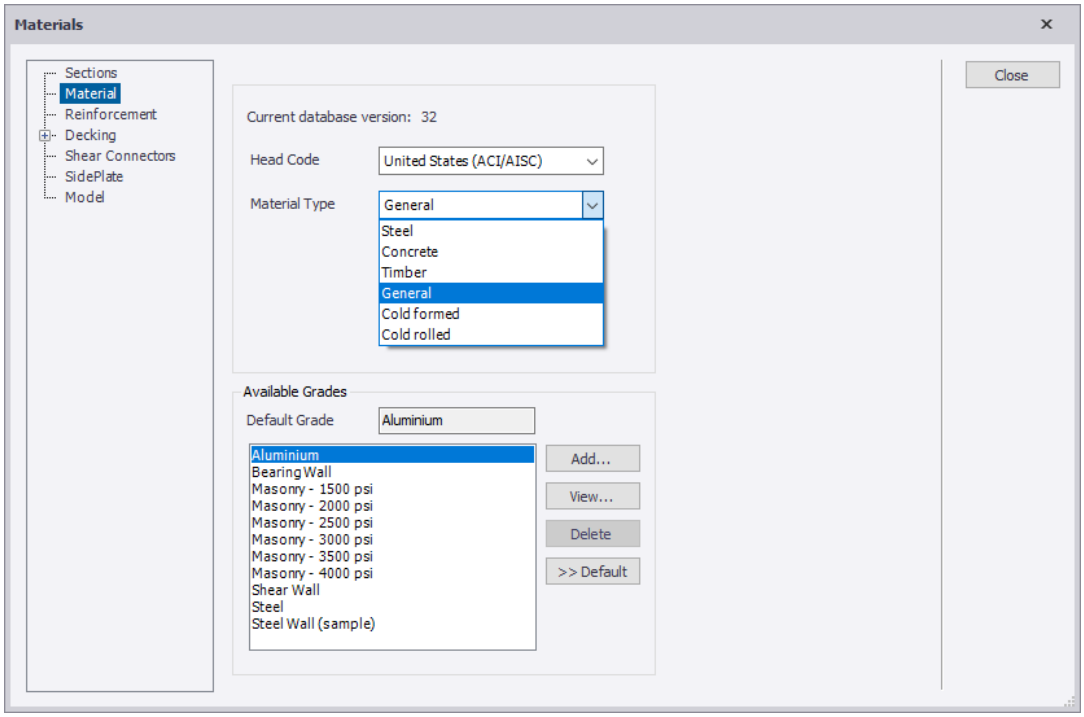
Material type

The **Grade** lists all the available **General** materials in the Materials database under the current regional code.



NOTE Care should be taken when choosing the grade, as some general materials in the database may not be appropriate for meshed walls. Meshed walls resist loads in all directions and isotropic stiffness properties are assumed.

If the grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 1267), taking care to first select the General material type as shown below.



Geometric rules

- Meshed walls are defined as quadrilaterals in a single plane that can be vertical or sloping, (unlike mid-pier concrete walls which must be rectangular in a vertical plane).
- Openings (page 204) are permitted in meshed walls.
- The alignment and offsets in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- Concrete meshed walls can be included in cores, General meshed walls cannot.

Sub-division of meshed walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Create rectangular walls in a level view

1. On the **Model** tab, click the arrow below  **General Wall**.
2. Go to the **Properties** window.

3. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. In the model, click the start point of the wall.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

7. Click the end point of the wall.

   Tekla Structural Designer creates the wall.

Create rectangular walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below  **General Wall**.
2. Go to the **Properties** window.
3. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
4. Ensure the **Rectangular** property is selected.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click a point to define a corner of the wall.
7. Click a point to define the opposite corner. Tekla Structural Designer creates the wall between the selected two points.

Create quadrilateral walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below  **General Wall**.
2. Go to the **Properties** window.
3. Ensure the **Rectangular** property is unselected.

4. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
5. Adjust the top and base level for the wall, if necessary.
6. Adjust the remaining properties, such as the wall thickness, if necessary.
7. Click the point where the base of the wall should start.
8. Click the point where the base of the wall should end.
9. Click the point where the top of the wall should start.
10. Click the point where the top of the wall should end. Tekla Structural Designer creates the wall between the selected four points.

See also

General wall properties (page 1452)

How meshed walls are represented in solver models (page 554)

Specify extensions and releases (page 202)

Create and modify wall supports (page 203)

Create door or window openings (page 204)

Create slabs and decks

This section focuses on the operations required to create slabs and decks.

- Overview of the slab model (page 217)
- Create slab items (page 221)
- Create slab or mat openings (page 223)
- Add overhangs to existing slab or mat edges (page 225)
- Apply curved edges to existing slab items (page 227)
- Create column drops (page 227)
- Specify the material for general slab types (page 228)
- Split and join slabs and mats (page 229)
- Modify slab/panel span direction (page 231)

Related information

See also

Slab item properties (page 1458)

Modify slab items and panels by moving a node (page 116)

Modify slab items by moving an edge (page 117)

Video: Creating and editing slabs (US customary units)

Video: Creating and editing slabs (metric units)

Overview of the slab model

Slabs and slab items

Tekla Structural Designer stores slab data in the form of parent **slabs**, each consisting of one or more individual panels, called **slab items**. Slab items can be either connected or separated from each other. However, they must be on the same level.

Every parent **slab** has a unique name. A slab name used at one level can not be re-used at a different level.

Every parent **slab** has general properties of **slab type**, associated **deck type** and **decomposition** as follows:

Slab Type	Deck Type	Decomposition
Slab on beams	• Reinforced concrete • Post tension	• One-way***** • Two-way
Flat slab	• Reinforced concrete • Post tension	• Two-way
Precast	• Precast concrete planks	• One-way*****
Composite slab	• Profiled metal decking	• One-way*****
General	• Steel plate • Timber • General	• One-way***** • Two-way
Foundation mat	• Reinforced concrete • Post tension	• Two-way
***** 1-way slabs with **Analyze slab** selected in their properties are treated instead as 2-way with orthotropic properties. The default Y-factor of 0.01 making them highly orthotropic, (100 times weaker transverse to the intended span direction).		

Slab items initially inherit their properties from the parent **slab**. However, once the slab items have been created, you can modify them to amend certain properties individually: for example you can change the **rotation angle**, override the **slab depth**, or **exclude a panel from the diaphragm**. Once you have overridden the slab depth, you can also apply a vertical offset in order to model a slab step.

NOTE If you amend other properties, such as material grades or general design parameters you will find they are updated not just for the individual slab item, but for all other slab items within the parent slab.

The relationship between **slabs** and **slab items** means that at the initial modeling stage you can create slabs over a wide area. While creating slabs, you do not have to consider sub-sections of the slab - you can simply create one big expanse of slab.

When it comes to design, you need to conceptualize the slab as a series of design panels, (i.e. **slab items**). Each **slab item** will have its own design settings and its own design results. You can select different reinforcement in different slab items. You also have to consider pattern loading, where some panels are loaded, and others not.

When results are later presented in calculations and drawings, you can specifically reference the design slab items.

Comparison of slab types

The different slab types available in Tekla Structural Designer are compared in the below table.

<table>
<tr><th></th><th>Slab on beams</th><th>Flat slab</th><th>Precast</th><th>Composite</th><th>General</th></tr>
<tr><td rowspan="3">Deck Type</td><td>• Reinf. concrete
• Post tension</td><td>• Reinf. concrete
• Post tension</td><td rowspan="2">Precast concrete planks</td><td rowspan="2">Profiled metal decking</td><td rowspan="2">• Steel plate
• Timber
• General</td></tr>
<tr><td colspan="2">Note: Design of post tension slabs is beyond scope</td></tr>
<tr><td colspan="5">See: Create slab items (page 221)</td></tr>
<tr><td rowspan="2">Decomposition</td><td>• 1-way*
• 2-way</td><td>2-way</td><td>• 1-way*</td><td>• 1-way*</td><td>• 1-way*
• 2-way</td></tr>
<tr><td colspan="5">* 1-way slabs with Analyze slab selected in their properties are treated instead as 2-way with orthotropic properties. The default Y-factor of 0.01 making them highly orthotropic, (100 times weaker transverse to the intended span direction).
See: One-way and two-way load decomposition (page 626)</td></tr>
</table>

	Slab on beams	Flat slab	Precast	Composite	General
Diaphragm action	• rigid • semi rigid • none	• rigid • semi rigid • none	• rigid • semi rigid • none	• rigid • semi rigid • none	• rigid • semi rigid • none
	See: Overview of diaphragm action in roof panels and slabs (page 619)				
Load	all panel load types	all panel load types	all panel load types	all panel load types	all panel load types
	See: Apply panel loads (page 352)				
Modification factors for 2-way spanning slabs	yes	yes	n/a	n/a	yes
	See: Analysis Settings>Modification factors (page 772)				
Meshed in 3D analysis and grillage chasedown analysis	optional	optional	optional	optional	optional
	See: Define whether two-way slabs are meshed for 3D building analysis and grillage chasedown analysis (page 585)				
Meshed in FE Chasedown analysis	• 1-way: optional • 2-way: yes	yes	optional	optional	• 1-way: optional • 2-way: yes
	See: FE meshed slabs (page 584)				
Designed in Tekla Structural Designer	• 1-way: beyond scope • 2-way: yes	yes	beyond scope	beyond scope	beyond scope
	See: Design slabs and mats, run punching shear checks (page 800)				

Vertical alignment

The vertical alignment of all slabs in a given level is determined by the construction level type specified in the Construction Levels dialog. The possible types are:

- **T.O.S.** (Top Of Steel): the base of each slab item aligns with the level.
- **S.S.L.** (Structural Slab Level): the top of each slab item aligns with the level.
- **T.O.F.** (Top Of Foundation): the top of each slab item aligns with the level.

Vertical offsets

A vertical offset can be specified after checking 'Override slab depth' in the slab item properties.

These offsets are not structurally significant in slabs as they have no effect on the analysis mesh position relative to the top of the slab in the solver model. However, the mesh properties would reflect any change in the slab thickness.

Slab edges

Slab edges can be either straight or curved.

See: Apply curved edges to existing slab items (page 227)

Slab openings

You can create openings of any shape within slabs. These are only considered when the slab is meshed for load decomposition or analysis.

See: Create slab or mat openings (page 223)

Column drops

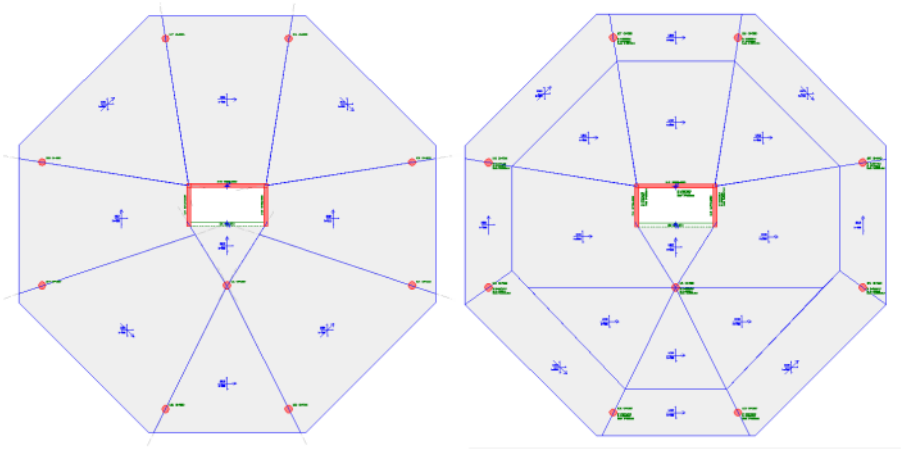
Column drop panels are slab items with an increased thickness. In flat slabs, at points where the slab is supported by columns, you can use column drops to thicken the slab.

See: Create column drops (page 227)

Panel sub-division

Regardless of how the slabs and slab items are initially created, you can further divide or re-form them with the **Slab Split** and **Slab Join** commands. There are several reasons why you may choose to use the previously mentioned commands, related to adding steps, loading patterns, and designing panels.

For flat slabs in particular, the way that slabs are split for the purposes of pattern loading is a matter of engineering judgement. The views below view two options that two different engineers might both justifiably choose for the same slab perimeter.



See: Split and join slabs and mats (page 229)

Create slab items

Slab items are individual slab panels that form parent slabs. You can create slab items either by bay, or by points. For more information, see the following instructions.

Select the slab type and specify slab properties

1. On the **Model** tab, click the arrow under **Slabs**.
2. In the list that appears, select the desired slab type.
3. In the **Properties** window, define the slab item parameters for the selected slab type.
4. Ensure that the **Slab** and **Select bays** properties are set according to your needs.

Specify the parent slab to which the slab item belongs

1. In the **Properties** window, go to the **Slab** property.
2. Click the arrow on the right side of **Slab**.
3. In the list that appears, select whether you want to:
   - Create a new slab

- Add to existing slab

  NOTE When you select this option, at the point of creating the slab item, Tekla Structural Designer checks if there is a existing slab in the plane with same properties.

  If Tekla Structural Designer finds an identical slab, the new slab item will be added to it.

  If Tekla Structural Designer finds two identical slabs, the closest one is used.

  If Tekla Structural Designer does not find an identical slab, a new parent slab is created.

- Manually select the parent slab

  NOTE This option is only available in 2D views after one or more slab items have been placed in that view.

4. When the slab item properties are defined, you can place the item either by bays or by picking points.

Create slab items by bay

You can only create slab items by bay in 2D Views.

1. In the **Properties** window, ensure that the **Select bays** option is selected.
2. Do one of the following:

To	Do this
To add an individual slab item	a. Click the outline of a bay bounded by beams or walls.
To add slab items into all bounded bays	• Drag a box that encompasses the bays. **NOTE** Dragging the box from left to right places items in those bays totally enclosed by the box. Dragging the box from right to left places items in all bays that are either enclosed by the box, or cross it. Holding down **Shift** while dragging creates a line instead of a box. Tekla Structural Designer places slab items in all bays that cross the line.

Create slab items by points

You can create slabs in both 2D and 3D Views.

1. In the **Properties** window, ensure that the **Select bays** option is cleared.

2. Click the start point of the slab item.
3. Click other points that define the slab item outline.
4. Double-click the final point to create the slab item.

Create slab or mat openings

When you have created slabs or mats, you can create different kinds of openings to them according to your needs. You can define rectangular, circular, and irregular openings. For more information on how to create and delete slab and mat openings, see the following instructions.

Simple openings

You can quickly define simple openings within existing slabs. Simple openings are rectangular or circular in plan.

The **Slab Opening** command is located on the list in the **Slabs** group. The command is only active in 2D views.

Slab openings can:

- Cross more than one slab item or slab
- Be overlaid or joined to create openings which together have shapes other than rectangular
- Cut across a stepped edge
- Be applied to level an sloping slabs

Openings cannot:

- Be applied to one-way spanning slabs
- Reside within or cut a column drop

Irregular Openings

Alternatively, you can create more complex openings by using construction lines and constructing slab items around an irregular shape.

Create rectangular openings

1. Open a 2D view of the level containing the slab item or mat panel within which you want to create an opening.
2. Do one of the following:

To	Do this
Create slab openings	a. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group. b. In the list that appears, select  **Slab Opening**.

Create mat openings	• On the **Foundations** tab, click **Mat Opening**.

The opening properties are viewed in the **Properties** window.

3. In the **Properties** window, set the **Opening Type** to **Rectangular**.
4. If necessary, specify a rotation angle to rotate the opening on plan.
5. Click within the outline of an existing slab item or mat panel to define the first corner of the opening, or press **F2** to define its exact position.
6. Drag the mouse pointer to the opposite corner of the opening.
7. Click the opposite corner of the opening, or press **F2** to define its exact position.

   Tekla Structural Designer creates the opening.

Create circular openings

1. Open a 2D view of the level containing the slab item or mat panel within which you want to create an opening.
2. Do one of the following:

To	Do this
Create slab openings	a. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Opening**.
Create mat openings	• On the **Foundations** tab, click **Mat Opening**.

The opening properties are viewed in the **Properties** window.

3. In the **Properties** window, set the **Opening Type** to **Circular**.
4. Click within the outline of an existing slab item or mat panel to define the center of the opening, or press **F2** to define its exact position.
5. Drag the mouse pointer to define the radius of the opening, or press **F2** to define the exact radius.

   Tekla Structural Designer creates the opening.

Delete slab or mat openings

1. In the **Structure** tree, open the **Slab Openings** branch.
2. Right-click the name of the slab or mat opening that you want to delete.

3. In the context menu, select **Delete**.

   Tekla Structural Designer deletes the opening.

Add overhangs to existing slab or mat edges

At times, the edge of a slab or mat may extend beyond a grid line, either to the edge of the beam which supports the edge of the slab, or around the perimeter of the building to meet the inside face of the cladding. In order to take these cases in to account, Tekla Structural Designer allows you to define overhangs to the edges of a slab. An overhang may extend across many slab items in one slab and can be curved, or tapered if required.

Any loads that you define over an overhang are included in the total loading on your building.

You can only create overhangs in 2D views.

Add an overhang along the full length of a supporting beam

NOTE This method can only be used if a beam exists along the entire slab/mat edge to which the overhang is to be added.

1. Open a 2D view of the level containing the slab/mat for which you want to create the overhang.
2. Do one of the following:

To	Do this
Create an overhang over a supporting beam to a slab edge	a. On the **Model** tab, click the arrow in the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Overhang**. c. In the **Properties** window, check **LengthOfBeam**. d. If necessary, in the **Properties** window, modify the width and other properties of the overhang. e. Click on the beam over which the overhang is to extend.
Create an overhang over a supporting beam to a mat edge	• On the **Foundations** tab, the arrow under **Mat Opening**. • In the list that appears, select **Mat Overhang**. • In the **Properties** window, check **LengthOfBeam**. • If necessary, in the **Properties** window, modify the width and other properties of the overhang.

	• Click on the beam over which the overhang is to extend.

Tekla Structural Designer creates the overhang along the length of the beam.

Add an overhang to a slab or mat edge between two points

NOTE In order to define a slab overhang between two points, you must have already defined the slab to which it applies.

1. Open a 2D view of the level containing the slab/mat for which you want to create an overhang.
2. Do one of the following:

To	Do this
Create an overhang to a slab edge	a. On the **Model** tab, click the arrow in the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Overhang**.
Create an overhang to a mat edge	• On the **Foundations** tab, the arrow under **Mat Opening**. • In the list that appears, select **Mat Overhang**.

   The slab or mat overhang properties are viewed in the **Properties** window.
3. Ensure the **LengthOfBeam** property is unchecked.
4. If necessary, in the **Properties** window, modify the width and other properties of the overhang.
5. Click along the edge of an existing slab item or mat panel to define the start point of the overhang, or press **F2** to define its exact position.
6. Click along the same edge to define the end point of the overhang, or press **F2** to define its exact position.

   Tekla Structural Designer creates the overhang between the two points.

Add a curved overhang to a slab or mat edge

While following one of the above procedures to add an overhang along the length of a supporting beam, or between two points:

1. In the **Properties** window, uncheck the **Edge parallel** property.
2. In the **Properties** window, define the **Curvature** of the overhang.
3. Continue to create the overhang in the normal way.

Add a tapered overhang to a slab or mat edge

While following one of the above procedures to add an overhang along the length of a supporting beam, or between two points:

1. In the **Properties** window, check the **Tapered** property.
2. Use the **Width1** property to define the length of overhang at end 1.
3. Use the **Width2** property to define the length of overhang at end 2.
4. Continue to create the overhang in the normal way.

Apply curved edges to existing slab items

You may sometimes need to create slab items that have curved edges instead of linear ones. For detailed instructions to do so, see the following paragraphs.

1. Select the slab items that you want to modify.
2. According to you needs, do one of the following:

To	Do this
Apply the same degree of curvature to all selected slab edges	a. In the **Properties** window, go to **All edges**. b. Clear the **Linear** option. c. Define the required curvature. A positive value creates an inward curve, whereas a negative value creates an outward curve.
Apply curvature to a specific edge	a. In the **Properties** window, go to the properties of the required edge. The edges are numbered. b. Clear the **Linear** option. c. Define the required curvature. A positive value creates an inward curve, whereas a negative value creates an outward curve.

Tekla Structural Designer redraws the slab item with the specified curvature.

Create column drops

In order to increase punching resistance, you can insert drop panels, or column drops, within concrete slabs at points where they are supported by columns. Column drops are slab thickenings that can be positioned above the slab, below the slab, or both. Column drops are rectangular in plan, and aligned to the column axes. Column drops are always positioned centrally over the supporting column.

The extent of the column drop is limited by the extent of the slab boundary. The cut back drop shape may not be rectangular.

NOTE In order to define a slab drop you must have already defined the concrete slab to which it applies.

1. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group.
2. In the list that appears, select **Column Drop**.

   The column drop properties are viewed in the **Properties** window.
3. If necessary, in the **Properties** window, modify the properties of the column drop.
4. Click an existing column connected to a concrete slab panel to create a single column drop, or drag a box around multiple columns to create a series of column drops.

   NOTE To insert the column drop correctly, ensure that the type of the construction level containing the slab is **S.S.L.**, not **T.O.S.**

Specify the material for general slab types

General slab types can have steel plate, timber, or general deck types. The material grade properties for each deck type can either be selected from the material database or they can be user-defined.

Choosing the grade from the material database

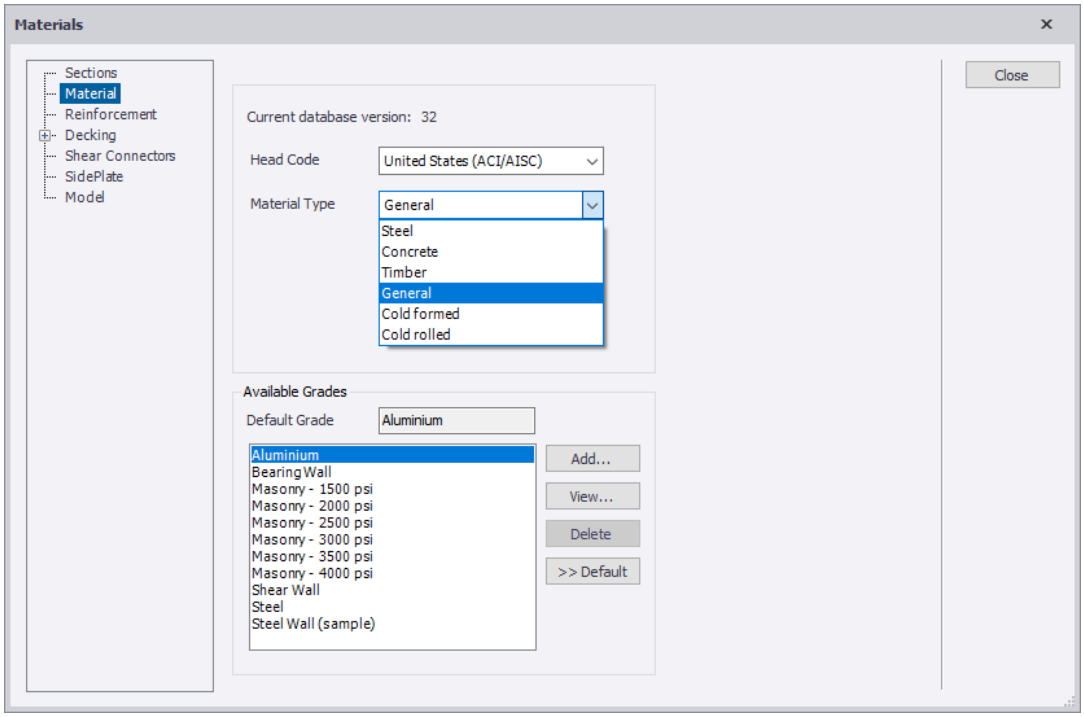
If the **Deck type** is set to:

- **Steel plate** - the **Grade** lists all the available **Steel** materials in the Materials database under the current head code.
- **Timber** - the **Grade** lists all the available **Timber** materials in the Materials database under the current regional code.

  NOTE When using a timber deck type, the program will initially default to a material override option rather than displaying a Grade. This is because the Poisson's ratio in the timber materials database is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. You can disable the override to select a timber grade, but you must then enable it again to enter a valid Poisson's ratio (other grade properties such as density and modulus of elasticity will be retained).

- **General** - the **Grade** lists all the available **General** materials in the Materials database under the current regional code.

If the general material grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 1267), taking care to first select the General material type as shown below.



Override material properties option

Unlike other slab types, general slab types each have an option to override material properties. This option is provided primarily to allow the existing properties of steel or timber deck slabs created in older versions of Tekla Structural Designer (pre 2020 SP5) to be retained, but it can also be used to enter the material properties directly if required.

NOTE When creating new timber decks, the program will default to the override option. This is because the Poisson's ratio in the timber materials database is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. You can disable the override to select a timber grade, but you must then enable it again to enter a valid Poisson's ratio (other grade properties such as density and modulus of elasticity will be retained).

Split and join slabs and mats

Tekla Structural Designer allows you to both sub-divide existing slab items into smaller items, and merge slab items into larger items.

NOTE The commands for splitting and joining slab items are only accessible in 2D views.

Split slab and mat items

1. Do one of the following:

To	Do this
Split slabs	• On the **Model** tab, click  **Slab Split**.
Split mats	• On the **Foundations** tab, click  **Mat Split**.

2. Hover the mouse pointer over the slab or mat edge, grid point, or other point where you want to start the split.

   NOTE The points used to define the cut line can be outside the boundary of the slabs being split. Thus, they do not need to be on the slab edges.

3. Click the start point of the split.
4. Click the second point of the split on either the same or other item or panel.
5. Do one of the following:
   - To continue the split, click subsequent split points.
   - To end the split, click the second point of the split again.

   Tekla Structural Designer splits any slab items entirely crossed by the split along the cut line.

Join slab and mat items

RESTRICTION You can only join slab items that share a common edge.

1. Do one of the following:

To	Do this
Split slabs	• On the **Model** tab, click  **Slab Join**.
Split mats	• On the **Foundations** tab, click  **Mat Join**.

2. Select the first of the slab items that you wish to join.

   NOTE The slab items selected later will adopt the properties of this first slab item.

3. Select the second slab item.

   Tekla Structural Designer joins the selected slab items to create a new one.

4. Select additional slab items as required, or press **Esc**.

Modify slab/panel span direction

The span direction of slab items or wall/roof panels can be edited via the **Properties** window. The span direction of individual slab items and wall/roof panels can also be edited graphically.

NOTE Provided SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow.

Modify a single span direction graphically

The span direction can be edited graphically , in both 2D and 3D views.

1. Select the slab/panel item.
2. Select the span direction symbol. A yellow command prompt is displayed and the **Properties** window lists the X direction mode options of **Align with edge** and **Define by 2 points**
3. Select the desired mode.
4. Select either a slab edge (of any slab item) or two points, depending on the selected mode.

   The slab direction is automatically set to the identified angle.

Modify span direction via the Properties window

1. Select the slab/ panel item(s).
2. In the **Properties** window enter the required rotation angle/X direction. The slab span direction is updated to the new angle.

Create trusses and joists

These topics introduce you to the methods of creating trusses and joists.

We recommend you familiarize yourself with how to:

- Create trusses (page 231)
- Create steel joists (page 234)

Create trusses

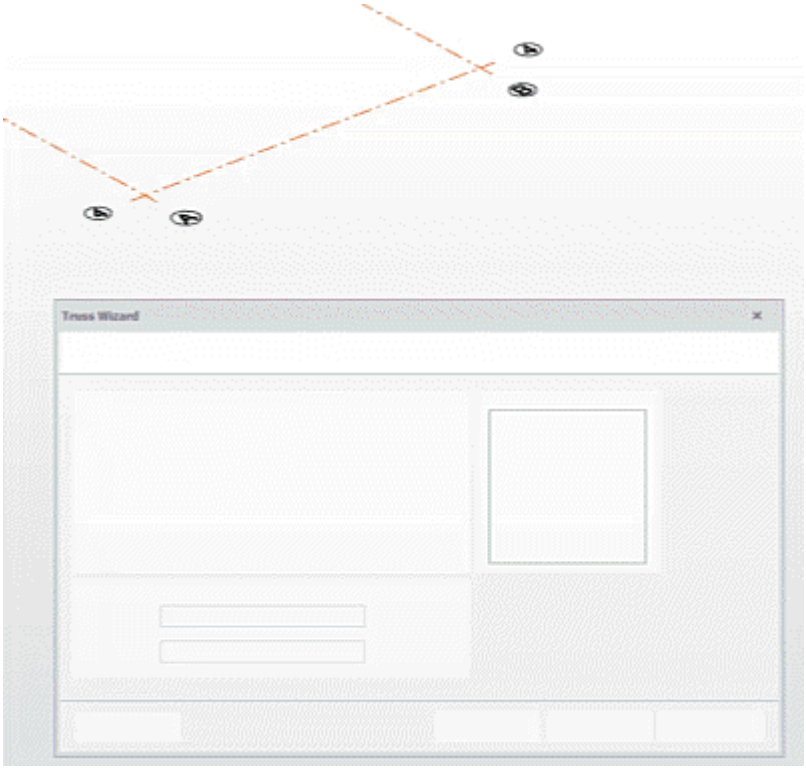
A truss wizard can be used to create a 2D, or a space truss, using steel, timber or cold formed elements. Alternatively you can manually create a free form truss without using the truss wizard.

Create a truss

The **Truss Wizard** helps you to create trusses in your models. For detailed instructions on how to use the **Truss Wizard** to create trusses, see the following instructions.

1. On the **Model** ribbon, open the truss/space/freeform list in the Steel, Timber, or Cold Formed group according to your needs.
2. Click **Truss** (for steel and cold formed), or **Truss** (for timber).

   The **Truss Wizard** opens.



3. Select the truss shape.
4. In the model, click the start point of the truss.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. In the model, click the end point of the truss.
6. In the **Truss Wizard**, click **Next**.
7. Specify the truss alignment parameters.
8. Click **Next**.
9. Specify the truss details.
10. Click **Finish**.

Create a space truss

Tekla Structural Designer allows you to create linear and planar space trusses. You can create space trusses in a 2D or 3D view. For more information, see the following instructions.

1. On the **Model** ribbon, open the truss/space/freeform list in the Steel, Timber, or Cold Formed group according to your needs.
2. Click **Space**

   The **Space Truss Wizard** opens.
3. Define the truss type, the alignment, and the number or bays.
4. Click **Next**.
5. In the model, click the truss location points:
   - For a linear truss, click the start and end points.
   - For a planar truss, click the four corners of the truss.
6. In the **Space Truss Wizard**, specify the truss width and height.

   If you are creating a planar truss, Tekla Structural Designer calculates the width automatically.
7. Click **Next**.
8. Define whether the truss should be straight or curved.
9. Click **Finish**.

Create a free form truss

You can create a free form truss by placing a series of truss members in the required shape. For more information, see the following instructions.

RESTRICTION Free form trusses cannot be created in a 3D view, or a 2D level view.

1. Open the 2D frame view in which you want to create the truss.

2. On the **Model** ribbon, open the truss/space/freeform list in the Steel, Timber, or Cold Formed group according to your needs.
3. Click **Free Form**
4. Click the start point of the first truss member.
5. Click the end point of the first truss member.
6. Repeat steps 4 and 5 to place each truss member as required.
7. When the truss geometry is complete, press **Esc** to finish.

Modify the geometry of an existing truss

After creating a truss you can modify its geometry by using the **Edit** command. For more information, see the following instructions.

1. Hover the mouse pointer over the desired truss so that it becomes highlighted.
2. Right-click the truss.
3. In the context menu, select **Edit [element name]**.

   Depending on the type of the truss, either the **Truss Wizard** or the **Space Truss Wizard** opens.
4. Modify the truss properties according to your needs.
5. To save the changes, click **Finish**.

Modify the properties of existing trusses

You can modify the section sizes, material grades, section orientations, and other truss properties in the **Properties** window. For more information, see the following instructions.

1. Move the mouse pointer over the desired truss, so that it becomes highlighted.
2. Click the truss.
3. In the **Properties** window, modify the truss properties according to your needs.

Related information

See also

Video: Creating and editing trusses (US customary units)

Video: Creating and editing trusses (metric units)

Create steel joists

Steel joists, or bar joists, are a specific type of members used in the United States. They are simply supported secondary members that, in turn, support slab and roof loads. Steel joists are constrained to standard types specified by

the US Steel Joist Institute, and standardized in terms of span, depth and load carrying capacity. To create steel joists in your model, see the following paragraphs.

Specify the section for Standard and Special joist types

1. On the **Model** tab, click **Steel Joist**.
2. In the **Properties** window, specify the **Steel joist type**
3. Click the arrow on the right side of **Steel joist type**.
4. In the list that appears, select either **Standard** or **Special** as required.
5. Select the **Section** parameter.
6. Click the arrow on the right side of **Section**.
7. In the list that appears, select **<New\Edit...>** The **Select Section** dialog box opens.
8. Select the desired joist type and size.
9. Click **Select**.
10. In the **Properties** window, adjust the remaining parameters according to your needs.

Specify the section for the Girder joist type

1. On the **Model** tab, click **Steel Joist**.
2. In the **Properties** window, select the **Steel joist type** parameter.
3. Click the arrow on the right side of **Steel joist type**.
4. In the list that appears, select **Girder**.
5. Select the **Section** parameter.
6. Click the arrow on the right side of **Section**.
7. In the list that appears, select **<New\Edit...>** The **Girder Section** dialog opens.
8. Either:
   - Enter the girder properties and then click **OK**.
   - Click **Add** if you want to add another girder section.
9. In the **Properties** window, adjust the remaining parameters according to your needs.

Create a steel joist

1. Select the joist type and size.

2. Click the start point of the joist.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

3. Click the end point of the joist.

NOTE The ends of steel joists can be on different levels.

Move a steel joist

You can modify the joist position in both 2D and 3D views.

1. Select the joist.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

Create portal frames

Tekla Structural Designer allows you to create both single-span and multi-span portal frames. After creating the portal frame, you can modify the properties of either the entire portal frame, or a portal frame member. For more information, see the following paragraphs.

Create a single or multi-span portal frame

1. On the **Model** tab, click **Portal Frame**.
2. In the **Properties** window use **Span Count** to specify the number of spans.
3. Click the start point of the first span of the portal frame.

   The start point sets the local X and Y coordinate origin for the portal frame.
4. Click the end point of the last span of the portal frame.

The start point defines the positive local X axis direction for the portal frame.

NOTE The end point must lie in the same construction level as the start point.

The **Portal Frame** dialog box opens.

5. On the **Spans** page, select the first span and click **Edit...**
6. Define the eaves level, select the frame type, and complete the frame geometry for the span.
7. Click **OK**.
8. According to your needs, do one or all of the following:
   - On the **Rafters** page, review the rafter section sizes.
   - On the **Columns** page, review the column section sizes.
   - On the **Haunches** page, specify eaves and apex haunches. See:Portal frame haunch geometry (page 238)
   - On the **Bases** page, specify the base fixity, and adjust the base levels according to your needs.

     NOTE You can also specify bases as *nominally pinned* or *nominally fixed* if required, as follows:

     a. close the **Portal Frame** dialog box,

     b. open the appropriate **Support** in the Portal Frame **Properties** window,

     c. set the Rotational stiffness x **Type** as *nominally pinned* or *nominally fixed*,

     d. enter the stiffness %

     For more details, see: Partial fixity of column bases (page 252).

   - Use the **Valleys**, **Ties**, **Tie Members**, **Parapets**, and **Parapet Members** pages to specify any additional elements.
9. To create the portal frame, click **OK**.

Modify the properties of an existing portal frame

- According to your needs, do one of the following:

To	Do this
Modify the overall frame properties	1. Hover the mouse pointer over the desired portal frame, so that it becomes highlighted. 2. Right-click the portal frame.

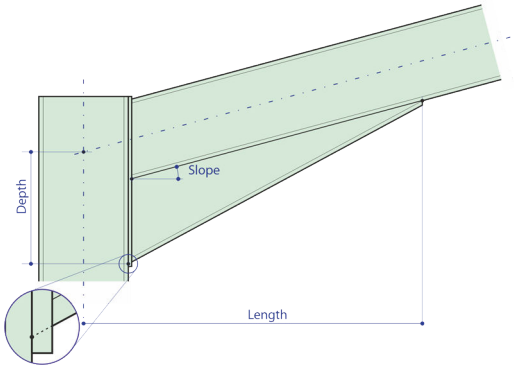
	3. In the context menu, select **Edit [element name]**. The **Portal Frame** dialog box opens. 4. Modify the properties according to your needs. 5. Click **OK**.
Modify the properties of an individual portal frame member	1. Hover the mouse pointer over the desired portal frame member, so that it becomes highlighted. 2. In the **Select Entity** tooltip, scroll until the desired member is highlighted. 3. In the model, right-click the member. 4. In the context menu, select **Edit [element name]**. The **Properties** dialog box opens. 5. Modify the properties according to your needs. 6. Click **OK**.

Add copy or mirror spans in an existing portal frame

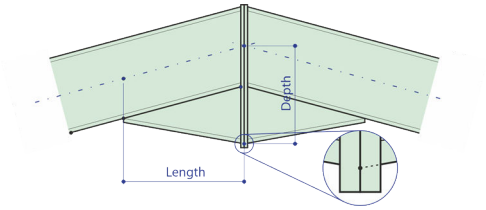
1. Open the **Portal Frame** dialog box (as described in the above topic).
2. Go to the **Spans** page of the **Portal Frame** dialog box.
3. Use the available buttons to introduce additional spans:
   - To introduce a new span at the end of the frame, click **Add**.
   - To insert a new span below the currently highlighted span, click **Insert**.
   - To copy the currently highlighted span to a pre-existing target span, click **Copy...**
   - To mirror the currently highlighted span to a pre-existing target span, click **Mirror...**
4. Click **OK**.

Portal frame haunch geometry

Eaves haunch dimensions



Apex haunch dimensions



Create members with cold rolled sections

Tekla Structural Designer allows you to create multiple characteristic types of cold rolled section members. The types are track, stud, and joist (US), or eaves beam, purlin, and rail (UK), depending on the language that you are using.

RESTRICTION You can only model and analyze cold rolled members in Tekla Structural Designer, but you cannot design them.

Create a cold rolled section member

1. On the **Model** tab, click any steel member type, for example, **Steel Beam**.
2. In the **Properties** window, set the **Characteristic** property to the desired cold rolled section member type.

   The properties in the **Properties** window are updated to match the selected cold rolled section member type.
3. In the **Properties** window, adjust the properties of the cold rolled section member according to your needs.

4. Click the start point of the member.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the member.

   Tekla Structural Designer creates the member.

Modify the position of a cold rolled section member

You can modify a cold rolled section member in both 2D and 3D views.

To move an entire cold rolled section member, see: Move and rotate objects (page 271).

1. Click the member that you want to modify.
2. Click the end node that you want to move.
3. Click the new position of the selected end node.

   The end node moves to the selected position.

Create wall and roof panels

Wall panels, also referred to as wind walls, allow you to apply loads calculated by the **Simple Wind Loading Generator** and **Wind Wizard** to your structure. Wall panels do not introduce any structural strength or stiffness to your structure. If you wish to introduce walls that resist gravity, or lateral loads, you must define them as concrete walls.

Roof panels allow loads placed on a sloping plane to be decomposed back to the supporting structure. Area loads on roofs can act either vertically, or normal to the roof plane. To create and modify roof panels, see the following instructions.

Create rectangular wall panels

1. On the **Model** tab, click **Wall Panel**.
2. In the **Properties** window, ensure the **Rectangular** property is selected.
3. Click the start point to define a corner of the panel.
4. Click a point to define the opposite corner. Tekla Structural Designer creates a wall panel between the selected two points.

Create non-rectangular wall panels

1. On the **Model** tab, click **Wall Panel**.
2. In the **Properties** window, ensure the **Rectangular** property is unselected.
3. Click the start point of the panel.
4. Click the remaining points of the panel.
5. To define the end point of the panel, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates a wall panel between the selected points.

Create wall panels with parapets

NOTE To ensure the wind analysis accounts for the parapet correctly, a wall panel with a parapet should be modeled in two parts:

1. Create an ordinary wall panel up to the roof level
2. Create a second wall panel above the roof level and marking the panel as a parapet

1. Open a frame view in which you want to create the wall.
2. Create the wall panel below the roof level normally.
3. Create the wall panel above the roof level normally.

   NOTE You may need to create a new construction level to define the top level of the parapet.

4. Press the **Esc** key.
5. Hover the mouse pointer over the second wall panel, so that it becomes highlighted.
6. Click the second wall panel.
7. In the **Properties** window, select the **Is a parapet wall** option.

See also: Parapet wall panel load decomposition (page 1505)

Modify the properties of a wall panel

You can modify the properties of a panel wall in both 2D and 3D Views.

1. Hover the mouse pointer over the panel, so that i becomes highlighted.

2. Click the panel to select it.

   TIP If necessary, add further roof panels to the selection by holding down the **Ctrl** key and clicking the panels.

3. Go to the **Properties** window.
4. Modify the properties according to your needs.

   Tekla Structural Designer automatically applies the changes to all the selected panels.

Create roof panels

1. On the **Model** tab, click **Roof Panel**.
2. Click the start point of the panel.
3. Click the remaining points of the panel.
4. To define the end point of the panel, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates a roof panel between the selected points.

Modify the properties of roof panels

You can modify the properties of roof panels in both 2D and 3D Views.

1. Hover the mouse pointer over the panel, so that it becomes highlighted.
2. Click the panel to select it.

   TIP If necessary, add further roof panels to the selection by holding down the **Ctrl** key and clicking the panels.

3. Go to the **Properties** window.
4. Modify the properties according to your needs.

   Tekla Structural Designer automatically applies the changes to all the selected panels.

Create supports

Supports allow you to constrain points in your structure positionally and rotationally. In addition, you can change the fixity provided at an existing support by modifying the support properties.

Create a single support

RESTRICTION	You can only place supports at existing grid points.

1. On the **Model** tab, click **Support**.

   The support will adopt the properties currently displayed in the **Properties** window.
2. If necessary, in the **Properties** window, adjust the support properties.
3. Click the grid point where you want to create the support.

Create a rotated or inclined support using 3 grid points

You can create a rotated support as follows:

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that the **3 Grid Points** option is selected.
4. Click the grid point where you want to create the support.
5. To define the support direction along the x axis, click a second point.
6. Lastly, define the plane containing the x and y axes by clicking a third point. The support is created with its x axis in the direction specified by the second point, and its y axis at 90 degrees to the x axis within the plane defined by the first, second and third points.

SEE ALSO	Rotated and inclined supports (page 245)

Create spring supports

You can create linear and non linear spring supports in a similar fashion.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that in the direction where you want the spring to act, the degree of freedom is set to **Free**.
4. In the appropriate stiffness properties, do one of the following:
   - Select the **Spring Linear** option, and define a single stiffness value that acts in both the positive and negative directions.

- Select the **Spring Non-linear** option, and define two stiffness values, one to act in the positive direction and another to act in the negative direction.

5. Click the grid point where you want to create the support.

Create nominally pinned or nominally fixed supports

A support placed under a column is a special case which can be specified as nominally pinned, or nominally fixed if required.

NOTE A validation error is produced if a nominally pinned or nominally fixed support is placed at any other location.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that the Mx and/ or My directions about which you require partial fixity are set to **Free**.
4. In the appropriate rotational stiffness properties, do one of the following:
   - Select the **Nominally pinned** option, and define the required stiffness value as a percentage of the column stiffness 4EI/L.
   - Select the **Nominally fixed** option, and leave the stiffness value as 100% of the column stiffness 4EI/L.
5. Click the grid point where you want to create the support.

SEE ALSO Partial fixity of column bases (page 252)

Modify support properties

You can modify support properties in both 2D and 3D views.

1. Hover the mouse pointer over the desired support, so that it becomes highlighted.
2. Click the support.

   TIP You can add further supports to the current selection by holding down the **Ctrl** key and clicking the additional supports.

3. Go to the **Properties** window.
4. Modify the support properties according to your needs.

   Tekla Structural Designer automatically applies any changes to the selected supports.

SEE ALSO	• Rotated and inclined supports (page 245) • Partial fixity of column bases (page 252)

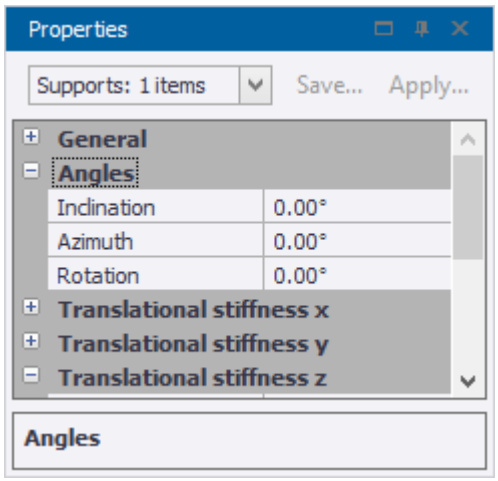
Rotated and inclined supports

A support placed under a column or wall is constrained to have its x and y axes aligned to the column/wall principal axes and its z axis vertical. If the column/wall is subsequently rotated the support will rotate with it.

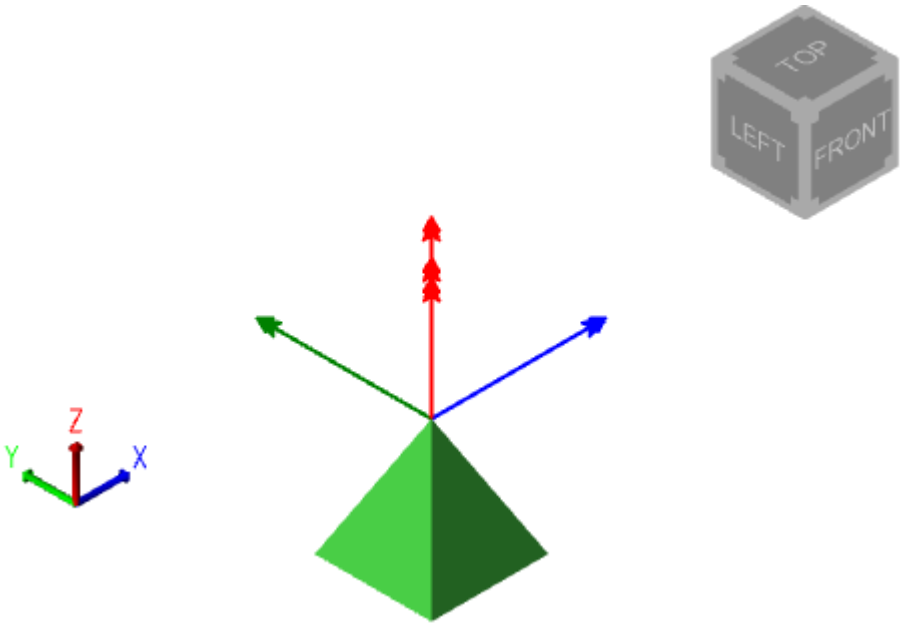
All other supports can be orientated as required by setting **inclination**, **azimuth** and **rotation** angles in the support properties, where:

- **Inclination** - rotates the support about **Global X**
- **Azimuth** - rotates an inclined support about **Global Z**
- **Rotation** - rotates the support about its **local z axis**

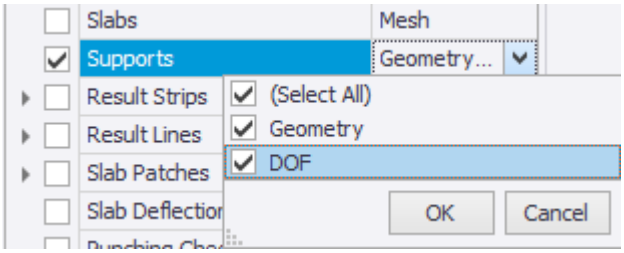
By default, for a new support the above values are zero.



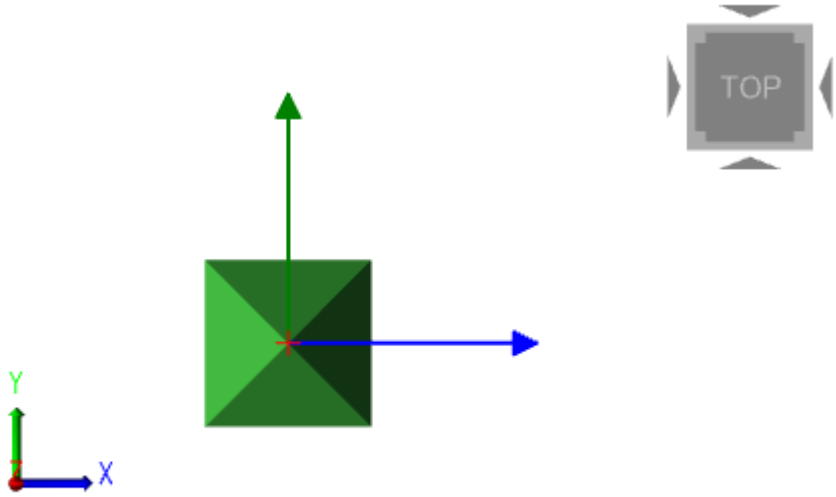
Accepting the defaults creates a support with its local x, y and z axes aligned to Global X, Y and Z.



TIP To see the support local x, y, and z axes as shown above, open the **Scene Contents** window and select **Supports > DOF**.

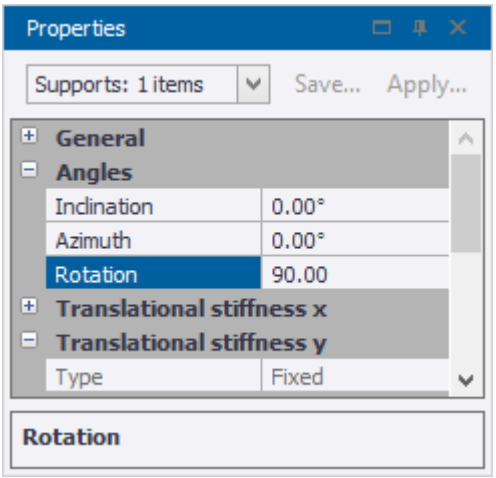


When viewed from above this alignment can be seen more clearly:

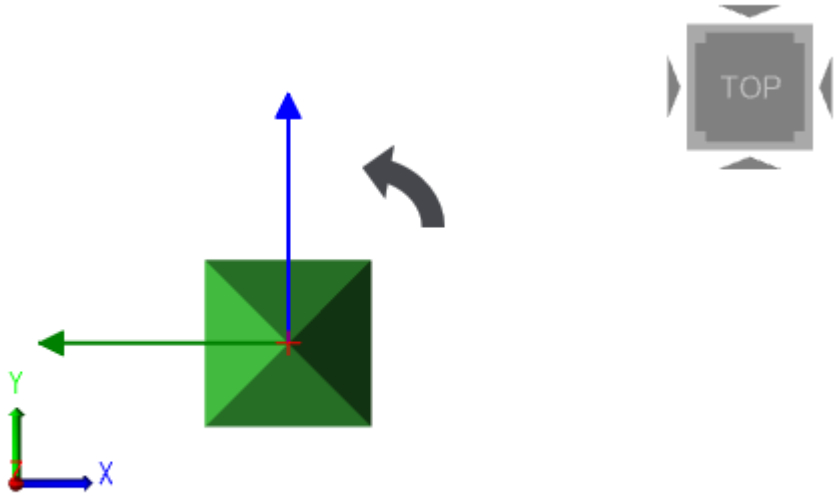


Apply a rotation (only)

To just rotate a support on plan specify a **Rotation** in the support properties.

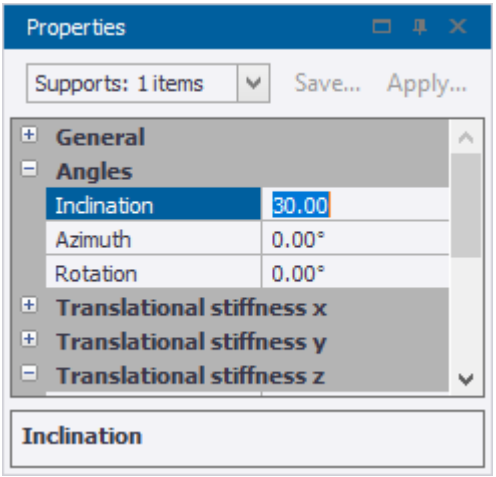


A positive rotation rotates the support about its **local z axis**. This is an anti-clockwise rotation when viewed from above.

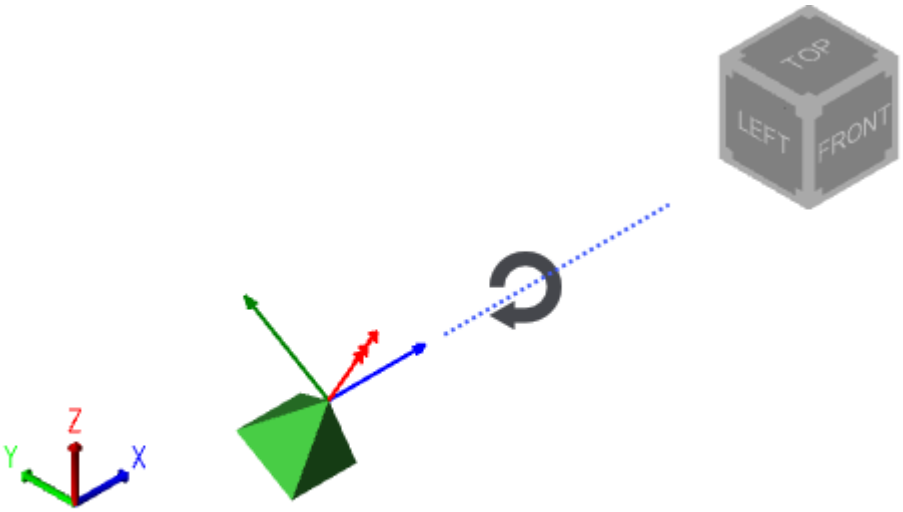


Apply an inclination

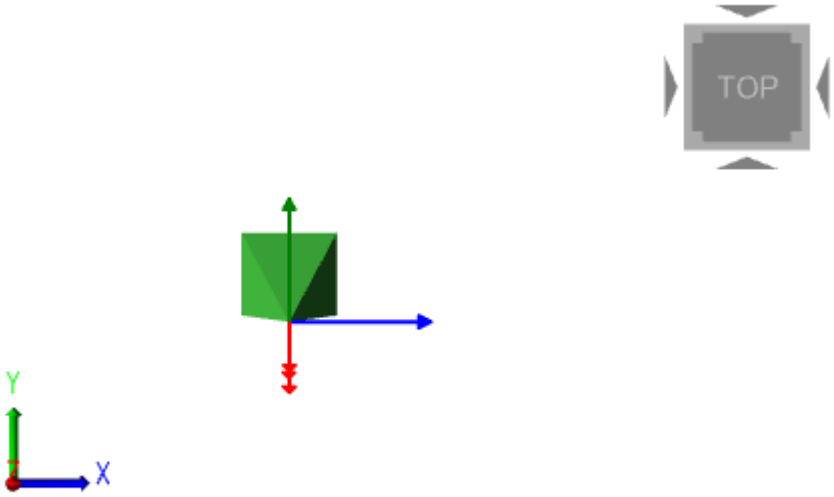
To incline a support specify an **Inclination** in the support properties.



A positive inclination rotates the support about **Global X**. This is a clockwise rotation when looking in the positive Global X direction.

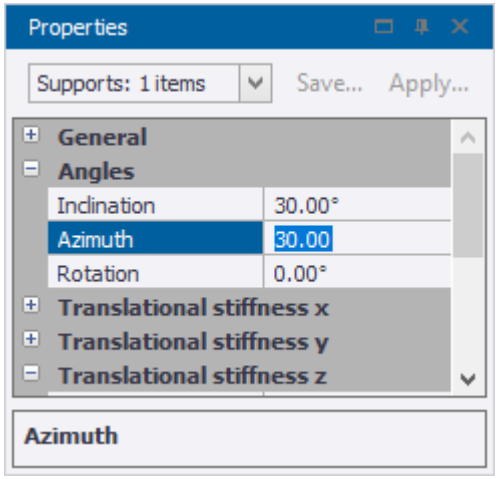


In a top down view a support inclined at 30 degrees would be shown as follows.



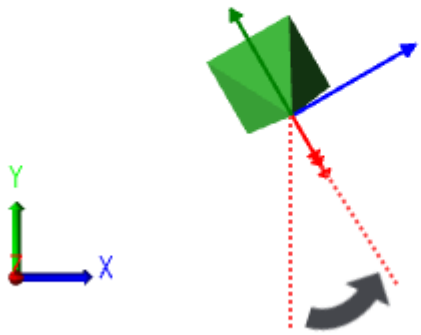
Apply an azimuth to an inclined support

To rotate an inclined support about Global Z specify an **Azimuth** in the support properties.



An Azimuth can only be applied to an inclined support. It rotates the support about **Global Z**, which is an anti-clockwise rotation when viewed from above.

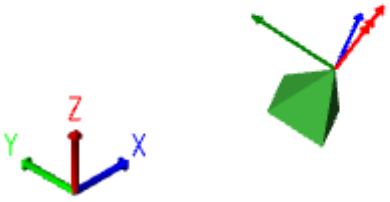




NOTE When viewed from above the **local z axis** appears to point in the direction of **Global X** for an **Azimuth** of 90 degrees, or **Global Y** for an **Azimuth** of 180 degrees.

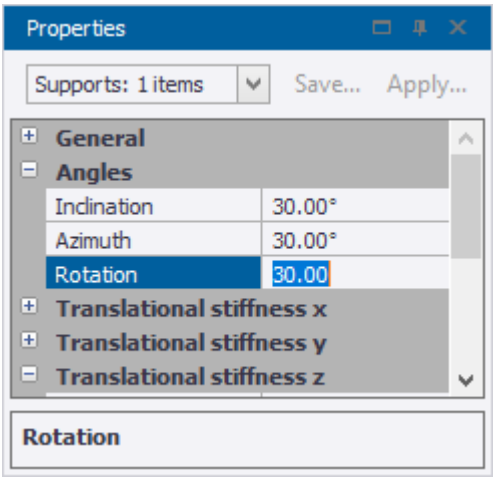
Switching back to an isometric view, for an inclination of 30 degrees and an azimuth of 30 degrees, the support would be aligned as follows.



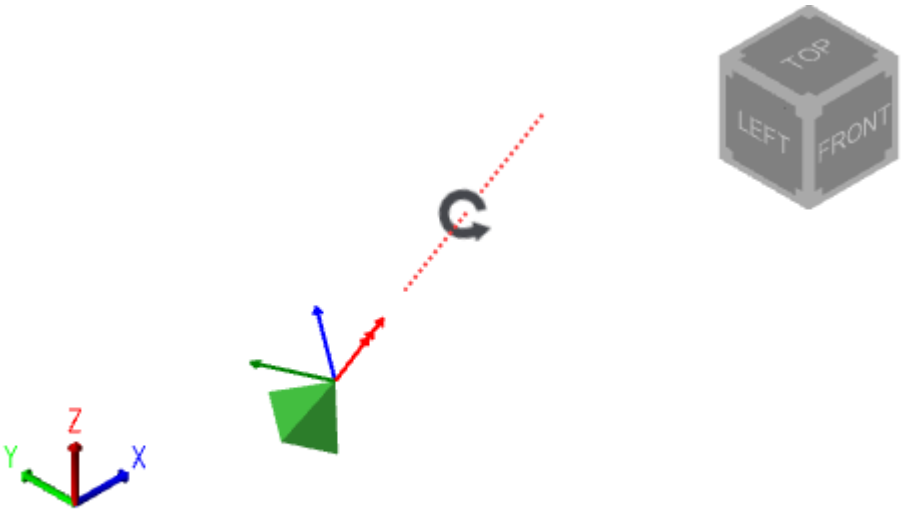


Apply a rotation to an inclined support

To rotate an inclined support about **local z** specify a **Rotation** in the support properties



A positive rotation rotates the support about its **local z axis**, which is a clockwise rotation when looking in the positive local z direction.



NOTE Once an **Inclination** has been applied, a rotation about the local z axis is **not** the same as a rotation about Global Z.

Partial fixity of column bases

Two additional types of rotational linear spring are provided to allow partial fixity to be modelled, these are:

- Nominally pinned
- Nominally fixed

These are specifically provided for supports under columns (of any material), but will result in a validation error if placed under walls, or if they are used for any other supports.

The support stiffness is based on the column properties (E*I/L)

- E = Young's Modulus of the column
- I = relevant bending stiffness (Ixx or Iyy) of the column
- L = distance from the support to the first column point (stack) that is on a Construction Level checked as a Floor in the Levels dialog, i.e. combined length of all the stacks until a floor is found.

NOTE Where no Floor has been defined in the column above the support, then L is taken as total length of column.

Partial fixity spring stiffness is calculated as follows for each of the two bending releases Mx and My:

- Nominally pinned (spring stiffness) - x% * 4 * E*I/L (default x% = 10%)
- Nominally fixed (spring stiffness) - x% * 4 * E*I/L (default x% = 100%)

Since the spring stiffness is dependent upon stack height and column stiffness (E and I), the spring stiffness will change if any changes are made to column stack height, column E or I values.

In addition, since for steel, Auto Design can change the column size (and hence I value) the spring stiffness will change with any change in column size.

Create analysis elements

Linear and non-linear elements and springs can be modeled in Tekla Structural Designer as analysis elements.

See also

Analysis Element properties (page 1419)

Element types (page 254)

Create analysis elements

1. On the **Model** tab, click  **Element**.
2. In the **Properties** window, select the Element type (page 254) that you require.
3. In the **Properties** window, adjust the material and/or properties of the element according to your needs.
4. Click the start point of the element.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the element.

   Tekla Structural Designer creates the element.

Create analysis element springs

1. On the **Model** tab, click **Element**.

   The element will adopt the properties currently displayed in the **Properties** window.
2. In the **Properties** window, select the spring Element type (page 254) that you require.
3. Enter the the material and properties of the element according to your needs.
4. Click the start point of the element.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
5. Click the end point of the element.

   Tekla Structural Designer creates the element.
6. Select the newly created element.
7. In the **Properties** window, enter the spring stiffness properties that you require.

Modify the position of analysis elements

You can modify the position of an individual analysis element in both 2D and 3D views.

1. Select the element that you want to move.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.
3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

Element types

This topic introduces the different element types available when creating Analysis Elements

Beam

The **Beam** element type is defined as a 2 node element with 6 potential releases; 3 translational (Fx, Fy, Fz) and 3 rotational (Mx, My, Mz); at each end.

Material properties and the following section properties are required:

- Ax
- A parallel to minor
- A parallel to minor
- Ix
- I major
- I minor

Truss

The **Truss** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Material properties and the area of the section are required.

This element type acts in both tension and compression.

Tension only

The **Tension only** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Material properties and the area of the section are required.

This element type acts in tension only.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements

Compression only

The **Compression only** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Material properties and the area of the section are required.

This element type acts in compression only.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements

Linear axial spring

The **Linear axial spring** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

An axial **Spring Stiffness** property is required.

This element type acts in both tension and compression.

Linear torsional spring

The **Linear torsional spring** element type is defined as a 2 node element which has torsional (Mx) fixity at each end. The remaining 5 degrees of freedom; 3 translational (Fx, Fy, Fz) and 2 rotational (My, Mz); are released at each end.

A rotational **Spring Stiffness** property is required.

Non-linear axial spring

The **Non-linear axial spring** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Seperate tension and compression **Spring Stiffness** properties are required and also the maximum tension and compression values.

NOTE Non-linear elements require non-linear analysis. If linear analysis is performed they will be treated as linear elements.

Non-linear torsional spring

The **Linear torsional spring** element type is defined as a 2 node element which has torsional (Mx) fixity at each end. The remaining 5 degrees of freedom; 3 translational (Fx, Fy, Fz) and 2 rotational (My, Mz); are released at each end.

Seperate tension and compression torsional **Spring Stiffness** properties are required and also the maximum torsional tension and compression values.

NOTE Non-linear elements require non-linear analysis. If linear analysis is performed they will be treated as linear elements.

Link

A basic link element is implemented in Tekla Structural Designer, defined as a 2 node element with 6 springs; 3 translational (Fx, Fy, Fz) and 3 rotational (Mx, My, Mz); at end 2.

This element type is used internally to effect partial fixity in beams.

Dimensions and measurements

Dimensions allow you to show distances between the appropriate points in your structure. The dimension lines are included on any drawings you create. You can also measure distances and angles without creating dimensions.

Related information

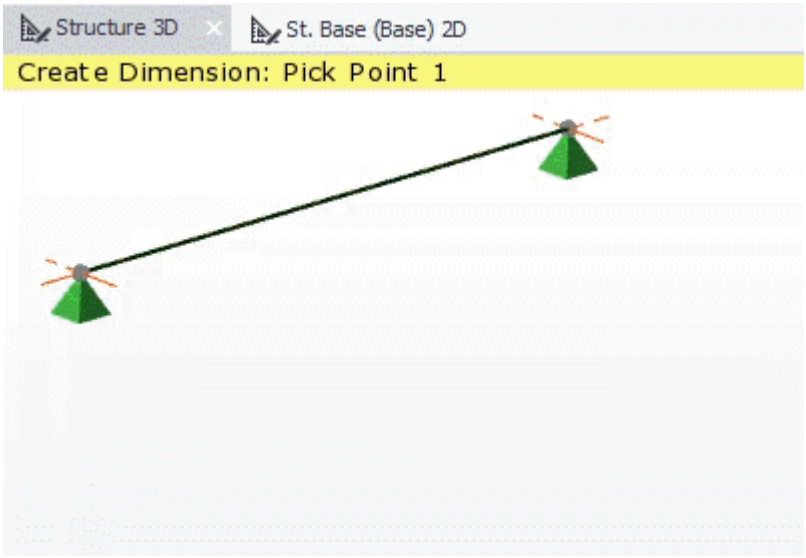
See also

Video: Dimensions and measuring (US customary units)

Video: Dimensions and measuring (metric units)

Create a dimension between existing construction points

The default dimensioning option (i.e. with the**Freehand Dimension** checkbox cleared) allows you to create a dimension between any existing construction points, or gridline intersections in the model.



1. On the **Model** tab, click **Dimension**.
2. In the **Properties** window ensure the **Freehand Dimension** checkbox is cleared.
3. Click **Point 1** at the start of the dimension, this must be an existing construction point, or grid point.
4. Click **Point 2** at the end of the dimension, this must be an existing construction point, or grid point.

   Tekla Structural Designer shows a line between the selected points.

5. Move the line to the point where you want the dimension line to lie. (This line can be positioned so as to define the projected dimension in X, the projected dimension in Y, or the parallel dimension between the two points.
6. Click to create the dimension.

Create a freehand dimension between any two points in a plane

By selecting the **Freehand Dimension** checkbox in the **Properties** window you are able to create dimensions between any two coordinates in a specified plane.

NOTE Because freehand dimensions are plane based, it is recommended they are created from 2D views.

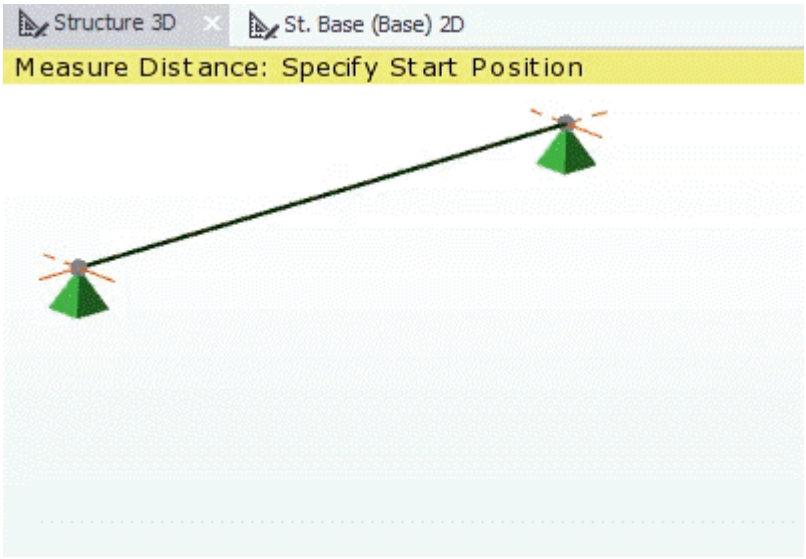
---

1. On the **Model** tab, click  **Dimension**.
2. In the **Properties** window ensure that **Freehand Dimension** is selected.
3. Select a suitable **Reference Point** in the plane in which you want to create the dimension.
4. Pick freehand **Point 1** in the reference plane to define the start of the dimension.
5. Pick freehand **Point 2** in the reference plane to define the end of the dimension.

   Tekla Structural Designer shows a line between the selected points.
6. Move the line to the point where you want the dimension line to lie. (This line can be positioned so as to define the projected dimension in X, the projected dimension in Y, or the parallel dimension between the two points.
7. Click to create the dimension.

Measure a distance

To know the exact distances or angles between different points in the model, you can use the Tekla Structural Designer measuring commands.



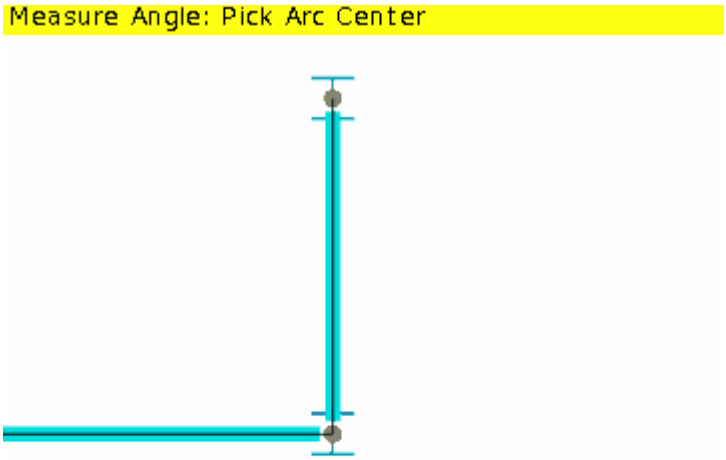
1. On the **Model** tab, click **Measure**.
2. Click a node to define the start position.
3. Click a second node to define the end position.

The distance between the nodes is displayed in the current view. To clear the measurement, press **Esc**.

Measure an angle

To know the exact distances or angles between different points in the model, you can use the Tekla Structural Designer measuring commands.

Angles can only be measured in 2D scene views.



1. On the **Model** tab, click **Measure Angle**.
2. Click a node to define the arc center.
3. Click a second node to define the start position.
4. Click a third node to define the end position.

Both the clockwise and anti-clockwise angles between the start and end position are displayed in the current view. To clear the measurement, press **Esc**.

2.3 Create isolated foundations

You can create reinforced pad bases or pile caps under columns, and strip bases under walls.

Before you can create a pile cap, the pile catalogue must contain at least one pile type.

- Create pad bases (page 261)
- Create strip bases (page 262)
- Create a pile type catalogue (page 262)
- Create pile caps (page 263)

Create pad bases

A reinforced pad base is an isolated foundation that supports a single column.

Create a pad base under a specific column

1. On the **Foundations** tab, click **Pad Base Column**.
2. In the **Properties** window, adjust the pad base properties according to your needs.
3. Click anywhere in the wall under which you want to place the pad base.

   Tekla Structural Designer places a pad base under the selected column.

Create multiple pad bases

1. On the **Foundations** tab, click **Pad Base Column**.
2. In the **Properties** window, adjust the pad base properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the columns under which you want to place pad bases.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places pad bases under all columns within the box.

Override the loaded area to be used in the pad base punching check

When you create a pad base, initially the punching shear loaded area is based on the area of the column section being supported. If you require the punching check to consider an unmodeled base plate or pedestal you can do so by specifying a **User loaded area** as follows:

1. Select the pad base.
2. In **Properties** window, go to the **Foundation Parameters** section and select **User loaded area**.
3. Specify a **Loaded Area Breadth** and **Loaded Area Depth** appropriate to the base plate or pedestal.

See also

Design isolated foundations (page 818)

Create strip bases

A strip base is an isolated foundation that supports a single wall. To place strip bases in your model, see the following instructions.

Create a strip base under a specific wall

1. On the **Foundations** tab, click  **Strip Base Wall**.
2. In the **Properties** window, adjust the strip base properties according to your needs.
3. Click anywhere in the wall under which you want to place the strip base.

   Tekla Structural Designer places a strip base under the selected wall.

Create multiple strip bases

1. On the **Foundations** tab, click  **Strip Base Wall**.
2. In the **Properties** window, adjust the strip base properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the walls under which you want to place strip bases.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places strip bases under all walls within the box.

See also

Design isolated foundations (page 818)

Create a pile type catalogue

In order to use pile foundations in your model, you must create a catalogue of pile types. To do so, see the following instructions.

1. On the **Foundations** tab, click  **Catalogue**.

   The (page 1571) opens.
2. To create a new pile type, click **Add...**

   The **Edit Pile Type** dialog box opens.

3. On the different pages of the **Edit Pile Type** dialog box, define the properties of the new pile type.
4. Click **OK**.

   Tekla Structural Designer creates the pile type.
5. Repeat steps 2–4 as necessary.

TIP In the **Pile Catalogue**, you can also modify or delete the existing pile types:

- To modify a pile type, select the pile type and click **Edit...**
- To delete a pile type, select the pile and click **Delete**.

NOTE Material information defined in the (page 1571) has no design significance. It is assigned solely to allow embodied carbon to be calculated and reported for the 'Pile' construction type.

Create pile caps

A pile cap is an isolated piled foundation that supports a single column.

NOTE Before you can create pile caps, the **Pile Catalogue** must contain at least one pile type.

To create pile types, see Create a pile type catalogue (page 262).

Create pile cap under a specific column

1. On the **Foundations** tab, click  **Pile Cap Column**.
2. In the **Properties** window, adjust the pile cap properties according to your needs.
3. Click anywhere in the column under which you want to place the pile cap.

   Tekla Structural Designer places a pile cap under the selected column.

Create multiple pile caps

1. On the **Foundations** tab, click  **Pile Cap Column**.
2. In the **Properties** window, adjust the pile cap properties according to your needs.

3. Move the mouse pointer to a corner of an imaginary box that will encompass the columns under which you want to place pile caps.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places pile caps under all columns within the box.

Create a user-defined pile arrangement

If necessary, you can modify the pile arrangement in a pile cap and create user-defined pile arrangements. Creating user-defined pile arrangements can be particularly useful when you want to check a pile cap where the pile positions on site do not exactly match the originally specified arrangement.

Do the following:

1. In the model, select the pile cap that you want to modify.
2. In the **Properties** window, go to **Pile arrangement**.
3. Click the **...** button next to **Pile arrangement**.

   The **Pile Arrangement** dialog box opens.
4. In the **Pile Arrangement** dialog box, select the **User Defined Arrangement** option.

   NOTE If the **Auto-design piles** option is selected in the **Properties** window, Tekla Structural Designer asks you whether you want to turn off the option. Click **Yes**.

5. Specify the pile arrangement properties according to your needs:
   a. Select the pile type.
   b. Define the pile length and width.
   c. If necessary, define the eccentricity of the pile cap centroid from the centroid of the column it supports in X and Y directions.

   TIP To add more piles into the arrangement, click **Add**.

6. If necessary, in the table below the properties, adjust the position of piles by defining the eccentricity of the pile centroid from the pile cap centroid.
7. Click **OK**.

   Tekla Structural Designer creates the pale arrangement that you defined.

Override the loaded area to be used in the pile cap punching check

When you create a pile cap, initially the punching shear loaded area is based on the area of the column section being supported. If you require the punching check to consider an unmodeled base plate or pedestal you can do so by specifying a **User loaded area** as follows:

1. Select the pile cap.
2. In **Properties** window, go to the **Foundation Parameters** section and select **User loaded area**.
3. Specify a **Loaded Area Breadth** and **Loaded Area Depth** appropriate to the base plate or pedestal.

See also

Design isolated foundations (page 818)

2.4 Create mat foundations

Mat foundations support multiple columns and walls. Mats can be ground bearing, supported on piles, or both. Tekla Structural Designer contains the following commands for creating mat foundations:

- **Minimum Area**: creates an overhanging polygonal mat to minimize the required area
- **Rectangular**: creates an overhanging rectangular mat either at a specified angle to the global axes or at the smallest rectangular area aligned to the global axes
- **Strip**: creates a constant width mat along a series of points that do not have to be on the same line
- **Area**: creates an overhanging polygonal mat when you identify points around its outline
- **Bays**: creates a mat with no overhang when you click within a closed grid area
- **Pile**: creates individual piles underneath existing mats
- **Pile Array**: creates an array of piles underneath existing mats

Click the links below to find out more:

- Create mats (page 266)
- Place piles and pile arrays in mats (page 267)

See also

Create slab or mat openings (page 223)

Add overhangs to existing slab or mat edges (page 225)

Create column drops (page 227)
Split and join slabs and mats (page 229)

Create mats

In Tekla Structural Designer, you can create various types of mat foundations, including minimum area mats, rectangular mats, strip mats, area mats, and mats within bays. Circular mats and mats with curved edges can be created by following the same instructions used for creating mats within bays.

For detailed information on creating mats, see the following instructions.

Create a minimum area or rectangular mat

1. On the **Foundations** tab, click either  **Minimum Area** or  **Rectangular**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. Do one of the following:

To	Do this
Create a mat in a 3D view	a. Click anywhere on the first column or wall to be supported. b. Click to define the other necessary columns and walls. c. To create the mat, click the last column or wall again or press **Enter**.

To	Do this
Create a mat in a 2D view	a. To select multiple columns and walls, hold down the left mouse button and drag a box from right to left to encompass the columns/walls under which you want to create the mat. b. To add individual columns and walls, click each member individually. c. To create the mat, click one of the previously selected members or press **Enter**.

Create a strip mat

1. Open a 2D view at the base level where you want to place the strip mat.
2. On the **Foundations** tab, click  **Strip**.

3. In the **Properties** window, enter the width (measured from the strip center line to the edge).
4. Adjust the other mat properties according to your needs.
5. In the model, click the start point of the strip.
6. Click to define the other necessary points.
7. To create the strip, click the last point again or press **Enter**.

Create an area mat

Either click once again on the last point, or press <Enter> to create the mat.

1. On the **Foundations** tab, click  **Area**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. In the model, click a construction point at a corner of the area.
4. Click to define the remaining corner points.
5. To create the mat, click the last point again or press **Enter**.

Create a mat within bays

1. On the **Foundations** tab, click **Bays**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. To select a bay, do one of the following:
   - Click within an enclosed area defined by grid lines, or grid arcs.
   - Hold down the left mouse button and drag a box around an enclosed area defined by grid lines, or grid arcs.

See also

Place piles and pile arrays in mats (page 267)

Design and check mat foundations (page 809)

Place piles and pile arrays in mats

A piled mat can either be supported on piles alone, or by a combination of ground springs and piles. Piles can either be placed individually at specific locations within the mat, or an array of equally spaced piles can be created. To place piles in mats and define their inclination, see the following instructions.

Specify if a piled mat is ground bearing

1. Click the desired mat.
2. In the **Properties** window, go to **Soil Parameters**.
3. Do one of the following:

To	Do this
Create a mat supported on piles alone	Clear the **Use Ground Bearing Springs** option.
Create a mat supported by a combination of ground spring and piles	Ensure that the **Use Ground Bearing Springs** option is selected.

Place an individual pile in a mat

1. On the **Foundations** tab, click  **Pile**.
2. In the **Properties** window, specify the Pile type:
   a. either by selecting an existing type (if a suitable type has already defined in the Pile catalogue (page 1571)),
   b. or by selecting <New>... (to add a new pile (page 1572) to the Pile type catalogue)
3. Click the mat in which you want to place the pile.
4. Click the pile position in the selected mat.

   Tekla Structural Designer places the pile in the selected position.

Place a pile array in a mat

1. On the **Foundations** tab, click  **Pile Array**.
2. In the **Properties** window, specify the Pile type:
   a. either by selecting an existing type (if a suitable type has already defined in the Pile catalogue (page 1571)),
   b. or by selecting <New>... (to add a new pile (page 1572) to the Pile type catalogue)
3. In the **Properties** window, adjust spacing and pile direction according to your needs.

   TIP To preview the array, hover the mouse pointer over a mat.

4. Click an individual mat, or hold down the left mouse button and drag a box around multiple mats.

5. If necessary, add further mats to the selection by clicking or boxing them.
6. To create the mat, click the last mat you selected again.

   Tekla Structural Designer places the piles in the mat.
7. Repeat steps 2–4 to place more piles or press **Esc** to finish placing piles.

Specify the pile direction of an inclined pile

1. On the **Foundations** tab, click  **Pile**.
2. In the **Properties** window, do one of the following:

To	Do this

Specify the direction as a vector	1. Adjust the X and Y components of the vector.
	
	TIP If you do not want the pile to slope in the global X or Y direction, leave the appropriate value as 0.
	
	2. Type a negative value in the Z component field.
	
	For example:
	
	To create an inclined pile of 45 degrees in positive X, define the components as follows:
	
	`X = 1.0, Y = 0.0, Z = -1.0.`
	
	To create an inclined pile of 45 degrees in negative Y, define the components as follows:
	
	`X = 0.0, Y = 0.5, Z = -1.0`

Specify the direction as an angular measurement	**NOTE** You can only specify the direction as an angular measurement before the pile is first positioned. 1. Select the **Direction by Angles** option. 2. Define the inclination of the pile from the vertical.

To	Do this

	3. Define the azimuth (the horizontal angle measured clocwise about the positive global Z direction.
	4. If necessary, define the rotation about the longitudinal axis of the pile.
	NOTE Since the pile is only checked for axial capacity, the rotation angle has no effect on the pile checks. However, you can define the rotation to vary how Tekla Structural Designer displays the moment and shear force values of the pile.
	For example:
	To create an inclined pile of 45 degrees in positive X, define the angles as follows:
	`inclination = 45 degrees, azimuth = 90 degrees`
	To create an inclined pile of 30 degrees in negative Y, define the angles as follows:
	`inclination = 30 degrees, azimuth = 0.0 degrees`

2.5 Edit the model

After creating the model and the necessary members within it, you may need to make some modifications.

For example you may want to:

- Edit the properties of (page 111) entities

- Re-position entities (page 112) by moving nodes or edges

Additional model editing commands are located on the **Edit** tab, these allow you to:

- copy objects (page 271) and loads
- move objects (page 271) or move the model
- mirror objects (page 272)
- delete objects (page 277)
- join and split members
- Automatically join all concrete beams (page 279)
- Reverse member axes and panel faces (page 280)
- use cutting planes to hide a part of your model
- remove any unused unused slopes, frames, construction and grid lines
- merge planes (page 300)
- Rationalize the model (page 298)
- create infill members
- edit or add free points
- Planarize (page 302)

Copy and rotate objects

To copy, or copy and rotate objects, see the following instructions.

1. Select the objects that you want to copy.
2. On the **Edit** tab, click **Copy**.
3. If you want to rotate the object when you copy it, in the **Properties** window, type the rotation about the Z axis.
4. Click the reference node.
5. Click the point where you want to copy the reference node.

   Tekla Structural Designer moves the selected member to the new location, and rotates them according to the specified rotation value.
6. Place more copies in your model, or press **Esc** to finish.

Move and rotate objects

To move, or move and rotate objects, see the following instructions.

1. Select the objects that you want to move.

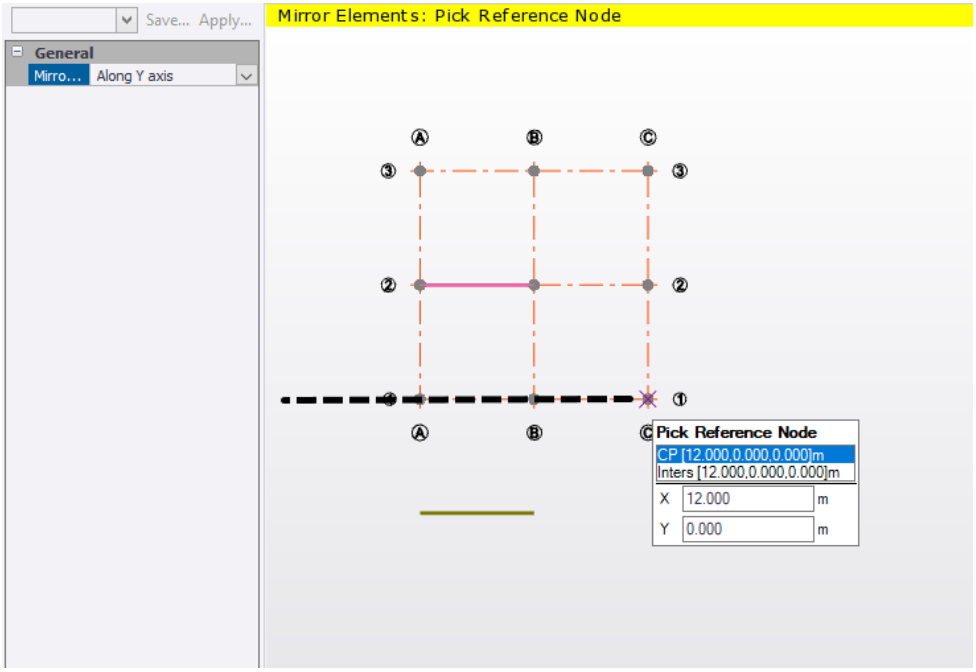
2. On the **Edit** tab, click **Move**.
3. If you want to rotate the object while moving it, in the **Properties** window, type the rotation about the Z axis.
4. Click the reference node.
5. Click the point where you want to move the reference node.

   Tekla Structural Designer moves the selected member to the new location, and rotates them according to the specified rotation value.

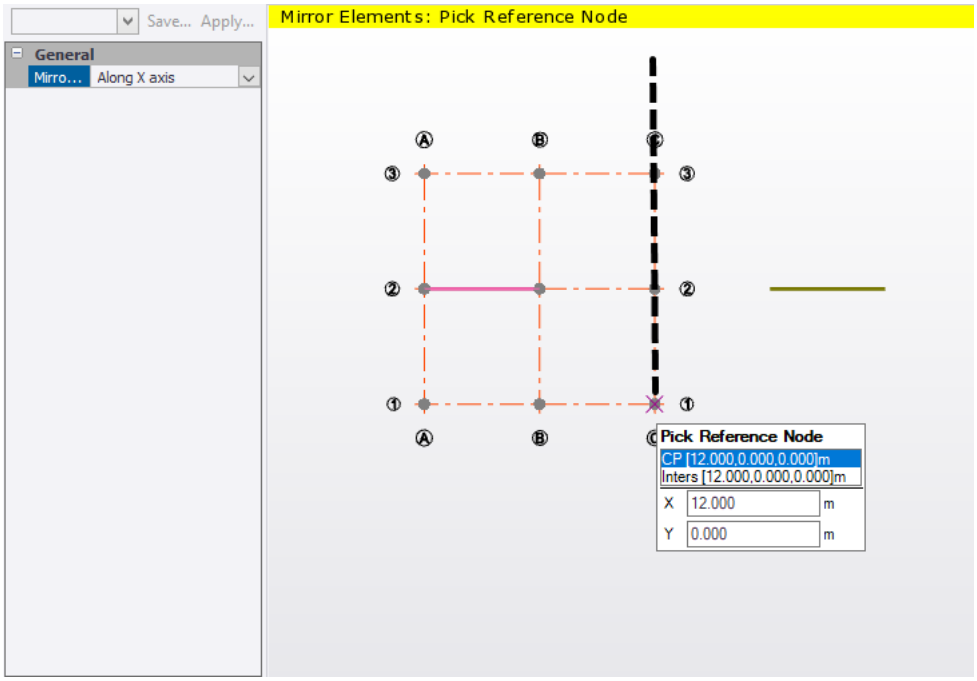
Mirror objects to new locations

To copy existing elements by mirroring them, see the following instructions.

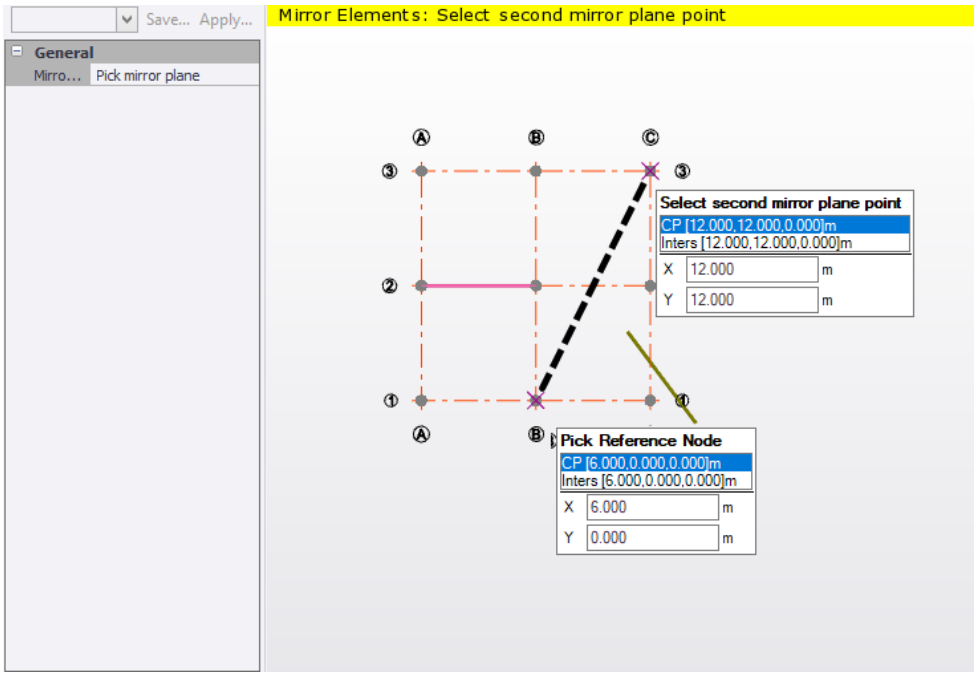
1. Select the objects that you want to mirror.
2. On the **Edit** tab, click **Mirror**.
3. In the **Properties** window, select the desired **Mirror mode**:
   - **Along Y axis** mirrors the objects about a global XZ plane defined by a single reference point.



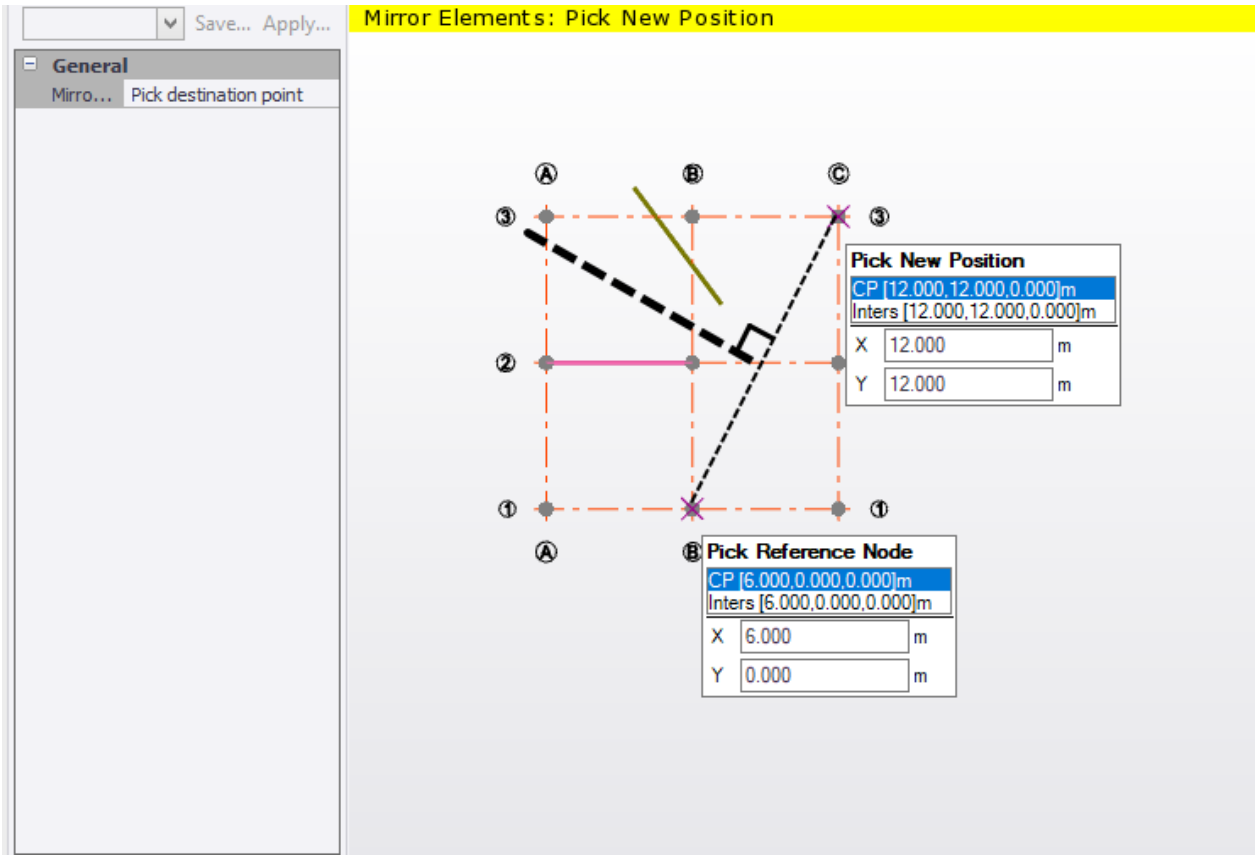
- **Along X axis** mirrors the objects about a global YZ plane defined by a single reference point.



- **Pick mirror plane** mirrors the objects about a plane defined between two points.



- **Pick destination point** mirrors the objects about a plane which perpendicularly bisects a line defined between two points.



4. In the model, click the reference points.
5. If you are using the **Pick mirror plane** or **Pick destination point** mode, click the second point to define the mirror plane.

See also

Copy and rotate objects (page 271)

Copy loads

Tekla Structural Designer allows you to quickly apply the same loads to multiple members by using the **Copy Loads** command.

Copy all member loads from one span to another

1. On the **Edit** tab, click **Copy Loads**.
2. Go to the **Properties** window.

3. Set **Mode** to **Copy Span Loading**.
4. Select whether you want to copy loads in the current loadcase or in all loadcases.

   NOTE If you select the current loadcase, remember to select the required loadcase in the .

5. In the model, click the span that contains the member loads that you want to copy.
6. Click the span that you want to apply the loads to. The source span and the destination span do not need to be the same length.

   Tekla Structural Designer copies the member loads to the selected span.
7. Do one of the following:
   - Click additional spans to continue applying the loads.
   - Press **Esc** to select a different loaded element span to copy loads from.
   - Press **Esc** twice to finish.

Only copy one member load to another span

1. On the **Edit** tab, click **Copy Loads**.
2. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
3. In the model, click the member load that you want to copy.
4. Click the span that you want to apply the load to. The source span and the destination span do not need to be the same length.

   Tekla Structural Designer copies the member load to the selected span.
5. Do one of the following:
   - Click additional spans to continue applying the loads.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy panel area, level, and slab loads

1. On the **Edit** tab, click **Copy Loads**.
2. In the , select the desired loadcase.
3. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
4. Click the area, slab, or level load that you want to copy.
5. Click the panel that you want to apply the load to.

   Tekla Structural Designer copies the load to the selected panel.

6. Do one of the following:
   - Click additional panels to continue applying the loads.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy panel point, line, and patch loads

1. On the **Edit** tab, click **Copy Loads**.
2. In the , select the desired loadcase.
3. In the **Properties** window, set **Mode** to **Copy Plane Loads**.
4. Click the panel point, line, or patch load that you want to copy.

   TIP If necessary, you can click to add other loads to the selection, or remove them from the selection by clicking them once more.

   A red circle indicates the original reference point for the selected loads.
5. To apply the loads, do one of the following:
   - Click a panel node to define a new reference point at that node.

     The loads are applied at the same offset from the new reference point.
   - Click anywhere within a panel boundary at a different level to define a new reference point directly above or below the original reference point.

     The loads are applied at the same offset from the new reference point.

     WARNING If you choose a new reference point that results in the loads being applied outside the panel area, Tekla Structural Designer does not apply the loads to the model. In this situation, a warning appears during validation.
6. Do one of the following:
   - Click to define further reference points to continue applying the loads. .
   - Press **Esc** to select a different loads to copy.
   - Press **Esc** twice to finish.

Copy structure loads

1. On the **Edit** tab, click **Copy Loads**.
2. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.

3. In the model, click the structure load to be copied.
4. Click the position you want to apply the load to.

   Tekla Structural Designer copies the load to the new location.
5. Do one of the following:
   - Click additional locations to continue applying the load.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy loads to another loadcase

1. On the **Edit** tab, click **Copy Loads**.
2. In the , select the loadcase from which you want to copy the loads.
3. Go to the **Properties** window.
4. Set **Mode** to **Copy Loads to another Loadcase**.
5. Select the loadcase that you want to copy the loads to.
6. In the model, click the load that you want to copy.

   TIP If necessary, you can click to add other loads to the selection, or remove them from the selection by clicking them once more.
7. To copy the loads to the loadcase displayed in the **Properties** window, press **Enter**.
8. Do one of the following:
   - Press **Esc** to select additional loads, and then press **Enter** to copy them.
   - Press **Esc** twice to finish.

Delete entities

1. Select the entities that you want to delete. For selection methods, see: Select entities (page 100).
2. On the **Edit** tab, click **Delete**

   NOTE **Delete** can also be accessed from the Quick Access toolbar and some of the context menus. You can also press **Delete** on the keyboard.

   If a single entity was selected it is deleted, otherwise a dialog allows you to filter the selection before the deleting.

3. Uncheck any items in the dialog that you don't want to delete.
4. Click **OK** The remaining checked items in the dialog are deleted.

NOTE Items cannot be deleted if they are being used to reference other items. For example grid and construction lines cannot be deleted until the members etc. that have been placed using them have also been deleted.

Join and split members

You can use the **Join** command for joining discontinuous members, thus creating a continuous member. The **Split** command, on the other hand, allows you to split continuous members of any material.

See also

Automatically join all concrete beams (page 279)

Join members

You can use the **Join** command to manually join concrete beams, even when the **Allow automatic join end** options are not selected. This is because the **Allow automatic join end** only applies to the automatic joining that occurs during design process or when you run the **Beam Lines** command.

RESTRICTION You can only join:

- Two similar members if that the angle between the members is less than 45 degrees in both plan and elevation.
- Beams that have an end point in common.

  If Tekla Structural Designer fails to join two beams, their end points have probably been defined using different construction or grid lines.

1. On the **Edit** tab, click **Join**.
2. Hover the mouse pointer over the member that you want to join to another member.

Both the original member and the member to which it will be joined become highlighted. The point where they will be joined is indicated by a red dot.

NOTE If Tekla Structural Designer tries to join the wrong end, move the mouse pointer towards the other end of the member until Tekla Structural Designer indicates the correct end.

3. Click the highlighted members to join them.
4. Click additional members to join them, or press **Esc** to finish.

Split members

RESTRICTION You can only split members that have previously been joined.

1. On the **Edit** tab, click **Split**.
2. Hover the mouse pointer over previously joined members that you want to split.

   The member is highlighted, and Tekla Structural Designer uses a red dot to indicate the point where the member will split.
3. According to your needs, do one of the following:
   - Click the member to split it at the indicated point.
   - Move the mouse pointer further along the member to identify other points where the member could be split.

NOTE If the member being split is a concrete beam, Tekla Structural Designer splits the member into two separate beams. The first beam has the **Allow automatic join end 2** option cleared, whereas the second one has the **Allow automatic join end 2** option cleared. This prevents Tekla Structural Designer from automatically making the two beams continuous again when the model is designed.

Automatically join all concrete beams

Tekla Structural Designer automatically forms concrete beam lines as part of the combined analysis and design process. However, if you would prefer to have greater control yourself, you can run the **Beam Lines** command manually. This way, you can verify that continuous beam lines are formed as you intend before proceeding with the design.

When run manually, the **Beam Lines** command applies to all concrete beam members in the model, irrespective of whether they are selected or not.

NOTE Discontinuous concrete beam ends can only be joined if the **Allow automatic join end** option is selected for the appropriate beam ends at the join. Then, Tekla Structural Designer only joins the ends if all the following criteria are met:

- The angle in plan at which the two beams meet is less than the **Limiting join angle in plan** specified in **Model Settings** --> **Beam Lines**
- The angle in elevation at which the two beams meet is less than the **Limiting join angle in elevation** specified in **Model Settings** --> **Beam Lines**
- The amount by which the cross sectional areas of the two beams overlap is greater than the **Minimum section overlap** percentage specified in **Model Settings** --> **Beam Lines**

In addition, if either of the two beam ends being joined is pinned, Tekla Structural Designer does not join them unless the **Join pinned beam end** option is selected in **Model Settings** --> **Beam Lines**.

To join the concrete beams in the model automatically, do the following:

- On the **Edit** tab, click  **Beam Lines**.

See also

Join and split members (page 278)

Reverse member axes and panel faces

Sometimes when creating a model, you may end up creating beams or wall panels that run or face in the wrong direction. This may lead to problems in the analysis phase. In order to fix these kinds of inconsistencies, you can use the **Reverse** command.

Reverse the local axis of a beam

You can easily end up with beams running forwards and backwards if a consistent approach has not been adopted when selecting start and end nodes. Beams that run in different directions can result in confusing force diagrams. By manually reversing the axes of beams, you can make all beams run left to right and bottom to top in a plan view.

To reverse a beam, do the following:

1. On the **Edit** tab, click **Reverse**.
2. Click a beam to reverse its direction.

Reverse the outward face of a wind panel

The front of each wall should be facing outwards in order to correctly determine the wind direction relative to the wall. Ensure that all the outward faces are displayed in the color assigned to the front of the wind wall.

If a wall is facing in the wrong direction, reverse it by doing the following:

1. On the **Edit** tab, click **Reverse**.
2. Click a wind panel to reverse its direction.

Create cutting planes

Cutting planes enable you to focus on a specific area of the model.

Everything on the positive side of the plane is hidden from the view, making it easier to work on remaining visible area.

Related information

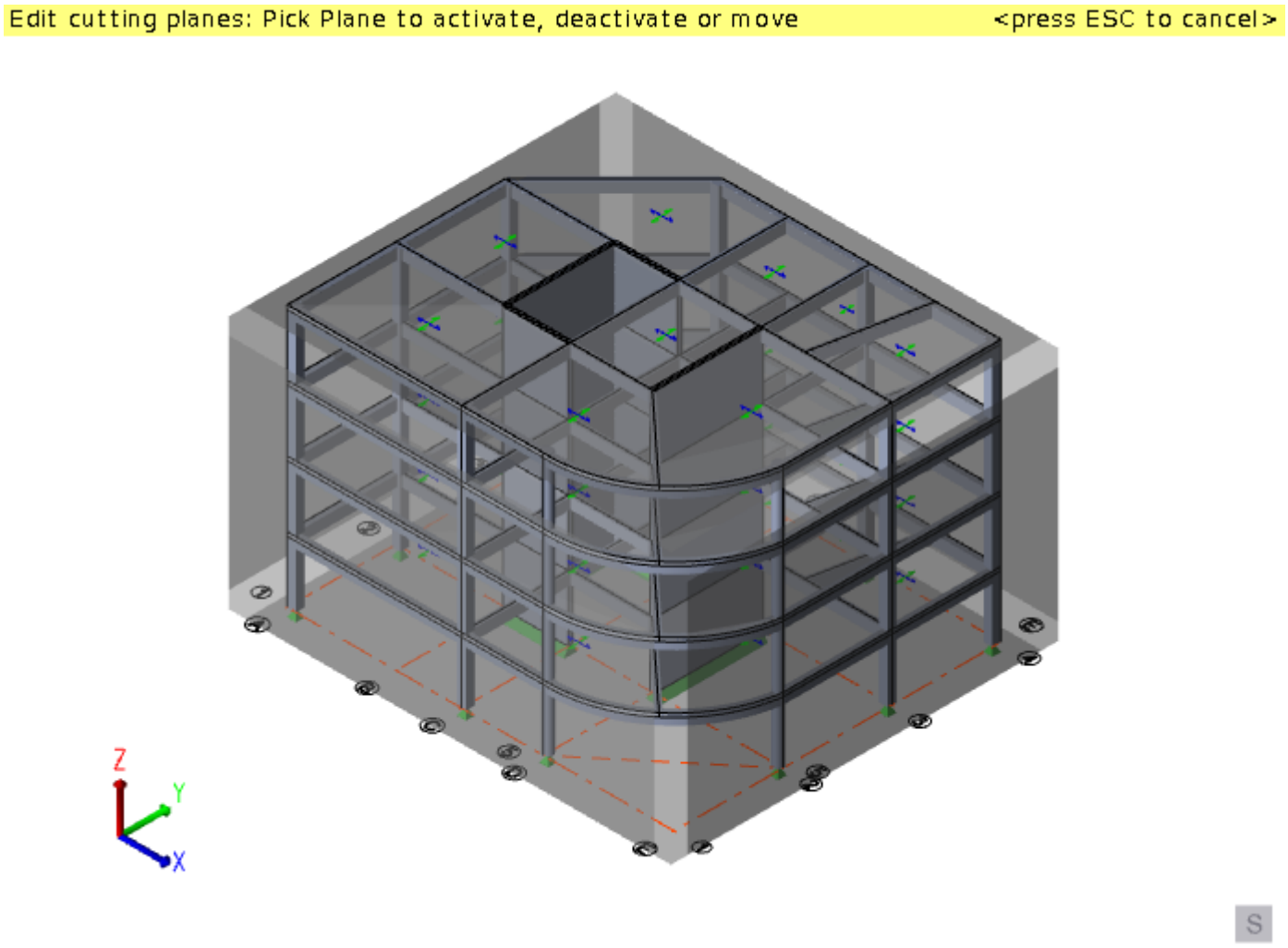
Related video:

How to use the Cutting Plane command?

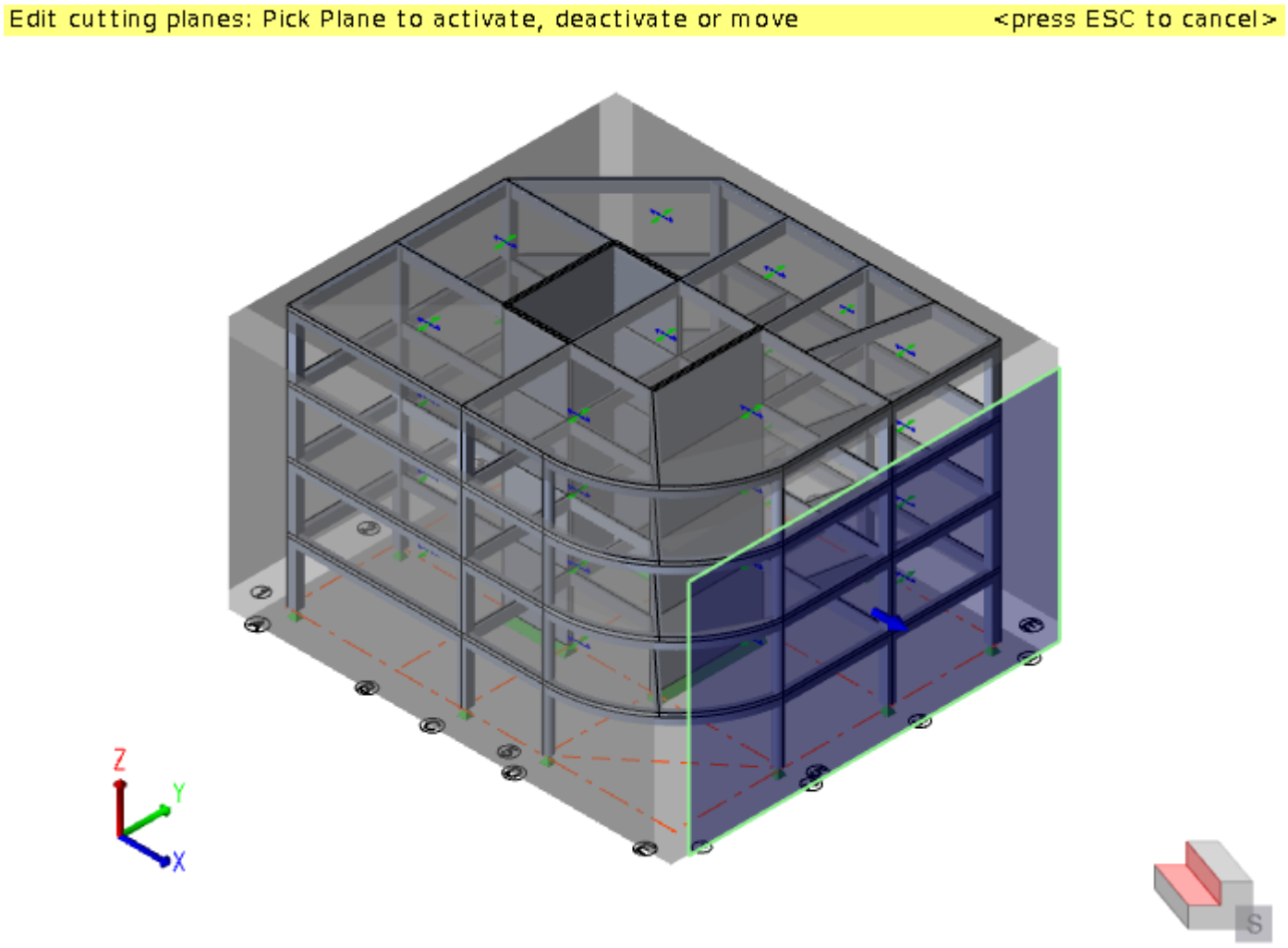
Position a cutting plane

1. On the **Edit** tab, click **Cutting Planes**.

   Tekla Structural Designer displays six cutting planes, initially forming a cube around the extents of the model.



2. Click a cutting plane to activate it.



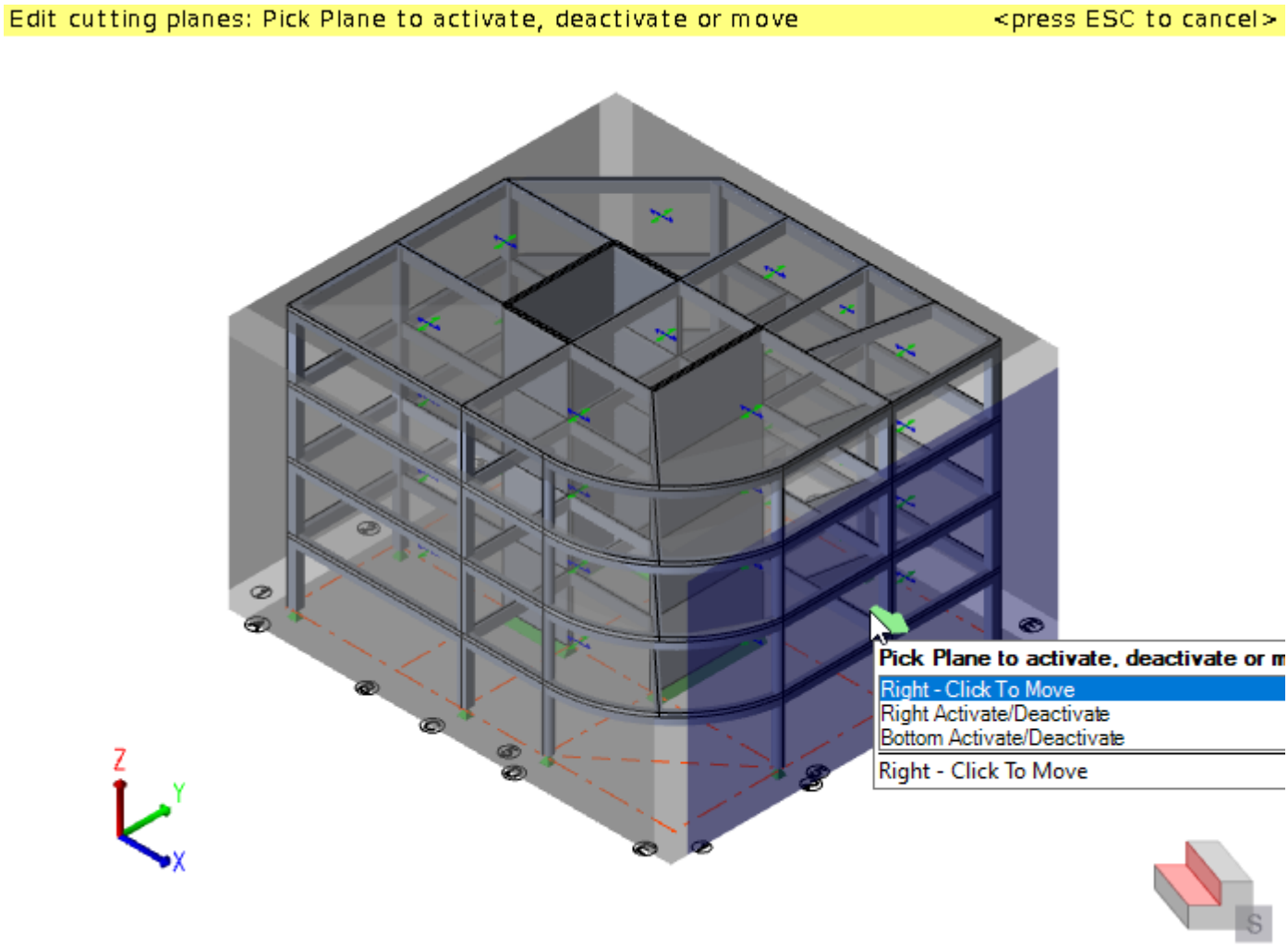
When a cutting plane is active a  symbol is displayed in the corner of the view.

The active cutting plane faces are shown in a different color. (By default blue indicates the positive side of the plane and red the reverse side).

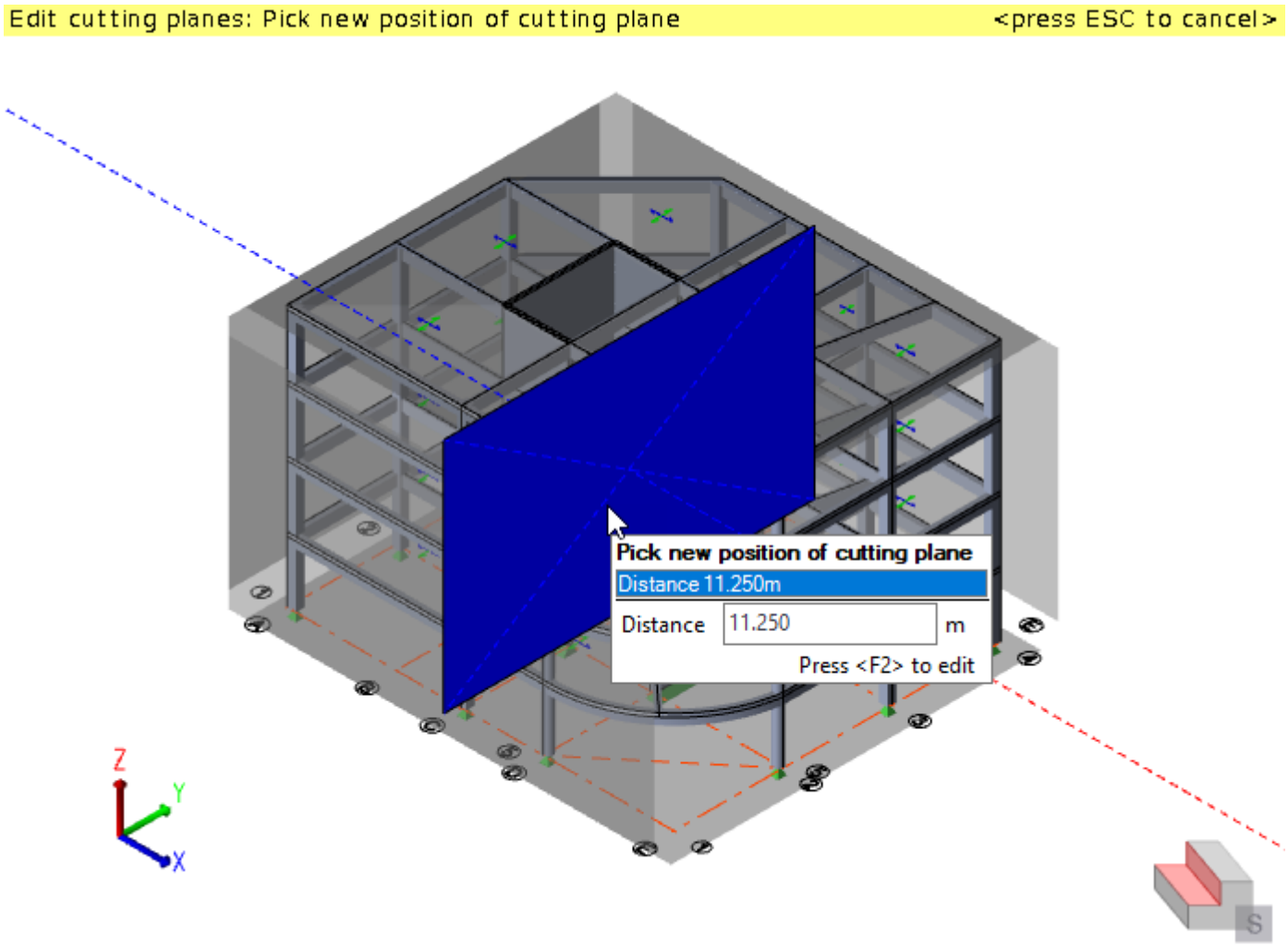
Anything on the positive side of the plane will be hidden from the view.

An arrow projects from the center of the cutting plane which can be used to reposition the plane.

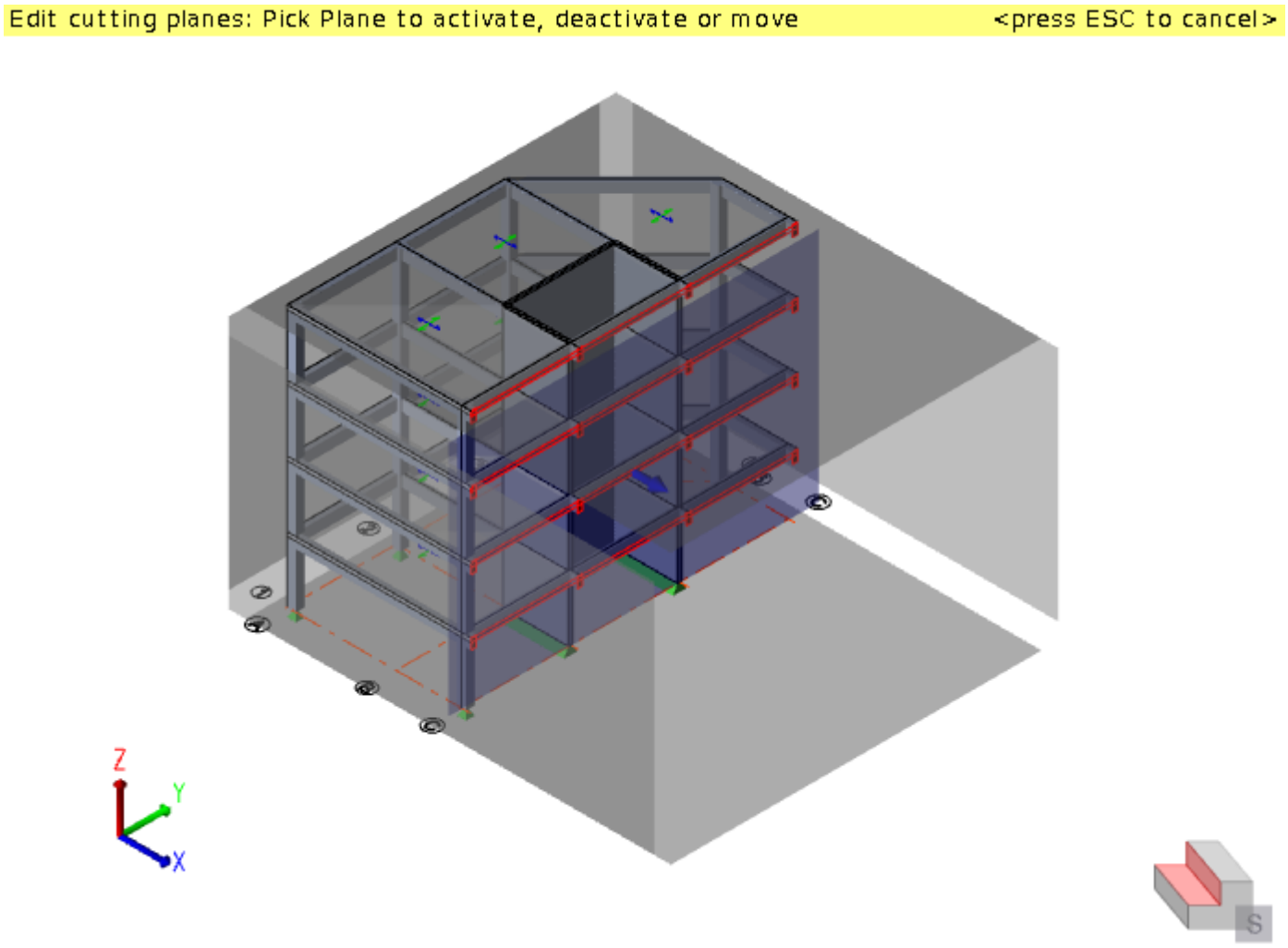
3. Click the arrow at the center of the cutting plane.



A dashed line indicates the direction in which you can move the plane.

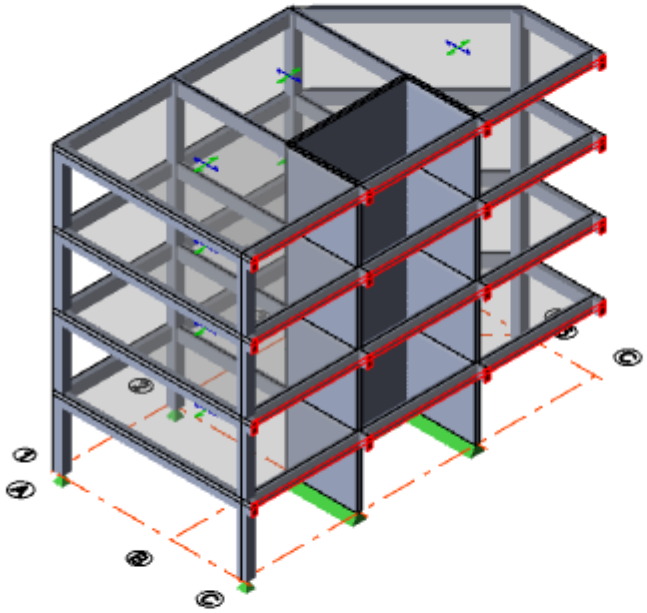


4. Click the desired position of the plane, or press **F2** to type the exact distance. Everything to the positive side of the plane is hidden.



At this point you can activate and move further cutting planes if required, or press ESC to end the command.

5. Press ESC



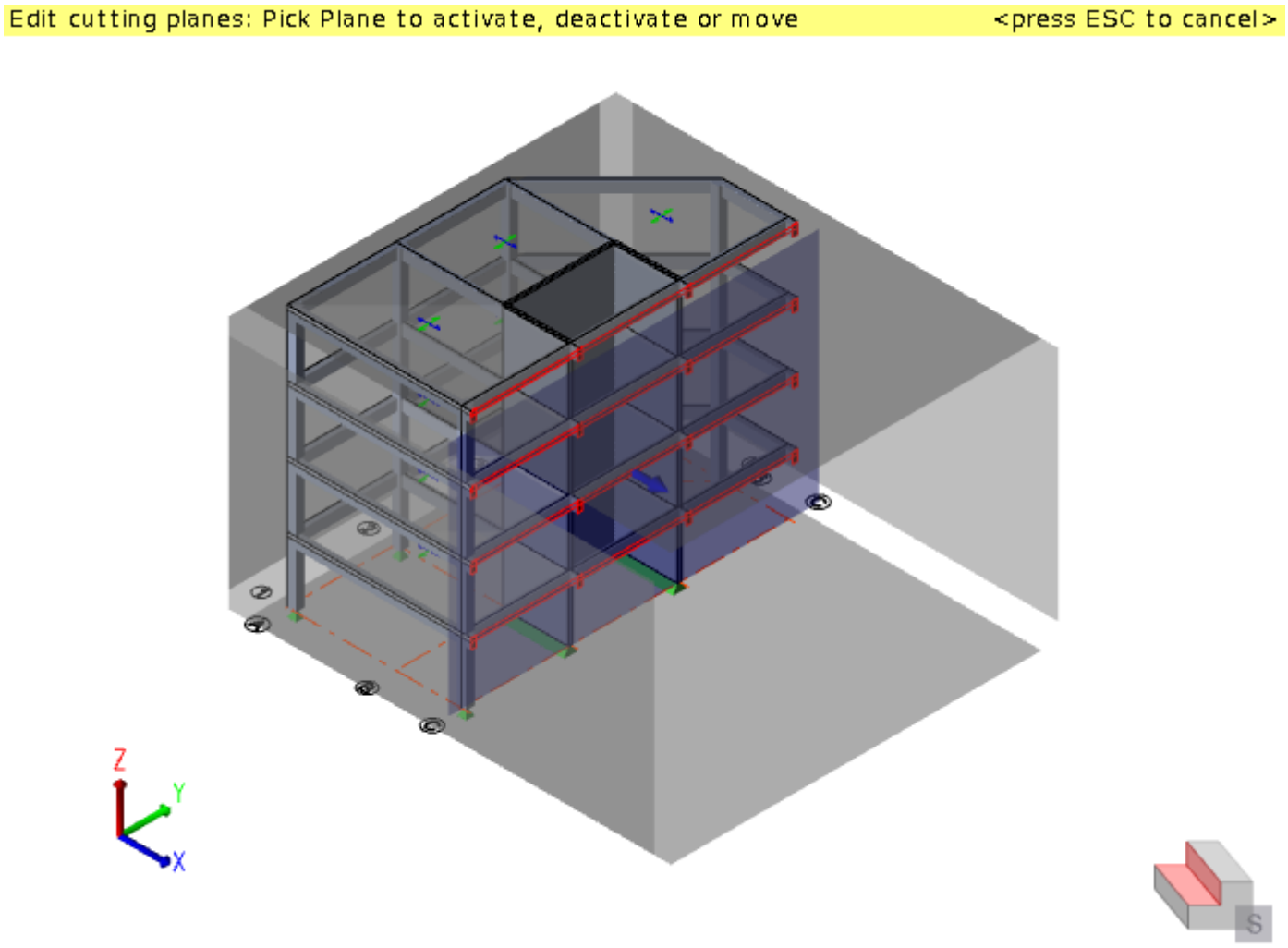


The six cutting planes are hidden, but the cutting planes icon remains in the bottom corner of the view to indicate that a plane is still activated.

Deactivate a cutting plane

The  symbol in the corner of the view indicates a cutting plane is active.

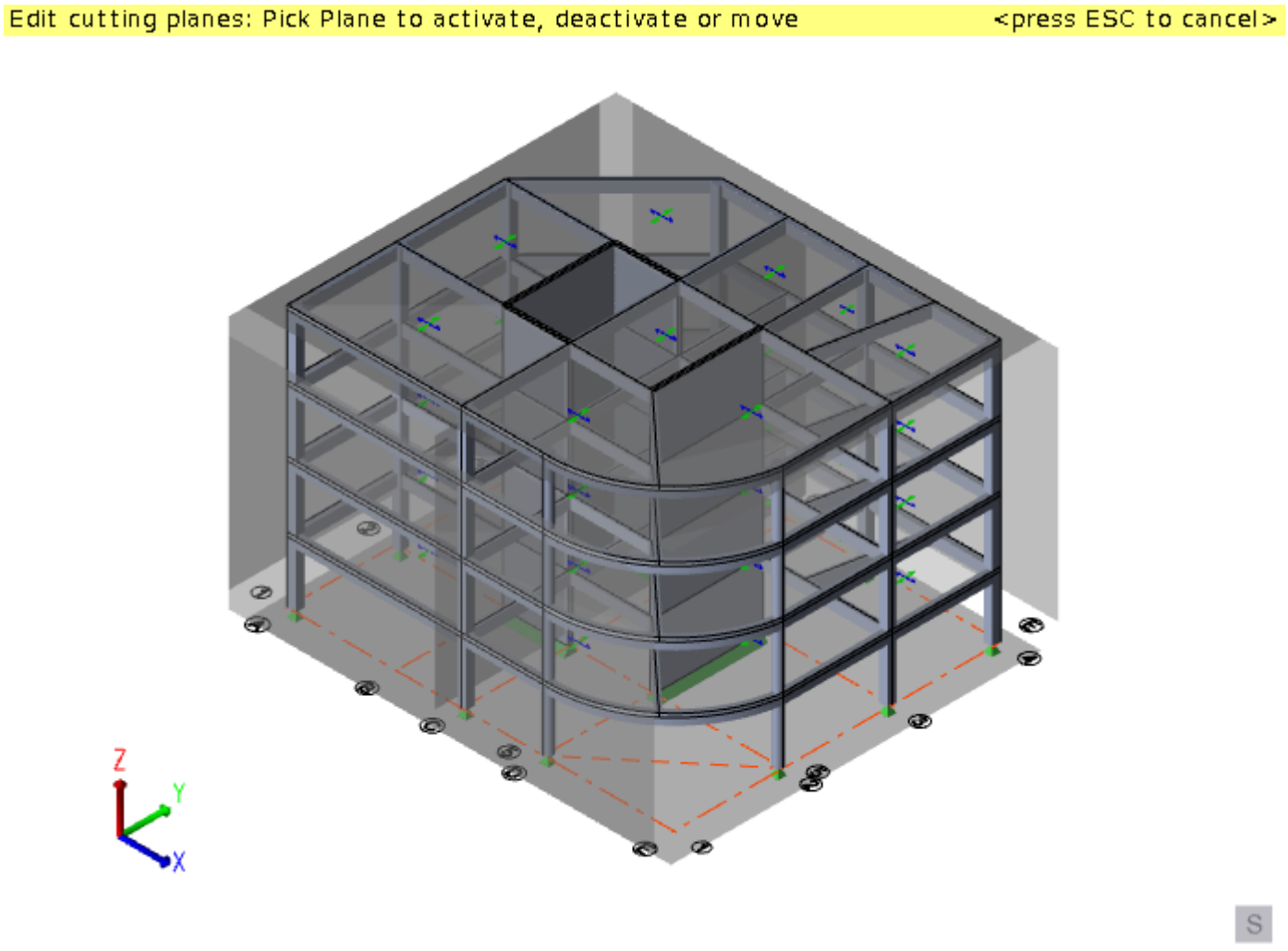
1. If cutting planes are not visible, click **Cutting Planes** on the **Edit** tab to display them.



Tekla Structural Designer displays the cutting planes. Grey planes are deactivated, colored planes are activated.

2. Click an active cutting plane to deactivate it.

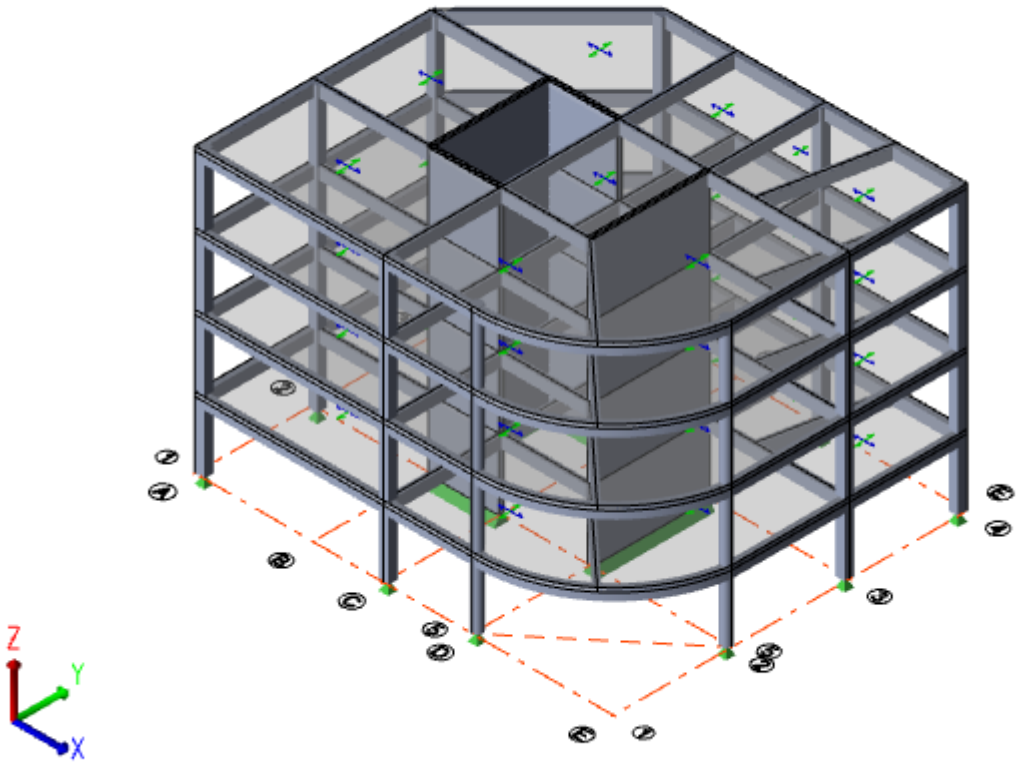
TIP If more than one cutting plane is listed in the tooltip, use the up or down cursor key to ensure the required one has been highlighted before picking it.



The plane returns to its original gray color and everything on its positive side is redisplayed.

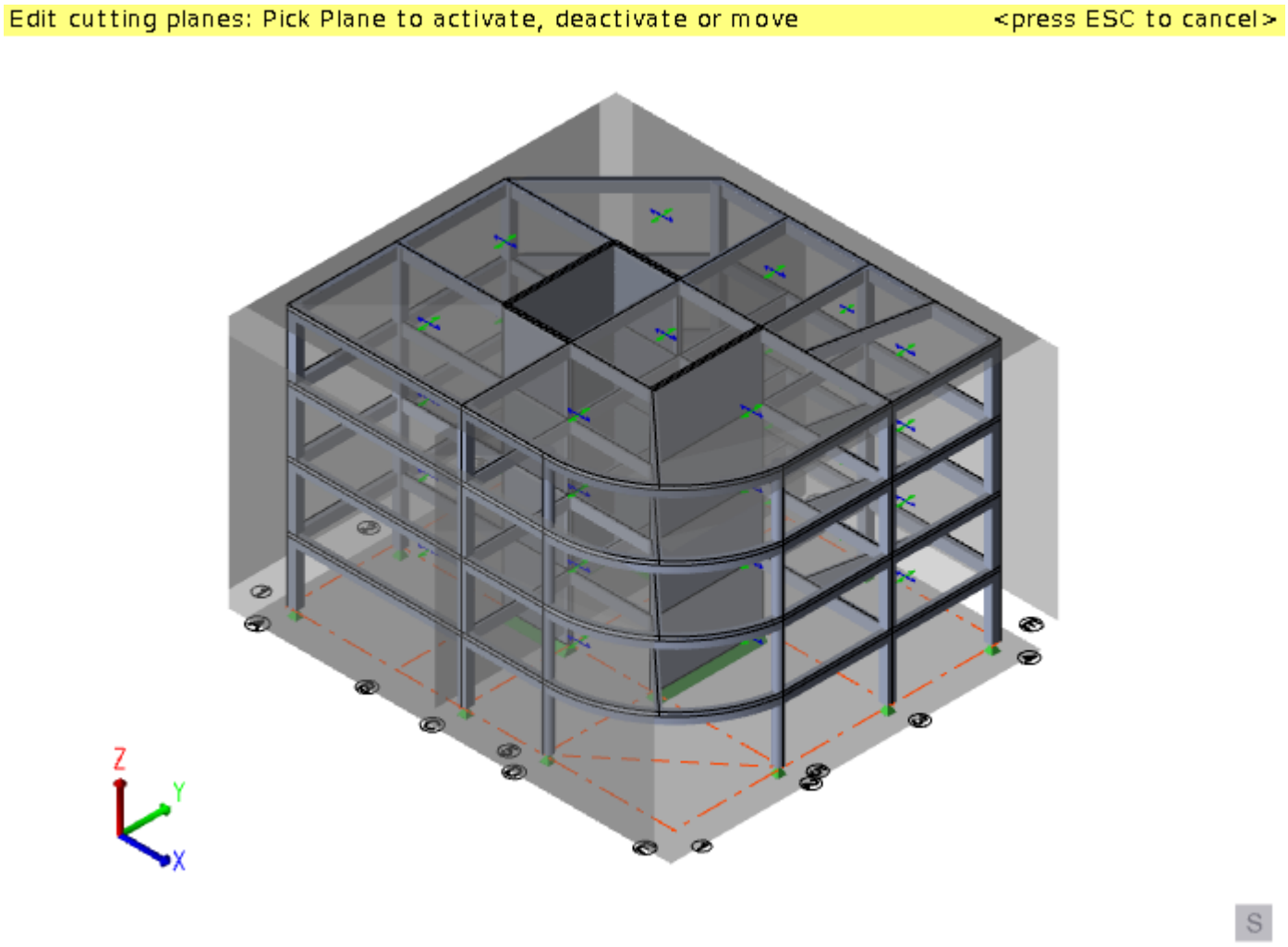
When all cutting planes have been deactivated the  symbol is removed.

3. Press ESC to end the command.



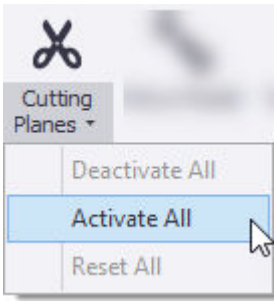
The cutting planes are hidden.

Activate and deactivate all cutting planes



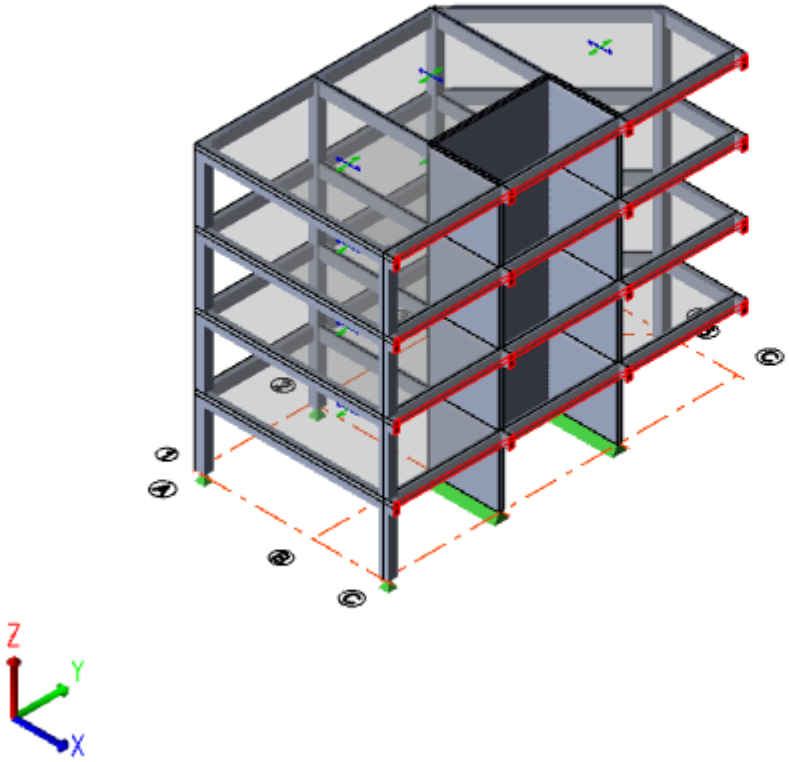
Once cutting planes have been positioned as required, you can quickly toggle the display to switch between having them activated or deactivated.

1. Click the **Cutting Planes** droplist on the ribbon and select **Activate All**



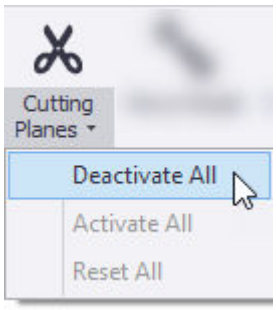
The cutting planes are activated.

2. Press ESC to end the command.

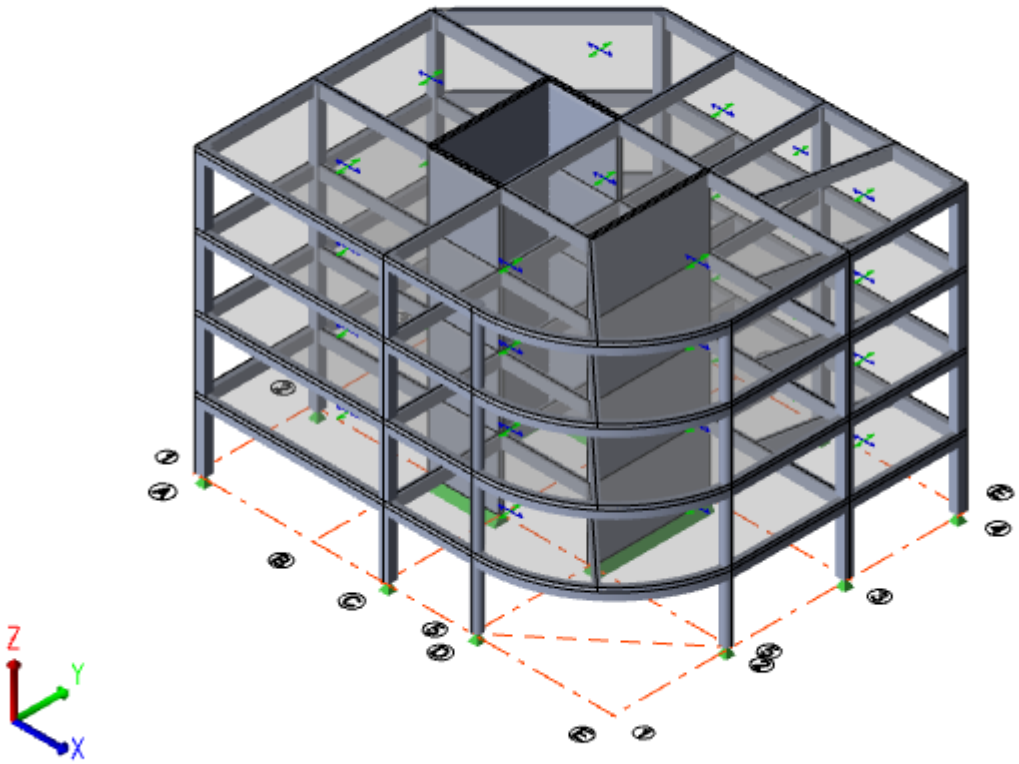




3. Click the **Cutting Planes** droplist on the ribbon and select **Deactivate All**



All six cutting planes are deactivated.

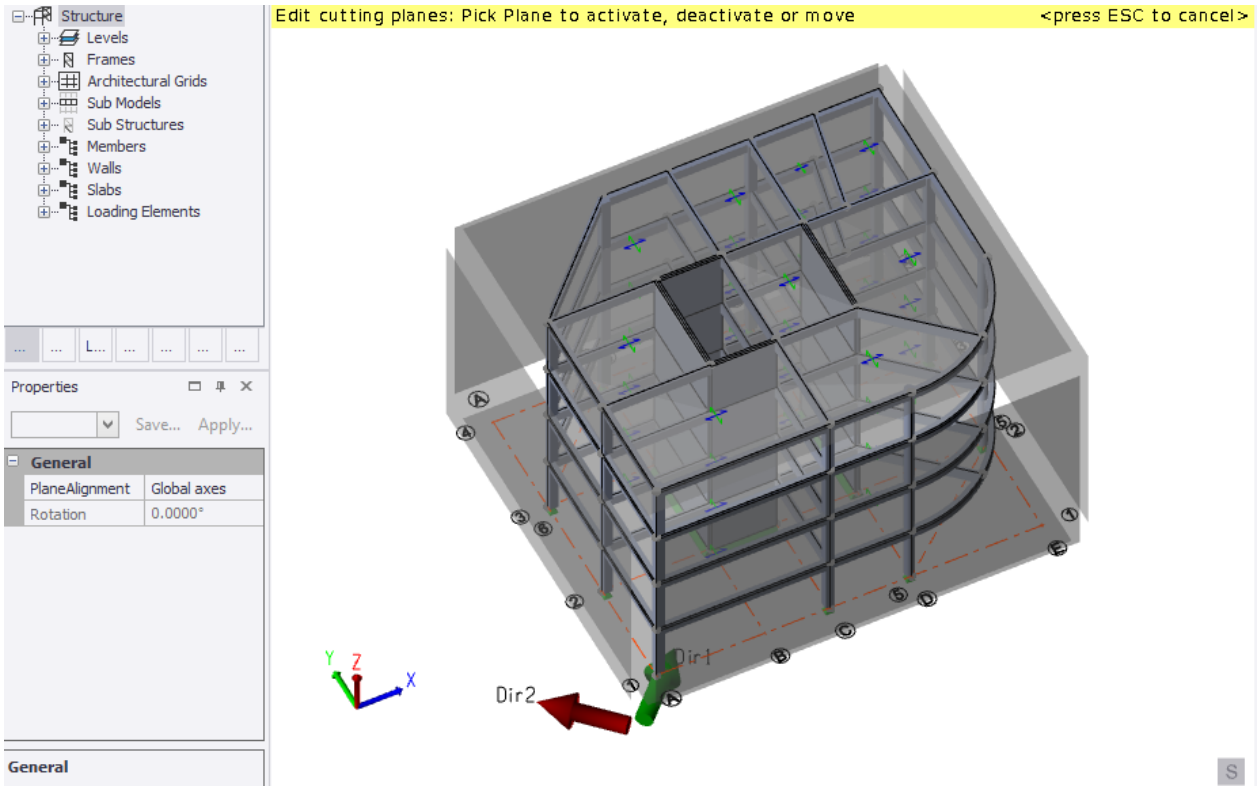


Align all cutting planes

Use the **Properties** window to specify how the cutting planes are to be aligned on plan.

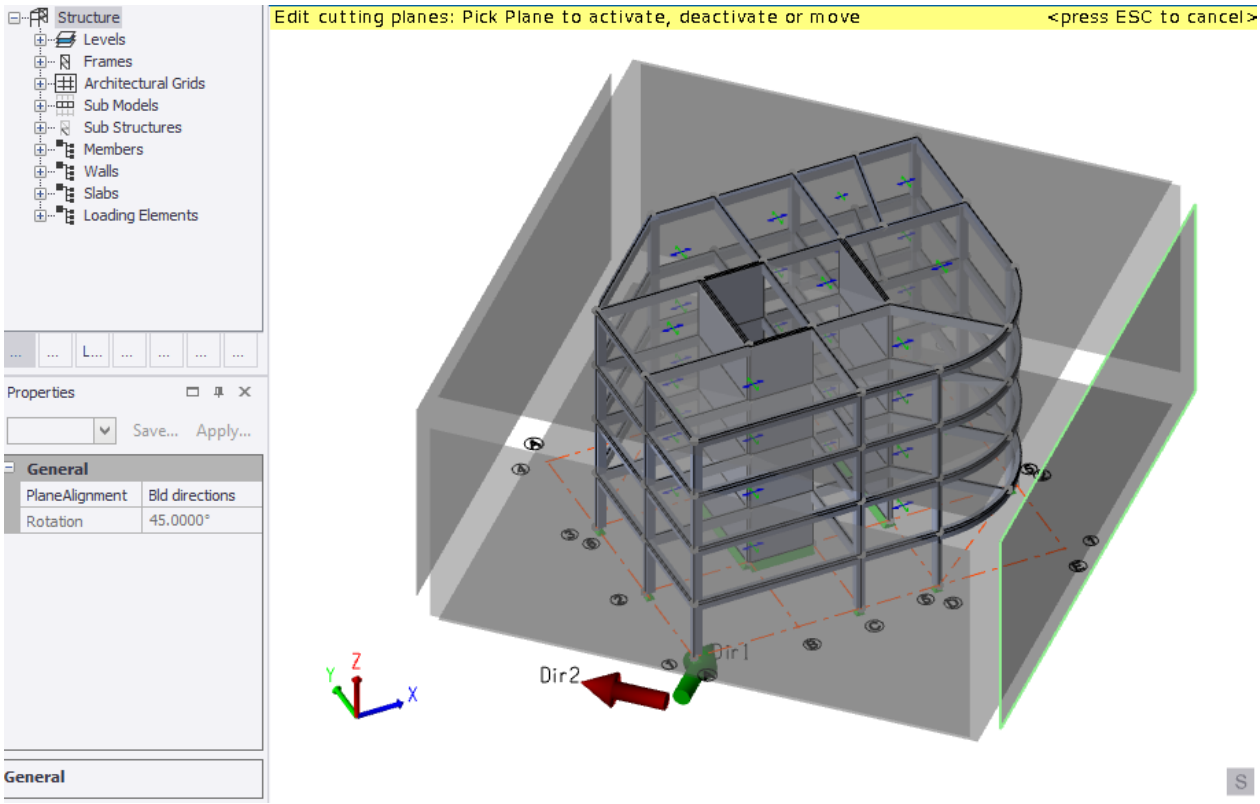
Cutting planes can be aligned on plan to the global axes, the building directions, or a custom angle.

1. In the **Properties** window, set **PlaneAlignment** to 'Global axes'.



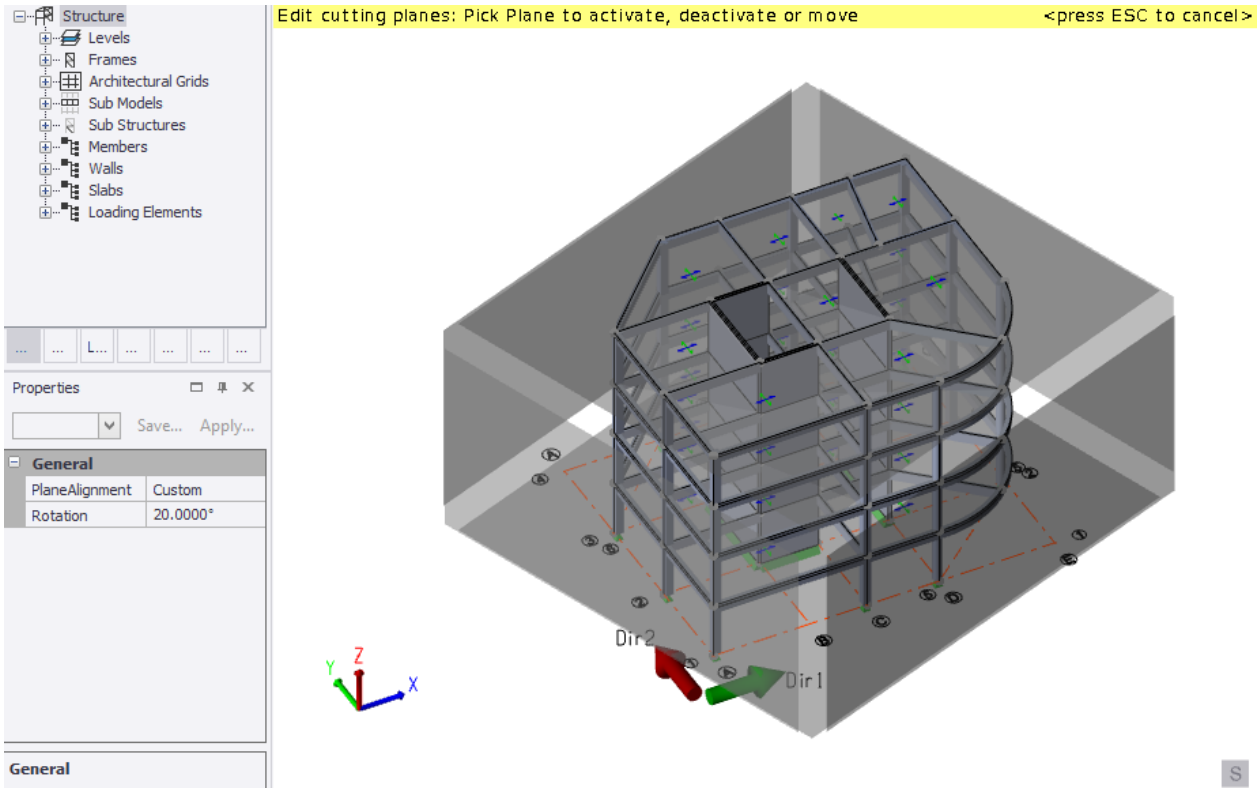
The cutting planes align to the global X and Y axes.

2. In the **Properties** window, set **PlaneAlignment** to 'Bld directions'.



The cutting planes align to the Dir 1 and Dir 2 axes.

3. In the **Properties** window, set **PlaneAlignment** to 'Custom' and enter the **Rotation** required.

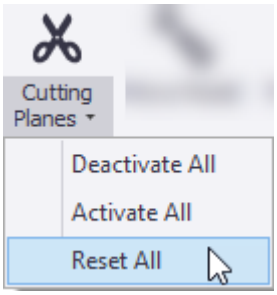


The cutting planes align to rotation angle specified.

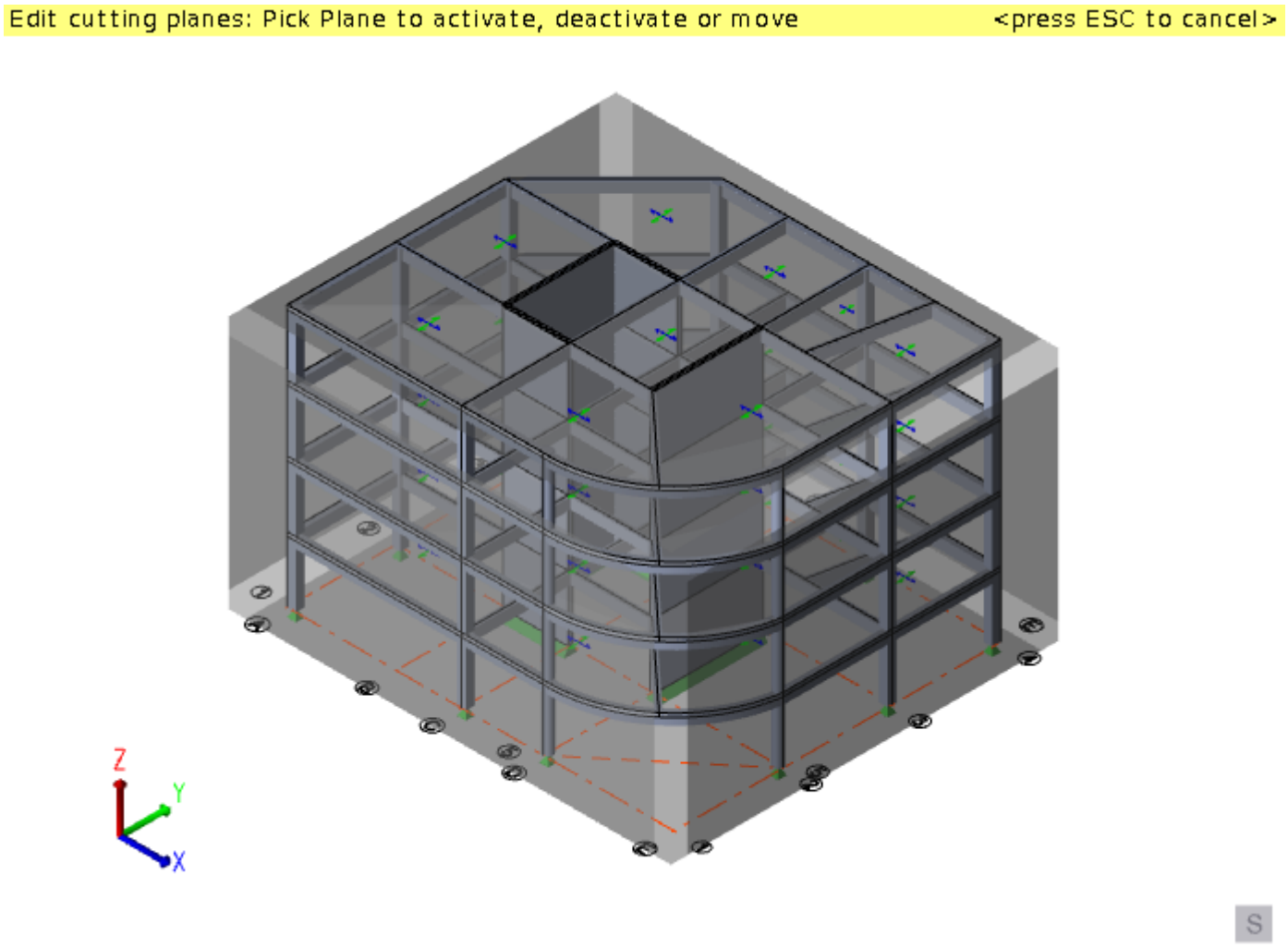
Reset all cutting planes

After having positioned cutting planes, if you want to return them all to the default starting location, they can be reset.

1. Click the **Cutting Planes** droplist on the ribbon and select **Reset All**



All the cutting planes are reset to their original positions.



2. Press ESC to end the command.

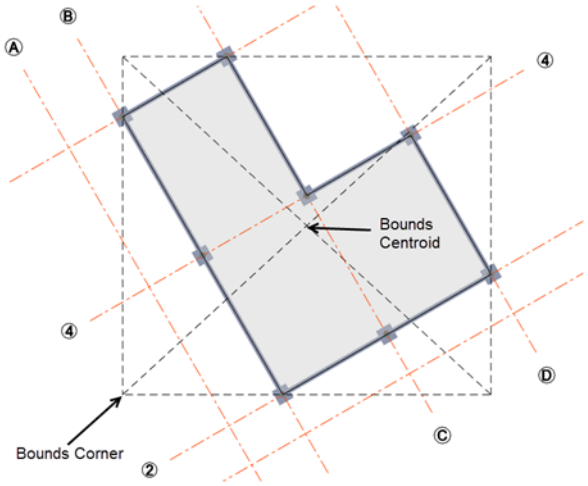
Move the model or the DXF shadow

If necessary, you can relocate the entire model to a new coordinate in the XY plane by using the **Move Model**. You can relocate the DXF shadow of the model similarly with the **Move DXF Shadow** command.

Move the model

In order to move the model, you first have to decide which reference point to use.

You can determine the reference points by drawing an imaginary box aligned to X and Y around the extents of the model, as shown in the following image:



Bounds Corner equals the lower left hand corner of the imaginary box, whereas **Bounds Centroid** equals the centroid of the imaginary box.

1. On the **Edit** tab, click **Move Model**.
2. Set the reference point to either **Bounds Corner** or **Bounds Centroid**.
3. Enter the required target coordinate of the reference point.
4. Click **Move**.

Move the DXF shadow

The **Move DXF Shadow** command is only available if you have previously imported a DXF shadow.

1. On the **Edit** tab, click **Move DXF Shadow**.
2. Select an existing point in the DXF shadow as the reference point.
3. Click the new position of the reference point.

Rationalize the model

In order to remove all unused sloped planes, frames, grids, and construction lines in your model, you can use the **Rationalize** command. If necessary, the **Rationalize** command also allows you to update all grid and construction lines to extend only a short length beyond the point where they are required. After running the command, you can see a summary of the changes in the **Process Window**.

Delete unused sloped planes, frames, grids, and construction lines

1. On the **Edit** tab, click **Rationalize**.

   The **Rationalize** dialog box opens.

2. Select the unused items that you want to remove.
3. Click **OK**.

   Tekla Structural Designer deletes the unused items of the selected types, and their associated items.

Update grid and construction line length

1. On the **Edit** tab, click **Rationalize**.

   The **Rationalize** dialog box opens.
2. Select **Shrink/Extend Grid & Construction lines**.
3. Click **OK**.

   Tekla Structural Designer updates all grid and construction lines in the model to extend a fixed length (500 mm / 1'8") beyond the point where they are required.

Create infill members

In order to quickly place a pattern of infill members into selected plays in a level, sloped plane, or frame, you can use the **Create Infills** command.

NOTE Infills can only be created within four sided enclosed bays with members or meshed/mid-pier shear walls attached to all sides. All suitable bays are shaded green when **Create Infills** is selected.

RESTRICTION You cannot create infills in 3D views.

Define the infill properties and pattern

1. Open a 2D scene view.
2. On the **Edit** tab, click  **Create Infills**.
3. Go to the **Properties** window.
4. In **Define Beams**, select one of the following:
   - **By number** allows you to specify the number of equally spaced members in the pattern.
   - **By spacing** allows you to specify the exact positions of the beams, separating them by a comma.
   - **By max spacing** allows you to create as many equally spaced beams in the pattern as possible without exceeding the max spacing.
5. In **Direction**, select one of the following:

- **Perpendicular** allows you to have the members drawn perpendicular to the highlighted edge member.
- **Parallel with left** allows you to have the members drawn parallel to the edge member that connects to the end 1 of the highlighted edge member.
- **Parallel with right** allows you to have the members drawn parallel to the edge member that connect to the end 2 of the highlighted edge member.

6. Define the element parameters according to your needs.

TIP If you have saved infill properties to a named property set, you can recall them by selecting the set in the drop list at the top of the **Properties** window.

Place the pattern in a single bay

NOTE Infills can only be created within four sided enclosed bays with members or meshed/mid-pier shear walls attached to all sides. All suitable bays are shaded green.

1. Hover the mouse pointer over the required bay, adjacent to the required edge member.
2. If necessary, change the orientation by moving the cursor to a different edge member, or by adjusting the general parameters in the **Properties** window.
3. Click the bay to create the pattern.

Place the pattern in multiple bays

NOTE Infills can only be created within four sided enclosed bays with members or meshed/mid-pier shear walls attached to all sides. All suitable bays are shaded green.

1. Drag a box around the required bays to select them.
2. Hover the mouse pointer over one of the bays, adjacent to the required edge member.
3. If necessary, change the orientation by moving the cursor to a different edge member, or by adjusting the general parameters in the **Properties** window.
4. Click to create the same pattern in all the selected bays.

Merge planes

Where a model contains very close planes, as can often occur in imported models, for example, the Merge Planes command allows the you to merge two planes by selecting a destination plane and then a source plane.

1. On the **Edit** tab, click **Merge Planes**.

   Existing Level, Frame, and Slope planes are displayed in the active view.

   NOTE If the view is cluttered by having all the plane types displayed simultaneously, and you only want to merge Levels, consider switching off Frames and Slopes in Scene Content.

2. Pick the destination plane.
3. Pick the source plane.

   If the planes can be merged, everything on the source plane will be moved to the destination plane, and the source plane will be deleted. If the merge is not possible, a message is given in the Process window and the model will be unchanged.

Create and manage free points

Free points can be used to connect beams, columns, braces etc. without having to define an intersection between grid lines or construction lines.

NOTE Once a free point has an item connected to it, if the item is moved a new point is created and the existing free point remains.

Create a free point

1. Make sure that Free Points are visible in Scene Content.
2. Right click in the view and pick **Add free point...** from the context menu.
3. Enter the point co-ordinates and click **OK**.

Adding, moving or deleting free points from the Edit tab

1. On the **Edit** tab, click **Free Points...**

   A Free Points dialog is displayed.

2. Click **Add** to add a new point, **Delete** to remove an existing point, or click in the table to edit the co-ordinates of existing points.

Related video

Free point defined by XYZ coordinates

Planarize

Planarize is a tool for moving nodes of selected entities into a target plane.

Related information

Related video

Correct modeling errors using Planarize

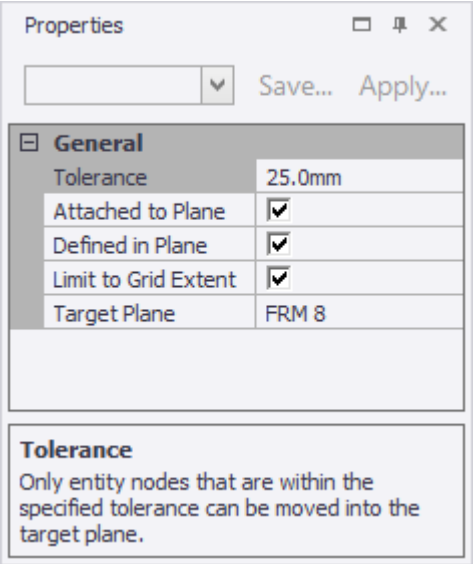
Overview

The Tekla Structural Designer Support Team occasionally see models with geometry issues such as entities that are very close to - but not quite - co-planar. Such small inconsistencies can cause issues with many operations, such as load decomposition and analysis, and should be avoided. Such issues rarely result from modeling directly in Tekla Structural Designer, but may more commonly occur for import of models with geometry issues from other applications.

Planarize can assist you in resolving these inconsistencies by allowing you to move nodes of not-quite co-planar entities into the 'correct' target plane, as selected by you.

To run **Planarize**, you right-click on the target grid line (the one on which entities are correctly modeled) in any view and select **Planarize FRM** *<grid ref>*

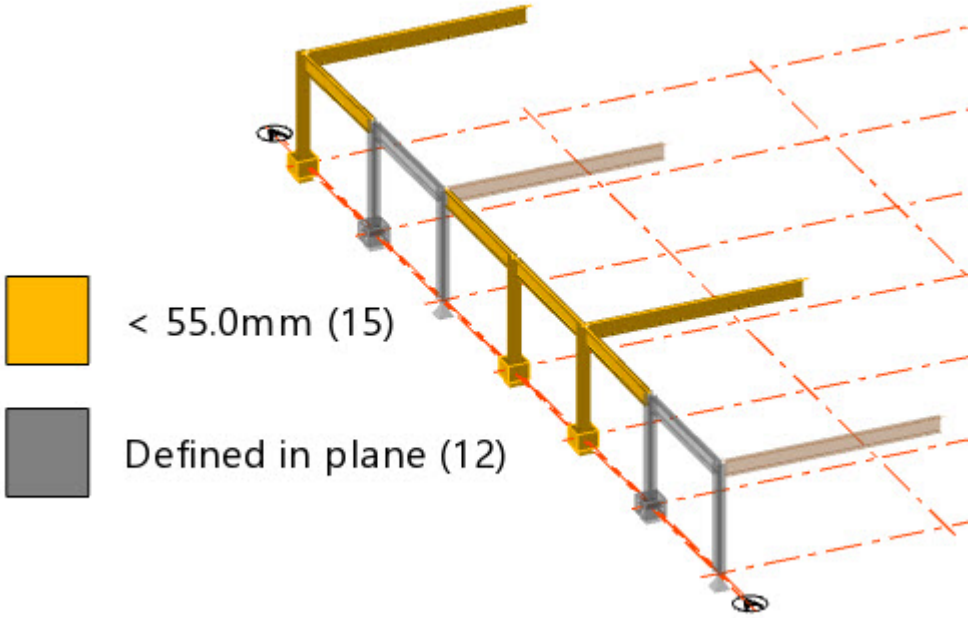
This identifies the **target plane** into which nodes can be moved, and opens the Planarize properties.



The properties enable you to:

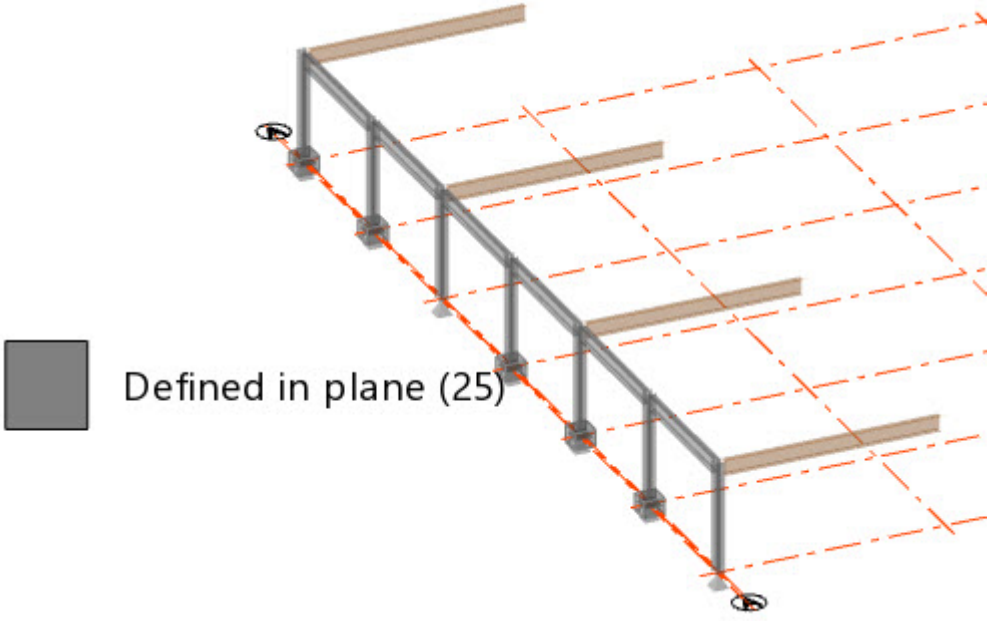
- define the **tolerance** (default 25mm) for entity nodes to be considered near enough to the target plane to be able to be moved into it.
- display or hide entities already **attached to**, or **defined in** the target plane.
- **Limit to Grid Extent**, by default this property is selected which restricts the length of the target plane to the extent of the grid line (see this example (page 319)).
- confirm the **target plane** into which nodes will be moved.

In the 3D scene view, different colors are used to distinguish between entities attached to, close to, and defined in the plane.

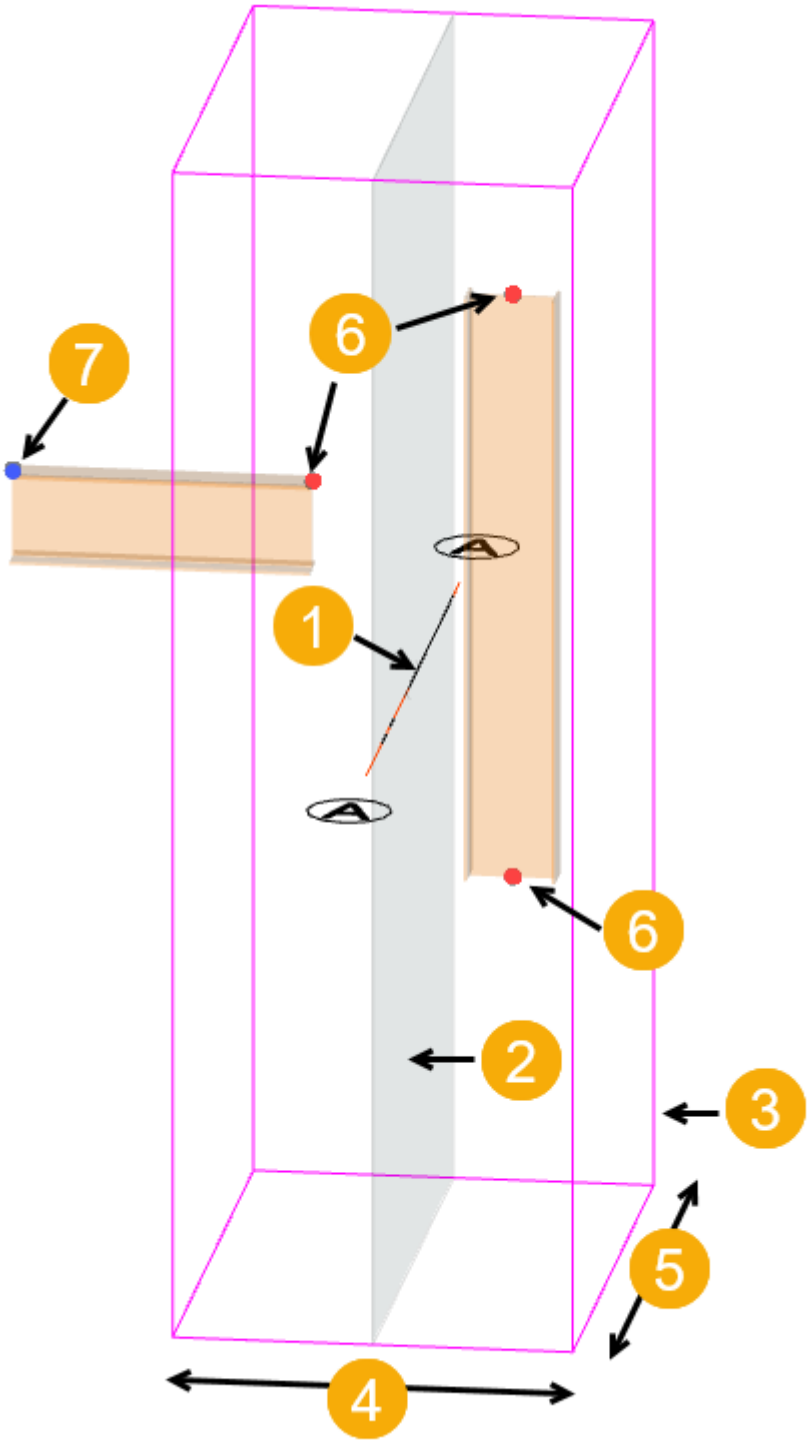


In this view you can select entities close to the target plane in order to move their **adjacent nodes** (i.e. not quite co-planar nodes) into the target plane.

- As entities are selected they change color to indicate their adjacent nodes have moved and are now defined in plane.

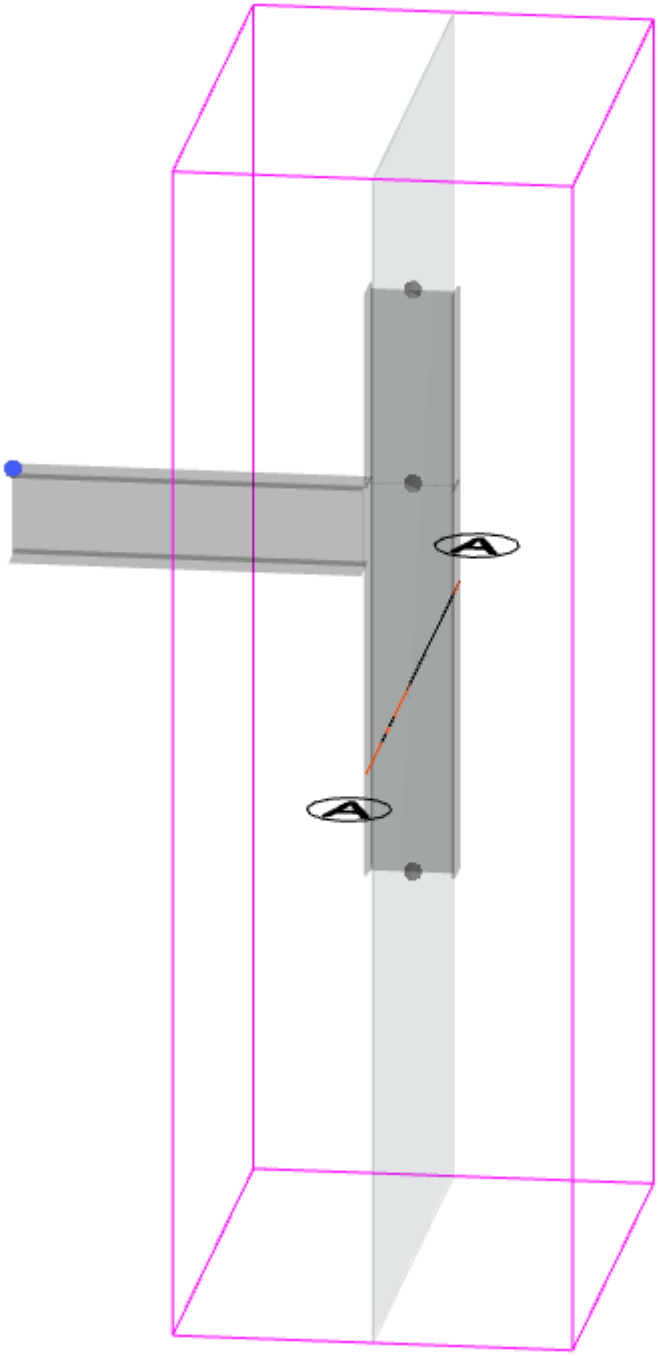


Note that adjacent nodes must lie within an adjacent zone around the plane. This zone (which vertically continues to the model extents) is illustrated in the figure below.



1 planarized grid line	
2 target plane	
3 adjacent zone	
4 width of zone = 2 x **Tolerance**	
5 length of zone = length of grid line	**NOTE** if **Limit to Grid Extent** unselected, zone length increases to model extents
6 adjacent nodes	
7 non-adjacent node	

When entities are selected, only their **adjacent nodes** move into the target plane, their **non-adjacent nodes do not move**.



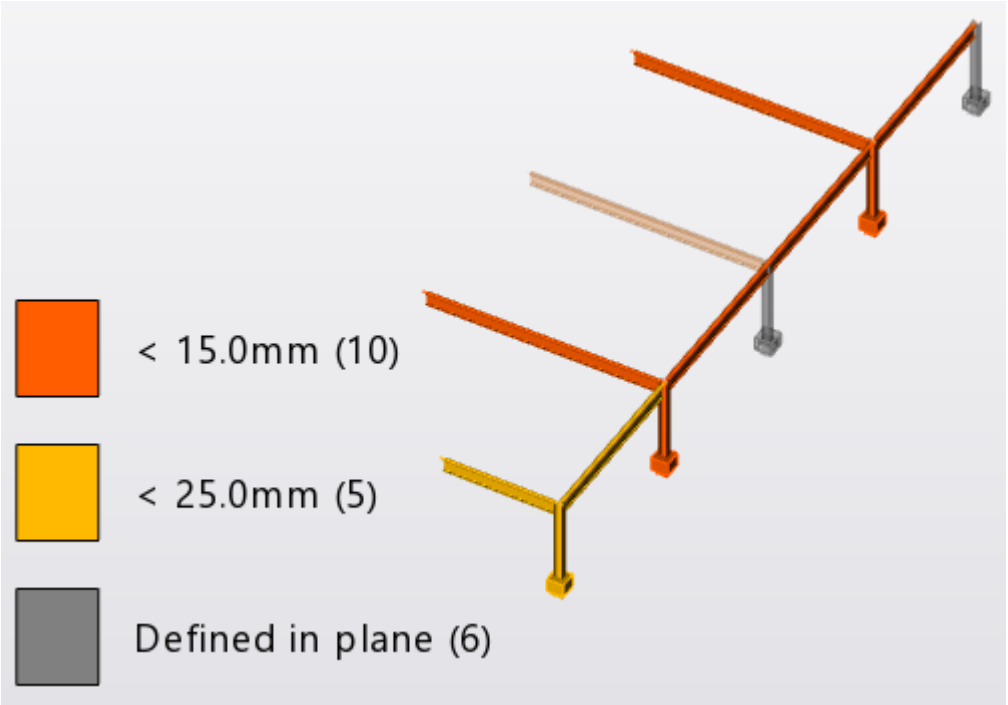
NOTE An example illustrating the use of **Planarize** to resolve validation errors is available here (page 312).

Move nodes into the plane of a grid line

To do this, run **Planarize** from a 2D or 3D scene view:

1. Hover the cursor over the target grid line to highlight it.
2. Right click and select **Planarize FRM** *<grid ref>*.

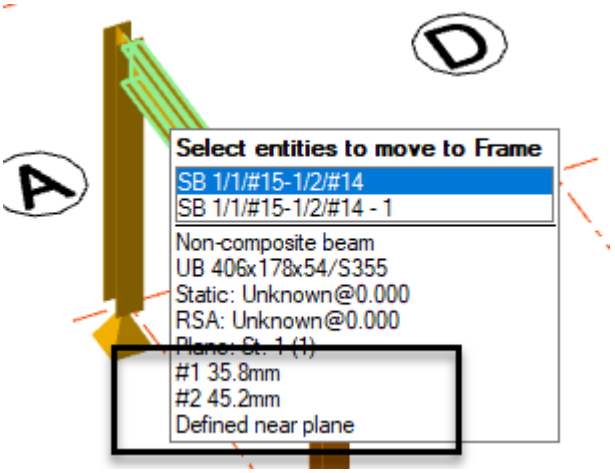
   Different colors and a legend are used to identify entities defined in, attached to, and near to the target plane.



   - Entities *defined in* the target plane are indicated in gray.
   - Entities with nodes *near to* the plane (i.e. within the tolerance) are indicated in one or more subsequent colors. The color being determined by the distance to the *furthest* node. These are the nodes that can be moved in to the plane.
   - The figure in brackets after each legend band indicates the number of entity *nodes* within the band (not the number of entities).
3. In the **Properties** window set the **tolerance** as required.

   NOTE Tolerance should typically be kept small to allow the correction of small mis-alignment errors only.

4. Hover the cursor over a color coded entity to display its tooltip. The tooltip lists the entity nodes (#1, #2 etc.) within the tolerance and shows their distance from the target plane.



5. Select the entity to move the entity nodes listed in the tooltip into the target plane.

NOTE Only the nodes indicated in the tooltip are moved from their current location into the target plane, other nodes connected to the entity but outside the **tolerance** are not moved.

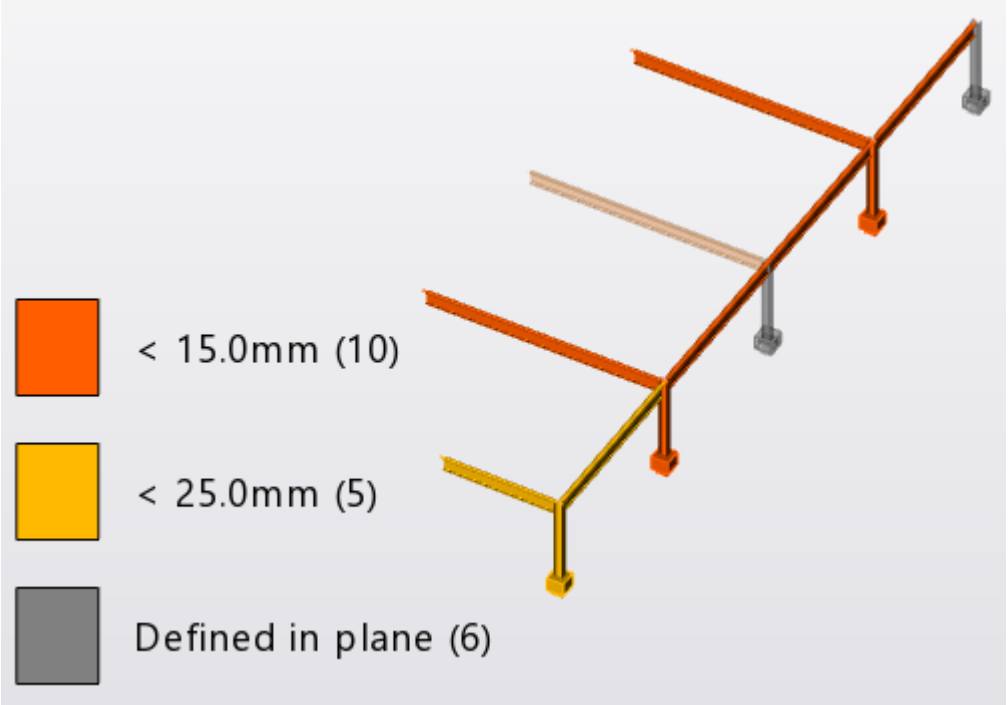
NOTE If moving the nodes results in an entity being defined in the target plane, a new frame will be automatically created for the target plane if it didn't already exist.

Move nodes into the plane of a frame

To do this, run **Planarize** from the Project Workspace:

1. Expand the Frames branch In the Structure tree.
2. Right click on the required frame and select **Planarize FRM** <*frame ref*>.

Different colors and a legend are used to identify entities defined in, attached to, and near to the target plane.

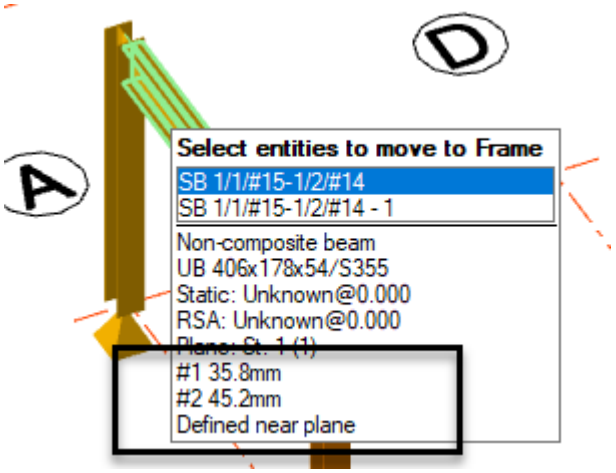


- Entities *defined in* the target plane are indicated in gray.
- Entities with nodes *near to* the plane (i.e. within the tolerance) are indicated in one or more subsequent colors. The color being determined by the distance to the *furthest* node. These are the nodes that can be moved in to the plane.
- The figure in brackets after each legend band indicates the number of entity *nodes* within the band (not the number of entities).

3. In the **Properties** window, set the **tolerance** as required.

NOTE Tolerance should typically be kept small to allow the correction of small mis-alignment errors only.

4. Hover the cursor over a color coded entity to display its tooltip. The tooltip lists the entity nodes (#1, #2 etc.) within the tolerance and shows their distance from the target plane.



5. Select the entity to move the entity nodes listed in the tooltip into the target plane.

NOTE Only the nodes indicated in the tooltip are moved from their current location into the target plane, other nodes connected to the entity but outside the **tolerance** are not moved.

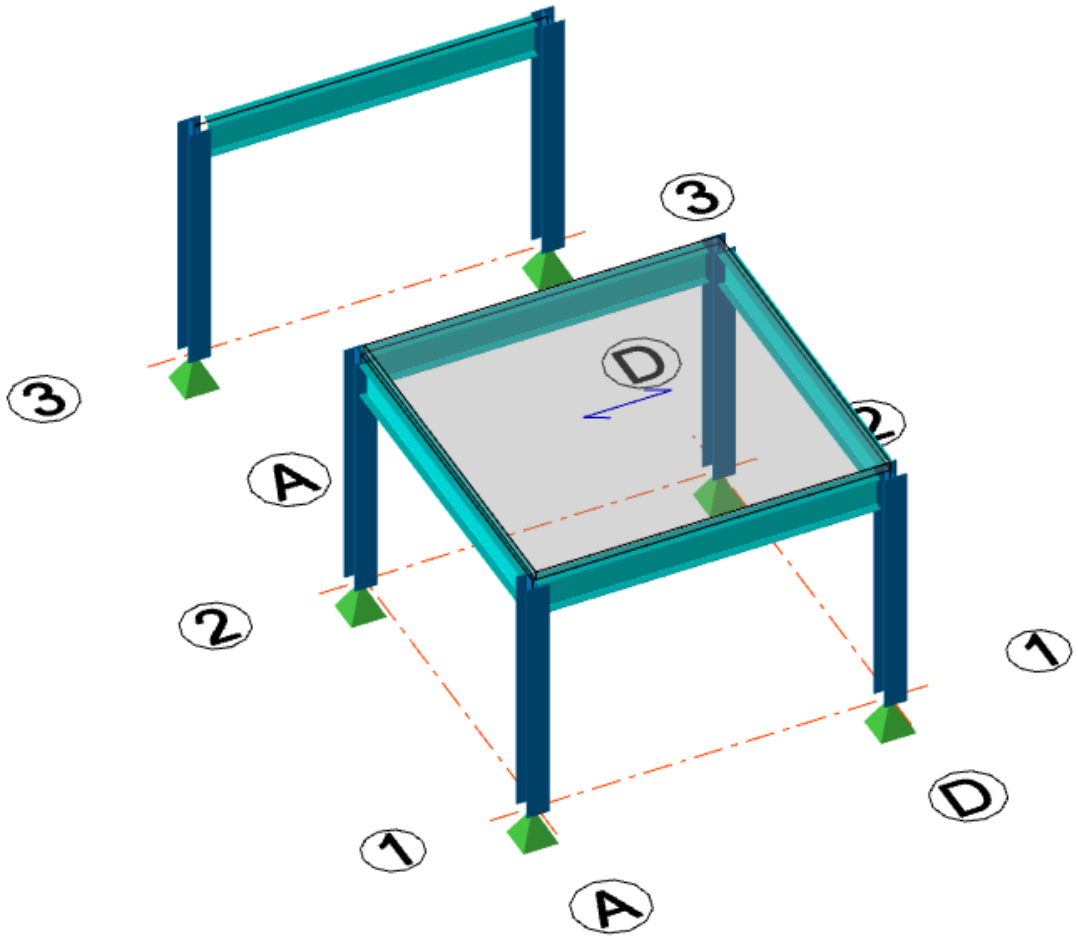
Planarize properties

Item	Description
Tolerance	As the Tolerance is increased, more nodes become available for moving into the plane.
Attached to plane	• On - displays entities with nodes attached to the target plane • Off - (default) hides entities with nodes attached to the target plane
Defined in plane	• On - (default) displays entities entirely in the target plane • Off - hides entities entirely in the target plane
Limit to Grid Extent	• On - (default) only those entities adjacent to the grid line defining the target plane are considered • Off - the target plane is projected to the model extents so that all entities within the tolerance are considered

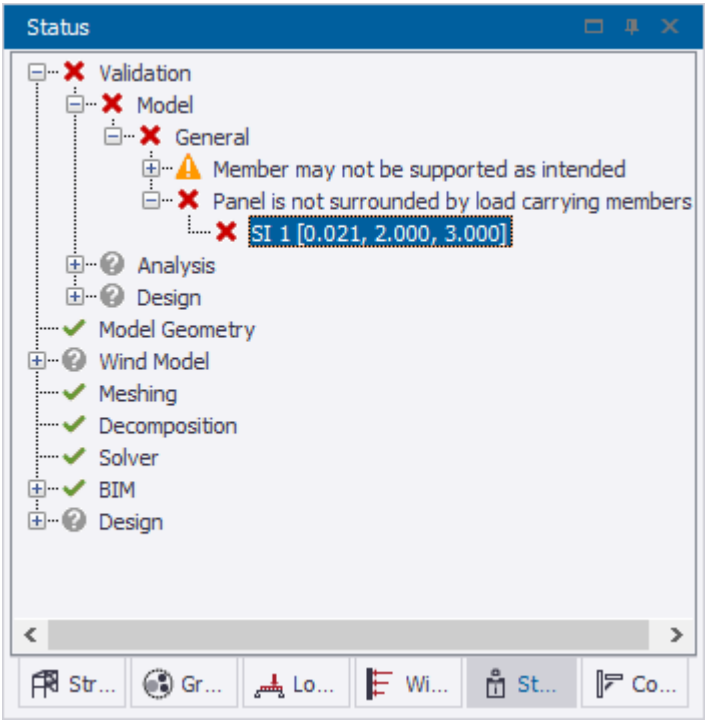
Item	Description
Target Plane	The plane into which nodes will be moved. (Pre-selected by choosing an existing frame or grid line.)

Planarize Example

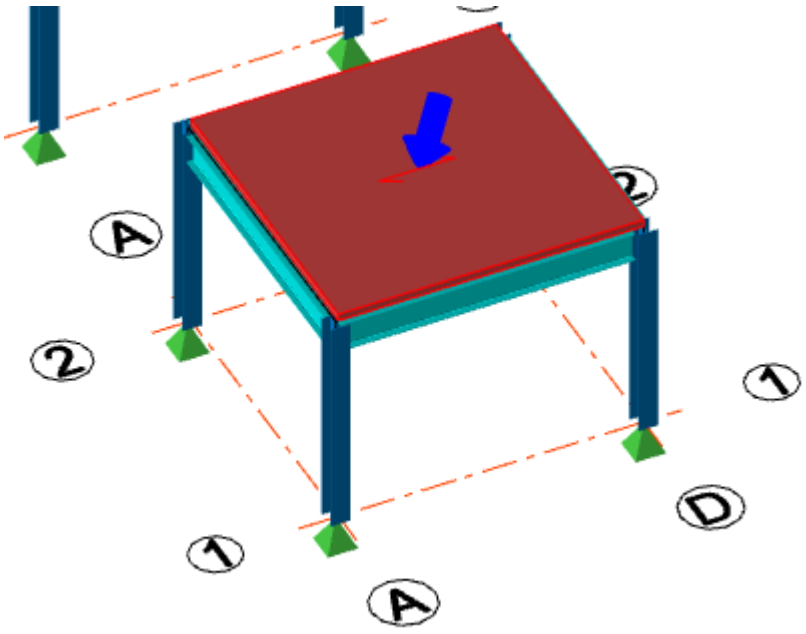
Consider the model shown below.



When the model is validated an error is displayed.



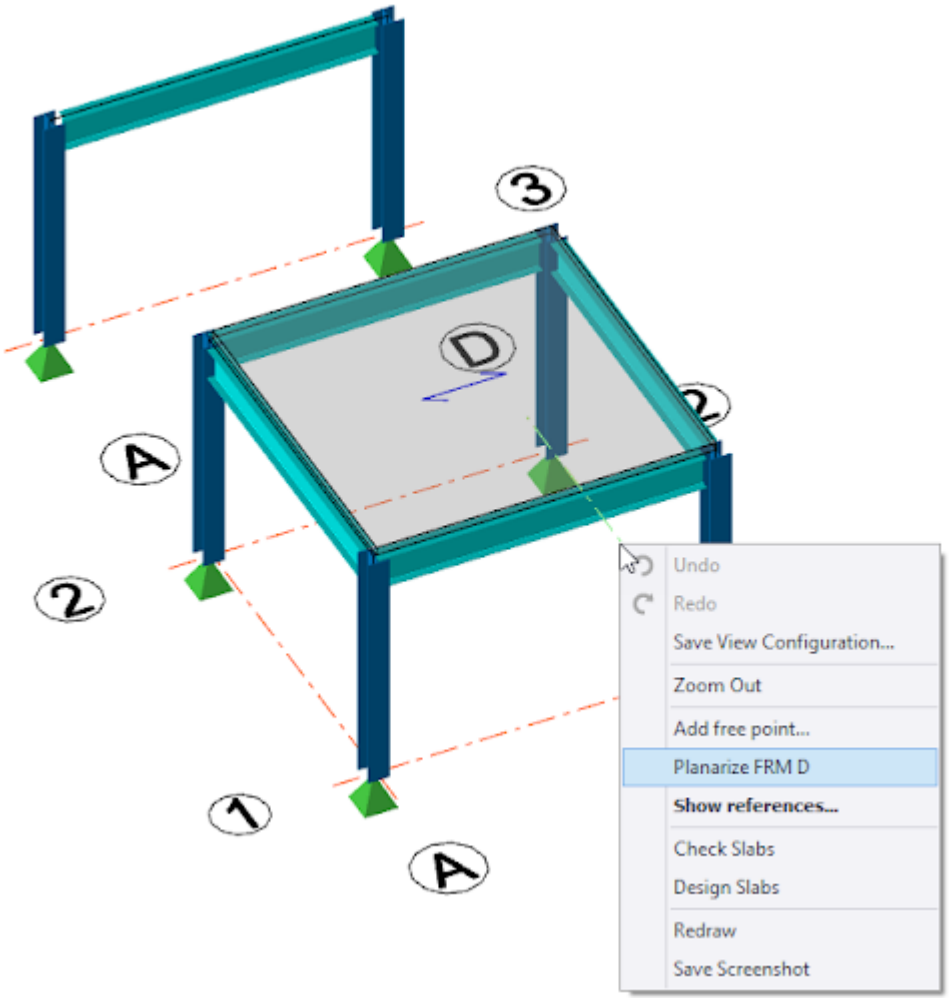
The error indicates that the roof panel is not being supported as intended by the edge beams.



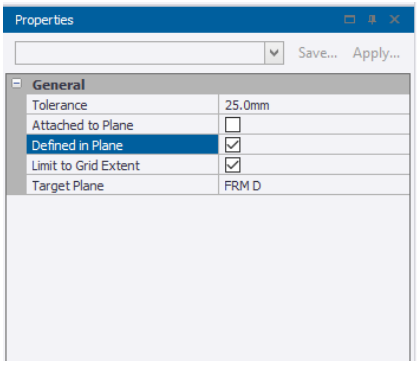
The roof panel should be supported along the edges defined by grids **D** and **A**. To confirm if this is the case both of these planes can be checked in turn using **Planarize**

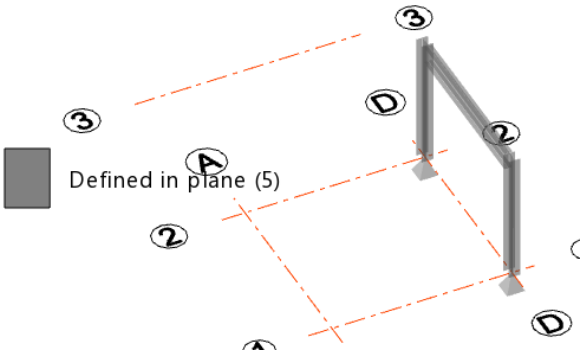
The process for doing this is as follows:

1. By hovering the cursor over grid line D, then right clicking, **Planarize FRM D** can be selected from the context menu.

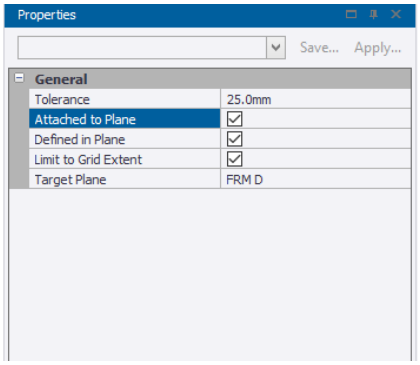


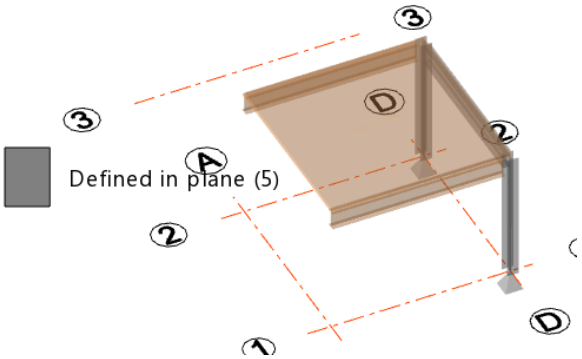
All of the edge members that should exist in the plane of grid line D are colored gray to indicate they are 'defined in plane' - this is correct.



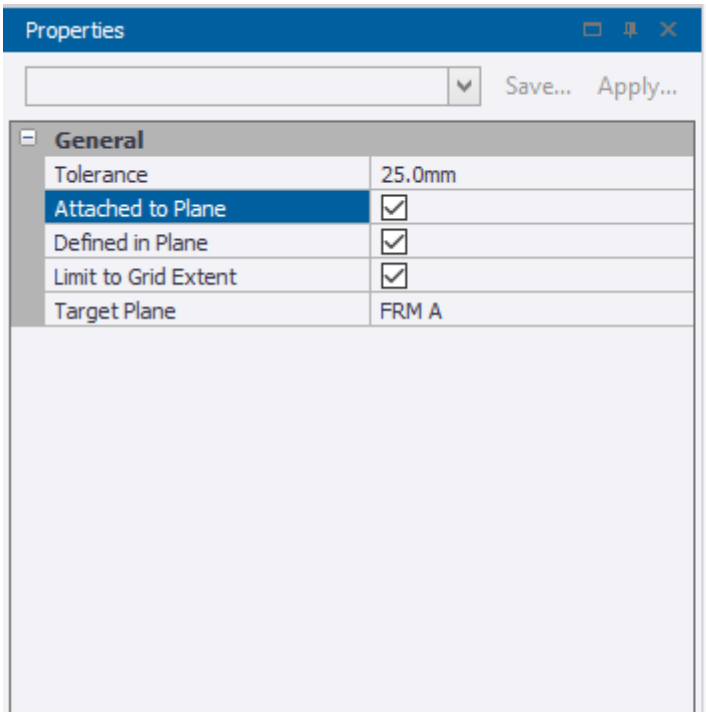


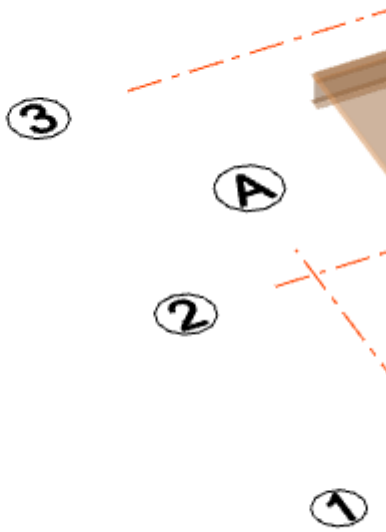
2. By clicking ‘Attached to plane’ in the Properties Window, the roof panel and transverse roof are displayed in brown - again this is correct.





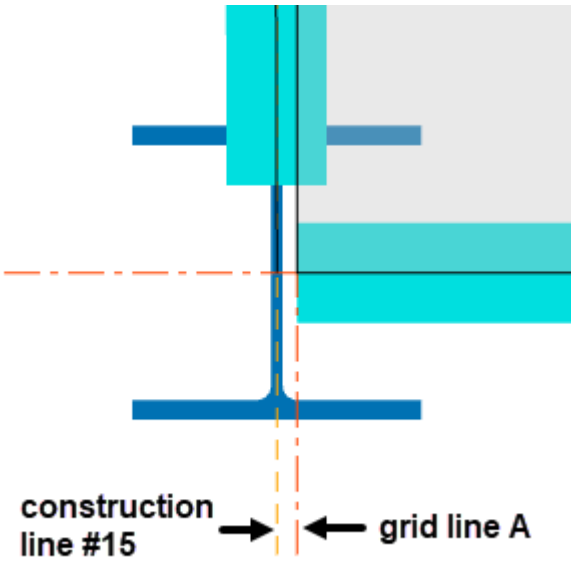
3. When the above process is repeated for grid line **A** a different result is obtained because this is where the modeling error exists.





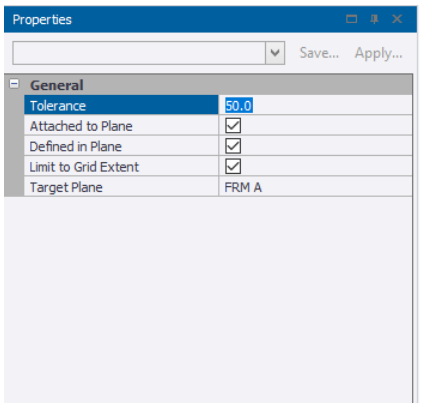
Neither of the columns or the beam that should have been colored gray to indicate they are in the plane of frame A are shown at all. The reason for this is that they were incorrectly placed using an adjacent parallel construction line (which isn't shown in the above view).

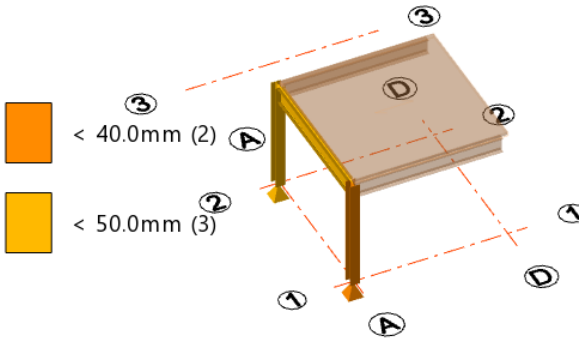
The view below shows the error in more detail - the beam and the two columns that should be referenced to grid line A are instead currently referenced to construction line #15 which runs almost but not quite parallel to grid line A.



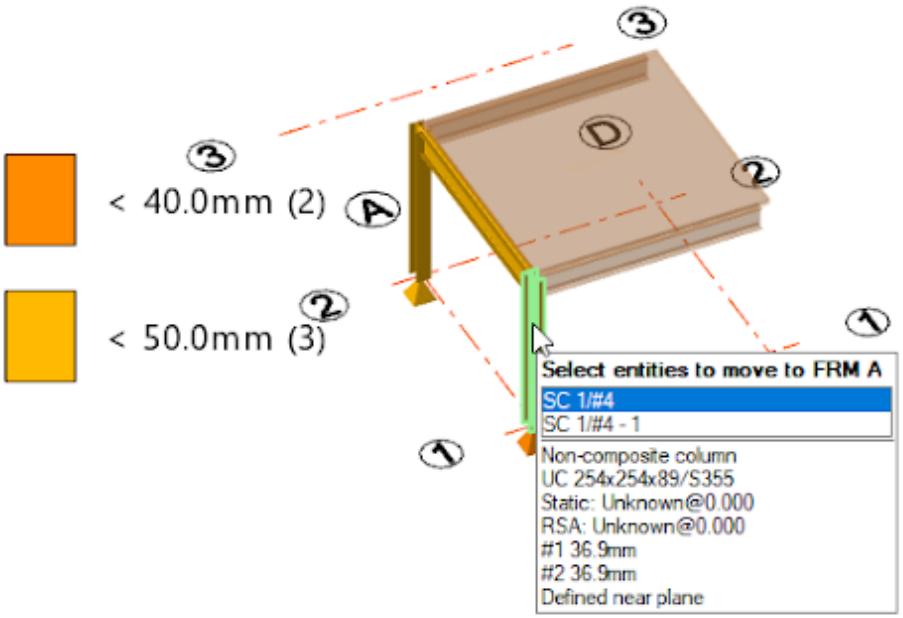
NOTE The beam and the two columns referenced to construction line **#15** are not initially displayed at all (in any color) when FRM A is planarized. This is because they lie just outside the current tolerance setting of 25mm.

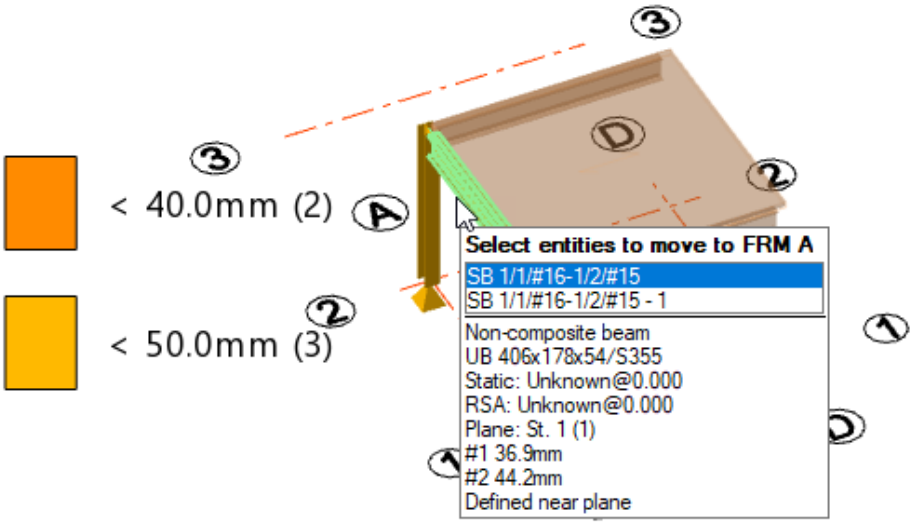
4. If the tolerance is increased to 50mm, the entities referenced by **#15** are now shown.

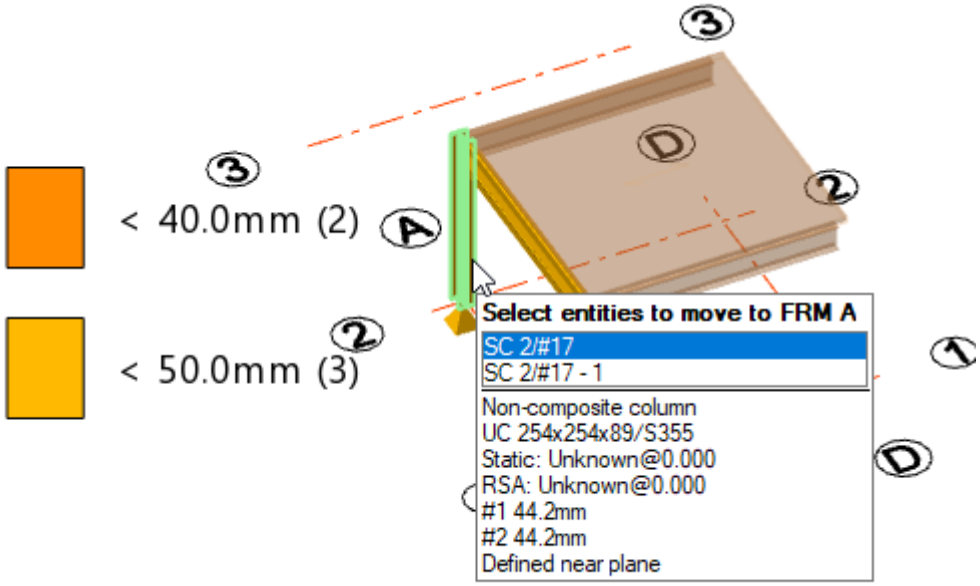




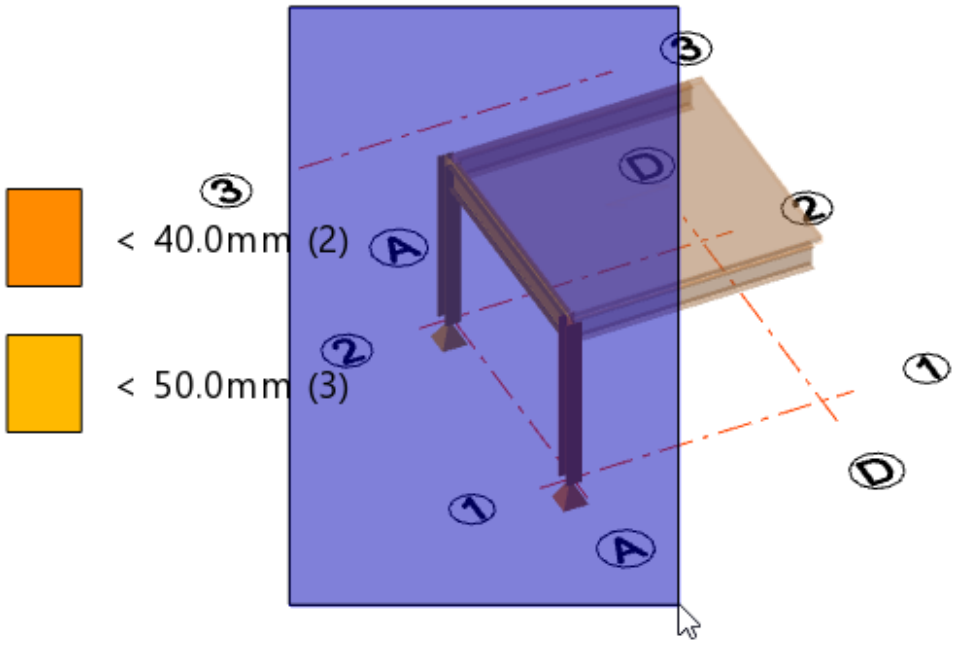
5. Hovering the cursor over the two columns and then the beam in turn, the tooltips reveal how far away the entity nodes are from the target plane.



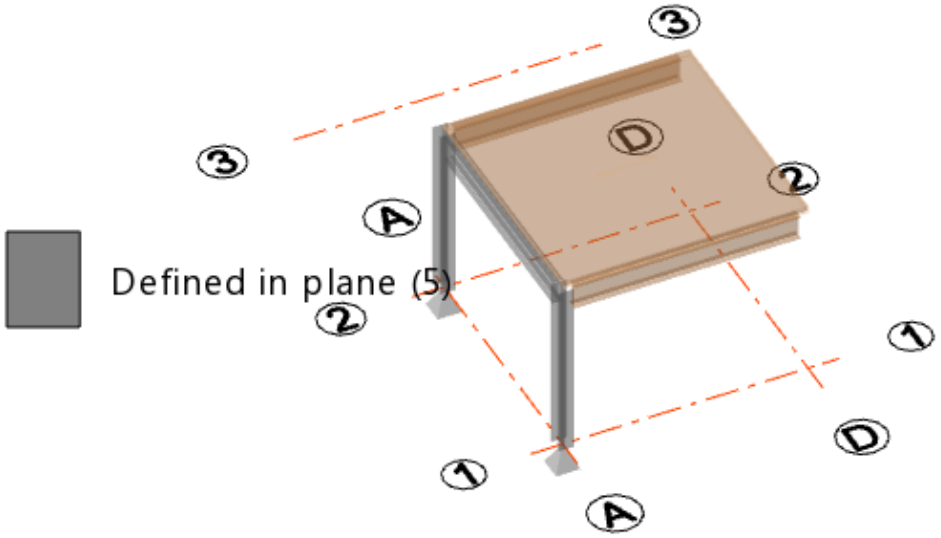




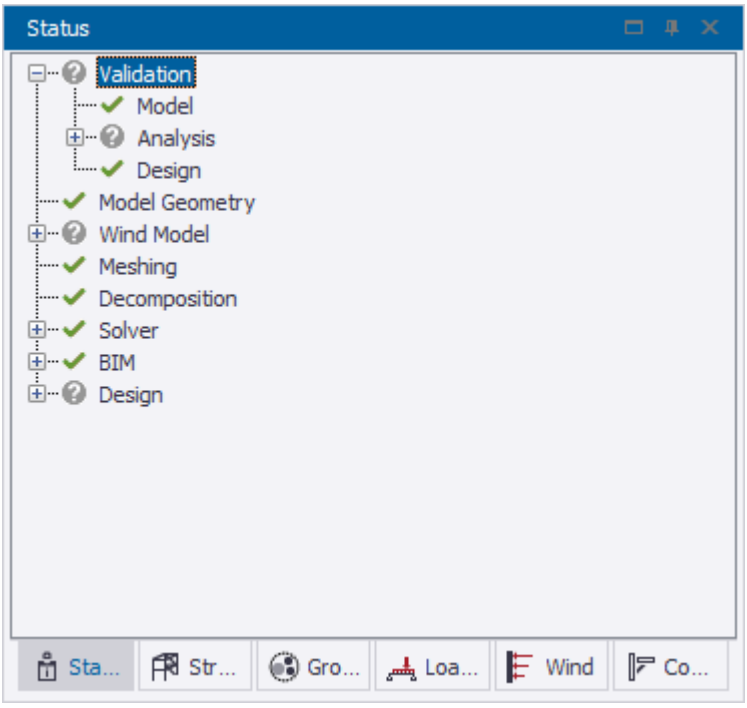
6. Because they now lie within the tolerance they are near enough to be moved into the plane of FRM A. They can all be selected in one operation by dragging a box from right to left around them.



7. Once selected, the nodes defining each of the three members are moved into the plane of FRM A.

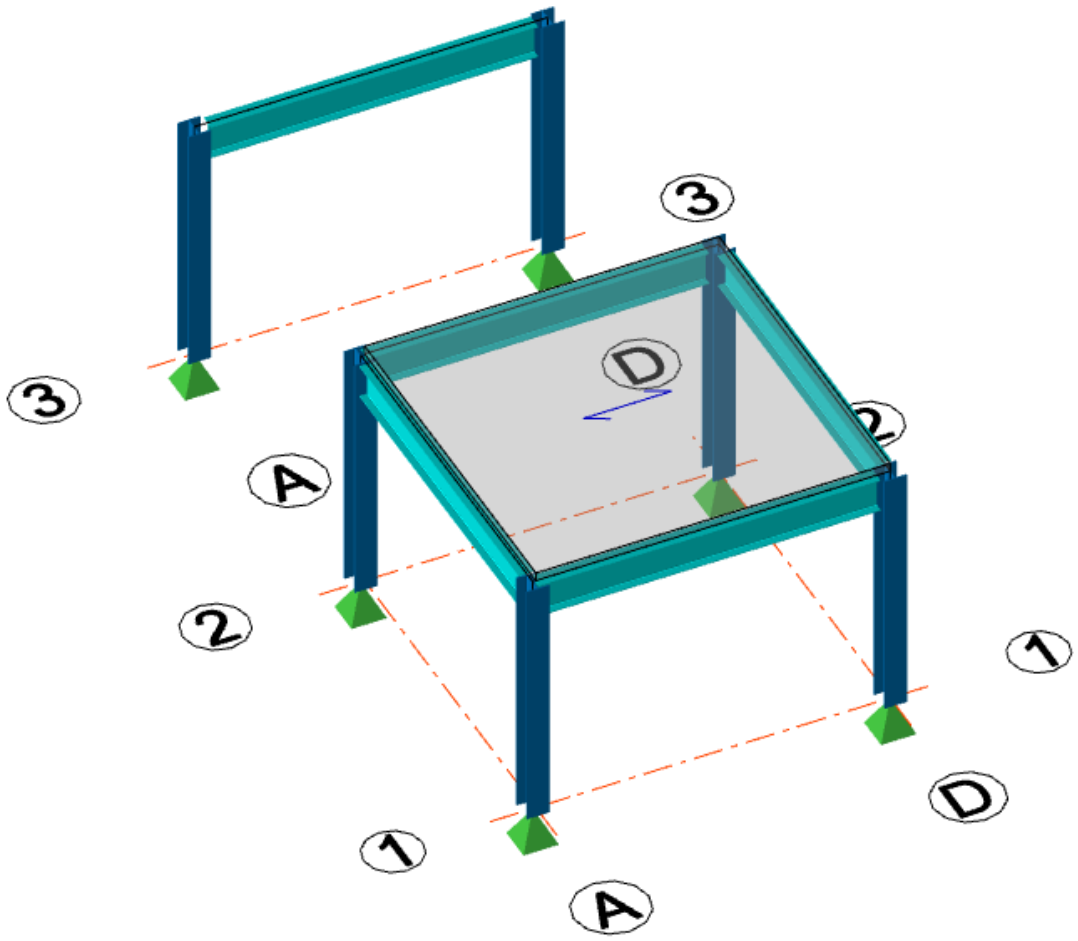


The model can now be validated, analyzed and designed without any further issues.

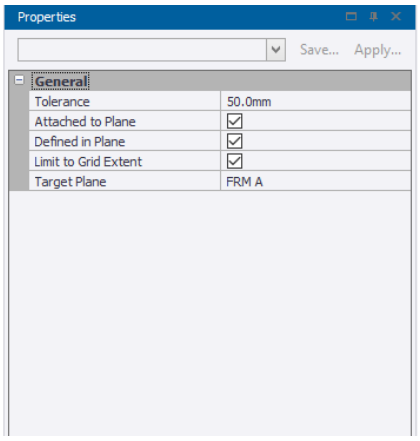


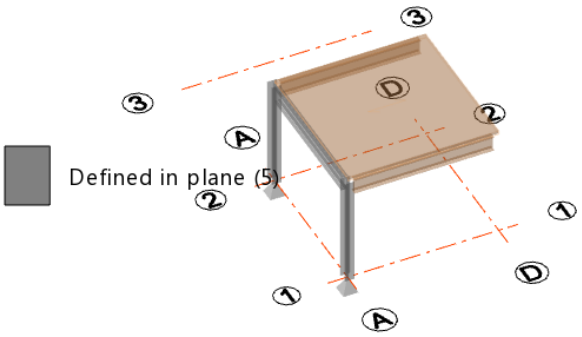
Link to Grid Extent

As long as the **Link to Grid Extent** property is selected, **Planarize** will look for entity nodes within the specified tolerance to either side of the target grid line, but is restricted to look no further than the two ends of the grid line.

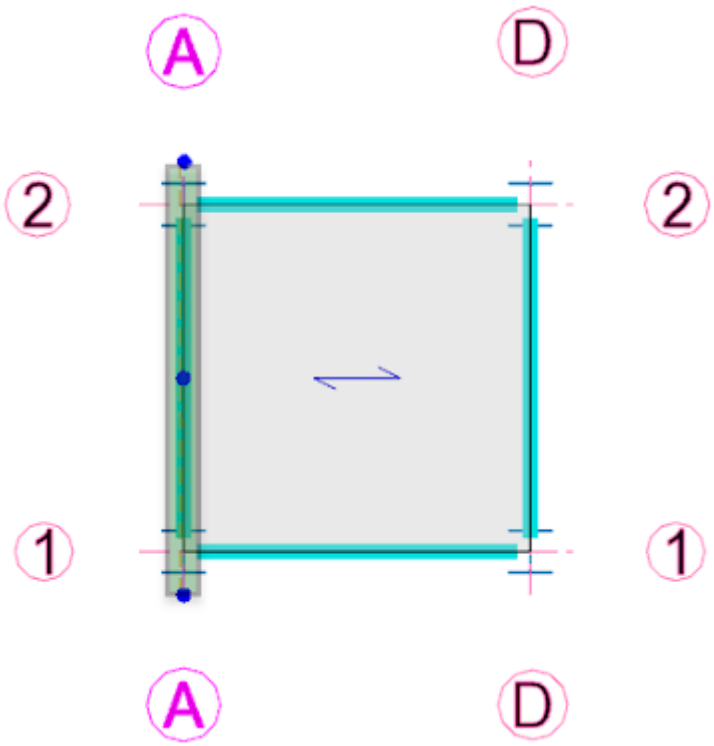


1. When grid line A is planarized, by default **Link to Grid Extent** is selected.

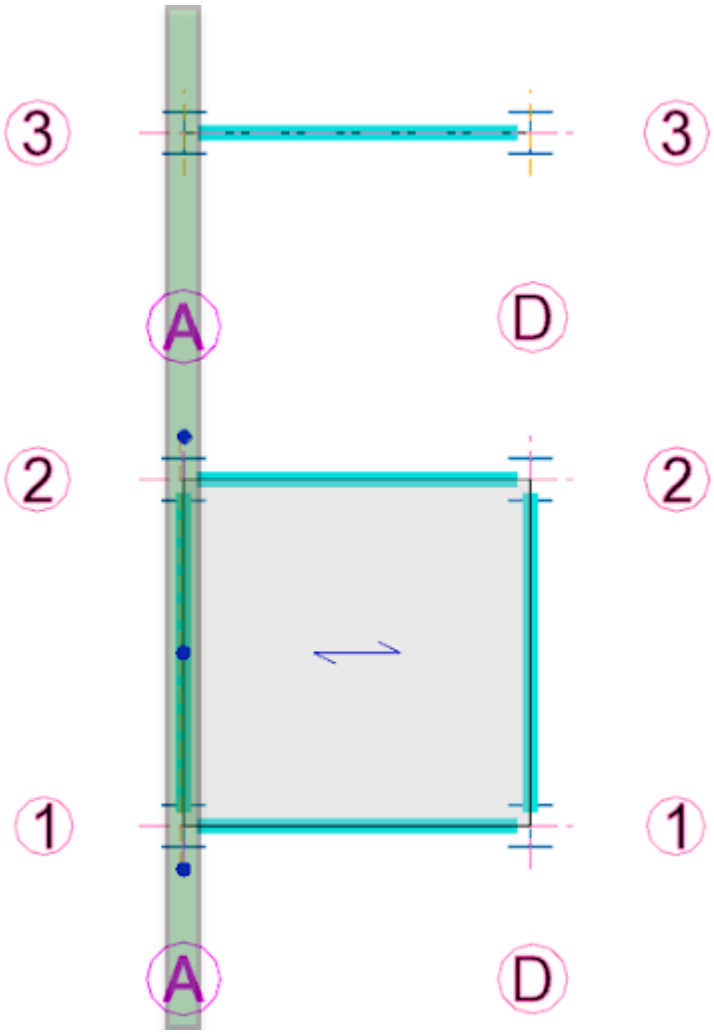




Consequently the potential zone in which entity nodes can be located is indicated by the green shaded box shown below.

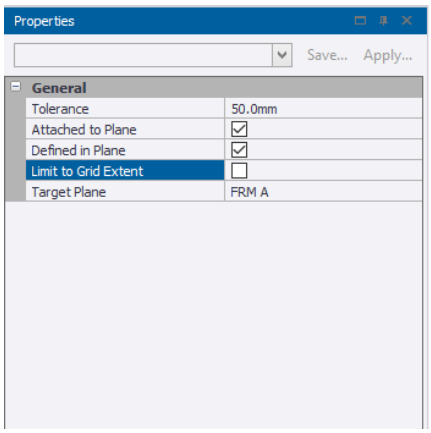


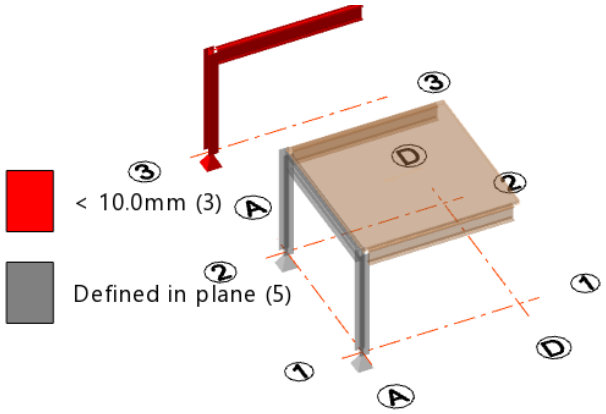
If **Link to Grid Extent** is unselected, the zone is allowed to extend to the model extents as shown below.



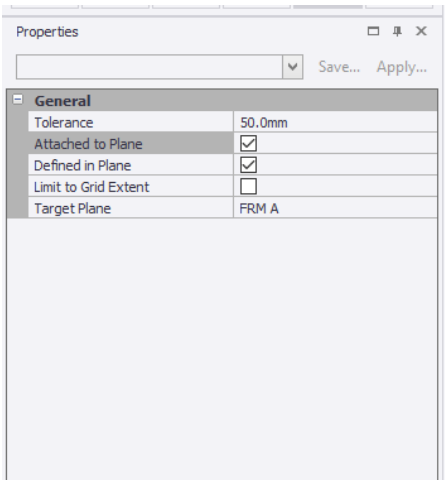
As a result other entities might potentially be identified that can also be moved into the target plane.

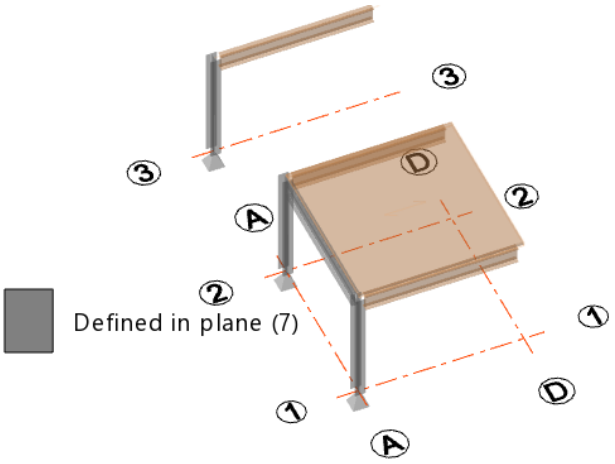
In this case a column and beam on grid line 3 have nodes that now lie within the shaded zone.





When these are selected they move so that they are now defined in the (projected) plane of grid line A.





2.6 Validate the model

You can validate the model at any time in order to trap errors that will cause the solver to fail before the model is submitted for analysis. Validation is also automatically performed during design.

NOTE For a list of validation issues and how to resolve them, refer to this support article.

Run model validation

- On the **Model** or **Load** tab, click **Validate**.

  Tekla Structural Designer performs the validation checks. If your model contains any issues, warning messages appear.

Adjust the conditions considered in model validation

1. On the **Home** tab, click **Model Settings**.
2. Go to the **Validation** page.
3. Select the conditions that you want Tekla Structural Designer to consider during validation checks.
4. Click **OK**.

3 Apply loading

The majority of loads can be applied and managed directly from the **Load** tab on the ribbon. Loading related settings, minimum lateral loads, global imperfections and live/imposed load reductions are all defined via Model settings (page 1243).

Click the below links to find out more:

- Create and manage loadcases, groups, combinations, envelopes and patterns (page 324)
-
-
- Apply panel, member, and structure loads (page 351)
- Apply wind loading (page 372)
- Apply snow loading (page 426)
- Apply seismic loading (page 434)

NOTE The application of ancillaries and equipment loads which are specific to industrial structures are covered elsewhere, for further details see:

3.1 Create and manage loadcases, groups, combinations, envelopes and patterns

Loadcases, groups, combinations and envelopes are all managed in the Loading dialog box (page 1557). Load patterns are reviewed via **Update Patterns** on the **Load** tab.

- Create and manage loadcases (page 325)
- Create load groups (page 329)
- Create and manage load combinations (page 338)

- Create and manage load patterns (page 348)

Create and manage loadcases

Before applying loads to your model, you must first define the loadcases within which the loads will be contained.

To create and manage loadcases, see:

- Create loadcases (page 325)
- Activate reductions in live or imposed loadcases (page 325)
- Renumber and/or reorder all loadcases (page 329)

See also

Import loadcases and combinations from a spreadsheet (page 343)

Create loadcases

When you create a new model, Tekla Structural Designer automatically creates a loadcase whose type is **Self weight - excluding slabs**. You cannot access the loadcase because Tekla Structural Designer automatically calculates the loads within it using the objects in your structure. Tekla Structural Designer also creates three other loadcases that are initially empty. However, you will almost certainly need to create other loadcases that contain the loads that your building must withstand.

1. On the **Load** tab, click  **Loadcases**.

   The Loading dialog box (page 1557) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Click **Add**.
3. In **Loadcase Title**, name the loadcase.
4. In **Type**, select the desired loadcase type.
5. Select whether the loadcase is included when you automatically generate load combinations.
6. Click **OK**.

   Tekla Structural Designer adds the new loadcase to the list of loadcases in the **Loading** list.

Activate reductions in live or imposed loadcases

When you create a loadcase whose type is **Live** (US) or **Imposed** (other regional codes), you can allow Tekla Structural Designer to automatically

calculate load reductions in accordance with the percentages specified on the Load reductions page in **Model Settings**. To do so, see the following instructions.

Overview of live or imposed load reductions

Live load reductions (US), or imposed load reductions (other regional codes) are established during pre-analysis for use in subsequent column and wall design, and when the regional code is set to ACI/AISC - beam design also.

- For regional codes other than ACI/AISC, the level of imposed load reduction to consider for beams can be set manually in the beam properties. This is especially useful for more economic design of transfer beams supporting a number of floors.

Reductions are only applied to those live/imposed loadcases that have had the **Live/Imposed Load Reductions** box selected on the **Loadcases** page in the Loading dialog (page 1557).

The reduction percentage to be applied is specified on the Live/imposed load reduction settings (page 1251) page in **Model Settings**. This percentage can differ depending on the number of floors being supported.

To cater for additional floors that are carried but that have not been included in the model an Assume extra floors supported value can be specified in the column and wall properties.

The methodology for live/imposed load reduction differs between national codes of practice:

Head Code: EC or BS

Levels can be designated either as "to be" or "not to be" included in the determination of the load reductions through Count floor as supported check boxes for each level in the column and wall properties. This feature enables what appears to be a roof to be counted as a floor, or conversely allows a mezzanine floor to be excluded from the number of floors considered for any particular column or wall. The moments from fixed ended beams framing into a column or wall are never reduced.

For more information, see: and

Head Code: ACI/AISC

Before undertaking member design, Live and Roof Live loads are multiplied by a reduction factor R for roof live loads and other live loads independently. This reduced load is then used in combination to create design forces. The reduction factor is related to the tributary area of load carried by the particular member and also the K_{LL} factor, where K_{LL} comes from Table 4-2 in ASCE 7-05/ ASCE 7-10 or Table 4.7-1 in ASCE 7-16 .

Essentially:

- Interior and exterior cols (no cantilever slabs) $K_{LL} = 4$
- Edge and interior beams (no cantilever slabs) $K_{LL} = 2$

- Interior beams (with cantilever slabs) K_{LL} = 2
- Cantilever beams K_{LL} = 1
- Edge cols (with cantilever slabs) K_{LL} = 3
- Corner cols (with cantilever slabs) K_{LL} = 2
- Edge beams (with cantilever slabs) K_{LL} = 1

As it is not possible to assess where cantilever slabs are and what they are attached to - you are able to change the K_{LL} factor for all column/wall stacks and beam spans as required in the Properties Window.

The default settings are:

- Columns/walls = 4
- Beams = 2
- Cantilevers = 1

For more information, see:

Head Code: AS

Before undertaking member design, imposed loads are multiplied by a reduction factor $\psi_{a.}$

This reduced load is then used in combination to create design forces. The reduction factor is related to the tributary area of load carried by the particular member.

For more information, see:

Activate reductions in live and roof live loadcases (US)

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog box (page 1557) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Select the **Live** or **Roof Live** loadcase to which you want to apply the reductions.
3. Select the **Live Load Reductions** option.
4. Click **OK**.

Activate reductions in imposed loadcases (other regional codes)

NOTE You cannot activate reductions in loadcases whose type is set to **Roof Imposed**.

1. On the **Load** tab, click  **Loadcases**.

   The Loading dialog box (page 1557) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Select the **Imposed** loadcase to which you want to apply the reductions.
3. Select the **Imposed Load Reductions** option.
4. Click **OK**.

Set up minimum lateral loadcases

If the regional code being worked to has a minimum lateral loads requirement, these loads can be set up via **Model Settings** as follows:

1. On the **Home** tab, click **Model Settings**.
2. In the **Model Settings** dialog box, click **Minimum Lateral Loads**.
3. Select the number of cases to set up by clicking **Case A**, or **Cases A & B**.
4. Select the gravity load type to be used for the calculation.
5. Select whether the gravity load used for the calculation should be factored or unfactored.
6. Specify the percentage of the gravity load to be applied as a minimum lateral load in each direction.
7. Click **OK**.

Minimum lateral load loadcases are added to the list of loadcases in the **Loading** list.

NOTE You must nominate a gravity load combination to be used for the minimum lateral load calculations (page 342) before these loadcases can be added to load combinations.

Set up global imperfections

Horizontal forces to cater for global imperfections are set up via **Model Settings** as follows:

1. On the **Home** tab, click **Model Settings**.
2. In the **Model Settings** dialog box, click the **Global Imperfections** page.
3. Specify the parameters as required for the regional code being worked to.

4. Click **OK**.

The global imperfection loadcases available for selection on the **Combinations** page in the **Loading** list are configured according to the parameters that have been specified.

Renumber and/or reorder all loadcases

When you delete loadcases from the Loading dialog box (page 1557), the remaining loadcases retain their original loadcase number. If necessary, you can renumber/reorder the remaining loadcases in sequence. For more information, see the following instructions.

1. On the **Loading** tab of the **Project Workspace**, right-click  **Loadcases**.
2. In the context menu, select **Reorder**. The loadcases are automatically reordered and the numbering updated to suit.
3. In the dialog you then have the opportunity to move loadcases up or down the order manually if required.

NOTE The same **Reorder** command can also be accessed directly from the (page 1557).

Create load groups

Load groups facilitate the generation of combinations (primarily for design of industrial structures where many loading scenarios must be considered).

Overview of load groups

Features of load groups

- Load groups are an aid to building combinations - both manually and using the generator - and their use is entirely optional.
- Once a combination is built using load groups, the link to the load group is lost – the combination is made up only of loadcases. Only loadcases and combinations are analyzed – load groups are not.
- Each load group holds items of a single load type - e.g. “Dead” or “Live”...etc. - and can contain both multiple loadcases and other load groups
- The fundamental load groups setting is the “Class” which is either “Inclusive” or “Exclusive”
    - When “Inclusive” – all loadcases are added at once into a combination

- When “Exclusive” – loadcases are used one at a time in combinations. Thus for example, where an Exclusive load group contains four loadcases, the generator will produce four combinations for all required combinations which include the group’s load type (e.g. “Imposed”), each containing only one of the group’s four cases.

The load groups process

To make use of load groups you would proceed as follows:

1. Define Loadcases as normal
2. If required, create load groups from loadcases and/or other load groups
3. Create combinations by combining loadcases and/or load groups

The end result is loadcases and combinations (built up of factored loadcases). These are then run through analysis and design.

Create load groups

1. On the **Load** tab, click  **Load Groups**.

   The Loading dialog box (page 1557) opens on the **Load Groups** page. On this page, you can see all currently existing load groups.
2. Click **Add**.
3. Enter the **Load Group Title**.
4. In **Class**, select the desired load group class.
   - Inclusive - all loadcases are added at once into a combination
   - Exclusive – loadcases are used one at a time in combinations
5. Click the load group name in the left hand panel of the dialog to display the available loadcases.
6. Select each loadcase in turn to be included and click the right arrow button to copy it into the load group.

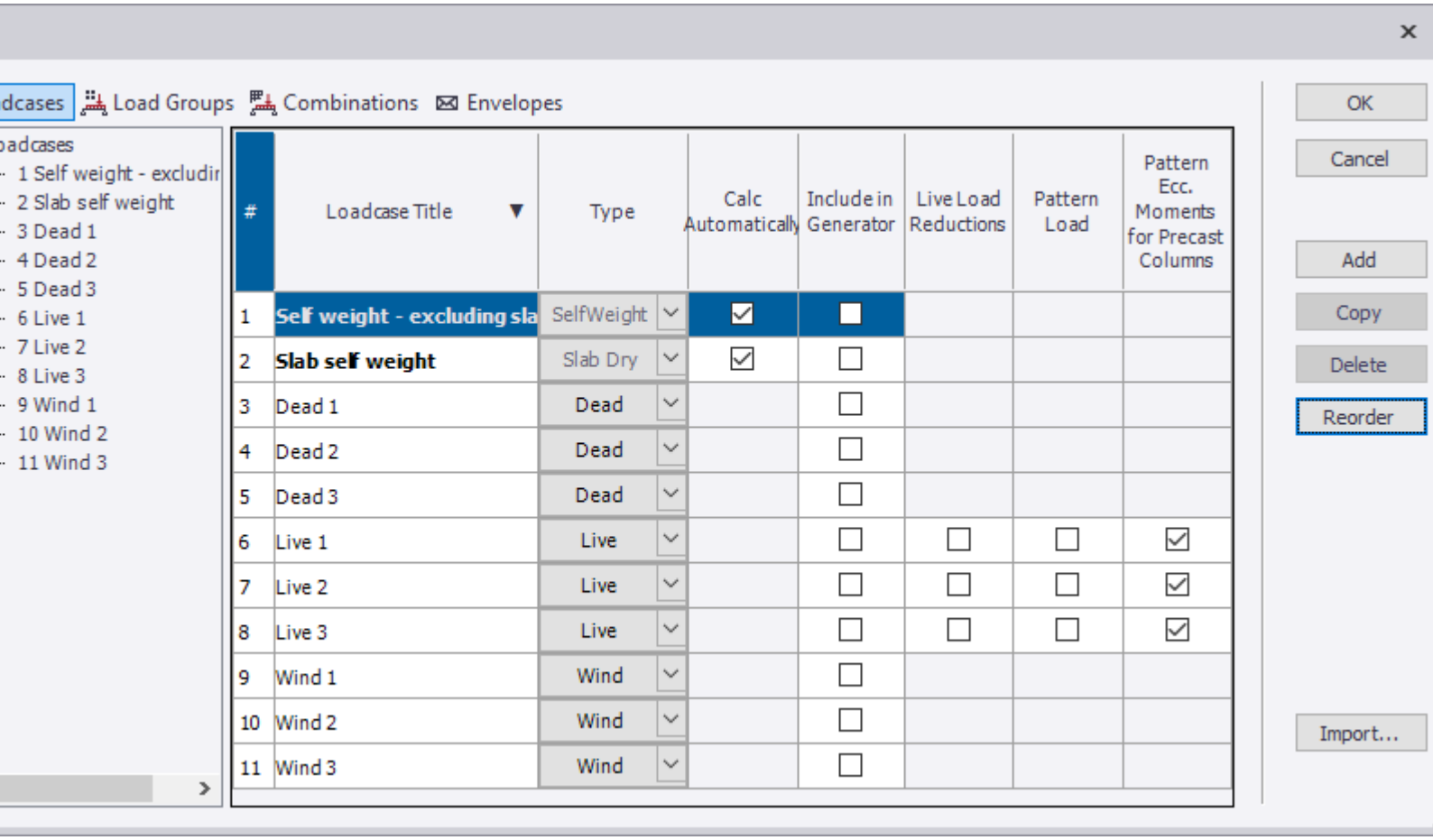
   NOTE A load group can only contain items of one load type.
7. Click **OK**.

   Tekla Structural Designer adds the new load group to the list of load groups.

Inclusive and exclusive load groups example

In this simplified example, loadcases will be arranged into inclusive and exclusive load groups. The resulting load groups will then be used to create some combinations.

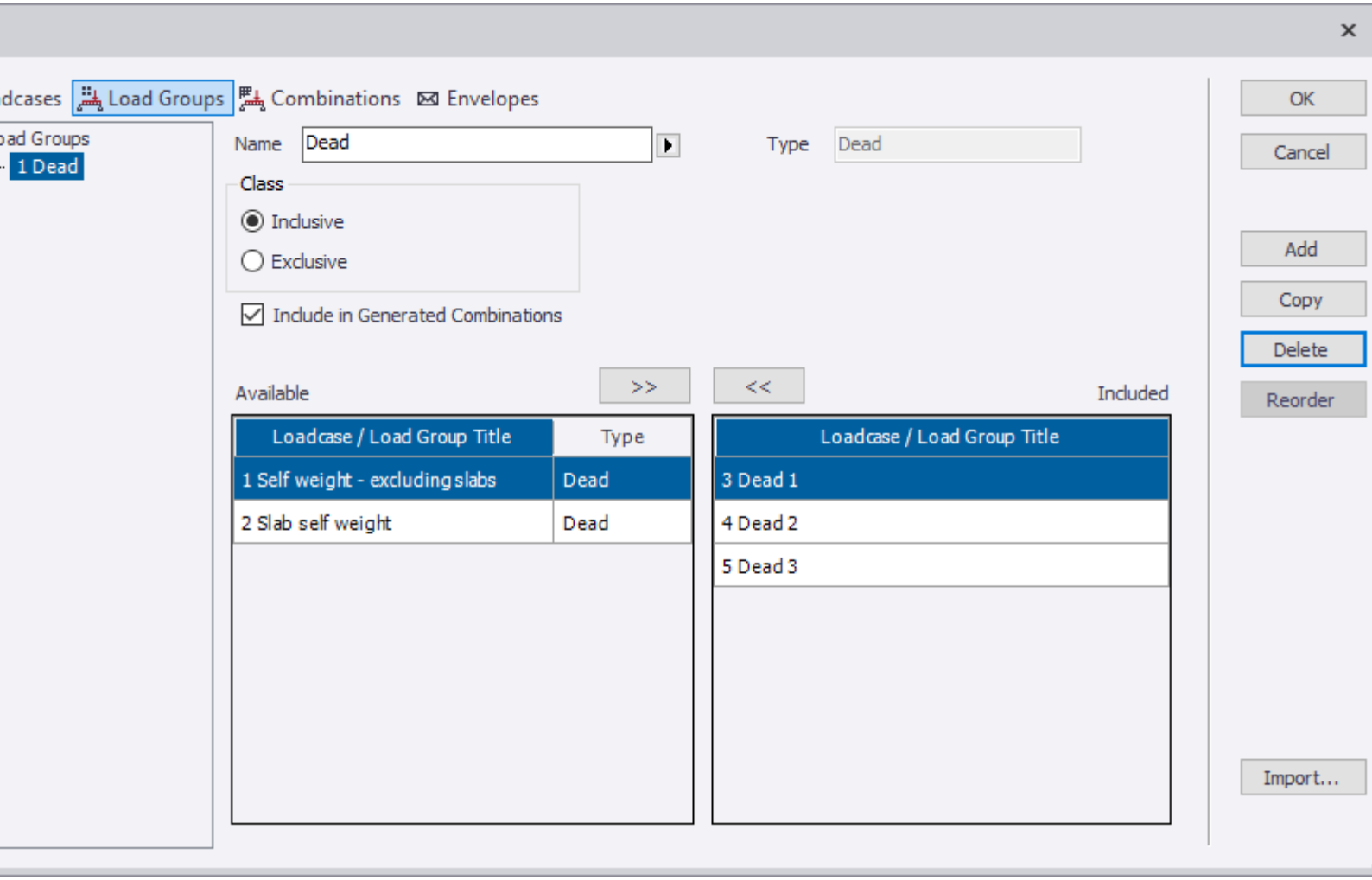
You can work through the example yourself by creating the loadcases shown below.



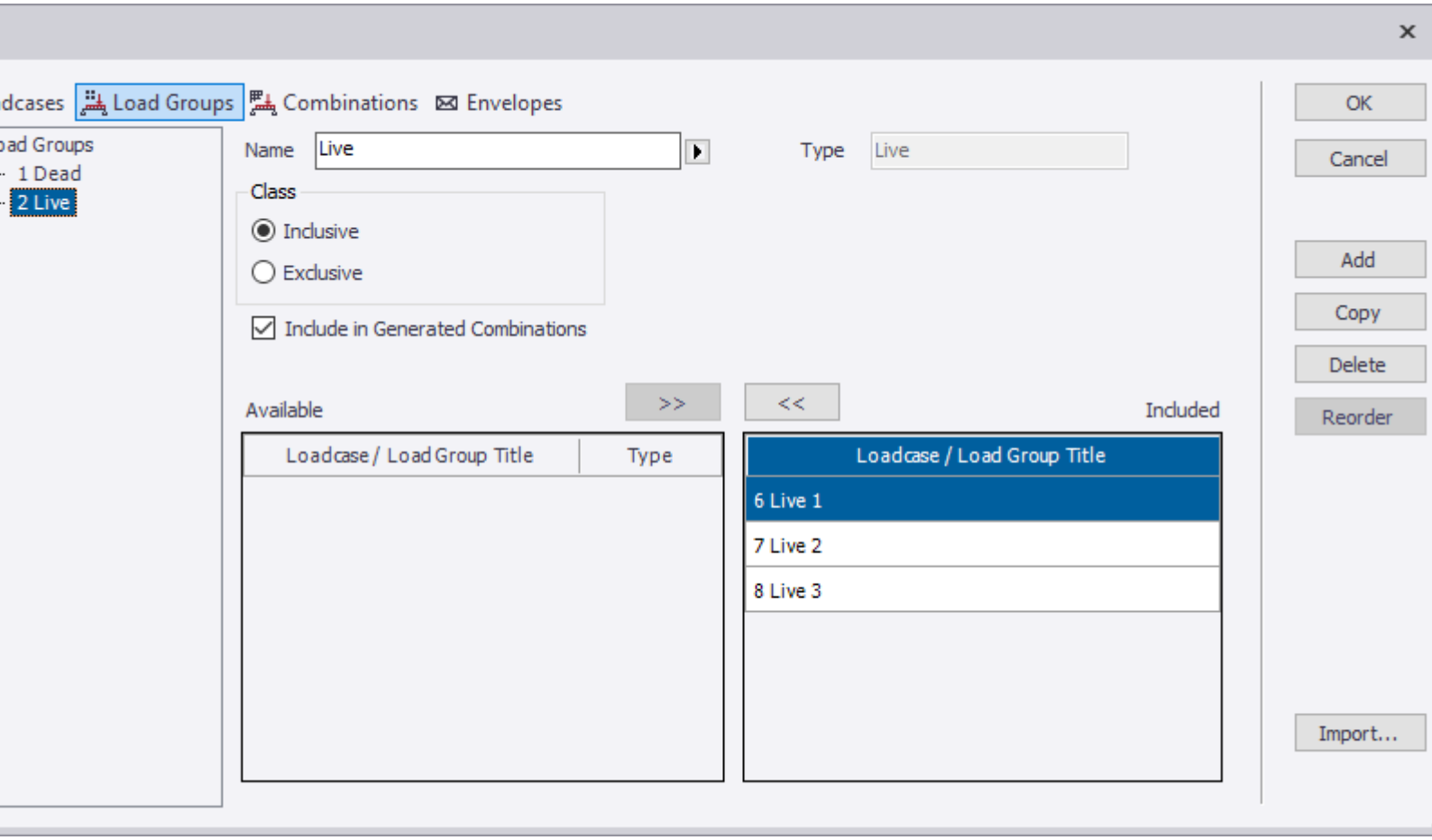
#	Loadcase Title	Type	Calc Automatically	Include in Generator	Live Load Reductions	Pattern Load	Pattern Ecc. Moments for Precast Columns
1	**Self weight - excluding sla**	SelfWeight	☑	☐			
2	**Slab self weight**	Slab Dry	☑	☐			
3	Dead 1	Dead		☐			
4	Dead 2	Dead		☐			
5	Dead 3	Dead		☐			
6	Live 1	Live		☐	☐	☐	☑
7	Live 2	Live		☐	☐	☐	☑
8	Live 3	Live		☐	☐	☐	☑
9	Wind 1	Wind		☐			
10	Wind 2	Wind		☐			
11	Wind 3	Wind		☐			

Once the above loadcases have been specified, the procedure is as follows:

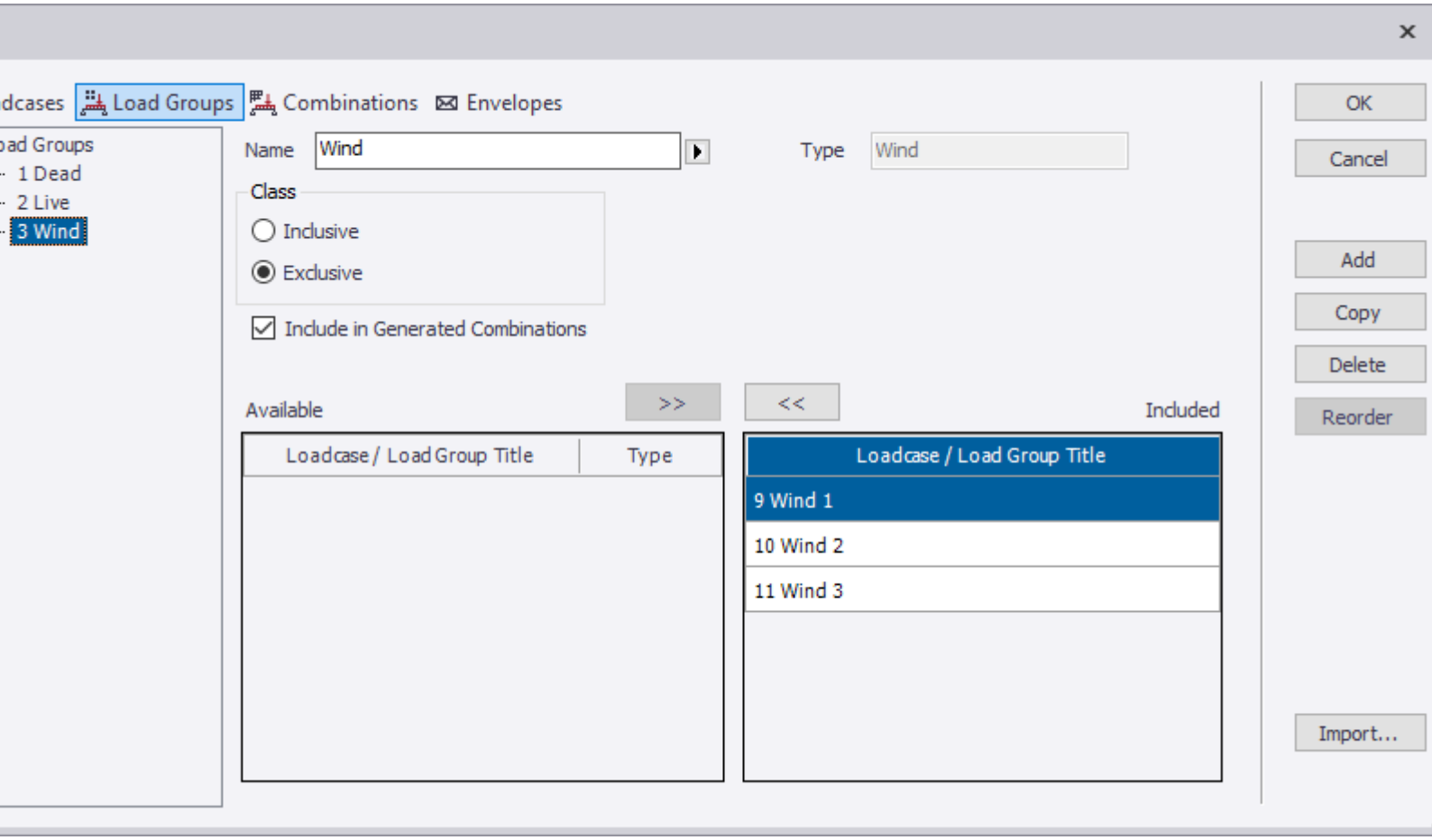
1. From the **Load Groups** tab, a ***Dead*** load group is added and set as ***Inclusive***. From the available loadcases **Dead 1**, **Dead 2**, and **Dead 3** are selected.



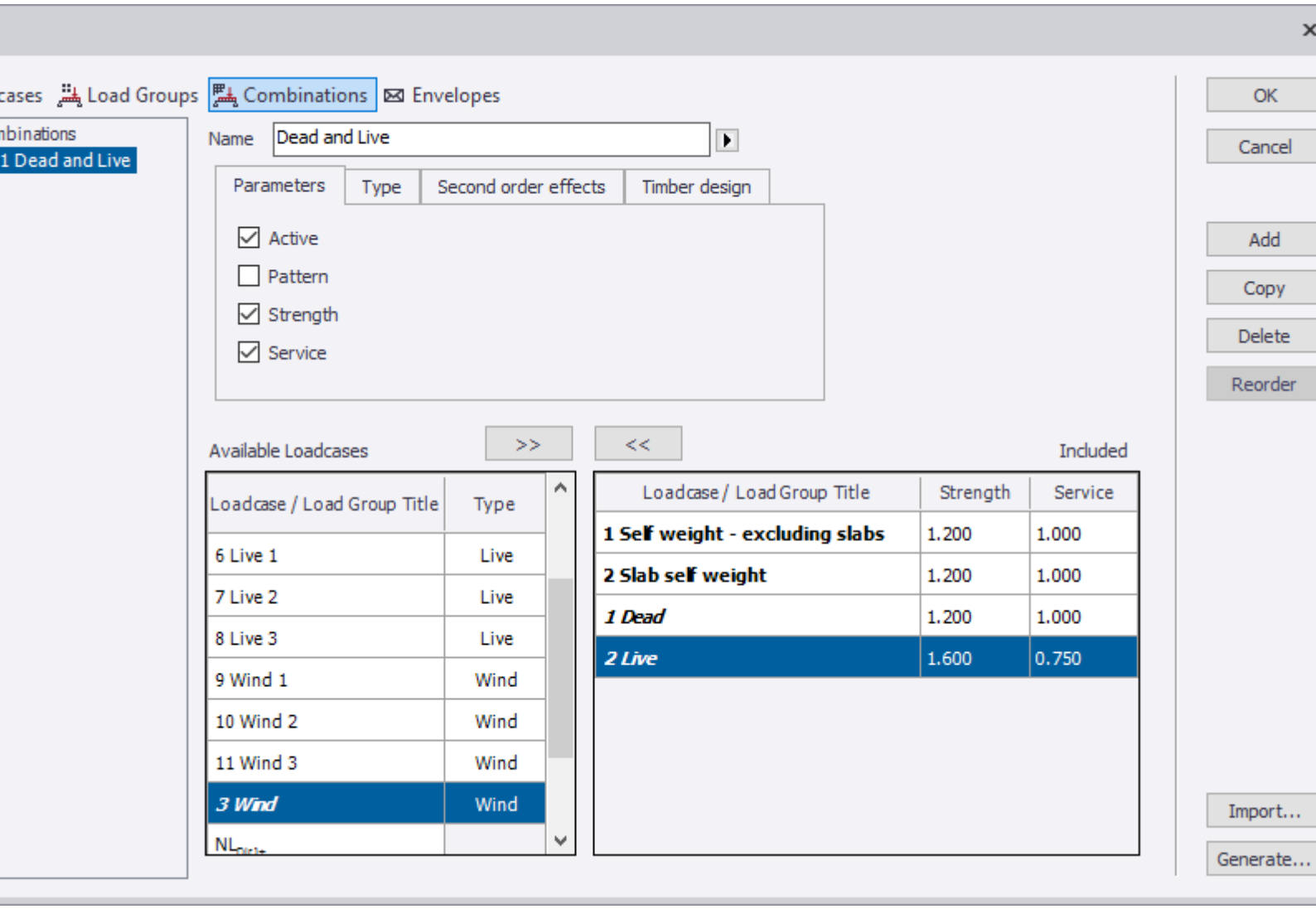
2. Next a ***Live*** load group is added and again set as ***Inclusive***. From the available loadcases **Live 1**, **Live 2**, and **Live 3** are selected.



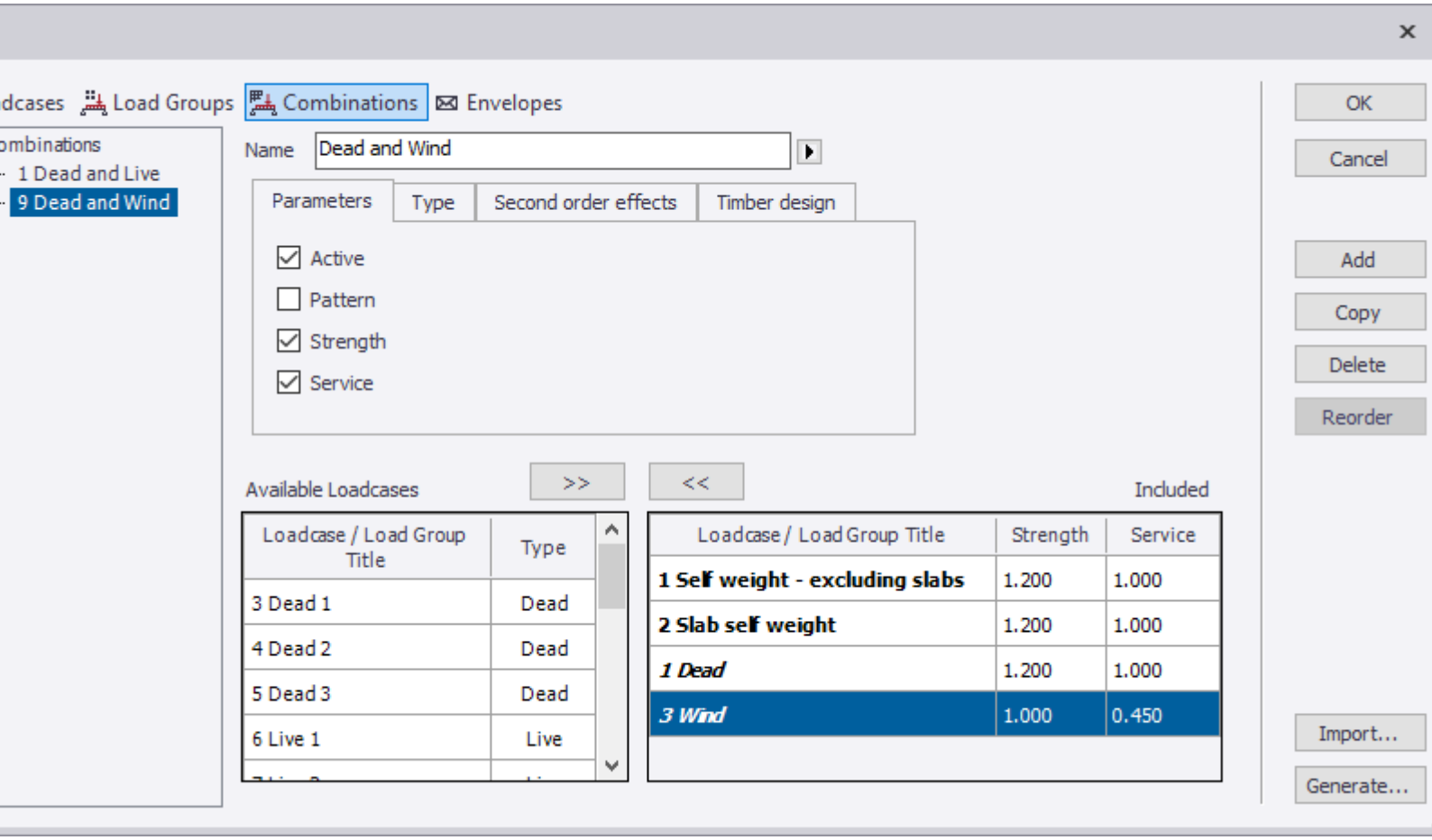
3. Finally a ***Wind*** load group is added. From the available loadcases **Wind 1**, **Wind 2**, and **Wind 3** are selected. Because there is a requirement for each of these loadcases to be placed in a separate combination the class is set as ***Exclusive***.



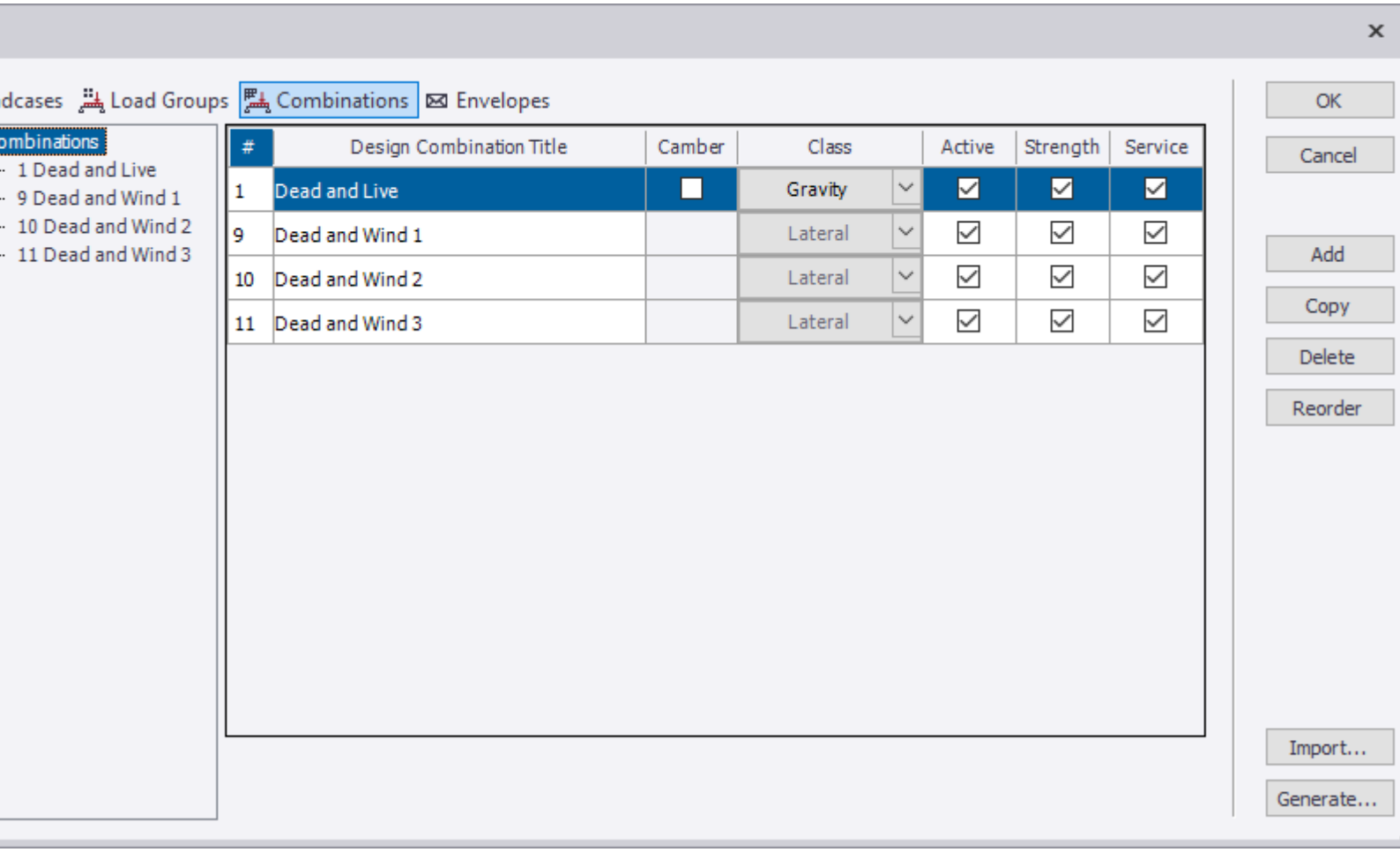
4. From the **Combinations** tab, a **Dead and Live** combination is added. From the available loadcases the **Self weight - excluding slabs** and the **Slab self weight** loadcases are selected. From the same list the ***Dead*** load group and the ***Live*** load group are also selected. (Note that in the "Available" and "Included" lists load groups are shown in bold italics.)



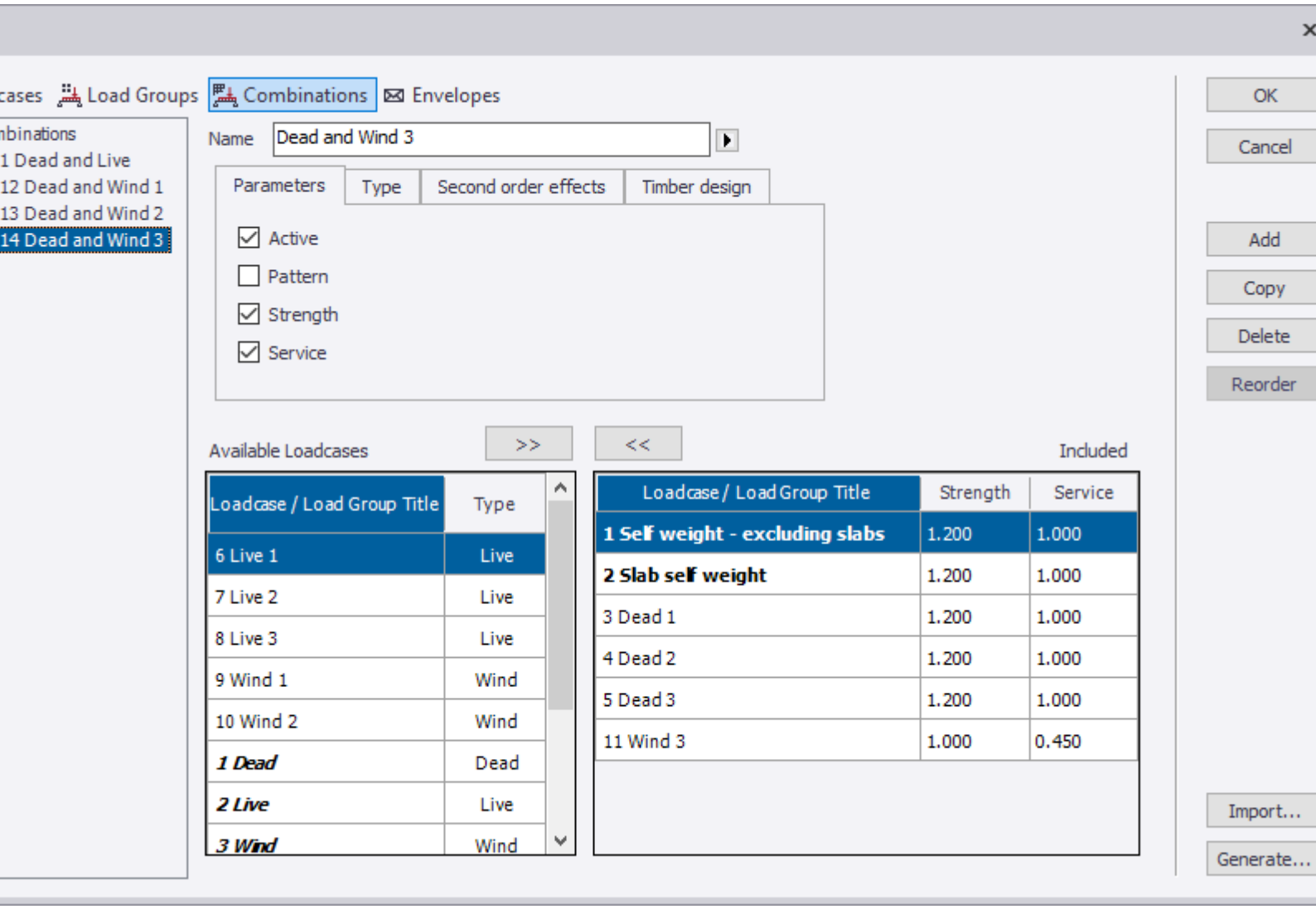
5. Next a **Dead and Wind** combination is added. For this combination the self weight loadcases, the ***Dead*** load group and the ***Wind*** load group are selected.



6. The Loading dialog then has to be closed and reopened in order to create the required combinations from the load groups. Because the ***Wind*** load group was exclusive, three separate combinations have been created to cater for each of the group's three wind loadcases.



7. Once combinations have been created from load groups in this way, the load groups are replaced in the list of included loadcases by the loadcases themselves, (with appropriate factors according to load type).



Create and manage load combinations

Load combinations allow you to assemble sets of loadcases, applying the appropriate factors for the strength and service condition. These factors are specific to the design code that you are using.

To create and manage load combinations, see:

- Load combination classes (page 339)
- Generate load combinations automatically (page 340)
- Create load combinations manually (page 341)
- Create modal mass combinations (page 343)
- Import loadcases and combinations from a spreadsheet (page 343)

- Renumber and/or reorder all load combinations (page 347)

Load combination classes

Combinations fall into nine different classes, with a number of options available for each of the classes:

Combination class	Description	Active	Strength	Service
Construction Stage	Describes the loading that will apply for the pre-composite stage design of composite steel beams.	N/A	N/A	N/A
Gravity	Automatically assigned to combinations consisting of gravity loadcases only (self weight, dead, slab dry, slab wet, imposed, roof imposed, snow).	On/Off	On/Off	On/Off
Lateral	Automatically assigned to combinations containing lateral loadcases (e.g. notional loads or wind), with or without gravity loadcases.	On/Off	On/Off	On/Off
Seismic	Automatically assigned to combinations containing seismic loadcases, with or without gravity and/or lateral loadcases.	On/Off	On/Off	On/Off
Seismic RSA	Automatically assigned to combinations containing seismic RSA loadcases, with or without gravity and/or lateral loadcases. Typically created by running the seismic wizard.	On/Off	On/Off	On/Off
Base Shear	Created by running the seismic wizard, This class is reserved for the Effective Seismic Weight (ASCE 7) or Seismic Inertia (Eurocodes).	On	N/A	N/A
Modal Mass	Describes the mass that will be considered during modal (vibration) analysis.	On/Off	N/A	N/A

Combination class	Description	Active	Strength	Service
Staged Construction	Describes the loading that will apply to each of the construction stage models considered in a staged construction analysis	N/A	N/A	N/A
Footfall Mass	Created in the footfall analysis view and describes the mass that will be considered during footfall modal analysis.	On/Off	N/A	N/A

- **Active**: Selecting and clearing the option switches the combination on and off for analysis and design.
- **Strength**: If the **Strength** option is not selected and the combination is active, the combination is not assessed for strength checks.
- **Service**: If the **Service** options is not selected and the combination is active, the combination is not assessed for serviceability checks.

Generate load combinations automatically

The easiest way to create load combinations is to generate the combinations automatically using the **Combination Generator**.

NOTE **Construction Stage** combinations must be created manually.

Open the Combination Generator

1. On the **Load** tab, click  **Combination**.

   The Loading dialog box (page 1557) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.

2. Click **Generate...**

   The **Combination Generator** dialog box opens appropriate to the regional code being worked to.

On completion of the Combination Generator

Once you have specified the necessary information within the combination generator you are returned to the **Combinations** page. On this page you can:

- Rename combinations.
- Review the options that have been applied to an individual combination.

- Specify if camber is to be applied to a combination.
- Choose which combinations are active for design.
- Choose which combinations are to be checked for strength during design.
- Choose which combinations are to be assessed for deflection during design.

TIP To review the options that have been applied to an individual combination, click the combination name in the list on the left side of the **Loading** dialog box.

Create load combinations manually

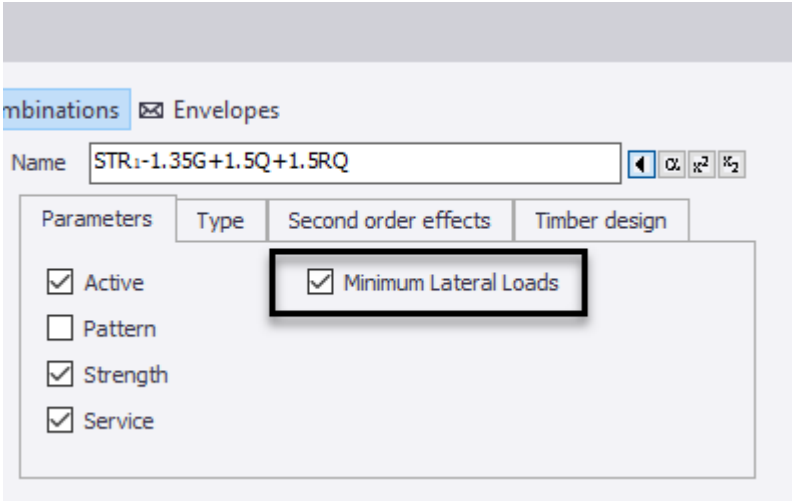
If necessary, you can create load combinations manually. In this case, Tekla Structural Designer uses a default factor for each loadcase when you add it to the combination.

1. On the **Load** tab, click  **Combination**.

   The Loading dialog box (page 1557) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. Click **Add**.
3. In **Design Combination Title**, name the combination.
4. In **Class**, select the combination class.
5. Depending on the combination class that you selected, do some of the following:
   - Select **Active** in order to include the combination in analysis or design.
   - Select **Strength** in order to assess the combination for design.
   - Select **Service** in order to assess the combination for deflection.
6. In the left side pane of the **Loading** dialog box, click the combination name.
7. Select the loadcases that you want to add in the combination, and click **>>**.

NOTE If Min Lateral Load cases (of type MLL) exist, you cannot add these to your combination until one of the existing gravity combinations has nominated for calculating the values. To do this, select the

required combination and check 'Minimum Lateral Loads' on the Parameters tab as shown below.



8. Click **OK** to save the load combination.

   Tekla Structural Designer adds the new load combination to the list of combinations in the **Loading** list.

   TIP To review the factors and options that have been applied to a combination, click the combination name on the left side pane of the **Loading** dialog box.

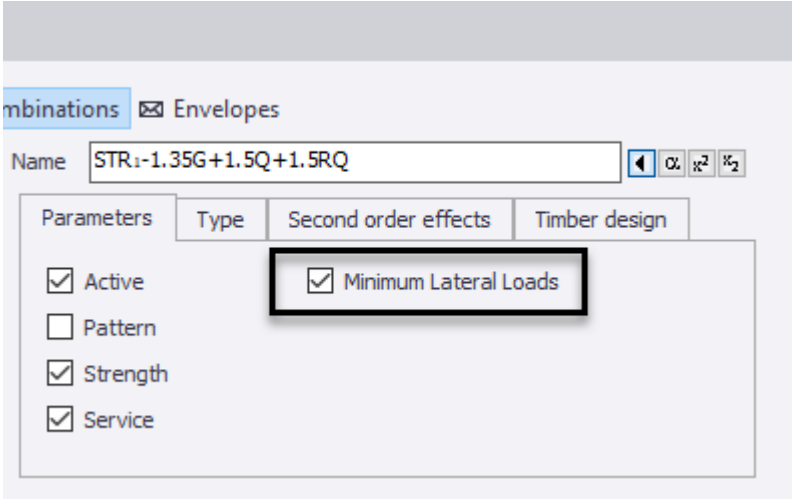
Nominate a gravity combination to be used for minimum lateral load calculations

If minimum lateral loads are required, you must nominate a gravity load combination to be used for the minimum lateral load calculations.

1. On the **Load** tab, click **Combination**.

   The Loading dialog box (page 1557) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.

2. Select an existing gravity load combination in the left hand pane.
3. On the Parameters tab, check 'Minimum Lateral Loads'.



4. Click **OK**.

Create modal mass combinations

1. On the **Load** tab, click **Combination**.

   The Loading dialog box (page 1557) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. Click **Add**.
3. In **Design Combination Title**, name the combination.
4. In **Class**, set the combination class to **Modal Mass**.
5. In the left side pane of the **Loading** dialog box, click the combination name.
6. Select the loadcases that you want to add in the combination, and click **>>**.
7. On the **Applied mass** tab, set the directions to be considered, and if necessary, specify the level below which mass can be ignored.
8. On the **Second order effects** tab, define the amplifier that you want to apply to the combination.
9. Click **OK** to save the load combination.

   Tekla Structural Designer adds the new load combination to the list of combinations in the **Loading** list.

Import loadcases and combinations from a spreadsheet

Loadcases and combinations defined in an Excel Spreadsheet can be imported via the 'Import...' button on the Loading dialog.

The data must be in the form of a matrix of combination factors in which loadcases form the row headers and combinations the column headers or vice-versa. Where a matrix value is blank the associated case is omitted from the combination. The load type can be set by including in the loadcase name "DL" for dead load, "LL" for Imposed and "Wind" for Wind (without quotes).

The selected spreadsheet's Worksheets are automatically listed and one is selected for import.

Flexible controls allow definition of whether Loadcases are listed in a column and combinations in a row or vice-versa and selection of the case and combination column/row sources. Identified cases and combinations can be reviewed prior to import and the case Type can be edited as necessary.

NOTE The following default/ automatically determined loadcases are not supported by this feature and would need to be manually added to combinations after import; Self weight - excluding slabs, Slab self weight, Notional Loads (EHF, NHF and NL).

Example

We want to import the following loadcases and combinations:

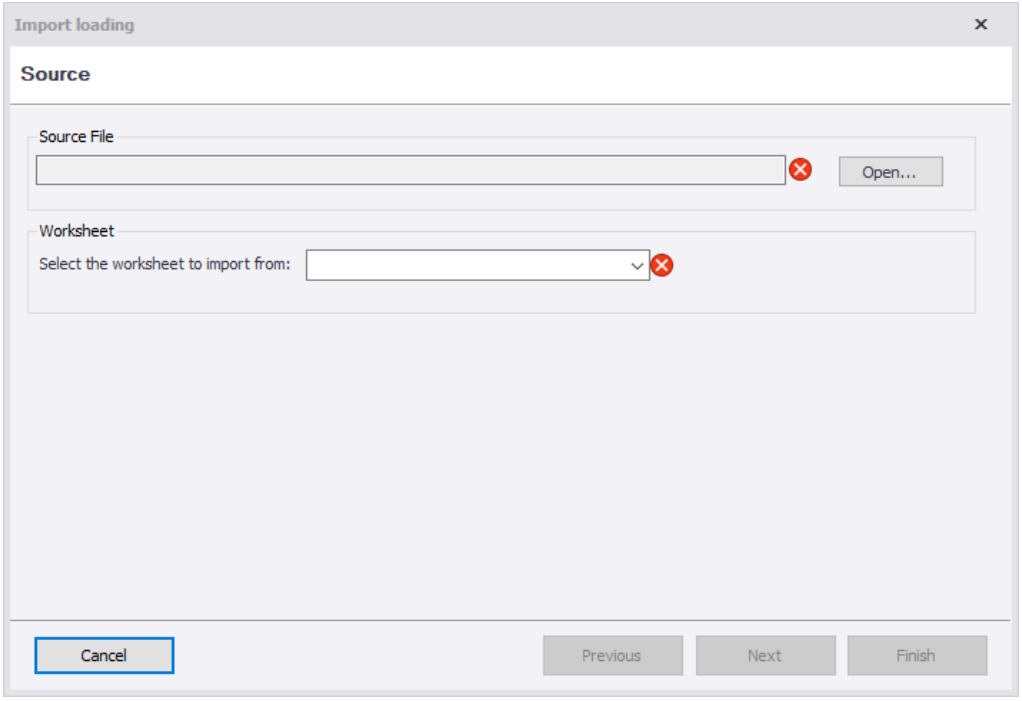
Combination Name	Loadcase	Strength Factor
dead + imposed	DL Dead	1.35
	LL Imposed	1.5
Dead+Imp+Wind Accomp	DL Dead	1.35
	LL Imposed	1.5
	Wind	0.75
Dead+Imp Accomp +Wind	DL Dead	1.35
	LL Imposed	1.05
	Wind	1.5
Dead+Wind	DL Dead	1.35
	Wind	1.5

1. In an Excel spreadsheet a matrix of loadcases and combinations is set out like this:

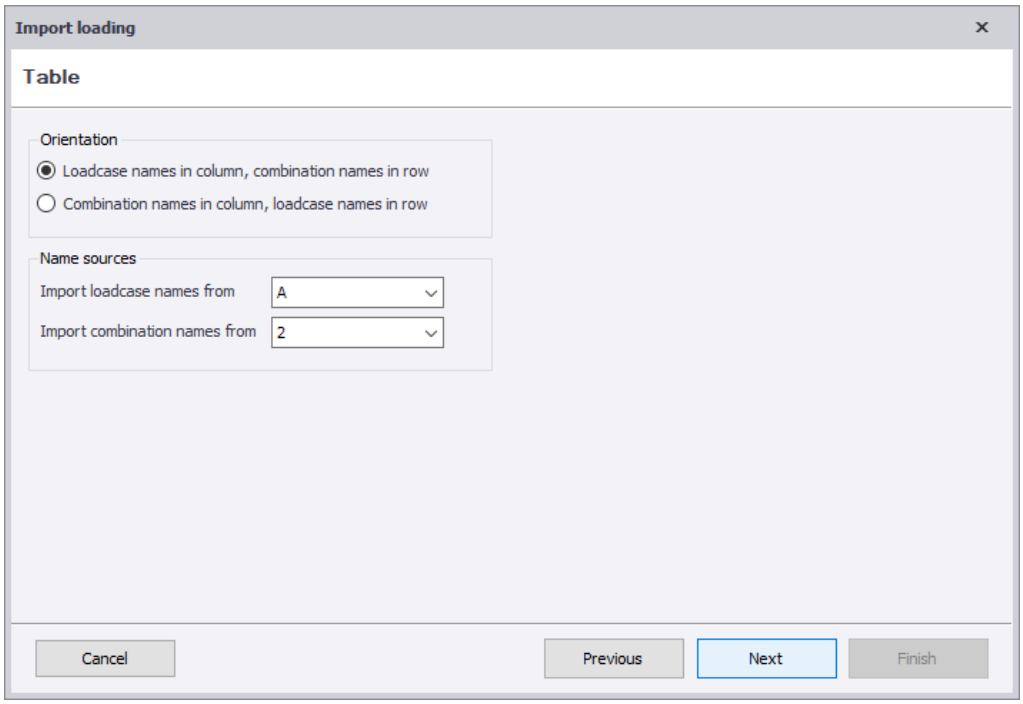
	A	B	C	D	E
1		Combinations			
2	Cases	Dead+Imposed	Dead+Imp+Wind Accomp	Dead+Imp Accomp+Wind	Dead+Wind
3	DL Dead	1.35	1.35	1.35	1.35
4	LL Imposed	1.50	1.50	1.05	
5	Wind		0.75	1.50	1.50
6					

NOTE An empty cell is used to indicate a Loadcase is not to be included in particular Combination.

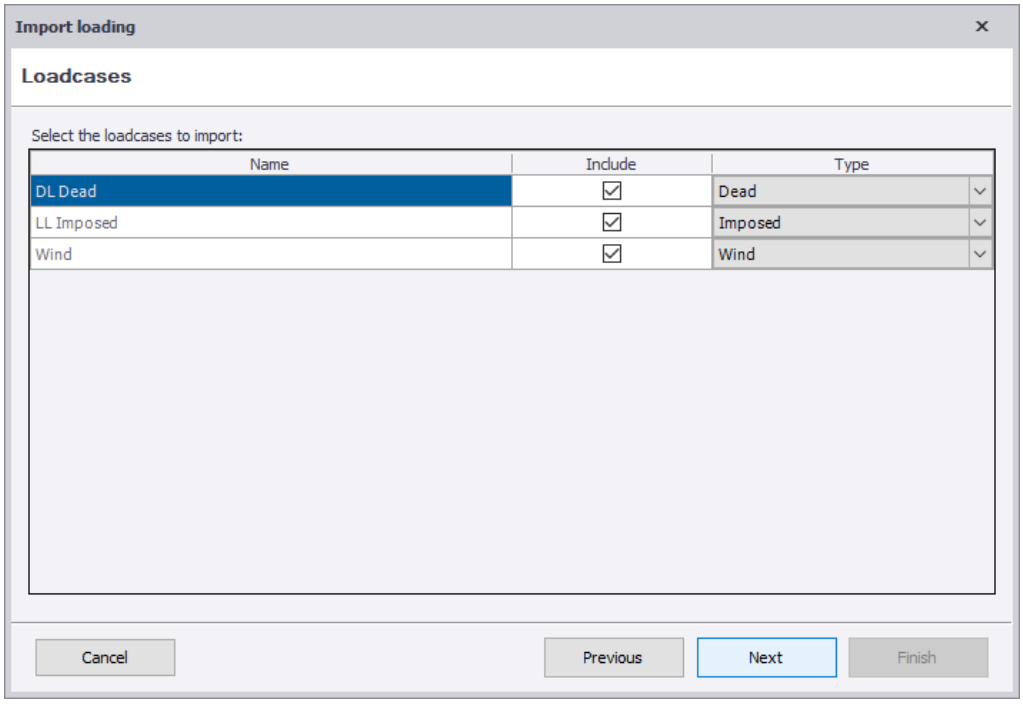
2. Starting from either the Loadcases or Combinations dialog in Tekla Structural Designer, click Import to start the Import loading wizard, the first page of which is:



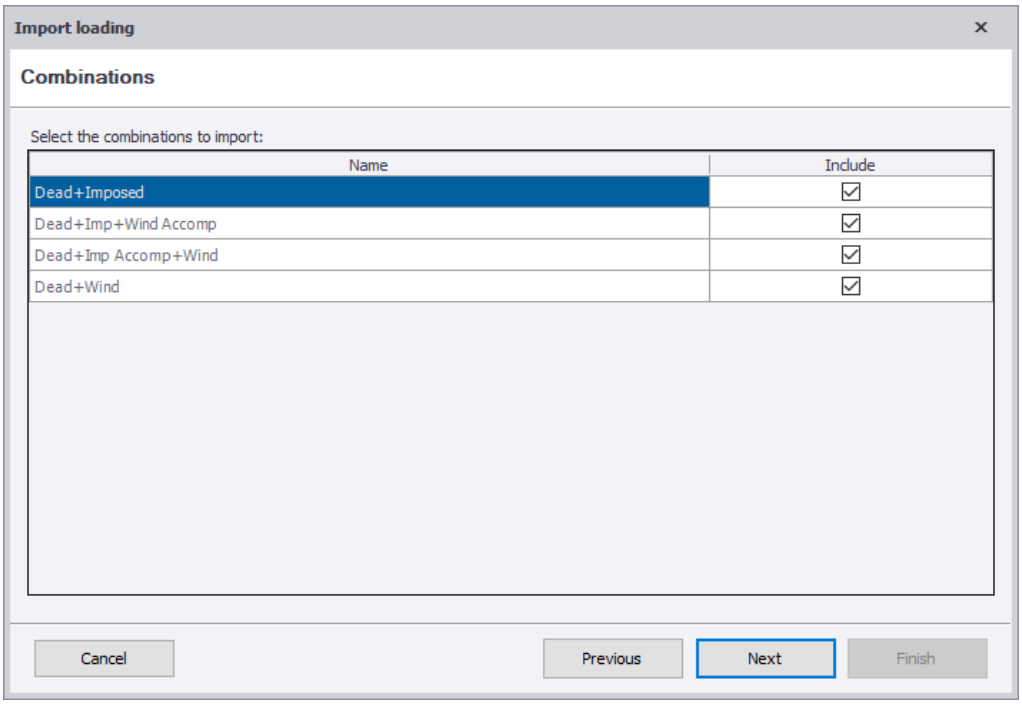
3. Click **Open ...** to navigate to the Excel file and the Sheet within the Excel file which contains the matrix and then click **Next**.
4. In the next page of the dialog you then choose the orientation of your matrix and confirm the Excel column & row which contain the Loadcase and Combination names (i.e. the matrix column and row headers). For this example the correct orientation and name sources are:



5. The next page of the dialog allows you to confirm/amend Loadcase options (a) **Include** (in the imported loadcases) - default is Include checked on, and (b) **Type** (the Loadcase type from Dead, Slab Dry, Slab Wet, Live, etc) - default is Dead for those names that can't be identified automatically:



6. The next page of the dialog allows you to confirm/amend the Combination option **Include** (in the imported combinations) - default is checked on:



7. The final page of the dialog summarizes the number of loadcases and combinations that will be imported, after which click **Finish** to complete the import process.
8. In the Loading dialog review the combinations, amend the combination classes where required, and if necessary include additional loadcases (e.g. automatically determined self weight).

Renumber and/or reorder all load combinations

When you delete load combinations from the **Loading** dialog box, the remaining load combinations retain their original loadcase number. If necessary, you can reorder the remaining load combinations in sequence and reorder them if required. For more information, see the following instructions.

1. On the **Loading** tab of the **Project Workspace**, right-click **Combinations**.
2. In the context menu, select **Reorder**. The combinations are automatically reordered and the numbering updated to suit.
3. In the dialog you then have the opportunity to move combinations up or down the order manually if required.

NOTE The **Reorder** command can also be accessed directly from the (page 1557).

Create envelopes

You can use envelopes to view analysis results for multiple combinations simultaneously. When you do so, Tekla Structural Designer displays the maximum positive and negative values along each member from any combination included in the envelope.

NOTE When using envelopes, note the following:

- If you have defined patterned load combinations, you only need to include the base case pattern combination in the envelope. This way, Tekla Structural Designer automatically includes all pattern combinations derived from the base case in the envelope.
- You can include gravity, lateral, and seismic combinations in the same envelope.
- You should not include seismic RSA combinations in envelopes, as Tekla Structural Designer currently cannot display the results.

Create envelopes

1. On the **Load** tab, click **Envelope**.

   The Loading dialog box (page 1557) opens on the **Envelopes** page. This page displays all currently existing envelopes.
2. Click **Add**.
3. Name the envelope.
4. On the left side pane of the **Loading** dialog box, click the name of the envelope.
5. Select the load combinations that you want to add in the envelope, and click **>>**.
6. Click **OK** to save the envelope.

Create and manage load patterns

Having live/imposed loads applied only to a portion of the structure can produce a more unfavorable loading than the "fully loaded" condition. More unfavorable loading conditions can be considered in Tekla Structural Designer by the use of load patterns.

Overview of load patterns

NOTE When you apply pattern loading to live/imposed loads, the factors of the loaded and unloaded spans are specific to the design code that you are using.

The basic steps of applying pattern loads

1. Set individual live/imposed loadcases to be patterned according to your needs.

   These loadcases are referred to as fully loaded pattern loadcases.

2. Set the gravity combinations that contain live/imposed loadcases to be patterned according to your needs.

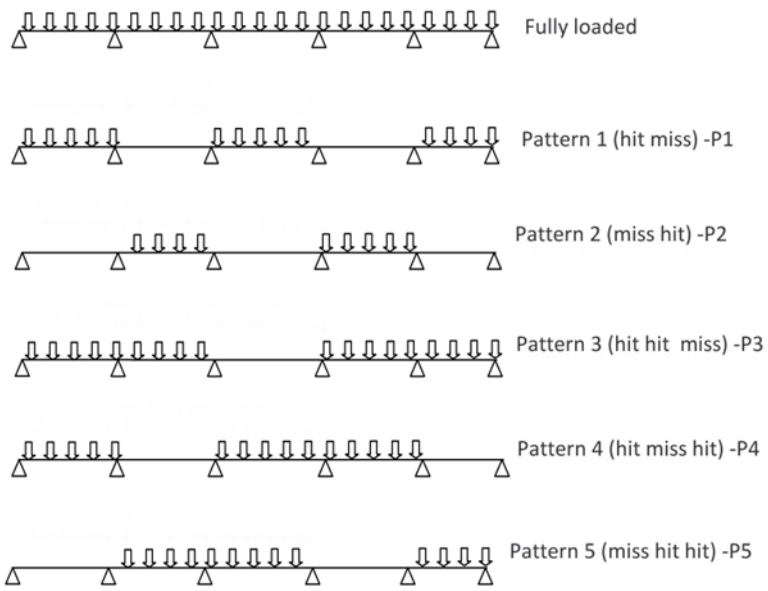
   These gravity combinations are referred to as base case pattern combinations.

After load decomposition, the building analysis automatically sets up the pattern cases for continuous *concrete* beams, (cases are also set up for continuous *steel* beams if the 'Use load patterning for steel beams' option is selected in Model Settings> Loading > General (page 1248)). Patterns are set up in Dir1 and Dir2 directions as follows:

- 10 pattern cases for each pattern gravity combination
- 5 pattern cases for beams along Dir1 and 5 for beams along Dir2

NOTE By default, patterns are only applied to beam loads, and slab loads that have been decomposed on to beams. Loads applied to meshed slabs should be manually patterned using engineering judgement.

Pattern loadcases



A pattern combination containing patterned live/imposed loadcases results in 11 combinations: the base case combination, and 10 pattern combinations derived from the base combination.

If you later modify the building geometry, you must run **Update Patterns** to ensure the load patterns reflect the changes.

NOTE Tekla Structural Designer contains a set of 10 pattern combinations associated with each fully loaded pattern combination. The pattern combinations are the same for beams and slabs.

The rules of applying load patterns to slabs and beams

Slab load patterning only applies where two-way slabs have been meshed in the solver model:

- in FE chasedown analysis
- in 3D analysis where two-way slabs are set as meshed

Therefore, the slab load pattern setting has no affect on:

- one-way spanning slabs
- two-way slabs not meshed in 3D analysis, as the slab loads are being decomposed to beams and walls prior to creation of the solver model
- two-way slab in Grillage chasedown analysis, as the slab loads are being decomposed to beams and walls prior to creation of the solver model

Therefore, in 3D analysis and Grillage chasedown analysis:

- When a beam is set to **Full Load**, it receives the full decomposed load from adjacent unmeshed two-way slabs, irrespective of whether the slabs themselves are set to **Full Load** or **Min Load**.
- When a beam is set to **Min Load**, it receives the min decomposed load from adjacent unmeshed two-way slabs, irrespective of whether the slabs themselves are set to **Full Load** or **Min Load**.

Apply patterning to live/imposed loadcases

If necessary, you can apply load patterning to live/imposed loadcases in a combination. In order to do so, see the following instructions.

1. On the **Load** tab, click  **Loadcases**.

   The Loading dialog box (page 1557) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Click the live/imposed loadcase to which you want to apply patterning.
3. Select the **Pattern Load** option.
4. Click **OK**.

Apply patterning to load combinations

You can only apply load combinations to gravity combinations, as lateral and seismic combinations do not consider pattern loading. For detailed information on how to apply pattern loading to load combinations, see the following instructions.

1. On the **Load** tab, click  **Combination**.

   The Loading dialog box (page 1557) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. On the left side pane of the **Loading** dialog box, select the name of the load combination.
3. In **Parameters**, ensure that the **Pattern** option is selected.
4. Click **OK**.

NOTE If you wish, you can use pattern loading for every gravity combination in your model. However, doing so may create many additional combinations.

Update load patterns

Tekla Structural Designer applies patterning to beam loads automatically. However, if you need to apply patterning to slab loads for slab design or other purposes, you must update the load patterns manually.

1. On the **Load** tab, click  **Update Patterns**.
2. In the **Properties** window, select each pattern in order to review the loading status of the existing beams and slabs.
3. To switch the loading status of a slab in the current pattern, click, or box around the slab(s) in the model.

The pattern that has been defined is applied to every imposed/live loadcase that is set to be patterned.

3.2 Apply panel, member, and structure loads

Each of these loads are applied from the **Load** tab.

- Apply panel loads (page 352)
- Apply member loads (page 357)
- Apply structure loads (page 361)

-
- Decompose panel loads (page 370)

See also

Copy loads (page 274)

Apply panel loads

You can use panel loads to apply loads to slab items, walls, roof panels, and wind wall panels.

The following different panel load types are available:

Panel load type	Use
Point load	Can be applied anywhere within an individual or across multiple slab items, walls, roof panels, or wind wall panels.
Patch load	
Polygonal load	
Area load	Entirely covers a slab item, wall, roof panel, or wall panel.
Slab load	Entirely covers all slab items in a parent slab.
Level load	Entirely covers all parent slabs in a level.

Create point loads

1. Open a 2D view of the level within which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Point**.
5. In the **Properties** window, adjust the load details according to your needs.
6. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

7. Click the load position, or press **F2** to type the exact position.

NOTE If you move the slab item or panel by manually selecting & re-positioning its nodes, the point load does not move with the slab or panel.

However, if you move any of the grid lines defining the reference node, the load will move as well.

Create line loads

1. Open a 2D view of the level within which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Line**.
5. In the **Properties** window, adjust the load details according to your needs.
6. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

7. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.
8. Click the end point of the load, or press **F2** to type the exact position.

   Note that when you type the position using the keyboard, the end point is an offset (X, Y) from the load start position.

Create patch loads

1. Open a 2D view of the level within which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Patch**.
5. In the **Properties** window, adjust the load details according to your needs.
6. In the model, click a reference node from which the load position can be offset.

The load position can be the start or end point of any member at the level.

TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

7. Click to specify a corner position of the load, or press **F2** to type the exact corner position.
8. Click to specify the size of the load, or press **F2** to type the size.

   The size is specified as the offset dimensions from the selected corner position.
9. Click to specify the rotation angle of the load, or press **F2** to type the rotation angle.

Create polygonal loads

1. Open a 2D view of the level within which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Polygon**.
5. In the **Properties** window, adjust the load details according to your needs.
6. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.
7. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.
8. Click a corner point of the load, or press **F2** to type the exact position.
9. Repeat step 7 according to your needs.
10. When the polygon is complete, do one of the following:
    - Press **Esc**.
    - Click the first corner that you defined.

    TIP To modify the shape of an existing polygonal load the following:

    a. In the model, select the load.

b. Click one vertex of the load.

c. Click the point where you want to move the vertex.

Create perimeter loads

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Perimeter**.
5. In the **Properties** window, adjust the load details according to your needs.

TIP If necessary, you can select one or both of the following options:

- **Create as line loads**: creates the load as a series of separate line loads along each external edge.
- **Ignore openings**: only creates the load around external perimeters, and not around internal opening perimeters.

6. Click any slab or mat item.

   Tekla Structural Designer creates the load around the external perimeter of every continuous area of slabs at the selected level.

Create variable patch loads

NOTE Variable patch loads do not have to be quadrilaterals. Instead, they can be any polygonal shape.

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Var. Patch**.
5. In the **Properties** window, specify three load values to describe the variable loads.
6. Adjust the remaining load details according to your needs.

7. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

8. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.

9. Click a corner point of the load, or press **F2** to type the exact corner position.
10. Repeat step 8 according to your needs.
11. When the polygon is complete, do one of the following:
    - Press **Esc**.
    - Click the first corner that you defined.
12. Click one of the corners of the polygon to specify the position of **Load 1**.
13. Click another corner to specify the position of **Load 2**.
14. Click a third corner to specify the position of **Load 3**.

    TIP To modify the shape of an existing patch load the following:

    a. In the model, select the load.

    b. Click one vertex of the load.

    c. Click the point where you want to move the vertex.

Create area loads

1. Open a 2D view of the level within which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Area**.
5. In the **Properties** window, adjust the load details according to your needs.
6. Click the panel to which you want to apply the load.

Create variable area loads

Variable area loads can only be applied to non-horizontal slab items and roof panels, as well as wall panels and concrete walls. Therefore, they cannot be applied in a 2D view.

1. Open the 3D view, frame, or sloped plane view in which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Var. Area**.
5. In the **Properties** window, adjust the load details according to your needs.
6. Click the slab item or panel to which you want to apply the load.

Create slab loads

1. Open the view in which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Slab**.
5. In the **Properties** window, adjust the load details according to your needs.
6. To apply the load to all the slab items within a parent slab, in the model, click any slab item within the parent slab.

Create level loads

Click any slab panel in order to apply the load to all slabs within the level.

1. Open the view in which you want to apply the load.
2. Set the view regime to  **Structural View**.
3. In the , select an appropriate loadcase.
4. On the **Load** tab, click **Level**.
5. In the **Properties** window, adjust the load details according to your needs.
6. To apply the load to all slabs within the level, in the model, click any slab item.

Apply member loads

Tekla Structural Designer allows you to create and apply the following types of member loads:

- Full-length uniformly distributed loads (Full UDLs) (page 358)
- Partial-length uniformly distributed loads (UDLs) (page 358)
- Partial-length variable distributed loads (VDLs) (page 358)
- Trapezoidal loads (page 359)
- Point loads and moment loads (page 360)
- Full-length torsional uniformly distributed loads (UDLs) (page 360)
- Partial-length torsional uniformly distributed loads (UDLs) (page 361)
- Partial-length torsional variable distributed loads (VDLs) (page 361)

The above loads can be applied to all one-dimensional member types apart from braces and truss internals, (these latter two types being prevented from having loads applied within their length).

Create full-length UDLs

Full-length uniformly distributed loads (UDLs) can be applied to all one-dimensional member types (with the exception of braces and truss internals) as follows:

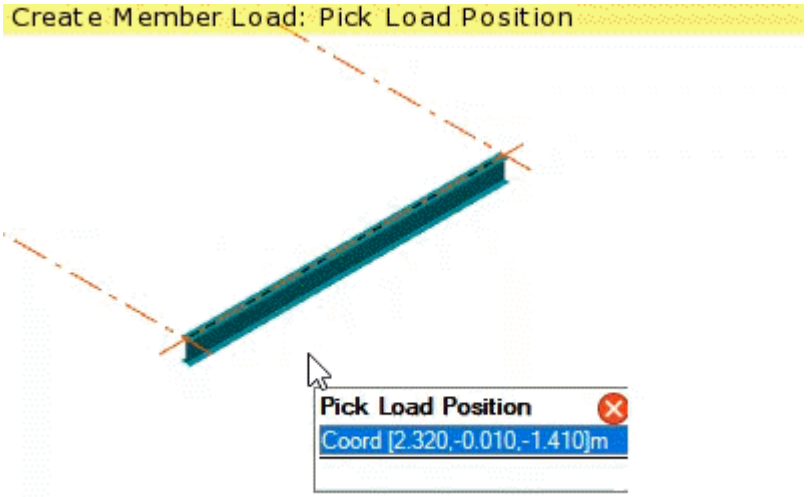
1. Open an appropriate view and set the view regime to  **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Full UDL**.
4. In the **Properties** window, adjust the load details according to your needs.
5. To apply the load, click anywhere along the member.

Create partial-length UDLs or VDLs

Uniformly and variably distributed loads (UDLs and VDLs) can be applied to all one-dimensional member types (with the exception of braces and truss internals) as follows:

1. Open an appropriate view and set the view regime to  **Structural View**.
2. In the , select an appropriate loadcase.

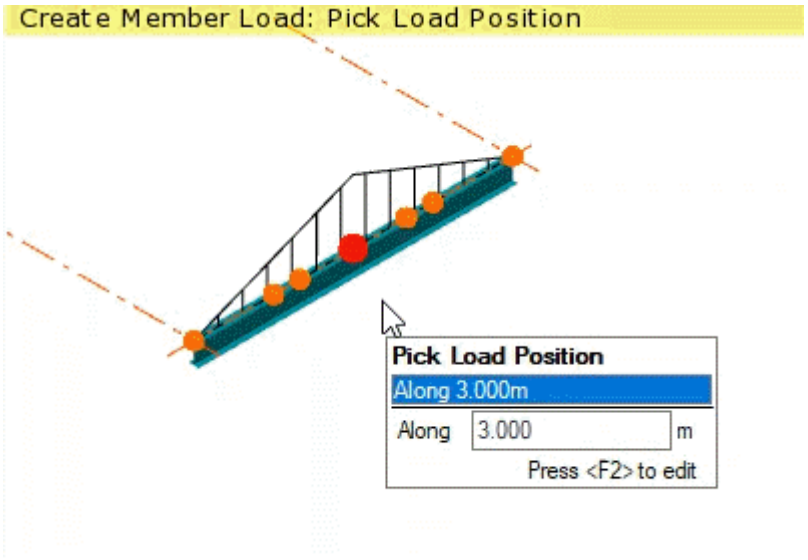
3. On the **Load** tab, click **UDL** or **VDL**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Hover the mouse pointer over the member to which you want to apply the load.
6. Click the start point of the load along the member, or press **F2** to type the exact position.
7. Click the end point of the load along the member, or press **F2** to type the exact position.



Create trapezoidal loads

Trapezoidal loads can be applied to all one-dimensional member types (with the exception of braces and truss internals) as follows:

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Trapezoidal**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Hover the mouse pointer over the member that you want to apply the load to.
6. Click the load position along the member, or press **F2** to type the exact position.



The load position defines the point at which the symmetrical trapezoidal load reaches its maximum intensity.

Create point loads and moment loads

Point loads and moment loads loads can be applied to all one-dimensional member types (with the exception of braces and truss internals) as follows:

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, in the **Member Loads** group click **Point** or **Moment**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Hover the mouse pointer over the member to which you want to apply the load.
6. Click the load position along the member, or press **F2** to type the exact position.

Create full-length torsional UDLs

Full-length torsional uniformly distributed loads can be applied to all one-dimensional member types (with the exception of braces and truss internals) as follows:

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Torsion Full UDL**.
4. In the **Properties** window, adjust the load details according to your needs.
5. To apply the load, click anywhere along the member.

Create partial-length torsional UDLs and VDLS

Partial-length torsional uniformly and variably distributed loads can be applied to all one-dimensional member types (with the exception of braces and truss internals) as follows:

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Torsion UDL** or **Torsion VDL**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Hover the mouse pointer over the member to which you want to apply the load.
6. Click the start point of the load along the member, or press **F2** to type the exact position.
7. Click the end point of the load along the member, or press **F2** to type the exact position.

Apply structure loads

The five types of structure loads that you can apply are:

- Diaphragm loads: applied at any point within a rigid or semi-rigid diaphragm.
- Equipment loads: applied to equipment, (in loadcases other than the dedicated 'equipment' ones).
- Nodal loads (page 369): applied at solver node locations.
- Temperature loads (page 369): global rises in temperature applied to elements or panels.

- Settlement loads (page 369): translations or rotations applied to supports in the support UCS system.

Diaphragm loads and diaphragm load tables

Diaphragm loads are Building Direction 1, Building Direction 2, and Mz torsion loads applied at any point within a rigid or semi-rigid diaphragm.

Typically these might be used for the application of externally determined level loads (for example from Wind Tunnel testing of large/ unusual structures).

The **Diaphragm** load command is available when a 2D plan view is active - it can be used to apply a load to any diaphragm within the current view.

If you have multiple loads to apply at different levels, the **Diaphragm Table** of loads provides a quicker means of doing this - either by typing the loads and their positions directly into the table, or by pasting the data into the table from a spreadsheet.

Add a diaphragm load in a 2D view

To apply a load to any diaphragm within the current 2D view:

Related video

Diaphragm loading

1. Open a 2D view of the level containing the diaphragm to which you want to apply the load and set the view regime to  **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click  **Diaphragm**
4. In the **Properties** window, adjust the load details according to your needs.
5. Click the load position, or press **F2** to type the exact position.

Add a diaphragm load in a 3D view

To apply a load to any diaphragm within the current 3D view:

Related video

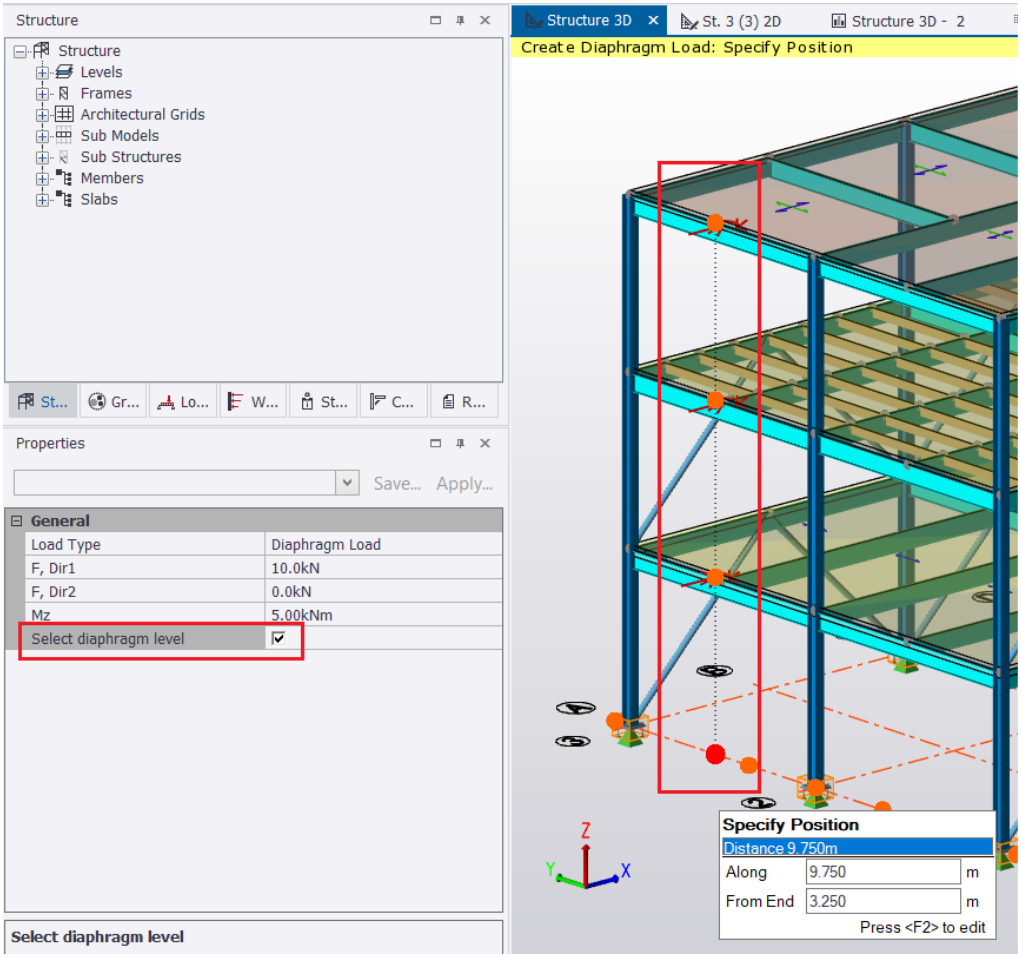
Diaphragm loading

1. Open a 3D view containing the diaphragm to which you want to apply the load and set the view regime to  **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click  **Diaphragm**

4. In the **Properties** window, adjust the load details according to your needs.
5. At the level of the diaphragm, click an existing construction point to apply the load at that position, or hover over an existing member to apply the load at a distance along the member (press **F2** to type the exact position).

Add a diaphragm load in a 3D view by selecting the diaphragm level
In this method, the load can be applied to a diaphragm at any level by selecting an existing point at any level:

1. Open a 3D view containing the diaphragm to which you want to apply the load and set the view regime to  **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click  **Diaphragm**
4. In the **Properties** window, adjust the load details according to your needs.
5. In the **Properties** window, check **Select diaphragm level**
6. At any level, click an existing construction point, or hover over an existing member to specify a distance along the member (press **F2** to type the exact position). A vertical line is displayed with nodes along it at each diaphragm position above or below the identified position.

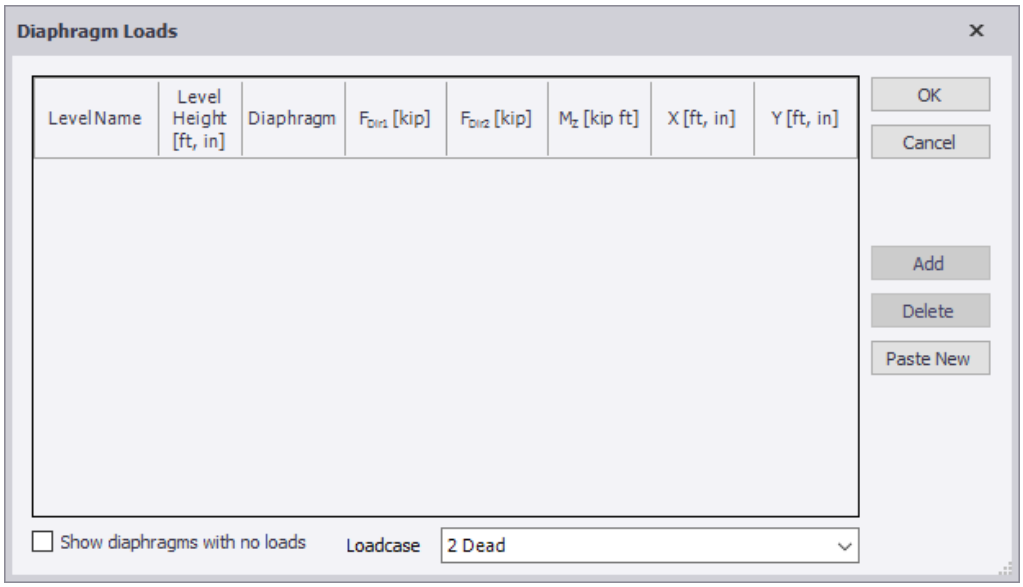


7. Click on a node to apply the load to the diaphragm at that level.

Add multiple loads using the diaphragm load table

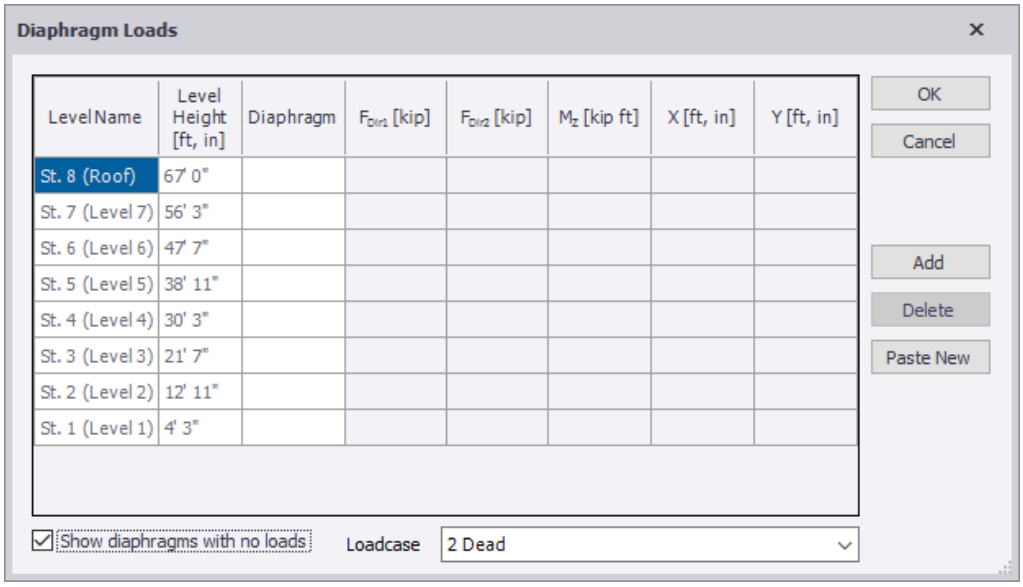
The diaphragm loads table provides a way to quickly apply multiple diaphragm loads at different levels, either by typing the loads and their positions directly into the table, or by pasting the data into the table from a spreadsheet.

1. On the **Load** tab, click **Diaphragm Table**.



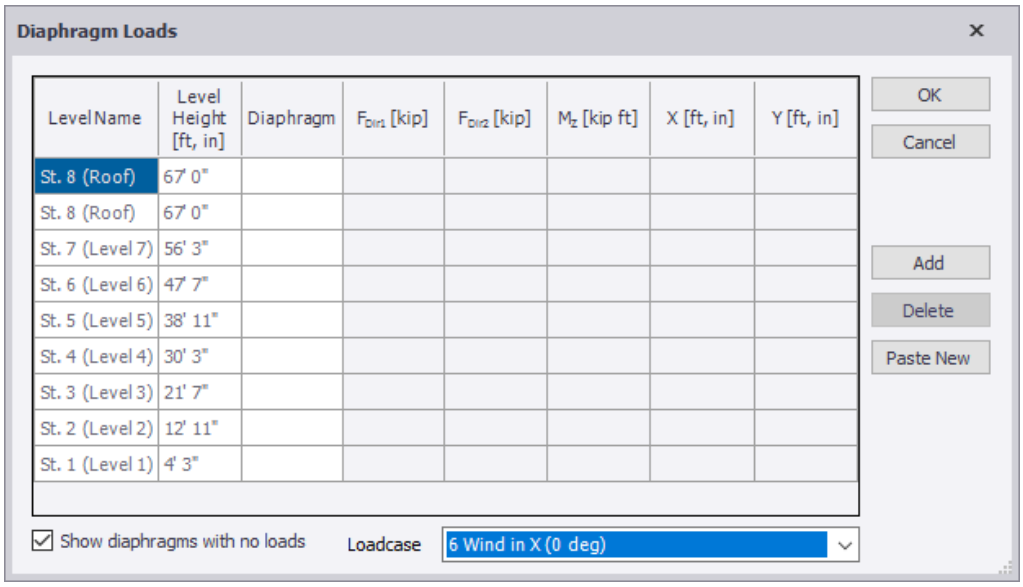
The Diaphragm Loads table opens in a dialog. If any loads had already been added (via the Diaphragm Load command, or the Diaphragm Loads table) these would be listed.

2. In order to see all the separate diaphragms at each level to which loads could be applied, click the **Show diaphragms with no loads** box.

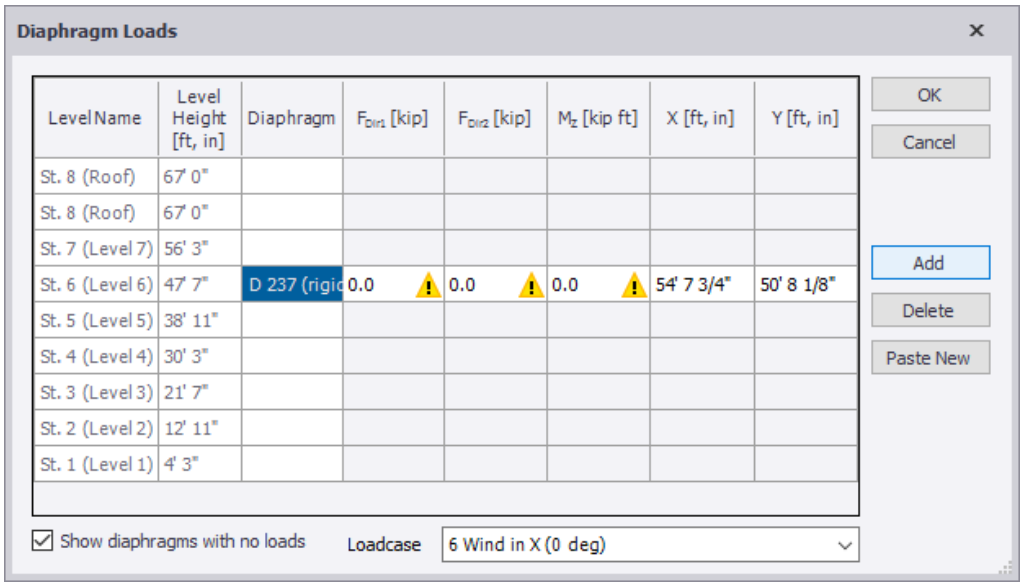


Each level containing a diaphragm is listed, (if more than one diaphragm exists at the same level a separate row is created for each).

3. In the **Loading** list at the bottom of the dialog, select the loadcase within which you want to apply the load.



4. In the table, click on the diaphragm level to which you want to apply the load, then click **Add**.



A row is displayed for entering the load. The initial default for the load position is simply the middle point in X and Y of the diaphragm extremities.

5. Enter the load values and the actual X and Y coordinates.
6. Add further rows of loading data as required, either at the same level or other levels.

NOTE Multiple loads can be applied to a single diaphragm if required.

7. Click **OK** when done.

   The loads are displayed on the model iat their defined locations.

NOTE To edit existing loads in the table, simply open it once more. To delete loads from the table, select the diaphragm in the table and click **Delete**.

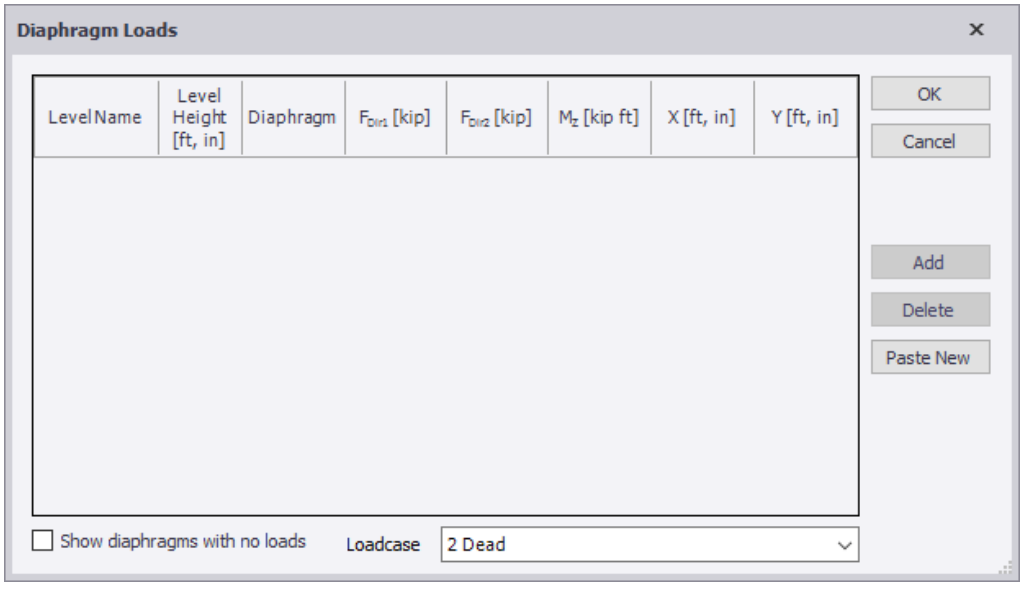
Paste multiple loads into the diaphragm load table

If you have been provided with the loading data in the form of a spreadsheet you can simply paste the data directly into the table.

NOTE The data must includes a Z (Level height) column and you must ensure that it is in the following order (which is not the same order as in the diaphragm loads table):

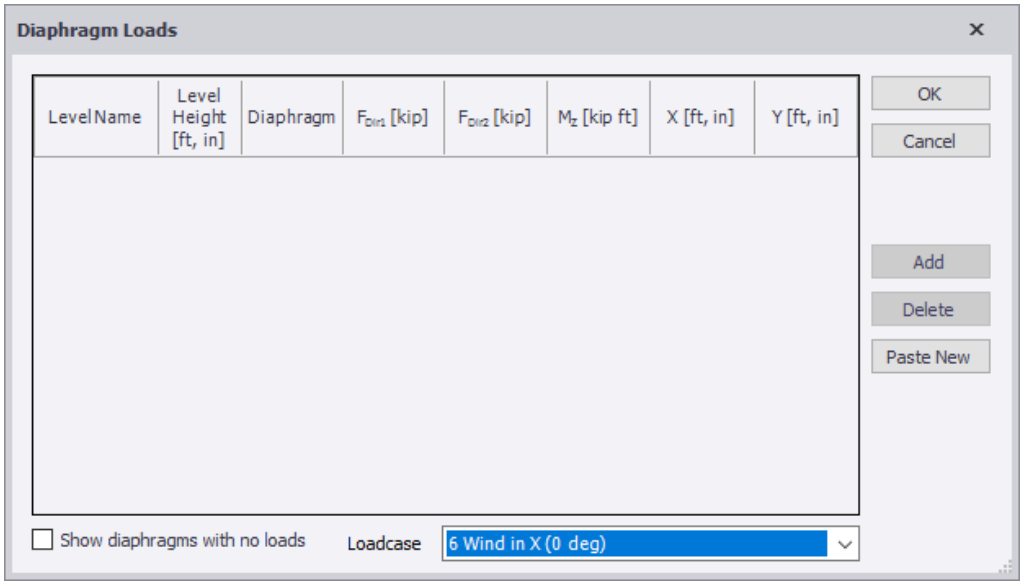
1. Force in direction 1
2. Force in direction 2
3. Torsion moment
4. Position X
5. Position Y
6. Position Z

1. Open the spreadsheet containing the data to be pasted and copy the contents to the clipboard.
2. In Tekla Structural Designer, on the **Load** tab, click **Diaphragm Table**.

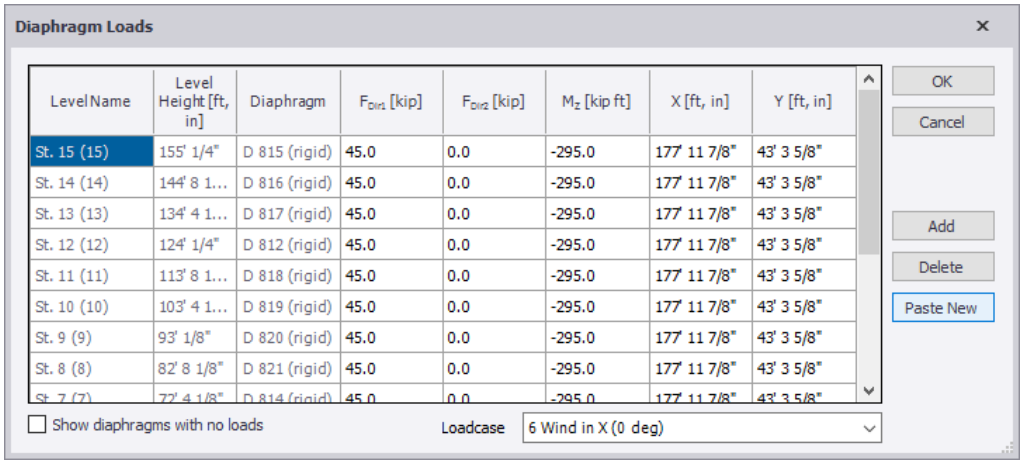


   The Diaphragm Loads table opens in a dialog.

3. In the **Loading** list at the bottom of the dialog, select the loadcase within which you want to apply the load.



4. Click **Paste New**.



Provided the spreadsheet contents were in the expected format, the loads and their positions are pasted into the table.

NOTE Existing diaphragm loads will be deleted / replaced by the pasted data.

5. Click **OK** when done.

Related information

See also

Video: Diaphragm loading

Manual wind loads (page 423)

Create nodal loads

To create nodal loads at the solver nodes in your model, see the following instructions.

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Nodal**.
4. In the **Properties** window, adjust the load details according to your needs.
5. To define the load position, click the desired node.

Create temperature loads

Temperature loads are global rises in temperature that you can apply to individual elements or panels in the model. For more information on how to create temperature loads, see the following instructions.

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Temperature**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click anywhere along the element to apply the load.

Create settlement loads

Settlement loads are translations or rotations that you can apply to a support. To create settlement loads, see the following instructions.

1. Open an appropriate view and set the view regime to **Structural View**.
2. In the , select an appropriate loadcase.
3. On the **Load** tab, click **Settlement**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a supported node to place the load.

Modify panel, member, and structure loads

To modify the properties of existing loads, see the following instructions.

1. In the model, select the load that you want to modify.
2. In the **Properties** window, adjust the load details according to your needs.

Delete panel, member, and structure loads

If necessary, you can delete existing loads in your model. In order to do so, see the following instructions.

1. Ensure that the loadcase containing the load is displayed in the **Loading** list.
2. On the **Quick Access Toolbar** at the top of the window, click **Delete**, then either:
   - Click the load that you want to delete.
   - Use an area selection (page 102) to select multiple loads.
     - A dialog box lists all the entities within the selection you made
     - Uncheck those you don't want to delete
     - Click **OK**

Decompose panel loads

TIP Although Tekla Structural Designer carries out load decomposition automatically when you click to analyze the structure, you have the option to decompose loads manually by using the **Decomposition** command.

In the context of big or complex models, decomposing manually can potentially save time, as it allows you to check that panel loads have been decomposed as you intend before running the analysis.

Decompose panel loads for an individual construction level

1. Open a 2D view of the desired level. (page 64)
2. Click the '2D' toggle button at the bottom right corner of the view to display it in 3D.
3. On the **Load** tab, click **Decomposition**.

Tekla Structural Designer generates an FE mesh within the two-way slab panels, and applies the resulting decomposed loads to the supporting members.

TIP In case you cannot see the changes in the model at this point, ensure that you have selected the correct settings in **Scene Content** - See **View decomposed loads** below.

Decompose panel loads to all required levels

1. Open the **Structure 3D** view.
2. On the **Load** tab, click  **Decomposition**.

   Tekla Structural Designer generates an FE mesh within the two-way slab panels, and applies the resulting decomposed loads to the supporting members.

   TIP In case you cannot see the changes in the model at this point, ensure that you have selected the correct settings in **Scene Content** - See **View decomposed loads** below.

View decomposed loads graphically

1. Open a 3D view of the model, or a 2D view displayed in 3D.
2. In **Scene Content**, go to **Loading**.
3. Select the **Decomposed** option.
4. According to your needs, do one of the following:

To	Do this
View the decomposed load values	• In the cell on the right side of **Decomposed**, select both **Geometry** and **Text**.
View the decomposed loads without load values	• In the cell on the right side of **Decomposed**, only select **Geometry**.

5. In the **Loading** list, select the loadcase.

NOTE Decomposed loads do not exist for 2-way slab items at levels where the **Mesh slabs in 3D analysis** option has been selected.

At levels where this is not the case, you can view the decomposed loads. However, you cannot see any shell results from the FE load decomposition.

View applied and decomposed member loads in a table

1. Open a 3D view of the model, or a 2D view displayed in 3D.
2. Right-click on the member, and from the context menu select **Show Member Loading**. Loads that have been applied directly to the member, and loads that have been decomposed to the member are displayed in a table. The 'In Proj.' checkbox is used to indicate applied loads applied in projection as opposed to along the element.
3. To filter the data by Loadcase, Source, Direction, or Type:
   a. Click the appropriate column header to filter by (Loadcase..., Source..., Direction..., or Type...)
   b. From the drop list that appears clear the categories that you don't want to be displayed.
   c. Click the Close button under the drop list.

3.3 Apply wind loading

Wind loads can be applied to a Tekla Structural Designer model in a variety of ways.

The most appropriate method to use will generally depend on the type and complexity of the structure.

Automatic generation of wind loads using the wind wizard

- This method determines detailed local pressures on panels
- It is country / code specific (not supported for all locations) - in such cases the "simple wind loads" method below is applicable.
- Some users feel it is more time consuming and detailed than is necessary for overall building load consideration - in such cases the "simple wind loads" method below may be preferable.
- Optionally, it can also be used to apply open structure wind loads (page 417)
- Optionally, for the US regional code, it can also be used to generate minimum design wind loads.

For further details, see: Wind wizard pressure loading (page 373)

The simple wind loads method

- A user defined simple pressure loading profile
- Includes a "Paste" option to easily apply a height varying pressure profile
- Includes two load application options:
  - Apply to diaphragms (requires the existence of diaphragms)

    - Apply to all windward wall panels, and all items marked as "open structure (page 417)" within the extents of the load projection.
- Potentially could be used where wind load is derived from testing, but generally diaphragm loading is used for this.

For further details, see: Simple wind pressure loading (page 396)

Manually applied wind loads

- Wind loads can be manually applied to any member, panel, or diaphragm (as panel, member, nodal, or diaphragm loads)
- Diaphragm loading is commonly used to apply the loads returned from wind tunnel testing, copy/paste functionality supports the rapid application of such loading.

For further details, see: Manual wind loads (page 423)

Open structure wind loads

- When working on an open structure, wind loads can be applied directly to members, ancillaries and equipment via their properties. In this case the wind pressures used to calculate the open structure wind loads can either be determined using the wind wizard, or using simple wind loads.

For further details, see: Open structure wind loads (page 417)

Wind wizard pressure loading

NOTE The Wind wizard is not currently available for the AS:1170.2 wind code variant.

Enclosed structures

The wind wizard method is typically used for enclosed structures and requires at least one wall panel, or roof panel to exist before it becomes available.

In this method, you 'clothe' the structure in wind and roof panels, and then create a wind model by running the wind wizard.

The wizard creates wind zone loads that are subsequently decomposed to the structure during analysis. Wind loadcases are then created for the required directions.

For further details, see:

- Wind wizard workflow (page 374)
- Wind model loadcases (page 377)
- Wind model load decomposition (page 378)
- Instructions for creating, editing and reviewing the wind model (page 387)

Open structures

The wind wizard method can also be used to apply loads to members marked as "open structure".

For further details, see:

- Wind wizard workflow for open structures (page 392)
- Instructions for applying loads to open structures using the wind wizard (page 395)

Wind wizard workflow

The basic steps required to run the wind wizard are as follows:

Clothe the structure in wind wall and roof panels

The Wind Model calculations depend on the geometry and inter-connectivity of the wall panels and roof panels that envelope the building. You must therefore define the model together with its wall and roof panels before you run the **Wind Wizard...**.

NOTE You can, should you wish, use Tekla Structural Designer purely for wind assessment - by setting up a model consisting only of wall panels and roof panels (no members). Tekla Structural Designercan then determine the wind loading on the building envelope.

In order to get the best results you should ensure that you define the largest possible sizes for the wind wall and roof panels. You may compromise the results if you define many small panels rather than one large one. (The calculation of the reference height in particular can be unconservative.)

Applying Wall Panels

A single wall panel is determined to be a single planar surface. The outward face is vitally important for determining the wind direction relative to the wall, that is windward or leeward.

It is recommended that you check the outward faces are as you intend by ensuring they are all shaded in the same color (the one assigned to 'Wind Wall - Front' in Settings > Scene). The inward faces will all be shaded in a different color. To correct any mistakes, choose the **Edit** --> **Reverse** command and then click once on a wall panel to switch its direction. Note that connected wall panels are checked to ensure that the normal directions are consistent whenever automatic zoning is carried out, for example at the end of the **Wind Wizard...**. If there is a problem it is indicated on the **Project Workspace** --> **Wind** tab, with affected panels being marked.

Once a wall panel has been placed the following additional panel properties can be specified:

- **Rotation angle** - defines the span direction, 0° is horizontal and 90° is vertical.
- **Is a parapet wall** - you can indicate whether the wall panel is a parapet or not.

  NOTE If a building face comprises a parapet above a wall, you should not attempt to model this as a single wall panel. It should be input as an upper and lower panel, with the upper panel being set as a parapet.

- **Gap** - (regional codes: EC and BS only) where the gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funnelling. A zero gap value explicitly means ignore funnelling, for example where this building and the adjacent one are sheltered by upwind buildings
- **Solidity** - (regional codes: EC and BS only) If you set the wall panel as a parapet, then you also need to indicate the Solidity of the parapet. (Wall panels that are not parapets automatically adopt a solidarity of 1.0).
- **Decompose to** - for wall panels that are not parapets, you can indicate how the wall load is decomposed on to supporting members. See Wind model load decomposition (page 378)

To set this information as you require, select the wall panels and then use the **Properties** window to make changes.

Applying Roof Panels

A single roof panel is determined to be a single planar surface. The orientation of a roof panel is automatically determined when placed based upon the slope vector - the line of maximum roof slope.

Initially the roof type (page 427) is set to 'Default'. This is interpreted as Flat if the roof slope < 5 degrees, otherwise it is interpreted as Monopitch. You should select the roof panel and then use the **Properties** window to adjust the roof type as necessary for all other situations (i.e. For Duopitch, Hip Main, Hip Gable or Mansard).

The span direction is also set in the **Properties** window, this is defined as an angle, where 0° is parallel to the X axis and 90° is parallel to the Y axis.

Perform the gravity design

We recommend that you perform an analysis and design at this stage for the gravity loading only, but this is not essential.

Run the wind wizard

Once the model has been 'clothed' in wall panels and roof panels, the Wind Wizard (located on the **Load** tab) guides you through the process of intelligently 'applying' wind to the resulting building envelope.

The wizard uses databases where appropriate (depending on the wind code) to determine the appropriate wind details for your structure location.

Having defined the wind directions in which you are interested, on completion of the wizard the appropriate wind zones on the roofs and walls of your structure are automatically calculated. At this point the **Update Zones**, **Wind Loadcases**, and **Delete Wind** commands on the **Load** tab become available.

NOTE

- EC1991 1-4 Wind wizard
- ASCE 7 Wind wizard
- BS6399-2 Wind wizard
- IS 875 (Part 3) Wind Wizard
-

Review the wind zones

The resulting wind model is accessed from the Project Workspace Wind tab. Wind Views can also be opened as required for each wind direction.

From here, you can set the type of each roof to achieve the correct zoning, and can then tailor the zoning to account for particular features in more detail, if you so require.

Define the wind loadcases

The Wind Loadcases dialog (located on the **Load** tab) can then be used to automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases the appropriate wind pressures are calculated on each zone.

Once a wind loadcase has been defined the **Decomposition** command on the **Load** tab becomes available. This can be used to confirm how the wind loads are decomposed on the structure prior to running the analysis, although this step is not essential as the decomposition will be automatically performed as part of the analysis process anyway.

NOTE It is assumed that the wind loads are developed to assess the overall stability of the structure and for member design. The wind loads have not been specifically developed for the design of cladding and fixings.

Related topics

Wind model loadcases (page 377)

Review wind zone loads

Wind zones can be graphically displayed for each wind direction from the appropriate Wind View. Once the wind loadcases have been created you can also display the wind pressures and zone loads for each loadcase.

Combine the wind loadcases into design combinations

Combine the wind loadcases into design combinations in the usual way.

Perform the static design

Run a static design from the **Design** tab.

Wind model loadcases

Once the wind wizard has been run, the next step is the creation of appropriate wind loadcases. This is achieved using the **Wind Loadcases** dialog which is accessed from the ribbon by going to the **Load** tab and clicking **Wind Loadcases**.

NOTE The **Wind Loadcases** dialog is only available after the wind wizard has been run.

Auto generate loadcases

The **Auto** button within the **Wind Loadcases** dialog automatically generates wind loadcases.

NOTE The **Auto** button is greyed out and hence unavailable, if wind loadcases have already been defined. These would need to be deleted to use the **Auto** button again.

Add loadcases

Alternatively, you can create loadcases manually using the **Add** button and proceeding as follows:

- specify which direction the loadcase will be created
- set default values for all the zone loads generated in the loadcase
- the loadcase name is generated automatically, (but can be overwritten if required by unselecting the Auto checkbox)

Try to use engineering judgment to identify the critical loadcases for design. This ensures that the number of load combinations to be considered are minimized. Use the **Delete** button to remove wind loadcases you do not wish to consider.

Once generated, the wind loadcases are standard loadcases which can be included in combinations in the normal manner.

Wind model load decomposition

Click the links below to find out about decomposition options available for panels when using the wind model method.

- Wall panel load decomposition (page 378)
- Roof panel load decomposition (page 382)
- Alternative wind load decomposition method for complex models (page 384)
- Limitations of wind decomposition to diaphragms (page 402)

Wall panel load decomposition

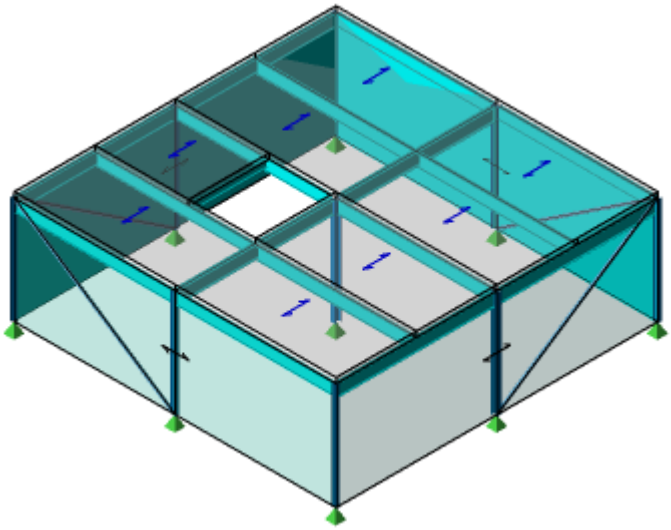
This topic will discuss the decomposition that occurs when considering wind load applied to wall panels.

Decomposition options

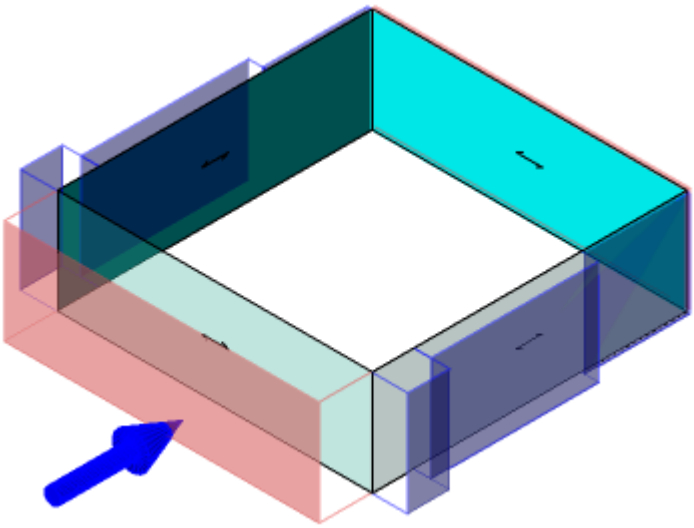
Wind zone loads are decomposed on to the structure according to the **Decompose to** wall panel property. The available options are:

- Members
- Nodes (default)
- Rigid Diaphragms

To demonstrate the effect of the different **Decompose to** values, consider a braced steel frame clad in wind wall panels as shown below:



For wind direction 0, the wind model produces Zone loads as follows:



The topics below illustrate how the zone loads shown above are decomposed using the three different **Decompose to** values.

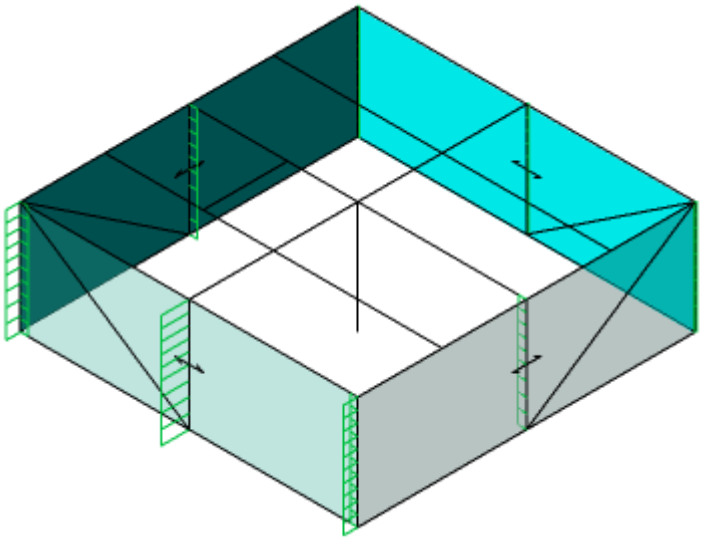
Decompose to Members

Decomposition to members is similar to the roof panel decomposition, the span direction of the wind wall panel determines the direction of the one way decomposition.

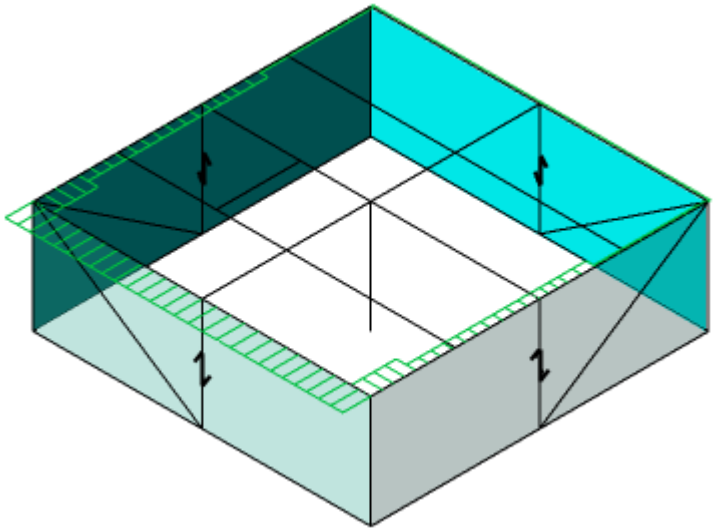
All element types within the wind wall plane are considered with the exception of bracing members.

Decomposition to members allows the generation of UDL's on portal stanchions and gable posts without the need to model side rails.

For the model shown above, choosing **Decompose to Members** produces the following load decomposition:



Decomposition to Members (rotation angle 0 degrees)



Decomposition to Members (rotation angle 90 degrees)

Decompose to Nodes

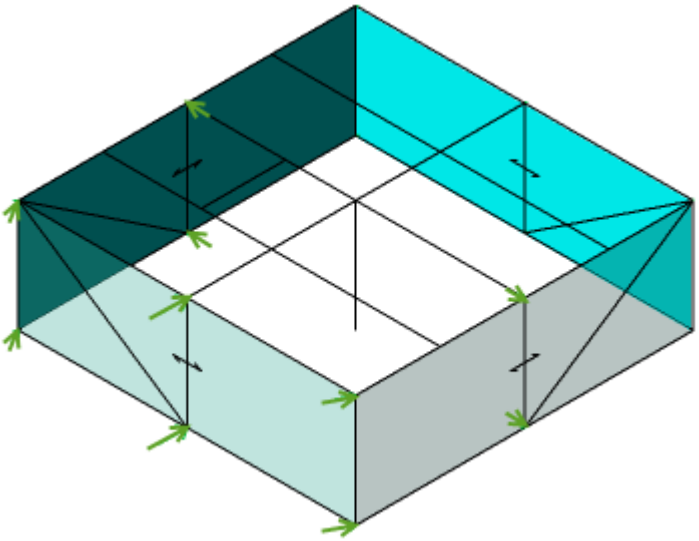
Decomposition to nodes is the **default** setting and results in nodal loads on the supporting members. This setting is typically appropriate to avoid lateral loads on simple beams.

All element types within the wind wall plane are considered with the exception of bracing members.

The initial decomposition is the same as for members, with the direction of the one way decomposition being specified by the span direction of the panel. A second decomposition stage is then undertaken to convert the member loads to nodal loads:

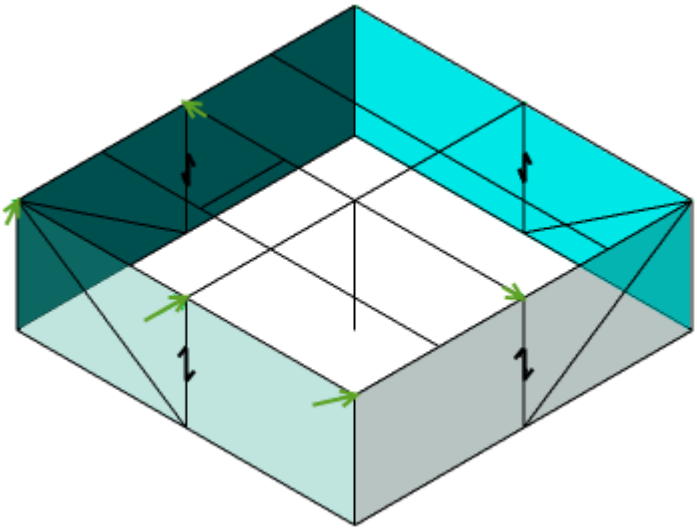
- Full/partial UDLs and VDLs on elements (lengths of beams/columns between nodes) are distributed back to nodes as if the elements were simply supported at either end.
- The final nodal load is the sum of all incoming element loads.

For the model shown above, choosing **Decompose to Nodes** produces the following load decomposition:



collapse

Decomposition to Nodes (rotation angle 0 degrees)



Decomposition to Nodes (rotation angle 90 degrees)

NOTE In the example above, when the rotation angle is 0 degrees, some of the nodal loads are applied directly to supports.

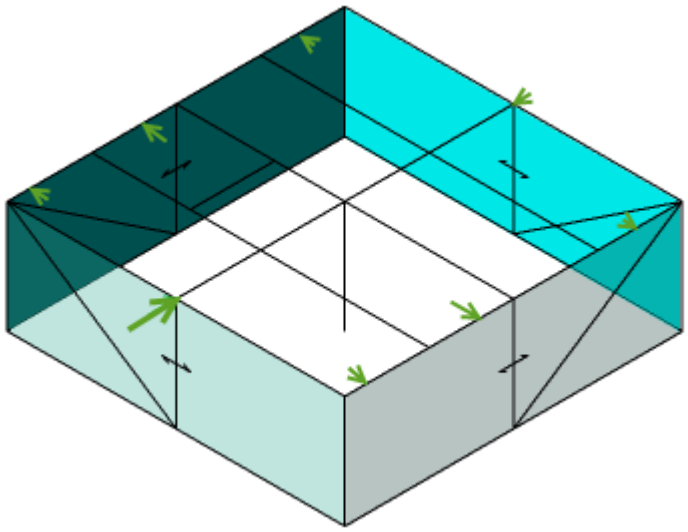
Decompose to Rigid Diaphragms

Decomposition to rigid diaphragms does not consider the span direction of the wall panel (so the rotation angle is irrelevant).

It is particularly useful for flat-slab structures, as the alternative **Decompose to Member** or **Node** decomposition methods require supporting members that may not exist in the model.

NOTE All rigid diaphragms within the wind wall height are considered for decomposition irrespective of whether they are physically connected to the wind wall.

For the model shown above, choosing **Decompose to Rigid Diaphagm** produces the following load decomposition:



Decomposition to Diaphragms - each Zone load is decomposed as a separate point load on the diaphragm

Validation of Panels set to Rigid Diaphragm Decomposition

The following validation checks are performed on wind wall panels set to **Decompose to Rigid Diaphragms**:

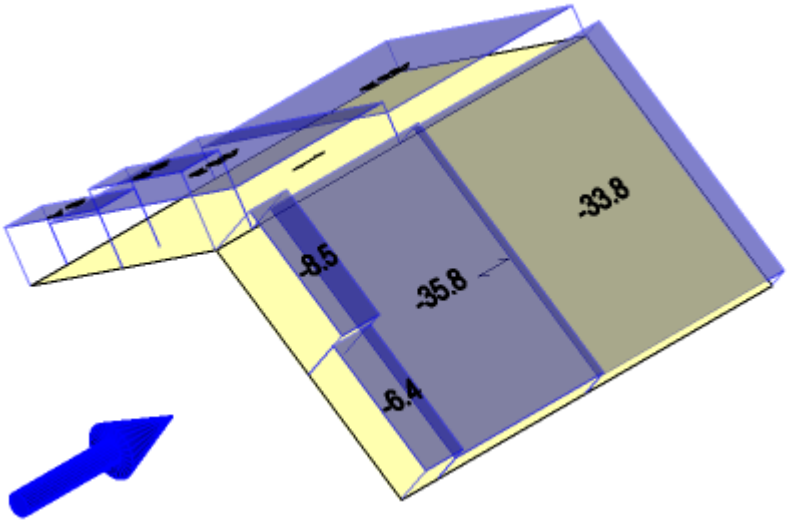
- Each panel must be rectangular
- The top level of every wall panel must align with a rigid diaphragm
- Each panel may be sub-divided into zones, but only by horizontal lines
- Unlike for **Decompose to Member** or **Node**, each wall panel does not need to have supporting members along its edges.

Roof panel load decomposition
This topic will discuss the decomposition that occurs when considering wind load applied to roof panels.

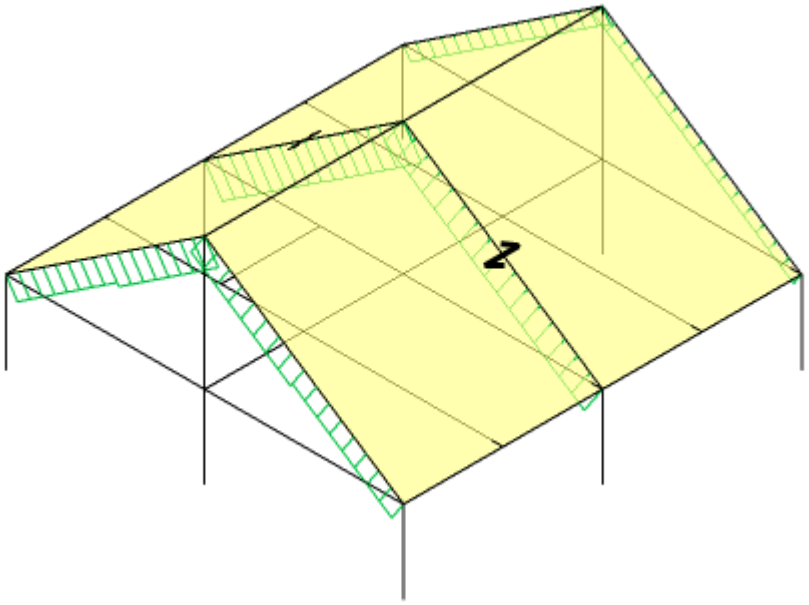
Roof panel load decomposition

The direction of the one way decomposition of the wind zone loads to roof members is determined by the span direction (i.e. rotation angle) of the roof panel.

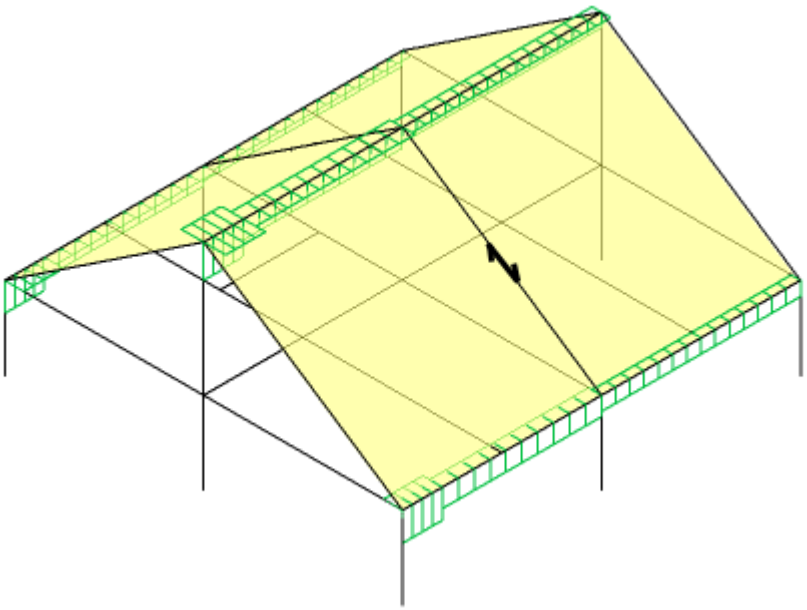
All types of elements within the roof panel plane (except bracing) are considered during the load decomposition.



Zone loads on roof.



Roof panel decomposition (rotation angle 0 degrees)



Roof panel decomposition (rotation angle 90 degrees)

Alternative wind load decomposition method for complex models

For complex models the placement of wind walls required by the Wind model method can be a time consuming operation.

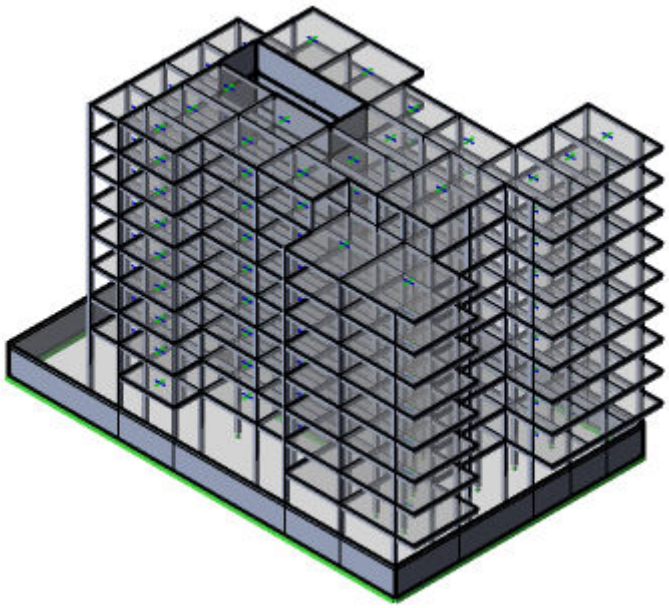
In such a situation (provided the model has suitable rigid diaphragms), the following alternative offers a quicker and simpler way to apply approximate wind loads:

- Use engineering judgment to clothe the structure with an arrangement of simplified wind wall panels around its bounding box,
- set the wind walls to decompose to rigid diaphragms,
- proceed with the Wind model method

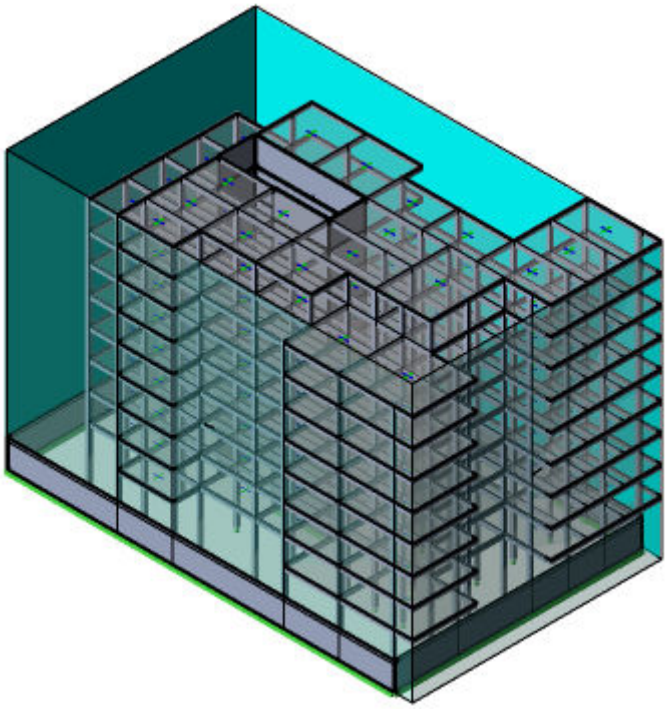
Simplified wind wall panels example

In theory, because wind walls set to **Decompose to Rigid Diaphragms** don't have to physically connect to the diaphragms, even a very complex model could be clothed with just four simplified wind walls defined along its bounding box, as illustrated below:

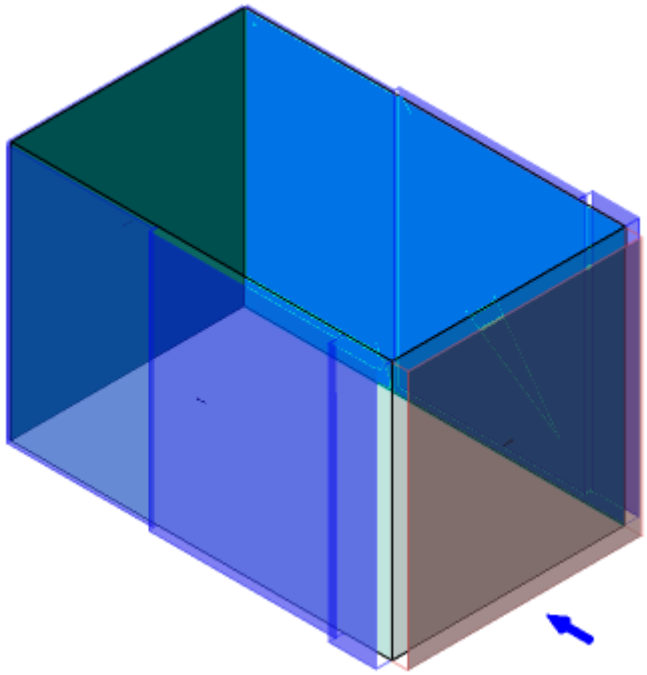
- Complex model prior to placement of wind walls



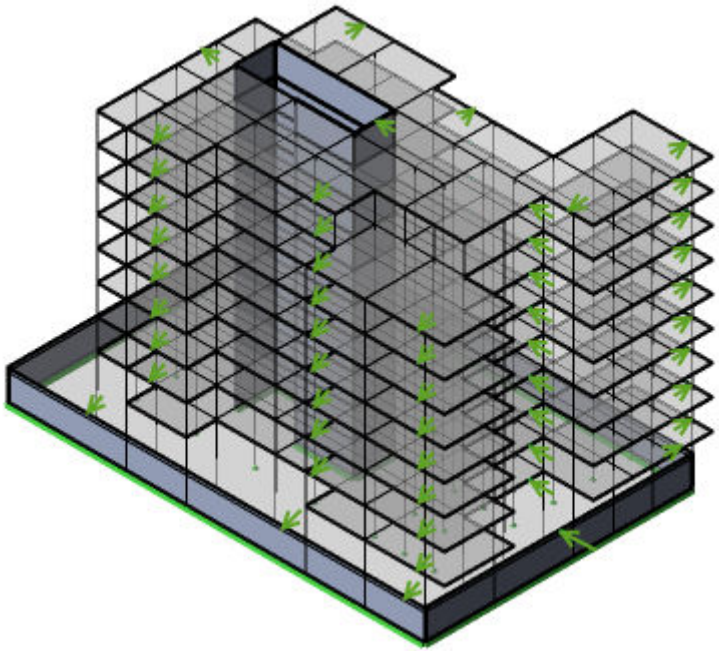
- Simplified wind walls placed around the "bounding box"



- Simplified zone loads after running the wind wizard



- Resulting point loads on rigid diaphragms after decomposition



Such an approach could be used to rapidly establish approximate wind loads (based on the rectangular building envelope) which could then be refined at a later stage if necessary.

NOTE If this method is adopted you are strongly advised to review the wind zones that are formed, and the resulting decomposed loads to ensure they meet your expectations - it will not give good results for all models.

If the decomposed loads produced by the above approach are not satisfactory, you might decide to take greater control by applying the loads manually.

Instructions for creating, editing and reviewing the wind model

When creating and editing your wind model you will need to perform the following tasks:

- Create a wind model and wind loads (page 387)
- Modify wind zones and wind zone loads (page 389)
- Create and manage wind loadcases (page 391)

Create a wind model and wind loads

NOTE The wind wizard is not currently available for the AS:1170.2 wind code variant.

NOTE The determination of the wind speeds, the pressures, and the zones is rigorous. However, do remember that the final wind loads that are adopted are your responsibility.

Run the Wind Wizard

NOTE You must define at least one wall or roof panel before running the **Wind Wizard**.

1. On the ribbon, go to the **Load** tab and click  **Wind Wizard...**

   The **Wind Wizard** opens.

   NOTE The **Wind Wizard** input varies according to the regional code that you are using. For details, see:

   -
   -
   -
   -

2. Define the necessary information for the wind model. To go to the next page in the wizard, click **Next**.

3. Once you have defined all the necessary information, click **Finish**.

NOTE After running the **Wind Wizard**, you can review the roof and wall zones for each wind direction.

After you have created the wind model with the **Wind Wizard**, you can open a wind view to graphically display the wind zones and loading that apply for a particular wind direction.

Add wind directions

1. On the ribbon, go to the **Load** tab and click  **Wind Wizard...**

   The **Wind Wizard** opens.
2. Click **Next** until you are on the **Results** page.
3. Click **Add Dir.**
4. Specify the properties of the new wind direction.
5. Click **Finish**.

NOTE Remember to add new wind loadcases and design combinations to incorporate the wind loading for the new direction into your calculations.

Delete wind directions

1. On the ribbon, go to the **Load** tab and click  **Wind Wizard...**

   The **Wind Wizard** opens.
2. Click **Next** until you are on the **Results** page.
3. Click the wind direction that you want to delete.
4. Click **Del Dir.**
5. Click **Finish**.

NOTE Remember to update the existing wind loadcases and design combinations to remove the details for the wind direction you have deleted from your calculations.

Delete the entire wind model

To delete the entire wind model and start the wind modeling process from scratch, do the following:

1. On the ribbon, go to the **Load** tab and click **Delete Wind**.

   Tekla Structural Designer deletes all the previously defined wind directions and wind loadcases.

See also

Modify wind zones and wind zone loads (page 389)

Create and manage wind loadcases (page 391)

Modify wind zones and wind zone loads

In order to view and modify the wind zones and loading that apply to a particular wind direction, you have to open the appropriate wind view. Then, you can modify the wind zones and wind zone loads according to your needs.

Open a wind view

NOTE Wind views are only available if you have already created the wind model with the **Wind Wizard**.

---

1. In the **Project Workspace**, go to the  **Wind** tab.
2. Right-click the wind direction that you want to view.
3. In the context menu, click **Open View**.

   Tekla Structural Designer opens the selected wind view and adds a **Zone Loads** tab on to the ribbon, which allows you to review wind zone data graphically.

View wind zones

1. Open a wind view that displays the details of the desired wind direction.
2. On the ribbon, go to the **Zone Loads** tab and click **Wind Zones**.

   The wind view displays the zones that are applied to the structure for the selected wind direction.

Modify wind zones

1. Open a wind view that displays the details of the desired wind direction.
2. In the **Project Workspace**, go to the  **Wind** tab.
3. For the wind direction in question, expand the **Roof Zones** or **Wall Zones** branch.

4. In the desired branch, right-click the panel label containing the zone that you want to edit.
5. In the context menu, select **Edit Zones...**

   The **Zone Properties** dialog box opens.
6. Modify the wind zones according to your needs.
7. Click **OK**.

View the wind zone loads

NOTE Remember to update the zone loads when you make changes to your model.

1. Open a wind view that displays the details of the desired wind direction.
2. On the ribbon, go to the **Zone Loads** tab and click **Zone Loads**.
3. In the **Loading** list, select the desired wind loadcase.

   Tekla Structural Designer displays the loads that are applied to the structure for this loadcase in this wind direction.

Modify wind zone loads

1. Open a wind view that displays the details of the wind direction whose zone loads you want to modify.
2. On the ribbon, go to the **Zone Loads** tab and click **Zone Loads**.
3. In the **Loading** list, select the desired wind loadcase.

   Tekla Structural Designer displays the loads that are applied to the structure for this loadcase in this wind direction.
4. In the model, click the wind zone that you want to change.
5. In the dialog that opens, clear the **Use Default Values** option.
6. Modify the values that determine the loads.

   TIP If you want to reduce the net pressure for benecifial loads to zero, select the **Beneficial Load** option.
7. To update the zones, **OK**.

Update wind zone loads

When you change the roofs or walls of your structure, Tekla Structural Designer does not automatically update the changes to existing wind zoning.

Updating the zoning is manual because you may wish to make more alterations before you recalculate the zoning.

Once you have completed your changes, incorporating them and recalculating the zoning details is simple. To reinstate the zoning after making changes, do the following:

1. On the ribbon, go to the **Zone Loads** tab and click **Update Zones**.

   The wind zoning calculations run in the background. Once the calculations are complete, Tekla Structural Designer views the new zoning layout for your structure.

   NOTE If you have defined your own zone layout for any roof or wall, the existing zones will be maintained when you make changes. Make sure to update these zones appropriately.

See also

Create a wind model and wind loads (page 387)

Create and manage wind loadcases (page 391)

Create and manage wind loadcases

Once you have defined the basic wind data for your model, and calculated wind zoning in the **Wind Wizard**, you can define the required wind loadcases manually. For detailed instructions, see the following paragraphs.

Create wind loadcases

1. On the ribbon, go to the **Zone Loads** tab and click **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.

2. Do one of the following:

   - Click **Add** to add the details of each wind loadcase individually.
   - Click **Auto** to generate standard wind loadcases for the wind directions that you have defined in the **Wind Wizard**.

     NOTE You cannot use the **Auto** option once you have created other wind loadcases.

Delete wind loadcases

1. On the ribbon, go to the **Zone Loads** tab and click **Wind Loadcases**.

2. Click the wind loadcase that you want to delete.
3. Click **Delete**.
4. Repeat steps 2 and 3 to delete further loadcases.
5. Click **OK** to close the **Wind Loadcases** dialog box.

NOTE Remember to update your design combinations, so that they take the deleted wind loadcases into account.

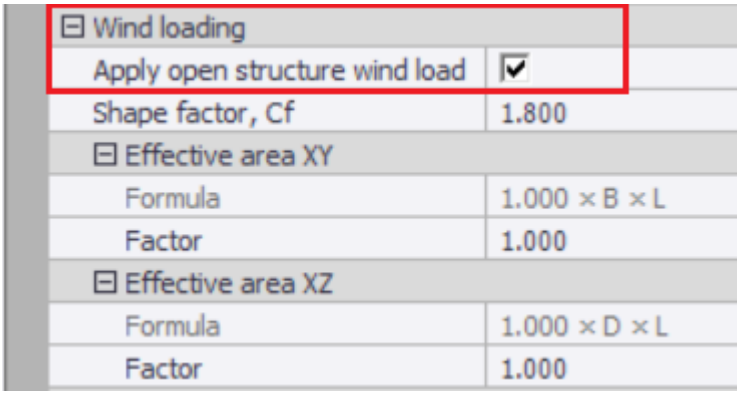
Wind wizard workflow for open structures

The basic steps required to apply the wind wizard to open structures are as follows:

NOTE Application of the wind wizard to open structures is not currently available for the British Standard BS 6399-2, or Australian AS:1170.2 wind code variants.

Select members to have open structure wind load applied

The first step is to confirm the **Apply open structure wind load** property is set as required for those structural members, ancillaries, and equipment to which the loads are to be applied. (By default this property is *off* for structural members, but *on* for ancillaries and equipment.)



It can be toggled *on* and *off* directly in the properties window as above, or it can be set graphically via Show/Alter State > (page 1011).

For members and ancillaries, when it set *on*, additional properties become available, allowing the shape factor and effective area factors used in the wind load calculation to be customized.

For members only, each of these properties can be set for all stacks/spans, or individual stacks/spans.

Run the wind wizard

Having selected the members to have open structure wind loads applied, the next step is to run the Wind Wizard (page 395).

On the first page of the wizard you must select the **Apply Open Structure Wind Load** option.

NOTE The above option is only available if at least one entity has been selected to have open structure wind load applied.

Step-through the remaining pages of the wizard to generate wind model.

The wizard uses databases where appropriate (depending on the wind code) to determine the appropriate wind details for your structure location.

Related topics

EC1991 1-4 Wind wizard

ASCE 7 Wind wizard

BS6399-2 Wind wizard

IS 875 (Part 3) Wind Wizard

Define the wind loadcases

The Wind Loadcases dialog can then be used (page 396) to define the loadcase information for the directions you require.

NOTE If you have a totally open structure without any wall or roof panels the C_{pi} values are immaterial and can be set to 0.0.

Once the loadcases have been defined, wind loads (page 417) are automatically calculated for those entities selected to have open structure wind load.

NOTE It is assumed that the wind loads are developed to assess the overall stability of the structure and for member design. The wind loads have not been specifically developed for the design of cladding and fixings.

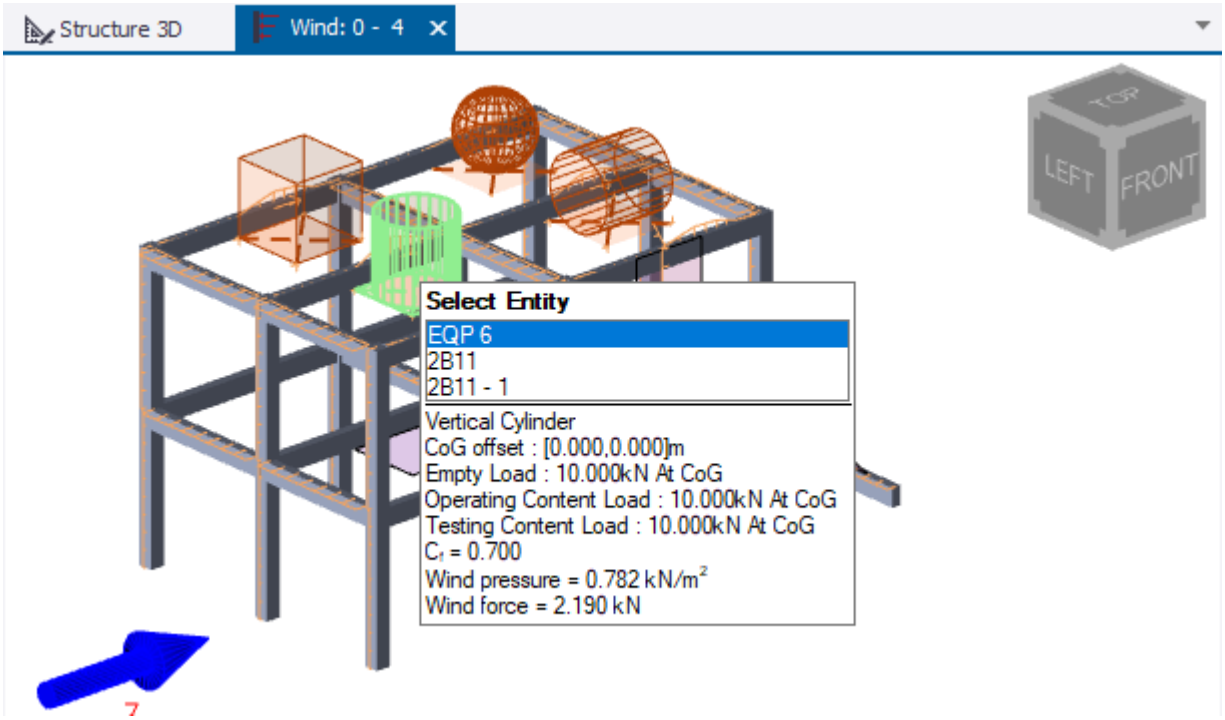
Related topics

Wind model loadcases (page 377)

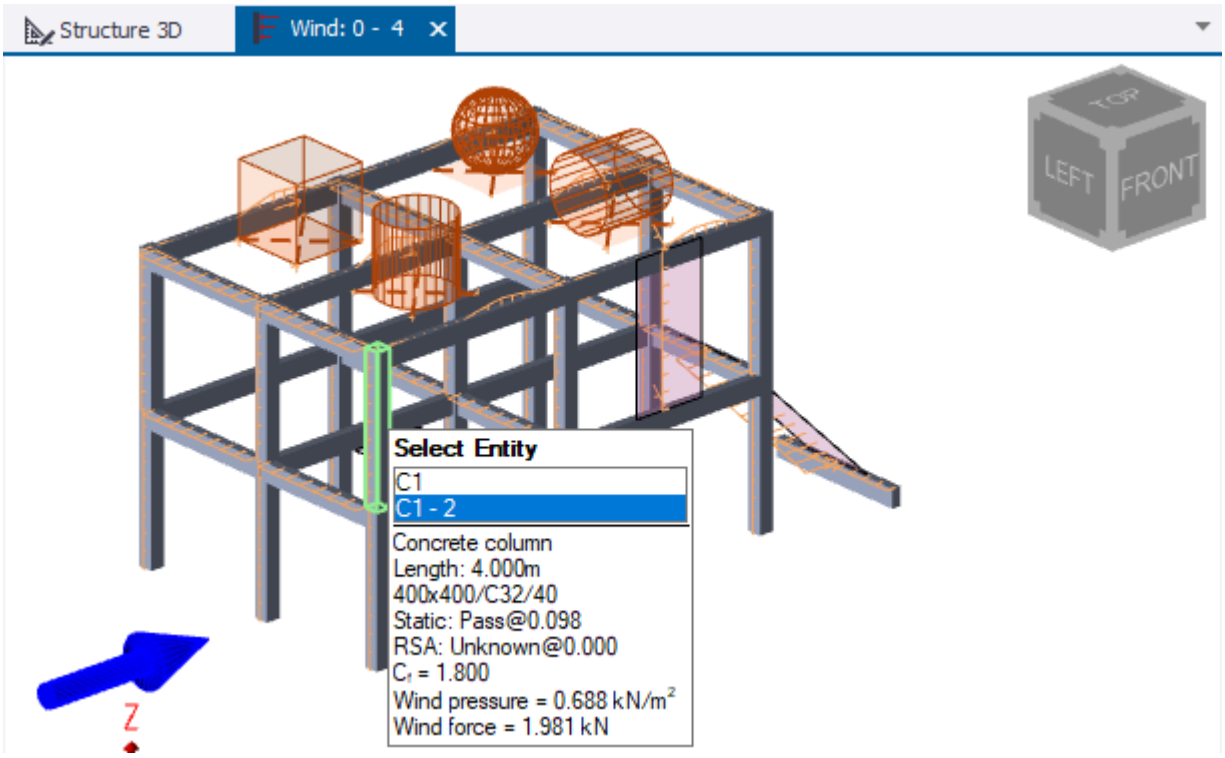
Review wind loads

The open structure wind loads on members, ancillaries and equipment can be reviewed by opening a wind view for the required direction from the Project Workspace Wind tab and selecting the required loadcase.

Hovering the cursor over an entity displays wind loading information in the Select Entity tooltip. The shape factor, wind pressure and wind force are all listed.



For members, the same information is displayed by hovering over the required stack/span.



Combine the wind loadcases into design combinations

Combine the wind loadcases into design combinations in the usual way.

Perform the static design

Run a static design from the **Design** tab.

Create an open structure wind loads report

These loads are automatically included in the Open Structure Wind Load report (page 1084).

Instructions for applying loads to open structures using the wind wizard

When using the wind wizard to apply loads to open structures you will need to perform the following tasks:

Apply open structure wind load to selected entities

Open structure wind loads can only be applied to structural members, ancillaries, and equipment.

1. Select the members to which the load is to be applied.
2. In the **Properties** window, under **Wind loading**, select **Apply open structure wind load**

   Additional properties then become available, allowing the shape factor and effective area factors used in the wind load calculation to be customized.
3. Edit these additional properties, if required.

> **NOTE** The **Apply open structure wind load** property is already selected by default for ancillaries and equipment, but can be deselected if required.

Run the wind wizard

> **NOTE** You must apply an open structure wind load to at least one entity before running the **Wind Wizard**.

1. On the **Load** tab, click  **Wind Wizard...**

   The **Wind Wizard** opens.

NOTE The **Wind Wizard** varies slightly according to the regional code that you are using.

2. On the first page of the wizard select the **Apply Open Structure Wind Load** option.
3. Step-through the remaining pages of the wizard to define the necessary information for the wind model.
4. Once you have defined all the necessary information, click **Finish**.

NOTE After you have created the wind model with the **Wind Wizard**, you can open a wind view to graphically review the open structure wind loads for each wind direction.

Define wind loadcases

1. On the **Load** tab, click  **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.
2. Do one of the following:
   - Click **Add** to add the details of each wind loadcase individually.
   - Click **Auto** to generate standard wind loadcases for the wind directions that you have defined in the **Wind Wizard**.

     NOTE You cannot use the **Auto** option once you have created other wind loadcases.

Simple wind pressure loading

The simple wind load method can be used as an alternative to the wind wizard (page 373), either where this may not be available or appropriate for certain geographical locations/ country codes, or where the engineer considers it is more detailed than necessary for the stage of design and/ or overall building load considerations.

Simple wind loads are defined as pressures that can vary vertically acting over a given width of the structure. These are decomposed on to the structure in one of two ways:

- Apply to Diaphragms
- Apply to Wall/ Roof Panels & Open Structure Members

Apply to Diaphragms

When using this option, the simple wind load is decomposed as point loads on to any rigid diaphragms between the top and bottom level of the simple wind load. Wall and roof panels are ignored.

When the option to apply to diaphragms is enabled the following further option is provided to control the application of the wind load:

- **Ignore diaphragms on lowest level** - If for some reason there are diaphragms at the ground level, then you may decide to check this option in order to ensure no load is lost directly to the foundations.

For further details, see:

- Instructions for applying simple wind loads to diaphragms (page 398)
- Simple wind load example (diaphragms) (page 400)
- Limitations of wind decomposition to diaphragms (page 402)

Apply to Wall/Roof Panels & Open Structure Members

When using this option the simple wind load is applied to all windward wall panels, all roof panels, and all items marked as "open structure (page 417)" within the extents of the load projection.

- Wall and roof panels have area loads applied which are then decomposed on to the structure members.
- Open structure items (i.e. those members, ancillaries and equipment entities for which the “Apply open structure wind load” property has been enabled) have loads applied directly.

For wall and roof panels the load application is broadly similar to that of the wind wizard, with the following notable differences:

- Load is applied only to the windward outer surface of wall and roof panels - not leeward or those parallel to the load direction.
- The applied area load *direction* is essentially that of the Simple wind, not Normal to panel surfaces.
- For wall panels, loads are decomposed as member loads for panels set to Decompose to Nodes, as well as when set to decompose to Members (the decompose to Diaphragms option is followed).

When the option to apply to wall/ roof panels & open structure members is enabled there are the following further options to control the application and extent of the wind load:

- **Pressure > At highest point of item/ Average between highest and lowest point** - the operation of these options briefly is as follows:
  - **At highest point of item** -wall/roof panels & open structure members are loaded using the pressure at the top of the item. (Click here (page 1587) for an illustration).

    - **Average between highest and lowest point** - wall/roof panels & open structure members are loaded using the average pressure between the top and bottom of the item. (Click here (page 1587) for an illustration).
- **Extend wind to top of building** - when enabled, wall/roof panels & open structure members receive the wind load, irrespective of whether they lie within, or outside the load height. Otherwise only those wall/roof panels & open structure members that lie within the load height receive load. (Click here (page 1588) for an illustration.)
- **Extend wind over full building width** - when enabled, wall/roof panels & open structure members see the wind load, irrespective of whether they lie within, or outside the load height. Otherwise only those wall/roof panels & open structure members that lie within the load width receive load. (Click here (page 1590) for an illustration.)

NOTE Validation warnings will be issued for the minimum required overlap of simple wind and for the case where a panel or member is loaded by more than one simple wind.

For further details, see:

- Instructions for applying simple wind loads to wall/roof panels and open structure members (page 411)
- Simple wind load example (panels) (page 413)
- Simple wind load workflow for open structures (page 416)

Instructions for applying simple wind loads to diaphragms

When working with simple wind pressure loads that you want to apply to diaphragms you will be required to perform some or all of the following tasks:

Related task

Instructions for applying simple wind loads to wall/roof panels and open structure members (page 411)

Create loadcases for simple wind loads

1. On the **Load** tab, click  **Loadcases**.

   The **Loading** dialog box opens.
2. On the **Loadcases** page, click **Add**.
3. Name the new loadcase.
4. Set the type to **Wind**.
5. Click **OK**.

Create simple wind loads

RESTRICTION You can only create simple wind loads in the **Structure 3D** view.

NOTE In order access the Simple Wind Loading dialog box (page 1583), you must first create and select the loadcase where you want to add the simple wind loads.

1. In the **Loading** list, select a manually created wind loadcase.
2. On the **Load** tab, click **Simple Wind**.
3. In the model, click to define the start point of the load width.
4. Click to define the end point of the load width.

   The Simple Wind Loading dialog box (page 1583) opens. The dialog box allows you to specify a single area wind load from the lowest level of the building up to the highest level that contains a rigid diaphragm.

   TIP The wind load does not have to begin at the lowest level of the building.

   If you have one or more levels below ground, you can adjust the value of the lowest level in the Simple Wind Loading dialog box (page 1583).

5. In the **Apply to** droplist, select **Diaphragms** - to have the load decompose to diaphragms only.
6. Specify the required load profile manually, or paste the load profile from a spreadsheet (page 1586).
7. To create a stepped profile up the height of the building, use the **Insert Above**, **Insert Below**, **Quick Above...** and **Quick Below...** buttons and then specify the load for each level.
8. When the load is complete, click **OK**.

Modify simple wind load vertical properties

1. Double-click anywhere in an existing simple wind load.

   The Simple Wind Loading dialog box (page 1583) opens.
2. Modify the load properties according to your needs.
3. Click **OK**.

Modify the simple wind load width

1. Select the simple wind load that you want to modify.
2. Select one of the two nodes that define the load width.

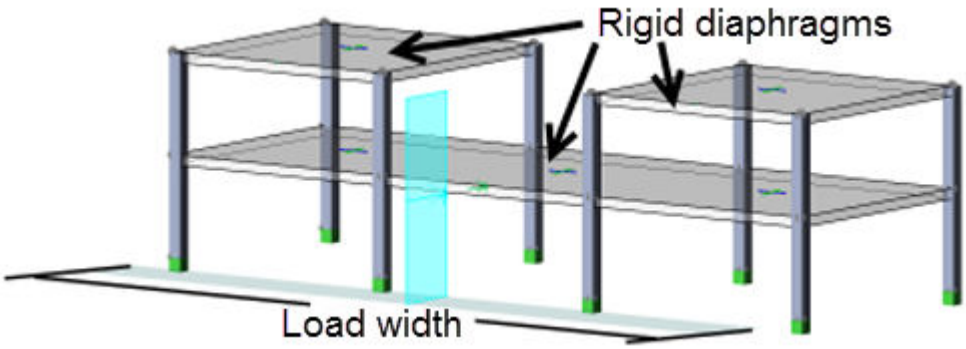
3. Click where you want to move the selected node.

   Tekla Structural Designer moves the selected node to the new location, and regenerates the load.

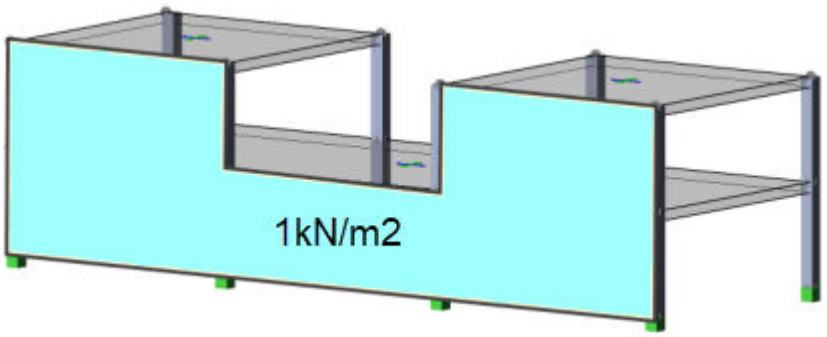
Simple wind load example (diaphragms)

Each simple wind load is defined as an area load over a defined width and height.

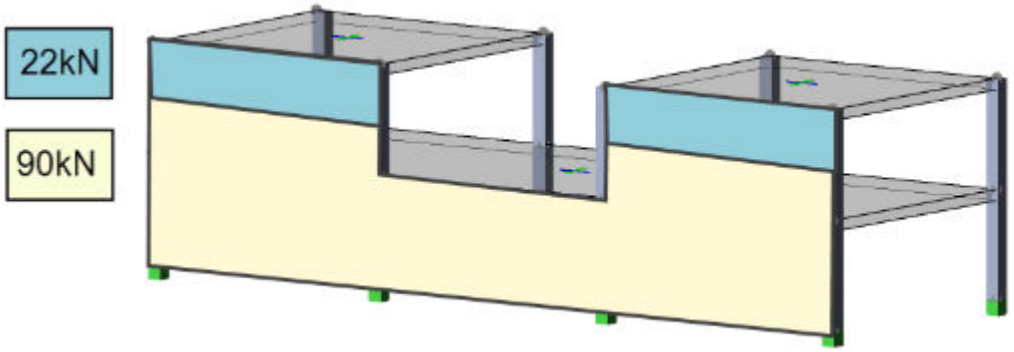
In the two story example shown below, a 1kN/m^2 load is applied over the full 22.5m width and 6m height of the building.



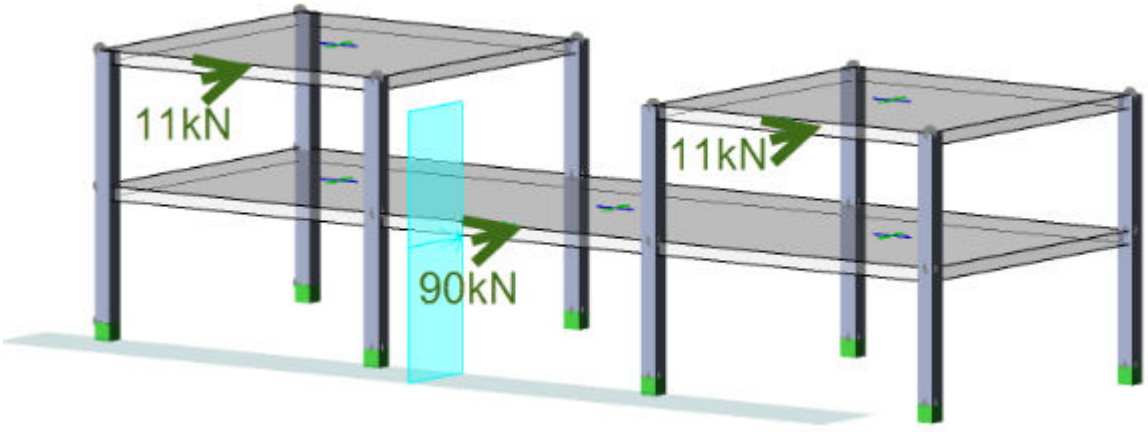
- The load defaults to being uniform over the full height (as shown above), but levels can be inserted to cater for a stepped loading profile if required. The inserted levels do not have to coincide with actual building levels.
- The top of the load should align with a rigid diaphragm - if not a validation warning is generated - this is to ensure the loading is distributed as correctly as possible.
- During decomposition an imaginary 'bounding box' is formed around the building - only the load width within the building profile is considered for decomposition.



- Load within the bounding box that hits the structure (the blue shaded area above) is decomposed to point loads on rigid diaphragms only - it does **not** get decomposed to semi-rigid diaphragms.
- All rigid diaphragms on the top or bottom level or anywhere in-between are considered, with the area load being divided between the levels before it is decomposed. In the example the area load is split between 1st and 2nd floor levels so that a total of 22kN is to be decomposed above and 90kN below.



- The load is then decomposed to the diaphragms at each level in proportion to the width of each diaphragm. Each load being applied as a nodal load in the direction of the simple wind load at the mid point of the projected load.



- If there are no suitable diaphragms on the top level, the load is applied at the next level down.
- Similarly if the 'Ignore diaphragms on lowest level' box is checked on the Simple Wind Loading dialog or there are no suitable diaphragms on the bottom level, the load is applied at the next level up.

NOTE If for some reason there are diaphragms at the ground level, then you may decide to check the 'Ignore diaphragms on lowest level' box in order to ensure no load is lost directly to the foundations.

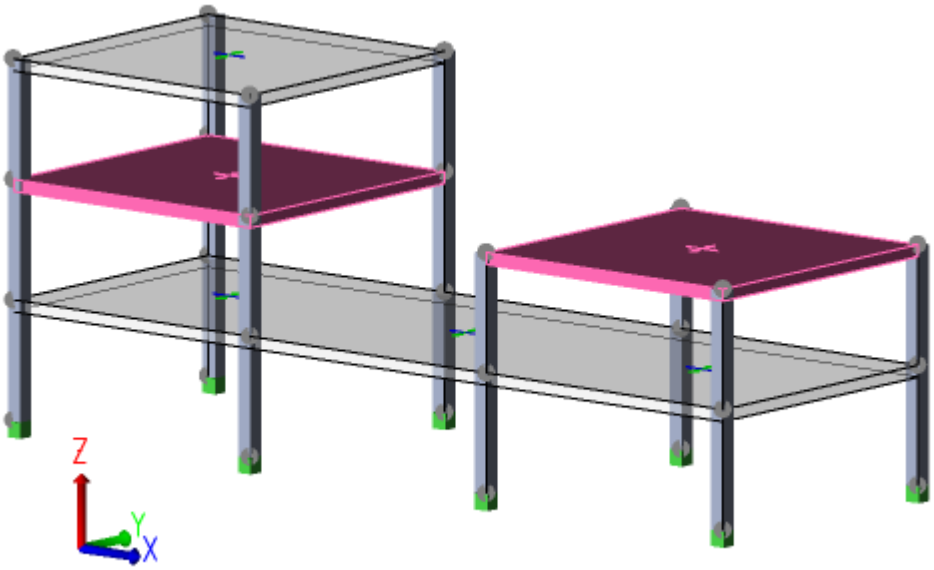
Limitations of wind decomposition to diaphragms

Certain building shapes need extra consideration if simple wind loads have been applied, or if wind loads have been applied to wall panels set to decompose to diaphragms.

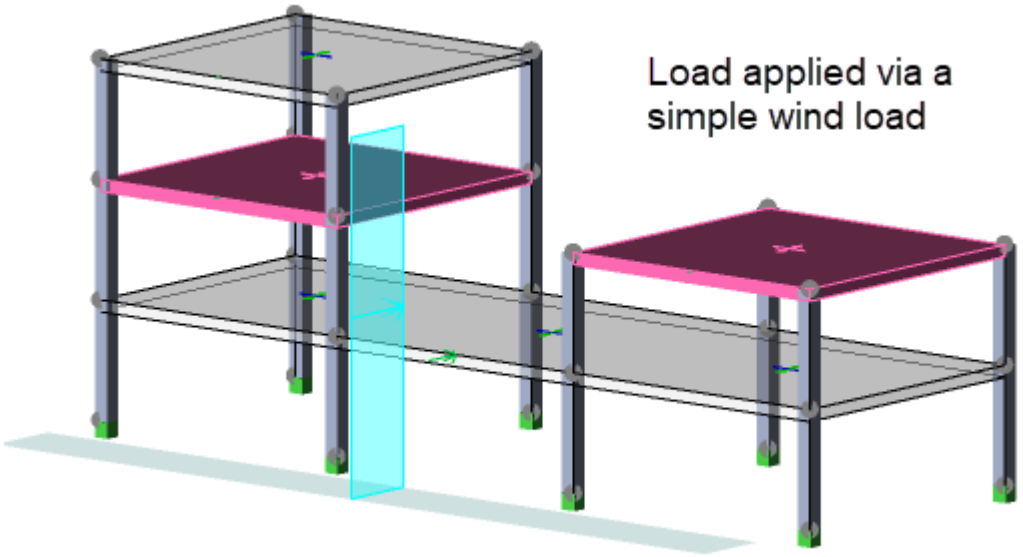
As illustrated by the following examples, buildings containing discrete towers (and thus containing disconnected rigid diaphragms) are a particular concern.

Wind load perpendicular to disconnected diaphragms

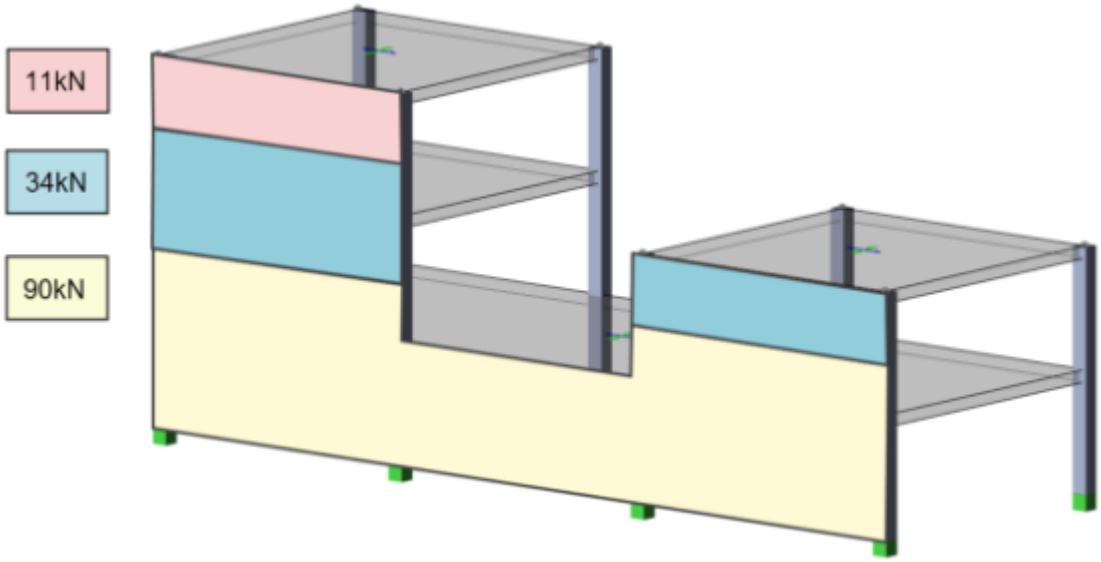
In this example wind load is to be applied in the Global Y direction, perpendicular to the disconnected diaphragms that exist in the highlighted slabs at the second floor level shown below.



An issue arises when the wind load is applied as an area load that has to be decomposed to both diaphragms. This could happen when the load is applied either via a Simple Wind load, or a wind wall panel:

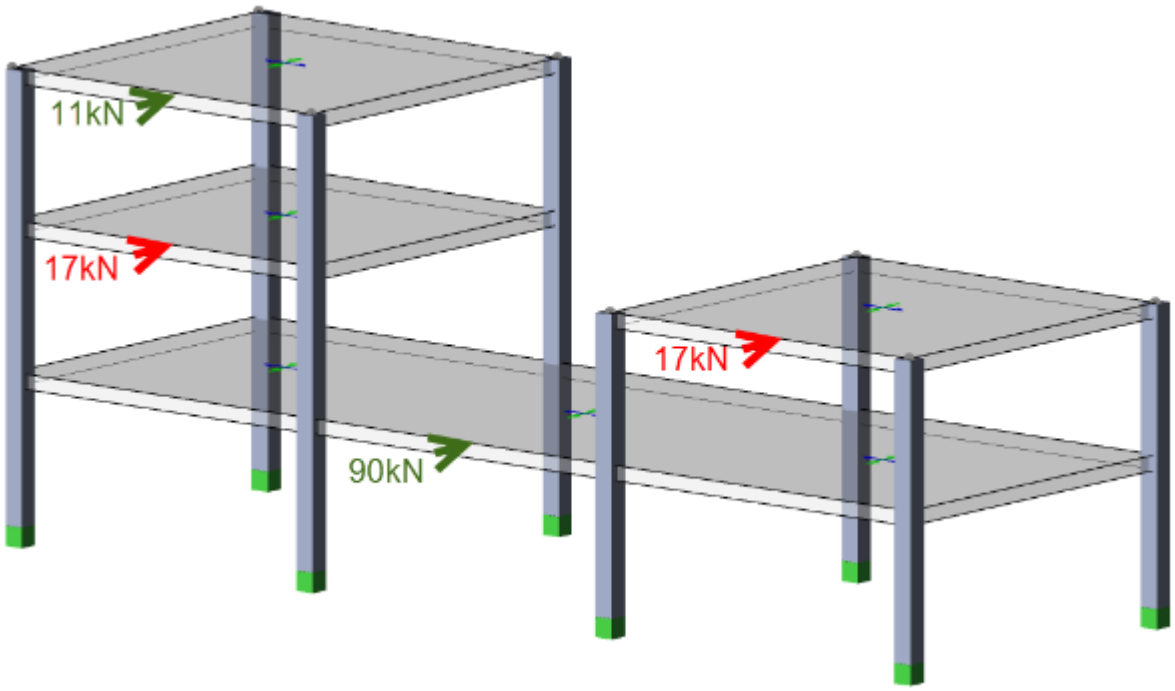


Irrespective of the method used to apply it, the area load within the building profile is shared between levels prior to decomposition.

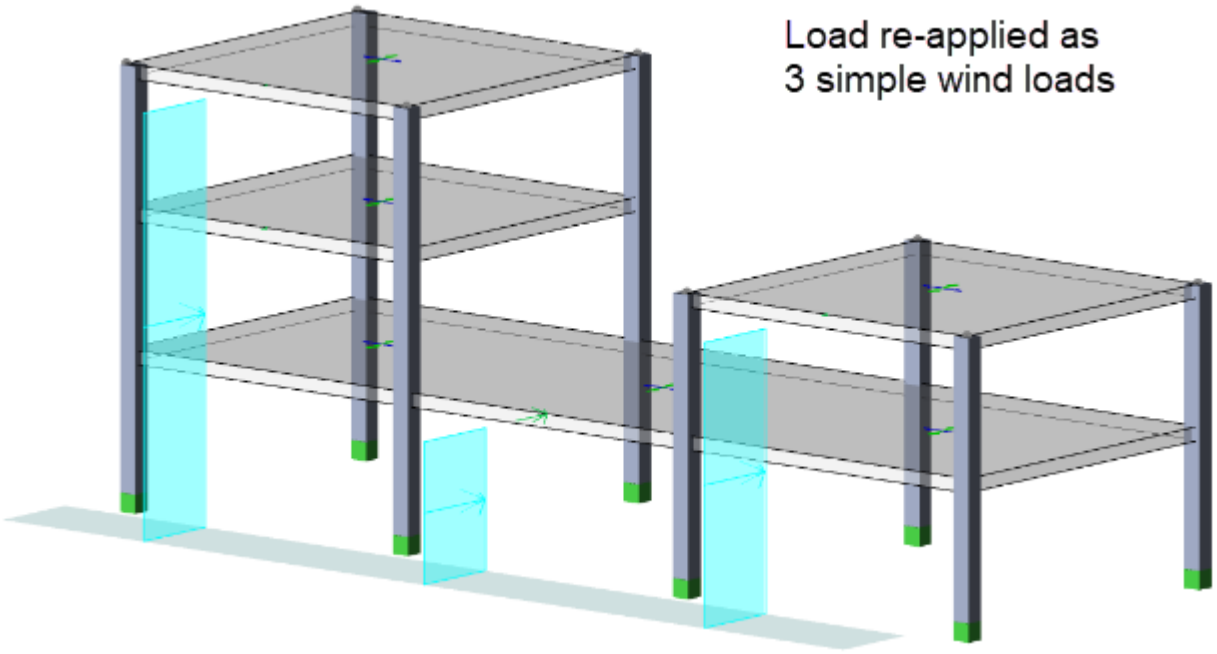


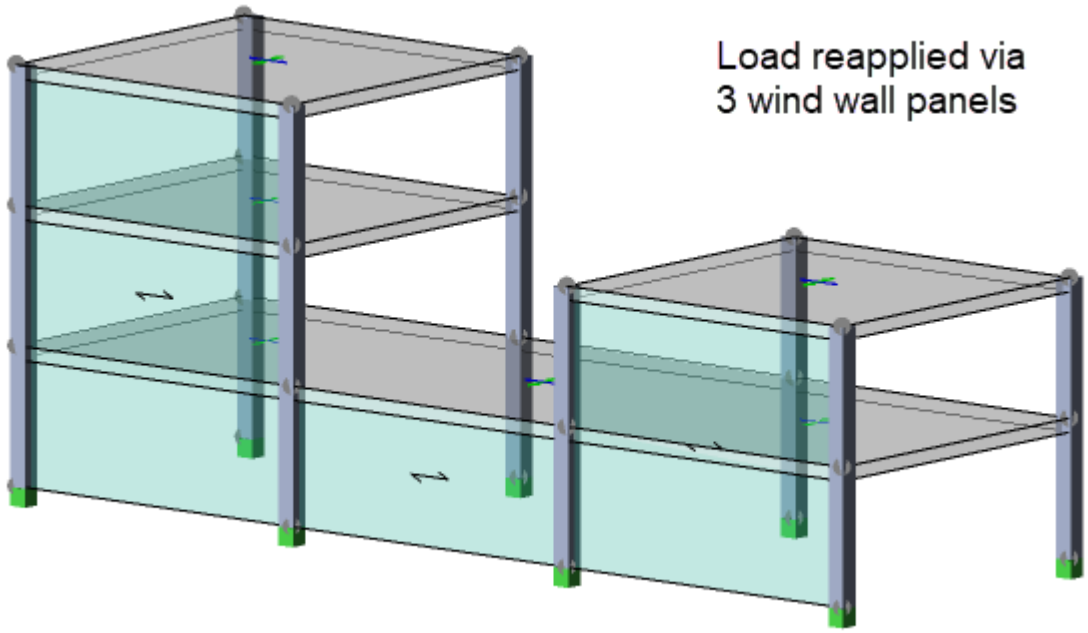
The load is then decomposed to the diaphragms at each level **in proportion to the width of each diaphragm**.

In this case because both diaphragms the second floor level are of equal width, the load is shared equally between them. This is not satisfactory as more of the load should have been applied to the left hand diaphragm in this case.

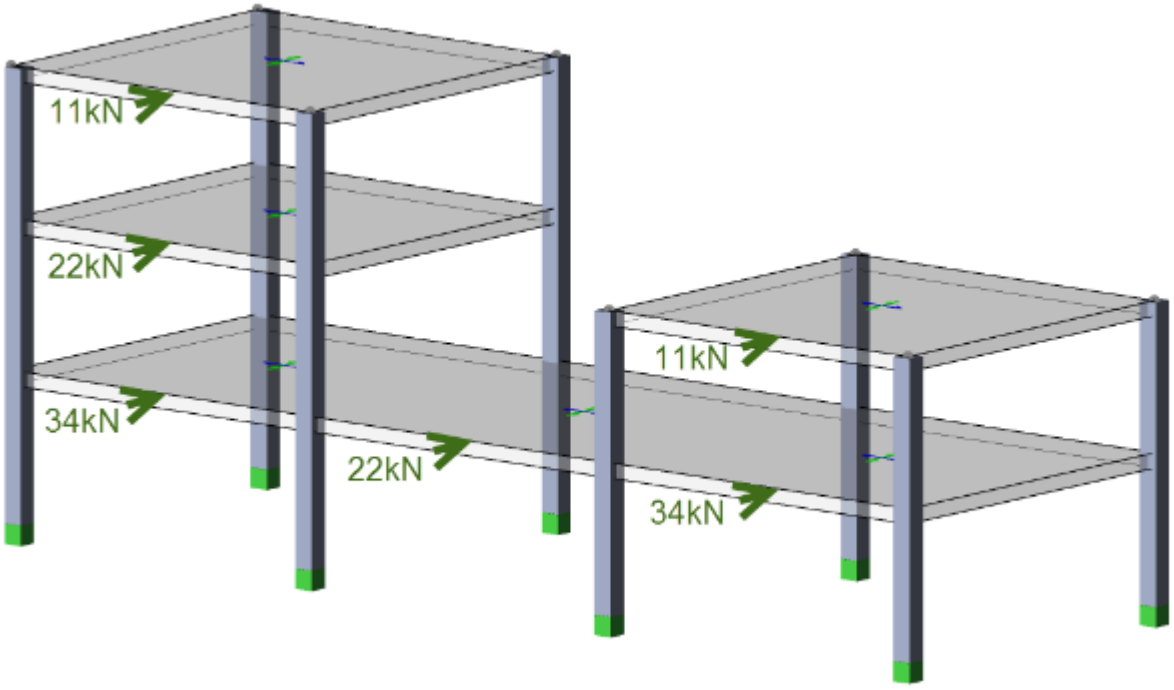


The workaround varies according to the method of loading, but basically involves replacing the original load with separate loads in each bay:



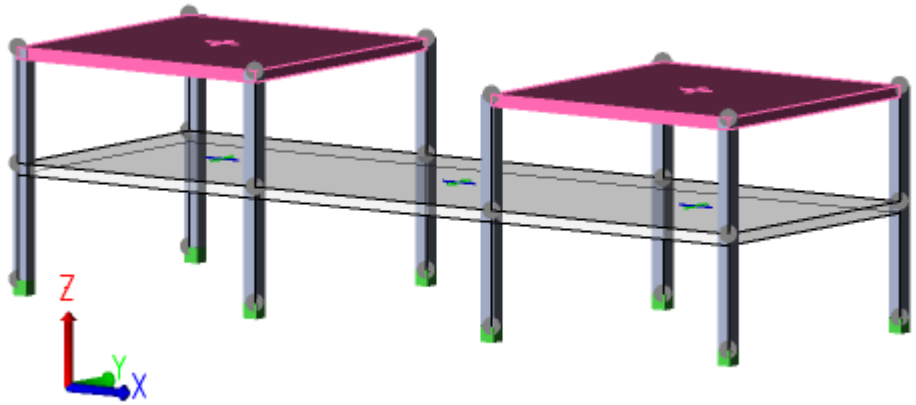


In both the above cases, the load is then decomposed as originally intended.

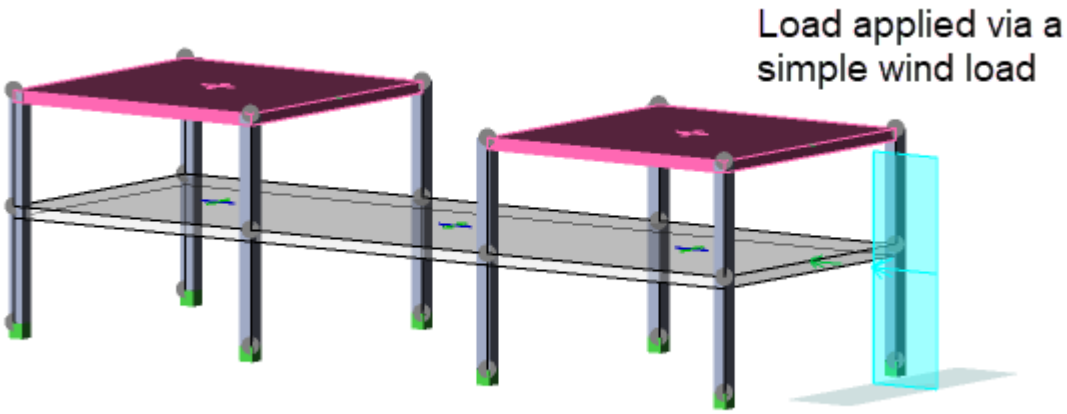


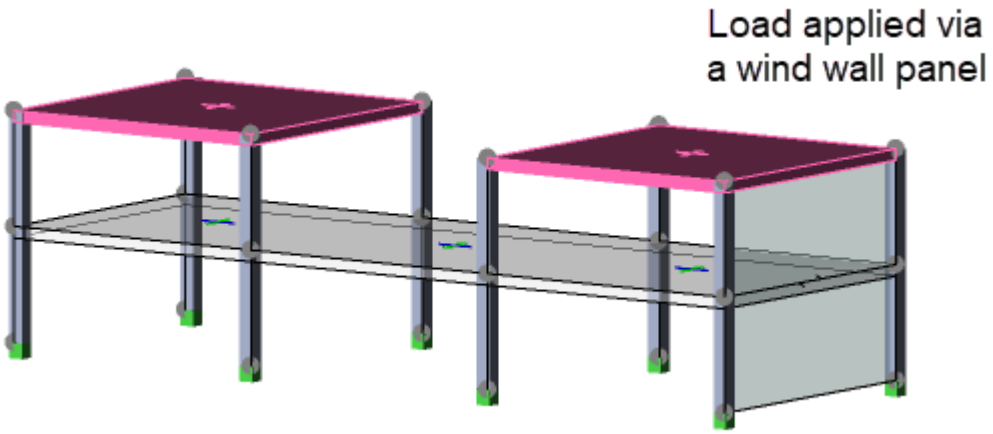
Wind load parallel to disconnected diaphragms

In this example although the two blocks are now the same height, another issue arises when the wind load is applied in the Global X direction, i.e. parallel to the disconnected diaphragms at the second floor level:

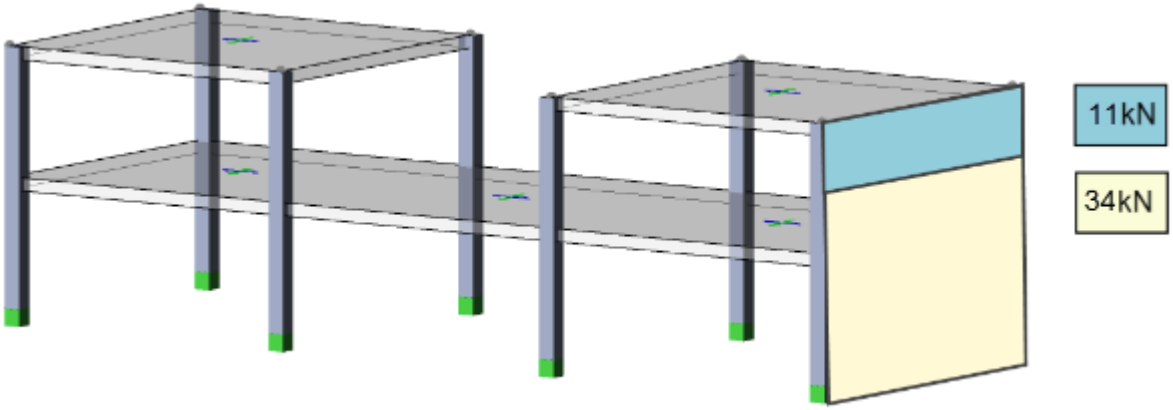


The issue arises because one diaphragm is 'hidden' from the applied load by the other diaphragm. The issue occurs irrespective of whether the load is applied via a Simple Wind load, or a wind wall panel:

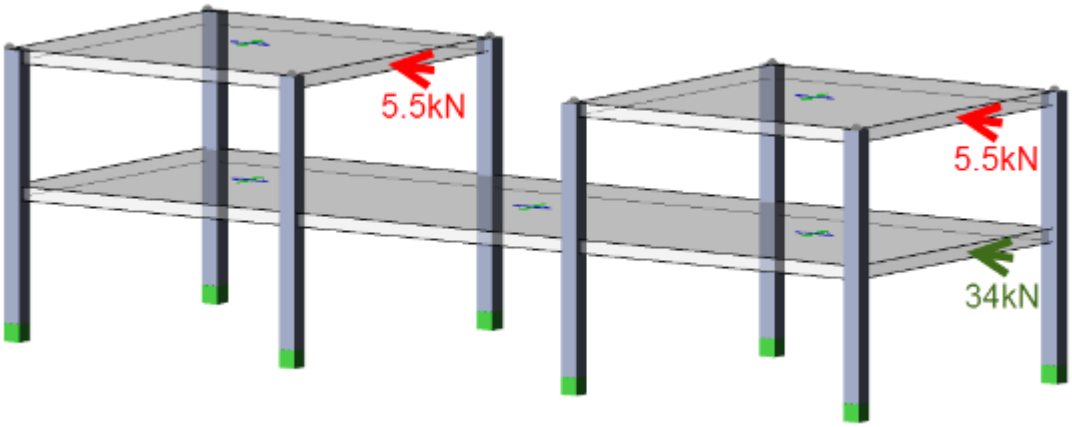




The area load within the building profile is first shared between levels prior to decomposition.



At the second floor level, instead of all the load being decomposed to the diaphragm facing the wind; because it is decomposed **in proportion to the width of each diaphragm at that level,** it ends up being shared equally to both.



To avoid the load being shared equally:

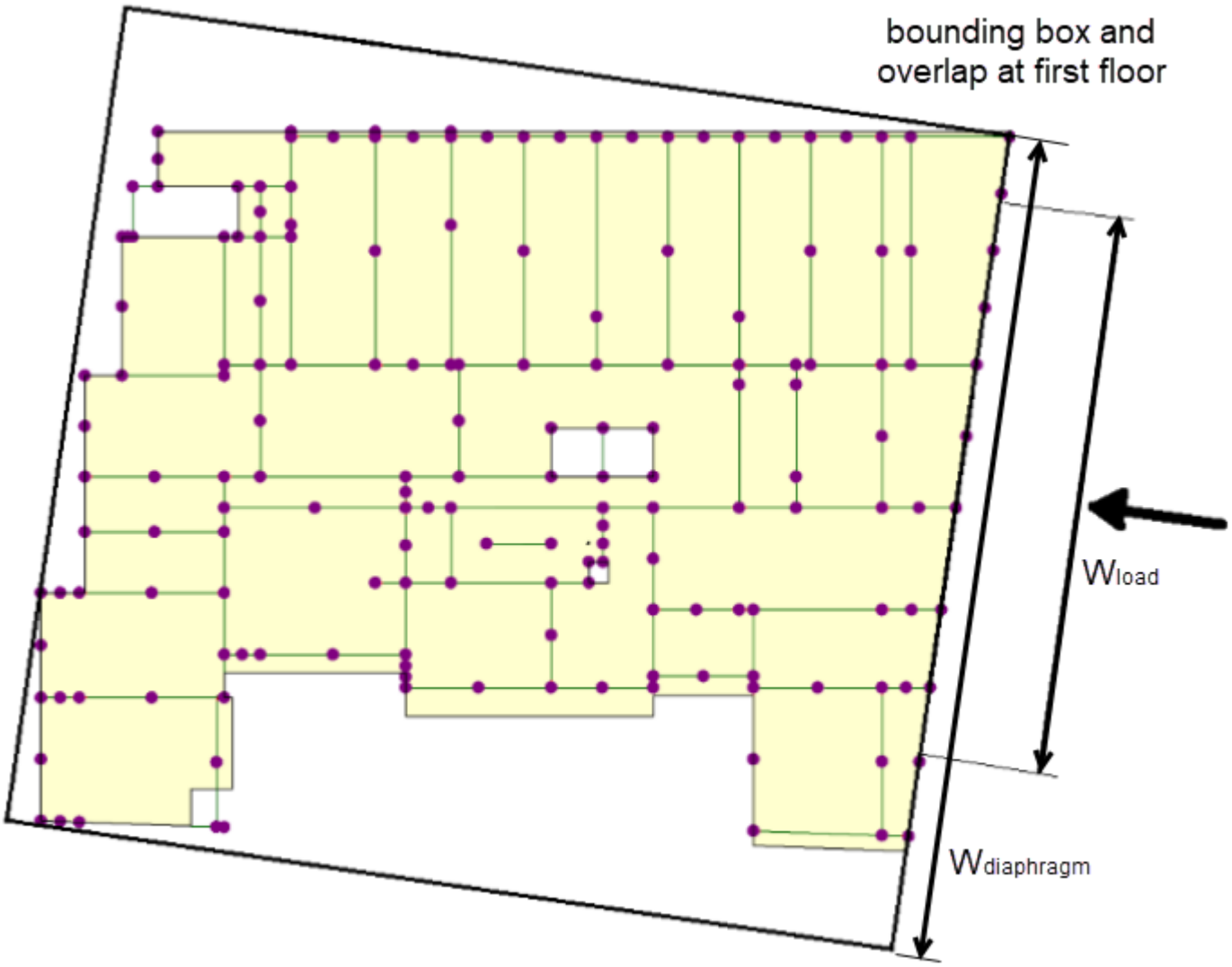
- if using wind wall panels - you would need to decompose to members or nodes instead of to diaphragms
- if using Simple Wind loads - there is no workaround, you would have to manually input the loads as diaphragm loads instead.

Diaphragms less than 5% of simple wind load width ignored

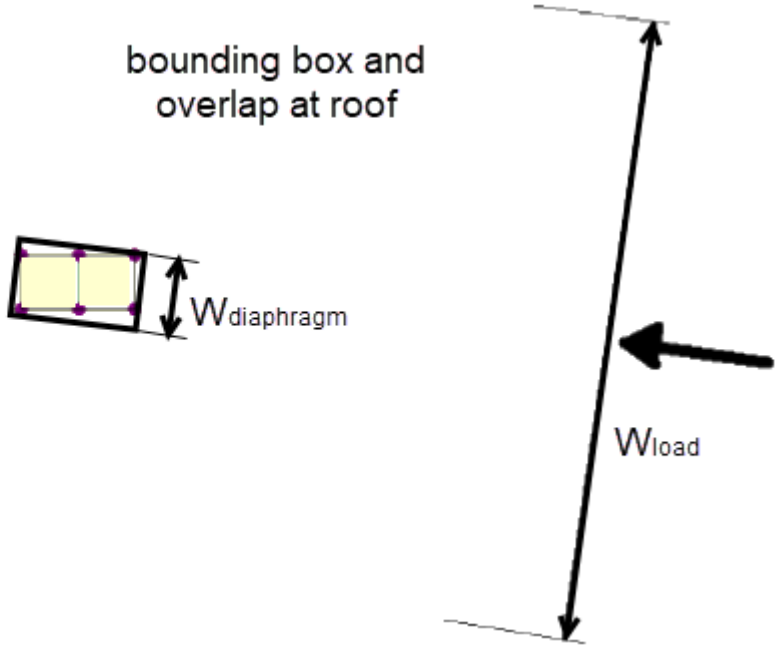
In Tekla Structural Designer, if the projected width of a rigid diaphragm in the direction that the load is acting is less than 5% of the simple wind load width it is assumed to be ineffective.

Consider the following example which consists of 4 levels and has a simple wind load applied over the full height of the building.

The first 3 levels are identical. The width of the diaphragm perpendicular to the load exceeds the width of the load, so the diaphragm at each of these levels is effective.



This changes at the roof (level 4). The only structure at this level is the lift core as shown below.

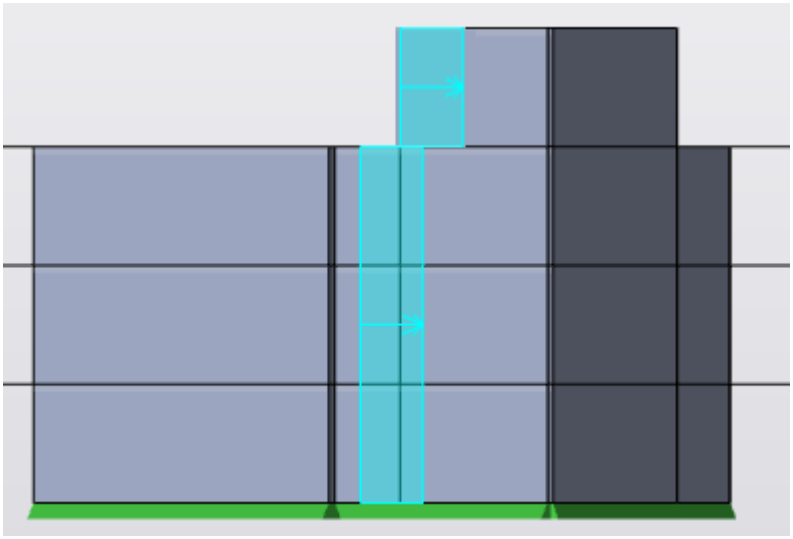


At this level the width of the diaphragm is now significantly smaller than the width of the load. In this case, because it is less than 5% of the load width, the diaphragm is ignored. As it is the only diaphragm at the top level a "Wind load profile applied above top diaphragm" warning message is displayed against that level in the Simple Wind Loading dialog when the load is created.

As it is only a warning it does not prevent the analysis from proceeding, however the diaphragm load applied to level 3 is significantly larger than it would otherwise have been.

This is because the *full* width is used for the loaded area above and below level 3 when calculating the diaphragm load to be applied to it.

To have the load calculated more accurately, the workaround would be to reduce the top level of the original simple wind load to align with level 3, then apply a second wind load from levels 3 to level 4 with a reduced width equal to that of the lift core.



Instructions for applying simple wind loads to wall/roof panels and open structure members

When working with simple wind pressure loads that you want to apply to wall/roof panels and/or open structure members you will be required to perform some or all of the following tasks:

Related task

Instructions for applying simple wind loads to diaphragms (page 398)

Set open structure entities to have wind load applied

NOTE This stage is only required if you are applying simple wind loads to open structure entities (I.e. structural C or equipment). If you are applying simple wind loads to wall/roof panels you do not need to select them - you can skip this stage.

1. Select the structural members, ancillaries, or equipment to which the load is to be applied.
2. In the **Properties** window, under **Wind loading**, select **Apply open structure wind load**

The above property is already selected by default for ancillaries and equipment, (but can be deselected if required).

TIP Instead of setting the above via the **Properties** window, it can be set graphically via Show/Alter State > (page 1011).

For members/ancillaries, once **Apply open structure wind load**has been selected, additional properties become available to allow the default shape factor and effective area factors used in the wind load calculation to be customized if required. For members, these properties can either be set for all stacks/spans, or individual stacks/spans.

Create loadcases for simple wind loads

1. On the **Load** tab, click  **Loadcases**.

   The **Loading** dialog box opens.
2. On the **Loadcases** page, click **Add**.
3. Name the new loadcase.
4. Set the type to **Wind**.
5. Click **OK**.

Create simple wind loads

RESTRICTION You can only create simple wind loads in the **Structure 3D** view.

NOTE In order access the **Simple Wind Loading** dialog box, you must first create and select the loadcase where you want to add the simple wind loads.

1. In the **Loading** list, select a manually created wind loadcase.
2. On the **Load** tab, click **Simple Wind**.
3. In the model, click to define the start point of the load width.
4. Click to define the end point of the load width.

The Simple Wind Loading dialog box (page 1583) opens. The dialog box allows you to specify a single area wind load from the lowest level of the building up to the highest level that contains a rigid diaphragm.

TIP The wind load does not have to begin at the lowest level of the building.

If you have one or more levels below ground, you can adjust the value of the lowest level in the Simple Wind Loading dialog box (page 1583).

---

5. In the **Apply to** droplist, select **Wall/Roof Panels & Open Structure Members**
6. Select the required Pressure (page 1587) option from the droplist.
7. Choose whether to extend the wind to the top the building (page 1588).
8. Choose whether to extend the wind across the building width (page 1590).
9. Specify the required load profile manually, or paste the load profile from a spreadsheet (page 1586).
10. To manually create a stepped profile up the height of the building, use the **Insert Above**, **Insert Below**, **Quick Above...** and **Quick Below...** buttons and then specify the load for each level.
11. When the load is complete, click **OK**.

Modify simple wind load vertical properties

1. Double-click anywhere in an existing simple wind load.

   The Simple Wind Loading dialog box (page 1583) opens.
2. Modify the load properties according to your needs.
3. Click **OK**.

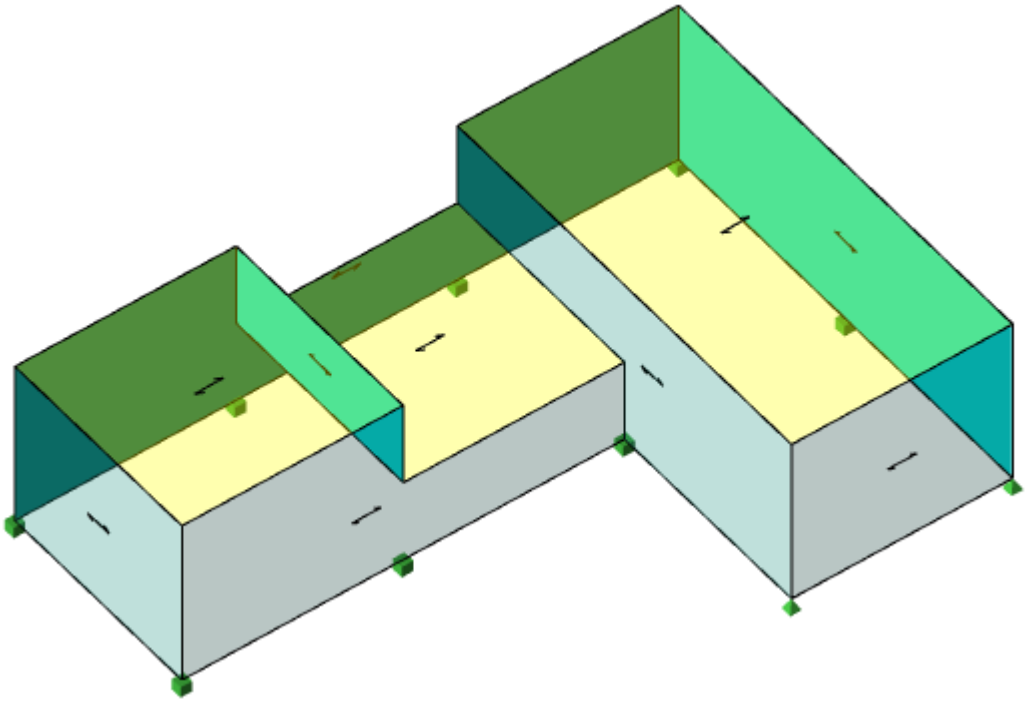
Modify the simple wind load width

1. Select the simple wind load that you want to modify.
2. Select one of the two nodes that define the load width.
3. Click where you want to move the selected node.

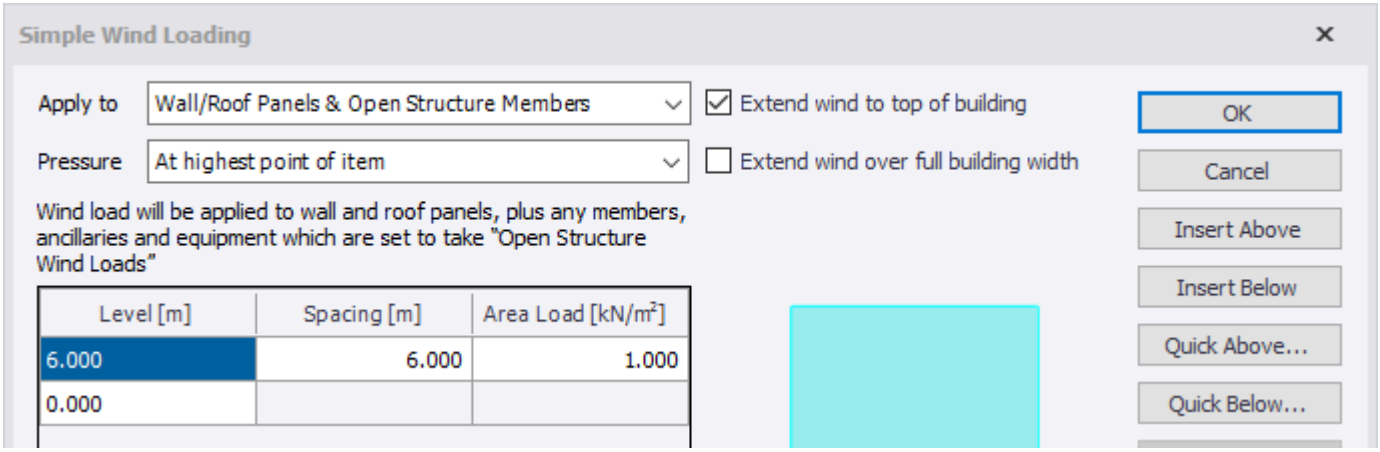
   Tekla Structural Designer moves the selected node to the new location, and regenerates the load.

Simple wind load example (panels)

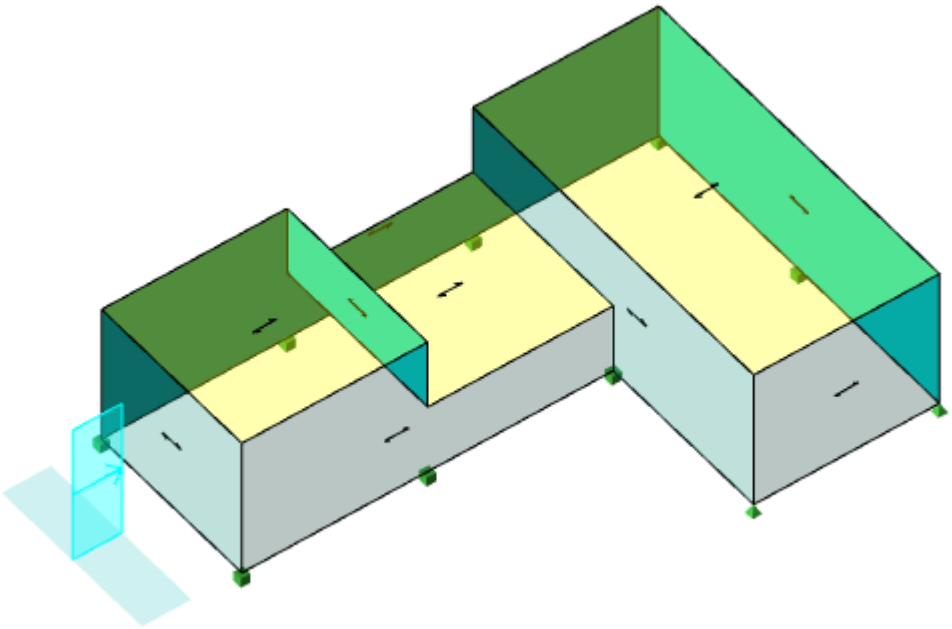
The example consists of a two level building clad in wall and roof panels.



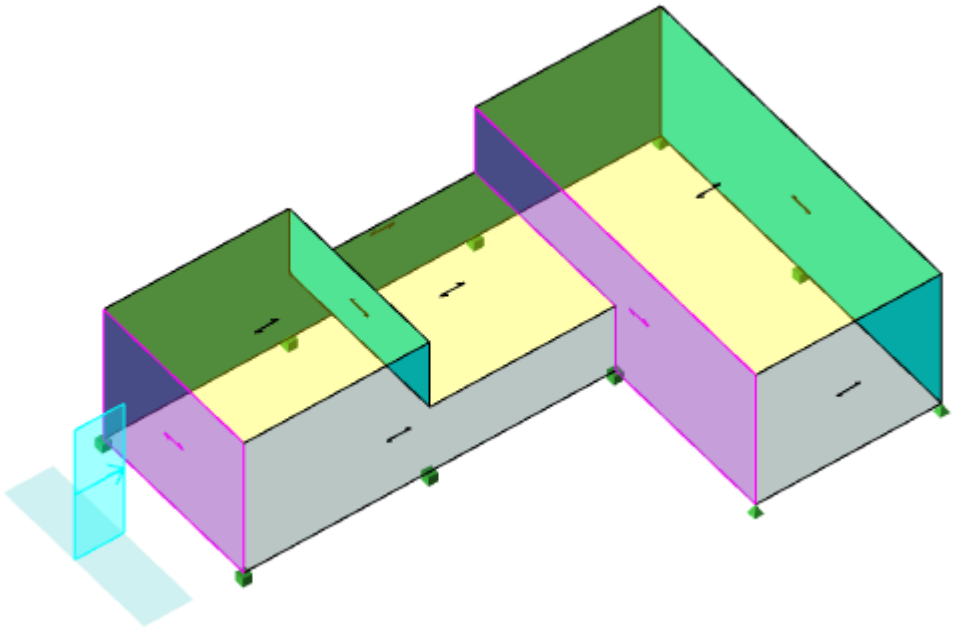
A simple wind load is applied at one end of the building. The load is uniform, applied to wall panels, and extends over the full building height, but not the full building width (as shown below).



The resulting simple wind load is as shown below:

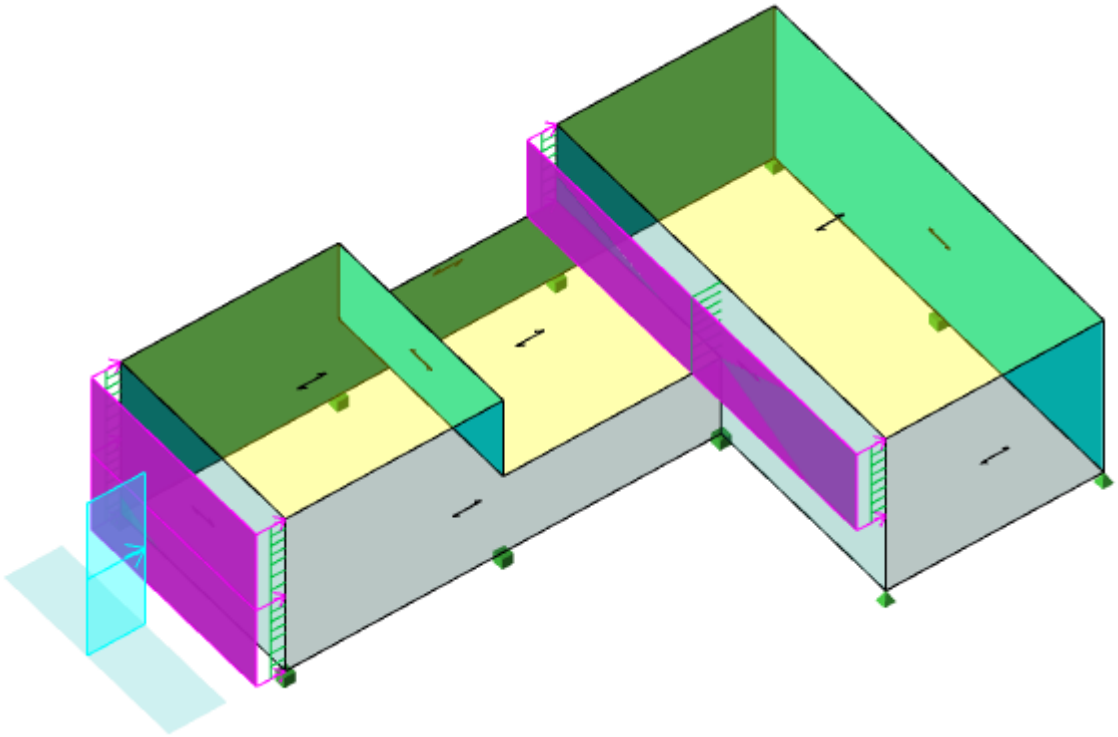


During analysis the simple wind load is applied to all panels within the load width and height that are facing the wind. For this example only the two panels highlighted below meet these criteria.



After analysis, the actual loads applied to the panels are highlighted below. These are in the form of polygonal area loads. (Because Decomposed Loads

are switched on in Scene Content these are also being displayed on the members supporting the two panels).



NOTE The following points should be noted in the above view:

- When applying simple wind loads to panels, the model is treated as an open structure. Therefore the simple wind load is 'seen' by both of the windward facing panels.
- The simple wind load has exactly the same area as the left hand panel and is applied over the panel's full width and height.
- Although the simple wind load is less wide than the right hand panel it still gets applied over the panel's full width. This is because only part of a panel width is required to overlap with the simple wind load width for it to be applied over the full panel width.
- Vertically, the simple wind load width does not overlap with the right hand panel until Level 1, thus it is only applied to the panel above that level.

Simple wind load workflow for open structures

The basic steps required to apply this method are as follows:

1. Select entities to have open structure wind load applied. (page 411)

2. Create loadcases for simple wind loads (page 398)
3. Create simple wind loads (page 412)
4. Combine the wind loadcases into design combinations
5. Perform the static design
6. Create an Open Structure Wind Load report (page 1084)

Open structure wind loads

For a wind pressure acting in a specified direction, open structure wind loads can be calculated for structural members, ancillaries or equipment as follows:

- **Structural members (beams, columns etc)** - wind pressure is determined, a distributed load is then calculated from this pressure, a shape factor and the area exposed to the wind (including fire protection if relevant)
- **Ancillary pipework and cable trays** - wind pressure determined, a distributed load is then calculated from this pressure, a shape factor and a calculated effective area (based on largest pipe for pipework and height for cable trays) (including insulation if relevant)
- **Ancillary ladders, guardrails and staircases** - wind pressure determined, a distributed load is then calculated from this pressure, a shape factor and a user defined effective area.
- **Equipment** - wind pressure determined, a point load is then calculated from this pressure, a shape factor and an effective area.

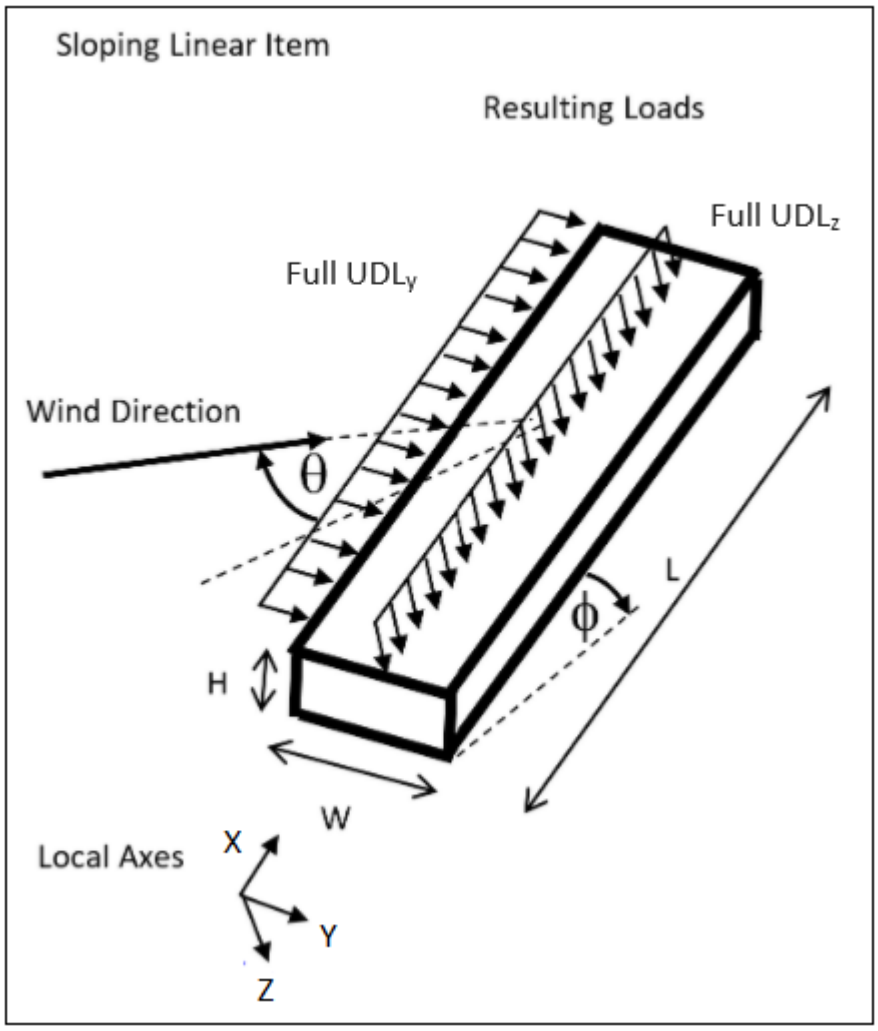
Open structure wind load workflow

The pressure can be applied by running the wind wizard to create a wind model, or, by creating a simple wind load. A workflow is provided for each.

- Wind wizard workflow for open structures (page 392)
- Simple wind load workflow for open structures (page 416)

Wind load calculation for members and ancillaries

Wind load is determined as two full length UDLs, one acting in local y (minor) and one acting in local z (major) on the member.

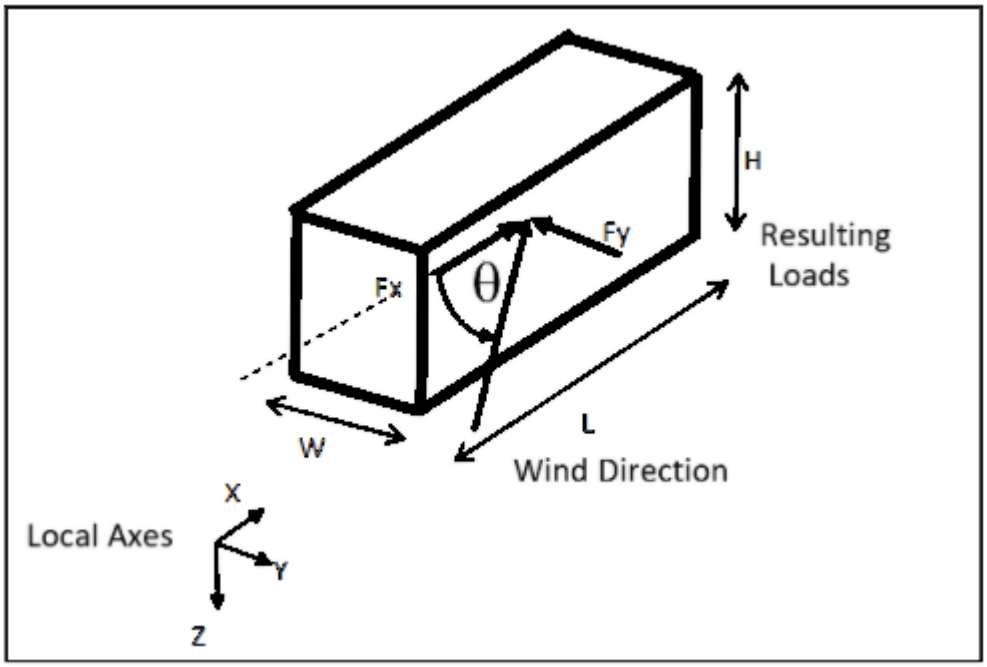


Full UDL_{Major}		$z \times C_f \times \text{Effective Area}_{xy} \times \text{Cos}(\theta) \times \text{Sin}(\varphi) / L$
Full UDL_{Minor}		$z \times C_f \times \text{Effective Area}_{xz} \times \text{Sin}(\theta) / L$
where		
z		wind pressure calculated at the height of the member for that wind direction (for vertical member average is taken at top and bottom level)
C_f		shape factor
Effective $Area_{xy}$		effective area of member in xy

Effective $Area_{xz}$		effective area of member in xz
θ		angle of the wind to the member x axis projected vertically into the horizontal plane
φ		angle of the member to the horizontal plane
L		length of the member

Wind load calculation for equipment

Wind load is determined as a single point load and applied to the CoG of the equipment at mid height.



F_{Wind}		z x C_f x Effective $Area_{xz}$ x Sin(θ) + Effective $Area_{yz}$ x Cos(θ))
F_x		F_{Wind} x Cos(θ)
F_y		F_{Wind} x Sin(θ)
where		
z		wind pressure calculated at the height of the equipment for that wind direction

C_f		shape factor =
Effective $Area_{xy}$		effective area of equipment in xy =
Effective $Area_{xz}$		effective area of equipment in xz =
θ		angle of the wind to the equipment x axis projected vertically into the horizontal plane =
L		length of the equipment =

NOTE All wind loads considered are horizontal, no vertical loading is considered.

Shape factors and effective area factors

The default shape factors (C_f) come from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE. These shape factors, and the effective area factors are assigned to members/ ancillaries and equipment as follows:

- **Members/ancillaries**
  - To specify the defaults for new members/ancillaries:
    - Edit the factors under **Model Settings > Loading > Wind Loading**
  - To edit the factors for existing members/ancillaries:
    - Select the member/ancillary
    - In the **Properties** window, under **Wind loading** select **Apply open structure wind load**
    - Edit the factors as required
- **Equipment**
  - The shape factors and effective area factors assigned to equipment are determined from **Table 1** below and cannot be edited.

Table 1: Equipment shape factors and effective area factors

Equipment type	Shape factor	Effective Area Side Elevation	Effective Area End Elevation
Sphere	0.5	π x $(D/2)^2$	π x $(D/2)^2$
Vertical cylinder	ASCE 7 Fig 6-21 see **Table 2** below	D x L	D x L
Horizontal cylinder -	ASCE 7 Fig 6-21 see **Table 2** below	D x L	-

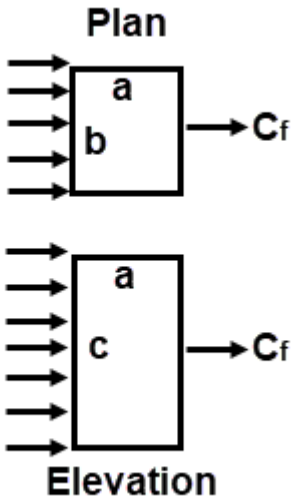
Table 1: Equipment shape factors and effective area factors			
Equipment type	**Shape factor**	**Effective Area Side Elevation**	**Effective Area End Elevation**
perpendicular wind			
Horizontal cylinder - longitudinal wind rounded end	0.5	-	$\pi \times (D/2)^2$
Horizontal cylinder - longitudinal wind flat end	1.2	-	$\pi \times (D/2)^2$
Cuboid	see **Table 3** below	H x L	H x B

Table 2: C_f for cylinders				
Cross-section	**Surface**	**L/D**		
		1	**7**	**25**
D x $\sqrt{q_z}$ > 5.3 (metric units)	Moderately smooth	0.5	0.6	0.7
D x $\sqrt{q_z}$ > 2.5 (US customary units)	Rough	0.7	0.8	0.9
	Very rough	0.8	1.0	1.2
D x $\sqrt{q_z}$ > 5.3 (metric units) D x $\sqrt{q_z}$ > 2.5 (US customary units)	All	0.7	0.8	1.2

NOTE

- in metric units D in m and q_z in N/m^2
- in US customary units D in ft and q_z in psf
- interpolation can be used between h/D values

Table 3: C_f for cuboids



c = h

a = L or B whichever is in line with the wind direction

b = B or L whichever is perpendicular to the wind direction

c/b	a/b								Max C_f	
	0.00	**0.25**	**0.5**	**1.00**	**1.50**	**2.00**	**2.50**	**3.00**	**C_f**	**a/b**
0.25	1.20	1.20	1.00	0.92	0.91	0.91	0.91	0.90	1.23	0.20
0.5	1.16	1.21	1.16	0.97	0.92	0.91	0.91	0.90	1.21	0.34
1.00	1.14	1.15	1.18	1.16	1.02	0.95	0.92	0.90	1.20	0.70
2.00	1.16	1.17	1.19	1.17	1.04	0.97	0.93	0.92	1.21	0.70
5.00	1.21	1.22	1.24	1.22	1.11	1.03	0.99	0.97	1.27	0.70
8.00	1.28	1.29	1.35	1.30	1.17	1.09	1.04	1.01	1.36	0.70
10.00	1.31	1.34	1.41	1.36	1.24	1.14	1.08	1.04	1.46	0.70

NOTE This Table is generated from Figure 1 of ESDU (1978) Fluid Forces, Pressures and Moments on Rectangular Blocks, ESDU Data Item 71016, Engineering Sciences Data Unit, London.

Assumptions and limitations

Assumptions

- All wind loads are assumed to act perpendicular to the members, equipment and ancillaries

- Applicable for all 1D structural members irrespective of their characteristic except
  - analysis elements
  - DELTABEAM
  - FABSEC beams

Limitations

- Only currently applicable for US, EC and IS wind codes
- Cannot be applied on structural walls
- Shielding effect is not considered
- There are no frictional wind forces accounted on any surfaces

Open structure wind loads report

An Open Structure Wind Load report (page 1084) can be generated for the open structure loads in user defined wind load cases on members, ancillaries and equipment.

Manual wind loads

Wind loads can be applied directly to the structure using panel, member, nodal, or diaphragm loads.

Diaphragm loading is commonly used to apply the loads returned from wind tunnel testing; Tekla Structural Designer permits the easy flow of information required for this, including:

- transfer of information out from Tekla Structural Designer to the wind specialists
- copy/paste functionality to support the rapid application of loads returned

Click the links below to find out more:

- Wind tunnel testing overview (page 423)
- Exporting wind tunnel data workflow (page 424)
- Using imported wind tunnel information (page 426)

Wind tunnel testing overview

There are four steps required in the process of using Tekla Structural Designer design within the process of wind tunnel testing.

1. Tekla Structural Designer creates a “Tekla Structural Designer - Wind Tunnel Report” from a vibration analysis to be used by the wind specialists.

   See: Exporting wind tunnel data workflow (page 424)

2. The “Tekla Structural Designer - Wind Tunnel Report” is sent to the wind specialists who undertake the wind tunnel tests. At the end of the tests, they create a “Wind Tunnel - Load Report” which they send back to the structural designer.
3. The designer manipulates the “Wind Tunnel - Load Report” to generate 24 loadcases with F_{Dir1}, F_{Dir2} & M_Z loads applied to the centres of mass of the rigid diaphragms at each floor/level.

   See: Using imported wind tunnel information (page 426)

4. The 24 wind loadcases are combined in regular combinations to be included as part of the design process in Tekla Structural Designer

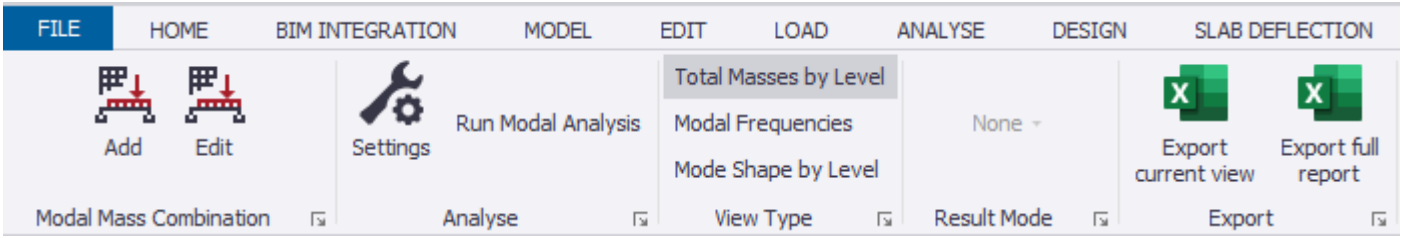
Exporting wind tunnel data workflow

Once the model has been created and had loads applied in Tekla Structural Designer, the basic workflow for generating and exporting wind tunnel data is described below.

Display the Wind Tunnel Data ribbon

The workflow is initiated by clicking **Wind Tunnel** on the Analyze ribbon.

- This enables the **Wind Tunnel Data** ribbon below:



- You can work through this from left to right...

Add modal mass combinations

1. On the **Wind Tunnel Data** tab, click  **Add**.

   The Loading dialog box (page 1557) opens on the **Combinations** page with a new **Modal Mass** class of combination added in the left hand pane.
2. Rename the combination as required.
3. Select the loadcases that you want to add in the combination, and click **>>**
4. On the **Applied mass** tab, set the directions to be considered, and if necessary, specify the level below which mass can be ignored.
5. Click **OK** to save the mass combination.

   Tekla Structural Designer adds the mass combination to the list of combinations in the **Loading** list.

Review the modal analysis settings

1. On the **Wind Tunnel Data** tab, click **Settings**

   The **Analysis Settings** dialog opens on the 1st Order Modal (page 763) page.
2. Select the total number of modes to be calculated.
3. Review and modify the other analysis settings according to your needs.

Run a modal analysis

1. On the **Wind Tunnel Data** tab, click **Run Modal Analysis**.

   Tekla Structural Designer analyzes the model.

Review the results

After modal analysis the following results are available and can be reviewed as tabular data:

- Masses, Center of Mass, and Mass Moment of Inertia by level,
- Modal Frequencies,
- Mode shape by level

Mode shapes can also be displayed graphically (page 701) if required by switching to a **Results View**.

Export the report

To export all the tabular results that can be viewed in the **Wind Tunnel Data View**, click **Export full report**.

Alternatively, you can export just the tabular results that are displayed in the current view by clicking **Export current view**.

Using imported wind tunnel information

The Tekla Structural Designer workflow for importing and using the wind tunnel information provided by wind specialists is as follows:

Create loadcases for the diaphragm loads

The wind tunnel report typically contains 24 loadcases with F_{Dir1}, F_{Dir2} & M_Z loads applied to the centres of mass of the rigid diaphragms at each floor/level.

Before the loads can be imported from the report, the engineer has to manually create wind loadcases to hold them.

Apply the diaphragm loads in each loadcase

Diaphragm loads can either be pasted into Tekla Structural Designer or entered manually.

NOTE Diaphragm loads can only be pasted at floors/levels with one or more horizontal rigid or semi rigid diaphragm.

See: Diaphragm loads and diaphragm load tables (page 362)

Create load combinations

Load combinations are created manually. The Wind Tunnel loadcases behave as any other "Wind" type loadcase when being used in combinations.

Analysis and design

A standard analysis and design process is followed, in which:

- the graphical analysis results for wind tunnel loadcases and combinations being shown in exactly the same way as for any other wind type loadcases and combination,
- wind tunnel loadcases are used in drift checks in the same way as other wind type loadcases,
- wind tunnel loadcases are used in design combinations in the same way as other wind type loadcases.

3.4 Apply snow loading

Snow loads are applied via the snow wizard, the functionality of which varies according to the regional code being worked to:

- US regional code - snow model data is generated, snow loadcases are set up, loads are created in the balanced loadcase

- EC regional code - snow model data is generated, snow loadcases are set up, loads are created in the undrifted loadcase
- AS, BS or IS regional code - snow loadcases are set up

Irrespective of the regional code, a typical workflow for running the snow wizard is as follows:

1. Prior to running the snow wizard ensure that roofed areas of the model are 'clad' with roof panels that have had their roof panel types (page 427) set appropriately.
2. Run the snow wizard (page 429).

   This results in a series of snow loadcases being set up.
3. Manually apply loads to loadcases on completion of the snow wizard (page 431) as necessary.
4. Include the snow loadcases in combinations in the usual manner - either using the Combination Generator or manually.

Snow load tasks

Run the snow wizard (page 429)

Apply loads to loadcases on completion of the snow wizard (page 431)

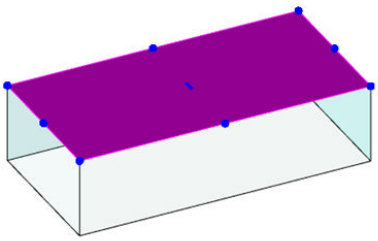
Update snow loads (page 433)

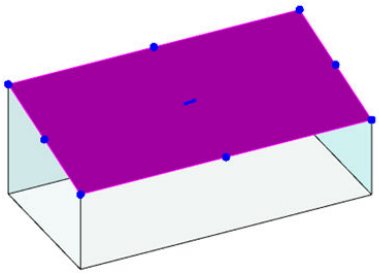
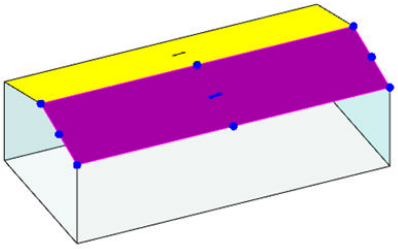
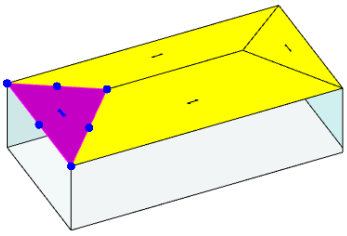
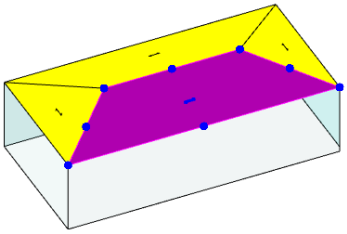
Delete the snow model (page 434)

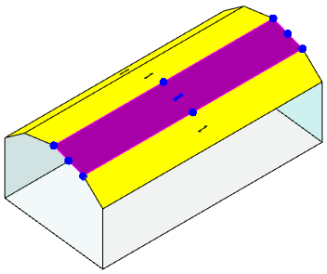
How do I define snow loading? (Playlist)

Roof panel types

For wind loading calculations it is important that the correct roof type has been assigned to each roof panel, the following **RoofType** choices being available.

RoofType	Example
Default (if roof slope < 5 degrees) and **Flat**	

RoofType	Example
Default (if roof slope ≥ 5 degrees) and **Monopitch**	
Duopitch	
Hip Gable	
Hip Main	

RoofType	Example
Mansard	

For snow loading calculations, Tekla Structural Designer only has to distinguish between monopitch and pitched roofs. This is determined from the **RoofType** as follows:

RoofType	For snow loading this is considered as:
Default	Monopitch
Flat	Monopitch
Monopitch	Monopitch
Duopitch	Pitched
Hip Gable	Pitched
Hip Main	Pitched
Mansard	Pitched

Run the snow wizard

The snow wizard that is run will vary according to the regional code being worked to.

How do I define snow loading? (Playlist)

Run the snow wizard (ASCE 7)

To run the ASCE 7 version of the snow wizard you should firstly ensure the snow loading code has been set as ASCE 7 in **Home > Model Settings > Design Codes**, then:

1. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

   The opens.

2. On the **Basic Data** page of the snow wizard, define the basic data.

3. Click **Loadcases** to access the second page. From here you specify the snow loadcases that you want to generate, and the number of drift directions to be considered.
4. Click **OK**.

The basic snow data and snow loadcases are set up, and the snow loads within the balanced snow loadcase are created.

NOTE Loads in drift snow loadcases are not created and must therefore be manually applied. For more information, see Apply loads to loadcases on completion of the snow wizard (page 431).

Run the snow wizard (Eurocode)

To run the snow wizard:

To run the Eurocode version of the snow wizard you should firstly ensure the correct snow loading code national annex has been set in **Home > Model Settings > Design Codes**, then:

1. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

   The opens.
2. On the **Basic Data** page of the snow wizard, define the basic data.
3. Click **Loadcases** to access the second page. From here you specify the snow loadcases that you want to generate, and the number of drift directions to be considered.
4. Click **OK**.

The basic snow data and snow loadcases are set up, and the snow loads within the undrifted snow loadcase are created.

NOTE Loads in drift snow loadcases are not created and must therefore be manually applied. For more information, see Apply loads to loadcases on completion of the snow wizard (page 431).

Run the snow wizard (AS, BS, or IS)

To run the snow wizard:

To run these versions of the snow wizard you should firstly ensure the appropriate snow loading code has been set in **Home > Model Settings > Design Codes**, then:

1. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

   The snow wizard opens.
2. Specify whether you want to generate a balanced snow loadcase, and also the number of unbalanced and drift snow loadcases to be created.

3. Click **OK**.

The requested snow loadcases set up.

NOTE Loads within the snow loadcases must be manually applied. For more information, see Apply loads to loadcases on completion of the snow wizard (page 431).

Apply loads to loadcases on completion of the snow wizard

On completion of the snow wizard loadcases will have been set up for four types of snow load - namely

- balanced or undrifted snow load (even UDL)
- unbalanced or drifted snow load (wind blown leeward and windward UDLs)
- valley drifted snow (wind blown VDLs on the two adjacent slopes in a valley)
- local drifted snow (VDL on UDL - snow load blown against a parapet, eaves step or object)

If working to the US or EC regional code - loads in the unbalanced or drifted snow loadcases have to be applied manually as uniform snow, valley snow, or local drift snow loads.

If working to the AS, BS, or IS regional code - loads in all snow loadcases have to be applied manually as uniform snow, valley snow, or local drift snow loads.

NOTE Any snow loads that are not appropriate for the selected loadcase cannot be selected.

Apply uniform snow loads

1. In the **Loading** list, select a snow loadcase generated by the wizard appropriate for uniform snow.
2. On the **Load** tab, click **Snow Load** --> **Uniform Snow**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click the relevant roof panel.

NOTE In the unbalanced or drifted cases, setting **Roof type** option to **Pitched** allows you to select whether the wind type of the load should be **Windward** or **Leeward**.

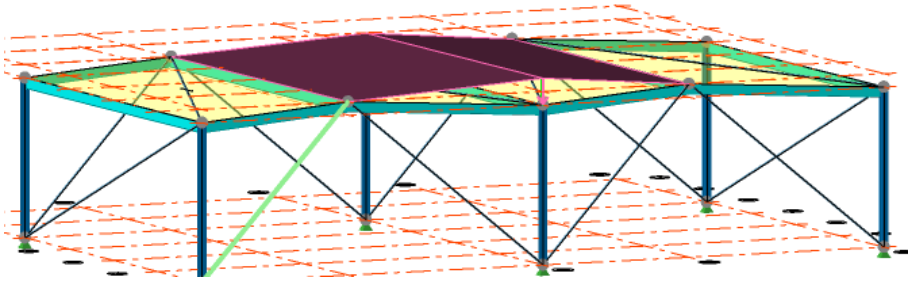
Apply a valley snow load

1. In the **Loading** list, select a snow loadcase generated by the wizard appropriate for valley snow.
2. On the **Load** tab, click **Snow Load** --> **Valley Snow**.
3. Select the left roof panel of the valley in which the load is applied.

   Tekla Structural Designer automatically determines the right panel, and calculates the valley load.

TIP If necessary, you can use apply a user override to the calculated load. Even if you rerun the **Snow Wizard**, Tekla Structural Designer maintains any user overrides that you have applied.

NOTE Graphic displays depict the loading magnitude of a snow load, and not depth of loading. Remember this when reviewing the load that has been calculated on structures like the low pitched valley model shown below:



Although the above image appears to indicate snow load rising above the ridges, the display actually views the loading magnitude.

Apply a local drift snow load

1. In the **Loading** list, select a snow loadcase generated by the wizard appropriate for local drift snow.
2. On the **Load** tab, click **Snow Load** --> **Local Drift Snow**.
3. Go to the **Properties** window.
4. Specify the drift type, and then enter the height, and any other parameters required to define the drift.
5. In the model, select the roof panel edge where the drift is to form, taking care to click close to the required edge.

TIP The edge closest to the cursor is highlighted, if this isn't where you want the drift to form move the cursor toward the edge required before clicking to select.

6. Click the start point of the drift on the highlighted edge.
7. Click the end point of the drift on the highlighted edge.

TIP If necessary, you can use apply a user override to the calculated load. Even if you rerun the **Snow Wizard**, Tekla Structural Designer maintains any user overrides that you have applied.

8. If you have another drift load to apply to the same panel in the same loadcase, repeat the above steps starting from step 3.

NOTE If working to the US regional code, when applying the second and subsequent drift loads to the same roof panel, a **Do not apply balanced load** setting is automatically enabled in the load properties. (This setting was disabled for the first drift load to ensure the balanced load gets applied.) The second and subsequent drift loads use the balanced load of the first drift to determine their drift load and extent. Working in this way, it is not necessary to override the load value in order to apply the second and subsequent drift loads.

Override snow loads

You can apply user overrides for snow load values and drift lengths, if necessary.

1. Select the snow load.
2. In the **Properties** window, select the **Override load value** option.
3. Specify the override value.

   Tekla Structural Designer maintains the user overrides, even if you rerun the **Snow Wizard**.

Update snow loads

If you have created a snow model using the **Snow Wizard**, and later decide to modify the roof geometry, the snow loads need to be recalculated. To do so, see the following instructions.

- On the **Load** tab, click **Snow Load** --> **Update Snow Loads**.

See also

Delete the snow model (page 434)

Delete the snow model

Occasionally, you may need to delete the entire snow model, and start the process from scratch. In order to do so, see the following instructions.

- On the **Load** tab, click **Snow Load** --> **Delete Snow**.

  Tekla Structural Designer resets the basic data that was previously defined in the **Snow Wizard**, and deletes the snow loadcases.

3.5 Apply seismic loading

The seismic wizard is used to define all the parameters required to set up the seismic loadcases and combinations, as outlined below.

Select the seismic loading code

By default the action code for seismic loading is already set to match the regional code that is being worked to, but you can choose to generate seismic loads to a different action code if required. You might choose to do this for example if your selected region has no Seismic Loading code (and hence no seismic wizard variant), such as "United Kingdom (BS)".

WARNING If you change the action code for seismic loading so that it does not match relevant regional code, you as the user need to be absolutely dear as to the implications of making this change - loading codes and design codes are developed with balanced hidden and exposed safety factors. You must fully understand the implications of running a mismatched loading and design code if you make this change.

- Generated seismic loads are based on the selected **Seismic Loading** action code.
- Generated seismic load *combinations* are based on the **Combinations** action code (which is determined by the Region assigned to the model).
  - Thus no seismic combinations will be created by the Combination Generator for regional codes the action code of which does not consider seismic loading, such as British Standards. In such cases the engineer must manually create seismic combinations.

- Seismic Drift checks are based on the selected **Seismic Loading** action code (which is NOT determined by the Region assigned to the model).
- Conventional static design of members (to the selected Resistance codes) will be performed for the results of the seismic combinations.
- However, for regional codes that feature *specific* seismic member design checks for members of Seismic Force Resisting Systems - such as the USA regional code and seismic checks to AISC 341 and the seismic provisions of ACI 318 - these checks will NOT be performed in the case of mismatched seismic loading and design codes.

---

1. On the **Home** tab, click  **Model Settings**.

   The **Settings** dialog box opens.
2. Go to **Design Codes**.
3. On the **Action Codes** tab, select the Seismic Loading code as required.

   RESTRICTION The seismic wizard is not currently available for the Australian AS:1170.4 loading code variant.
4. Click **OK**.

Create seismic loads in the seismic wizard

1. On the **Load** tab, click  **Seismic Load** --> **Seismic Wizard...**

   The (page 436) opens.

   NOTE The parameters in the **Seismic Wizard** vary according to the loading code that you have selected in **Model Settings**.
2. In the **Seismic Wizard**, define the necessary parameters for the seismic loading and loadcases. To go to the next page, click **Next**.
3. Once you have defined the parameters, click **Finish**.

   The **Combination Generator** dialog box opens.
4. In the **Combination Generator** dialog box, define the seismic load combinations.
5. Click **Finish**.

Display the horizontal design spectrum

After running the **Seismic Wizard**, you can view the horizontal design spectrum.

1. On the **Load** tab, click  **Seismic Load** --> **Horizontal Spectrum**.
2. If necessary, in the **Properties** window, you can switch **Direction** between **Dir 1** and **Dir 2**.

Delete seismic loads

- On the **Load** tab, click **Seismic Load** --> **Delete Seismic**.

  Tekla Structural Designer deletes all the previously defined seismic loads in your model.

See also

The seismic wizard (page 436)

The seismic wizard

The seismic wizard is run to specify the seismic analysis method (RSA or ELF) and to set up the seismic loadcases and combinations.

The information that is entered in the seismic wizard will vary according to the seismic loading code that is being worked to:

- Using the ASCE 7 seismic wizard (page 436)
- Using the UBC 1997 seismic wizard (page 445)
- Using the Eurocode EN1998-1:2004 seismic wizard (page 452)
- Using the IS1893 seismic wizard (page 460)
- Code spectra and site specific spectra (page 465)
- Seismic loadcases (page 478)

Using the ASCE 7 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

- Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the ASCE 7 Horizontal Design Spectrum (page 468).
Site Specific Spectra (user defined - based on S_d and T)	User defined spectra are appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 465).
Site Specific Spectra (user defined - generic curve)	User defined spectra are appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve manually, or by pasting data from a spreadsheet. **NOTE** The spreadsheet data must be in the form of period vs acceleration values and in ascending order of period (the input dialog features an

Property	Description
	automatic Sort command if the latter is not the case). See: Code spectra and site specific spectra (page 465).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE 7-10 (not ASCE 7-05) there is an additional value of C_s calculated for a building < 5 stories and where T <0.5s. In this instance S_s is taken as 1.5 and the additional value of C_s should be used to give a final value of C_s. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of stories	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Site Occupancy	
Site class	This field allows you to set the appropriate soil conditions, A-E - (A - Hard rock, B - Rock, C - Very dense soil/soft rock, D - Stiff soil, E - Soft soil. as defined in ASCE 7-05 & ASCE-10 Table 20.3-1.
Occupancy/Risk Category	This field allows you to set the appropriate occupancy category, I-IV, as defined in ASCE 7-05 &-10 Table 1-1.

Property	Description
Importance Factor, I_e	This is automatically derived from the occupancy class.
Override checkbox	Checking this box allows you to override the importance factor, (some countries require different Importance Factors to those given in the US codes).
Seismic Design Category (SDC)	
Seismic Design Category (SDC)	User editable (range A-F)
Alternative seismic design category determination	Select this checkbox in order to use the alternative determination from IBC2009 1613.5.6.1 / IBC2012 1613.5.6.
User Defined SDC	Select this checkbox in order to specify your own SDC Category
Include vertical seismic load effect	This checkbox is only active for SDC B. It can be used to set the vertical seismic load effect to zero if required.
Max earthquake spectral response acceleration	
S_s – short period (0.2s)	This figure can be determined from the maps found in ASCE 7-05 &-10 (units % of g, range 1 – 500)
S_1 – 1.0s period	This figure can be determined from the maps found in ASCE 7-05 &-10 (units % of g, range 1 – 500)
Design spectral response spectral acceleration	
S_{DS} – short period	This is automatically determined from SS and the site class, according to ASCE 7-05&-10
S_{D1} – 1s period	This is automatically determined from S1 and the site class, according to ASCE 7-05&-10
Next	Unless the SDC is type A, clicking **Next** takes you to the **Structure Irregularities page** described below; for SDC type A only, clicking **Next** takes you to the **Seismic Loading page**.

Structure Irregularities page

For SDC types B to F this page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure • Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Use orthogonal seismic combinations	By default, the orthogonal seismic combination procedure is only activated when a **Plan irreg 5 - Non parallel systems** irregularity has been selected. However, by checking this box you can directly specify the use of orthogonal seismic combinations, (in which the seismic loadcases for both directions are included) when needed.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

For SDC types B to F this page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived from ASCE 7-05&-10 clause 12.8.2.1.
User defined fundamental period T	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period	
Long period transition period, T_L	This figure can be determined from figs 22-15, 16, 17, 18, 19 and 20 found in ASCE 7-05 or figs 22-12, 13, 14, 15 and 16 found in ASCE 7--10.
T_S	This value is derived ($T_s = S_{D1}/S_{DS}$)
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Steel moment resisting frames, Concrete moment resisting frames, Eccentrically braced steel frames or All other structural systems (ref ASCE 7-05 / ASCE 7-10 Table 12.8-2
Approx fundamental period, T_a	This is automatically derived from ASCE 7-05&-10 clause 12.8.2.1.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

For SDC types B to F this page is used to determine the response modification coefficient, R, and other factors in the X and Y directions.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	

Property	Description
System	This field allows you to set the appropriate system, (eg. bearing wall, building frame, moment resisting frame etc.) from ASCE 7-05 &-10 Table 12.2-1.
Type	This field allows you to set the appropriate type for the chosen system from ASCE 7-05 &-10 Table 12.2-1.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
System over-strength factor	This is automatically derived, but you are given the facility to edit the calculated value.
Deflection amplification factor, C_d	This is automatically derived, but you are given the facility to edit the calculated value.
Redundancy factor, ρ	This is automatically derived from the SDC, (SDC B-C ρ = 1.0, SDC D-F ρ = 1.3) but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those loadcases that you want to contribute to the Structure Seismic Dead Weight.

NOTE The effective seismic weight combination is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used
- define appropriate localization parameters for using the ASCE 7 code outside of the US.

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Structure Type	Select the structure type for determining the allowable story drift factor.
Risk Category	Displays the risk category that was selected on an earlier page of the wizard.
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift is determined from the above two parameters. Click 'override' if you want to enter your own value.
Include localization - Check this box in order to display the below 'ELF & RSA' and 'RSA only' localization settings.	
ELF & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (ASCE 7 code value = 1/(R/I)).
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (ASCE 7 code value V = Cs x W and Cs = SDS / (R /Ie)).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (ASCE 7 code value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination),(ASCE 7 factor = 0.2 x S_{DS} x Dead loads).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation, (ASCE 7 code value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard.

Property	Description
	NOTE It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (ASCE 7 code value: 85% x V / V_t).
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (ASCE 7 code value: 85% x V / V_t). **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 478) and open the **ASCE 7 Seismic Combination Generator** described below.

ASCE 7 Seismic Combination Generator

The Combination Generator sets up seismic combinations from the following ASCE 7 "core" combinations that include seismic loads.

ASD - Seismic Combinations

5) 1.0D + 0.714E

6) 1.0D + 0.75L + 0.536E + 0.75S

8) 0.6D + 0.714E

NOTE In ASCE, in the above 0.714 equates to 0.7 and 0.536 equates to 0.525. Tekla Structural Designer utilizes the ASCE factors - you can however change them if required.

LRFD - Seismic Combinations

5) 1.2D + 1.0E + 1.0L + 0.2S

7) 0.9D + 1.0E

(Where E is the seismic component of the loading in the combination.)

Property	Description
Initial Parameters page	

Property	Description
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this checkbox to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Using the UBC 1997 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure in accordance with the Uniform Building Code.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

1. Click Home > Model Settings > Design Codes
   a. Set the Design Code for Seismic Loading to UBC
   b. Click OK
2. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the UBC Horizontal Design Spectrum.
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 465).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE 7-10 (not ASCE 7-05) there is an additional value of C_s calculated for a building < 5 stories and where T <0.5s. In this instance S_s is taken as 1.5 and the additional value of C_s should be used to give a final value of C_s. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of stories	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Soil Profile Occupancy	
S - Soil Profile Type	This field allows you to set the appropriate soil conditions, A-E - (A - Hard rock, B - Rock, C - Very dense soil/soft rock, D - Stiff soil, E - Soft soil. As defined in UBC97 Table 16-J.
Occupancy Category	This field allows you to set the appropriate occupancy category, I-IV. As defined in UBC97 Table 16-K.

Property	Description
Importance Factor, I	This is automatically derived from the occupancy category.
Override checkbox	Checking this box allows you to override the importance factor, (some countries require different Importance Factors to those given in the US codes).
Seismic Zone	
Seismic Zone	This field allows you to set the appropriate zone, I-4. As defined in UBC97 Figure 16-2
Z - Seismic Zone Factor	This is automatically derived for the seismic zone.
Near Source (for Seismic Zone 4 only)	
Seismic Source Type	This field allows you to set the appropriate source type, A-C.
Distance to Seismic Source	The distance to the seismic source.
Next	Clicking **Next** takes you to the **Structure Irregularities page** described below.

Structure Irregularities page

For SDC types B to F this page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Static Force Procedure

Property	Description
	• Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

For SDC types B to F this page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived. If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Steel moment resisting frames, Concrete moment resisting frames, Eccentrically braced steel frames or All other structural systems
Approx fundamental period, T_a	This is automatically derived.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

This page is used to determine the response modification coefficient, R, and other factors in the directions 1 and 2.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	
System	This field allows you to set the appropriate system, (eg. bearing wall, building frame, moment resisting frame etc.) from UBC97 Table 16-N.
Type	This field allows you to set the appropriate type for the chosen system from UBC97 Table 16-N.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
System over-strength factor	This is automatically derived, but you are given the facility to edit the calculated value.
Redundancy factor drift, ρ_{drift}	This factor has to be entered manually.
Redundancy factor, ρ	This factor has to be entered manually.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those loadcases that you want to contribute to the Structure Seismic Dead Weight.

NOTE The effective seismic weight combination is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used

- define appropriate localization parameters for using the UBC code outside of the US

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Structure Type	Select the structure type for determining the allowable story drift factor.
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift is varies according to the Structure Type. Click 'override' if you want to enter your own value.
Include localization - Check this box in order to display the below 'ELF & RSA' and 'RSA only' localization settings.	
SFP & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (UBC code value = 1/R).
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (UBC code value $V = C_V \times I \times W / (R \times T)$).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (UBC code value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination), (UBC factor = 0.5 x Ca x I x Dead loads for strength design and 0.0 for allowable stress design).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation, (UBC code value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard.

Property	Description
	NOTE It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, UBC code values: Code ground motion = 90% x V / V_{Design} Site specific ground motion = 80% x V / V_{Design} Irregular structures = 100% x V / V_{Design}
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, UBC code values: Code ground motion = 90% x V / V_{Design} Site specific ground motion = 80% x V / V_{Design} Irregular structures = 100% x V / V_{Design} **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 478) and open the **UBC Seismic Combination Generator** described below.

UBC Seismic Combination Generator

The Combination Generator sets up seismic combinations from the following UBC 1997 "core" combinations that include seismic loads.

ASD - Seismic Combinations

5) 1.0D + 0.714E

6) 1.0D + 0.75L + 0.536E + 0.75S

8) 0.6D + 0.714E

NOTE In ASCE, in the above 0.714 equates to 0.7 and 0.536 equates to 0.525. Tekla Structural Designer utilizes the ASCE factors - you can however change them if required.

LRFD - Seismic Combinations

5) 1.2D + 1.0E + 1.0L + 0.2S

7) 0.9D + 1.0E

(Where E is the seismic component of the loading in the combination.)

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this checkbox to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Using the Eurocode EN1998-1:2004 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

- Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the EN 1998-1 Horizontal Design Spectrum (page 470).
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 465).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic inertia combination is determined.
Number of storeys	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Ground acceleration	
Region	Only applies for the Malaysia NA: • Peninsular Malaysia, • Sarawak, • Sabah
Reference Peak Ground Acc, a_{gR}	Refer to PD6698:2009

Property	Description
Design Ground Acc, a_g	a_{gR} x γI.
Importance & Ground	
Importance class	For Base Eurocode and UK NA: I- IV as defined in BS EN 1998-1:2004 - Table 4.3. For Singapore NA: Ordinary or Special
Ground type	For Base Eurocode and UK NA: A-E For Malaysia NA: Rock, Stiff Soil, Flexible Soil For Singapore NA: C, D, S_1 (A - Rock, B - Very dense soil/gravel/clay, C - Deposits of dense/ medium dense soil, D - Loose to medium soil, E - Surface alluvium. Ref BS EN 1998-1:2004 - Table 3.1.)
Importance Factor, γ_I	This is automatically derived from the occupancy class.
Spectrum type	For Base Eurocode and UK NA: Spectrum Type - 1 or 2 BS EN 1998-1:2004 - Cl 3.2.2.2.1(P) For Malaysia NA: Spectrum Type - 1 For Singapore and Norway NA: (No choice)
Lower bound factor, β	This is automatically derived.
Upper limit of the period of the constant spectral acceleratio n branch, T_c	This is automatically derived.
Structural Ductility Class	Low, Medium or High - for whole building (not directional) If ag ≤ 0.78 m/s2 or if ag x S ≤ 0.98 m/s2 - Structure suitable for Low Seismicity (where S is the soil factor) If not - Structure suitable for Medium or High Seismicity
Site natural period, T_S (Malaysia NA only)	This is only required if the ground type is Flexible Soil.
Elastic response	This is only required if the ground type is Flexible Soil.

Property	Description
spectral displacement, S_{DR} (Malaysia NA only)	
Next	Clicking **Next** takes you to the **Structure regularity page** described below.

NOTE The Norway NA for EC1998 has the following differences:-

Soil factors specific to Norway

- φ factor for Imposed load types A, B and C = 1.0 (not 0.8)

- γI importance factors specific to Norway- Spectrum specific to Norway (TB, TC and TD factors)

- q values for Ductility Class Medium use for both Medium and High

Structure regularity page

For Medium and High structural ductility this page is used to indicate any irregularities in plan or elevation. (For Low structural ductility only the irregularities in elevation are displayed.)

Property	Description
Structure Plan Regularity - Cl 4.2.3.2	Check the appropriate boxes to define any plan irregularities.
Structure Elevation Regularity - Cl 4.2.3.3	Check the appropriate boxes to define any elevation irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure • Use Modal Response Spectrum Analysi **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

This page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_A	If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures (ref BS EN 1998-1 Cl 4.3.3.2.2)
Approx fundamental period, T_A	This is automatically derived.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_A, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Behaviour Factor page** described below.

Behaviour Factor page

For Low Ductility Class, q = 1.5, (but the value can be changed by the user) - all other fields on the page are dimmed and cannot be changed.

For Medium or High Ductility Classes, this page is used to determine the Behaviour Factor in direction 1 and 2.

Property	Description
Ductility Class	Medium or High.
Structure Type	Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures

Property	Description
Frame Type	This options displayed here depend on the structure type selected above.
α_u/ α_l	This is a user defined multiplication factor - for the structure
User defined q	For Medium or High Ductility Classes, select this checkbox in order to edit the calculated q value.
Behaviour Factor, q	This is automatically derived, but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Seismic Inertia Combination page** described below.

Seismic Inertia Combination page

This page is used to set up the loadcases for the Seismic Inertia combination.

You should include those loadcases that you want to contribute to the effective seismic weight of the structure.

NOTE The Seismic Inertia combination is used to develop the seismic design loading and is classed as a modal mass combination for the modal analysis. It is not used in any other analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used
- define appropriate localization parameters

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Structure Type	Select the structure type for determining the allowable story drift factor.
Importance Class	Displays the risk category that was selected on an earlier page of the wizard.
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift is determined from the above two parameters. Click 'override' if you want to enter your own value.
-	
ELF & RSA	
Scale Factor	This field allows you to override the code value for the scale factor, (Eurocode value = 1/q).

Property	Description
between elastic and design spectra (Dir1 and Dir2)	
Design Static Base Shear, V (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (Eurocode value $F_B = S_d (T_l) \times m \times \lambda$).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (Eurocode value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination),((Eurocode - No vertical loads considered - factor = 0.0).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation, (Eurocode value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (Eurocode - No base shear adjustment).
Scaling of Deflections for Drift Checks (%	This field allows you to set your required scaling factors, (Eurocode - No adjustment).

Property	Description
of Base Shear) (Dir1 and Dir2)	**NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 478) and open the **EC8 Seismic Combination Generator** described below.

EC8 Seismic Combination Generator

The Combination Generator uses the two “core” combinations from EC8 that include seismic loads as below:

Equation 6.12a

$G_{kj} + A_{Ed} + \psi_{2,i} \times Q_{k,i}$

Equation 6.12b

$G_{kj} + A_{Ed}$

Provided the generate option is selected these will then be used, with factors as specified, to generate an appropriate number of combinations to cover for direction +/- Dir1 and Dir2 with +/- eccentricities.

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this checkbox to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.

Property	Description
Next	Click **Next** to specify the service combinations.
Finish	Click **Finish** to generate the Seismic combinations.

Using the IS1893 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure in accordance with the Indian IS1893 code.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

1. Click Home > Model Settings > Design Codes
   a. Set the Design Code for Seismic Loading to IS1893 (Part 1) and select the year required
   b. Click OK
2. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the IS893 (Part 1) Horizontal Design Spectrum.
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 465).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The SZone Factor is determined on this page.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of storeys	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Zone & Site	
Seismic Zone	This field allows you to set the appropriate zone, II-V.
Site class	This field allows you to set the appropriate soil conditions, I-III.
Importance Factor, I	This is automatically derived from the occupancy class.
Next	Clicking **Next** takes you to the **Structure Irregularities page** described below.

Structure Irregularities page

This page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure

Property	Description
	• Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

This page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived. If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select the structural type.
Approx fundamental period, T_a	This is automatically derived.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

This page is used to determine the response modification coefficient, R, and other factors in the directions 1 and 2.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	
Type	This field allows you to set the appropriate type.
System	This field allows you to set the appropriate system for the chosen type.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those loadcases that you want to contribute to the Structure Seismic Dead Weight.

NOTE The effective seismic weight combination is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used
- define appropriate localization parameters

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift. Click 'override' if you want to enter your own value.
Include localization - Check this box in order to display the below 'ELF & RSA' and 'RSA only' localization settings.	

Property	Description
ELF & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor.
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear.
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combinatio n factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination).
RSA only	(only available if RSA has been selected)
% Mass participatio n for RSA	This field allows you to set your own required % mass participation. **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors

Property	Description
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors. **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 478) and open the **Seismic Combination Generator** described below.

Seismic Combination Generator

The Combination Generator sets up seismic combinations from the "core" combinations that include seismic loads.

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this checkbox to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Code spectra and site specific spectra

Code Spectra

Different analysis procedures are provided for determining the elastic design response spectrum, the first of which is simply to adopt the code spectra appropriate to the Seismic Loading action code selected for the model.

- Default spectra (page 466)

Site Specific Spectra (user defined)

In addition to the code spectra, Tekla Structural Designer also allows you to define your own site specific spectra by specifying significant periods and acceleration. This procedure is applicable to both ELF and RSA analysis.

The curve equations used for the different Seismic Loading action codes are as follows:

- ASCE 7 Horizontal Design Spectrum (page 468)
- EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA) (page 470)
- EN 1998-1 Horizontal Design Spectrum (Malaysia NA) (page 472)
- IS893 (Part 1) Horizontal Design Spectrum (page 474)
- ASCE 7/UBC Horizontal Design Spectrum - Taiwan (page 476)

Site Specific Spectra (user defined - generic curve)

A third route is to set a site specific generic curve, either manually, or by pasting data from a spreadsheet. This procedure requires an RSA to be run (not ELF). This has been added primarily for remote markets using the following US Seismic Loading action codes:

- ASCE 7-05
- ASCE 7-10
- ASCE 7-16

It is not available for UBC, EC or IS Seismic Loading action codes as these codes do not define the process to be used with a generic spectrum.

User defined RSA curve

Default spectra
The following details the initial values of the parameters for spectra for each of the seismic regional codes:

ASCE - Horizontal Design Spectrum

Limits	Equation	Segment	Ref
$0 \le T \le T_0$	S_{DS} x (0.4 + 0.6 x T / T_0) x (1/(R/I_e)	Start point/ linear	11.4.5 (1)
$T_0 \le T \le T_s$	S_{DS} x (0.4 + 0.6 x T / T0) x (1/(R/Ie)	Constant	11.4.5 (2)
$T_s \le T \le T_L$	(S_{D1}/T)/(R/I)	Constant/T	11.4.5 (3)
$T_L \le T \le 4s$	(S_{D1} x T_L / T^2)/(R/I)	Constant/T^2	11.4.5 (4)

UBC - Horizontal Design Spectrum

Limits	Equation	Segment
$0 \le T \le T_0$	C_a + 1.5 x T / T_0 x (2.5 / R-1)	Start point/linear
$T_0 \le T \le T_s$	2.5 x C_a /R	Constant
$T_s \le T$	C_v / T/R	Constant/T

EC - Horizontal Design Spectrum (UK, Singapore and Europe)

Limits	Equation	Segment	Ref
$0 \le T \le T_B$	a_g x S x [2/3 + T/T_B x (2.5 / q - 2/3)]	Start point/ linear	3.13
$T_B \le T \le T_C$	a_g x S x 2.5 / q	Constant	3.14
$T_C \le T \le T_D$	a_g x S x T_C/T x 2.5 / q	Constant/T	3.15
$T_D \le T \le 4s$	a_g x S x 2.5 / q x (T_C x T_D) / T^2	Constant/T^2	3.16

EC - Horizontal Design Spectrum (Malaysia)

For **Rock soil** sites

Limits	Equation	Segment
$0 \le T \le T_C$	((2 x π)^2 x γI x S_{DR}(1.25) / (T_C x T_D)) / q	Constant
$T_C \le T \le T_D$	((2 x π)^2 x γI x S_{DR}(1.25) / (T x TD)) / q	Constant/T
$T_D \le T \le 4s$	((2 x π)^2 / T2 x [λI x S_{DR}(1.25) + γI x m_r x (T - T_D))] / q	Constant/T^2

For **Stiff soil** sites

Limits	Equation	Segment
$0 \le T \le T_C$	((2 x π)^2 x γI x S_{DR}(1.25) x 1.5 / (T_C x T_D)) / q	Constant
$T_C \le T \le T_D$	((2 x π)^2 x γI x S_{DR}(1.25) x 1.5 / (T x TD)) / q	Constant/T

Limits	Equation	Segment
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda I \times S_{DR}(1.25) \times 1.5 + \gamma I \times m_r \times (T - T_D))] / q$	Constant/T^2

For **Flexible soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	$((2 \times \pi)^2 \times \gamma I \times S_{DR} (1.5T_S) \times 3.6 / (T_C \times T_D)) / q$	Constant
$T_C \leq T \leq T_D$	$((2 \times \pi)^2 \times \gamma I \times S_{DR}(1.5T_S) \times 3.6 / (T \times TD)) / q$	Constant/T
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda I \times S_{DR}(1.5T_S) \times 3.6 + \gamma I \times m_r \times (T - T_D))] / q$	Constant/T^2

IS - Horizontal Design Spectrum

For **rocky, or hard soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.40$	(2.50)	Constant/T
$0.40 \leq T \leq 4.00$	(1.00/T)	Constant/T^2

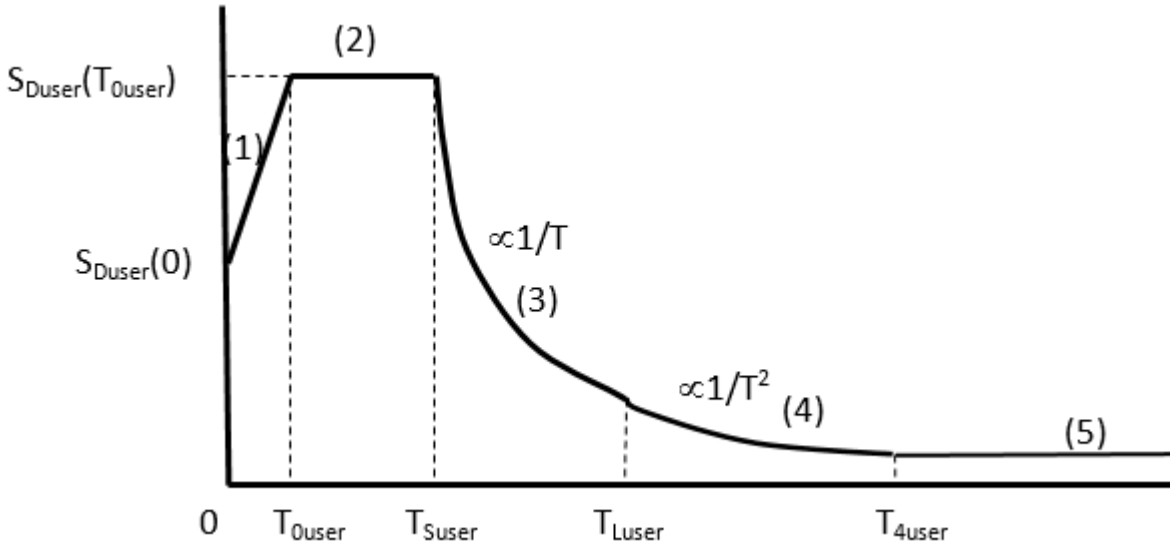
For **medium soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.55$	(2.50)	Constant/T
$0.55 \leq T \leq 4.00$	(1.36/T)	Constant/T^2

For **soft soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.67$	(2.50)	Constant/T
$0.67 \leq T \leq 4.00$	(1.67/T)	Constant/T^2

ASCE 7 Horizontal Design Spectrum



NOTE ASCE 7 has either sloped second segment or horizontal 4th segment - not both.

Parameters

- $S_{Duser}(0)$ – units g
- $S_{Duser}(T_{0user})$ – units g
- T_{0user} – units sec
- T_{Suser} – units sec
- T_{Luser} – units sec
- T_{4user} – units sec, default = 4s

Input limits

- $S_{Duser}(0) > 0$
- $S_{Duser}(T_{0user}) > 0$ if $T_{0user} > 0$
- $0s \leq T_{0user} < T_{Suser} < T_{Luser} \leq T_{4user}$

Curve Equations

Design Response Spectrum curves for $S_a(g)$ are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{0user}$	$S_a(g)(T) = S_{Duser}(0) + ((S_{Duser}(T_{0user}) - S_{Duser}(0)) \times T / T_{0user})$
(2) – straight line	$T_{0user} \le T \le T_{Suser}$	$S_a(g)(T) = S_{Duser}(T_{0user})$
(3) – curve	$T_{Suser} \le T \le T_{Luser}$	$S_a(g)(T) = S_{Duser}(T_{0user}) \times T_{Suser} / T$
(4) – curve	$T_{Luser} \le T \le T_{4user}$	$S_a(g)(T) = S_{Duser}(T_{0user}) \times T_{Suser} \times T_{Luser} / T^2$
(5) – straight line continued from (4)		

Adjusted Design Response Spectrum curves for $S_a(g)/(R/I_e)$ are defined by

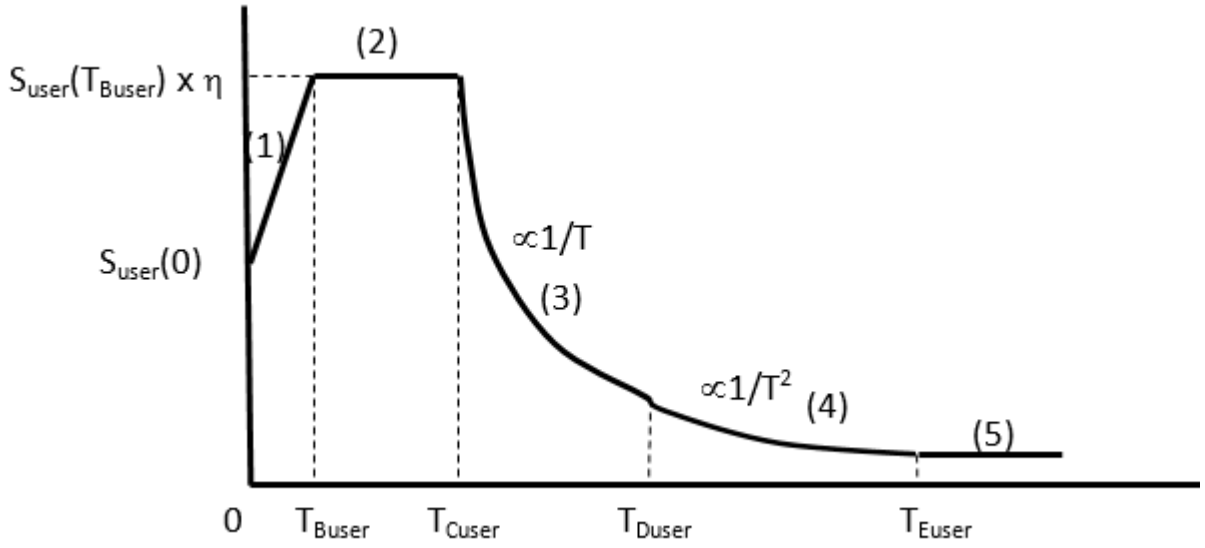
NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{0user}$	$S_D(T) = [S_{Duser}(0) + ((S_{Duser}(T_{0user}) - S_{Duser}(0)) \times T / T_{0user})] /(R/I_e)$
(2) – straight line	$T_{0user} \le T \le T_{Suser}$	$S_D(T) = S_{Duser}(T_{0user}) / (R/I_e)$
(3) – curve	$T_{Suser} \le T \le T_{Luser}$	$S_D(T) = S_{Duser}(T_{0user}) \times T_{Suser} / T / (R/I_e)$
(4) – curve	$T_{Luser} \le T \le T_{4user}$	$S_D(T) = S_{Duser}(T_{0user}) \times T_{Suser} \times T_{Luser} / T^2 / (R/I_e)$
(5) – straight line continued from (4)		

NOTE If $T_{0user} = 0$, then no line (1) exists

If $T_{Luser} = T_{4user}$ then no line (4) exists

EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA)



Parameters

The following input parameters as shown in the above figure are required:

- $S_{user}(0)/a_g$
- $S_{user}(T_{Buser})/a_g$
- T_{Buser}
- T_{Cuser}
- T_{Duser}
- T_{Euser}

See the curve equation tables below for more details.

Input limits

- $S_{user}(0) > 0$
- $S_{user}(T_{Buser}) > 0$
- $0s \leq T_{Buser} < T_{Cuser} < T_{Duser} \leq T_{Euser}$

Elastic Response Spectrum Curve Equations

Elastic Response Spectrum curves for S_e/a_g are defined by:

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_e/a_g(T) = S_{user}(0) + ((S_{user}(T_{Buser}) \times \eta - S_{user}(0)) \times T / T_{Buser})$

Line	Limits	Equation
(2) – straight line	$T_{Buser} \le T \le T_{Cuser}$	$S_e/a_g(T) = S_{user}(T_{Buser}) \times \eta$
(3) – curve	$T_{Cuser} \le T \le T_{Duser}$	$S_e/a_g(T) = S_{user}(T_{Buser}) \times \eta \times T_{Cuser}/ T$
(4) – curve	$T_{Duser} \le T \le T_{Euser}$	$S_e/a_g(T) = S_{user} (T_{Buser}) \times \eta \times T_{Cuser} \times T_{Duser}/ T^2$
(5) – straight line continued from (4)		

Notes:

- The same curve applies for both Dir1 and Dir2
- $S_e/a_g(T)$ is the elastic response spectrum to design ground acceleration ratio
- η is the damping correction factor
- See EC8 Clause 3.2.2.2 for more details

Design Spectrum Curve Equations

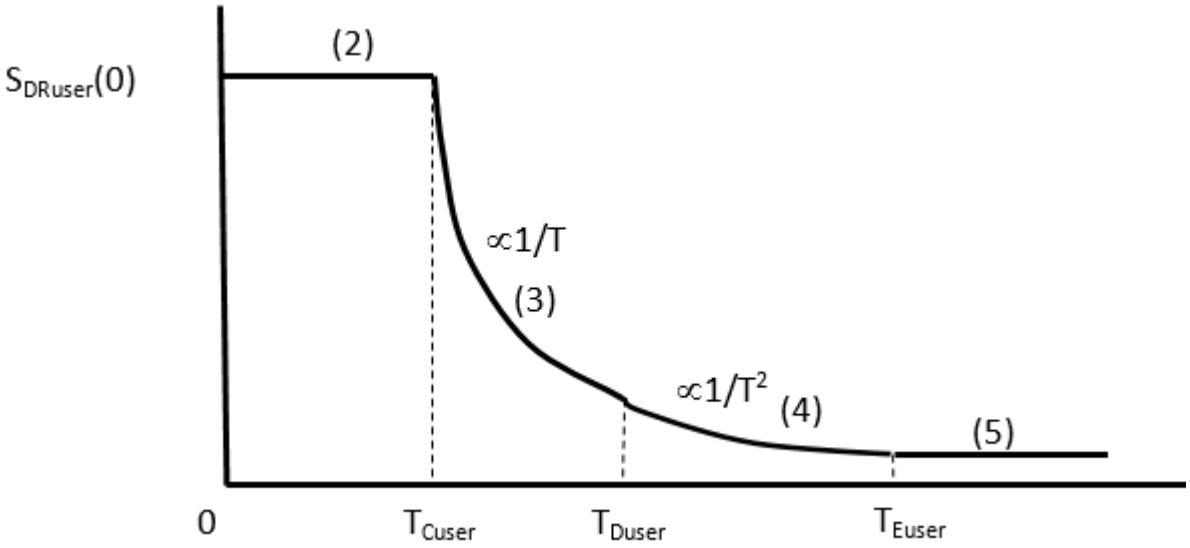
Design Spectrum for Elastic Analysis curves for S_e/a_g are defined by:

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{Buser}$	$S_d/a_g(T) = [2/3 \times S_{user}(0) + ((S_{user}(T_{Buser}) / q - 2/3 \times S_{user}(0)) \times T / T_{Buser})]$
(2) – straight line	$T_{Buser} \le T \le T_{Cuser}$	$S_d/a_g(T) = S_{user}(T_{Buser}) / q$
(3) – curve	$T_{Cuser} \le T \le T_{Duser}$	$S_d/a_g(T) = S_{user}(T_{Buser}) \times T_{Cuser}/ T / q$
(4) – curve	$T_{Duser} \le T \le T_{Euser}$	$S_d/a_g(T) = S_{user} (T_{Buser}) \times T_{Cuser} \times T_{Duser}/ T^2/ q$
(5) – straight line continued from (4)		

Notes:

- The same curve applies for both Dir1 and Dir2
- $S_d/a_g(T)$ is the design spectrum to design ground acceleration ratio
- If $T_{Buser} = 0$, then no line (1) exists
- If $T_{Duser} = T_{Euser}$ then no line (4) exists

EN 1998-1 Horizontal Design Spectrum (Malaysia NA)



Parameters

- $S_{DRuser}(0)$ – units g
- $S_{DRuser}(T_{Buser})$ – units g
- T_{Buser} – units sec
- T_{Cuser} – units sec
- T_{Duser} – units sec
- T_{Euser} – units sec, default = 4s

Input limits

- $S_{DRuser}(0) > 0$
- $S_{DRuser}(T_{Buser}) > 0$
- $0s \leq T_{Buser} < T_{Cuser} < T_{Duser} \leq T_{Euser}$

Curve Equations

Elastic Response Spectrum curves for $S_e/a_g$are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_e/a_g(T) = S_{DRuser}(0) + ((S_{DRuser}(T_{Buser}) - S_{DRuser}(0)) \times T / T_{Buser})$

Line	Limits	Equation
(2) – straight line	$T_{Buser} \le T \le T_{Cuser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser})$
(3) – curve	$T_{Cuser} \le T \le T_{Duser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} / T$
(4) – curve	$T_{Duser} \le T \le T_{Euser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} \times T_{Duser} / T^2$
(5) – straight line continued from (4)		

Design Spectrum for Elastic Analysis curves for S_d/a_g are defined by

NOTE same curve for both Dir1 and Dir2

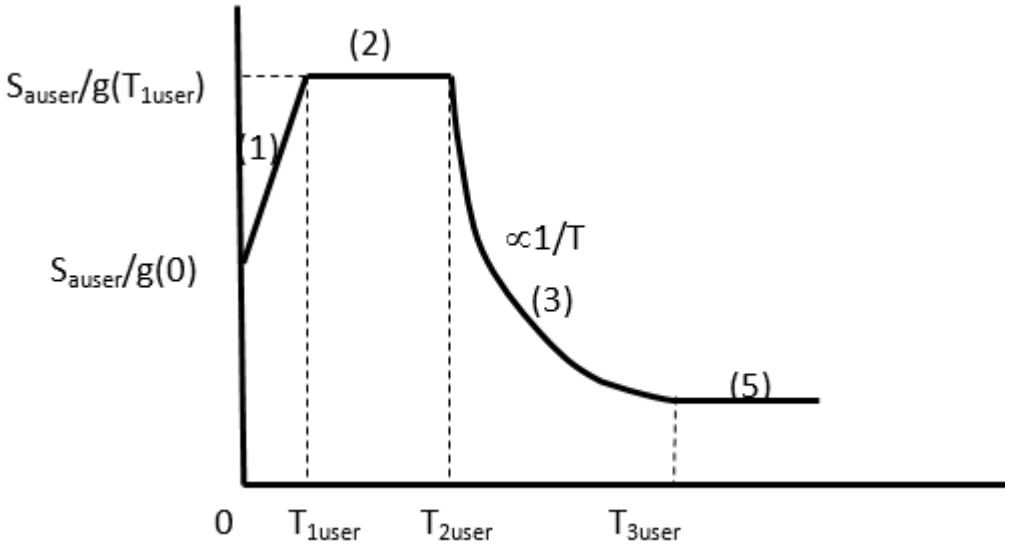
Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{Buser}$	$S_d/a_g(T) = [S_{DRuser}(0) + ((S_{DRuser}(T_{Buser}) - S_{DRuser}(0)) \times T / T_{Buser})] / q$
(2) – straight line	$T_{Buser} \le T \le T_{Cuser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) / q$
(3) – curve	$T_{Cuser} \le T \le T_{Duser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} / T / q$
(4) – curve	$T_{Duser} \le T \le T_{Euser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} \times T_{Duser} / T^2 / q$
(5) – straight line continued from (4)		

NOTE If $T_{Buser} = 0$, then no line (1) exists(as in the example above)

If $T_{Duser} = T_{Euser}$ then no line (4) exists

NOTE The above by default does not quite equate to Line (4) in the Malaysian NA because there is an extra term $(2 \times \pi)^2 / T^2 \times \gamma l \times m_r \times (T - T_D)$ that tweaks the $1/T^2$ curve slightly. In Sarawak, this term is non-existent, in Peninsular Malaysia it does not exist for Flexible soils. In the Tekla Structural Designer User defined spectrum it is also non-existent.

IS893 (Part 1) Horizontal Design Spectrum



NOTE In this example T3user = T4user so there is no line (4)

Parameters

- Damping $factor_{user}$
- $S_{auser}/g(0)$ – units g
- $S_{auser}/g(T_{1user})$ – units g
- T_{1user} – units sec
- T_{2user} – units sec
- T_{3user} – units sec, default = 4s
- T_{4user} – units sec, default = 4s

Input limits

- $S_{auser}/g(0) > 0$
- $S_{auser}/g(T_{1user}) > 0$
- $0s \leq T_{1user} < T_{2user} < T_{3user} \leq T_{4user}$

Curve Equations

Average Response Spectrum curves for S_a/g are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{1user}$	$S_a(g)(T) = \text{Damping factor}_{user} \times [S_{auser}/g(0) + ((S_{auser}/g(T_{1user}) - S_{auser}/g(0)) \times T / T_{1user})]$
(2) – straight line	$T_{1user} \le T \le T_{2user}$	$S_a(g)(T) = \text{Damping factor}_{user} \times S_{auser}/g(T_{1user})$
(3) – curve	$T_{2user} \le T \le T_{3user}$	$S_a(g)(T) = \text{Damping factor}_{user} \times S_{auser}/g(T_{1user}) \times T_{2user}/ T$
(4) – curve	$T_{3user} \le T \le T_{4user}$	$S_a(g)(T) = \text{Damping factor}_{user} \times S_{auser}/g(T_{1user}) \times T_{2user} \times T_{3user}/ T^2$
(5) – straight line continued from (4)		

Design Response Spectrum curves for $A_h(g)$ (=$S_a/g/((2xR)/(ZxI))$) are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{1user}$	$A_h(g)(T) = \text{Damping factor}_{user} \times [S_{auser}/g(0) + ((S_{auser}/g(T_0) - S_{auser}/g(0)) \times T / T_{1user})]$ But not less than Z/2
(2) – straight line	$T_{1user} \le T \le T_{2user}$	$A_h(g)(T) = \text{Damping factor}_{user} \times S_{auser}/g(T_{1user}) / ((2xR)/(ZxI))$
(3) – curve	$T_{2user} \le T \le T_{3user}$	$A_h(g)(T) = \text{Damping factor}_{user} \times S_{auser}/g(T_{1user}) \times T_{2user}/ T / ((2xR)/(ZxI))$
(4) – curve	$T_{3user} \le T \le T_{4user}$	$A_h(g)(T) = \text{Damping factor}_{user} \times S_{auser}/g(T_{1user}) \times T_{2user} \times T_{3user}/ T^2 / ((2xR)/(ZxI))$
(5) – straight line continued from (4)		

NOTE If $T_{1user} = 0$, then no line (1) exists

If $T_{3user} = T_{4user}$ then no line (4) exists

ASCE 7/UBC Horizontal Design Spectrum - Taiwan

Taiwan code Figure 2

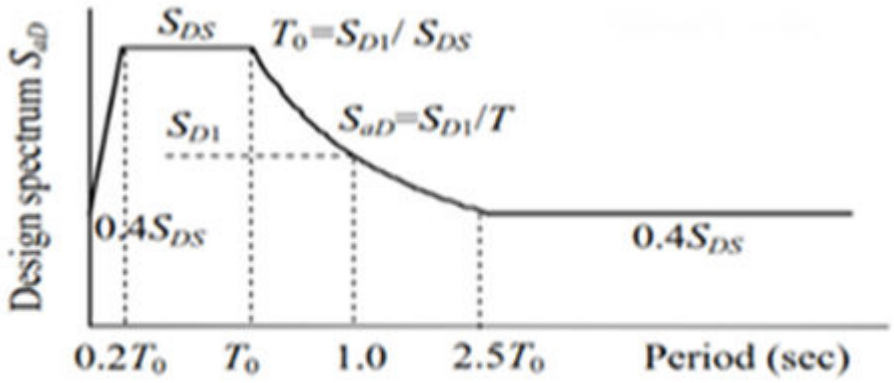


Figure 2. Design response spectrum developed by site-adjusted parameters S_{DS} and S_{D1}

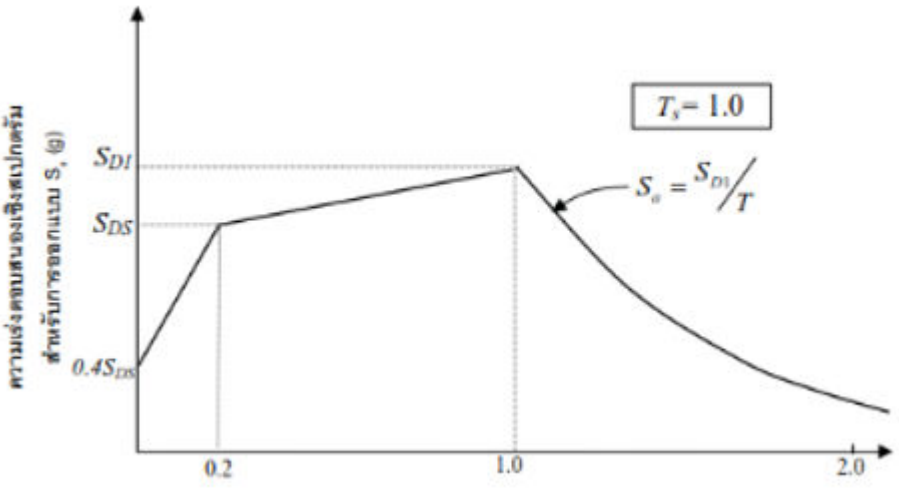
Specifically, the last section of curve (Segment 4) extends horizontally from $2.5T_0$ with a value of 0.4 x S_{DS}

NOTE The above curve can be obtained by selecting Horizontal 4th Segment on the Site Specific Spectra page of the Seismic Wizard.

ASCE 7/UBC Horizontal Design Spectrum - Thailand

When $S_{d1} > S_{ds}$

Thailand code Figure 1-4-3



Specifically, the previously flat section of curve (Segment 2) rises from S_{DS} at T = 0.2 to S_{D1} At T = 1.0

NOTE The above curve can be obtained by selecting Sloped 2nd Segment on the Site Specific Spectra page of the Seismic Wizard.

Seismic loadcases

On completion of the Seismic Wizard process, seismic loadcases are automatically created and applied in the two building directions, Dir1 and Dir2. Seismic torsion loadcases are also created.

Equivalent Lateral Force (ELF) Loadcases

For structures not requiring application of accidental seismic torsion (eg ASCE7 SDC A)

The following lateral seismic loadcases are created:

- Seismic Dir1 - loadcase type: Seismic
- Seismic Dir2 - loadcase type: Seismic

For all other structures:

The following lateral seismic loadcases are created:

- Seismic Dir1 - loadcase type: Seismic
- Seismic Dir2 - loadcase type: Seismic

The following seismic torsion loadcase is also created:

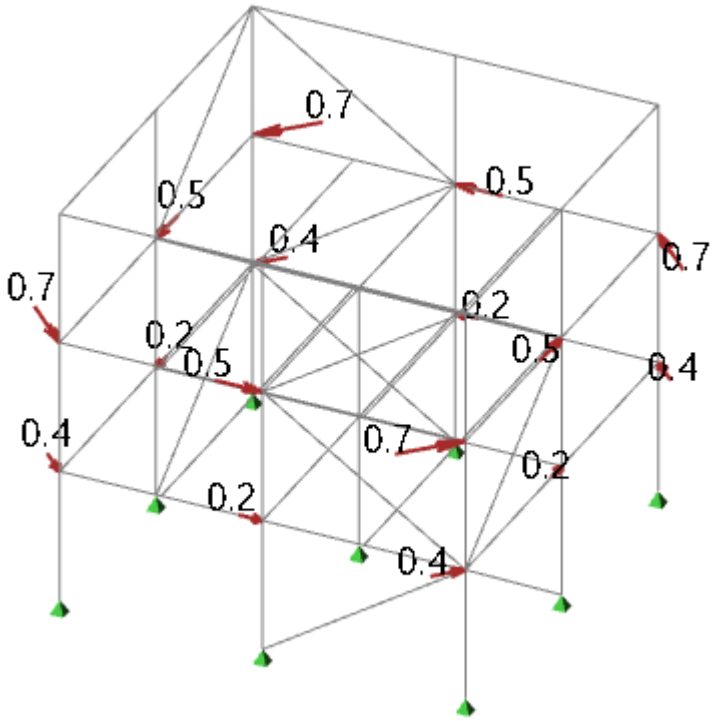
- Seismic Torsion - loadcase type: Seismic

NOTE + and - factors are applied in the combinations to achieve the possible permutations - +Dir, -Dir, +torsion and -torsion.

Viewing the Seismic Forces

After analysis, the magnitudes and distributions of the seismic forces can be viewed directly in the Results View via the “Seismic” button on the Results ribbon after selecting the required a seismic loadcase.

Note that, for seismic torsion loadcases created by the ELF method, a system of lateral, radial “push-pull” forces are applied to the structure to produce the overall torsion.



The Seismic Design report includes a table giving the total level loads $\mathbf{F_i}$ and eccentricity values **Ecc** used to calculate these forces.

Response Spectrum Analysis (RSA) Loadcases

The following lateral seismic loadcases are created:

- Seismic Dir1 - loadcase type: Seismic RSA
- Seismic Dir2 - loadcase type: Seismic RSA

The following seismic seismic torsion loadcase is also created:

- Seismic Torsion - loadcase type: Seismic

NOTE In Tekla Structural Designer the seismic torsion loading and analysis is static, even for the RSA method; only the lateral RSA analysis is 'dynamic' i.e. a modal analysis + RSA. This is a common method of seismic analysis i.e. dynamic lateral analysis + static torsion.

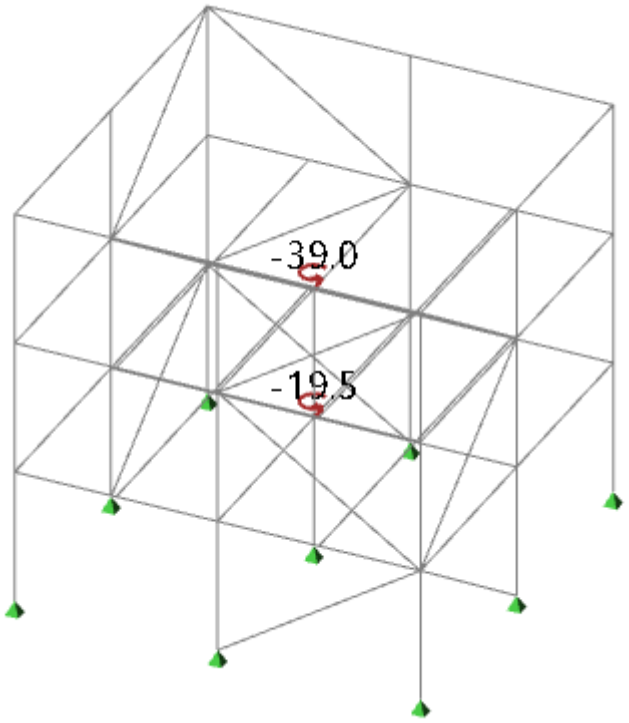
NOTE Enveloping is used in the combinations to achieve the possible permutations - +Dir, -Dir, +torsion and -torsion.

Viewing the Seismic Forces

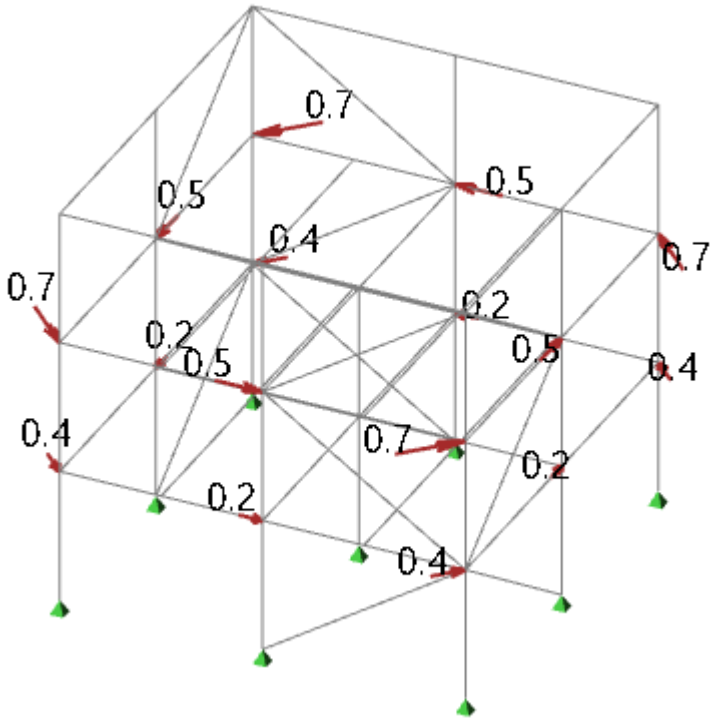
After analysis, the magnitudes and distributions of the seismic forces can be viewed directly in the Results View via the “Seismic” button on the Results ribbon after selecting the required a seismic loadcase.

Note that, for seismic torsion loadcases created by the RSA method, the application of these forces depends on the diaphragm options that have been applied.

- For a rigid diaphragm single point torsion loads are applied.



- For semi-rigid diaphragms or for levels with no diaphragms, radial “push-pull” forces are applied to the structure to produce the overall torsion.



The Seismic Design report includes a table giving the total level loads $\mathbf{F_i}$ and eccentricity values **Ecc** used to calculate these forces.

4 Analyze models

With Tekla Structural Designer you can perform an extensive range of different analyses.

Click the links below to find out more:

- Analysis types (page 482)
- Linear, non-linear and buckling analyses (page 494)
- Modal analysis (page 508)
- RSA seismic analysis (page 515)
- Solver models (page 537)
- Analysis model choices (page 568)
- Analysis results (page 662)
- Analysis settings (page 760)

Related information

See also

Background to the Analysis/Design procedures (eLearning)

4.1 Analysis types

NOTE Before running analysis you should familiarize yourself with the analysis limitations and assumptions (page 516).

The following analysis types can be run in Tekla Structural Designer:

1st order linear

1st order linear static analysis is suitable for structures where secondary effects are negligible. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads • Preparing loadcases and combinations • Meshing and diaphragms • Applying loads • Resolving vertical loads for application of global imperfections The actual pre-analysis processes performed will vary depending on the specific model that has been defined.
3D analysis	A traditional frame first-order linear analysis of the entire 3D model. Nonlinearity Included: • Geometric: No • Material: No

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Related task: Run 1st order linear analysis (page 495)

1st order non-linear

This is a nonlinear analysis with loading applied in a single step.

It is suitable for structures where secondary effects are negligible and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D analysis: First-order linear	A 1st order linear analysis (see above), is automatically performed prior to the non-linear analysis.
3D analysis: First-order non-linear	A traditional frame first-order non-linear analysis of the entire 3D model, with an option to mesh floors. Nonlinearity Included: • Geometric: No • Material: Yes

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Related task: Run a 1st order non-linear analysis (page 495)

1st order modal

This is an unstressed modal analysis which can be used to determine the structure's natural frequencies.

The structure is assumed to be in an unstressed state and nonlinear elements are constrained to act linearly.

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads • Preparing loadcases and combinations • Meshing and diaphragms • Applying loads The actual pre-analysis processes performed will vary depending on the specific model that has been defined.

Process	Description
3D analysis First-order Modal	A first-order modal analysis of the entire 3D model, with an option to mesh floors. Nonlinearity Included: • Geometric: No • Material: No

Related concept: Modal analysis (page 508)

Related task: Run a 1st order modal analysis (page 514)

2nd order linear

This is a 2-stage P-Delta analysis which is suitable for structures where secondary effects are of comparable magnitude to primary effects. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D analysis: First-order linear	A 1st order linear analysis (see above), is automatically performed prior to the 2nd order linear analysis.
3D analysis: Second-order linear	A traditional frame 2nd order linear analysis of the entire 3D model, with an option to mesh floors. Nonlinearity Included: • Geometric: Yes • Material: No

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Related task: Run a 2nd order linear analysis (page 495)

2nd order non-linear

This is a nonlinear analysis with loading applied in a single step.

It is suitable for structures where secondary effects are of comparable magnitude to primary effects and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D analysis: First-order linear	A 1st order linear analysis (see above), is automatically performed prior to the 2nd order non-linear analysis.
3D analysis: Second-order non-linear	A traditional frame 2nd order non-linear analysis of the entire 3D model, with an option to mesh floors. Nonlinearity Included: • Geometric: Yes • Material: Yes

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Related task: Run a 2nd order non-linear analysis (page 496)

2nd order buckling

This is a linear buckling analysis which can be used to determine a structure's susceptibility to buckling.

The stressed state of the structure is determined from linear analysis; therefore nonlinear elements are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads

Process	Description
	• Preparing loadcases and combinations • Meshing and diaphragms • Applying loads The actual pre-analysis processes performed will vary depending on the specific model that has been defined.
3D analysis Second-order buckling	A second-order buckling analysis of the entire 3D model. Nonlinearity Included: • Geometric: Yes • Material: No

Related task: Run a 2nd order buckling analysis (page 496)

FE chasedown

This analysis type cannot be run in isolation, it is only performed when it is required as part of one of the following batch analysis/design processes:

- **Analyze All (Static)**
- **Design Steel (Gravity)**
- **Design Steel (Static)**
- **Design Concrete (Gravity)**
- **Design Concrete (Static)**
- **Design All (Gravity)**
- **Design All (Static)**

It is only required in the above processes if the model contains meshed slabs, (these can be either two-way spanning slabs, or one-way spanning slabs with the "Analyze slab" property selected).

Process	Description
FE chasedown	Separate analyses are performed for a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. See: Solver Model used for FE Chasedown. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always considered for timber member design and slab design. They can also optionally be used for: steel

Process	Description
	member design; concrete beam, concrete column and concrete wall design. **NOTE** You can use (page 866) to turn off 'Design for FE Chasedown analysis results' for steel members and concrete members/walls if required. For steel this option is located under Design Settings > Steel > General (page 875). For concrete this option is located under Concrete > Cast-in-place (page 878), then **Beam/ Column/Wall > General Parameters** (as appropriate).

A significant consideration when opting to design for the FE chasedown results is that the slabs will tend to carry a significant proportion of the load direct to the columns.

Consequently, for beam design in particular, it is unlikely that an FE chasedown could result in a more critical set of design forces than would be already catered for by a Grillage chasedown.

NOTE If duplicate levels have been specified in the Construction Levels dialog separate sub models are created and analyzed for the source and every duplicate level. This ensures that the increasing load carried by the vertical members in the lower sub models is catered for. In turn this can cause small differences in the analysis results (and consequently the design) for these sub models.

Accounting for lateral loading in chasedown results

It is important to note that the chasedown analysis procedure is only valid for gravity loads. The chasedown analysis results for any lateral loadcase (wind / EHF) or from the direct analysis of any combination that includes a lateral loadcase are not valid.

Therefore in order to generate a set of forces suitable for design, the chasedown analysis results are merged with the 3D building analysis combination results as follows:

1. Start with the building analysis combination result
2. Identify all gravity cases used in the combination and the relevant load factor
3. For each included gravity loadcase:
   a. Subtract the 1st order linear building analysis result multiplied by the relevant load factor
   b. Add the chasedown result multiplied by the relevant load factor
4. For results with live/imposed load reduction, subtract the relevant % of the chasedown result for each reducible loadcase.

Following this procedure means that chasedown analysis of lateral loadcases or combinations is not required.

NOTE This procedure is only applied to beam, column, and wall-line results, but not to 2D nodal results. For this reason it is not possible to calculate or display 2D element chasedown results for combinations that include lateral loadcases.

Related eLearning: Background to the Analysis/Design procedures

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Grillage chasedown

We know from experience that a 3D Analysis on its own does not give the gravity results engineers have traditionally used or want - staged construction analysis reduces but doesn't resolve this. Therefore, in Tekla Structural Designer a grillage chasedown is used to emulate a traditional analysis and establish an alternative second set of design forces for concrete beams, columns and walls.

Grillage chasedown analysis cannot be run in isolation, it is only performed when it is required as part of the following processes:

- **Analyze All (Static)**
- **Design Steel (Gravity)**
- **Design Steel (Static)**
- **Design Concrete (Gravity)**
- **Design Concrete (Static)**
- **Design All (Gravity)**
- **Design All (Static)**

In each of the above processes grillage chasedown is only performed if the model contains concrete beams, columns, or walls, or if it has been requested for steel member design.

Process	Description
Grillage chasedown	Separate analyses are performed for a series of 3D sub models each containing the members between two horizontal planes,

Process	Description
	floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. See: Solver Model used for Grillage Chasedown **NOTE** You can use the options located under Design Settings > Steel > General (page 875) to turn off 'Grillage Chasedown' for steel members.

Accounting for lateral loading in chasedown results

It is important to note that the chasedown analysis procedure is only valid for gravity loads. The chasedown analysis results for any lateral loadcase (wind / EHF) or from the direct analysis of any combination that includes a lateral loadcase are not valid.

Therefore in order to generate a set of forces suitable for design, the chasedown analysis results are merged with the 3D building analysis combination results as follows:

1. Start with the building analysis combination result
2. Identify all gravity cases used in the combination and the relevant load factor
3. For each included gravity loadcase:
   a. Subtract the 1st order linear building analysis result multiplied by the relevant load factor
   b. Add the chasedown result multiplied by the relevant load factor
4. For results with live/imposed load reduction, subtract the relevant % of the chasedown result for each reducible loadcase.

Following this procedure means that chasedown analysis of lateral loadcases or combinations is not required.

NOTE This procedure is only applied to beam, column, and wall-line results, but not to 2D nodal results. For this reason it is not possible to calculate or display 2D element chasedown results for combinations that include lateral loadcases.

Related eLearning: Background to the Analysis/Design procedures

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Analyze All (Static)

This is batch process that includes the full series of analyses required to enable a static design to be performed.

- 3D analysis
  - 1st order linear/non-linear
  - 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)
- FE chasedown (if required)
- Grillage chasedown (if required)

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads • Preparing loadcases and combinations • Meshing and diaphragms • Applying loads • Resolving vertical loads for application of global imperfections The actual pre-analysis processes performed will vary depending on the specific model that has been defined.
3D analysis	A traditional frame analysis of the entire 3D model, with an option to mesh floors. This analysis generates a first set of results that can be used for the design of beams, columns, walls and slabs. You can control whether a first, or a second order 3D Analysis is run by making the appropriate selection on the (page 866) page in (page 866). The actual options that are presented will vary depending on the design code being worked to. The choice of linear or non-linear analysis is made automatically: • if the model has entirely linear properties a linear analysis is performed, • else if any non-linear properties are detected a non-linear analysis is performed.

Process	Description
FE chasedown	An FE chasedown analysis is only performed if required, (see above FE Chasedown topic) .
Grillage chasedown	A Grillage chasedown analysis is only performed if required, (see above Grillage chasedown topic).
Stability checks	Sway (page 828)/Drift (page 823) checks and Wind Drift (page 836) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Related eLearning: Background to the Analysis/Design procedures

Related concept: Features of 3D analysis and chasedown analysis (page 506)

Related concept: Why chasedown analyses are included in Analyze All (Static) (page 499)

Related task: Run Analyze All (Static) (page 497)

3D only (Static)

This is a batch process similar to **Analyze All (Static)** but which excludes chasedowns to save time during scheme design, (for example while addressing overall stability, sway, drift, wind drift, etc.)

The processes performed when this analysis type is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis process begins.
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads • Preparing loadcases and combinations • Meshing and diaphragms • Applying loads • Resolving vertical loads for application of global imperfections The actual pre-analysis processes performed will vary depending on the specific model that has been defined.
3D analysis	A traditional frame analysis of the entire 3D model, with an option to mesh floors. This analysis generates a first set of

Process	Description
	results that can be used for the design of beams, columns, walls and slabs. You can control whether a first, or a second order 3D Analysis is run by making the appropriate selection on the (page 866) page in (page 866). The actual options that are presented will vary depending on the design code being worked to. The choice of linear or non-linear analysis is made automatically: • if the model has entirely linear properties a linear analysis is performed, • else if any non-linear properties are detected a non-linear analysis is performed.
Stability checks	Sway (page 828)/Drift (page 823) checks and Wind Drift (page 836) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Related eLearning: Background to the Analysis/Design procedures

Related task: Run 3D only (Static) (page 498)

1st order RSA seismic

This is a Modal Response Spectrum Analysis used to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis

Related task: Run a 1st order RSA seismic analysis (page 515)

2nd order RSA seismic

This is a Modal Response Spectrum Analysis used to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis

Related task: Run a 2nd order RSA seismic analysis (page 515)

Analyze All (RSA)

This is the same Modal Response Spectrum Analysis that would be carried out as part of Design All (RSA), but with no design. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Related task: Run Analyze All (RSA) (page 515)

Rigorous slab deflection analysis

Tekla Structural Designer's rigorous slab deflection analysis provides a rigorous approach to slab deflection estimation. It comprises (i) Sequential Loading Analysis, (ii) iterative cracked section analysis and (iii) simple modelling of creep and shrinkage effects.

See:

Related information

See also

Background to the Analysis/Design procedures (eLearning)

Linear, non-linear and buckling analyses

These analysis types are all run from the **Analyze** ribbon.

Click the links below to find out more:

- Run a 1st order linear or non-linear analysis (page 494)
- Run a 2nd order linear or non-linear analysis (page 495)
- Run a 2nd order buckling analysis (page 496)
- Run Analyze All (Static) (page 497)
- Run 3D only (Static) (page 498)
- Run FE chasedown or grillage chasedown analysis (page 499)
- Why chasedown analyses are included in Analyze All (Static) (page 499)
- Features of 3D analysis and chasedown analysis (page 506)

See also

Apply and manage analysis settings (page 1263)

Analysis results (page 662)

Analysis limitations and assumptions (page 516)

Run a 1st order linear or non-linear analysis

To run either a linear or non-linear 1st order analysis on your model, see the following instructions.

Run 1st order linear analysis

1. On the **Analyze** tab, click **1st Order Linear**.

   The **Select loading** dialog box opens.

2. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.

3. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

Run a 1st order non-linear analysis

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **1st Order Non-Linear** page.

3. If necessary, adjust the convergence criteria and relaxation factors.

4. Click **OK**.

5. On the **Analyze** tab, click **1st Order Non-linear**.

   The **Select loading** dialog box opens.

6. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.

7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

See also

Apply and manage analysis settings (page 1263)

Analysis results (page 662)

Run a 2nd order linear or non-linear analysis

In order to run either a linear or a non-linear 2nd order analysis on your model, see the following instructions.

Run a 2nd order linear analysis

1. On the **Analyze** tab, click **2nd Order Linear**.

   The **Select loading** dialog box opens.

2. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.

3. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

Run a 2nd order non-linear analysis

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **2nd Order Non-linear** page.

3. If necessary, adjust the convergence criteria and relaxation factors.

4. Click **OK**.

5. On the **Analyze** tab, click **2nd Order Non-linear**.

   The **Select loading** dialog box opens.

6. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.

7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

See also

Apply and manage analysis settings (page 1263)

Analysis results (page 662)

Run a 2nd order buckling analysis

In order to run a buckling analysis on your model, see the following instructions.

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **2nd Order Buckling** page.

3. If necessary, adjust the buckling options.

4. Click **OK**.
5. On the **Analyze** tab, click **2nd Order Buckling**.

   The **Select loading** dialog box opens.
6. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.
7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

See also

Apply and manage analysis settings (page 1263)

Buckling factors (view as tabular data) (page 723)

Display analysis results in a Results View (page 662)

Run Analyze All (Static)

Static designs can only be performed provided suitable analysis results exist - while these results are created when you run a combined analysis and member design, in some situations you may prefer to run the analysis separately and then selectively design parts of the model as required.

The **Analyze All (Static)** process facilitates this as it performs all the analyses for static loadcases and active static load combinations that are required to enable the selective designs to be carried out.

As indicated in the table below, the analyses included in the process vary according to the model content and the options specified on the (page 866) page of **Design Settings**.

Analysis type	Included in Analyze All (Static)?
1st order 3D analysis (linear or non-linear)*	Always included
2nd order analysis (linear or non-linear)*	Included if second-order analysis has been specified on the (page 866) page of **Design Settings**
Grillage chasedown	Included if one or more concrete members exist
FE chasedown	Included if two-way spanning slabs exist

Analysis type	Included in Analyze All (Static)?
1st order staged construction analysis (linear or non-linear)* or 2nd order staged construction analysis (linear or non-linear)*	According to if 'First-order' or 'Second-order' analysis and 'Include staged construction analysis' have been specified on the (page 866) page of **Design Settings** If 'Include staged construction analysis' have been specified on the (page 866) page of **Design Settings** this will be included (as 1st or 2nd order according to the 'Analysis' setting on the same page.)
* *If the model contains non-linear elements the analysis will be non-linear, otherwise it will be linear*	

To run **Analyze All (Static)**:

1. Open the Process window (page 83). (This step is optional but can provide feedback on the analyses as they are performed.)
2. On the **Analyze** ribbon, click  **Analyze All (Static)**.

   At the end of the analyses if a 2D or 3D view is active it is switched to a **Results View**.

See also

Analysis results (page 662)

Run 3D only (Static)

The **3D only (Static)** analysis process can be used to save time during scheme design, for example while addressing overall stability, sway, drift, wind drift, etc.

As indicated in the table below, the analyses included in the process vary according to the model content and the options specified on the (page 866) page of **Design Settings**.

Analysis type	Included in 3D only (Static)?
1st order 3D analysis (linear or non-linear)*	Always included
2nd order analysis (linear or non-linear)*	Included if second-order analysis has been specified on the (page 866) page of **Design Settings**
Grillage chasedown	Included if one or more concrete members exist
FE chasedown	Included if two-way spanning slabs exist

Analysis type	Included in 3D only (Static)?
1st order staged construction analysis (linear or non-linear)* or 2nd order staged construction analysis (linear or non-linear)*	According to if 'First-order' or 'Second-order' analysis and 'Include staged construction analysis' have been specified on the (page 866) page of **Design Settings** If 'Include staged construction analysis' have been specified on the (page 866) page of **Design Settings** this will be included (as 1st or 2nd order according to the 'Analysis' setting on the same page.)
* *If the model contains non-linear elements the analysis will be non-linear, otherwise it will be linear*	

To run **3D only (Static)**:

1. Open the Process window (page 83). (This step is optional but can provide feedback on the analyses as they are performed.)
2. On the **Analyze** ribbon, click **3D only (Static)**.

   At the end of the analyses if a 2D or 3D view is active it is switched to a **Results View**.

See also

Analysis results (page 662)

Run FE chasedown or grillage chasedown analysis

FE chasedown and grillage chasedown analyses are performed automatically as part of the **Analyze All (Static)** process if they are needed.

- FE chasedown analysis is performed if the model contains two-way slabs, or one-way slabs with the "Analyse slab" property selected.
- Grillage chasedown analysis is performed if the model contains one or more concrete members

See also

Run Analyze All (Static) (page 497)

Analysis results (page 662)

Why chasedown analyses are included in Analyze All (Static)

The **Analyze All (Static)** command, (and the **Design All** commands also), each perform a 3D analysis to determine a set of design forces which can then be

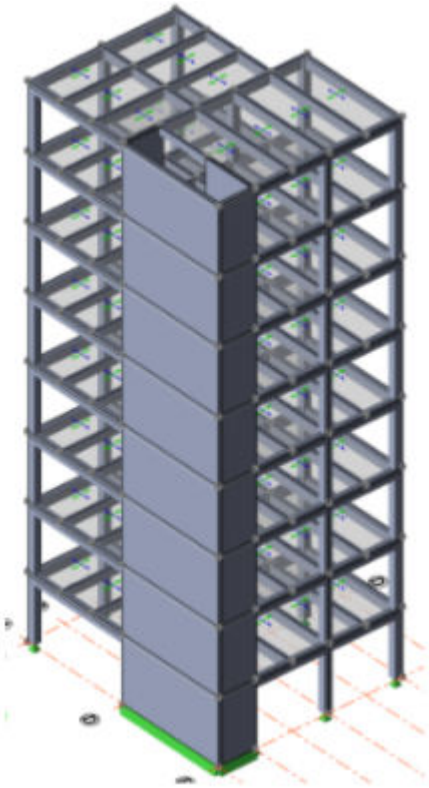
used to design the members. So why does these commands also perform chasedown analyses in many situations?

The answer relates to expectation of results - in particular with regard to the following:

- Sway effects under pure gravity loading
- Transfer beam designs
- Differential axial deformation

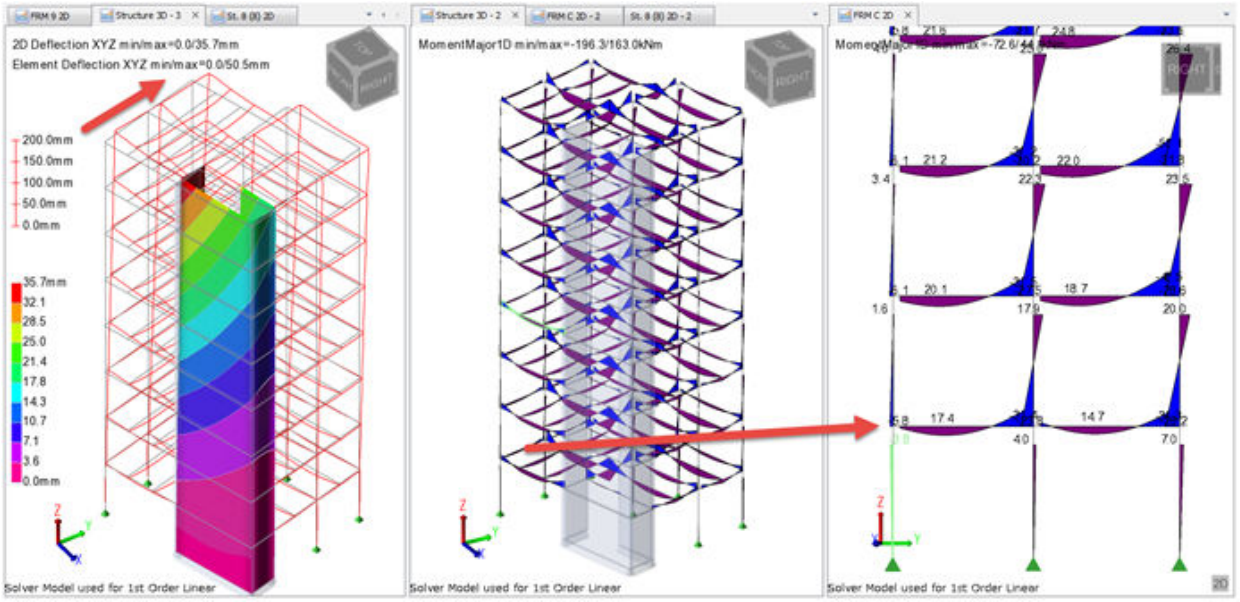
Sway effects under pure gravity loading

Consider this 8 story model, when you click **Analyze All (Static)**, grillage and FE chasedown solver models are created in addition to the 3D analysis model.

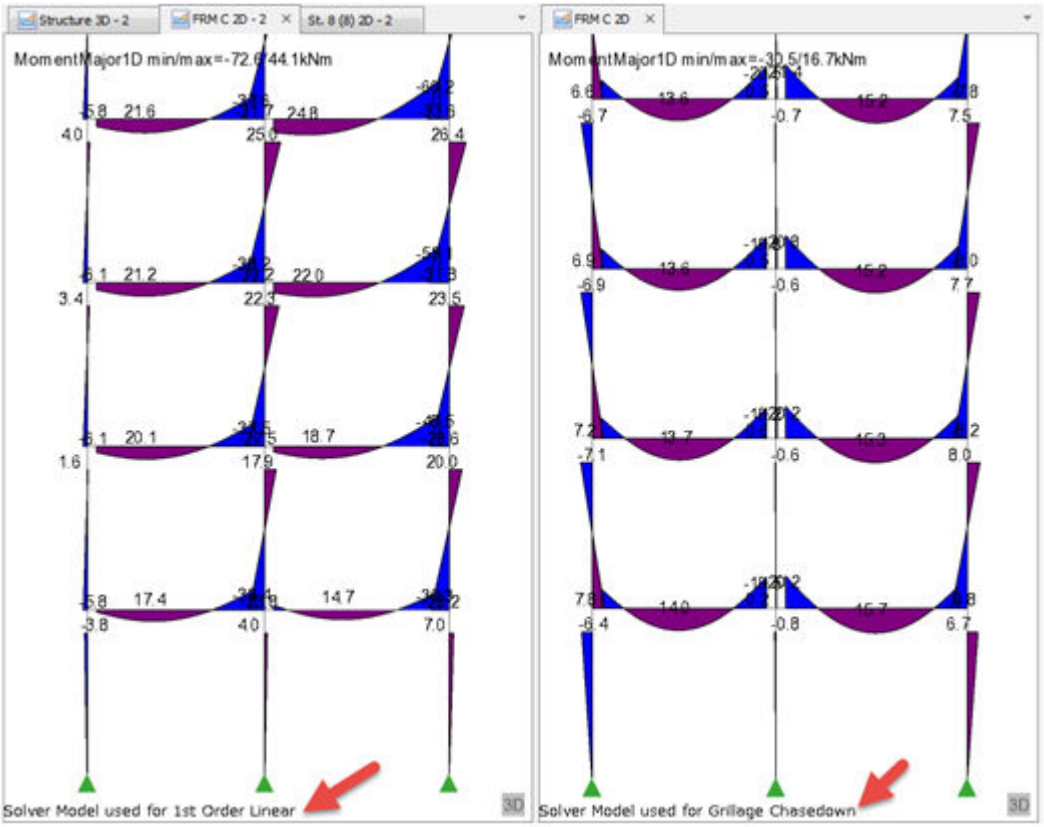


The Identified end frame below is resisting the sway - the design moments shown would be given by any analysis software.

It is an extreme example - but this result does not fit with traditional engineering expectation



The 3D analysis results (below left) can be compared with those from grillage chasedown (below right).



The grillage chasedown results are more in line with expectation:

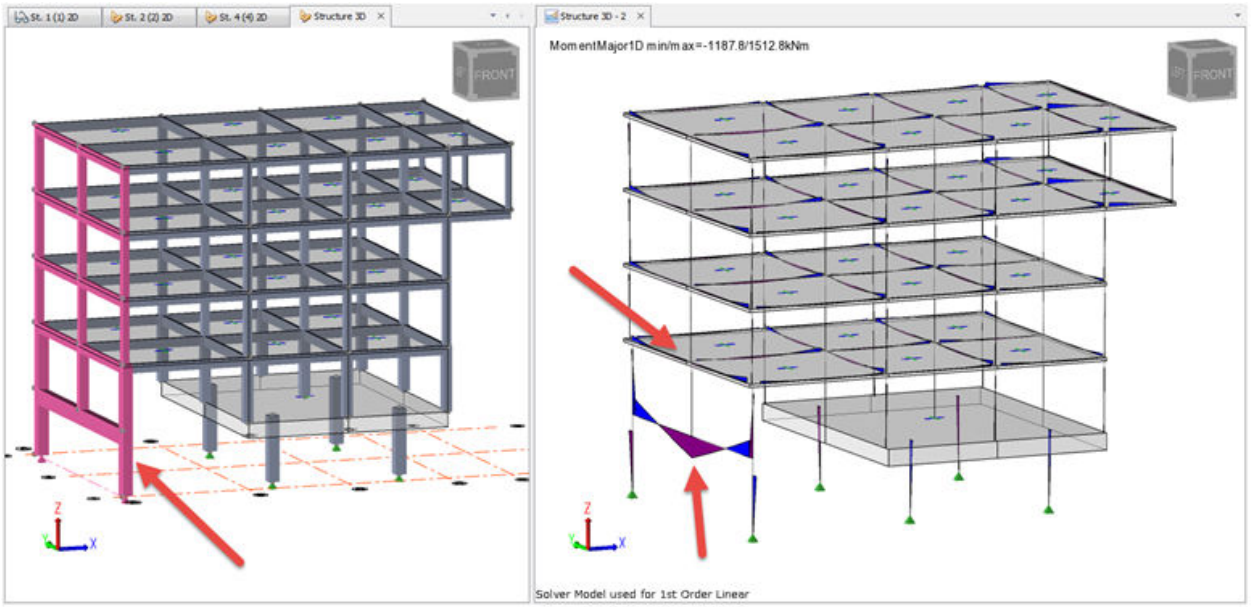
- -ve moment at LH end
- Similar moment profile at every level

Another example of why chasedown is wanted follows below....

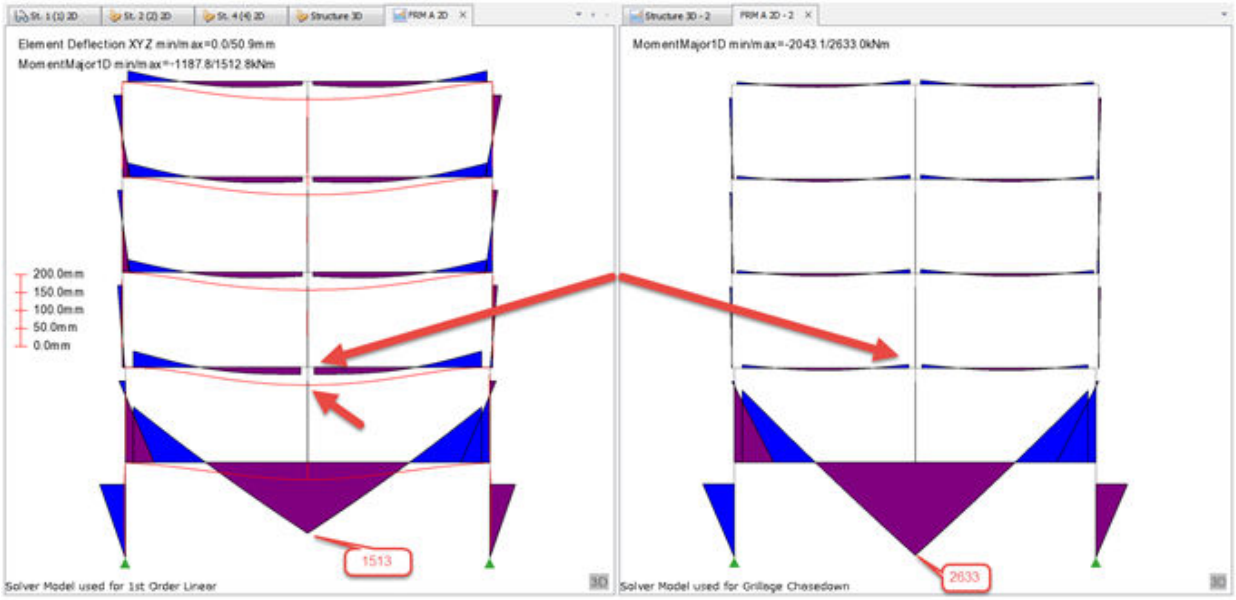
Transfer beam designs

Focus on the transfer beam in the highlighted frame.

At first glance the results look ok, but lets look more closely...



The 3D analysis results (below left) can be compared with those from grillage chasedown (below right).



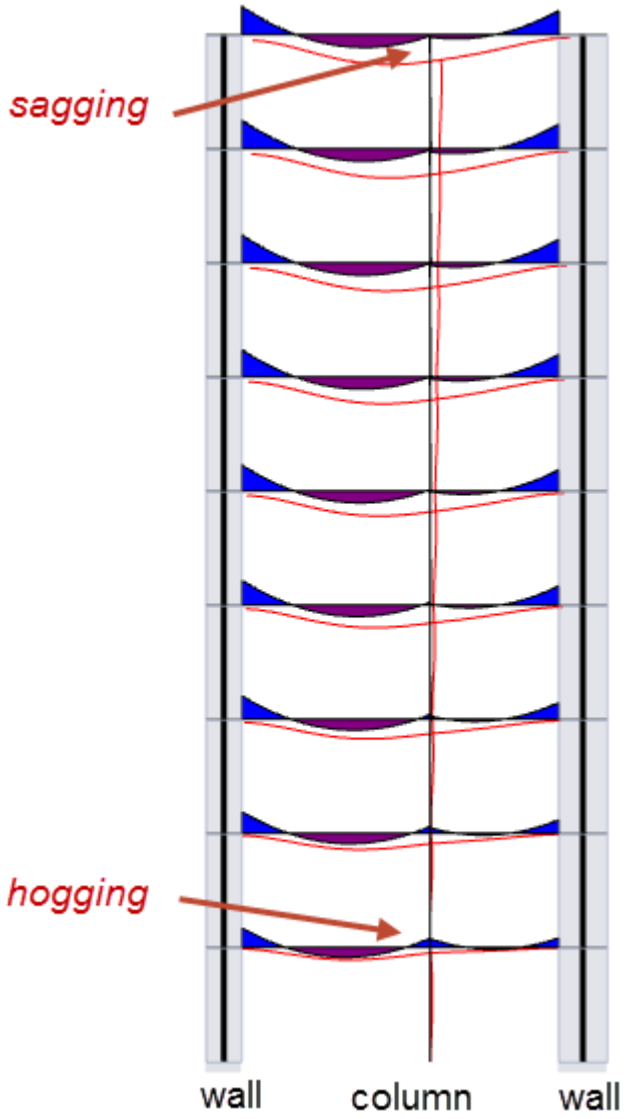
- 3D analysis - frame deflects and loads are shared according to stiffness.
- Grillage chasedown - loads are collected floor by floor and then applied to the transfer beam - much higher moment in the transfer beam. Once again this is more in line with traditional expectation.

A third example of why chasedown is wanted follows below....

Differential axial deformation (axial shortening)

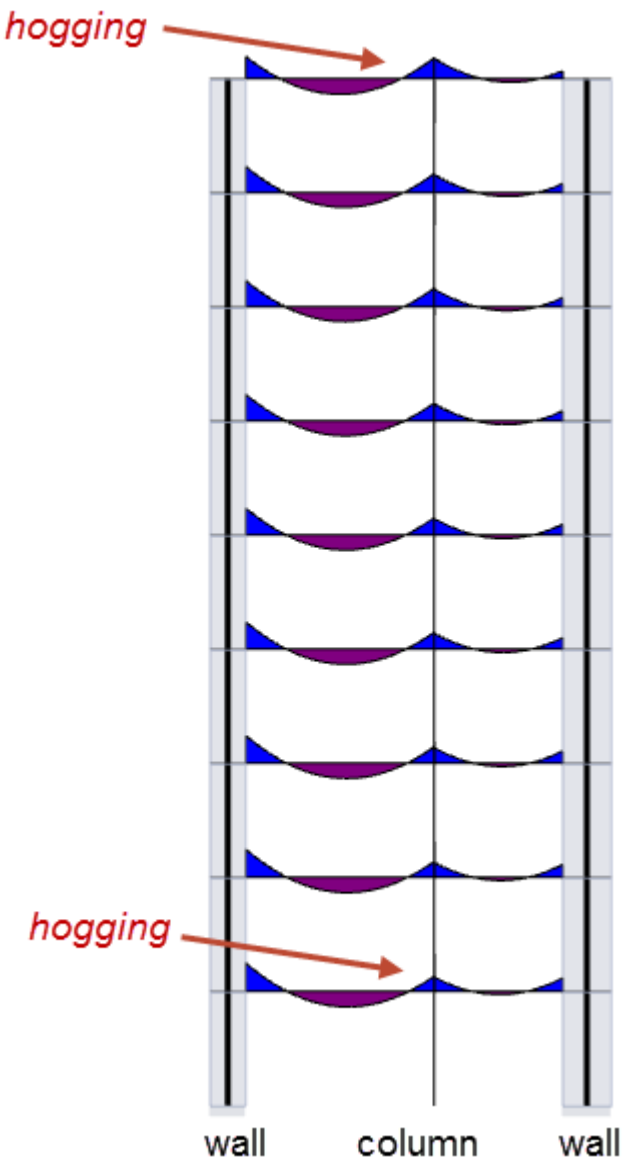
In this case a two span beam sits on walls at each end and a column in the center.

3D Analysis Results:



- Column is more highly stressed than walls and deflects downwards shedding load back to the walls.
- You end up with completely different moment profiles on beams that would traditionally be considered and designed as identical.

Grillage chasedown Results:



• the sub-models at each level are almost identical and you get very similar results over the height of the building.

Findings from the above examples

- Grillage chasedown emulates a more traditional design style where continuous beams or sub-frames are considered in isolation.
- The 3 examples show where 3D analysis gives results that are not liked (based on traditional design expectation).
- But once you start to think about it you may conclude that actually, the 3D analysis result should not be ignored.

- By running **Analyze All (Static)** each analysis type is performed and members can be simultaneously designed for each set of results.

NOTE Deliberately extreme examples have been used to illustrate these effects and in real models the differences between the sets of results might not be as dramatic.

Features of 3D analysis and chasedown analysis

The following table compares the features of the three analysis types that can be used to generate sets of forces suitable for static design.

NOTE In the table the term '3D Analysis' refers to any of 1st-order linear, 1st order non-linear, 2nd order linear, 2nd order non-linear analysis types.

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
Examples / When useful?	Gravity and Lateral analysis (Notional/Wind/ Seismic)	“Beam & Slab” buildings	Flat slab and “Beam & Slab” buildings
Special Features	• Pattern loading • Automatic EC2 sway sensitivity assessment and sway amplification • Automatically centralised analysis wires (improved rigid offsets / rigid zones) • Option to mesh slabs in the 3D analysis	• Mimics traditional design approach (sub-frame analysis) • Pattern loading	• Mimics traditional design approach (isolated floor analysis) • Slab Pattern loading
Benefits	• Member Design considers sway and differential axial	Member design based on traditional sub frame is considered simultaneously	• Member design based on traditional sub frame is considered simultaneously

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
	deformation effects. • Caters for slabs that contribute to the lateral load resisting system	with that for 3D Analysis	with that for 3D Analysis • Irregular slab panel design automatically catered for
Analysis Model	3D model of entire building: • either meshed 2-way slabs, • or, slab loads decomposed to beams	series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • either meshed 2-way slabs, • or, slab loads decomposed to beams	series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • all 2-way slabs meshed
Analysis Method	Whole model in one pass	Each sub model sequentially from top to bottom – chasing member loads down	Each sub model sequentially from top to bottom – chasing member loads down
Analysis Type	• First order • First order - Kamp • Second order - P-D	First order	First order
Supports	External supports as defined by the user	Ends of members above/below each sub model are automatically supported	Ends of members above/below each sub model are automatically supported
Loading	Gravity and Lateral Loads	Gravity Loads only	Gravity Loads only
Forces for design			
RC Slab	Yes– All Combs	No forces	Yes – All Gravity loadcases

Feature	3D Analysis	Grillage Chasedown	FE Chasedown
RC Beam	Yes– All Combs	Yes – All Gravity loadcases	Optional – All Gravity loadcases
RC Column	Yes– All Combs	Yes – All Gravity loadcases	Optional – All Gravity loadcases
RC Wall	Yes– All Combs	Yes – All Gravity loadcases	Optional – All Gravity loadcases
Steel/Composite Members	Yes – All Combs except patterns	Not required	Not required
Timber Members (via Tedds)	Yes	Yes	Yes
Foundation design	Yes – All Combs except patterns	Yes – All Gravity loadcases	Yes – All Gravity loadcases

Related information

See also

Background to the Analysis/Design procedures (eLearning)

1st order non-linear (page 483)

2nd order linear (page 485)

2nd order non-linear (page 485)

Grillage chasedown (page 489)

FE chasedown (page 487)

Why chasedown analyses are included in Analyze All (Static) (page 499)

Modal analysis

What is modal analysis?

Tekla Structural Designer's modal analysis is an analysis of **undamped and simple-harmonic** vibrations.

To formulate a solution we begin by considering the discrete dynamic equation used in finite element analysis:

$[M]\{\ddot{u}\} + [C]\{u\} + [K]\{u\} = \{F\}$

Where *[M]* is the mass matrix, *[C]* is the damping matrix and *[K]* the stiffness matrix. All three matrices are constant in linear dynamics. *{ü}*, *{u}* and *{u}* are respectively the acceleration vector, velocity vector and the displacement vector. All three vectors vary as a function of time, as does *{F}* which is the load-vector (sometimes referred to as the forcing function).

Thinking about this equation in physical terms; *[M]{ü}* is the inertial force (from Newton's 2nd law), *[C]{u}* the damping force and *[K]{u}* the internal elastic force (i.e. strain energy).

By making some simplifications - such as that damping is zero - this can be reduced to:

$[M]\{\ddot{u}\} + [K]\{u\} = 0$

A physical explanation of this equation is that, for a valid vibration mode at any instant of time, the inertial forces (*[M]{ü}*) and internal elastic forces (*[K]{u}*) are equal and opposite, while no external force is applied to the system. This is known as *free vibration*. Free vibration isn't related to static loading (other than where this defines mass) and will identify frequencies at which the structure can naturally resonate, based on its mass distribution and stiffness alone.

Solutions for the above equation have the following form:

$\{u\}(t) = \{U\}e^{i\omega t}$ *and* $\{\ddot{u}\}(t) = -\omega^2\{U\}e^{i\omega t}$

Substituting and dividing by $e^{i\omega t}$ (which represents the time-response and is simply a sine wave since the behaviour is simple harmonic) we obtain the ***eigenvalue equation***:

$([K] - \omega^2[M])\{U\} = \{0\}$

In which ω^2 is the ***eigenvalue*** and *{U}* the ***eigenvector***.

Dividing by *{U}* and rearranging gives the following equation, which you may recognize as being of the same form as the fundamental equation for frequency (valid only for a single degree of freedom (SDOF) system) $f \propto \sqrt{(k/m)}$:

$\omega = \sqrt{([K]/[M])}$

Note the eigenvalue equation contains no terms for force or acceleration.

Tekla Structural Designer's modal analysis is a solution of this eigenvalue equation for the specific model. The method of solution is highly complex - if you are interested, details can be found in any good reference on vibration theory (we can recommend ***Vibration: Fundamentals and Practice*** Clarence W. de Silva, Boca Raton: CRC Press LLC, 2000). There are many (numerical) methods of solution, no fewer than three of which are implemented in TSD; Jacobi, Subspace and FEAST (the selection of the method used being automated).

The solution to the eigenvalue equation is the vector *{U}* - which is the ***mode shape*** - with a corresponding frequency which is the square root of the eigenvalue; $\omega = \sqrt{\omega^2}$. The mode shape is just that; a *shape*. It can have any *amplitude* - modal analysis does not give this, just the shape. So it does not produce 'real' deflections or forces.

Note that ω is the angular (or 'circular') frequency in terms of radians and so must be converted (by dividing by 2π) to be in Hz.

There is not a single solution to the eigenvalue equation (other than for a single degree of freedom (SDOF) system) - a structure which has many DOFs will have multiple mode shapes*****. Each mode shape occurs at a very specific frequency called the *natural frequency* of the mode in question.

*****In reality the number of modes of even a simple structure is potentially infinite, however in the finite element method solution it is finite and is governed by the ***discretization*** of the numerical model i.e. how many elements and nodes it contains.

- We can easily calculate the number of mode shapes for any numerical model - it is the number of free nodes (i.e. those not at a support) multiplied by the number of DOF's (i.e. mass directions) at each node. In 3D space and using the consistent mass matrix (the default), there are 6 DOFs (3 translational and 3 rotational). Hence a model with 100 nodes will have 100*6 = 600 modes.
- From this you will appreciate that even a relatively small model will have a large number of modes, while a large model will have 1000's. However we are generally only interested in the lower modes - i.e. those with lower frequencies - which a) will involve the majority of the structure and b) are most likely to be excited (by whatever forcing function is in play - e.g. an earthquake or human movement) and thus problematic.

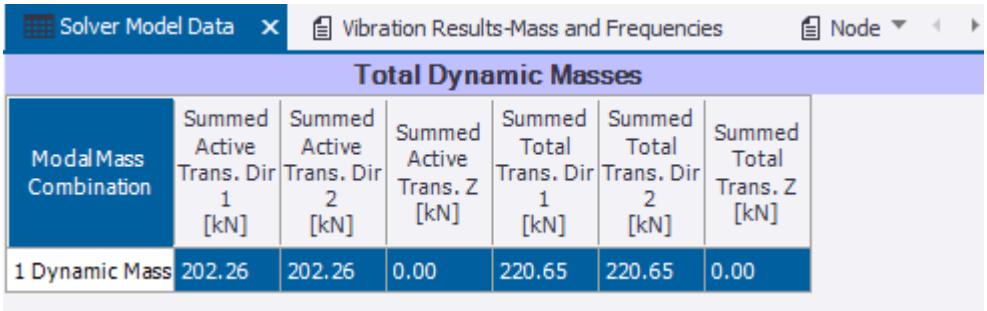
So in summary; modal analysis in Tekla Structural Designer is an idealized solution which finds the natural frequencies of a structure that does not consider damping or dynamic loading 'input' to your structure, which may occur in the real world. The results it produces are a list of modes - in order of ascending frequency - together with their: frequency; period; mass participation % and modal mass.

Active mass and mass participation

In a 1st order modal analysis mass is assigned to nodes of the analysis model. In simple terms (neglecting rotation terms for the consistent mass matrix) half of each element mass is assigned to each node it is attached to.

Mass that is assigned to a translational support cannot go anywhere - i.e. it is not "active".

Summed active translational mass



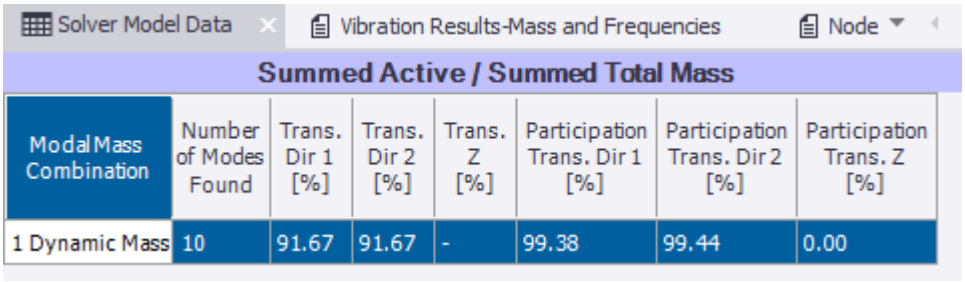
Modal Mass Combination	Summed Active Trans. Dir 1 [kN]	Summed Active Trans. Dir 2 [kN]	Summed Active Trans. Z [kN]	Summed Total Trans. Dir 1 [kN]	Summed Total Trans. Dir 2 [kN]	Summed Total Trans. Z [kN]
1 Dynamic Mass	202.26	202.26	0.00	220.65	220.65	0.00

Reported in the Dynamic Masses (page 720) table, this is the actual total active mass for each direction, but expressed in terms of force units rather than mass.

Summed total translational mass

Reported in the Dynamic Masses (page 720) table (see above), this is the total system mass for each direction, again expressed in terms of force units rather than mass.

Translation %


Solver Model Data | Vibration Results-Mass and Frequencies | Node

Summed Active / Summed Total Mass

Modal Mass Combination	Number of Modes Found	Trans. Dir 1 [%]	Trans. Dir 2 [%]	Trans. Z [%]	Participation Trans. Dir 1 [%]	Participation Trans. Dir 2 [%]	Participation Trans. Z [%]
1 Dynamic Mass	10	91.67	91.67	-	99.38	99.44	0.00

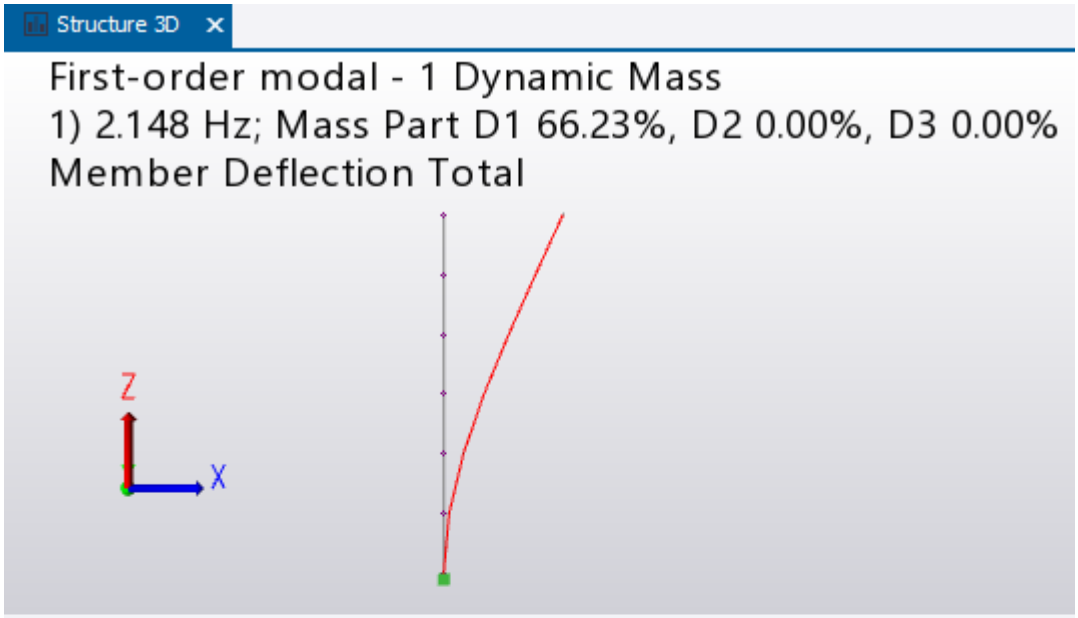
Reported in the Summed Mass (page 720) table, this is the proportion of mass that is active for each direction. For a building this will usually be close to but not quite 100% as some mass always goes to the supports.

Translation % = (Summed Active Mass / Summed Total Translational Mass) x 100

Participation translation %

Reported in the Summed Mass (page 720) table (see above), this is the sum of **Mass Participation %** (reported in the Modal Frequencies (page 720) table,

as shown below) for all modes for each direction.



Solver Model Data | Vibration Results-Mass and Frequencies | Nodes and Mode shapes

Modal Frequencies

Mode Number	Period [sec]	Frequency [Hz]	Error [%]	Mass Participation Dir 1 [%]	Mass Participation Dir 2 [%]	Mass Participation Z [%]	Modal Mass Dir 1 [kN]	Modal Mass Dir 2 [kN]	Modal Mass Z [kN]
1	0.466	2.148	0.00	66.23	0.00	0.00	56.42	56.42	56.42
2	0.349	2.862	0.00	0.00	66.27	0.00	56.47	56.47	56.47
3	0.077	12.980	0.00	20.71	0.00	0.00	65.14	65.14	65.14
4	0.058	17.210	0.00	0.00	20.78	0.00	65.42	65.42	65.42
5	0.029	35.016	0.00	7.08	0.00	0.00	87.17	87.17	87.17
6	0.022	46.060	0.00	0.00	7.11	0.00	87.55	87.55	87.55
7	0.015	65.427	0.00	3.53	0.00	0.00	123.58	123.58	123.58
8	0.012	84.978	0.00	0.00	3.51	0.00	122.29	122.29	122.29
9	0.010	100.225	0.00	1.84	0.00	0.00	105.31	105.31	105.31
10	0.008	128.037	0.00	0.00	1.77	0.00	106.06	106.06	106.06
				Σ= 99.38	Σ= 99.44				

Mass participation is used in seismic analysis - in simple (not literal) terms it is a measure of the *momentum* of a mode and so a predictor of the base shear it produces.

Design codes stipulate that the sum for all the modes should be ≥ 90% for seismic analysis usually for two orthogonal lateral directions.

Modal mass

After running a 1st order modal analysis, modal masses for each mode are available in the Modal Frequencies (page 720) table.

In Tekla Structural Designer the modal mass, M_i is given by the following matrix equation:

$M_i = \{ \Psi \}_i^T [M] \{ \Psi \}_i$

Where $\{\Psi\}_i$ is the unity-scaled mode shape (often termed mode vector) of the i^{th} mode (i.e. any single mode) and *[M]* is the mass matrix. The meaning of the unity-scaled mode shape is that the (numerically) largest modal displacement is set to unity and all other displacements are scaled accordingly. The term $\{\Psi\}_i$ is used to differentiate this mode shape from the mass-normalized shape $\{\Phi\}_i$ which is the mode shape actually reported by Tekla Structural Designer.

This equation comes from modal analysis theory. It may also be termed "generalized mass".

Another way to state this equation, which is found in some design guides, is a summation equation for point masses and their associated modal displacements for a system of discretized mass distribution:

We have from CCIP-016:

$$\hat{m}_j = \sum_{i=1}^{N} \mu^2_{j,i} m_i$$

where i is each on N points on the structure, having mass m_i at which the mode shape $\mu_{j,i}$ is the j^{th} mode is known.

According to this reference - "Conceptually, the modal mass can be thought of as the mass of an equivalent single degree of freedom system... which represents the j^{th} mode."

It can be seen how this is equivalent to the matrix equation given above. Actually Tekla Structural Designer makes use of a shortcut calculation since it already has mode shapes which are normalized to mass.

The mass-normalized mode shape $\{ \Phi\}_i$ and the unity-normalized mode shape $\{\Psi\}_i$ are related as follows:

$$\{\Phi\}_i = \frac{1}{\sqrt{M_i}} \{\psi\}_i$$

From this we can state the following, where Φ_2 is the largest modal displacement from the mass-normalized mode shape (which we already have from the Tekla Structural Designer modal analysis):

$$\Phi_2 = \frac{1}{\sqrt{M_i}} \times 1$$

Hence:

$$\text{Modal mass} = M_i = \frac{1}{\Phi_2^2}$$

Run a 1st order modal analysis

In order to run a modal analysis on your model, see the following instructions.

NOTE To run a modal analysis, your model must contain an active modal mass combination.

For more information, see Create modal mass combinations (page 343).

---

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the page (page 763).
3. If necessary, adjust the analysis options.
4. On the **Analyze** tab, click **1st Order Modal**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** opens to allow you to display the mode shapes graphically. Tabular results can also be viewed in a **Solver Model Data View**.

See also

Apply and manage analysis settings (page 1263)

Display mode shapes (page 701)

Summed mass (view as tabular data) (page 720)

Dynamic masses (view as tabular data) (page 720)

Active masses by node (view as tabular data) (page 721)

Modal frequencies and modal masses (view as tabular data) (page 720)

RSA seismic analysis

In order to run either a 1st or 2nd order seismic analysis on your model, see the following instructions.

Run a 1st order RSA seismic analysis

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the **1st Order Seismic** page.
3. If necessary, adjust the analysis options.
4. Click **OK**.
5. On the **Analyze** tab, click **1st Order RSA Seismic**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

Run a 2nd order RSA seismic analysis

NOTE While RSA analysis itself is always 1st order, in this analysis type the RSA results are enveloped around 2nd order static analysis results.

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the **1st Order Seismic** page.
3. If necessary, adjust the analysis options.
4. Click **OK**.
5. On the **Analyze** tab, click **2nd Order RSA Seismic**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

Run Analyze All (RSA)

1. On the **Analyze** ribbon, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the **1st Order Seismic** page.

3. If necessary, adjust the analysis options.
4. Click **OK**.
5. On the **Analyze** tab, click **Analyze All (RSA)**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process the **Results View** and the **Results** tab open to allow you review the analysis results graphically.

See also

Apply and manage analysis settings (page 1263)

RSA seismic results (page 701)

Analysis results (page 662)

Analysis limitations and assumptions

Certain specific limitations and assumptions relating to the various analysis types are expanded upon below:

Linear analysis of structures containing material nonlinearity

If a structure containing nonlinear springs or nonlinear elements is subjected to a linear (i.e. 1st or 2nd order linear, 1st order modal, or 2nd order buckling) analysis, then the nonlinear springs/elements are constrained to act linearly as described below:

Nonlinear spring supports

In each direction in which a nonlinear spring has been specified, a single value of stiffness is applied which is taken as the greater of the specified -ve or +ve stiffness.

Any specified maximum capacities of the spring are ignored.

Tension only, or compression only elements

If either tension only or compression only element types have been specified, they are constrained to act as truss element types instead.

Nonlinear axial spring, or nonlinear torsional spring elements

If either of these element types have been specified, they are constrained to act as linear axial spring, or linear torsional spring element types instead.

A single value of stiffness is applied which is taken as the greater of the specified -ve or +ve stiffness.

Any specified maximum capacities of these spring elements are ignored.

Tension only X braces

It is essential that the "X Brace" pattern is used to input cross braces as brace pairs rather than creating them individually. For linear analysis one of the

braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

To determine which brace in each pair is inactivated the program pushes the structure simultaneously in the positive direction 1 and positive direction 2. The brace that goes into tension retains its full stiffness, while the compression brace becomes inactive.

If the above process fails to determine which of the pair goes into tension and which is inactivated then a shear is applied to the structure and the braces are re-assessed.

Analysis of structures containing geometric nonlinearity

It is assumed that where secondary effects are significant (for example the structure is close to buckling), the engineer will elect to undertake a 2nd order analysis. If a 1st order analysis is performed any secondary effects will be ignored.

Analysis of structures containing curved beams

The member analysis for curved members in the plane of the curve is approximated by joining the values at the nodes, which are correct. For detailed analysis of curved members it is your responsibility to ensure sufficient discretization. More refined models can be achieved, if required, by decreasing the maximum facet error.

Analysis of compound (plated) steel beams and columns

Compound (plated) sections are 2 chords or more connected by battens, lattice or welded.

- Static calculations for these section types are the same as for solid sections. The section characteristics are calculated in the basis of the actual section.
- The torsional constant of a compound section is calculated as a total of torsional constants of the chords in a compound section.
- Plane section remain plane.
- The material is homogeneous, isotropic and linearly elastic.
- Saint Venant's principle applies.

Member Deflections

There is a known issue when calculating member deflection profiles in combinations which can affect the following analysis types:

- 2nd Order Linear

- 1st Order Nonlinear
- 2nd Order Nonlinear

This occurs when the structures behavior is significantly nonlinear because the deflection profile is currently based on linear superposition of the loadcases within it. Clearly as structural response becomes more nonlinear the assumption that deflections can be superposed becomes less valid. This can cause a deflected profile to be calculated which deviates from the correct profile. The deviation can become significant if loadcases fail to solve, but the combination succeeds in solving, as components of the deflected shape are missing entirely. It is suggested that for the three analysis types listed member deflections in combinations be used with caution and engineering judgment.

It should be noted that this limitation only affects member deflection profiles between solver nodes. All other results, including member force profiles and deflection at the solver nodes are correct.

Torsion load analysis - relative angle of twist

Any section that is subject to torsional moment will rotate through an angle, θ. If the cross-section is non-circular this will also be accompanied by warping.

To be able to determine stresses on members subject to torsional moments it is necessary to determine θ (and for "Open" sections its derivatives also).

For single span pinned steel beams only: a torsion load analysis is performed which enables θ (and its derivatives) to be calculated and made available in the Load Analysis View.

For open sections: a more accurate approach is used to determine θ and its derivatives. The 'cases' in DG9, SCI P057 & SCI P385 being used which depend on the end conditions and loading conditions on the beam.

This more accurate analysis is carried out for the following open sections:

i. I Symmetric

ii. I Asymmetric

iii. I Plated (including Westok Plated)

iv. Channel

v. Westok cellular beyond scope

For all other sections not mentioned above (including compound sections): a "Standard" analysis is carried out to determine θ only using the following equation:

$1 / GI_T * \int T(x)$

Where:

I_T = torsion constant

G = shear modulus of steel

T(x) = function of torsion moment

- 2nd Order Linear
- 1st Order Nonlinear
- 2nd Order Nonlinear

Unstable Structures

Flat Slab Structures

If a concrete structure exists with only flat slabs and columns (i.e. no beams and no shear walls), and the slab is modeled with a diaphragm this is an unstable structure, assuming that the concrete columns are pinned at the foundation level (current default).

To prevent the instability you should mesh the slabs, as the resulting model does then consider the framing action that results from the interaction of the slabs and columns.

Analysis verification examples

A small number of verification examples are included in this section. Our full automatic test suite for the Solver contains many hundreds of examples which are run and verified every time the Solver is enhanced. These verification examples use SI units unless otherwise stated.

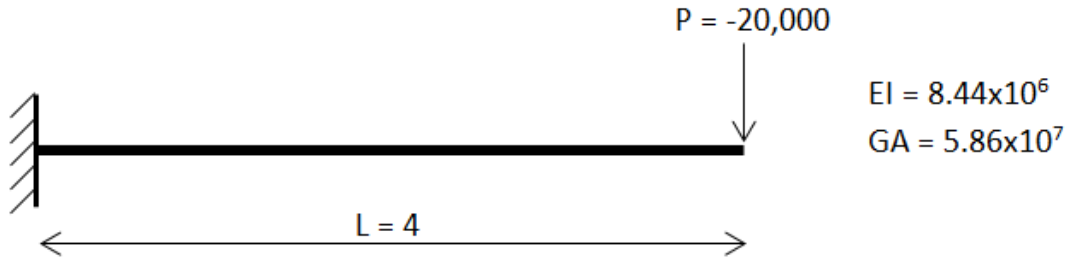
- 1st order linear - Simple cantilever (page 520)
- 1st order linear - Simply supported square slab (page 520)
- 1st order linear - Simply supported orthotropic square slab (page 522)
- 1st order linear - 3D truss (page 524)
- 1st order linear - Thermal load on simply supported beam (page 525)
- 1st order nonlinear - Simple cantilever (page 525)
- 1st order nonlinear - Nonlinear supports (page 526)
- 1st order nonlinear - Displacement loading of a plane frame (page 527)
- 2nd order linear - simple cantilever (page 528)
- 2nd order linear - Simply supported beam (page 529)
- 2nd order nonlinear - Tension only cross brace (page 531)
- 2nd order nonlinear - Compression only element (page 532)
- 1st order modal - Simply supported beam (page 533)
- 1st order modal - Bathe and Wilson eignenvalue problem (page 533)
- 2nd order buckling - Euler strut buckling (page 534)

- 2nd order buckling - Plane frame (page 535)

1st order linear - Simple cantilever

Problem definition

A 4 long cantilever is subjected to a tip load of 20,000.



Assumptions

Flexural and shear deformations are included.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Support Reaction	-P	20,000	20,000	0%
Support Moment	PL	-80,000	-80,000	0%
Tip Deflection	$\frac{PL^3}{3EI}+\frac{PL}{GA}$	-0.0519	-0.0519	0%

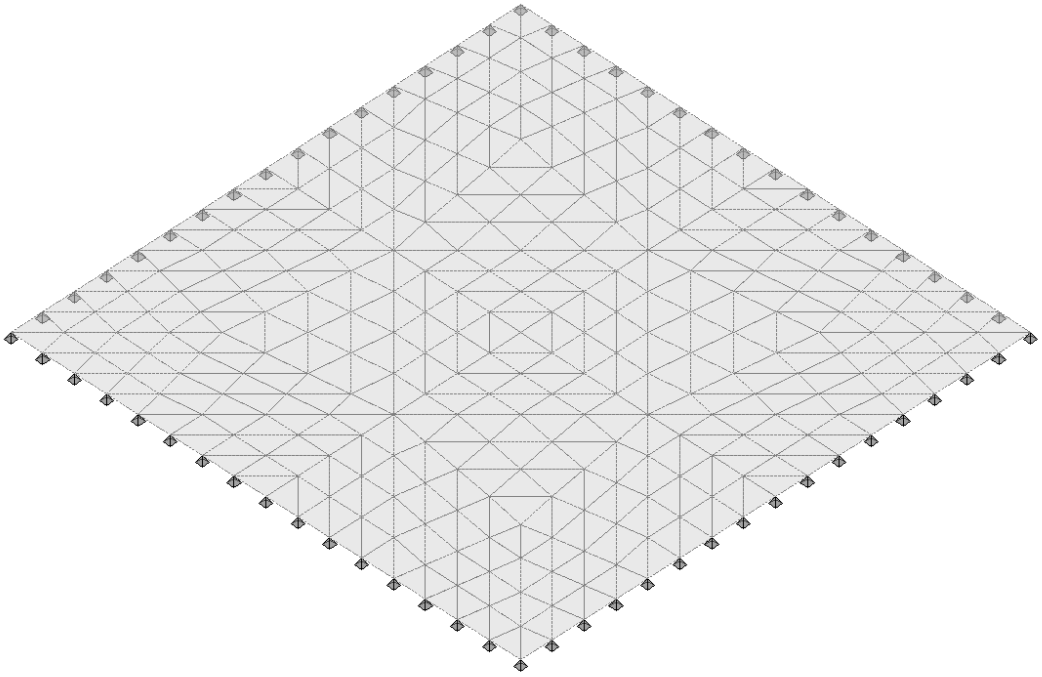
Conclusion

An exact match is observed between the values reported by the solver and the values predicted by beam theory.

1st order linear - Simply supported square slab

Problem definition

Calculate the mid span deflection of an 8x8 simply supported slab of 0.1 thickness under self-weight only. Take material properties $E=2x10^{11}$, $G=7.7x10^{10}$ and $\rho=7849$.



Assumptions

A regular triangular finite element mesh is used with sufficient subdivision. Flexural and shear deformation is included, and the material is assumed to be isotropic.

Key results

The mid-span deformation is calculated using Navier's Method.

$$w = \frac{16q_0}{\pi^6 D}\sum_{m=1}^{9}\sum_{n=1}^{9}\frac{\sin\frac{m\pi x}{a}\sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2}$$

$$M_x = M_y = \frac{16q_0}{a^2\pi^4}\sum_{m=1}^{9}\sum_{n=1}^{9}\frac{m^2\sin\frac{m\pi x}{a}\sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2} + \nu\frac{16q_0}{b^2\pi^4}\sum_{m=1}^{9}\sum_{n=1}^{9}\frac{n^2\sin\frac{m\pi x}{a}\sin\frac{n\pi y}{b}}{mn\left(\frac{m^2}{a^2}+\frac{n^2}{b^2}\right)^2}$$

Result	Theoretical value	Comparison 1	Solver value	% error
Mid-span deflection	$7.002\text{x}10^{-3}$	$6.990\text{x}10^{-3}$	$7.031\text{x}10^{-3}$	0.43%
Mid-span Moment	23616	23708	23649	0.14%

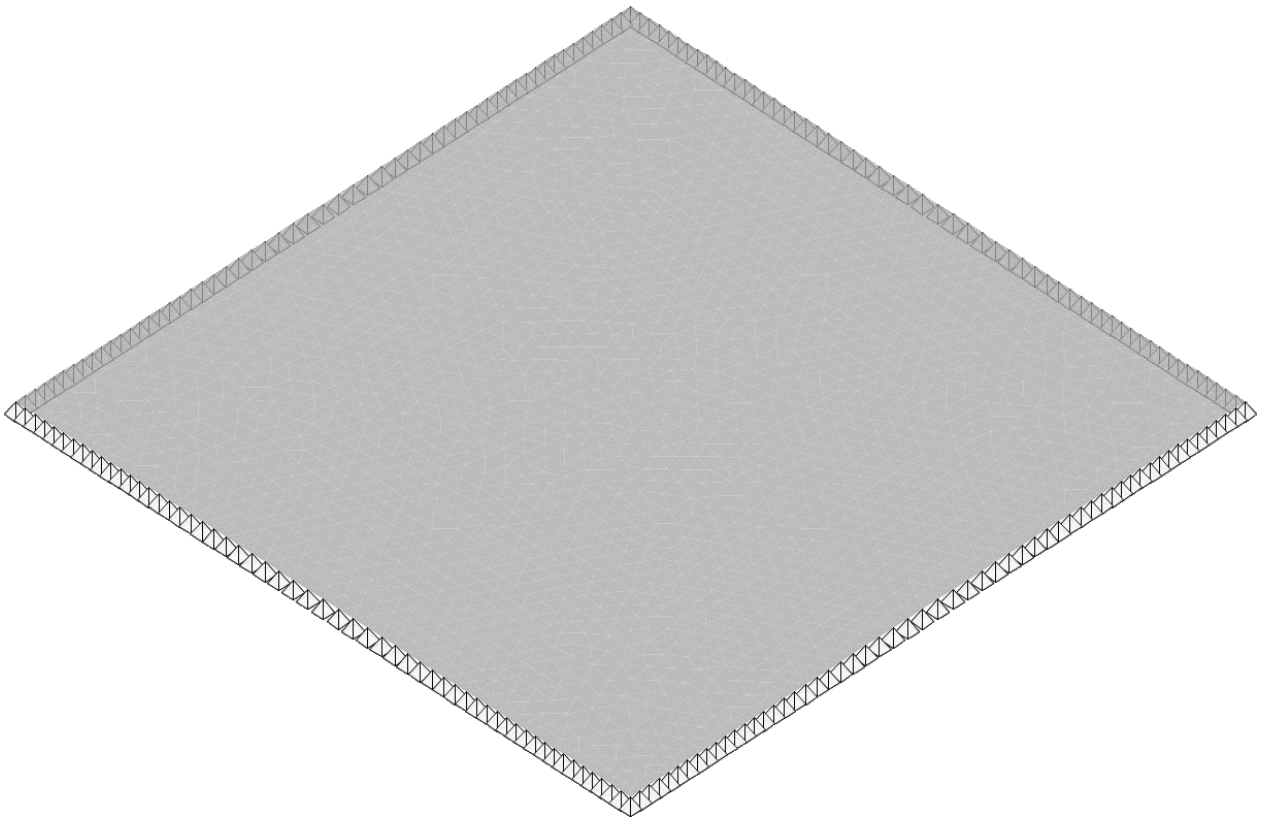
Conclusion

An acceptable match is observed between the theoretical values and the solver results. An acceptable match is also observed between the solver results and those obtained independently.

1st order linear - Simply supported orthotropic square slab

Problem definition

Calculate the mid-span deflection of a 10m by 10m simply supported slab of 25mm thickness with 10N/m^2 UDL applied to it. The material properties the slab were assumed to be as follow: Ex = 33.5 MPa, E_y = 33500 MPa, G_{xy} = 523 MPa, ν_x = 0.4000, and ν_y = 0.0004.



FEM model

A regular triangular finite element mesh is used with a mesh size of 0.25m.

Analytical solution

The deflection was calculated using the solution obtained by Timoshenko and Woinowsky-Kreiger. The solution uses Navier's method, which employs the double trigonometric series equation:

$$w(x,y) = \frac{16q_0}{\pi^6} \sum_{\substack{m=1,3,5,\dots \\ n=1,3,5,\dots}}^{\infty} \frac{\sin\frac{m\pi x}{a} \cos\frac{n\pi y}{b}}{mn\left(D_x \frac{m^4}{a^4} + H \frac{2m^2n^2}{a^2b^2} + D_y \frac{n^4}{b^4}\right)}$$

where:

w - deflection,

a = b - side lengths

x and y - coordinates of the point of interest

$$D_x = \frac{E_x h^3}{12\left(1 - \nu_x \nu_y\right)}$$

$$D_y = \frac{E_y h^3}{12\left(1 - \nu_x \nu_y\right)}$$

$$H = \nu_x D_x + 2\frac{G_{xy} h^3}{12} = \nu_y D_y + 2\frac{G_{xy} h^3}{12}$$

Results

Value	Analytical solution (mm)	Solver result (mm)	% Error
Mid-span deflection	29.799	29.795	0.01

Conclusion

A satisfactory convergence of the analytical and solver's results was observed.

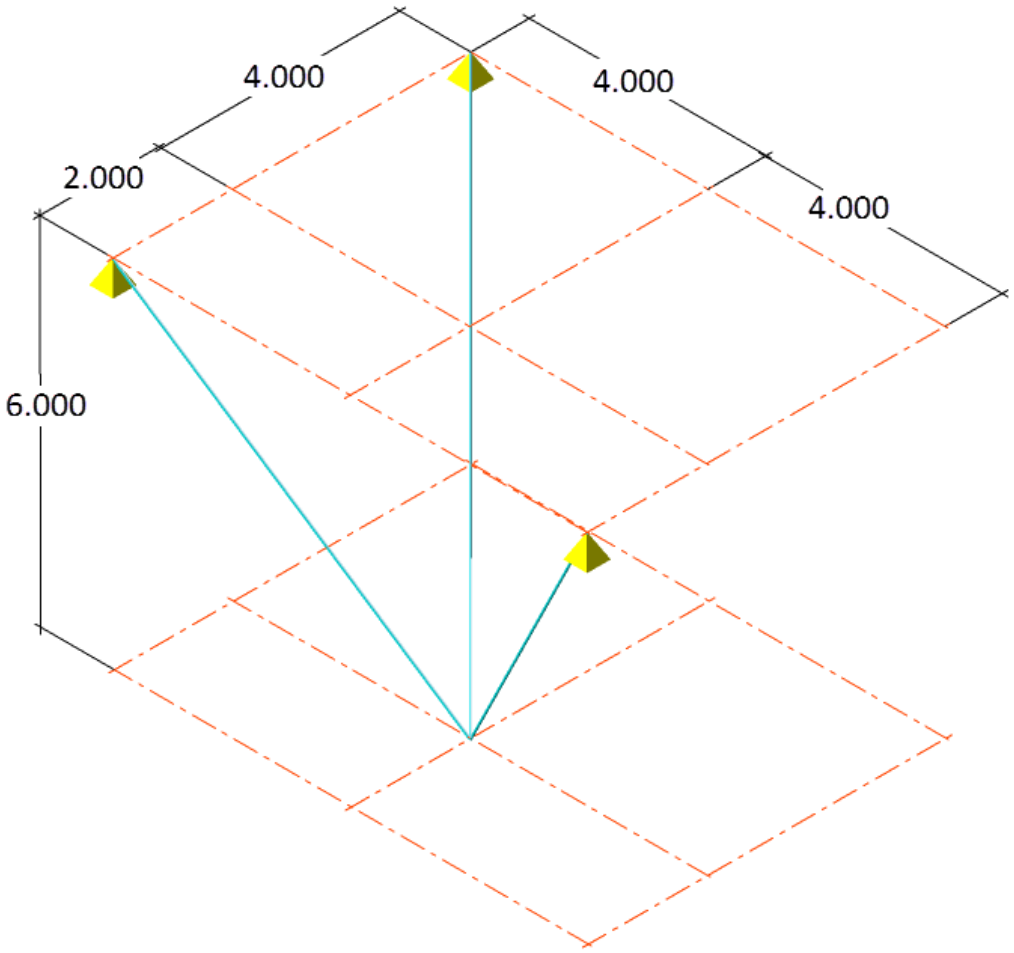
References

Timoshenko, S. and Woinowsky-Krieger, S. 1987. *Theory of Plates and Shells*. 2nd Edition. McGraw-Hill Book Company, Inc. New York, NY, USA.

1st order linear - 3D truss

Problem definition

Three truss members with equal and uniform *EA* support an applied load of -50 applied at the coordinate (4, 2, 6). The start of each truss member is fixed and are located at (0, 0, 0), (8, 0, 0) and (0, 6, 0) respectively. Calculate the axial force in each element.



Key results

The results for this problem are compared against those published by Beer and Johnston, and against another independent analysis package.

Result	Beer and Johnston	Comparison 1	Solver value	% error
(0, 0, 0) - (4, 2, -6)	10.4	10.4	10.4	0.00 %
(8, 0, 0) - (4, 2, -6)	31.2	31.2	31.2	0.00 %
(0, 6, 0) - (4, 2, -6)	22.9	22.9	22.9	0.00 %

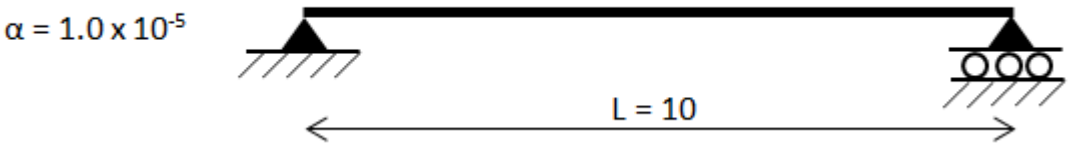
Conclusion

An exact match is observed between the values reported by the solver and those reported by Beer and Johnston.

1st order linear - Thermal load on simply supported beam

Problem definition

Determine the deflection, *U*, due to thermal expansion at the roller support due to a temperature increase of 5. The beam is made of a material with a thermal expansion coefficient of 1.0×10^{-5}.



Assumptions

The roller pin is assumed to be frictionless.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Translation at roller	$U = \Delta T \times \alpha \times L$	5×10^{-4}	5×10^{-4}	0%

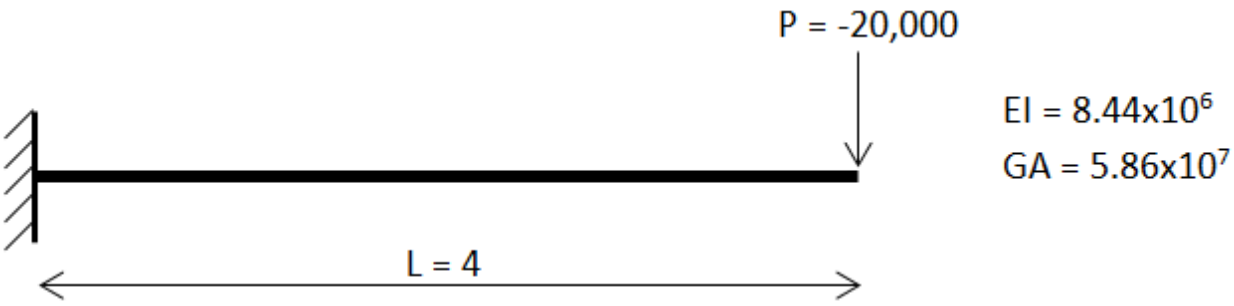
Conclusion

An exact match is shown between the theoretical result and the solver result.

1st order nonlinear - Simple cantilever

Problem definition

A 4 long cantilever is subjected to a tip load of 20,000.



Assumptions

Flexural and shear deformations are included.

Key results

Result	Theoretical formula	Theoretical value	Solver value	% error
Support Reaction	-P	20,000	20,000	0 %
Support Moment	-PL	-80,000	-80,000	0 %
Tip Deflection		-0.0519	-0.0519	0 %

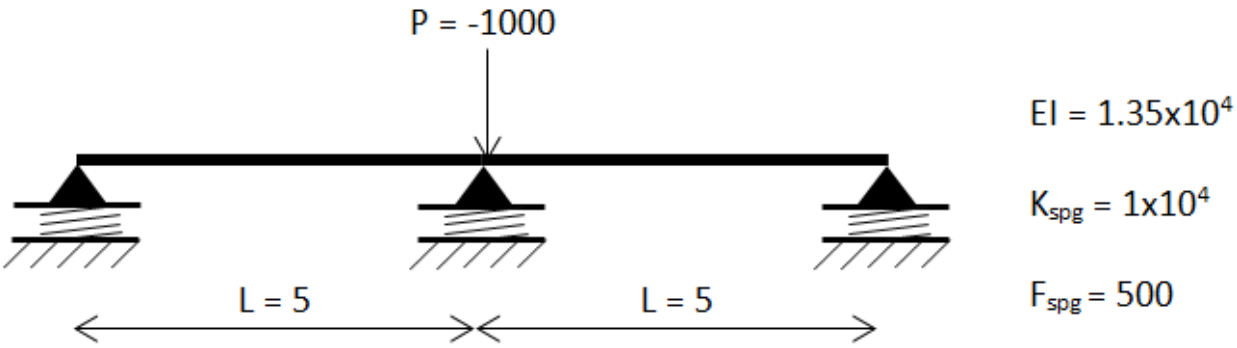
Conclusion

An exact match is observed between the values reported by the solver and the values predicted by beam theory.

1st order nonlinear - Nonlinear supports

Problem definition

A 10 long continuous beam is simply supported by three translational springs as shown. All springs have a maximum resistance force of 500. Calculate the reaction forces and deflection at each support.



Assumptions

Axial and shear deformations are ignored.

Key results

Result	Comparison 1	Solver value
LHS Reaction	250	250
Centre Reaction	500	500
RHS Reaction	250	250
LHS Displacement	-0.025	-0.025
Centre Displacement	-0.797	-0.797
RHS Displacement	-0.025	-0.025

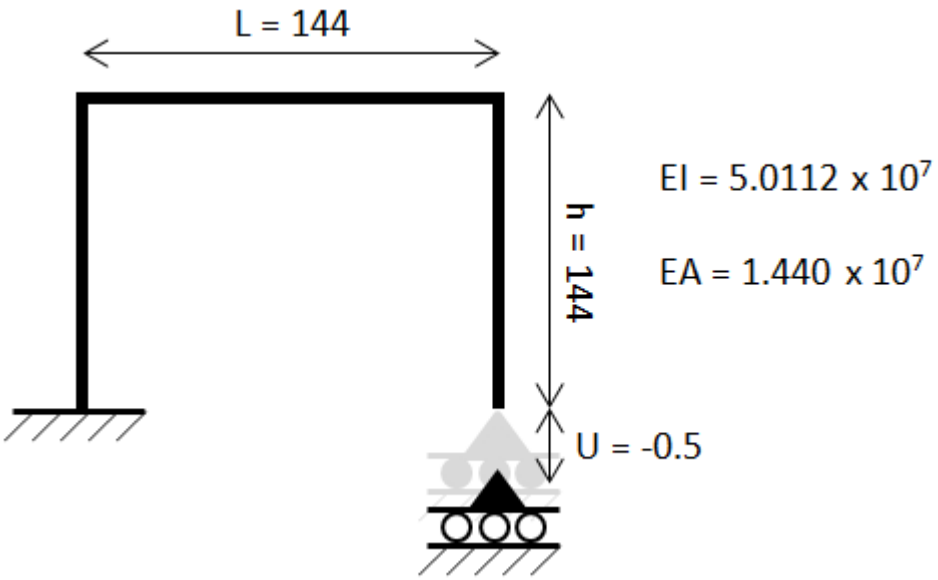
Conclusion

An exact match is shown between the solver and the independent analysis package.

1st order nonlinear - Displacement loading of a plane frame

Problem definition

Calculate the reaction forces of the plane moment frame shown below with the applied displacement *U*.



Assumptions

All elements are constant and equal *EI*. Axial and shear deformations are ignored; to achieve the former - analytically the cross sectional area was increased by a factor of 100,000 to make axial deformation negligible.

Key results

Results were compared with two other independent analysis packages.

Result	Comparison 1	Comparison 2	Solver value
LHS Vertical Reaction	6.293	6.293	6.293
LHS Moment Reaction	-906.250	-906.250	-906.250
RHS Vertical Reaction	-6.293	-6.293	-6.293

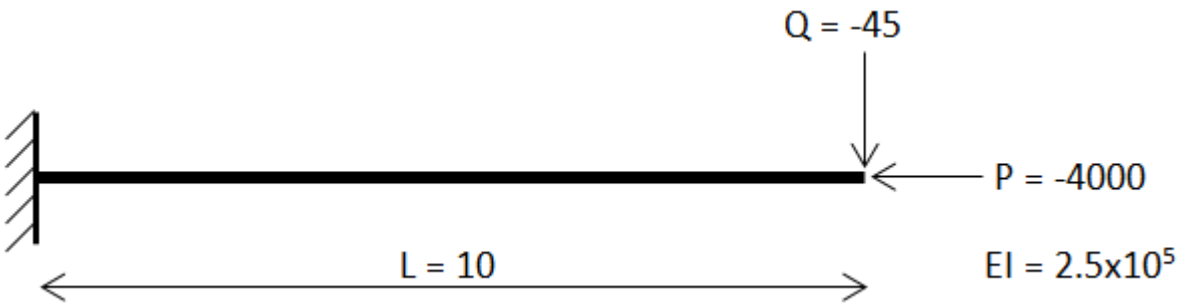
Conclusion

An exact match is shown between the solver and the two independent analysis packages.

2nd order linear - simple cantilever

Problem definition

A 10 long cantilever is subjected to a lateral tip load of 45 and an axial tip load of 4000.



Assumptions

Shear deformations are ignored. Results are independent of cross section area; therefore any reasonable value can be used. Second order effects from stress stiffening are included, but those caused by update of geometry are not. The beam is modelled with only one finite element, (if more elements had been used the result would converge on a more exact value).

Key results

Results were compared with an independent analysis package.

Result	Comparison	Solver value
Tip Deflection	-0.1677	-0.1677
Base Moment Reaction	-1121	-1121

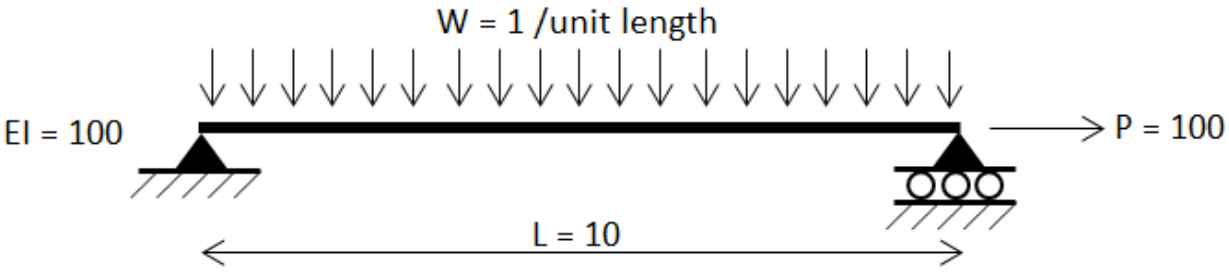
Conclusion

An exact match is observed between the values reported by the solver and the values reported in “Comparison”.

2nd order linear - Simply supported beam

Problem definition

Determine the mid-span deflection and moment of the simply supported beam under transverse and tensile axial load.



Assumptions

Shear deformations are excluded. Results are independent of cross section area; therefore any reasonable value can be used. The number of internal nodes varies from 0-9.

Key results

The theoretical value for deflection and moment are calculated as:

$$Y_{max} = -0.115 = \frac{5wL^4}{384EI} \times \frac{\frac{1}{\cosh U} - 1 + \frac{U^2}{2}}{\frac{5}{24}U^4}$$

$$M_{max} = -0.987 = \frac{wL^2}{8} \times \frac{2(\cosh U - 1)}{U^2 \cosh U}$$

Where U is a variable calculated:

$$U = 5 = \sqrt{\frac{PL^2}{4EI}}$$

No. internal nodes	Solver deflection	Solver deflection %	Solver moment	Solver moment %
1	-0.116	0.734 %	-0.901	8.631 %
3	-0.115	0.023 %	-0.984	0.266 %
5	-0.115	0.004 %	-0.986	0.042 %
7	-0.115	0.001 %	-0.986	0.013 %

No. internal nodes	Solver deflection	Solver deflection %	Solver moment	Solver moment %
9	-0.115	0 %	-0.986	0.005 %

Conclusion

As the element is subdivided the result converges to the correct theoretical value.

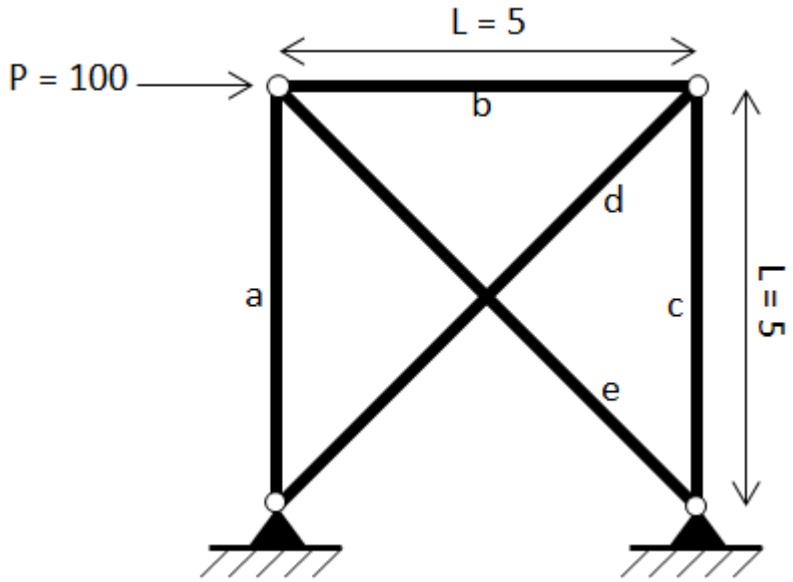
Reference

Timoshenko. S. 1956. *Strength of Materials, Part II, Advanced Theory and Problems*. 3rd Edition. D. Van Nostrand Co., Inc. New York, NY.

2nd order nonlinear - Tension only cross brace

Problem definition

Calculate the axial forces of the elements a-e shown in the 5x5 pin jointed plane frame shown below. Elements d and e can resist tensile forces only.



Assumptions

All elements are constant and equal EA. A smaller value of EA will increase the influence of second order effects, whereas a larger value will decrease the influence.

Key results

Under the applied loading element *e* becomes inactive. The theoretical formulas presented below are obtained using basic statics. Note that a positive value indicates tension. These results assume no 2nd order effects; this

requires the value of *EA* to be sufficiently large to make the 2nd order effect negligible.

Result	Theoretical formula	Theoretical value	Solver value	% Error
a	0	0	0	0
b	-P	-100	-100	0
c	-P	-100	-100	0
d	$P\sqrt{2}$	141.42	141.42	0
e	0	0	0	0

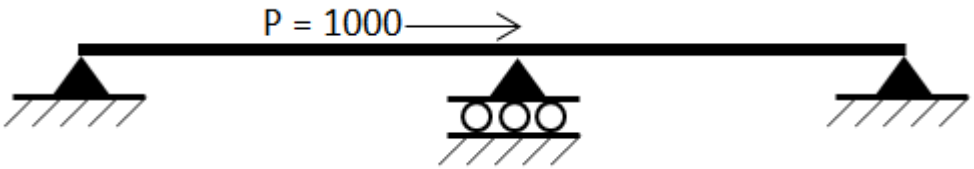
Conclusion

An exact match is observed between the values reported by the solver and the values predicted using statics. A 1st order nonlinear analysis can be used, with any section sizes, to confirm this result without second order effects.

2nd order nonlinear - Compression only element

Problem definition

Calculate the reaction forces for the compression only structure shown below.



Assumptions

All elements are constant and equal *EA*, and can resist only compressive forces

Key results

Under the applied loading the element on the left becomes inactive, therefore all applied loading is resisted by the support on the right.

Result	Theoretical formula	Theoretical value	Solver value
LHS Reaction	0	0	0
RHS Reaction	-P	-1000	-1000

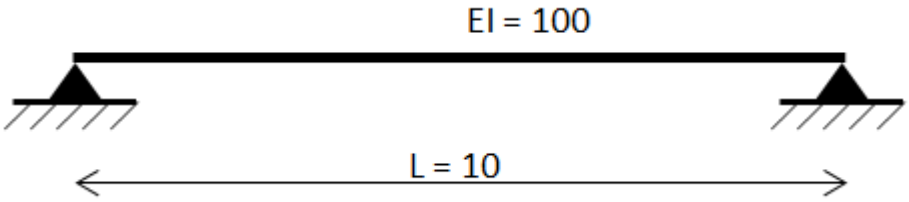
Conclusion

An exact match is observed between the values reported by the solver and the theoretical values.

1st order modal - Simply supported beam

Problem definition

Determine the fundamental frequency of a 10 long simply supported beam with uniform *EI* and mass per unit length equal to 1.0.



Assumptions

Shear deformations are excluded. The number of internal nodes varies from 0-5. Consistent mass is assumed.

Key results

The theoretical value for the fundamental frequency is calculated as:

$$\omega = 0.9870 = \sqrt{\left(\frac{\pi}{10}\right)^4 \frac{100}{1}} = \sqrt{\left(\frac{\pi}{L}\right)^4 \frac{EI}{m/L}}$$

Where *m* is the total mass of the beam.

No. internal nodes	Solver value	% error
0	1.0955	10.995 %
1	0.9909	0.395 %
2	0.9878	0.081 %
3	0.9872	0.026 %
4	0.9871	0.011 %
5	0.9870	0.005 %

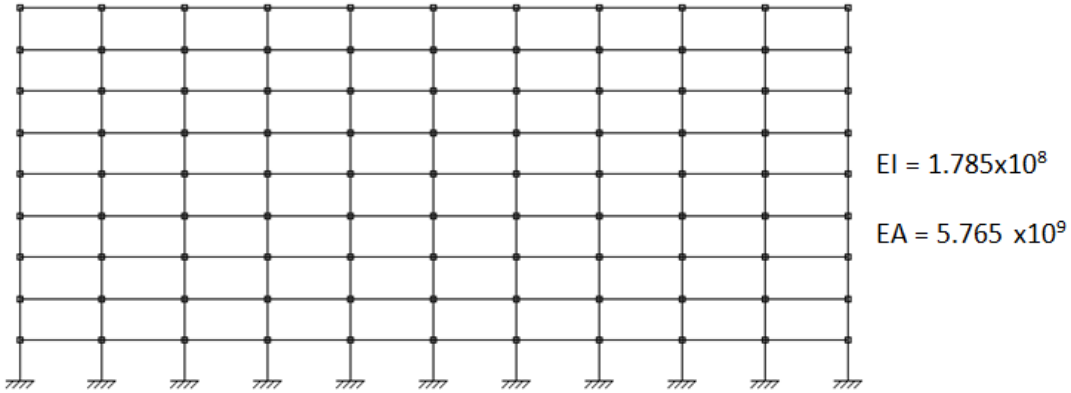
Conclusion

As the element is subdivided the result converges to the correct theoretical value.

1st order modal - Bathe and Wilson eignenvalue problem

Problem definition

A 2D plane frame structure has 10 equal bays each measuring 6.096m wide and 9 stories 3.048m tall. The column bases are fully fixed. All beams and columns are the same section, which have a constant mass/unit length equal to 1.438×10^5. Calculate the first three natural frequencies (in Hz) of the structure under self-weight.



Assumptions

Shear deformations are excluded. Each beam/column is represented by one finite element. Consistent mass is assumed.

Key results

The results for this problem are compared with those published by Bathe and Wilson and against an independent analysis package.

Mode	Bathe and Wilson	Comparison	Solver value
1	0.122	0.122	0.122
2	0.374	0.374	0.375
3	0.648	0.648	0.652

Conclusion

The results show a good comparison with the original published results and against the other analysis packages.

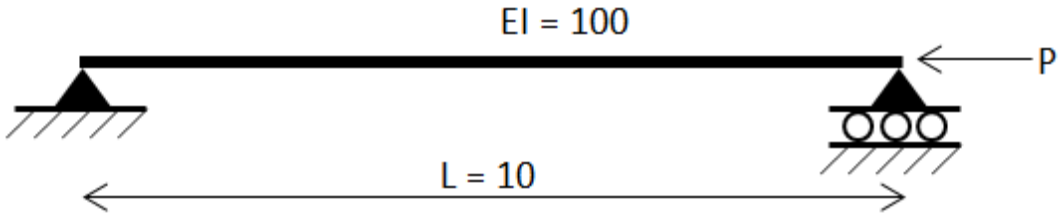
References

Bathe, K.J. and E.L. Wilson. 1972. *Large Eigen Values in Dynamic Analysis.* Journal of the Engineering Mechanics Division. ASCE Vol. 98, No. EM6. Proc. Paper 9433. December.

2nd order buckling - Euler strut buckling

Problem definition

A 10 long simply supported beam is subjected to an axial tip load of P.



Assumptions

Shear deformations are excluded. The number of internal nodes varies from 0-5.

Key results

The theoretical value for the first buckling mode is calculated using the Euler strut buckling formula:

$$\lambda = 9.869 = \frac{\pi^2 EI}{L^2}$$

With P = -1.0 the following buckling factors are obtained

No. internal nodes	Solver value	% error
0	12.000	21.59 %
1	9.944	0.75 %
2	9.885	0.16 %
3	9.875	0.05 %
4	9.872	0.02 %
5	9.871	0.01 %

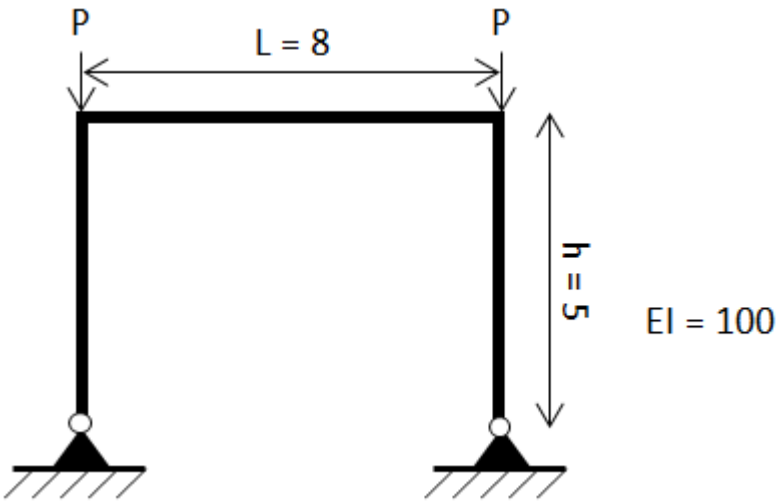
Conclusion

As the element is subdivided the result converges to the correct theoretical value.

2nd order buckling - Plane frame

Problem definition

Calculate the buckling factor of the moment frame shown below.



Assumptions

All elements are constant and equal *EI*. Axial deformations are ignored; to achieve this the cross section area is set to 1000. The number of elements per member is varied between 0 and 5.

Key results

The theoretical buckling load is calculated by

$$P_{cr} = 6.242 = \frac{(kL)^2 EI}{h^2}$$

where

$$kL\tan(kL) = 1.249 = \frac{6h}{L}$$

Which can be solved using Newtons method and five iterations

$$kL_{n+1} = kL_n - \frac{f(kL_n)}{f'(kL_n)} = kL_n - \frac{kL_n\tan(kL_n) - \frac{6h}{L}}{kL_n\sec^2(kL_n) + \tan(kL_n)}$$

No. internal nodes/ members	Solver value	% error
0	6.243	0.17 %
1	6.243	0.01 %
2	6.242	0.00 %
3	6.242	0.00 %

No. internal nodes/ members	Solver value	% error
4	6.242	0.00 %
5	6.242	0.00 %

Conclusion

A good match is shown between the solver and theory. The discrepancy decreases as the level of discretization is increased.

References

Timoshenko, S. and J. M. Gere. 1961. *Theory of Elastic Stability*. 2nd Edition. McGraw-Hill Book Company.

4.2 Solver models

A separate solver model is created for each analysis type that has been run. These can each be displayed in 2D or 3D by opening a Solver View (page 549).

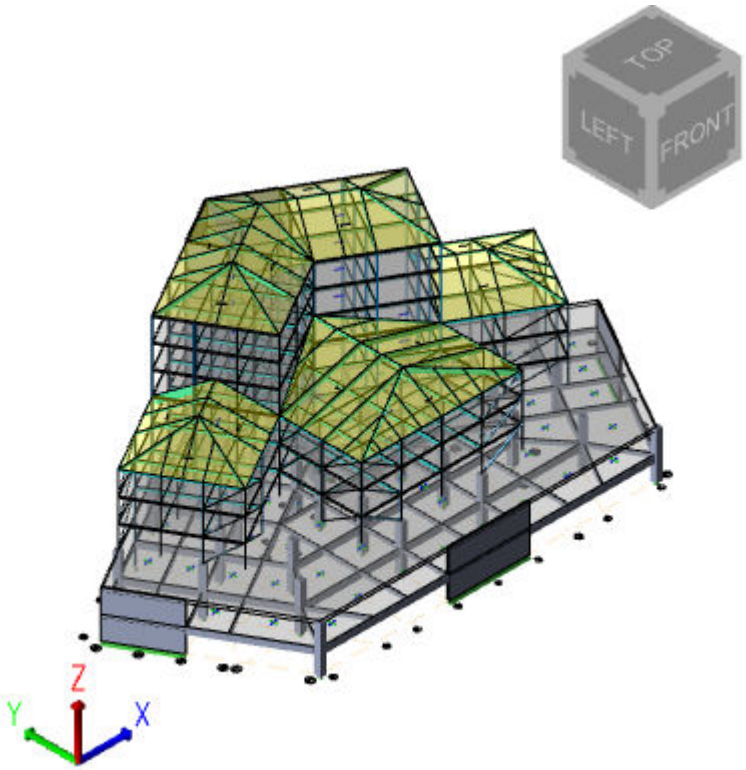
Click the links below to find out more:

- What is a solver model? (page 537)
- Open a Solver View (page 549)
- Solver model types (page 540)
- Solver element (1D) types (page 551)
- Solver element (2D) types (page 553)
- How meshed walls are represented in solver models (page 554)
- How mid-pier walls are represented in solver models (page 559)
- How shear only walls are represented in solver models (page 561)
- How bearing walls are represented in solver models (page 565)

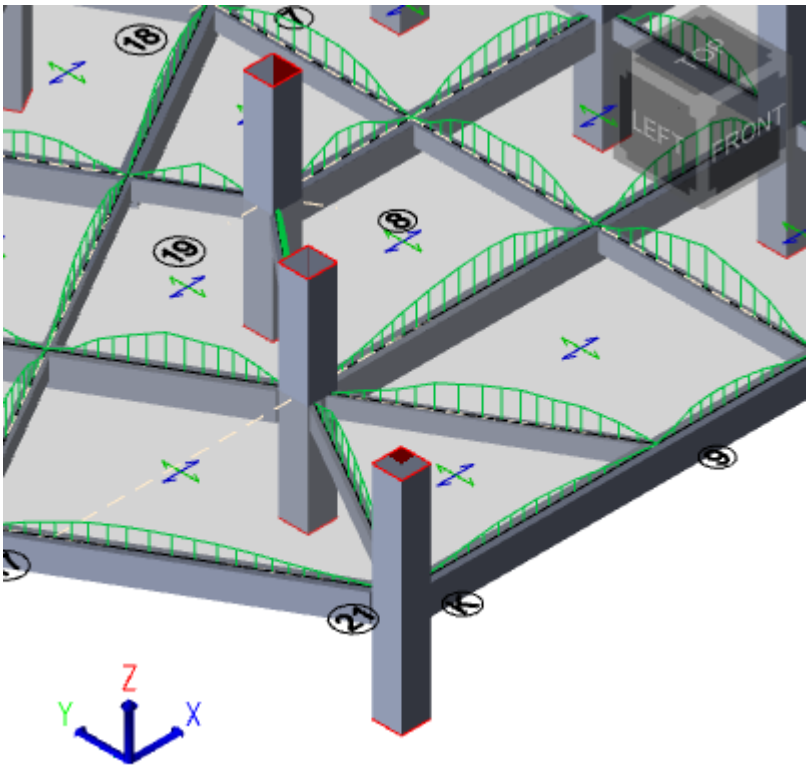
What is a solver model?

When you use Tekla Structural Designer to model, analyze, and design structures, you will become familiar with the following concepts:

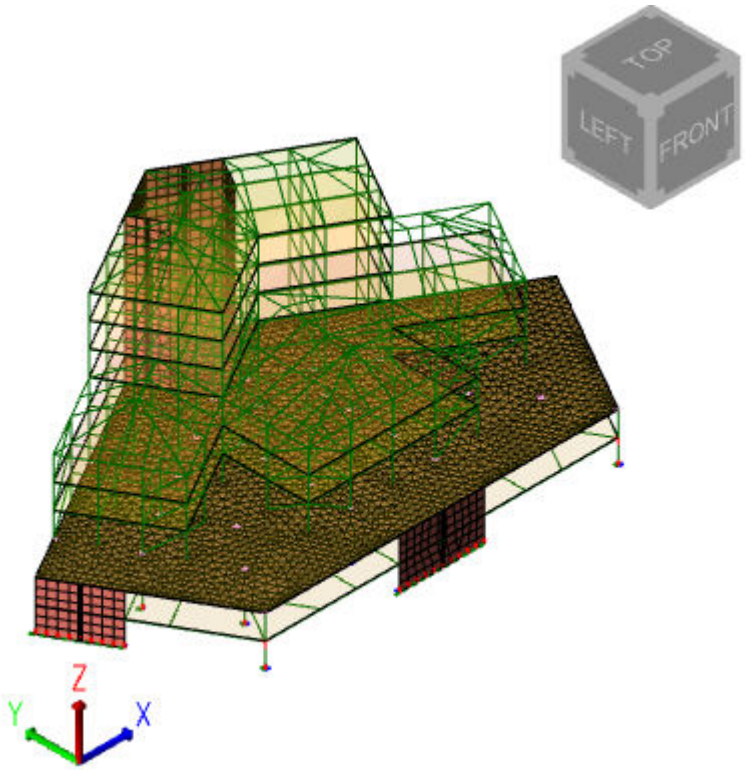
A *structural model* is a 3D model that includes the physical members/objects you create using Tekla Structural Designer, and information related to them. Each member/object in the physical model will exist in the completed structure.



The structural model also contains information about the loadcases and combinations that act on the physical members/objects.



A *solver model* is a 3D model that is created from the structural model. It is used for analyzing structural behavior, and for design.



When you create a solver model, Tekla Structural Designer generates the following analysis objects and includes them in the solver model:

- Solver nodes
- Support conditions for nodes
- 1D solver elements
- 2D solver elements

See also

Open a Solver View (page 549)

Solver model types (page 540)

Solver model types

You can review the different solver models by opening a Solver View (page 549) and then choosing the solver model type required from the **Solver** ribbon, (or from the right-click menu).

Working Solver Model

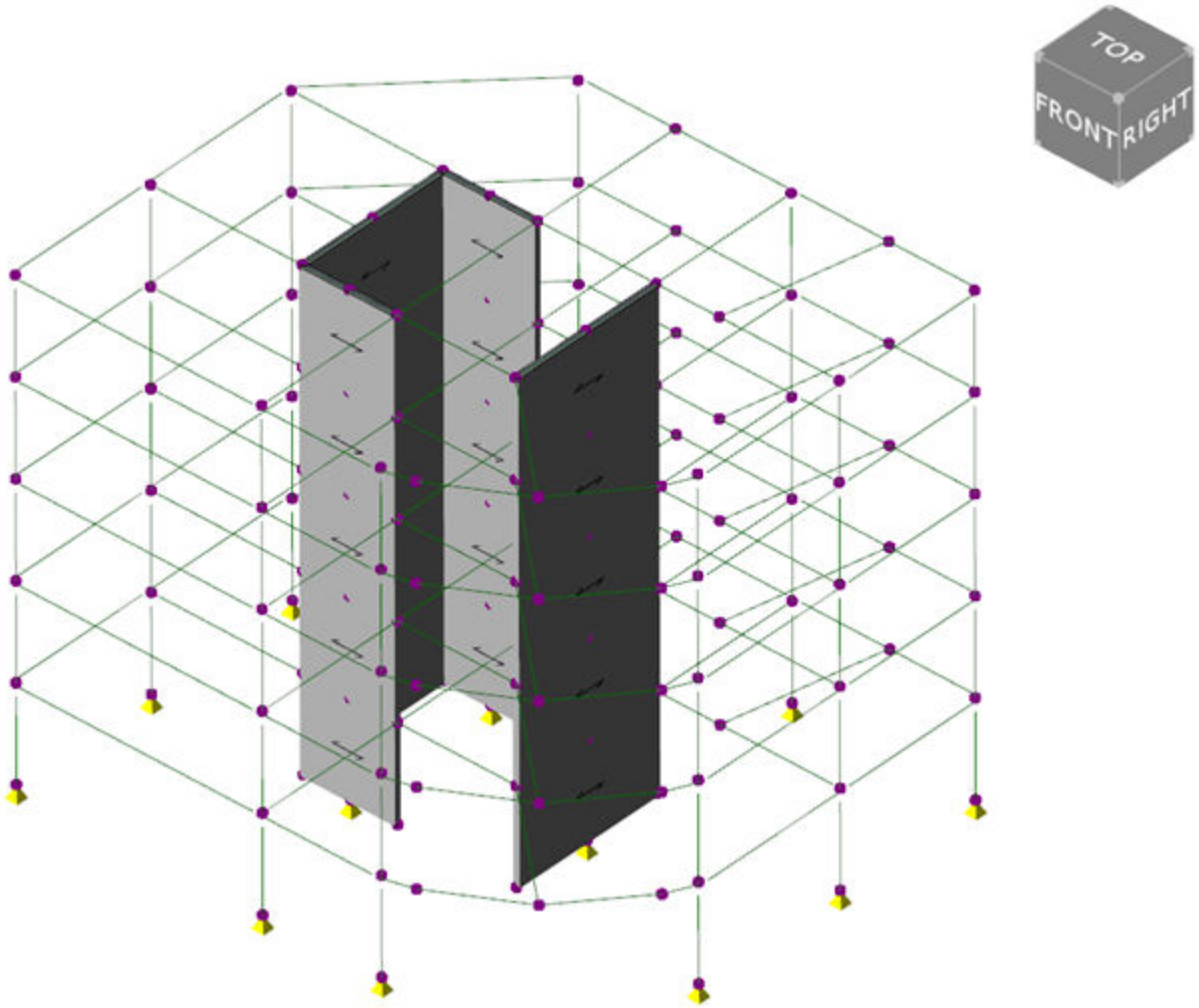
The **Working Solver Model** shows the model in its form prior to any analysis.

Although 1D elements, solver nodes, and diaphragms are displayed, 2D elements are not (even when you have chosen to mesh slabs/walls). This is because 2D elements are only formed at the point of analysis.

Solver Model used for 1st Order Linear and 2nd Order Linear

The solver model used for 1st order and 2nd order linear analysis potentially features a mix of 1D analysis elements, FE meshes and diaphragms as follows:

- beams and columns are modelled as 1D analysis elements
- walls are either mid-pier analysis elements, or FE meshes
- slabs (optionally) form rigid, or semi rigid, diaphragms in floors
- 1-way slabs have their loads decomposed on to supporting members at a preliminary stage of the analysis.
- 2-way slabs are (typically) not meshed, in which case they will also have their loads decomposed on to supporting members at a preliminary stage of the analysis.
- 2-way slabs (optionally) can be meshed, this is only recommended for special cases, typically where slabs participate in the lateral load stability system, e.g. transfer slabs
- supports are user defined



Any FE meshes in the solver model are formed using the mesh parameters in place for the most recent run of the chosen analysis (i.e. 1st order linear or 2nd order linear).

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

2-way slabs meshed

Optionally you can choose to mesh all 2-way slabs - making a fully meshed model (both walls and floors) possible.

This is generally not recommended as it will inevitably increase the model size, (and potentially the time to solve for large models), although it might be considered that a fully meshed model behaves more "correctly" where slabs are considered to be part of the lateral load resisting system of the structure.

It is more likely that you will choose to mesh specific floor levels only (e.g. transfer levels), keeping other levels unmeshed.

Solver Model used for 1st Order Non Linear and 2nd Order Non Linear

These solver models are basically the same as those used for 1st order analysis with the exception that they will also feature non linear elements.

Any FE meshes in these solver models are formed using the mesh parameters in place for the most recent run of the chosen analysis (i.e. 1st order non-linear or 2nd order non-linear).

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis types after changes to either geometry or loading will prevent you from displaying results for these models.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 1st Order Modal

Any FE meshes in this 3D solver model are formed using the mesh parameters in place for the most recent run of 1st order modal analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 2nd Order Buckling

Any FE meshes in this 3D solver model are formed using the mesh parameters in place for the most recent run of 2nd order buckling analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

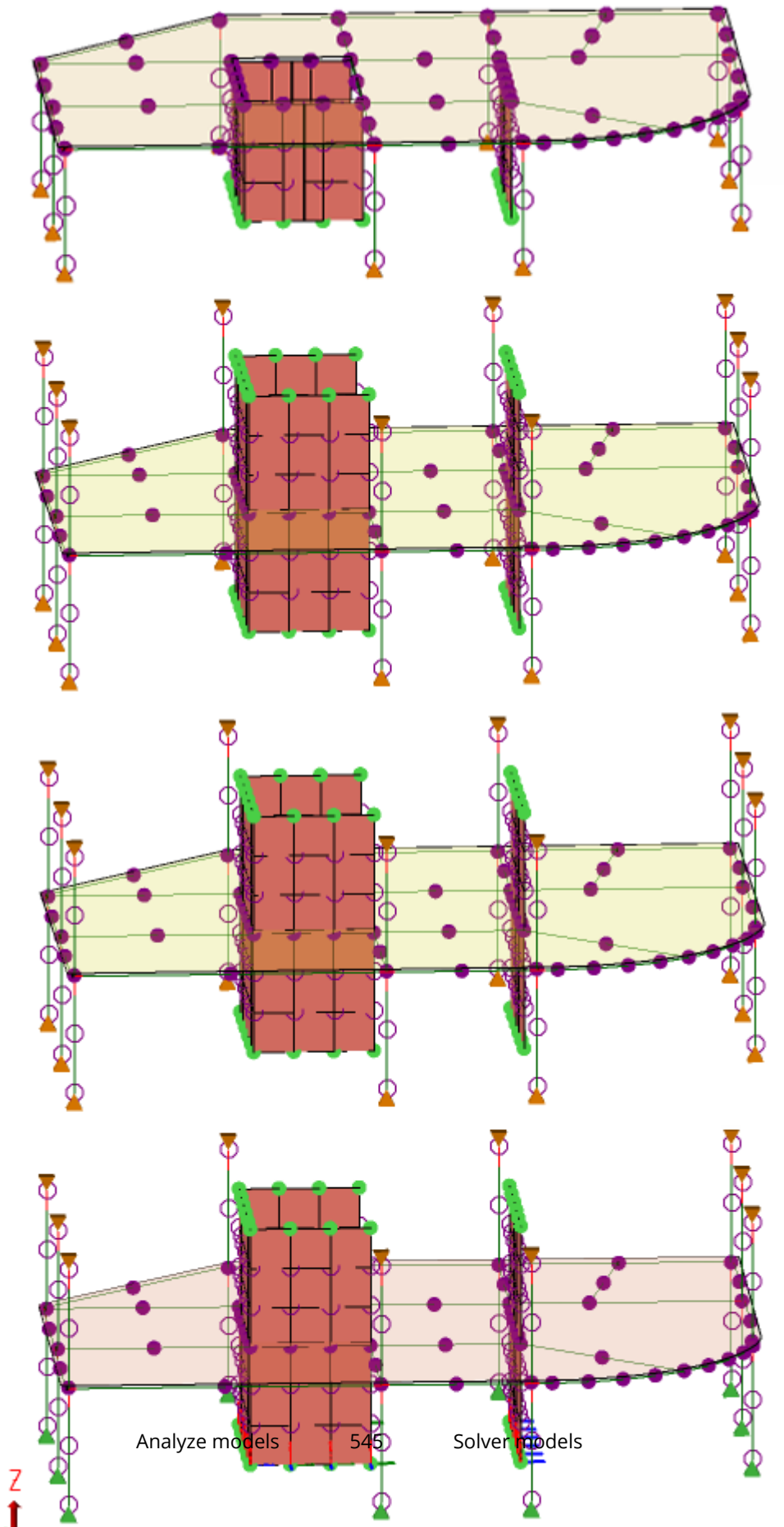
Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for Grillage Chasedown

In grillage chasedown a 3D sub model is formed for each floor including those columns and walls that connected to the floor.

The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

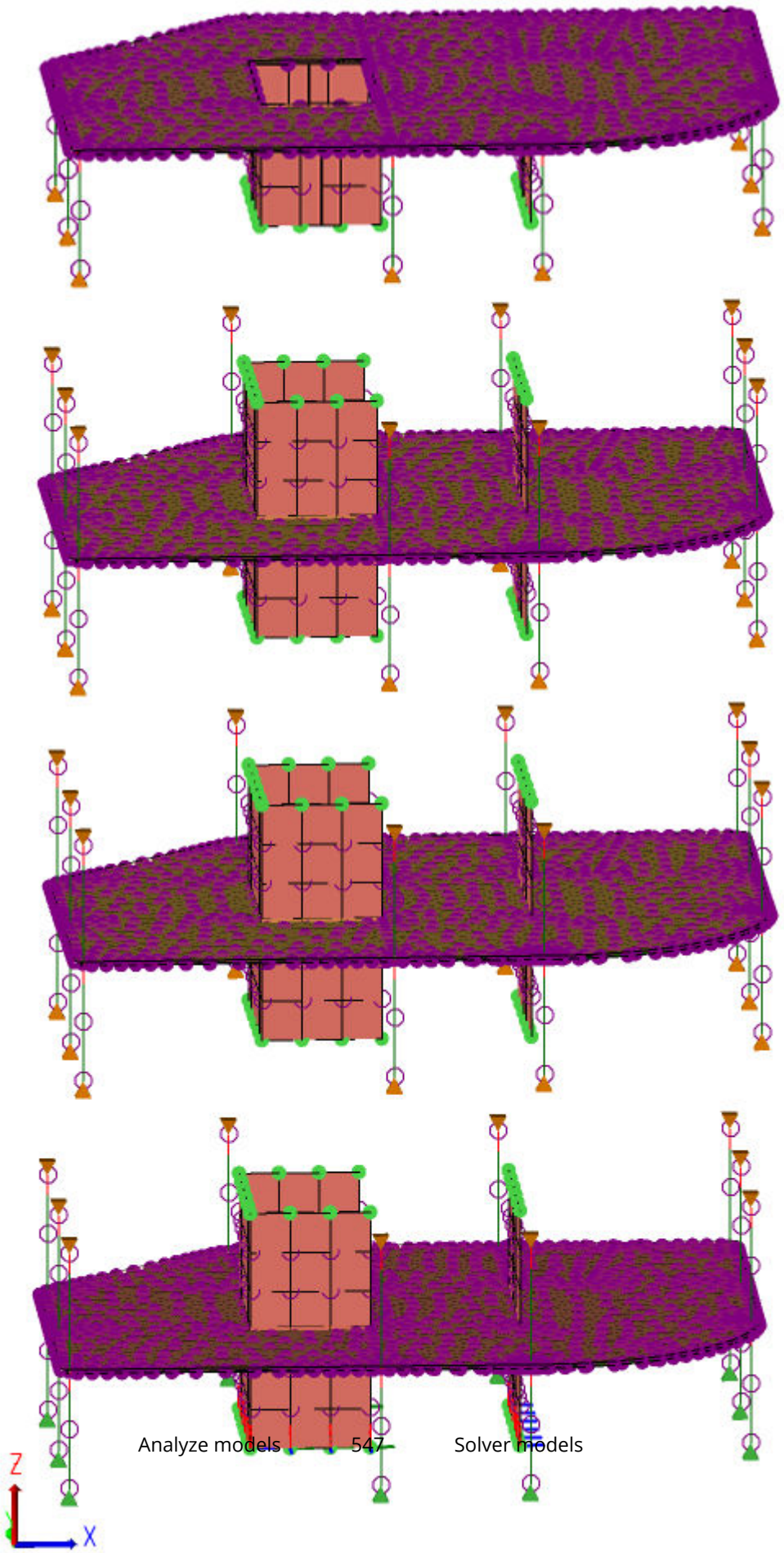

Z
X

they have been set as meshed for 3D Analysis. For all other slabs **load decomposition** is carried out prior to the analysis.

Solver Model used for FE Chasedown

FE chasedown is similar to grillage chasedown, with 3D sub models being formed at each level; the one difference being that in the FE chasedown the two-way slabs are always meshed.

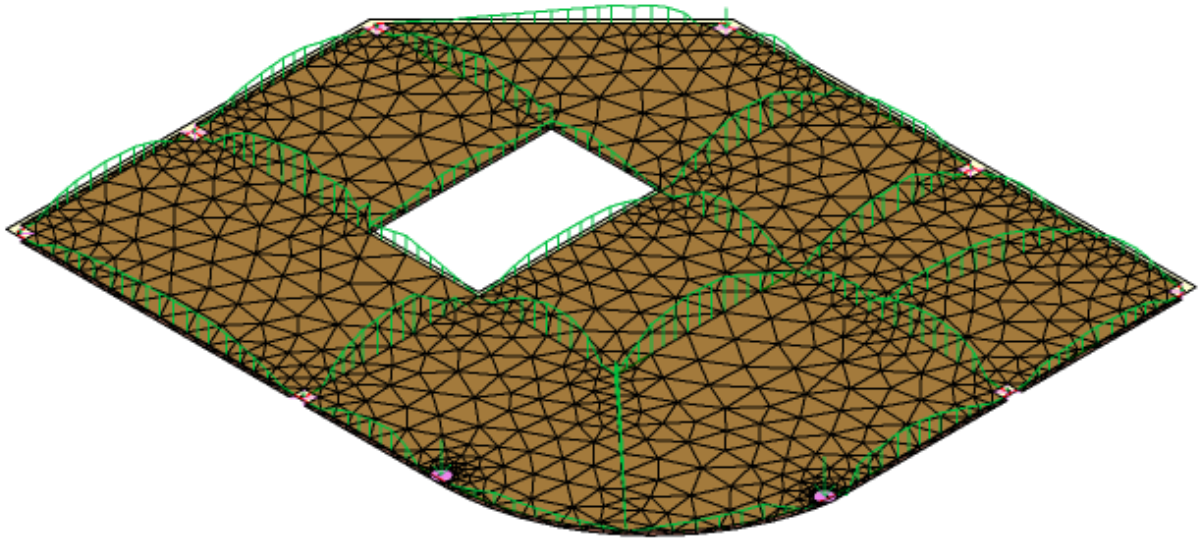
The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.


Z
X

Solver Model used for Load Decomposition

At each level, (provided you have not checked the Mesh 2-way Slabs in 3D Analysis option), a solver model is created solely for the purpose of decomposing slab and panel loads back on to the supporting members. As these load decomposition models are only used during the pre-analysis stage, by default they are not retained.

However, if you want to examine the load decomposition model used at a given level (page 634) this is possible by editing the level properties prior to analysis and selecting **keep solver model**.



Solver Model used for Load Decomposition

The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

Refresh Solver Model

The solver model currently displayed is updated to reflect changes that have occurred in the working solver model since the last analysis. The 2D element mesh is also updated to reflect the current mesh parameters. Previous analysis results are also erased for this solver model

See also

Solver element (1D) types (page 551)

Solver element (2D) types (page 553)

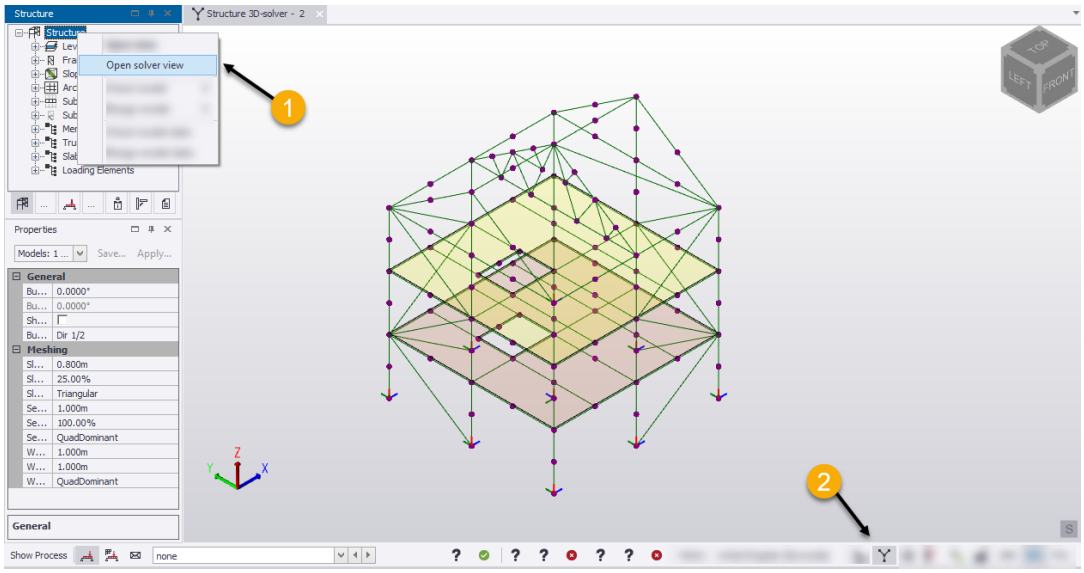
How meshed walls are represented in solver models (page 554)

How mid-pier walls are represented in solver models (page 559)

How shear only walls are represented in solver models (page 561)
How bearing walls are represented in solver models (page 565)

Open a Solver View

You can open a new solver view from the **Structure tree context menus (1)**, or you can switch an existing view to a solver view by clicking **Solver View (2)** in the **Status bar**.



Open a 3D Solver View

1. Open the **Structure** tab in the **Project Workspace**.
2. In the **Structure** tree, right-click **Structure**.
3. From the context menu, select **Open Solver View**.
   A structure 3D-solver view is displayed.

You can now select the solver model type (see below).

Open a 2D Solver View

1. Open the **Structure** tab in the **Project Workspace**.
2. In the **Structure** tree, right-click on the required **Level**, **Frame** or **Slope**.
3. From the context menu, select **Open Solver View**.
   A 2D-solver view of the level, frame. or slope is displayed.

You can now select the solver model type (see below).

Change an existing view to a Solver View

1. Open an appropriate 2D or 3D view.
2. To change the view type of the view, do one of the following:
   - Right-click the view tab and in the context menu, select  **Solver view**.
   - In the **Status bar** at the bottom of the window, click  **Solver View**.

You can now select the solver model type (see below).

Select the solver model type

1. In the Method list on the **Solver** ribbon, select the required solver model type (page 540). The solver model that was created the last time analysis was run is displayed.

   Select **Refresh Solver Model** if changes have been made and you want to see the model that would be created if analysis were re-run now.
2. If a staged construction method has been selected, select the construction or design stage required from the Stage list.
3. To see properties that have been assigned; select solver nodes, solver elements (1D) or solver elements (2D) as required. Properties for the selected objects are then displayed in the **Properties** window.

NOTE The properties displayed are the ones used the last time the analysis was run, after the application of any modification factors. (If you change modification factors you need to re-run the analysis in order to see the changed effects both in terms of analysis results and in the properties reported in the solver model view.)

NOTE If viewing a chasedown solver model in a *Structure* solver view, the section/material properties for 2D solver elements are not displayed. In order to see these properties you would need to open an appropriate sub-model solver view instead.

NOTE When **Only Existing** is selected on the **Solver** ribbon, only models created by last analysis runs are available for selection. When unselected you can also view models that would be created for some analysis types that have not yet been run.

See also

Solver model types (page 540)

Solver element (1D) types (page 551)

Solver element (2D) types (page 553)

Solver element (1D) types

Eight different 1D solver element Types are available in Tekla Structural Designer as follows:

Beam

An element in any material, with user defined area and inertia properties, and user-definable end releases - used in **all solver models** for:

- Columns (any material)
- Beams (any material)
- Truss top, bottom and side members (any material)
- Mid-pier concrete wall: wall-beam, and wall-column elements
- Bearing wall: wall-beam elements
- Analysis Elements (any material) with element type: Beam

Truss

An element in any material, with user-defined cross sectional area, and pinned ends (releases not being editable) - used in **all solver models** for:

- Braces (any material) that have not been set as tension or compression only
- Truss internal members (any material) that have not been set as tension or compression only
- Bearing wall: wall-column elements
- Analysis Element (any material) with element type: Truss

Truss 1D solver elements are also used in **linear solver models** only for:

- Braces (any material) that have been set as tension or compression only
- Truss internal members (any material) that have been set as tension or compression only

Tension only

A pin ended member in any material, with user-defined cross sectional area, that can only transfer tension. This is a non-linear element which requires non-linear analysis - hence used in **non-linear solver models only** for:.

- Braces (any material) that have been set as tension only
- Truss internal members (any material) that have been set as tension only
- Analysis Element (any material) with element type: Tension only

Compression only

A pin ended member in any material, with user-defined cross sectional area, that can only transfer compression. This is a non-linear element which requires non-linear analysis -hence used in **non-linear solver models only** for:

- Braces (any material) that have been set as compression only
- Truss internal members (any material) that have been set as compression only
- Analysis Element (any material) with element type: Compression only

Linear axial spring

An element that deflects linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness. These are specified as:

- Analysis Element (any material) with element type: Linear axial spring

Linear torsional spring

An element that rotates linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness. These are specified as:

- Analysis Element (any material) with element type: Linear torsional spring

Non-linear axial spring

An element that deflects non-linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness. These are used in **non-linear solver models only** for:

- Analysis Element (any material) with element type: Non-linear axial spring

Non-linear torsional spring

An element that rotates non-linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness. These are used in **non-linear solver models only** for:

- Analysis Element (any material) with element type: Non-linear torsional spring

Solver element (2D) types

Three 2D solver element types are available and applied in Tekla Structural Designer as follows:

Shell

Shell elements are used for the following slabs and decks when meshed, provided they are two way spanning and their diaphragm option is not set to Semi-rigid:

- • Flat slab,
  - Slab on beams,
  - Steel deck,
  - Timber deck,
  - General deck,
  - Foundation mat

SemiRigid

SemiRigid elements are used for the following slabs and decks when meshed, if their diaphragm option is set to Semi-rigid:

- • Flat slab,
  - Slab on beams,
  - Steel deck,
  - Timber deck,
  - General deck,
  - Foundation mat

Wall

Wall elements are used for walls when the Wall Type is set to Meshed Shear Wall, i.e.

- • Concrete Meshed Shear Wall,

- General Meshed Shear Wall

How meshed walls are represented in solver models

Meshed walls have isotropic stiffness properties and resist loads in all directions.

Wall beam elements

Wall beam elements are inserted into walls primarily to collect slab mesh nodes and line elements. For meshed walls, they are generated along the top of the wall and also at intermediate levels where an object (e.g. slab, beam, truss) is physically connected. A wall beam element is also generated along the bottom edge of the wall if Generate Support is not selected. The wall beam element has only a small minor axis inertia (100cm^4 in metric units, 2.4in^4 in US Customary units) to prevent it from adding undue lateral stiffness, particularly above wall openings.

Sloping wall beam elements can be generated by sloping top or bottom edges or connected sloping slabs.

Where horizontal wall beam elements are required, they are generated across the entire width of the wall at that level.

Wall beam elements can also be created where certain properties, (e.g. thickness), differ across panel boundaries.

2D solver elements

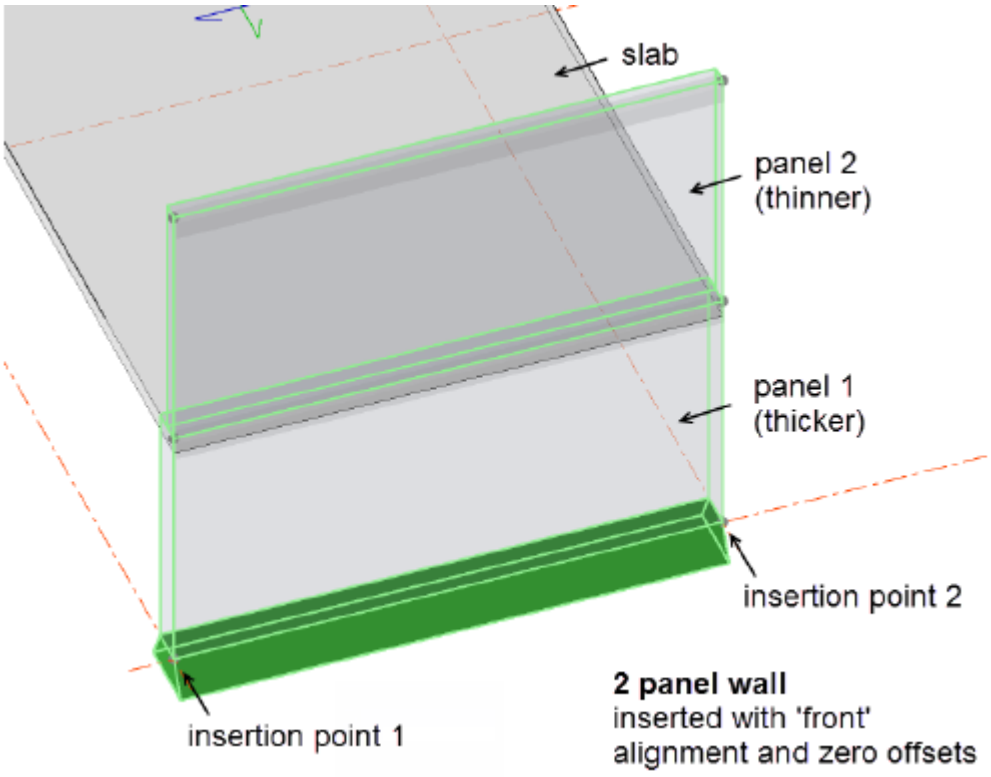
For meshed walls the type of 2D solver element used will depend on whether the wall mesh type is set to Quad only, Tri only, or, Quad dominant.

Modification Factors

Different modification factors applied to meshed walls in the analysis depending on the Material type that has been applied.

Meshed concrete wall example

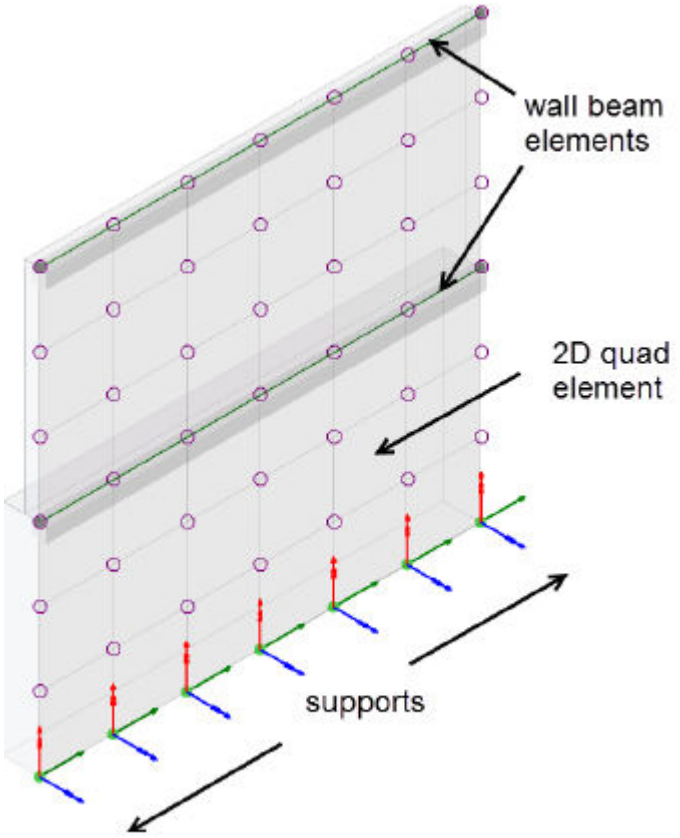
Consider the following two stack wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



1D and 2D solver elements for each wall panel are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified.

Quad only

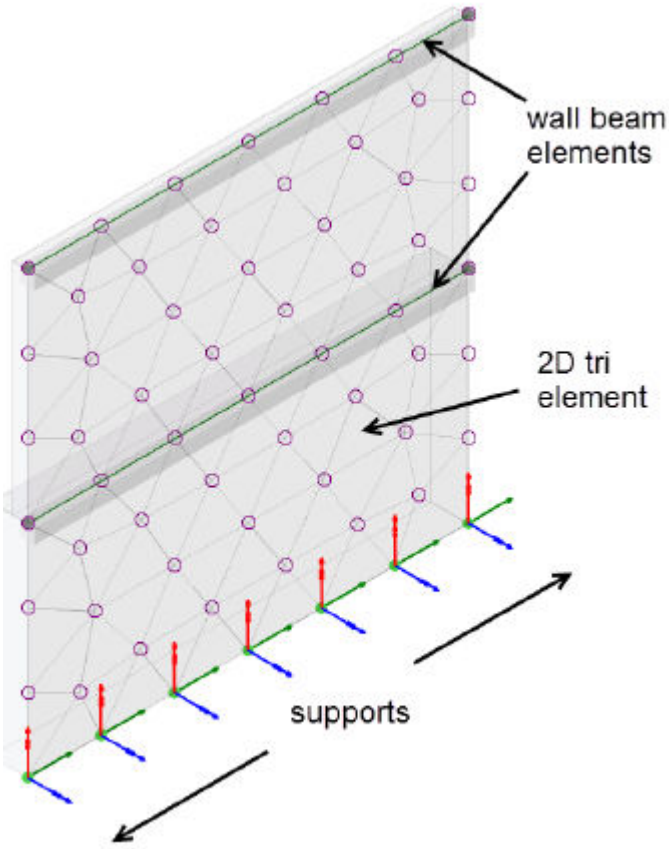
In this two stack example, when the Wall Mesh Type is set to Quad only, solver elements are formed as shown below:



NOTE To see solver elements, solver nodes and 2D elements: open a Solver View, and then in Scene Content check 1D Elements> Geometry, 2D Elements> Geometry and Solver Nodes> Geometry.

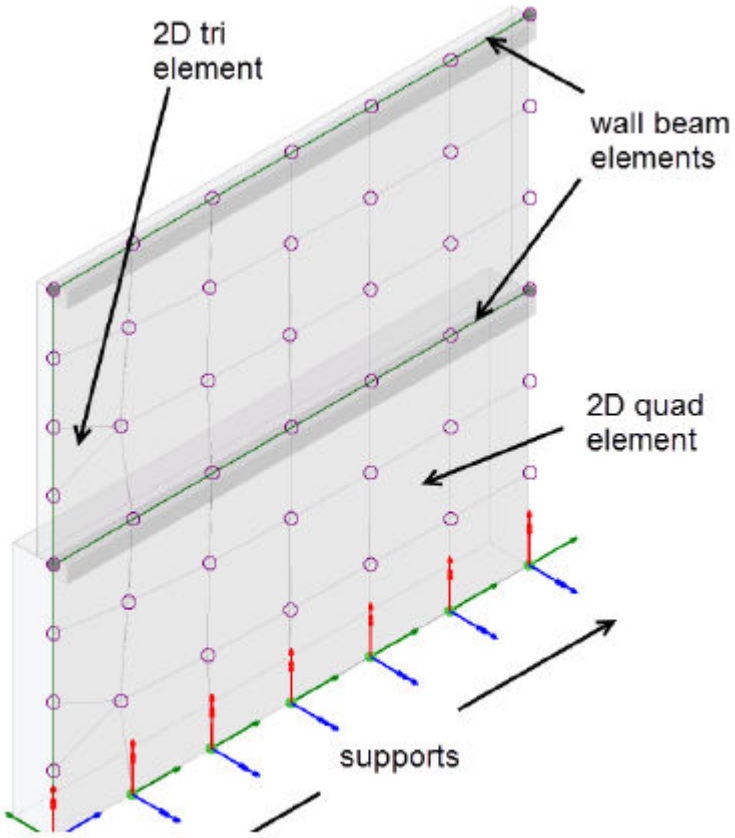
Tri only

In this two stack example, when the Wall Mesh Type is set to Tri only, solver elements are formed as shown below:



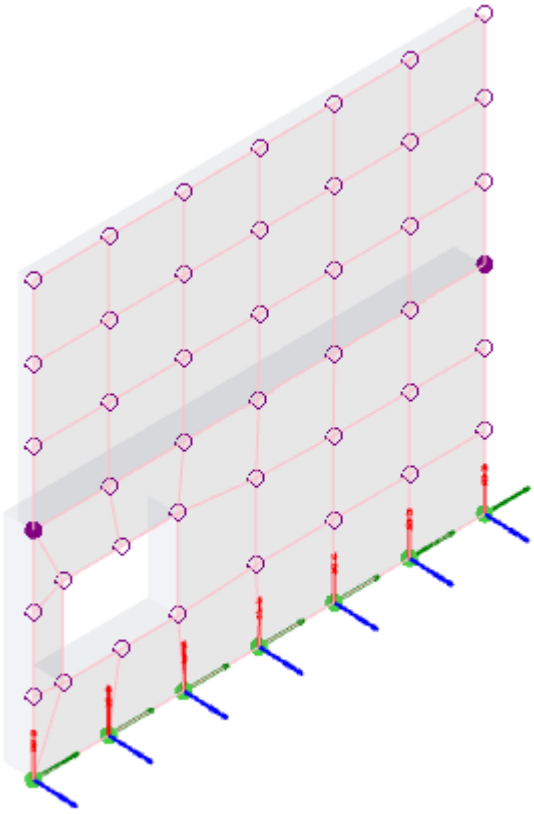
Quad dominant

In this two stack example, when the Wall Mesh Type is set to Quad dominant, solver elements are formed as shown below:



Wall openings

When wall openings are introduced, the mesh will adjust to form around the openings.



See also: Meshed wall openings analysis model (page 641)

See also

FE meshed walls (page 635)

Meshed wall openings analysis model (page 641)

How mid-pier walls are represented in solver models

NOTE There are no advantages to be gained from using mid-pier walls in place of meshed walls. Mid-pier walls are only retained in Tekla Structural Designer for continuity reasons, (so that old models containing them can still be opened). For new models we recommend meshed shear walls are used instead.

Wall beam and wall column elements

Wall beam elements are inserted into walls primarily to collect slab mesh nodes and line elements. For mid-pier walls, they are generated along the top and bottom edges of the wall and also at intermediate levels where an object (e.g. slab, beam, truss) is physically connected.

NOTE Only horizontal wall beam elements can be generated in mid-pier walls - sloping wall beam elements cannot be generated - this will be indicated by an error in validation.

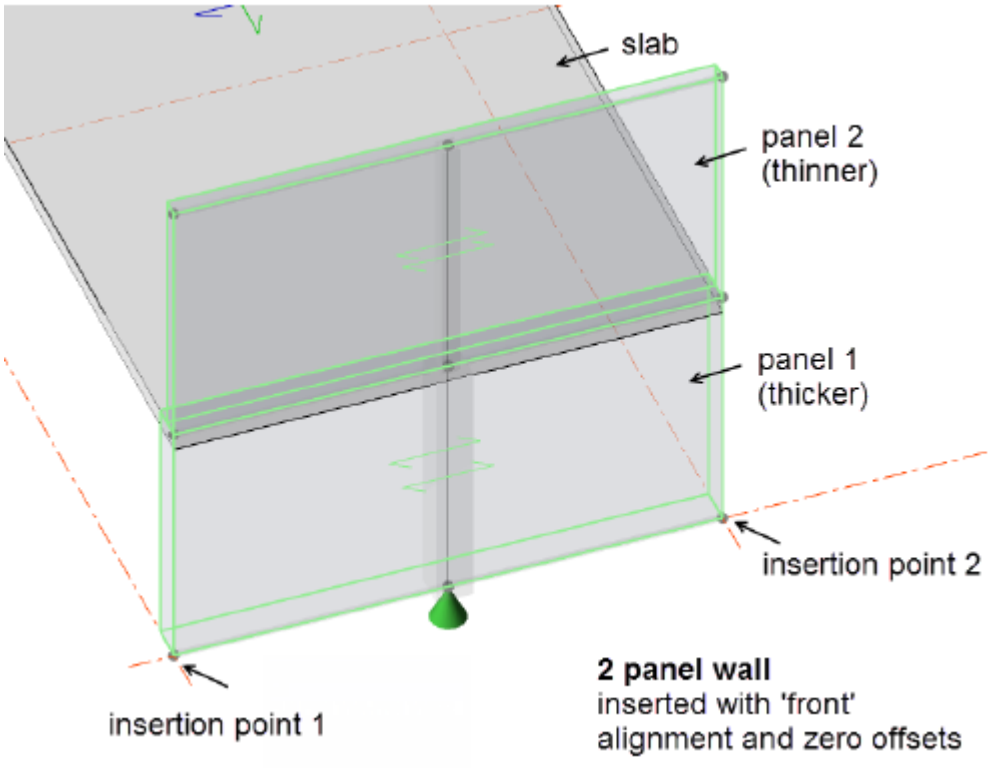
Each mid-pier wall object also has a single vertical wall column element in the middle of the wall, from the top to the bottom level.

Modification Factors

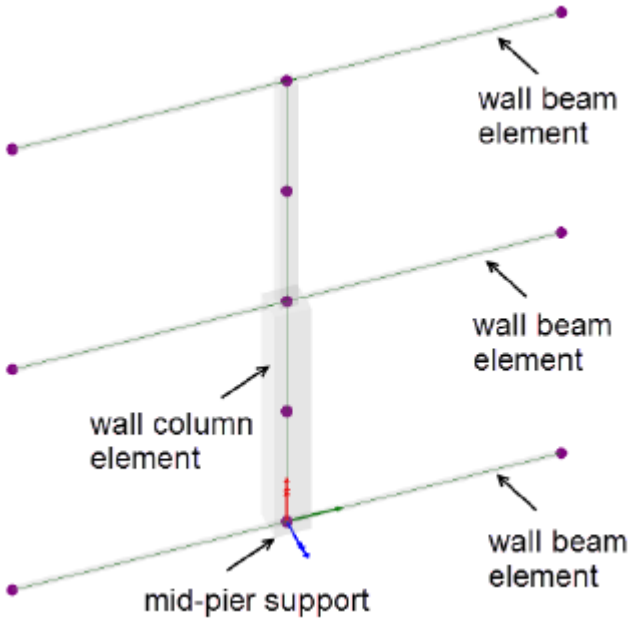
The modification factors applied to concrete walls in the analysis depend on whether they have been specified as meshed or mid-pier.

Mid-pier wall example

Consider the following two stack mid-pier wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



The wall beam and wall column elements are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified, so for this example, the elements are formed as shown below:



NOTE To see wall beam elements, wall column elements and solver nodes: open a Solver View, and then in Scene Content select 1 D Elements> Geometry and Solver Nodes> Geometry.

How shear only walls are represented in solver models

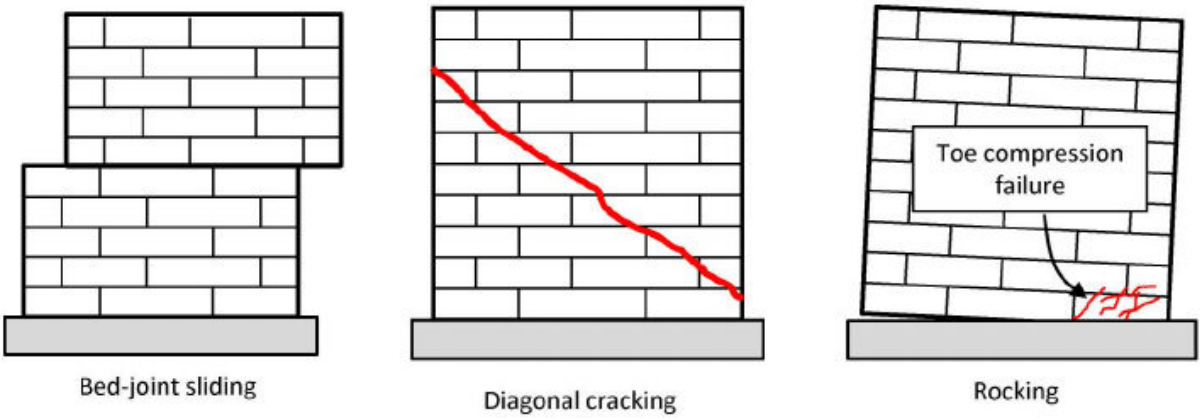
Shear only walls resist in-plane shear only and have no out of plane stiffness or load bearing resistance.

Background

The ‘shear only wall’ is configured to allow, as the name implies, shear forces only to be resisted. These arise from frame action in the lateral load resisting system when subject to lateral loads. The behaviour is typical of unreinforced masonry walls built into a steel or concrete frame. Under lateral loading the top of the column adjacent to the wall panel bears on a relatively short length of wall, creates compression in some width of masonry in the diagonal and exits the wall panel at the toe of the opposite corner.

A masonry wall panel resists this force couple at each corner primarily as a shear panel. The failure mechanism either follows a stepped pattern through the joints or by shear bond along the bed joints. The compression strut manifests as a failure in diagonal tension across the bed joints in a stepped fashion. Failure can also occur by local crushing in the top left or bottom right

of the wall (or vice versa). These failure modes are depicted in the figure below.



The third failure mode mentioned, crushing in the corners, is very difficult to model and is believed to have a minor influence on overall behaviour. It is thought to be self compensating to some extent because when the masonry begins to crush more length of wall is brought into play.

The diagonal cracking and sliding in the bed joint are the primary effects of the wall acting as a shear panel. This is the only behaviour that 'shear only walls' are able to model.

The compression strut that is also inferred by the diagonal cracking generates push-pull forces in the frame and complementary axial forces in the beams. These are incorporated in the Tekla Structural Designer implementation by the use of special 'Link Elements'.

The oft used and simplest model of a single brace or pair of braces from corner to corner of the wall panel can adequately represent the lateral stiffness of the infilled frame but introduces unwanted axial forces (from gravity loads), particularly in columns. The Tekla Structural Designer implementation is a significant improvement on this simplest model and requires only the determination of the spring stiffness associated with the shear behaviour.

Crisafulli (2007)[1] provides a formula for the stiffness, k_s, of the shear spring as,

k_s	=	$\gamma_s \times A_{ms} \times E_m/d_m \times \cos 2\theta$
Where,		
A_{ms}	=	total area of equivalent strut based on a width of strut of the order of ¼ to ⅓ of the diagonal length of the panel

E_m	=	Elastic modulus of the masonry
d_m	=	diagonal length of the wall panel
θ	=	the angle of the 'strut' with the horizontal

The factor γ_s is the proportion of total stiffness that is assigned to the spring whilst the remainder is provided in the Crisafulli model by a pair of 'masonry struts'. In Tekla Structural Designer γ_s is 1.0 i.e. all of the stiffness is provided by the spring whilst the force effects of the 'masonry struts' are replicated by the special 'Link Element'.

Other formulations for the spring stiffness are likely to exist in the literature.

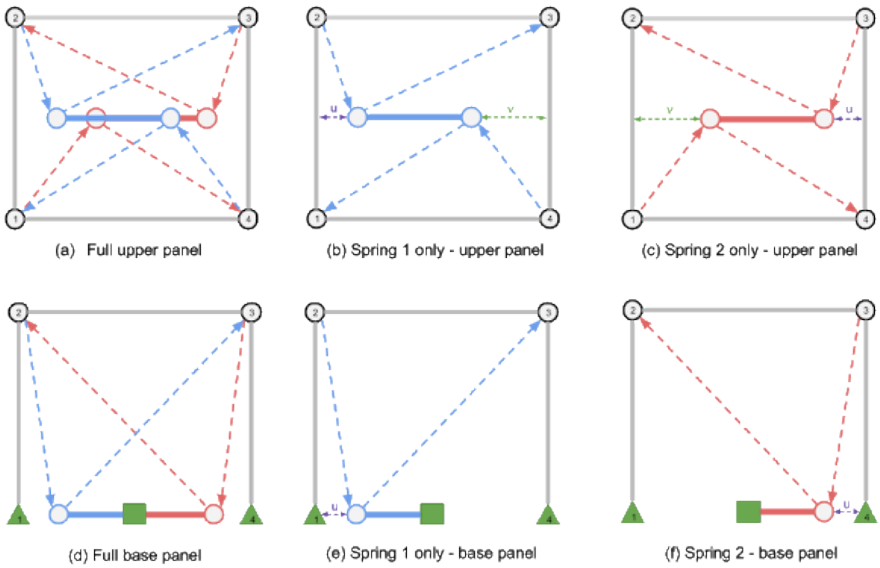
In Tekla Structural Designer, the spring is a either a linear or non-linear uniaxial spring, the former being used when all diaphragms are 'Rigid'.

One of the consequences of the configuration of 'shear only walls' adopted in Tekla Structural Designer is that no loads can be applied out of plane and no members can be connected into the main body of the wall panel. Similarly, the wall panel must be completely surrounded by column and beam members to ensure transfer of lateral loads. For this type of shear wall the head detail is assumed to be such that there is no load transfer from the beam above to the head of the wall.

1 *Crisafulli F. J. and Athol J. C., Proposed macro-model for the analysis of infilled frame structures, Bulletin of the New Zealand Society for Earthquake Engineering, Vol. 40 No. 2 June 2007.*

Solver model in Tekla Structural Designer

Shear only wall panels are modelled using two axial springs between 'panel nodes' connected to 'corner nodes' by link elements. How these are configured differs for interstory panels and the base panel.



Configuration of shear only walls for interstory and base panels. (a) & (d) show the full panel, (b) & (e) show the right to left spring assembly, (c) & (f) show the left to right spring assembly.

NOTE The axial spring and link elements are only shown in the Solver View used for analysis. The above illustrations are not to scale, the actual u and v dimensions being 10mm (25/64 in.) and 20mm (50/64in.)

Interstory panels

For each interstory panel, two springs, each with a pair of nodes are created and connected to the ‘corner’ nodes where the panel connects to the beam-column node. The connection is made using four special ‘Link Elements’. The orientation of the axial spring means there is stiffness only in the plane of the wall, specifically only in the horizontal direction. The Link Elements coordinate systems and their degrees of freedom are configured such that the panel operates in-plane and is stable out-of-plane whilst not generating any untoward moments and forces.

Base panel

Where a base panel exists, a single fixed base is created. Two springs are created at the same level, and connected to the ‘corner’ nodes where the panel connects to the beam-column node using two Link Elements. The orientation of the axial springs means there is stiffness only in the plane of the wall, specifically only in the horizontal direction. The Link Element coordinate system and their degrees of freedom are configured such that the panel operates in-plane and is stable out-of-plane whilst not generating any untoward moments and forces. Only horizontal reaction is produced at the

support and this must be distributed manually along the wall footing if required.

Walls supported on meshed slabs and foundation mats are treated as base panels with the springs and consequent forces applied to a ‘seeded’ node in the mesh.

Self weight

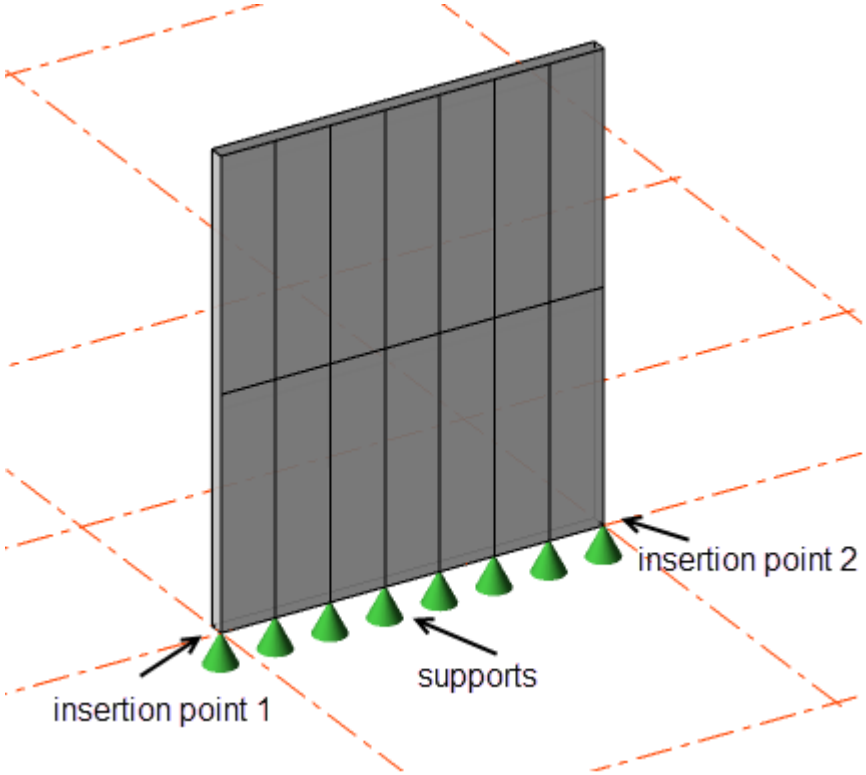
The self weight of each panel is automatically calculated by Tekla Structural Designer and applied to the supporting beam. For a base panel this applied directly to the wall support. For meshed slabs and foundation mats the wall is treated as a base panel and the self weight is applied to a ‘seeded’ node in the mesh.

How bearing walls are represented in solver models

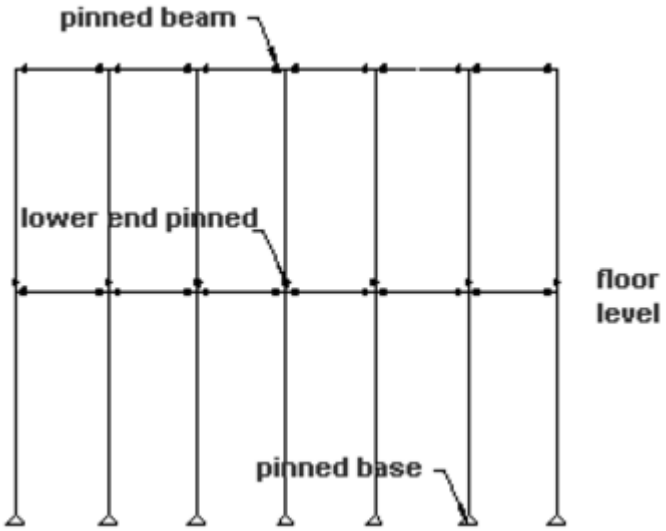
Bearing walls have no in-plane or out-of-plane stiffness and resist vertical load only.

For bearing walls the alignment (Front, Middle, or Back) specified in the wall properties is not structurally significant as it has no effect on the positioning of the solver elements in the solver model.

Consider the two stack bearing wall shown below.

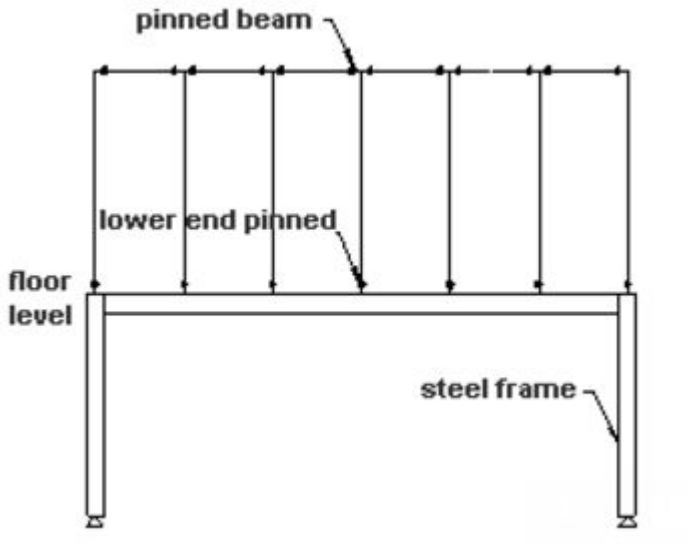


The solver model for this wall is formed using a series of vertical "wall column" and horizontal "wall beam" solver elements. The beams have pinned ends and are placed at the top of the wall spanning between the columns. The next panel above is pinned to the one below and similarly the lower end of a column is pinned to a supporting beam. At the lowest level the column is 'fixed' to a pinned support.

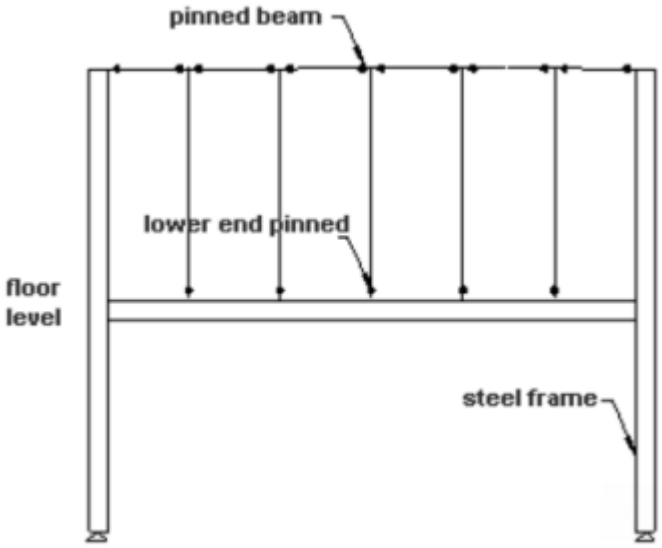


Members supported by the wall either (fortuitously) bear directly on one of the wall columns or on one of the wall beams at the head of the wall. All wall columns and wall beams in an individual bearing wall are given properties automatically by Tekla Structural Designer, based on the width of the bearing wall with which they are associated.

If the bearing wall did not continue to the lowest level, but was instead supported by a transfer beam, then at the lowest level the wall columns would have pinned ends and no supports would be introduced.



For bearing walls that are defined between other vertical column members e.g. steel columns, the wall columns at the edge of the panel are omitted and the associated wall beam is connected to the steel column (for example) and the adjacent wall column - as below.



Wall columns at the edge of the panel are also omitted when it is defined between concrete walls.

Irrespective of whether the wall spans between other vertical column members or not - any load applied to the wall beam at the edge of the panel is shared between the end column and the first internal column. This can result in some load being `lost' directly into the supports.

Load transfer in the bearing wall model is not the same as it would be in for example, a masonry wall. A point load applied at the top of a masonry wall would result in a distributed load on any beam supporting the masonry wall, whereas in a bearing wall the supporting beam would be subjected to a pair of point loads, (or possibly even a single point load if the applied load coincides exactly with a wall column location).

Self weight of the bearing wall is concentrated in the wall beams so seismic weight is concentrated at the top of the wall and not split between the floor above and below.

4.3 Analysis model choices

This section looks at how your choices of settings, entity types, and properties affects the solver models that are created.

Click the links below to find out more:

- Beam end releases and partial fixity (page 568)
- Column releases (page 571)
- Rigid offsets and rigid zones in concrete members (page 572)
- Meshed slabs (page 584)
- Sub models (page 616)
- One-way and two-way load decomposition (page 626)
- Diaphragm action in roof panels and slabs (page 619)
- Meshed walls (page 635)
- Cracked, partially cracked, and uncracked concrete members (page 650)
- The Direct Analysis Method (page 655)

Related information

See also

Background to the Analysis/Design procedures (eLearning)

Beam end releases and partial fixity

The following beam end release options are provided:

- **Pin** - Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Fully fixed** - with the following major (My), and minor (Mz) **stiffness** sub-options to allow definition of partial fixity in each direction if required:

    - **Fixed** - Encastré, all degrees of freedom fixed.
    - **Spring linear** - allows you to specify a linear spring stiffness value.
    - **Partially fixed** - allows you to specify partial fixity as a percentage of the member flexural stiffness (%4EI/L)

      See instructions below for setting the Partially fixed release type.
- **Moment (pin Mz)** - Major axis moment connection, and pinned about the minor axis.
- **Moment (pin My)** - Minor axis moment connection, and pinned about the major axis.

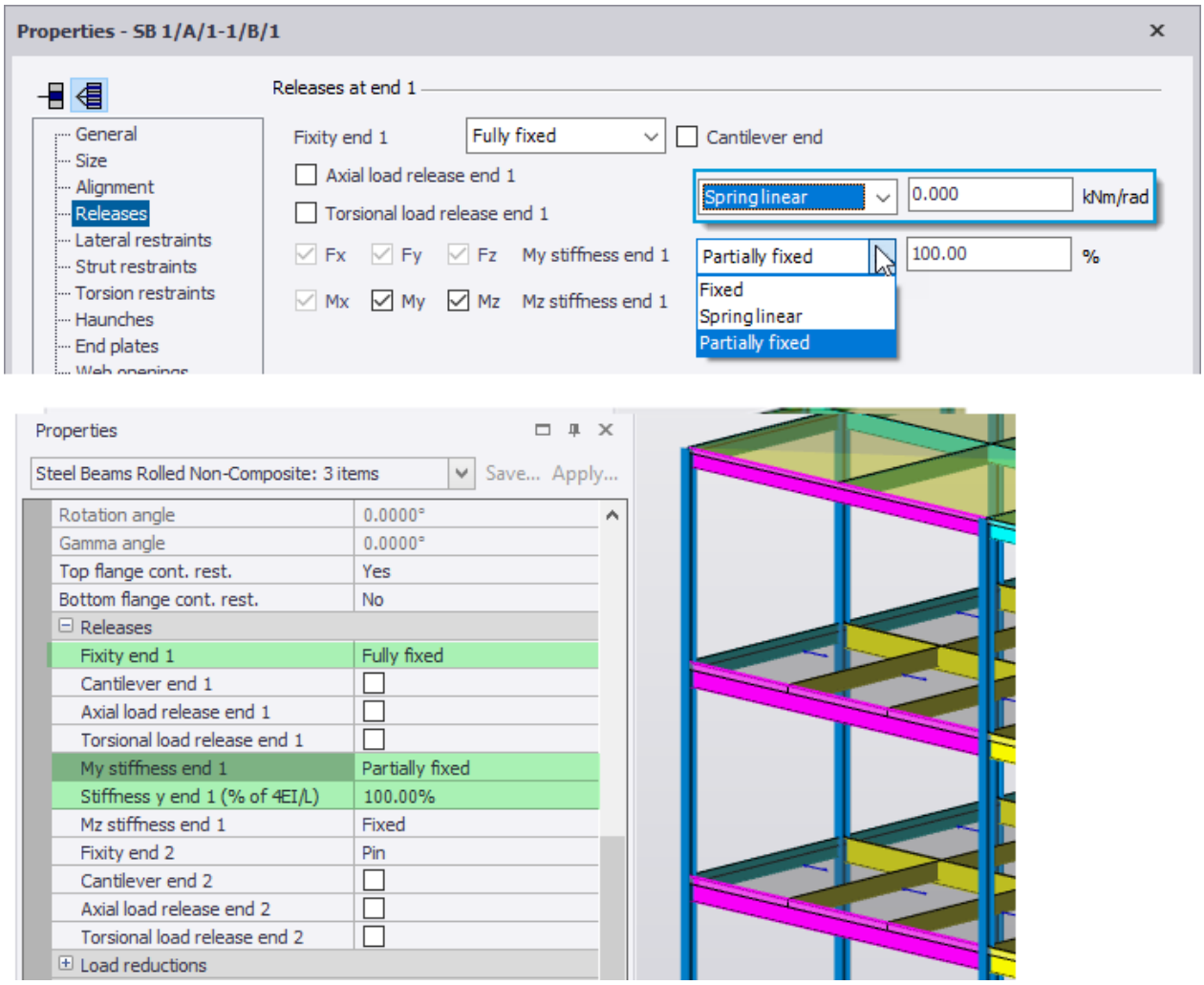
  See instructions below for setting the Moment (pin My) release type.
- **Fully fixed (cantilever end)** - Denotes a cantilever end. It is achieved by selecting Cantilever end.
    - When Cantilever end is selected the remote end of the same span is automatically set to fully fixed.
- **Continuous** - This setting is automatically applied when a continuous beam is created and effectively creates a non-editable fully fixed connection between the spans of the continuous member. The connection can only be edited by splitting the beam.

In addition to the above release options you are also able to apply a **torsional** release at either end by checking the appropriate box. Similarly an **axial** release can be applied to beams of all materials apart from concrete.

Set Partially fixed release type

The partial fixity option is especially useful for example when modeling precast concrete structures in which it is common to consider some amount of end fixity, but not full end fixity (which is commonly assumed for in-situ concrete buildings).

- To apply the **Partially fixed** setting, first set the end Fixity to **Fully fixed** to enable the fixity options of **Spring linear** and **Partially fixed**.
    - For **Spring linear** a value of stiffness in moment/rad is directly entered.
    - For **Partially fixed** a % value is entered. Note that this is the % of the member flexural stiffness (4EI/L) and not of full fixity (which is infinitely stiff). The resulting stiffness is automatically calculated and applied in the analysis model.



End fixity settings can also be applied and verified efficiently via Review View > Show/Alter State, using the Rotational Stiffness (page 997) Attribute.

- Use Set mode to rapidly apply or edit the % Partial fixity and Rotational stiffness values.
- Use Review mode, to display the values in an automatic color-coded range.

Related video

Automatic modelling of partial fixity

Set Moment (pin My) release type

This (unusual) release type is not available in the **Properties** window and can only be specified as follows:

1. Right-click the beam to display the context menu.

2. Choose Edit
3. From the Beam Property Dialog open the Releases page.
4. Check the Mz and uncheck the My degree of freedom at the desired end as required.

Set releases in a Review View

While beam end releases can be set directly in the **Properties** window/ **Properties** dialog they can also be set or toggled graphically in a **Review View** as follows:

- Use the Show/Alter State Cantilever end (page 978) attribute to review or apply cantilever ends
- Use the Show/Alter State Fixed/Pinned (page 963) attribute to review or apply pinned/fixed ends
- Use the Show/Alter State Rotational stiffness (page 997) attribute to review or apply partial fixity/rotational springs

Column releases

The following column release options are provided:

Moment releases

- **Pinned** - Pinned to the adjacent stack and connecting members. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Fixed** - fully fixed
- **Fixed, and set as a cantilever end** - Cantilever ends can be applied to the top or bottom stacks. When applied, both ends of the chosen stack are set to fully fixed.
- **Moment (pin My)** - Pinned about the y axis only
- **Moment (pin Mz)** - Pinned about the z axis only.

NOTE While **Moment (pin My)** and **Moment (pin Mz)** releases are less usual, for continuous columns composed of two or more stacks, they have been provided in response to customer feedback, to allow any/all column stacks to be pinned in one direction whilst the other direction remains fixed.

When such a release is applied, a Warning status is issued in the Column Property dialog. This is intentional and does not prevent analysis or design - as the Warning tooltip states, it is to prompt the engineer to consider the connection details at this location in the column.

Translational releases

- Fy and Fz releases are prevented in all stacks i.e. no ‘roller’ connections.

- Fx (axial) release allowed at the top of the topmost stack only.
- Mx (torsional release allowed in all stacks.

Set releases in a Review View

While column releases can be set directly in the **Properties** window/ **Properties** dialog they can also be set or toggled graphically in a **Review View** as follows:

- Use the Show/Alter State Fixed/Pinned (page 963) attribute to review or apply pinned/fixed ends
- Use the Show/Alter State Cantilever end (page 978) attribute to review or apply cantilever ends

Rigid offsets and rigid zones in concrete members

Solver elements for most members are created directly between the member insertion points - they do not take into account major and minor snap points, or any offsets that might have been specified in the member properties. The exception to this rule is that solver elements for concrete columns and concrete beams do take into account snap points and offsets - **Rigid offsets** are then automatically introduced where necessary to connect the solver elements.

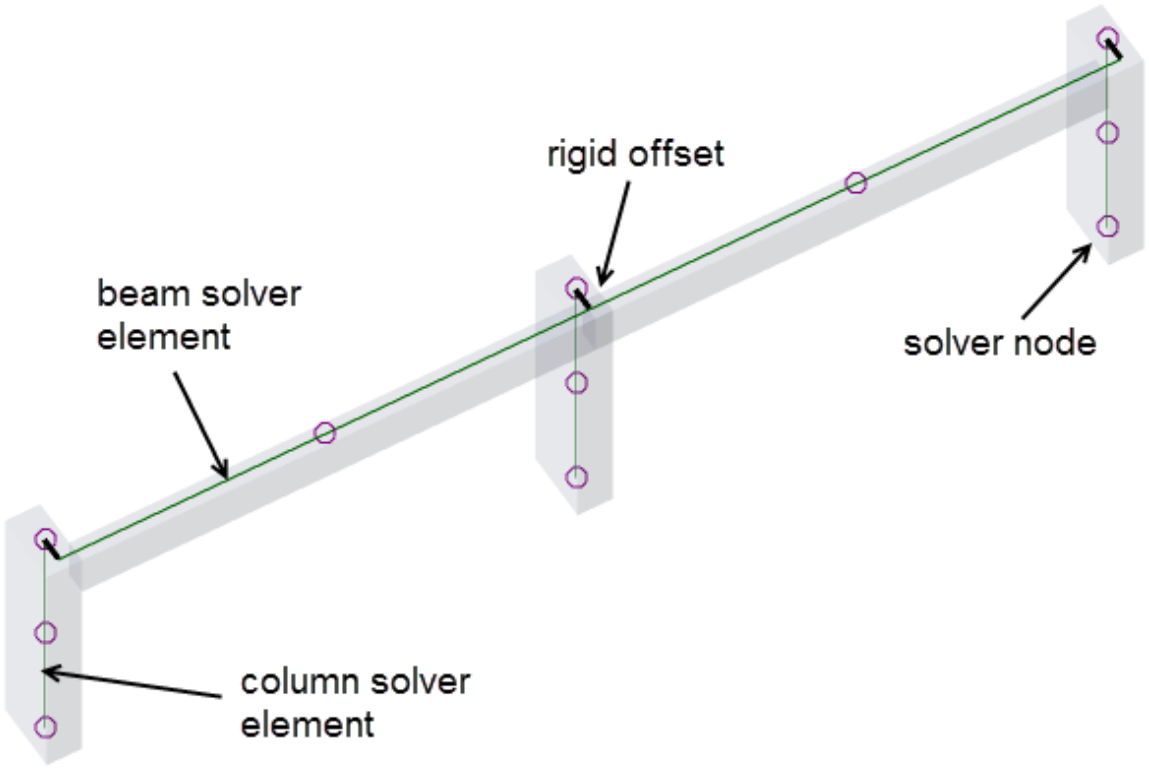
NOTE The rules applied to insertion of solver elements for concrete columns are different to those that are applied to concrete beams. For concrete structures this enables you to simplify the grid layout but then employ offsets to position the members exactly.

Design codes also allow engineers to assume parts of concrete beams and columns are rigid, leading to more efficient designs. Tekla Structural Designer uses **Rigid zones** to cater for this where columns and beams are connected and where beams are connected to other beams. Columns can have rigid zones when they are the supporting or supported member, but beams will only have rigid zones when they are the supported member.

Rigid offsets

For concrete beams and columns rigid offsets are automatically applied to the start and end of solver elements as required to ensure that the solver model is properly connected.

This will be necessary whenever the 1D solver elements are not co-linear. A typical example of this occurs when concrete edge beams are aligned to be flush with the face of the supporting columns, as shown below:



TIP To see rigid offsets: open a Solver View, and then in Scene Content ensure that 1D Elements> RigidOffsets is selected.

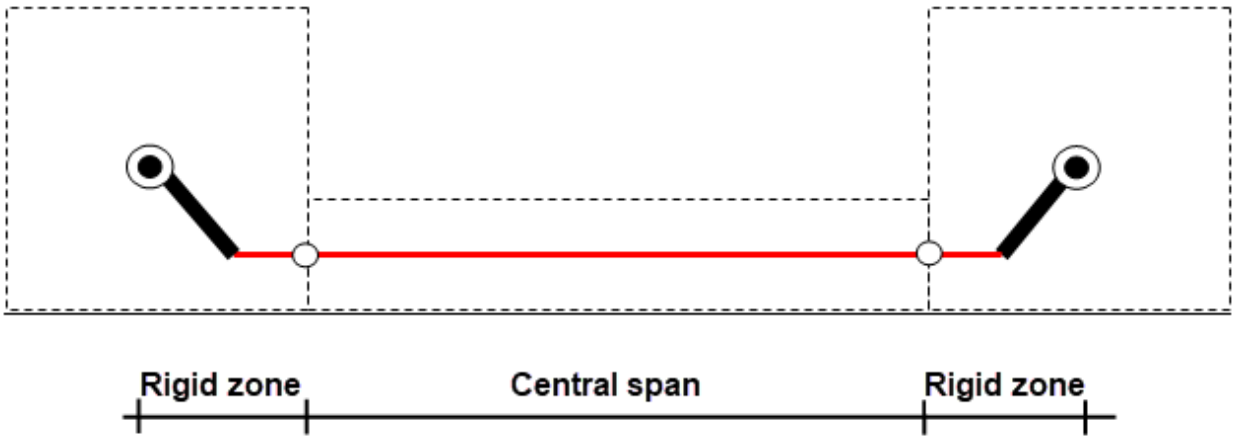
Click the below link to view a couple of examples illustrating the use of rigid offsets in concrete columns and beams:

- Rigid offsets examples (page 574)

Rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections. The proportion of the zone which is modeled as rigid (the thick black line shown below) is specified as a percentage, the remaining portion of the rigid zone (the red line inside the rigid zone) remains

elastic. The proportion of the rigid zone that is rigid is specified in Model Settings and can vary between 0 - 100%



As shown above, the elastic portion of the rigid zone is aligned with the central span solver element.

In most situations in order to get an efficient design you would want rigid zones to be applied. You can however choose not to consider them by checking the **Rigid zones not applied** option that is provided in Model Settings, this will deactivate them throughout the model. You can also selectively deactivate rigid zones at specific column/beam connections by unchecking the **Apply rigid zones** option that is provided in the column properties under the **Design control** heading.

- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the position of releases in analysis model is affected, and member start and end points for design are also adjusted.

There is a significant difference between Rigid Zones Not Applied and Rigid Zones Applied with 0% rigidity. The total elastic length of a member is the same in the two models, but the position of releases and start/end of design members will be different.

Rigid zones should not be confused with rigid offsets which are used to ensure that the analysis model is properly connected, i.e. it is possible to have rigid offsets in the model even if rigid zones are turned off.

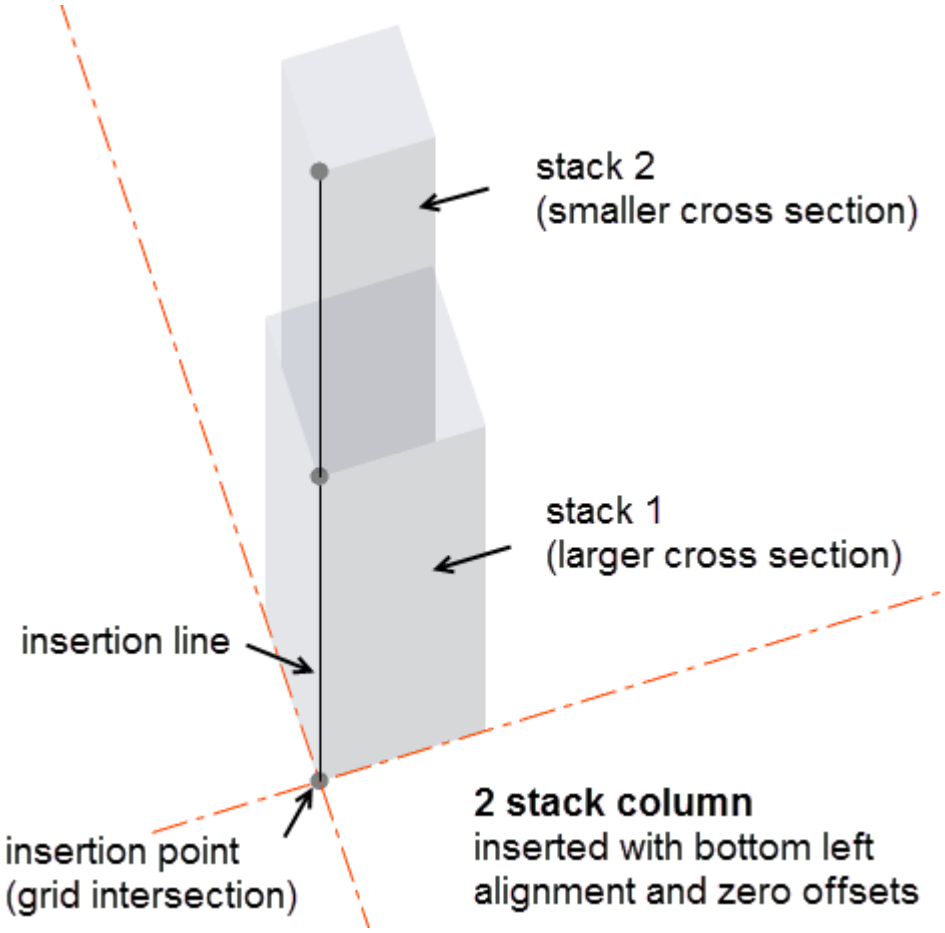
Click the below link to view a couple of examples illustrating the use of rigid zones in fixed and pin ended beams:

- Rigid zones examples (page 578)

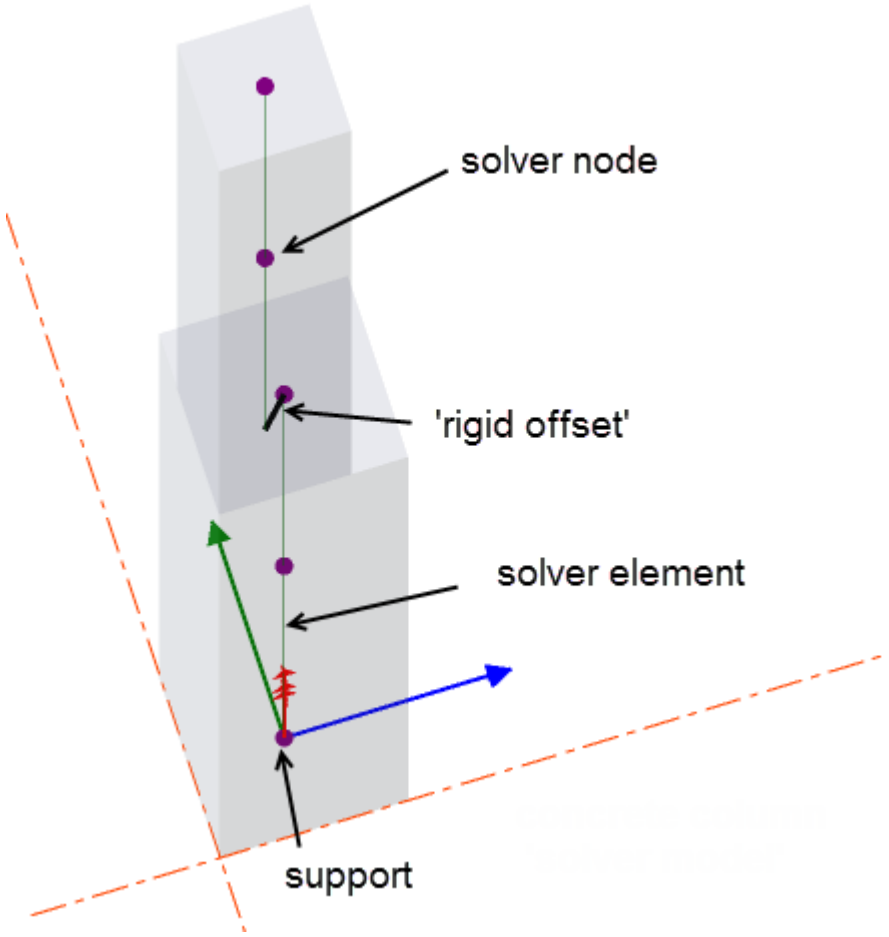
Rigid offsets examples

Rigid offsets example 1 - concrete column

Consider the two stack concrete column shown below - this has been inserted with its alignment properties set to bottom left so that the outer column faces remain flush despite a smaller section being introduced in stack 2.



Since solver elements for concrete columns always take into account any snap points or offsets, they will always be located at the centroid of each stack - thus they do not necessarily coincide with the insertion line used to position the column originally. In this example the centroid position shifts from one stack to the next which causes a “rigid offset” to be created automatically to connect the solver elements. Similar rigid offsets would also be created as required to connect incoming beams into the column centroids.

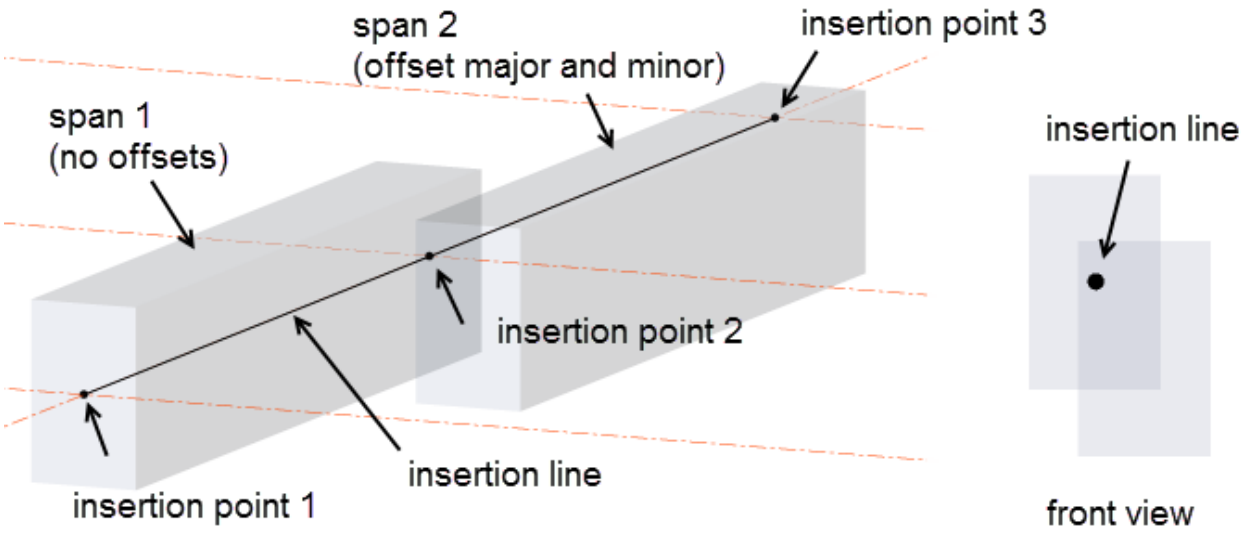


TIP To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset** and **Solver Nodes> Geometry**.

As a consequence of this method of modeling, you are freer to simplify the grid layout in order to create the structure more effectively, and then employ column offsets to position each column exactly, knowing that during analysis the program automatically assumes the column is located at its centroid as shown in the plan view.

Rigid offsets example 2 - concrete beam

Consider the two span concrete beam shown below - this has been inserted with both major and minor axis offsets applied to span 2 only.



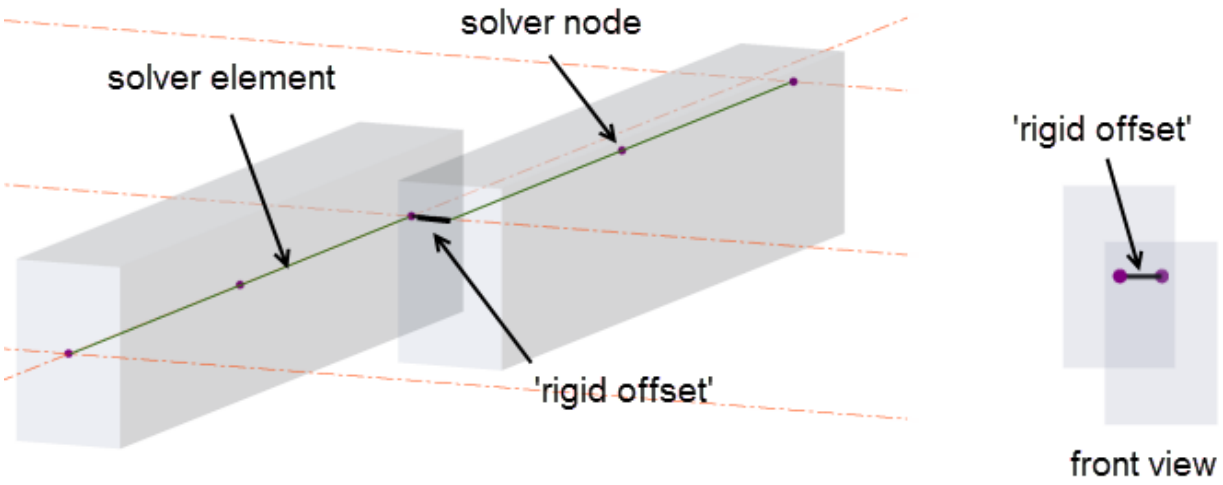
NOTE For concrete beams:

- The **minor** snap points and offsets **are** structurally significant and have an effect on the positioning of the 1D solver elements.
- The **major** snap points and offsets **are not** structurally significant.

In the minor direction beam solver elements are always located at the center of each beam section - as beam widths or minor offsets may vary, this may result in the introduction of lateral rigid offsets to make the connection between spans.

In the major direction beam solver elements are always created at the same level as the insertion line used to position the beam.

Consequently, for this particular example a lateral rigid offset is required to make the connection between the two spans.

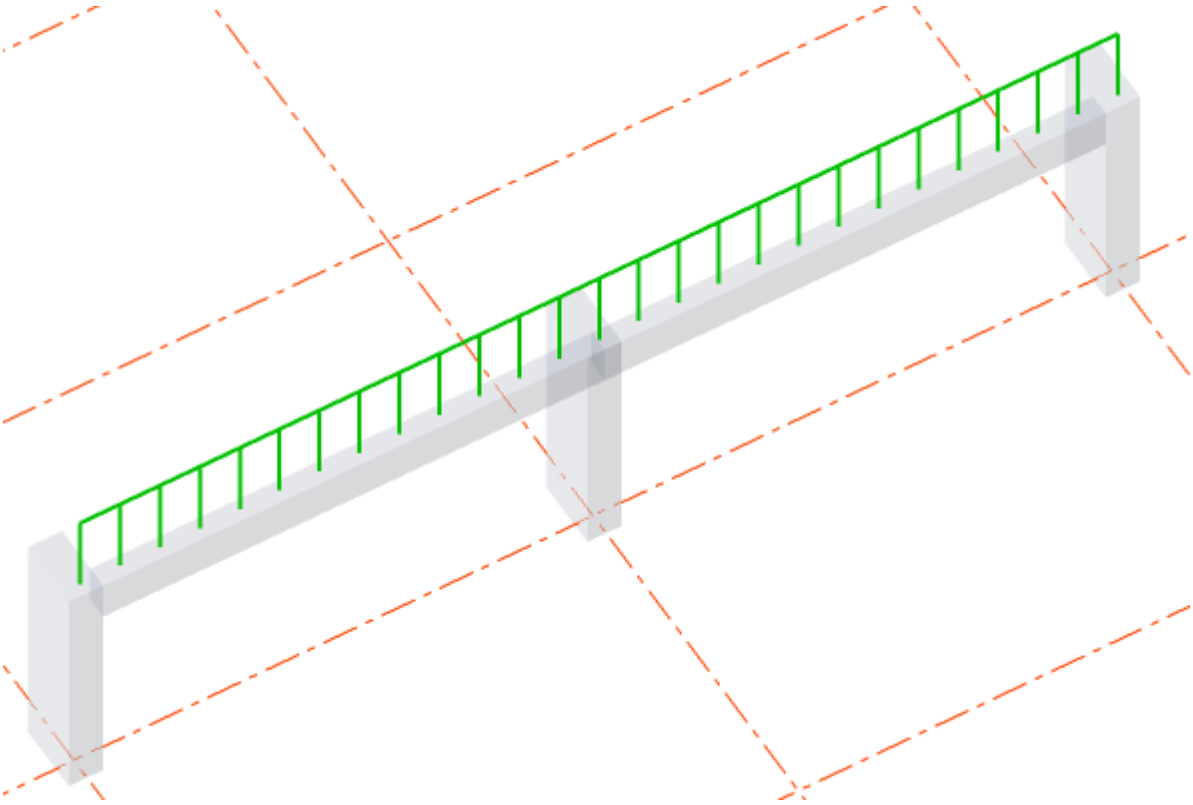


TIP To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset** and **Solver Nodes> Geometry**.

Rigid zones examples

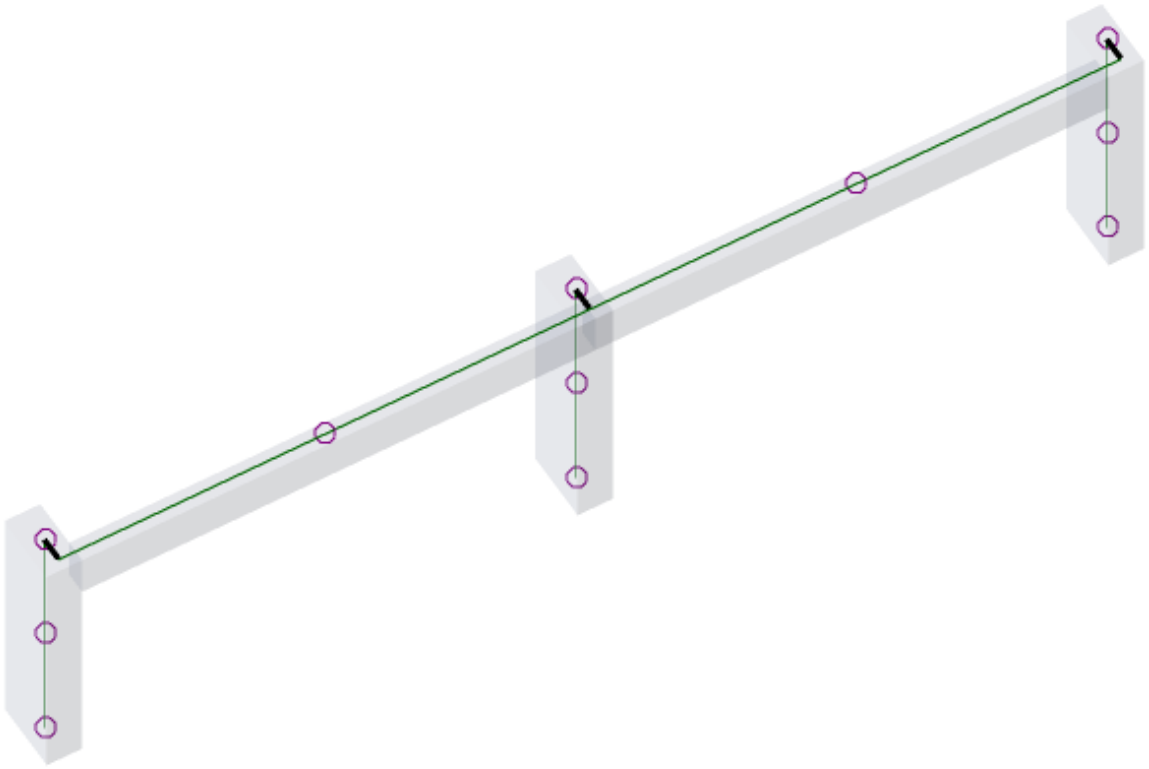
Rigid zones example 1 - fixed ended beam

Consider the following 2 span beam supported on columns and loaded with a udl:

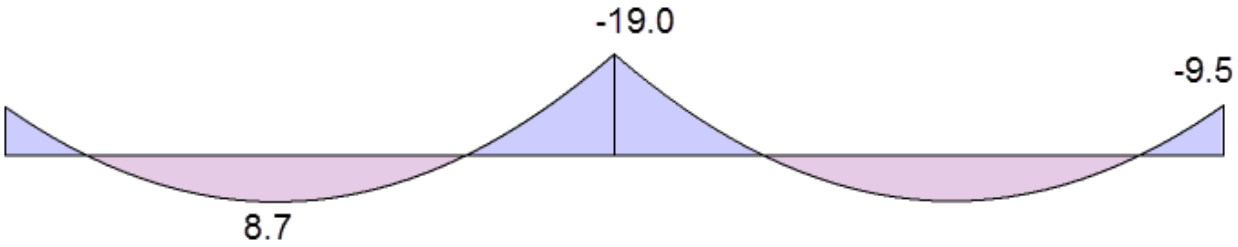


Rigid Zones Not Applied

The analysis model is simply constructed from the solver elements with rigid offsets applied as necessary to connect the beam solver elements to the column solver elements.



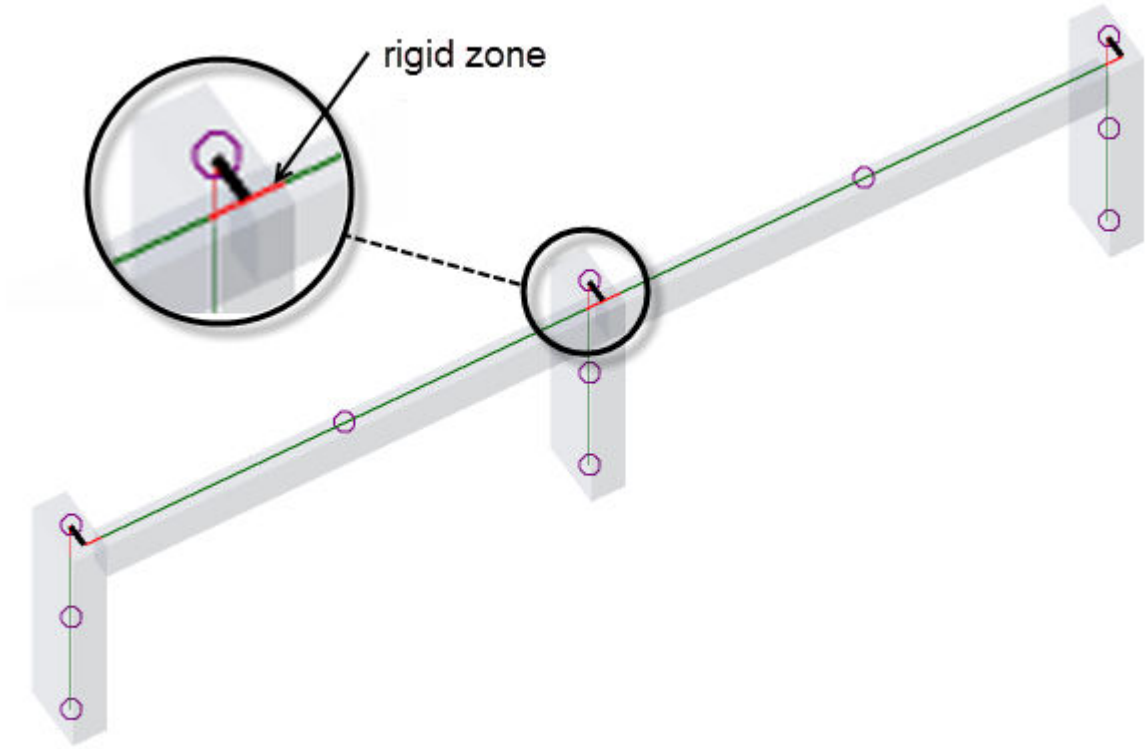
The resulting beam bending moment diagram is as follows:



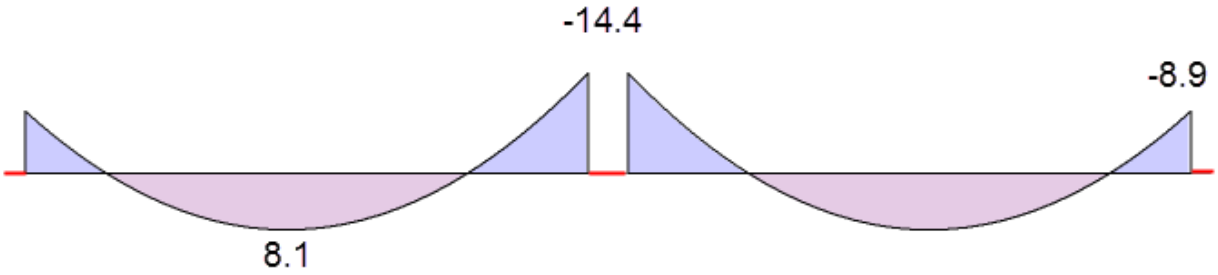
Rigid Zones Applied (default 50%)

Rigid Zones are activated via Model Settings, and this is also where the percentage of rigidity of the zones can be specified. Initially for this example it will be left at the default of 50%.

The revised solver model is as shown below, note the rigid zones that have been formed where the columns and beams connect:



The beam bending moment diagram for the revised model is as shown below.

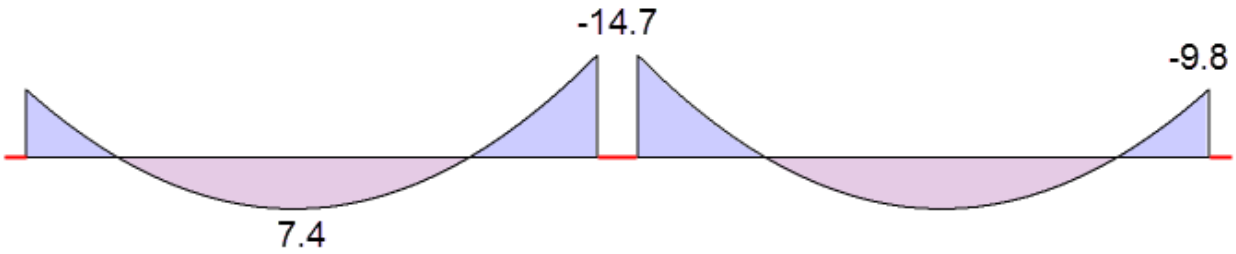


NOTE The above diagram was generated from a Results View to illustrate that a "gap" is formed in the diagram where there are rigid zones. It should be noted that when the same result is displayed in a Load Analysis View the gap is removed, leaving only the non-rigid length of the member displayed.

We might expect the extra stiffness introduced at the supports to increase the hogging moments and reduce the sagging moments, however because the element end forces are now reported at the rigid zone boundaries (rather than the ends of the solver elements) - in this example the main effect is that the hogging moment over the central column support is substantially reduced.

Rigid Zones Applied (100%)

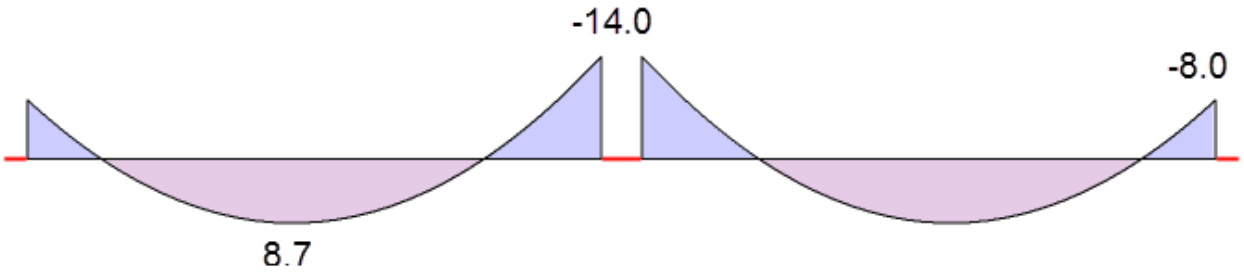
To investigate the effect of the percentage rigidity an additional run is made with the percentage rigidity increased to 100%. The bending moment diagram that results is shown below:



As expected the extra stiffness introduced at the supports increases the hogging moments and reduce the sagging moments in comparison to the run at 50%.

Rigid Zones Applied (0%)

If the percentage rigidity is reduced to 0% the bending moment is as shown below:



If this result is compared to the run in which rigid zones were not applied, it is clear that although the sagging moments are identical, the hogging moments that are reported are not the same. This is because, although the total elastic length of a member is the same in the two models, the position of the start and end of design members is different (being taken at the rigid zone boundaries when applied).

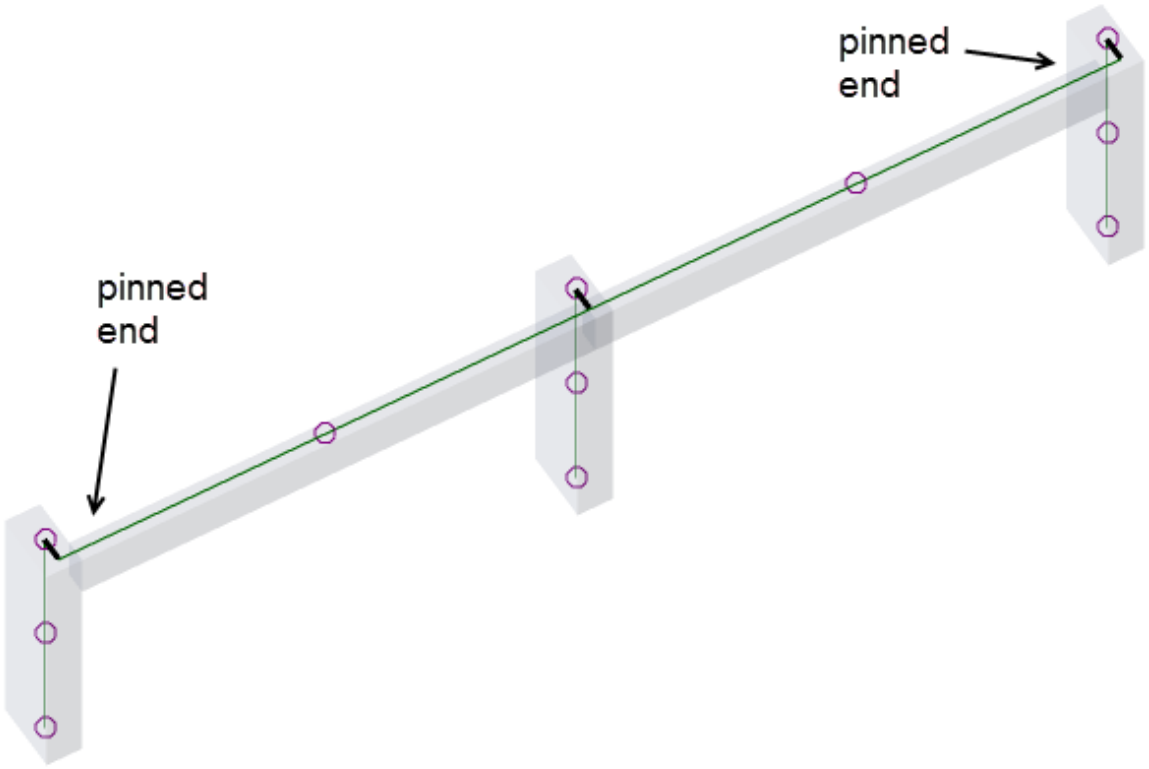
Rigid zones example 2 - pin ended beam

When rigid zones are applied to a pin ended member, the end release is shifted from the end of the solver element to the rigid zone boundary.

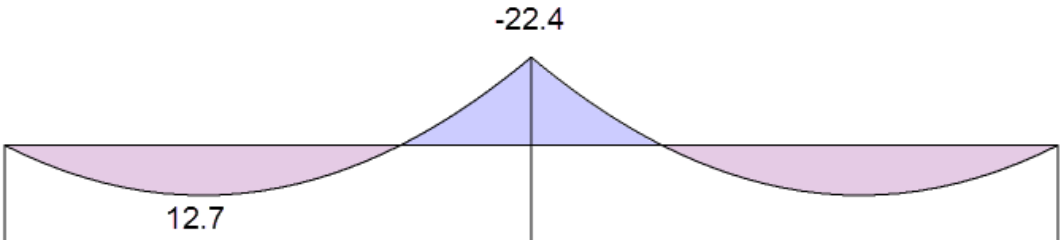
To illustrate this effect the model illustrated in Rigid zones example 1 is modified to have pinned connections introduced at the two remote ends of the beam.

Rigid Zones Not Applied

The analysis model is constructed from the solver elements with rigid offsets applied to connect the beam and column solver elements. Releases are formed at the two remote ends of the beam solver elements.



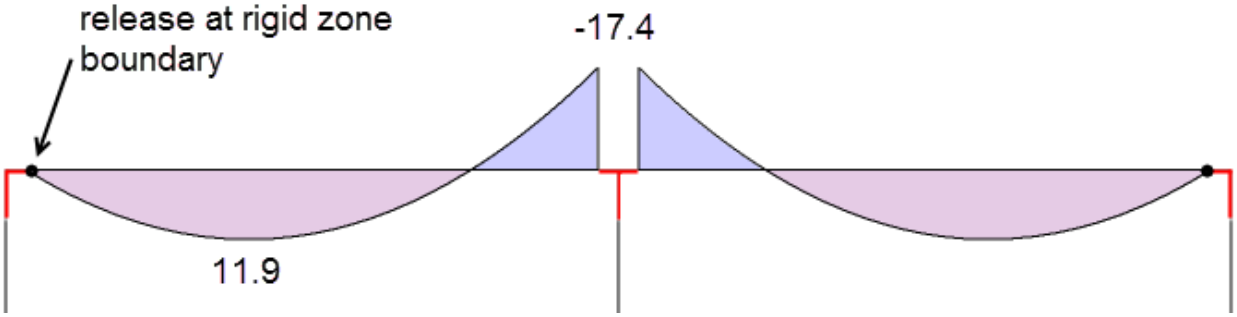
The beam bending moment diagram is as follows:



Rigid Zones Applied (0%)

For comparison, rigid zones are then introduced, (with 0% rigidity in order to keep the total elastic length of the beams the same in both models).

Because the releases are moved to the rigid zone boundaries, this has the effect of reducing the moments in the beams.



FE meshed slabs

At the levels where two-way spanning[1] slabs exist, Tekla Structural Designer applies FE meshing as follows:

- Two-way spanning slabs are **always** meshed in the FE chasedown analysis that occurs as part of the static design process.
- Two-way slabs are by default **not** meshed for 3D building analysis or grillage chasedown analysis. This ensures the quickest solution time. A 3D pre-analysis process (in which the slabs are meshed) is used to decompose loads from the slabs on to supporting members.
- Alternatively, you can choose to mesh two-way slabs for 3D building analysis and grillage chasedown analysis, either at all, or selected construction levels. In this case Tekla Structural Designer does not perform the load decomposition during 3D pre-analysis for the two-way slabs at those levels.

In all of the above mentioned situations, the slab mesh density is set globally, but can be overridden at individual levels if required.

By default two-way slabs are created with **isotropic** analysis properties; **othotropic** properties can be specified if required. This could be particularly useful when designing: precast slab units, profiled steel decking, timber decks on floor joists, and cross-laminated timber (CLT) elements.

Click the links below to find out more:

- Define whether two-way slabs are meshed for 3D building analysis and grillage chasedown analysis (page 585)
- Adjust global slab mesh properties (page 586)

[1] *One-way spanning slabs are also treated as two-way spanning if **Analyze slab** is selected in slab item properties*

- Apply different mesh properties at different levels (page 586)
- Review the slab mesh before the analysis (page 587)
- How slab properties and features impact on meshing (page 587)
- Slab meshing controls (page 592)
- Orthotropic slabs (page 601)

Footnotes

Define whether two-way slabs are meshed for 3D building analysis and grillage chasedown analysis

To ensure the quickest solution time, two-way[1] spanning slabs are by default **not** meshed for 3D building analysis or grillage chasedown analysis. In this case, the diaphragm option (none, semi-rigid, or rigid) in the slab properties determines their axial rigidity and a 3D pre-analysis process is used to decompose loads from the slabs on to supporting members. For more information about load decomposition, see: One-way and two-way load decomposition (page 626).

If you prefer, you can instead choose to mesh the slabs, either at all, or selected construction levels.

NOTE Two-way spanning slabs[1] are **always** meshed in the FE chasedown analysis that occurs as part of the static design process.

Use meshed two-way slabs in at all construction levels

1. Go to the **Project Workspace**.
2. In the **Structure** tree, select  **Levels**.
3. In the **Properties** window, select the **Mesh slabs in 3D analysis** option.

Use FE decomposed slab loads at all construction levels

1. Go to the **Project Workspace**.
2. In the **Structure** tree, select  **Levels**.
3. In the **Properties** window, clear the **Mesh slabs in 3D analysis** option.

[1] *One-way spanning slabs are also treated as two-way spanning if **Analyze slab** is selected in slab item properties*

Use meshed two-way slabs at selected construction levels

1. In the **Structure** tree, expand the  **Levels** branch.
2. Click one of the desired construction levels.
3. According to your needs, either select or clear the **Mesh slabs in 3D analysis** option.
4. Repeat step 3 for all the desired construction levels.

Footnotes

Adjust global slab mesh properties

Initially, the same mesh properties are applied globally to all meshed slabs. In order to adjust the global slab mesh properties, see the following instructions.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click  **Structure**.
3. Go to the **Properties** window.
4. Accept or adjust the **Slab Mesh Size**.
5. Accept or adjust the **Slab Uniformity Factor**. At 100% uniformity all slabs in the mesh will be of approximately equal area.

   TIP To optimize solution time, you can use a coarser mesh during design development. Later, at the final design stage, you can switch to a more refined mesh.

6. Accept or adjust the **Slab Mesh Type**:
   - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
   - **QuadOnly**: only consists of quadrilateral 2D elements.
   - **Triangular**: only consists of triangular 2D elements.

See also

Apply different mesh properties at different levels (page 586)

Apply different mesh properties at different levels

If you need to apply different mesh parameters at a specific level, you can do so by creating sub models. For more information, see the following instructions.

1. Create a sub model. (page 617)
2. In the **Structure** tree, expand the  **Sub Models** branch.
3. Select the sub model that you created.
4. In the **Properties** window, select the **Override model's** option.
5. Adjust the mesh size and uniformity factor according to your needs.

Review the slab mesh before the analysis

In order to review the existing slab mesh in your model before running a particular analysis, see the following instructions.

1. In the **Status bar** at the bottom of the window, click  **Solver View**.

   The solver view opens.
2. Right-click anywhere in the solver view.
3. In the context menu, go to **Solver models**.
4. In the list, select the solver model appropriate to the analysis that you want to run.

   If the slab mesh is applicable to the selected solver model, it is displayed.

   NOTE The slab mesh is not displayed in the working solver model. This is because the working solver model displays the model in its form before any analysis, and 2D elements are only formed during the analysis.

See also

View and manage solver models (page 537)

How slab properties and features impact on meshing

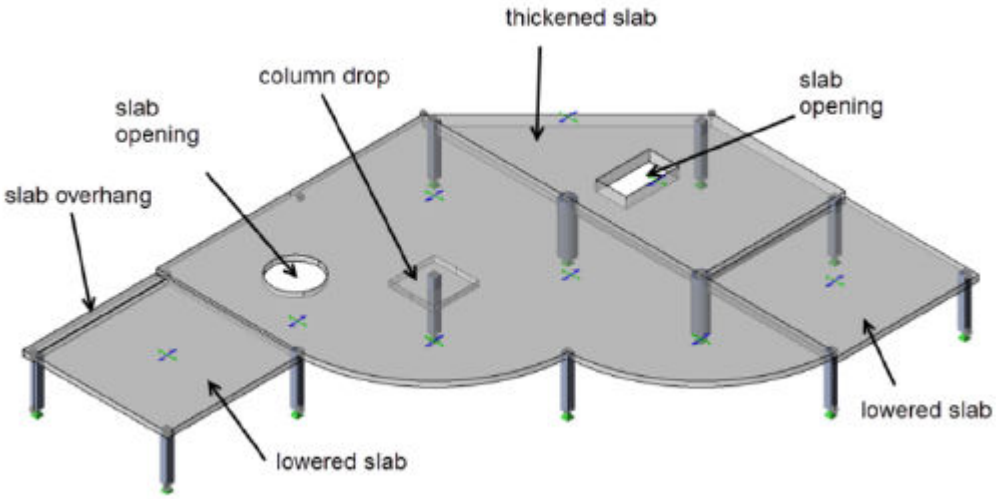
The slab mesh is always created in a single plane, but the resulting mesh is can be affected by the slab properties and features as described below.

Slab thickness, vertical offsets, column drops and openings

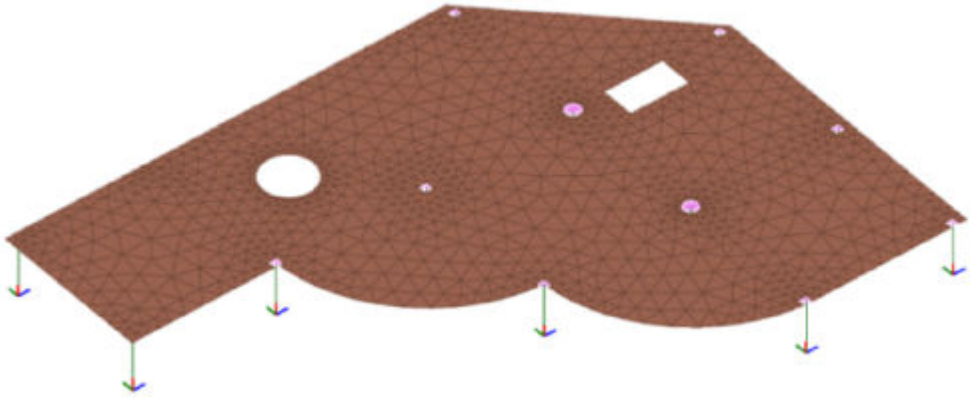
The mesh of shell 2D solver elements is always created in the same plane, irrespective of whether slabs have different thicknesses, slabs items have been raised/lowered via vertical offsets, or column drops have been applied. Shells

are not created inside slab openings and any loads placed within openings are not applied to the model.

Consider the example shown below. This features curved slab boundaries, circular and rectangular openings, thickened slab panels, lowered slab panels and a slab overhang. A column drop panel has also been inserted at one of the locations where the slab is supported by a column.



In the resulting FE solver model, since vertical offsets are not structurally significant the analysis mesh is formed at the same level relative to the top of the slab. The mesh properties do however reflect the change to the slab thicknesses in the different slab areas.



NOTE Beam solver elements and slab meshes can only be offset vertically from one another by being defined in different construction levels.

Other slab properties

Rotation Angle

Different slab items in the same slab can have different rotation angles.

This property is used for the following where appropriate:

- Span direction for 1-way load decomposition
- To determine the 2D solver element local axes in the solver model
- Bar direction for Slab on Beam and Flat Slabs.

Include in Diaphragm

This property is only available when the Diaphragm option is Semi-Rigid or Rigid. Individual slab items in the same slab can be included or excluded as required.

The effect of excluding a slab item depends on the Diaphragm option as follows:

- Semi-Rigid - excluded slab items are not meshed with semi-rigid 2D solver elements
- Rigid - internal nodes not considered in the nodal constraints

NOTE Where 2 items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

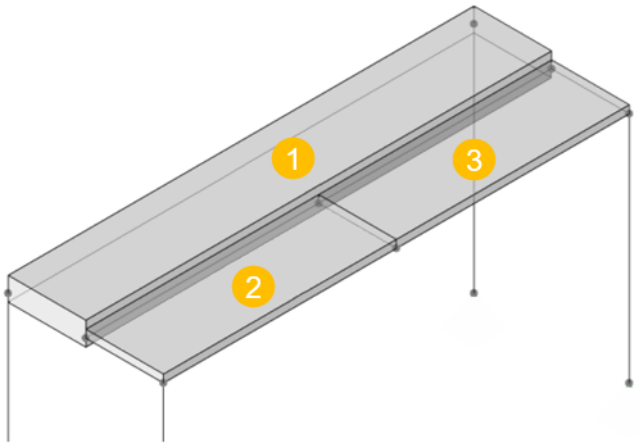
NOTE When a slab item is excluded from the diaphragm this has no effect on the mesh of shell 2D solver elements used in some of the solver models for 2-way spanning slabs.

Mesh groups

To facilitate meshing, Tekla Structural Designer automatically gathers slab items and features together into mesh groups, and meshes them as a single entity. A mesh group contains one or more slab items with identical analysis attributes. Since slab depth is a key analysis attribute, by definition a slab step, or a column drop, will produce an additional mesh group.

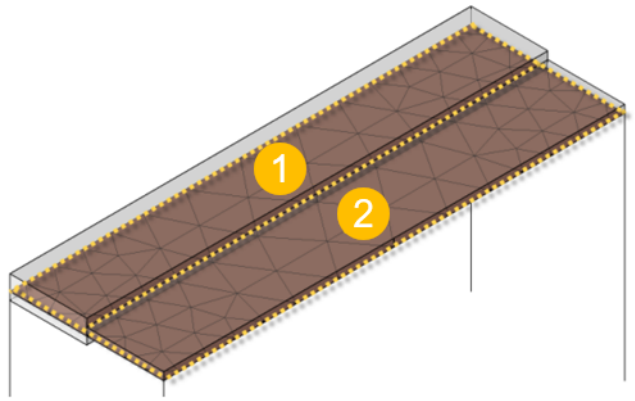
Example: Mesh groups at a slab step

In the following image, you can see three separate slab items:



1. d = 300 mm
2. d = 100 mm
3. d = 100 mm

Although there are three slab items in the previous image, there are only two different slab depths. That is why Tekla Structural Designer only creates two mesh groups, as shown in the following image:



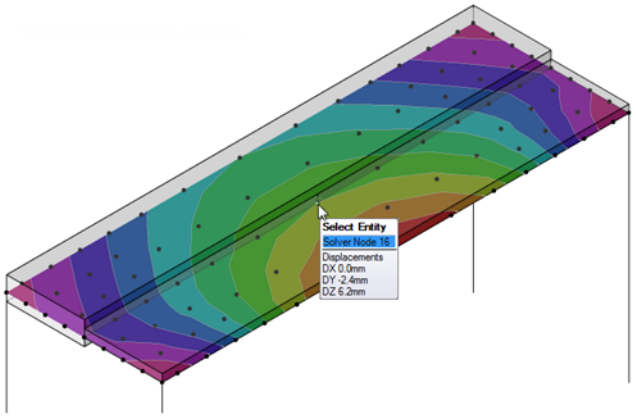
1. mesh group 1
2. mesh group 2

Discontinuity of force contours at slab steps and column drops

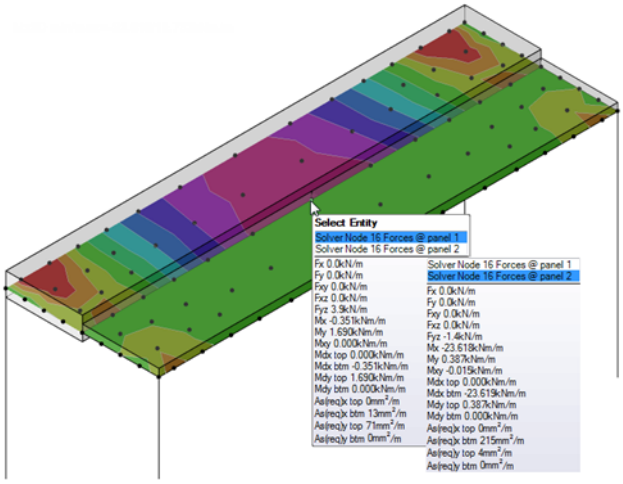
When slab items on either side of a slab step are placed into different mesh groups, the solver nodes along the boundary are shared by both groups. Each node on the boundary reports a single value of deflection, but two values of force, one for each group. That is why there will be a discontinuity of force contours along the boundary.

See the following examples:

Example: Deflection contours (no discontinuity)



Example: Moment contours, discontinuity along boundary



The force discontinuity is a genuine result. The slab items share the same curvature at the step and have the same elastic modulus, so the moment must be directly proportional to the inertia of each slab item.

NOTE Other programs may average the value across the boundary when generating the contours. However, we prefer the approach of Tekla Structural Designer because averaging would result in an unrealistically high design of the thinner slab.

Mesh group boundary warnings

Meshing may fail or produce undesirable results when there is challenging mesh group boundary geometry. In this case, Tekla Structural Designer warns you about the source points source of meshing issues.

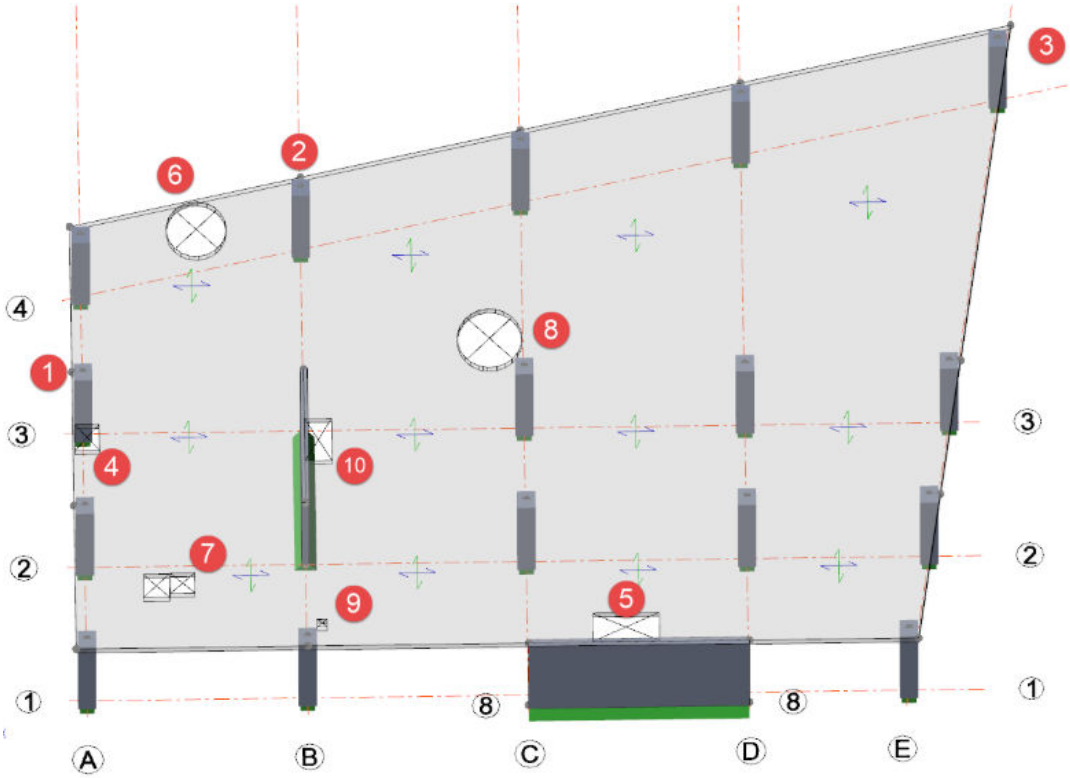
Examples of possible warning triggers are:

- short edges
- distance between a hole and an edge
- small area enclosed by a mesh group

Slab meshing controls

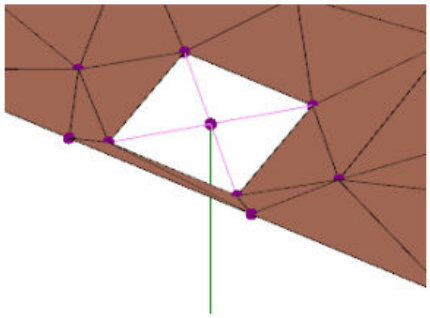
Meshing issues tend to be caused when the area being meshed ends up with really narrow strips of slab, for example when the slab edge ends up just outside the edge of a column.

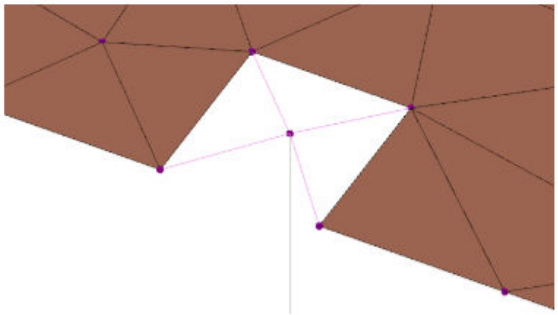
On the **Meshing** page of Analysis options (page 760) certain meshing tolerance controls can be adjusted when required to solve these type of issue.



The model shown above has various challenging situations which are discussed below along with an indication of whether changing the automatic merging distances in Analysis Settings can have an impact:

1. Column face close to parallel slab edge - (yes)

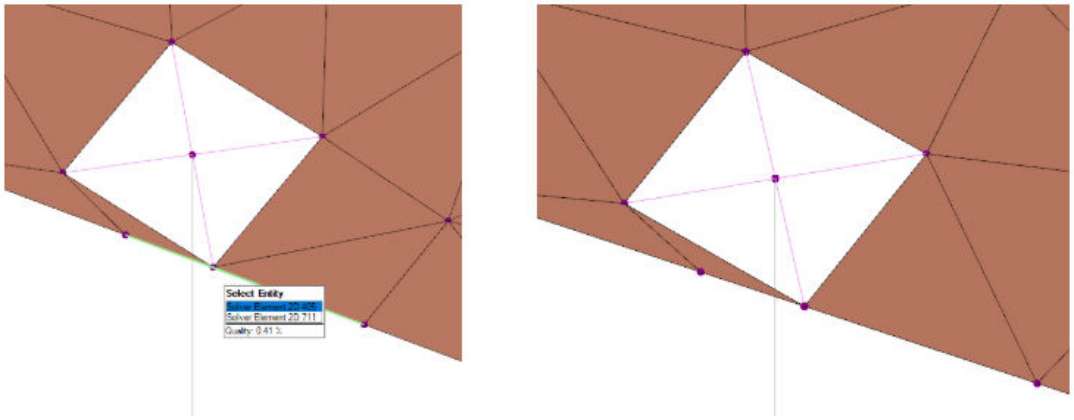




In this example the slab edge is positioned 50mm outside the column face. The view above left shows the analysis model you would get when the Column boundary to slab edge merging distance is < 50mm. The view above right shows the model you get if the merging distance is set to 50mm or above. The column corner nodes merge (move to) the slab edge eliminating the narrow strip.

The simplified analysis model avoids mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

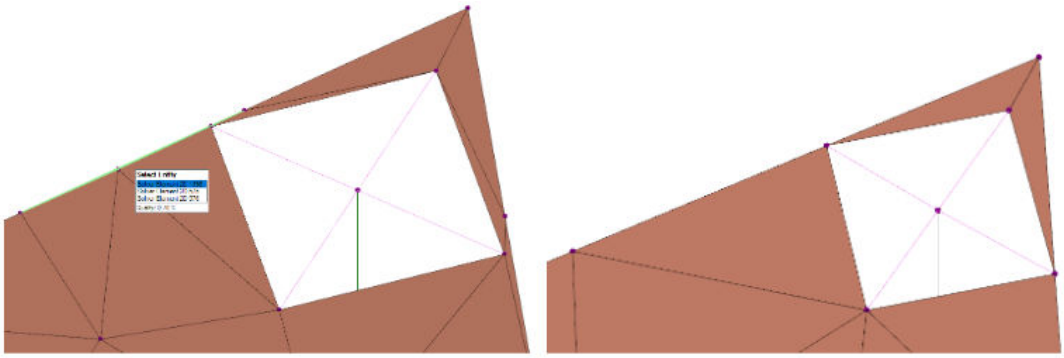
2. Column face close to inclined slab edge - (yes)



In this example the slab edge sits just a few mm outside the column corner. The view above left shows analysis model where a very poor quality element runs past the corner, (you would only end up with this model if the Column boundary to slab edge merging distance were to be set less than the default 5mm. The view above right shows the model you get if the merging distance is set to the default 5mm or above. One column corner node merges (moves to) the slab edge eliminating the narrowest part of strip.

The simplified analysis model avoids mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

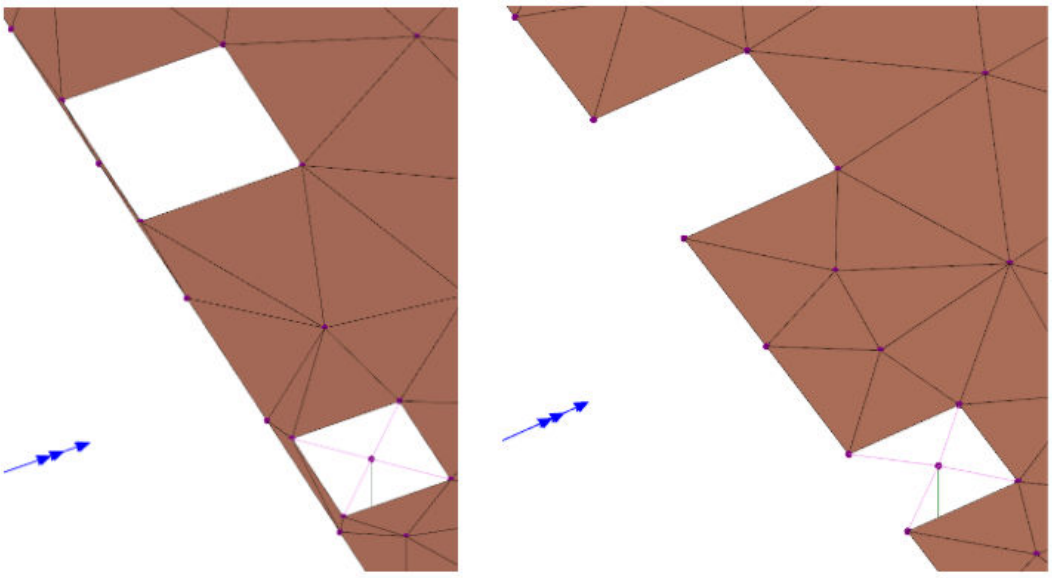
3. Column close to multiple non-parallel slab edges - (yes)



This is just another variation on positions 1 and 2. The view above left shows analysis model you would end up with if the Column boundary to slab edge merging distance were to be set less than the default 5mm. The view above right shows the model you get provided the default merging distance (5mm) is retained - meshing is simplified avoiding mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

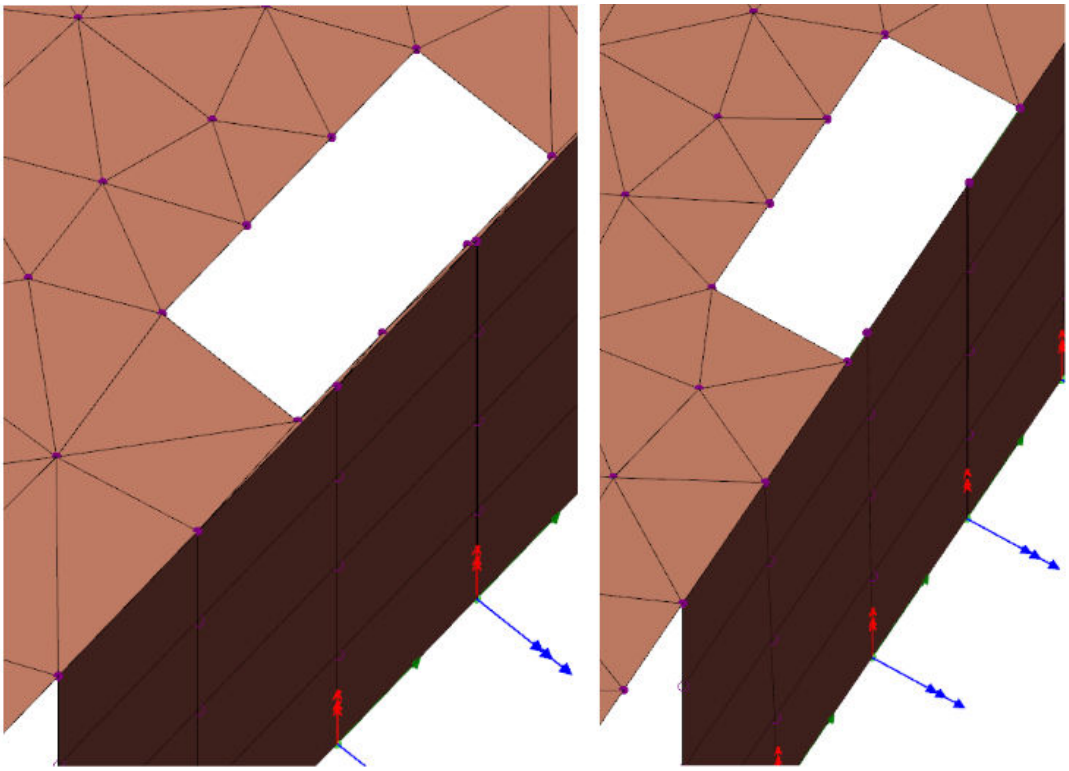
4. Opening near slab edge - (yes)

In the following examples merging will make things better, but the reality is that a slab will never be cast with thin isolated strips at the edge - you cannot effectively reinforce a 50 to 100mm wide strip of slab. In all these the ideal solution is that the modelling should reflect the reality of realistic minimum edge distances.



A hole is defined approximately 20mm from the slab edge. The effect is the same as at the column cut-out boundary - above on the right shows how with the Opening to slab edge merge distance set to 50mm the opening corners move to the slab edge. This creates a better analysis model.

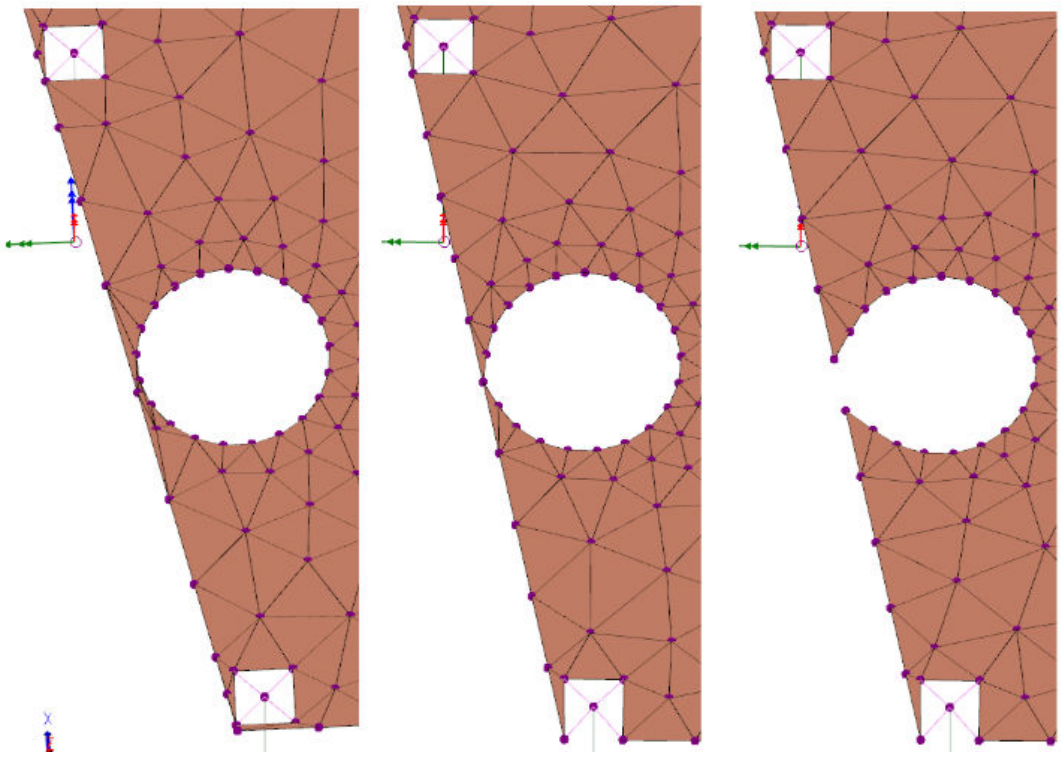
5. Opening near wall inserted along slab edge - (yes)



This is basically the same as the previous situation except that a wall happens to be defined along the edge. In such cases the user is often tempted to extend the opening into the physical width of the wall but does not extend it right to the wall insertion line.

There are pros and cons to both models. On the left the narrow meshed strip of slab can give mesh quality warnings and errors. On the right the slab mesh is better, but the model relies a bit more heavily on the wall beam to transfer moments. However, the overall minor axis moment transferred to the wall only drops from 75.7 to 72.5 kNm.

6. Circular opening near slab edge - (yes)

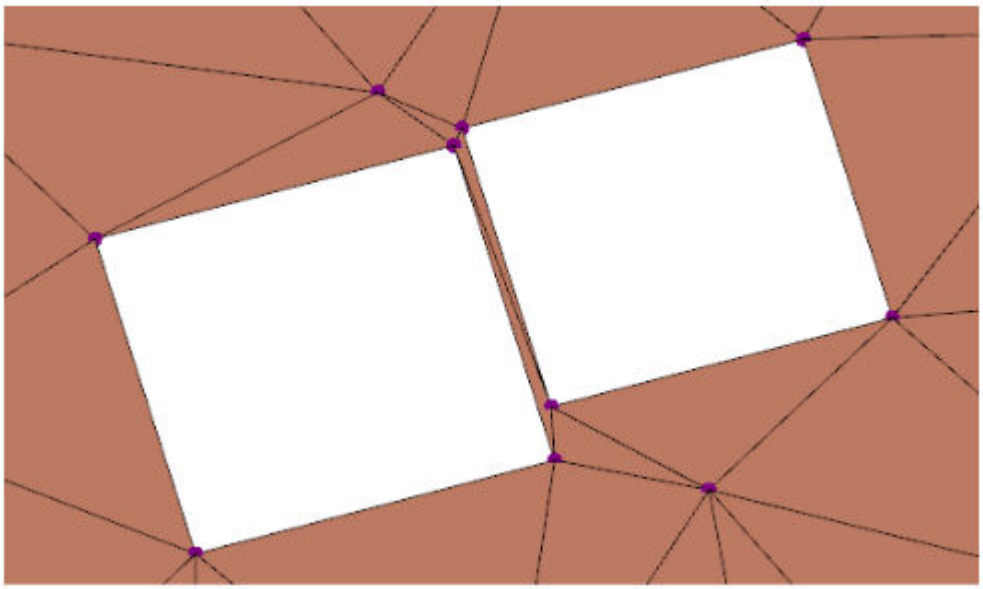


Above left shows the sort of meshing that will occur when a circular opening is created near to the slab edge. Above center shows an example where closest point on the circle is merged to edge. The meshing looks a bit better but this is still not a great model because you will get strange local results at the connecting node. On the right where the Opening to slab edge merge distance is set to 100mm the edge gets completely broken - this is probably the best model.

7. Closely spaced openings - (no)

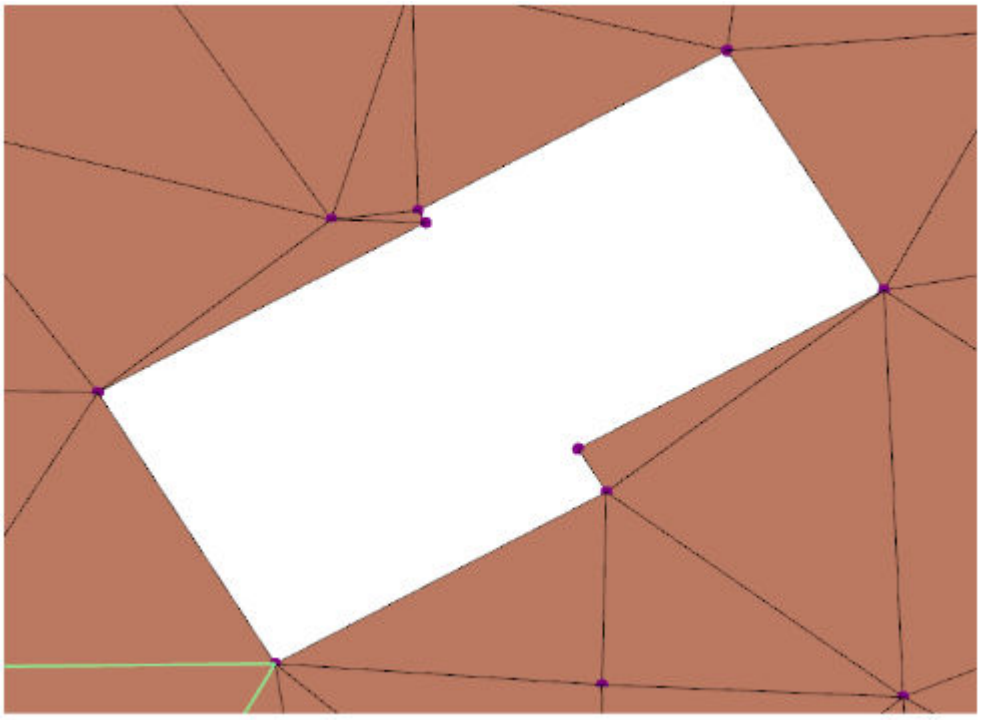
These are all examples where the merging settings have no effect.

In each case the problems can generally be resolved by adjusting the openings so that they overlap slightly.

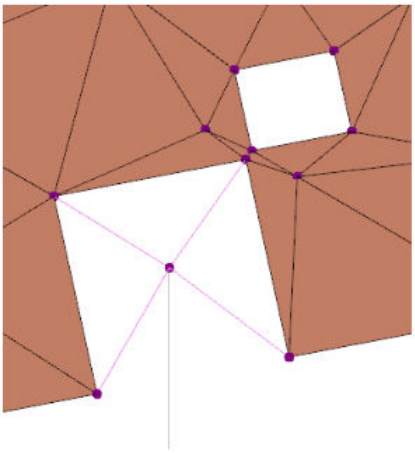


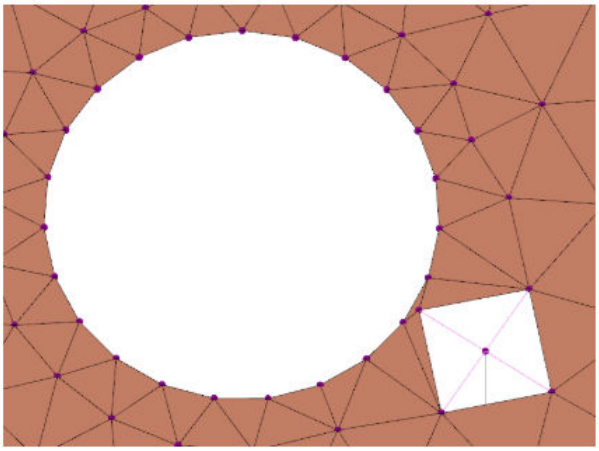
Above - shows meshing when a narrow (unbuildable?) strip of slab is left between openings.

Below shows the model when the openings are manually adjusted to eliminate this strip.

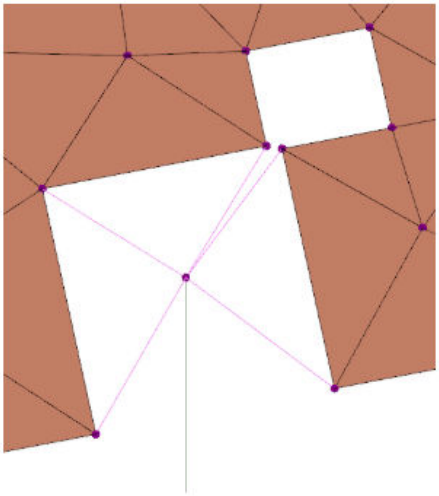


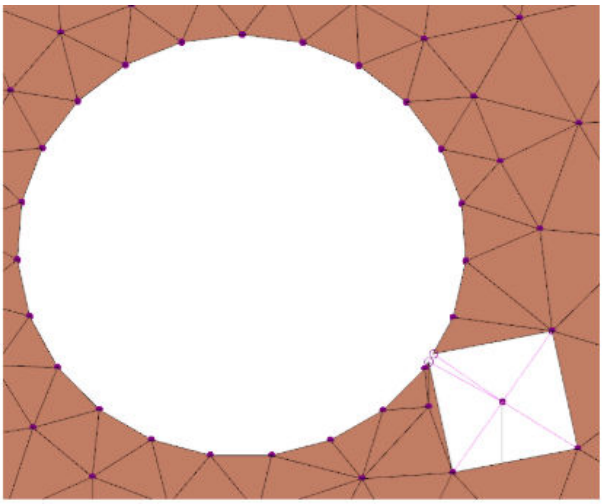
8 and 9. Opening near column - (no)



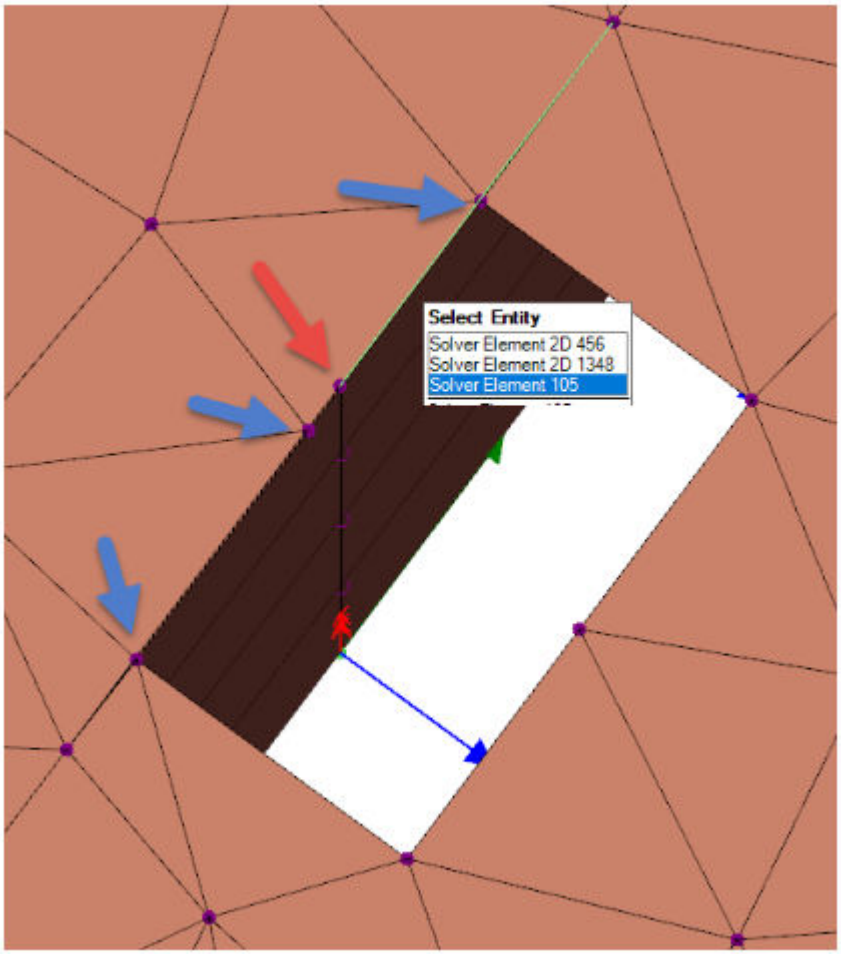


In both these cases it would again be better to adjust the openings so that they overlap the column boundaries as suggested below.



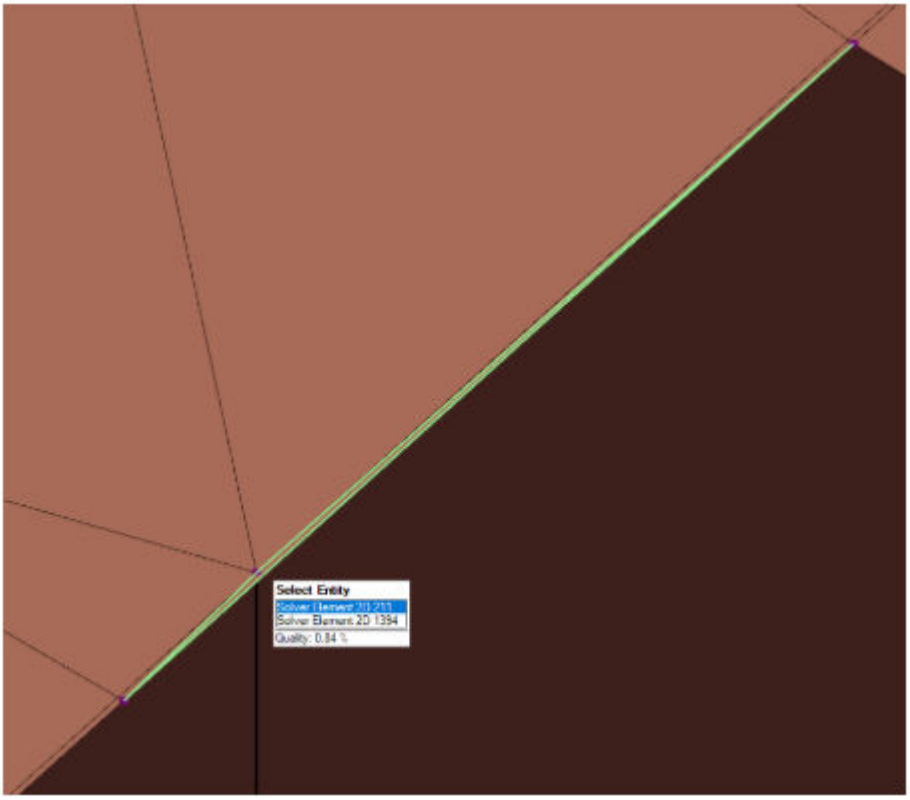


10. Opening near wall with slab on other side - (no, unless certain conditions apply)

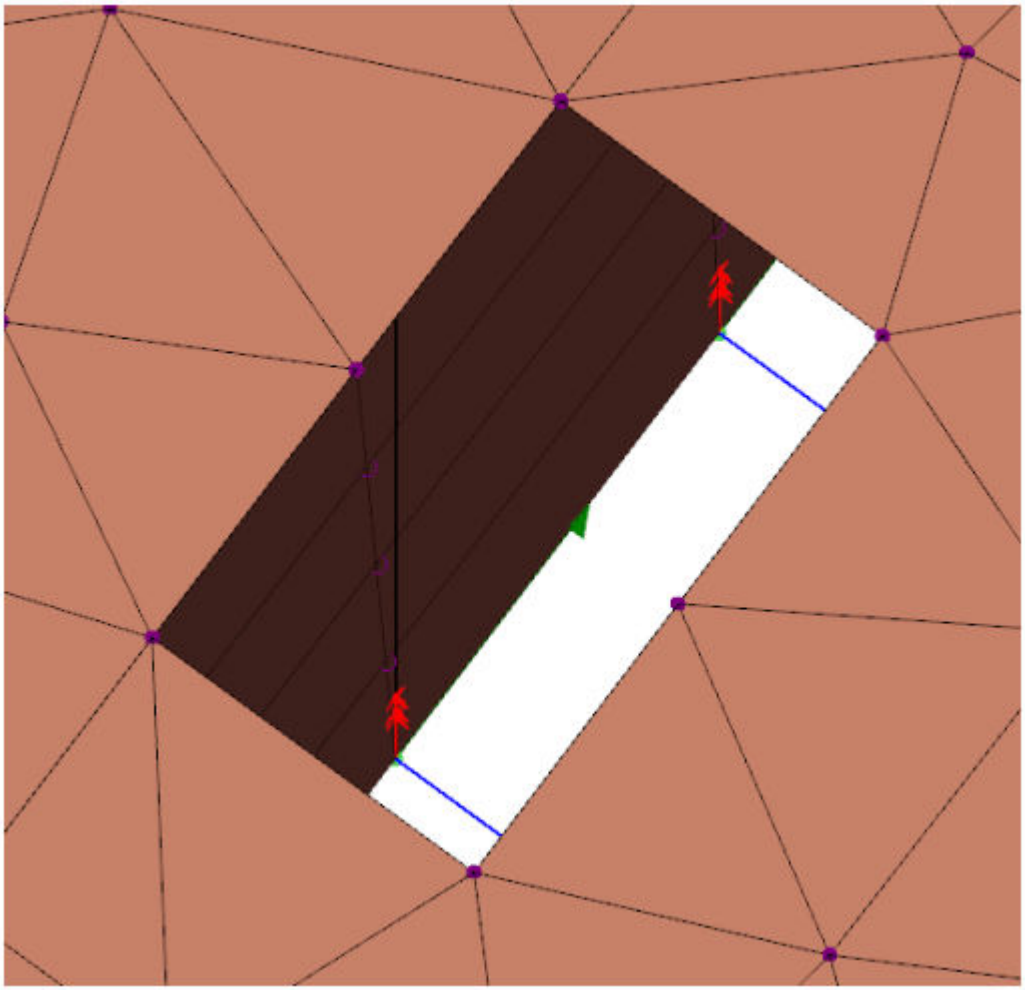


In this case the opening is defined with an edge that is approximately 4mm from the insertion line of the wall. With the Hardpoint to edge distance tolerance at the default of 5mm this creates a model that is poorly connected along the edge of the opening. The node highlighted in red is not included in the slab meshing (because it is just 4mm away from the edge). The 3 slab nodes highlighted in blue are not directly connected to the wall.

If the hard point distance is reduced to 1mm then the model changes as shown below, the node marked in red above is now included in the slab mesh so the adjacent nodes have a better connection to the wall.



However, the best model would be obtained if the opening edge got exactly aligned with the wall, this can actually be quite hard to achieve in the manual positioning of openings. A workaround at present is to force a “slab mesh boundary” along the line of the wall by making the slab on one side slightly different to the slab on the other - then the automatic mergings will create a model as shown below.



Orthotropic slabs

There are two ways to define an orthotropic meshed slab in Tekla Structural Designer: you either specify relative stiffness factors, or you enter material properties manually.

Both methods (as described in more detail below) require you to select the **Orthotropic** option in the slab properties. This option actually has two practical uses:

- It enables different stiffnesses to be set for the two orthogonal (X and Y) directions of slabs thus making them *orthotropic* rather than *isotropic* (same stiffness in both directions). This can be used to model slab elements that exhibit orthotropic material properties and are assumed to behave predominantly as “one-way spanning”, such as precast slab units, profiled steel decking, timber decks on floor joists and cross-laminated timber (CLT) elements.

- For isotropic slabs, it also provides a way (as described below) to override the global stiffness modification factor with a modification factor specific to the selected slab only.

Related information

Related video

Orthotropic Properties

Define an orthotropic slab using relative stiffness factors

The stiffness modification X factor and Y factor can be set in the *Analysis parameters* property grid. They are visible only when *Orthotropic* and *Calculate properties* options are selected and can be set as follows:

1. Select any slab item within the slab of interest.
2. In the **Properties** window, navigate to **Slab parameters > Analysis parameters**
3. Select **Orthotropic**
4. Enter the required values of **X factor** and **Y factor**.

   Adjusting the factors in this way allows you to automatically set material orthotropicity using the method described in Using the 'Calculate properties' functionality. Decreasing one of the factors can have a significant impact on internal forces in the slab item, as well as on the load transfer.

Example: In a particular model in order to achieve highly orthotropic behavior you might enter an X factor of 0.001 while maintaining the Y factor as 1.0 as shown below.

Analysis parameters	
Orthotropic	☑
Calculate properties	☑
X factor	0.001
Y factor	1.000
Ex	34N/mm^2
Ey	33500N/mm^2
Gxy	523N/mm^2
Gxz | Gyz	13958N/mm^2
νx	0.400
νy	0.000
Apply modification factors	☑

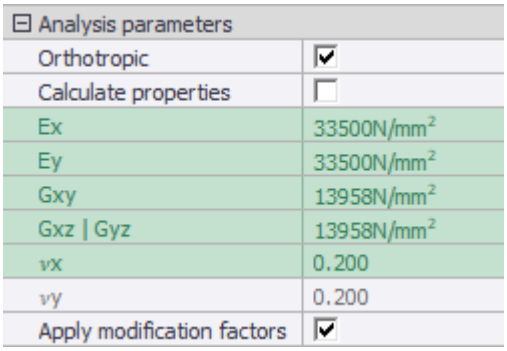
Define an orthotropic slab by entering material properties manually

As an alternative to calculating the orthotropic material properties using X and Y factors, the properties can be also set manually if required. The properties take default values based on their state prior to deselecting the checkbox.

To increase the slab item's orthotropicity, except adjusting Young's moduli in the local X and Y directions, respectively, one might consider modifying the in-plane shear modulus, Gxy, as well.

To define properties manually in this way:

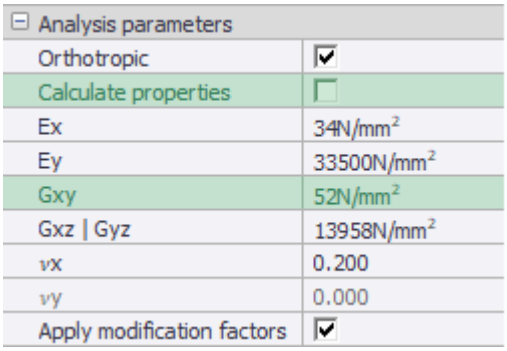
1. Select any slab item within the slab of interest.
2. In the **Properties** window, navigate to **Slab parameters > Analysis parameters**
3. Select **Orthotropic**
4. Unselect **Calculate properties**



The properties take default values based on their state prior to deselecting the checkbox.

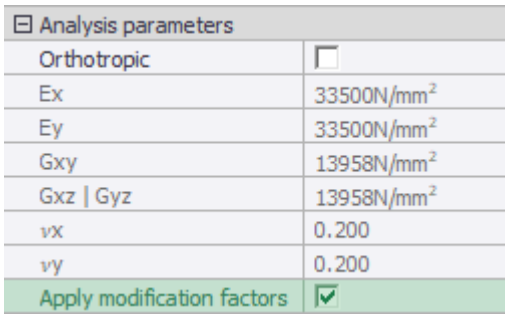
5. Enter the required values of Young's moduli **Ex** and **Ey**.

Example: In a particular model in order to achieve highly orthotropic behavior you might reduce Ex while maintaining the Ey unchanged. To increase the slab item's orthotropicity, one might consider modifying the in-plane shear modulus, Gxy, as well.



Override global stiffness modification factor with slab specific factors

By default the **Apply modification factors** option is selected, so that the stiffness modification factors defined in Analysis Settings > Modification Factors (page 772) are applied.



If required, the above option can be cleared to enable the analysis parameters to be passed directly to the solver unmodified.

Furthermore, by selecting the **Orthotropic** and the **Calculate properties** options you are then provided access to X and Y factors which can be used in the place of the global stiffness modification factor. Although the implication is that you will apply these to make the slab *orthotropic* there is nothing to prevent you from specifying identical X and Y values. In this case, due to lack of differences between X and Y factors, the slab will be analyzed as an *isotropic* one. When used in this way, the **Orthotropic** property can be activated for isotropic slabs solely to override the global stiffness modification factor with a slab specific factor.

This is done as follows:

1. Select any slab item within the slab of interest.
2. In the **Properties** window, navigate to **Slab parameters > Analysis parameters**
3. Select **Orthotropic**
4. Select **Calculate properties**
5. Clear **Apply modification factors**
6. Enter the *same* value of **X factor** and **Y factor**.

Example: In a particular model the global factor might be 0.2 for typical thin flat slabs, but you could have a reason to override this with 0.3 for some much

thicker transfer slab. This would be achieved by entering both the X factor and Y factor as 0.3 as shown below.

Analysis parameters	
Orthotropic	☑
Calculate properties	☑
X factor	0.300
Y factor	0.300
Ex	10050N/mm^2
Ey	10050N/mm^2
Gxy	4188N/mm^2
Gxz | Gyz	4188N/mm^2
νx	0.200
νy	0.200
Apply modification factors	☐

See also

Orthotropic slab implementation (page 605)

"One-way" spanning orthotropic slab example (page 607)

Further orthotropic slab modeling considerations (page 613)

Orthotropic slab implementation

The following properties are required when specifying an orthotropic slab:

- E_x - Young's modulus in the local X direction
- E_y - Young's modulus in the local Y direction
- G_{xy} - Shear modulus in the local XY plane (in-plane)
- G_{xz} - Shear modulus in the local XZ plane (out-of-plane)
- G_{yz} - Shear modulus in the local YZ plane (out-of-plane)
- ν_x - Poisson's ratio in the local X direction
- ν_y - Poisson's ratio in the local Y direction

NOTE No distinction is made between G_{xz} and G_{yz} out-of-plane shear moduli.

Using the 'Calculate properties' functionality

The material properties are determined based on the X and Y stiffness modification factors, denoted as ν_x and ν_y, respectively. The factors are applied to Young's modulus, E_{db}, and Poisson's ratio, ν_{db}, defined in the materials database. The shear modulus defined in the materials database, G_{db}, is ignored, as the in-plane and out-of-plane shear moduli are derived using the formulae shown below.

Young's moduli in local X and Y directions

$$E_x = \alpha_x E_{db}$$

$$E_y = \alpha_y E_{db}$$

Poisson's ratios in local X and Y directions

$$\nu_x = \frac{\alpha_y}{\bar{\alpha}} \nu_{db}$$

$$\nu_y = \frac{\alpha_x}{\bar{\alpha}} \nu_{db}$$

where

$$\bar{\alpha} = \frac{\alpha_x + \alpha_y}{2}$$

In-plane shear modulus

The modulus is established using Huber's formula:

$$G_{xy} = \frac{\sqrt{E_x E_y}}{2\left(1 + \sqrt{\nu_x \nu_y}\right)}$$

Out-of-plane shear moduli

$$G_{xz} = G_{yz} = \frac{\max\left(E_x, E_y\right)}{2(1+\bar{\nu})}$$

where

$$\bar{\nu} = \frac{\nu_x + \nu_y}{2}$$

Manually setting material properties

The material properties can be set manually when the 'Calculate properties' checkbox is unchecked. The properties take default values based on their state prior to deselecting the checkbox.

The Poisson's ratio in the local Y direction is not directly editable, and is calculated automatically based on the editable parameters using the following formula:

$$\bar{\nu} = \frac{\nu_x + \nu_y}{2}$$

When manually defining properties of an orthotropic slab item, it is recommended to treat the local X direction as representing the minor direction, i.e. characterized by lower stiffness. This allows modifying the corresponding Poisson's ratio, which is not possible when the local Y direction is chosen as the minor direction. In case the local Y direction of a slab item needs to represent the minor direction, it's advisable to rotate the item's LCS, by offsetting the current X direction by 90^{o} in *Slab item properties -> General*. Please refer to Slab item properties (page 1458) page for more information.

See also

"One-way" spanning orthotropic slab example (page 607)

Further orthotropic slab modeling considerations (page 613)

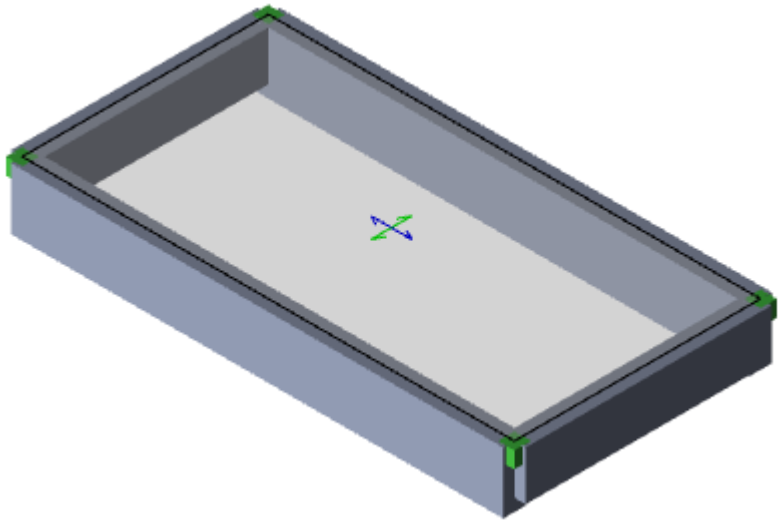
1st order linear - Simply supported orthotropic square slab (page 522)

"One-way" spanning orthotropic slab example

Defining orthotropic material properties for a two-way spanning slab allows you to reliably model the behavior of a one-way spanning slab. This will be investigated in the topics below with the aid of an example model.

Example model

The example initially consists of a two-way spanning slab item supported by beams along all four edges. The beams have fully fixed supports at each end. The slab item is defined with its X-axis parallel to its long edge. A full area load is applied to the slab.



The slab is set as orthotropic by selecting the appropriate option in the *Analysis parameters* property grid.

Analysis parameters	
Orthotropic	☑
Calculate properties	☑
X factor	1.000
Y factor	1.000
Ex	33500N/mm^2
Ey	33500N/mm^2
Gxy	13958N/mm^2
Gxz | Gyz	13958N/mm^2
νx	0.200
νy	0.200
Apply modification factors	☑

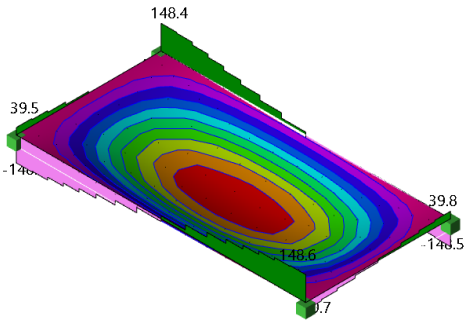
Adjusting the stiffness modification factor

The stiffness modification X and Y factors are only visible only when *Orthotropic* and *Calculate properties* options are selected.

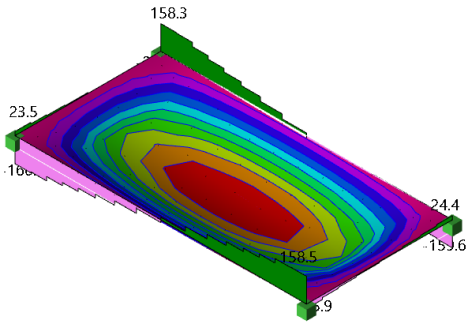
Analysis parameters	
Orthotropic	☑
Calculate properties	☑
X factor	0.001
Y factor	1.000
Ex	34N/mm²
Ey	33500N/mm²
Gxy	523N/mm²
Gxz | Gyz	13958N/mm²
νx	0.400
νy	0.000
Apply modification factors	☑

Adjusting the factors allows you to automatically set material orthotropicity using the method described in Using the 'Calculate properties' functionality. Adjusting one of the factors can have a significant impact on internal forces in the slab item, as well as on the load transfer.

It can be seen below how reducing the X factor from 0.1 (left) to 0.001 (right) impacts the results.



Max deflection: 17.6 mm



Max deflection: 19.3 mm

Adjusting the material properties

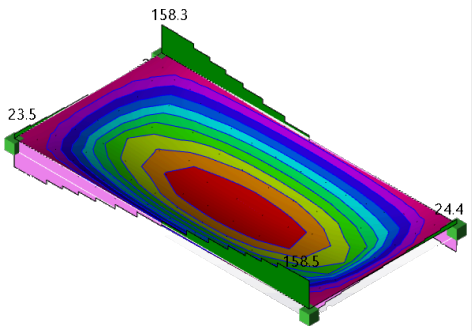
As an alternative to calculating the orthotropic material properties using X and Y factors, the properties can be set manually if required. This can be achieved by deselecting the *Calculate properties* option in the *Analysis parameters* property grid. The properties take default values based on their state prior to deselecting the checkbox.

Analysis parameters	
Orthotropic	☑
Calculate properties	☐
Ex	33500N/mm²
Ey	33500N/mm²
Gxy	13958N/mm²
Gxz | Gyz	13958N/mm²
νx	0.200
νy	0.200
Apply modification factors	☑

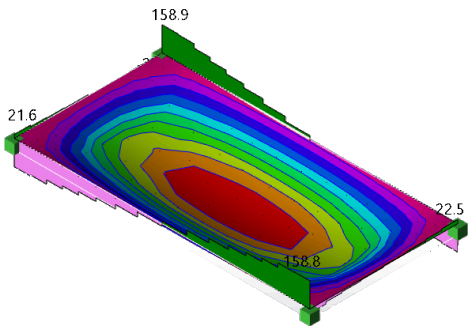
To increase the slab's orthotropicity, after adjusting Young's modulus in the local X direction (to replicate the effect of the X factor), one might consider modifying the in-plane shear modulus, Gxy, as well. For example the in-plane shear modulus could be reduced from 523 N/mm^2 (left) to 52.3 N/mm^2.

Analysis parameters	
Orthotropic	☑
Calculate properties	☐
Ex	34N/mm²
Ey	33500N/mm²
Gxy	52N/mm²
Gxz | Gyz	13958N/mm²
νx	0.400
νy	0.000
Apply modification factors	☑

How this impacts the results is shown below. Shear modulus 523 N/mm^2 (left) and 52.3 N/mm^2 (right).



Max deflection: 19.3 mm

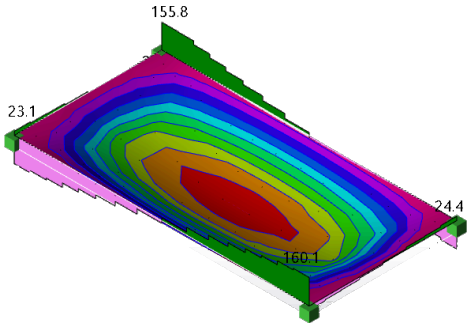


Max deflection: 19.5 mm

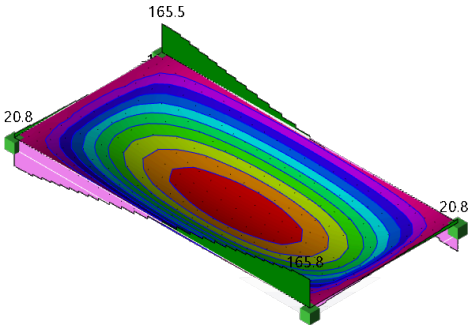
Adjusting the mesh size

Decreasing global or sub-model slab mesh size can lead to the convergence of the FEM results with the analytical solution. It is recommended that the mesh size is no greater than 10% of the shortest edge of the slab.

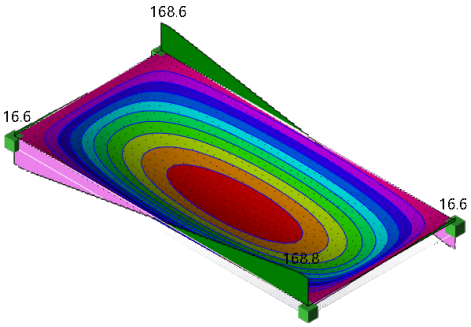
The example below shows four different mesh sizes, equal to 20%, 10%, 5%, and 1% of the shortest slab edge length, respectively.



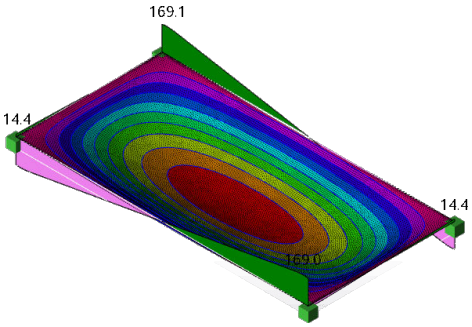
20% - Max deflection: 19.7 mm



10% - Max deflection: 19.6 mm



5% - Max deflection: 19.6 mm



1% - Max deflection: 19.6 mm

NOTE Decreasing the mesh size leads to convergence of the FEM results with the true solution; however, it is important to note that this comes at the price of the solver's performance.

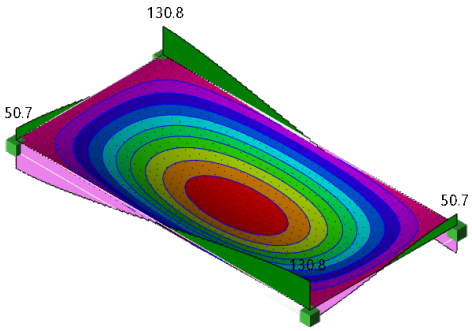
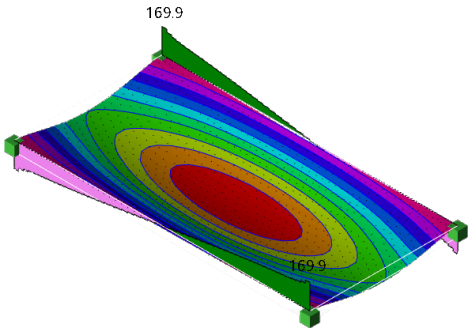
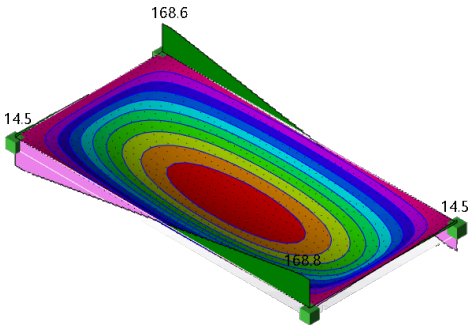
For further details, see Adjust global slab mesh properties (page 586) and Apply different mesh properties at different levels (page 586).

Isotropic vs orthotropic comparison

For this comparison three versions of the model are considered: (i) isotropic slab properties with the slab supported along all four edges (ii) isotropic slab properties with the slab only supported along the two shorter edges, (the

longer beams having been deleted) and (iii) highly orthotropic slab properties with the slab supported along all four edges.

The shears in the supporting beams and the slab deflections are shown below.

Isotropic slab item - *4 edges of support*	**Isotropic slab item** - *2 edges of support*
	
Max deflection: 14.1 mm	Max deflection: 15.9 mm
Orthotropic slab item	
	
Max deflection: 19.7 mm	

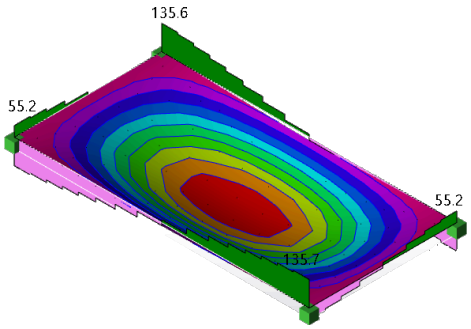
It is normal that in the case of the orthotropic slab some load is transferred onto shorter supporting beams. This has been minimized by adjusting the mesh size and the in-plane shear modulus. The majority of the load is transferred on the beams perpendicular to the principal direction of the slab, resulting in a difference of under 0.8% when compared to the case of the isotropic slab supported along two edges.

Applying/not applying modification factors

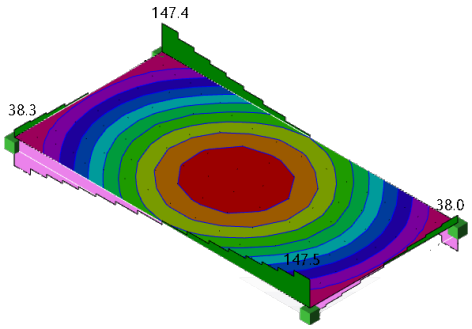
In order to ensure that the material properties as defined in the *Analysis parameters* property grid are passed directly to the solver, a checkbox is provided to allow you to disable the global modification factors for the slab.

For *isotropic* slabs, this functionality also provides a way to Override global stiffness modification factor with slab specific factors (page 604).

When the example model is set to have isotropic properties, the results below show the effect of switching the Apply modification factors option on (left), and off (right).



Max deflection: 13.8 mm



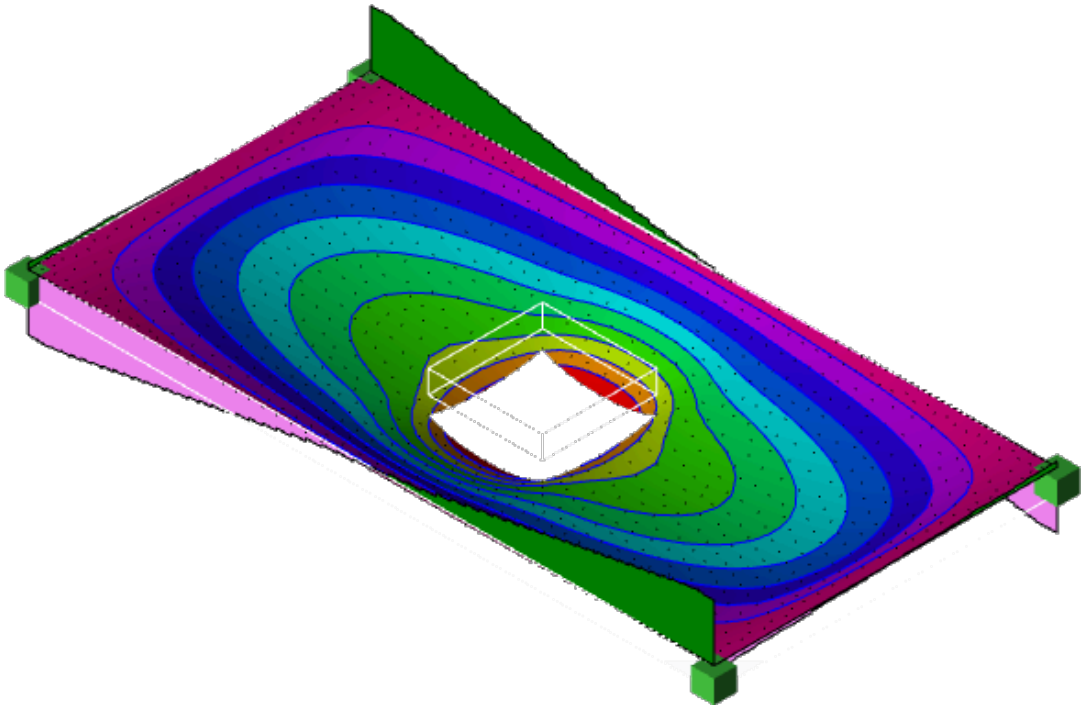
Max deflection: 1.8 mm

Further orthotropic slab modeling considerations

When modeling two-way spanning slabs exhibiting strong orthotropic behavior, there are situations that need to be carefully considered. These include, among others: (i) openings, (ii) free edges perpendicular to the principal direction of the slab item, (iii) cantilevers in the weak direction of the slab item, and (iv) concentrated points of load or support. All those cases, if not modeled correctly, can lead to unexpectedly large deflections and nodal instabilities.

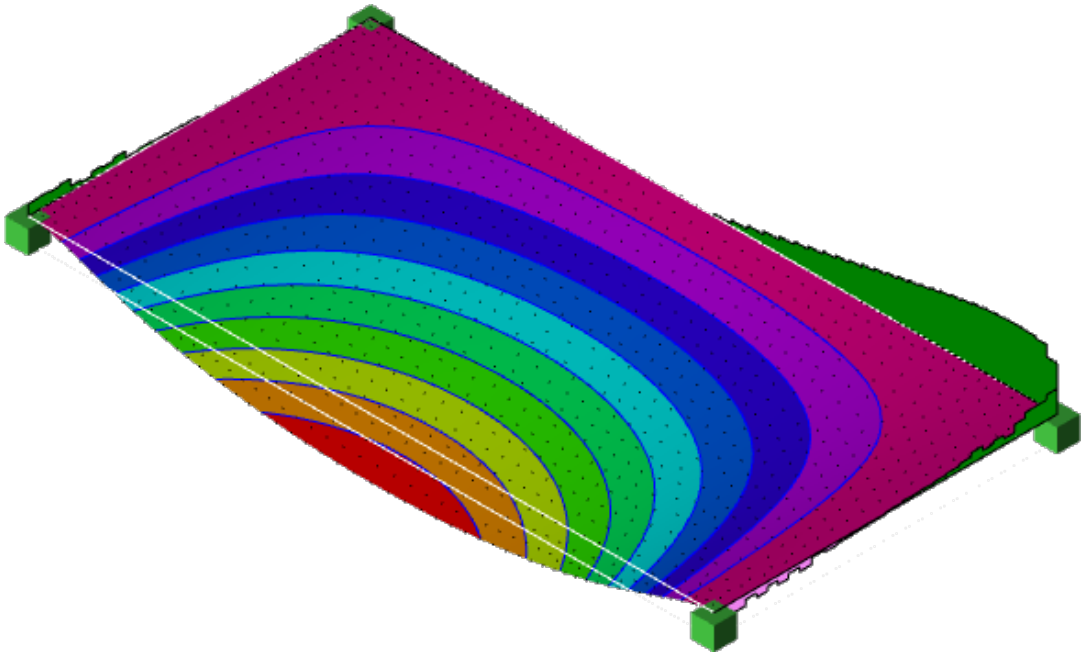
Openings

Creating an opening not supported by any additional beams can lead to increased deflections around the perimeter of the opening.



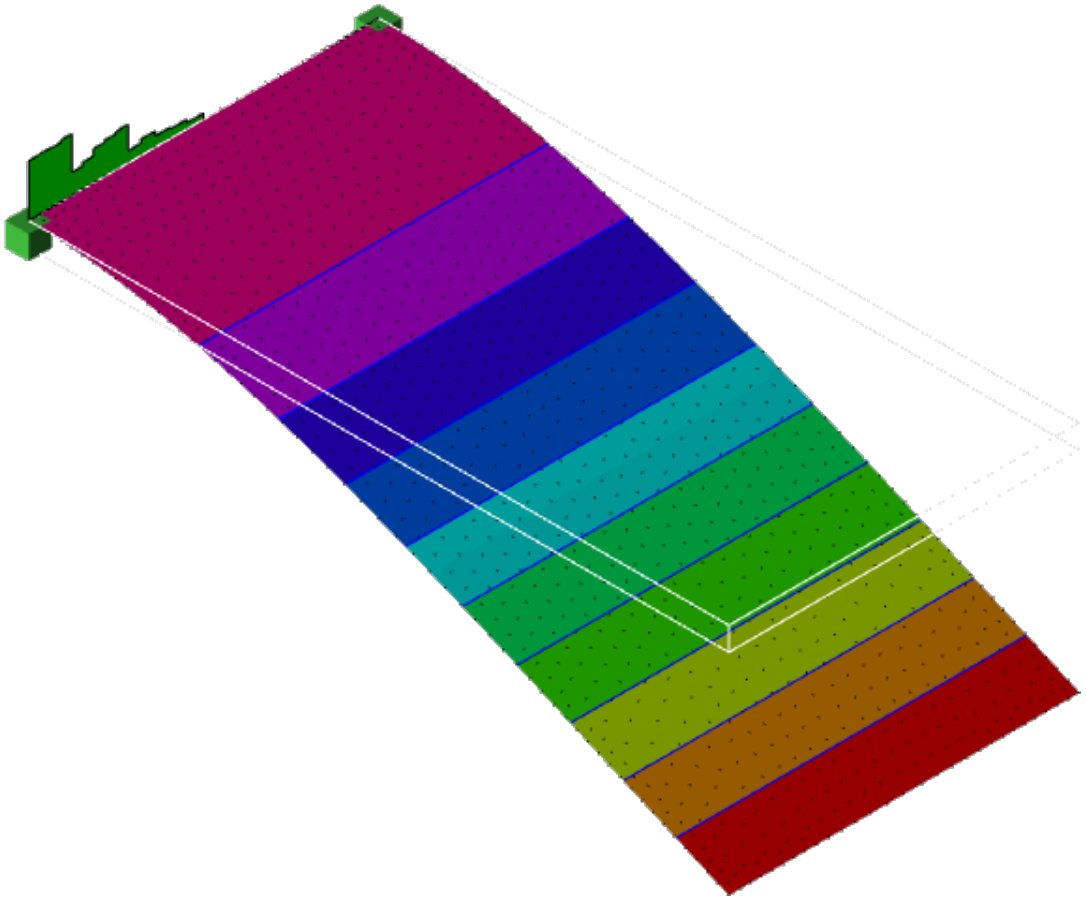
Free edges

Having an orthotropic slab with an unsupported edge perpendicular to the principal direction of the slab can lead to extreme deflections along that edge.



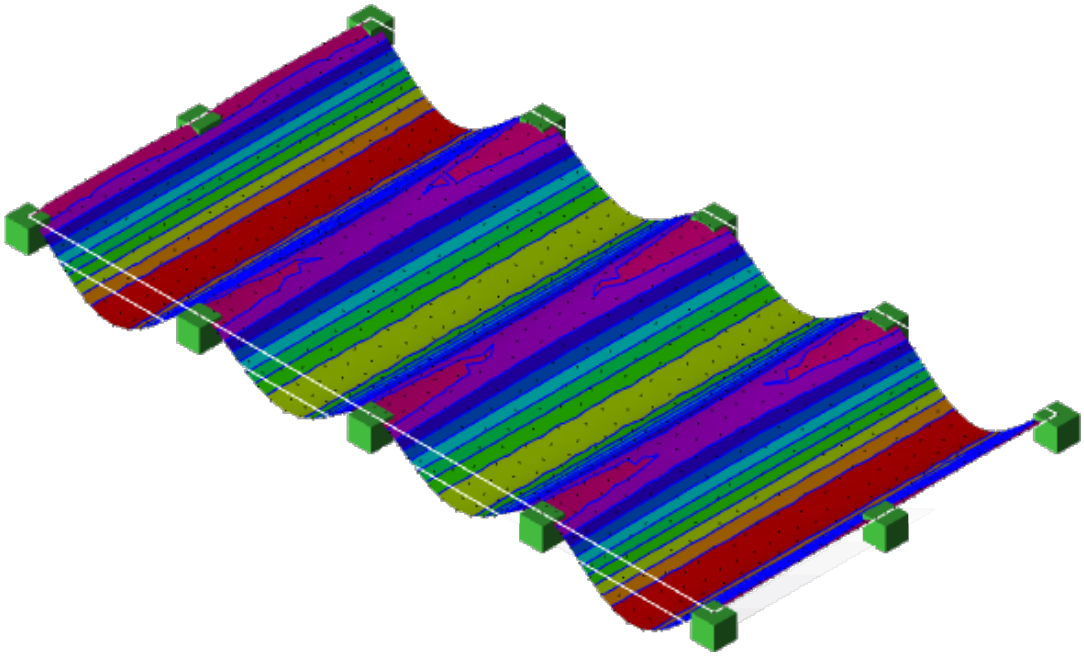
Cantilevers

Strongly orthotropic slabs are unable to maintain stability if only supported along one of the edges perpendicular to the weaker direction of the slab.



Point loads and supports

Supporting an orthotropic slab by point supports or applying concentrated point loads can lead to increased deflections around the point of loads/ supports



Sub models

In both grillage and FE chasedown analysis, (performed as part of the Analyze All, Design Concrete, or Design All processes), Tekla Structural Designer performs the analysis one sub model at a time. The topmost sub model being analyzed first, and then its support reactions are applied as loads for the analysis of the sub model below. The sequence continues until all sub models have been analyzed down to the foundation level.

Slab mesh parameters can be specified individually for each sub model if required.

As part of the chasedown analysis process, (if they don't already exist), sub models are automatically created for every level specified as a **Floor** in the **Construction Levels** dialog box.

You can also define sub models manually in the **Sub Models** dialog box and you can modify the default support conditions applied to individual sub-models, if necessary.

Definitions of sub model characteristics

- **Sub model**: a part of the 3D model between two horizontal sub model divide planes.

  Each sub model contains all members entirely between the two horizontal planes. For the columns, wall and braces split by a divide plane, the stacks and brace length above the top plane are included in the sub model, as are the stacks and brace length below the lower plane.
- **Sub model divide planes**: horizontal planes that you can add, delete or move in the 3D structure. Sub model divide planes are notional and infinite. Tekla Structural Designer only allows the planes to cut through the structure where they only split the following members:
  - Column stacks
  - Wall stacks
  - Steel braces
- **Sub model supports**: the artificial supports that Tekla Structural Designer defines for the column and wall stack ends and braces that pass through the divide planes
- **Structure supports**: the supports that the user has defined in the 3D structure
- **Column and wall stacks**: the span length of a column or wall.
- **Volume of the sub model**: the 3D space that exists between any two adjacent sub model divide planes.

Basic rules of sub models

When all sub models are considered together, they form the complete structure. Only column stacks, wall stacks, and braces that are split by sub model divide planes can be in several sub models. This means that:

- Each member in the 3D model is in at least one sub model.
- A sub model cannot contain a member that is already in another sub model, unless that member is a column, wall, or brace divided by a sub model divide plane.
- A sub model must contain at least one beam member, one truss member, or one slab item.

See also:

Create sub models (page 617)

Open a 3D view of a sub model (page 618)

Delete sub models (page 618)

Create sub models

Sub models allow you to apply different slab mesh parameters within your structure. To create sub models, see the following instructions.

1. In the **Structure** tree, double-click  **Sub Models**.

   The **Sub Models** dialog box opens.

2. According to your needs, do one of the following:

To	Do this
Automatically create sub models	• Click **Generate**.
Manually create sub models	• According to your needs, click **Insert Above**, or **Insert Below**.

3. If necessary, modify the height above the base of each level in the **Level** field.
4. Click **OK**.

See also

Sub Models dialog box (page 1593)

Sub Model Properties (page 1337)

Open a 3D view of a sub model

To display a sub model in its own 3D view, see the following instructions.

1. In the **Project Workspace**, open the **Structure** tree.
2. Expand the  **Sub Models** branch.
3. Double-click the sub model.

   Tekla Structural Designer opens a 3D view of the selected sub model.

   TIP To open a solver view of the sub model, right-click the sub model, and select **Open solver view**.

Delete sub models

If necessary, you can delete existing sub models in the **Sub Models** dialog box. For detailed information on deleting sub models, see the following instructions.

1. In the **Structure** tree, double-click  **Sub Models**.

   The **Sub Models** dialog box opens.

2. Select the sub model that you want to delete.

3. Click **Delete**.

   Tekla Structural Designer deletes the sub model, and the floors are transferred to the sub model immediately above the deleted one.

Diaphragm action in roof panels and slabs

Roof panels and slabs both have the potential to act as a diaphragms.

Both rigid and semi-rigid diaphragms can be modelled.

- Overview of diaphragm action in roof panels and slabs (page 619)
- Managing diaphragm action in roof panels and slabs (page 622)

Overview of diaphragm action in roof panels and slabs

Roof panels and slabs will both act as a diaphragms provided they have **Include in diaphragm** checked in their properties.

While **Include in diaphragm** is the only property used to determine diaphragm action in roof panels, for slab items the diaphragm properties are also determined by the **solver model** under consideration and the following properties:

- **Diaphragm option** slab item property
- **Decomposition** slab item property
- **Mesh 2-way Slabs in 3D analysis** level or slope property

How these and other choices affect the diaphragm model are described in the topics below.

Include in Diaphragm

Roof panels:

The **Include in diaphragm** choice works as follows:

- On - a diaphragm is created within the roof panel using a mesh of semi-rigid 2D solver elements
- Off - no diaphragm is created

Slab items:

For slab items the **Include in diaphragm** choice is only available when the **Diaphragm option** is Semi-Rigid or Rigid. Individual slab items can then be included or excluded as follows:

- On - a diaphragm is created within the slab item which may take the form of a shell mesh, a semi-rigid mesh, or nodal constraints
- Off - no diaphragm is created

NOTE Where 2 slab items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

NOTE When a slab item is excluded from the diaphragm this has no effect on the mesh of shell 2D solver elements used in some of the solver models for 2-way spanning slabs.

Diaphragm option

The choice of **Diaphragm option** is set in the slab item properties. The chosen option is applied to **all** slab items in the same slab.

The available options are:

- Rigid
- Semi-rigid*****
- None

***** See the following **Decomposition** section for further details of the solver models in which semi-rigid as opposed to shell 2D solver elements are adopted.

How your choice affects the 2D solver element types used depends on the method of decomposition selected.

NOTE For roof panels there is no choice of diaphragm option - it is always treated as semi-rigid.

Decomposition

While roof panels are always one-way spanning, for slab panels you can choose the **Decomposition** method in the slab item properties. The chosen option is applied to **all** slab items in the same slab.

Two options are available: Two-Way Spanning or One-Way Spanning - although for certain slab types the value is fixed as follows:

- Composite Slab - One-Way only
- Precast Slab - One-Way only
- Slab on Beams - Two-Way or One-Way
- Flat Slab - Two-Way only
- Steel Deck - One-Way or Two-Way
- Timber Deck - One-Way only

One-way spanning slabs are unmeshed unless the **Diaphragm option** is set to Semi-rigid, in which case they will be meshed with semi-rigid 2D solver elements.

NOTE Semi-rigid 1-way spanning slabs cannot be designed.

Two-way spanning slabs always adopt a mesh of **shell** 2D solver elements for the FE chasedown and FE Load Decomposition solver models. However for the 3D Analysis and Grillage chasedown solver models they are:

- **unmeshed** - if the Diaphragm option is set to None,
- meshed with **semi-rigid** 2D solver elements - if the Diaphragm option is set to Semi-rigid, or,
- meshed with **shell** 2D solver elements - if the Diaphragm option is set to Rigid.

NOTE Semi-rigid 2-way spanning slabs can be designed, but only using the results from the FE Chasedown, not the 3D Analysis or Grillage Chasedown.

Mesh 2-way Slabs in 3D Analysis

Two-way spanning slabs are always meshed with **shell** 2D solver elements in FE Chasedown and Load decomposition solver models. When the **Mesh 2-way Slabs in 3D Analysis** property (specified in the Level, or Slope properties) is checked the same meshing parameters are then extended to the Grillage chasedown and 3D Analysis solver models.

Summary of diaphragm constraint and mesh type configurations

The configurations of mesh and nodal constraints applied to each solver model resulting from the different permutations of the **Decomposition**, **Diaphragm option**, and **Mesh 2-way slabs in 3D analysis** properties are recapped in the table below.

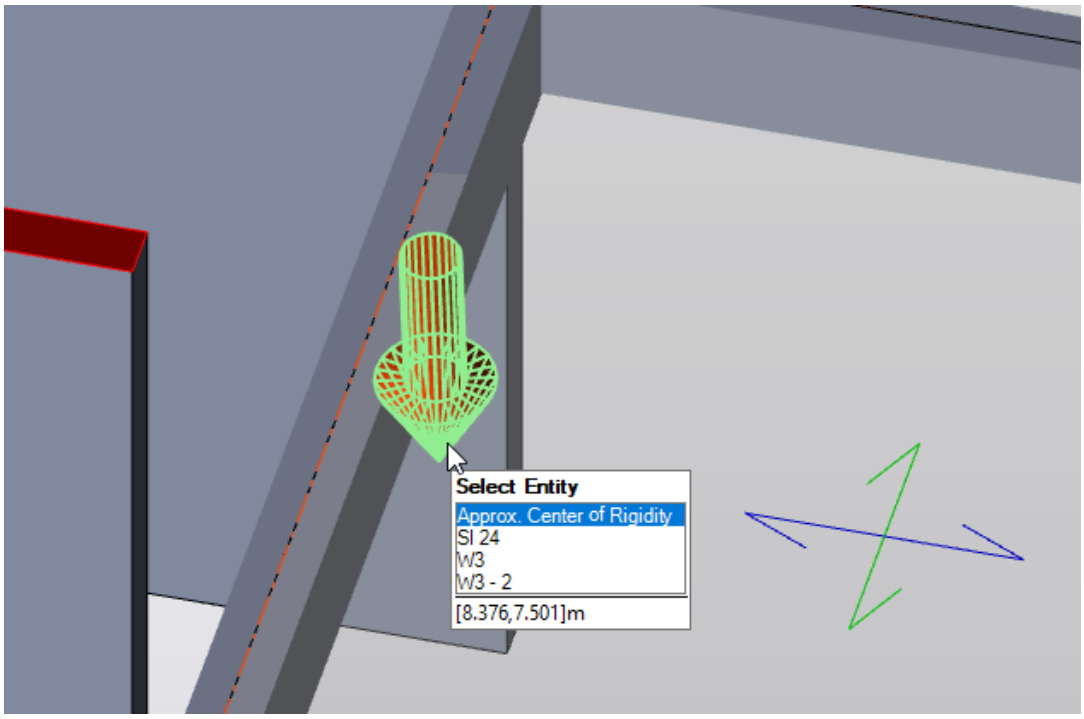
Decom-position	Diaphragm Option	Mesh 2-way slabs in 3D Analysis	FE Load Decomposition & FE Chasedown Models	Grillage Chasedown & 3D Analysis Models
1-way	None	Not Applicable	No mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Not Applicable	Semi-Rigid mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
	Rigid	Not Applicable	No mesh; Nodal constraints	No mesh; Nodal constraints
2-way	None	Yes	Shell Mesh; no nodal constraints	Shell Mesh; no nodal constraints
		No	Shell Mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Yes	Shell mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints

		No	Shell Mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
	Rigid	Yes	Shell Mesh; Nodal constraints	Shell Mesh; Nodal constraints
		No	Shell Mesh; Nodal constraints	No Mesh; Nodal constraints

Diaphragm Center of Rigidity

Any given diaphragm (rigid or semi rigid) has a “center of rigidity” or bending stiffness based on the stiffness of the structure that supports it, (i.e. the column stacks, wall stacks and braced bays that connect to the diaphragm from below).

In any 2D or 3D Scene View, the approximate center of rigidity for each diaphragm can be displayed from the Scene Content side pane by selecting **Center of Rigidity**. If you then hover the cursor over the resulting center of rigidity arrow, its approximate X and Y global coordinate is displayed in the tooltip.



NOTE In Tekla Structural Designer the Center of Rigidity that is displayed is only relevant to the area of the diaphragm and the wall/column stacks/braced bays supporting that diaphragm - not the structure as a whole.

Managing diaphragm action in roof panels and slabs

Tekla Structural Designer forms diaphragms in every slab item of a parent slab when **Diaphragm option** is set to **Rigid** or **Semi-rigid**.

When a rigid diaphragm is specified, Tekla Structural Designer automatically applies nodal constraints to the associated slab nodes.

In turn, when a semi-rigid diaphragm is specified, Tekla Structural Designer creates a mesh of semi-rigid 2D elements within the slab.

Activate rigid diaphragm option within a slab

By default, Tekla Structural Designer automatically creates rigid diaphragm within slabs. If necessary, you can review and modify the setting for specific slabs as follows:

1. Go to the **Project Workspace**.
2. In the **Structure** tree, expand the  **Slabs** branch.
3. Select the slab that contains the diaphragm.
4. In the **Properties** window, set **Diaphragm option** as required.

NOTE You can set the **Diaphragm option** as required in both slab item properties and parent slab properties. Either way, all slab items within the parent slab adopt the new setting.

Activate semi-rigid diaphragm option within a slab

You can form semi-rigid diaphragms in both one-way and two-way spanning slabs. In addition, you can review and modify the diaphragm properties by selecting the slab within which they are located.

1. Go to the **Project Workspace**.
2. On the **Structure** tree, expand the  **Slabs** branch.
3. Select the desired slab.
4. In the **Properties** window, set **Diaphragm option** to **Semi-rigid**.
5. If necessary, to adjust the flexibility of the diaphragm, type a value in the **Divide stiffness by** option.

TIP You can set the **Diaphragm option** as required in both slab item properties and parent slab properties. Either way, all slab items within the parent slab adopt the new setting.

Activate semi-rigid diaphragm option within a roof panel

You can form semi-rigid diaphragms in roof panels as follows:

1. Go to the **Project Workspace**.
2. On the **Structure** tree, expand the  **Roofs** branch.
3. Select the desired roof panel.
4. In the **Properties** window, select **Include in diaphragm**.

NOTE You cannot form a *rigid* diaphragm within a roof panel.

Adjust semi-rigid diaphragm flexibility

When the Diaphragm option is Semi-Rigid a **Divide stiffness by** value can be set in the slab item properties. The same value is applied to **all** slab items in the same slab.

The stiffness determined from the material properties and slab thickness is divided by this value in order to adjust semi-rigid diaphragm flexibility.

Adjust global semi-rigid mesh properties

When a semi-rigid mesh is created the 2D solver element shape (triangular or quad), the degree of mesh uniformity, and mesh type adopted are obtained from the **Structure Properties**. These parameters can be overridden for individual sub-models by setting different values in the respective **Sub Model Properties**.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click  **Structure**.
3. Go to the **Properties** window.
4. Accept or adjust the **Semi-Rigid Mesh Size**.
5. Accept or adjust the **Semi-Rigid Uniformity Factor**.
6. Accept or adjust the **Semi-Rigid Mesh Type**:
   - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
   - **QuadOnly**: only consists of quadrilateral 2D elements.
   - **Triangular**: only consists of triangular 2D elements.

NOTE Beam elements are not split by semi-rigid 2D element nodes.

Apply different semi-rigid mesh properties at different levels

If you need to apply different semi-rigid mesh parameters at a specific level, you can do so by creating sub models.

1. Create a sub model. (page 617)

2. In the **Structure** tree, expand the  **Sub Models** branch.
3. Select the sub model that you created.
4. In the **Properties** window, select the **Override model's** option.
5. Adjust the semi-rigid mesh size, uniformity factor and type according to your needs.

Identify the nodes constrained by rigid diaphragms
In order to see which nodes are constrained by diaphragms, see the following instructions.

1. In the **Status bar** at the bottom of the window, click  **Solver View**.

   A solver view opens. The rigid and semi-rigid diaphragms are represented as different colored shaded planes.

   Nodes that are constrained by a rigid diaphragm:

   - Must lie within, or be on the edge of, the shaded rigid diaphragm plane.
   - Must be solid rather than hollow, or excluded.

   Therefore, the following nodes are not constrained by a rigid diaphragm:

   - Solid nodes that lie outside the shaded rigid diaphragm plane.
   - Hollow, or excluded, nodes.

Exclude individual nodes from a rigid diaphragm
If necessary, you can exclude specific nodes from a rigid diaphragm. In order to do so, see the following instructions.

1. In the status bar at the bottom of the window, click  **Review View**. A review view opens.

2. On the **Review** tab, click  **Diaphragm On/Off**.

   The nodes are displayed as follows:

   - Included nodes are only constrained if they lie within or on the edge of a rigid diaphragm. If they lie outside the rigid diaphragm, they are not constrained by it.
   - Excluded nodes are always unconstrained.

3. Click a node to switch whether it is included in the diaphragm or not.

Exclude slab items from a diaphragm
By default, Tekla Structural Designer forms a rigid diaphragm in all the individual slab items within a slab. However, you can decide to exclude specific slab items. In order to do so, see the following instructions.

1. In the structure view, select the slab item that you want to exclude.
2. In the **Properties** window, clear the **Include in diaphragm** option.

TIP You can also exclude individual slab items graphically in the Review View. On the **Review** tab, click **Diaphragm On/Off** and then click the slab item to include/exclude.

One-way and two-way load decomposition

Apart from when slabs are set to be meshed in the analysis, any panel loads applied to slab panels and roof panels have to be decomposed on to supporting members prior to analysis.

Potential load decomposition methods

In traditional hand calculations a "tributary area" (sometimes called "yield line") loading approach would have been adopted to determine the decomposed loads, but this has limitations when dealing with complex geometry such as:

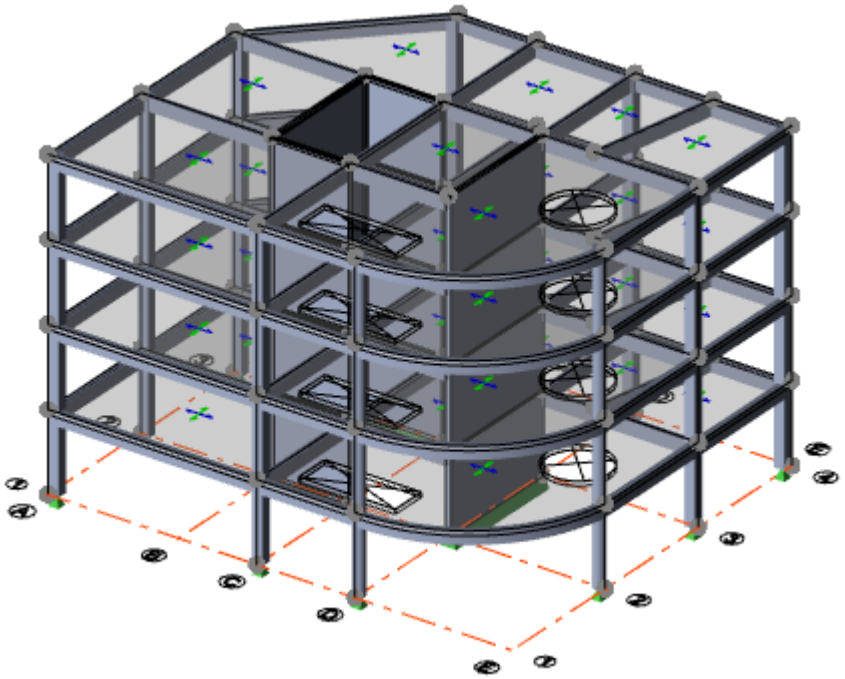
- Slabs not supported on all edges
- Complex panel shapes
- Panels with openings

Also the "tributary area" approach can only approximately handle point, line, and patch loads (by converting them to area loads).

Because of these limitations the "tributary area" method is not used in Tekla Structural Designer - instead a method referred to as FE decomposition is applied instead. This is based on finite element analysis.

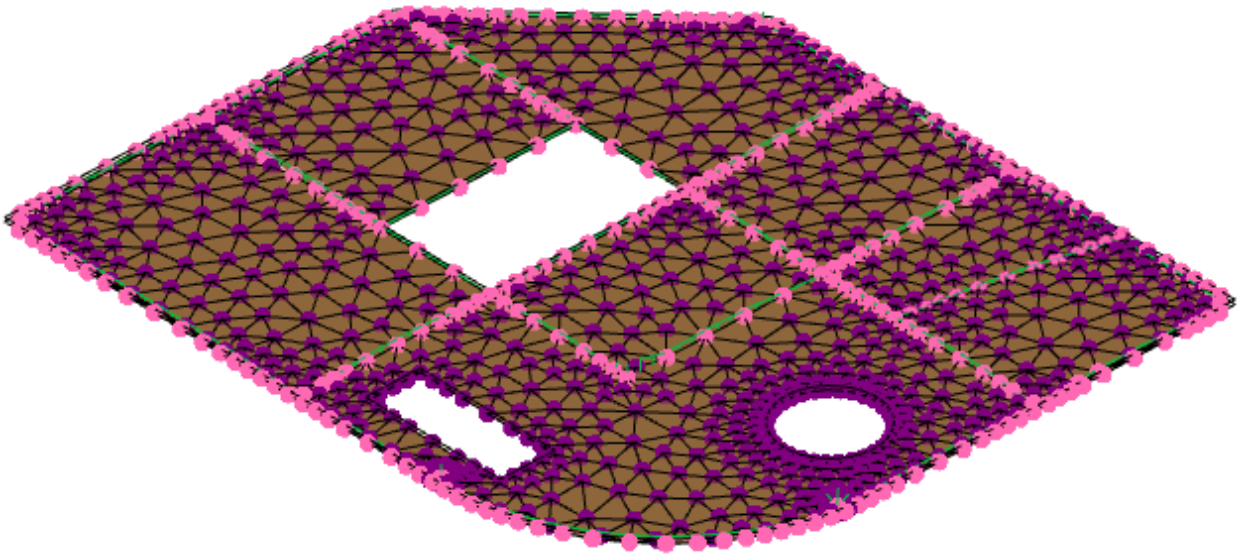
FE decomposition model

The FE decomposition model can be demonstrated using the following two-way slab on beams example.

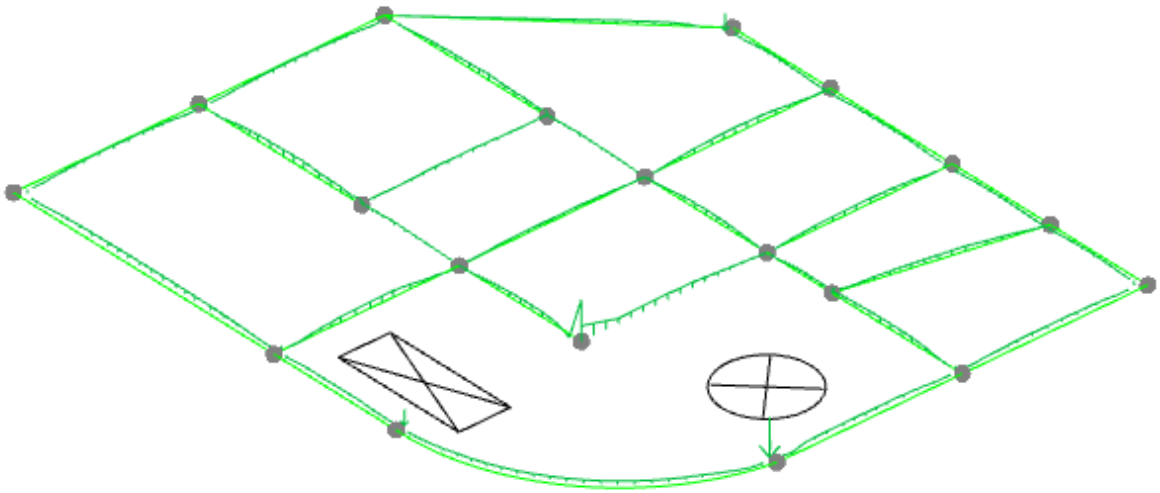


For this type of structure:

- A separate FE decomposition model is created for every floor (or sloped plane).
- Beam column and wall nodes in each FE decomposition model (shown selected in pink below) are all rigidly supported.



• Each FE decomposition model is analysed and the reactions at the rigidly supported nodes are turned back into VDLs along beams and walls, and point loads on columns.

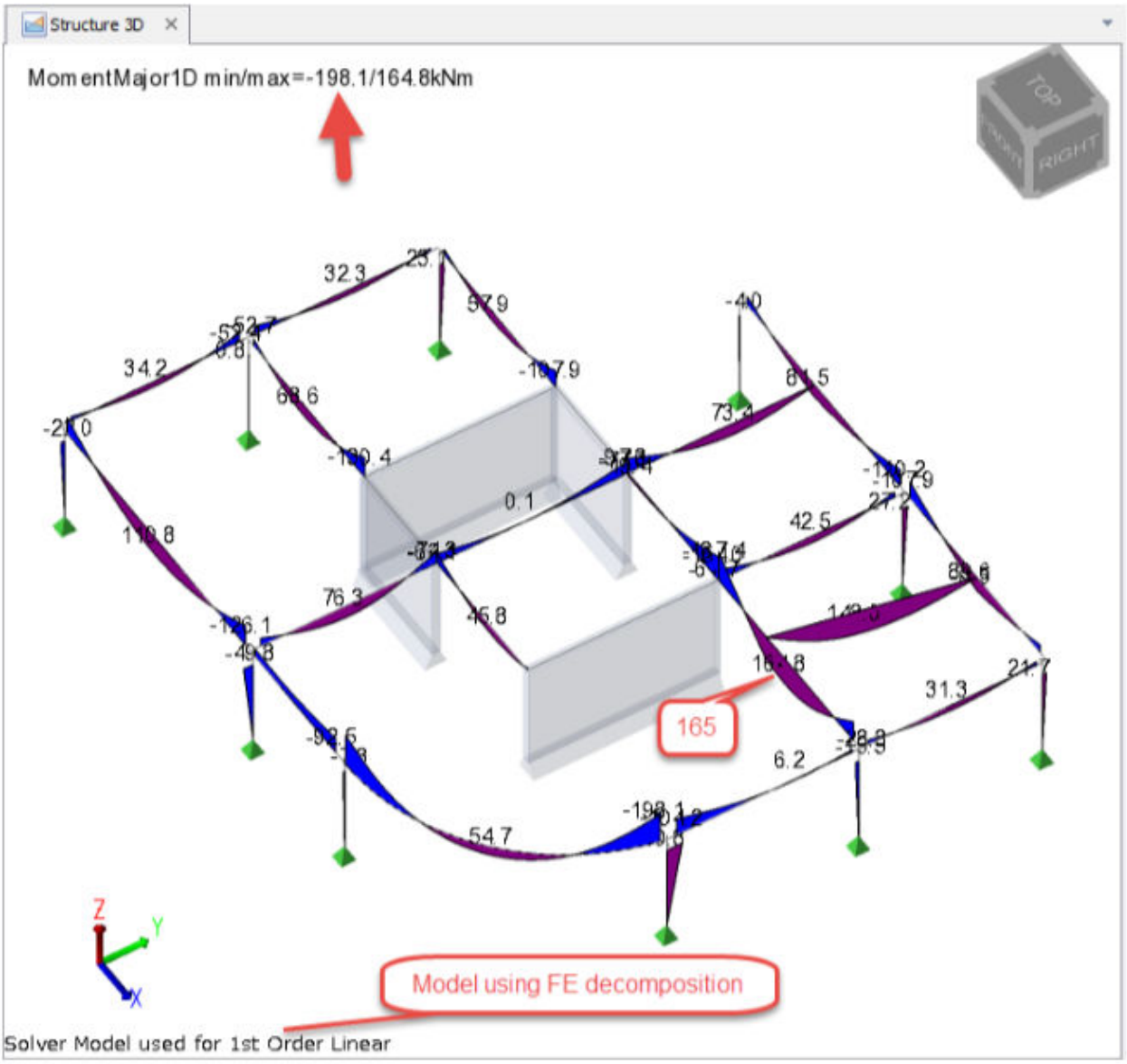


A common question

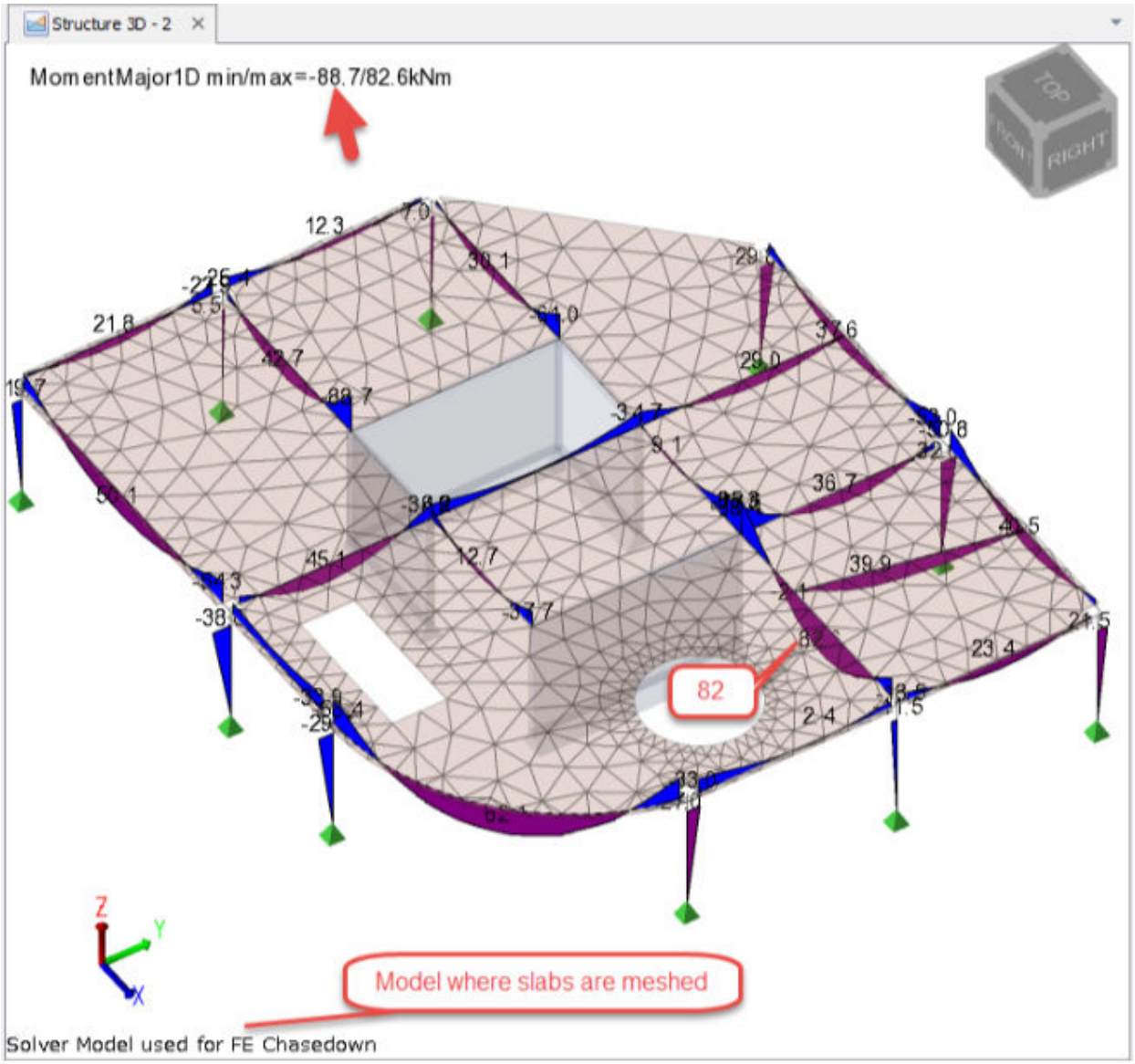
So if the slabs have to be meshed for the FE decomposition during pre-analysis, why not do away with decomposition entirely and just mesh the slabs when performing 3D analysis of the entire model instead?

Because this gives results that you don't like....

In the first run of the model below the slabs are left unmeshed, this requires the loads on the two-way slabs to be FE decomposed prior to analysis. The bending moments from the resulting 3D Analysis are as shown, (max hogging -198, max sagging 165):



The above results can be compared against a second run of the model in which the 2-way slabs are set to be meshed in the 3D analysis. FE decomposition is no longer required and the bending moments from the resulting 3D Analysis are as shown, (max hogging -89, max sagging 82)



In this second run, because the slabs are included in the 3D analysis model, some of the load is being transferred directly to the supporting columns and walls via the slabs themselves. While this is not wrong, it goes against the engineer's expectation - which would be to design the beams on the basis that they transfer all the load to the supporting members.

Controlling decomposition via the 'Mesh slabs in 3D Analysis' level property

As stated in the previous section, the engineer's expectation would be to design the beams on the basis that they transfer all the load to the supporting members. This is achieved in Tekla Structural Designer because (by default),

slabs are **not** meshed in the 3D analysis and grillage chasedown solver models.

Note however that by switching on the **Mesh slabs in 3D analysis** level property the engineer can allow some of the load to be transferred directly to the supporting columns and walls via the slabs.

	Solver model under consideration	
Decomposition method in slab properties	**3D Analysis and Grillage chasedown models**	**FE chasedown model**
Two-way	**Mesh slabs in 3D analysis OFF** (default): • load decomposition performed (2-way)[1]	• load decomposition **not** performed[2]
	Mesh slabs in 3D analysis ON: • load decomposition **not** performed[2]	
One-way (with **Analyze slab OFF**)	**Mesh slabs in 3D analysis OFF** • load decomposition performed (1-way)[1]	• load decomposition performed (1-way)[1]
	Mesh slabs in 3D analysis ON • load decomposition performed (1-way)[1]	
One-way (with **Analyze slab ON**)	**Mesh slabs in 3D analysis OFF** • load decomposition performed (2-way with orthotropic properties)[1]	• load decomposition **not** performed[2]
	Mesh slabs in 3D analysis ON • load decomposition **not** performed[2]	
[1] - Beams designed on the basis that they transfer all the load to the supporting members [2] - Beams designed on the basis that slabs transfer some of the load to the supporting members		

Decomposition rules for the different types of panel and overlapping panels

The way in which Tekla Structural Designer decomposes panel loads depends on how the slabs/panels are modeled and how they are spanning.

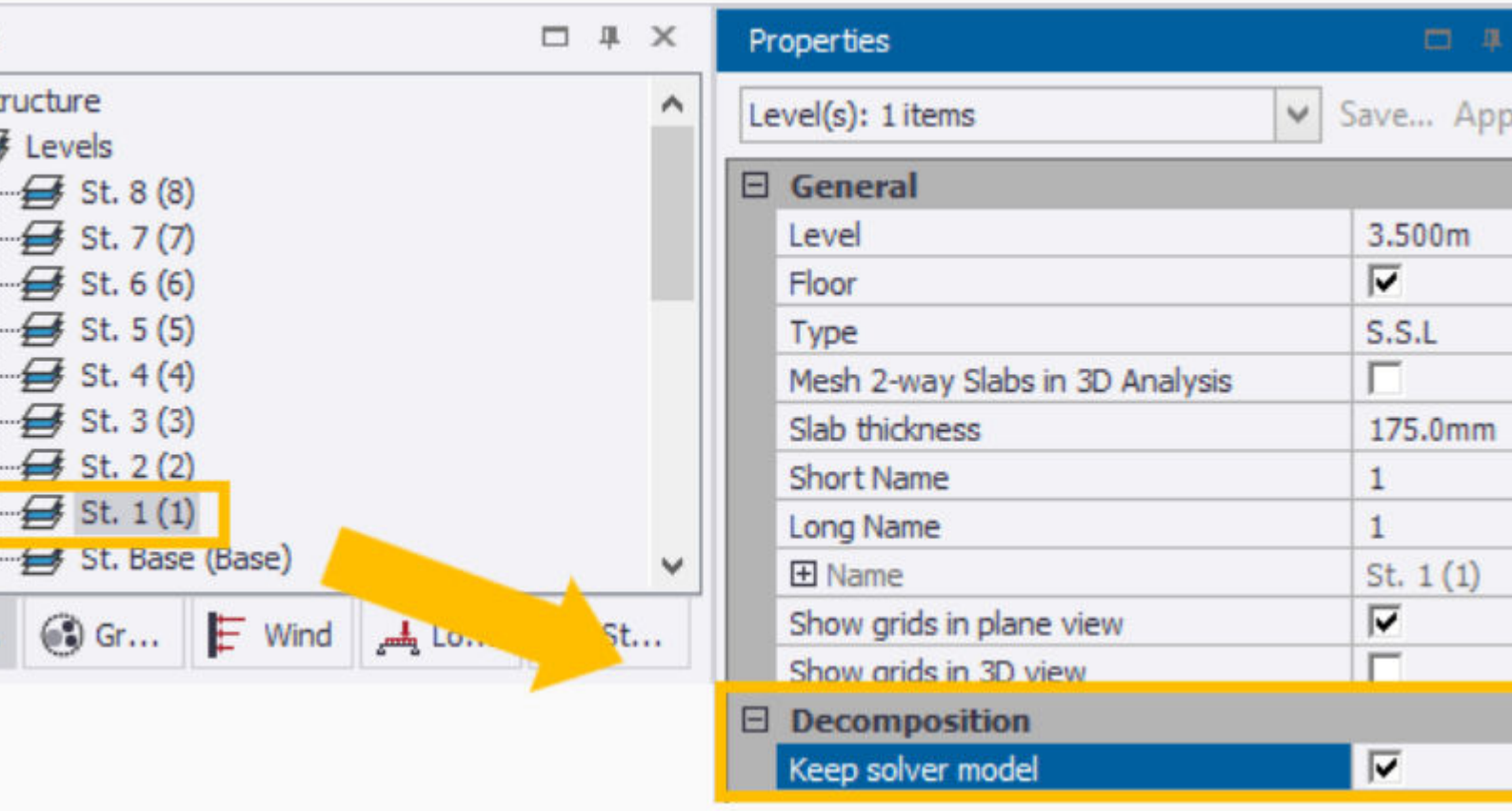
Slab/Panel type	Decomposition
One-way spanning slab item	• At levels where the **Mesh slabs in 3D analysis** option is selected, load decomposition is not required for those slabs for which the **Analyze slab** slab item property is also selected. **NOTE** Where openings have been defined, any portion of a panel load that lies within the opening will not be decomposed. • In all other situations, panel loads applied to the slab are 1-way decomposed directly on to supporting members/two-way spanning slab edges before performing 3D analysis. The rotation angle of the slab item determines the decomposition direction. **NOTE** Where openings have been defined, these are ignored and thus have no impact on load decomposition.
Roof panel	Panel loads applied to the roof panel are 1-way decomposed directly on to supporting members/two-way spanning slab edges before performing 3D analysis. The rotation angle of the roof panel determines the decomposition direction.
Two-way spanning slab item	• At those levels where the **Mesh slabs in 3D analysis** option is selected, load decomposition is not required.

Slab/Panel type	Decomposition
	• At other levels, any loads applied to two-way slabs are automatically 2-way decomposed back to supporting members during the 3D pre-analysis stage, immediately prior to the 3D analysis stage. **NOTE** Where openings have been defined, any portion of a panel load that lies within the opening will not be decomposed.
Roof panel overlapping a one-way spanning slab item	Panel loads are 1-way decomposed directly on to supporting members/ two-way spanning slab edges before performing 3D analysis. The rotation angle of the **slab item** (and not the roof panel) determines the decomposition direction. **NOTE** Any openings in 1 way slabs are ignored and thus have no impact on load decomposition.
Roof panel overlapping a two-way spanning slab item	• If there are no openings in the slab, decomposition is as per **Two-way spanning slab item** type described above. • Where openings exist, any panel load applied inside the opening is first one-way decomposed in the direction defined by the rotation angle of the roof panel on to the two-way slab at the edge of the opening. It is then further decomposed back to supporting members as per **Two-way spanning slab item** type described above. **NOTE** This does not apply when the panel load is an area load that has been applied to the slab item (as

Slab/Panel type	Decomposition
	opposed to the roof panel), or a level load, or a slab load. In each of these instances the load will automatically be boxed out around the opening.

How to view the solver model used for load decomposition

A separate FE decomposition solver model is created for each plane, but this can only be viewed provided **Keep solver model** has been selected in the level properties.



1. Open a 2D view of the required level.
2. Display the Level properties in the
   a. A separate FE decomposition solver model will only be used provided you have not chosen **Mesh 2-way Slabs in 3D Analysis**

b. Furthermore, you will only be able to view this FE decomposition solver model provided **Keep solver model** has been selected.

3. If necessary, re-run the analysis to create the FE decomposition solver model.
4. Set the view regime to  **Solver View**
5. From the right-click context menu select **Solver Models > Solver Model used for Load Decomposition**

FE meshed walls

If concrete walls have been modeled as meshed, when the structure is analyzed they will initially adopt the mesh type and size specified in Structure Properties (page 1331). You can however override these in order to apply a user defined mesh to individual walls if necessary. Extra consideration of the analysis model may be required if openings have been placed within meshed walls.

By default FE meshed walls are created with **isotropic** analysis properties. **Othotropic** properties can be specified if required to model walls that are assumed to predominantly "one-way spanning", such as masonry wall panels and cross-laminated timber (CLT) panels.

Click the links below to find out more:

- Mesh type and size guidelines for FE walls (page 635)
- Adjust the wall mesh parameters specified in structure properties (page 639)
- Apply different mesh parameters to individual walls (page 640)
- Review the wall mesh before the analysis (page 640)
- Meshed wall openings analysis model (page 641)
- Orthotropic walls (page 644)

See also

How meshed walls are represented in solver models (page 554)

Mesh type and size guidelines for FE walls

When creating walls, you first have to choose whether to use the mid-pier or meshed modeling option. Since mid-pier walls have a number of limitations the expectation is that in most cases the meshed option will be preferred.

Assuming you choose the meshed option, you then also need to consider what is an appropriate mesh size, and what type of 2D element (i.e. QuadDominant,

Quad, or Triangular) is most appropriate. The defaults in Tekla Structural Designer are:

- wall mesh horizontal and vertical size: 1m (3')
- 2D element type: QuadDominant

Benchmarking quad and triangular meshes against the equivalent mid-pier model and theory

A small laterally loaded wall model is investigated in order to benchmark the quad and triangular mesh types against an equivalent mid-pier model and theoretical results.

Model definition

A 12 tall x 6 long x 0.2 thick cantilever wall is subjected to a horizontal tip load of 100 and vertical tip load of -1000.

$E = 13.4 \times 10^6$

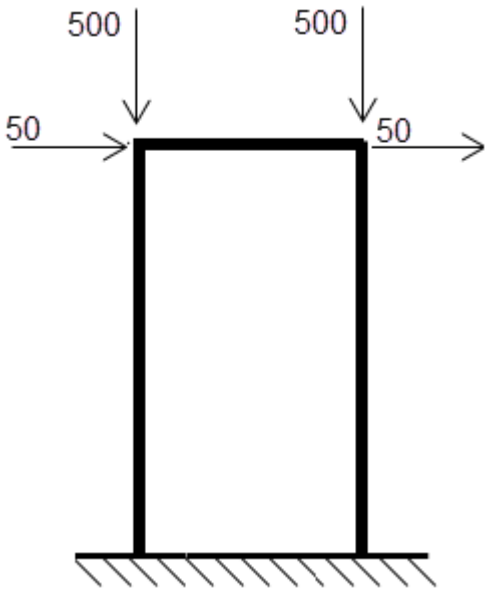
$G = 5583 \times 10^3$

$P = 100$

$L = 12$

$h = 6$

$t = 0.2$



Assumptions

Flexural and shear deformations are included.

Theory

Lateral deflection of vertical cantilever with rectangular section of length L, shear area A_{vx} and horizontal tip load P:

$$v(x) = \frac{PL^3}{3EI_x} + \frac{PL}{GA_{vx}}$$

Substituting for the wall thickness, t and depth, h this becomes:

$$v(x) = \left[\frac{4PL^3}{Eth^3}\right] + \left[\frac{6PL}{5Gth}\right]$$

NOTE The first term accounts for the deflection due to *flexure*, the second term accounts for the deflection due to *shear*. A usual reference for this is: https://en.wikipedia.org/wiki/Timoshenko%E2%80%93Ehrenfest_beam_theory

---

Models

Three models are initially considered as follows:

1. FE meshed wall (quad elements - default mesh size)
2. FE meshed wall (triangular elements - default mesh size)
3. Mid-pier wall

The FE meshed model parameters are then varied to investigate potential mesh sensitivity.

NOTE Because the model consists of a single panel the QuadDominant and Quad meshing options result in identical meshes of quad elements only. In more complex models the QuadDominant option will result in a predominantly quad mesh, but with occasional triangular elements inserted where necessary.

Key results

Result	Theoretical formula	Theoretical value	FE meshed quad elements	FE meshed tri elements	Mid-pier
Mesh size			72 elements	170 elements	N/A

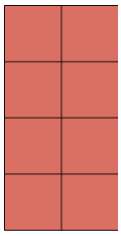
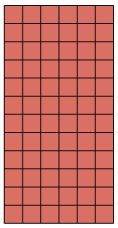
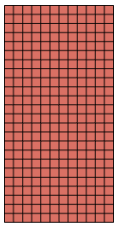
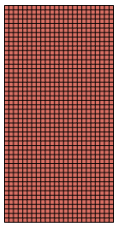
Result	Theoretical formula	Theoretical value	FE meshed quad elements	FE meshed tri elements	Mid-pier
Lateral Support Reaction	-P	100	100	100	100
Support Moment	PL	-1,200	-1,200	-1,200	-1,200
Lateral Deflection	$\left[\frac{4PL^3}{Eth^3}\right]+\left[\frac{6PL}{5Gth}\right]$	0.001409	0.001380	0.001333	0.001409

Comment on above results

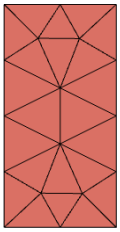
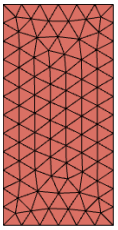
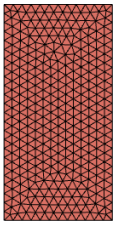
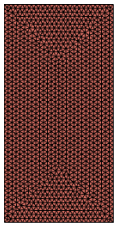
As is to be expected, the lateral deflection reported for the mid-pier model exactly matches the value predicted by beam theory. What may at first seem surprising is that the lateral deflections reported for both the meshed models are slightly different. It must be borne in mind that while the answer given by beam theory is an accepted close approximation, it is assuming that plane sections remain plane, so a small difference in the deflection results when using an FE meshed model might be expected.

Mesh sensitivity study (quad elements)

To investigate the sensitivity of the lateral deflections to mesh size the analysis is repeated using a range of mesh sizes.

Result	Beam Theory	Quad elements			
		8 shells	72 shells (default)	288 shells	1152 shells
					
Lateral Deflection	0.001409	0.001334	0.001380	0.001386	0.001388

As shown above, the results for quad elements do not indicate any significant mesh sensitivity over a wide range of mesh sizes.

Result	Beam Theory	Triangular elements			
		22 shells	170 shells (default)	666 shells	2652 shells
					
Lateral Deflection	0.001409	0.001065	0.001333	0.001373	0.001384

The above results for triangular elements show a much greater mesh sensitivity. The model with 666 shells is the first that could be regarded as a reasonably converged result.

Conclusion

When using FE meshed elements for walls, quad elements are to be preferred as they are not particularly sensitive to the mesh size that is adopted. The same can not be said to be true for triangular elements. For this reason we would suggest using either the **Quad dominant**, or **Quad only** mesh types, for walls, with the **Tri only** mesh type being non-preferred.

While mid-pier walls have been shown to give accurate results for this simplistic example, they have significant limitations in real models, one of the main ones being that they do not cater for wall openings. For that reason they are also not generally recommended.

Adjust the wall mesh parameters specified in structure properties

Initially, the same mesh parameters are applied to all meshed walls in the structure. In order to adjust these, see the following instructions.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click  **Structure**.
3. Go to the **Properties** window.
4. If necessary, in **Wall Mesh Type**, change the shape of the wall mesh:
   - **QuadDominant**: (Default) mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
   - **QuadOnly**: only consists of quadrilateral 2D elements.

- **Triangular**: only consists of triangular 2D elements.

  NOTE The **Triangular** option is *not* generally recommended for modeling walls. While for out of plane results there is little difference between quads and triangular elements, for in-plane results triangular elements are constant stress and quads are bi-linear.

5. Adjust the horizontal and vertical sizes of the wall mesh according to your needs.

See also

Apply different mesh properties to an individual wall (page 640)

Apply different mesh parameters to individual walls

If you need to apply different wall mesh parameters to specific walls, you can override the model mesh properties, and modify the properties of the individual wall. For more information, see the following instructions.

1. In the model, select the walls that you want to modify.
2. Go to the **Properties** window and locate the parameters under the **Meshing** heading.
3. Select **Override model's**
4. If required, in **Wall Mesh Type**, change the shape of the wall mesh:
   - **QuadDominant**
   - **QuadOnly**
   - **Triangular**
5. Adjust the horizontal and vertical sizes of the wall mesh according to your needs.

Review the wall mesh before the analysis

In order to review the existing slab mesh in your model before running a particular analysis, see the following instructions.

1. In the **Status bar** at the bottom of the window, click  **Solver View**.

   The solver view opens.
2. Right-click anywhere in the solver view.
3. In the context menu, go to **Solver models**.
4. In the list, select the solver model appropriate to the analysis that you want to run.

If the wall mesh is applicable to the selected solver model, it is displayed.

NOTE The wall mesh is not displayed in the working solver model. This is because the working solver model displays the model in its form before any analysis, and 2D elements are only formed during the analysis.

See also

View and manage solver models (page 537)

Meshed wall openings analysis model

Limitations of wall openings

1. If you have specified a door or window opening in a wall panel you must model the wall as a meshed wall, otherwise a "Walls with openings have a mid-pier" validation error is displayed and the analysis will not proceed.
2. A "Panel contains openings - these are ignored in design" warning will always be issued when a wall containing openings is designed.

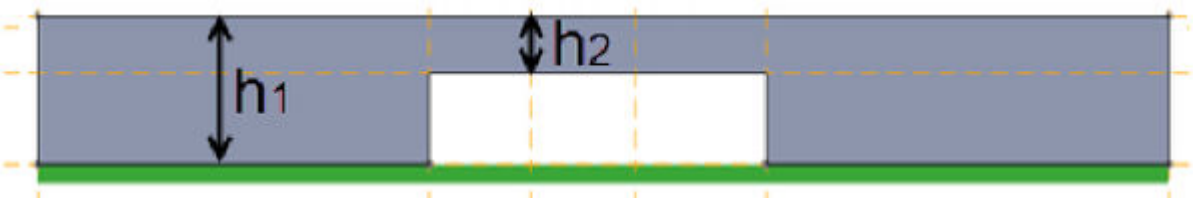
   When you encounter this warning we recommend that you use result lines to determine the reinforcement requirements around the opening. For further details, see the topic Manage, display and design result lines (page 698), and the related video Interactive design using Result lines.

   As well as taking stock of the design implications; you need also to consider if the analysis model is appropriate, as potentially it may not reflect your original intention. In certain situations the **Alternative model for wall openings** (described below) may prove to be a better solution.

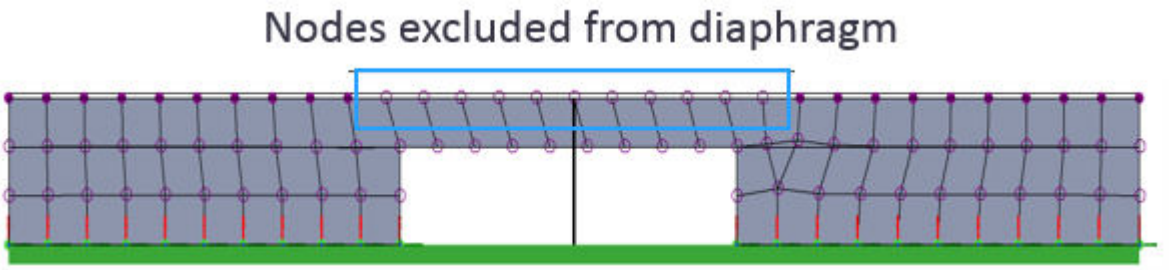
Analysis model applied to meshed wall panels with openings

If an opening is introduced in a meshed wall, the properties of the "lintel" wall beam directly above the opening are automatically adjusted in order to prevent the panel being unrealistically stiff. The adjustments that are applied are as follows:

• wall beam properties in the lintel use the lintel depth (h_2), rather than the panel depth (h_1)



• wall beam nodes in the lintel are removed from the slab diaphragm



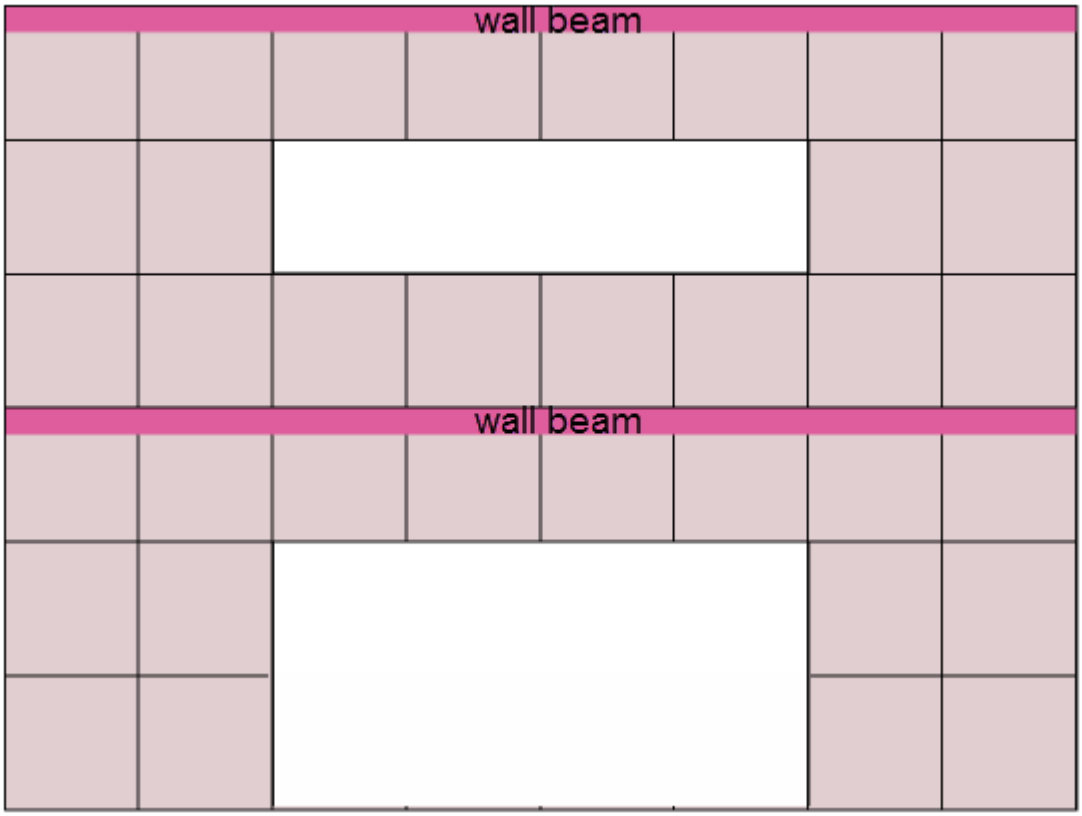
Modeling in this way the lintel becomes less stiff resulting in improved wall results when subject to gravity and lateral loading.

Alternative model for wall openings

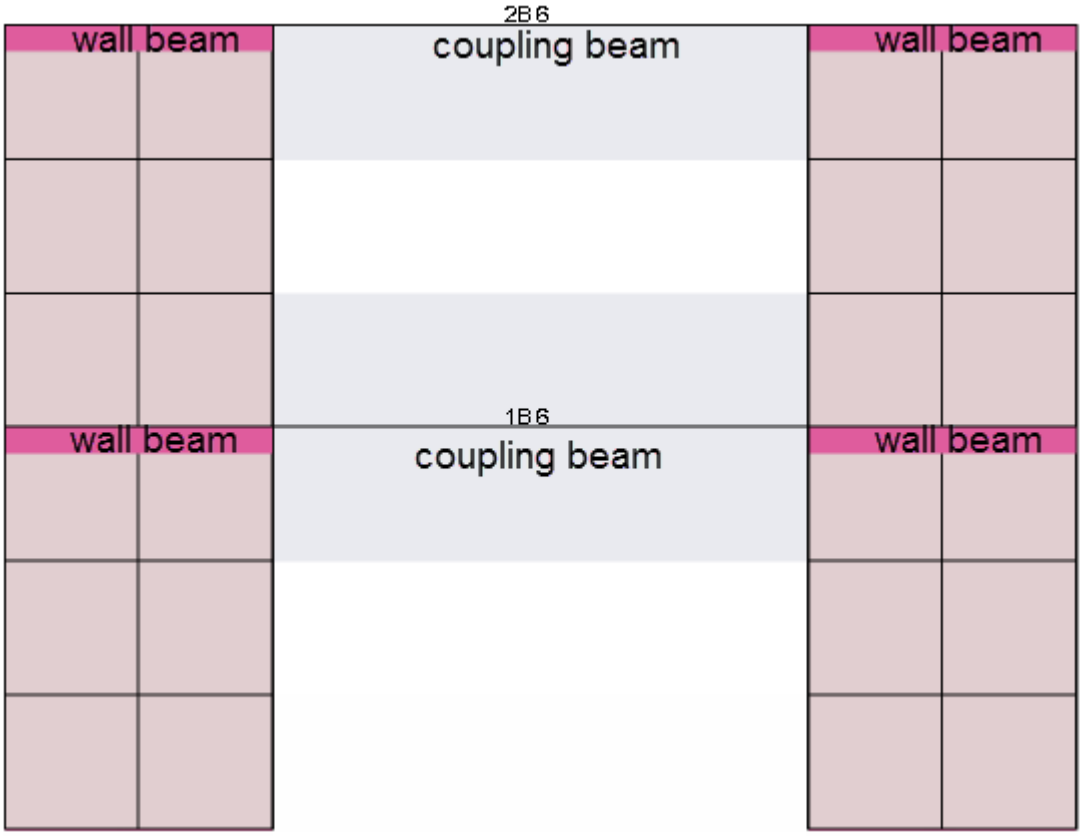
If the presence of an opening would form a beam like strip above or below the opening, you are advised to create separate wall panels to each side of the opening and then model the strip between the panels with a connecting beam ('coupling beam').

This method can be demonstrated by considering the below example, consisting of a two story wall with a large opening at each level.

If the openings were to be created as a window and door the resulting model would be as shown:



However, by separating the wall into discrete panels and inserting coupling beams you obtain an alternative model as below:



Such an idealisation enables the panels either side of the openings to be designed for their respective forces and enables the strips between the openings to be designed as beams.

Of course, this approach will require some additional detailing, but that would have been the case anyway had the openings been added and subsequently ignored by the design.

See also

Create door or window openings (page 204)

Manage, display and design result lines (page 698)

Orthotropic walls

There are two ways to define an orthotropic meshed shear wall in Tekla Structural Designer: you either specify relative stiffness factors, or you enter material properties manually.

Both methods (as described in more detail below) require you to select the **Orthotropic** option in the wall properties. This option actually has two practical uses:

- It enables different stiffnesses to be set for the two orthogonal (X and Y) directions of walls thus making them *orthotropic* rather than *isotropic* (same stiffness in both directions). This can be used to model walls that exhibit orthotropic material properties such as masonry wall panels and cross-laminated timber (CLT) panels.
- For isotropic walls, it also provides a way (as described below) to override the global stiffness modification factor with a modification factor specific to the selected wall only.

Define an orthotropic masonry wall using section adjustment factors

This option is only available for the US (AISC/ACI) region and then only when the wall construction material has been defined using Tedds.

1. Select the wall to be edited.
2. In the **Properties** window, open:
   - **All panels > Analysis & design control** - if you want to apply the same orthotropic properties to all panels in a multi-panel wall, or,
   - **Panel <*panel number*> > Analysis & design control** - if you want to apply the orthotropic properties to a specific panel only.
3. Select **Orthotropic**
4. Set **Calculation** to **Apply section adjustment factors**
   - When **Orthotropic** is unselected, the E and G values that are displayed are the isotropic values given in the materials database.
   - When **Orthotropic** and **Apply section adjustment factors** are selected, because the wall construction material has been fully defined using Tedds, sufficient data exists to allow the orthotropic properties to be automatically determined in Tekla Structural Designer. Values will be populated when the analysis is completed.

Define an orthotropic masonry wall by entering material properties manually

As an alternative to calculating the orthotropic material properties using X and Y factors, the properties can be also set manually if required. The properties take default values based on their state prior to deselecting the checkbox.

To increase the wall panel's orthotropicity, except adjusting Young's moduli in the local X and Y directions, respectively, one might consider modifying the in-plane shear modulus, Gxy, as well.

To define properties manually in this way:

1. Select the wall to be edited.
2. In the **Properties** window, open:
   - **All panels > Analysis & design control** - if you want to apply the same orthotropic properties to all panels in a multi-panel wall, or,

- **Panel <*panel number*> > Analysis & design control** - if you want to apply the orthotropic properties to a specific panel only.

3. Select **Orthotropic**
4. Set **Calculation** to **No calculation**
5. Enter the required values of Young's moduli **Ex** and **Ey**.

Example: In a particular model in order to achieve highly orthotropic behavior you might reduce Ey while maintaining the Ex unchanged. To increase the wall panel's orthotropicity, one might consider modifying the in-plane shear modulus, Gxy, as well.

Define an orthotropic masonry wall using X and Y relative stiffness factors

1. Select the wall to be edited.
2. In the **Properties** window, open:
   - **All panels > Analysis & design control** - if you want to apply the same orthotropic properties to all panels in a multi-panel wall, or,
   - **Panel <*panel number*> > Analysis & design control** - if you want to apply the orthotropic properties to a specific panel only.
3. Select **Orthotropic**
4. Set **Calculation** to **Apply X and Y factors**
5. Enter the required values of **X factor - along the length of the wall** and **Y factor - along the height of the wall**.

   Adjusting the factors in this way allows you to automatically set material orthotropicity using the method described in Using the 'Calculate properties' functionality. Decreasing one of the factors can have a significant impact on internal forces in the wall panel, as well as on the load transfer.

Example: In a particular model in order to achieve highly orthotropic behavior you might enter a Y factor of 0.001 while maintaining the X factor as 1.0.

Define an orthotropic wall (other than masonry) by entering material properties manually

As an alternative to calculating the orthotropic material properties using X and Y factors, the properties can be also set manually if required. The properties take default values based on their state prior to deselecting the checkbox.

To increase the wall panel's orthotropicity, except adjusting Young's moduli in the local X and Y directions, respectively, one might consider modifying the in-plane shear modulus, Gxy, as well.

To define properties manually in this way:

1. Select the wall to be edited.
2. In the **Properties** window, open:

- **All panels > Analysis & design control** - if you want to apply the same orthotropic properties to all panels in a multi-panel wall, or,
- **Panel <*panel number*> > Analysis & design control** - if you want to apply the orthotropic properties to a specific panel only.

3. Select **Orthotropic**
4. Clear **Calculate properties**
5. Enter the required values of Young's moduli **Ex** and **Ey**.

Example: In a particular model in order to achieve highly orthotropic behavior you might reduce Ey while maintaining the Ex unchanged. To increase the wall panel's orthotropicity, one might consider modifying the in-plane shear modulus, Gxy, as well.

Define an orthotropic wall (other than masonry) using X and Y relative stiffness factors

1. Select the wall to be edited.
2. In the **Properties** window, open:
   - **All panels > Analysis & design control** - if you want to apply the same orthotropic properties to all panels in a multi-panel wall, or,
   - **Panel <*panel number*> > Analysis & design control** - if you want to apply the orthotropic properties to a specific panel only.
3. Select **Orthotropic**
4. Select **Calculate properties**
5. Enter the required values of **X factor - along the length of the wall** and **Y factor - along the height of the wall**.

   Adjusting the factors in this way allows you to automatically set material orthotropicity using the method described in Using the 'Calculate properties' functionality. Decreasing one of the factors can have a significant impact on internal forces in the wall panel, as well as on the load transfer.

Example: In a particular model in order to achieve highly orthotropic behavior you might enter a Y factor of 0.001 while maintaining the X factor as 1.0.

Override global stiffness modification factor with wall specific factors

Selecting the **Orthotropic** property provides access to X and Y factors which can be used override the global stiffness modification factor. Although the implication is that you will apply these to make the wall *orthotropic* there is nothing to prevent you from specifying identical X and Y values. In this case, due to lack of differences between X and Y factors, the wall will be analyzed as an *isotropic* one. When used in this way, the **Orthotropic** property can be

activated for isotropic walls solely to override the global stiffness modification factor with a wall specific factor.

This is done as follows:

1. Select the FE meshed wall(s) to be have an override applied.
2. In the **Properties** window, navigate to **All panels > Analysis & design control**
3. Select **Orthotropic**
4. Select the calculation method:
    a. For concrete walls, select **Calculate properties**
    b. For masonry walls, in the **Calculation** list, select **Apply X and Y factors**
5. Enter the *same* value of **X factor** and **Y factor**.
6. Unselect **Apply modification factors** The value entered will be used for the selected walls in the place of the global stiffness modification factor.

See also

Orthotropic wall implementation (page 648)

Orthotropic wall implementation

When a wall is selected as orthotropic, the wall beam element is treated as isotropic with material properties similar to that of orthotropic walls in 'Y' direction i.e.

$E_{X_beam} = E_{Y_beam} = E_{Y_wall}$

A wall beam element common between two wall panels adopts the property of the wall panel with the lesser E_y value.

The rest of the properties for analysis listed below are calculated in a similar way to those for orthotropic two way meshed slabs.

- E_x - Young's modulus in the local X direction
- E_y - Young's modulus in the local Y direction
- G_{xy} - Shear modulus in the local XY plane (in-plane)
- G_{xz} and G_{yz} - Shear modulus in the local XZ and local YZ planes (out-of-plane).
- G_{yz} - Shear modulus in the local YZ plane (out-of-plane)
- ν_x - Poisson's ratio in the local X direction
- ν_y - Poisson's ratio in the local Y direction

Using the 'Calculate properties' functionality

The material properties are determined based on the X and Y stiffness modification factors, denoted as α_x and α_y, respectively. The factors are applied to Young's modulus, E_{db}, and Poisson's ratio, ν_{db}, defined in the materials database. The shear modulus defined in the materials database, G_{db}, is ignored, as the in-plane and out-of-plane shear moduli are derived using the formulae shown below.

Young's moduli in local X and Y directions

$$E_x = \alpha_x E_{db}$$

$$E_y = \alpha_y E_{db}$$

Poisson's ratios in local X and Y directions

$$\nu_x = \frac{\alpha_y}{\bar{\alpha}} \nu_{db}$$

$$\nu_y = \frac{\alpha_x}{\bar{\alpha}} \nu_{db}$$

where

$$\bar{\alpha} = \frac{\alpha_x + \alpha_y}{2}$$

In-plane shear modulus

The modulus is established using Huber's formula:

$$G_{xy} = \frac{\sqrt{E_x E_y}}{2\left(1 + \sqrt{\nu_x \nu_y}\right)}$$

Out-of-plane shear moduli

$$G_{xz} = G_{yz} = \frac{\max\left(E_x, E_y\right)}{2(1 + \bar{\nu})}$$

where

$$\bar{\nu} = \frac{\nu_x + \nu_y}{2}$$

Manually setting material properties

The material properties can be set manually when the Calculate properties checkbox is cleared. The properties take default values based on their state prior to clearing the checkbox.

The Poisson's ratio in the local Y direction is not directly editable, and is calculated automatically based on the editable parameters using the following formula:

$$\bar{\nu} = \frac{\nu_x + \nu_y}{2}$$

See also

Orthotropic walls (page 644)

Cracked, partially cracked, and uncracked concrete members

Assuming concrete sections (walls, columns and beams) are cracked has a direct affect on analysis - smaller Modification Factors (page 772) are applied to cracked sections, (typically a cracked member is assumed to have half the

stiffness of an uncracked member), causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption.

The *Assume cracked* setting in the member properties is used to specify cracked/uncracked status. It can be set for individual concrete beam spans, column stacks and wall panels as follows:

- **Yes** - the section is fully (100%) cracked
- **No** - the section is uncracked (0%)
- **Partially** - the section is partially cracked (anywhere between 0 - 100%)

While it can be set directly in the member properties, *Assume cracked* can also be set or toggled graphically (page 973) in a Show/Alter State Review View. For meshed walls a Review Wall Stress (page 974) feature is provided to speed up this process.

Usage of cracked, partially cracked, and uncracked

This feature is of especial use in the design of tall buildings, for which the engineer needs to particularly assess and modify the stiffness of the lateral force resisting system (usually composed primarily of concrete (shear) walls) to control lateral displacements. The stiffness of concrete members is determined in large part by their state of cracking - which can vary between the 'extreme' conditions of entirely uncracked and fully cracked.

You have the option to specify the (fully) cracked and uncracked conditions for concrete members (beams, columns and walls) and control their stiffness accordingly and separately via Modification Factors.

NOTE The default values of these **for the Building Analysis** assume the loading is long term and should be modified when considering short term loading - for more see this TUA article Where do the default values for cracked and uncracked properties of concrete come from?.

You also have the option to specify an intermediate state, partially cracked i.e. neither uncracked nor fully cracked, which is applicable to beams, columns and walls (meshed). The partially cracked state is set by a simple degree of cracking % for which the stiffness modification factor is automatically calculated and applied.

Tall Building Companion Features - as mentioned, this feature adds to and can be considered in the context of a number of other Tekla Structural Designer features targeted particularly at the design of tall buildings. These are:

- Dynamic Analysis - Wind Tunnel Data Report Generation
- Design for high strength concrete grades (Eurocode) - for more on this see the Tekla Structural Designer 2019i Release Notes
- Stresses in 2D elements (page 687)

- 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls
- Diaphragm loads and diaphragm load tables (page 362)

Partially cracked modification factor

Assume cracked can be set to:

- Yes - the section is fully (100%) cracked
- No - the section is uncracked (0%)
- Partially - the section is partially cracked
  - Percentage - anywhere between 0 - 100%

The **partially cracked** modification factor ($ModFactor_{Partially\ Cracked}$) is automatically calculated by linear interpolation for the specified percentage (**x %**) between the factors set for uncracked ($ModFactor_{unCracked}$) and cracked ($ModFactor_{Cracked}$) as follows:

- $ModFactor_{Partially\ Cracked} = ModFactor_{unCracked}$ - **x%** * ($ModFactor_{unCracked}$ - $ModFactor_{Cracked}$)
  - Thus for example for the default short term (Modal Analysis & RSA) concrete meshed wall modification factors of cracked = 0.5 and uncracked = 1.0, a Partially cracked value of 50% gives a modification factor = 1.0 - 0.5*(1.0 - 0.5) = 0.75.
  - The member stiffnesses are then adjusted accordingly using the calculated partially cracked modification factor values:
    - Meshed Walls - the factor is applied to properties E, G and/ or t as appropriate in the same manner as the cracked/ uncracked factors (specific to the factors currently set in the model)
    - Beams & Columns - the factor is applied to section properties in the same manner as the cracked/ uncracked factors (specific to the factors currently set in the model).

Determining wall cracked properties

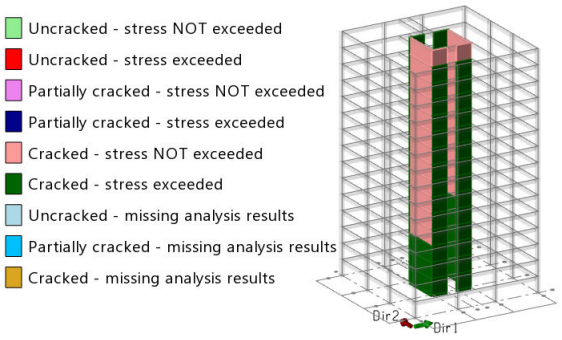
For engineers investigating the sway characteristics of taller buildings the cracked/uncracked status of walls is a complex topic of critical importance.

A couple of challenges faced by engineers are:

1. If using general stiffness adjustments - what are the realistic adjustment values for cracked and uncracked panels?
2. How can the cracked/uncracked status of wall panels be easily determined?

Tekla Structural Designer provides a Review Wall Stress (page 974) feature in Show/Alter State to significantly assist with the latter task for meshed panels.

This graphically displays the current cracked/uncracked status of each panel and indicates in which panels the stress threshold has and has not been exceeded. It then allows you to update the cracked status to be compatible with the threshold.



Use of the Review Wall Stress feature requires a degree of engineering judgment - you will need to make your own choices with regard to:

- strength or service level stress?
- basing the stress threshold on max instead of in-plane stress contours?
- whether the panel cracks for any level of stress, or could a higher threshold be considered?

You will also need to be familiar with the limitations and assumptions (page 975) that apply.

Workflow for reviewing wall cracked properties

To determine an appropriate cracked status for wall panels, you could adjust the following basic workflow to suit your needs:

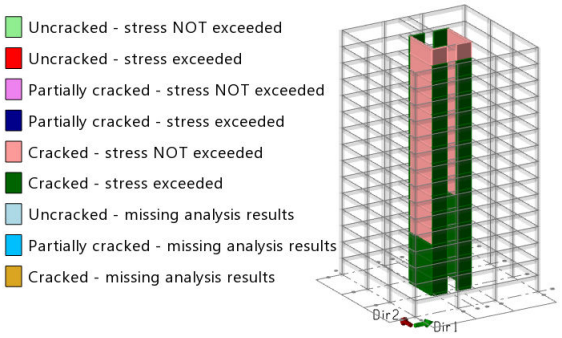
1. Analyze the model to determine some initial stresses.
2. Open a Review View and click Show/Alter State.
3. In the Show/Alter State **Properties** window:
   - Set the *Attribute* to **Assume cracked**
   - Set the *[M]ode* to Review Wall Stress (page 974)
   - Set the *Result type* to **Strength** or **Service** as you deem appropriate.
   - Set the *Stress type* to **In-plane tension - Y** or **Max tension - Y** as you deem appropriate.

   NOTE If using **In-plane tension - Y** contours:

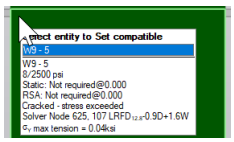
- these give an average stress through the thickness of the wall and therefore exclude local stress concentrations that exist as a result of out of plane bending
  - (engineers often choose to ignore out of plane bending when considering the overall cracked status of panels)

- Enter the *Stress threshold* that you want to work to.

In the below example, the most likely scenario where everything was initially set as cracked is shown; in lower panels this is a correct setting, in the higher ones they could be changed to uncracked.

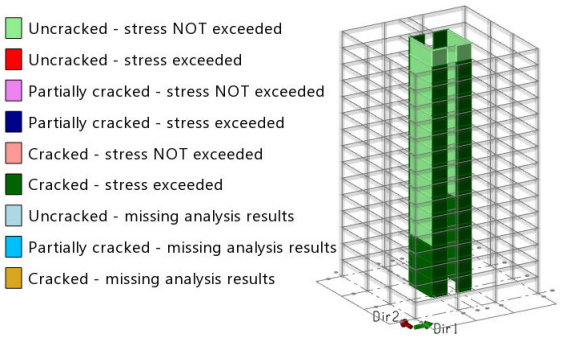


4. By hovering the cursor over an individual panel a tooltip is displayed. This reports the maximum stress value from all nodes in the panel and the case/combination in which it occurs. This value is used to determine if the stress threshold has been exceeded.



NOTE To investigate the reported stresses in more detail you might choose to open a Results View in order to display stresses (page 687) for specific envelopes or combinations.

5. At this point you could choose to manually set or toggle the cracked/uncracked status (page 973) of the individual panels, however **Auto update** provides a quicker way to make all panels compatible.
6. To make all panels automatically compatible:
   - Click Auto update to expose the Auto update button.
   - Click the Auto update button



Each panel is updated as follows:

- uncracked panels in which the stress threshold has been not exceeded are unaffected,
- uncracked panels in which the stress threshold has been exceeded are set to cracked,
- cracked panels in which the stress threshold has been exceeded are unaffected,
- cracked panels in which the stress threshold has not been exceeded are set to uncracked.

7. Reanalyze the model to take into account the revised cracked/uncracked properties.
8. Review the results once more and make further adjustments if required.

See also

Modify assumed cracked settings (page 972)

The Direct Analysis Method

AISC developed the Direct Analysis Method (DAM) as a solution to meeting stability analysis and design requirements in a modern way that is most suitable for implementation in analysis and design software. This method is not limited in its application, applies to all buildings when designing to US codes, and is the most general and accurate approach provided. The requirements for the DAM include:

- *Second-order analysis*: A second-order analysis which considers both P-Δ and P-δ effects is required.
- *Initial imperfections*: The effects of initial imperfections of the structure geometry are considered by applying notional loads, which are lateral loads proportional to the gravity loads applied at each framing level.
- *Inelasticity*: The axial and flexural stiffnesses of members that contribute to the stability of the structure are required to be reduced. This is to account

for the effects of residual stresses that lead to inelastic softening before the members reach their design strength.

- *Effective Length Factor*: Setting K to be 1.0 can be allowed because the effects for which it was meant to compensate (initial imperfections and inelasticity) have already been accounted for in the method.

DAM implementation in Tekla Structural Designer

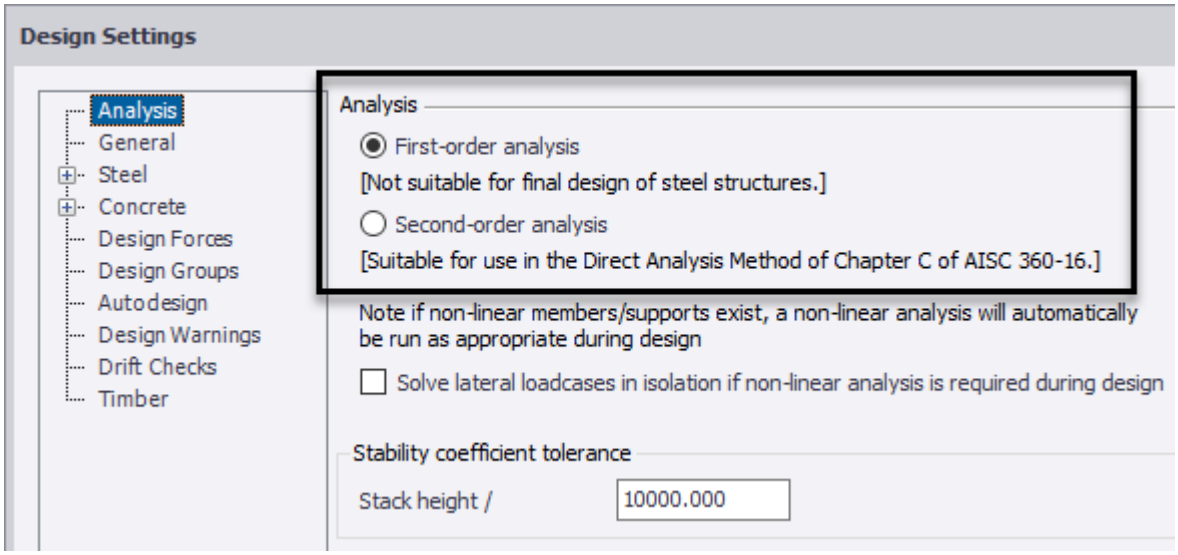
Tekla Structural Designer implements the Direct Analysis Method fully, using a rigorous second-order analysis. The following table describes how the AISC requirements have been implemented.

AISC Required effect to be considered	Tekla Structural Designer implementation
Flexural, shear, and axial deformations	A general 3D analysis is performed which considers all required deformations.
All component and connection deformation that contribute to the lateral displacement of the structure	All moment connections are assumed to be fully restrained. As is typical of most analysis programs, it is rationalized that these deformations do not contribute significantly to the stability of the structure and are therefore not directly included in the analysis.
Second-order effects (both P-Δ and P-δ)	The Chen and Lui rigorous second-order analysis is implemented. The two-cycle iterative method automates a two-pass analysis procedure during which nodal displacements are used to determine 'stress stiffening' in structural elements. The resulting matrix accommodates the P-Δ and P-δ effects as well as accounting for 'stress stiffness'. See also: What are the "2nd Order" Analysis options listed on the Analyze Ribbon in TSD? Are they the same as "P-Delta" analysis?
Geometric imperfections	Notional loads are calculated and applied wherever there's a column stack or wall panel division based on the gravity load for each appropriate load combination.
Member stiffness reductions due to residual stresses	All relevant members have reduced axial and flexural stiffnesses, the

AISC Required effect to be considered	Tekla Structural Designer implementation
	default for this being 0.8. The option to take t_b = 1.0 and increase the notional load factor is exercised.

Choice of analysis type

When designing to US codes, irrespective of whether you are performing Gravity Design (page 780) or Static Design (page 781), two options exist for the analysis, the choice being controlled via Design Settings > Analysis (page 866)



- **First-order analysis**

  This option is not suitable for final design of steel members. A standard linear elastic static analysis is included within Tekla Structural Designer to assist the user with steel structures that prove to be unstable and fail to complete second-order analysis.

  The results from the first-order analysis can be utilized to "pre-design" all members, after which you are more likely to have sections that will be sufficient for second-order analysis to solve.

  The first-order analysis is also used to determine both building drifts and member deflections in order to determine structure stability, seismic drift and serviceability design. The stability coefficient (Δ_2/Δ_1) is based on an approximate calculation of the second-order deflections of the structures based on the first-order analysis results.

- **Second-order Analysis**

  This option is suitable for use in the Direct Analysis Method. Second-order analysis is performed using a two step iterative method incorporating a geometric stress stiffness matrix. An initial linear static analysis is performed, the stiffness matrix is updated to account for forces derived from initial displacements and finally a second linear static analysis is performed using the modified stiffness matrix.

  This second-order analysis accurately determines both:

  - P-Δ (P-big delta) effects resulting from loads acting on the global deformation of the entire structure or a part of the structure
  - P-δ (P-little delta) effects resulting from the line of action of the axial force in a member acting at an eccentricity due the deformed shape of the member.

  The forces resulting from this analysis are utilized in the member design.

  Also during the design process first-order analysis is used to determine both building drift and member deflections in order to determine structure stability, seismic drift and serviceability design.

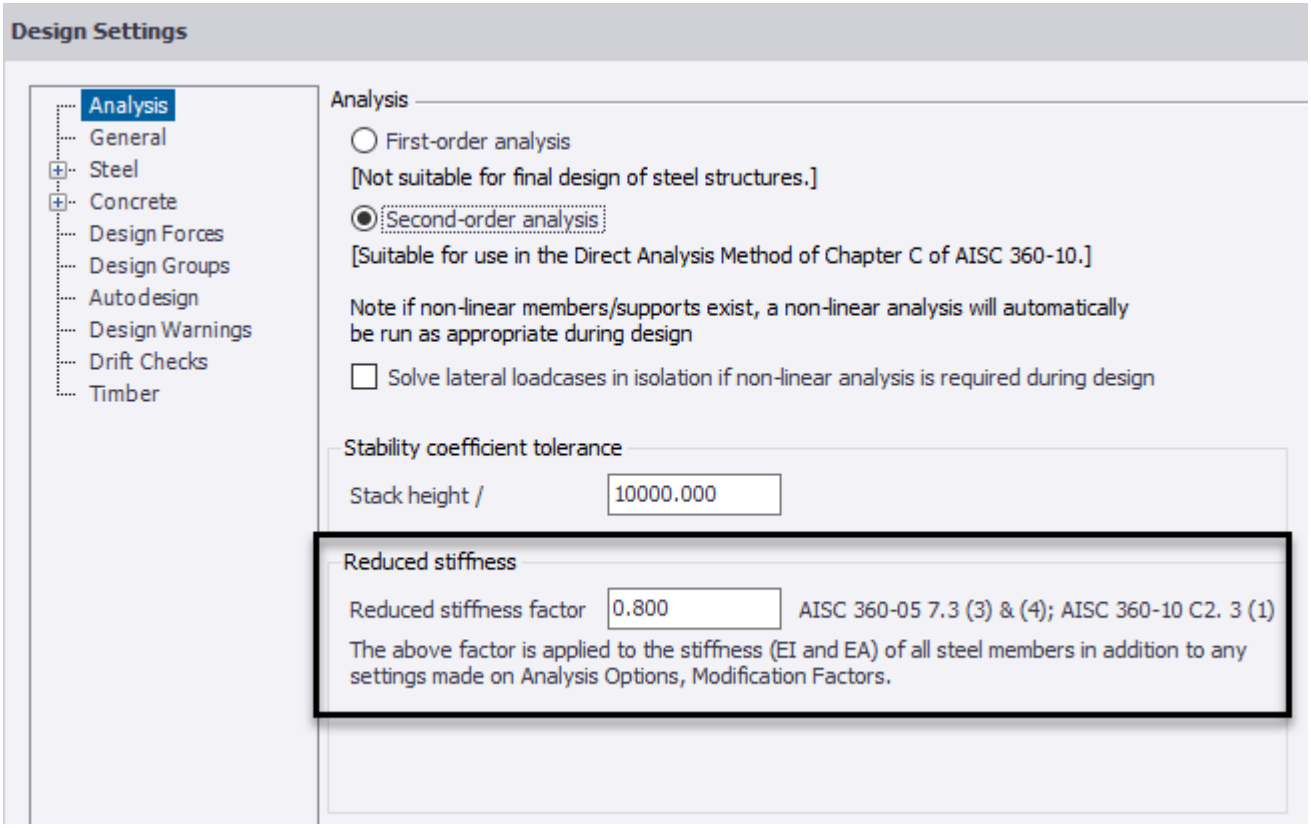
  The stability coefficient (Δ_2/Δ_1) is based on a combination of results from both the second and first-order analyses.

  NOTE There is potential for buckling to occur during 2nd order analysis. See the following article for more details: What is 2nd Order Buckling Analysis? How and when would I use it?

Analysis model adjustments

Certain adjustments should be made to the structural model in order to prepare it for analysis – these are controllable by the user.

- For both first and second-order analysis
  - concrete member section properties should be set as cracked, and suitable Modification Factors should be applied via Analysis settings (page 760), (see ACI 318-05 10.11.1)
- For second-order analysis
  - 2005 AISC Specification 7.3 (3) & (4) requires that for those members whose stiffness is considered to contribute to the lateral stability of the structure, a reduced flexural stiffness (EI) and a reduced axial stiffness (EA) must be used in the analysis. In Tekla Structural Designer, a reduction factor of 0.8 (default) is automatically applied to the stiffness (EI and EA) of all steel members. You are able to review this default stiffness factor and adjust it if you require via Design Settings > Analysis (page 866).



NOTE The above factor is applied in addition to any Modification Factors specified for steel members in Analysis settings (page 760).

Additionally, in order to adhere to the Direct Analysis Method, certain extra enhancements are made to the analysis model:

- For both first and second-order analysis
  - All load combinations require a minimum lateral load at each level of the structure of 0.2% (2005 AISC Specification 7.3 (2)). In addition, Tekla Structural Designer assumes τ_b = 1.0, so an additional 0.1% is required (2005 AISC Specification 7.3 (3)).

    It is the user's responsibility to include notional loads in combinations using the combinations wizard. for these Tekla Structural Designer automatically calculates the additional notional load to be 0.3% of the gravity component in each combination. (Although the user is provided with an option in Model Settings to override this with their own percentage values if they require.)

    Hence in combinations that have notional loads included by the user, Tekla Structural Designer automatically calculates the additional notional load to be 0.3% of the gravity component in each combination.

(Although the user is provided with an option in Model Settings to override this with their own percentage values if they require.)

- For second-order analysis
  - For designs using ASD, prior to second-order analysis all loads are automatically factored up by 1.6 and before design, all design forces are reduced by a factor of 1.6 as required by 2005 AISC Specification 7.3 (1).

Suggested workflow

When applying the Direct Analysis Method the following approach is recommended. This helps to avoid spending unproductive time on analysis issues and a potentially overly conservative design:

1. Never just construct a model (or import one e.g. from Revit) and attempt design using second-order analysis straight away (by setting **Design > Settings > Analysis > Second-order**).
2. Always conduct a full **First-order** analysis and design for both gravity and lateral combination to size members for these by setting **Design > Settings > Analysis > First-order**. This should ensure that:
   a. There are no mechanisms - troubleshoot and remove any mechanisms that come to light at this stage by following this article Guide to resolving Mechanisms.
   b. Section sizes are sensible and adequate for First-order design forces. They will then usually be adequate for second-order analysis without buckling (though not always). See this article for more about this; I have analysis Warnings and/ or Errors about buckling. Why is this happening and what do I do about it?
3. It is a good idea to assess displacements and Drift check results after this initial design also as these can highlight parts of the structure that are poorly braced or not well connected to the lateral force resisting system (LFRS). However bear in mind that these are not final results - second-order analysis may lead to increased section sizes which will moderate drift results.
4. Only when all members are passing at this stage, and overall displacements and drift results appear reasonable, should you proceed to second-order analysis and design. Set **Design > Settings > Analysis > Second-order** and first perform a “check” design, by turning off auto-design for all members then running a Design (Static) command, and assess the results of this.
5. Members that now fail at this stage are most likely to be members in the LFRS since second-order magnification effects will tend to be largest in these. If there are only a few failing members, consider skipping to step 6. Otherwise, set ONLY the failing members to auto-design on, then run a full model design (Design (Static)).

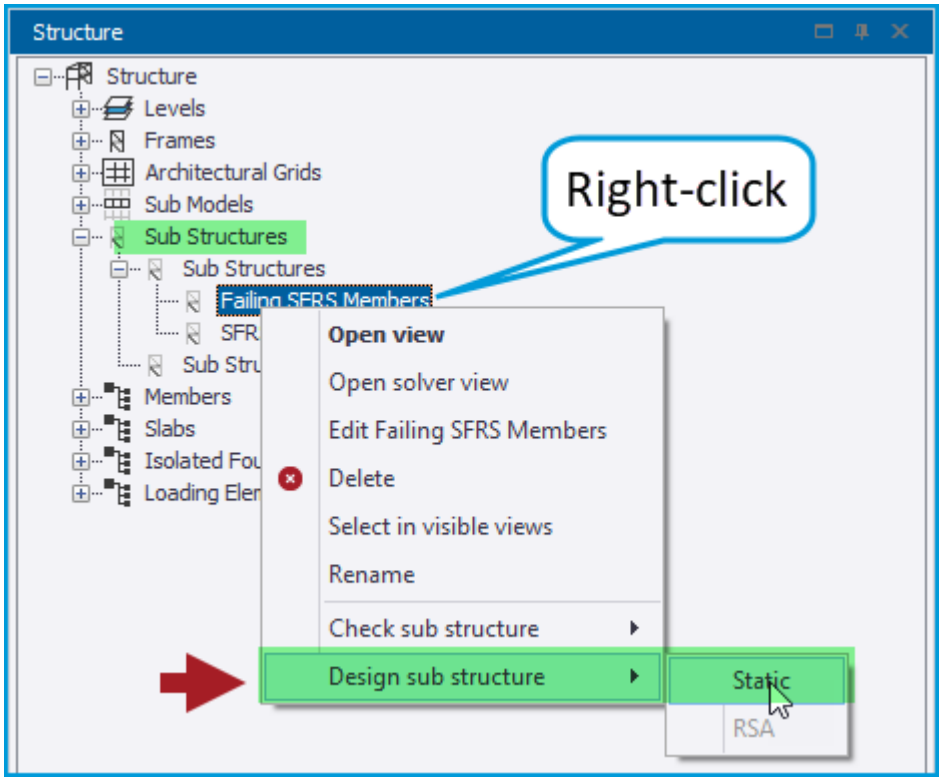
a. When this is complete, assess the efficiency of the section sizes assigned to the auto-designed members by examining their utilization ratios (URs) (tip - Design review filters (page 951) can help with this). If the UR's are low, it may be that pre-size sections (which are assigned to all auto-design members for the first analysis run) are causing excessive second-order magnification of design forces leading to inefficient section selection.

b. Though it is rare, buckling issues may occur during the second-order analysis if the pre-size sections are inadequate. The "Prevent out of plane instability" setting can be used to mitigate this in some circumstances as described in this support article.

6. If there are only a few members failing the second-order check-design - or if excessive second-order magnification or buckling of pre-size sections is occurring - we recommend conducting a 'manual' iterative second-order analysis and design as follows:

a. After setting second-order analysis for design and running a "check" design as described in step 4) above, add any failing members to a sub structure and set them to auto-design on.

b. Run a design of this sub-structure ONLY from the Structure tree context menu as shown in the picture below. Note that when this is complete by default this will set auto-design off for the members in it (but check that this is the case).



c. Run a “check” design of the entire structure - this updates the global analysis results for the new sections which may produce a change in member forces so some members may now fail again.

d. Set any failing members (ONLY) back to auto-design, update the sub structure if necessary, then design it once more.

e. Iterate steps c) and d) until all members are passing.

7. This "manual" iterative approach avoids the potential for pre-size sections to fail analysis due to buckling or cause excessive second-order amplification leading to a suboptimal design solution.

   a. Note however that you may still need to manually size members to satisfy Drift checks or, for members of a SFRS, to pass specific seismic design checks (members are not auto-sized for these).

4.4 Analysis results

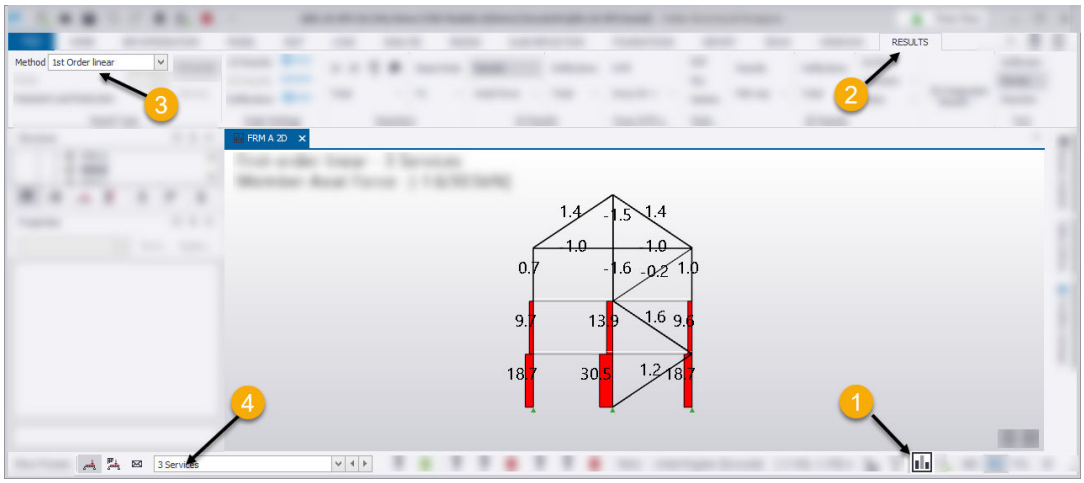
Once analysis has been performed, you can display the analysis results in a number of ways.

Click the links below to find out more:

- Display analysis results in a Results View (page 662)
- Display tabular analysis results in a Solver Model Data view (page 707)
- Display member analysis results in a Load Analysis View (page 748)
- Display loading summaries in the Project Workspace (page 758)

Display analysis results in a Results View

To display graphical analysis results, first switch the view to the  **Results View** regime **(1)**, then click the **Results** tab **(2)**.



After selecting the analysis method and loading (page 664), results can then be displayed by making selections from the ribbon.

Ribbon Group	Click the link to find out more
Reactions	Display reactions (page 665)
1D Results	Display 1D results (page 667)
	Display 1D deflections (page 667)
	Animate 1D and 2D deflections (page 668)
Sway Drift and Story Shear	Display sway, drift, and story shear (page 668)
Notional Loads	Display notional forces, minimum lateral loads and seismic equivalent lateral forces (page 684)
2D Results	Display forces in 2D elements (page 685)
	Display stresses in 2D elements (page 687)
	Display 2D deflections (page 691)
	Animate 1D and 2D deflections (page 668)
	Display AsReq contours (page 692)
2D Integrated Results **NOTE** Result lines and result strips must be created before they can be displayed.	Display wall lines (page 692)
	Display core lines (page 692)
	Manage, display and design result lines (page 698)
	Manage and display result strips (page 693)
Mode Shapes	Display mode shapes (page 701)

Ribbon Group	Click the link to find out more
RSA Results	RSA seismic results (page 701)

If required, adjustments can be applied to the diagrams as follows:

- To change contour intervals and colors, see Customize the display of 2D contours (page 705)
- To adjust the amplitude of the diagrams, see Change result diagram scale settings (page 706)
- In 2D views, it is sometimes necessary to switch to an isometric projection, see Display 2D view in isometric projection (page 707)

NOTE When Tekla Structural Designer has performed several analysis types, the results of each analysis are held separately. Therefore, there is no need to re-perform a particular analysis to recall its results.

Set the analysis type and loading for viewing analysis results

When Tekla Structural Designer has performed several analysis types, the results of each analysis are held separately. Therefore, there is no need to re-perform a particular analysis to recall its results.

1. On the ribbon, go to the **Results** tab.
2. In the **Result Type** group, select the required analysis method.

   NOTE If you select: 1st order modal, 2nd order buckling, 1st order RSA seismic or 2nd order RSA seismic analyses, you should also choose the **Mode** to be displayed.

3. In the **Result Type** group, if required, click **Imposed/Live (US) Load Reduction** if you want to take into account reductions when viewing axial forces in a loadcase or combination in which reductions have been applied.
4. In the **Result Type** group, click **Strength Factors**, or **Service Factors**, as required, to control which factors are used when displaying results for combinations.
5. In the **Loading** list, click either the  **Select Loadcase**,  **Select Combination**, or  **Select Envelope** button.
6. In the **Loading** list, select the desired loadcase, combination, or envelope.

You can now proceed to select the diagram to be displayed.

See also

Display analysis results in a Results View (page 662)

Display reactions

The instructions below describe how to graphically display reactions at individual supports and beam ends. For instructions on how to view a summary of the total reactions for the whole model see: Display loading summaries in the Project Workspace (page 758).

Support reactions

1. Open a view and switch the view regime to  **Results View**. On the ribbon, the **Results** tab is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**.
4. In the **Result Type** group, if you are viewing a loadcase or combination to which load reductions have been applied, you can select whether you want to view the results with the **Imposed/Live Load Reduction** applied, or not.
5. in the **Reactions** group on the **Results** tab, click **Settings**  if required, to select the required axis system. To find out more about the axis system choices, see: Working with foundation reaction axis systems (page 726)
6. In the **Reactions** group, select the reactions to be displayed:

Option	Description
1D	Reactions at supports under 1D elements (e.g. column support reactions),
2D	Reactions at supported nodes of 2D elements (individual meshed wall supports and mat foundation supports),
	Distributed wall reactions (Fz) - available for self-supported meshed, mid-pier, bearing and shear-only wall panels with horizontal bottom levels.

Option	Description
	When active the nodal support reactions exclusive to the wall are hidden. Summary of assumptions, • Wall panel has enough height to ensure that all loading that contributes to the reaction is far enough from the support such that the principle of St. Venant is considered valid, • Material is perfectly homogeneous and isotropic, • Elastic limit is nowhere exceeded and 'E' is same in tension and compression.
	Integrated reactions at core supports. When active support reactions exclusively from 1D and 2D core members are hidden.

7. For **Distributed wall reactions** the droplist beneath should be set to **Total**, for other support reactions you can select the desired reaction from the droplist.

Option	Description
Fx	support local x axis reaction (corresponds to **Fminor** in the Foundation Reactions Report),
Fy	support local y axis reaction ((corresponds to **Fmajor** in the Foundation Reactions Report),
Fz	support local z axis reaction (corresponds to **Fvert** in the Foundation Reactions Report),
Mx	support local x axis moment (corresponds to **Mmajor** in the Foundation Reactions Report).
My	support local y axis moment (corresponds to **Mminor** in the Foundation Reactions Report)

Option	Description
Mz	support local z axis moment (corresponds to **Mtor** in the Foundation Reactions Report)
Fxyz	support local reactions in **Fx** and **Fy** and **Fz**
Mxyz	support local reactions in **Mx** and **My** and **Mz**
Total	all support local reactions (**Fx**, **Fy**, **Fz**, **Mx**, **My** and **Mz**.

8. From the **Text** group select **Reaction** to display values on the diagram.

Beam end reactions

1. Open a view and set the view regime to  **Results View**.
   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Reactions** group, select **Beam Ends**, then from the droplist beneath select the desired reaction.
4. From the **Text** group select **Reaction** to display values on the diagram.

Display 1D results

1. Open a view and set the view regime to  **Results View**.
   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **1D Results** group, select **Results**, then from the droplist underneath it, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

See also

Change result diagram scale settings (page 706)

Display 2D view in isometric projection (page 707)

Display 1D deflections

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **1D Results** group, select **Deflections**, then from the droplist underneath it, select the desired deflection type.

   NOTE 1D deflections for loading envelopes cannot be displayed for the model in a Results View, however they can be displayed for an individual member in a Load Analysis View (page 749).
4. From the **Text** group select **Deflection** to display values on the diagram.

See also

Change result diagram scale settings (page 706)

Animate 1D and 2D deflections (page 668)

Animate 1D and 2D deflections

Once the required deflections are displayed, they can be animated as follows:

1. Right click in the view and from the context menu select **Animate**.

   The animation commences.
2. If required you can change the number of frames per second, the cycle duration and the amplitude by adjusting the sliders and clicking **Apply** in the **Animation** dialog.
3. To end the animation, simply close the **Animation** dialog.

   NOTE Animations are available for static displacements and those for modal and buckling analysis.

Display sway, drift, and story shear

Sway

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).

3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the list beneath select **Sway Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Relative sway

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the list beneath select **Relative Sway Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Wind drift

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the list beneath select **Wind Drift Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Story Forces

Story forces (Dir 1 and 2) (page 672) can be displayed as follows:

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the list beneath select **Story Forces Dir 1** or **Story Forces Dir 2** as required.

   Story forces are displayed for the selected direction.

NOTE By default small story force values are not displayed. You can control the threshold for this in **Model Settings** --> **Graphics View Settings** by adjusting the **Do not display values of story shear minimum below** value.

CAUTION Story force values are only displayed at column nodes, at the common node of a brace and column, and at the common node of an A or V brace pair. For more details, see: Limitations (page 681)

Story Shears

Story shears (Dir 1 and 2) (page 674) can be displayed as follows:

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the list beneath select **Story Shears Dir 1** or **Story Shears Dir 2** as required.

   Story shears are displayed for the selected direction.

NOTE By default small story shear values are not displayed. You can control the threshold for this in **Model Settings** --> **Graphics View Settings** by adjusting the **Do not display values of story shear minimum below** value.

CAUTION Story shear values are only displayed at column nodes, at the common node of a brace and column, and at the common node of an A or V brace pair. For more details, see: Limitations (page 681)

Wall/Braced Bay Story Shears

Wall/braced bay story shears (Dir 1 and 2) (page 675), or Wall/braced bay story shears (in-plane) (page 677) can be displayed as follows:

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the list beneath select **Wall/Braced Bay Story Shears In-Plane**, **Wall/Braced Bay Story Shears Dir 1** or **Wall/Braced Bay Story Shears Dir 2** as required. Cumulative story shears to be resisted by **walls** and **braced bays** are displayed for the selected direction. Story shears resisted by **moment frames** are not shown.

NOTE By default small story shear values are not displayed. You can control the threshold for this in **Model Settings** --> **Graphics View Settings** by adjusting the **Do not display values of story shear minimum below** value.

See also

Understanding story shears and story forces (page 671)

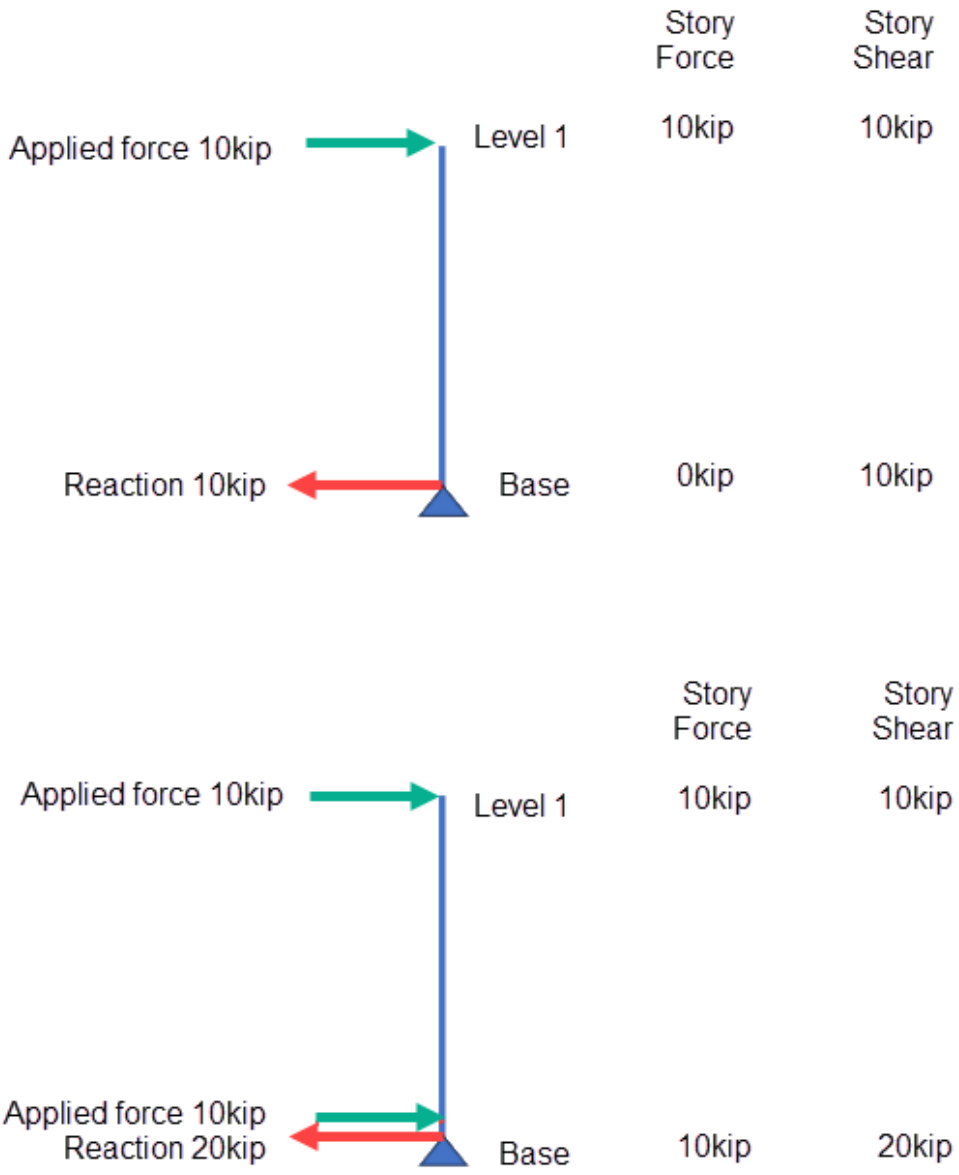
Understanding story shears and story forces

Illustration of story forces and story shears

In general terms:

- **Story Forces** are the lateral forces applied at the story (a floor level), ignoring any support forces at that level.

- **Story Shears** are the cumulative shears from the structure above and at the story (a floor level), ignoring any support forces at that level.



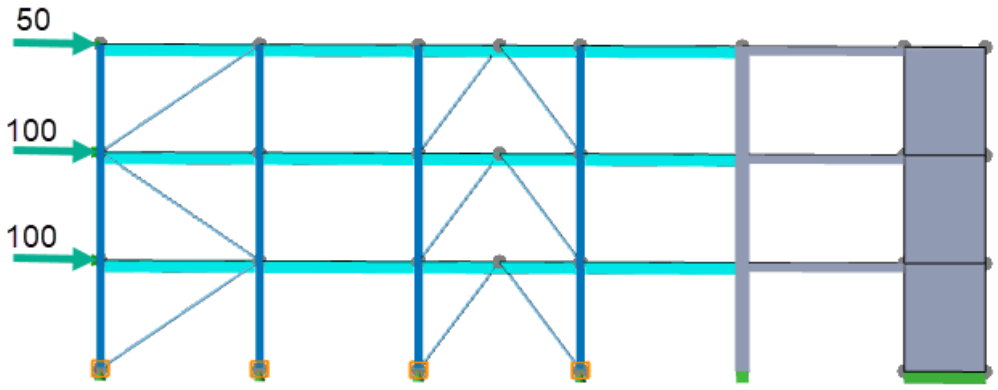
Story forces (Dir 1 and 2)

The graphical display of story forces shows the distribution of lateral forces coming into/ out of each level, typically at points where the members of the lateral force resisting system (LFRS) – be it braced bays, moment frames or

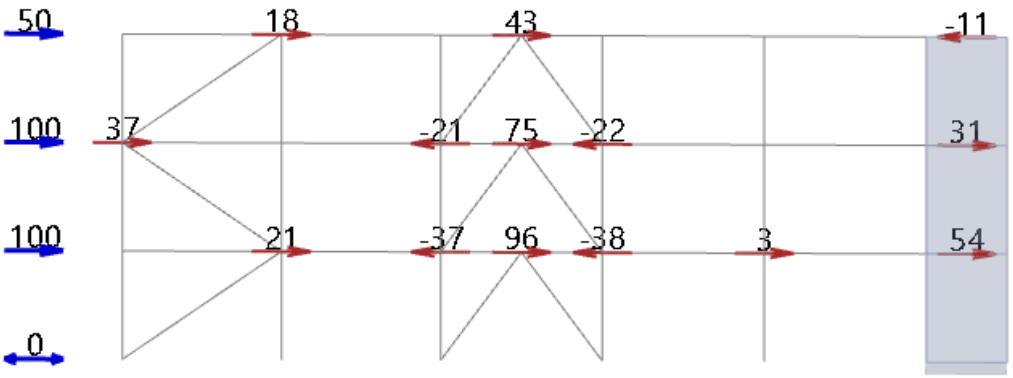
shear walls – are attached. Note that story forces are only depicted at levels and not in between levels.

For an example of story forces, consider the following Dir 1 frame, in which:

- the first and third bays are braced
- the fifth bay is a moment frame
- the last bay is a shear wall
- lateral loads are applied as shown



By selecting (page 669) the following results are displayed:



- Red arrows show the story forces at each level at those points where the LFRS is attached, (where the change in shear takes place).
- Each arrow indicates the direction of loading. The value is positive if the arrow acts in the selected building direction and negative if it acts against the selected building direction.
- Blue arrows drawn outside the structure show the summations at each level of lateral forces ***applied at that level***, **lateral forces applied between levels are *not* considered in these summations.**

NOTE By default small story shear/force values are not displayed. You can control the threshold for this in **Model Settings** --> **Graphics View Settings** by adjusting the **Do not display values of story shear minimum below** value.

The summation of story forces in each direction can also be shown in a Solver Model Data view, (by following these instructions (page 718)).

Story Forces

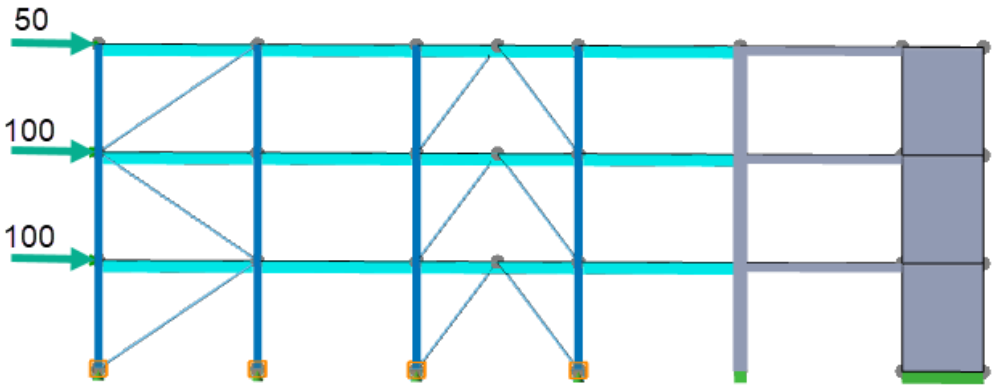
Level [ft, in]	Reference	Σ Shear Dir 1 [kip]	Σ Shear Dir 2 [kip]
36′ 0″	St. 3 (eaves)	50	0
24′ 0″	St. 2 (2)	100	0
12′ 0″	St. 1 (1)	100	0
0″	St. Base (Base)	-	-

Story shears (Dir 1 and 2)

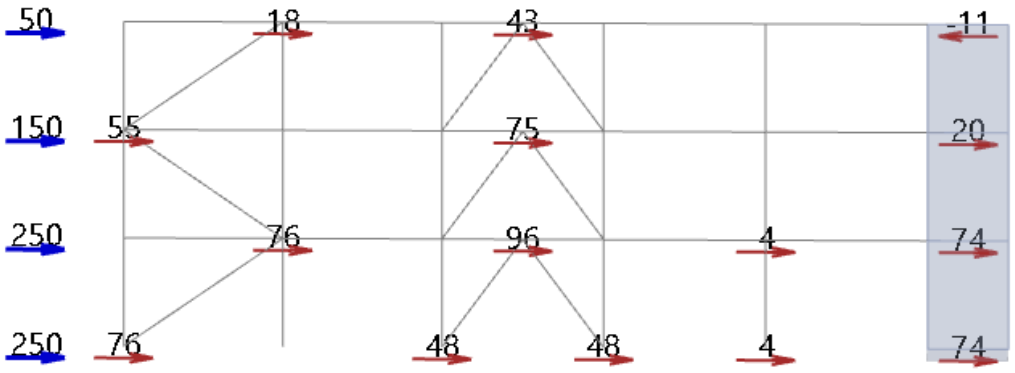
Story shear is the shear load that has to be resisted by the structure immediately below the level by the lateral force resisting system (LFRS) – be it braced bays, moment frames or shear walls.

The graphical display of Dir 1 and Dir 2 story shears shows the *cumulative* horizontal shear resulting from all loads applied above and at each level in the selected building direction. Note that these shears are only depicted at levels and not in between levels.

For example, consider the frame from the previous section:



By selecting (page 670) the following results are displayed:



Story shears are shown immediately below the level to which they relate.

- Red arrows show the individual cumulative shears. (The value at each node being the sum of shears from elements connected below the node.)
- Each arrow indicates the direction of loading. The value is positive if the arrow acts in the selected building direction and negative if it acts against the selected building direction.
- Blue arrows drawn outside the structure show the total shear load to be resisted by the structure immediately below each level.

NOTE If you experience an unexplained jump in total shear load from one level to the next, check to see if it can be explained by lateral forces applied between the levels. (See "Limitations" section below for more details).

NOTE By default small story shear/force values are not displayed. You can control the threshold for this in **Model Settings** --> **Graphics View Settings** by adjusting the **Do not display values of story shear minimum below** value.

The summation of story shears in each direction can also be shown in a Solver Model Data view, (by following these instructions (page 719)).

Story Shears

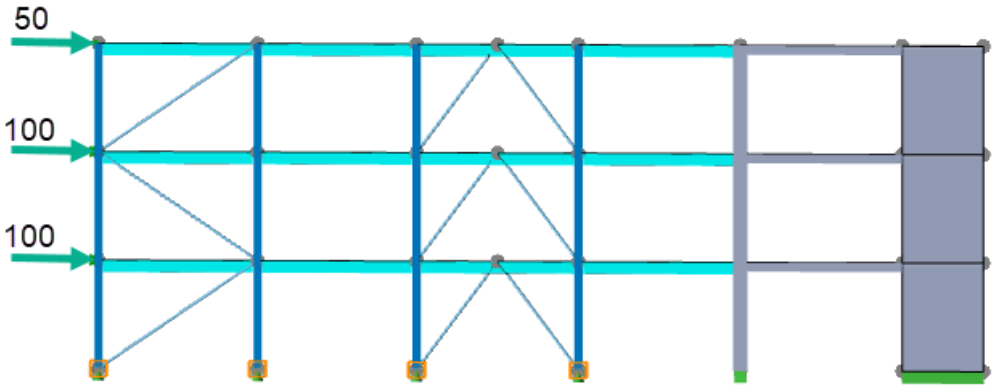
Level [ft, in]	Reference	Σ Shear Dir 1 [kip]	Σ Shear Dir 2 [kip]
36′ 0″	St. 3 (eaves)	50	0
24′ 0″	St. 2 (2)	150	0
12′ 0″	St. 1 (1)	250	0
0″	St. Base (Base)	250	0

Wall/braced bay story shears (Dir 1 and 2)

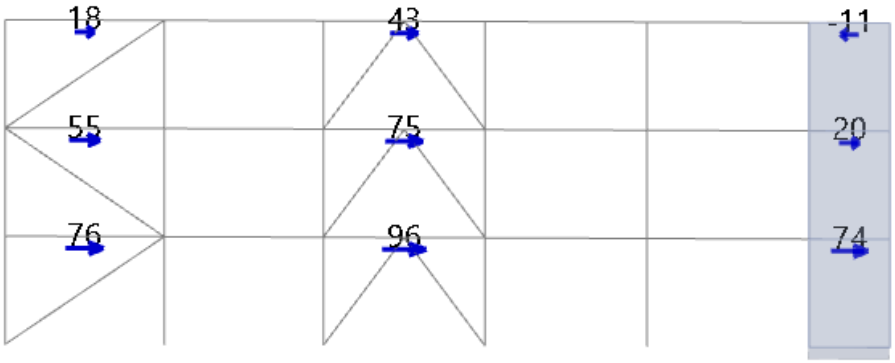
The graphical display of Dir 1 and Dir 2 wall/braced bay story shears shows the cumulative horizontal shear resulting from all loads applied above and at each level in the selected building direction that is resisted in a vertical plane *by*

braced bays or shear walls only – columns that are not attached to braces, and moment frames do not have any values shown.

For example, consider the frame from the previous sections:



By selecting (page 670) the following results are displayed:



- Blue arrows show the cumulative shears at each level in each braced bay or shear wall.
- Each arrow indicates the direction of loading. The value is positive if the arrow acts in the selected building direction and negative if it acts against the selected building direction.

This diagram is easier to work with than the “Story Shears Dir 1" and "Story Shears Dir 2” diagrams because:

- a single shear value is displayed at each level in each bay
- moment frame values are hidden
- values on columns not attached to braced bays are hidden.

Wall/braced bay story shears can also be shown in a Solver Model Data view, (by following these instructions (page 719)). This view shows the values in each direction as well as the in-plane values.

Wall/Braced Bay Storey Shears						
Level [ft, in]	Level Name	Frame Name	Wall/Braced Bay Reference	Dir1 Wall/Braced Bay Storey Shears [kip]	Dir2 Wall/Braced Bay Storey Shears [kip]	In-plane Wall/Braced Bay Storey Shears [kip]
36′ 0″	St. 3 (eaves)	FRM 1	SBP 13	43	0	43
36′ 0″	St. 3 (eaves)	FRM 1	SBR 2/A/1-3/B/1	18	0	18
36′ 0″	St. 3 (eaves)	FRM 1	W2 - 3	-11	0	-11
36′ 0″	St. 3 (eaves)		Not captured by walls and braced bays	1	0	-
24′ 0″	St. 2 (2)	FRM 1	SBP 12	75	0	75
24′ 0″	St. 2 (2)	FRM 1	SBR 1/B/1-2/A/1	55	0	55
24′ 0″	St. 2 (2)	FRM 1	W2 - 2	20	0	20
24′ 0″	St. 2 (2)		Not captured by walls and braced bays	0	0	-
12′ 0″	St. 1 (1)	FRM 1	SBP 7	96	0	96
12′ 0″	St. 1 (1)	FRM 1	SBR Base/A/1-1/B/1	76	0	76
12′ 0″	St. 1 (1)	FRM 1	W2 - 1	74	0	74
12′ 0″	St. 1 (1)		Not captured by walls and braced bays	4	0	-

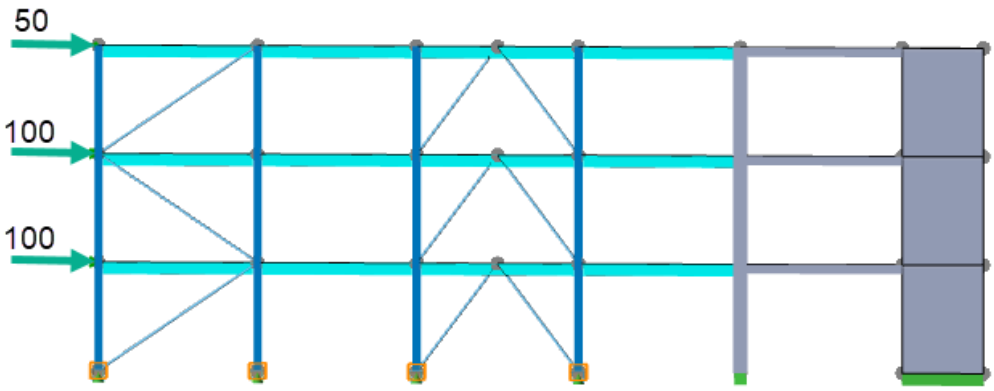
The in-plane values are described in the next section.

NOTE Some (usually small) amount of shear can develop in columns (or other systems) that are not identified by the "Wall/Braced Bay" subsets. These are reported at each level in each direction in the above Solver Model Data view as "Not captured by wall and braced bays" values. This is entirely normal, and if you want to investigate these you must look at the "Story Forces" or "Story Shears" Dir 1 and Dir 2 graphical displays.

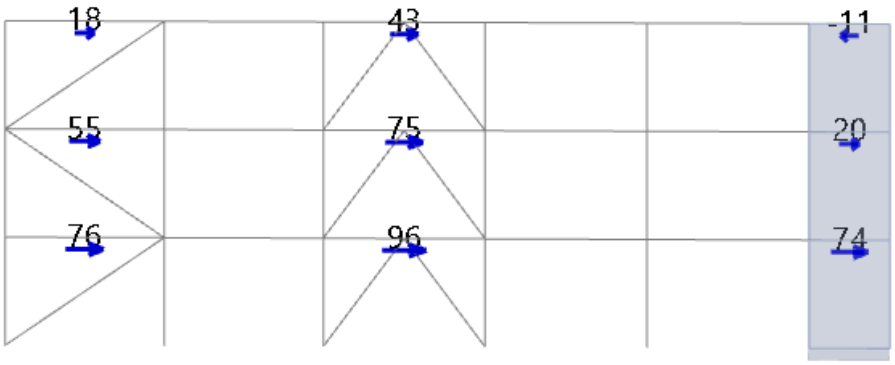
Wall/braced bay story shears (in-plane)

The graphical display of in-plane wall/frame story shears shows the cumulative horizontal shear resulting from all loads applied above and at each level that is resisted by *braced bays or shear walls in the plane in which they are aligned* – columns that are not attached to braces (e.g. columns in moment frames) do not have any values shown.

For example, consider the frame from the previous sections:



By selecting (page 670) the following results are displayed:



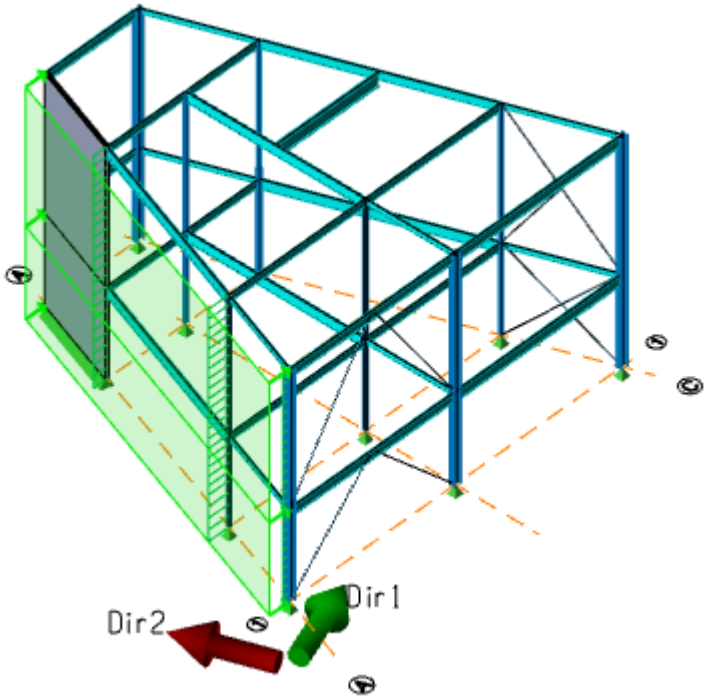
Because the braced bays and shear walls in this example are aligned to Dir 1 they are identical to the **Wall/Braced Bay Story Shears Dir 1** in the previous section.

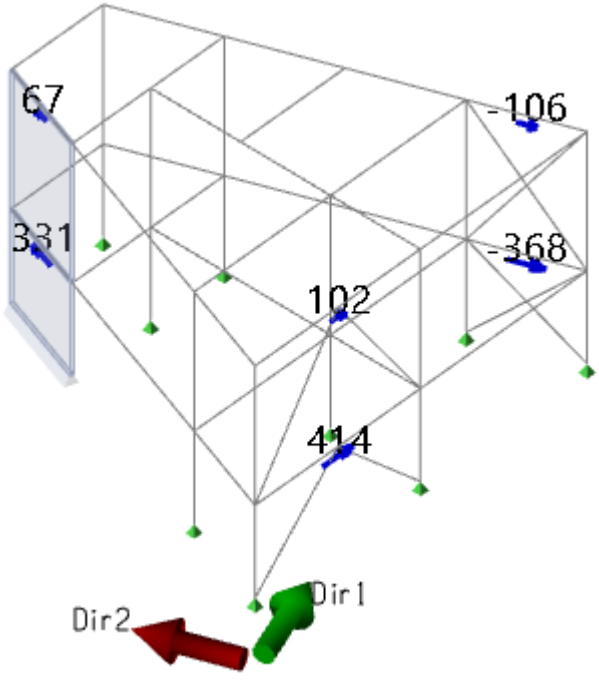
The main advantage of the “Wall/Braced Bay Story Shears In-Plane” graphical display when compared to the “Wall/Braced Bay Story Shears Dir 1, Dir 2” displays is that it shows the values of interest irrespective of how braced bays and walls are aligned, and the view isn't cluttered by out-of-plane values that could be ignored.

To demonstrate, consider this second example shown (below left), in which:

- a wall has been defined in Frame A
- braced bays have been defined in Frames 1 and C
- none of the frames align with the building directions Dir 1 and Dir 2
- a lateral load has been applied perpendicular to Frame A

By selecting (page 670) you can immediately see (below right) the forces to be resisted by each lateral force resisting system (LFRS).





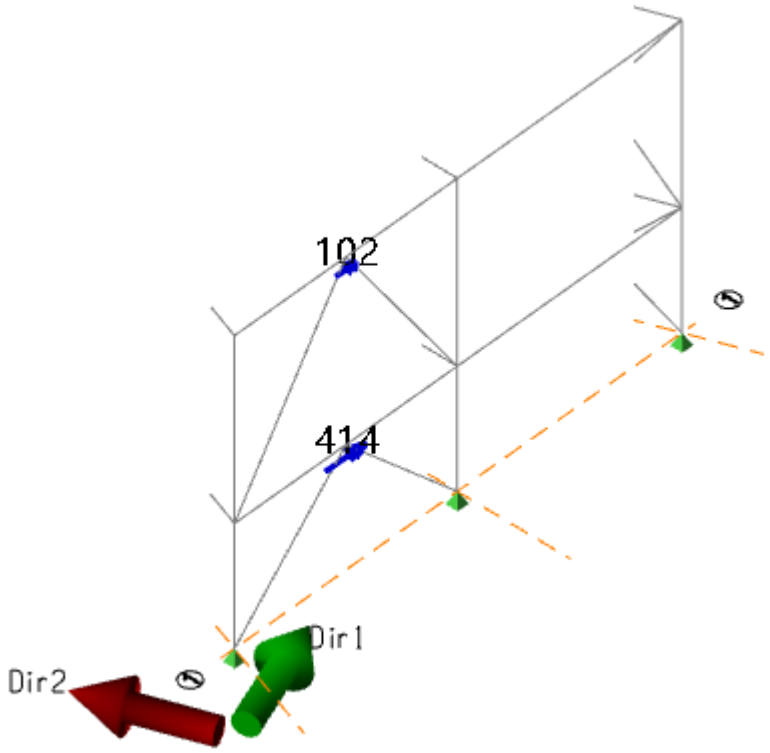
The arrows show the direction of loading. The sign convention applied to values is:

- if the vertical plane in which an arrow lies is more closely aligned with Dir 1 (within +-45 deg of Dir 1), the value is positive if the arrow acts towards Dir 1 and negative if it acts against Dir 1

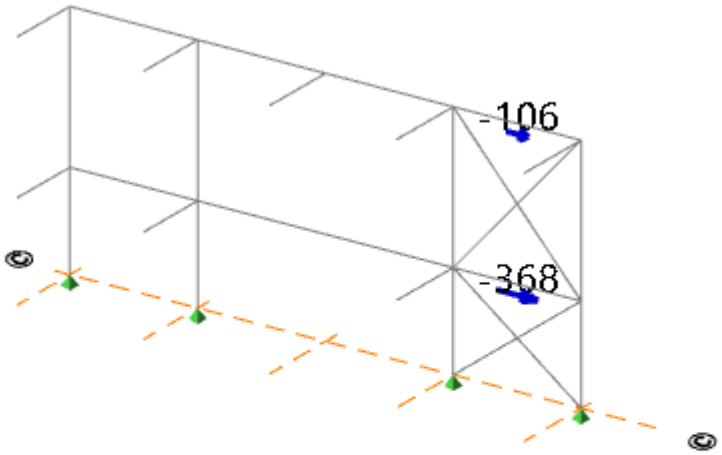
- otherwise the arrow will be more closely aligned with Dir 2, in which case the value is positive if the arrow acts towards Dir 2 and negative if it acts against Dir 2.

Applying this convention to the above example:

- Frame 1 which is within +-45 deg of Dir 1, the arrows in the braced bay act towards Dir 1 so their values are positive.



- Frame C is within +-45 deg of Dir 2, the arrows in the braced bays act against Dir 2 so their values are negative.





The above wall/braced bay story shears can also be shown in a Solver Model Data view, (which can be displayed by following these instructions (page 719)).

Wall/Braced Bay Storey Shears

Level [m]	Level Name	Frame Name ▲	Wall/Braced Bay Reference	Dir1 Wall/Braced Bay Storey Shears [kN]	Dir2 Wall/Braced Bay Storey Shears [kN]	In-plane Wall/Braced Bay Storey Shears [kN]
10.000	St. 2 (Roof)		Not captured by walls and braced bays	-65	25	-
4.000	St. 1 (Floor 1)		Not captured by walls and braced bays	-63	20	-
10.000	St. 2 (Roof)	FRM 1	SBP 8	97	-33	102
4.000	St. 1 (Floor 1)	FRM 1	SBP 7	388	-145	414
10.000	St. 2 (Roof)	FRM A	W1 - 2	-27	109	67
4.000	St. 1 (Floor 1)	FRM A	W1 - 1	138	315	331
10.000	St. 2 (Roof)	FRM C	SBP 3	5	-106	-106
4.000	St. 1 (Floor 1)	FRM C	SBP 4	25	-368	-368

NOTE Some (usually small) amount of shear can develop in columns (or other systems) that are not identified by the "Wall/Braced Bay" subsets. These are reported at each level in each direction in the above Solver Model Data view as "Not captured by wall and braced bays" values. This is entirely normal, and if you want to investigate these you must look at the "Story Forces" or "Story Shears" Dir 1 and Dir 2 graphical displays.

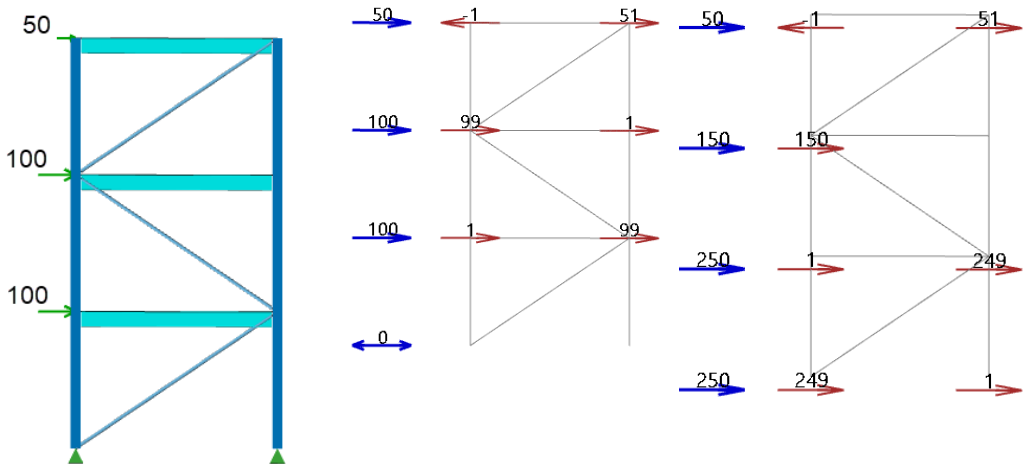
Lateral loads applied between levels

Any lateral loads (point loads, udls etc.) applied between levels do not get included in the story force totals that are drawn in blue outside of the structure.

Therefore, if you experience an unexplained jump in total shear load from one level to the next, check to see if it can be explained by lateral forces applied between the levels.

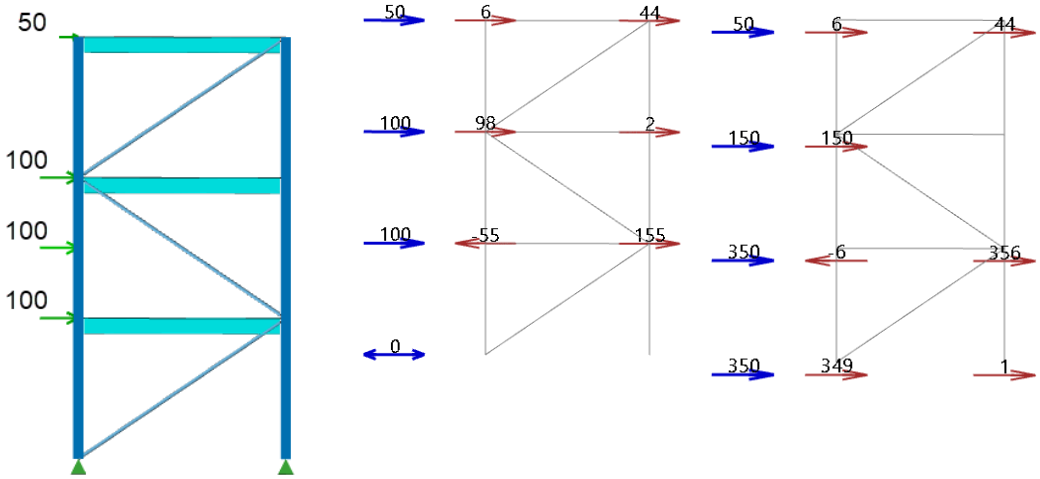
This limitation is demonstrated in the following examples.

Example 1: Load applied at levels only



Applied loads	Story Forces	Story Shears
50, 100, 100 applied at levels 3, 2 and 1 respectively	Story force totals (in blue) equate to applied loads	Story shear totals (in blue) equate to cumulative story force totals

Example 2: Load applied between levels



Applied loads

Extra 100 applied between level 2 and 1

Story Forces

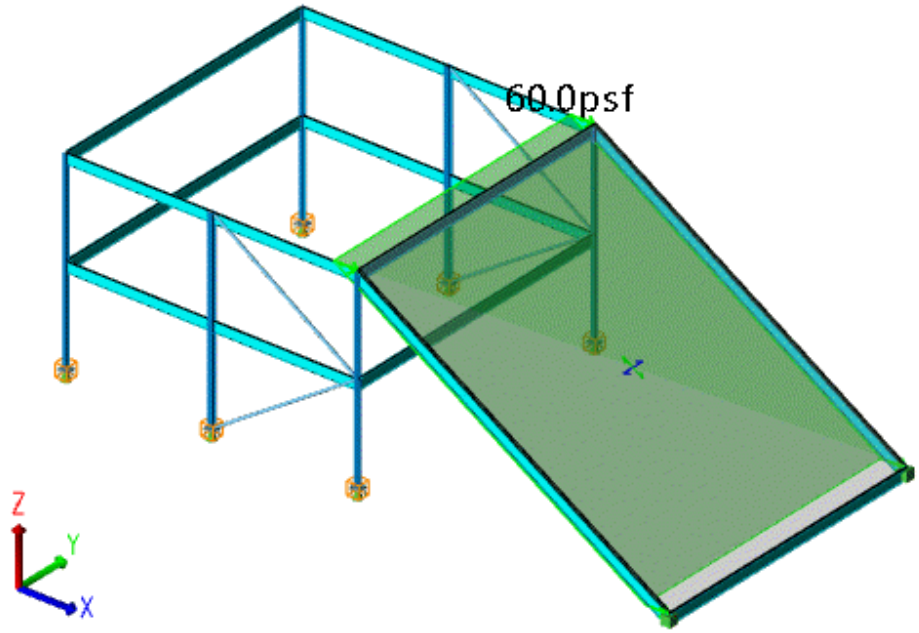
Totals (in blue) equate to loads applied **at levels** only - 100 applied between level 2 and 1 not included in totals

Story Shears

Story shear totals (in blue) **do not** equate to cumulative story force totals

Sloping slabs

Story forces and story shears will not be correct if a sloping slab passes through a floor with no connection to the floor.



They are not correct because the forces at the level in the floor are not calculated in this situation.

See also

Display sway, drift, and story shear (page 668)

Display tabular analysis results in a Solver Model Data view (page 707)

Display notional forces, minimum lateral loads and seismic equivalent lateral forces

To view the notional forces, minimum lateral loads, or seismic equivalent lateral forces that apply to a particular load combination or loadcase, see the following instructions.

View the magnitude and direction of EHF

RESTRICTION	The command is only applicable to the Eurocode regional code.

1. Open a view and set the view regime to **Results View**.

   The **Results** ribbon is displayed.
2. In the **Loading** list, select the required EHF combination.
3. On the **Results** tab, click **EHF**.

View the magnitude and direction of NL

RESTRICTION The command is only applicable to the ACI/AISC regional code.

1. Open a view and set the view regime to  **Results View**.
   The **Results** ribbon is displayed.
2. In the **Loading** list, select the required NL combination.
3. On the **Results** tab, click **NF**.

View the magnitude and direction of NHF

RESTRICTION The command is only applicable to the BS, IS and Australian regional codes.

1. Open a view and set the view regime to  **Results View**.
   The **Results** ribbon is displayed.
2. In the **Loading** list, select the required NHF combination.
3. On the **Results** tab, click **NHF**.

View the magnitude and direction of MLL (minimum lateral load)

1. Open a view and set the view regime to  **Results View**.
   The **Results** ribbon is displayed.
2. In the **Loading** list, select the required MLL loadcase or combination.
3. On the **Results** tab, click **MLL**.

View the magnitude and direction of seismic equivalent lateral forces

1. Open a view and set the view regime to  **Results View**.
   The **Results** ribbon is displayed.
2. In the **Loading** list, select the required seismic loadcase or combination.
3. On the **Results** tab, click **Seismic**.

Display forces in 2D elements

Display forces

To display forces graphically:

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **2D Results** group, select **Forces**, then from the droplist beneath select the force required.

SEE ALSO > FE results by node (page 713) (displayed in a Tabular Data View)
> FE results by shell (page 714) (displayed in a Tabular Data View)

Which forces can be displayed?

Option	Desription
Mdx top	Wood Armer bending along panel x axis, top surface
Mdx bottom	Wood Armer bending along panel x axis, bottom surface
Mdy top	Wood Armer bending along panel y axis, top surface
Mdy bottom	Wood Armer bending along panel y axis, bottom surface
Fx	Axial force in panel x axis
Fy	Axial force in panel y axis
Fxy	Complimentary in-plane shear
Fxz	Shear in panel z axis in the panel xz plane
Fyz	Shear in panel z axis in the panel yz plane
Fz_{max}	Maximum shear force
Mx	Bending along panel x axis
My	Bending along panel y axis
Mxy	Plate torsional moment
M_1	Maximum principal moment
M_2	Minimum principal moment

Calculation of the maximum and minimum principal moments

The maximum and minimum principal moments are calculated from the Mx, My and Mxy moments at the nodes as follows:

$M_1 = (Mx + My + \sqrt{((Mx - My)^2 + 4 \times Mxy^2)}) / 2$

$M_2 = (Mx + My - \sqrt{((Mx - My)^2 + 4 \times Mxy^2)}) / 2$

Calculation of the maximum shear force

The maximum shear force is calculated from the Fxz and Fyz forces at the nodes as follows:

$Fz_{max} = \sqrt{(Fxz^2 + Fyz^2)}$

Display stresses in 2D elements

Display stresses

To display stresses graphically:

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **2D Results** group, select **Stresses**, then from the droplist beneath select the result required.

Which stresses can be displayed?

The topmost item in the droplist, is simply the bearing pressure. This can only be viewed for foundation mats.

The next 6 stresses can be viewed for slabs or mats and are calculated directly from the nodal forces and moments:

1. σ_x top - in-plane axial stress in the x direction, top surface
2. σ_y top - in-plane axial stress in the y direction, top surface
3. τ_{xy} top - in-plane shear stress in xy direction, top surface
4. σ_x bottom - in-plane axial stress in the x direction, bottom surface
5. σ_y bottom - in-plane axial stress in the y direction, bottom surface
6. τ_{xy} bottom - in-plane shear stress in xy direction, bottom surface

---

NOTE The above stresses are available for loadcases & combinations, but not envelopes.

The next 4 stresses are determined from the above stresses:

1. σ_x max tension - maximum tension stress in the x direction for both surfaces
2. σ_y max tension - maximum tension stress in the y direction for both surfaces
3. σ_x max compression - maximum compression stress in the x direction for both surfaces
4. σ_y max compression - maximum compression stress in the y direction for both surfaces

---

NOTE The above 4 stresses are available for loadcases, combination & envelopes.

The center stress values are calculated directly from the forces and moments:

1. σ_x in-plane - in-plane axial stress in the x direction, center
2. σ_y in-plane - in-plane axial stress in the y direction, center

The last 4 stresses are determined from the above 2 stresses:

1. σ_x in-plane tension - maximum tension stress in the x direction center
2. σ_y in-plane tension - maximum tension stress in the y direction center
3. σ_x in-plane compression - maximum compression stress in the x direction center
4. σ_y in-plane compression - maximum compression stress in the y direction center

How might these results be used?

Users performing the design of structures with concrete core walls are interested to know which panels within the walls are cracked. Which panels are cracked can be determined by comparing the maximum tensile stress in each panel to the concrete tensile strength.

Tekla Structural Designer calculates stress values from the gross section properties (ignoring the reinforcement). To determine cracked panels, you can see the maximum tension (and compression) stress in each direction for each panel, across loadcases, combinations and envelopes.

Calculation of in-plane axial and shear stress

For loadcases and combinations, by using the 2D element thickness, stresses (based on the gross section properties) can be calculated from the forces at the nodes:

$\sigma_x top = F_x / t + 6M_x / t^2$

$\sigma_y top = F_y / t + 6M_y / t^2$

$\tau_{xy}top = F_{xy} / t + 6M_{xy} / t^2$

$\sigma_x bottom = F_x / t - 6M_x / t^2$

$\sigma_y bottom = F_y / t - 6M_y / t^2$

$\tau_{xy}bottom = F_{xy} / t - 6M_{xy} / t^2$

$\sigma_x in\text{-}plane = F_x / t$

$\sigma_y in\text{-}plane = F_y / t$

NOTE
- Tension stresses are positive
- Compression stresses are negative

The process for enveloping the above values is the same as that used for other envelopes. For each of the items, a pair of values is found, these are the minimum & maximum values across all loadcases and combinations.

Calculation of maximum tension and compression stress for loadcases and combinations

For loadcases and combinations, the maximum tension and compression values are determined for a specific direction by finding the maximum or minimum of the top and bottom stresses in that direction:

σ_x max tension = Max ($\sigma_x top$, $\sigma_x bottom$, 0.0)

σ_y max tension = Max ($\sigma_y top$, $\sigma_y bottom$, 0.0)

σ_x max compression = Min ($\sigma_x top$, $\sigma_x bottom$, 0.0)

σ_y max compression = Min ($\sigma_y top$, $\sigma_y bottom$, 0.0)

σ_x in-plane tension = Max ($\sigma_x in\text{-}plane$, 0.0)

σ_y in-plane tension = Max ($\sigma_y in\text{-}plane$, 0.0)

σ_x in-plane compression = Min ($\sigma_x in\text{-}plane$, 0.0)

σ_y in-plane compression = Min ($\sigma_y in\text{-}plane$, 0.0)

For envelopes, the maximum tension and compression values are determined by applying the above equations to the enveloped values. Envelopes yield two values for each of the 4 entries in the dropdown.

Calculation of maximum tension and compression stress for envelopes

For envelopes, the maximum tension and compression values are determined by applying the above equations for loadcases and combinations to the enveloped values.

Envelopes yield two values for each of the 4 entries in the droplist.

For **tension stresses** (x or y - only x shown for brevity) the values are returned are:

σ_x max tension = m_1 / m_2 , where:

m_1 = Min ($\sigma_x$top $_{\text{max across all cases \& combs}}$, $\sigma_x$bottom $_{\text{max across all cases \& combs}}$, 0.0)

m_2 = Max ($\sigma_x$top $_{\text{max across all cases \& combs}}$, $\sigma_x$bottom $_{\text{max across all cases \& combs}}$, 0.0)

For **compression stresses** (x or y - only x shown for brevity) the values returned are:

σ_x max compression = m_1 / m_2 , where:

m_1 = Min ($\sigma_x$top $_{\text{min across all cases \& combs}}$, $\sigma_x$bottom $_{\text{min across all cases \& combs}}$, 0.0)

m_2 = Max ($_x$top $_{\text{min across all cases \& combs}}$, $\sigma_x$bottom $_{\text{min across all cases \& combs}}$, 0.0)

In summary the values visible in the tooltip are:

σ_x max tension	=	Min ($\sigma_x$top$_{\text{max across all cases \& combs}}$, $\sigma_x$bottom$_{\text{max across all cases \& combs}}$, 0.0) / Max ($\sigma_xtop_{\text{max across all cases \& combs}}$, $\sigma_x$bottom$_{\text{max across all cases \& combs}}$, 0.0)
σ_y max tension	=	Min ($\sigma_y$top$_{\text{max across all cases \& combs}}$, $\sigma_y$bottom$_{\text{max across all cases \& combs}}$, 0.0) / Max ($\sigma_ytop_{\text{max across all cases \& combs}}$, $\sigma_y$bottom$_{\text{max across all cases \& combs}}$, 0.0)
σ_x max compression	=	Min ($\sigma_x$top$_{\text{min across all cases \& combs}}$, $\sigma_x$bottom$_{\text{min across all cases \& combs}}$, 0.0) / Max ($\sigma_xtop_{\text{min across all cases \& combs}}$, $\sigma_x$bottom$_{\text{min across all cases \& combs}}$, 0.0)
σ_y max compression	=	Min ($\sigma_y$top$_{\text{min across all cases \& combs}}$, $\sigma_y$bottom$_{\text{min}}$

		across all cases & combs, 0.0) / Max ($\sigma_y top_{min}$ across all cases & combs, $\sigma_y bottom_{min}$ across all cases & combs, 0.0)

Key points when using stress values

1. In Walls X direction is horizontal in plane of wall and Y is vertical
2. "Top"/"Bottom" is dependent on shell local axis system, but if you are only concerned about max values you don't need to worry about this - use the "max" options which consider both faces.
3. For engineers wanting to consider tensile stresses in walls the "σ_y in-plane tension" option will be of greatest interest.
   - This is based purely on the *membrane* tension stress (i.e. ignoring out of plane bending effects).
   - This can be viewed for enveloped results
   - It is very easy to see walls/panels in which no tension stress is developing.
   - In a full 3D view it may be difficult to assess whether a particular cracking stress level is exceeded. Viewing results in 2D views or sub structures may be helpful here.
   - It should be clear that this is based on the concrete section only - reinforcement content is not considered.

Display 2D deflections

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **2D Results** group, select **Deflections**, then from the droplist beneath select the desired deflection type.
4. From the **Text** group select **Deflection** to display values on the diagram.

See also

Customize the display of 2D contours (page 705)

Change result diagram scale settings (page 706)

Animate 1D and 2D deflections (page 668)

Display AsReq contours

AsReq contours can either be displayed as values, or as a pass/fail threshold. Displaying as a pass/fail threshold helps to visualize minimum patch sizes when optimizing panel and patch reinforcement as it highlights the specific areas in which the existing reinforcement is not sufficient.

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).

   NOTE Typically the analysis method will be FE chase-down
3. In the **2D Results** group , select **As Req**.
4. From the first droplist, select the desired direction and slab face.
5. From the second droplist, select **Value**, or **Pass/Fail** as required.

See also

Customize the display of 2D contours (page 705)

Change result diagram scale settings (page 706)

Display wall lines

Tekla Structural Designer automatically creates a wall line at the centroid of every meshed shear wall to facilitate the display of wall forces.

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **2D Integrated Results** group, select **Wall Lines**, then from the droplist in the same group, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the wall line.

   TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 707).

Display core lines

Tekla Structural Designer automatically creates a core line at the centroid of each concrete core to facilitate the display of core forces.

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. Set the analysis type and loading for viewing analysis results (page 664).
3. In the **2D Integrated Results** group, select **Core Lines**, then from the droplist in the same group, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the core line.

NOTE All forces in the results are rotated to be in the axis system of the core line.

TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 707).

See also

Create concrete cores (page 205)

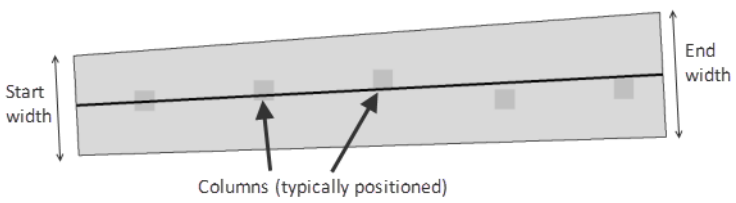
Manage and display result strips

You can place user-defined result strips across 2D element meshes. For these strips, Tekla Structural Designer determines the force and moment results from the shell/plate/membrane nodal analysis results. These results can then be used for design purposes, typically for slab design.

Engineering judgement is required when positioning the strips to ensure suitable design forces are obtained.

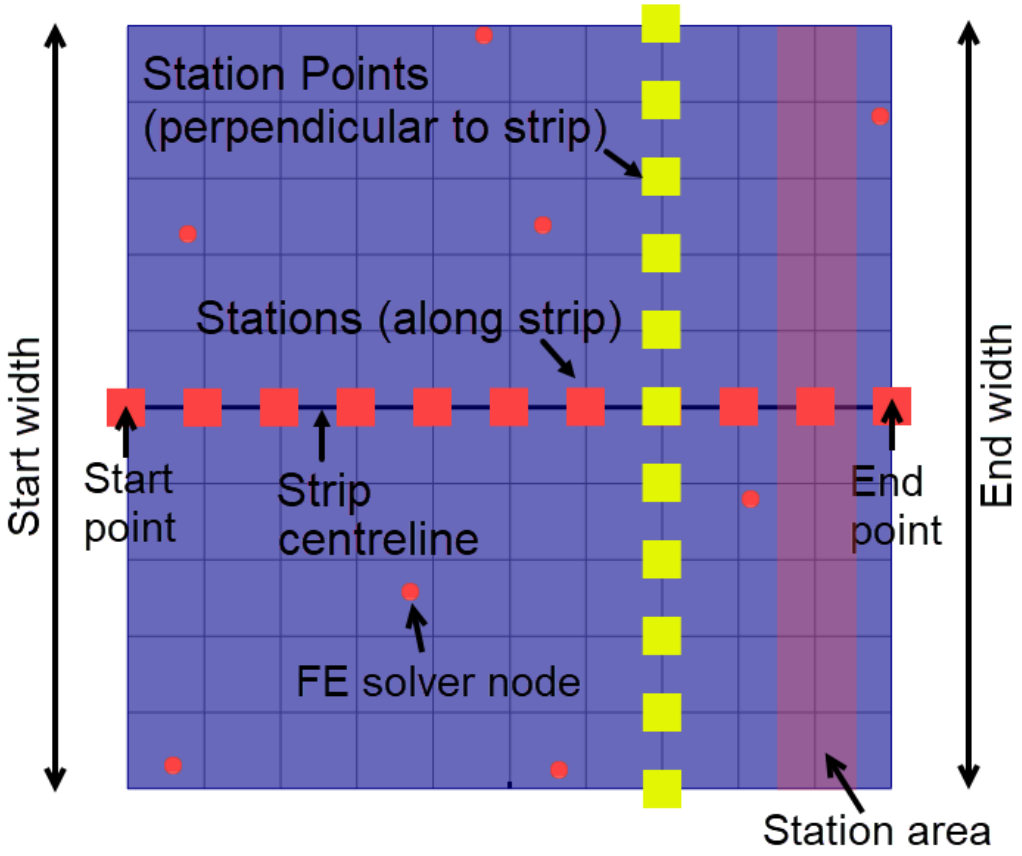
Once a strip has been placed it can be moved if required, or copied in order to create new strips.

By default, strips have parallel edges, but tapering strips can also be defined as shown in the following image:



Overview

The following parameters are used to define a result strip.



Stations and station points

- The **Start point** and **End point** define the strip centreline and direction.
- **Stations** are created at a regular user defined spacing/unit length along the centreline (and also at any points of interest encountered along the centreline).
- **Station Lines** run perpendicular to each station. They extend to the full strip width, or to the edge of the unbroken FE mesh if this is shorter.
- **Station Points** are created at a regular user defined spacing/unit length across the width of the station line.

Determination of forces

- **Station point forces** are determined at every station point by interpolation from surrounding FE solver node values.
  - Station point values represent peak values for the user defined grid spacing, (only being significantly lower than nearby solver node values if there is a dramatic change in value between adjacent nodes).

- **Station forces** are determined from the station point forces in accordance with the strip's **Result Type** property:
  - **Average** - average of the station point values.
  - **Maximum** - maximum of:
    1. All station point values
    2. Any FE solver node found within the "station area"
  - **Centreline** - the station point value on the strip centreline.

Create result strips

1. Open a 2D view of the FE mesh where you want to place the strip.
2. In the **Status bar**, click  **Results View**.

   On the ribbon, the **Results** tab opens.
3. In the **2D Integrated Results** group, click  **Result Strip**.
4. In the **Properties** window, adjust the result strip properties (page 1491) according to your needs.
5. Click the start point of the strip.
6. Click the end point of the strip.

   TIP Neither the start or the end point have to match nodes in the mesh.

   Tekla Structural Designer creates a strip between the selected points.
7. Do one of the following:
   - Continue placing further strips.
   - Press **Esc** to exit the command.

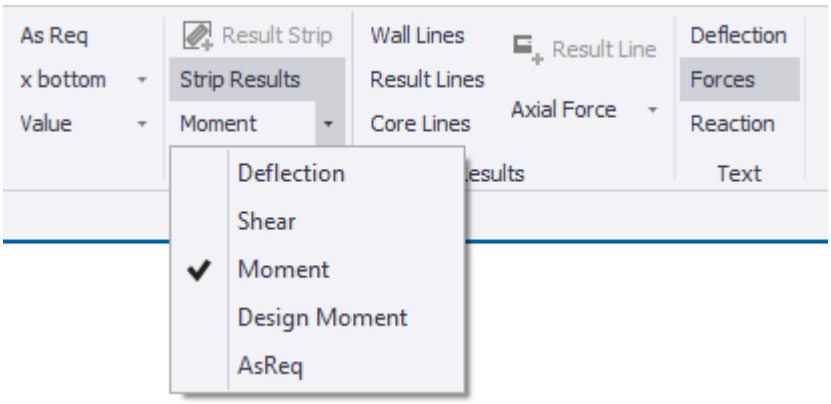
View the results in a 3D results view

NOTE All forces in the strip results are rotated to be in the axis system of the result strip.

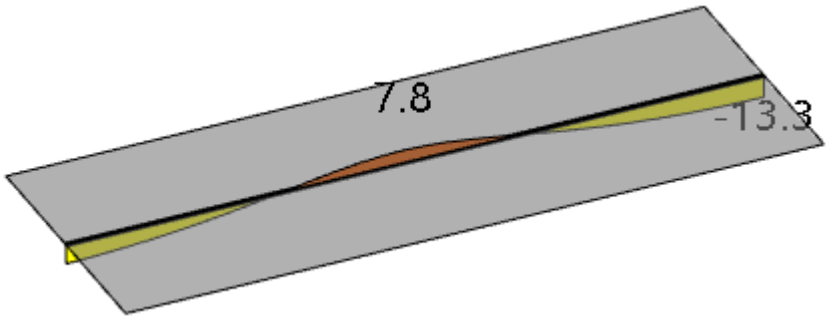
Once you have created a result strip in the model, you can obtain results for it without re-running the analysis. In order to view the results, do the following:

1. Open a 3D view containing the strips whose results you want to view.
2. In the **Status bar**, click  **Results View**. On the ribbon, the **Results** tab is displayed.

3. Using the Method list in the **Result Type** group, select an appropriate analysis method (one in which the slabs containing the strips will have been meshed in the analysis).
4. In the **Loading** list, select the loadcase or combination that you want to display.
5. In the **2D Integrated Results** group, select **Strip Results**
6. In the **2D Integrated Results** group, select the effect that you want to view from the Strip Results list.



Tekla Structural Designer displays the selected effect on the strip, and provided the appropriate Deflection or Forces option has been selected from the **Text** group of the ribbon, its maximum positive and negative values, are also indicated.



View the results in a strip diagrams view

NOTE All forces in the strip results are rotated to be in the axis system of the result strip.

In order to view the results in a strip diagrams view, do the following:

1. Right-click the strip you want to view.

2. In the context menu, select . **Open strip diagrams view**

   Tekla Structural Designer opens the result strip view and a **Loading Analysis** tab is added to the ribbon.

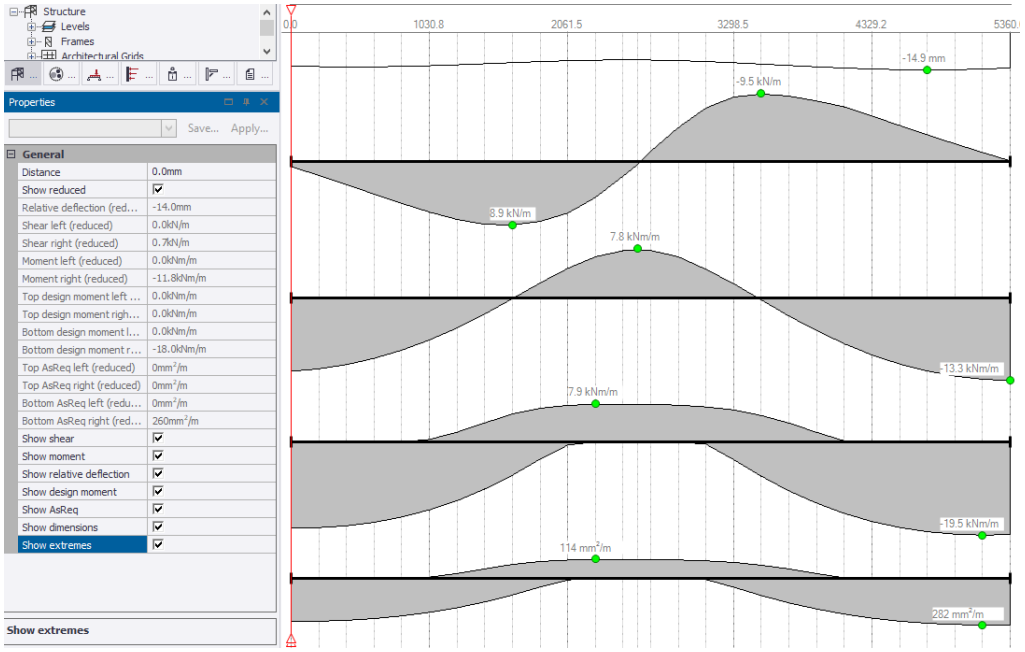
3. In the **Loading** list, select the loadcase, combination or envelope whose results you want to view.

4. On the ribbon, select an appropriate analysis method (one in which the slabs containing the strips will have been meshed in the analysis) using the list in the **Result Type** group.

5. If you are viewing the results for a load combination or envelope, in the **Result Type** group on the ribbon, select whether you want to view the results based on strength or service factors.

6. In the **Direction** group, select the axis type (**Axial**, **Major**, or **Minor**).

   Tekla Structural Designer displays the results according to the selected options.



   You can then further control the results in the **Properties** window according to your needs.

Create a report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. Click **Add** and type the report name in the **Active Style** field.

3. In **Chapters and Options**, locate **Analysis > Result Strip Forces** and drag it into the **Report Structure**.

4. To filter the output apply a model filter (page 1069) and/or loading filter (page 1069) as required.
5. Right-click on **Result Strip Forces** and select **Analysis Method** from the context menu. Check that the selected method is one in which the the slabs containing the strips have been meshed.
6. Click **OK**.

See also

Design and check patches (page 810)

Manage, display and design result lines

You can place user-defined result lines across 2D element meshes. For these lines, Tekla Structural Designer determines the force and moment results from the shell/plate/membrane nodal analysis results. These results can then be used to assess design solutions, typically for the design of wall panels with openings.

Engineering judgement is required when positioning the lines to ensure suitable design forces are obtained.

Create result lines

1. Open a 2D view containing the wall or slab within which you want to place the strip.
2. In the **Status bar**, click  **Results View**. The **Results** toolbar opens.
3. In the **2D Integrated Results** group, click  **Result Line**.
4. Click the start point of the line.
5. Click the end point of the line.

   TIP Neither the start or the end point have to match nodes in the mesh.

   Tekla Structural Designer creates a result line between the selected points.
6. Do one of the following:
   - Continue placing further result lines.
   - Press **Esc** to exit the command.

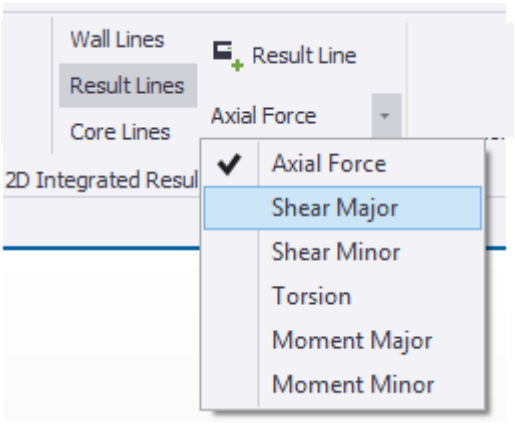
View the analysis results for result lines

NOTE An analysis must be performed after adding or editing result lines in order to obtain up to date results.

To view the results, do the following:

1. Open a view containing the result lines whose results you want to view.
2. In the **Status bar**, click **Results View**.

   The **Results** toolbar opens.
3. Set the analysis type and loading for viewing analysis results (page 664).
4. In the **2D Integrated Results** group, click **Result Lines**, then from the droplist in the same group, select the desired result.
5. From the droplist in the **2D Integrated Results** group, select the effect that you want to view.



6. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the result line.

NOTE All forces in the results are rotated to be in the axis system of the result line.

TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 707).

See also:

2D result lines (how to view as tabular results) (page 711)

Interactively design result lines

1. Open a view containing the result line to be designed.
2. Hover the mouse pointer over the result line that you want to design.

   The **Select Entity** tooltip appears.
3. In the **Select Entity** tooltip, navigate to the result line (RL) name by using the arrow keys.

4. Right-click the result line.
5. In the context menu, select **Interactive Design...** (as a column section, or as a wall section as required).

   The selected (column or wall) interactive design dialog is displayed with the Additional Design Cases page populated with the result line results.
6. Modify the reinforcement parameters as required to achieve a satisfactory design, and click the **Check** button to examine the detailed design results.
7. Click **OK** to save the designed reinforcement to the result line.

NOTE For help on using the interactive dialogs, see and .

Features of result line design

- The interactive design dialog features full interactive manual selection of both lateral and vertical reinforcement, interaction diagrams and auto-design and check options, just as for a regular wall/ column section interactive design.
- The wall length/ column depth considered is the Result Line length.
- Result lines are not constrained to cross sections in the horizontal plane - vertical sections can be used above/between openings to investigate forces and reinforcement requirements in “coupling beams”.
  - The section design always considers the main bars (running perpendicular to the cross section) as being on the inner layer, from a design perspective this will tend to be conservative but the engineer should give this some consideration when working with non-horizontal sections.
  - The design does not consider the existing reinforcement specified in wall properties - only that which is defined in the Interactive design dialog.
  - Currently this feature is not linked with Reports and so it is envisaged output will be via screenshots of the interactive design and check results dialogs.
- The same result line can be used to interactively design as a column, and as a wall. Both designs are saved to the result line independently.

Delete result lines

TIP To delete a result line, ensure that Result Lines are switched on in **Scene Content**.

1. Open a view containing the result line that you want to delete.
2. In the **Quick Access Toolbar**, click **Delete**.

3. In the model, click the result line that you want to delete.

   Tekla Structural Designer deletes the selected result line.

Related video

Interactive design using Result lines

Display mode shapes

1. Open a view and set the view regime to  **Results View**.

   The **Results** ribbon is displayed.
2. In the **Result Type** group, select the required analysis type, e.g. 1st Order modal.
3. In the **Loading** list, select the required loadcase or combination.
4. In the second list of the **Result Type** group, select the desired mode.
5. Still in the **Result Type** group, select between **Mass Nomalized** and **Unity Nomalized** results.

   NOTE In the graphical view the only visible difference between Mass and Unity Normalized results is the tooltip information displayed as you hover over nodes (there is no change to mode shape).
6. Select the diagram that you want to display.

See also

Display analysis results in a Results View (page 662)

RSA seismic results

For information on how RSA seismic results are displayed for different loadcases and combinations, see the following paragraphs.

Result Type

When the result type is 1st or 2nd order RSA seismic, the results that can be displayed depend on the type of the currently selected loadcase or combination.

Mode Shapes

Mode shapes can be displayed for:

- Effective seismic weight combination:
  - List of all modes returned by the modal analysis

Mode shapes are not displayed for:

- RSA torsion loadcases
- Static loadcases included in the RSA seismic combination
- RSA seismic loadcases and combinations

SEE ALSO • Display mode shapes (page 701)

Support Reactions

Support reactions are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at each support and the result is displayed as both positive and negative. All relevant modes: A standard diagram is displayed.
RSA torsion loadcases	A standard diagram is displayed.
Static loadcases included in the RSA seismic combination	A standard diagram is displayed.
Effective seismic weight combinations	Not available
RSA seismic combinations	The diagram displays two values at each support: • Static value + seismic value • Static value - seismic value

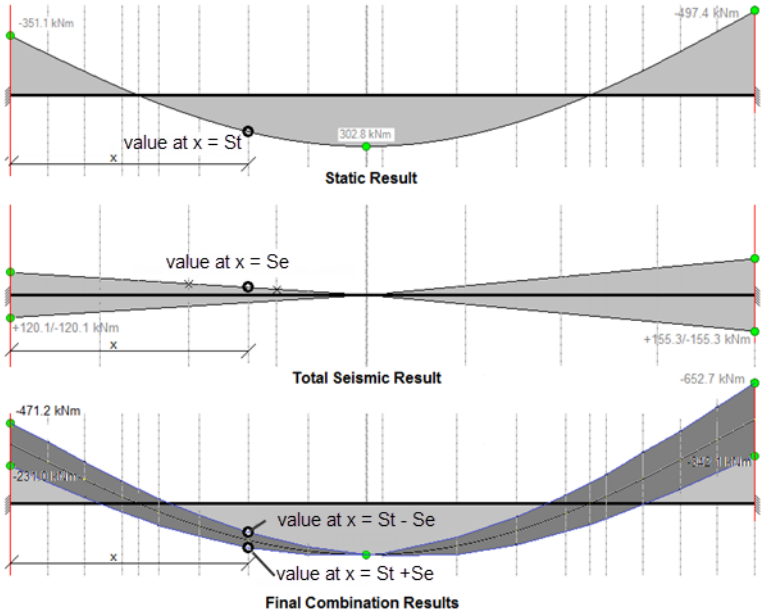
SEE ALSO • Display reactions (page 665)

1D Element Results

1D element results (and Load Analysis View results) are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at various points along each member and displayed on both the positive

Loadcase or combination	Display method
	and negative side of the diagram. Therefore, the diagrams are always symmetrical about the base line. All relevant modes: A standard enveloped diagram is displayed.
RSA torsion loadcases	Displayed as per 1st order linear analysis.
Static loadcases included in the RSA seismic combination	Displayed as per 1st order linear analysis.
Effective seismic weight combination	Not available
RSA seismic combinations	An envelope is drawn displaying the seismic results above and below the static result: • Baseline goes through the static values • Top line = static value + seismic value • Bottom line = static value - seismic value



SEE ALSO

- Display 1D results (page 667)

Story Shear

Story shears are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at each position of interest and the result is displayed as both positive and negative. All relevant modes: A standard diagram with a single value at each point of interest is displayed.
RSA torsion loadcases	Displayed as per 1st order linear analysis.
Static loadcases included in the RSA seismic combination	Displayed as per 1st order linear analysis.
Effective seismic weight combination	Not available
RSA seismic combinations	The diagram displays two values at each point of interest: • Static value + seismic value • Static value - seismic value

SEE ALSO • Display sway, drift, and story shear (page 668)

Seismic Forces

The magnitude and distribution of forces for the seismic torsion loadcases can be displayed.

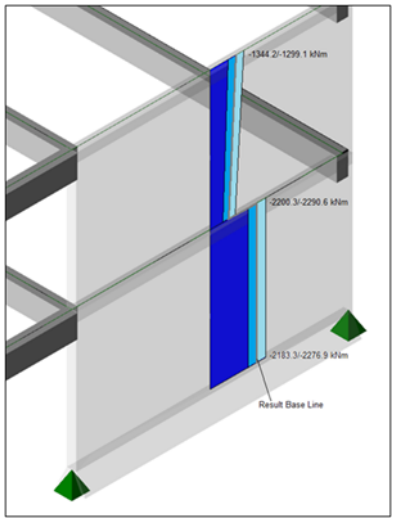
Note that a static analysis is used for seismic torsion irrespective of whether an ELF or RSA method has been used.

SEE ALSO • View the magnitude and direction of seismic equivalent lateral forces (page 685)

Concrete Wall Results

Concrete wall results are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at points along the wall line and displayed on both the positive and negative side of the diagram. Therefore, the diagrams are always symmetrical about the wall line. All relevant modes: A standard diagram is displayed.
RSA torsion loadcases	A standard diagram is displayed.
Static loadcases included in the RSA seismic combination	A standard diagram is displayed.
Effective seismic weight combinations	Not available
RSA seismic combinations	An envelope is drawn displaying the seismic results above and below the static result: • Baseline goes through the static values • Top line = static value + seismic value • Bottom line = static value - seismic value



Customize the display of 2D contours

By default, all contour diagrams consist of 10 evenly sized contours, each accounting for 10% of the total range. If necessary, you can increase or decrease the number of contours, and also change the size and the color of individual contours.

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.

   NOTE If you intend to make the changes to the contours in the current model, ensure that you are making changes to the active settings set.

2. Go to **Scene** --> **Contours** .
3. According to your needs, do one or more of the following:

To	Do this
Add new contours	• Click **Split**.
Delete existing contours	• Click **Delete**.
Modify the size of contours	• In the **Size [%]** column, type new values in the appropriate cells.
Change the color of contours	a. In the **Color** column, click the color that you want to change. b. Select a new color. c. Click **OK**.
Revert to the default contour settings	• Click **Reset**.

4. Click **OK**.

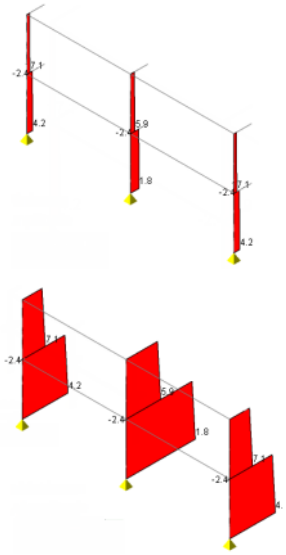
See also

Display analysis results in a Results View (page 662)

Change result diagram scale settings

- Move the sliders in the **Scale Settings** group according to adjust the diagram amplitude according to your needs.

The below example shows the effect on an axial force diagram when the 1D Results slider is increased from minimum to maximum.



See also

Display analysis results in a Results View (page 662)

Display 2D view in isometric projection

Because the diagrams are plotted on each element in the planes in which they act when you are working in a 2D view, you will need switch on an isometric display to see the out of plane forces.

If the 2D view is currently displayed in plan, the **2D**/**3D** toggle button at the bottom right corner of the screen is labeled **3D**.

- To change the current view, click the **2D**/**3D** toggle button.

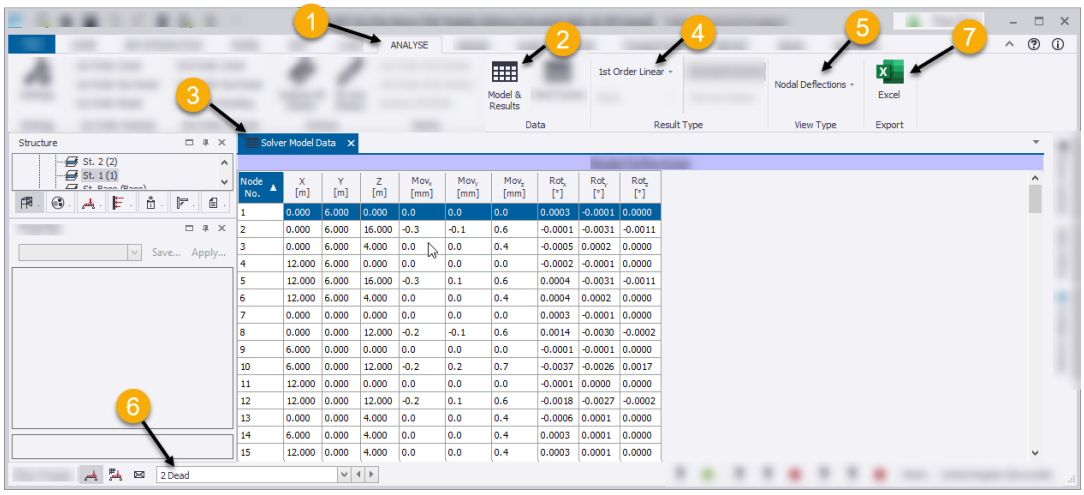
See also

Display analysis results in a Results View (page 662)

Display tabular analysis results in a Solver Model Data view

To display tabular model and results data, first go to the **Analyze** ribbon **(1)**.

Then click  **Model & Results (2)**.



This opens a **Solver Model Data** view **(3)** and adds extra commands to the ribbon to allow you to choose a **Result Type (4)** and a **View Type (5)**. Note that some view types are only available for specific result types.

For most view types you will also need to select the **loading (6)**.

Once a view type is displayed, the data can be sorted by any column into ascending (▲) or descending (▼) order simply by clicking a column header.

Data can be exported to Excel to facilitate further result processing by clicking **Excel (7)**.

To find out more about the view types, see below:

Nodes

To view node coordinates and degrees of freedom:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.

2. In the upper list of the **Result Type** group, select the solver model result type required.

3. In the **View Type** list, select **Nodes**.

   Node coordinates and degrees of freedom are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Nodes** chapter.

Elements

To view solver element properties:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select the solver model type required.
3. In the **View Type** list, select  **Elements**. Solver element properties are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Elements** chapter.

Nodal forces

NOTE This view only allows you to view support reactions referenced by node - to view support reactions referenced by support or by member supported, refer instead to the instructions for viewing **Foundation reactions** tabular data (located under a separate heading further down in this topic).

To view support reactions referenced by node:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Nodal Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select **Imposed Load Reduction** (or **Live Load Reduction** if working to US regional codes) if you want to view the reactions with imposed/live reductions applied.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the reactions for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Nodal Forces** chapter.

Nodal deflections

To view nodal deflections as tables:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Nodal Deflections**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Nodal Deflections** chapter.

Element end forces

To view element end forces:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Element End Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.

5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**.

   Tekla Structural Designer displays the raw analysis results of element end forces. This means that no axial load reductions have been applied.

NOTE If an asterisk is shown next to an element number, this signifies that the results are actually output at the end of a rigid arm that you have modeled, rather than at the node itself.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Element End Forces** chapter.

Wall lines

This table includes the major/minor shears and moments, axial force, and torsion at five locations for each stack within each wall line.

It can be viewed as follows:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Wall Lines**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Wall Line Forces** chapter.

2D result lines

To view the results of the result lines in your model as tables:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select 2D **Result Lines**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > 2D Result Line Forces** chapter.

Core lines

This table includes the major/minor shears and moments, axial force, and torsion at the bottom, mid-point and top for each stack within each core line.

It can be viewed as follows:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Core Lines**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Core Line Forces** chapter.

Shear only walls

To view panel shear results for shear only walls as tables:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select 2D **Shear Only Walls**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Shear Only Wall Forces** chapter.

FE results by node

NOTE **Sign conventions:** In the **FE results by node** table the membrane results Fx and Fy are reported in the traditional analysis convention where tension is positive.

To view the FE results by node as tables:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **FE Results by Node**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select **Imposed Load Reduction** (or **Live Load Reduction** if working to US regional codes) if you want to view the results with imposed/live reductions applied.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > FE Results by Node** chapter.

FE results by shell

To view the FE results by shell as tables:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **FE Results by Shell**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select **Imposed Load Reduction** (or **Live Load Reduction** if working to US regional codes) if you want to view the results with imposed/live reductions applied.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > FE Results by Shell** chapter.

FE slab results by panel

This table gives the max (i.e. maximum positive value of the actions) and min (i.e. absolute maximum negative value of the actions) results for panel nodes, result strips and result lines within each slab panel.

NOTE **Sign conventions:** In the **FE results by node** table the membrane results Fx and Fy are reported in the traditional analysis convention where tension is positive. In the **FE slab results by panel** table this is swapped to the convention traditionally used in design codes/ standards where compression is positive.

To view the results:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select an appropriate analysis method (one in which the slabs containing the strips will have been meshed in the analysis).
3. In the **View Type** list, select **FE Slab Results by Panel**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select **Imposed Load Reduction** (or **Live Load Reduction** if working to US regional codes) if you want to view the results with imposed/live reductions applied.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

If required, you can apply a **Model Filter** to filter the results by slab, level, or by a saved report filter. For instructions on how to do this see: Add and apply a new filter (page 725).

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Analysis > FE Slab Results by Panel** chapter.

FE wall results by panel

This table gives the max (i.e. maximum positive value of the actions) and min (i.e. absolute maximum negative value of the actions) results for panel nodes, wall lines and result lines within each wall panel.

NOTE **Sign conventions:** In the **FE results by node** table the membrane results Fx and Fy are reported in the traditional analysis convention where tension is positive. In the **FE wall results by panel** table this is swapped to the convention traditionally used in design codes/ standards where compression is positive.

To view the results:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select an appropriate analysis method.

3. In the **View Type** list, select **FE Wall Results by Panel**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select **Imposed Load Reduction** (or **Live Load Reduction** if working to US regional codes) if you want to view the results with imposed/live reductions applied.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed accordingly.

If required, you can apply a **Model Filter** to filter the results by wall, or by a saved report filter. For instructions on how to do this see: Add and apply a new filter (page 725).

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Analysis > FE Wall Results by Panel** chapter.

1D member forces

To view the forces along 1D members:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **1D Member Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select whether you want **Live Load Reduction** (for US codes), or **Imposed Load Reduction** (other codes) to be on or off.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed for all members of all materials according to the selections that have been made.

If required, you can apply a **Model Filter** to filter the results by material type, characteristic, construction, fabrication, or by a saved report filter. For instructions on how to do this see: Add and apply a new filter (page 725).

You can also apply **Table Options** to change the number of positions along each member at which results are reported. For instructions on how to do this, see: Working with member forces table options (page 723).

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Member Forces** chapter.

Foundation reactions

This table gives the foundation reactions for member and wall supports. The support co-ordinates and the supported member/wall reference, rotation and size are also included. (Where a single support is supporting more than one member, each member reference will be included.)

- For meshed wall supports the reactions are reported for each support node along the wall.
- If an envelope is selected two rows may be generated for each support. The first row containing the maximum positive values for each of the 6 reaction directions and the second row the maximum negative values.
- If an RSA load is selected two results (one positive and one negative) will be output in a single row for each of the 6 reaction directions.

To view foundation reactions referenced by support:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select an appropriate analysis method.
3. In the **View Type** list, select **Foundation Reactions**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**.
6. If the chosen loading has live load reductions activated (US), or imposed load reductions activated (other codes); in the **Result Type** group select **Live Load Reduction** (US), or **Imposed Load Reduction** (other codes) if you want to view the results with the reductions applied. Results are displayed accordingly.

If required, you can also:

- Apply a **Model Filter** to filter the results by column, wall, or by a saved report filter. For instructions on how to do this see: Add and apply a new filter (page 725)

- Use **Table Options** in order to change the axis system for reporting the reactions. For instructions on how to do this see: Working with foundation reaction axis systems (page 726)

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Foundation Reactions > Foundation Reactions** chapter.

Pile forces

To view the forces in piles:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Pile Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.
5. In the **Result Type** group, select whether you want **Live Load Reduction** (for US codes), or **Imposed Load Reduction** (other codes) to be on or off.
6. In the **Result Type** group, (unless you are viewing loadcase results), select whether you want to view the results for **Strength Factors**, or **Service Factors**. Results are displayed for all piles according to the selections that have been made.

If required, you can apply a **Model Filter** to filter the results by pile type, pile cap, foundation mat, loading, or by a saved report filter. For instructions on how to do this see: Add and apply a new filter (page 725).

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Pile Forces** chapter.

Story forces

To view story forces (page 671):

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.

2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Story Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination. Story forces are displayed according to the selections that have been made.

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Story Forces** chapter.

Story shears

To view story shears (page 671):

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Story Shears**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination. Story shears are displayed according to the selections that have been made.

NOTE If required, the same results can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Story Shears** chapter.

Wall/Braced Bay Story shears

To view wall/braced bay story shears (page 671):

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. Using the droplist in the **Result Type** group, select the required linear, non-linear, chasedown or RSA type.
3. In the **View Type** list, select **Wall/Braced Bay Story Shears**.

4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination. Wall/braced bay story shears are displayed according to the selections that have been made.

Modal frequencies

To view the modal frequencies and modal masses of a particular load combination:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the upper list of the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
4. In the **View Type** list, select **Modal Frequencies**.
5. In the **Loading** list, select the desired load combination.

   Modal frequencies and modal masses of the selected load combination are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Modal Frequencies** chapter.

Summed mass

To view the summed total mass of the modal mass combinations:

1. On the **Analyze** ribbon, click  **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
3. In the **View Type** list, select **Summed Mass**.

   The total mass of the modal mass combinations are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Summed Mass** chapter.

Dynamic masses

To view the total dynamic masses for modal mass combinations:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
3. In the **View Type** list, select **Dynamic Masses**. The total dynamic masses for the modal mass combinations are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Dynamic Masses** chapter.

Active masses by node

To view the dynamic active masses of the nodes:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
3. In the **View Type** list, select **Active Masses by Node**.

   The dynamic active masses of the nodes in your model are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Active Masses by Node** chapter.

Total masses by node

To view total masses by node:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.

3. In the **View Type** list, select **Total Masses by Node**.

   The dynamic total masses of the nodes in your model are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Total Masses by Node** chapter.

Total masses by level

To view total masses by level:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
3. In the **View Type** list, select **Total Masses by Level**.

   The total masses by level are displayed accordingly.

Mode shape by node

To view mode shape by node:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the **Result Type** group, select **2nd Order Buckling**, **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
3. In the **View Type** list, select **Mode shape by Node**.
4. In the second list of the **Result Type** group, select the desired mode.
5. Still in the **Result Type** group, select between **Mass Nomalized** and **Unity Nomalized** results.

   The mode shapes by node are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Mode Shapes by Node** chapter.

Mode shape by level

To view mode shape by level:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the **Result Type** group, select **1st Order Modal**, **1st Order RSA Seismic**, or **2nd Order RSA Seismic** as required.
3. In the **View Type** list, select **Mode shape by Level**.
4. In the second list of the **Result Type** group, select the desired mode.
5. Still in the **Result Type** group, select between **Mass Nomalized** and **Unity Nomalized** results.

   The mode shapes by level are displayed accordingly.

Buckling factors

To view buckling factors:

1. On the **Analyze** ribbon, click **Model & Results**.

   A **Solver Model Data** view is the active view and a **Result Type** group and a **View Type** group are available in the **Analyze** ribbon.
2. In the upper list of **Result Type** group, select **2nd Order Buckling**.
3. In the **View Type** list, select **Buckling Factors**.
4. In the **Loading** list, select the desired loadcase or combination.

   The buckling factors are displayed accordingly.

NOTE If required, the same information can be output as a report by configuring a model report (page 1066) which includes the **Analysis > Buckling Factors** chapter.

Working with member forces table options

The **Member Forces Table Options** dialog box controls the number of positions along members at which forces and deflections are reported in tabular data views and reports.

Set number of positions to be reported for a 1D Member Forces tabular data view

1. On the **Analyze** tab, click **Table Options**.
2. In the dialog, configure the number of positions as required.

- Separate numbers of positions can be specified for beam and column members.
- The minimum number of beam and column positions is 2, which will report at the two ends of the member only.
- The maximum number of beam and column positions is 50.
- In addition to reporting at fixed positions, points of interest can also be reported. These are locations where a maximum or minimum value of any major/minor force, or major/minor deflection occurs along the member. By default points of interest are automatically included, (but they can be excluded if required).
- If a fixed position is close to a point of interest it can be skipped if required. How close it needs to be to be skipped is defined as a ratio of the span.

3. Click **OK**.

The 1D Member Forces view is reconfigured to the number of positions specified.

Set number of positions to be reported for a Member Forces report

1. On the **Report** tab, click **Model Report...**.

   The **Report Contents** dialog box opens.
2. Select a report configured to include **Analysis > Member Forces** in its Report Structure
3. In Report Structure, right click **Analysis > Member Forces**.
4. In the context menu, select **Settings...**
5. In the dialog, configure the number of positions as required.
   - Separate numbers of positions can be specified for beam and column members.
   - The minimum number of beam and column positions is 2, which will report at the two ends of the member only.
   - The maximum number of beam and column positions is 50.
   - In addition to reporting at fixed positions, points of interest can also be reported. These are locations where a maximum or minimum value of any major/minor force, or major/minor deflection occurs along the member. By default points of interest are automatically included, (but they can be excluded if required).
   - If a fixed position is close to a point of interest it can be skipped if required. How close it needs to be to be skipped is defined as a ratio of the span.
6. Click **OK**.

When the report is generated the 1D Member Forces are reconfigured to the number of positions specified.

Working with model filters

In large structures **Solver Model Data** views often contain a lot of data. You can reduce this for certain view types by applying a **Model Filter**.

NOTE **Model Filters** are only available on the **Analyze** tab when the **View Type** is set to **1D Member Forces**, or **FE Slab Results by Panel**, or **FE Wall Results by Panel**.

Add and apply a new filter

1. Click  **Model Filter** on the **Analyze** tab.
2. Click **Add filter...** in the **Filter** dialog box.
3. Click **New**
4. Choose the required filter category from the list. The chosen filter name appears as a new row in the **Filter** dialog box with three adjacent buttons.
   
5. To configure the filter, click on its name to see the items that are currently selected, then select/unselect individual items as required. You can also click  to select all items, or to unselect all items.

If this is the only filter in the dialog box, the view is automatically filtered to display results for the filtering criteria.

If multiple filters are listed, you are given two options:

- **Match all rules** - displays only those results that match **all** of the filtering criteria
- **Match any rule** - displays results that match **any** of the filtering criteria

---

NOTE When you add a new filter from the **Analyze** tab, it is also automatically made available as a report filter on the **Report** tab (provided it is a type suitable for reports).

Turn off a filter

Initially member force results are displayed for all members of all materials. If you want to display results for selected materials only, you can do so by setting up a materials filter as follows:

1. Click **Model Filter** on the **Analyze** tab. All the filters that are currently applied are listed.
2. Click next to the filter you want to turn off.
3. Close the **Filter** dialog box.

NOTE Clicking does not delete the filter, it just turns it off in the current solver model data view.

Turn on an existing filter

1. Click **Model Filter** on the **Analyze** tab.
2. Click **Add filter...** in the **Filter** dialog box.
3. Click **Saved Report Filter**
4. Choose the required filter category from the list. The chosen filter name appears as a new row in the **Filter** dialog box.

NOTE You can turn on existing filters that have been set up from the **Report** tab in addition to those set up from the **Analyze** tab.

If only one filter is listed in the dialog box, the view is automatically filtered to display results for the selected items in the filter.

If there are multiple filters listed, then how they are displayed depends on your choice at the bottom of the dialog box.

- **Match all rules** - displays only those results that match all of the filters
- **Match any rule** - displays results that match any of the filters

Rename a filter

While filters can be applied from both the **Analyze** tab and the **Report** tab, they can only be renamed from the **Report** tab.

1. On the **Report** tab, in the **Filters** group, select the filter type. The existing filters of the chosen type are listed under **Available filters**.
2. Select the filter to be renamed, and then enter the name required.
3. Click **OK**

Working with foundation reaction axis systems

The following options are available for setting the axis system that foundation reactions align to:

- Local Support (default)
- Building Direction
- Angle to Global (X and Y)

The method for setting the axis system varies depending on whether you want to display the reactions in a graphical results view, a tabular data view, a report, or a drawing. Instructions for each method are provided below.

Also given below in the 'Understanding the axis system options' section are details of the sign conventions applicable along with some examples.

Report Foundation Reactions in multiple axis systems

Set the axis system in a graphical results view

Once you have chosen the support reactions to be displayed in a graphical results view (page 665), you can select the axis system as follows:

1. in the **Reactions** group on the **Results** tab, click **Settings**  .
2. In the dialog, select the required axis system. To find out more about the axis system choices, see **Understanding the axis system options** below.
3. Click **OK**.

The Reactions graphical results view is reconfigured to display the reactions in the selected axis system.

Set the axis system for a tabular data view

Once you have displayed support reactions in a tabular data view (page 717), you can select the axis system as follows:

1. On the **Analyze** tab, click **Table Options**.
2. In the dialog, select the required axis system. To find out more about the axis system choices, see **Understanding the axis system options** below.
3. Click **OK**.

The Foundation Reactions tabular data view is reconfigured to display the reactions in the selected axis system.

Set the axis system for a report

1. On the **Report** tab, click **Model Report...**.

   The **Report Contents** dialog box opens.

2. Select a report configured to include **Foundation Reactions** in its Report Structure
3. In Report Structure, right click **Foundation Reactions > Foundation Reactions**.
4. In the context menu, select **Settings...**
5. In the dialog, select the required axis system. To find out more about the axis system choices, see **Understanding the axis system options** below.
6. Click **OK**.
7. Click **OK** once more to close the Report Contents dialog.

When the report is generated the reactions are reported in the selected axis system.

Set the axis system for a drawing

1. On the **Draw** tab, click **Settings**.

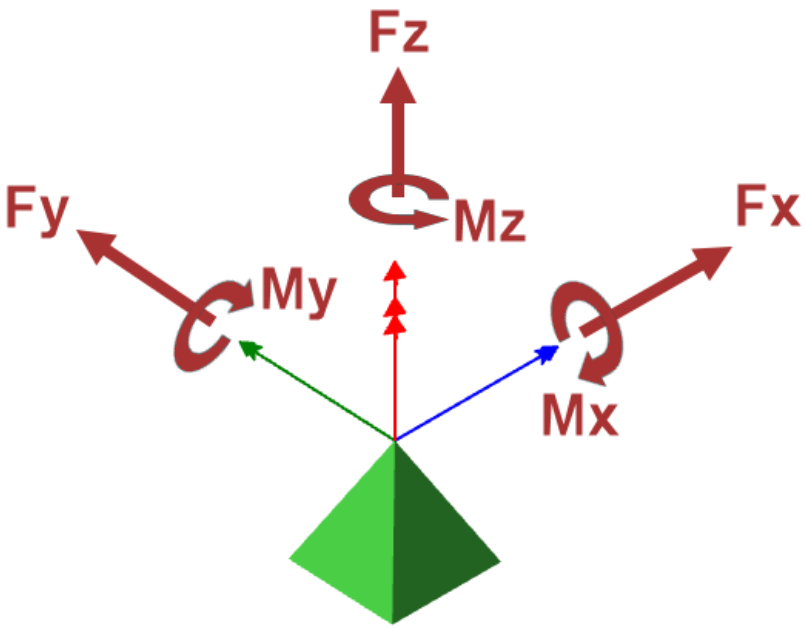
   The **Drawing Settings** dialog box opens.
2. Click **Options** --> **Planar Drawings** --> **Foundation Reactions**
3. On the **General** tab, select the required axis system.
4. In the context menu, select **Settings...**
5. In the dialog, select the required axis system. To find out more about the axis system choices, see **Understanding the axis system options** below.
6. Click **OK**.

When a Foundation Reactions drawing is generated the reactions are reported in the selected axis system.

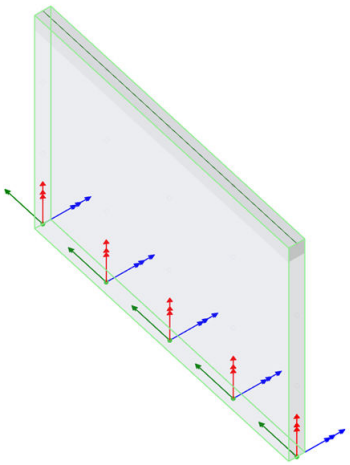
Understanding the axis system options

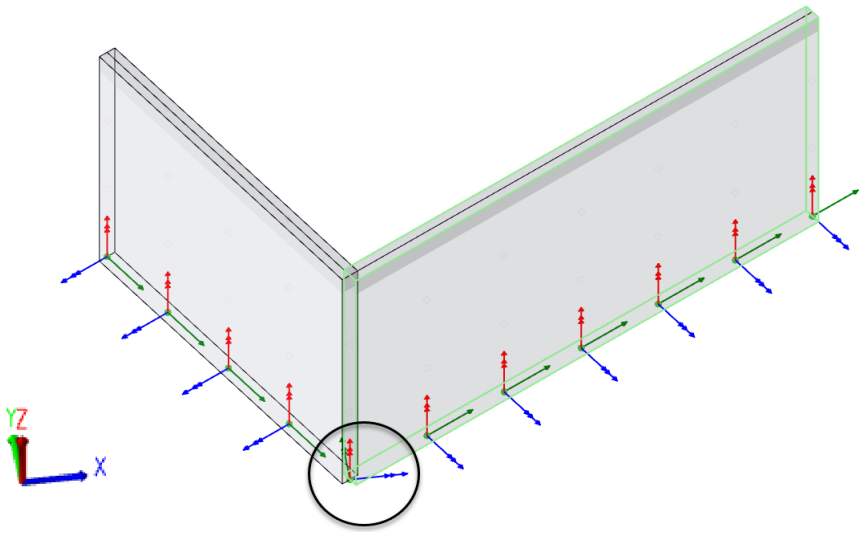
Local Support (default)

The sign convention for support reactions using this axis system is as shown below, with positive moments about an axis acting in a clockwise direction when viewed in the direction of the axis:



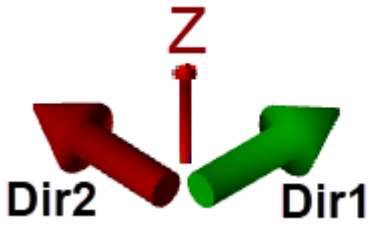
For columns and walls the support local axes are determined by the column/wall orientation.

Column	Wall
	
• Local x aligns to the column major axis • Local y aligns to the column minor axis	• Local x aligns to the wall major axis • Local y aligns to the wall minor axis

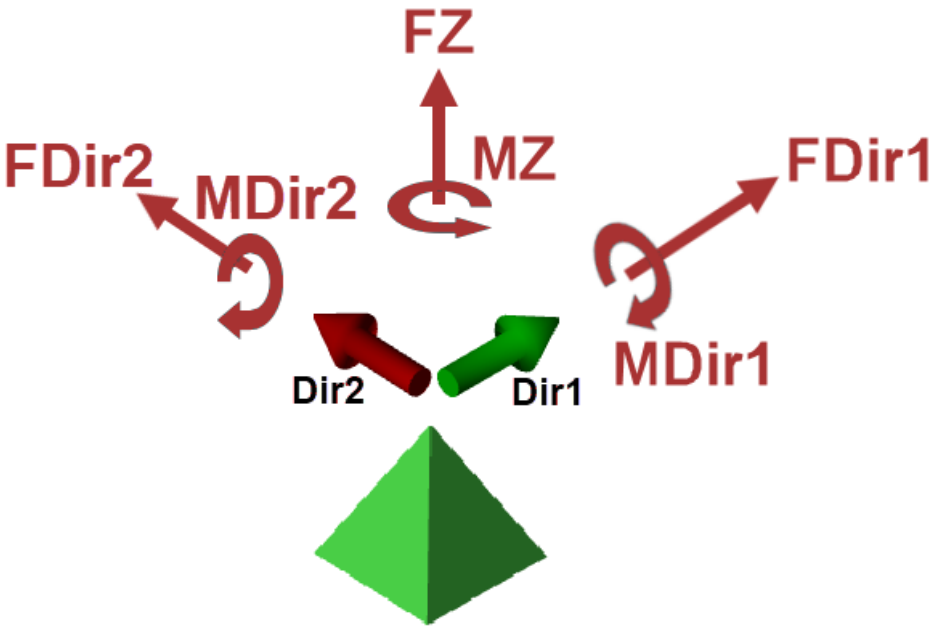
Special case - intersection of two connecting walls

• The shared support is aligned with the global axis system • When local reactions are output to a tabular data view or report, the values at the shared support are split equally between the walls and reported in the same local axis system as the other supports in each respective wall. For more details, see: **Case study (connecting walls)** below.

Building Direction

The Building Direction arrows can be displayed by selecting **Show Building Direction Arrows** in the **Structure Properties** window. These Dir1 and Dir2 arrows indicate the orthogonal horizontal reaction directions when the **Building Direction** axis system for reporting reactions is selected. The third reaction direction is aligned to the Global Z axis.



The sign convention for support reactions using this axis system is as shown below, with positive moments about an axis acting in a clockwise direction when viewed in the direction of the axis:

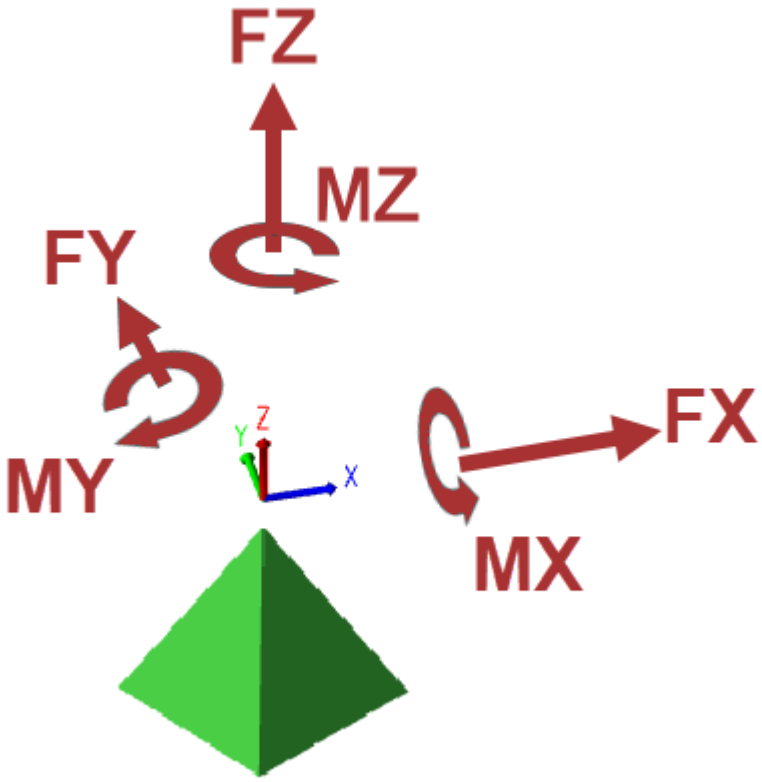


Angle to Global (X and Y): when angle = 0 degrees

The Global Axis direction arrows are permanently displayed in 3D scene views.



The sign convention for support reactions using this axis system is as shown below, with positive moments about an axis acting in a clockwise direction when viewed in the direction of the axis:



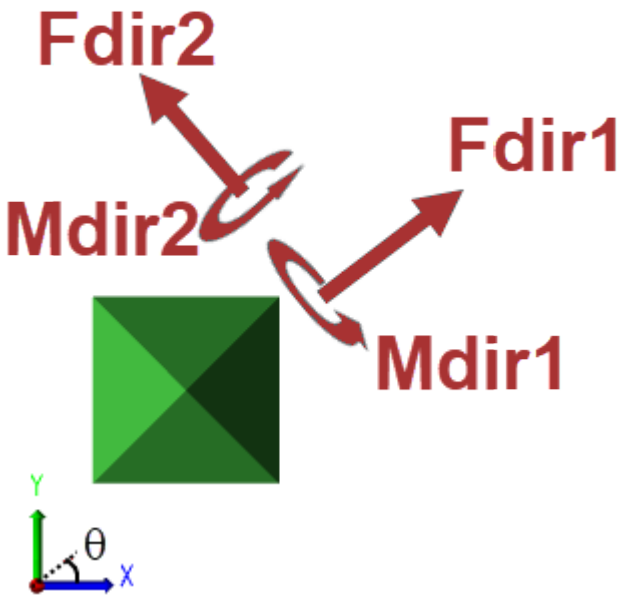
Angle to Global (X and Y): for any other angle

While they are not actually displayed, the dir1 and dir2 arrows that apply when an angle to global has been specified are as follows:



- dir1 is at an anticlockwise angle θ from Global X when looking down in plan.
- dir2 is at 90 degrees to dir1
- The third reaction direction is aligned to the Global Z axis.

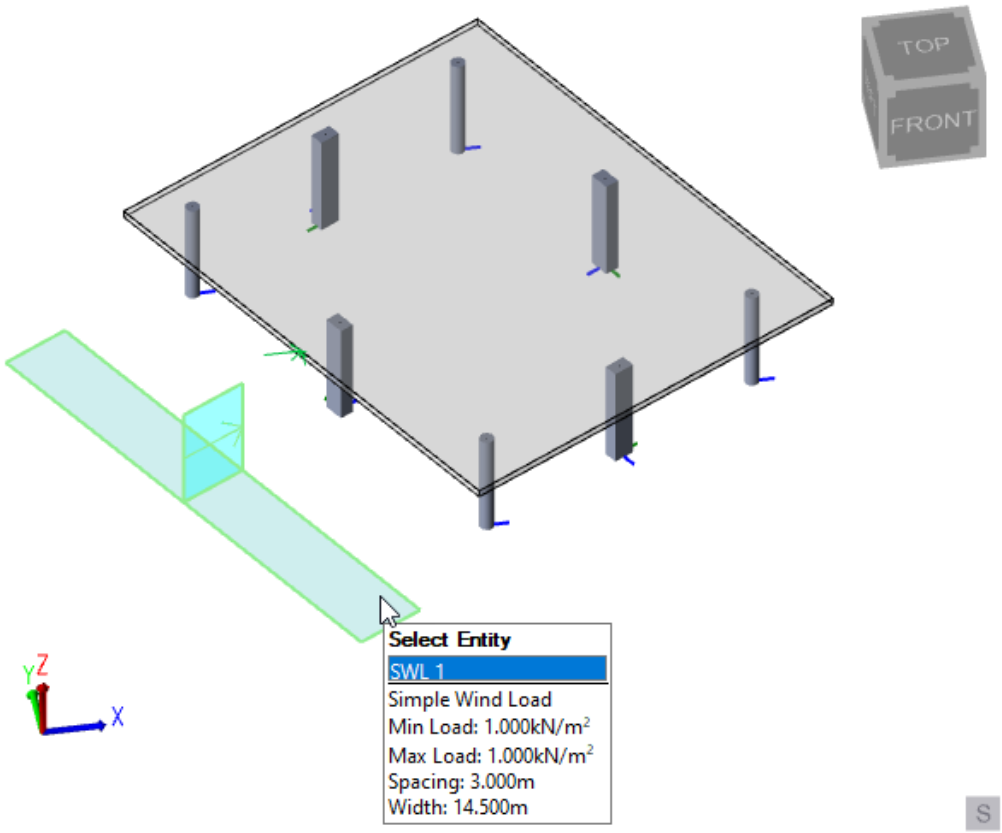
The sign convention for support reactions using this axis system is as shown below, with positive moments about an axis acting in a clockwise direction when viewed in the direction of the axis:



Columns example

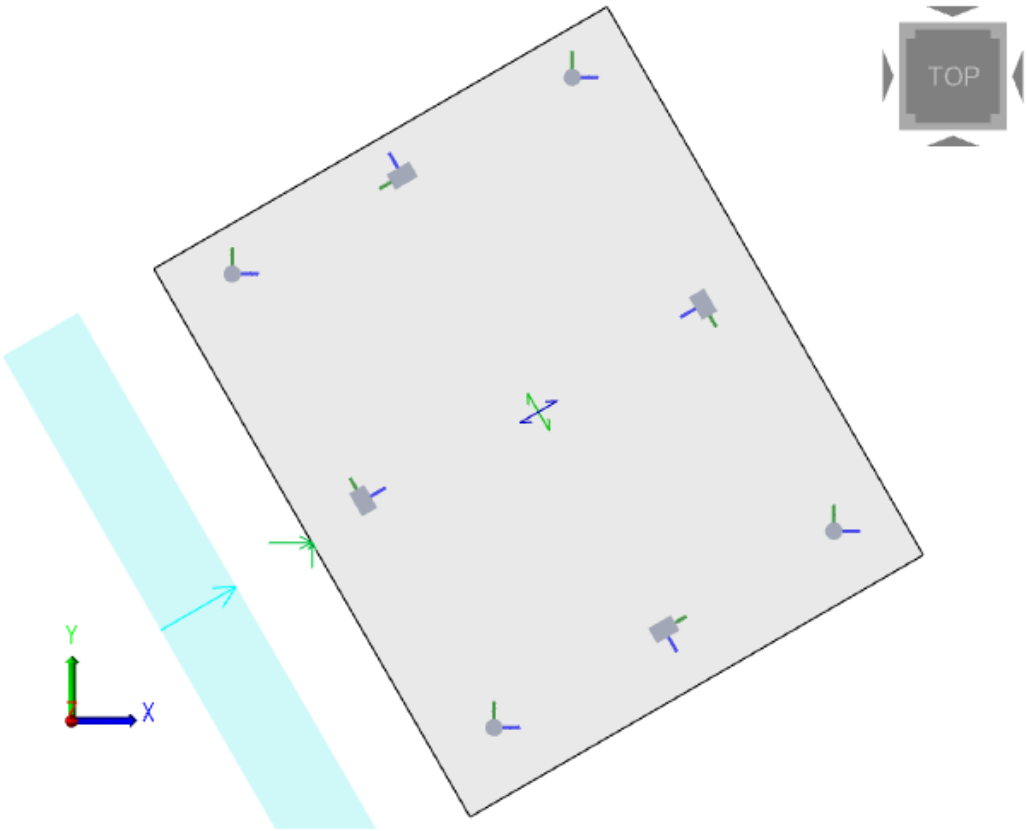
Consider the following structure consisting of a slab supported on eight columns. The building direction has been defined at an angle of 30 degrees to

the global axes. We will investigate the reactions resulting from a wind load which has been applied to one side.



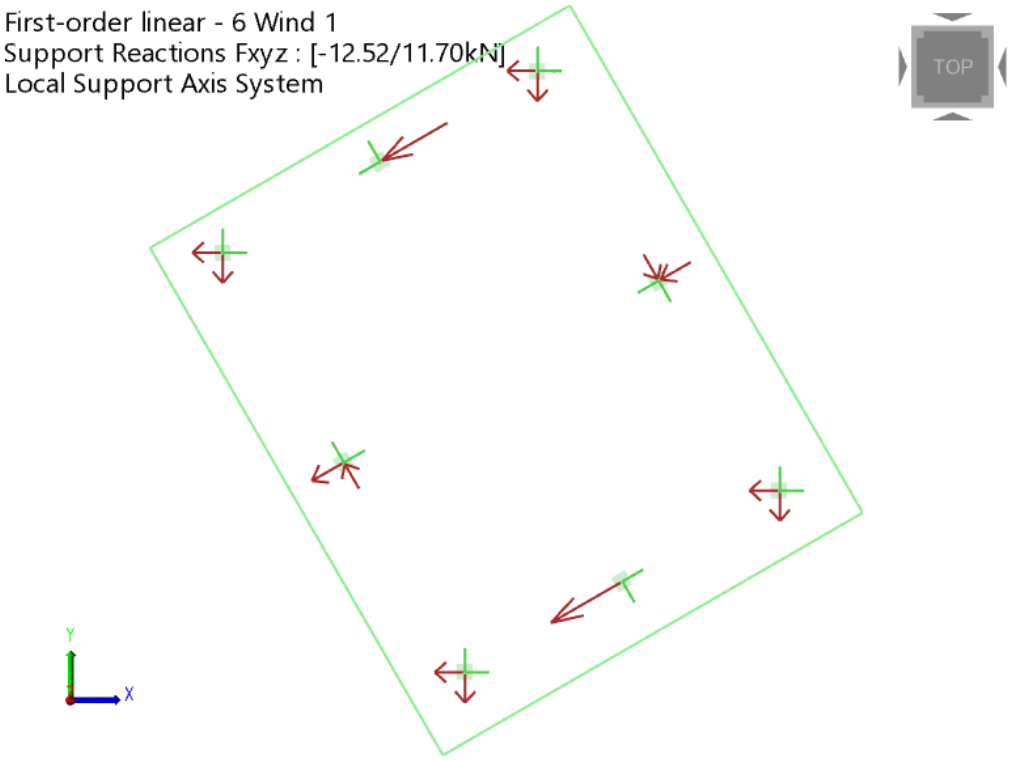
The total wind load is 1.0kN/m^2 * 14.5m * 3m = 43.5kN

When viewed from above you can see how the local axis system of each support is determined by the rotation angle of the column being supported.

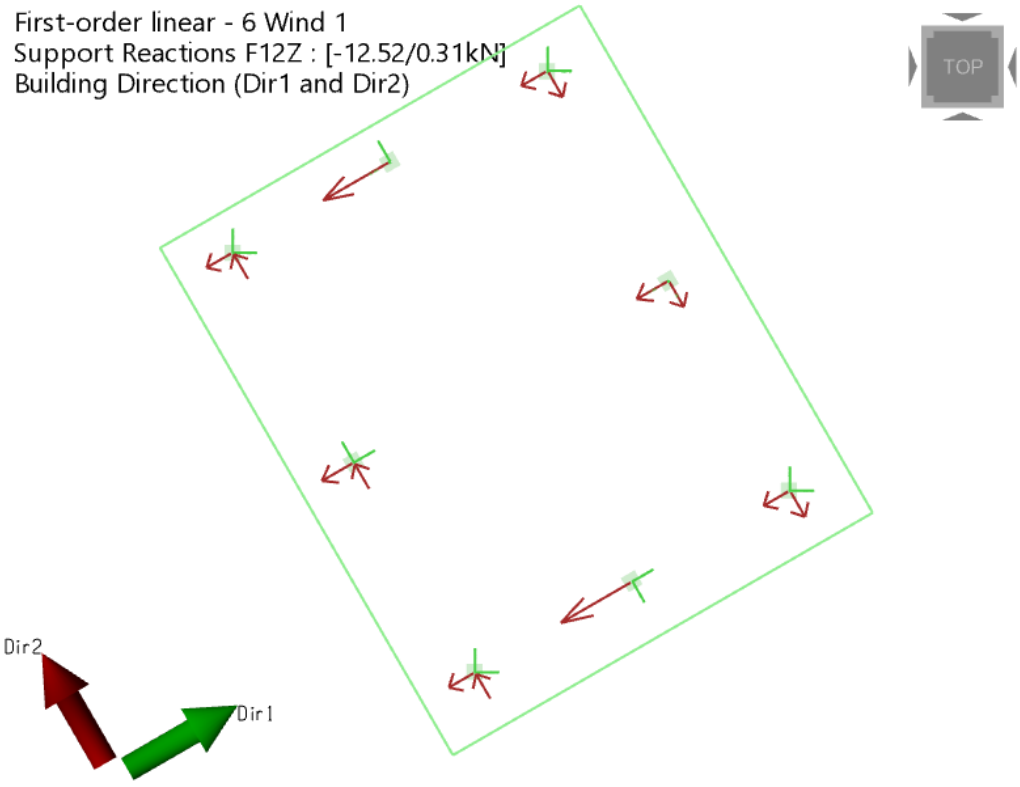


In the Results View, Foundation Reactions can be displayed in each of the axis systems.

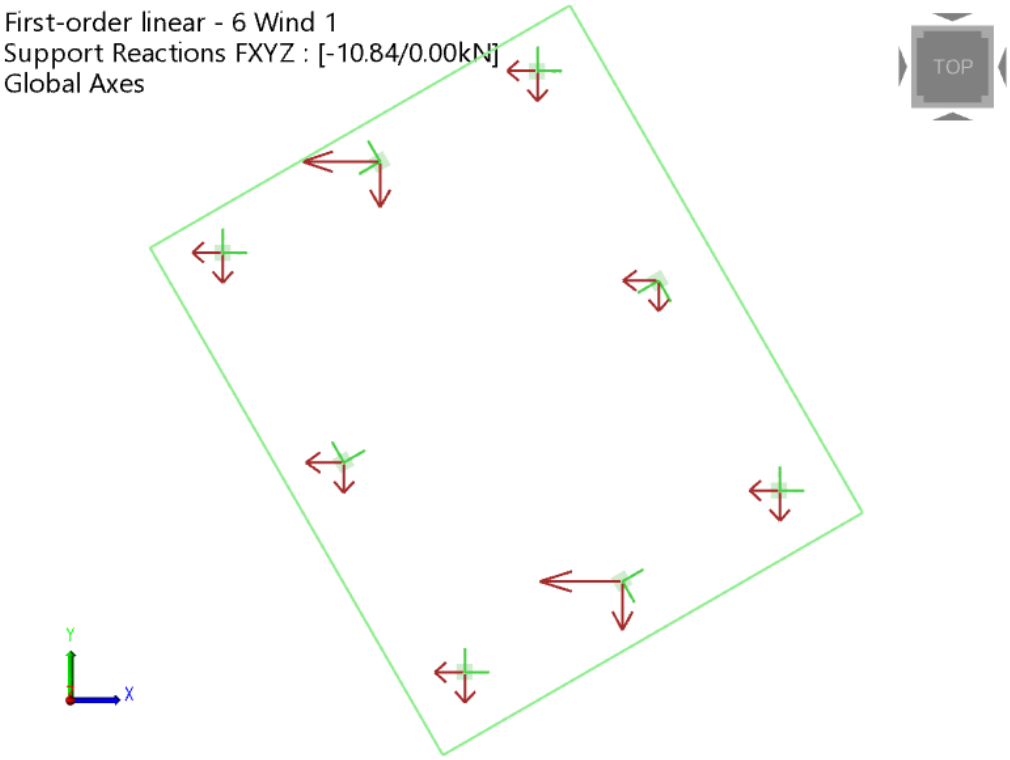
- In the **Local Support** axis system the **Fx** and **Fy** reactions would be displayed as follows:



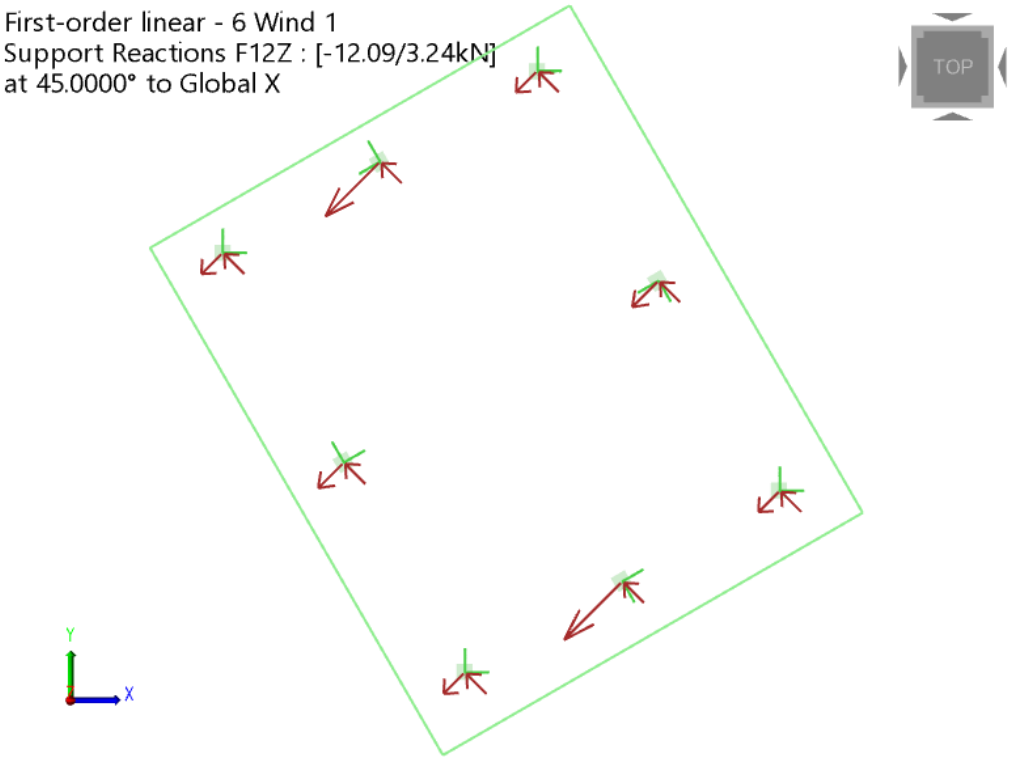
- In the **Building Direction** axis system the **Dir1** and **Dir2** reactions would be displayed as follows:



- In the **Angle to Global** axis system (with angle set to 0 degrees) the **FX** and **FY** reactions would be displayed as follows:



- In the **Angle to Global** axis system (with angle set to 45 degrees) the **F1** and **F2** reactions would be displayed as follows:



The reactions can also be displayed in a Tabular Data View.

- In the **Local Support** axis system the tabular reactions would be displayed as follows:

Solver Model Data

Foundation Reactions, Local Support Axis System

Support	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
SUP A/1	0.000	0.000	0.000	C5	400 Dia	0.0000	-1.9	-1.0	0.0	3.148	-5.777
SUP A/2	-3.000	5.196	0.000	C1	400x600	30.0000	-5.4	0.3	0.0	-0.924	-16.198
SUP A/4	-6.000	10.392	0.000	C9	400 Dia	0.0000	-1.8	-1.0	0.0	2.933	-5.403
SUP B/1	3.897	2.250	0.000	C6	400x600	-60.0000	0.0	-12.5	0.0	37.561	0.000
SUP B/4	-2.103	12.642	0.000	C10	400x600	120.0000	0.0	11.7	0.0	-35.098	0.000
SUP D/1	7.794	4.500	0.000	C8	400 Dia	0.0000	-1.9	-1.1	0.0	3.429	-5.615
SUP D/2	4.794	9.696	0.000	C4	400x600	-150.0000	5.4	0.3	0.0	-0.924	16.198
SUP D/4	1.794	14.892	0.000	C12	400 Dia	0.0000	-1.7	-1.1	0.0	3.213	-5.241

Because the local axes are all in different directions, there is nothing to be gained from summing the above reactions.

- In the **Building Direction** axis system the tabular reactions would be displayed as follows:

Solver Model Data

Foundation Reactions, Building Direction (Dir1 and Dir2)

Support	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_{Dir1} [kN]	F_{Dir2} [kN]	F_z [kN]	M_{Dir1} [kNm]	M_{Dir2} [kNm]
SUP A/1	0.000	0.000	0.000	C5	400 Dia	30.0000	-2.2	0.1	0.0	-0.162	-6.577
SUP A/2	-3.000	5.196	0.000	C1	400x600	30.0000	-5.4	0.3	0.0	-0.924	-16.198
SUP A/4	-6.000	10.392	0.000	C9	400 Dia	30.0000	-2.0	0.1	0.0	-0.162	-6.146
SUP B/1	3.897	2.250	0.000	C6	400x600	30.0000	-12.5	0.0	0.0	0.000	-37.561
SUP B/4	-2.103	12.642	0.000	C10	400x600	30.0000	-11.7	0.0	0.0	0.000	-35.098
SUP D/1	7.794	4.500	0.000	C8	400 Dia	30.0000	-2.2	-0.1	0.0	0.162	-6.577
SUP D/2	4.794	9.696	0.000	C4	400x600	30.0000	-5.4	-0.3	0.0	0.924	-16.198
SUP D/4	1.794	14.892	0.000	C12	400 Dia	30.0000	-2.0	-0.1	0.0	0.162	-6.146

When the above reactions are summed, FDir1 equates to -43.5kN, and FDir2 equates to 0kN.

Solver Model Data

Foundation Reactions, Local Support Axis System

Support	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
SUP A/1	0.000	0.000	0.000	C5	400 Dia	0.0000	-1.9	-1.0	0.0	3.148	-5.777
SUP A/2	-3.000	5.196	0.000	C1	400x600	30.0000	-5.4	0.3	0.0	-0.924	-16.198
SUP A/4	-6.000	10.392	0.000	C9	400 Dia	0.0000	-1.8	-1.0	0.0	2.933	-5.403
SUP B/1	3.897	2.250	0.000	C6	400x600	-60.0000	0.0	-12.5	0.0	37.561	0.000
SUP B/4	-2.103	12.642	0.000	C10	400x600	120.0000	0.0	11.7	0.0	-35.098	0.000
SUP D/1	7.794	4.500	0.000	C8	400 Dia	0.0000	-1.9	-1.1	0.0	3.429	-5.615
SUP D/2	4.794	9.696	0.000	C4	400x600	-150.0000	5.4	0.3	0.0	-0.924	16.198
SUP D/4	1.794	14.892	0.000	C12	400 Dia	0.0000	-1.7	-1.1	0.0	3.213	-5.241

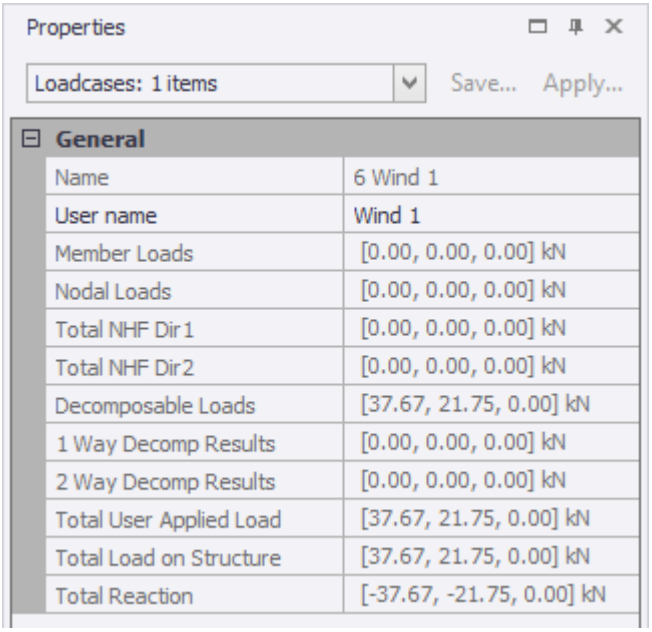
- In the **Angle to Global** axis system (with angle set to 0 degrees) the reactions would be displayed as follows:

Solver Model Data

Foundation Reactions, Global Axes

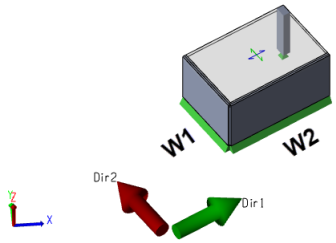
Support	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_x [kN]	F_Y [kN]	F_z [kN]	M_x [kNm]	M_Y [kNm]
SUP A/1	0.000	0.000	0.000	C5	400 Dia	0.0000	-1.9	-1.0	0.0	3.148	-5.777
SUP A/2	-3.000	5.196	0.000	C1	400x600	0.0000	-4.8	-2.4	0.0	7.299	-14.490
SUP A/4	-6.000	10.392	0.000	C9	400 Dia	0.0000	-1.8	-1.0	0.0	2.933	-5.403
SUP B/1	3.897	2.250	0.000	C6	400x600	0.0000	-10.8	-6.3	0.0	18.781	-32.529
SUP B/4	-2.103	12.642	0.000	C10	400x600	0.0000	-10.1	-5.8	0.0	17.549	-30.395
SUP D/1	7.794	4.500	0.000	C8	400 Dia	0.0000	-1.9	-1.1	0.0	3.429	-5.615
SUP D/2	4.794	9.696	0.000	C4	400x600	0.0000	-4.5	-3.0	0.0	8.899	-13.566
SUP D/4	1.794	14.892	0.000	C12	400 Dia	0.0000	-1.7	-1.1	0.0	3.213	-5.241

When the above reactions are summed, FX and FY equate to the Total Reaction figures given in the Project Workspace Loading Summary, (as the Total Reaction is always reported according to the Global Axis system.



Walls example

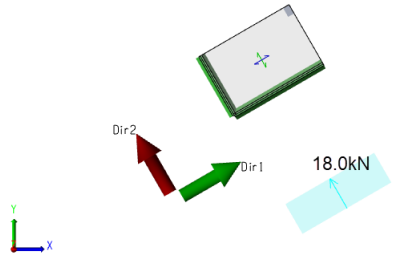
Consider the following structure consisting of a slab supported on two connected walls and a column. The building direction has been defined at an angle of 30 degrees to the global axes.



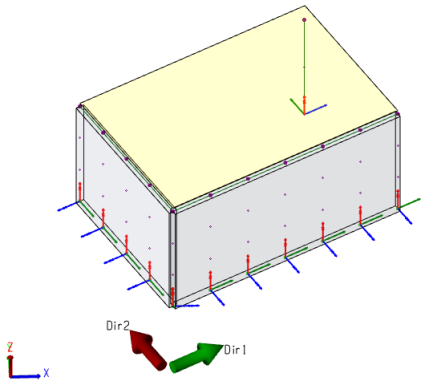
It can be seen that:

- The Building Directions (Dir 1 and Dir 2) are not aligned to the Global Axis System.
- Wall W1's major axis is aligned with Building Direction Dir 1
- Wall W2's major axis is aligned with Building Direction Dir 2

We will investigate the reactions resulting from a wind load of 18kN is applied to the front face of the building so that it acts in Building Dir 2.

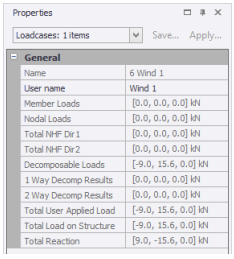


When **Support > DOF** is switched on in **Scene Content** it can be seen that they are aligned as follows:

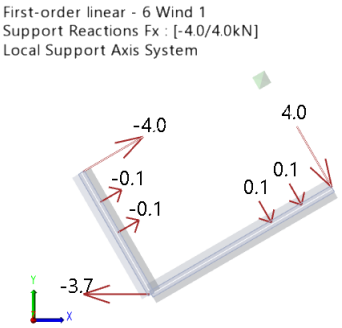


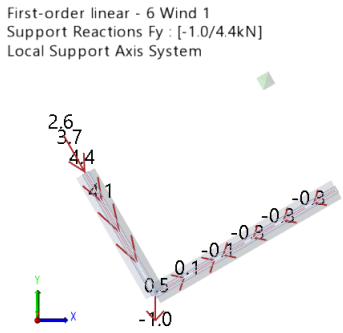
- Shared support at the point where the walls connect:
  - Local x is aligned to global X
  - Local y is aligned to global Y
- Remaining supports:
  - Local x is aligned to each wall's major axis
  - Local y is aligned to each wall's minor axis

After analysis the total global X,Y, and Z reactions can be obtained from the Project Workspace Loading Window.



- In the **Local Support** axis system the **Fx** and **Fy** reactions would be displayed in the Results View as follows:





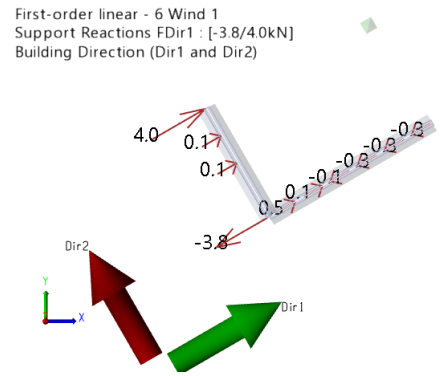
At the shared support the local **Fx** and **Fy** reactions align with the global axes, whereas all the other supports they align to the wall major and minor axes.

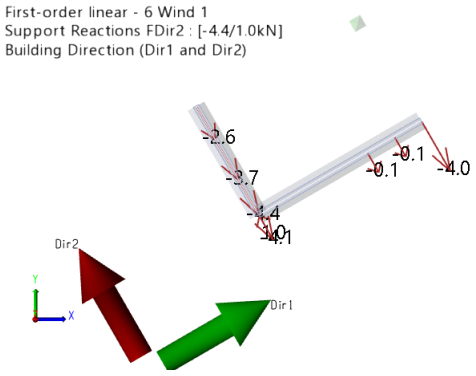
When the same results are displayed in a tabular data view, the reaction at the shared support (Reference 28 in the table below) is split equally and then converted to the local axis system applicable to each wall.

Foundation Reactions, Local Support Axis System

Support ▲	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
SUP B/3	1.173	9.568	-3.000	C8	400x600	30.0000	0.0	0.0	0.0	0.0	0.0	-10.8
W1	-4.000	6.928	-3.000	12	4000×200	-150.0000	-4.0	2.6	8.1	0.0	0.0	-0.1
W1	-3.500	6.062	-3.000	18	4000×200	-150.0000	-0.1	3.7	7.1	0.0	0.0	-0.2
W1	-3.000	5.196	-3.000	20	4000×200	-150.0000	-0.1	4.4	1.6	0.0	0.0	-0.2
W1	-2.500	4.330	-3.000	25	4000×200	-150.0000	-0.0	4.1	-2.8	0.0	0.0	-0.2
W1	-2.000	3.464	-3.000	28	4000×200	-150.0000	1.9	-0.5	-4.3	-0.1	-0.0	-0.1
W2	-2.000	3.464	-3.000	28	6000×200	-60.0000	-0.5	-1.9	-4.3	-0.0	0.1	-0.1
W2	-1.134	3.964	-3.000	30	6000×200	-60.0000	0.0	0.5	-4.4	0.0	0.0	-0.2
W2	-0.268	4.464	-3.000	33	6000×200	-60.0000	0.0	0.1	-2.0	0.0	0.0	-0.2
W2	0.598	4.964	-3.000	36	6000×200	-60.0000	0.0	-0.1	-0.9	0.0	0.0	-0.2
W2	1.464	5.464	-3.000	39	6000×200	-60.0000	0.1	-0.3	0.0	0.0	0.0	-0.2
W2	2.330	5.964	-3.000	42	6000×200	-60.0000	0.1	-0.3	1.0	0.0	0.0	-0.2
W2	3.196	6.464	-3.000	45	6000×200	-60.0000	4.0	-0.3	1.1	0.0	0.0	-0.1

- In the **Building Direction** axis system the **FDir1** and **FDir2** reactions would be displayed in the Results View as follows:





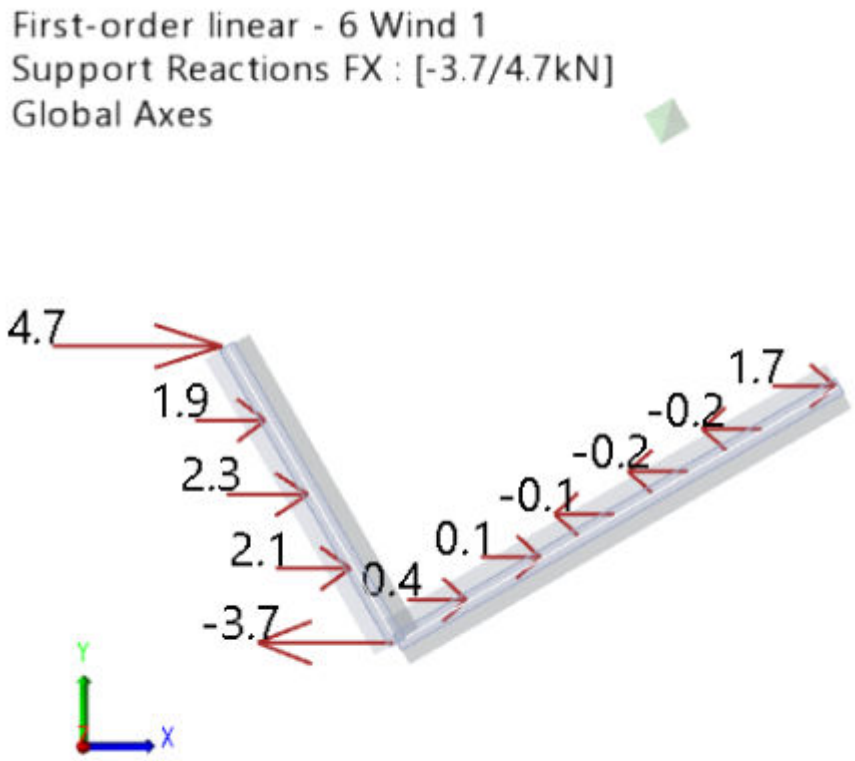
When the same results are displayed in a tabular data view, the reactions at the shared support are output in a single row. (All rows are output in the Building Direction axis system).

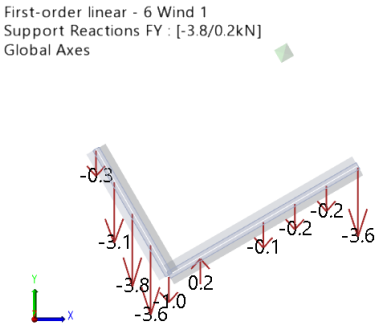
Foundation Reactions, Building Direction (Dir1 and Dir2)

Support	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_{Dir1} [kN]	F_{Dir2} [kN]	F_z [kN]	M_{Dir1} [kNm]	M_{Dir2} [kNm]	M_z [kNm]
SUP B/3	1.173	9.568	-3.000	C8	400x600	30.0000	0.0	0.0	0.0	0.0	0.0	-10.8
W1	-4.000	6.928	-3.000	12	4000×200	30.0000	4.0	-2.6	8.1	0.0	0.0	-0.1
W1	-3.500	6.062	-3.000	18	4000×200	30.0000	0.1	-3.7	7.1	0.0	0.0	-0.2
W1	-3.000	5.196	-3.000	20	4000×200	30.0000	0.1	-4.4	1.6	0.0	0.0	-0.2
W1	-2.500	4.330	-3.000	25	4000×200	30.0000	0.0	-4.1	-2.8	0.0	0.0	-0.2
W2	-1.134	3.964	-3.000	30	6000×200	30.0000	0.5	-0.0	-4.4	0.0	0.0	-0.2
W2	-0.268	4.464	-3.000	33	6000×200	30.0000	0.1	-0.0	-2.0	0.0	0.0	-0.2
W2	0.598	4.964	-3.000	36	6000×200	30.0000	-0.1	-0.0	-0.9	0.0	0.0	-0.2
W2	1.464	5.464	-3.000	39	6000×200	30.0000	-0.3	-0.1	0.0	0.0	0.0	-0.2
W2	2.330	5.964	-3.000	42	6000×200	30.0000	-0.3	-0.1	1.0	0.0	0.0	-0.2
W2	3.196	6.464	-3.000	45	6000×200	30.0000	-0.3	-4.0	1.1	0.0	0.0	-0.1
W2 W1	-2.000	3.464	-3.000	28	6000×200 4000×200	30.0000	-3.8	1.0	-8.6	0.1	0.1	-0.2

In the tabular data view, the sum of the reactions (FDir1 = 0kN, FDir2 = 18kN) equate to the applied load.

- In the **Angle to Global** axis system (with angle set to 0 degrees) the **FX** and **FY** reactions would be displayed in the Results View as follows:



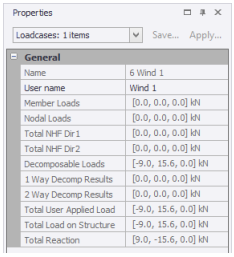


When the results are displayed in a tabular data view, the reactions at the shared support are output in a single row. (All rows are output in the Global Axes system).

Foundation Reactions, Global Axes

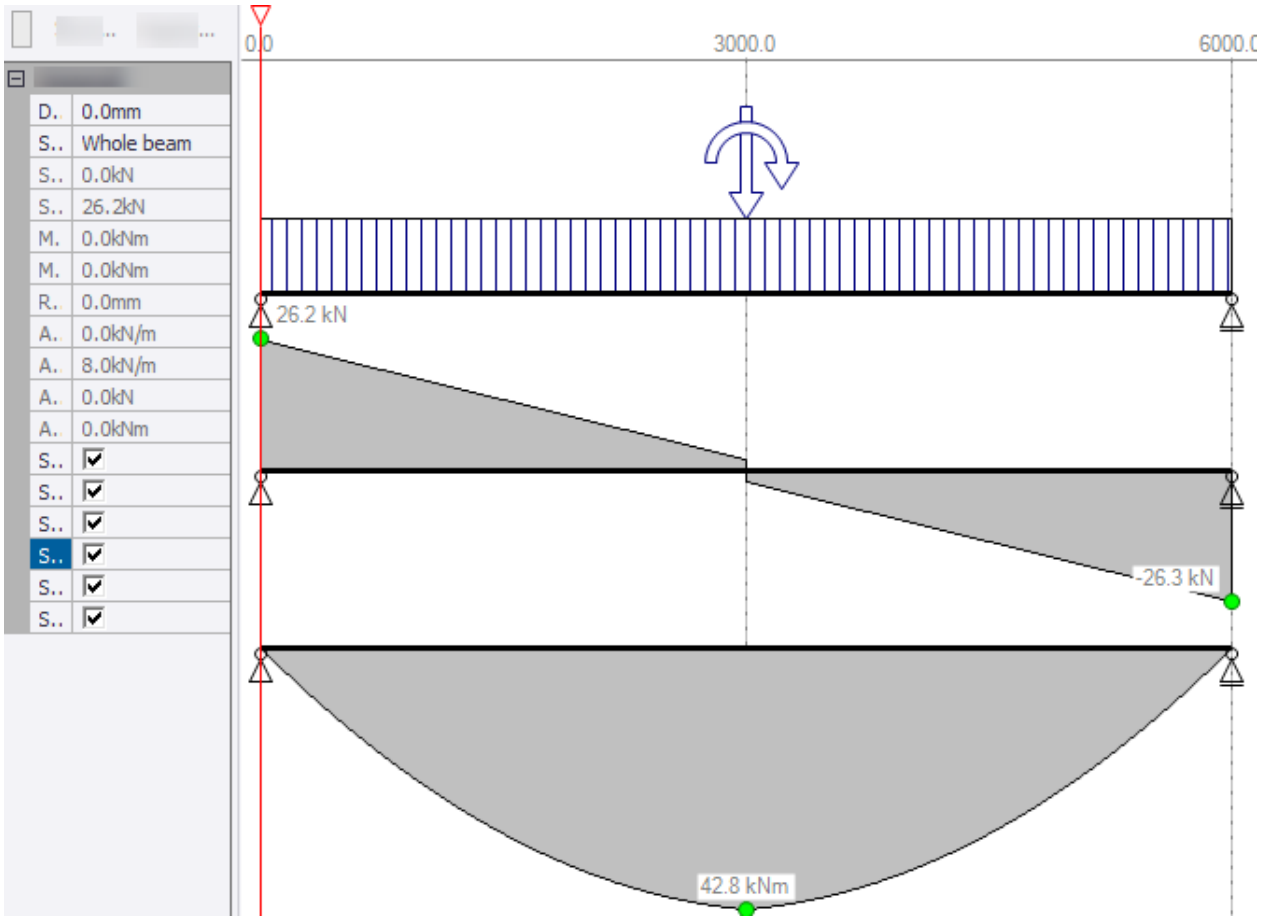
Support	X [m]	Y [m]	Z [m]	Reference	Size	Result rotation [°]	F_X [kN]	F_Y [kN]	F_Z [kN]	M_X [kNm]	M_Y [kNm]	M_Z [kNm]
SUP B/3	1.173	9.568	-3.000	C8	400x600	0.0000	0.0	0.0	0.0	0.0	0.0	-10.8
W1	-4.000	6.928	-3.000	12	4000×200	0.0000	4.7	-0.3	8.1	0.0	0.0	-0.1
W1	-3.500	6.062	-3.000	18	4000×200	0.0000	1.9	-3.1	7.1	0.0	0.0	-0.2
W1	-3.000	5.196	-3.000	20	4000×200	0.0000	2.3	-3.8	1.6	0.0	0.0	-0.2
W1	-2.500	4.330	-3.000	25	4000×200	0.0000	2.1	-3.6	-2.8	0.0	0.0	-0.2
W2	-1.134	3.964	-3.000	30	6000×200	0.0000	0.4	0.2	-4.4	0.0	0.0	-0.2
W2	-0.268	4.464	-3.000	33	6000×200	0.0000	0.1	0.0	-2.0	0.0	0.0	-0.2
W2	0.598	4.964	-3.000	36	6000×200	0.0000	-0.1	-0.1	-0.9	0.0	0.0	-0.2
W2	1.464	5.464	-3.000	39	6000×200	0.0000	-0.2	-0.2	0.0	0.0	0.0	-0.2
W2	2.330	5.964	-3.000	42	6000×200	0.0000	-0.2	-0.2	1.0	0.0	0.0	-0.2
W2	3.196	6.464	-3.000	45	6000×200	0.0000	1.7	-3.6	1.1	0.0	0.0	-0.1
W2 W1	-2.000	3.464	-3.000	28	6000×200 4000×200	0.0000	-3.7	-1.0	-8.6	0.1	0.1	-0.2

The sum of the reactions (FX = 9kN, FY = 16.6kN, FZ = 0kN) equate to the global reaction values in the Project Workspace Loading Window.



Display member analysis results in a Load Analysis View

Analysis results for individual members and walls can be displayed in a **Load Analysis View**.



Open a Load Analysis View

1. Right-click the member whose loading and analysis results you want to view.
2. In the context menu, select **Open Load Analysis View**.

   Tekla Structural Designer opens the load analysis view and a **Loading Analysis** tab is added to the ribbon.
3. In the **Loading** list, select the loadcase, combination or envelope whose results you want to view.
4. On the ribbon, select the analysis type using the list in the **Result Type** group.
5. If you are viewing the results for a load combination or envelope, in the **Result Type** group on the ribbon, select whether you want to view the results based on strength or service factors.
6. In the **Direction** group, select the axis type (**Axial**, **Major**, or **Minor**).

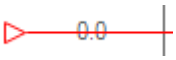
Tekla Structural Designer displays the results according to the selected options.

You can then further control the results in the **Properties** window as follows:

- If the member is continuous:
  - If viewing results for a column, set the stack property to display either results for the whole column, or an individual stack.
  - If viewing results for a beam, set the span property to display either results for the whole beam, or an individual span.
- If the loadcase, combination or envelope has live/imposed load reductions applied:
  - If viewing **Axial** results for a column, click **Show axial force reduced** to display the results with live/imposed load reductions applied.If viewing **Major** or **Minor** results for a beam, click **Show reduced** to display the results with live/imposed load reductions applied.

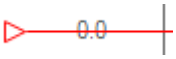
Load Analysis View properties for columns

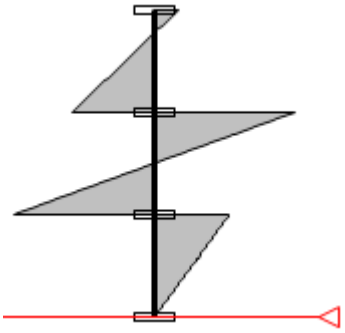
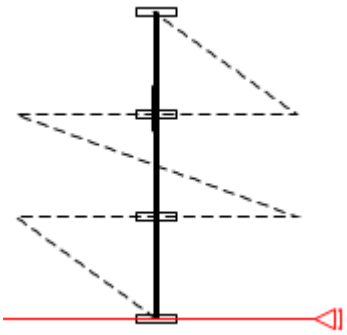
Result Type: Axial

Property	Description
Distance	The results can be reported at any position along the column, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider  to the position required. **NOTE** For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.
Axial force above	The axial force in the column immediately above the cross section at the distance specified.
Axial force below	The axial force in the column immediately below the cross section at the distance specified.

Property	Description
Axial force reduced above	The axial force in the column immediately above the cross section at the distance specified, taking into account live/imposed load reductions.
Axial force reduced below	The axial force in the column immediately below the cross section at the distance specified, taking into account live/imposed load reductions.
Torsion moment above	The torsion in the column immediately above the cross section at the distance specified.
Torsion moment below	The torsion in the column immediately below the cross section at the distance specified.
Show axial force	If cleared, the axial force diagram is removed from the view.
Show axial force reduced	If cleared, the reduced axial force diagram is removed from the view.
Show torsion moment	If cleared, the torsion diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor

Property	Description
Distance	The results can be reported at any position along the column, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider  to the position required. **NOTE** For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.

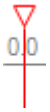
Property	Description
Shear above	The major or minor shear force immediately above the cross section at the distance specified.
Shear below	The major or minor shear force immediately below the cross section at the distance specified.
Moment above	The major or minor moment immediately above the cross section at the distance specified. The Moments diagram is solid and shaded. 
Moment below	The major or minor moment immediately below the cross section at the distance specified.
Ecc. Moment above	The major or minor moment due to eccentricity immediately above the cross section at the distance specified. The eccentricity moments diagram is dashed and unshaded. 
Ecc. Moment below	The major or minor moment due to eccentricity immediately below the

Property	Description
	cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load above	The applied distributed load in the major or minor direction immediately above the cross section at the distance specified.
Applied load below	The applied distributed load in the major or minor direction immediately below the cross section at the distance specified.
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.
Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflections diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Load Analysis View properties for beams

Result Type: Axial

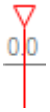
Property	Description
Distance	The results can be reported at any position along the beam, either by typing the distance directly into the **Properties** window, or, on the

Property	Description
	diagram, by dragging this slider shown below to the position required.  **NOTE** For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.
Axial force left	The axial force in the beam immediately to the left of the cross section at the distance specified.
Axial force right	The axial force in the beam immediately to the right of the cross section at the distance specified.
Torsion moment left	The torsion in the beam immediately to the left of the cross section at the distance specified.
Torsion moment right	The torsion in the beam immediately to the right of the cross section at the distance specified.
Relative angle of twist	The relative angle of twist (due to torsion) in the beam cross section, at the distance specified.
Angle of twist derivative	A droplist allowing the selection of the derivative of angle of twist: • First • Second • Third (This property is displayed for “Open” sections only).
Angle of twist derivative left	The relative angle of twist (due to torsion) in the beam cross section,

Property	Description
	immediately to the left of the distance specified. (This property is displayed for “Open” sections only).
Angle of twist derivative right	The relative angle of twist (due to torsion) in the beam cross section, immediately to the right of the distance specified. (This property is displayed for “Open” sections only).
Show axial force	If cleared, the axial force diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show torsion moment	If cleared, the torsion diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show relative angle of twist	If cleared, the relative angle of twist diagram is removed from the view.
Show angle of twist derivative	If cleared, the angle of twist derivative diagram is removed from the view. (This property is displayed for “Open” sections only).
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor, (plus Major Principal and Minor Principal for angle sections)

Property	Description
Distance	The results can be reported at any position along the beam, either by typing the distance directly into the **Properties** window, or, on the

Property	Description
	diagram, by dragging this slider shown below to the position required.  **NOTE** For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.
Show reduced	If selected, the forces that are reported take account of any live/ imposed load reductions that might apply.
Shear left	The major or minor shear force immediately to the left of the cross section at the distance specified.
Shear right	The major or minor shear force immediately to the right of the cross section at the distance specified.
Moment left	The major or minor moment immediately to the left of the cross section at the distance specified.
Moment right	The major or minor moment immediately to the right of the cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load left	The applied distributed load in the major or minor direction immediately to the left of the cross section at the distance specified.
Applied load right	The applied distributed load in the major or minor direction immediately to the right of the cross section at the distance specified.

Property	Description
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.
Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflection diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

RSA Seismic Results in a Load Analysis View

Loading Analysis Views for 1st or 2nd Order RSA Seismic result types use the same rules as those applied to multi-member Results Views for the same result types, i.e. as follows:

RSA Seismic Loadcases

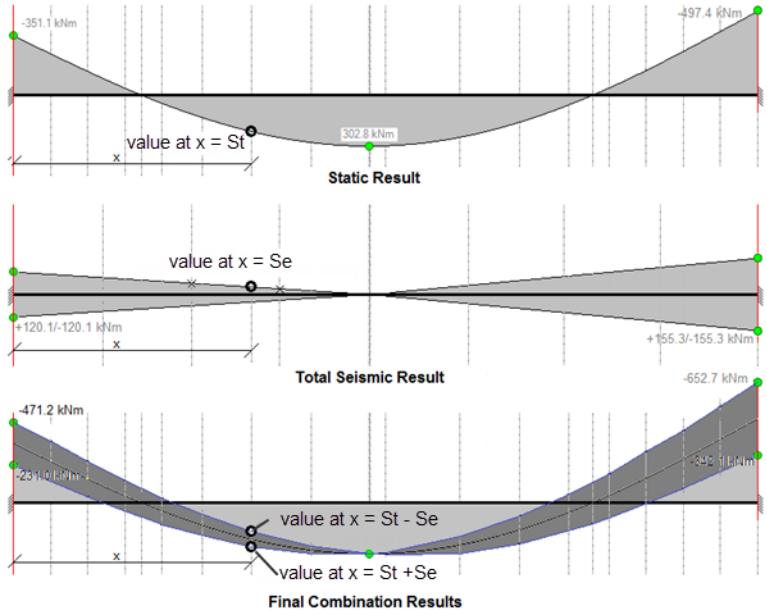
- Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options
    - Absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram, (so that the diagrams are always symmetrical about the base line).
- All relevant modes - a standard enveloped diagram is displayed
- RSA Torsion Loadcases - displayed as per 1st order linear analysis
- Static Loadcases included in the RSA Seismic Combination - displayed as per 1st order linear analysis
- Effective Seismic Weight Combination - not available

RSA Seismic Combinations

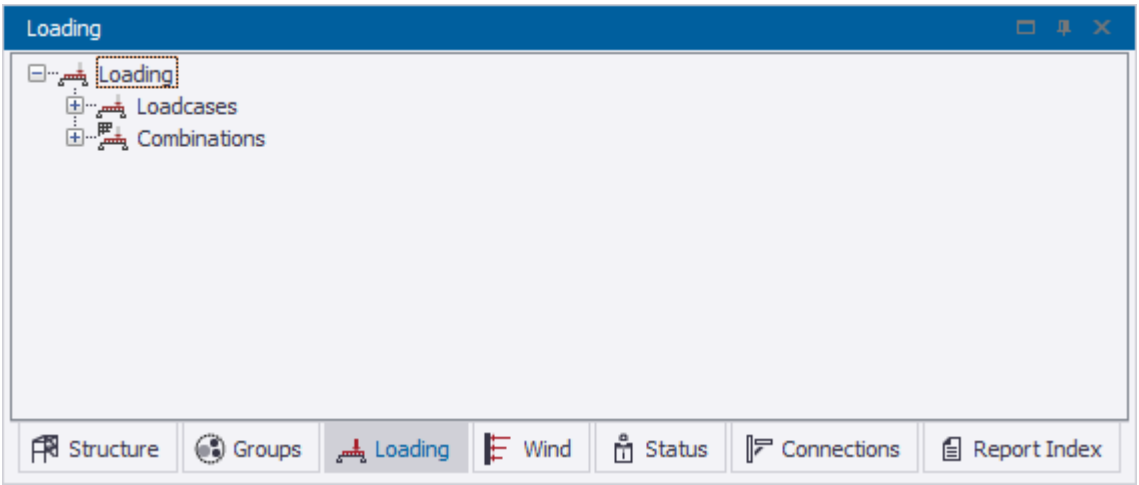
A droplist is provided to allow you to view:

- Design Profile - derived from the Static+Seismic result, the Design Profile is always plotted on the same side of the base line as the Static Only result
- Static Only - displayed as per 1st order linear analysis

- Seismic Only - absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram
- Static+Seismic - an envelope is drawn showing the seismic results above and below the static result.
  - Base line is through the static values
  - Top line is static value + seismic value
  - Bottom line is static value - seismic value



Display loading summaries in the Project Workspace



The **Loading** tree in the **Project Workspace** can be used to view:

- Total loads on the structure and total reactions for each loadcase and combination analysed.
- Loadcase or combination status, (a pass status indicates the above totals equate to each other).
- Notional forces and any minimum lateral loads
- Combination load duration factors used in timber design.

View the load status of a loadcase or combination

1. In the **Project Workspace**, select the **Loading** tab.
2. In the **Loading** tree, select the required loadcase or combination. The status of the selected loadcase or combination is displayed in the **Properties** window as follows:
    - ✓ total reaction equates to the total load on structure
    - ✖ total reaction does not equate to the total load on structure
    - ? total reaction is not available

How load status is determined

Under **General** there are summations (in global X, Y & Z) of:

- **Member Loads**
- **Nodal Loads**
- **Total NHF Dir 1**
- **Total NHF Dir 2**
- **Decomposable Loads** (i.e. loads applied to panels)
- **1 Way Decomposition Results**
- **2 Way Decomposition Results**
- **Total User Applied Load** (this can be greater than the total Member, Nodal and Decomposable loads if Diaphragm Loads have been applied).
- **Total Load on Structure**
- **Total Reaction** - from the 3D analysis

Under **FE Chasedown** and **Grillage Chasedown**(if they have been performed) there are summations of:

- **Total Reaction**
- For each **sub-model**:
    - **Load from above**

- **Load Applied**
- **Reaction**

The following cross checks are performed on the above data:

- **Total User Applied Load** should equate to the **Total Load on Structure**.
- **Total Load on Structure** should equate to the **Total Reaction** for 3D analysis reported in the **General** section.
- **Total Load on Structure** should equate to the **Total Reaction** for FE chasedown analysis reported in the **FE ChaseDown** section.
- For each sub-model in the **FE ChaseDown** section the **Load Applied** when added to the **Load from above** should equate to the **Reaction**
- **Total Load on Structure** should equate to the **Total Reaction** for Grillage chasedown analysis reported in the **Grillage ChaseDown** section.
- For each sub-model in the **Grillage ChaseDown** section the **Load Applied** when added to the **Load from above** should equate to the **Reaction**

A ✖ status is assigned if the **Total User Applied Load**, **Total Load on Structure** and **Total Reaction** do not balance*.

* Within a small tolarance of: Max (50N, 0.25% TotalReaction).

View the magnitude of minimum lateral loads

NOTE Minimum lateral loads are only displayed if they have been activated in Model Settings

1. In the **Project Workspace**, select the **Loading** tab.
2. In the **Loading** tree, select the required **Min Lateral Load** loadcase.
3. The magnitude of the load is listed under **Nodal Loads** in the **Properties** window.

View the magnitude of notional forces

NOTE The notional forces are calculated according to the Global Imperfections settings in Model Settings.

1. In the **Project Workspace**, select the **Loading** tab.
2. In the **Loading** tree, select the required loadcase or combination.
3. The magnitude of the notional forces are listed under **Total NHF Dir 1** and **Total NHF Dir 2** in the **Properties** window.

4.5 Analysis settings

Click **Analyze** --> **Settings** to edit the following analysis settings specific to the current model.

After they have been edited, click Save, Load, Cancel, or OK (page 1263) according to your requirements.

Click **Home** --> **Settings** --> **Analysis Settings** to edit the Global settings (page 1232).

1st order non-linear settings

Item	Description
Maximum number of iterations	The number of iterations to perform. Default = 100.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.0001.
Relative	Default: ON.
Relaxation Factors	Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either zero or tension force. The process is entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances. • **Use relaxation factors for tension only elements**: Default = cleared • **Minimum relaxation factor**: Default = 0.1

Item	Description
	• **Maximum relaxation factor**: Default = 0.5 If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. **NOTE** Relaxation factors are not activated by default, as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

2nd order non-linear settings

Item	Description
Maximum number of iterations	The number of iterations to perform. Default = 100.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.0001.
Relative	Default: ON.
Relaxation Factors	Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either

Item	Description
	zero or tension force. The process is entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances. • **Use relaxation factors for tension only elements**: Default = cleared • **Minimum relaxation factor**: Default = 0.1 • **Maximum relaxation factor**: Default = 0.5 If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. **NOTE** Relaxation factors are not activated by default, as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

1st order modal settings

Item	Description
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Automatic** (default option): Initially uses the **Subspace** option to find the lowest modes. If the criteria (either mass or number of modes) is not fulfilled, **FEAST** is then automatically used to find

Item	Description
	higher modes until the stopping criteria is fulfilled. • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **FEAST**: uses the FEAST algorithm to effectively calculate all the eigenvalues within a specific range. Suitable for any size structure. For more information, see: http://www.ecs.umass.edu/~polizzi/feast/.
Mass Model	Allows you to select how the mass model of the analysis is displayed. The options are: • **Consistent** • **Lumped**
Mass reporting	For users that wish to determine a center of mass for each floor, the **Simple** mass option is available to simplify the calculation. A single value of mass for each node is listed in the Tabular Data for both the "Active Masses by Node" & "Total Masses by Node" view types. The simple mass is calculated by averaging the mass values in the active directions. **NOTE** It is recommended that the simple mass option is used only when the lumped mass model is used.

Item	Description
Stopping Criteria	Stopping criteria prevent analysis continuing forever. If either of the criteria (**Maximum number of modes** or **Stopping Frequency**) are met, the analysis will not look for any more modes.
Modes	Allows you to select one of the following options: • **Automatic number of modes**: If the option is selected, you must specify the mass participation required in each direction. You can optionally specify an initial number of modes that should be close to the actual number required in order to speed up the analysis process. • **Total number of modes**: If **Automatic number of modes** is cleared, you can specify the total number of modes required. The default value is 10.
Jacobi *(Only displayed for the Jacobi Extraction method)*	Allows you to adjust the following settings: • **Maximum number of sweeps**: A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed. • **Sweep tolerance**: At the end of each sweep, values are checked against the previous sweep results. If the difference is less than the tolerance specified here, the result is converged, and the analysis is complete.
Subspace	Allows you to adjust the following settings: • **Maximum number of iterations**: The number of iterations to perform. • **Tolerance**: At the end of each iteration, values are checked

Item	Description
	against the previous iteration results. If the difference is less than the tolerance specified here, the result is converged, and the analysis is complete.
FEAST	Allows you to adjust the following settings: • **Initial search range**: Specifies the initial range of values FEAST will search for eigenvalues in. • **Overestimation multiplier**: Specifies the initial guess for the subspace dimension within each range; an overestimate of the predicted number of modes in the range. • **Maximum modes in range**: The maximum number of modes in the range. If more modes are found in a range, the range is split into several smaller ranges. • **Minimum search range**: When a range is smaller than the value specified here, it will no longer be split, even if the maximum number of modes is greater than that allowed.

2nd order buckling settings

Item	Description
Maximum number of iterations	The number of iterations to perform. Default = 1000.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00001.
Maximum number of sweeps	A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the

Item	Description
	maximum number of sweeps allowed. Default = 50.
Sweep tolerance	At the end of each sweep, values are checked against the previous sweeps results. If the difference is less than the tolerance determined here the result is converged, and the analysis is complete. Default = 1.0E-12.
Total number of modes	Allows you to specify the total number of modes required. Default = 10.
Show negative buckling factors	Default = OFF.
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **Automatic** (default option): Tekla Structural Designer determines the most appropriate extraction method for the structure.

1st order seismic settings

Item	Description
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Automatic** (default option): Initially uses the **Subspace** option to find the lowest modes. If the criteria (either mass, or number of

Item	Description
	modes) is fulfilled, **FEAST** is then automatically used to find higher modes until the stopping criteria is fulfilled. • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **FEAST**: uses the FEAST algorithm to effectively calculate all the eigenvalues within a specific range. Suitable for any size structure. For more information, see: http://www.ecs.umass.edu/~polizzi/feast/.
Mass Model	Allows you to select how the mass model of the analysis is displayed. The options are: • **Consistent** • **Lumped**
Mass reporting	For users that wish to determine a center of mass for each floor, the **Simple** mass option is available to simplify the calculation. A single value of mass for each node is listed in the Tabular Data for both the "Active Masses by Node" & "Total Masses by Node" view types. The simple mass is calculated by averaging the mass values in the active directions. **NOTE** It is recommended that the simple mass option is used

Item	Description
	only when the lumped mass model is used.
Stopping Criteria	Stopping criteria prevent analysis continuing forever. If either of the criteria (**Maximum number of modes** or **Stopping Frequency**) are met, the analysis will not look for any more modes.
Modes	Allows you to modify the following options: • **Initial number of modes**: In order to speed up the analysis process, you can specify an initial number of modes you expect to be required to achieve the required participation. The value specified here should be close to the actual number required because if you enter too few or too many modes, the analysis may take longer. • **Mass participation for RSA**: Allows you to specify the mass participation required in each direction. If this isn't achieved before the stopping criteria apply, the RSA analysis will still be performed, but a warning will be issued. • **Min. Mass participation for RSA**: If the minimum participation isn't achieved before the stopping criteria apply, the RSA analysis is not performed. **NOTE** The options in **Stopping Criteria** overrule both the number of modes and mass percentage.
Jacobi	Allows you to adjust the following settings: • **Maximum number of sweeps**: A sweep is a transformation of every

Item	Description
	off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed. • **Sweep tolerance**: At the end of each sweep, values are checked against the previous sweeps results. If the difference is less than the tolerance determined here the result is converged, and the analysis is complete.
Subspace	Allows you to adjust the following settings: • **Maximum number of iterations**: The number of iterations to perform. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete.
FEAST	Allows you to adjust the following settings: • **Initial search range**: Specifies the initial range of values FEAST will search for eigenvalues in. • **Overestimation multiplier**: Specifies the initial guess for the subspace dimension within each range; an overestimate of the predicted number of modes in the range. • **Maximum modes in range**: The maximum number of modes in the range. If more modes are found in a range, the range is split into several smaller ranges. • **Minimum search range**: When a range is smaller than the value specified here, it will no longer be split, even if the maximum

Item	Description
	number of modes is greater than that allowed.
Modal Combination Method	To determine the representative maximum response of interest for a loadcase, the relevant values for each relevant mode are combined by using the modal combination method specified here. Note that once modes have been combined, the relative signs are lost. The options are: • **Complete Quadratic Combination (CQC)**: Suitable for models where modes are closely spaced or well spaced. • **Square Root of Summation of Squares (SRSS)**: Suitable only for models where modes are well spaced.

Iterative cracked section analysis settings

Item	Description
Global Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 200. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00100. • **Relative**: Default: ON. **TIP** To resolve global convergence failures: • Increase the value in **Maximum number of iterations**.

Item	Description
	• Decrease the value in **Tolerance**.
Local Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 500. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.000001. • **Smoothing Parameter**: A property only available for the Eurocode regional code. Default = 0.005. **TIP** To resolve local convergence failures: • Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**. • If you are using the Eurocode regional code, increase the value in **Smoothing Parameter**.

Modification factors

Different factors can be applied for each of the different materials in the model in order to adjust the following properties:

- E: Young's modulus
- G: Shear modulus
- $I_{torsion}$: Section inertia about local X

- I_{major}: Section inertia about local Y
- I_{minor} - Section inertia about local Z
- Area: Section area in compression or tension
- $A_{parallel\ to\ minor}$: Section shear area in local Y
- $A_{parallel\ to\ major}$ - Section shear area in local Z
- t: shell thickness (applicable to concrete only)

The factors also vary according to the member types, (and in the case of concrete members whether they are cracked or not).

For concrete members in particular, design codes can require that analysis stiffness adjustment factors are applied since the appropriate properties to use in analysis are load and time dependent.

Although default modification factors for each material are provided in the settings sets to reflect the design code being worked to, you should check that these are appropriate for your particular analysis model.

For more details, see .

NOTE If you make changes to any of the modification factors, analysis must be repeated.

Meshing settings

The meshing options can be adjusted in order to fine-tune the meshes that are produced in slabs and walls. The default settings are generally appropriate, but they could potentially be adjusted if 2D element quality errors are being created during validation.

For examples of the types of issue that can be resolved by adjusting the meshing tolerance controls, see: Slab meshing controls (page 592)

Composite steel beams settings

Item	Description
Inertia used for loading analysis list	Allows you to specify the inertia to be used in the global analysis of the model for a composite steel beam: • Steel beam (default option) • Long term composite

Item	Description
	• Short term composite **NOTE** • If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. • Relaxation factors are not activated by default as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree. The inertia has different effects on different regional codes: • US regional code: Long and short term composite inertia only apply to rolled symmetric I sections without web openings. For all other sections, the standard steel beam inertia is used regardless of the analysis option selected. • EC regional code: Long and short term composite inertia only apply to: • Symmetric I rolled sections without web openings • I plated sections without web openings

Item	Description
	• Westok plated sections without web openings • Fabsec sections without web openings For all other sections, the standard steel beam inertia is used regardless of the analysis option selected. • BS regional code: Long and short term composite inertia only apply to: • Symmetric I rolled sections without web openings • Asymmetric I rolled sections without web openings • I plated sections without web openings • Westok plated sections without web openings • Fabsec sections without web openings For all other sections, the standard steel beam inertia is used regardless of the analysis option selected. • AUS and IS regional codes: Design of composite beams is not currently supported for the regional codes.

5 Design models

You can design most of your model's members, walls, slabs and foundations directly in Tekla Structural Designer, to get started, see:

- Design steel members and cast-in-place concrete beams, columns and walls (page 776)
- Design slabs and mats, run punching shear checks (page 800)
- Design isolated foundations (page 818)
- Check floor vibration (page 819)
- Check drift, sway, seismic drift, wind drift, and overall displacements (page 822)

If your model contains timber members, precast members, or masonry walls these can be designed using Tekla Tedds. For further details, see:

- Design timber members, precast members, and masonry walls using Tekla Tedds (page 849)

If your model contains steel connections, some connection types can be checked directly in Tekla Structural Designer whereas others can be designed or checked using Tekla Connection Designer or third party applications. For further details, see:

-

See also

Design settings (page 866)

5.1 Design steel members and cast-in-place concrete beams, columns and walls

Before commencing the design you should ensure that members have their Autodesign (page 778) property set appropriately. You should then also take some time to ensure Design settings (page 866) are set as required.

If working to the US regional code you should be aware of additional considerations in relation to The Direct Analysis Method (page 655).

Having set the properties and settings as required you can then proceed to rapidly assess all steel members and/or concrete members/walls in your model from the **Design** tab by running a combined analysis and member design (page 779). The necessary analysis is automatically performed as part of the batch design process.

Alternatively, you may prefer to concentrate on the design of specific members only. To do this you would first run an analysis (e.g. Analyze All (Static) (page 497) or 3D only (Static) (page 498)) to generate an appropriate set of design forces, after which you could then selectively check (page 790), or selectively design (page 792) just those members you are interested in.

If working to either the Eurocode, US, or India regional code, you may also want to introduce more/less conservatism to the design process by specifying user defined utilization ratios (page 797).

Additional requirements may be necessary if designing members and walls for the results of a seismic analysis. For further details, see in the Seismic analysis and design handbook

See also

The Direct Analysis Method (page 655)

Apply and modify design settings

The design options in Tekla Structural Designer allow you to adjust the way in which your model is designed.

The different Design settings (page 866) subpages enable you to, for example:

- set which analysis type is run for the design, (on the Analysis subpage)
- set reinforcement and other parameters for concrete design, (on the Concrete subpage)
- set the levels at which forces can be ignored, (on the Design Forces subpage)
- set which members to design using groups, (on the Design Groups subpage)

- control how Autodesign settings will be reset after the design, (on the Auto design subpage)

Modify design settings in the current project

1. On the **Design** tab, click  **Settings**.
2. Modify the Design settings (page 866) according to your needs.
3. After making the changes, do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes as defaults for future projects, click **Save...**

   TIP To revert to the default design settings of the active settings set, click **Load...**

Modify design settings defaults for future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Go to the Design settings (page 866) page.
4. Select the settings set that you want to modify.
5. Modify the settings according to your needs.
6. To save the changes to the settings set, click **OK**.

See also

Design settings (page 866)

Autodesign versus check design

Before you begin the design process you should select a suitable design mode for each member in your model by selecting an appropriate Autodesign setting.

Autodesign can be set in the member properties and operates as follows:

- **Autodesign on:**

  For steel members, you select the desired section type. Tekla Structural Designer then determines a suitable size for the selected section type.

  For concrete members, you assign the desired section size.Tekla Structural Designer then automatically determines a suitable reinforcement configuration.

- **Autodesign off:**

  You assign the desired section size (for steel members), or section size and reinforcement (for concrete members). Then, Tekla Structural Designer determines if the section or reinforcement is sufficient.

You can control if the Autodesign setting gets reset to off at the end of the design process via (page 927).

TIP To quickly review and update the Autodesign setting applied to all members in the model, open a review view and use the command (page 961) to switch between the two modes.

NOTE The Autodesign setting is only considered when you choose to design multiple members, as explained below:

- If you choose to run a **combined analysis and design** (Design Steel, Design Concrete, Design All); or choose a **selective design** of multiple members (Design model, Design plane, Design Selection etc.) members will either be designed or checked according to their individual Autodesign setting.
- If you choose a **check** (Check model, Check plane, Check Selection, Check Member etc.) each member will be checked irrespective of its Autodesign setting.
- If you **design** an individual member (Design Member, Design Wall etc.) the member will be designed irrespective of its Autodesign setting.

See also

Overview of combined analysis and member design (page 779)

Overview of combined analysis and member design

After creating and loading a model, you can proceed directly to a static design to rapidly design every steel and/or concrete member and wall for active gravity, static and ELF seismic load combinations.

For larger steel models you might potentially find it more efficient to adopt a two step process consisting of a gravity design followed by a static design.

If you have set up RSA load combinations via the Seismic Wizard these are catered for by running an RSA design.

Related task

Run combined analysis and member design (page 786)

Gravity Design

Gravity Design uses the results from a 3D analysis to design* every steel and concrete member and wall in the model for **all active gravity combinations**. (For steel and composite members, this includes both “gravity only” and “gravity and lateral” members).

The 3D analysis can be either first or second order, depending on the settings specified on the page (page 866) in (page 866).

NOTE If working to the US regional code and applying the The Direct Analysis Method (page 655) refer to the suggested workflow (page 660) for guidance on setting the analysis type appropriately.

Although your final design should consider all combinations, designing for gravity combinations only can be a useful way to rapidly pre-size the members in the model that are not subjected to lateral loads.

Because you may not have created lateral systems at this stage, the column nodes that are not in a rigid floor diaphragm are, by default, fixed horizontally. To change the default setting for this go to the page (page 874) in (page 866).

Once members have been sized acceptably for gravity combinations, a Static Design can then be run to design the remaining combinations.

* Each member is either designed or checked according to its individual Autodesign setting (page 778).

Gravity Design Process

The processes performed when a Gravity design is run can be summarized as follows:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis and design process begins.

Process	Description
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads • Preparing loadcases and combinations • Meshing and diaphragms • Applying loads • Resolving vertical loads for application of global imperfections The actual pre-analysis processes performed will vary depending on the specific model that has been defined.
3D analysis	A traditional frame analysis of the entire 3D model, with an option to mesh floors. This analysis generates a first set of results that can be used for the design of beams, columns, walls and slabs. You can control whether a first, or a second order 3D Analysis is run by making the appropriate selection on the page (page 866) in (page 866). The actual options that are presented will vary depending on the design code being worked to. The choice of linear or non-linear analysis is made automatically: • if the model has entirely linear properties a linear analysis is performed, • else if any non-linear properties are detected a non-linear analysis is performed.
Member Design	Members are designed in accordance with the selected gravity design command: • **Design Steel (Gravity)** - designs or checks “gravity only” and “gravity and lateral” steel and composite members • **Design Concrete (Gravity)** - designs or checks concrete beams, columns and walls • **Design All (Gravity)** - designs or checks “gravity only” and “gravity and lateral” steel and composite members, plus concrete beams, columns and walls
Stability checks	Sway (page 828)/Drift (page 823) checks and Wind Drift (page 836) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

Static Design

Static Design uses the results from a 3D analysis to design* every steel and concrete member and wall in the model for **all active gravity, lateral and ELF seismic combinations**. The 3D analysis can be either first or second order, depending on the settings specified on the page (page 866) in (page 866).

NOTE If working to the US regional code and applying the The Direct Analysis Method (page 655) refer to the suggested workflow (page 660) for guidance on setting the analysis type appropriately.

If the model contains concrete members the results from Grillage and FE chasedown analyses are also used to establish a second and third set of design forces for those members. FE chasedown results are only generated if the model contains FE meshed slabs.

You can use (page 866) to turn off 'Design for FE Chasedown analysis results' for a specific concrete member type if required. This option is located under (page 878), then **Beam, Column, or Wall > General Parameters** as appropriate.

Members are designed for the appropriate design requirements as follows:

- “gravity only” members for all active gravity combinations, (For more information, see: Design individual members for gravity only (page 796))
- other members and walls for all active gravity, lateral and seismic combinations,
- any members or walls defined as being in Seismic Load Resisting Frames, that are subject to seismic combinations (as opposed to seismic RSA combinations) and that meet the relevant criteria for the appropriate seismic provisions.

* Each member is either designed or checked according to its individual Autodesign setting (page 778).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis. For further details, see in the Seismic analysis and design handbook.

Static Design Process

The main steps in the Static Design process are summarized below:

Process	Description
Model validation	Model validation (page 323) checks are performed to detect specific issues before the analysis and design process begins.
3D pre-analysis processes	In preparation for analysis a number of pre-analysis processes are undertaken as necessary, such as: • Decomposing slab and wall loads • Preparing loadcases and combinations

Process	Description
	• Meshing and diaphragms • Applying loads • Resolving vertical loads for application of global imperfections The actual pre-analysis processes performed will vary depending on the specific model that has been defined.
3D analysis	A traditional frame analysis of the entire 3D model, with an option to mesh floors. This analysis generates a first set of results that can be used for the design of beams, columns, walls and slabs. You can control whether a first, or a second order 3D Analysis is run by making the appropriate selection on the page (page 866) in (page 866). The actual options that are presented will vary depending on the design code being worked to. The choice of linear or non-linear analysis is made automatically: • if the model has entirely linear properties a linear analysis is performed, • else if any non-linear properties are detected a non-linear analysis is performed.
FE chasedown	An FE chasedown analysis is only performed if required, see: FE chasedown (page 487)
Grillage chasedown	A Grillage chasedown analysis is only performed if required, see: Grillage chasedown (page 489)
Member Design	Members are designed in accordance with the selected static design command: • **Design Steel (Static)** - designs or checks “gravity only” and “gravity and lateral” steel and composite members • **Design Concrete (Static)** - designs or checks concrete beams, columns and walls • **Design All (Static)** - designs or checks “gravity only” and “gravity and lateral” steel and composite members, plus concrete beams, columns and walls
Stability checks	Sway (page 828)/Drift (page 823) checks and Wind Drift (page 836) checks are performed for all columns and walls, (apart from any that have been manually excluded from the check

RSA Design

RSA Design is used to design* any members defined as being in Seismic Load Resisting Frames, that are subject to seismic RSA combinations.

RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 436) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

---

* Each member is either designed or checked according to its individual Autodesign setting (page 778).

Choice of Design Steel, Design Concrete, or Design All

The appropriate design command depends on the materials you have used in the model.

In simple terms:

- If your model consists of steel members only, you can run the design by using the **Design Steel** commands.
- If your model consists of concrete members only, you can run the design by using the **Design Concrete** commands.
- If your model consists of a mix of both concrete and steel members, you can run the design by using the **Design All** commands.

TIP For structures that are mostly steel but have a few concrete members: instead of running Design All, you could run Design Steel (in order to focus on the steel design) before switching to Design Concrete for the remaining members.

In this way during the steel design phase you are not running grillage and FE chasedown analyses when they are not required .

For more details on how each command affects the analysis process, see the following:

Design steel

- Performs a 3D analysis.
- Does not perform grillage chasedown or FE chasedown analysis.
- Designs or checks all steel elements.
- Does not design or check concrete beams, columns or shear walls.

Design concrete

- Performs a 3D analysis and a grillage chasedown analysis.
- May also be required to perform an FE chasedown analysis.

- Designs or checks all concrete beams, columns and shear walls.
- Does not design or check steel elements.

Design all

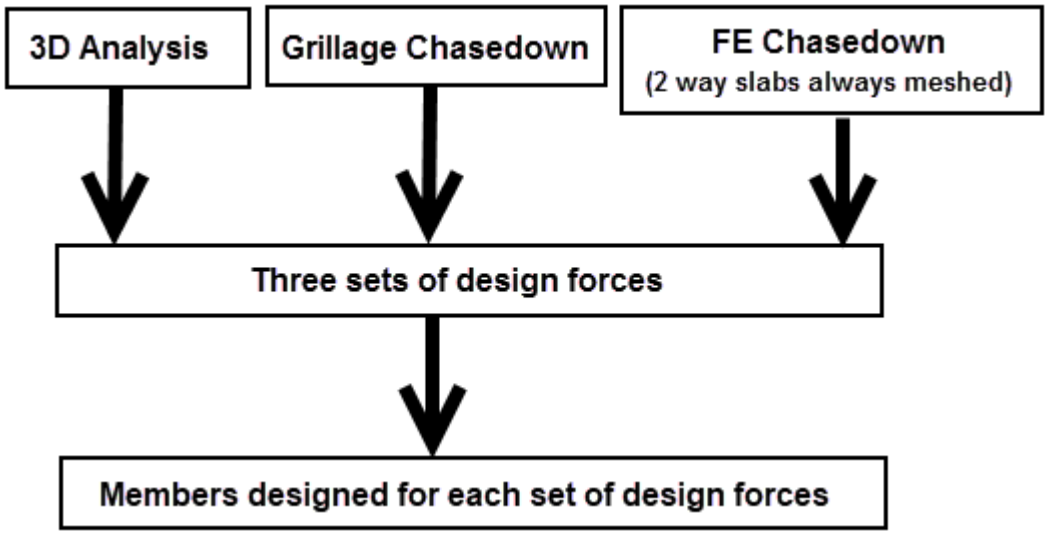
- Performs a 3D analysis and a grillage chasedown analysis.
- May also be required to perform an FE chasedown analysis.
- Designs or checks all concrete beams, columns and shear walls.
- Designs or checks all steel elements.

Design force sets used for steel and concrete member design

Steel Member Design Forces

The 3D analysis results are the only results set used in steel member design.

Concrete Member Design Forces



Up to three sets of analysis results will exist for concrete member design as follows:

- **3D analysis** results will always be used to design all beams, columns and walls.
- **Grillage chasedown** results will exist for gravity loadcases if the model contains any concrete beams, in which case they will also be used to design all beams, columns and walls.
- **FE chasedown** results for gravity loadcases will also exist if the model contains 2-way spanning slabs.

By default these results will also used for concrete beam, column and wall design.

NOTE You can use **Design Settings** to turn off 'Design for FE Chasedown analysis results' for concrete beams, columns or walls if required. This option is located under (page 878), then **Beam/Column/Wall > General Parameters** (as appropriate).

Run combined analysis and member design

Run a combined analysis and design in order to rapidly check/design every concrete member or steel member, or every concrete member and steel member, for active gravity, static, or RSA load combinations.

Run Design Steel (Gravity)

- On the **Design** tab, click  **Design Steel (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all steel beams and columns in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design Steel (Static)

- On the **Design** tab, click  **Design Steel (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all steel beams and columns in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design Steel (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 436) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

- On the **Design** tab, click  **Design Steel (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all steel beams and columns in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design Concrete (Gravity)

- On the **Design** tab, click  **Design Concrete (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all concrete beams, columns and walls in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design Concrete (Static)

- On the **Design** tab, click  **Design Concrete (Static)**.

Tekla Structural Designer analyzes all static combinations and then designs all concrete beams, columns and walls in the model.

After design, the original member design mode assigned to each member can either be retained or updated. The action that is taken is controlled via (page 927).

On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design Concrete (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 436) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

- On the **Design** tab, click  **Design Concrete (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all concrete beams, columns, and walls in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design All (Gravity)

- On the **Design** tab, click  **Design All (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all steel and concrete beams and columns and all concrete walls in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset

auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via (page 927).

On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design All (Static)

- On the **Design** tab, click  **Design All (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all steel and concrete beams and columns and all concrete walls in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Run Design All (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 436) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

- On the **Design** tab, click  **Design All (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all steel and concrete beams and columns and all concrete walls in the model.

  After design, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via (page 927).

  On completion of the design the ribbon switches to the **Review** tab and a Review View opens to allow you to inspect the pass/fail status and utilization ratio of each member.

Check selected members and walls

There are various ways to check specific members only, using the results of the analyses that have already been performed.

Using a selective approach can be a more efficient way to work with large models.

Check an individual member, wall, or core

- Do one of the following:

To	Do this
Check a member/ wall from a 2D or 3D view	1. Hover the mouse pointer over the member. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right click and select **Check Member > Static/RSA**, or **Check Wall > Static/RSA** as required.
Check a member from the Structure Tree	1. In the Members branch locate the member reference. 2. Right click and select **Check Member > Static/RSA** as required.
Check a wall from the Structure Tree	1. In the Walls branch locate the wall reference. 2. Right click and select **Check Wall > Static/RSA** as required.
Check all members in a core from the Structure Tree	1. In the Cores branch locate the core reference. 2. Right click and select **Check member > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check selected members and walls

1. In the active view, select the members to check.
2. Right-click, then in the context menu, select **Check Selection > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members in a level, slope, or frame

- In the **Project Workspace**, click **Structure** tab.
- In the **Structure** tree, do one of the following:

To	Do this
Check all members in a construction level	Right click the required **Level** and select **Check plane > Static/RSA** as required.

Check all members and walls in a frame	Right click the required **Frame** and select **Check plane > Static/RSA** as required.
Check all members in a slope	Right click the required **Slope** and select **Check plane > Static/RSA** as required.

NOTE These commands use the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Members** tree.
3. In the context menu, select **Check members > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all walls

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Walls** tree.
3. In the context menu, select **Check walls > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members and walls

1. In the **Project Workspace**, click **Structure** tab.
2. Right click on the **Structure** branch.
3. In the context menu, select **Check model > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members of a particular section or type

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Members** tree, right-click the section type or size you want to check.
3. In the context menu, select **Check members > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members in a group

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, right-click the member group you want to check.
3. In the context menu, select **Check Group > Static/RSA** as required.

NOTE
- **Check Groups** uses the existing analysis results, (even if the analysis status is 'Out of Date').

  It is not available unless the **Design Groups** option is checked for the member type in **Design Settings**.

Check all members and walls in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Check sub structure > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design selected members and walls

There are various ways to design specific members only, using the results of the analyses that have already been performed.

Using a selective design approach can be a more efficient way to work with large models.

NOTE In all of the selective design methods, each member in the selection will be individually checked or designed according to its individual autodesign setting, (while also taking account of any member design groups that are active). The exception being that if there is only a single member in the selection it will always be designed irrespective of its autodesign setting.

Design an individual member, wall, or core

Design Member designs an individual member (taking into account any member design groups that are active).

Design Wall designs an individual wall.

Both these commands ignore the auto-design setting of the member/wall. (i.e. a design is always performed even if the autodesign property is off.)

- Do one of the following:

To	Do this
Design a member/wall from a 2D or 3D view	1. Hover the mouse pointer over the member/wall. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right click and select **Design Member > Static/RSA**, or **Design Wall > Static/RSA** as required.
Design a member from the Structure Tree	1. In the Members branch locate the member reference. 2. Right click and select **Design Member > Static/RSA** as required.
Design a wall from the Structure Tree	1. In the Walls branch locate the wall reference. 2. Right click and select **Design Wall > Static/RSA** as required.
Design all members in a core from the Structure Tree	1. In the Cores branch locate the core reference. 2. Right click and select **Design member > Static/RSA** as required.

NOTE The **Design Member** and **Design Wall**commands both use the existing analysis results, (even if the analysis status is 'Out of Date').

Design selected members and walls

Design Selection performs a check or design of each entity in the current selection, according to each entity's autodesign setting, (taking into account any member design groups that are active).

1. In the active view, drag a box to make your selection.
2. Right-click, then in the context menu, select **Design Selection > Static/RSA** as required.

   The selected members/walls are designed/checked.

   If any members in the selection are in a design group, other members in the same group are updated and checked as required.

NOTE The **Design Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Interactively design a concrete member

Interactive design is only available for concrete beams columns and walls. A dialog opens showing the existing design of the highlighted member for the currently supplied reinforcement. From here you can interactively modify the reinforcement and instantly see the result.

1. In the active view, hover the mouse pointer over the concrete member you want to interactively design.
2. Right click and select **Interactive Design > Static/RSA** as required.

   The interactive dialog opens from where the member can be designed.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

Design all members in a level, slope, or frame

Design plane performs a check or design of each entity in the plane, according to each entity's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Design all members in a construction level	Right click the required **Level** and select **Design plane > Static/RSA** as required.
Design all members and walls in a frame	Right click the required **Frame** and select **Design plane > Static/RSA** as required.
Design all members in a slope	Right click the required **Slope** and select **Design plane > Static/RSA** as required.

NOTE
- The **Design plane** command uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It requires at least one member in the selection to be in auto-design mode.
- If any grouped members are designed and the grouped member design option is on, all members of the group are considered in the design even if they are not in the selected level, slope, or frame.

Design all members

Design members performs performs a check or design of every member according to each member's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Members** tree.
3. In the context menu, select **Design members > Static/RSA** as required.

NOTE
- **Design members** uses the existing analysis results, (even if the analysis status is 'Out of Date').

- It is not available until at least one member is in auto-design mode.

Design all walls

Design walls performs performs a check or design of every wall according to each wall's autodesign setting.

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Walls** tree.
3. In the context menu, select **Design walls > Static/RSA** as required.

NOTE
- **Design walls** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one wall to be in auto-design mode.

Design all members and walls

Design model performs performs a check or design of every member/wall in the model according to the individual member/wall autodesign settings, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. Right click the **Structure** branch.
3. In the context menu, select **Design model > Static/RSA** as required.

NOTE
- **Design model** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member is in auto-design mode.

Design all members of a particular section or type

Design members performs performs a check or design the selected members according to each member's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Members** tree, right-click the section type or size you want to design.
3. In the context menu, select **Design members > Static/RSA** as required.

NOTE
- **Design members** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member is in auto-design mode.

Design all members in a group

Design Group performs performs a check or design of the selected member group according to the autodesign settings in the group.

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, right-click the member group you want to design.
3. In the context menu, select **Design Group > Static/RSA** as required.

NOTE
- **Design Group** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available unless the **Design Groups** option is checked for the member type in **Design Settings**.

Design all members and walls in a sub structure

Design sub structure performs performs a check or design of all members/ walls in the selected sub structure according to their individual autodesign settings, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to design.
3. In the context menu, select **Design sub structure > Static/RSA** as required.

NOTE
- **Design sub structure** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member in the sub structure is in auto-design mode.

Design individual members for gravity only

By default, when a full static design is run, all members are designed for both gravity and lateral combinations.

You can however tell Tekla Structural Designer to only consider gravity combinations for the design of specific members. This is achieved by checking **Gravity only** in an individual member's properties, i.e.

- **Gravity only** checked - designed for gravity combinations only
- **Gravity only** unchecked - designed for all combinations types: gravity, lateral and seismic

The **Gravity only** setting is available for beams and columns in all materials, although you would typically only ever consider checking it for steel and composite members.

Axial and minor axis checks are not performed on steel beams set to gravity only.

Setting steel columns that do not help resist lateral loads to be designed for gravity loads only, will reduce the overall design time.

Engineering judgment will be required when identifying members as being 'gravity only'.

For example:

- if an inclined braced member connects to a beam, axial force in the brace (from both gravity and lateral loads) puts the beam into bending and therefore the beam should be designed for both gravity and lateral loads.
- potentially, beams in a sloping roof would also need to be designed for both gravity and lateral load

NOTE If a composite beam is identified as being designed for both gravity and lateral combinations, it will only be designed for gravity loads acting through the web. Minor axis bending and axial loads are not considered. A warning is provided if the ignored loads exceed a preset limit.

See also

Overview of combined analysis and member design (page 779)

Apply user defined utilization ratios

By default designs and checks are performed against a utilization ratio (U/R) of 1.0.

To facilitate design optimization Tekla Structural Designer allows you to specify user defined U/Rs on an individual member basis (currently for Eurocode, US, and India regional codes only).

NOTE Utilization ratio = actual performance value / maximum allowable performance value.

Overview of user defined U/R

When applied to the autodesign process user defined U/Rs introduce a measure of:

- Conservatism - if the value is less than 1.0
- Non conservatism - if the value is greater than 1.0

By not applying user defined U/Rs to the check process, you are still able to work to the original U/R of 1.0 when working in check mode.

User defined U/Rs can be applied to the following objects:

- Members; steel and concrete columns and beams, composite beams, steel joists and braces. This includes separate settings for each span/stack/panel of continuous beams/columns/walls.
- Steel Trusses - includes; Truss member top/bottom, Truss member side/internal.
- Portal frames (Individual members within Frames are treated as steel columns and steel beams as appropriate).
- Concrete Walls and Slabs including panels, patches, and punching shear checks.
- Foundations; Pad Bases (Spread Footings), Strip Bases, Pile Caps, Mat and Piled Mat Foundation slabs and Piles.

Apply user defined U/R for autodesign only

Applying a value of utilization ratio < 1.0 for autodesign only introduces a measure of conservatism into the autodesign process but doesn't fail the member until the ratio default value of 1.0 is exceeded.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window

2. Expand **Utilization ratio** and select **Apply (to autodesign)**, but do **not** select **Apply to check**
3. Enter the **Ratio limit** as required.

   NOTE For continuous beams/columns/walls you can apply separate ratio limits for the individual spans/stacks/panels if required.

When an **Autodesign** is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. When a **Check** is performed, the check will pass provided the utilization ratio is less than 1.0.

Apply user defined U/R for autodesign and check

Applying a value of utilization ratio < 1.0 for autodesign and check introduces a measure of conservatism into the autodesign process and fails the member when the ratio value that was set is exceeded.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window

2. Expand **Utilization ratio** and select both**Apply (to autodesign)** and **Apply to check**
3. Enter the **Ratio limit** as required.

> **NOTE** For continuous beams/columns/walls you can apply separate ratio limits for the individual spans/stacks/panels if required.

When an **Autodesign** is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. When a **Check** is performed, the check will pass provided the utilization ratio is less than the ratio limit.

Related information

See also

Review and modify user defined utilization ratios (page 1006)

Video: User defined utilization ratio

Validate the model for design issues

If required, you can manually validate the model for design issues before the model is submitted for design in order to trap errors that will cause the design to fail.

> **NOTE** For a list of validation issues and how to resolve them, refer to this support article.

Run design validation

> **NOTE** As design validation is performed automatically as part of all the combined analysis and design processes, there is no requirement to run design validation manually beforehand in those cases. You might however elect to run the validation manually when performing selective member and wall checking or design.

> **NOTE** Design validation cannot be performed if model validation is outdated.

- On the **Design** tab, click  **Validate**.

  Tekla Structural Designer performs the design validation checks. If your model contains any design validation issues, warning messages appear in the **Status** tab of the **Project Workspace**.

Adjust the conditions considered in design validation

1. On the **Home** tab, click **Model Settings**.
2. Go to the **Validation** page.
3. Select the conditions that you want Tekla Structural Designer to consider during validation checks.
4. Click **OK**.

5.2 Design slabs and mats, run punching shear checks

From the **Design** tab on the ribbon you can batch design all concrete slabs, slab patches and run all punching shear checks. As an alternative to batch slab design, once analysis has been performed you might choose to selectively design or check selected slabs and slab patches as required. Concrete mats can be designed in a similar way, but from the **Foundations** tab.

Slabs and mats can only be designed once a suitable analysis has been run to generate the required design forces. Because a certain amount of user interaction is required, they are not included in the **Design All** or **Design Concrete** combined analysis and design processes.

The slab and mat foundation batch design processes are quite similar and can be broken down into the following main steps:

1. Create slab patches over columns, beams, walls, or panels as required
2. Design slabs/ design mats (page 809) (the areas that lie outside the patches)
3. Design patches (the areas inside the patches)
4. Create punching shear checks as required
5. Run punching shear checks (page 815)

Create and modify patches

You can apply rectangular patches of reinforcement to individual slab items to act in addition to the background reinforcement. After selecting the patch type, you can place patches in the model by clicking or boxing around elements in 2D or 3D views.

Create column patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Column**.

3. In the **Properties** window, adjust the patch properties according to your needs.
   a. To specify the patch size, type values in the **Lx** and **Ly** fields.
   b. To specify the layer of the patch, select the desired option in **Surface**.
   c. To have the reinforcement automatically designed, select the **Autodesign** option.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place a patch over a specific column	• Click a column node within the slab. Tekla Structural Designer creates a patch to the selected column.

To	Do this
Place multiple patches	a. Move the mouse pointer to one corner of an imaginary box that will encompass the columns to which you want to create patches. b. Hold down the left mouse button. c. Drag the mouse pointer to the opposite corner of the box. d. Release the mouse button. Tekla Structural Designer creates patches to all columns entirely within the box.

Create beam patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Beam**.
3. In the **Properties** window, define the required patch width (perpendicular to the beam span) and the center slab width.

Tekla Structural Designer recalculates the two end strips accordingly. The strips cannot be modified.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place a patch over a specific beam span	• Click the required beam span. Tekla Structural Designer creates a patch centered on and orientated to the beam center line.

To	Do this
Place multiple patches	a. Move the mouse pointer to one corner of an imaginary box that will encompass the beams to which you want to create patches. b. Hold down the left mouse button. c. Drag the mouse pointer to the opposite corner of the box. d. Release the mouse button. Tekla Structural Designer creates patches to all beams entirely within the box.

Create wall patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Wall**.
3. In the **Properties** window, adjust the patch properties according to your needs.
   a. To specify the layer of the patch, select the desired option in **Surface**.
   b. To have the reinforcement automatically designed, select the **Autodesign** option.
   c. Define the required patch width perpendicular to the wall.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Create a patch along a specific wall	a. In the **Properties** window, set **Create Mode** to **Single Patch Along Wall**. b. Click the required wall. Tekla Structural Designer creates a patch, centered on and orientated to the wall center line.

To	Do this
Place an internal patch and two end patches along a specific wall	a. In the **Properties** window, set **Create Mode** to **Internal With End Patches**. b. Click the required wall. Tekla Structural Designer creates an internal patch and two end patches, centered and orientated to the wall center line.

To	Do this
Place an end patch at one end of a specific wall:	a. In the **Properties** window, set **Create Mode** to **End Patch at Wall End**. b. Click near the required end of the wall. Tekla Structural Designer creates a patch at the end of the wall that is closest to the point that you clicked.

To	Do this
Place an internal patch part way along a specific wall	a. In the **Properties** window, set **Create Mode** to **Internal Patch**. b. Click to define the start point of the patch along the required wall. c. Click to define the end point of the patch along the required wall. Tekla Structural Designer creates a patch between the selected points.

To	Do this
Create patches along multiple walls	a. In the **Properties** window, set **Create Mode** to **Single Patch Along Wall** or **Internal With End Patches**.

	b. Move the mouse pointer to one corner of an imaginary box that will encompass the walls to which you want to create patches. c. Hold down the left mouse button. d. Drag the mouse pointer to the opposite corner of the box. e. Release the mouse button. Tekla Structural Designer creates patches to all walls entirely within the box.

Create panel patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Panel**.
3. In the **Properties** window, adjust the patch properties according to your needs.
   a. To specify the patch size, type values in the **Lx** and **Ly** fields.
   b. To specify the layer of the patch, select the desired option in **Surface**.
   c. To have the reinforcement automatically designed, select the **Autodesign** option.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place the patch at the centroid of a specific panel	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is selected. b. Click anywhere within the required panel. Tekla Structural Designer creates a patch at the panel centroid.

To	Do this
Place the patch elsewhere within a specific panel	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is cleared.

	b. Still in the **Properties** window, do one of the following: • To define the patch by its corners, select the **Define corner points** option. • To define the patch by its center, clear the **Define corner points** option. c. In the panel, click the points that define the patch. Tekla Structural Designer creates a patch at the selected position.

To	Do this
Create multiple centroid patches	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is selected. b. Move the mouse pointer to one corner of an imaginary box that will encompass the panels to which you want to create patches. c. Hold down the left mouse button. d. Drag the mouse pointer to the opposite corner of the box. e. Release the mouse button. Tekla Structural Designer creates patches to the centroids of all panels entirely within the box.

Modify patch properties

1. Hover the mouse pointer over the patch that you want to modify.

   The **Select Entity** tooltip appears.

2. In the **Select Entity** tooltip, use the arrow keys to navigate to the patch name and press **Enter**.
3. In the **Properties** window, modify the patch properties according to your needs.
   a. To specify the patch size, type values in the **Lx** and **Ly** fields.
   b. To specify the layer of the patch, select the desired option in **Surface**.
   c. To have the reinforcement automatically designed, select the **Autodesign** option.
   d. To specify the directions of the reinforcement that you want to design, select the appropriate option in **Consider Strips**.
   e. To specify the type and direction of the reinforcement that you want to design, select the appropriate option in **Reinforcement**.
   f. To specify the required width for each strip, type a value in the **Width** field.

g. To specify whether each strip should be designed for the average or maximum design force, select the appropriate option in **Design Force**.

Resize patches

1. Hover the mouse pointer over the patch that you want to resize.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, use the arrow keys to navigate to the patch name and press **Enter**.
3. Hover the mouse pointer over the patch edge or corner that you want to adjust.

   The **Select Entity** tooltip appears.
4. In the **Select Entity** tooltip, use the arrow keys to navigate to the edge or corner node and press **Enter**.
5. Hold down the left mouse button and drag to move the edge or corner.

See also

Design and check slabs (page 806)

Design and check slabs

After creating any necessary patches, you can determine the top and bottom reinforcement requirements in all the slab item areas that lie outside of the patch areas. Because patch areas are excluded from slab item design, you should create patches before designing slab items.

NOTE Slabs can only be designed/checked after slab design moments have been established from a suitable analysis.

Check an individual slab item

- Do one of the following:

To	Do this
Check an individual slab from a 2D or 3D view	1. Hover the mouse pointer over the slab. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Check Panel**.

To	Do this

Check an individual slab from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs branch, locate the slab reference. 3. Right click and select **Check Panel**.

NOTE **Check Panel** uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all slab items

1. In the **Project Workspace**, click **Structure** tab.
2. Right click on the **Structure** branch.
3. In the context menu, select **Check model slabs** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all slab items on a single floor

- Do one of the following:

To	Do this
Check all slabs in a floor from a 2D view	1. Open a 2D view of the floor that you plan to design. 2. Right-click anywhere in the view. 3. In the context menu that appears, select **Check Slabs**. Tekla Structural Designer checks the reinforcement of all slab items on the floor.

To	Do this
Check all slabs in a floor from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Levels branch, right click the required **Level** and select **Check plane slabs**.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all slab items in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure containing the slabs you want to check.

3. In the context menu, select **Check sub structure slabs**.

   Tekla Structural Designer checks the reinforcement of all slab items on the floor.

Design an individual slab item

- Do one of the following:

To	Do this
Design an individual slab from a 2D or 3D view	1. Hover the mouse pointer over the slab. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Design Member**.

To	Do this
Design an individual slab from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs branch, locate the slab reference. 3. Right click and select **Design panel**.

NOTE **Design panel** uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design all slab items

- On the **Design** tab, click  **Design Slabs**.

  Tekla Structural Designer designs or checks all slab items in the model according to their autodesign settings.

Design all slab items on a single floor

- Do one of the following:

To	Do this
Design all slabs in a floor from 2D view	1. Open a 2D view of the floor that you plan to design. 2. Right-click anywhere in the view. 3. In the context menu that appears, select **Design Slabs**.

To	Do this
Design all slabs in a floor from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab.

	2. In the Levels branch, right click the required **Level** and select **Design plane slabs**.

Design all slab items in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure containing the slabs you want to design.
3. In the context menu, select **Design sub structure slabs**.

   Tekla Structural Designer designs all slab items in the sub structure, possibly selecting new reinforcement for them.

See also

Design and check patches (page 810)

Design and check mat foundations

TIP There are several steps in the mat/piled mat procedure that must be undertaken before running one of the below design/check commands. Click the appropriate design workflow link at the bottom of this page for details.

Design or check all mats in the model

1. On the **Foundations** tab, click **Design Mats**.

   Tekla Structural Designer designs or checks all mats in the model according to their auto design settings.

Check all mats in a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the scene view.
3. In the context menu, select **Check Slabs**.

   Tekla Structural Designer checks all slab items (including mat foundations) in the floor regardless of their autodesign settings.

Design all mats in a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the scene view.

3. In the context menu, select **Design Slabs**. Tekla Structural Designer designs all slab items (including mat foundations) in the floor regardless of their autodesign settings.

Check an individual mat

1. Hover the mouse pointer over the mat that you want to check.
2. Right-click the mat.
3. In the context menu that appears, select **Check Panel**.

   Tekla Structural Designer displays the results of the check in a new dialog box.

Design an individual mat

1. Hover the mouse pointer over the mat that you want to design.
2. Right-click the mat.
3. In the context menu that appears, select **Design Member**.

   Tekla Structural Designer displays the results of the design in a new dialog box.

Design and check patches

When that the slab item designs resulting from the slab or mat design process are satisfactory, you can then proceed to design patches. You should design slab items before designing patches, as the additional patch reinforcement requirement takes account of the existing level of reinforcement provided by the slab items.

NOTE For each strip within each patch, Tekla Structural Designer determines the area of the required steel ($A_{s,reqd}$) and the area of the provided steel ($A_{s,prov}$). The $A_{s,prov}$ calculation always considers the patch reinforcement, but if necessary and allowed, you can also include additional background reinforcement in the calculation.

Check an individual patch

1. Hover the mouse pointer over the patch that you want to check.
2. Right-click the patch.
3. In the context menu that appears, select **Check Slab Patch**.

   Tekla Structural Designer displays the results of the check in a new dialog.

Check all patches in the model

- The same command can be accessed in two ways:
  - In a 3D Structural View, right click on the background and choose **Check Patches** from the context menu.
  - In the **Project Workspace** Structure tree, right-click on the **Structure** branch and choose from the context menu.

  Tekla Structural Designer checks the reinforcement of all patches in the model.

Check all patches on a single floor

1. Open a 2D view of the floor that you plan to design.
2. Right-click anywhere in the view.
3. In the context menu that appears, select **Check Patches**.

   Tekla Structural Designer checks the reinforcement of all patches on the floor.

Design an individual patch

1. Hover the mouse pointer over the patch that you want to design.
2. Right-click the patch.
3. In the context menu that appears, select **Design Slab Patch**. Tekla Structural Designer displays the results of the design in a new dialog.

   The results of the design are displayed in a new dialog.

Design or check all patches in the model

- The same command can be accessed in three ways:
  - On the **Design** tab, click  **Design Patches**
  - In a 3D Structural View, right click on the background and choose **Design Patches** from the context menu.
  - In the **Project Workspace** Structure tree, right-click on the **Structure** branch and choose from the context menu.

  Tekla Structural Designer designs or checks all patches in the model according to their autodesign settings.

Design all patches on a single floor

1. Open a 2D view of the floor that you want to design.

2. Right-click anywhere in the view.
3. In the context menu that appears, select **Design Patches**.

   Tekla Structural Designer designs all patches on the floor, possibly selecting new reinforcement for them.

Create punching checks

Punching checks are the items to which the punching shear information are attached. You can create them at various locations in flat slabs and foundation mats.

Create punching checks in a flat slab

To create punching checks in a flat slab, do the following:

1. On the ribbon, go to the **Design** tab.
2. Select  **Punching Check**.
3. In the **Properties** window, adjust the check properties according to your needs.

   NOTE The tension reinforcement defaults to the slab **Top** surface when the check is created from the **Design** tab. (If created from the **Foundations** tab the only difference is that it defaults to the **Bottom** surface.)

4. To create checks in either a 2D or 3D scene view, do one of the following:
   - Hover the mouse pointer over the desired column node, wall node, or point load. When the **Pick Punching Check Location** tooltip appears, click to add the punching check at the selected node.
   - Hold down the left mouse button and drag a box to add punching checks to all potential check locations within the box.

Create punching checks in a foundation mat

To create punching checks in a foundation mat, do the following:

1. On the ribbon, go to the **Foundations** tab.
2. Select  **Punching Check**.
3. In the **Properties** window, adjust the check properties according to your needs.

NOTE The tension reinforcement defaults to the mat **Bottom** surface when the check is created from the **Foundations** tab. (If created from the **Design** tab the only difference is that it defaults to the **Top** surface.)

4. To create checks in either a 2D or 3D scene view, do one of the following:
   - Hover the mouse pointer over the desired column node, wall node, or point load. When the **Pick Punching Check Location** tooltip appears, click to add the punching check at the selected node.
   - Hold down the left mouse button and drag a box to add punching checks to all potential check locations within the box.

Specify the axis orientation for punching checks applied to point loads

When you apply a punching check to a **point load**, by default the check Y and Z axes are automatically orientated to align with global X and Y.

If this is not appropriate you can manually define the punching check Z axis orientation as follows:

1. In the **Properties** window, select **User loaded area**.
2. Specify the **Loaded Area Orientation** angle in relation to the global Y axis.

NOTE The above doesn't apply for checks to columns because in that case the check Y and Z axes are automatically orientated to align with the column major and minor axes. The two axis systems are locked together, so if the column is rotated, the punching check axes also rotate. For more information, see:

Override the punching check loaded area

When you apply a punching check **to a point load**, the default loaded area is taken as 4in x 4in (US Customary units), or 100mm x 100mm (metric units).

When you apply a punching check **to a column**, the loaded area is based on the column section - the default being the section boundary box.

There can be certain situations where the above defaults may not be appropriate, (for example when the check is applied to the bottom of a column attached to a base plate or pedestal).

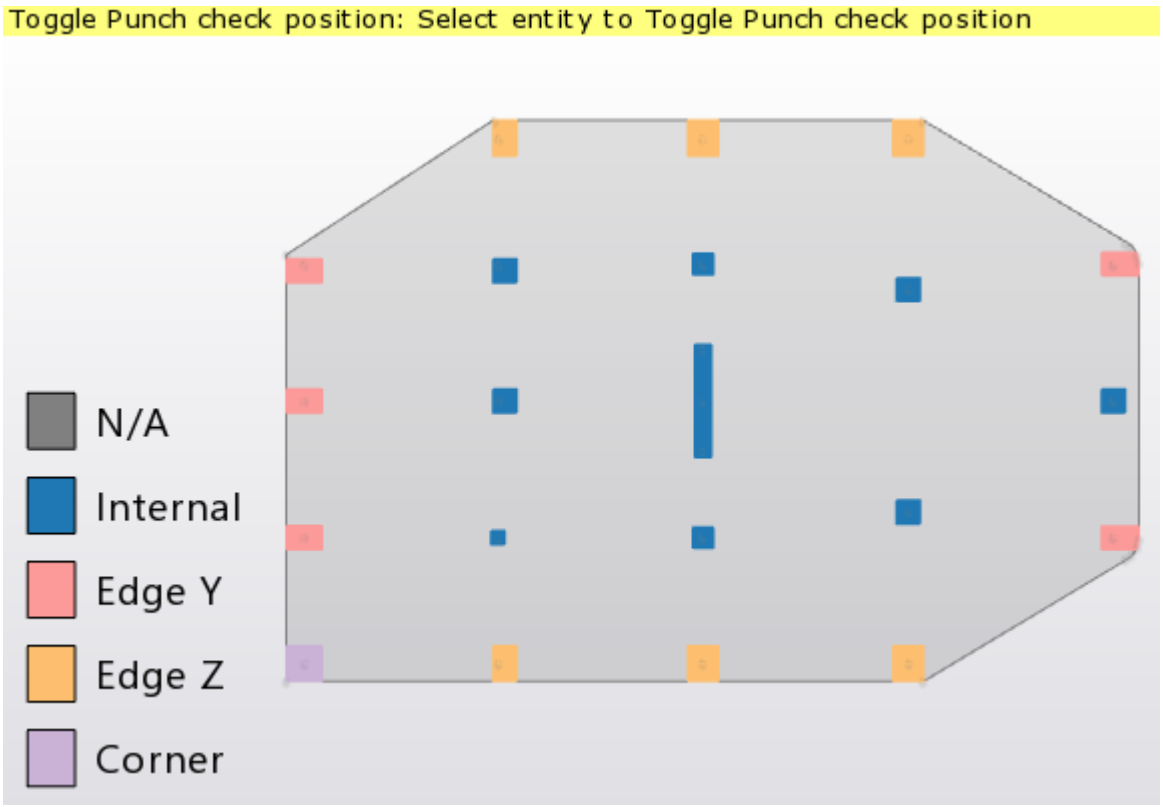
In such situations, you can override the default and specify a **User loaded area** as follows:

1. In the **Properties** window, select **User loaded area**.
2. Specify the **Loaded Area Breadth** and **Loaded Area Depth**.

Confirm punching check positions

When you apply a punching check, the program automatically detects whether a slab is present in each of punching check axis directions. In the **Properties** window, **Slab in y positive/negative** and **Slab in z positive/negative** are set accordingly. Based on these, the punching check is assumed to be at an **Internal**, **Edge Y**, **Edge Z** or **Corner** position.

It is very easy to graphically review and modify the check position (page 988) from a Review View as shown in the image below.



Alternatively, you could manually adjust the position from the **Properties** window as follows:

1. Select **Slab Override**
2. Change **Slab in y positive/negative** and **Slab in z positive/negative** as required.

Specify stud rail reinforcement

RESTRICTION Stud rail reinforcement is only available for Eurocode and ACI regional codes.

To specify the reinforcement:

1. Create a punching check item as instructed above.
2. Right-click the punching check item.
3. In the context menu, select **Edit Reinforcement**. The Punching shear reinforcement properties dialog box (page 1575) is displayed.
4. Select the **Use reinforcement** option.
5. Select whether you want to autodesign (page 1578) the reinforcement, and, if auto-designing, select the required reinforcement starting from (page 1578) parameter.
6. Select the reinforcement parameters (page 1579) according to your needs.
7. Adjust the stud rail parameters (page 1580) according to your needs.

   NOTE For more information about rail layout arrangements, see: .

8. Provided the analysis has been performed, you can run the punching shear calculations for the specified layout by selecting **Check...** on the right hand side of the dialog box.
9. When the layout is acceptable, click **OK** The reinforcement data is saved to the punching check item **Properties** window.

See also

Design and check punching shear (page 815)

Design and check punching shear

Once you have created your punching check items (page 812) you can proceed to design or check them. You can design/check individually, by floor, or for the whole model, (although some of the methods described below are only available when working to the Eurocode or ACI regional code).

NOTE As the checks are dependent on using the correct levels of slab reinforcement (typically provided by patch reinforcement), you should only design or check punching shear after you have designed the slabs (page 806) and designed the patches (page 810) at the check locations.

Design an individual punching check item

NOTE This method is only available when working to the Eurocode or ACI head code.

- Do one of the following:

To	Do this

Design an individual punching check item from within the Punching Shear Reinforcement Properties dialog	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Edit Reinforcment**. 4. Select Auto-design 5. Select **Check...**.

To	Do this
Design an individual punching check item from a 2D or 3D view	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Design Punching Shear**.

To	Do this
Design an individual punching check item from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs> Punching Checks branch, locate the check reference. 3. Right click and select **Design Punching Shear**.

Tekla Structural Designer displays the results of the design in a new dialog box.

Design all punching check items on a floor

NOTE This method is only available when working to the Eurocode or ACI head code.

1. In the **Project Workspace**, click **Structure** tab.
2. In the Levels branch, right click the required **Level** and select **Design Punching Shear**.

Design all punching check items

- Do one of the following:

To	Do this
Design all punching check items from the ribbon	1. On the **Design** tab, click  **Design Punching Shear**.

To	Do this

Design all punching check items from the Structure Tree	**NOTE** This method is only available when working to the Eurocode or ACI regional code. 1. In the **Project Workspace**, click **Structure** tab. 2. Right click the Structure branch and select **Design Punching Shear**.

- If working to the Eurocode or ACI regional code, Tekla Structural Designer performs punching shear checks for all punching check items in the model based on their individual auto-design setting.
- If working to other regional codes, Tekla Structural Designer checks all punching check items in the model.
- If the punching check item does not connect to a flat slab, punching shear is flagged as beyond scope.

Check an individual punching check item

- Do one of the following:

To	Do this
Check an individual punching check item from a 2D or 3D view	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Check Punching Shear**.

To	Do this
Check an individual punching check item from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs> Punching Checks branch, locate the check reference. 3. Right click and select **Check Punching Shear**.

To	Do this
Check an individual punching check item from within the Punching Shear Reinforcement Properties dialog	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Edit Reinforcment**. 4. Unselect Auto-design 5. Select **Check...**.

Check all punching check items on a floor

1. In the **Project Workspace**, click **Structure** tab.
2. In the Levels branch, right click the required **Level** and select **Check Punching Shear**.

Check all punching check items

1. In the **Project Workspace**, click **Structure** tab
2. Right click the Structure branch and select **Check Punching Shear**.

See also

Review punching check status (page 949)

5.3 Design isolated foundations

You can design isolated foundations (reinforced pad bases, strip bases, and pile caps) either in a batch or individually. Unreinforced pad bases are currently beyond scope.

NOTE Before bases or pile caps can be designed, you have to run an analysis to establish the design forces. You can run either the **Analyze All (Static)** command or one of the combined analysis and design commands (such as **Design All**).

Design or check all pad bases and strip bases

1. On the **Foundations** tab, click  **Design Pad Bases**.

   Tekla Structural Designer designs or checks all pad bases and strip bases in the model according to their individual auto-design settings. In the process, Tekla Structural Designer considers all active static and RSA combinations .

Design or check all pile caps

1. On the **Foundations** tab, click  **Design Pile Caps**.

   Tekla Structural Designer designs or checks all pile caps in the model according to their individual auto-design settings. In the process, Tekla Structural Designer considers all active static and RSA combinations .

Check an individual isolated foundation

1. Hover the mouse pointer over the foundation that you want to check.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, navigate to the foundation name by using the arrow keys.
3. Right-click the foundation.
4. In the context menu, select **Check Member**. Tekla Structural Designer displays the results of the check in a new dialog box.

Design an individual isolated foundation

1. Hover the mouse pointer over the foundation that you want to design.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, navigate to the foundation name by using the arrow keys.
3. Right-click the foundation.
4. In the context menu, select **Design Member**. Tekla Structural Designer displays the results of the design in a new dialog box.

5.4 Check floor vibration

To establish the response of a floor to dynamic excitation you can either apply a simplified floor vibration check (as described here), or you might choose to perform a more rigorous footfall assessment.

NOTE Tekla Structural Designer's footfall assessment (which can be performed to either of the SCI P354, CCIP 016, or AISC DG11 design guides) is described elsewhere, see: .

If you apply a simplified floor vibration check, it will be applied over a user-defined rectangular or polygon shaped slab area by using the **Vibration**

Check command in the **Floor Vibration** group. In order to create the check, you need to identify a primary beam, secondary beam and critical slab item. In addition, you must specify associated data in order to perform the floor vibration calculation.

Create floor vibration check items

Floor vibration check items are the objects to which the floor vibration check information and calculations are attached.

NOTE Floor vibration check items can only be created in 2D Views.

1. Go to the **Design** tab on the ribbon and click  **Vibration Check**.
2. Go to the **Properties** window.
3. In **FloorPlateDefinition**, select whether you want to create a rectangular or polygonal slab.
   a. If you selected the **Rectangular** option, adjust the local x angle to define the angle at which the rectangle will be drawn, if necessary.
4. To create a check area, in the model, do one of the following depending on the slab shape:
   - **Rectangular**: Click once to define the first corner of the rectangle, and again to define the opposite corner.
   - **Polygon**: Click to define the corners of the polygon. In order to close the shape, click the first point again.
5. Click to define the primary beam.

   The primary beam properties appear in the **Properties** window.
6. Click to define the secondary beam.

   The secondary beam properties appear in the **Properties** window.
7. Click to define the critical slab item.

   NOTE If the critical slab item is a composite slab, Tekla Structural Designer automatically defines the slab item properties. In other cases, you need to input the properties yourself.

   The critical slab item properties appear in the **Properties** window.
8. In the **Properties** window, review and adjust the check item properties.

9. Click anywhere in the model to create the check item.

   Tekla Structural Designer highlights the primary beam, secondary beam and critical slab item of the current check item.

10. Create new checks, or press **Esc** to finish creating floor vibration check items.

Create floor vibration checks that consider two or three adjoining spans

Creating two-span or three-span floor vibration checks is almost identical to that required for single spans. When creating the check item, do the following:

- In the **Properties** window, set the **Adjoining Spans** property of a primary and/or secondary beam to **Two span** or **Three span**.

NOTE If the **Two span** option is selected, note the following:

- When you hover the mouse pointer to select the beam, only beams of two or more spans are available.
- When a beam is highlighted, note that the beam directly under the mouse pointer will become the critical beam, and the second highlighted beam will become the adjoining beam.
- The second highlighted beam (the adjoining beam) will be the beam closest to the mouse pointer position. That's why, in order to highlight the adjoining beam at a particular end of the critical beam, you can move the mouse pointer toward that end of the critical beam.

NOTE If the **Three span** option is selected, note the following:

- When you hover the mouse pointer to select the beam, only beams of three or more spans are available to be highlighted,
- When a beam is highlighted, note that the beam directly under the mouse pointer will become the critical beam, and the second highlighted beam will become the adjoining beam.
- The second highlighted beam (the adjoining beam) will be the beam closest to the mouse pointer position. That's why, in order to highlight the adjoining beam at a particular end of the critical beam, you can move the mouse pointer toward that end of the critical beam.

Modify the properties of existing floor vibration check items

1. In **Scene Content**, ensure that Floor Vibration Checks have been turned on.
2. Hover the mouse pointer over the slab area where the floor vibration check item lies.

   The **Select Entity** tooltip appears.
3. Navigate to the floor vibration check item name by using the arrow keys, and click **Enter**.

   The floor vibration check properties appear in the **Properties** window.
4. In the **Properties** window, adjust the check item properties according to your needs.

Check vibration for all floor vibration check items

- Go to the **Design** tab on the ribbon and click **Check Floor Vibration**.

  Tekla Structural Designer performs floor vibration checks for all floor vibration check items in the model.

Check floor vibration for an individual floor vibration check item

1. Hover the mouse pointer over the floor vibration check item that you want to check.

   The **Select Entity** tooltip appears.
2. If necessary, navigate to the floor vibration check item name by using the arrow keys.
3. Right-click the check item.
4. In the context menu, select **Check Floor Vibration**. Tekla Structural Designer displays the results of the design in a new dialog box.

5.5 Check drift, sway, seismic drift, wind drift, and overall displacements

As part of the static design process the model is automatically checked on a level by level basis for drift (page 823) (ACI/AISC), or sway (page 828) (other regional codes). Also any seismic load combinations are checked for seismic

drift (page 832) and wind load combinations checked for wind drift (page 836)/overall wind drift (page 840).

Click the links below to find out more:

- Drift check (page 823)
- Sway check (page 828)
- Seismic drift check (page 832)
- Wind drift check (page 836)
- Overall wind drift check (page 840)
- Check overall displacement (page 842)

Drift check

By default, this check is applied to all columns (of all materials) and all walls at every level in the model.

NOTE The drift check is only relevant to models that use the ACI/AISC regional code.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Reporting Level** option is displayed in Level Properties (page 1333). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the story drift in each direction (i.e. the difference between top and bottom deflection of the stack) is calculated, and from this the stability coefficient is determined as follows:

Stability coefficient = Δ_2/Δ_1

Where

Δ_2= second order drift

Δ_1= first order drift

A Warning status is displayed against any stack in which the ratio exceeds 1.71 and a Beyond Scope status is applied if it exceeds 2.85 (as this indicates the structure lacks stiffness in whole or in part).

NOTE Stability coefficients can only be calculated accurately by performing a second order analysis, however indicative values can still be displayed when only a first order analysis has been performed. These are determined from the column's elastic critical load factor, λ:

$(\Delta_2/\Delta_1) = 1/(1- 1/\lambda)$

Approximate second order drift:

$\Delta_2= \Delta_1/(1- 1/\lambda)$

A workflow for running the check is outlined below:

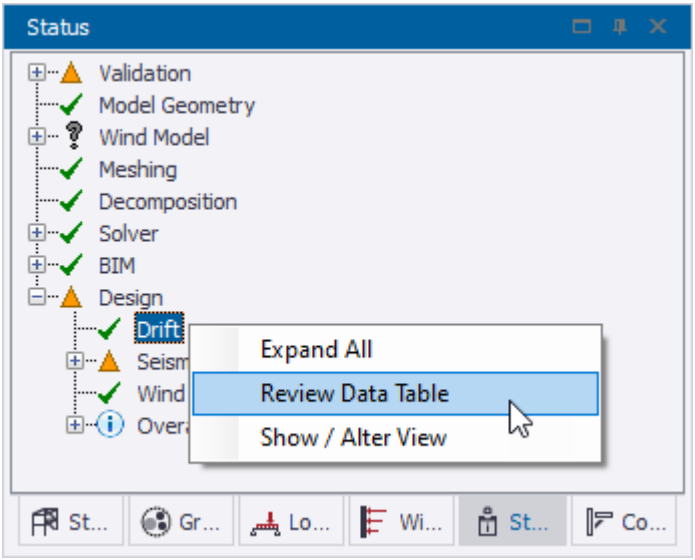
Run the check

Load combinations are automatically checked on a level by level basis for drift by running a static analysis or design.

Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 59), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



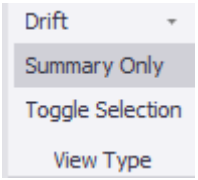
From this menu you can display the drift check summary in a Review Data Table (page 1020).

Review Data

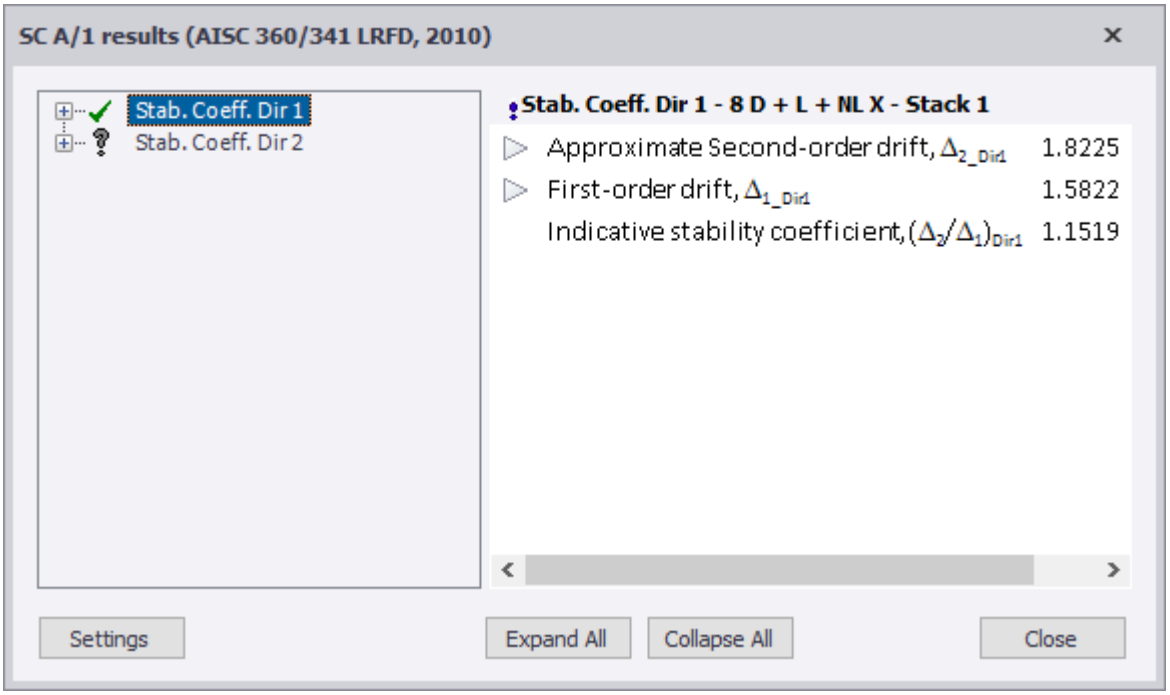
Drift

Level	Ref.	Stack	Combination Dir 1	Drift 1st order Dir 1 [mm]	Drift 2nd order Dir 1 [mm]	Ratio Dir 1	Combination Dir 2	Drift 1st order Dir 2 [mm]	Drift 2nd order Dir 2 [mm]	Ratio Dir 2	Status	Details
St. 2 (2)	SC A/1	2	8 D + L + NL X	0.7615	0.8131	1.0678	-	-	-	-	Pass	Details...
St. 1 (1)	SC A/4	1	8 D + L + NL X	1.5822	1.8225	1.1519	-	-	-	-	Pass	Details...

By unselecting the **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.



The checks can be investigated in more detail by clicking the **Details** button at the end of any row.



This displays the stability coefficients in both directions (RatioDir1,RatioDir2). These are calculated as follows:

Stability coefficient = Δ_2/Δ_1

Where

Δ_2= second order drift

Δ_1= first order drift

A Warning status is displayed against any stack in which the ratio exceeds 1.71 and a Beyond Scope status is applied if it exceeds 2.85 (as this indicates the structure lacks stiffness in whole or in part).

Stability coefficients can only be calculated accurately by performing a second order analysis, however indicative values can still be displayed when only a first order analysis has been performed. These are determined from the column's elastic critical load factor, λ:

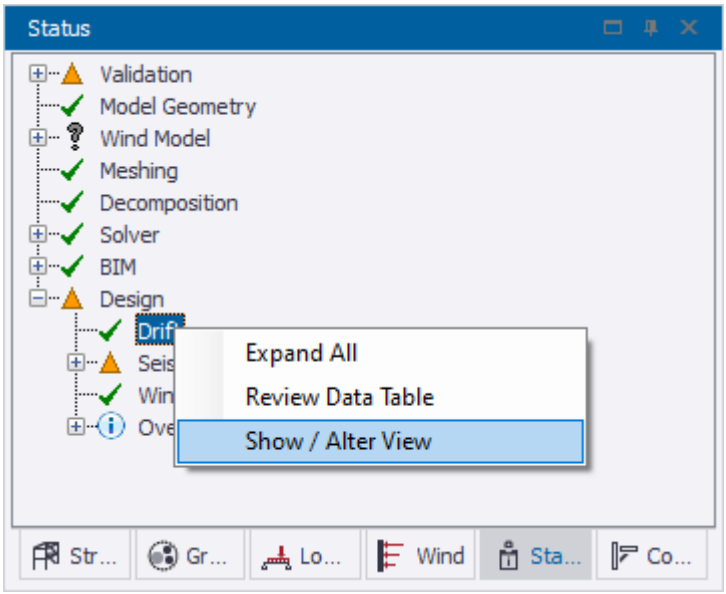
$(\Delta_2/\Delta_1) = 1/(1- 1/\lambda)$

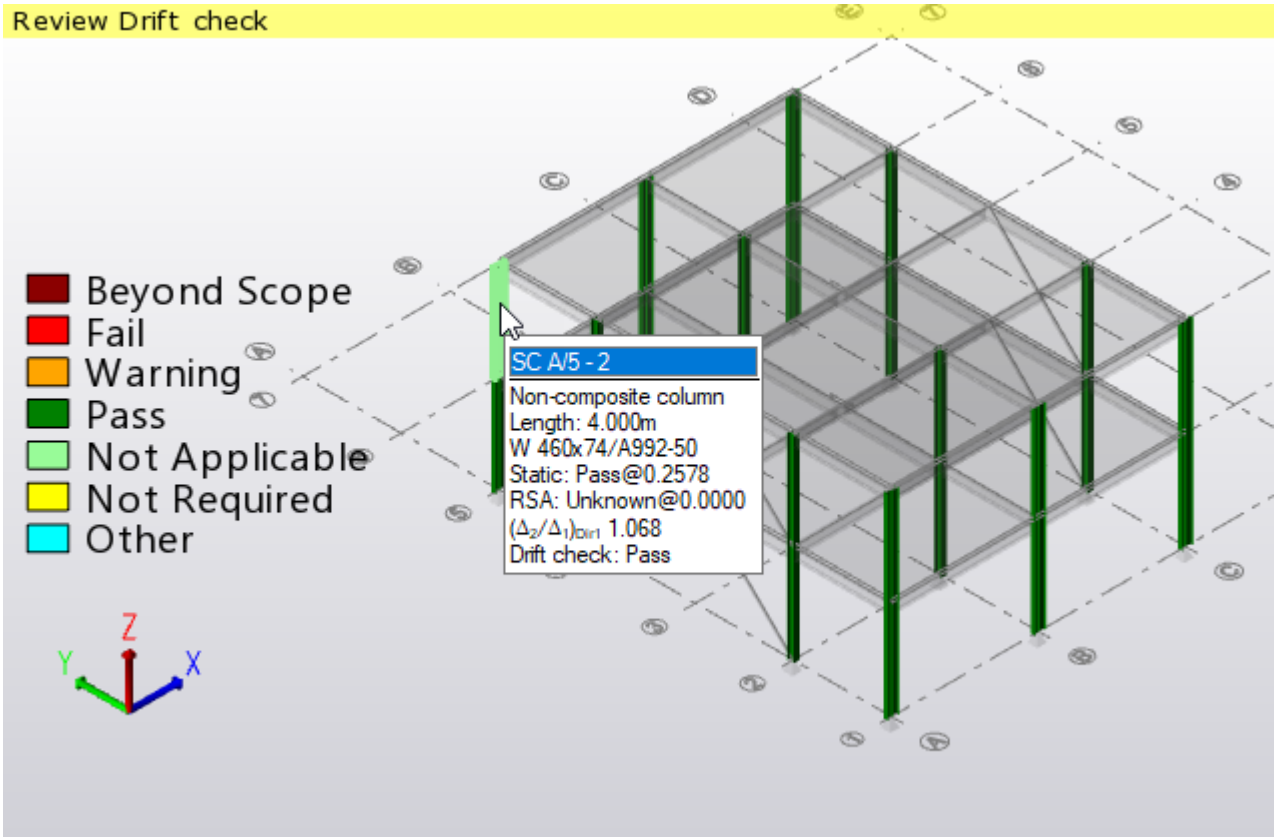
Approximate second order drift:

$\Delta_2= \Delta_1/(1- 1/\lambda)$

Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 981).





The color coded legend makes it easy to see individual failing members. By hovering the cursor over a member, its drift results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have the option to switch off checks for specific stacks (page 981) if you require.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 845), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 842) where they are less than a specified limit. Stacks check lengths can also be manually adjusted (page 846) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 845) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The drift check results table is available either by creating a Building Analysis & Drift Checks report (page 1078), or by including the Analysis>Drift chapter in a custom report.

Sway check

By default, this check is applied to all columns (of all materials) and all walls at every level in the model.

NOTE The sway check is not relevant to models that use the ACI/AISC regional code.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Reporting Level** option is displayed in Level Properties (page 1333). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the story sway in each direction (i.e. the difference between top and bottom deflection of the stack) is calculated and from this the elastic critical load factor is determined.

A workflow for running the check is outlined below:

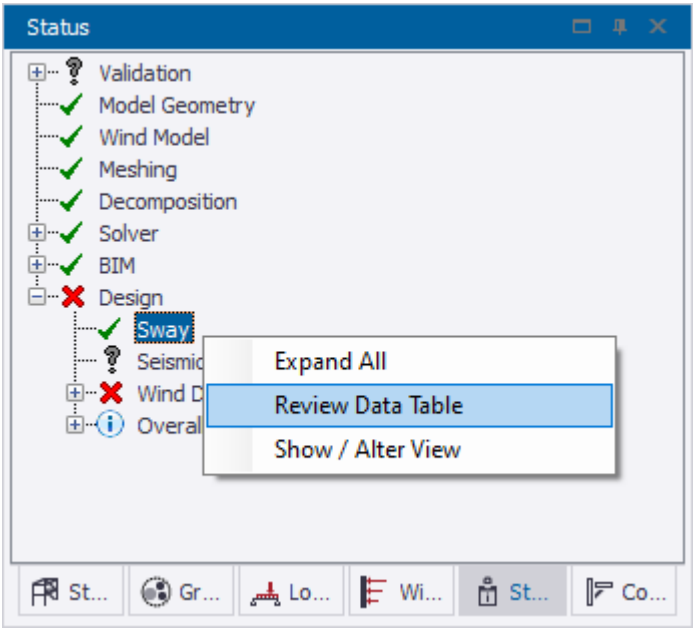
Run the check

Load combinations are automatically checked on a level by level basis for sway by running a static analysis or design.

Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 59), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



From this menu you can display the sway check summary in a Review Data Table (page 1014).

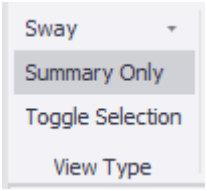
Structure 3D | Review Data

Critical Sway

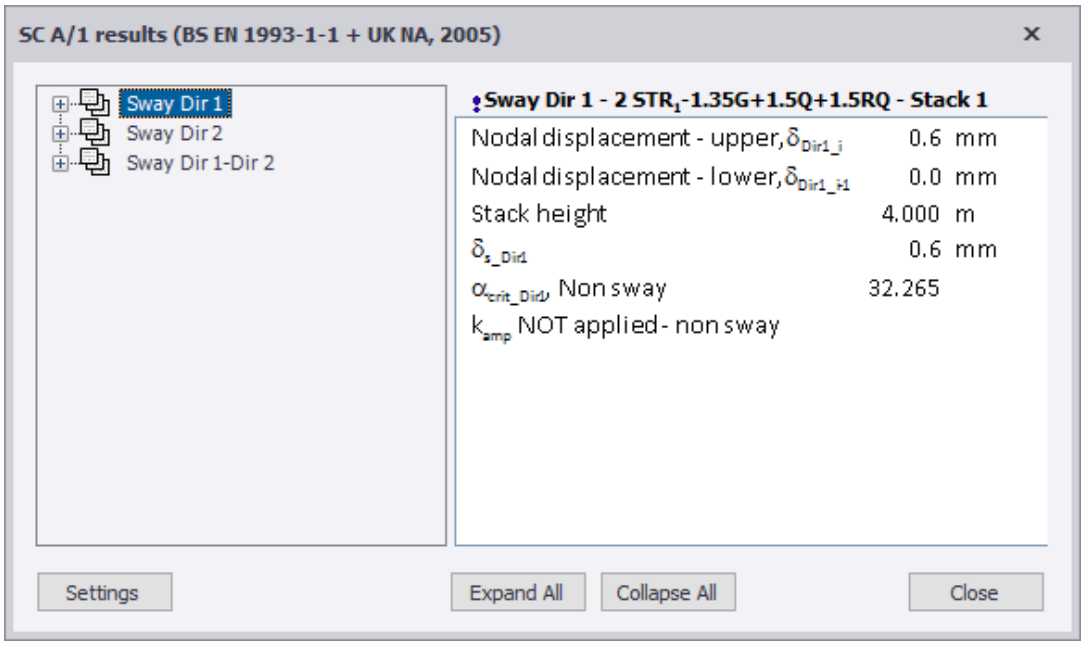
Level	Ref.	Stack	Combination Dir 1	α_{Dir1}	Combination Dir 2	α_{Dir2}	Combination Dir 1/2	Twist	Status	Details
Top	SC C/2	3	2 STR_1-1.35G+1.5Q+1.5RQ	110.659	2 STR_1-1.35G+1.5Q+1.5RQ	132.257	2 STR_1-1.35G+1.5Q+1.5RQ	1.017	✓ Pass	Details...
Top	SC B/3	3	2 STR_1-1.35G+1.5Q+1.5RQ	143.294	2 STR_1-1.35G+1.5Q+1.5RQ	84.744	4 $STR_{6.2}$-1.35G+1.5Q+1.5ψ_0S+1.5ψ_0W+EHF_{Dir1+}	1.044	✓ Pass	Details...
St. 2 (2)	SC A/1	2	2 STR_1-1.35G+1.5Q+1.5RQ	54.427	2 STR_1-1.35G+1.5Q+1.5RQ	57.568	2 STR_1-1.35G+1.5Q+1.5RQ	1.003	✓ Pass	Details...
St. 2 (2)	SC C/1	2	2 STR_1-1.35G+1.5Q+1.5RQ	54.427	2 STR_1-1.35G+1.5Q+1.5RQ	55.859	2 STR_1-1.35G+1.5Q+1.5RQ	1.003	✓ Pass	Details...
St. 1 (1)	SC A/1	1	2 STR_1-1.35G+1.5Q+1.5RQ	32.265	2 STR_1-1.35G+1.5Q+1.5RQ	33.091	2 STR_1-1.35G+1.5Q+1.5RQ	1.001	✓ Pass	Details...
St. 1 (1)	SC C/1	1	2 STR_1-1.35G+1.5Q+1.5RQ	32.265	2 STR_1-1.35G+1.5Q+1.5RQ	32.630	2 STR_1-1.35G+1.5Q+1.5RQ	1.001	✓ Pass	Details...

In this example, at the Level "Top", steel column "SC C/2" is critical for Dir 1 and steel column "SC B/3" is critical for Dir 2. For completeness of the table both Dir 1 and Dir 2 are shown for both columns.

By unselecting the **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.

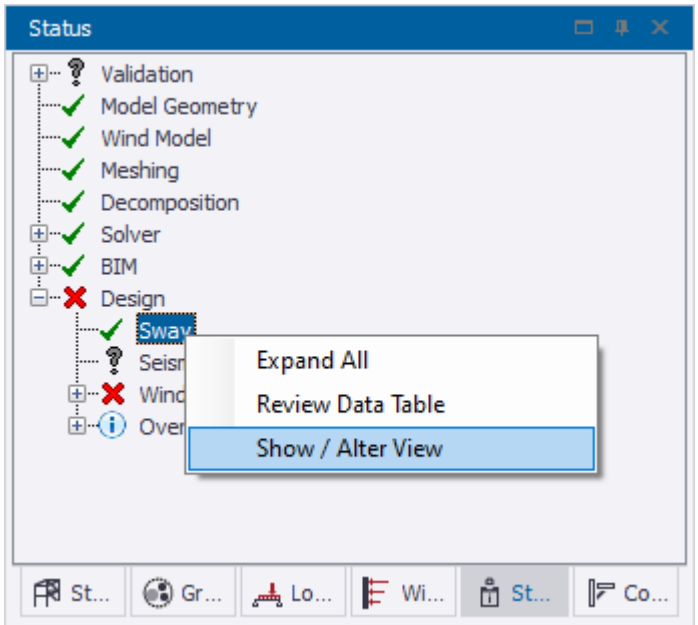


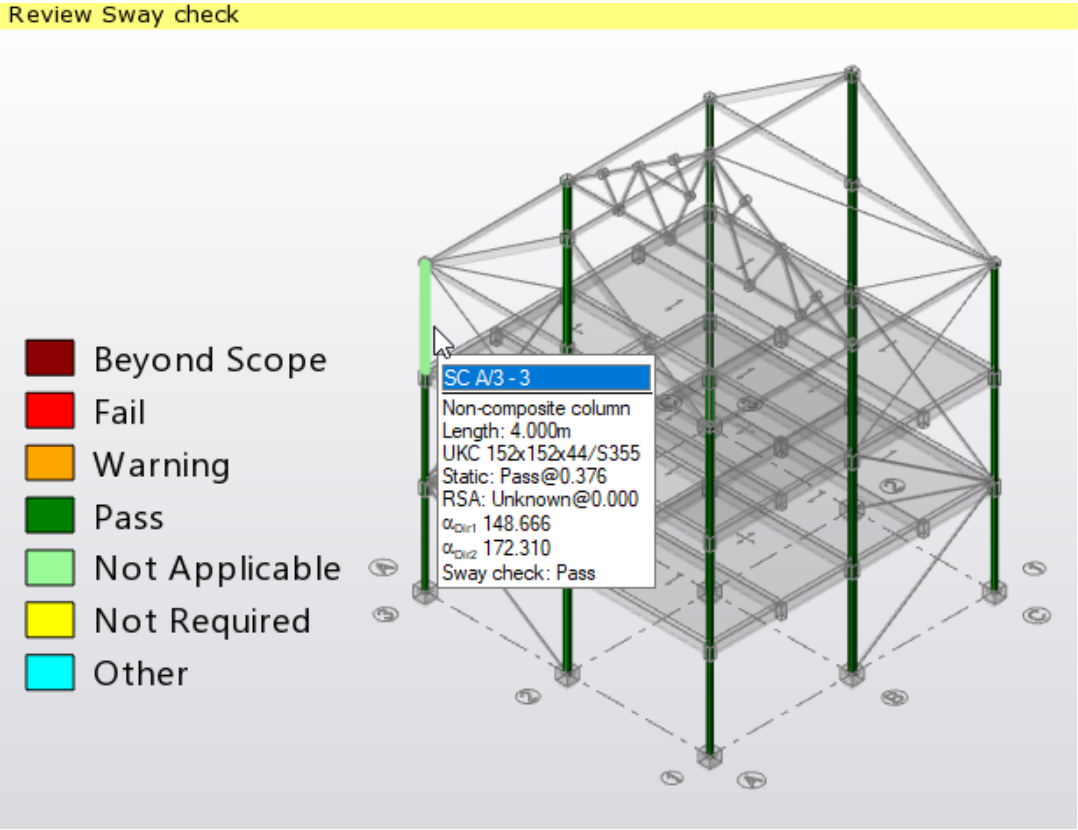
The individual stack by stack checks can be investigated in more detail by clicking the **Details** button at the end of any row.



Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 1005).





The color coded legend makes it easy to see any individual failing members. By hovering the cursor over a member, its sway results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have the option to switch off checks for specific stacks (page 1005) if you require.

If required, sway values can also be displayed graphically (page 668) in a Results View.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 844), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 842) where they are less than a specified limit. Stacks check lengths can also be be manually adjusted (page 846) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 845) if you decide they are inappropriate, (for example if the

story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The sway check results table is available either by creating a Building Analysis & Drift Checks report (page 1078), or by including the Analysis>Sway chapter in a custom report.

Seismic drift check

By default this check is applied to all columns (of all materials), and all walls at every level in the model.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Reporting Level** option is displayed in Level Properties (page 1333). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the seismic stability coefficient θ in each direction is checked against the limit θ_{max} and then the stack design story drift Δ_{des} is determined in each direction and checked against the allowable story drift Δ_a

A workflow for running the check is outlined below:

Configure the check and set the limit

The check is configured on the Localization page of the **Seismic Wizard**.

An option is provided to override the allowable story drift factor and a second option allows for the seismic drift check to be skipped entirely if required.

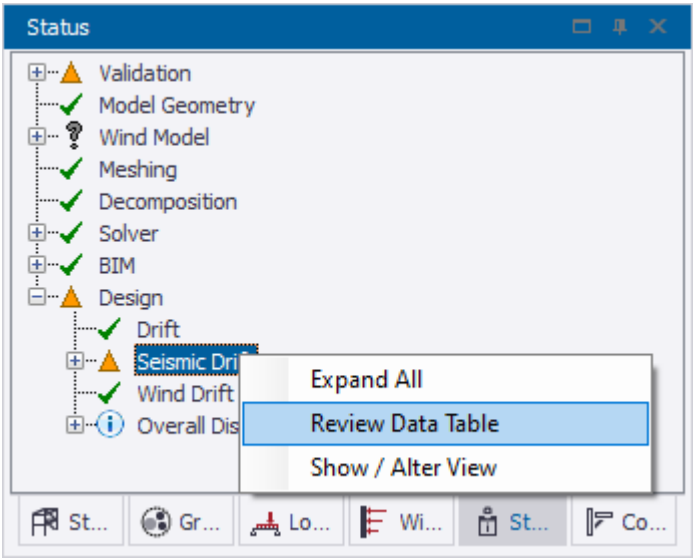
Run the check

Seismic load combinations are automatically checked on a level by level basis for drift by running an appropriate analysis or design.

Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 59), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



From this menu you can display the seismic drift check summary in a Review Data Table (page 1023), an ASCE example of this table is shown below.

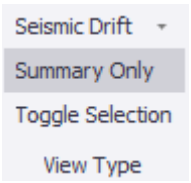
In the summary, either one or two rows of results are output for each level; when two rows are output the critical drift ratio in Dir 1 is given in one row, and the critical drift ratio in Dir 2 in the other, (for completeness of the table, in each row the value in the other direction is also shown even though it might not be critical).

Review Data

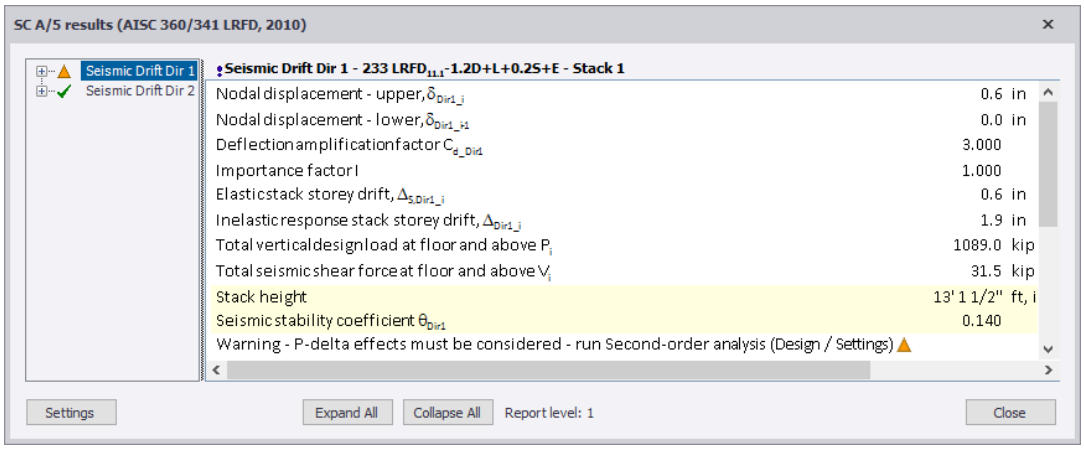
Seismic Drift

Level	Ref.	Stack	Combination Dir 1	Drift Ratio Dir 1	θ_{Dir1} Ratio	Combination Dir 2	Drift Ratio Dir 2	θ_{Dir2} Ratio	Status	Details
St. 2 (2)	SC A/5	2	234 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.324	0.335	238 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.026	0.025	Pass	Details...
St. 2 (2)	SC A/1	2	233 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.324	0.335	238 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.026	0.025	Pass	Details...
St. 1 (1)	SC A/5	1	234 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.719	0.854	238 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.032	0.041	Warning	Details...
St. 1 (1)	SC A/1	1	233 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.719	0.854	238 $LRFD_{11.5}$-1.2D+L+0.2S+E	0.032	0.041	Warning	Details...

By unselecting the **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.

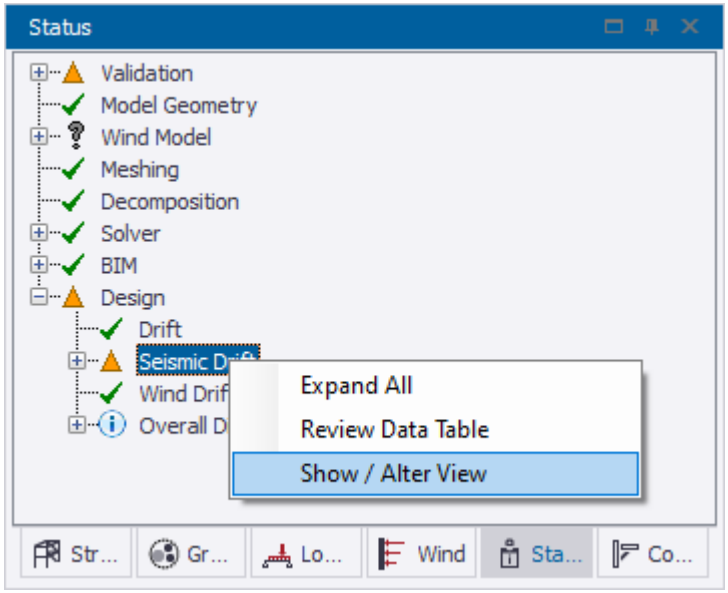


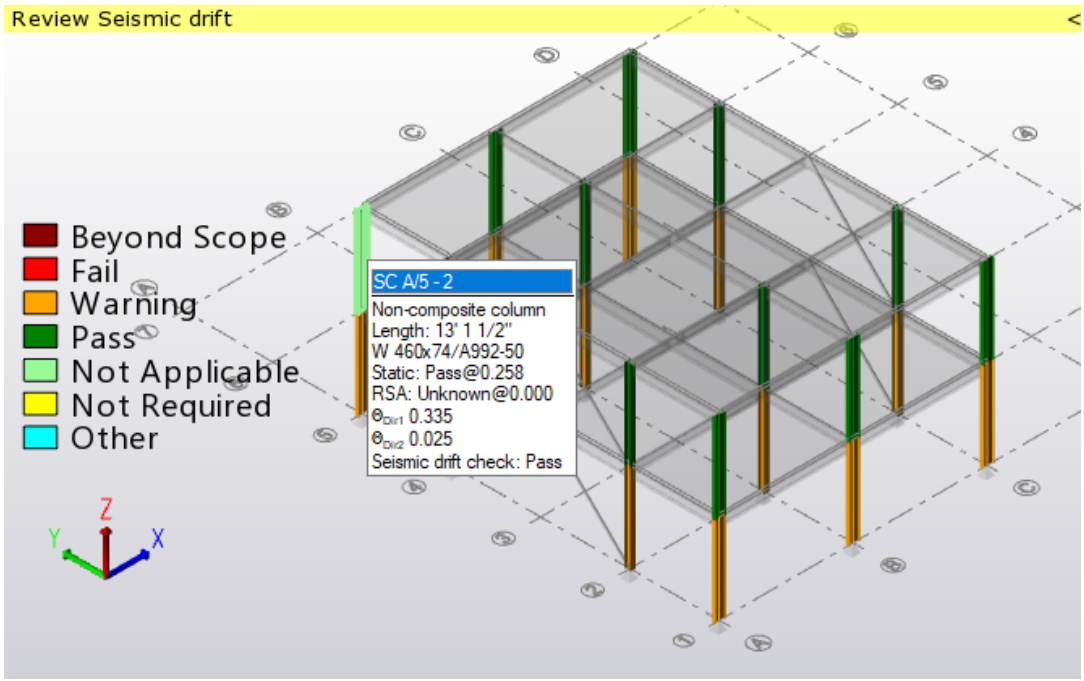
The individual stack by stack checks can be investigated in more detail by clicking the **Details** button at the end of any row.



Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 998).





The color coded legend makes it easy to see individual failing members. By hovering the cursor over a member, its seismic drift results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have the option to switch off checks for specific stacks (page 998) if you require.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 845), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 842) where they are less than a specified limit. Stacks check lengths can also be be manually adjusted (page 846) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 845) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The seismic drift check results table is available either by creating a Seismic Design report (page 1084), or by including the Analysis>Seismic Drift chapter in a custom report.

Wind drift check

By default this check is applied to all columns (of all materials), and all walls at every level in the model.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Reporting Level** option is displayed in Level Properties (page 1333). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the lateral drift resulting from the analysis is determined for each wind load combination using the combination SLS (Service Level) factors - (which can be < 1.0). This is then compared against a user-defined limit.

A workflow for running the check is outlined below:

Configure the check and set the limit

The check is configured from **Design Settings**.

On the **Drift Checks** page (if working to the ACI/AISC codes) or **Sway & Drift Checks** page (if working to any other regional code) you can enter the limit required.

NOTE By default the limit is set as 1/300 of the story height, in line with Eurocode 3 recommendations.

Individual limits can also be applied as overrides to selected stacks (page 846) if required.

By default the check is performed in the resultant wind drift direction only, but by unselecting **Check for Resultant Wind Drift** it can be performed in each of Building Directions 1 & 2 if required.

By default the check only considers the effects of the wind loadcase(s) in wind combinations, but by unselecting **Check wind cases only** it will consider the effects of all loadcases in wind combinations (which would include drift induced by gravity loads).

Run the check

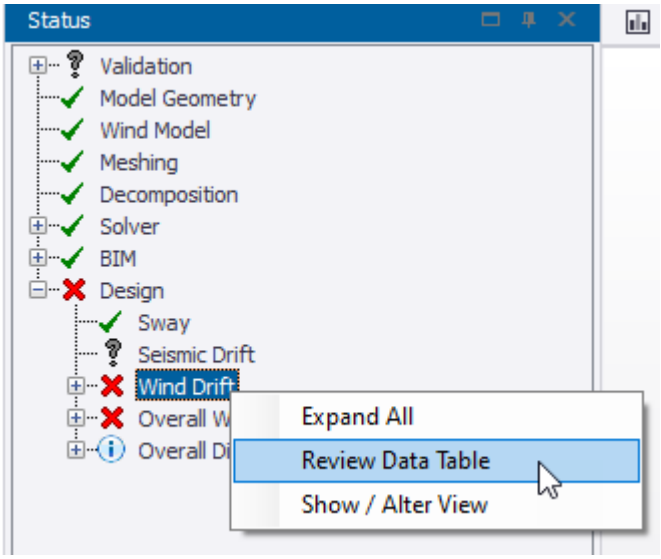
Wind load combinations are automatically checked on a level by level basis for wind drift during the structure static design, and also when any 3D analysis is

run in isolation. If a sub-set of combinations are considered for analysis then only those combinations are considered in the wind drift checks, (allowing engineers working on larger structures to investigate and optimize the lateral load resisting systems more rapidly).

Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 59), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



From this menu you can display the wind drift check summary in a Review Data Table (page 1026), an example of this table for a Eurocode model is shown below.

Review Data

Wind Drift

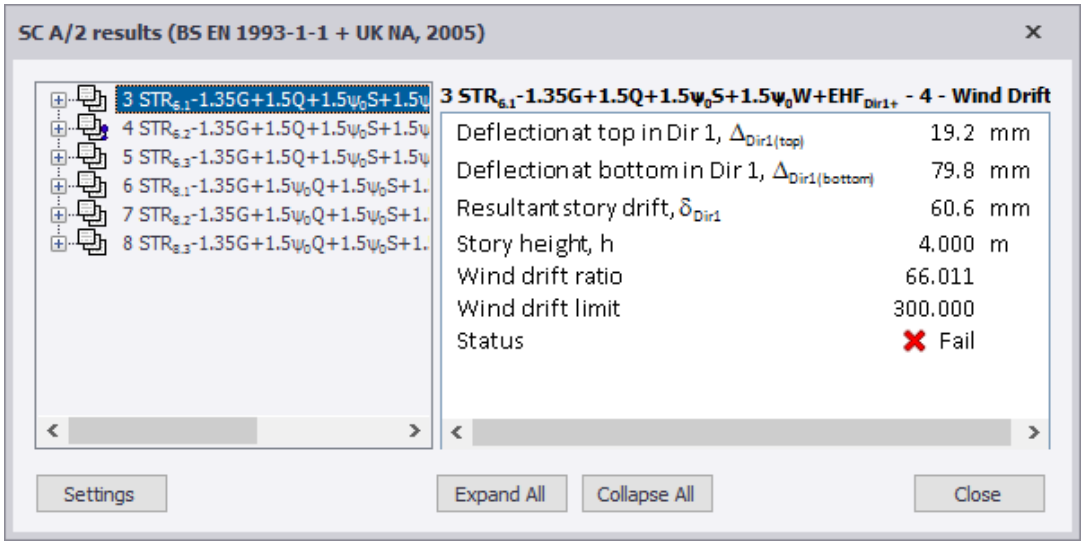
Level	Ref.	Stack	Combination	Deflection Dir 1 [mm]	Deflection Dir 2 [mm]	Drift Dir 1 [mm]	Drift Dir 2 [mm]	Ratio Limit	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2	Details
Top	SC A/2	3	7 $STR_{g,2}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1}	80.3	-0.8	68.5	0.0	300.000	58.398	104534.687	Fail	Pass	Details...
Top	SC B/3	3	8 $STR_{g,3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1}	-0.9	24.7	0.1	13.5	300.000	70942.136	296.342	Pass	Fail	Details...
St. 2 (2)	SC A/1	2	7 $STR_{g,2}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1}	11.8	-0.9	5.3	0.4	300.000	755.091	8906.868	Pass	Pass	Details...
St. 2 (2)	SC C/1	2	8 $STR_{g,3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1}	-0.9	11.3	0.5	5.1	300.000	7741.316	777.898	Pass	Pass	Details...
St. 1 (1)	SC A/1	1	7 $STR_{g,2}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1}	6.5	-0.4	6.5	0.4	300.000	610.699	9869.817	Pass	Pass	Details...
St. 1 (1)	SC C/1	1	8 $STR_{g,3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1}	-0.4	6.1	0.4	6.1	300.000	9910.736	652.584	Pass	Pass	Details...

In this example, at the Level "Top", steel column "SC A/2" is critical for Dir 1 and steel column "SC B/3" is critical for Dir 2. For completeness of the table both Dir 1 and Dir 2 values are shown for both columns.

By unselecting the above **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.

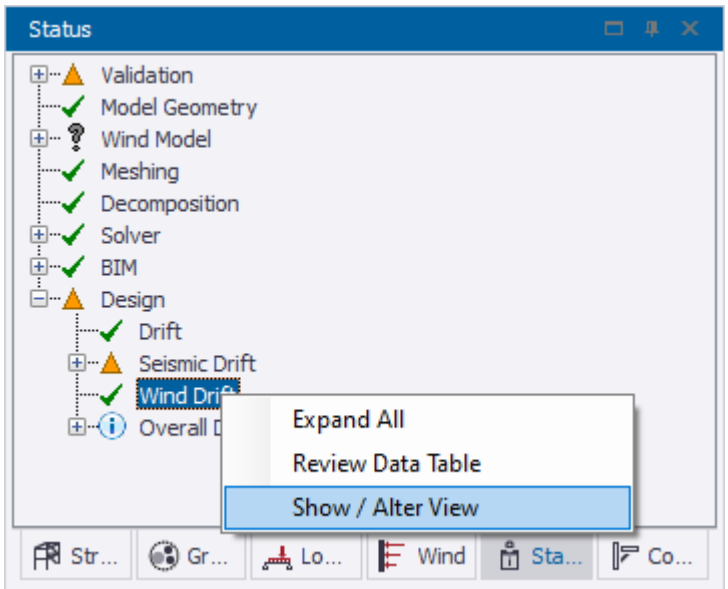


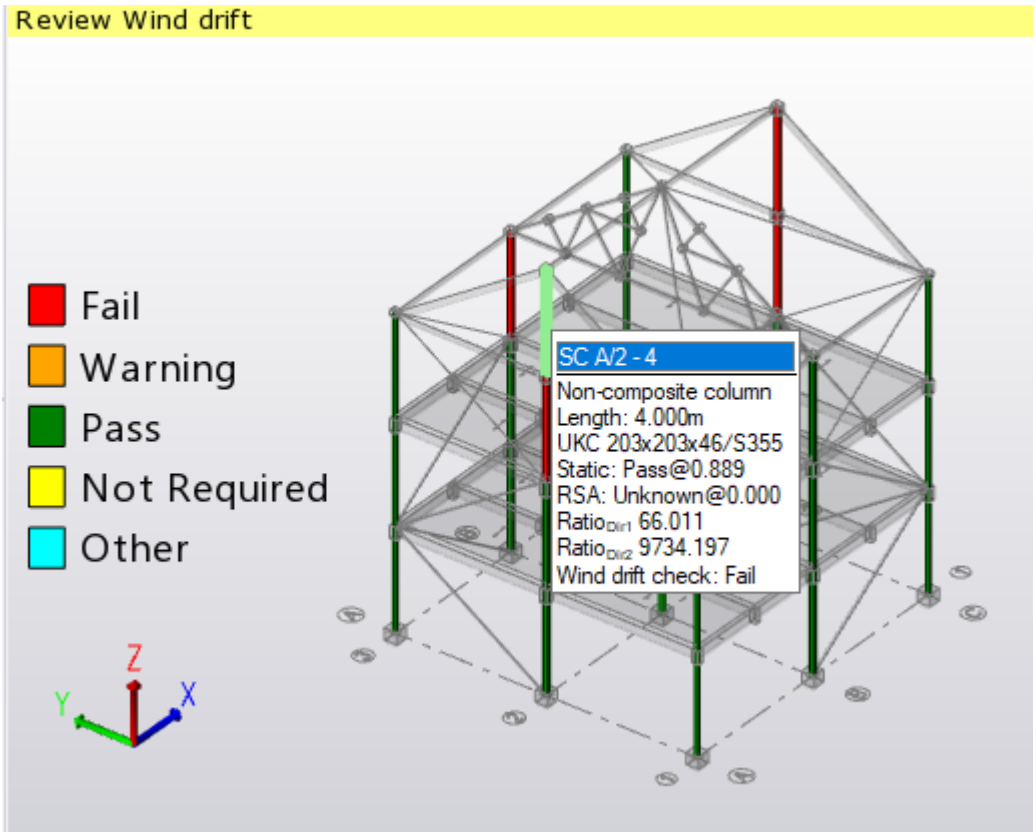
The individual stack by stack checks can be investigated in more detail by clicking the **Details** button at the end of any row.



Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 1010).





This color coded legend makes it easy to see individual failing members. By hovering the cursor over a member, its wind drift results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have options to switch off checks (page 1010), or set a different drift limit (page 1010) for specific stacks if you require.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 845), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 842) where they are less than a specified limit. Stacks check lengths can also be be manually adjusted (page 846) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 845) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The wind drift check results table is available either by creating a Building Analysis & Drift Checks report (page 1078), or by including the Analysis>Wind Drift chapter in a custom report.

NOTE The Wind Drift chapter has a Settings option that can be used to dramatically reduce the number of report pages. By default all columns and walls are output, but you can specify instead the most critical 'XX' columns and walls, (where 'XX' is a number of your choice).

The same Settings option also allows you to choose between reporting only the critical stack/all stacks, and reporting only the critical combination/all combinations.

Overall wind drift check

When activated, by default this check considers all columns (of all materials), and all walls at every level in the model.

In this check the overall wind drift ratio, (H / Δ_{max}) is compared against a user defined limit.

For levels set as floors a **Reporting Level** option is displayed in Level Properties (page 1333). This is used in the above calculation as follows:

H	=	Distance between the highest and lowest levels with **Reporting Level** selected.
Δ_{max}	=	Maximum lateral deflection at the highest level with **Reporting Level** selected.

By default the Δ_{max} is calculated in the resultant wind drift direction only:

$\Delta_{max} = \Delta_{max,Resultant}$

By unselecting in Design Settings (page 843) it is calculated as follows:

$\Delta_{max} = \max [\Delta_{max, Dir1}, \Delta_{max, Dir2}]$

A workflow for running the check is outlined below:

Configure the check and set the limit

The check is activated from **Design Settings**.

On the **Drift Checks** page (if working to the ACI/AISC regional code) or **Sway & Drift Checks** page (if working to any other regional code), select **Check for Overall Wind Drift** and then enter the limit required.

NOTE By default the check is activated for new models with the limit set as 500.

By default the check is performed in the resultant wind drift direction only, but by unselecting **Check for Resultant Wind Drift** it can be performed in each of Building Directions 1 & 2 if required.

By default the check only considers the effects of the wind loadcase(s) in wind combinations, but by unselecting **Check wind cases only** it will consider the effects of all loadcases in wind combinations (which would include drift induced by gravity loads).

Run the check

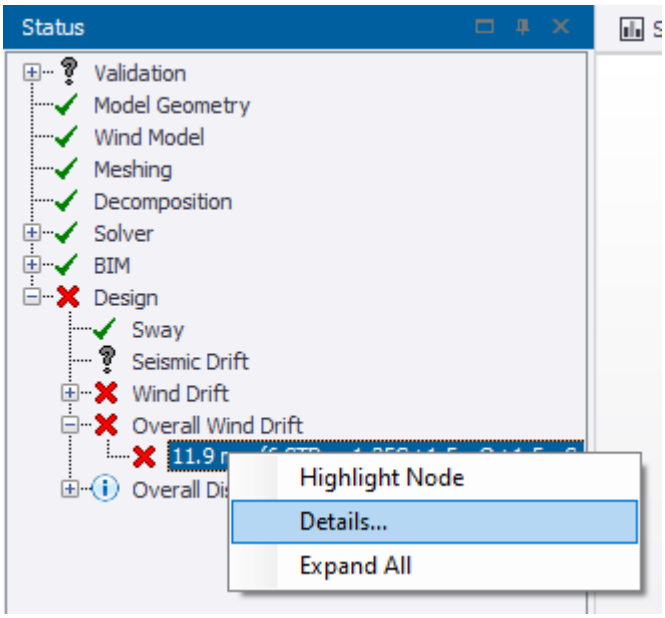
Provided the check has been activated, wind load combinations are automatically checked for overall wind drift during the structure static design, and also when any 3D analysis is run in isolation. If a sub-set of combinations are considered for analysis then only those combinations are checked, (allowing engineers working on larger structures to investigate and optimize the lateral load resisting systems more rapidly).

In performing the checks, the maximum lateral deflection (Δ_{max}) associated with the highest level of active check for drift is determined for each combination using the combination SLS (Service Level) factors - (which can be < 1.0). The ratio of H / Δ_{max} is then compared against a user-defined limit.

Review the check status and details

Once the check has been run, an **Overall Wind Drift** status is reported in the **Design** branch of the Status Tree (page 59). Beneath the status the node with maximum wind drift deflection value and its associated combination are listed.

Right clicking on the value opens a context menu, from which you can choose **Highlight Node** to see where the maximum occurs in a solver view.



You can also choose Details... to open a dialog which shows the results of the check.

Check overall displacement

By expanding the **Design** branch of the Status Tree (page 59), you are able to review the maximum positive and negative overall displacement results from the 3D Analysis for both Strength and Service combinations.

Displacements can also be viewed graphically in the Results View by using the buttons on the Deflections group.

Common tasks for sway and drift checks

Automatically merge short stacks

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of sway/drift, wind drift and seismic drift checks only, you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

A setting is provided in **Design Settings** to automatically do this when stacks are less than a given length.

- With the setting off, the check considers all stacks of all columns/walls apart from those that have been manually excluded.
- With the setting on, any stacks that are less than the limit specified are merged with the stack below, and the check is performed on the merged stack length. The check is not performed on single stack columns/walls that are less than this limit .

To turn the setting on:

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC regional code)
   - **Sway & Drift Checks** (if working to any other regional code)
3. Select **Merge short stacks**.
4. Enter the minimum stack length, (below which the stack will be merged with the next adjacent stack).
5. Click **OK**

Set the wind drift limit

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC regional code)
   - **Sway & Drift Checks** (if working to any other regional code)
3. Enter the **Wind Drift Limit** required.
4. Click **OK**

The limit is applied to all columns and walls, but can be overridden if required for specific members via in the member properties.

Choose resultant or directional wind drift checks

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC regional code)
   - **Sway & Drift Checks** (if working to any other regional code)
3. Select or unselect **Check for Resultant Wind Drift**.
   - Selected - the wind drift limit is checked against the resultant wind drift

- Unselected - two checks are performed, one each for Building Directions 1 & 2.

4. Click **OK**

Consider wind cases only for the wind drift check

A **Check wind cases only** design setting (default on) is provided which applies to the check as follows:

- With the setting on, the Wind Drift check only considers the effects of the wind loadcase(s) in wind combinations.
- With the setting off, the check considers the effects of all loadcases in wind combinations (which would include drift induced by gravity loads).

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC regional code)
   - **Sway & Drift Checks** (if working to any other regional code)
3. Select or unselect **Check wind cases only** as required.
4. Click **OK**

Switch off sway checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the sway check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Sway & Drift Checks** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel
3. Unselect **Sway/Seismic drift checks**

Switch off drift checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the drift check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Drift Checks** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall

- under the required **Stack** or **Panel** - to exclude a specific stack/panel

3. Unselect **Drift/Seismic drift checks and consider for seismic torsion**

Switch off seismic drift checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the seismic drift check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.

2. Expand the appropriate **Sway & Drift Checks*** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel
3. Unselect:
   - **Drift/Seismic drift checks and consider for seismic torsion** (if working to the ACI/AISC regional code)
   - **Sway/Seismic drift checks** (if working to any other head code)

NOTE *If working to the ACI/AISC regional code this heading is shortened to **Drift Checks**

Switch off wind drift checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the wind drift check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.

2. Expand the appropriate **Sway & Drift Checks*** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel
3. Unselect **Wind drift check**

NOTE *If working to the ACI/AISC regional code this heading is shortened to **Drift Checks**

Switch off tabular results for an entire level

The sway/drift, seismic drift and wind drift check results for all column stacks and wall panels at a specific level can be excluded from the tabular review data as follows:

1. Go to the **Project Workspace**.
2. In the **Structure** tree, expand  **Levels** then select the level at which you want the tabular results to be switched off.
3. In the **Properties** window, unselect the **Reporting Level** option.

Override the wind drift limit for selected columns/walls

The wind drift limit specified in Design Settings can be overridden for selected column stacks or wall panels as follows;

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Sway & Drift Checks*** heading:
   - under **All stacks**, **All Panels** - to override the entire column/wall
   - under the required **Stack** or **Panel** - to override a specific stack/panel
3. Select **Override Wind Drift Limit**
4. Enter the required **Wind drift ratio limit**.

NOTE *If working to the ACI/AISC regional code this heading is shortened to **Drift Checks**

Adjust column stack or wall panel check lengths

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of sway/drift, wind drift and seismic drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

To do this manually proceed as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Under the required **Stack** or **Panel** group, unselect the following setting:
   - **Drift Checks > Merge with stack below** (if working to the ACI/AISC head code)

- **Sway & Drift Checks > Merge with stack below** (if working to any other head code)

Considerations for non-linear models with Tension Only bracing

For non-linear models with Tension Only bracing it is essential that the 'X Brace' pattern is used to input the braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

Perform checks

The checks are performed either by running Analyze All (Static) from the Analyze ribbon, or by running any of the Design (Static) commands from the Design ribbon.

Review tabular results in a Data Table

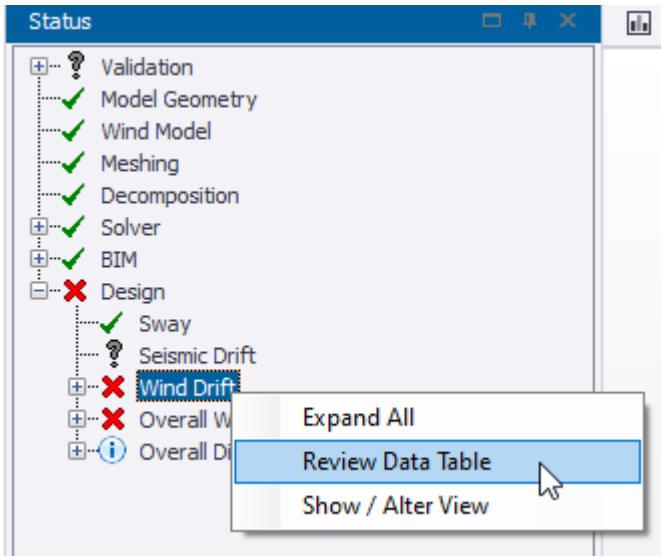
Tabular results can either be reported in summary, or in detail.

- Summary - the column/wall with most critical result from all combinations is reported for each direction at each level
- Detailed - the critical result in each direction at each level is reported for all columns/walls

The results can also be filtered by material type, characteristic and fabrication as required.

To access the tabular results from the Project Workspace Status Tree proceed as follows:

1. Expand the Design branch of the Project Workspace Status Tree.
2. Right-click on the required check.
3. Select **Review Data Table**



For further information, see:

- Review drift check tabular results (page 1019)
- Review sway check tabular results (page 1013)
- Review seismic drift check tabular results (page 1022)
- Review wind drift check tabular results (page 1025)

Review results graphically in a Show / Alter State view

To access the graphical results from the Project Workspace Status Tree proceed as follows:

1. Expand the Design branch of the Project Workspace Status Tree.
2. Right-click on the required check.
3. Select **Show / Alter View**

For further information, see:

- Review and modify sway checks (page 1005)
- Review and modify drift checks (page 981)
- Review and modify seismic drift (page 998)
- Review and modify wind drift checks (page 1010)

Create a report

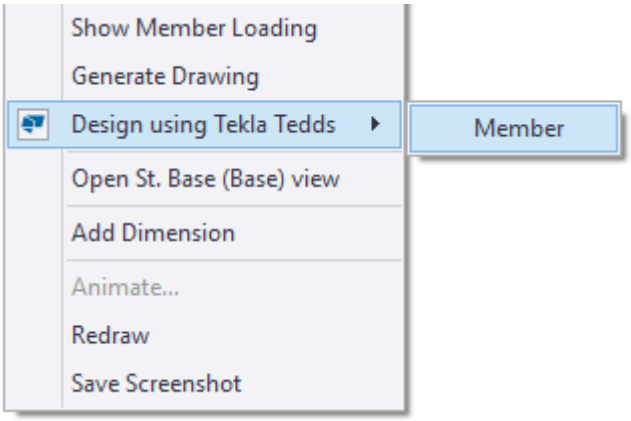
Each of the results tables can be included in printed output by adding the Drift, Sway, Seismic Drift and Wind Drift chapters from the Analysis branch into your model report.

5.6 Design timber members, precast members, and masonry walls using Tekla Tedds

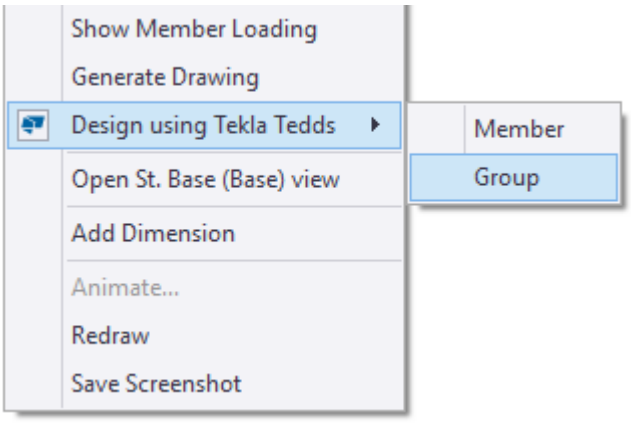
Tekla Structural Designer integrates with **Tekla Tedds** to take advantage of its design capabilities for timber members (to US codes and Eurocodes), precast members (to Eurocodes only), and masonry walls (to US codes only).

Design and check timber members and precast members

Provided an appropriate design code has been selected, once an analysis has been performed each timber or precast member will have a Design using Tekla Tedds (page 851) command available when you right click it in the Project Workspace Structure Tree, or in a Scene view as shown below.

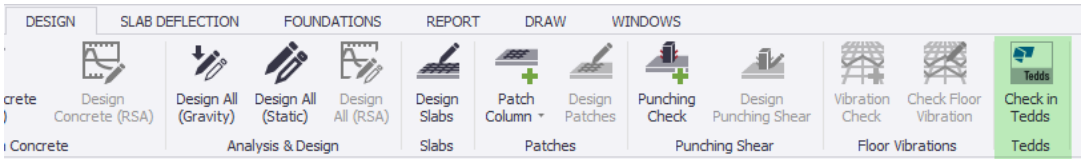


If group design has been activated for the specific member type in Design Settings an additional option is provided to design the group.

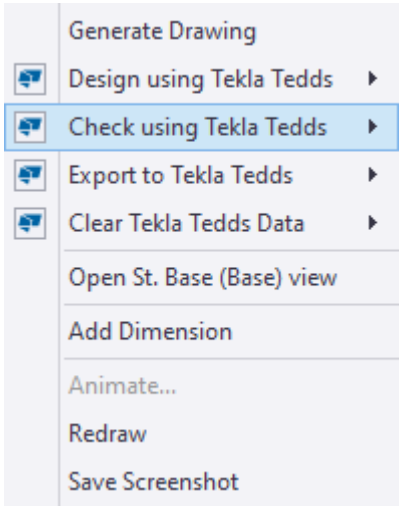


Other design options are available giving you the flexibility to design the whole model, a selection, a specific member type, or a substructure. For details, see: Design using Tekla Tedds (page 851)

Once the design of the member or group has been completed, the design data is attached to member/group and the Check in Tedds (page 860) command becomes available on the **Design** tab



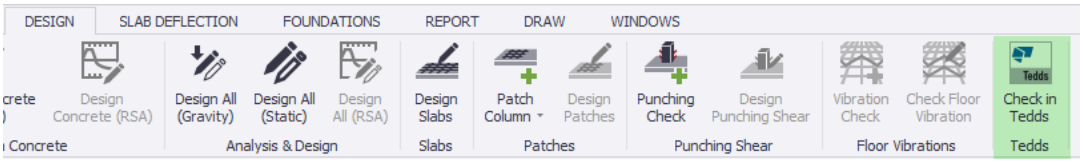
Check options are added to the right click context menus, giving you the flexibility to check the whole model, a selection, a specific member type, or a substructure. For details, see: Check using Tekla Tedds (page 859)



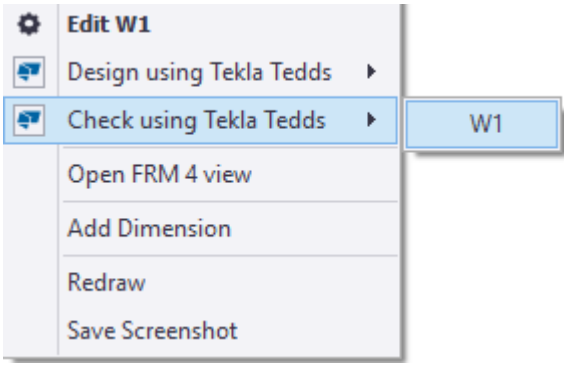
In addition Export to Tekla Tedds (page 1183) and Clear Tekla Tedds Data (page 865) commands are added to the right click context menu.

Design and check masonry walls

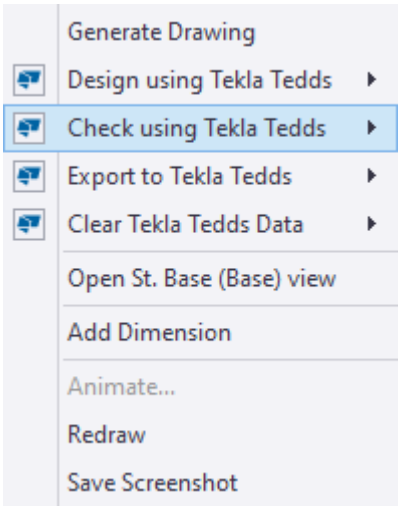
Provided an appropriate design code has been selected and suitable load combinations have been defined (and analyzed), the Check using Tekla Tedds (page 859) command can be run from the **Design** ribbon to rapidly access all of the masonry walls in the model using a default basic set of Tedds data.



Other check and design options are also available, giving you the flexibility to check or design an individual wall, a selection, or a substructure. For details, see: Check using Tekla Tedds (page 859) and Design using Tekla Tedds (page 851)



Once a wall has been checked or designed the Tedds data is attached to wall and additional options to Export to Tekla Tedds (page 1183) or Clear Tekla Tedds Data (page 865) are added to the right click menu.



Design using Tekla Tedds

The following options for designing in Tekla Tedds are available:

- Design using Tekla Tedds> Model
- Design using Tekla Tedds> Member
- Design using Tekla Tedds> <*Masonry wall name*>
- Design using Tekla Tedds> Group
- Design using Tekla Tedds> Selection
- Design using Tekla Tedds> Substructure
- Design using Tekla Tedds> <*Substructure name*>

Where to find each design option

Design using Tekla Tedds> Model

- In the Groups Tree, right-click on Groups, Design, or Detailing

Design using Tekla Tedds> Member

- right-click on a *highlighted* precast or timber member,

Design using Tekla Tedds> Group

- right-click on a *highlighted* precast or timber member for which group design is active (provided that there is more than one member in the group to which the highlighted member belongs)

Design using Tekla Tedds> Selection

- right-click on a selection of one or more precast or timber members or masonry walls.

Design using Tekla Tedds> Substructure

- right-click on a specific timber member type, precast member type , or masonry wall in the Structure Tree

Design using Tekla Tedds> <*Substructure name*>

- right-click on a sub structure containing precast or timber members, or masonry walls

Understanding each of the design options

Command	Description
Design using Tekla Tedds > Model	Launches Tekla Tedds and performs interactive designs on all timber and precast members in the model taking into account group design settings.

Command	Description
Design using Tekla Tedds > Member	• *If group design is active* - Launches Tekla Tedds and performs an interactive design on the selected member. Member design uses the critical forces in the selected member only. All other members remaining in the associated design group will be check designed for the assigned Tekla Tedds properties and their individual forces. (It should be noted that some members may fail if the critical member was not initially selected for the design process.) Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model. • *If group design is inactive* - Launches Tekla Tedds and performs an interactive design on the selected member. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.
Design using Tekla Tedds > *<Masonry wall name>*	Launches Tekla Tedds and performs an interactive design on the selected masonry wall.
Design using Tekla Tedds > Group	Launches Tekla Tedds and performs an interactive design on the selected group. Group design uses the *group* critical forces. This is followed by a check design on each group member for the assigned Tekla Tedds properties and their individual forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.
Design using Tekla Tedds > Selection	Launches Tekla Tedds and: • performs an interactive design for each masonry wall in the selection • *If group design is active* - performs an interactive design for each timber or precast member group in the selection, using *group* critical forces. All members in the

Command	Description
	associated design group (including those that are not in the selection will be check designed for the assigned Tekla Tedds properties and their individual forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model both for selected members and members in groups associated with the selection. • *If group design is inactive* - performs an interactive design for each timber or precast member in the selection using the member's individual critical forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.
Design using Tekla Tedds > Substructure	This option is available in the Structure Tree to enable design of all timber elements of a particular type, all precast elements of a particular type, or all masonry walls.
Design using Tekla Tedds > <*Substructure name*>	This extra option becomes available when working in a sub structure. Designs all timber elements, precast elements and masonry walls in the substructure. • *If group design is active* - Launches Tekla Tedds and performs an interactive design for each group in the sub structure, using *group* critical forces. All members in the associated design group (including those that are not in the sub structure will be check designed for the assigned Tekla Tedds properties and their individual forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model both for sub structure members and members in the same groups but not in the sub structure.

Command	Description
	• *If group design is inactive* - Launches Tekla Tedds and performs an interactive design for each member in the sub structure using the member's individual critical forces. Section changes made within Tekla Tedds are imported back into the Tekla Structural Designer model.

To design all timber and precast members

1. In the **Project Workspace**, click **Group** tab.
2. In the tree, right-click **Groups**.
3. In the context menu, select **Design using Tekla Tedds > Model**

   Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design section (s1) in the first member to be designed for forces at that section.

   NOTE To find out more about this and the other **Design using Tekla Tedds** commands, see **Understanding each of the design options** above.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
   - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
   - If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
   - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
5. If the member is in a group and group design is active, all other members in the design group are then check designed for the assigned Tekla Tedds properties and their individual forces.
6. Tekla Structural Designer opens the Tekla Tedds design calculation to allow the next member to be designed.
   - Continue in the same way until all the members have been designed.

At the end of the process the status of the member(s) is shown in a Review View.

NOTE **Design using Tekla Tedds > Model** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design a timber or precast member

1. Hover the mouse pointer over the member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right click and select **Design using Tekla Tedds > Member**

   Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design section (s1) in the member to be designed for forces at that section.

   NOTE To find out more about this and the other **Design using Tekla Tedds** commands, see **Understanding each of the design options** above.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
   - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
   - If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
   - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
5. If the member is in a group and group design is active, all other members in the design group are then check designed for the assigned Tekla Tedds properties and their individual forces.

At the end of the process the status of the member(s) is shown in a Review View.

NOTE You can also run **Design using Tekla Tedds > Member** by right-clicking the member reference in the Structure Tree.

NOTE **Design using Tekla Tedds > Member** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design a masonry wall

1. Hover the mouse pointer over the wall to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right click and select **Design using Tekla Tedds > <*masonry wall name*>**

   Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design case (DC1) in the wall to be designed for forces in that case.

   NOTE To find out more about this and the other **Design using Tekla Tedds** commands, see **Understanding each of the design options** above.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design case (DC2) from the Design section droplist.
   - Continue in the same way, adjusting parameters as necessary for each design case until a satisfactory design is achieved then click **Finish**.

At the end of the process the status of the wall(s) is shown in a Review View.

NOTE **Design using Tekla Tedds > <*masonry wall name*>** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design a timber or precast member group

In grouped design, one member of the group is designed for the worst case forces from all members in the group. The design is then copied to the other group members and each member is checked. The status for each member is then displayed in a Review View.

1. Hover the mouse pointer over a group member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. From the right-click menu, choose **Design using Tekla Tedds > Group** Tekla Structural Designer collects the worst case forces from all members in the group and then opens the Tekla Tedds precast design calculation to allow the first design section (s1) in the selected member to be designed for forces at that section.

   NOTE To find out more about this and the other **Design using Tekla Tedds** commands, see **Understanding each of the design options** above.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
    - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
5. If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
    - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
6. After the selected group member has been designed, a check is then automatically performed on the remaining group members, resulting in an individual status and utilisation for each member in the group. The status of the group members is displayed in a Review View.

To design selected timber or precast members or masonry walls

1. In the active view make your selection.
2. Right-click, then in the context menu, select **Design using Tekla Tedds > Selection**
    - Tekla Structural Designer opens the Tekla Tedds design calculation to allow the first design section (s1) in the first member/wall of the selection to be designed for forces at that section.

    NOTE To find out more about this and the other **Design using Tekla Tedds** commands, see **Understanding each of the design options** above.

3. Adjust the parameters as required to achieve a satisfactory design, then select the next design section (s2) from the Design section droplist.
    - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
4. If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
    - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member/wall has been designed.
5. Tekla Structural Designer re-opens the Tekla Tedds design calculation to allow the next selected member/wall to be designed.
6. Continue in the same way until all members/walls in the selection have been designed.
7. The status of the members is shown in a Review View.

NOTE The **Design using Tekla Tedds > Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

To design timber or precast members or masonry walls in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Design using Tekla Tedds > <*Sub structure name*>**

   NOTE To find out more about this and the other **Design using Tekla Tedds** commands, see **Understanding each of the design options** above.

4. Adjust the parameters as required to achieve a satisfactory design, then select the next design section/case from the droplist.
   - Continue in the same way, adjusting parameters as necessary for each design section until a satisfactory design is achieved for the whole span/stack, then click **Finish**.
5. If it is a multi-span/stack member, the Tekla Tedds design calculation will re-open to allow the next span/stack to be designed.
   - Continue the design in the same way, clicking **Finish** after each span/stack, until the entire member has been designed.
6. Tekla Structural Designer re-opens the Tekla Tedds design calculation to allow the next member or wall in the sub structure to be designed.
7. Continue in the same way until all members/walls in the sub structure have been designed.
8. The status of the members is shown in a Review View.

NOTE The **Design using Tekla Tedds > <*Sub structure name*>** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Check using Tekla Tedds (page 859)

Check using Tekla Tedds

The following options for checking in Tekla Tedds are available:

- Check using Tekla Tedds> Model
- Check using Tekla Tedds> Member
- Check using Tekla Tedds> Group
- Check using Tekla Tedds> Selection
- Check using Tekla Tedds> Substructure
- Check using Tekla Tedds> <*Substructure name*>

NOTE For timber members and precast members, each of the above **Check in Tedds** options only become active after Design using Tekla Tedds (page 851) has been performed.

Where to find each check option

Check using Tekla Tedds> Model

- in the Groups Tree, right-click on Groups, Design, or Detailing

Check using Tekla Tedds> Member

- right-click on a *highlighted* precast or timber member which has already been designed

Check using Tekla Tedds> Group

- right-click on a *highlighted* precast or timber member for which group design is active (provided that there is more than one member in the group to which the highlighted member belongs , and the group has already been designed)

Check using Tekla Tedds> Selection

- right-click on a selection of one or more precast or timber members or masonry walls.

Check using Tekla Tedds> Substructure

- in the Structure Tree, right-click on a specific timber member type or precast member type which have already been designed, or masonry wall

Check using Tekla Tedds> <*Substructure name*>

- right-click on a sub structure containing precast or timber members which have already been designed, or masonry walls

Understanding each of the check options

Command	Description
Check in Tedds (from the **Design** tab), or **Check using Tekla Tedds > Model**	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks all masonry walls and all those members in the model where a linked Tekla Tedds calculation has been created. • For masonry walls this command runs the necessary Tedds calculations in the background using the latest analysis results and a default basic set of Tedds data • For timber and precast members this command can be used after any the members have been initially designed. It is intended for use where a general re-check for all objects is required following changes to the model which require a reanalysis process (e.g. changing the loading applied onto the model). This command should ideally be used at the end of the design cycle and ensures that the previously designed timber, or precast concrete elements still meet the design criteria of the model.
Check using Tekla Tedds > Member	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the member with the pre-saved design data. No other members in the associated Design Group will be checked.
Check using Tekla Tedds > <*Masonry wall name*>	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the wall with the pre-saved design data.
Check using Tekla Tedds > Group	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks each member in the selected group with the pre-saved design data.

Command	Description
Check using Tekla Tedds > Selection	Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the selected members/walls with the pre-saved design data.
Check using Tekla Tedds > Substructure	This option is available in the Structure Tree to enable checking of all timber elements of a particular type, all precast elements of a particular type, or all masonry walls.
Check using Tekla Tedds > <*Substructure name*>	This extra option becomes available when working in a sub structure - Launches Tekla Tedds within the background (the user is not exposed to the interface) and checks the selected sub structure members with the pre-saved design data.

To check all timber or precast members

NOTE This command is only available for those members that have already been designed.

1. On the **Design** tab, click  **Check in Tedds**.
   - Tekla Structural Designer checks all the existing Tekla Tedds design calculations in the background for the current set of forces.
   - The updated status of the members is shown in a Review View.

NOTE To find out more about this and the other **Check using Tekla Tedds** commands, see **Understanding each of the check options** above.

NOTE This command is also available in the **Project Workspace** by selecting **Check using Tekla Tedds > Model** from the **Group** branch of the **Group** tab.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check a timber or precast member

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. Hover the mouse pointer over the member to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right click and select **Check using Tekla Tedds > Member** Tekla Structural Designer checks the existing Tekla Tedds precast design calculation in the background for the current set of forces.

   At the end of the process the status of the member(s) is shown in a Review View.

NOTE To find out more about this and the other **Check using Tekla Tedds** commands, see **Understanding each of the check options** above.

NOTE You can also run **Check using Tekla Tedds > Member** by right-clicking the member reference in the Structure Tree.

NOTE **Design using Tekla Tedds > Member** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check a masonry wall

1. Hover the mouse pointer over the wall to *highlight* it in a 2D or 3D view.
2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. Right click and select **Check using Tekla Tedds > *<masonry wall name>*** Tekla Structural Designer checks the existing Tekla Tedds masonry wall design calculation in the background for the current set of forces.

NOTE **Check using Tekla Tedds > *<masonry wall name>*** uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check a timber or precast member group

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. Hover the mouse pointer over a group member to *highlight* it in a 2D or 3D view.

2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip.
3. In the context menu, select **Check using Tekla Tedds > Group**
   - Tekla Structural Designer checks the existing Tekla Tedds design calculations in the background for the current set of forces.
   - The updated status of the group members is shown in a Review View.

NOTE To find out more about this and the other **Check using Tekla Tedds** commands, see **Understanding each of the check options** above.

To check selected timber or precast members or masonry walls

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. In the active view, drag a box to make your selection.
2. Right-click, then in the context menu, select **Check using Tekla Tedds > Selection**
   - Tekla Structural Designer checks the existing Tekla Tedds design calculations in the background for the current set of forces.
   - The updated status of the members is shown in a Review View.

     NOTE To find out more about this and the other **Check using Tekla Tedds** commands, see **Understanding each of the check options** above.

NOTE The **Check using Tekla Tedds > Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

To check timber or precast members or masonry walls in a sub structure

NOTE **Check using Tekla Tedds** is only available for those members that have already been designed.

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Check using Tekla Tedds > <*Sub structure name*>**
   - Tekla Structural Designer checks the existing Tekla Tedds design calculations in the background for the current set of forces.

- The updated status of the sub structure members/walls is shown in a Review View.

NOTE To find out more about this and the other **Check using Tekla Tedds** commands, see **Understanding each of the check options** above.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also

Design using Tekla Tedds (page 851)

Clear Tekla Tedds Data

Command	Description
Clear Tekla Tedds Data > Model	Clears all existing data from all the associated Tekla Tedds calculations in the model.
Clear Tekla Tedds Data > Member	Clears all existing data from the associated Tekla Tedds calculation for the highlighted member.
Clear Tekla Tedds Data > Group	Clears all existing data from the associated Tekla Tedds calculation for the group to which the highlighted member belongs.
Clear Tekla Tedds Data > Selection	Clears all existing data from the associated Tekla Tedds calculation for the selected members.
Clear Tekla Tedds Data > <*Substructure name*>	This extra option becomes available when working in a sub structure - Clears all existing data from the associated Tekla Tedds calculation for the selected sub structure members.

NOTE If you change the Design Settings or section shape, you would need to clear the Tekla Tedds data from existing members in order for the new settings to apply when the members are redesigned.

When a model is reanalyzed, there is no need clear the Tekla Tedds data to get the analysis results to update in the Tedds calculation, (the latest analysis results are always used).

5.7 Design settings

Click **Design** --> **Settings** to edit the following design settings specific to the current model.

After they have been edited, click Save, Load, Cancel, or OK (page 1263) according to your requirements.

Click **Home** --> **Settings** --> **Design Settings** to edit the Global settings (page 1232).

- Analysis (page 866)
- General (page 874)
- Steel > General (Eurocode only) (page 875)
- Steel > Composite Beams (page 876)
- Steel > Steel Joists (page 877)
- Concrete (Cast-in-place) (page 878)
- Concrete (Precast - Eurocode only)
- Design Forces (page 910)
- Design Groups (page 926)
- Autodesign (page 927)
- Design Warnings (AISC/ASC only) (page 932)
- Design Checks (IS only) (page 936)
- Sway and Drift Checks (page 938)
- Fire checks (Eurocode only) (page 939)
- Design Settings > Timber (page 940)

Design Settings > Analysis

The settings displayed on this page are dependent on the regional code being worked to.

United States (ACI/AISC)

Analysis	
First-order/second-order analysis radio buttons	For 3D building analysis and staged construction analysis these allow you to select if a first or second order analysis is performed. For the US region the options presented are:

Analysis	
	• First-order analysis • Second-order analysis **TIP** For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design.

NOTE In addition to the 3D building analysis results from the method selected above, FE and grillage chasedown results are also generally considered, with the following exceptions:

- **Steel members** - FE and grillage chasedown results are *never* considered.
- **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).

Solve lateral loadcases in isolation if non-linear analysis is required during design	Selecting this option allows you to try to solver lateral loadcases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral loadcases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral loadcases, you can either run the analysis directly on the **Analyze** ribbon, or select this option if you know that the analysis does not fail.

Stability coefficient tolerance	
Stack height / ratio	This ratio allows you to adjust the value of deflection that can safely be ignored. Selecting the option prevents any small deflections from causing erroneously high stability coefficients in stability coefficient calculations.

NOTE If the second order drift is less than the tolerance specified in **Stack height /**, the stability coefficient value is returned as N/A with a note that states that the drift is small enough to be ignored.

Reduced Stiffness	
Reduced stiffness factor	This factor is only exposed once the analysis is set to second-order. It is applied to the stiffness (EI and EA) of all steel members in addition to any settings made on Analysis Options, Modification Factors. For correct design to the AISC Specification using the DAM, it should be set to 0.8.

NOTE As an alternative to setting the analysis to first-order to explore the reason for any second order analysis failure, it is possible to alter this factor. If you set it to a value of say 10, this will stiffen both the Modulus of elasticity (E) and the shear modulus of elasticity (G) by a factor of 10 in the second order analysis. Although the results will not be able to be used for a valid design, it should now be possible to run the analysis to see which member might fail a design and hence be the cause of the analysis instability. This factor can then be reduced towards 0.8 for further investigation.

Staged Construction Analysis	
Include staged construction analysis	Select this option to include a staged construction analysis in the analysis and design process. For details of the implications this can have on the "envelope" of forces used for design, see:
3D building analysis **FE chasedown** **Grillage chasedown**	These options only become available if **Include staged construction analysis** has been selected. Select the results that should continue to be considered for member design alongside the staged construction results.

NOTE When staged construction analysis is selected the following exceptions apply when selecting the other results to be considered for member design:

- **Steel members** - Only 3D building analysis results are considered, (staged construction, FE and grillage chasedown results are *not* considered.)
- **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).

All Eurocode regions

Analysis	
First-order/second-order analysis radio buttons	For 3D building analysis and staged construction analysis these allow you to select if a first or second order analysis is performed. The options presented for the Eurocode regions are: • First-order (Elastic) analysis • Second-order (P-Δ) analysis - BS EN 1993-1-1 Cl 5.2.2 (6)B - Amp. forces method • Second-order (P-Δ) analysis **TIP** For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design.
NOTE In addition to the 3D building analysis results from the method selected above, FE and grillage chasedown results are also generally considered, with the following exceptions: • **Steel members** - FE and grillage chasedown results are *never* considered. • **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).	
Solve lateral loadcases in isolation if non-linear analysis is required during design	Selecting this option allows you to try to solver lateral loadcases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral loadcases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral loadcases, you can either run the analysis directly on the **Analyze** ribbon, or select this option if you know that the analysis does not fail.

Staged Construction Analysis	
Include staged construction analysis	Select this option to include a staged construction analysis in the analysis and design process. For details of the implications this can have on the "envelope" of forces used for design, see:
3D building analysis **FE chasedown** **Grillage chasedown**	These options only become available if **Include staged construction analysis** has been selected. Select the results that should continue to be considered for member design alongside the staged construction results.

NOTE When staged construction analysis is selected, if the FE and/or grillage chasedown results are also considered, the following exceptions apply:

- **Steel members** - FE and grillage chasedown results are *never* considered.
- **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).

All British Standards regions

Analysis	
First-order/second-order analysis radio buttons	For 3D building analysis and staged construction analysis these allow you to select if a first or second order analysis is performed. The options that are presented for the British Standards regions are: • First-order (Elastic) analysis • Second-order (P-Δ) analysis - BS 5950-1:2000 k_{amp} approach • Second-order (P-Δ) analysis **TIP** For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design.

Analysis	
NOTE In addition to the 3D building analysis results from the method selected above, FE and grillage chasedown results are also generally considered, with the following exceptions: • **Steel members** - FE and grillage chasedown results are *never* considered. • **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).	
Solve lateral loadcases in isolation if non-linear analysis is required during design	Selecting this option allows you to try to solver lateral loadcases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral loadcases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral loadcases, you can either run the analysis directly on the **Analyze** ribbon, or select this option if you know that the analysis does not fail.

Staged Construction Analysis	
Include staged construction analysis	Select this option to include a staged construction analysis in the analysis and design process. For details of the implications this can have on the "envelope" of forces used for design, see:
3D building analysis **FE chasedown** **Grillage chasedown**	These options only become available if **Include staged construction analysis** has been selected. Select the results that should continue to be considered for member design alongside the staged construction results.
NOTE When staged construction analysis is selected the following exceptions apply when selecting the other results to be considered for member design: • **Steel members** - Only 3D building analysis results are considered, (staged construction, FE and grillage chasedown results are *not* considered.) • **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).	

India

Analysis	
First-order/second-order analysis radio buttons	For 3D building analysis and staged construction analysis these allow you to select if a first or second order analysis is performed. The options that are presented for the India are: • First-order (Elastic) analysis • Second-order (P-Δ) analysis **TIP** For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design.
NOTE In addition to the 3D building analysis results from the method selected above, FE and grillage chasedown results are also generally considered, with the following exceptions: • **Steel members** - FE and grillage chasedown results are *never* considered. • **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).	
Solve lateral loadcases in isolation if non-linear analysis is required during design	Selecting this option allows you to try to solver lateral loadcases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral loadcases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral loadcases, you can either run the analysis directly on the **Analyze** ribbon, or select this option if you know that the analysis does not fail.

Staged Construction Analysis	
Include staged construction analysis	Select this option to include a staged construction analysis in the analysis and design process. For details of the implications this can have on the "envelope" of forces used for design, see:

Staged Construction Analysis	
3D building analysis **FE chasedown** **Grillage chasedown**	These options only become available if **Include staged construction analysis** has been selected. Select the results that should continue to be considered for member design alongside the staged construction results.

NOTE When staged construction analysis is selected the following exceptions apply when selecting the other results to be considered for member design:

- **Steel members** - Only 3D building analysis results are considered, (staged construction, FE and grillage chasedown results are *not* considered.)
- **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).

Australia

Analysis	
First-order/second-order analysis radio buttons	For 3D building analysis and staged construction analysis these allow you to select if a first or second order analysis is performed. The options that are presented for the Australia are: • First-order (Elastic) analysis • Second-order (P-Δ) analysis - AS 4100-1998 sway amplification approach • Second-order (P-Δ) analysis **TIP** For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design.

NOTE In addition to the 3D building analysis results from the method selected above, FE and grillage chasedown results are also generally considered, with the following exceptions:

- **Steel members** - FE and grillage chasedown results are *never* considered.

Analysis	
• **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).	
Solve lateral loadcases in isolation if non-linear analysis is required during design	Selecting this option allows you to try to solver lateral loadcases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral loadcases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral loadcases, you can either run the analysis directly on the **Analyze** ribbon, or select this option if you know that the analysis does not fail.

Staged Construction Analysis	
Include staged construction analysis	Select this option to include a staged construction analysis in the analysis and design process. For details of the implications this can have on the "envelope" of forces used for design, see:
3D building analysis **FE chasedown** **Grillage chasedown**	These options only become available if **Include staged construction analysis** has been selected. Select the results that should continue to be considered for member design alongside the staged construction results.
NOTE When staged construction analysis is selected the following exceptions apply when selecting the other results to be considered for member design: • **Steel members** - Only 3D building analysis results are considered, (staged construction, FE and grillage chasedown results are *not* considered.) • **Concrete members** - FE chasedown results can *optionally be not* considered (by unselecting the FE Chasedown option in Design Settings > Concrete > *(member type)* > General Parameters).	

Design Settings > General

Setting	Description
Fix column nodes horizontally	Allows you to automatically apply lateral translational fixed supports to column nodes for the 3D building analysis performed during the gravity design processes. The option is useful at the early design stage when lateral systems have not been established. The automatic lateral supports can remove unwanted lateral displacements that may prevent the analysis from completing and produce unrealistic forces.

NOTE The option does not apply to:

- The column nodes that are in a rigid floor diaphragm
- An analysis run as a separate process
- The **Analyze All (Static)** command
- The 3D building analysis performed during any **Design (Static)** command

Design Settings > Steel > General

Partial safety factors (Eurocode and relevant National Annexes only)	
Use user defined partial safety factors	When **Use user defined partial safety factors** is selected, the options to define partial safety factors are enabled. Otherwise, the corresponding country defaults will be used. **NOTE** Default for this setting is unselected.
Resistance of cross-sections, γ_{M0}	Specify a user-defined value for this partial safety factor as required.
Resistance of members to instability, γ_{M1}	Specify a user-defined value for this partial safety factor as required.

Partial safety factors (Eurocode and relevant National Annexes only)	
Resistance of cross-sections to tension, γ_{M2}(EC3 - Part 1-1)	Specify a user-defined value for this partial safety factor as required.
Resistance of joints, γ_{M2}(EC3 - Part 1-8)	Specify a user-defined value for this partial safety factor as required.

FE Chasedown	
Design Steel Beams **Design Composite Beams** **Design Steel Columns** **Design Steel Braces**	If FE chasedown results exist they are considered in the design of the selected member types.

Grillage Chasedown	
Design Steel Beams **Design Composite Beams** **Design Steel Columns** **Design Steel Braces**	If Grillage chasedown results exist they are considered in the design of the selected member types.

Design Settings > Steel > Composite Beams

Setting	Description
Tolerance of rectilinearity	The calculation of the effective width of composite beams is only performed if they lie within the tolerance on rectilinearity adjusted here. The default tolerance is 15 degrees. At greater angles, you will be prompted to type the effective width manually.
Update effective width prior to design check	Allows you to determine the effective width each time auto or check design is run. **NOTE** Unless this box is selected, you will be unable to manually override the effective width on the Floor construction and Natural

Setting	Description
	frequency pages of the beam properties.

Design Settings > Steel > Steel Joists

NOTE Steel joists are only applicable in the US.

Setting	Description
Uniform load tolerance	If the calculated percentage load tolerance for steel joists is less than this value, the load is classified as Uniform.
Equivalent load tolerance	If the calculated percentage load tolerance for steel joists is greater than this value, the load is classified as Non-uniform. **NOTE** If 0% equivalent load tolerance is specified, only pure uniform loading is accepted.
Maximum sum of concentrated loads	Allows you to specify a user-defined maximum sum of concentrated loads that limits K, LH and DLH joists. If any concentrated loads exceed the specified maximum sum, the relevant load combination is classified as Non-uniform.
Maximum eccentricity of zero shear	In the equivalent uniform load method, the position of the point of zero shear relative to the center span point of the joist is determined. The option allows you to specify the maximum eccentricity of zero shear value in the equivalent uniform load method. If the position is located outside the value, the load is classified as Non-uniform.

Setting	Description
Deflection increase to allow for shear	Deflections due to live load are calculated using the following effective inertia value: $I_{effective} = I_{gross} / (1 + R)$, where R = the deflection increase to allow for shear effects (defaulted to 15%) and applied to KCS only. The option allows you to adjust the value of R, (the deflection increase to allow for shear, percentage).

Design Settings > Concrete (Cast-in-place)

General

Setting	Description
Limitation on concrete cylinder strength for shear/torsion design (Eurocode only)	
Normal weight concrete limit **Lightweight concrete limit**	For shear/torsion design the cylinder strength and its use in all derived mechanical properties is limited (but 'overridable') for specific national annexes. Default national annex values of these limits for normal weight/ lightweight respectively: • UK, Eire, Singapore, Malaysia: 50/50 • Norway: 65/55 • EU, Finland, Sweden: 90/90 (no limit stated) **NOTE** Changing the regional code/ settings set does NOT update these values automatically.

Setting	Description
Reinforcement anchorage length parameters (used in the calculation of the ultimate bond stress, from which anchorage lengths are determined)	
Plain Bars Bond Quality Modifier	Default value 0.5.
Deformed Bars Bond Quality Modifier (ACI only)	Default value 1.0.
Type-1 Bars Bond Quality Modifier (Eurocode only)	Default value 0.8.
Type-2 Bars Bond Quality Modifier (Eurocode only)	Default value 1.0.

Beam

Reinforcement Settings

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Longitudinal bars	Allows you to adjust the following settings applied to longitudinal bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the longitudinal bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the longitudinal bar design process. • **Minimum side bar size**: Sets the minimum allowable side bar size that can be used in the longitudinal bar design process. • **Minimum top steel clear spacing**: Sets the minimum allowable top steel clear spacing that can be used in the longitudinal bar design process. • **Minimum bottom steel clear spacing**: Sets the minimum

Setting	Description
	allowable bottom steel clear spacing that can be used in the longitudinal bar design process. • **Maximum tension steel spacing**: Sets the maximum allowable tension steel spacing that can be used in the longitudinal bar design process. • **Maximum compression steel spacing**: Sets the maximum allowable compression steel spacing that can be used in the longitudinal bar design process. • **Use single bars when beam width <=**: Single bars are only permitted in beams whose width is less than the width specified. • **Steel overstrength factor**: Sets the value that accounts for the reinforcement steel yielding overstrength in seismic design. Only applies to the ACI regional code.
Short span maximum length	Spans smaller than the value set here are treated as short spans. Support bars of short spans are merged with the span bars.

Detailing Settings

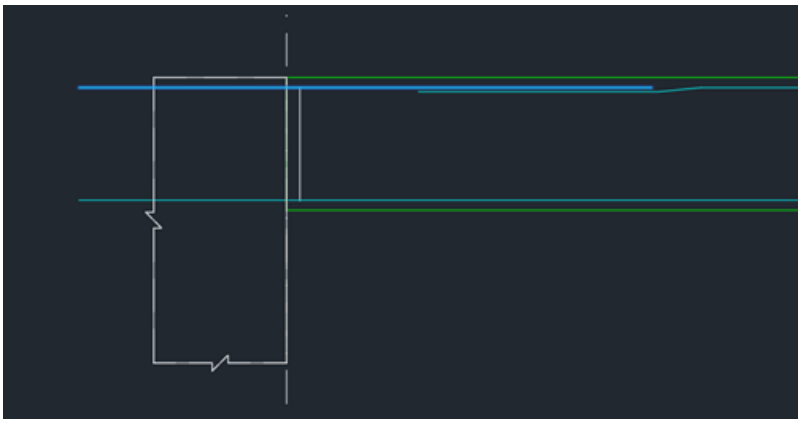
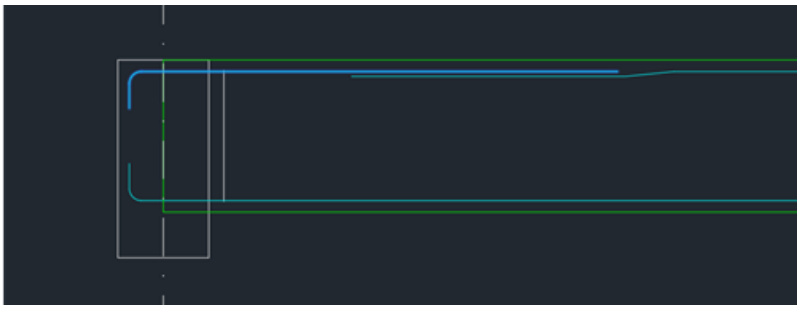
Longitudinal Bars	
Use same size bars in multilayer arrangements	Allows you to use the same bar size in each layer.
Use same number of bars in each layer	Allows you to use the same number of bars in each layer.

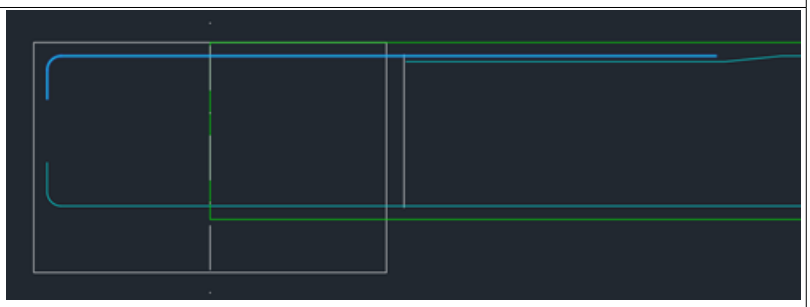
Cut Length	
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.

Longitudinal Bar Curtailment	
Merge identical longitudinal bars if appropriate	Allows you to merge bars of the same size, number, and position, provided that the total length of the merged

Longitudinal Bar Curtailment	
	bar does not exceed the max allowable bar length. **TIP** The max bar length can be verified on the **Reinforcement** page of the **Materials** dialog box. To do so, click an available bar size and then, click **View...** **NOTE** Top bar patterns 1 and 2 do not have any bars that you could merge.
Extend top longitudinal support bars symmetrically	Extends the support bars symmetrically to both spans based on the larger effective span length; but only if the spans vary by less than the percentage specified.
Extend top longitudinal support bars by anchorage length	Adds anchorage lengths to the calculated extension lengths.
Min anchorage length {value} x dia	Allows you to control the minimum anchorage length as a multiple of bar diameter. **NOTE** The option is only available for the Eurocode regional code.
Min anchorage length of plain bars {value} x dia	Allows you to control the minimum anchorage length as a multiple of bar diameter when the rib type is plain. **NOTE** The option is only available for the ACI regional code.

End Support Curtailment	
Extend top span bars to end support	Extends the top span bars of the first and last spans to the end supports.
Use 'U' bars at end support if appropriate	Allows you to replace the top and bottom bars at the end support region with 'U' bars under certain conditions. The bars that are joined or merged to create the 'U' bars

End Support Curtailment	
	depend on the top and bottom patterns selected for the beam. The anchorage lengths for the resulting 'U' bars are taken as the lengths required for the pair of bars that made the 'U' bar.
Extend and bob all bars for supporting member widths up to	This setting has no effect when the supporting member is an in-plane wall, as in this case straight bars are always used in the beam end support region.  When the supporting member is a beam or column, typically, the top and bottom bars will be bent (bobbed) up or down in the beam end support region.  This setting only applies to situations where wider supporting beams or columns have been used. • If the value you specify for this setting is greater than the supporting member width a bobbed bar is used.

End Support Curtailment	
	 • If the value you specify is less than the supporting member width, (and the actual supporting beam/ column width can accommodate the calculated straight bar anchorage length) straight bars are used. 

Stirrups (Links)	
Select same bar size in support region and in the span	Allows you to use the same stirrup (link) bar size in the support and span regions.
Select symmetrical stirrup in support region	Allows you to use the same stirrup (link) arrangement (bar size and spacing) in both supports.

Top / Bottom Longitudinal Bar Pattern

Standard Pattern Setup and Default Pattern	
Longitudinal Bar Pattern	Allows you to select a pattern to be viewed and modified.
Longitudinal Default Pattern	Allows you to select the pattern that is applied to new beams when they are first created. **NOTE** The **Longitudinal Default Pattern** option cannot be used to change the pattern applied to existing beams.

Standard Pattern Setup and Default Pattern	
	Instead, you should modify the beam properties.

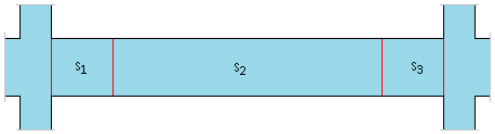
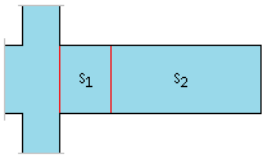
Continuous Span + Cantilever (Backspan) tab	
Allows you to define how the selected longitudinal bar pattern is applied to continuous spans and cantilever backspans. An interactive diagram updates to match the changes you make.	
Bar (Bar ref.)	Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs.
Region T_1, T_2, B_1, B_2 etc.	The top and bottom regions in which bars are applied are defined as a percentage of the span length.
Bar lapping **(Bottom patterns only)**	Select whether you want to lap the bottom bars in the middle of the support region (inside support) or at the face of the support region (outside support).

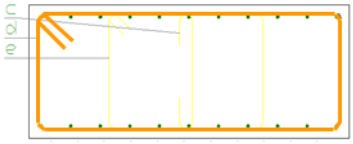
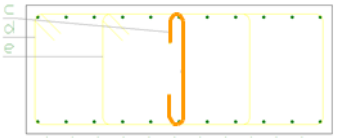
Single Span tab	
Allows you to define how the selected longitudinal bar pattern is applied to single spans. An interactive diagram is also displayed and updates to match the changes you make.	
Bar (Bar ref.)	Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs.
Region T_1, T_2, B_1, B_2 etc.	The top and bottom regions in which bars are applied are defined as a percentage of the span length.

Cantilever tab	
Allows you to define how the selected longitudinal bar pattern is applied to cantilevers. An interactive diagram updates to match the changes you make.	
Bar (Bar ref.)	Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs.

Cantilever tab	
Region T_1, T_2, B_1, B_2	The top and bottom regions in which bars are applied are defined as a percentage of the span length.

Stirrup (Link) Settings

Setting	Description
Normal	When considering shear, the design shear checks are performed in each of 3 regions S_1, S_2, and S_3 as shown below. In each region, the maximum vertical shear from all load combinations is determined and this maximum value used to determine the shear reinforcement required in that region.  • **Region S1**, **Region S2**, **Region S3**: The regions are defined as fixed proportions of the clear span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically. • **Stirrup Type**: Select either open or closed stirrups in the list.
Cantilever	In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2 as shown below. 
Minimum bar size	Sets the minimum allowable bar size that can be used in the design process.
Maximum bar size	Sets the maximum allowable bar size that can be used in the design process.

Setting	Description
Minimum spacing	Sets the minimum allowable stirrup spacing that can be used in the design process.
Maximum spacing	Sets the maximum allowable stirrup spacing that can be used in the design process.
Spacing increment	The designed stirrup spacings are multiples of this value.
Maximum stirrup leg spacing across beam	Allows you to determine if single stirrups, double stirrups, or more are required, depending on the width of the beam.
Use single outside stirrup	Allows you to use a single outside link with additional links added as required (as per stirrup d shown dotted below). 
Accept single leg internal stirrup	Allows the use of single leg internal links (as per stirrup c shown dotted below). 
Optimize stirrup design regions where possible	In this case in the central region S_2, shear reinforcement is provided to meet the minimum of the code requirement or user preference whilst in regions S_1 and S_3, designed shear reinforcement is required. The position and length of region S_2 is determined from considerations of the shear resistance of the concrete cross-section and this then enables the lengths of regions S_1 and S_3 to be determined.

General Parameters

Reinforcement anchorage length parameters (Eurocode, Australia and India only)	
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0.

Fixity (ACI and BS only)	
Partial fixity coefficient, β **(β_1 for BS)**	A coefficient applied to the maximum positive moment in the beam span (excluding support positions) to set a user-defined minimum design moment for beam support regions. Allowable range is 0.0 to 1.0. Default value is 0.25

Cover and spacing check parameters	
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom,\ lim}$. The full calculation is: $c_{nom,\ lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode, Australia and India regional codes.

Deflection (ACI only)	
Long term deflection period	Allowable range is 3 to 60 months. Default value is 60 months.
Time at which brittle finishes are introduced	Allowance range is 1 to 6 months. Default value is 1 month.

FE Chasedown	
Design Beams for FE Chasedown analysis results	Allows you to specify that the beams are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Tolerance on rectilinearity	The calculation of the effective width is only carried out for concrete beams if they lie within the tolerance on rectilinearity set here. The default tolerance is 15 degrees. At greater angles you will be prompted to enter the effective width manually.

Effective flanges calculation	
Tolerance on rectilinearity	The calculation of the effective width is only carried out for concrete beams if they lie within the tolerance on rectilinearity set here. The default tolerance is 15 degrees. At greater angles you will be prompted to enter the effective width manually.

Deflection method and Shrinkage (Australia only)	
Span to depth ratio	This approach to controlling deflection involves satisfaction of the maximum span to depth ratio specified in cl 8.5.4. According to this approach, there is no requirement to calculate deflection, provided beam effective depth exceeds minimum specified value. This approach can be used only when following conditions are satisfied: • Beam cross-section shall be uniform • Beams shall be fully propped during construction • Beams shall be subjected to uniformly distributed load with imposed load (Q) does not exceed dead load (G)
Simplified method	In the simplified deflection calculation procedure actual short-term deflection is calculated using the mean value of modulus of elasticity of concrete at appropriate age (E_{cj}) and an effective second moment of area of member (I_{ef}).The additional long term deflection caused by creep and shrinkage may be approximated by multiplying the short term deflection caused by the sustained load by deflection multiplier k_{cs}
Design time since commencement of drying, **Final drying basic shrinkage strain**, **Shrinkage strain cofficient**	When the simplified method is selected, these values are required for the calculation of autogenous shrinkage strain and drying shrinkage strain.

Column

Reinforcement Layout

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Vertical bars	Allows you to adjust the following settings applied to vertical bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the vertical bar design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a column is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the concrete section area. • **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the concrete section area. • **Steel overstrength factor**: Sets the value that accounts for the reinforcement steel yielding overstrength in seismic design.

Setting	Description
	Only applies to the ACI regional code.
Tie bars (Link bars) (Link bars)	Allows you to adjust the following settings applied to tie (link) bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a column is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are.
Multiple Layers	For circular column sections only, if autodesign requires a 2nd layer of reinforcement you can control the **layer spacing**. Autodesign will use a larger spacing if needed - and it is checked in the check results.
Aspect ratio change point	Allows you to specify a value that determines which rectangular sections are considered to have a low aspect ratio and which are considered to have a high aspect ratio.
Use double ties (links) for low aspect ratio	Allows you to use double ties (links) where applicable instead of single ties for sections with a low aspect ratio.
Use double ties (links) for high aspect ratio	Allows you to use double ties (links) where applicable instead of single ties for sections with a high aspect ratio.

Setting	Description
Use triple ties (links) for low aspect ratio	Allows you to use triple ties (links) where applicable instead of double or single ties for sections with a low aspect ratio.
Use cross ties (links) for low aspect ratio	Allows you to use cross ties (links) where applicable instead of triple, double or single ties (links) for sections with a low aspect ratio.

Detailing Settings

Setting	Description
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Kicker dimension	Allows you to specify the height of column kicker cast above the slab level.
Assumed foundation depth for starter bars	Allows you to control the length of starter bars at the base of the column in drawings.
Join identical bars where possible	Allows you to merge bars together only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.
Number of span region ties (links) required to use separate regions	Allows you to specify the minimum number of span region ties (links) that are necessary for a span region to be indicated on the detail, and in consequence, standardize the spacing on the detail.
Provide ties (links) through full foundation depth	Allows you to draw ties (links) through the full depth of the foundation.
Foundation penetration depth of ties (links)	Allows you to specify the penetration depth of ties (links) into the foundation. **NOTE** The option is only available in the If the **Provide ties (links) through full foundation depth** option is cleared.
Provide ties (links) through floor depth for internal walls	For walls restrained by flat slabs: ties (links) are always provided through

Setting	Description
	the floor depth irrespective of this setting. For walls restrained by beam and slab: ties (links) are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When the option is cleared, they are provided up to the soffit of the shallowest beam depth.
Tie bar type (Link bar type)	Allows you to select the bar type that you want to use.

General Parameters

Setting	Description
Bar sizes no smaller than stack above	Allows you to ensure that bar sizes do not reduce in lower stacks.
Match bar position to stack above	Allows you to have the starting arrangement for longitudinal bars match the arrangement of the bars in the stack above if the section geometry matches.
Increase main bar size preferentially	Allows you to have the corner bars increased in preference to the intermediate bars. All bars start off at the same size (unless the initial bar size is driven by the current arrangement or the stack above), but when the check fails, the corner bars will be increased in size if all bar sizes are the same. Otherwise, the intermediate bars will be increased in size. This means that when the final design is produced, either all bars will be the same size or the corner bars will be one size larger than the intermediate bars. Alternatively, if you require all bar sizes to be the same size, clear the option. This way, all bar sizes are increased together.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the

Setting	Description
	anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode regional code.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom,\ lim}$. The full calculation is: $c_{nom,\ lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode regional code.
Design Columns for FE Chasedown analysis results	Allows you to specify that the columns are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Wall

Reinforcement Layout

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Vertical bars	Allows you to adjust the following settings applied to vertical bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process.

Setting	Description
	• **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a wall is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the concrete section area. • **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the concrete section area.
Vertical bars in end zone	Allows you to adjust the following settings applied to vertical bars in the end zone area: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process in the end zone area. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process in the end zone area. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the end zone area.

Setting	Description
	• **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the end zone area.
Horizontal bars	Allows you to set the minimum allowable bar size that can be used in the design process of horizontal bars and the minimum allowable area of reinforcement as a ratio of the end zone area.
Tie (Link)/horizontal bars	Allows you to set the minimum and maximum allowable bar sizes that can be used in the bar design process and specify the value whose multiples the designed bar spacings are.
Tie (Link)/confinement bars	Allows you to set the minimum and maximum allowable bar sizes that can be used in the design process.
Confinement bars in end zone	Allows you to set the minimum and maximum allowable bar sizes that can be used in the design process.
Substitute loose bars if mesh inadequate	Allows you to use additional loose bars in the end zones when the mesh is inadequate.

Detailing Settings

Setting	Description
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Kicker dimension	Allows you to specify the height of column kicker cast above the slab level.
Assumed foundation depth for starter bars	Allows you to control the length of starter bars at the base of the wall in drawings.
Join identical bars where possible	Allows you to merge bars together only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.
Number of span region ties (links) required to use separate regions	Allows you to specify the minimum number of span region ties (links) that are necessary for a span region to be indicated on the detail, and in

Setting	Description
	consequence, standardize the spacing on the detail.
Provide ties (links) through full foundation depth	Allows you to draw ties (links) through the full depth of the foundation.
Foundation penetration depth of ties (links)	Allows you to specify the penetration depth of ties (links) into the foundation. **NOTE** The option is only available in the If the **Provide ties (links) through full foundation depth** option is cleared.
Provide ties (links) through floor depth for internal walls	For walls restrained by flat slabs: ties (links) are always provided through the floor depth irrespective of this setting. For walls restrained by beam and slab: ties (links) are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When the option is cleared, they are provided up to the soffit of the shallowest beam depth.
Tie bar type (Link bar type)	Allow you to select the bar type that you want to use.
Tie bar type in end zones (Link bar type in end zones)	

General Parameters

Setting	Description
Bar sizes no smaller than stack above	Allows you to ensure that bar sizes do not reduce in lower stacks.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode regional code.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the

Setting	Description
	minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode regional code.
Design Walls for FE Chasedown analysis results	Allows you to specify that the walls are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Slab

Reinforcement Layout

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Minimum spacing (slab auto-design)	Allows you to set the minimum allowable clear bar spacing used in the slab item auto design process. As the option only applies to slab items, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more

Setting	Description
	readily pass by reducing minimum spacing check failures in the patches.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Maximum bar length	Allows you to set the maximum length of a bar.
Bar length increment	Allows you to set a value whose multiples the bar lengths are.
Use mesh where possible	Select to use mesh where possible in bars.
Make bob for top steel of cantilevers	Allows you to apply a bob to the end of the top steel in cantilevers.
Auto selection of outer bars	Allows you to automatically select the bars in the outer layer.

Patches

Setting	Description
Use mesh where possible	Select to use mesh where possible in patches.
Patch Size	Allows you to set the default patch size to be used.
Center strip width	Allows you to set the width of the center strip in beam patches.
Edge strip width as proportion of patch width	Allows you to control how the patch width is split between edge and center strips in column patches.

General Parameters

Setting	Description
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the

Setting	Description
	anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is not displayed when working to British Standards.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom,\ lim}$. The full calculation is: $c_{nom,\ lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode, Australia and India regional codes.
Allowance for additional detailing bars	Allows you to specify a percentage by which the total mass of bars for slabs or mats shown in material listing reports and in slab or mat detailing drawings is increased.

Punching Shear

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement bars	Allows you to set the minimum and maximum allowable bar sizes that can be welded to the rail in the design process and set the maximum allowable diagonal spacing for radial layouts.
Spacing along rail	Allows you to set the minimum and maximum allowable bar spacing along each rail and specify a value whose multiples the bar spacings are.

Setting	Description
Spacing to first reinforcement line	Allows you to set the minimum and maximum spacing from the column face to the first reinforcement line and define a value whose multiples the spacings from the column face are.
Auto-design Method	Allows you to select whether the auto design method is to minimize the bar size or minimize the number of rails and specify the minimum number of studs per face that will be used when auto-designing from minima.
Starting number of studs per face	In auto-design this is the minimum number of studs per face that auto-design will come up with for a valid design
Diagonal spacing between rails	Allows you to set the maximum diagonal spacing between the outermost studs in the last critical perimeter for circular arrangements.
Spacing between rails in Y direction	Allows you to set the minimum and maximum allowable rail spacing in the Y direction and specify a value whose multiples the bar spacings in the Y direction are.
Spacing between rails in Z direction	Allows you to set the minimum and maximum allowable rail spacing in the Z direction and specify a value whose multiples the rail spacings in the Z direction are.
Spacing from last line to outer perimeter	Allows you to set the maximum spacing from the last reinforcement line to the outer perimeter.

Foundations > General

Setting	Description
Minimum distance from column/ wall face to base/cap edge	Allows you to specify the minimum distance from the column or wall face to the base edge or cap edge.
Use presumed bearing capacity method for pad bases and mats (EN 1997-1 cl.6.5.2.4)	Tekla Structural Designer can undertake foundation bearing capacity checks on pad bases and

Setting	Description
	strip footings using two alternative methods allowed in BS EN 1997-1 1. Ultimate Bearing Pressure (default) 2. Presumed Bearing Resistance By selecting this option, the second method is used,and an allowable bearing pressure is checked against serviceability loads. The presumed bearing resistance for the check is specified in the pad base properties (no 'STR' and 'GEO' capacities). **NOTE** The option is only available for the Eurocode regional code.

Foundations > Isolated Foundations

Reinforcement Layout

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.

Setting	Description
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Use mesh where possible	Select to use mesh where possible in bars.

Foundation Size

Setting	Description
Minimum depth	Allows you to set the minimum allowable foundation depth.
Maximum depth	Allows you to set the maximum allowable foundation depth.
Auto-design depth increment	Allows you to set the depth increment.
Minimum side length	Allows you to set the minimum allowable side length.
Maximum side length / strip width	Allows you to set the maximum side length under columns as a distance.
Maximum side length ratio under columns	Allows you to set the maximum side length ratio under columns.
Rounding increment for footing dimensions	Allows you to specify a value to which the overall footing size is rounded.
Default 3-pile cap shape	Allows you to define whether the default shape is triangular or rectangular when 3 piles are used.

General Parameters

Pad Base tab	
Reinforcement Anchorage Length Parameters	
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode regional code.
Cover and Spacing Check Parameters	
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Pad Base tab	
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode regional code.
Passive Resistance	
Consider passive resistance of soil for sliding check	This allows you to control whether the passive resistance of the soil is included in the sliding check, (in addition to the sliding resistance due to friction).
Pad base shear reactions at pinned bases: **Ignorable percentage of passive pressure**	This allows you to ignore small shear forces at pinned bases. The default of 1% is typically a negligible load level which would commonly be ignored. If you do not want to ignore any shear forces, set the value to zero. For further details, see:
Design Safety Factors	
FOS for sliding and uplift checks	Allows you to edit the design safety factor for these checks. **NOTE** This group is only displayed for the ACI/AISC and Indian regional codes.

Pile Cap tab	
Reinforcement Anchorage Length Parameters	
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode regional code.
Cover and Spacing Check Parameters	

Pile Cap tab	
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode regional code.
Design Parameters	
Tolerance for pile position variation	Allows you to specify the allowable deviation value of the pile position from its original plan position. The value is used in the design moment calculations.

Piles

Setting	Description
Minimum number of piles for pile caps under columns	Minimum and maximum numbers of piles that apply when a pile cap under a column has been set to auto design the piles.
Maximum number of piles for pile caps under columns	
Minimum spacing option list	Allows you to select whether the minimum spacing check uses a set minimum spacing value or a multiple of the pile width or pile circumference.
Minimum spacing of piles	Allows you to set the minimum spacing value that you can select in the **Minimum spacing option** list.
Minimum pile edge distance	Allows you to set the minimum distance from the pile face to edge of foundation.
Pile auto-design method list	Allows you to select to use the smallest number of high capacity piles or more piles of lowest capacity when a pile cap has been set to auto design.

Setting	Description
Limit maximum critical section size	When selected, allows you to specify a maximum critical section size as a multiple of the pile dimension.
Use pile capacity for pile punching and shear checks	When selected, allows you to use pile capacity for pile punching and punching shear checks. When unselected, the pile load is used.
Check piles for lateral load	When selected, a pile lateral capacity check is performed.

Foundations > Mat Foundations

Reinforcement Layout

Setting	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Minimum spacing (slab auto-design)	Allows you to set the minimum allowable clear bar spacing used in the panel auto design process. As the option only applies to slab items, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more readily pass by reducing minimum spacing check failures in the patches.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that

Setting	Description
	can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Maximum bar length	Allows you to set the maximum length of bar.
Bar length increment	Allows you to set a value whose multiples the bar lengths are.
Auto selection of outer bars	Allows you to automatically select the bars in the outer layer.
Use mesh where possible	Select to use mesh where possible in bars.

Patches

Setting	Description
Use mesh where possible	Select to use mesh where possible in patches.
Patch Size	Allows you to set the default patch size to be used.
Center strip width	Allows you to set the width of the center strip in beam patches.
Edge strip width as proportion of patch width	Allows you to control how the patch width is split between edge and center strips in column patches.

General Parameters

Setting	Description
Maximum Bond Quality Coefficient	Allows you to specify a value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode regional code.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Setting	Description
Allowance for deviation	Allows you to specify a value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode regional code.

Piles

Setting	Description
Use pile capacity for punching check	When selected, allows you to use pile capacity for the pile punching check. When unselected, the pile load is used.
Check piles for lateral load	When selected, a pile lateral capacity check is performed.

Design Settings > Concrete (Precast - Eurocode only)

Beam

Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement settings	Allows you to adjust the **Reinforcement Class** and **nominal cover** settings applied to beam reinforcement:

Top / Bottom Longitudinal Bar Pattern

Buttons, commands, or options	Description
Longitudinal Bar Pattern	Allows you to select a pattern to be viewed and modified.

Buttons, commands, or options	Description
Longitudinal Default Pattern	Allows you to select the pattern that is applied to new beams when they are first created. **NOTE** The **Longitudinal Default Pattern** option cannot be used to change the pattern applied to existing beams. Instead, you should modify the beam properties.
Continuous Span tab	Allows you to define how the selected longitudinal bar pattern is applied to continuous spans. An interactive diagram updates to match the changes you make. • Bars: Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs. • Region fields: The regions in which bars are applied are defined as a percentage of the span length. **NOTE** The Top and Bottom Region fields dictate how many **design sections** appear in the Tedds Calculation - a different design section is required at each interface between regions.
Pinned Span tab	Allows you to define how the selected longitudinal bar pattern is applied to single pinned spans. An interactive diagram is also displayed and updates to match the changes you make.
Cantilever	Allows you to define how the selected longitudinal bar pattern is applied to cantilevers. An interactive diagram updates to match the changes you make.

Stirrup (Link) Settings

Buttons, commands, or options	Description
Normal	When considering shear, the design shear checks are performed in each of 3 regions S_1 , S_2, and S_3. In each region, the maximum vertical shear from all load combinations is determined and this maximum value

Buttons, commands, or options	Description
	used to determine the shear reinforcement required in that region. • **Region S1**, **Region S2**, **Region S3**: The regions are defined as fixed proportions of the span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically.
Cantilever	In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2.

General Parameters

Buttons, commands, or options	Description
Design Beams for FE Chasedown analysis results	Allows you to specify that the beams are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Design precast beams for lifting forces	Allows you to specify that the beams are also to be designed for lifting forces, (the lifting check being optional in the Tedds calculation).

Column

Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement settings	Allows you to adjust the **Reinforcement Class** and **nominal cover** settings applied to column reinforcement:

General Parameters

Buttons, commands, or options	Description
Design Columns for FE Chasedown analysis results	Allows you to specify that the columns are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Design precast column for lifting forces	Allows you to specify that the columns are also to be designed for a set of lifting forces, (the lifting check being optional in the Tedds calculation).

Design Settings > Design/Reporting Forces

Ignore forces for member design below

NOTE Values entered in the **Ignorable Forces Below** section:

- **are** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Setting	Description
Ignore Forces Below	Allow you to specify appropriate negligible or nominal force levels to prevent 3D analysis from exposing small forces that are normally ignored in design. When the small forces in the 3D analysis are below the specified levels, they are ignored and the design in Tekla Structural Designerproceeds automatically.

Ignore deflections for reporting

Setting	Description
Deflection	Allow you to specify appropriate negligible deflections that are normally ignored. When all the deflection values on a diagram are below the specified level, the deflection diagram is not reported.

Concrete beams (Eurocode only)

Setting	Description
Torsion force % of concrete resistance	Allows you to limit the maximum design value of the torsional resistance moment ($T_{Rd,max}$) to a % of the concrete torsional resistance moment ($T_{Rd,c}$).

Minimum brace force for reporting and connection design

NOTE The axial force entered in this section:

- **is** considered when creating **bracing forces** reports/drawings in Tekla Structural Designer
- **is** considered when performing **connection resistance checks** in Tekla Structural Designer
- **is** considered when designing **connections** in Tekla Connection Designer, or IDEA StatiCa
- **is not** considered when designing **steel** braces in Tekla Structural Designer
- **is not** considered when designing **timber** or **precast** members via Tekla Tedds

These settings provide flexibility to control minimum design values used in the following:

- Member end force reports/drawings
- Connection resistance checks
- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Axial force	Click the links to see examples of how the minimum entered here is used in: • Member end force reports/ drawings (page 915) • Connection resistance checks (page 922) • Exported connection forces (page 924)

Minimum beam end force for reporting and connection design

NOTE Values entered in this section:

- **are** considered when creating **member end force** reports/ drawings and **foundation reaction** reports/drawings in Tekla Structural Designer
- **are** considered when performing **connection resistance checks** in Tekla Structural Designer
- **are** considered when designing **connections** in Tekla Connection Designer, or IDEA StatiCa
- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

These settings provide flexibility to control minimum design values used in the following:

- Member end force reports/drawings
- Connection resistance checks
- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments are used in: • Member end force reports/ drawings (page 915) • Connection resistance checks (page 922)

Setting	Description
	• Exported connection forces (page 924)

Brace/beam end rounding increments

NOTE Values entered here:

- **are** considered when creating **member end force** reports/drawings and **foundation reaction** reports/drawings in Tekla Structural Designer
- **are** considered when performing **connection resistance checks** in Tekla Structural Designer
- **are** considered when designing **connections** in Tekla Connection Designer, or IDEA StatiCa
- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

These settings provide flexibility to control the rounding increments used in the following:

- Member end force reports/drawings
- Connection resistance checks
- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Rounding Increment for Force Rounding Increment for Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Member end force reports/drawings (page 915) • Connection resistance checks (page 922) • Exported connection forces (page 924)

Minimum foundation reactions for reporting and connection design

These settings provide flexibility to control minimum design values and rounding increments used in the following:

- Foundation reaction reports/drawings
- Connection resistance checks
- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Foundation Reactions Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Foundation reaction reports/ drawings (page 918) • Exported connection forces (page 924)
Rounding Increment for Force Rounding Increment for Moment	

Minimum design forces

NOTE Values entered in the **Minimum Design Forces** section:

- **are** considered when creating **member end force** reports/ drawings and **foundation reaction** reports/drawings in Tekla Structural Designer
- **are** considered when performing **connection resistance checks** in Tekla Structural Designer
- **are** considered when designing **connections** in Tekla Connection Designer, or IDEA StatiCa
- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

---

These settings provide flexibility to control minimum design values and rounding increments used in the following:

- Member end force reports/drawings
- Foundation reaction reports/drawings
- Connection resistance checks

- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Brace Force Axial Force	Click the links to see examples of how the minimum entered here is used in: • Member end force reports/ drawings (page 915) • Connection resistance checks (page 922) • Exported connection forces (page 924)
Beam End Forces Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Member end force reports/ drawings (page 915) • Connection resistance checks (page 922) • Exported connection forces (page 924)
Rounding Increment for Force Rounding Increment for Moment	
Foundation Reactions Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Foundation reaction reports/ drawings (page 918) • Exported connection forces (page 924)
Rounding Increment for Force Rounding Increment for Moment	

How Minimum Design Forces are used

The following topics illustrate how Minimum Design Forces are applied in different areas of Tekla Structural Designer.

Member end force reports/drawings

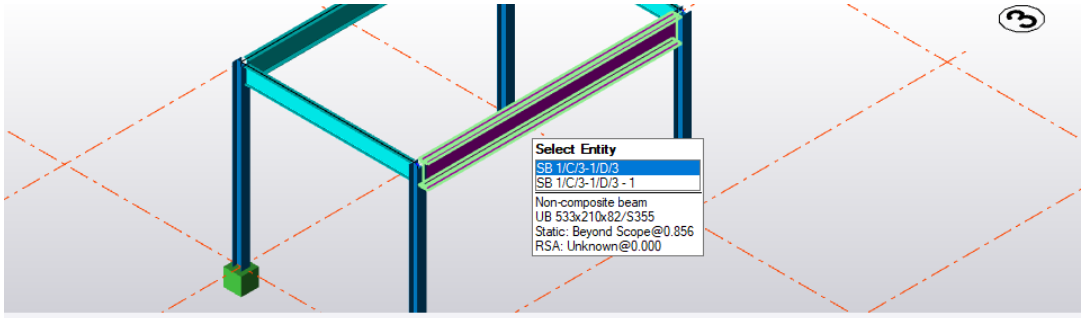
Tekla Structural Designer reports member end forces (Beam end forces/ Bracing forces) based on analysis results.

The 'Minimum Design Forces' inputs under **Brace Force/Beam End Forces** allow the user to specify minimum values for the reported forces, and also apply rounding increments to them.

NOTE **While minimum design values and rounding increments are applied to the member end force reports/drawings, they have no impact on steel and concrete member design in Tekla Structural Designer, or timber and precast design using Tekla Tedds.**

Consider a fixed beam with axial force + major/minor shear + torsion + major/ minor moment.

Forces/moments are reported in the Beam End Force report.



Member End Forces

Beams

End 1/End 2 + Coincident, First-order linear, All combinations

St. 1 (1)

Reference	Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
SB 1/C/3-1/D/3	1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR₁ -1.35 G+1.5 Q+1.5 RQ	-6.224	10.438	163.878	16.3	10.4	7.0
				2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR₁ -1.35 G+1.5 Q+1.5 RQ	6.224	5.762	175.650	-16.3	24.9	7.0

The user wants the forces rounded to the nearest 5kN and moments to the nearest 10kNm so inputs 5kN in the 'Rounding Increment for Force' and 10kNm in the 'Rounding Increment for Moment' under **Beam End Forces**.



The Beam End Forces report displays rounded value of forces and moments.

Beam End Forces

Member End Forces

Beams

End 1/End 2 + Coincident, First-order linear,All combinations

St. 1 (1)

Reference	Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
SB 1/C/3-1/D/3	1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR$_1$ -1.35 G+1.5Q+1.5RQ	-10.000	15.000	165.000	16.3	20.0	10.0
				2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR$_1$ -1.35 G+1.5Q+1.5RQ	10.000	10.000	180.000	-16.3	30.0	10.0

NOTE The rounding increment for moment applies to major axis and minor axis moments, but does not apply to torsional moments (M_x).

The user might want the forces reported to be not less than minimum values, for example they might enter 200kN in the 'Major Axis Shear Force' under '**Beam End Forces**.

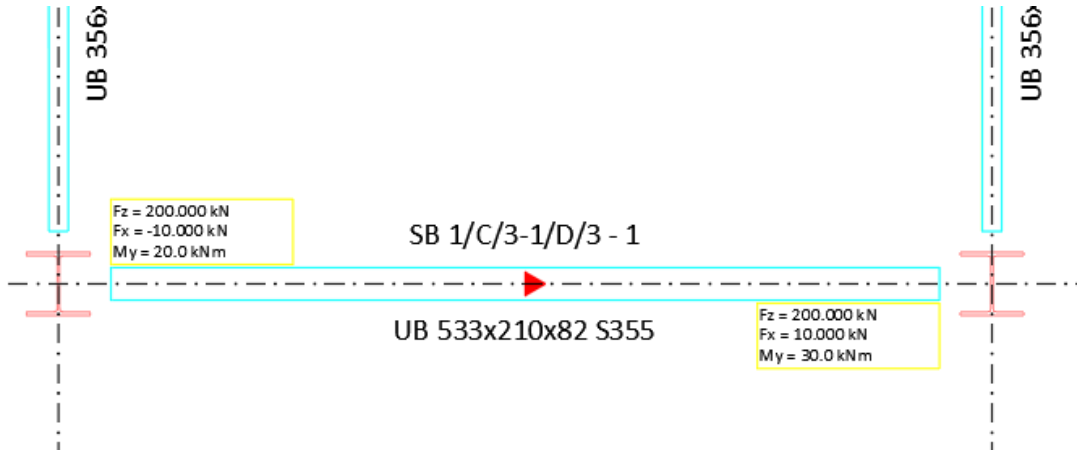


This results in the report being updated as follows.

Beam End Forces

Reference	Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
SB 1/C/3-1/D/3	1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR$_1$ -1.35G+1.5Q+1.5RQ	-10.000	15.000	200.000	16.3	20.0	10.0
				2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR$_1$ -1.35G+1.5Q+1.5RQ	10.000	10.000	200.000	-16.3	30.0	10.0

The changes to the rounding increments and mimimum force are also reflected in the Beam End Forces drawing.



Foundation reaction reports/drawings

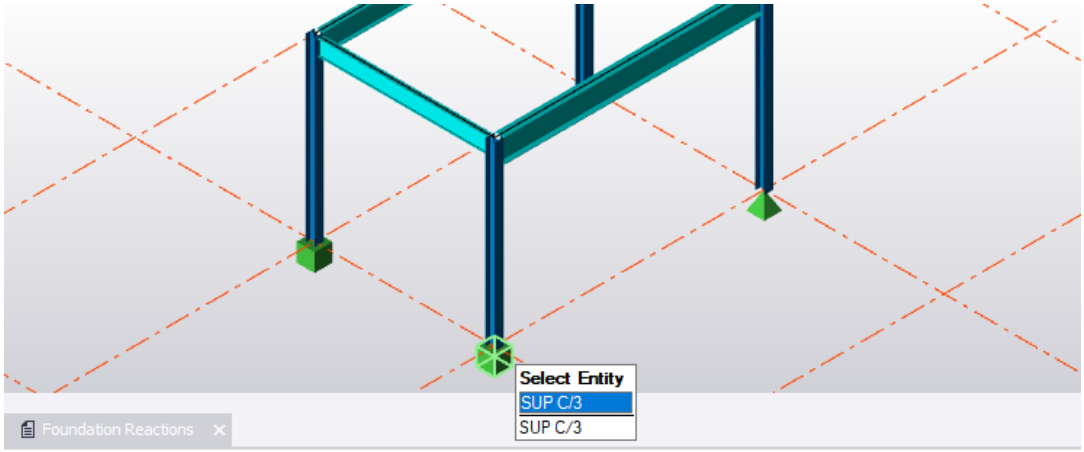
Tekla Structural Designer reports foundation reactions based on analysis results.

The 'Minimum Design Forces' inputs under **Foundation Reactions** allow the user to specify minimum values for the reactions, and also apply rounding increments to them.

NOTE Changed force/moment values are for reporting purposes only.

Consider a fixed end support.

Forces/moments are reported in the Foundation Reactions report.



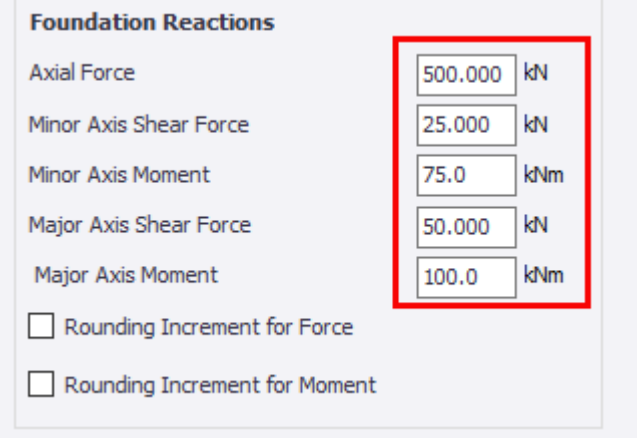
FoundationReactions

FoundationReactions, First-order linear, Strength Factors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	147.140	40.500	-40.500	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	75.824	0.000	0.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	148.911	0.000	0.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	104.596	0.000	0.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	278.195	3.468	-7.276	-30.2	-18.7	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	282.895	6.971	0.002	-34.6	0.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	289.967	4.071	-6.224	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	282.895	1.691	-0.002	0.0	0.0	0.0

The user wants these reported values to be not less than certain minimum values, so inputs the required minimums under **Foundation Reactions**.



The Foundation Reactions report shows minimum design forces/moments specified by the user.

Foundation Reactions

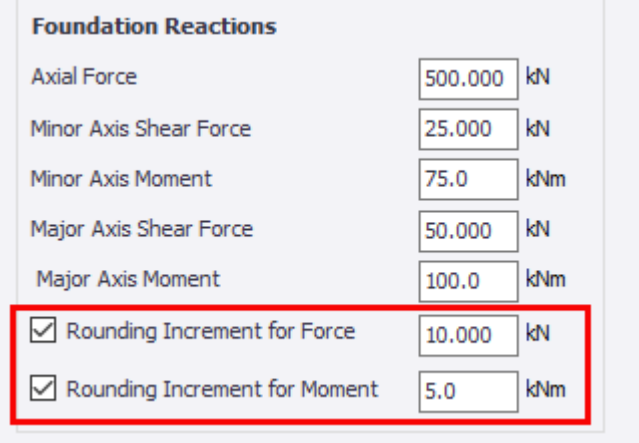
FoundationReactions

FoundationReactions, First-order linear, Strength Factors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-40.500	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	-25.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	-100.0	-75.0	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	25.000	-100.0	-75.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	0.0

The user also wants the forces rounded to the nearest 5kN and moments to the nearest 10kNm so inputs 5kN in the 'Rounding Increment for Force' and 10kNm in the 'Rounding Increment for Moment' under **Foundation Reactions**.



When the report is updated, any values exceeding the minimums are rounded up accordingly.

Foundation Reactions

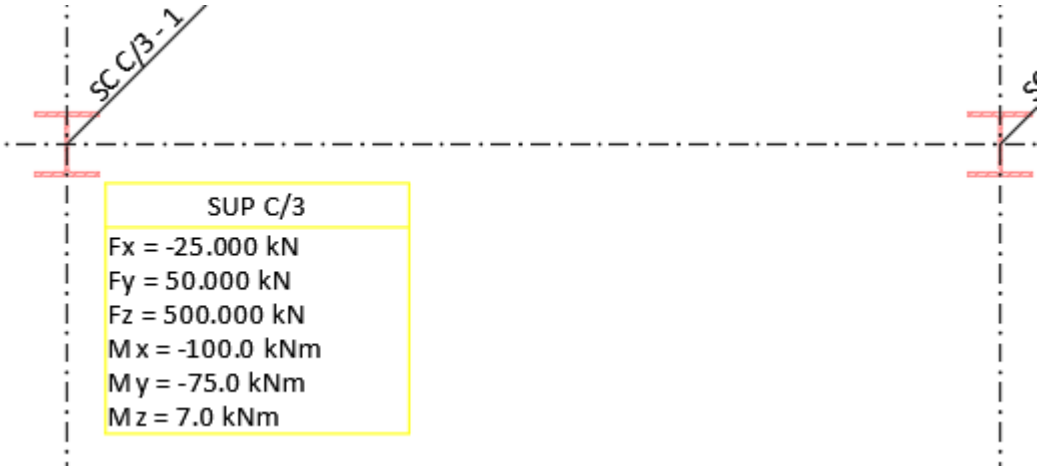
FoundationReactions

FoundationReactions, First-order linear, StrengthFactors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-50.000	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	-25.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	-100.0	-75.0	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	25.000	-100.0	-75.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	0.0

The changes are also reflected in the Foundation Reactions drawing, which now shows the same minimum values of design force and moment.



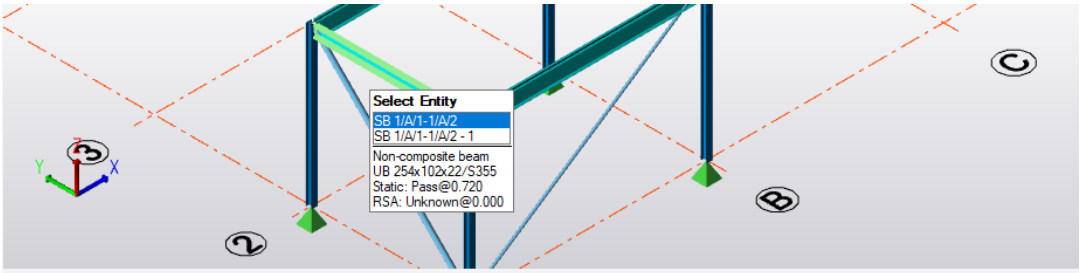
Connection resistance checks

Tekla Structural Designer performs connection resistance checks for simple beams and braces which use design forces from the analysis results.

The 'Minimum Design Forces' inputs under **Brace Force/Beam End Forces** allow the user to specify minimum values for these forces, and also apply rounding increments to them.

Consider a simple beam connection.

Review Data of Connection Resistance shows the following details.

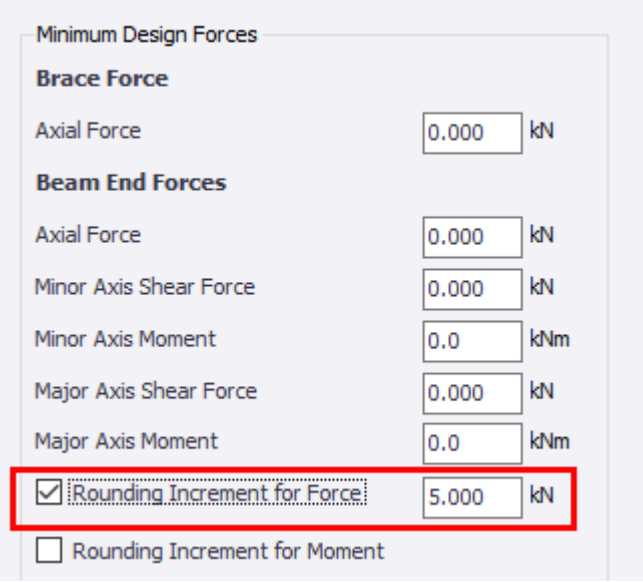


Review Data

Connection Resistance

Reference	Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization	Status
SB 1/A/1-1/A/2	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	0	1	2	74.000	0.575	✓ Pass
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	1	1	2	74.000	0.575	✓ Pass
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	2	1	2	72.000	0.591	✓ Pass
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Full Depth End Plate	0	1	2	301.000	0.141	✓ Pass
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	0	1	2	158.000	0.270	✓ Pass
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	1	1	2	158.000	0.270	✓ Pass
	UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	2	1	2	99.000	0.430	✓ Pass

The user wants the forces rounded to the nearest 5kN so inputs 5kN in the 'Rounding Increment for Force' under **Beam End Forces**.



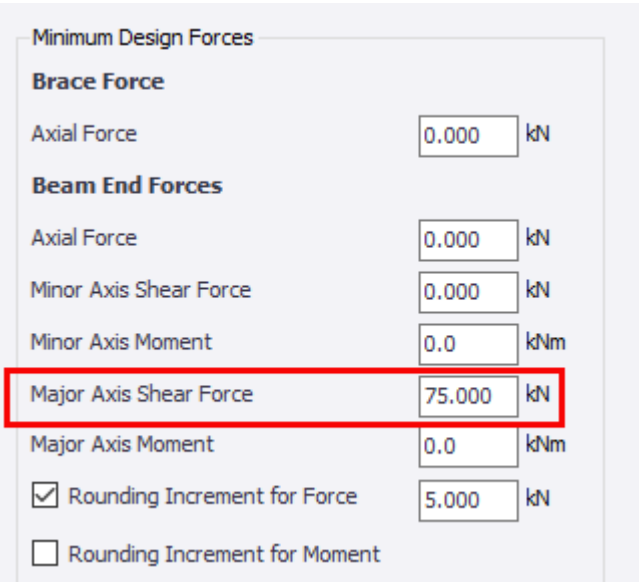
By closing and re-opening the Connection Resistance Review Data, the connection resistance check is updated and now uses the rounded value.

Review Data

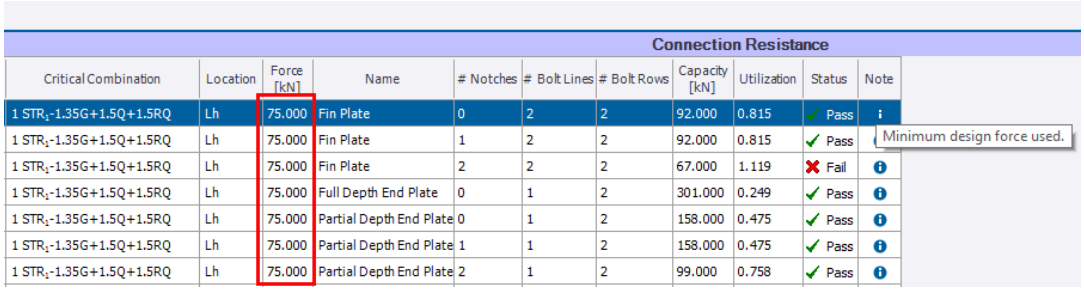
Connection Resistance

Reference	Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization	Status
SB 1/A/1-1/A/2	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	0	1	2	74.000	0.608	✓ Pass
	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	1	1	2	74.000	0.608	✓ Pass
	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	2	1	2	72.000	0.625	✓ Pass
	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Full Depth End Plate	0	1	2	301.000	0.150	✓ Pass
	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	0	1	2	158.000	0.285	✓ Pass
	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	1	1	2	158.000	0.285	✓ Pass
	UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	2	1	2	99.000	0.455	✓ Pass

The user also wants a minimum design force of 75 kN for connection resistance check so inputs 75 kN in the 'Major Axis Shear Force' under **Beam End Forces**.



By closing and re-opening the Connection Resistance Review Data once again, the connection resistance check is updated to use the specified minimum design force.



Connection Resistance

Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization	Status	Note
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Fin Plate	0	2	2	92.000	0.815	Pass	i
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Fin Plate	1	2	2	92.000	0.815	Pass	
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Fin Plate	2	2	2	67.000	1.119	Fail	
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Full Depth End Plate	0	1	2	301.000	0.249	Pass	
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	0	1	2	158.000	0.475	Pass	
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	1	1	2	158.000	0.475	Pass	
1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	2	1	2	99.000	0.758	Pass	

Minimum design force used.

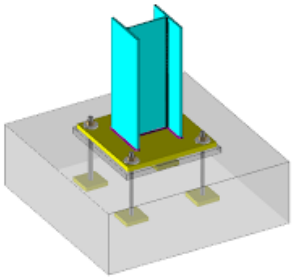
Because the minimum design force is being used this is indicated in the table under the 'Note' column.

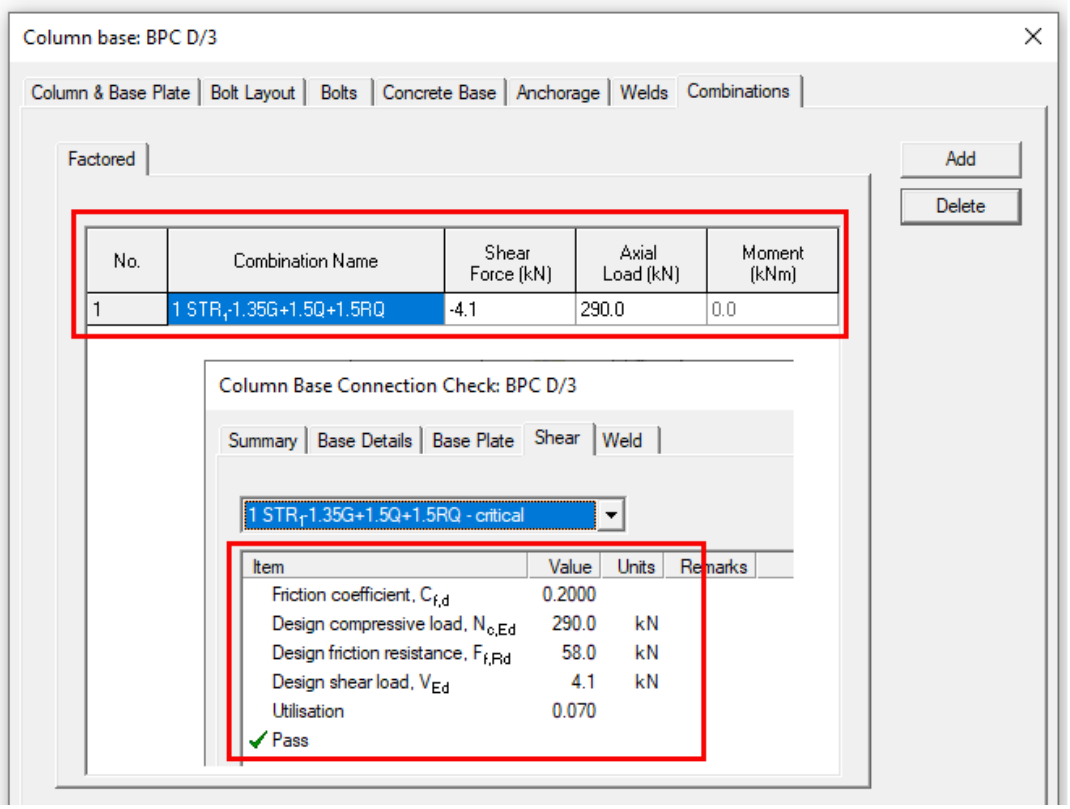
Exported connection forces

Forces exported to Tekla Connection Designer or IDEA StatiCa from Tekla Structural Designer are based on analysis results.

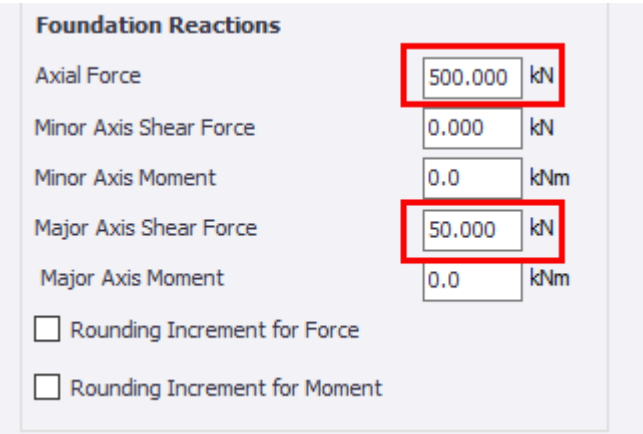
The ‘Minimum Design Forces’ inputs allow the user to specify minimum values for the exported forces, and also apply rounding increments to them.

Consider a base plate connection exported to Tekla Connection Designer and designed for axial load and major shear.





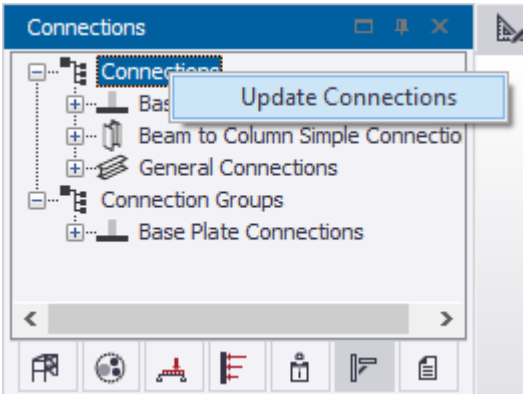
The user wants to design the base plate connection for a minimum shear of 50 kN and for a minimum axial load of 500 kN, so inputs these values under **Foundation Reactions**.



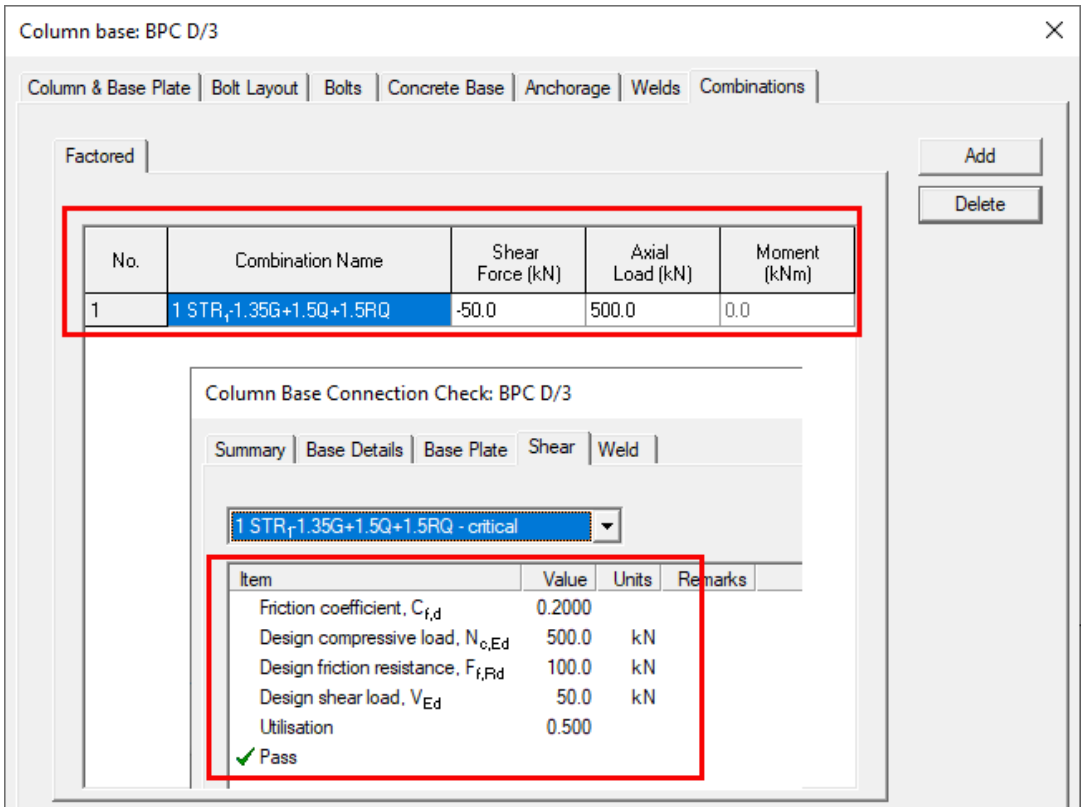
NOTE For other connection types the values under **Brace Forces/Beam End Forces** would be edited instead.

Rounding increments could also be applied if required. These would only apply to forces and moments exceeding the above minimums.

To apply the new minimums to the existing connections, it is necessary to click **Update Connections** from the Project Workspace.



The base plate connection is now designed for the minimum shear of 50 kN and for the minimum axial load of 500 kN specified by the user.



Design Settings > Design Groups

Setting	Description
Members to design using groups	Allows you to select the member types for which you want to apply grouped design. When the members are grouped, only one member in the group is designed. This design is then copied to the remaining members in the group, so that they can be checked. Any failing member in the group is redesigned, and the process is repeated until all members pass, or a design is not possible.

Design Settings > Autodesign

Steel/Cold Formed Members

Setting	Description

Reset Autodesign to off...	Allows you to control what happens to individual steel and concrete member auto design settings and isolated foundation auto design settings at the end of the design process. The options are:
	• **Always**: the auto design setting is automatically cleared at the end of the design process, so that each member is put into check mode.
	• **Never**: the autodesign settings is always retained as it is at the end of the design process.
	• **When check status is at worst**:
	  • **Pass**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass.
	  • **Warning**: The auto design setting is only automatically cleared at the end of the

Setting	Description
	design process for members with design status Pass or Warning. • **Fail**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, or Fail. • **Invalid**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail or Invalid. • **Beyond Scope**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail, Invalid, or Beyond Scope. **TIP** The most practical use of **When check status is at worst** is to set it to **Pass** and start with all members in auto design mode. At the end of the first design run, passing members would be set to check mode, allowing you to focus on the remaining members still in auto design mode.
Beam design section step back *[US Regional code only]*	This option allows for the sections order file loop in subsequent iterations to start up to 5 sections before the one identified as passing in the prior iteration, occasionally improving design efficiency but with a potential to impact design times particularly for larger models. • Applies to the autodesign of steel beams only, both non-composite and composite, • Applies to static design only,

Setting	Description
	• Defaults to 0 and can be adjusted to consider up to 5 section step backs.

Isolated Foundations

Setting	Description

Reset Autodesign to off...	Allows you to control what happens to individual steel and concrete member auto design settings and isolated foundation auto design settings at the end of the design process. The options are:
	• **Always**: the auto design setting is automatically cleared at the end of the design process, so that each member is put into check mode.
	• **Never**: the autodesign settings is always retained as it is at the end of the design process.
	• **When check status is at worst**:
	  • **Pass**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass.
	  • **Warning**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass or Warning.
	  • **Fail**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, or Fail.
	  • **Invalid**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail or Invalid.

Setting	Description
	• **Beyond Scope**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail, Invalid, or Beyond Scope. **TIP** The most practical use of **When check status is at worst** is to set it to **Pass** and start with all members in auto design mode. At the end of the first design run, passing members would be set to check mode, allowing you to focus on the remaining members still in auto design mode.

Concrete Members

Setting	Description

Reset Autodesign to off...	Allows you to control what happens to individual steel and concrete member auto design settings and isolated foundation auto design settings at the end of the design process. The options are:
	• **Always**: the auto design setting is automatically cleared at the end of the design process, so that each member is put into check mode.
	• **Never**: the autodesign settings is always retained as it is at the end of the design process.
	• **When check status is at worst**:
	  • **Pass**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass.
	  • **Warning**: The auto design setting is only automatically cleared at the end of the

Setting	Description
	design process for members with design status Pass or Warning. • **Fail**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, or Fail. • **Invalid**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail or Invalid. • **Beyond Scope**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail, Invalid, or Beyond Scope. **TIP** The most practical use of **When check status is at worst** is to set it to **Pass** and start with all members in auto design mode. At the end of the first design run, passing members would be set to check mode, allowing you to focus on the remaining members still in auto design mode.

Composite beam Connector auto-layout

Setting	Description
Reset Auto layout to off...	Allows you to control what happens to individual composite beam auto layout settings at the end of the design process. The options are: • **Always**: the auto layout setting is automatically cleared at the end of the design process.

Setting	Description
	• **Never**: the auto layout setting is always retained as it is at the end of the design process.
Prefer fewer connectors over section weight *[US Regional code only]*	This option allows the user to switch the emphasis of the composite design, in terms of it's efficiency, between having the smallest possible number of studs or having the lighter possible section, typically one in detriment of the other. • Applies to the autodesign of steel composite beams in particular the studs, • Applies to static design only, • Applies to beams with decking ribs parallel to the beam only, • The default is to prefer less studs over a lighter section.

Design Settings > Design Warnings (AISC/ASC only)

Warnings used as guidance during the design of a structure to US seismic provisions are in effect by default. These can be de-activated as found useful from this page.

Concrete Seismic Warnings	
Specified concrete grade is too low	• Compliance with ACI 318-08 and ACI 318-11 Section 21.1.4.2, "(..) f'_c, shall not be less than 3000 psi." • Applies to normal weight reinforced concrete columns and beams part of Special Moment Frames • Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls • Prompts the designer to increase the concrete grade of the member
Specified longitudinal reinforcement grade too high	• Compliance with ACI 318-11 Section 21.1.5.2, "(..) reinforcement resisting earth-quake induced

Concrete Seismic Warnings	
	flexure, axial force, or both, shall comply with ASTM A706, Grade 60." • Applies to normal weight reinforced columns and beams part of Special Moment Frames • Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls • Prompts the designer to use lower longitudinal reinforcement grades
Column at the end of the beam is not in the same seismic frame	• Used to validate if the frame of which the beam is part of is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete beams part of a seismic force resisting system • Applies to both SRFS type and direction • Checks both ends of each beam span • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam at the end of the column is not in the same seismic frame	• Used to validate if the frame of which the column is part off is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete columns part of a seismic force resisting system • Checks the top of each column stack • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam support region size is too small	• Compliance with ACI 318-08 and ACI 318-11 Section 21.3.4.2, "(..) hoops shall be provided over lengths not less than 2h measured from the face of the supported member toward the midspan"

Concrete Seismic Warnings	
	• Applies to normal weight reinforced concrete beams part of Intermediate and Special Moment Frames • Comes into effect if the length of the end shear design regions is smaller than the requirement • Prompts the designer to review the shear design regions - Design options
High seismic to conventional demand on non-SFRS beams **High seismic to conventional demand ratio of non-SFRS columns** **High seismic to conventional demand of non-SFRS walls**	• Used to recognize when a member has a significant contribution to the building's lateral force resistance but has not been assigned to the SFRS with the required ductility properties • Specifically apply to each member type • The highest non-seismic design ratio from bending and shear design is obtained • The highest seismic design ratio from bending and shear design is obtained • Comes into effect when the ratio between seismic and non-seismic values is higher than a given threshold (default = 1.0) • The maximum seismic/non-seismic ratio threshold that triggers the warning is set through the **Seismic to conventional minimum demand ratio** field described below • Prompts the designer to either include the member in the SFRS or review the structure layout to reduce the contribution of the member to the lateral force resistance
Non-SFRS design requirements applicable in SDC D-F	• Compliance with ACI 318-08 and ACI 318-11 Sections 21.13.1, "Requirements of section 21.13

Concrete Seismic Warnings	
	apply to members not designated as part of the seismic-force-resisting system in structures assigned to SDC D, E and F" • Applies to normal weight reinforced concrete beams, column and walls not included in the SFRS • Comes into effect in Seismic Design Category, SDC D or above • Prompts the designer to apply code prescribed requirements on top of current design

Steel Static Warnings	
Additional notional loads required when alpha Pr/Py > 0.5, C2.3	• Compliance with 7.3 (3) of AISC 360-05 and C2.3 of AISC 360-10 requires the stiffness of certain members to be reduced. When the ratio of the required axial compressive force (Pr) and the axial yield strength (Py = Fy Ag) is > 0.5 then this stiffness must be reduced further • As an alternative which is the approach adopted in Tekla Structural Designer, an additional notional load of 0.001 Yi can be applied instead • When the program identifies that the ratio of Pr/Py exceeds 0.5 a warning is given to alert the designer • If this additional notional load has been dealt with by the designer then this warning is no longer relevant and this option allows you to switch off the warning

Concrete Seismic	
Seismic to conventional minimum demand ratio	• Sets the maximum seismic/non-seismic ratio threshold, which is used to trigger the **High seismic**

Concrete Seismic	
	to conventional demandwarnings listed above • Default: 1.000

Design Settings > Design Checks (IS only)

Warnings used as guidance during the design of a structure to IS seismic provisions are in effect by default. These can be de-activated as found useful from this page.

Concrete Seismic Warnings	
Requirement of seismic provisions of non-SFRS members	• Applies to normal weight reinforced concrete beams, column and walls not included in the SFRS • Prompts the designer to apply code prescribed requirements on top of current design
Structure in seismic zone II - Clause 1.1.1	See IS 13920 2016 Clause 1.1.1
Specified concrete grade is too low - Clause 5.2	• Prompts the designer to increase the concrete grade of the member
Specified longitudinal reinforcement grade too high - Clause 5.3.1 (b)	• Prompts the designer to use lower longitudinal reinforcement grades
Column at the end of the beam is not in the same seismic frame	• Used to validate if the frame of which the beam is part of is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete beams part of a seismic force resisting system • Applies to both SRFS type and direction • Checks both ends of each beam span • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam at the end of the column is not in the same seismic frame	• Used to validate if the frame of which the column is part off is correctly set-up as a seismic frame

Concrete Seismic Warnings	
	• Applies to normal weight reinforced concrete columns part of a seismic force resisting system • Checks the top of each column stack • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam axial stress is too high - consider as compression member - Clause 6.1	See Clause 6.1
Beam support region size is too small - Clause 6.1.1, 6.1.2	• Comes into effect if the length of the end shear design regions is smaller than the requirement • Prompts the designer to review the shear design regions - Design options
Beam effective width at support is too large/not calculated - Clause 6.1.4	• Comes into effect if the effective width at support is larger than the requirement, or not calculated
Beam depth is too high - Clause 6.1.4	See Clause 6.1.4
Column axial stress limit - consider as flexural member - Clause 7.1	See Clause 7.1
Column cross sectional dimension is too small - Clause 7.1.1, 7.1.2	See Clauses 7.1.1, 7.1.2
Column width is too small in Beam-Column joint- Clause 9.1.3	See Clause 9.1.3
Wall thickness is too small - Clause 10.1.2, 10.1.3	See Clause 10.1.2
Wall width is too small in Wall-Beam joint- Clause 9.1.3	See Clause 9.1.3

Concrete seismic checks	
Wall minimum horizontal reinforcement - Clause 10.1.6 Table 1 **Wall maximum allowed horizontal bar diameter - Clause 10.1.8**	• These options can be used to skip wall design seismic horizontal and vertical reinforcement checks

Concrete seismic checks	
Wall minimum vertical reinforcement - Clause 10.1.6 Table 1 **Wall maximum allowed vertical bar diameter - Clause 10.1.8**	

Steel Static Warnings	
Shear buckling is too high - Clause 8.4.2.1	See Clause 8.4.2.1

Design Settings > Drift Checks/Sway & Drift Checks

Sway & drift check settings

Setting	Description
Merge short stacks	With this setting off the Sway/Drift, Wind Drift & Seismic Drift checks are performed for all column stacks and wall panels irrespective of their stack lengths. With this setting on you can specify a minimum stack length. **NOTE** The 'Merge short stacks' option does not apply to columns/walls where you have elected to merge stacks manually (via the column/wall properties). The following logic is applied to merging stacks when the setting is on: 1. Start at the topmost stack of the column/wall. 2. If the stack length is less than the merge limit then merge with the stack below. 3. Check again to see if then new length exceeds the merge limit, if not then merge again with the next stack below. 4. Repeat step 3 as required until the stack length exceeds the merge limit. 5. For subsequent stacks repeat steps 2 and 3 as required until the length exceeds the merge limit. 6. When the bottommost stack is reached, if this needs to be merged, merge with the stack above.

Setting	Description
	NOTE If a single stack column or single panel wall length is less than the merge short stacks limit it is not considered for sway/drift, wind drift, seismic drift checks.
Check wind cases only	Allows you to set that the wind drift check performed on all columns only considers the effect of the wind loadcases in each wind combination. Clear the setting to consider the effects of all loadcases in wind combinations, such as drift induced by gravity loads. For more information, see: Consider wind cases only for the wind drift check (page 844)
Wind Drift Limit	The limit applied to all columns and walls. **NOTE** This can be overridden for specific members as required in the member properties. **NOTE** A re-analysis is required if the limit is changed. For more information, see: Set the wind drift limit (page 843)
Check for Resultant Wind Drift	The wind drift check can either be performed in two directions (Dir 1 & 2), or for the resultant wind drift direction only. For new models the default is to check for the resultant direction only. For more information, see: Choose resultant or directional wind drift checks (page 843)

Design Settings > Fire check (Eurocode only)

Setting	Description
Time interval for critical temperature iteration if 'Unprotected'	default 5 sec
Time interval for critical temperature iteration if 'Protected'	default 30 sec

Setting	Description
NOTE The required time of fire exposure is specified for each beam individually in the steel beam (page 1338) (default R30).	

Design Settings > Timber

The settings allow you to apply your own set of timber design defaults that broadly apply to all members. These are incorporated into Tedds calculations as default variables, which can then be edited within the calculation if required.

Timber (US regional code)

Setting	Description
Service condition	NDS: Service condition • Dry (default) • Wet
Temperature range	• up to 100 degF (default) • 100 degF to 125 degF • 125 degF to 150 degF
Is section incised	• yes • no (default) The incisions made when applying a preservative treatment result in a loss of area and section modulus. When you indicate that the section is incised, appropriate incising factors are applied in the Tedds calculation.
Include long term effects in total load deflection check	Whan a Tedds design is performed for a timber member or group and a total load deflection check has been specified in the timber member properties, this setting is used to control whether long term effects due to creep are included in the check. • yes (default) • no

Timber (Eurocode)

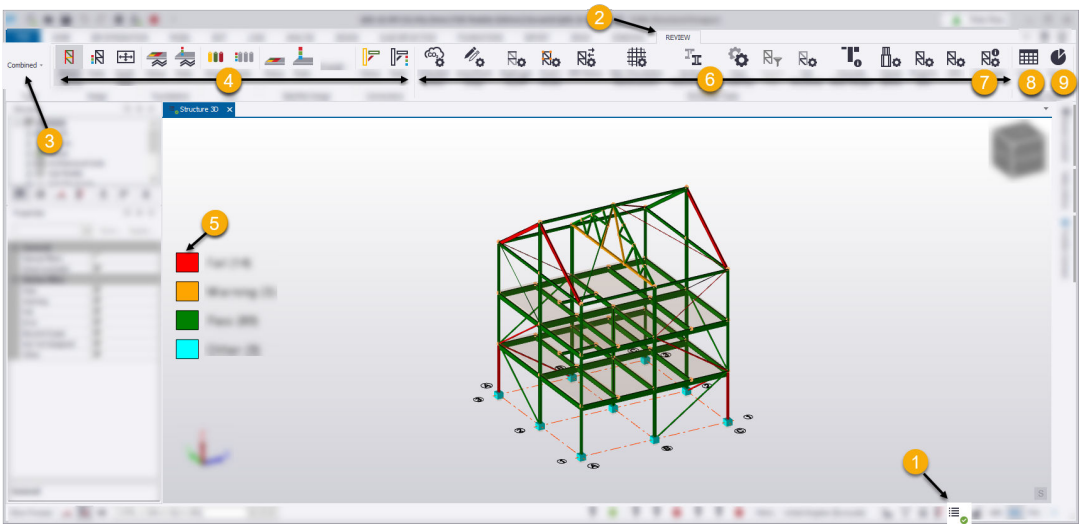
Setting	Description
Service class	• 1 (default) • 2 • 3
Individual grade stamp	• yes • no (default) Depending on the National Annex being worked to, the partial factor for material properties can be reduced when timber has been individually marked.
Include long term effects in total load deflection check	Whan a Tedds design is performed for a timber member or group and a total load deflection check has been specified in the timber member properties, this setting is used to control whether long term effects due to creep are included in the check. • yes (default) • no

6 Review models

The **Review** view enables you to graphically display the design status of members, slabs, foundations and connections. In the same view you can also interrogate or modify a variety of model parameters and properties.

You can also open a **Review Data** view in order to display tables of Sway/Drift Results, Design Summaries, Material Lists and much more.

The main components in the **Review** view are numbered below:



To switch a view to the **Review** view regime click **Review View (1)** in the **Status bar**, then click the **Review** ribbon tab **(2)**.

- To graphically review the design status or ratio (page 943) of objects in the view, select the design **type (3)** (Static, RSA, or Combined) then choose the required **Status** or **Ratio** icon **(4)** from the ribbon. This causes a legend **(5)** to be displayed and objects in the view are color coded accordingly.
- To graphically review and edit the properties (page 958) of objects in the view, select a category from the **Show/Alter State** group **(6)**. Depending on the category, it can either be reviewed with the aid of a legend, or edited with the aid of the **Properties** window.

    - A particularly useful category is (page 970) **(7)**. For this category you can select from a list of **Attributes** in the **Properties** window before graphically reviewing or editing the chosen attribute.

- To open a tabular (page 1011) view, select  **Tabular Data (8)**.

- To open a (page 1059) view, select  **Charts (9)**. In this view you can display charts of embodied carbon data.

6.1 Review designs

By graphically reviewing the design status and ratios you are able to quickly assess the design efficiency.

Click the links below to find out more:

- Review member design (page 943)
- Review foundation and pile design (page 945)
- Review slab and mat design (page 946)
- Design review filters (page 951)

Review member design

Review member design status

1. Activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
    - Static
    - RSA
    - Combined (i.e. Static and RSA together)
3. On the ribbon, in the **Design** group, click **Status**.

Color codes display the design status of each member using the following classifications:

Status	Description
Error	The member currently cannot be designed because an error has occurred.
Beyond Scope	The member cannot be designed because its design is beyond the scope of Tekla Structural Designer or Tekla Structural Designer/Tedds.
Fail	The member has failed one or more design checks.
Warning	Although the member has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
Pass	The member has passed all design checks.
Other	The member has not been designed.

Review member design ratios

1. Activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA
   - Combined (i.e. Static and RSA together)
3. On the ribbon, in the **Design** group, click **Ratio**.

   Color codes display the design ratio of each member. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates members that either are beyond scope or have yet to be designed.

Review member depth ratios

1. Activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.

- Static
- RSA
- Combined (i.e. Static and RSA together)

3. On the ribbon, in the **Design** group, click **Depth Ratio**.

   All steel and concrete beams are color coded to indicate their span to depth utilization ratios.

See also

Design review filters (page 951)

Review design summary tabular results (page 1012)

Review foundation and pile design

Review foundation or pile status

1. Activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA
   - Combined (i.e. Static and RSA together)
3. Either in the **Foundations** group or the **Piles** group of the ribbon, click **Status**.

   Color codes display the design status of each foundation or pile using the following classifications:

Status	Description
Error	The foundation or pile currently cannot be designed because an error has occurred.
Beyond Scope	The foundation or pile cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Fail	The foundation or pile has failed one or more design checks.
Warning	Although the foundation or pile has passed the design checks, one or more warnings have been issued.

Status	Description
	Review the warnings before deciding if you need to take further action.
Pass	The foundation or pile has passed all design checks.
Other	The foundation or pile has not been designed.

Review foundation or pile ratios

1. Activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA
   - Combined (i.e. Static and RSA together)
3. Either in the **Foundations** group or the **Piles** group of the ribbon, click **Ratio**.

   Color codes display the design ratio of each foundation or pile. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates foundations and piles that are either beyond scope or have yet to be designed.

See also

Design review filters (page 951)

Review design summary tabular results (page 1012)

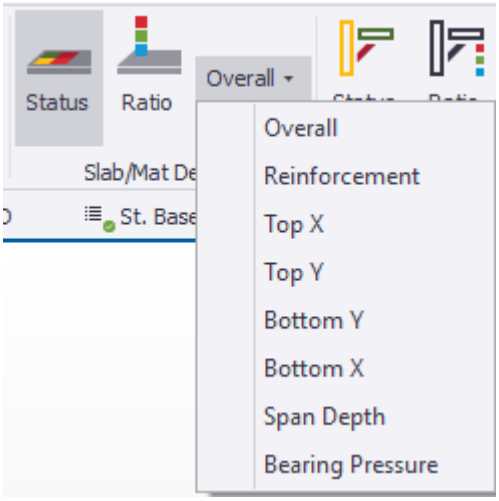
Review slab and mat design

Review slab and mat design status

1. Open a 2D view at the level of the slab or mat and activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA

- Combined (i.e. Static and RSA together)

3. In the **Slab/Mat Design** group on the ribbon, click **Status**.
4. In the same group, click the drop list to display the filtering options.



Filter	Description
Overall	The governing Top X, Top Y, Bottom X, Bottom Y, or span depth result is displayed.
Reinforcement	The governing Top X, Top Y, Bottom X, or Bottom Y result is displayed.
Top X	The result is only displayed for top reinforcement in the X direction.
Top Y	The result is only displayed for top reinforcement in the Y direction.
Bottom X	The result is only displayed for bottom reinforcement in the X direction.
Bottom Y	The result is only displayed for bottom reinforcement in the Y direction.
Span Depth	The span to depth result is displayed.
Bearing Pressure	The bearing pressure result is displayed.

5. Select the filtering option you require.

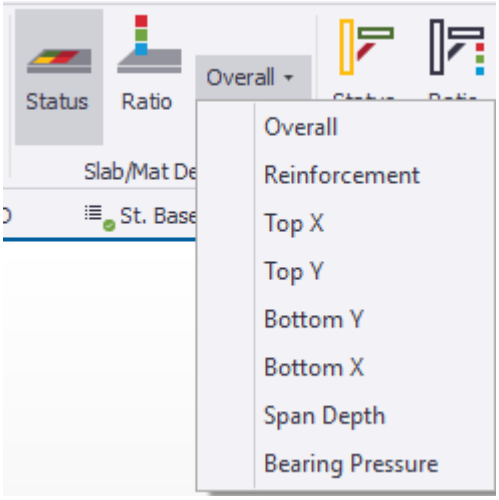
   Color codes display the design status of each slab and mat for the selected filtering option.

   Tekla Structural Designer uses the following classifications:

Status	Description
Error	The slab item currently cannot be designed because an error has occurred.
Beyond Scope	The slab item cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Fail	The slab item has failed one or more design checks.
Warning	Although the slab item has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
Pass	The slab item has passed all design checks.
Other	The slab item has not been designed.

Review slab and mat design ratios

1. Open a 2D view at the level of the slab or mat and activate a **Review View** by clicking  **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA
   - Combined (i.e. Static and RSA together)
3. In the **Slab/Mat Design** group on the ribbon, click **Ratio**.
4. In the same group, click the drop list to display the filtering options.



Filter	Description
Overall	The governing Top X, Top Y, Bottom X, Bottom Y, or span depth result is displayed.
Reinforcement	The governing Top X, Top Y, Bottom X, or Bottom Y result is displayed.
Top X	The result is only displayed for top reinforcement in the X direction.
Top Y	The result is only displayed for top reinforcement in the Y direction.
Bottom X	The result is only displayed for bottom reinforcement in the X direction.
Bottom Y	The result is only displayed for bottom reinforcement in the Y direction.
Span Depth	The span to depth result is displayed.
Bearing Pressure	The bearing pressure result is displayed.

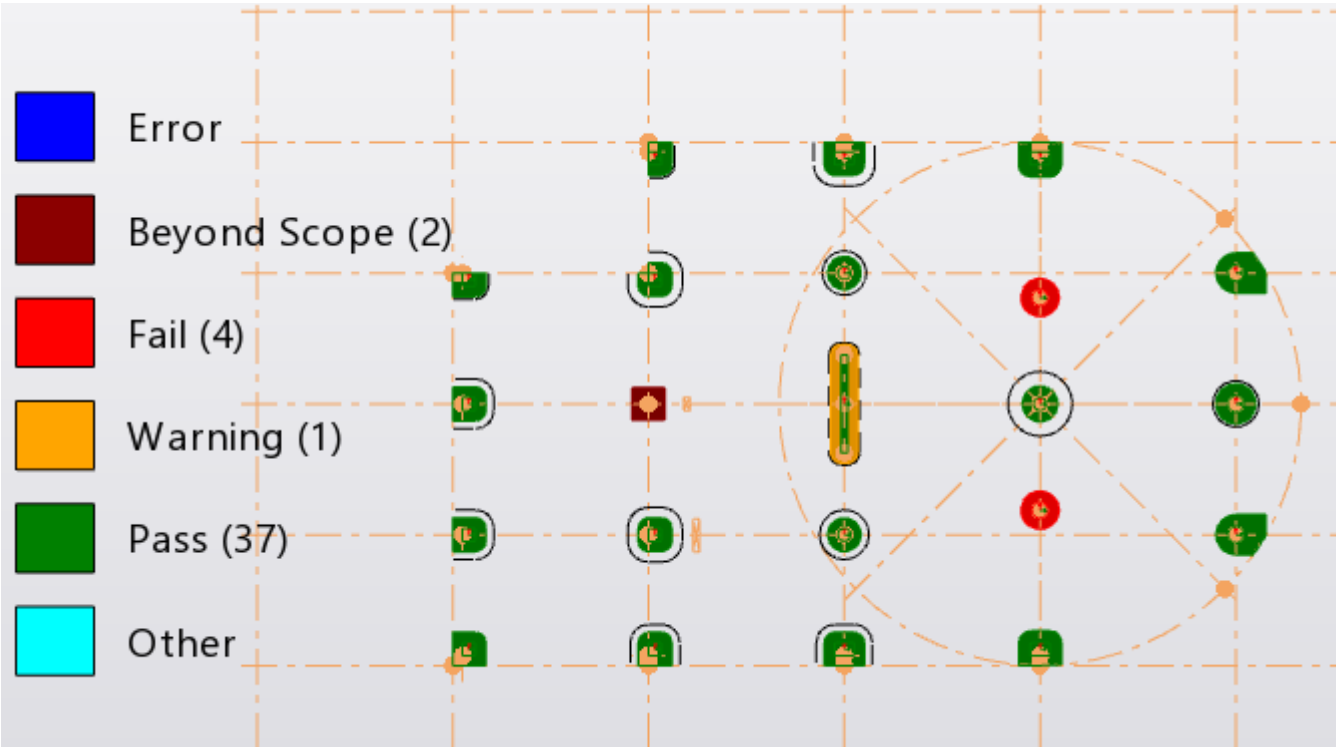
5. Select the filtering option you require.

   Color codes display the design ratio of each slab item for the selected filtering option. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates slab items that are either beyond scope or have yet to be designed.

Review punching check status

NOTE In order to see punching check status you must first switch off Slab Items and Slab Patches in Scene Content.

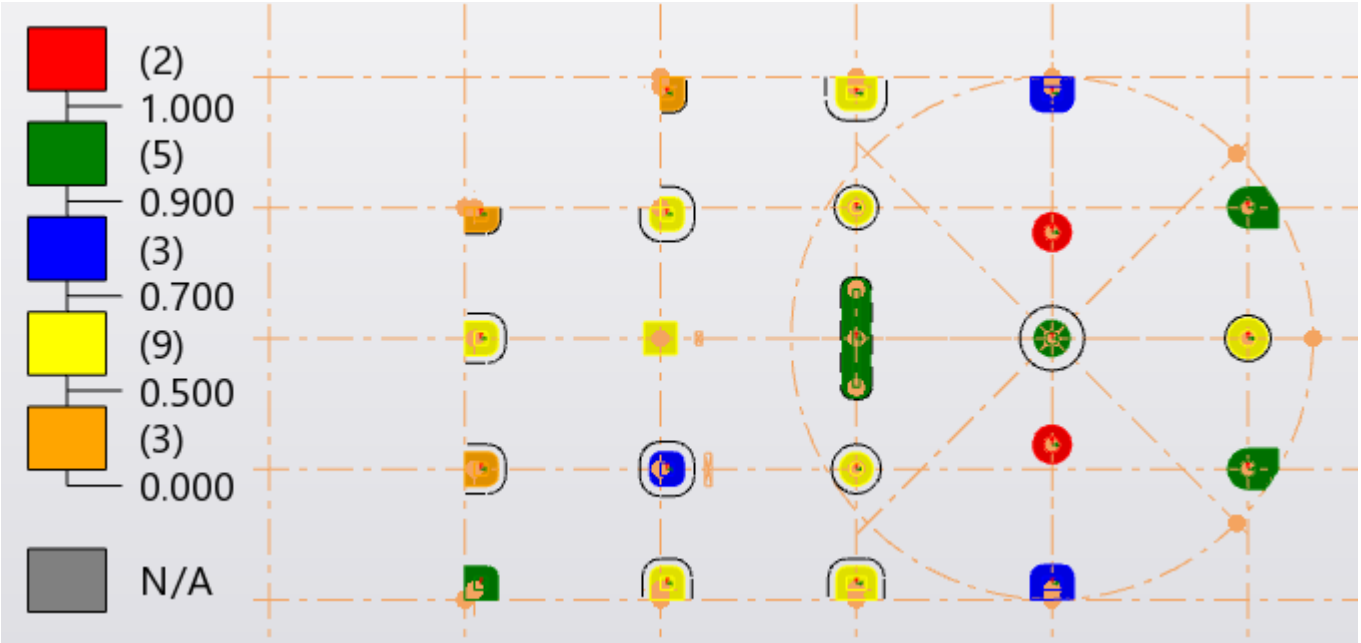
1. Open a 2D view at the level of the slab or mat and activate a **Review View** by clicking **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA
   - Combined (i.e. Static and RSA together)
3. In the **Slab/Mat Design** group on the ribbon, click **Status**. Color codes display the design status of each punching check.



Review punching check ratio

NOTE In order to see punching check ratio you must first switch off Slab Items and Slab Patches in Scene Content.

1. Open a 2D view at the level of the slab or mat and activate a **Review View** by clicking **Review View** in the **Status bar**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type to review.
   - Static
   - RSA
   - Combined (i.e. Static and RSA together)
3. In the **Slab/Mat Design** group on the ribbon, click **Ratio**. Color codes display the design ratio of each punching check.



See also

Design review filters (page 951)

Review design summary tabular results (page 1012)

Design review filters

Design review filters help the user to focus on specific results.

The filters are displayed in the **Properties** window when **Status** or **Ratio** are selected from any group on the **Review** ribbon.

The following filter types are available and can be used individually or in combination:

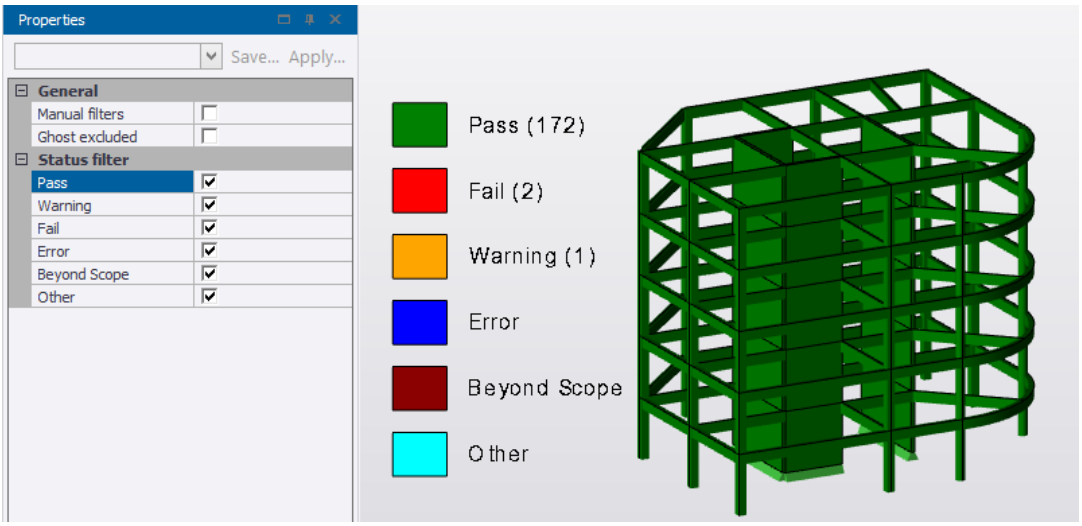
- **Status filter**

- **Utilization ratio filter**
- **Entity type filter**

TIP These filters can be particularly useful when working with large models.

Working with the Status filter

The **Status filter** is always active in the **Properties** window when **Status** is selected from the **Review** ribbon.



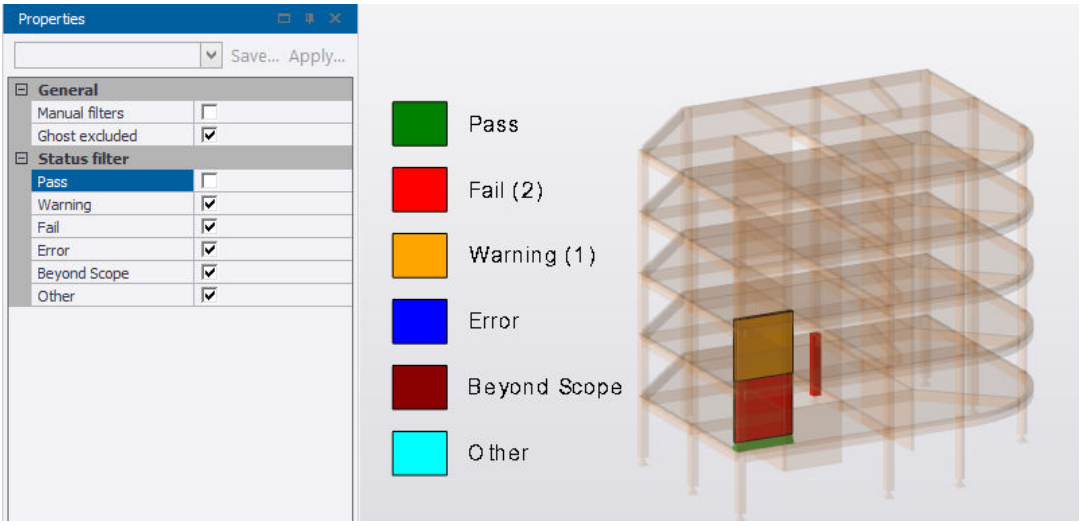
All entities are assigned a design status from the below list:

- **Pass** - The entity has passed all design checks.
- **Warning** - Although the entity has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
- **Fail** - The entity has failed one or more design checks.
- **Error** - The entity currently cannot be designed because an error has occurred.
- **Beyond Scope** - The entity cannot be designed because its design is beyond the scope of Tekla Structural Designer.
- **Other** - The entity has not been designed.

In the above model the legend indicates that two members are failing and another has a warning but their locations are not immediately apparent. Deselecting a status in the **Properties** window causes entities of that status to be hidden in the **Review View**, so in this instance deselecting the Pass status helps to focus on where the problems are occurring.



By also selecting **Ghost excluded** the filtered results can be displayed in relation to the rest of the model.

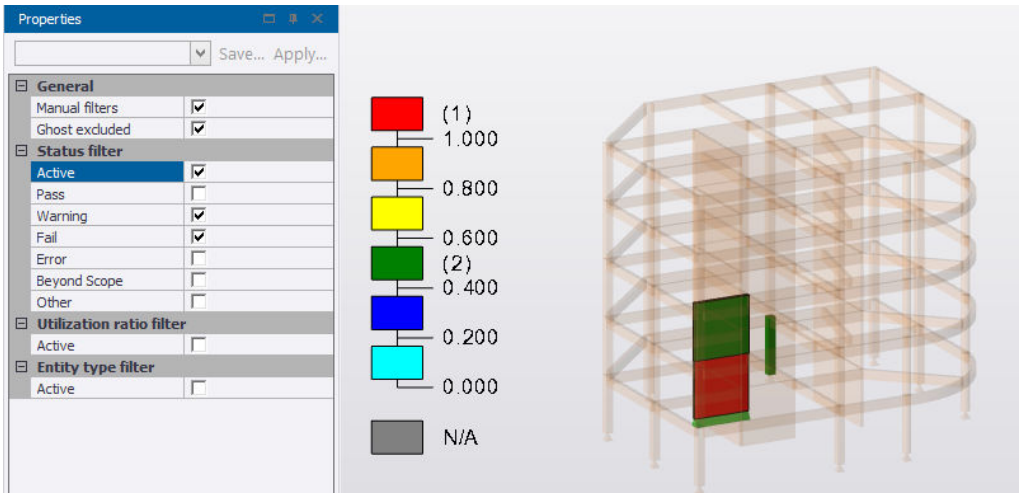


If required, you can also apply the **Status filter** when investigating ratios. This is achieved as follows:

1. From the **Review** ribbon select **Ratio**
2. Select **Manual filters** in the **Properties** window.

   The **Status filter** group is displayed.
3. Select **Active** and choose the statuses by which to filter the display of ratios.

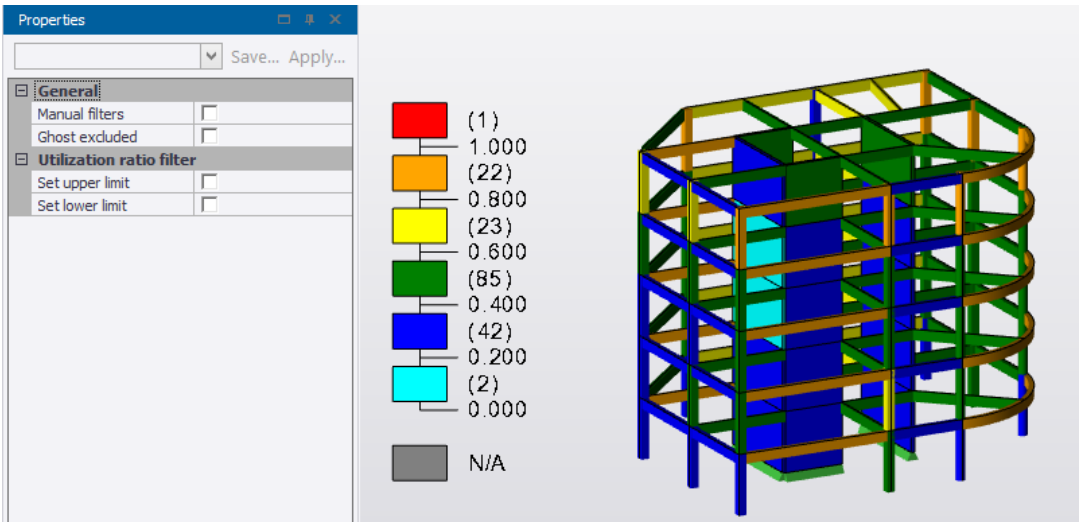
   The result is that ratios are then only displayed for those members in the chosen **Status filter** categories.



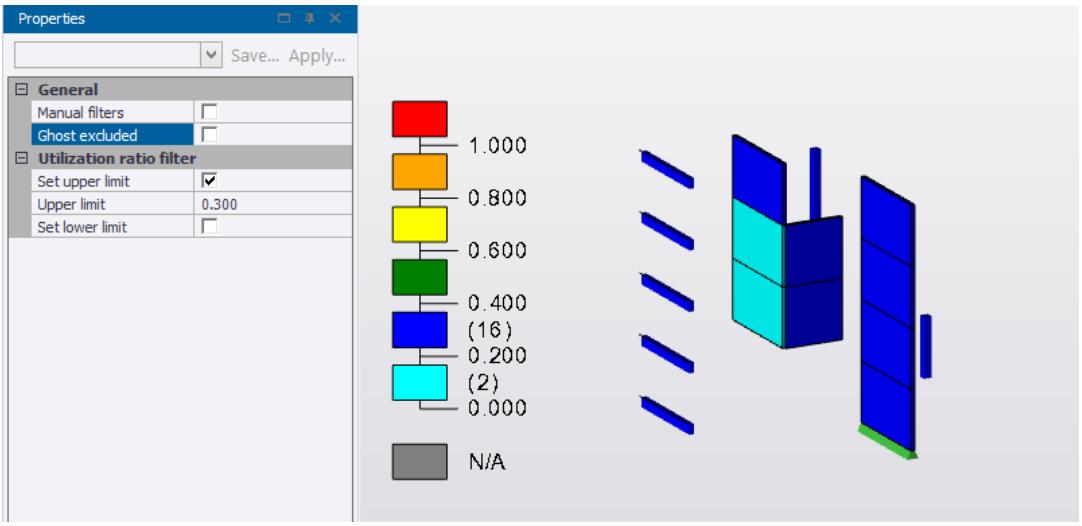
Working with the Utilization ratio filter

All entities that have been designed are assigned a utilization ratio. The **Utilization ratio filter** allows you to display only those entities within a given utilization range.

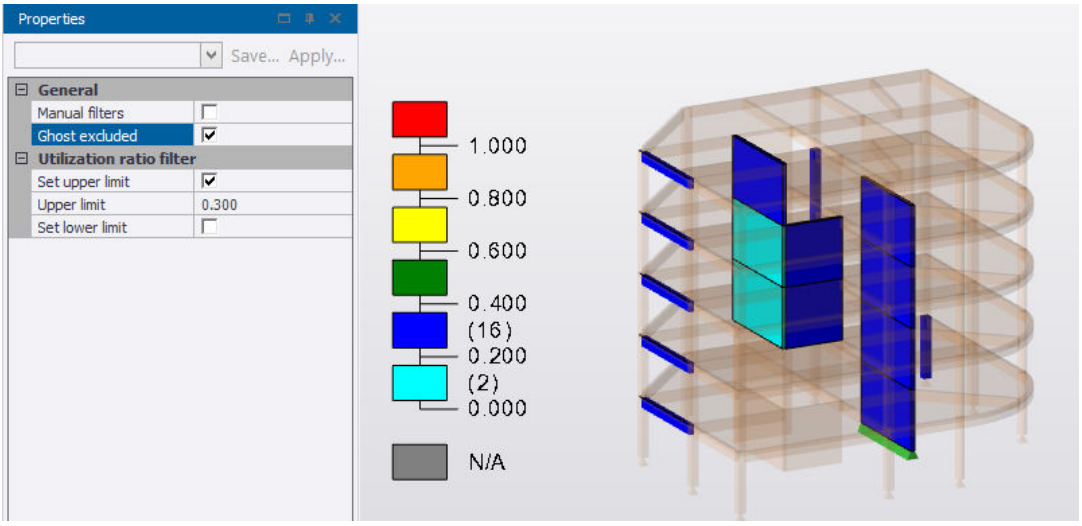
This filter is always active in the **Properties** window when **Ratio** is selected from the **Review** ribbon.



Initially entities with any ratio are displayed, but by selecting upper and/or lower limits, entities outside the defined range are hidden in the **Review View**.



By selecting **Ghost excluded** the filtered results can be displayed in relation to the rest of the model.

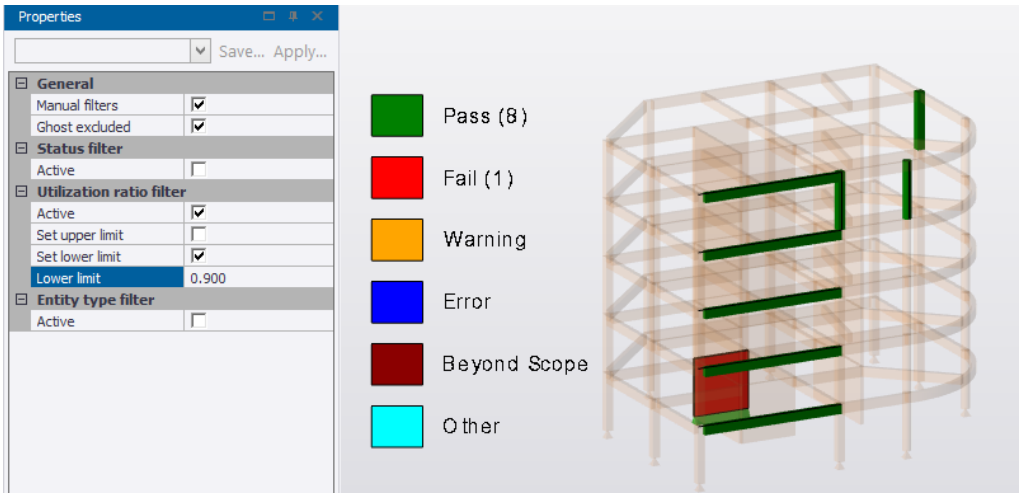


If required, you can also apply the **Utilization ratio filter** when investigating statuses. This is achieved as follows:

1. From the **Review** ribbon select **Status**
2. Select **Manual filters** in the **Properties** window.

   The **Utilization ratio filter** group is displayed.
3. Select **Active** and set the upper and/or lower limits by which to filter the display.

   The result is that statuses are then only displayed for those members within the specified utilization ratio limits.



Working with the Entity type filter

Unless they have been unselected in **Scene Content**, all relevant entity types are initially displayed when either **Status** or **Ratio** are selected from the **Review** ribbon.

You can use the **Entity type filter** to focus on specific types only, it is made active as follows:

1. Select **Manual filters** in the **Properties** window

   The **Entity type filter** group is displayed.

2. Select **Active**

A list of designable entity types is displayed in the **Properties** window.

Deselecting an entity type from the list causes entities of that type to either be hidden in the **Review View**, or displayed as ghosted if **Ghost excluded** is selected.

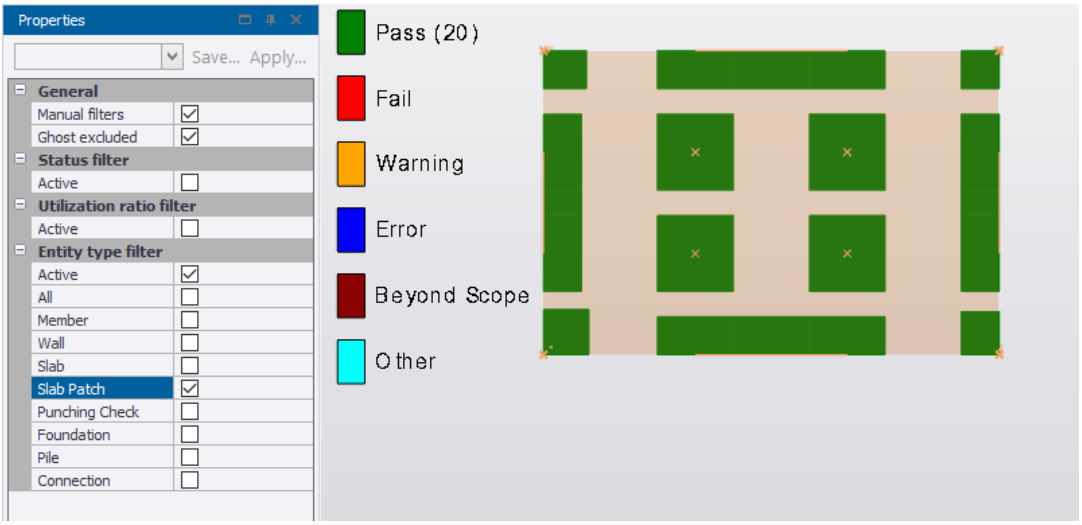
NOTE Clicking 'All' toggles between all entity types being selected, or unselected.

Example:

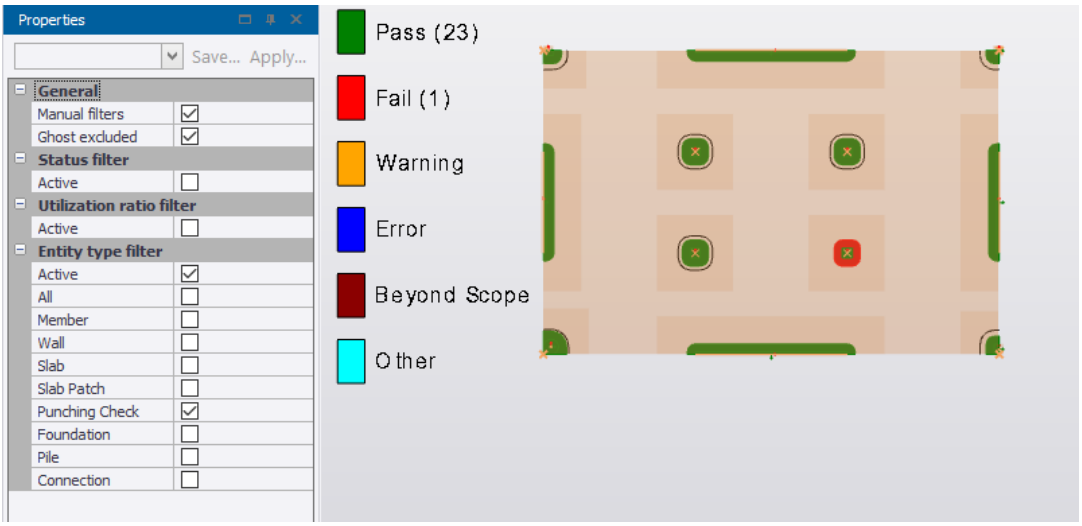
The below view shows slab status at a particular level. Initially, the **Entity type filter** is not active and as a result the **slab**, **slab patch**, and **punching check** statuses are all displayed making interpretation difficult. Note that a Fail is reported, but it is not immediately apparent where it is.



By activating the **Entity type filter** you can investigate the status for specific entity types. The below view is filtered for patches only, which are all passing.



The below view is filtered for punching checks only and clearly reveals where the Fail occurs.



6.2 Review model properties (show/alter state)

The **Show/Alter State** group of commands on the **Review** ribbon allow you to graphically review and modify various model properties. For each of the commands, entities are color coded in accordance with a legend, and the entity color changes to reflect any changes as they are made.

Operations are accessed by first selecting the appropriate command from the ribbon, and then, when necessary selecting an appropriate attribute in the **Properties** window.

In the ribbon, click:

- **Embodied Carbon** to access:
  - (page 979) - to graphically review and override the carbon factors applied to the model
  - (page 1008) - to focus in on high carbon usage and inefficiency
- **Auto/Check Design** to access:
  - (page 961) - to graphically review or modify autodesign settings of a member or slab
  - (page 1004) - to modify the stud auto layout setting of composite beams
- **Diaphragm On/Off** to access:
  - - to graphically review or modify diaphragm settings
  - (page 983) - to review and modify the gravity only setting of beams and columns
  - (page 990) - to review and modify lateral restraint settings of steel beams, columns, trusses and portal frames
- **Fixed/Pinned** to access:

    - (page 978) - to graphically review or modify cantilever ends
    - (page 963) - to graphically review or modify member end fixities
    - (page 997) - to review and modify rotational stiffness settings of beam ends
- (page 964) - to graphically assess the BIM Status, and also to exclude members and panels from the import/export process
- - to graphically review and edit or rationalize panel, patch or foundation reinforcement
- **Section / Material Grade** to access:
    - (page 966) - to graphically copy section sizes
    - (page 966) - to graphically copy material grades
- **Copy Properties** to access:
    - (page 967) - to copy all properties between members
    - (page 980) - to review and copy specified deflection limits between steel beams
    - (page 966) - to graphically copy material grades
    - (page 989) - to copy the connector layout from one beam to others
    - (page 966) - to graphically copy section sizes
    - (page 999) - to copy shear connector properties between composite beams
    - (page 1005)
    - - to review and modify user defined attributes applied to members
    - (page 1009) - to copy web opening properties between members
    - (page 1009) - to copy Westok opening properties between members
- (page 968) - to graphically review and modify report filters
- (page 968) - to graphically review and modify sub structures
- (page 969) - to graphically set whether flange are considered
- (page 969) - to graphically review and modify splice positions within steel columns.
- - to graphically review or apply property sets to members, slab items and roof panels.
- - to graphically review and modify the values of user defined attributes that have been attached to the model.
- **Show/Alter State** to access:
    - (page 971) - to set single span members to be active/inactive
    - (page 967) - to copy all properties between members

- (page 972) - to set concrete members as either cracked or uncracked
- (page 961) - to review or modify autodesign settings
- (page 964) - to graphically assess BIM Status, and also to exclude members/panels from the import/export process
- (page 975) - to modify slenderness settings in each direction for concrete columns and walls
- (page 976) - to review and apply camber
- (page 978) - to mark beam and column ends as cantilevers
- (page 980) - to review and copy specified deflection limits between steel beams
- - to graphically review or modify diaphragm settings
- - to review check status, or specify members to be checked
- (page 981) - to review and apply fire proofing
- (page 963) - to graphically review or modify member end fixities
- (page 983) - to review and modify the gravity only setting of beams and columns
- (page 984) - to review and set imposed load reductions
- (page 986) - to review and set live load reductions (US regional code)
- (page 966) - to graphically copy material grades
- (page 988) - to set the composite beams to which an effective width override applies
- (page 988) - to modify assumed punching shear check positions
- (page 989) - to copy the connector layout from one beam to others
- (page 990) - to review and modify lateral restraint settings of steel beams, columns, trusses and portal frames
- (page 997) - to review and modify rotational stiffness settings of beam ends
- (page 966) - to graphically copy section sizes
- (page 998) - to review check status, or specify members to be checked
- - review or modify SFRS type and direction
- (page 999) - to copy shear connector properties between composite beams
- (page 999) - to review the status and locations where SidePlate connections have been applied to steel beams
- - to review and copy specified size constraints for steel beams, columns, and braces

- (page 1001) - to review construction stages in order to define the content and duration of each stage
- (page 1004) - to modify the stud auto layout setting of composite beams
- - to review check status, or specify members to be checked
- (page 1005) - to copy composite beam transverse reinforcement from one beam to others
- - to review and modify user defined attributes applied to members
- - to review and modify user defined utilization ratios
- (page 1009) - to copy web opening properties between members
- (page 1009) - to copy westok opening properties between members
- - to review check status, specify members to be checked and set the limit
- (page 1011) - to set the members/ancillaries/equipment to which open structure wind loading applies

---

TIP You can use **Ctrl** + **A** as a shortcut to quickly scroll the **Attribute** list.

Modify autodesign settings

You can graphically modify autodesign settings of members and slabs from a **Review View**.

To modify the autodesign settings of a member or slab, do the following:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, either:
   a. Click **Auto/Check Design**, or,
   b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

   Members and slabs are color coded according to their auto design setting.
3. In the **Properties** window, select the appropriate mode.
4. Do one of the following:
   - Click an individual entity to modify its auto design setting.
   - Hold down the left mouse button and drag a box from left to right to modify the auto design setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the auto design setting of all members within the box or cut by the box.

Review and modify diaphragm settings

The **Diaphragm On/Off** command on the **Review** tab allows you to select whether the slab items, roof panels, and nodes in the model are included

Review the diaphragm settings

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or
   b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

   Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.

Modify the diaphragm settings of slab items or roofs

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or
   b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

   Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.
3. In the **Properties** window, set **Entity type** to **Slab Item** or **Roof**.
4. Do one of the following:
   - Click an individual slab item or roof panel to switch between including and excluding the element from the diaphragm.
   - Hold down the left mouse button and drag a box from left to right to switch between including and excluding slab items and roof panels within the box from the diaphragm.
   - Hold down the left mouse button and drag a box from right to left to switch between including and excluding slab items and roof panels within the box or cut by the box from the diaphragm.

Include or remove solver nodes from the diaphragm

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or
   b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

   Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.
3. In the **Properties** window, set **Entity type** to **Solver Node**.
4. Do one of the following:
   - Click an individual slab item or roof panel to switch between including and excluding the element from the diaphragm.
   - Hold down the left mouse button and drag a box to switch between including and excluding the nodes within the box in diaphragm.

See also

Managing diaphragm action in roof panels and slabs (page 622)

Modify end fixity

The **Fixed/Pinned** command allows you to graphically review and modify the end fixity of all members in the model.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, select **Fixed/Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Fixed/Pinned** is selected.
5. With the **Mode** set as **Toggle**, do one of the following:
   - Click near one end of an individual member to switch the end fixity at that end between the valid types, (the tooltip indicates the current fixity at that end).
   - Click near the middle of an individual member to switch the end fixity at both ends between the valid types, (the tooltip indicates the current fixities at both ends).
     - If both ends currently have the same end fixity then both are changed to the next valid type.
     - If the ends currently have different end fixities then both are changed to pinned (regardless of their current fixity).

- Hold down the left mouse button and drag a box from left to right to alter the fixity settings of all members within the box.
- Hold down the left mouse button and drag a box from right to left to alter the fixity settings of all members within the box or cut by the box.

6. Alternatively, with the **Mode** set as **Set On**, select the **FixityType** required and then click near the end or middle of an individual member or drag a box, as above to apply the chosen fixity.

NOTE If you want to apply or remove a cantilever end, first select **Cantilever end** in the **Attribute** list. For details, see: Apply cantilever ends (page 978).

Modify BIM status

The **BIM Status** command allows you to graphically modify the BIM status and to exclude members and panels from the BIM import or export process.

To include or exclude members and panels from the BIM process, do the following:

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **BIM Status**.
3. Do one of the following:
   - Click an individual member or panel once to exclude it or twice to re-include it.
   - Hold down the left mouse button and drag a box from left to right to modify the exclude setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the exclude setting of all members within the box or cut by the box.

Copy or modify slab and foundation reinforcement

You can graphically review and modify the bar/mesh size and spacing applied in each layer and direction for slab items, mats, patches and isolated foundations. Tekla Structural Designer also allows you to copy reinforcement from one slab item, mat, patch or isolated foundation to another.

TIP If you open two **Review View**s side by side, you can then use the first one to modify the reinforcement while displaying the design status in the second one.

This way, each change you make to the reinforcement immediately updates the design status.

Copy reinforcement

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, either:
   a. Click **Auto/Check Design**, or,
   b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

   Members and slabs are color coded according to their auto design setting.
3. Ensure that auto design is set off for the slab items and patches that you want to modify.
4. On the **Review** tab, click **Slab / Foundation Reinforcement**.
5. Go to the **Properties** window.
6. Set **Entity type** to either **Slab Item**, **Slab Patch** or **Isolated Foundation**, depending on the reinforcement that you want to copy.
7. In **Reinforcement Direction**, select the direction that you want to modify.
8. In **Surface**, select the slab layer that you want to modify.
9. Click the slab item or patch whose reinforcement you want to copy.
10. Click the slab items or patches to which you want to copy the reinforcement.

    The slab reinforcement is copied to the selected slab items or patches.

TIP To ensure that the updated reinforcement is sufficient, in the **Slab/Mat Design** group, click **Status**.

Modify reinforcement

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, either:
   a. Click **Auto/Check Design**, or,
   b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

   Members and slabs are color coded according to their auto design setting.

3. Ensure that auto design is set off for the slab items and patches that you want to modify.
4. On the **Review** tab, click **Slab / Foundation Reinforcement**.
5. Go to the **Properties** window.
6. Set **Entity type** to either **Slab Item**, **Slab Patch** or **Isolated Foundation**, depending on the reinforcement that you want to modify.
7. In **Reinforcement Direction**, select the direction that you want to modify.
8. In **Surface**, select the slab layer that you want to modify.
9. If you want to modify bars, in **Bar Parameters**, select the properties that you want to modify.
10. In **Apply**, select the bar properties that you want to apply to the slab.
11. Click the slab item or patch whose reinforcement you want to update.

TIP To ensure that the updated reinforcement is sufficient, in the **Slab/Mat Design** group, click **Status**.

Copy section sizes

If necessary, you can graphically copy the section size applied to a member to other members in the model. For more information, see the following paragraphs.

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Section**.
5. Set **Mode** to **Copy**.
6. In **Material type**, select the desired material type.

NOTE You can use a characteristic filter to reduce the elements displayed in the model.

---

7. In the model, click the element from which you want to copy a section size.
8. Click the elements to which you want to copy the section size.

   The section size is copied to the selected elements.

Copy material grades

If necessary, you can graphically review the material grade applied to the members in your model. Tekla Structural Designer also allows you to copy the grade of one member to another member with the same material type . For more information, see the following paragraphs.

RESTRICTION You can only copy material grades to and from steel, cold formed, cold rolled, concrete, or timber members.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Material Grade**.
5. In **Material Type**, select the desired material type.

   TIP If necessary, use a characteristic filter to reduce the elements displayed in the **Review View**.

6. Set **Mode** to **Copy**.
7. Click the element whose grade you want to copy.
8. Click the elements to which you want to copy the grade.

   The material grade is copied to the selected elements.

Copy properties

The **Copy Properties** command allows you to graphically copy a specified element parameter (such as web openings, shear connectors, or transverse reinforcement) from a source member to other valid target members. To copy properties from one member to another, see the following instructions.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Copy Properties**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select the parameter that you want to copy.
5. In the model, click the member from which you want to copy the parameter.
6. Do one of the following:
   - Click an individual member to which you want to copy the parameter.

- Hold down the left mouse button and drag a box around multiple members to which you want to copy the parameter.

Tekla Structural Designer copies the parameter to the selected members.

Review and modify member filters

The **Report Filter** command allows you to graphically review and modify report filters that you have defined for the members in your model. The **Report Filter** command remains unavailable until you have defined a member filter.

TIP To define a report filter, do the following:

1. Go to the **Report** tab.
2. In the **Filters** group, click **Members**.
3. Define the member filter according to your needs.

---

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Report Filter**.
3. Do one of the following:
   - Click an individual member to switch between including and excluding the member from the selected member filter.
   - Hold down the left mouse button and drag a box from left to right to switch between including and excluding the members within the box from the selected member filter.
   - Hold down the left mouse button and drag a box from right to left to switch between including and excluding the members within the box or cut by the box from the selected member filter.

Review sub structures

The **Sub Structures** command allows you to graphically create, review and modify sub structures for modeling purposes. The command can be especially useful in large models, as individual sub structures can then be differentiated by color and worked on in separate views.

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Sub Structures**.

   The elements in the model are color coded based on whether they belong to sub structures.

See also

Create and manage sub structures (page 1296)

Review concrete beam flanges

The **Concrete Beam Flanges** command allows you to graphically review the flanges of concrete beams. See detailed instructions on how to review conrete beam flanges in the following paragraphs.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Concrete Beam Flanges**.

   Each concrete beam is color coded to indicate if its flanges are considered and flange widths determined.

Use of beam flanges

Review and modify column splice positions

The **Column Splices** command allows you to graphically review and modify splice positions within steel and timber columns. For detailed instructions on how to review and modify splice positions, see the following steps:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Column Splices**.

   All potential splice locations are color coded to indicate if they are on or off.
3. Do one of the following:
   - Click a potential splice location to alter its setting between on or off.
   - In a 2D view, hold down the left mouse button and drag a box to alter the setting of multiple splices simultaneously.

Review and apply property sets

The command provides a means to graphically review or apply property sets to entities.

- Review where property sets have been applied (page 1287)
- Apply property sets to existing entities (page 1286)

Copy or modify user-defined attributes

If necessary, you can graphically review and modify the values of attributes that have been attached to the model.

Show/alter state

Show/Alter State allows you to efficiently review and modify various model attributes. For each of the attributes entities are color coded in accordance with a legend. The entity color changes to reflect changes as they are made.

In the **Properties** window, you can use filters for most **Show/Alter State** categories. The filters allow you to control both the visibility of entities by their current setting and their adjustability by their type and characteristic. Inactive but visible entities are shaded a light transparent gray whose opacity you can also control, if necessary.

To access **Show/Alter State**, do the following:

1. Open a **Review** view.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. Choose the attribute required from the **Attribute** list:

Attribute	Description
(page 971)	set single span members to be active/inactive
(page 967)	copy all properties between members
(page 972)	set concrete members as either cracked or uncracked
(page 961)	review or modify autodesign settings
(page 964)	modify BIM status, or exclude members/panels from BIM import/export
(page 975)	modify slenderness settings in each direction for concrete columns and walls
(page 976)	review and apply camber
(page 978)	mark beam and column ends as cantilevers
(page 979)	allows you to graphically review and override the carbon factors applied to the model
(page 980)	review and copy specified deflection limits between steel beams
(page 962)	select the slab items, roof panels, and nodes to be included in diaphragms
	review check status, or specify members to be checked
(page 981)	review and apply fire proofing
(page 963)	review and modify member end fixity
(page 983)	review and modify the gravity only setting of beams and columns
(page 984)	review and set imposed load reductions
(page 986)	review and set live load reductions (US regional code)
(page 966)	review and copy the material grade between members
(page 988)	set the composite beams to which an effective width override applies

(page 988)	modify assumed punching shear check positions
(page 989)	copy the connector layout from one beam to others
(page 990)	review and modify lateral restraint settings of steel beams, columns, trusses and portal frames
(page 997)	review and modify rotational stiffness settings of beam ends
(page 966)	review and copy section sizes between members
(page 998)	review check status, or specify members to be checked
	review or modify SFRS type and direction
(page 999)	copy shear connector properties between composite beams
(page 999)	review the status and locations where SidePlate connections have been applied to steel beams
	review and copy specified size constraints for steel beams, columns, and braces
(page 1001)	review construction stages in order to define the content and duration of each stage
(page 1004)	modify the stud auto layout setting of composite beams
	review check status, or specify members to be checked
(page 1005)	modify the transverse reinforcement of composite beams
(page 970)	review and modify user defined attributes applied to members
	review and modify user defined utilization ratios
(page 1008)	allows you to graphically focus in on high carbon usage and inefficiency.
(page 1009)	copy web opening properties between members
(page 1009)	copy Westok opening properties between members
	review check status, specify members to be checked and set the limit
(page 1011)	set the members/ancillaries/equipment to which open structure wind loading applies

TIP You can use **Ctrl** + **A** as a shortcut to quickly scroll the **Attribute** list.

Modify active / inactive settings

Initially all members are active. If a member is set inactive it does not participate in the analysis but any load applied to (or decomposed) to it is still accounted for.

Only single span beams, braces, analysis elements, purlins and rails have the potential to be inactive. They each therefore have an **Active** setting in the properties which defaults to 'on', but can be unchecked. In the **Review View** these members are color coded allowing the same setting to be toggled graphically.

NOTE Members cannot be inactive if:

- They are multi-span
- They support another member (active or inactive)

To modify the active setting of a member graphically do the following:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State** .
3. Go to the **Properties** window.
4. In the **Attribute** list select **Active**.

   Members are color coded according to their active setting.
5. In the **Properties** window, select the appropriate mode.
6. Do one of the following:
   - Click an individual entity to modify its active setting.
   - Hold down the left mouse button and drag a box from left to right to modify the active setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the active setting of all members within the box or cut by the box.

Modify assumed cracked settings

You can graphically review and modify the assumed cracked setting of the concrete beams, columns and walls in a Review View using Show/Alter State - *Assume cracked*.

See also

Cracked, partially cracked, and uncracked concrete members (page 650)

Review assumed cracked settings

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Review**
6. Set the **Entity type**

   The member color coding indicates the cracked, partially cracked or uncracked status.

NOTE By hovering the cursor over an individual member or panel a tooltip is displayed. This reports the cracked status, and also the EI_{major}, EI_{minor} values. For partially cracked members/panels the percentage is also given.

Set as fully cracked or uncracked

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Set On**
6. Set **Assume cracked** to one of the following:
   - **Yes** - click or box around concrete member(s) to set to cracked.
   - **No**- click or box around concrete member(s) to set to uncracked.

   The member color coding updates to reflect the cracked or uncracked status.
7. Select **Entire member** to apply the new setting to all spans/stacks, or leave it unselected to update individual spans/stacks only.
8. Click or box around those concrete member(s) you want to set.

   The member color coding updates to reflect the new cracked or uncracked status.

Set as partially cracked

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Set On**
6. Set **Assume cracked** to **Partially**
7. Select **Set Percentage**
8. Enter the **Percentage** required.
9. Select **Entire member** to apply the new percentage to all spans/stacks, or leave it unselected to update individual spans/stacks only.
10. Click or box around those concrete member(s) you want to apply the percentage to.

    The member color coding updates to reflect the partially cracked status.

Toggle assumed cracked settings

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Toggle**
6. Click or box around those concrete member(s) you want to toggle.

   The assumed cracked status of the members toggles between cracked, partially cracked, and uncracked.

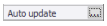
Review Wall Stress

To review/update the cracked status of all wall panels to be compatible with the stress levels for the loadcases/ combinations that have been analyzed, proceed as follows:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Review Wall Stress**.
6. Check the yellow command prompt to confirm which analysis type is being used for the review, (this will be the type currently specified in **Design Settings>Analysis**).

```
Review Wall Stress Assume cracked (First-order linear): Select entity to Set compatible
```

7. Set the **Result type** to control which factors to use in combinations for the review.
   - **Strength** factors, or
   - **Service** factors
8. Set the **Stress type** to be considered.
   - **Max tension - Y**, or
   - **In-plane tension - Y**
9. Set the **Stress threshold** above which walls should crack. The existing cracked/uncracked status of each panel is displayed, along with an indiation of whether the stress threshold has been exceeded.

10. To make panel status compatible with the stress threshold for one or more panels:
    a. click an individual panel, or
    b. box around multiple panels, or
    c. to make all panels compatible, click 

then click the Auto update button

    Panel status is made compatible, i.e.

    - If uncracked and the stress threshold has been exceeded, the status is changed to cracked.
    - If partially cracked and the stress threshold has been exceeded, the status is changed to cracked.
    - If cracked and the stress threshold has not been exceeded, the status is changed to uncracked.

Review Wall Stress - limitations and assumptions

NOTE The following limitations and assumptions apply to **Review Wall Stress**

1. Only meshed wall panels are considered, mid-pier walls are excluded.
2. The analysis type specified in **Design Settings>Analysis** determines which set of analysis results are used for the review.
3. For the above analysis type, the loadcases and combinations considered are those from the **last performed building analysis of that type.**
    - This gives you the flexibility to review the stresses for a subset of loadcases/combinations if required.
    - By running Analyse All you can ensure all active gravity, lateral and seismic (not seismic RSA) combinations have been considered.
        - Stresses are available for seismic but not for seismic RSA combinations - these are beyond scope. (Review Wall Stress is more particularly aimed at determining the cracked status of panels under wind loading).
4. Chasedown results are **not** considered.
5. Axial load reductions are always taken into account.
6. The maximum stress value from all nodes for all analysed cases/ combinations is reported in a tooltip for each panel. This value is used to determine if the stress threshold has been exceeded.
    - The reinforcement content in the panel is not considered in determining the stress value.
7. The loading droplist in the status bar does not filter the status for individual cases/combinations.

Modify slenderness settings

The **Braced/Bracing** command provides a means to graphically review and modify the slenderness settings in each direction for concrete columns and walls.

To modify slenderness settings, do the following:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Braced/Bracing**.
5. Select the direction under consideration.

   Members are color coded according to their slenderness setting in the selected direction.
6. Do one of the following:
   - Click an individual column or wall to modify its slenderness setting between braced and bracing.
   - Hold down the left mouse button and drag a box from left to right to modify the slenderness setting of all concrete columns and walls within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the slenderness setting of all concrete columns and walls within the box or cut by the box.

Review and set camber

The Show/Alter State **Camber** attribute allows you to graphically set and review the camber properties applied to steel beams.

Review camber

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Review**

   The member color coding indicates where and how camber has been applied to beams.

Set on

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Set On**
6. Set **Apply as** to **Value/Proportion of span/Proportion of deflection** as required.
   a. Enter the value or proportion information required to definer the camber.
7. Enter any beam length, or web thickness limits that you want to use to limit when the camber should be applied.
8. Select **Entire member** to apply the new setting to all spans, or leave it unselected to update individual spans only.
9. Click or box around those beam(s) you want to set.

   The member color coding updates to reflect the details that have been applied.

   - Members color coded as 'On' have the camber that is shown in the properties window.
   - Members color coded as 'Other details' have other camber properties. These are displayed in a tooltip when you hover the cursor over each member.

Set off

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Set Off**
6. Select **Entire member** to switch off camber for all spans, or leave it unselected to update individual spans only.
7. Click or box around those beam(s) for which you want to switch off camber.

   The member color coding updates to reflect the details that have been applied.

Toggle camber settings

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Toggle**
6. Click or box around those beam(s) you want to toggle.

   The camber setting toggles between on and off.

Copy camber settings

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Copy**
6. In the model, click the beam from which you want to copy the camber properties.
7. Click the beam to which you want to copy the camber properties. The camber properties are copied to the selected beam.
8. Continue copying the same camber properties by clicking the desired beams; or press **Esc** to select a new beam to copy the camber properties from; or press **Esc** twice to finish.

Apply cantilever ends

The **Cantilever end** command allows you to graphically apply cantilever ends where required to beams and columns.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Fixed/Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Cantilever end**.
5. Set **Mode** to **Set On**, **Set Off**, or **Toggle** as required.
6. Do one of the following:
   - Click an individual member end to set/unset a cantilever end as per the **Mode**.

- Hold down the left mouse button and drag a box from left to right to alter the cantilever end settings of all members entirely within the box.
- Hold down the left mouse button and drag a box from right to left to alter the cantilever end settings of all members within the box or cut by the box.

Review carbon factors

The Show/Alter State **Carbon factors** attribute allows you to graphically review and override the carbon factors applied to the model.

Review carbon factors

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Embodied Carbon**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Carbon factors**.
5. Set **Mode** to **Review**.
6. Set the **Carbon source category** to be reviewed

> **NOTE** Any entity in the model can have *multiple* carbon sources that relate to it.

Set override

> **NOTE** In general, rather than applying a carbon factor override you should consider adding it instead to the list of embodied carbon factors that can be applied.

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Carbon factors**.
5. Set **Mode** to **Set override**.
6. Set the **Carbon source category** to be overridden.
7. Enter the required **Carbon factor**.
8. In the model, click the entities to which the override applies.
9. Continue applying overrides or press **Esc** to finish.

Remove override

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Carbon factors**.
5. Set **Mode** to **Remove override**.
6. Set the **Carbon source category** override to be removed.
7. In the model, click the entities from which the override is to be removed.
8. Continue removing overrides or press **Esc** to finish.

Review and copy deflection limits

You can graphically review specified deflection limits and copy the limits from one steel, timber, or cold formed beam to others.

Review deflection limits

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Deflection Limits**.
5. Set **Mode** to **Review**.
6. Set **Load type** to the load type that you want to review.

Copy deflection limits

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Deflection Limits**.
5. Set **Mode** to **Copy**.
6. In the model, click the beam from which you want to copy the deflection limits.
7. Click the beam to which you want to copy the deflection limits.

   The deflection limits are copied to the selected beam.

8. Continue copying the limits by clicking the desired beams or press **Esc** to finish.

Review and modify drift checks

You can graphically review drift check results and set the members to be checked for columns and walls.

Review drift checks

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Drift check**.
5. Set **Mode** to **Review**.

Set drift checks on or off

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Drift check**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Apply fire proofing

The Show/Alter State **Fire proofing** attribute allows you to graphically set and review the fire proofing applied to 1D member types.

Review fire proofing

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.

3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Review**

   The member color coding indicates the fire proofing status. If working to Eurocodes the beams that have been set to have a fire resistance check are also indicated.

   NOTE By hovering the cursor over an individual member or panel a tooltip is displayed. This reports the cracked status, and also the EI_{major}, EI_{minor} values. For partially cracked members/panels the percentage is also given.

Set as protected

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Set On**
6. Set **Checks** to **Protected**
7. Select which of the protection properties (exposure, shape, thickness & density) you want to apply proofing of the
8. Select **Entire member** to apply the new setting to all spans/stacks, or leave it unselected to update individual spans/stacks only.
9. Click or box around those 1D member(s) you want to set.

   The member color coding updates to reflect the details that have been applied.

   - Members color coded as 'On' have the fire proofing that is shown in the properties window.
   - Members color coded as 'Other details' have other fire proofing properties. These are displayed in a tooltip when you hover the cursor over each member.

Set check for fire resistance (Eurocode only)

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Set On**
6. Set **Checks** to **Check for fire resistance**
7. Select **Set exposure** to specify whether the member is exposed on all sides or 3 sides only.
8. Select **Entire member** to apply the new setting to all spans, or leave it unselected to update individual spans only.
9. Click or box around those beam(s) you want to set.

   The member color coding updates to reflect the details that have been applied.

   - Members color coded as 'On' have the fire resistance that is shown in the properties window.
   - Members color coded as 'Other details' have other fire resistance properties. These are displayed in a tooltip when you hover the cursor over each member.

Toggle fire proofing settings

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Toggle**
6. If working to Eurocodes, choose whether you want to toggle the protection or fire check settings.
7. Click or box around those 1D member(s) you want to toggle.

   The member properties toggle between on and off.

Modify gravity only settings

If necessary, you can review and modify the gravity only setting of the beams and columns in your model. For more information, see the following paragraphs.

CAUTION If you specify a member as 'gravity only' it will only be designed/checked for active gravity combinations, all other active combinations are ignored. For more information, see: Design individual members for gravity only (page 796)

If necessary, you can review and modify the gravity only setting of the beams and columns in your model as follows;

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Gravity only**.

   All members are color coded to indicate their gravity only setting.
5. Do one of the following:
   - Click a steel beam or column to set its gravity only setting on or off.
   - Hold down the left mouse button and drag a box from left to right to set on or off the gravity only setting of all steel beams and columns within the box.
   - Hold down the left mouse button and drag a box from right to left to set on or off the gravity only setting of all steel beams and columns within the box or cut by the box.

See also

Design individual members for gravity only (page 796)

Review and set imposed load reduction

By selecting the **Imposed load reduction** attribute in **Show/Alter State** you can graphically review and set imposed load reduction parameters.

TIP Graphical editing will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review load reduction attributes

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Review**.
6. Set the **Load reduction attribute** as required:
   - Applied imposed load reduction factor
   - Reduce imposed loads by
   - Count the floor as being supported*
   - Number of floors carried*

* Not applicable if working to the Australian regional code.

The view updates to display the selected load reduction attribute information.

TIP Hover the cursor over a member to see all the above attributes in a tooltip.

Set 'Reduce imposed loads by' percentage for beams/slabs

To graphically reduce imposed loads on beams and slabs by a percentage proceed as follows:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Reduce imposed loads by**.
7. Enter the percentage.
8. Set the **Entity type** that you want to modify.
9. Click or box around entities in the Review view to apply the new percentage. The view updates to display the revised reduction percentages.

TIP Hover the cursor over entities that don't have the currently selected percentage reduction to see their attributes in a tooltip.

Set 'Count floor as being supported' for columns/walls

NOTE Not applicable if working to the Australian regional code.

To graphically set the 'Count floor as being supported' property at specific levels for columns and walls proceed as follows:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Set**.

6. Set the **Load reduction attribute** to **Count the floor as being supported**.
7. Check or uncheck the **Count the floor as being supported** checkbox as required.
8. Click on a column or wall just above/below a specific floor to set/unset the **Count the floor as being supported** property at that level for the selected column or wall.

TIP Set the **Entity type** to **Any**, **Column**, or **Wall** to work on specific entities only.

Set 'Assume extra floors supported' for columns/walls

NOTE Not applicable if working to the Australian regional code.

To graphically set the 'Assume extra floors supported' property for columns and walls proceed as follows:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Assume extra floors supported**.
7. Enter the **Assume extra floors supported** number.
8. Click or box around columns and walls to apply the new value.

Review and set live load reduction

By selecting the **Live load reduction** attribute in **Show/Alter State** you can graphically review and set live load reduction parameters.

TIP Graphical editing will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review load reduction attributes

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Live load reduction**

5. Set **Mode** to **Review**.
6. Set the **Load reduction attribute** as required:
   - Applied live load reduction factor
   - KLL
   - Reduce live loads by
7. If reviewing the applied live load reduction factor, choose the loadcase type to review:
   - Live
   - Roof Live

   The view updates to display the selected load reduction attribute information.

TIP Hover the cursor over a member to see all the above attributes in a tooltip.

Set 'Reduce live loads by' percentage for slabs

To graphically reduce live loads on slabs by a percentage proceed as follows:

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Live load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Reduce live loads by**.
7. Enter the percentage.
8. Click or box around slabs in the Review view to apply the new percentage. The view updates to display the revised reduction percentages.

TIP Hover the cursor over slabs that don't have the currently selected percentage reduction to see their attributes in a tooltip.

Set KLL factor

To graphically set the KLL property proceed as follows:

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **Live load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **KLL**.
7. Enter the KLL value.
8. Uncheck **Entire member** if you only want the value to apply to individual stacks/spans.
9. Click or box around entities to apply the new value.

TIP Set the **Entity type** to **Beam**, **Column**, or **Wall** to work on specific entities only.

Override effective width

The Show/Alter State **Override Effective Width** attribute allows you to graphically set the composite beams to which an effective width override applies.

NOTE For the width to be overridden the **Update effective width prior to design check** setting must be selected in **Design > Settings > Steel > Composite beam**.

---

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Override Effective Width** is selected.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the composite beams to which an effective width override will apply.
    a. If **Mode** is **Set On** - the setting is switched on
    b. If **Mode** is **Set Off** - the setting is switched off
    c. If **Mode** is **Toggle** - the setting toggles between on and off
7. Continue applying the setting or press **Esc** to finish.

Modify punching shear check position

If necessary, you can graphically modify the assumed punching shear check position. This allows you to determine the loaded perimeter when the perimeter is close to a free edge.

1. Open a view and switch the view regime to  **Review View**.

2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Punch check position**.
5. To change the punching shear check position, do one of the following:

To	Do this
Switch the punching shear check position between Edge Z and Internal	• Click the Z edge of a punching check item.
Switch the punching shear check position between Edge Y and Internal	• Click the Y edge of a punching check item.
Switch the punching shear check position between Corner, Edge Z, Edge Y and Internal	• Click a corner of a punching check item.

NOTE Clicking an internal punching check item does not change the punching shear check position.

TIP You can also box around multiple punching check item to switch their position. Do one of the following:

- Hold down the left mouse button and drag a box from left to right to switch the position setting of all punch check items within the box.
- Hold down the left mouse button and drag a box from right to left to switch the position setting of all punch check items within the box or cut by the box.

Copy quick connector layout

You can graphically copy the connector layout from one beam to another. For more information, see the following paragraphs.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Quick Connector Layout**.
5. Click on the beam with the layout that you want to copy (must be valid).

   The composite beams in the model are color coded to indicate the source and valid targets.
6. To select the beams to apply the layout to, do one of the following:
   - Click an individual valid target beam.
   - Hold down the left mouse button and drag a box from left to right to totally enclose the target beams.

- Hold down the left mouse button and drag a box from right to left to select target beams within the box or cut by the box.

The connector layout is copied from the source to the target beam(s).

Review and modify restraints

By selecting the **Restraints** attribute in **Show/Alter State** you can graphically review and modify continuous and discrete restraints for beams, columns, trusses and portal frames.

TIP Graphical editing of restraints will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review restraints

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Review**. For details of the restraint type choices etc. that are presented in Review mode, see the **Options overview** topics below.
6. Set the **Continuous** box as required:
   - Checked: shows continuous restraints, each sub-member being colour coded in accordance with the legend.
   - Unchecked: shows discrete restraints, each internal node at which other members connect being colour coded in accordance with the legend.
7. Set the Restraint Type that you want to review.
8. Set the Entity type that you want to review.

Toggle restraints

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Toggle**. For details of the restraint type choices etc. that are presented in Toggle mode, see the **Options overview** topics below.
6. Set the **Entity type** that you want to modify.

7. Leave the **Continuous** box checked to set continuous restraints along sub-members, or uncheck it to set discrete restraints at those points where other members connect.
8. Set the **Restraint Type** as required.
9. In the model:
   - click the member at the location where want to toggle the restraint.
   - drag a box around multiple members to toggle the restraint at all valid locations within the box.

   The type of restraint at the selected location(s) changes to the next restraint type shown in the legend. (Simply click again as necessary until the desired restraint type has been applied.)
10. Continue toggling the restraints by clicking the desired members or press **Esc** to finish.

Set restraints

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Set**. For details of the restraint type choices etc. that are presented in Set mode, see the **Options overview** topics below.
6. Set the **Entity type** that you want to modify.
7. Leave the **Continuous** box checked to set continuous restraints along sub-members, or uncheck it to set discrete restraints at those points where other members connect.
8. Set the **Restraint Type** as required.
9. Set the **Restraint** as required.
10. In the model:
    - click the member at the location where want to set the restraint.
    - drag a box around multiple members to set the same restraint at all valid locations within the box.
11. Continue setting the checks by clicking/boxing the desired members or press **Esc** to finish.

Options overview (US regional code)

The restraint options displayed in the **Properties** window depend on the **Entity type** filter. When this is set to either **Beam**, **Column**, or **Truss Top/ Bottom Member**, the options are:

Command or option	Description
[M]ode	• **Review** - sub-members/discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking sub-members/discrete points of restraint toggles the applicable restraints. • **Set** - clicking sub-members/discrete points of restraint sets the selected restraint.
Continuous	• Checked - enables the review/setting of **sub-member continuous restraints** • Unchecked - enables the review/setting of **discrete restraints** at those points where other members connect.
Restraint Type	The restraint type determines which page(s) of properties are being edited. • **LTB** - enables the review/setting of the check-boxes located on the **Restraints (LTB)** property page. • **Compression** - enables the review/setting of the check-boxes located on the **Restraints (Comp)** property page. • **LTB & Compression Out of Plane** - enables the simultaneous review/setting of check-boxes on both the the **Restraints (LTB)** and **Restraints (Comp)** property pages.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the selected restraint property page are checked and un-checked.
	For **LTB** restraint types (and Entity type Beam or Truss): • **Top** - Top flange check-box is checked, Bottom flange check-box is unchecked. • **Bottom** - Top flange check-box is unchecked, Bottom flange check-box is checked. • **Top and Bottom** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **LTB** restraint types (and Entity Type Column): • **Face A** - Face A check-box is checked, Face C check-box is unchecked.

Command or option	Description
	• **Face C** - Face A check-box is unchecked, Face C check-box is checked. • **Face A & C** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **Compression** restraint types: • **Out of Plane** - Out of plane check-box is checked, In plane check-box is unchecked. • **In Plane** - In plane check-box is checked, Out of plane check-box is unchecked. • **Out of Plane & In Plane** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **LTB & Compression Out of Plane** restraint type: • **Top & Bottom & Out of Plane** - Top flange, Bottom flange, and Out of Plane are checked. • **Top & Out of Plane** - Top flange and Out of Plane are checked, Bottom flange is unchecked. • **Bottom & Out of Plane** - Bottom flange and Out of Plane are checked, Top flange is unchecked. • **Unrestrained** - Top flange, Bottom flange, and Out of Plane are checked. **NOTE** For each of the above, the In Plane check-box is unaffected, retaining its existing setting.

Options overview (other regional codes)

The restraint options displayed in the **Properties** window depend on the **Entity type** filter. When this is set to either **Beam**, **Column**, or **Truss Top/ Bottom Member**, the options are:

Command or option	Description
[M]ode	• **Review** - sub-members/discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking sub-members/discrete points of restraint toggles the applicable restraints. • **Set** - clicking sub-members/discrete points of restraint sets the selected restraint.

Command or option	Description
Continuous	• Checked - enables the review/setting of **sub-member continuous restraints** • Unchecked - enables the review/setting of **discrete restraints** at those points where other members connect.
Restraint Type	The restraint type determines which page(s) of properties are being edited. • **Lateral** - enables the review/setting of the check-boxes located on the **Lateral restraints** property page. • **Strut** - enables the review/setting of the check-boxes located on the **Strut restraints** property page • **Lateral & Strut Minor** - enables the simultaneous review/setting of check-boxes on both the the **Lateral restraints** and **Strut restraints** property pages.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the selected restraint property page are checked and un-checked.
	For **Lateral** restraint types (and Entity type Beam or Truss): • **Top** - Top flange check-box is checked, Bottom flange check-box is unchecked. • **Bottom** - Top flange check-box is unchecked, Bottom flange check-box is checked. • **Top and Bottom** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **Lateral** restraint types (and Entity Type Column): • **Face A** - Face A check-box is checked, Face C check-box is unchecked. • **Face C** - Face A check-box is unchecked, Face C check-box is checked. • **Face A & C** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **Strut** restraint types: • Major - Major check-box is checked, Minor check-box is unchecked. • Minor - Minor check-box is checked, Major check-box is unchecked.

Command or option	Description
	• Major and Minor - both check-boxes are checked. • Unrestrained - both check-boxes are unchecked.
	For **Lateral & Strut Minor** restraint type: • **Top & Bottom & Minor** - Top flange, Bottom flange, and Minor are checked. • **Top & Minor** - Top flange and Minor are checked, Bottom flange is unchecked. • **Bottom & Minor** - Bottom flange and Minor are checked, Top flange is unchecked. • **Unrestrained** - Top flange, Bottom flange, and Minor are checked. **NOTE** For each of the above, the Major check-box is unaffected, retaining its existing setting.

Portal Frame options (US regional code)

The restraint options displayed in the **Properties** window when the **Entity type** filter is set to **Portal Frame** are:

Command or option	Description
[M]ode	• **Review** - discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking discrete points of restraint toggles the applicable restraints. • **Set** - clicking discrete points of restraint sets the selected restraint.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the restraint property pages are checked and un-checked. Rafters: • **Torsional** - Top flange, Bottom flange, and Out of Plane are checked. • **Outer** - Top flange is checked, Bottom flange is unchecked. • **Inner** - Top flange is unchecked, Bottom flange is checked.

Command or option	Description
	• **Unrestrained** - Top flange, Bottom flange, and Out of Plane are unchecked. Columns: • **Torsional** - Face A, Face C, and Out of Plane are checked. • **Outer** - Face A is checked, Face C is unchecked. • **Inner** - Face A is unchecked, Face C is checked. • **Unrestrained** - Face A, Face C, and Out of Plane are unchecked. **NOTE** For each of the above, any checkbox not specifically mentioned retains its existing setting.

Portal frames options (other regional codes)

The restraint options displayed in the **Properties** window when the **Entity type** filter is set to **Portal Frame** are:

Command or option	Description
[M]ode	• **Review** - discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking discrete points of restraint toggles the applicable restraints. • **Set** - clicking discrete points of restraint sets the selected restraint.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the restraint property pages are checked and un-checked. Rafters: • **Torsional** - Top flange, Bottom flange, and Minor are checked. • **Outer** - Top flange is checked, Bottom flange is unchecked. • **Inner** - Top flange is unchecked, Bottom flange is checked. • **Unrestrained** - Top flange, Bottom flange, and Minor are unchecked. Columns:

Command or option	Description
	• Torsional - Face A, Face C, and Minor are checked. • Outer - Face A is checked, Face C is unchecked. • Inner - Face A is unchecked, Face C is checked. • Unrestrained - Face A, Face C, and Minor are unchecked. **NOTE** For each of the above, any checkbox not specifically mentioned retains its existing setting.

Related video

Show/Alter State Restraints

Apply rotational stiffness to a beam end

The **Rotational stiffness** command allows you to graphically apply partially fixity/linear springs to the ends of beams.

Related video

Automatic modelling of partial fixity

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Fixed/Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Rotational stiffness**.
5. Set **Mode** to **Set On**.
6. Set **Direction** to **Major**, or **Minor** as required.
7. Set **Rotational stiffness** to **Partially fixed**, or **Spring linear** as required.
8. If **Partially fixed**, set the percentage fixity value; or if **Spring linear** set the spring stiffness value as required.
9. To apply the new setting do one of the following:
   - Click an individual beam end.
   - Hold down the left mouse button and drag a box from left to right to apply to all beam ends entirely within the box.
   - Hold down the left mouse button and drag a box from right to left to apply to both ends of all beams within the box or cut by the box.

Review and modify seismic drift

You can graphically review seismic drift check results and set the members to be checked for columns and walls.

Review seismic drift

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Seismic drift**.
5. Set **Mode** to **Review**.

Set seismic drift checks on or off

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Seismic drift**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Review and modify SFRS settings

If necessary, you can graphically review and modify the types and directions of seismic force resisting systems (SFRS) in the model. For more information, see the following paragraphs.

Review SFRS type and direction settings

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **SFRS**.

   Members and walls are color coded according to their SFRS type and direction settings.

Modify SFRS type & direction settings

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **SFRS**.

   Members and walls are color coded according to their SFRS direction setting.
5. Set **Mode** to one of the following:
   - **Update Type & Direction**: includes a member in the selected SFRS type & direction.
   - **Remove from SFRS**: removes the member from SFRS.
6. Click the desired member.

   Depending on the set mode, Tekla Structural Designer includes the member in the SFRS type & direction, or removes the member from SFRS.

Copy shear connectors

If necessary, you can copy the composite beam shear connector properties from a source beam to other composite beams. Do the following:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Shear Connectors**.
5. In the model, click the composite beam from which you want to copy the shear connector properties.
6. Click the composite beams to which you want to copy the shear connect properties.

   The properties are copied to the selected beams.

Modify SidePlates

The **SidePlates** command allows you to graphically review and modify the SidePlate connection status.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **SidePlates**.
3. Do one of the following:
   - Click on the first half of a beam to reset end 1, the second half to reset end 2 and the middle to reset both ends simultaneously.
   - Hold down the left mouse button and drag a box from right to left to reset all the beam ends within the box or cut by the box.

Review and copy size constraints

If necessary, you can graphically review specified size constraints for steel beams, columns, and braces, and copy the constraints from one steel member to others. For more information, see the following paragraphs.

Review size constraints

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Size Constraints**.
5. Set **Mode** to **Review**.
6. In **Size constraint**, select the size constraint that you want to review.

Copy size constraints

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Size Constraints**.
5. Set **Mode** to **Copy**.

   TIP Select the **By constraint type** option if you want to only copy an individual constraint type.

6. In the model, click the member from which you want to copy the size constraints.
7. Click the member to which you want to copy the size constraints.

8. Continue copying the size constraints by clicking the desired members or press **Esc** to finish.

Review and modify Staged Construction stages

You can graphically review construction stages in order to define the content and duration of each stage.

Review existing stage content and stage durations

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Staged Construction**.
5. Set the **Mode** to **New Structure by Stage**.

   Entities are color coded according to their construction stage. The duration of each stage (in weeks) is indicated in the legend.

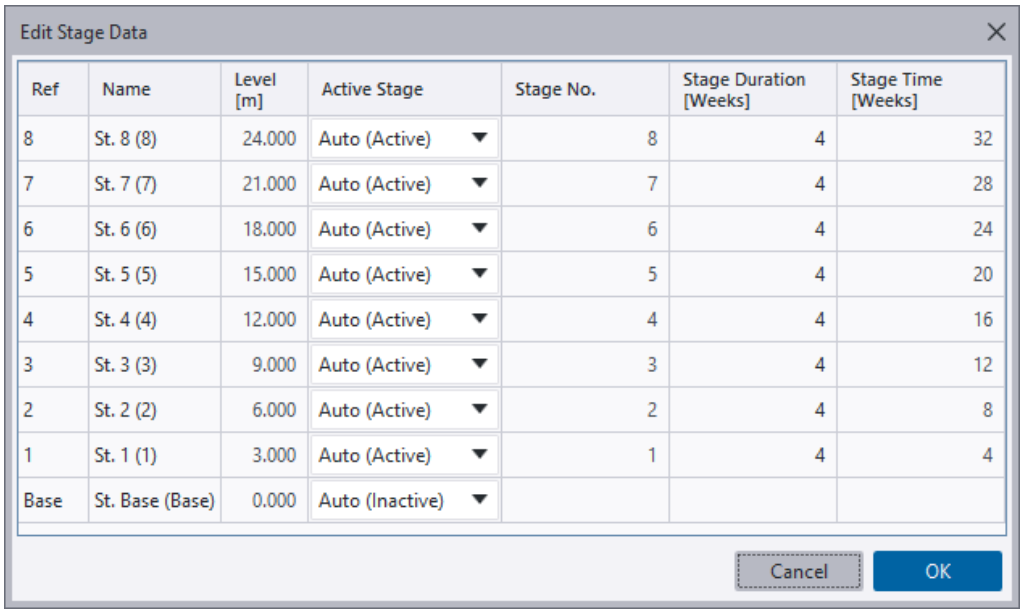
Change the number of stages

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Staged Construction**.
5. Set the **Mode** to **New Structure by Stage**.

   Entities are color coded according to their construction stage. The duration of each stage (in weeks) is indicated in the legend.
6. Under **Data** click **Stage Data** then click the **Edit** button.



   This displays the **Edit Stage Data** dialog box.



NOTE In the table each floor initially defaults to either Auto (Active), or Auto (Inactive):

- Auto (Active) - the floor level is an active stage because it contains at least one member, or has a member attached to it from below.
- Auto (Inactive) - the floor level is not an active stage because it doesn't contain at least one member, or have a member attached to it from below.
- The auto defaults can be overridden by manually setting a level to be an active or an inactive stage.

7. In the table, set **Active Stage** at the level of the stage to be removed to **Inactive**.

   The stage is removed and all entities at that level are moved to the next active stage.

NOTE If required, you can also use the table to edit individual stage durations.

Modify the duration of all stages

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Staged Construction**.

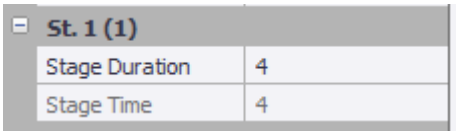
5. Under **Data** set **Stage Duration** to the value required.

   The specified duration is applied to all stages.

Modify the duration of an individual stage in the Properties window

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Staged Construction**.
5. Set the **Mode** to **Edit Stage**.
6. In the **Stage** list under **Options**, select the stage to edit.

   Entities in the selected stage are shown and the duration of the stage is indicated in the legend. A new section for the stage name is added to the **Properties** window, under which the selected stage's duration and stage time are displayed.

   In the **Properties** window entities in the currently selected stage are shown and the duration of the stage is indicated in the legend.
7. In the **Properties** window, under the stage name change **Stage Duration** to the value required. (Be careful not to change it in the **Data** section as this would change the duration of all stages).

   The specified duration is applied to the selected stage only.

NOTE You can also modify an individual stage's duration in **Add to Stage** and **Remove from Stage** modes.

Modify stage content - add to stage

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Add to Stage**.
5. In the **Stage** list under **Options**, select the stage to edit.

   Entities in the currently selected and previous stages are indicated in the legend. Entities that can be added to the current stage are shown as ghosted.

6. Select the entities to add to the stage.

   The selected entities are added to the stage.

NOTE You can also add entities to a stage in the same way in **Edit Stage** mode.

Modify stage content - remove from stage

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Remove from Stage**.
5. In the **Stage** list under **Options**, select the stage to edit.

   Entities in the currently selected and previous stages are indicated in the legend.
6. Select an entity from the current stage to remove it.

   The selected entity is removed from the current stage and added to the next stage.

   NOTE Entities cannot be removed from the final construction stage.

NOTE You can also remove entities from a stage when in **Edit Stage** mode.

Modify stud auto layout

If necessary, you can graphically modify the stud auto layout setting of composite beams in the model. For more information, see the following paragraphs.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Stud auto layout**.

   The composite beams in the model are color coded according to their auto layout setting.
5. Do one of the following:
   - Click an individual composite beam to alter its auto layout setting.

- Hold down the left mouse button and drag a box from left to right to alter the auto layout setting of all composite beams within the box.
- Hold down the left mouse button and drag a box from right to left to alter the auto design setting of all composite beams within the box or cut by the box.

Review and modify sway checks

You can graphically review sway check results and set the members to be checked for columns and walls.

Review sway checks

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Sway check**.
5. Set **Mode** to **Review**.

Set sway checks on or off

1. Open a view and switch the view regime to  **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Sway check**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Copy transverse reinforcement

If necessary, you can copy beam transverse reinforcement from one composite beam to others in your model. Do the following:

1. Open a view and switch the view regime to  **Review View**.

2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Transverse reinforcement**.

   The composite beams in the model are color coded based on whether you can copy transverse reinforcement properties from them.
5. Click the composite beam from which you want to copy transverse reinforcement properties.
6. Click the composite beams to which you want to copy the transverse reinforcement properties.

   The properties are copied to the selected beams.

Review and modify user defined U/R

From a **Review View** you can graphically review user defined utilization ratios, set the members to which they are applied, and set whether they apply to autodesign alone, or both autodesign and check.

Review user defined U/R

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.

Apply a user defined U/R for autodesign only

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Apply to Autodesign**.
6. Select **Set limit ratio**.
7. Specify the required **Ratio limit**
8. If required, set **Entity type** and **Material** to make the command easier to apply.
9. In the model, click those column/wall stacks, beam spans, or objects for which you want to apply the user defined U/R.

10. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Apply a user defined U/R for autodesign and check

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Apply to Autodesign+Check**.
6. Select **Set limit ratio**.
7. Specify the required **Ratio limit**
8. If required, set **Entity type** and **Material** to make the command easier to apply.
9. In the model, click those column/wall stacks, beam spans, or objects for which you want to apply the user defined U/R.
10. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Turn off user defined U/R

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Set Off**.
6. If required, set **Entity type** and **Material** to make the command easier to apply.
7. In the model, click those column/wall stacks, beam spans, or objects for which you want to turn off user defined U/R.
8. Continue by clicking the desired stacks/spans/objects or press **Esc** to finish.

Related information

See also

Apply user defined utilization ratios (page 797)

Video: User defined utilization ratio

Review utilization and embodied carbon

The Show/Alter State **Utilization and embodied carbon** attribute allows you to graphically focus in on high carbon usage and inefficiency.

Review utilization and embodied carbon

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Embodied Carbon**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Utilization and embodied carbon**.
5. Set the **Entity type** and **Variant** to review.
6. Set the **Carbon source category** to be review.

NOTE Any entity in the model can have *multiple* carbon sources that relate to it, for example:

- A concrete beam has concrete and reinforcement
- A composite beam has steel, shear studs, and potentially transverse shear reinforcement

For example

Set filters to focus on where the carbon impact might be reduced

1. Set the utilization filter as required.
   - Value below - only entities with utilization below the specified value are displayed
   - Value above - only entities with utilization above the specified value are displayed
   - Lowest X % - only the lowest specified percentage of utilizations are displayed
   - Highest X % - only the highest specified percentage of utilizations are displayed
2. Set the embodied carbon filter as required.
   - Value below - only entities with a carbon usage below the specified value are displayed
   - Value above - only entities with a carbon usage above the specified value are displayed
   - Lowest X % - only the lowest specified percentage of carbon usages are displayed

- Highest X % - only the highest specified percentage of carbon usages are displayed

Typically, filters would be applied to detect where *high* carbon usage occurs in combination with *low* utilization, as this informs where the carbon impact could be reduced.

For example: Highest 50% embodied carbon combined with lowest 50% utilization.

Copy web openings

If necessary, you can copy web opening properties from one member to others. Do the following:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Web Openings**.

   The members in the model are color coded based on whether you can copy web opening properties from them.
5. In the model, click the member from which you want to copy web opening properties.
6. Click the members to which you want to copy the web opening properties.

   The properties are copied to the selected members.

Copy Westok openings

If necessary, you can copy Westok opening properties from one member in your model to others. Do the following:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Westok Openings**.

   The members in the model are color coded based on whether you can copy Westok opening properties from them.
5. In the model, click the member from which you want to copy Westok opening properties.

6. Click the members to which you want to copy the Westok opening properties.

   The properties are copied to the selected members.

Review and modify wind drift checks

You can graphically review wind drift check results, set the members to be checked and set the check limits for columns and walls.

Review wind drift checks

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Review**.

Set wind drift checks on or off

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Set wind drift checks on and set the ratio limit

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.

5. Set **Mode** to **Set On**.
6. Select **Set limit ratio**.
7. Specify the required **Wind drift ratio limit**
8. In the model, click the column or wall for which you want to set the limit.
9. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Modify wind loading

The Show/Alter State **Wind Loading** attribute allows you to graphically set the members/ancillaries/equipment to which open structure wind loading applies.

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Wind Loading** is selected.
5. If you want to apply the setting to individual stacks/spans within members as opposed to the entire member, unselect **Entire member**.
6. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
7. In the model, click the members/ancillaries/equipment for which you want to set the check.
   a. If **Mode** is **Set On** - the setting is switched on
   b. If **Mode** is **Set Off** - the setting is switched off
   c. If **Mode** is **Toggle** - the setting toggles between on and off
8. Continue applying the setting or press **Esc** to finish.

6.3 Tabular review data

Tabular Data on the **Review** tab allows you to view design summaries, sway & drift results, material lists, connection resistances, embodied carbon and more in tabular views. The actual views available will depend on the regional code being worked to. Tabular review data can be filtered, sorted and exported to Excel.

- Design summary (page 1012)
- Sway (page 1013)
- Kamp (page 1017)

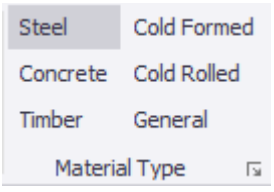
- Drift (page 1019)
- Seismic drift (page 1022)
- Wind drift (page 1025)
- Connection resistance
- Material list (page 1029)
- Embodied carbon detail (page 1054)
- Embodied carbon overview (page 1055)
- Floored area (page 1057)
- Review portal frame deflection (page 1058)
- Sort and filter tabular review data (page 1058)
- Export tabular review data to Excel (page 1059)

Review design summary tabular results

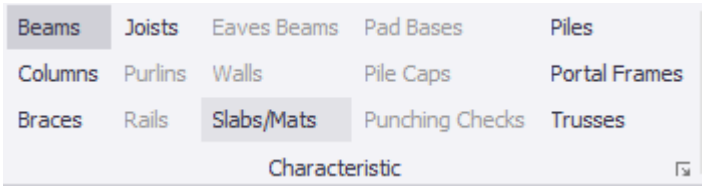
To review a tabular **Design Summary**:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Design Summary**
4. In the **Result Type** group, select the analysis type for which the design summary is required.
5. In the **Material Type** group, select the material.



   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.
6. In the **Characteristic** group, select the required characteristic.



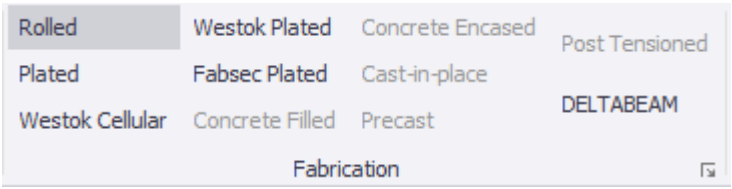
The available selections in the **Construction** group are reduced to those appropriate to the selected characteristic.

7. If applicable, in the **Construction** group, select the construction.



The available selections in the **Fabrication** group are reduced to those appropriate to the selected construction.

8. If applicable, in the **Fabrication** group, select the fabrication.



The tabular design summary is displayed for every entity of the selected material and type in the structure.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

See also

Export tabular review data to Excel (page 1059)

Review member design (page 943)

Review sway check tabular results

NOTE The sway check is not applicable if using the ACI/AISC regional code.

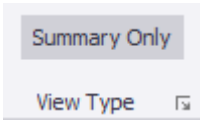
Sway check results can be reviewed by creating a **Sway** tabular data view.

Sway tabular data views:

- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical result for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 1058)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 1078) report which can then be printed.

Review sway check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Sway** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
5. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



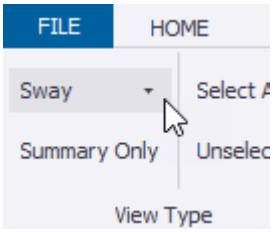
The sway checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Review sway check tabular results from a Review View

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

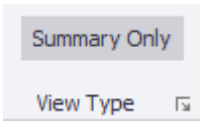
   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Sway**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.

- In the **Loading** list, select a specific combination.

6. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.

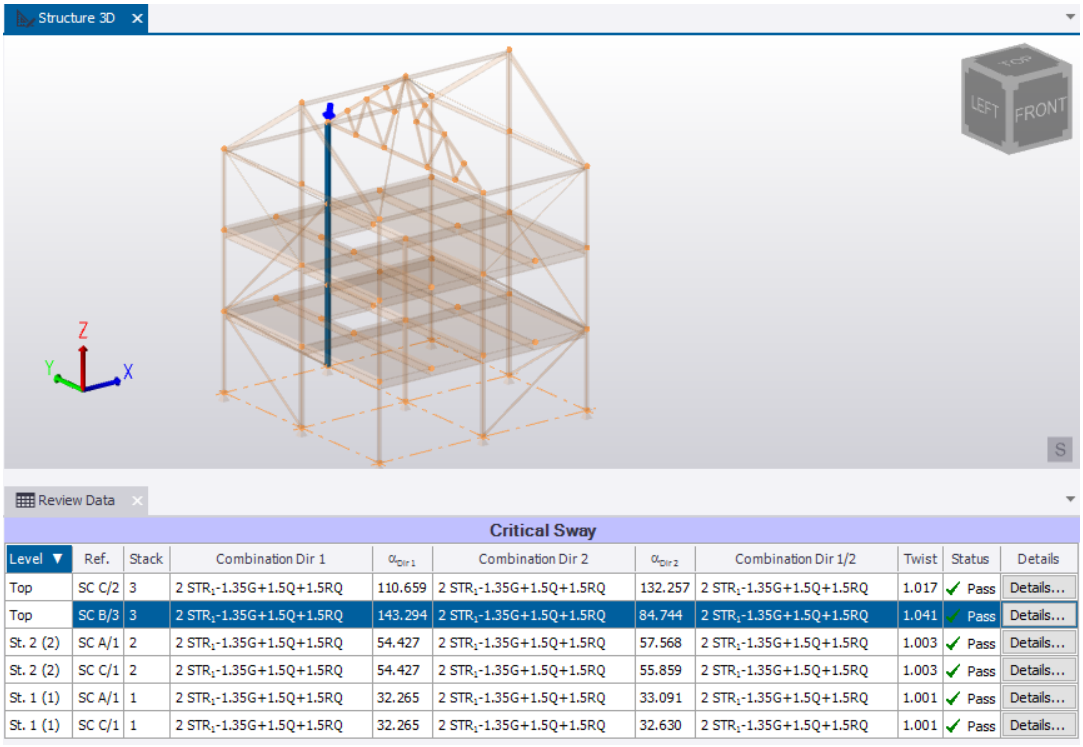


The tabular drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



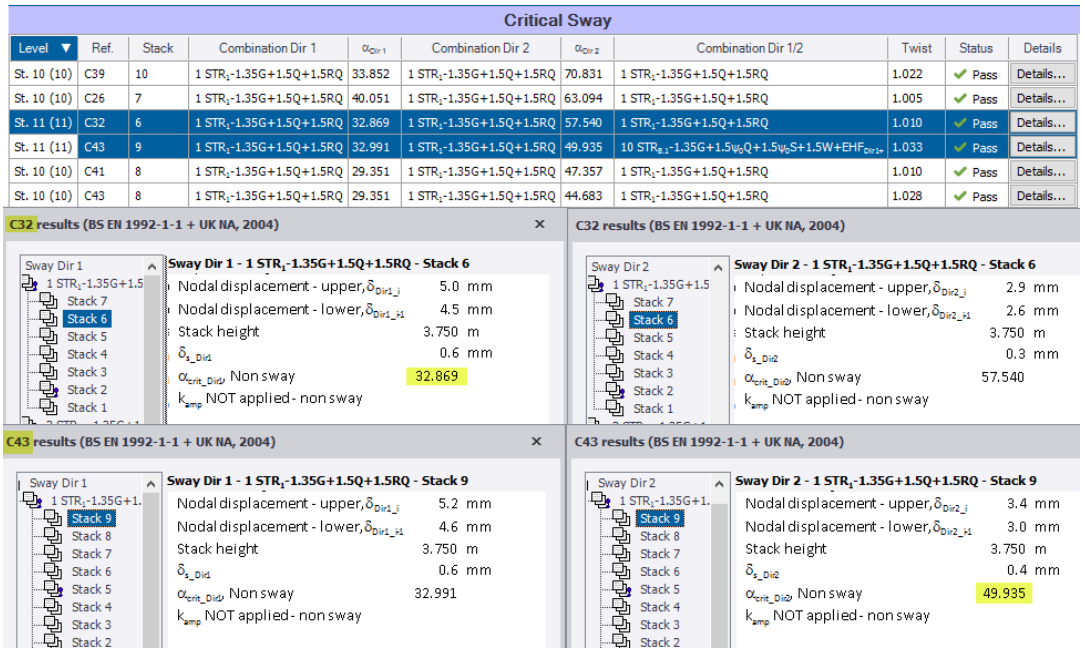
Level	Ref.	Stack	Combination Dir 1	α_{Dir1}	Combination Dir 2	α_{Dir2}	Combination Dir 1/2	Twist	Status	Details
Top	SC C/2	3	2 STR_1-1.35G+1.5Q+1.5RQ	110.659	2 STR_1-1.35G+1.5Q+1.5RQ	132.257	2 STR_1-1.35G+1.5Q+1.5RQ	1.017	✓ Pass	Details...
Top	SC B/3	3	2 STR_1-1.35G+1.5Q+1.5RQ	143.294	2 STR_1-1.35G+1.5Q+1.5RQ	84.744	2 STR_1-1.35G+1.5Q+1.5RQ	1.041	✓ Pass	Details...
St. 2 (2)	SC A/1	2	2 STR_1-1.35G+1.5Q+1.5RQ	54.427	2 STR_1-1.35G+1.5Q+1.5RQ	57.568	2 STR_1-1.35G+1.5Q+1.5RQ	1.003	✓ Pass	Details...
St. 2 (2)	SC C/1	2	2 STR_1-1.35G+1.5Q+1.5RQ	54.427	2 STR_1-1.35G+1.5Q+1.5RQ	55.859	2 STR_1-1.35G+1.5Q+1.5RQ	1.003	✓ Pass	Details...
St. 1 (1)	SC A/1	1	2 STR_1-1.35G+1.5Q+1.5RQ	32.265	2 STR_1-1.35G+1.5Q+1.5RQ	33.091	2 STR_1-1.35G+1.5Q+1.5RQ	1.001	✓ Pass	Details...
St. 1 (1)	SC C/1	1	2 STR_1-1.35G+1.5Q+1.5RQ	32.265	2 STR_1-1.35G+1.5Q+1.5RQ	32.630	2 STR_1-1.35G+1.5Q+1.5RQ	1.001	✓ Pass	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Understanding the Critical Sway Summary Only tabular results

The Critical Sway Review Summary Only data table reports by Level, so at each Level of the structure it shows the column and column stack which gives the worst $\alpha_{Dir\ 1}$ ***at that Level*** and worst $\alpha_{Dir\ 2}$ (usually a different column in a second row) ***at that Level***. However, for each of the columns/stacks it reports α in both Dir 1 and Dir 2.

Thus in the example below, for the Level St. 11 column C32 has critical $\alpha_{Dir\ 1}$ from stack 6 (α = 32.869) and column C43 has critical $\alpha_{Dir\ 2}$ from stack 9 (α = 49.935) (but the $\alpha_{Dir\ 1}$ and $\alpha_{Dir\ 2}$ are shown for both of the columns).



Level	Ref.	Stack	Combination Dir 1	$\alpha_{Dir\ 1}$	Combination Dir 2	$\alpha_{Dir\ 2}$	Combination Dir 1/2	Twist	Status	Details
St. 10 (10)	C39	10	1 STR1-1.35G+1.5Q+1.5RQ	33.852	1 STR1-1.35G+1.5Q+1.5RQ	70.831	1 STR1-1.35G+1.5Q+1.5RQ	1.022	Pass	Details...
St. 10 (10)	C26	7	1 STR1-1.35G+1.5Q+1.5RQ	40.051	1 STR1-1.35G+1.5Q+1.5RQ	63.094	1 STR1-1.35G+1.5Q+1.5RQ	1.005	Pass	Details...
St. 11 (11)	C32	6	1 STR1-1.35G+1.5Q+1.5RQ	32.869	1 STR1-1.35G+1.5Q+1.5RQ	57.540	1 STR1-1.35G+1.5Q+1.5RQ	1.010	Pass	Details...
St. 11 (11)	C43	9	1 STR1-1.35G+1.5Q+1.5RQ	32.991	1 STR1-1.35G+1.5Q+1.5RQ	49.935	10 STR8.1-1.35G+1.5ψ0Q+1.5ψ0S+1.5W+EHFDir1+	1.033	Pass	Details...
St. 10 (10)	C41	8	1 STR1-1.35G+1.5Q+1.5RQ	29.351	1 STR1-1.35G+1.5Q+1.5RQ	47.357	1 STR1-1.35G+1.5Q+1.5RQ	1.010	Pass	Details...
St. 10 (10)	C43	8	1 STR1-1.35G+1.5Q+1.5RQ	29.351	1 STR1-1.35G+1.5Q+1.5RQ	44.683	1 STR1-1.35G+1.5Q+1.5RQ	1.028	Pass	Details...

Review amplification factor Kamp

NOTE The kamp factor is not applicable if using the ACI/AISC regional code.

When the analysis method to be used in design has been set to the amplified forces method, the calculated amplification factor in both directions at each level can be reviewed by creating a **Kamp** tabular data view.

Kamp tabular data views:

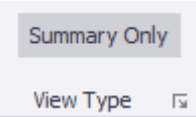
- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column with most critical result for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns for the selected combinations

- are displayed for all columns (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 1058)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, but by creating a report which includes the Analysis>Kamp chapter they can then be printed.

Review kamp tabular results from a Review View

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **kamp**
4. Choose if you want results displayed for **all** available materials, or a subset only:
   - To display results for all available materials, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



The tabular kamp factors are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review drift check tabular results

NOTE The drift check is only applicable if using the ACI/AISC regional code.

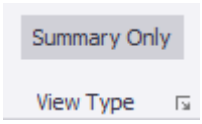
Drift check results can be reviewed by creating a **Drift** tabular data view.

Drift tabular data views:

- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical ratio for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 1058)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 1078) report which can then be printed.

Review drift check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Drift** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
5. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



The tabular drift checks are displayed as specified.

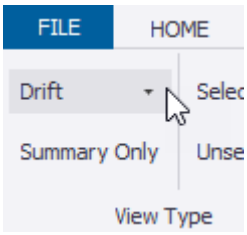
If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Review drift check tabular results from a Review View

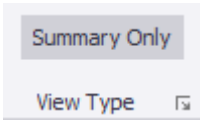
1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Drift**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.

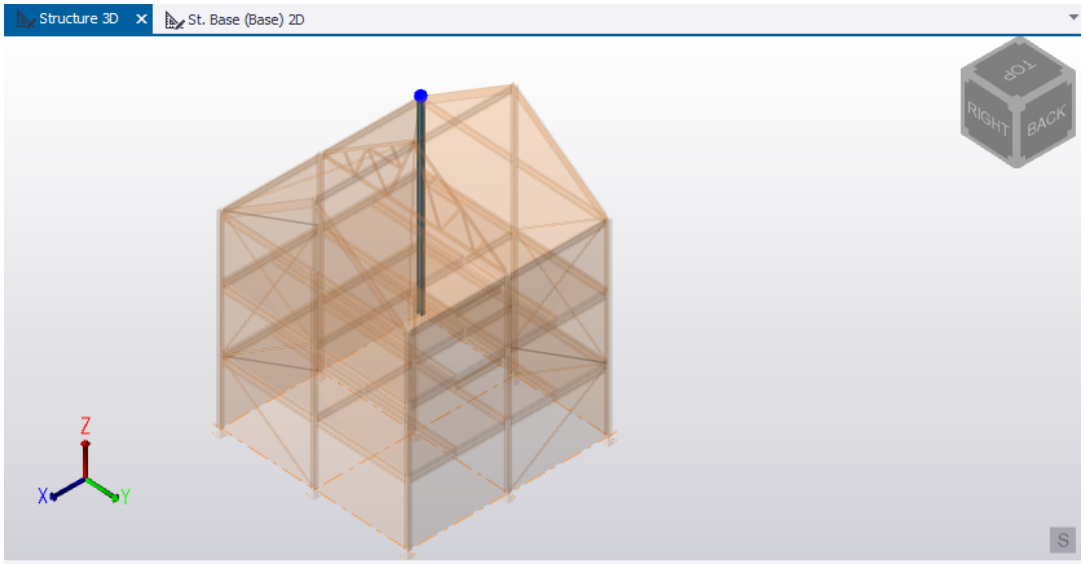


The tabular drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Drift

Level	Ref.	Stack	Combination Dir 1	Drift 1st order Dir 1 [mm]	Drift 2nd order Dir 1 [mm]	Ratio Dir 1	Combination Dir 2	Drift 1st order Dir 2 [mm]	Drift 2nd order Dir 2 [mm]	Ratio Dir 2	Status	Details
Top	SC C/2	3	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	43.7	44.2	1.012	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	2.3	2.3	1.003	✓ Pass	Details...
Top	SC B/3	3	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	0.8	0.8	1.002	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	4.5	4.5	1.005	✓ Pass	Details...
St. 2 (2)	SC A/1	2	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.2	1.2	1.011	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	2.7	2.8	1.013	✓ Pass	Details...
St. 2 (2)	SC C/1	2	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.2	1.2	1.011	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	2.7	2.7	1.013	✓ Pass	Details...
St. 1 (1)	SC A/1	1	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.5	1.5	1.019	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	3.5	3.5	1.022	✓ Pass	Details...
St. 1 (1)	SC C/1	1	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.5	1.5	1.019	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	3.4	3.5	1.023	✓ Pass	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review seismic drift check tabular results

Seismic drift check results can be reviewed by creating a **Seismic drift** tabular data view.

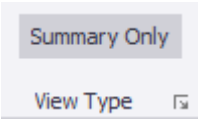
Seismic drift tabular data views:

- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical ratio for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations

- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 1058)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 1078) report which can then be printed.

Review seismic drift check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Seismic Drift** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
5. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



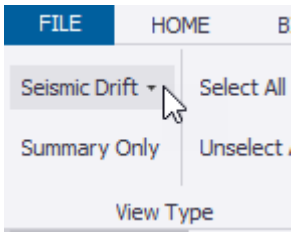
The tabular results are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

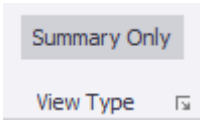
Review seismic drift check tabular results from a Review View

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Seismic drift**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.

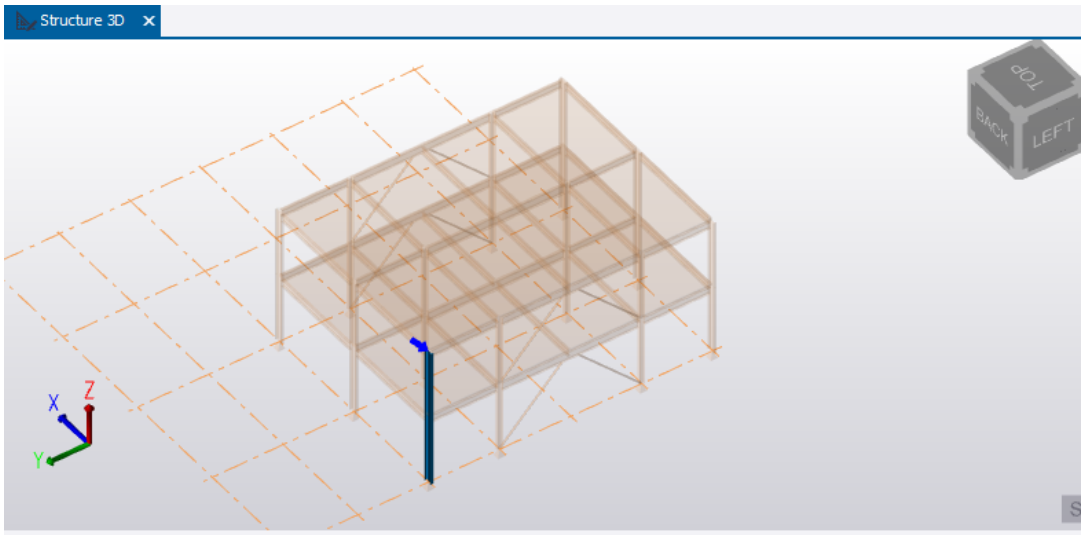


The tabular results are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Seismic Drift

Level	Ref.	Stack	Combination Dir 1	Drift Ratio Dir 1	$\theta_{Dir\,1}$ Ratio	Combination Dir 2	Drift Ratio Dir 2	$\theta_{Dir\,2}$ Ratio	Status	Details
St. 2 (2)	SC A/5	2	234 $LRFD_{11.2}$-1.2D+L+0.2S+E	0.3245	0.3351	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0257	0.0246	✓ Pass	Details...
St. 2 (2)	SC A/1	2	233 $LRFD_{11.1}$-1.2D+L+0.2S+E	0.3245	0.3351	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0257	0.0246	✓ Pass	Details...
St. 1 (1)	SC A/5	1	234 $LRFD_{11.2}$-1.2D+L+0.2S+E	0.7194	0.8536	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0322	0.0413	▲ Warning	Details...
St. 1 (1)	SC A/1	1	233 $LRFD_{11.1}$-1.2D+L+0.2S+E	0.7194	0.8536	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0322	0.0413	▲ Warning	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review wind drift check tabular results

Wind drift check results can be reviewed by creating a **Wind drift** tabular data view.

Wind drift tabular data views:

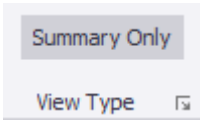
- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical ratio for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 1058)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 1078) report which can then be printed.

Review wind drift check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Wind Drift** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:

- In the **Loading** list, select **All Combinations**.
- In the **Loading** list, select a specific combination.

5. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



The tabular wind drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

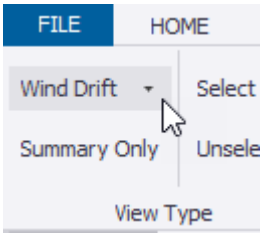
Review wind drift check tabular results from a Review View

1. Open a view and switch the view regime to **Review View**.

2. On the **Review** ribbon, click **Tabular Data**

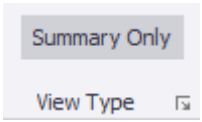
   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Wind drift**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).

5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.

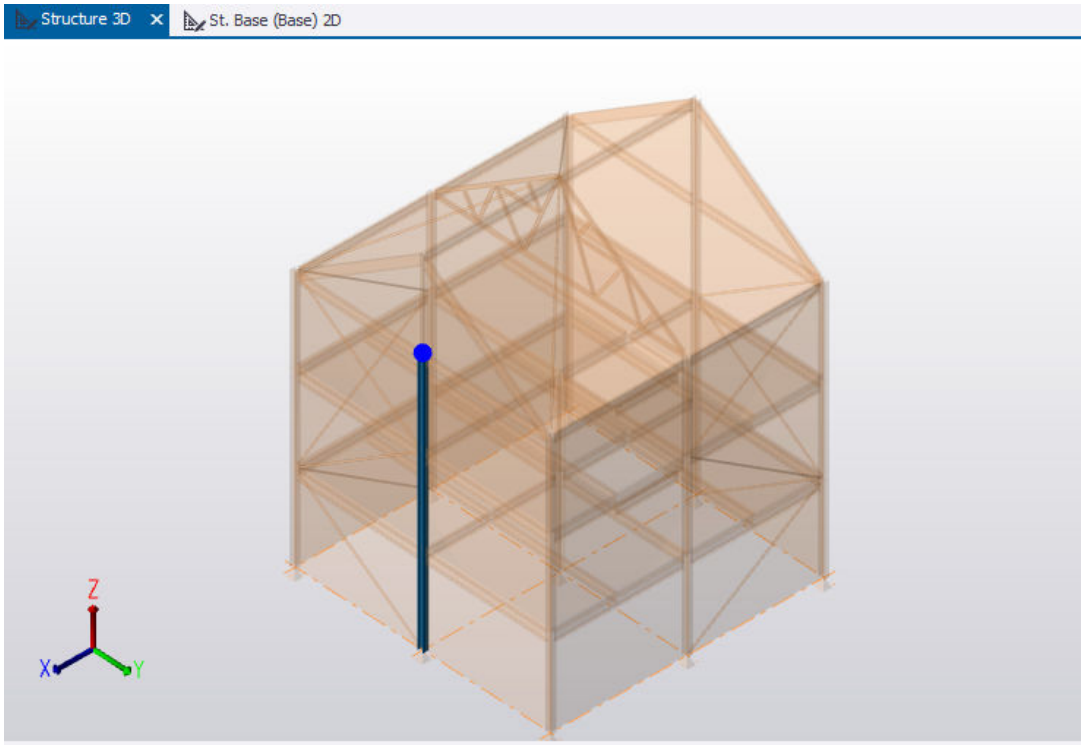


The tabular wind drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Level	Ref.	Stack	Combination	Deflection Dir 1 [mm]	Deflection Dir 2 [mm]	Drift Dir 1 [mm]	Drift Dir 2 [mm]	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2	Details
Top	SC C/2	3	50 LRFD$_{12.4}$-0.9D+W	26.2	4.8	26.3	1.1	152.012	3503.178	✗ Fail	✓ Pass	Details...
Top	SC B/3	3	50 LRFD$_{12.4}$-0.9D+W	-0.1	5.7	0.0	2.0	170421.311	1972.091	✓ Pass	✓ Pass	Details...
St. 2 (2)	SC A/1	2	47 LRFD$_{12.1}$-0.9D+W	3.7	-0.2	1.7	0.1	2370.157	41721.556	✓ Pass	✓ Pass	Details...
St. 2 (2)	SC C/1	2	50 LRFD$_{12.4}$-0.9D+W	-0.1	3.7	0.0	1.6	2.301E+06	2482.328	✓ Pass	✓ Pass	Details...
St. 1 (1)	SC A/1	1	47 LRFD$_{12.1}$-0.9D+W	2.0	-0.1	2.0	0.1	1998.949	53404.925	✓ Pass	✓ Pass	Details...
St. 1 (1)	SC A/1	1	50 LRFD$_{12.4}$-0.9D+W	-0.1	2.1	0.1	2.1	42301.068	1930.097	✓ Pass	✓ Pass	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review material list tabular results

The material content of the model can be readily assessed in the form of a **Material List**. This can be created and viewed either in a *tabular data view* or a *report*, according to your needs.

- Material List tabular data view:
  - created for a specific material and entity type

    - displayed on screen (not suitable for direct printing)
    - can be sorted in ascending/descending order of a column by clicking a column heading
    - row content can be located within a 3D view by double-clicking a row
    - can be exported to a spreadsheet
- Material List report:
    - created for *multiple* materials and entity types
    - displayed in a format suitable for printing
    - can be exported to a spreadsheet

Two levels of content can be generated:

- **Summary Only** - this is the default level and is very condensed, for example:
    - total quantities only for each beam/ column section size
    - total quantities for each slab
    - total weight of reinforcement (by mass, and by mass/unit volume)
- **Detailed** - this is a longer report, for example:
    - quantities for each span/ stack/ panel for each concrete beam / column / wall
    - quantities for each section size of a given length for beams and columns in other materials
    - quantities for each slab item
    - quantities for individual reinforcement bar/mesh types

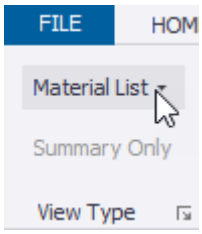
The topics in the section all relate to the tabular data view. To find out how to create a material list report, see: Material Listing report (page 1082)

Create material list tabular results

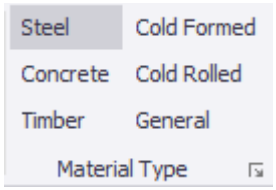
To create a material list as tabular data:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Material List**



4. In the **Material Type** group, select the material.



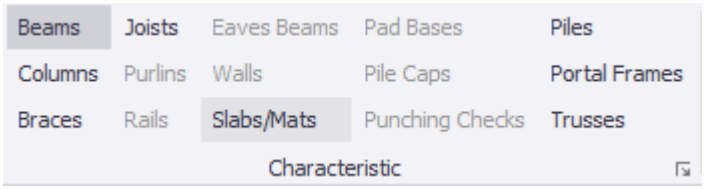
The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.

NOTE To find out exactly what summary and detailed level output is produced for each material type and characteristic, see:

- Steel material lists (page 1034)
- Concrete material lists (page 1039)
- Timber material lists (page 1050)
- Cold formed material lists (page 1051)
- General material lists (page 1052)

---

5. In the **Characteristic** group, select the required characteristic.



The available selections in the **Construction** group are reduced to those appropriate to the selected characteristic.

6. If applicable, in the **Construction** group, select the construction.



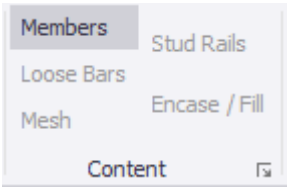
The available selections in the **Fabrication** group are reduced to those appropriate to the selected construction.

7. In the **Fabrication** group, select the fabrication.

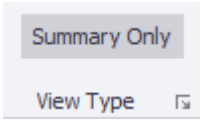


The available selections in the **Content** group are reduced to those appropriate to the selected fabrication.

8. In the **Content** group, select the content.



9. If you have selected **Members** in the **Content** group, you have the option to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



The tabular material list is displayed for every entity of the selected material and type in the structure.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Sort and filter tabular review data (page 1058)

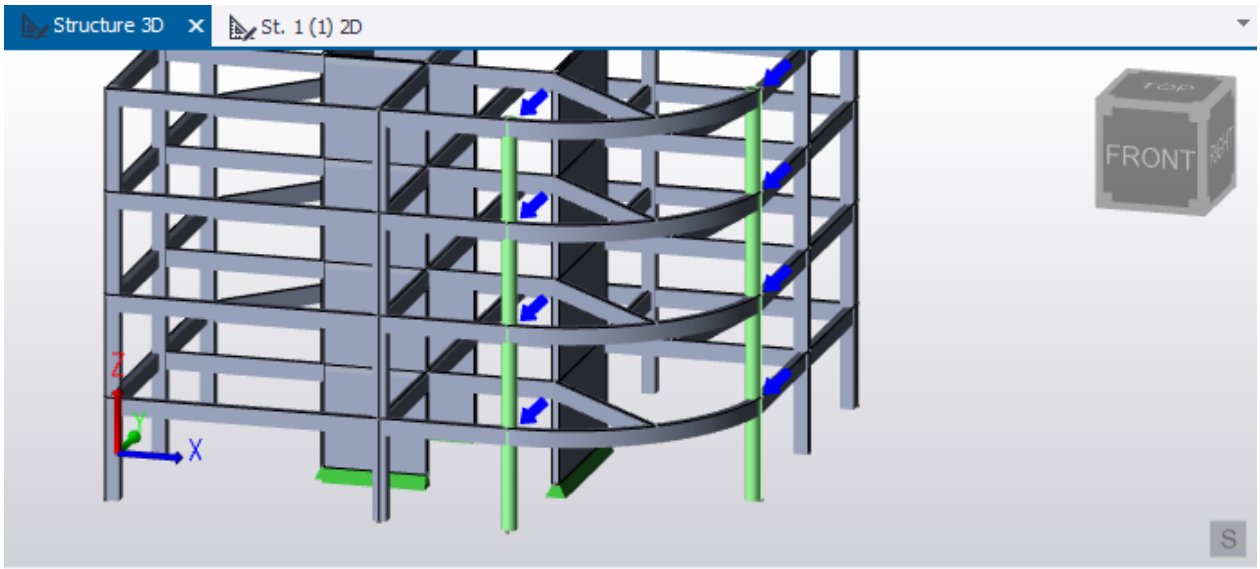
See also:

- Steel material lists (page 1034)
- Concrete material lists (page 1039)

- Timber material lists (page 1050)
- Cold formed material lists (page 1051)
- General material lists (page 1052)

Locate material list rows in a 3D view

Double-clicking on a specific row in the material list tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Material List

Section Geometry	Section Size	Grade	No.	Total Length [m]	Mass [kg]	Surface Area [m²]	Volume [m³]	Reinforcement [kg]	Reinforcement [kg/m³]
Rectangular	400x400	C32/40	32	96.000	38400.00	153.6	15.4	10315.35	672
Circular	450 Dia	C32/40	8	24.000	9542.59	33.9	3.8	1334.72	350
			40	**120.000**	**47942.59**	**187.5**	**19.2**	**11650.07**	**608**

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Export material list to Excel

1. Create the material list and ensure that it contains the necessary information.
2. On the **Review Data** ribbon tab, click **Excel**.

3. Click **Yes** to confirm exporting the report.

   The material list opens in Excel

Material lists for steel

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length
- Composite beam reports also include the number of connectors, the transverse reinforcement associated with the beam (at detailed level only), and the weight of reinforcement

Construction	Fabrication	Content	View Type	Description
Composite	• Rolled • Plated • Westok Plated • Fabsec Plated • DELTABEAM	Members	Summary	One row for each size: size, grade, number (of that size), no. of connectors, total length, mass, surface area, reinforcement
			Detailed	One row for each size & length: size, grade, number (of that size & length), no. of connectors, length, mass, surface area, reinforcement
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass

Construction	Fabrication	Content	View Type	Description
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Westok Cellular	Members	Summary	One row for each size: size, grade, number (of that size), no. of connectors, total length, gross mass, net mass, gross surface area, net surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), no. of connectors, length, gross mass, net mass, gross surface area, net surface area
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Mesh	One row for each type: type, unit mass, total area, total mass

Construction	Fabrication	Content	View Type	Description
Non-composite	• Rolled • Plated • Westok Plated • Fabsec Plated • DELTABEAM	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
	Westok Cellular	Members	Summary	One row for each size: size, grade, number (of that size), total length, gross mass, net mass, gross surface area, net surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, gross mass, net mass, gross surface area, net surface area

Columns, Braces, Joists

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	Fabrication	Content	View Type	Description
Columns	• Rolled • Plated	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
	• Concrete Filled • Concrete Encased	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass
		Encase/Fill	Detailed	One row for each size: size, grade, number (of

Characteristic	Fabrication	Content	View Type	Description
				that size), length, mass
Braces	Rolled	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
Joists	N/A	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	Description
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Portal Frames

Characteristic	Content	View Type
Portal Frames	Members	Detailed

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for concrete

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column. This can be useful for example when wanting to sort members by how much reinforcement they contain.

Beams and Columns

NOTE
- Summary reports provide total quantities for each section size
- Detailed reports provide quantities on a span/stack basis
  - When group design is active this reduces to every detailing group span/stack
- Reinforcement running between beam spans is shared
- Reinforcement running between column stacks is shared
- Reinforcement quantities are given in terms of weight and weight/unit volume
- Reinforcement bar lengths are as shown in the member detail drawings
- No additional detailing allowance has been applied

Fabrication	Content	View Type	Description
Cast-in-place	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Detail	One row for each span/stack: Group/Member ref., section geometry, size, grade, number (of that ref), length, mass, surface area, volume, reinforcement (mass),

Fabrication	Content	View Type	Description
			reinforcement (mass/unit volume)
	Loose Bars	Detail	One row for each type: type, unit mass, total length, total mass
Precast	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
		Detail	One row for each span: Member reference, section geometry, size, grade, number, length, mass, surface area, volume
Post-tensioned (Beams only)	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
		Detail	One row for each span: Member reference, section geometry, size, grade, number, length, mass, surface area, volume

Fabrication	Content	View Type	Description
Cast-in-place	Loose Bars	Detail	One row for each type: type, unit mass, total length, total mass

Walls

NOTE
- Summary reports provide total quantities for each wall
- Detailed reports provide quantities on a wall panel basis
- Reinforcement running between panels is shared
- Reinforcement quantities are given in terms of weight and weight/unit volume
- Reinforcement bar lengths are as shown in the member detail drawings
- No additional detailing allowance has been applied

Fabrication	Content	View Type	Description
Cast-in-place	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Detail	One row for each panel: Member reference, gross mass, mass, gross surface area, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)

Fabrication	Content	View Type	Description
	Loose Bars	Detail	One row for each type: type, unit mass, total length, total mass
	Mesh	Detail	One row for each type: type, unit mass, total area, total mass
Precast	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume
		Detail	One row for each panel: Member reference, gross mass, mass, gross surface area, surface area, volume
	Loose Bars	Detail	One row for each type: type, unit mass, total length, total mass
	Mesh	Detail	One row for each type: type, unit mass, total area, total mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

- Patch and punching reinforcement is approximately distributed by sharing it equally between slab items it touches - it is not split based on covered areas
- Reinforcement allows for laps at the slab item boundary
- An approximate user defined additional detailing allowance (specified in Design Settings for cast-in-place slabs) is applied to all reinforcement defined in any slab type.
- Punching reinforcement weight is approximated (based on ACI guidance)
- Precast and post-tensioned slabs are not designed, but the reinforcement that is specified in the slab properties is reported. The post-tensioned tendon information is not reported and must be allowed for in some other way.

Construc-tion	Fabri-cation	Content	View Type	Description
Composite	N/A	Members	Sum.	One row for each slab: level, slab, depth, manufacturer, reference, gauge, grade, mass, gross surface area, net surface area, volume, reinforcement
			Det.	One row for each slab item: level, slab, depth, manufacturer, reference, gauge, grade, mass, gross surface area, net surface area, volume, reinforcement
		Loose Bars	Det.	One row for each type: type, unit mass, total length, total

Construc-tion	Fabri-cation	Content	View Type	Description
				mass inc. allowance
		Mesh	Det.	One row for each type:
				type, unit mass, total area, total mass inc. allowance
Non-composite	Cast-in-place	Members	Sum.	One row for each slab:
				level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
			Det.	One row for each slab item:
				level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total

Construc-tion	Fabri-cation	Content	View Type	Description
				reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
		Loose Bars	Det.	One row for each type:
				type, unit mass, total length, total mass inc. allowance
		Mesh	Det.	One row for each type:
				type, unit mass, total area, total mass inc. allowance
	Precast	Members	Sum.	One row for each slab:
				level, slab, depth, manufacturer, reference, unit depth, topping, grade, mass, gross surface area, net surface area, volume, reinforcement
			Det.	One row for each slab item:
				level, slab, depth, manufacturer, reference, unit depth,

Construc-tion	Fabri-cation	Content	View Type	Description
				topping, grade, mass, gross surface area, net surface area, volume, reinforcement
		Loose Bars	Det.	One row for each type:
				type, unit mass, total length, total mass inc. allowance
		Mesh	Det.	One row for each type:
				type, unit mass, total area, total mass inc. allowance
	Post Tension	Members	Sum.	One row for each slab:
				level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)

Construc-tion	Fabri-cation	Content	View Type	Description
			Det.	One row for each slab item:
				level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
		Loose Bars	Det.	One row for each type:
				type, unit mass, total length, total mass inc. allowance
		Mesh	Det.	One row for each type:
				type, unit mass, total area, total mass inc. allowance

Punching checks

NOTE The weight of punching shear rails is approximated based on ACI guidance.

Content	View Type	Description
Stud Rails	Detail	One row for each rail ref: ref., diameter,stud height, studs per rail, stud spacing, rail length, total no. of rails, approx. mass

Pad Bases, Pile Caps and Piles

NOTE
- Reports provide quantities for each individual base
- Reinforcement quantities are given in terms of weight and weight/unit volume
- No additional detailing allowance has been applied

Characteristic	Content	View Type	Description
Pad Bases Pile Caps	Members	Detail	One row for each base: member reference, grade, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
	Loose Bars	Detail	One row for each type: type, unit mass, total length, total mass
	Mesh	Detail	One row for each type: type, unit mass, total area, total mass
Piles	Members	Detail	One row for each pile type: reference, dimension, shape, installation type, installation length,

Characteristic	Content	View Type	Description
			number, total length

Material lists for timber

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	View Type	Description
• Beams • Columns • Braces	Summary	One row for each size: size, grade, number (of that size), total length, mass
	Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	Description
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross

Characteristic	Content	View Type	Description
			surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for cold formed

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

Characteristic	Content	View Type	Description
• Beams • Columns • Braces	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
		Detailed	One row for each size & length: size, grade, number (of that size & length),

Characteristic	Content	View Type	Description
			length, mass, surface area

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for general materials

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	Content	View Type	Description
• Beams • Columns • Braces	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area

Characteristic	Content	View Type	Description
		Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area

Walls

Characteristic	Content	View Type	Description
Walls	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume
		Detailed	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Characteristic	Content	View Type	
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Review embodied carbon detail

Embodied carbon can be reviewed in depth by creating an **Embodied Carbon Detail** tabular data view.

Embodied Carbon Detail tabular data views:

- are displayed for one construction type at a time according to the material, characteristic, construction and fabrication type selected in the ribbon
- display a single row listing the total embodied carbon mass per member reference when **Summary Only** is selected
- display separate rows listing the quantities, units, carbon factors, and embodied carbon masses for each material that comprises the member reference when **Summary Only** is unselected
- can have filters applied (page 1058)
- are linked to the 3D view: double-clicking a row in the table locates the row entities in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, (but could be printed if exported to a spreadsheet).

For more details about the practical use of this feature, see: in the topic.

NOTE The embodied carbon mass totals are also included in the Review material list tabular results (page 1029).

NOTE Embodied carbon can also be reviewed in summary by creating an (page 1055) tabular data view.

Create embodied carbon detail tabular results

To display an embodied carbon detail as tabular data:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Embodied Carbon Detail**
4. Choose the construction type by selecting the required material, characteristic, construction and fabrication type from the ribbon.
5. Choose if you want summary or detailed results:
   - To display summary only, in the **View Type** group, select **Summary Only**.
   - To display more detailed results, unselect **Summary Only**.

   The results are displayed as specified.

   If required you can configure a filter to limit the display to selected levels, frames etc. To do this, see: Sort and filter tabular review data (page 1058)

Locate tabular data in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review embodied carbon overview

Embodied carbon can be reviewed in summary form by creating an **Embodied Carbon Overview** tabular data view.

Embodied Carbon Overview tabular data views:

- are either displayed for the whole model, or according to the materials, characteristics, construction and fabrication types selected in the ribbon
- can be displayed grouped by:
  - Construction type
  - Level & Construction type
  - Construction type & Material
  - Material
  - Level
  - Plane
- can have filters applied (page 1058) (selected UDAs, frames, or groups)
- are linked to the 3D view: double-clicking a row in the table locates the row entities in the 3D view
- can be exported to a spreadsheet (page 1059)
- are not suitable for direct printing, but the data can be included in an Embodied Carbon report (page 1080) which can then be printed.

For more details about the practical use of this feature, see: in the topic.

NOTE Embodied carbon can also be reviewed in more depth by creating an (page 1054) tabular data view.

Create embodied carbon overview tabular results

To display an embodied carbon overview as tabular data:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Embodied Carbon Overview**
4. Choose if you want results displayed for **all** available material and member types, or a subset only:

- To display results for all available material and member types, in the **View Type** group, click **Toggle Selection** so that all construction types are selected.
- To display results for a subset of construction types proceed as follows:
  a. In the **Material Type** group, select the material(s).
  b. In the **Characteristic** group, select the required characteristic(s).
  c. In the **Fabrication** group, select the fabrication type(s).

5. Using the droplist in the **View Type** group, choose how you want the results to be reported:
   - **Group by Construction Type**
   - **Group by Level & Construction Type**
   - **Group by Construction Type & Material**
   - **Group by Material**
   - **By Level - Summary**
   - **Group by Plane**

   The **Embodied Carbon Overview** is displayed grouped as specified.

Locate tabular data in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review floored area tabular results

To create a tabular floored area summary:

1. If necessary, switch the view regime to a **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Floored Area**.

A table of floored areas is displayed.

Review portal frame deflection

NOTE This option is only available for Eurocodes.

1. If necessary, switch the view regime to a **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Portal Frame Deflection**.

The horizontal and vertical deflections and deflection check status are reported for the selected portal frames for the selected load combinations. The limiting deflections that are displayed are calculated according to the limits defined in the Portal frame properties (page 1492) window.

Sort and filter tabular review data

You can sort review data, and you can also apply model filters to have it displayed selectively. The filters available depend on the view type displayed.

Sort tabular review data

The data can be arranged into ascending or descending order by any column simply by clicking on the required column header.

- Click on a column header once to arrange in ascending order (▲)
- Click on the same column header again to arrange in descending order (▼)
- Click on the same column header again to return to the original order.

Create and apply filters

1. In the **Filters** group of the **Review Data** ribbon tab, select the desired filter type.

   The **Select filter items** list opens.
2. In the **Select filter items** list, select the necessary options to define the filter requirements.
3. Click **OK**.

   Tekla Structural Designer applies the filter.

Edit filters

NOTE If a filter type is not applicable for the current tabular data view type, it will be unavailable.

1. On the **Review Data** tab, click either **Filter Items...**

   The **Select filter items** list opens.

2. In the **Select filter items** list, select the necessary options to define the filter requirements.

3. Click **OK**.

   Tekla Structural Designer applies the filter.

Export tabular review data to Excel

1. Create the tabular review data view you require (page 1011) and ensure that it contains the necessary information.

2. On the **Review Data** ribbon tab, click **Excel**.

3. Click **Yes** to confirm exporting the report.

   The tabular data opens in Excel

6.4 Charts

The **Charts** command on the **Review** tab opens a **Charts** tab on the ribbon and a **Review Charts** view.

Select the required chart type from the ribbon to display it in the view.

Click **Settings** on the ribbon to customize the appearance of the chart.

Display embodied carbon by level chart

To view this chart:

1. Open a view and switch the view regime to **Review View**.

2. On the **Review** ribbon, click **Charts**.

3. In the **Chart Type** group, select **Embodied Carbon by Level**

4. Click **Settings**. Select the appropriate options to configure the chart as required.

5. Click **OK**. The chart is displayed.

Display embodied carbon by construction type chart

To view this chart:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Charts**.
3. In the **Chart Type** group, select **Embodied Carbon by Construction Type**
4. Click **Settings**. Select the appropriate options to configure the chart as required.
5. Click **OK**. The chart is displayed.

Display embodied carbon by material chart

To view this chart:

1. Open a view and switch the view regime to **Review View**.
2. On the **Review** ribbon, click **Charts**.
3. In the **Chart Type** group, select **Embodied Carbon by Material**
4. Click **Settings**. Select the appropriate options to configure the chart as required.
5. Click **OK**. The chart is displayed.

Adding charts into reports

Charts can also be included in reports.

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. Select the required report in the **Available Styles** list.
3. Drag **Chart** from the **Chapters and Options** pane across to the **Report Structure** pane.
4. Right-click on **Chart** in the Report Structure pane and click **Settings**.
5. From the **Chart Settings** dialog box choose the chart type.
6. From the **Chart Settings** dialog box click the options button . Select the appropriate options to configure the chart as required.

7. Click **OK**.

7 Create reports and drawings

Having analyzed/designed your model in Tekla Structural Designer you can then output the results in the form of reports and/or drawings.

Click the links below to find out more:

- Create and modify reports (page 1062)
- Create drawings (page 1090)

7.1 Create and modify reports

You can tailor a wide range of different reports according to your needs by using the **Report** tab on the ribbon.

Click the following links to find out more:

- Report terminology (page 1063)
- Create reports (page 1065)
- Filter reports (page 1069)
- Format reports (page 1070)
- Navigate reports (page 1073)
- Export reports (page 1075)
- Print reports (page 1076)
- Example reports (page 1076)
- Report settings (page 1086)

NOTE Initially, only the **Contents** group and **Filters** group are displayed on the **Report** tab. Only when a report is displayed and the **Report View** is active, do the **Appearance** group, **Navigation** group, and **Export** group become available.

Report terminology

The following terms are used to described specific aspects of the report creation process in Tekla Structural Designer.

Model reports

Model reports are used to set up the printed output for either the entire structure or a part of it. Model reports are configured by selecting specific output categories (referred to as *chapters*) in a list of all the available output categories. You can include entire chapters, or just the headings within the chapter you require.

The combination of selected chapters is referred to as the *report structure*. If necessary, you can apply filters to individual headings in the report structure to limit the output that is produced.

Model report chapters include:

- Structure:

  Structure data (under headings such as **Loadcases** or **Wind Data**).

- Analysis:

  Analysis model properties, member forces, nodal forces (support reactions), nodal deflections, and other analysis results.

- Concrete, Steel, Timber, Cold Formed, Cold Rolled, and General Material:

  Member reports for each different member types. Each member report can be configured to contain as much or as little output as needed. You can also decide to include design summary tables for the members that Tekla Structural Designer can design.

- Member End Forces and Bracing Forces:

  These provide a useful way of reporting connection design forces.

- Foundation Reactions

- Picture:

  A 3D view of the entire structure. Applying model filters also allows you to view a picture of selected sub structures, frames, or levels. If necessary, you can also include applied loads in the picture by applying a load filter.

- View:

  The current display in any 2D or 3D view can be saved as a view configuration, and then, included in the model report as a View chapter.

- Analysis Diagram:

  A 3D diagram of the entire structure. Applying model filters also allows you to view a picture of selected sub structures, frames, or levels. If necessary,

you can also display the analysis results by applying an analysis method filter.

- Material Listing:

  Tabulated quantities of materials.

- Revision History:

  The revision history that has been recorded in the **Project Wiki**.

Member reports

Member reports are only output as part of a created model report or by right-clicking on an individual member and selecting **Report for Member**.

Member report chapters include:

- Picture:

  The 3D view that you receive by right-clicking a member and selecting **Open [element name] view**.

- Drawing:

  The DXF file that you receive by right-clicking a member and selecting **Generate Detailing Drawing...**

- Loading:

  The table that you receive by right-clicking a member and selecting **Show Member Loading**.

- Different force diagrams and Deflected Shape Diagram:

  The diagrams that you receive by right-clicking a member and selecting **Open Load Analysis View**.

Active model report

The active model report is the default report displayed in the list on the left corner of the **Report** tab, unless another report view is already active.

To specify the active model report, do the following:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. In the **Available Styles** list, select the desired report.
3. Click **>> Active**.

Active member report

The active member report is the report that is generated when you right-click a member in a 2D or 3D view and select **Report for Member**.

To specify the active model report, do the following:

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the desired report.
3. Click **>> Active**.

Active and inactive chapters

In the **Report Contents** dialog box, each chapter in the report structure is marked as either *active* or *inactive*. Only the chapters that are marked as *active* are included in the report that you generate.

Report filters

You can use filters to limit the amount of output in the report.

Tekla Structural Designer offers the following filters for different data types:

- Model filters:

  If necessary, you can filter the report for selected levels, frames, or planes. You can also filter certain data types for selected beams, columns, or walls. If you have applied user-defined attributes to your model, you can also use them to filter the report output.
- Loading filters:

  If necessary, you can filter the report for selected loadcases and combinations.

Available styles

A number of sample reports are available in the **Available Styles** list of the **Report Contents** dialog box. The reports serve as templates and can be modified to suit the model in question. If you cannot find a report that can be customized to meet your needs, you can add further reports to the list by clicking **Add**.

If you start a new project, the same default reports are available. However, they are reset to the default report structures. It is not currently possible to save customized reports from the **Report Contents** dialog box to apply them to other projects.

Create reports

Tekla Structural Designer allows you to create two kinds of reports: you can either create a report based on the entire model or an individual member in the model.

See also

Export reports (page 1075)

Print reports (page 1076)

Format reports (page 1070)

Configure a model report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. Select the report:
   - To select an existing report, click the report name in the **Available Styles** list.
   - To add a report, click **Add** and type the report name in the **Active Style** field.

3. Review the report structure and modify it according to your needs.
   - To include chapters and options in the report, drag them from the left column to the right column.
   - To remove unwanted chapters and options permanently from the report, drag them from the right column to the left column.
   - To exclude a specific chapter from the report but maintain the current report structure, right-click the chapter and clear the **Active** option.
   - To re-arrange the report order, drag and drop chapters within the right column.

4. To limit the output to selected levels, frames, planes, or sub structures, apply a model filter (page 1069).

   TIP You can further limit the output of many chapters by applying a loading filter (page 1069).

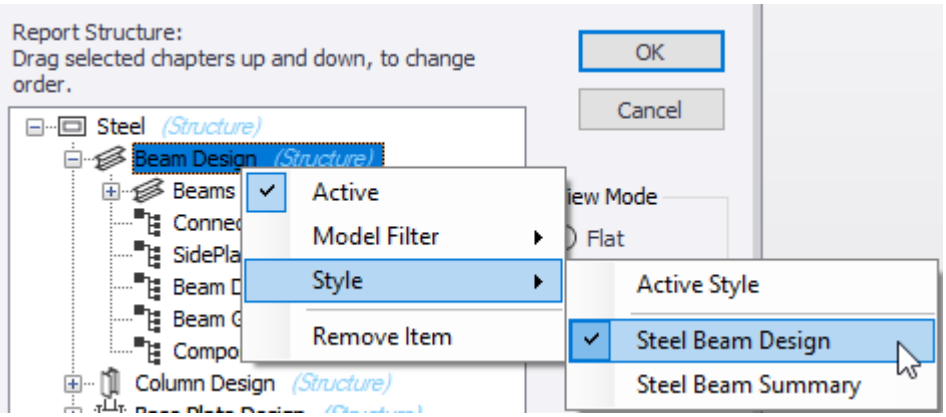
5. If you have created specific view configurations for the model and want to include them in the report, include a separate **View** chapter for each view configuration.

6. If the report structure includes any member chapters (such as beams, columns, or walls), select an appropriate member report style as follows:

   a. In the Report Structure, select the required chapter.

b. Right-click, and from the context menu go to **Style**

c. Click the desired member report style.



NOTE You can either choose a named style as shown above, or you can choose "Active Style" (in which case when you come to display the model report it will adopt whichever member report style is active at that time).

7. Click **OK**.

Display a model report

1. In the list on the far left of the **Report** tab, select the report that you want to view.
2. On the **Report** tab, click **Show Report**.

   Tekla Structural Designer displays the report in a new window. If the report contains loading analysis views of individual members (such as force diagrams or pictures), the views are displayed for each member type according to the selected member report style.

Configure a member report style

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Member Type** list, select the desired member.
3. Select the report:
   - To select an existing report, click the report name in the **Available Styles** list.
   - To add a report, click **Add** and type the report name in the **Active Style** field.

4. Review the report structure and modify it according to your needs.
   - To include chapters and options in the report, drag them from the left column to the right column.
   - To remove unwanted chapters and options permanently from the report, drag them from the right column to the left column.
   - To exclude a specific chapter from the report but maintain the current report structure, right-click the chapter and clear the **Active** option.
   - To re-arrange the report order, drag and drop chapters within the right column.
5. You can limit the output for diagram and end reaction chapters etc. by applying a loading filter (page 1069).
6. If you have included a drawing in the report structure, specify appropriate settings for the drawing as follows:
   a. Right-click the drawing in the report structure.
   b. Click **Settings...**
   c. Define the drawing settings according to your needs.
   d. Click **OK**.
7. Click **OK**

NOTE If several styles are listed in the **Available Styles** list, the one with **(active)** after its name will be the one that gets used when you display a Member Report.

To change this, select a different style in the **Available Styles** list and then click **>> Active**.

Display a member report

Before you display a member report you should ensure the correct member report style has been set as active. This can be done as follows:

1. On the **Report** tab, click **Member Report...**
2. In the **Member Type** list, select the desired member type.
3. In the **Available Styles** list, select the style that you want to use.
4. Click **>> Active**.
5. Click **OK**

You can then proceed to display the report as follows:

1. Hover the mouse pointer over the desired member.
2. Once the outline of the member is high-lighted, right-click the member.

3. In the context menu, select **Report for Member**.

   Tekla Structural Designer opens a report for the selected member in a new window.

Filter reports

Tekla Structural Designer contains two main categories of filters that you can apply to chapters in the report structure. The categories are model filters and loading filters. For more information, see the following paragraphs.

You can apply model filters to enable selective output based on selected:

- Levels
- Frames
- Planes
- Groups
- Members
- Trusses
- Portal frames
- UDAs

Using loading filters enables selective output based on selected:

- Loadcases
- Combinations
- Envelopes

See also

Configure a model report (page 1066)

Create filters

1. In the **Filters** group of the **Report** tab, select the desired filter type.

   The **Edit filters** dialog box opens.
2. Click **Add**.
3. Type a name for the new filter.
4. In the **Selected items** list, select the necessary options to define the filter requirements.
5. Click **OK**.

Tekla Structural Designer creates the filter. You can now apply the filter to a specific report.

TIP If you create a member filter, you can later review and modify it graphically by clicking **Report Filter** on the **Review** tab.

Apply filters

NOTE If a filter type is not applicable for the selected chapter, it will be unavailable.

1. On the **Report** tab, click either **Model Report...** or **Member Report...**

   The **Report Contents** dialog box opens.
2. Select the report that you want to filter.

   In the right column, any chapters that can be filtered are displayed in blue text.
3. Right-click a chapter.
4. In the context menu, select the desired filter type and filter name.

   TIP If you cannot find an appropriate filter, do the following:

   a. In the context menu, go to the filter type and click **Edit\New...**

   b. In the **Select filter** dialog box, click **Add**.

   c. Type the name of the filter in the **Active filter** field.

   d. In the **Selected items** list, select the necessary options to define the filter requirements.

   e. Click **OK**.
5. Click **OK**.

   Tekla Structural Designer applies the filter to the selected report chapter.

Format reports

Tekla Structural Designer provides you with default page header and footer layouts for your reports. However, you can also create, use and save different layouts according to your needs.

You can also modify the appearance of the report body. You can adjust, for example, paragraph styles, page margins and numbering, tables, and headers or footers.

See also

Create reports (page 1065)

Export reports (page 1075)

Print reports (page 1076)

Enter company details in the header

1. On the **Report** tab, create the desired report and click **Show Report**.

   The report is displayed.

2. On the **Report** tab, click **Edit Header**.

   The **Document headers/footers** dialog box opens.

3. Go to the **Headers** page.
4. In the **Available layouts** list, select an existing layout that contains an **Address** cell.
5. Go to the **Fields** page.
6. In the **Available fields** list, select **Address**.
7. In the list on the right, select the necessary company details and click **<<** to include them in the report.
8. To define the text displayed in each included field, select the field in the left **Available fields** list and type the desired text in the **Displayed value** field.
9. To insert the company logo in the header, select **Company Logo** in the left **Available fields** list and click **...** to browse to the desired image.

   TIP
   - Large logos are automatically resized to within the column width in which they are placed. However, using smaller images in PNG format saves memory and speeds up exporting reports.
   - To include more images in the header, click **Add Image Field**, type a name for the field and selected the desired image.

10. Click **OK**.

    The company details are displayed in the report header.

Enter project-specific details in the header

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.

2. On the **Project Summary** page, type the project details in the available fields.

3. Click **OK**.

   The project details are saved.

NOTE When you go back to the **Report** tab, remember to update the report to display the project details in the header.

Create a new header or footer layout

1. On the **Report** tab, create the desired report and click **Show Report**.

   The report is displayed.

2. On the **Report** tab, click **Edit Header**.

   The **Document headers/footers** dialog box opens.

3. According to your needs, go to the **Headers** or **Footers** page.
4. In the **Available layouts** section, click **New**.
5. Type a name for the new layout in the **Name** field.
6. According to your needs, do some of the following to adjust the header layout:

To	Do this
Remove an existing field and create an empty shell	• Hover the mouse pointer over the field in the **Current layout** section until the field name is highlighted. • Click the field that you want to remove it and replace it with an empty cell.

To	Do this
Place a field into an empty cell or replace an existing field	• Hold down the left mouse button and drag the field from the **Available fields** list into the desired cell.

To	Do this
Merge cells	• Hold down the left mouse button and drag over the cells that you want to merge. • To unjoin previously merged cells, right-click the merged cell and in the context menu, select **Unjoin**.

To	Do this
Modify the field alignment within a shell	• Right-click the required cell in the layout. • In the context menu, set the horizontal and vertical alignment options as required.

To	Do this
Insert or remove rows or columns or change their alignment	• Right-click the layout. • In the context menu, select the desired command.

To	Do this

Change column type, width and alignment in the layout	• Under the **Current layout** section, click **Edit...** • Adjust the column properties according to your needs. • Click **OK**.

7. Click **OK**.

   The new layout is created.

Adjust and apply report settings

Report settings allow you to modify the appearance of reports. You can adjust, for example, paragraph styles, page margins and numbering, tables, and headers or footers.

1. On the **Report** tab, click **Settings**.

   The **Settings** dialog box opens.

2. In the **Settings** dialog box, do one or more of the following to adjust the report settings:

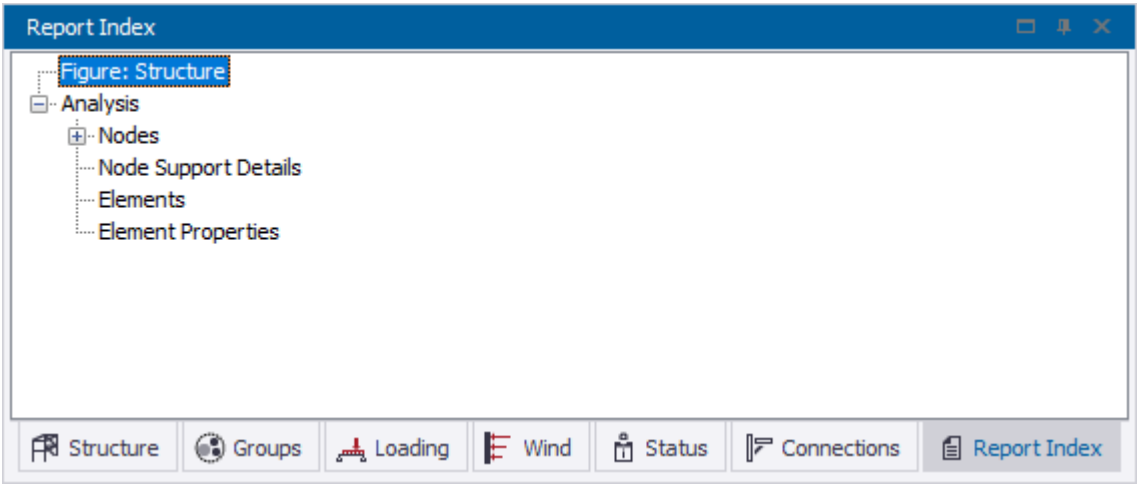
To	Do this
Customize the paragraph styles in the different areas of the report	Go to the **Styles** page and adjust the settings according to your needs.
Adjust the page margin width, page numbering, and margin frame	Go to the **Page Options** page and adjust the settings according to your needs.
Adjust the appearance of tables	Go to the **Table Options** page and adjust the settings according to your needs.
Define whether headers and footers are displayed, adjust image width and paragraph spacing, and control page breaks	Go to the **Document Options** page and adjust the settings according to your needs.
Adjust the appearance of fonts used in pictures and force diagrams	Go to the **Picture Fonts** page and adjust the settings according to your needs.

3. Click **OK**.

   Provided that you adjusted the report settings of an active settings set, the new report settings are applied to the report.

Navigate reports

Use the **Report Index** to quickly navigate to a specific section in a report.



On **Report** tab on the ribbon, you can also use the buttons in the Navigation group to move through the report pages.

Navigation using the Report Index

Open a report view then:

1. On the ribbon, go to the **Report** tab.
2. In the **Navigation** group, select **Report Index**

   The **Report Index** tab is displayed in the **Project Workspace**.
3. Click the **Report Index** tab in the **Project Workspace**.
4. Click on a specific heading or sub heading Report Index to move to that section in the report.

Navigation buttons on the Report tab

Open a report view then:

- Click  to move to the first page
- Click  to move to the previous page
- Click  to move to the next page
- Click  to move to the last page
- Click  to open the **Report Index** in the **Project Workspace**

Export reports

Tekla Structural Designer allows you to export your reports to PDF format, Microsoft Word, Excel, and Tekla Tedds. For detailed instructions, see the following paragraphs.

Export a report to PDF

1. Create the report and ensure that it contains the necessary information.
2. Ensure the Report View containing the report to be exported is the active view.
3. On the **Report** tab, click **PDF**.
4. Click **Yes** to confirm exporting the report.

   The report opens as a PDF file.

Export a report to Microsoft Word

1. Create the report and ensure that it contains the necessary information.
2. Ensure the Report View containing the report to be exported is the active view.
3. On the **Report** tab, click **Word**.
4. Click **Yes** to confirm exporting the report.

   The report opens in Microsoft Word.

Export a report to Excel

1. Create the report and ensure that it contains the necessary information.
2. Ensure the Report View containing the report to be exported is the active view.
3. On the **Report** tab, click **Excel**.
4. Click **Yes** to confirm exporting the report.

   The report opens in Excel

Export a report to Tekla Tedds

1. Create the report and ensure that it contains the necessary information.
2. Ensure the Report View containing the report to be exported is the active view.
3. On the **Report** tab, click **Tedds**.

4. Click **Yes** to confirm exporting the report.

   The report opens in Tekla Tedds

Print reports

To print any reports that you require, see the following instructions.

1. Create the report and click **Show Report**.

   Tekla Structural Designer displays the report.
2. On the **File** menu, go to **Print** --> **Print...**
3. Adjust the printing settings according to your needs.
4. Click **OK**.

Example reports

Tekla Structural Designer provides a number of standard reports that are installed by default. Even though the standard reports may not match your exact requirements, they can often serve as templates that you can modify according to your needs.

Beam End Forces report

The report is typically generated for steel or timber beams in order to provide connection design forces in one of the following configurations:

- End 1/End 2:

  6 forces are output in the support's local co-ordinate system for the selected loadcases, combinations, or envelopes. For each end of each beam span, a single row of data is output for loadcases, combinations, or envelopes in the selected loading filter (Fx, Fy, Fz, Mx, My, Mz). Select this option if you intend to design the connections at both ends of each beam independently.
- End 1/End 2 + Coincident:

  For each end of each beam span, a separate row is output for each loadcase, combination, or envelope according to the applied loading filter for the maximum and the minimum of each of the 6 forces that are output. In addition, the report contains the coincident forces in the other directions.

  Up to 12 rows of data can be output for a loadcase, combination, or envelope as follows:

  - maximum: Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - minimum: Fx (+ coincident Fy, Fz, Mx, My, Mz)

- maximum: Fy (+ coincident Fx, Fz, Mx, My, Mz)
- minimum: Fy (+ coincident Fx, Fz, Mx, My, Mz)
- maximum: Fz (+ coincident Fx, Fy, Mx, My, Mz)
- minimum: Fz (+ coincident Fx, Fy, Mx, My, Mz)
- maximum: Mx (+ coincident Fx, Fy, Fz, My, Mz)
- minimum: Mx (+ coincident Fx, Fy, Fz, My, Mz)
- maximum: My (+ coincident Fx, Fy, Fz, Mx, Mz)
- minimum: My (+ coincident Fx, Fy, Fz, Mx, Mz)
- maximum: Mz (+ coincident Fx, Fy, Fz, Mx, My)
- minimum: Mz (+ coincident Fx, Fy, Fz, Mx, My)

The loadcase, combination, or envelope names in which the maximum or minimum values occur are also reported.

- Higher End Only:

  For each beam span, two rows of data are output for the chosen loading filter: maximum (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz; minimum (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz. This configuration does not contain loadcase, combination, or envelope names because they could vary for each of the 6 values.

- Higher End Only + Coincident (initial default option):

  A separate row is output for the maximum and the minimum of each of the 6 forces, along with the coincident forces in the other directions. For each beam span, up to 12 rows of data can be output for the applied loading filter as follows:

  - maximum (end 1, end 2): Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - minimum (end 1, end 2): Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - maximum (end 1, end 2): Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - minimum (end 1, end 2): Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - maximum (end 1, end 2): Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - minimum (end 1, end 2): Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - maximum (end 1, end 2): Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - minimum (end 1, end 2): Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - maximum (end 1, end 2): My (+ coincident Fx, Fy, Fz, Mx, Mz)
  - minimum (end 1, end 2): My (+ coincident Fx, Fy, Fz, Mx, Mz)
  - maximum (end 1, end 2): Mz (+ coincident Fx, Fy, Fz, Mx, My)

- minimum (end 1, end 2): Mz (+ coincident Fx, Fy, Fz, Mx, My)

The loadcase, combination, or envelope names in which the maximum or minimum values occur are also reported.

TIP To change the default **Beam End Forces** report configuration, do the following:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select **Beam End Forces**.
3. In the **Chapters and Options** list, expand the **Member End Forces** group.
4. Hold down the left mouse button and drag the desired configurations into the report structure.
5. Click **OK**.

To display a **Beam End Forces** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Beam End Forces**.
3. Click **Show Report**.

   TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Bracing Forces report

Tekla Structural Designer provides the bracing design force data for all loadcases and combinations in steel buildings.

To display a **Bracing Forces** report:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Bracing Forces**.
3. Click **Show Report**.

   TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Building Analysis & Drift Checks report

By default this report contains the following tables:

- Loadcase summary

- Drift check or sway check results (depending on regional code)
- Wind drift check results

To display the **Building Analysis & Drift Checks** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Building Analysis & Drift Checks**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Building Design report

By default, the report contains the design results for the building at a summary level.

To display a **Building Design** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Building Design**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Building Loading report

By default, the report contains a picture of the model and the following tables of loading related input data:

- Action codes
- Resistance codes
- Combinations
- Wind data

To display a **Building Loading** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Building Loading**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Connection Resistance report

Tekla Structural Designer provides the bracing design force data for all loadcases and combinations in steel buildings.

To display a **Connection Resistance** report:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Connection Resistance**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Embodied Carbon report

The default report summarizes the embodied carbon in the structure in two tables:

- Embodied Carbon Overview by Level

  The level reference, level, embodied carbon mass, floored area, and embodied carbon mass per floored area are output for each level.

  > **NOTE** Level references are only included for those levels which have **Reporting Level** selected in Level Properties (page 1333).

- Embodied Carbon Overview by Construction Type

  The embodied carbon mass is output each construction type.

To display an **Embodied Carbon** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Embodied Carbon**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Foundation Reactions report

The report summarizes the foundation design forces and is available in two different configurations:

- Foundation Reactions (initial default option):

  6 forces are output in the support's local co-ordinate system for the selected loadcases, combinations, or envelopes.

- Foundation Reactions + Coincident

  The maximum and minimum value of force in the support's local co-ordinate system are output in separate rows for each of the 6 reaction directions (where a value exists in that direction). The loadcase/combination/envelope to which the maximum or minimum value relates is also output. In addition, the same row reports the coincident forces that exist in the other directions for the loadcase/combination/envelope in which the maximum or minimum value occurs.

  NOTE The table for the Foundation Reactions + Coincident option never contains more than 12 rows, irrespective of the number of selected loadcases or combinations. (Max. and min. in each of the 6 directions).

To display a **Foundation Reactions** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Foundation Reactions**.
3. Click **Show Report**.

   TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

   In addition to model and loading filters, you can also reconfigure settings for each of the two configurations. These settings provide options for:

   - choosing strength or service factors
   - choosing to display all rows, or non-zero rows only
   - excluding reactions from columns assigned to cores
   - excluding reactions from walls assigned to cores

4. If required you can also change the axis system that the reactions are being reported in. For instructions on how to do this, see: Working with foundation reaction axis systems (page 726)

Industrial Structure Loading report

By default, this report contains tables for the following industrial structure loading data:

- Line Ancillary
- Area Ancillary
- Equipment

To display an **Industrial Structure Loading** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Industrial Structure Loading**.
3. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

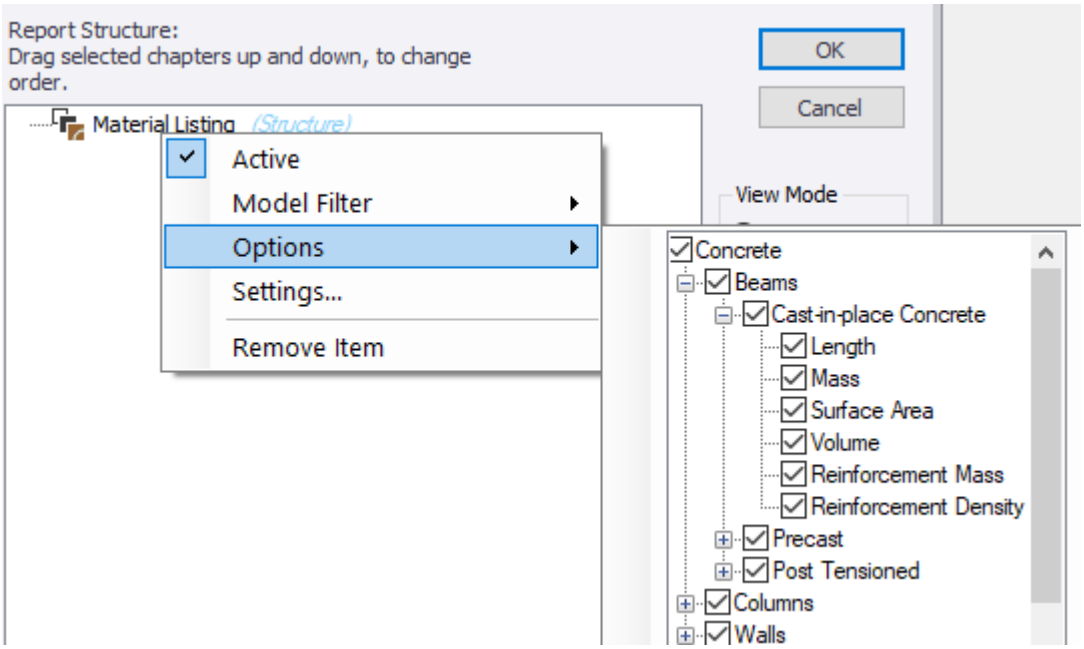
Material Listing report

To configure and then display a **Material Listing** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. Select **Material Listing** from the list in the **Contents** group.
3. Click **Model Report...**

   The **Report Contents** dialog box opens.
4. The default report is created for all materials and entity types in the model and includes all associated properties. If required you can configure which materials/entity types are to be included in the report and also choose to exclude properties that you don't require. This is done as follows:

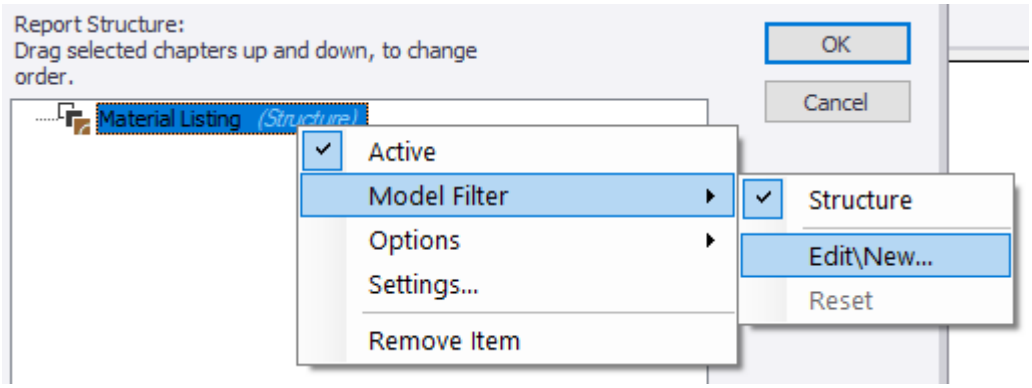
   a. Right-click on the **Material Listing** chapter in the Report Structure and choose **Options** from the context menu.



   b. Uncheck those materials/entity types and any properties that you don't want to include in the report.

5. The default report is created for the whole structure. If required you can configure a filter to limit the report to selected floor levels only, or other criteria as follows:

   a. right-click on the **Material Listing** chapter once again and choose **Model Filter > Edit/New**



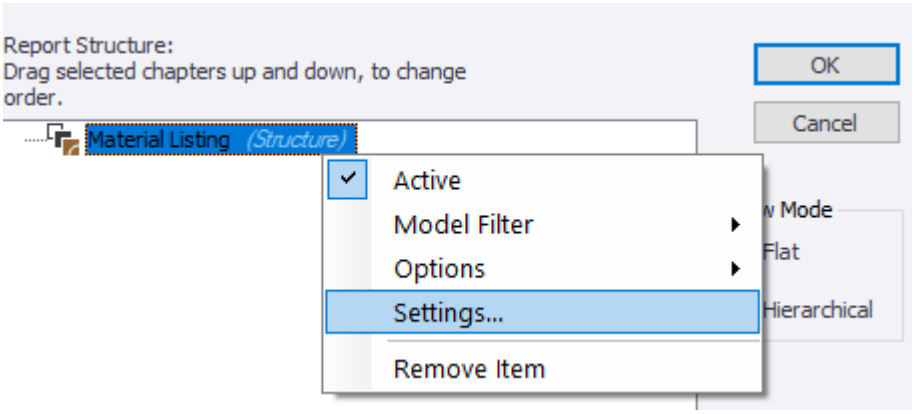
   b. Click Add

   c. Select the type of filter

   d. Select the items to be included

   e. Click OK

6. To choose between a summary or detailed report, right-click on the **Material Listing** chapter once again and choose **Settings**



   - Uncheck **Show summary only** for a detailed report
   - Leave it checked for a summary report

NOTE To find out exactly what summary and detailed level output is produced for each material type and characteristic, see:

- Steel material lists (page 1034)

- Concrete material lists (page 1039)
- Timber material lists (page 1050)
- Cold formed material lists (page 1051)
- General material lists (page 1052)

7. Click **OK** to close the settings dialog.
8. Click **OK** to close the **Report Contents** dialog. The report is displayed according to the configuration choices that you have applied.

Member Design Calcs report

By default, the report contains the design results for each member at a summary level.

To display a **Member Design Calcs** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Member Design Calcs**.
3. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Open Structure Wind Load report

By default, this report contains tables of the open structure wind loads applied to the following entities:

- Member
- Line Ancillary
- Area Ancillary
- Equipment

To display an **Open Structure Wind Load** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Open Structure Wind Load**.
3. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Seismic Design report

By default, the report contain the following tables:

- Seismic loading summary
  - When using the seismic code ASCE 7, torsional irregularity and amplification must always be checked for. To assist with this, accidental torsion amplification tables are included in the summary when amplification is required, irrespective of whether it is actually applied. An explanitory note is provided in the report when no amplification has been applied.
- Analysis seismic drift results

To display a **Seismic Design** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Seismic Design**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

Solver Model Data report

By default, the report contains a picture of the model and the following tables of analysis model input data:

- Nodes
- Node support details
- Elements
- Element members
- Element properties
- Contains to PDF

To display a **Solver Model Data** report, do the following:

1. On the ribbon, go to the **Report** tab.
2. In the list on the left side, select **Solver Model Data**.
3. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 1066).

See also

Create reports (page 1065)

Report settings

Click **Home** --> **Settings** --> **Report** to edit the following report settings.

NOTE Unlike other settings, changes made to **Report Settings** in the active settings set are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

You can also access the same settings when a report is displayed in the active window by clicking **Report** --> **Settings**.

Styles

Setting	Description
Report Styles list	Allows you to select a style used in reports whose font, color, or other properties you want to modify.
Font --> **Change...**	Allows you to change the font used for the currently selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the foreground color of the selected style.
Colors --> **Background** --> **Clear**	Allows you to use a clear background for the selected style.
Colors --> **Background** --> **Change...**	Allows you to change the background color of the selected style.
Vertical alignment	Allows you to modify the vertical alignment of the selected style.
Indentation	Allows you to determine the indentation of the selected style.
Line spacing	Allows you to determine the line spacing of the selected style.

Page Options

Setting	Description
Page margins	Allows you to adjust the margins on report pages.
Margin frame --> **Draw page margin frame**	Allows you to select whether Tekla Structural Designer creates a margin frame to separate the report text from the margins.
Margin frame --> **Color** --> **Change...**	Allows you to change the color of the margin frame.

Setting	Description
First page number	Allows you to determine the number from which the page numbers start.
Page number prefix	Allows you to specify a prefix to the page number.

Table Options

Setting	Description
Border Style	Allows you to specify the table border style. You can select to have no table borders, a single line border, or a double line border.
Border Properties --> **Color** --> **Change...**	Allows you to change the table border color.
Border Properties --> **Width**	Allows you to adjust the table border width.
Table width --> **[] percent of page width**	Allows you to specify the width of all tables included in reports.
Use landscape orientation for wide tables	When enabled (default) the report page orientation will automatically change to landscape mode when the table header will not fit in portrait orientation. **NOTE** You can reduce the need for landscape orientation and number of pages by; reducing page margins (default values may be quite wide); increasing the allowable table width from 90 to 95% of the available width

Document Options

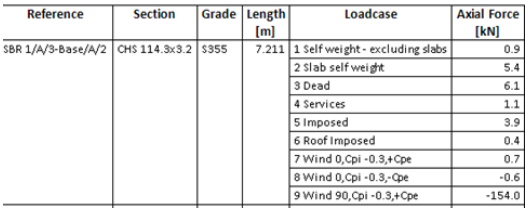
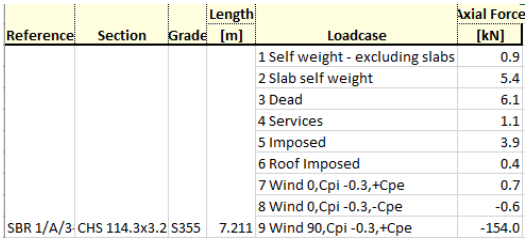
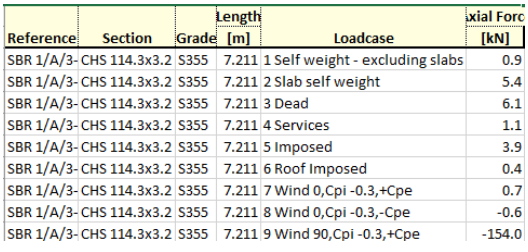
Setting	Description
Show document header	Allows you to include the header at the top of each report page.
Show border in document header	Draws a border around each field cell in the header.
Show document footer	Allows you to include the header at the top of each report page.

Setting	Description
Show border in document footer	Draws a border around each field cell in the footer.
Show document field description in line with the value	Allows you to display the field descriptions in line with the field values in each cell. If you do not select the **Show document field description in line with the value** option, the field values are displayed on a new line.
Underline document field cell	Underlines the cell value of each field.
Image width --> **[] percent of page width**	Allows you to specify the width of all images included in reports.
Paragraphs --> **Spacing**	Allows you to specify the paragraph spacing in reports.
Start each item on new page	Allows you to start each report chapter on a new page.
Start each member report on new page	Allows you to start each member report on a new page.

Picture Fonts

Setting	Description
Reset Fonts	Resets all the fonts to the default settings.
Font Name	Displays the report item to which the font applies.
Font	Allows you to change the font used for a report item.
Size	Allows you to adjust the size of the font.
Bold	Allow you to further format the font.
Italic	

Excel Export

Setting	Description
Disable cell merging	As shown below, columns of data are typically merged in reports to improve readability.  How this data appears when exported to Excel will depend on this setting. Provided cell merging is not disabled a similar result is obtained.  If the option to disable cell merging is selected, the data is repeated on each row. This is often necessary if you intend to manipulate the data in Excel. 

Reference	Section	Grade	Length [m]	Loadcase	Axial Force [kN]
SBR 1/A/3-Base/A/2	CHS 114.3x3.2	S355	7.211	1 Self weight - excluding slabs	0.9
				2 Slab self weight	5.4
				3 Dead	6.1
				4 Services	1.1
				5 Imposed	3.9
				6 Roof Imposed	0.4
				7 Wind 0,Cpi -0.3,+Cpe	0.7
				8 Wind 0,Cpi -0.3,-Cpe	-0.6
				9 Wind 90,Cpi -0.3,+Cpe	-154.0

Reference	Section	Grade	Length [m]	Loadcase	Axial Force [kN]
				1 Self weight - excluding slabs	0.9
				2 Slab self weight	5.4
				3 Dead	6.1
				4 Services	1.1
				5 Imposed	3.9
				6 Roof Imposed	0.4
				7 Wind 0,Cpi -0.3,+Cpe	0.7
				8 Wind 0,Cpi -0.3,-Cpe	-0.6
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	9 Wind 90,Cpi -0.3,+Cpe	-154.0

Reference	Section	Grade	Length [m]	Loadcase	Axial Force [kN]
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	1 Self weight - excluding slabs	0.9
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	2 Slab self weight	5.4
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	3 Dead	6.1
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	4 Services	1.1
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	5 Imposed	3.9
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	6 Roof Imposed	0.4
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	7 Wind 0,Cpi -0.3,+Cpe	0.7
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	8 Wind 0,Cpi -0.3,-Cpe	-0.6
SBR 1/A/3-	CHS 114.3x3.2	S355	7.211	9 Wind 90,Cpi -0.3,+Cpe	-154.0

Export to Tekla Tedds

Setting	Description
Report detail	Set the level of calculation detail to be exported: • None • Summary • Full

Setting	Description
Include summary table	Include a tabular summary of the calculation results.
Include section sketch	Include a sketch.

NOTE The above settings are used to configure the initial default output options applied to Tedds calculations when the calculation is first run. If the output options are changed within the calculation the updated options are saved to the calculation.

To generate a report you run the Export to Tekla Tedds (page 1183) command. This will apply the output options that are saved within the calculation. Therefore, if you want to update the output options in an existing calculation, re-run the calculation to change the output options and export to Tedds once more.

See also

Adjust and apply report settings (page 1073)

7.2 Create drawings

You can create drawings of individual levels and frames by using the commands on the **Draw** tab. As for single-member drawings, they are created by right-clicking the desired object and selecting the correct option in the context menu that appears. In addition, you can create drawings in batches using the **Drawing Management...** commands.

Click the following links to find out more:

- Drawing categories (page 1091)
- Adjust and apply drawing settings (page 1093)
- Create planar drawings (page 1097)
- Create member detail drawings (page 1103)
- Create slab and mat drawings (page 1107)
- Create foundation drawings (page 1108)
- Create concrete member schedule drawings (page 1110)
- Manage drawings in batches (page 1112)
- Manage schedule drawings in batches (page 1117)
- Drawing settings (page 1119)

Drawing categories

In Tekla Structural Designer there are five main drawing categories. Each of the categories contains different drawing variants, and each variant has a number of specific options that allows you to configure their appearance. In addition, each variant contains a set number of drawing layers that are switched on and off according to the selected layer configuration. The appearance of each layer is controlled by the layer style. Drawing options, drawing layers, and layer styles can be managed in drawing settings.

See the different drawing categories and variants in the following table:

Category	Drawing variant	Description
Planar drawings	General arrangement	General arrangement (GA) of 2D levels and frames.
	Foundation reactions	GA that also contains support reactions in order to assist foundation design.
	Loading plan	GA that also contains applied loads for the selected loadcase.
	Beam end forces	GA that also contains forces at the ends of steel beams in order to assist steel connection design.
	Column splice loads	Frame GA that also contains the splice loads at splice locations in steel columns in order to assist steel connection design.
Member details	Concrete beam detail	Beam reinforcement in elevation and section for each span. If necessary, you can also include a reinforcement quantity table.
	Concrete column detail	Column reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

Category	Drawing variant	Description
	Concrete wall detail	Wall reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.
	Non-concrete beam detail	Individual details for non-concrete beams.
	Non-concrete column detail	Individual details for non-concrete columns. When applicable, this can include the associated base plate detail.
	Base plate detail	Individual details for column base plates. Applicable to EC & US regional codes
Concrete member schedules	Concrete beam schedule	Tabular data created by building, by floor, or by selected beams. The information shown in the schedule is based on the design groups.
	Concrete column schedule	A table containing cross sections through each stack for the selected columns. If necessary, you can also include a reinforcement quantity table.
	Concrete wall schedule	A table containing cross sections through each stack for the selected walls. If necessary, you can also include a reinforcement quantity table.
Slabs and mats	Slab/mat layout	Slab item, patch and punching shear reinforcement requirements. If necessary, you can also include a reinforcement quantity table for the reinforcement displayed

Category	Drawing variant	Description
		with an added detailing allowance.
	Punching check detail	Punching shear reinforcement details for individual punching check locations.
Foundations	Isolated foundation detail	Individual foundation details in plan. If necessary, you can also include the detail in cross section and a reinforcement quantity table.
	Foundation layout	GA at foundation level displaying isolated bases, pile layouts, and mats. If necessary, you can also include isolated foundation details, a reinforcement quantity table, and an isolated foundation schedule.

Adjust and apply drawing settings

To adjust the drawing settings of the current project or set them as default settings for future projects, see the following instructions.

Adjust drawing settings in the current project

1. On the **Draw** tab, click **Settings**

   The **Drawing Settings** dialog box opens.

2. Review and modify the drawing settings (page 1119) according to your needs.

3. Do one of the following:

   - To apply the changes to the current project, click **OK**.
   - To save the changes to the selected settings set, click **Save...**
   - To revert to the drawing settings specified in the selected settings set, click **Load...**

Adjust drawing settings in future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Drawings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. Review and modify the drawing settings (page 1119) according to your needs.
5. To save the settings as defaults for future projects that use the selected settings set, click **OK**.

See also

Drawing settings (page 1119)

Create drawing scales

You can only create new drawing scales by accessing the **Settings** dialog box via the **Home** tab. For more information on creating scales, see the following instructions.

1. On the **Home** tab, click **Settings**.
2. Go to the **Drawings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. In the **Drawing Variant** list, select the drawing category that you want to configure.
5. Type the desired scale in the **Scale** field.
6. Click **Add**.

   The new scale is added to the **Available Styles** list.
7. Click **OK**.

TIP If necessary, you can also adjust the distance between independent drawing blocks by typing a value in the **Minimum Text Block Spacing** field. However, note that if the value is too great, text labels can end up far from the objects to which they refer.

Create, modify, or delete layer configurations

Layer configurations control which layers are displayed when the drawing is created. Tekla Structural Designer contains several default layer configurations that you can modify. If necessary, you can also create your own layer configurations.

Create a new layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** subpage.
3. In the **Drawing Variant** list, select the drawing category whose layer configurations you want to modify.
4. Click **Add**.
5. Name the new layer configuration.
6. Select or clear the layers according to your needs.
7. Click **OK**.

Modify a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** subpage.
3. In the **Available Configurations** list, select the configuration that you want to modify.
4. Select or clear the layers according to your needs.
5. Click **OK**.

Copy a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** subpage.
3. In the **Drawing Variant** list, select the drawing category whose properties you want to copy.
4. In the **Available Configurations** list, select the configuration that you want to copy properties from.
5. Click **Add copy...**

   The **Copy Drawing Item** dialog box opens.
6. Select the properties that you want to copy.
7. Name the copy.
8. Select or clear the layers according to your needs.
9. Click **OK**.

Delete a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** subpage.
3. In the **Drawing Variant** list, select the drawing category whose layer configurations you want to modify.
4. In the **Available Configurations** list, select the configuration that you want to remove.
5. Click **Remove**.

   The selected layer configuration is deleted for the selected drawing category.
6. Click **OK**.

Create, modify, or delete layer styles

Drawing styles control how a drawing is displayed, for example, what line types, fonts, and colors are used. Tekla Structural Designer contains several default layer styles that you can modify. If necessary, you can also create your own layer styles.

Create a new layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** subpage.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. Click **Add**.

   A new layer style is added to the **Available Styles** list.
5. Name the new layer style.
6. Adjust the layer properties according to your needs.
7. Click **OK**.

Modify an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** subpage.
3. In the **Available Styles** list, select the layer style that you want to modify.
4. Adjust the layer properties according to your needs.

5. Click **OK**.

Copy an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** subpage.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. In the **Available Styles** list, select the layer style that you want to copy properties from.
5. Click **Add copy...**

   The **Copy Drawing Item** dialog box opens.
6. Select the drawing item that you want to copy.
7. Click **OK**.
8. Name the copied layer style.
9. Adjust the layer properties according to your needs.
10. Click **OK**.

Delete an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** subpage.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. In the **Available Styles** list, select the layer style that you want to delete.
5. Click **Delete**.

   The selected layer configuration is deleted for the selected drawing category.
6. Click **OK**.

Create planar drawings

Five drawing variants are grouped together as planar drawings:

- **General Arrangement** - Layout drawing of a 2D level or frame.
- **Beam End Forces** - General arrangement that includes the forces at steel beam ends. (Use case: steel connection design)

- **Column Splice Loads** - Frame general arrangement that includes the loads at splice locations in steel columns. (Use case: steel connection design)
- **Foundation Reactions** - General arrangement that includes support reactions. (Use case: foundation design)
- **Loading Plan -** General arrangement that includes applied loads for a selected loadcase.

You can create a General Arrangement (page 1098) or Loading Plan (page 1102) at any time (from any view regime).

You can create a Beam End Forces (page 1099), Column Splice Loads (page 1100) or Foundation Reactions (page 1101) drawing for a chosen analysis type and loading (having first specified the analysis type required from the **Results** tab.)

NOTE Before you create a drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**
2. Go to **Options**.
3. Click the drawing type whose options you want to review or adjust.
4. Adjust the drawing options according to your needs.
5. Click **OK**.

See also

Create general arrangement drawings (page 1098)

Create loading plan drawings (page 1102)

Create beam end force drawings (page 1099)

Create column splice load drawings (page 1100)

Create foundation reaction drawings (page 1101)

Create general arrangement drawings

General arrangement drawings display 2D levels and frames, and can only be created in 2D views. To create general arrangement drawings, do the following:

1. Open a 2D view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. On the **Draw** tab, click **General Arrangement**.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.

5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 1094)

Create beam end force drawings

Beam end force drawings are general arrangement drawings that also contain support reactions. They are typically created to assist connection design in steel structures and can only be created in 2D views. To create beam end force drawings, do the following:

NOTE Beam end forces are not displayed for concrete beams.

1. Open a 2D results view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. On the **Results** tab; in the **Result Type** group, use the **Method** list to select the analysis type for which the results are to be included on the drawing.
3. In the **Loading** list, select the loadcase or combination that you want to display.
4. On the **Draw** tab, click **Beam End Forces**.

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.

- Clear the **Use automatic file name** option and name the file yourself.

9. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 1094)

Create column splice load drawings

Column splice drawings are general arrangement drawings that also contain applied loads for the selected loadcase. Column splice drawings are typically created to assist connection design in steel structures. To create column splice load drawings, do the following:

NOTE Column splice loads are not relevant in concrete structures, and therefore, they are not displayed for concrete beams.

---

1. Open a frame view containing the steel frame for which you want to see the column splice loads.
2. On the **Status bar**, click  **Results View**.
3. On the **Results** tab; in the **Result Type** group, use the **Method** list to select the analysis type for which the results are to be included on the drawing.
4. In the **Loading** list, select the loadcase or combination that you want to display.
5. On the **Draw** tab, click **Column Splice Loads**.

   TIP If the **Column Splice Loads** command is not active, ensure that the current view is displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
6. Select the layer configuration and layer style for the drawing.
7. Set the drawing scale.
8. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
9. Do one of the following:

- Accept the automatic file name.
- Clear the **Use automatic file name** option and name the file yourself.

10. Click **OK**.

    The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 1094)

Create foundation reaction drawings

Foundation reaction drawings are general arrangement drawings that also contain support reactions in order to assist foundation design. Foundation reaction drawings can only be created in 2D results views. To create foundation reaction drawings, do the following:

1. Open a 2D results view displaying the part of the model that you want to include in the drawing.

   Typically, the part is be the base construction level.

2. On the **Results** tab; in the **Result Type** group, use the **Method** list to select the analysis type for which the results are to be included on the drawing.
3. In the **Loading** list, select the loadcase or combination that you want to display.
4. On the ribbon, go to the **Draw** tab.
5. Click **Foundation Reactions**.

   The **DXF Export Preferences** dialog box opens.

6. Select the layer configuration and layer style for the drawing.
7. Set the drawing scale.
8. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
9. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
10. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

11. If required you can change the axis system that the reactions are being reported in. For instructions on how to do this, see: Working with foundation reaction axis systems (page 726)

See also

Create drawing scales (page 1094)

Create loading plan drawings

Loading plan drawings are general arrangement drawings that also contain applied loads for the selected loadcase. Loading plan drawings can only be created in 2D scene views. To create loading plan drawings, do the following:

1. Open a 2D view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. In the **Loading** list, select the loadcase that you want to display.
3. On the **Draw** tab, click **Loading Plan**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 1094)

Create member detail drawings

Member detail drawings include concrete beam detail drawings, concrete column detail drawings, concrete wall detail drawings, non-concrete beam detail drawings and non-concrete column detail drawings. For instructions on how to create different member detail drawings, see the following paragraphs.

NOTE Before you create a drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to **Settings** --> **Member Details**.
3. Click the drawing type whose options you want to review or adjust.
4. Adjust the drawing options according to your needs.
5. Click **OK**.

See also

Create drawing scales (page 1094)

Create concrete beam detail drawing

Concrete beam drawings display the beam reinforcement in elevation and section for each span. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the beam that you want to detail.
2. When the desired beam is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete column detail drawing

Concrete column drawings display the column reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the column that you want to detail.
2. When the desired column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete wall detail drawing

Concrete wall details drawings display the wall reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the wall that you want to detail.
2. When the desired wall is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.

6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create non-concrete beam detail drawing

Non-concrete beam detail drawings display individual steel beam details.

1. Hover the mouse pointer over the beam that you want to detail.
2. When the desired beam is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create non-concrete column detail drawing

Non-concrete column detail drawings display individual steel column details.

1. Hover the mouse pointer over the column that you want to detail.

2. When the desired column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create base plate detail drawing

Base plate detail drawings display individual column base plate details.

1. Hover the mouse pointer over the base plate that you want to detail.
2. When the desired base plate column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create slab and mat drawings

Slab and mat drawings refer to slab and mat layout drawings and punching shear check detail drawings. Slab and mat detail drawings are used to convey slab reinforcement and patch reinforcement requirements, and contain a quantity table for the reinforcement displayed with a detailing allowance added. Punching check detail drawings display an individual check detail in, and also contain the option to display the reinforcement quantities table.

See also

Create drawing scales (page 1094)

Create slab or mat layout drawings

1. Open a 2D scene view displaying the slabs or mats that you want to include in the drawing.
2. On the **Draw** tab, click **Slab/Mat Detailing**.

   NOTE If the **Slab/Mat Detailing** command is not active, ensure that you are in a 2D scene view.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create punching shear check detail drawings

1. Hover the mouse pointer over the punching shear check item that you want to detail.
2. When the desired punching shear check item is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create foundation drawings

Foundation drawings refer to isolated foundation detail drawings and foundation layout drawings. Isolated foundation detail drawings display an individual foundation detail in plan. If necessary, you can also display the detail in cross section and a reinforcement quantity table. As for foundation layout drawings, they display the piling layout and the layout of isolated foundations. If necessary, you can also display isolated foundation details, a reinforcement quatity table, an isolated foundation schedule, and a pile location table.

See also

Create drawing scales (page 1094)

Create isolated foundation detail drawings

NOTE Before you create a base cap detail drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to **Settings** --> **Isolated Foundation Detail**.
3. Adjust the drawing options on the **Content** and **Isolated Foundation Detail** sub pages according to your needs.
4. Click **OK**.

---

1. Hover the mouse pointer over the base cap or pile cap that you want to detail.
2. When the desired base cap or pile cap is highlighter, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create foundation layout drawings

1. Open a 2D scene view displaying the piles, bases or pile caps that you want to include in the drawing.
2. On the **Draw** tab, click **Foundation Layout**.

NOTE If the command is not active, ensure that you are in a 2D scene view displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.

5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete member schedule drawings

Concrete member schedule drawings include concrete beam schedules, concrete column schedules and concrete wall schedules. In beam schedule drawings, the reinforcement is listed on a span by span basis, and contractors or specialist detailing firms can use the information to produce the necessary bar bending schedules. As for concrete column schedule drawings, they display a cross section through each stack of the selected columns, and if necessary, allow you to include a reinforcement quantity table. Concrete wall schedule drawings display a cross section through each stack of the selected walls, and if necessary, also allow you to include a reinforcement quantity table

NOTE Before you create a schedule drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to **Settings**.
3. Adjust the drawing options on the sub pages according to your needs.
4. Click **OK**.

See also

Create drawing scales (page 1094)

Create concrete beam schedule drawings

You can create beam schedule drawings by building, by floor or by selected beams. The information shown in the schedule is based on the different design groups. Beam schedule drawings are created in dxf format even though

they do not include graphical information, so that you can add them to beam detail drawings.

1. Open a 2D scene view of a part displaying the beams that you want to include in a schedule. The part can be, for example, a construction level, frame, or sub model.
2. On the **Draw** tab, click **Beam Schedule**.

   NOTE If the **Beam Schedule** command is not active, ensure that you are in a 2D scene view that is displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete column schedule drawings

1. Open a 3D view or a frame view displaying the columns that you want to include in the schedule.
2. On the **Draw** tab, click **Column Schedule**.

   The **Select column schedule content** dialog box opens.
3. Select the columns to be included and click **OK**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:

- Accept the automatic file name.
- Clear the **Use automatic file name** option and name the file yourself.

8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete wall schedule drawings

1. Open a 3D view or a frame view displaying the walls that you want to include in the schedule.
2. On the **Draw** tab, click **Wall Schedule**.

   The **Select wall schedule content** dialog box opens.
3. Select the walls to be included and click **OK**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Manage drawings in batches

Although you can create individual drawings according to your needs, creating a batch of drawings at once is often more efficient, especially when working with larger models.

The **Drawing Management...** command on the **Draw** tab allows you to:

- Select the drawing variant that you want to manage.

- Either add a drawing sheet manually or generate a series of drawing sheets.
- Select the frames, levels, or members for which you want to create drawings.
- Arrange the drawings on the drawing sheet either in a linear or a grid arrangement.
- Select the loadcases or combinations for load-dependent drawing variants.
- Create drawing revisions.
- View the revision history of drawings.
- Reset reinforcement marks on concrete detail drawings in order to remove gaps in the bar mark numbering.

Create or generate drawings in batches

To easily create multiple drawings of any particular variant , be they base plate details, punching check details, concrete member details, see the following instructions:

To place multiple drawings of a single variant, (base plate detail, punching check detail, concrete member detail etc.) on to one or more drawing sheets, see the following instructions:

See also

Specify the drawing layout (page 1114)

Specify the loading for load-dependent drawings (page 1114)

View drawings (page 1115)

Review drawings (page 1116)

Reset reinforcement marks in concrete detail drawings (page 1115)

Create a new batch drawing manually

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. Click **Add**.

   A new drawing appears in the **Available Drawings** list.
4. Name the new drawing.
5. Click **Content...**

   The **Drawing Content** dialog box opens.

6. Drag the items that you want to include in the drawing from the left column to the right column.
7. Click **OK**.
8. Click **View Drawing...**

Generate a new concrete beam or column detail drawing

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Drawing Variant** list, select either **Concrete Beam Detail** or **Concrete Column Detail**.
3. Click **Generate**.

   New drawings are created with automatically generated content. One drawing contains typical beams or columns, wheras additional drawings can contain any ungrouped beams or columns.
4. If necessary, click the available drawings to rename them, and type a new name.
5. Click **View Drawing...**
6. Select the drawing you want to create from the available drawings list.
7. Click **View Drawing...**

Specify the drawing layout

To specify the layout of the drawings that you have created, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Layout...**

   The **Drawing Layout** dialog box opens.
5. Define the direction and arrangement of the layout.
6. Click **OK**.

See also

Specify the loading for load-dependent drawings (page 1114)

Specify the loading for load-dependent drawings

To specify the loading for load-dependent drawings, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Loading...**

   The **Select loading** dialog box opens.
5. Select the desired loadcases and combinations.
6. Click **OK**.

See also

Create or generate drawings in batches (page 1113)

Reset reinforcement marks in concrete detail drawings

Each bar geometry used in the model has an associated reinforcement mark. However, if a model is designed and some of the bars fall out of use, the mark assignment is still retained. As a result, there may be gaps in the marks and marks starting at high numbers. To avoid this, you can reset all the reinforcement marks in the model. Do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Drawing Variant** list, select either **Concrete Beam Detail** or **Concrete Column Detail**.
3. Click **Reset ALL Marks**.

   All reinforcement marks are reset.

See also

Create or generate drawings in batches (page 1113)

View drawings

To view the drawings that you have created in the **Drawing Management** dialog box, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **View Drawing...**

   The **DXF Export Preferences** dialog box opens.

5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create or generate drawings in batches (page 1113)

Review drawings

To apply a revision to a drawing, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Create Revision...**

   The **Create Drawing Revision** dialog box opens.
5. Name the revision and type a revision note.
6. Click **OK**.

   The drawings open in an available DXF tool.

See also

View the revision history of drawings (page 1116)

View the revision history of drawings

To view the formerly applied revisions to a drawing, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **History...**

   The **Drawing History** dialog box opens and allows you to view the revision history of the selected drawing.

See also

Review drawings (page 1116)

Manage schedule drawings in batches

Although you can create individual schedule drawings according to your needs, creating a batch of schedule drawings at once is also possible. For more information, see the following instructions.

The **Schedule Management...** command on the **Draw** tab allows you to:

- Select the drawing variant that you want to manage.
- Create new drawing sheets.
- Select the members that you want to include in the drawing.
- Arrange the drawing layout.
- Create drawing revisions.
- Reset reinforcement marks in order to remove gaps in the bar mark numbering.

Create new schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. Click **Add**.

   A new drawing appears in the **Available Drawings** list.
4. Name the new drawing.
5. Click **Content...**

   The **Drawing Content** dialog box opens.
6. Drag the items that you want to include in the drawing from the left column to the right column.
7. Click **OK**.

Specify the schedule drawing layout

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. Click **Layout...**

   The **Drawing Layout** dialog box opens.
3. Define the direction and arrangement of the layout.
4. Click **OK**.

View schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **View Drawing...**

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 1113), (allowing multiple drawings to be created on a single drawing sheet).

Reset reinforcement marks in schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Drawing Variant** list, select **Concrete Beam Schedule**, **Concrete Column Schedule**, or **Concrete Wall Schedule**.

3. Click **Reset ALL Marks**.

   All reinforcement marks are reset.

Review schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Create Revision...**

   The **Create Drawing Revision** dialog box opens.
5. Name the revision and type a revision note.
6. Click **OK**.

   The drawings open in an available DXF tool.

View the revision history of schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **History...**

   The **Drawing History** dialog box opens and allows you to view the revision history of the selected drawing.

Drawing settings

Click **Draw** --> **Settings** to edit the following drawing settings specific to the current model.

After they have been edited, click Save, Load, Cancel, or OK (page 1264) according to your requirements.

Click **Home** --> **Settings** --> **Drawings** to edit the Global settings (page 1232).

Export Preferences

NOTE The **Export Preferences** subpage is only available when drawing settings are accessed from the **Home** ribbon.

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose export preferences you want to modify.
Minimum Text Block Spacing	Allows you to define a minimum value between independent drawing blocks. **NOTE** If the value is too great, text labels can end up far from the objects to which they refer.
Available Scales list	Displays the existing drawing scales. **NOTE** You can add user-defined scales in the list by typing the scale in the **Scale** field and clicking the **Add** button, or delete them by selecting the scale and clicking the **Remove** button.
Scale	Allows you to create a user-defined drawing scale by typing it in the field.
Name	Allows you to specify a name for a user-defined drawing scale.

General

NOTE The **General** subpage is only available when drawing settings are accessed from the **Draw** ribbon.

Setting	Description
Default export directory	Allows you to specify the default path to be used in the **DXF Export Preferences** dialog.

Layer Configurations

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose layer configurations you want to modify. **NOTE** Use the subpages of the **Layer Configurations** subpage to modify the required drawing type.
Available Configurations list	Displays the available layer configurations and allows you to select or modify them. You can use the following buttons to modify the available layer configurations: • **Add**: creates an empty layer configuration that you can modify in the **Active Configuration** section. • **Add copy...**: allows you to copy a drawing item to create a new layer configuration. • **Remove**: deletes the selected layer configuration.
Active Configuration	Displays the name and layers in the selected layer configuration.
Name	Displays and allows you to modify the name of the selected layer configuration.
Layers list	Displays and allows you to select or clear the layers that are included in the selected layer configuration. **TIP** If you access the drawing settings from the **Draw** tab and modify them, consider saving the changes to a settings set, so that you can use them in later projects as well.

Layer Styles

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose layer styles you want to modify. **NOTE** Use the subpages of the **Layer Styles** subpage to modify the required drawing type.
Available Styles list	Displays the available layer styles and allows you to select or modify them. You can use the following buttons to modify the available layer styles: • **Add**: creates an empty layer style that you can modify in the **Active Style** section. • **Add copy...**: allows you to copy a drawing item to create a new layer style. • **Remove**: deletes the selected layer style.
Active Style	Displays the name and layers in the selected layer style.
Name	Displays and allows you to modify the name of the selected layer style.
Apply to All...	This button opens a dialog for rapidly applying a single color, report color, line type, font, or font size to all layers in the active layer style.
Layers list	Displays and allows you modify the appearance of layers that are included in the active layer style.
Description	The layer description is fixed and can't be changed.
Name	The layer name as it will appear in the drawing can be edited if required.
Is Merged, **Merged with**	By checking the box you are able to merge two or more layers together. All objects in the merged layer are drawn in the **Merged with** layer, adopting its line type, font and font size.

Setting	Description
Color	The colors specified here are used for the different layers in the dxf drawing output
Report Color	**Member Reports** have the facility to include a member drawing along with the calculations and diagrams etc. When a member report is printed the drawings adopt the report colors that have been specified here.
Line Type	Different line types can be specified for the different drawing layers as required.
Font	Different fonts can be specified for the different drawing layers as required.
Font Size	Different font sizes can be specified for the different drawing layers as required.

Options > Planar Drawings

General (all drawing variants)

Setting	Description
Hatching	Allows you to select how columns and walls are hatched in planar drawings. You can select or clear the following options: • **Show columns and walls above the level as hatched**: The columns and walls that continue above the current level are hatched. • **Show transfer columns and walls as cross hatched**: Transfer columns and walls are cross hatched.
Force and Moment Values	**NOTE** Applies to the following drawing variants: • Foundation Reactions • Beam End Forces

Setting	Description
	• Column Splice Loads Allows you to select which force and moment values are displayed in planar drawings. The options are: • **Strength factors**: Displays the strength factors in planar drawings. • **Service factors**: Displays the service factors in planar drawings. **Additionally factor by** allows you to specify a value by which the strength or service factors are factored in the selected drawing category.
Moment Values	**NOTE** Applies to the following drawing variants: • Foundation Reactions • Beam End Forces • Column Splice Loads Allows you to select which moment values are displayed in planar drawings. The options are: • **None**: Does not display any moment values. • **Only greater than**: Moment values are only displayed if their value is greater than the value specified here. • **All**: Displays all moment values, even if zero.
Shear Force Values	**NOTE** Foundation Reactions drawing variant only. Allows you to select which shear values are displayed in Foundation Reactions drawings. The options are: • **None**: Does not display any shear values.

Setting	Description
	• **Only greater than**: Shear values are only displayed if their value is greater than the value specified here. • **All**: Displays all shear values, even if zero.
Axial Force Values	**NOTE** Beam End Forces drawing variant only. Allows you to select which axial force values are displayed in planar drawings. The options are: • **None**: Does not display any axial force values. • **Only greater than**: Axial force values are only displayed if their value is greater than the value specified here. • **All**: Displays all axial force values, even if zero.
Display End Forces for	**NOTE** Beam End Forces drawing variant only. Allows you to select the members for which end forces are displayed.
Enveloped Reaction Values	**NOTE** Beam End Forces drawing variant only. **Include coincident forces**: when this option is selected, further options are presented allowing you to choose the max/min values and coincident forces to display when the beam end forces drawing is created for a loading envelope.

Beams (all drawing variants)

Setting	Description
Grouped Beam Labelling	Allow you to modify the labeling of beams when the beams have been

Setting	Description
	designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the beam label, or clear the option to use the design group name in the beam label. • **Include the beam name**: Select the option to include the beam name in the beam label for grouped beams.
Concrete Beam Labelling Position	Allows you to set the position of the beam label in relation to the beam in planar drawings. The options are: • **Above** • **Inside** • **Below**
Beam Mark Position	Allows you to adjust the appearance and position of the beam mark in relation to the brace in planar drawings. To display beam marks, select **Show beam mark**. The position options are: • **Above** • **Below**
Beam Attributes Position	Allows you to adjust the appearance and position of beam attributes in relation to the beam in planar drawings. To display beam attributes, select **Show beam attributes**. The position options are: • **Above** • **Below**
Beam Attributes	Allows you to select which beam attributes are displayed in planar drawings and how they are displayed. You can select or clear the following options: • **Show beam size in parentheses**: Places brackets around the beam size in the beam label.

Setting	Description
	• **Grade**: Displays the beam grade in planar drawings. • **Camber**: Displays the camber in steel beams. To modify the camber prefix, type the desired value in the **Camber prefix** field. • **Composite properties**: Displays composite beam properties. To change the separators inside which the number of studs is displayed, select the desired the **Stud separator** list. • **Transverse reinforcement**: Displays transverse reinforcement in planar drawings.

Braces (all drawing variants)

Setting	Description
Brace Mark Position	Allows you to adjust the appearance and position of the brace mark in relation to the brace in planar drawings. To display brace marks, select **Show brace mark**. The position options are: • **Above** • **Below**
Brace Attributes Position	Allows you to adjust the appearance and position of brace attributes in relation to the brace in planar drawings. To display brace attributes, select **Show brace attributes**. The position options are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in planar drawings.

Columns (all drawing variants)

Setting	Description
Grouped Column Labelling	Allow you to modify the labeling of columns when the columns have been designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the column label, or clear the option to use the design group name in the label. • **Include the column name**: Select the option to include the column name in the column label for grouped columns.
Column Mark Position	Allows you to adjust the appearance and position of the column mark in relation to the brace in planar drawings. To display column marks, select **Show column mark**. The position options are: • **Above** • **Below**

Column Attributes Position	Allows you to adjust the appearance and position of the column attribute in relation to the brace in planar drawings. To display column attributes, select **Show column attributes**. The position options are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

Column Attributes	Allows you to select which column attributes are displayed in planar drawings and how they are displayed. You can select or clear the following options: • **Show column size in parentheses**: Places brackets

Setting	Description
	around the column size in the column label. • **Grade**: Displays the column grade in planar drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

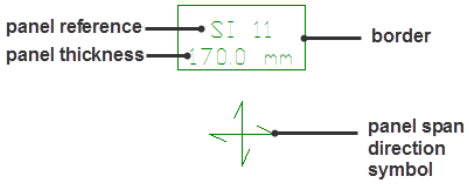
Walls (all drawing variants)

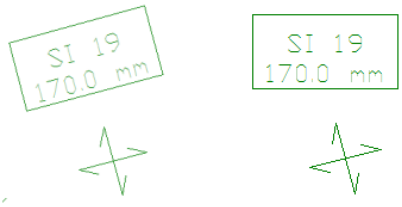
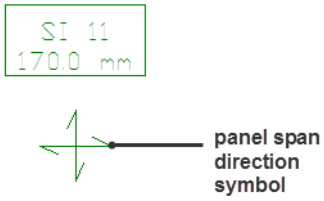
Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in planar drawings. The options are: • **Above** • **Inside** • **Below**
Show wall size in parentheses	Allows you to select whether brackets are placed around the wall size in the wall label.

Wall Reactions	Allows you to select whether or not to include distributed vertical reaction values of walls in the **Foundation Reactions** planar drawing. These are available in two different formats:
	• **Show distributed wall reactions table**: a table with the distributed wall reaction end values is included in the drawing,
	• **Show vertical reaction values**: values are added into the **Foundation Reaction Values** layer for each wall in the planar drawing,
	  • **Distributed (Maximum)**: Maximum absolute values of the vertical distributed reaction retaining sign,
	  • **Distributed (Average)**: Average value of the vertical distributed reaction along the length of the bottom analytical chord of the wall,

Setting	Description
	• **Distributed (Extremes)**: Extreme values at each end of the wall bottom analytical chord, • **Total**: Single integrated total vertical reaction of the wall.

Slabs/Mats (all drawing variants)

Setting	Description
Panel Labelling	Allows you to select which panel properties are included in panel labels and how panel labels are displayed in planar drawings.  You can select or clear the following options: • **Include panel reference**: Allows you to include the panel reference in the panel label. • **Include panel thickness**: Allows you to include the panel thickness in the panel label. • **Include surface offset (if non-zero)**: Allows you to include any surface offset that has been applied to the panel in the panel label. • **Include border around label**: Allows you to add a border around the label. • **Align label to panel reinforcement**: Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and clear the option to

Setting	Description
	achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 

Loads (Loading Plan drawing variant only)

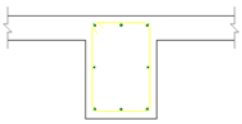
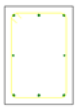
Setting	Description
Display size	Allows you to adjust the width and height of line, udl, and vdl loads and the size of point loads in planar drawings. The different options that you can adjust are: • **Width of line/UDL/VDL loads on plan**: Line, UDL, and VDL loads are drawn as hatched rectangles of fixed width when drawn in plan. The option allows you to adjust the width of the hatched rectangle. • **Height of line/UDL/VDL loads on plan**: Line, UDL, and VDL loads are drawn as hatched rectangles of fixed width when drawn in plan. The option allows you to adjust the height of the hatched rectangle. • **Point load marker size**: Allows you to adjust the size of point loads in planar drawings.

Setting	Description
Show dimensions for panel loads	Allows you to include the dimensions of panel loads in planar drawings.
Include dimensions to reference points	Allows you to include the dimensions from panel loads to their reference points in planar drawings.
Show dimensions for member loads	Allows you to include the dimensions of member loads in planar drawings.

Options > Member Details

Beam Detail > Content

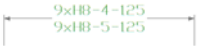
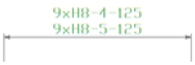
Setting	Description
Grouped Beams tab	
Show number of beams in group	If the beams in the model have been arranged into detailing groups, selecting the option causes the detailing number to be used as the member label instead of the beam reference. If detailing groups have not been used, the beam reference is always used as the member label.
Levels tab	
Show span levels	Allows you to display the span levels on the elevation.
Cross-sections tab	
Spans list	Allows you to select which cross-sections to display. You can decide to display no cross-sections, only the cross-sections for first spans of multi-span beams, or all spans.
Positions list	If cross-sections are displayed, the list allows you to select where they are positioned. You can decide to only position cross-sections for spans, for supports, or for supports and spans.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**

Setting	Description
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Show slab lines in section	Select the option to display slab lines in sections, as shown in the first image:  Clear the option to not display slab lines in sections, as shown in the second image: 
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Show steel bar layer information	Allows you to display steel bar layer information (B1, B2, T1, T2, and so on).
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL).
Anchorage lengths list	Allows you to select whether any required anchorage lengths are dimensioned, not dimensioned, or the dimension is replaced by a standard label (AL).
Axes list	Allows you to select whether axes are not displayed, are displayed above

Setting	Description
	the beam with dimensions, or are displayed below the beam with dimensions.
Additional bottom span bar positioning dimensions	Allows you to dimension any existing optional second span bars from the face of the support.
Support region length	If different stirrup regions have been used along a beam span, the option allows you to dimension the length of the support regions on the elevation.
First and last stirrups	Allows you to add dimensions from the face of the supports to the first and last stirrups.
Support columns and clear spans	Allows you to add dimensions showing the width of each supporting column and the clear beam span between supports.
Beam section	Allows you to dimension the beam depth and width on the cross-section.
Slabs in beam sections	Allows you to dimension the slab depth on the cross-section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in beam detail drawings.

Beam Detail > Style

Setting	Description
Beam Labels tab	
Print beam labels below the detail	Allows you to display the beam label centrally below each span. Clear the option to position the label immediately above each span.
Underline beam labels	Allows you to to underline the beam label on the elevation.
Label every span of multi-span beams	Allows you to display a label for each span in multi-span beams.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Restart section labels in each beam line	Allows you to restart the section labels for each line when multiple

Setting	Description
	beam lines are displayed on the same drawing sheet.
Add beam name label as prefix to section labels	Allows you to prefix each section label with the beam name.
Place cross-sections underneath the elevation	Allows you to to position cross-sections under the elevation.
Longitudinal Bars tab	
Draw bobs as shifted	Allows you to apply a slight offset to bobbed bars in drawings, so that the bobbed bars do not overlap each other in drawings an therefore, the drawings are easier to read.
Display only a single side bar in detail	Allows you to draw only a single side bar is drawn full length when multiple side bars are required in each face. Clear the option to draw all side bars full length.
Stirrups tab	
Draw stirrup labels in line	Allows you to display stirrup labels in line on the elevation, as shown in the first image:  Clear the option to display stirrup labels above the line, as shown in the second image: 
Print stirrup labels inside beam	Allows you to display the stirrup labels inside the beam. Clear the option to display the stirrup labels below the beam.
Stirrup label distance from bottom edge [n] % of the beam depth	Allows you to control the vertical position of stirrup labels that are positioned inside the beam.
Dimensions tab	
Lap and anchorage rounding increment	Allows you to control the rounding increment of lap and anchorage dimensions.

Column Detail > Content

Setting	Description
Grouped Columns tab	
Show number of columns in group	Allows you to display the number of columns in a column group.
Levels tab	
Show levels	Allows you to label the construction levels.
Cross-sections tab	
Show cross-sections	Allows you to display cross-sections through each stack.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL or CL).
Support region length	Allows you to dimension the support regions on the elevation.
Levels	Allows you to add dimensions between levels.

Setting	Description
Grid line offsets	Allows you to add dimensions from the grid to the column face on the elevation.
Connecting elements and clear heights	Allows you to add dimensions connecting elements and clear heights.
Column section	Allows you to add column dimensions to the column section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in column detail drawings.

Column Detail > Style

Setting	Description
Column Labels tab	
Underline column labels	Allows you to to underline the column label on the elevation.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Restart section labels in each column line	Allows you to restart the section labels for each line when multiple column lines are displayed on the same drawing sheet.
Add column name label as prefix to section labels	Allows you to prefix each section label with the column name.
Ties (Links) tab	
Draw tie (link) labels in line	Allows you to display tie labels in line on the elevation, as shown in the image on the left. Clear the option to display tie labels above the line, as shown in the image on the right. 
Dimensions tab	

Setting	Description
Lap dimension rounding increment	Allows you to control the rounding increment of lap dimensions.

Wall Detail > Content

Setting	Description
Levels tab	
Show levels	Allows you to label the construction levels.
Cross-sections tab	
Show cross-sections	Allows you to display cross-sections through each stack.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL or CL).
Support region length	Allows you to dimension the support regions on the elevation.
Levels	Allows you to add dimensions between levels.

Setting	Description
Grid line offsets	Allows you to add dimensions from the grid to the column face on the elevation.
Wall section	Allows you to add wall dimensions to the wall section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in wall detail drawings.

Wall Detail > Style

Setting	Description
Wall Labels tab	
Underline wall labels	Allows you to underline the wall label on the elevation.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Add wall name label as prefix to section labels	Allows you to prefix each section label with the wall name.
Don't label typical panel bars	Allows you to not label typical bars in cross-sections. Clear the option to label every bar in cross-sections.
Horizontal Bars and Ties (Links) tab	
Draw labels in line	Allows you to display tie labels in line on the elevation, as shown in the image on the left. Clear the option to display tie labels above the line, as shown in the image on the right. 
Label bars inside panel	Allows you to label bars inside the wall panel.
Dimensions tab	
Lap dimension rounding increment	Allows you to control the rounding increment of lap dimensions.

Options > Member Schedules

General

Setting	Description
NOTE The settings that you see on the **General** tab depend on which drawing category subpage you have selected in the left side pane.	
Item list	Allows you to select an item to view the text label that will applied to it in the concrete beam schedule drawing.
Text	Allows you to modify the text displayed in the beam schedule for the item selected in the **Item** list.
Size Column Format	Allows you to select whether to display the width of the beam or its height first in the size column.
Use single column for size	Allows you to display both the width and height in a single column.
Use single column for bottom bars	Allows you to display the bottom bars in a single column
Omit top middle bars column	Allows you to omit the top middle bars from the schedule drawing.
Use single column for stirrups	Allows you to display the stirrups in a single column.
Show design group name	Allows you to display only the design group name in the mark column.
Show grouped column number	Allows you to display column groups.
Include starter bars	Allows you to include starter bars in member schedule drawings.
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in member schedule drawings.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **Standard** • **IStructE**
Display bar marks	Allows you to display bar marks in cross-section labels.
Show outline of stack below	Allows you to display the outline of the column stack below.

Setting	Description
Dimension column section	Allows you to add column dimensions to the column section.
Dimension levels	Allows you to add dimensions between levels.
Show outline of panel below	Allows you to display the outline of the wall panel below.
Dimension wall section	Allows you to add wall dimensions to the wall section.

Bar Key

Setting	Description
Reference	Lists every reference that can potentially appear in the bar bending details table in concrete beam schedule drawings. **TIP** Hover the mouse pointer over a reference to see the bar and its associated note.
Use Custom Name	Allows you to use a custom name for the selected reference.
Custom Name	Allows you to specify the custom name that replaces the original reference in the bar bending details table if the

Options > Slabs and Mats > Layout

General

Setting	Description
Show columns and walls above the level as hatched	Allows you to hatch columns and walls that continue above the current level.
Show transfer columns and walls as cross hatched	Allows you to cross-hatch transfer columns and walls.

Beams

Setting	Description
Use detail group name	Allows you to use the detail group name in the beam label. Clear the option to use the design group name in the beam label instead.
Include the beam name	Allows you to include the beam name in the label for grouped beams.
Beam Labelling position	Allows you to set the position of the beam label in relation to the beam in slab and mat layout drawings. The options are: • **Above** • **Inside** • **Below**
Show beam mark	Allows you to display beam marks in slab and mat layout drawings. The position options for beam marks are: • **Above** • **Below**
Show beam attributes	Allows you to display beam attributes in slab and mat layout drawings. The position options for beam attributes are: • **Above** • **Below**
Show beam size in parentheses	Allows you to place brackets around the beam size in the beam label.
Grade	Allows you to display the beam grade in slab and mat layout drawings.

Braces

Setting	Description
Show brace mark	Allows you to display brace marks in slab and mat layout drawings. The position options for brace marks are: • **Above** • **Below**

Setting	Description
Show brace attributes	Allows you to display brace attributes in slab and mat layout drawings. The position options for brace attributes are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in slab and mat layout drawings.

Columns

Setting	Description
Use detail group name	Allows you to use the detail group name in the column label. Clear the option to use the design group name in the column label instead.
Include the column name	Allows you to include the column name in the label for grouped columns.
Show column mark	Allows you to display column marks in slab and mat layout drawings. The position options for column marks are: • **Above** • **Below**

Show column attributes	Allows you to display column attributes in slab and mat layout drawings. The position options for column attributes are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

Show column size in parentheses	Allows you to place brackets around the column size in the column label.

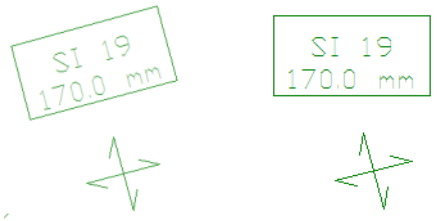
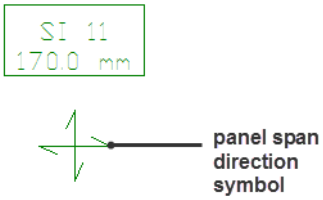
Setting	Description
Grade	Allows you to display the column grade in foundation layout drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

Walls

Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in slab and mat layout drawings. The options are: • **Above** • **Inside** • **Below**
Wall Attributes	Allows you to select whether brackets are placed around the wall size in the wall label.

Slabs/Mats

Setting	Description
Include panel reference	Allows you to include the panel reference in the panel label.
Include panel thickness	Allows you to include the panel thickness in the panel label.
Include surface offset (if non-zero)	Allows you to include any surface offset that has been applied to the panel in the panel label.
Include border around label	Allows you to add a border around the label.
Align label to panel reinforcement	Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and

Setting	Description
	clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 
Extend loose bar panel reinforcement lines across full panel	Allows you to draw bar panel reinforcement across the entire panel. Clear the option to display the loose bar reinforcement as shown in the following image:
Always show main bar layer for rectangular mesh	If the mesh is not a square mesh, the main bars are normally put on the outer layer of the drawing, and no text is required in the drawing. However, if the bars are not on the outer layer of the drawing, they are indicated by adding B2 (if bottom mesh) or T2 (if top mesh) aligned to the main bar direction. **NOTE** If a square mesh (has the same size and spacing of bars in both directions) is applied, a square mesh symbol is used, and bars are shown

Setting	Description
	equally spaced in both directions. If the mesh is not a square mesh (does not have the same size and spacing of bars in both directions), a rectangular mesh symbol is used. Bars are shown in both directions, but with closer spacing for the more closely spaced bars in the mesh.
Anchorage rounding increment	Allows you to specify the rounding value applied to the anchorage length.

Patches

Setting	Description
Show patches with no reinforcement	Allows you to display patches that have no reinforcement specified in drawings.
Draw full anchorage lengths	Allows you to display bars with full anchorage lengths.
Draw curtailed (indicative) anchorage lengths = max()	Allows you to draw curtailed (indicative) anchorage lengths instead of full anchorage lengths and specify their size.

Punching Shear

Setting	Description
Show punching reinforcement details	Allows you to include a detail to the side of the layout showing the punching shear reinforcement provided.
Don't show area of steel requirement where rails have been designed	Allows you to only display the area of steel requirement for punching shear check items where rails are not provided (such as walls or column drops). Clear the option to display the area of steel requirement for all punching check items.

Setting	Description
Hide punching reinforcement on the main layout	Allows you to hide the punching reinforcement on the main layout. The reinforcement is still displayed in the punching check detail drawing, provided that **Show punching reinforcement details** is selected.

Options > Slabs and Mats > Punching Check Detail

Content

Setting	Description
Dimensions tab	
Show column to first stud spacing	Allows you to add a dimension from the column face to the first stud.
Show stud spacing	Allows you to add dimensions displaying the stud spacings along the rail.
Show rail spacing	Allows you to add dimensions displaying the rail spacings.
Include schematic showing stud dimensions	Allows you to add a schematic displaying the stud width and height above the rail.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in punching check detail drawings.
Style subpage	
Show column as hatched	Allows you to hatch the column.
Underline punching check label	Allows you to underline the punching check label in punching check detail drawings.

Style

Setting	Description
Show column as hatched	Allows you to hatch the column.
Underline punching check label	Allows you to underline the punching check label in punching check detail drawings.

Options > Foundations > Layout

General

Setting	Description
Show columns and walls above the level as hatched	Allows you to hatch the columns and walls that continue above the current level.
Show transfer columns and walls as cross hatched	Allows you to cross-hatch transfer columns and walls.
Show pile location and loading table	Click **Columns** to choose which columns of tabular data are to be included in the table on the drawing. **NOTE** Only the single most critical load condition is reported in the table for each pile. - could be tension or compression - could be a gravity, wind, or seismic combination

Beams

Setting	Description
Use detail group name	Allows you to use the detail group name in the beam label. Clear the option to use the design group name in the beam label instead.
Include the beam name	Allows you to include the beam name in the label for grouped beams.
Concrete Beam Labelling Position	Allow you to modify the labeling of beams when the beams have been designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the label, or clear the option to use the design group name in the label. • **Include the beam name**: Select the option to include the beam

Setting	Description
	name in the label for grouped beams.
Show beam mark	Allows you to display beam marks in foundation layout drawings. The position options for beam marks are: • **Above** • **Below**
Show beam attributes	Allows you to display beam attributes in foundation layout drawings. The position options for beam attributes are: • **Above** • **Below**
Beam Attributes	Allows you to select which beam attributes are displayed in foundation layout drawings and how they are displayed. You can select or clear the following options: • **Show beam size in parentheses**: Places brackets around the beam size in the beam label. • **Grade**: Displays the beam grade in planar drawings. • **Camber**: Displays the camber in steel beams. To modify the camber prefix, type the desired value in the **Camber prefix** field. • **Composite properties**: Displays composite beam properties. To change the separators inside which the number of studs is displayed, select the desired the **Stud separator** list. • **Transverse reinforcement**: Displays transverse reinforcement in foundation layout drawings.

Braces

Setting	Description
Show brace mark	Allows you to display brace marks in foundation layout drawings. The position options for brace marks are: • **Above** • **Below**
Show brace attributes	Allows you to display brace attributes in foundation layout drawings. The position options for brace attributes are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in foundation layout drawings.

Columns

Setting	Description
Use detail group name	Allows you to use the detail group name in the column label. Clear the option to use the design group name in the column label instead.
Include the column name	Allows you to include the column name in the label for grouped columns.
Show column mark	Allows you to display column marks in foundation layout drawings. The position options for column marks are: • **Above** • **Below**

Show column attributes	Allows you to display column attributes in foundation layout drawings. The position options for column attributes are:
	• For position in elevation:
	  • **Above**
	  • **Below**

Setting	Description

	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

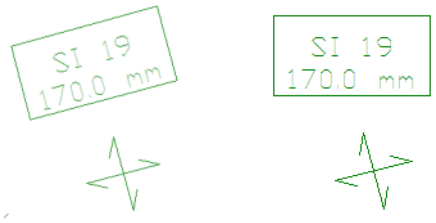
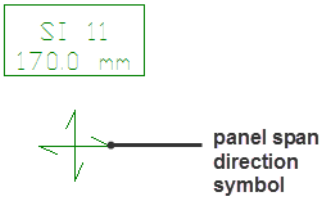
Show column size in parentheses	Allows you to place brackets around the column size in the column label.
Grade	Allows you to display the column grade in foundation layout drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

Walls

Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in foundation layout drawings. The options are: • **Above** • **Inside** • **Below**
Wall Attributes	Allows you to select whether brackets are placed around the wall size in the wall label.

Slabs/Mats

Setting	Description
Include panel reference	Allows you to include the panel reference in the panel label.
Include panel thickness	Allows you to include the panel thickness in the panel label.
Include surface offset (if non-zero)	Allows you to include any surface offset that has been applied to the panel in the panel label.
Include border around label	Allows you to add a border around the panel label.
Align label to panel reinforcement	Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and

Setting	Description
	clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 
Show pile location table	Allows you to include a table showing the pile locations in foundation layout drawings.
Show pile reference	Allows you to display the pile references in foundation layout drawings.

Isolated Foundations

Setting	Description
Show pile type name	Allows you to include the pile type names in the footing attributes displayed for each pile cap in foundation layout drawings.
Show foundation details	Allows you to include pad base or strip base and pile cap details, displaying the designed reinforcement, on the side of the foundation layout drawing.
Show reinforcement quantities table	Allows you to include the reinforcement quantity table for the pad bases and strip bases and pile caps displayed in foundation layout drawings.
Show pad/strip base schedule	Allows you to include the reinforcement schedule for the pad

Setting	Description
	bases or strip bases in foundation layout drawings.
Show pile cap schedule	Allows you to include the reinforcement schedule for the pile caps in foundation layout drawings.
Show allowable pile capacity	Allows you to include the allowable pile capacity table for the piles under pile caps in foundation layout drawings.
Show overall dimensions	Allows you to include the overall dimensions of the pad bases or strip bases and pile caps in foundation layout drawings.
Show pile spacings	Allows you to include the pile cap pile spacing dimensions in foundation layout drawings.
Include the foundation name	Allows you to include the pad base or strip base or pile cap name in the footing mark for grouped footings in foundation layout drawings.
Labelling Position	Allows you to control where the footing mark and attributes are displayed for isolated foundations. The position options are: • **Above** • **Below**

Options > Isolated Foundations > Detail

Content

Setting	Description
Grouped Foundations tab	
Show number of foundations in group	For isolated foundations designed using groups, selecting this option displays the number in the group in brackets after the group label.
Cross-sections tab	
Positions list	Allows you to select how cross-section positions are displayed. You can select to not display cross-section positions at all, display a length-wise

Setting	Description
	section or display cross-sections in both directions.
Pile Labels tab	
Show pile type name	Allows you to include the pile type name in the pile label.
Dimensions tab	
Show overall dimensions	Allows you to include the overall dimensions of the base/pile cap.
Show supported member dimensions	Allows you to include dimensions of the column or wall being supported.
Show pile spacings	Allows you to include dimensions from center to center of each pile for pile caps.
Quantities tab	
Show reinforcement quantities table	When a drawing is created for an *individual* isolated foundation, this option allows you to include the reinforcement quantity table. (When a Foundation Layout drawing is created the reinforcement quantities table is always included, so this option has no effect).

Style

Setting	Description
General tab	
Show supported member as hatched	Allows you to hatch columns and walls that are supported on the base pile cap in isolated foundation detail drawings.
Underline foundation label	Allows you to underline the foundation label on the detail in isolated foundation detail drawings.
Cross-sections tab	
Section label style list	Allows you to select the label style to be applied to cross-sections in isolated foundation detail drawings.
Restart labels in each foundation	Allows you to restart labeling from the beginning of each section when multiple foundations are included in the same drawing.

See also

Adjust and apply drawing settings (page 1093)

8 BIM integration

Data can be imported from other applications to create the Tekla Structural Designer model, or to create 3D reference models for checking compatibility with other disciplines.

The Tekla Structural Designer model can also be exported to other Trimble and 3rd party applications as required for a variety of purposes.

You can also use the **Trimble Connect** project collaboration tool to share information with other project stakeholders.

Click the following links to find out more:

- Import model data (page 1156)
- Working collaboratively with Trimble Connect
- Export to Trimble applications (page 1176)
- Export to and import from other applications (page 1186)

BIM fundamentals (Playlist)

8.1 Import model data

Tekla Structural Designer allows you to import model data from other applications.

Click the following links to find out more about importing a model into Tekla Structural Designer:

- Import a project from a Structural BIM Import file (page 1156)
- Import a project from a TEL file
- Import data from a 3D DXF file
- Import 3D reference models

Import a project from a Structural BIM Import file

1. On the **BIM Integration** tab, click **Structural BIM Import**.

   The **BIM Integration** wizard opens.

2. Use the **[...]** button to browse and select the appropriate CXL file.

   The remainder of the fields are populated with the settings read directly from the file.

3. Click **Next** to continue.

   The **Import Options** page shows the settings related to the file, it will indicate if this is a first time import or an update, and allows the user to choose Metric or Imperial for the units, a design code and the default level type. The level type can later be adjusted in the construction levels dialog if required.

4. Click **Next** to continue.

   The **Relocate Import Model** page allows for models to be moved to around the datum position in Tekla Structural Designer from a real world co-ordinate. The extent of the model can be seen from the information displayed on the right hand side of the dialog.

5. Click **Next** to continue.

   The **Integration Filter** will be displayed. The five options shown here allow you to verify if grids, levels etc. are to be imported. Please note that the settings held in the Delete Existing... and four Update... columns should not be used when performing a First Time Import and are only used when synchronizing changes into an existing Tekla Structural Designer model.

6. Click **Next** to continue.

   Mapping of materials to recognized grades will be shown now. Here you can view the different materials recognized in the incoming CXL file and the options to alter materials with a more preferred grade.

7. Click **Next** to continue (if available).

   Following on from Material mapping, the dialog for Decking mapping will be shown (if applicable). Again this allows users to view the decking types being imported and have the option of overwriting the details if required.

8. Click **Finish**

   After the file has completed processing, the model will be displayed within the Structural 3D view and you can then proceed with validation of the structure before applying any analytical information.

BIM fundamentals (Playlist)

Integration with Tekla Structures (Playlist)

Integration with Autodesk Revit (Playlist)

Revit-TSD Integration (eLearning)

Guidance notes for Integration between Tekla Structural Designer and Tekla Structures

Import a project from a TEL file

Before importing a project from a TEL file, you are advised to be aware of the restrictions. Then, you can proceed to follow the detailed instructions to perform the import.

Restrictions

When importing TEL files, note the following points:

Round tripping	Does not exist in TEL file imports. This means that the import data is used to create new objects in the model, not to update existing ones. All existing objects and data are maintained.

Data that is imported	• Project summary (new models only)
	  • Project name, engineer, etc.
	• Support conditions
	  • Any associated UCS
	  • Spring supports, including linear and non-linear
	NOTE In S-Frame, for non-linear spring the default is F_max = 0. This does not mean that the spring has zero capacity, but F_max is simply ignored.
	This is not the case in Tekla Structural Designer. In Tekla Structural Designer, F_max =0 means the spring has zero capacity. Hence for models with compression-only springs imported

		from S-Frame, all nonlinear spring supports with F_max = 0 will need editing, or analysis will fail.
		• 1D elements
		  • Imported as analysis elements. However, contiguous elements are not merged into members (straight or curved).
		  • Replicated by additional relatively stiff 1D elements.
		  NOTE Automatic supports are not created, for example, under columns.
		• Panels - area load only
		  • Created as roof or wall panels without openings
		• Panels - shell: tri, quad, or mixed
		  • Created as meshed concrete walls (vertical planes only) or 2-way spanning slab items
		  • Material properties are mapped manually during the import.
		  • Thickness
		• Panels - rigid or independent diaphragm
		  • Created as 1-way spanning slab items.
		  • Material properties are mapped manually during the import.
		  • Thickness

	• Panels - holes
	  • Created as slab or wall openings.
	  NOTE The holes must be rectangular for walls and rectangular or circular for slabs. Otherwise, Tekla Structural Designer will generate a warning.
	• Loadcases (linear only)
	• Nodal loads
	• Settlement loads
	• 1D element loads, including uniform temperature loads
	• Area panel loads, not including uniform temperature loads
	• Combinations

Exclusions	• Units: the Tekla Structural Designer model units are not changed to match the TEL file units. However, values are converted to the Tekla Structural Designer model units. • There is no special handling for 2D files. 2D files are imported in the same plane as they are defined in S-Frame (the X-Y plane), and default constraints are not imported. • Default constraints are not imported, and no warning is generated. Default constraints are supports applied to all nodes without exception internally during analysis. Default constraints are not displayed in the S-Frame interface. For models marked as 2D, default constraints restrict displacement to the X-Y plane and are as follows: Fz, My and Mz fixed. Default constraints may be manually applied in 3D

		models, and the S-Frame model can be examined to confirm their nature. The import does not replicate default constraints. To ensure equivalence, default constraints must be applied manually to all nodes, either in the S-Frame model prior to import, or in Tekla Structural Designer subsequently. • The import does not create any physical members. S-Frame physical members are treated like any other 1D element and imported as a single analysis element. In particular, Tekla Structural Designer does not merge contiguous elements into members (straight or curved) or identify columns, beams, and so on. No warnings are generated. In addition, for S-Frame physical models please note the following: • Intermediate nodes that do not form the ends of other elements are not imported. If such nodes have supports applied, the model will not be equivalent and should be adjusted to ensure equivalence. • If physical members have tapered sections, the sections should be sub-divided in S-Frame before importing to Tekla Structural Designer to produce an equivalent model. • Alternatively, the S-Frame model can be converted to an analytical model in S-Frame prior to import, using the S-Frame command for this. • Staged construction data is not imported, and no warning is generated. Typically, the entire model is imported, representing

	the last stage in which the model is complete. Otherwise, turn the **Staged Construction** setting in S-Frame off prior to import. This will remove all stages but the last one, and issue a warning to this effect. The model is then non-staged, and so should be valid for import.
	• Although the following can be modeled as single objects in Tekla Structural Designer, no attempt is made to import them as single objects from collections of S-Frame objects:
	  • Mid-pier walls
	  • Trusses
	  • Portal frames
	• Inactive elements are imported as inactive analysis elements of the beam type.
	NOTE Inactive elements are quite likely to originate from tension-only cross bracing in Fastrak Building Designer models. In this situation, we recommend that you click the warning to identify the relevant part of the model, delete both "braces", and create new braces using the specific X-brace pair.
	• Wall and strip integration lines are not imported. A warning is issued to this effect.
	• Tapered sections: a 1D element is imported. However, no tapered section dimension data is imported. A warning is issued to this effect.
	• Prestress data for 1D & 2D elements is not imported.

	• Percentage fixity data for 1D elements is not imported. • Non-linear spring data by graph: 1D elements and supports are imported, but the spring stiffnesses are set to 0. A warning is issued to this effect. • Non-structural alignment and offsets (cardinal point data) are ignored, and warnings are issued if they are non-zero. • Panels: general diaphragms, mat foundations, membranes and plates are excluded, and a warning is issued. • Diaphragm panel node exclusions are ignored. • 2D elements are excluded, and a warning is issued. • Meshing properties for panels are excluded without warning. • Shear Walls: only quadrilaterals can be created. • Diaphragm constraints are excluded, and a warning is issued to this effect. Any diaphragm constraints must be replicated in Tekla Structural Designer to ensure equivalence. • "Leader/follower" nodes are excluded, and a warning is issued. • Lumped mass is excluded. • Groups are excluded. • Notional load factors: NHF and EHF are added to combinations, with the sign indicating positive or negative for each direction. However, the actual value is ignored. The standard notional load calculation method and default strength factor are used.

	• Non-zero gravitational factors for Global X & Y are excluded, and a warning is issued. • Thermal gradient loads for 1D and 2D elements are excluded. • Moving loads are excluded. • Time history loads are excluded. • 2D elements are excluded, and a warning is issued. • RSA data is excluded.

Instructions

1. On the **BIM Integration** tab, click **'TEL' File Import**.

   The **'TEL' File Import** button is not active unless you have a Tekla Structural Designer document open.

   The **BIM Integration** wizard opens.
2. Click **...** to browse the .TEL file, and click **Next**.
3. If necessary, adjust the location and rotation of the import model, and click **Next**.
4. Select an appropriate material type and grade for each material using the lists.
5. Click **Finish**.

The model is now imported. Any associated warning messaged can be reviewed in the **BIM** sub-branch of the **Status tree**.

Import data from a 3D DXF file

See the following paragraphs for the restrictions of 3D DXF import, and the instructions on importing data from a 3D DXF file.

Restrictions

Note the following points when importing 3D DXF files:

- Round tripping does not exist in 3D DXF file imports. This means that the import data is used to create new objects in the model, not to update existing ones. All existing objects and data are maintained.
- Analysis elements are created from line segments in the selected layers of the DXF file as follows:
  - All lines in these layers become 1D analysis elements.

- Arcs and circles in the selected layers are excluded without a warning.
- Blocks are not handled, and no warnings are issued.
- Polylines in the selected layers are excluded without a warning.
- 3D solids in the selected layers are excluded without any warning.
- 2D faces for 3D objects in the selected layers are excluded without a warning.
- All ends of lines in the selected layers become nodes.
- Nodes are not introduced in the intersections of crossing lines.
- The following elements are not included in the import:
  - Reading of any text
  - Intelligence on "through members"
  - Gridlines and construction lines
  - 2D elements
  - Supports
  - Section properties
  - Materials
  - Loads
  - Combinations

Instructions

1. On the **BIM Integration** tab, click **3D DXF Import**.

   The **3D DXF Import** can only be selected in a 3D view.

   The **Open** dialog opens.
2. Browse to the required .dxf file.
3. Click **Open**.

   The **DXF Import Wizard** opens.
4. Select the layers and colors that you want to import.
5. If necessary, apply offsets and rotate the model before data is imported.
6. Click **OK**.

Any line segments found in the selected layers are now imported as analysis elements.

Import 3D reference models

You can accurately model and check compatibility with other disciplines during design decision making and execution through the use of TrimBIM or IFC based reference models, all from within Tekla Structural Designer, saving time and improving design accuracy.

Reference models are managed through the dedicated Reference Models side pane (page 81).

The same side pane is also used to import TrimBIM files to be used as reference models.

The Trimble Connect side pane (page 81) can be used to import both TrimBIM (.trb) files and IFC files to be used as reference models.

Import a TrimBIM file and use it as a reference model

1. Click the **Reference Models** side pane to open the **Reference Models** window.
2. In the **Reference Models** window, click **Import**. The **Import Reference Model** dialog box is displayed.
3. Select the **Source** as either:
   a. **Local file**, (if the file is accessible from your local drive), or
   b. **Trimble Connect**, (if the file is accessible from a Trimble Connect project).
4. Click  and then select the .TRB file from the source location. The selected reference model file is shown in the **Import Reference Model** dialog box.
5. Click **OK**. The **Edit Reference Model** dialog box is displayed.
6. Apply any required offsets or rotations and unselect any layers that you don't want to import, then click **OK**. The reference model is displayed in the Structure 3D view and the reference model trb is listed as "Visible" in the **Trim BIM** side pane.
7. In the **Trim BIM** side pane you can then click  to hide and  to show the reference model as required.

Use a Trimble Connect TrimBIM file or IFC file as a reference model

1. Click the **Trimble Connect** side pane to open the **Trimble Connect** window.
2. In the **Trimble Connect** window, if necessary link to the project (page 1169) containing the reference model(s).

3. Locate the required .trb or ifc file and then right-click on it.
4. From the context menu, select **Show reference model**. The **Edit Reference Model** dialog box is displayed.
5. Apply any required offsets or rotations and unselect any layers that you don't want to import, then click **OK**. The reference model is displayed in the Structure 3D view and the reference model file is listed as "Visible" in the **Trim BIM** side pane.
6. In the **Trim BIM** side pane you can then click  to hide and  to show the reference model as required.

Edit the offsets used, or layers displayed in a reference model

1. Click the **Reference Models** side pane to open the **Reference Models** window.
2. Right click on the required .trb or .ifc file and then select **Edit** from the context menu. The **Edit Reference Model** dialog box is displayed.
3. Apply any required offsets or rotations and unselect any layers that you don't want to be displayed, then click **OK**. The reference model is updated in the Structure 3D view.

Remove a reference model

1. Click the **Reference Models** side pane to open the **Reference Models** window.
2. Right click on the required .trb or .ifc file and then select **Delete** from the context menu. The reference model is removed.

8.2 Working collaboratively with Trimble Connect

Trimble Connect is a project collaboration tool allowing project stakeholders access to reliable, up-to-date project information. It is available as a cloud-based platform (Trimble Connect Web) and a Windows application (Trimble Connect for Windows). Projects are synchronized between the Windows app and the cloud.

NOTE To learn more about Trimble Connect, see:

- https://connect.trimble.com/
- https://trimbleconnect.support.tekla.com/

NOTE You need to have a Trimble Identity before you can start using Trimble Connect

Trimble Connect Project Explorer is used within Tekla Structural Designer to control the flow of information between the open model and a Trimble Connect project.

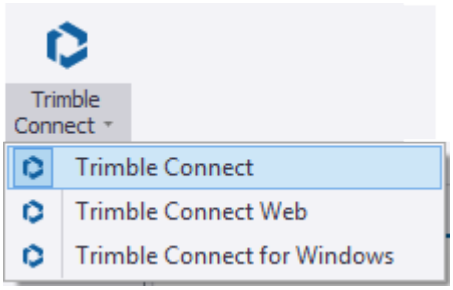
With Trimble Connect Project Explorer you can:

- link a Tekla Structural Designer model to a Trimble Connect project
- create and rename folders in a Trimble Connect project
- view a file list and rename files in a Trimble Connect project
- upload an IFC of the model to a Trimble Connect project
- upload drawings
- upload reports

NOTE A Tekla Structural Designer model can only be linked to a Trimble Connect project from the Trimble Connect Project Explorer and not from the web or Windows apps. The model must be linked before any information can move between Tekla Structural Designer and Trimble Connect.

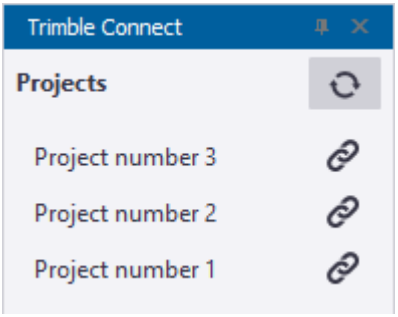
Launch Trimble Connect Project Explorer

1. Log in using your Trimble Identity, (if not already signed in).
2. On the **BIM Integration** tab, click the **Trimble Connect** droplist.
3. Select **Trimble Connect**



Trimble Connect Project Explorer opens, either as a docked window, or as a tab on the right of the interface.

Available projects are listed with link icons as below.



TIP If the project in which you want to collaborate is not shown, try clicking the refresh button to synchronise with the cloud. If it still not shown you would need somebody with the appropriate permissions to create it, or grant you access, using Trimble Connect Web, or Trimble Connect for Windows.

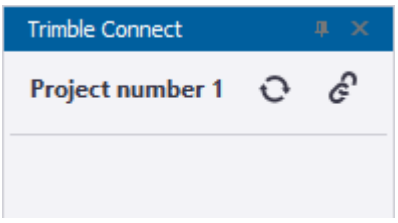
TIP If a 'Cannot find linked project' message is displayed, this indicates either that you do not have permission to view the project; or, that the project to which the model was previously linked has been deleted - in this situation you would need to click 'Unlink' before you are able to link the model with another project.

Link or unlink a project

1. To link to a project:

1. Click  next to the project you want to link to.

   When the model is linked, the project name appears at the top of the window and the icon changes as shown below.

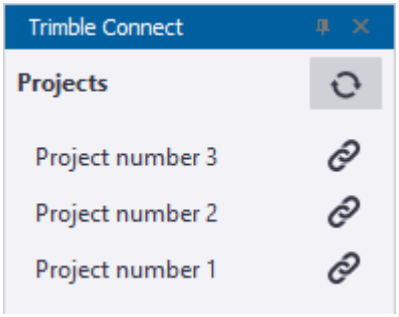


   Any folders or files in the Trimble Connect project that you have *read* access to are also displayed.

2. To unlink from a project:

2. Click  to unlink the project.

The full list of available projects is displayed once more.



Create folders, rename folders, rename files

When you link to a Trimble Connect project the existing project folder structure is displayed. You can add to this, if required.

NOTE You can only create but not delete folders in Trimble Connect Project Explorer. Folders can only be deleted in Trimble Connect Web, or Trimble Connect for Windows.

You can only rename folders and files to which you have *write* access.

---

1. To create a folder:
1. Right click in the Trimble Connect Project Explorer window.
2. Select **Create folder** and enter the folder name.
2. To create a subfolder:
3. Right click on an existing folder.
4. Click Create folder and enter the folder name.
3. To rename a file or folder:
5. Right click on the file/folder.
6. Select **Rename**

Upload an IFC file of a model

1. Right-click on the folder where you want to create the IFC.
2. Select **Upload New IFC file**
3. Follow the steps in the Export to IFC wizard to create the ifc file.
   a. Adjust the location and rotation of the model, and click **Next**
   b. Specify the export names of material grades, and click **Next**

c. Specify the ifc file name

The ifc file will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the IFC model in Trimble Connect.

NOTE If the model is subsequently updated, you can upload a new revision of the ifc file. To do this, right-click on the ifc file name and select **Upload new version of IFC**.

Upload a multi-member drawing

1. Open a view displaying the part of the model that you want to include in the drawing.
2. On the **Draw** ribbon, click the drawing type.

   The **DXF Export Preferences** dialog box opens.
3. Select **Upload to Trimble Connect**

   NOTE The above option is only available after the model has been linked to a Trimble Connect project.

   A default drawing name for the upload is displayed.

   If required, the drawing name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

   The **Create link between model member and uploaded drawing** option is not applicable to multi-member drawings, so this option cannot be selected.
4. Click **OK**

   The drawing will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the drawing in Trimble Connect.

NOTE The **Defaults** button on the **DXF Export Preferences** dialog box can be used to set a default file location, output target and link creation option. The options defined here will be true for the whole model.

Upload a single member drawing

1. Highlight the member for which you want to create a drawing.
2. Right click, then from the context menu select **Generate Drawing**

   The **DXF Export Preferences** dialog box opens.

3. Select **Upload to Trimble Connect**

   NOTE The above option is only available after the model has been linked to a Trimble Connect project.

   A default drawing name for the upload is displayed.

   If required, the drawing name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

4. To link the model member and drawing, click **Create link between model member and uploaded drawing**.

   NOTE The above option is only available after an IFC file of the model has been uploaded to the Trimble Connect project.

   See the *Link a drawing or report to an existing IFC* topic below for more details about this option.

5. Click **OK**

   The drawing will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the drawing in Trimble Connect.

NOTE The **Defaults** button on the **DXF Export Preferences** dialog box can be used to set a default file location, output target and link creation option. The options defined here apply for the whole model.

Upload a model report

1. Open the required report in Tekla Structural Designer.
2. On the **Report** ribbon, click **PDF Upload**

   NOTE Both **PDF Upload** and **Upload Settings** are only available after the model has been linked to a Trimble Connect project. (The latter can be used to set a default file location, output target and link creation option that applies for the whole model.)

   The **Upload Report** dialog box opens and a pdf name for the report is displayed.

   If required, the pdf name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

   The **Create link between model member and uploaded report** option is not applicable to model reports, so this option cannot be selected.

3. Click **OK**

   The pdf will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the report in Trimble Connect.

Upload a member report

1. Highlight the member for which you want to create a report.
2. Right click, then from the context menu select **Report for Member**

   The report is displayed in Tekla Structural Designer.
3. On the **Report** ribbon, click **PDF Upload**

   NOTE Both **PDF Upload** and **Upload Settings** are only available after the model has been linked to a Trimble Connect project. (The latter can be used to set a default file location, output target and link creation option that applies for the whole model.)

   The **Upload Report** dialog box opens and a pdf name for the report is displayed.

   If required, the pdf name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.
4. To link the model member and report, click **Create link between model member and uploaded report**.

   See the *Link a drawing or report to an existing IFC* topic below for more details about this option.
5. Click **OK**

   The pdf will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the report in Trimble Connect.

Link a drawing or report to an existing IFC

When drawings and reports are uploaded they are automatically linked to an existing IFC file as long as the appropriate option is selected (*Create link between model member and uploaded report / Create link between model member and uploaded drawing).*

It is important to **save** the Tekla Structural Designer model after exporting the IFC to Trimble Connect so that the link between the two is stored in the TSD model.

Linking reports

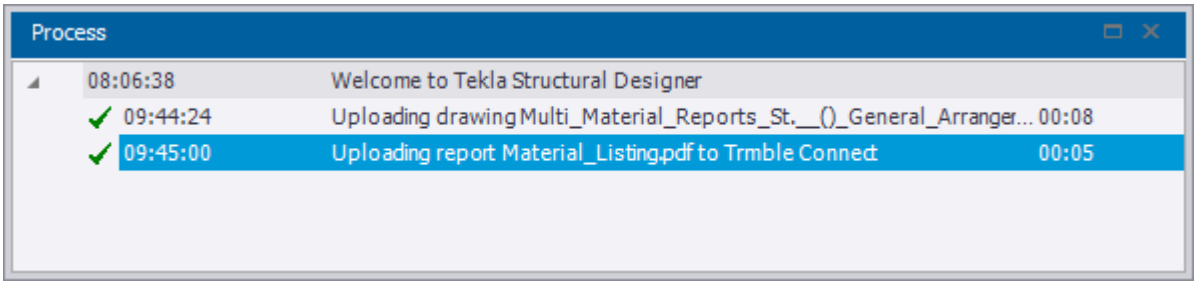
Linking of reports occurs for member reports only. If there are 3 IFC objects associated with one member (eg a 3 span beam) there will be a link created on each of the 3 IFC objects.

Linking drawings

Linking of drawings occurs for member drawings only. If there are 3 IFC objects associated with one TSD member (eg a 3 span beam) there will be a link created on each of the 3 IFC objects.

Check linking progress in the Process Window

The progress of the linking is logged in the process window. This will show the number of successful/unsuccessful links that have been created.



There are a few reasons why linking could be unsuccessful:

1. The member associated with the report/drawing does not exist in IFC model
2. Connection to the Trimble Connect project is lost
3. IFC model was not successfully uploaded

Import an uploaded IFC as a 3D reference model

Once an IFC file has been uploaded to Trimble Connect it can be imported to Tekla Structural Designer as a reference model.

1. In Tekla Structural Designer open a Structure 3D view.
2. In the Trimble Connect Project Explorer window, right-click on the IFC file to be imported as a reference model.
3. Select **Show TrimBIM model**.

   The reference model ifc is listed as "Loading" in the **Trim BIM** side pane and a **TrimBIM Model Import** dialog box is displayed. This dialog box allows you to reposition the reference model and selectively choose which layers to import.

4. In the **TrimBIM Model Import** dialog box apply any required offsets or rotations and unselect any layers that you don't want to import, then click **OK**. The reference model is displayed in the Structure 3D view and the reference model ifc is listed as "Visible" in the **Trim BIM** side pane.

5. In the **Trim BIM** side pane you can click  to hide and  to show the reference model as required.

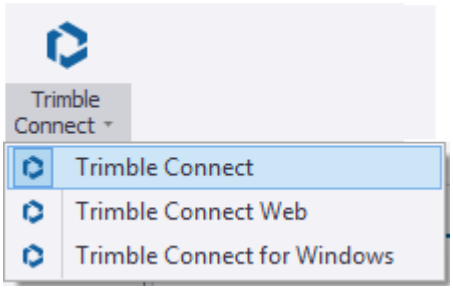
Open a model view of an uploaded IFC in Trimble Connect

1. In the Trimble Connect Project Explorer window, right-click on the IFC file to be viewed.
2. Select **Open in Trimble Connect Web** or **Open in Trimble Connect for Windows** as required.

   Trimble Connect opens the linked project and displays a view of the selected ifc.

Open Trimble Connect

1. To open Trimble Connect from the ribbon:
1. If not already signed in, click log in (at the top right of the interface).
2. On the **BIM Integration** tab, click the **Trimble Connect** droplist.
3. Select select **Trimble Connect Web**, or **Trimble Connect for Windows** as required.



   - **Trimble Connect Web:** This launches the in-browser web app. If the open model is associated with a Trimble Connect project, the project itself is opened, otherwise a default Trimble Connect page is opened.
   - **Trimble Connect for Window:** This launches the Windows application. Again, if the open model is associated with a Trimble Connect project then that project is opened.

2. To open Trimble Connect from **Trimble Connect Project Explorer**:

4. Right-click on a folder in the Trimble Connect Project Explorer window.
5. Select **Open in Trimble Connect Web**, or **Open in Trimble Connect for Windows** as required.

   The in-browser web or Windows app is launched and opened at the selected folder.

8.3 Export to Trimble applications

Click the following links to find out more about exporting to other Trimble applications:

- Export a model to Tekla Structures (page 1176)
- Export to Tekla Connection Designer (page 1177)
- Export to Tekla Portal Frame Designer (page 1178)
- Export to Tekla Tedds (page 1183)

Export a model to Tekla Structures

To export a model to Tekla Structures, see the following instructions.

1. Create your model in the usual way.
2. Ensure that **Autodesign** is unselected in the member properties for all member types. (This is necessary because the section size assigned to a member in "Autodesign" mode does not get included in the export - a default section would be assigned instead.)
3. Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 1333). This allows individual levels to be:
   - only included if the level is a floor
   - always included
   - never included
4. Ensure that in Model Settings > Structural BIM > Export (page 1259), the settings are as you require.
5. On the **BIM Integration** tab, click **Tekla Structures Export**.

   The **BIM Integration** wizard opens.
6. On the first page, click **Next**.
7. Adjust the location and rotation of the model, and click **Next**.
8. Select the items that are included in the model, and click **Next**.

9. Specify the export names of material grades, and click **Next**.
10. Specify the file name and location.
11. Select whether the file is exported for the first time, or whether you want to update an existing model.
12. Click **Finish**.
13. Start Tekla Structures and open the file to see your project.

NOTE Our integration links with Tekla Structures are always evolving and part of this is the use of the Tekla Structural Designer API to transfer model information.

For models containing designed rebar intent, it is not necessary to perform the process **Tekla Structures Export**. Instead the '.tsmd' format model file can be directly integrated into Tekla Structures.

The advantage of this process is that rebar objects will be created to suit the design intent using our Tekla Structures rebar set technology.

For more information on this topic, please refer to our document: Guidance notes for Integration between Tekla Structural Designer and Tekla Structures.

Integration with Tekla Structures (Playlist)

Export to Tekla Connection Designer

Before exporting connections to Tekla Connection Designer, you are advised to be aware of the limitations and also the recommended workflows for the different connection types.

NOTE Only valid base plate, column splice and moment connections listed in the Project Workspace Connections tree can be exported.

To export a single connection

NOTE Export requires a Tekla Connection Designer licence.

1. Right-click the connection in a Scene View.
2. Choose Export *(connection name)* to Tekla Connection Designer from the right-click menu.

   The Tekla Connection Designer application opens to allow the selected connection to be designed.

To export multiple connections

NOTE Export requires a Tekla Connection Designer licence.

1. Select the connections to be exported in a Scene View.
2. On the **BIM Integration** tab, click **TCD Export**.

   The Tekla Connection Designer application opens to allow the selected connections to be designed.

To return connection data from Tekla Connection Designer

Connection data from Tekla Connection Designer can be returned to Tekla Structural Designer as follows:

1. In Tekla Connection Designer click Connection> Return Connection to Tekla Structural Designer

   NOTE In the current release, Moment Connection data from Tekla Connection Designer is only partially returned. (e.g. Bolt layouts and endplate dimensions are not returned).

Export to Tekla Portal Frame Designer

Export to Tekla Portal Frame Designer workflow

It is important to follow the correct definition, loading and design sequence when frames are to be transferred for design to Tekla Portal Frame Designer.

1. Define frames/loading etc in Tekla Structural Designer.
2. Run Analyse All (Static).

   Analysis is necessary in order for the loading to be included in the export to Tekla Portal Frame Designer.

3. Export the frame to Tekla Portal Frame Designer to design it and check member stability.
4. Return revised sections/grades and haunch geometry back to the Tekla Structural Designer model.

NOTE Only section sizes, haunch dimensions and steel grades are returned to Tekla Structural Designer. Any other changes to the model data and loading are not returned.

WARNING Caution is required if the portal frame has been set out in Tekla Structural Designer using height to U/S Haunch, with haunches set to the

max. cut depth from the same size as the rafters. In this case, if revised rafter sections are returned to Tekla Structural Designer the eaves level of the frame would update accordingly. Any members or panels that were previously connected to the rafters would need to be edited to suit the updated eaves level.

How loading, restraints and supports are handled in the export

Loadcases and combinations

When you open the Loadcases dialog in Tekla Portal Frame Designer the first four loadcases are automatically created and named as follows:

- Self Weight
- Frame Dead Load
- Frame Service Load
- Frame Imposed Load

The loading in these loadcases will be taken from Tekla Structural Designer loadcases (providing the loadcase names have been specified exactly as per the below table):

Tekla Structural Designer Loadcase	Tekla Portal Frame Designer Loadcase
Self weight - excluding slabs	Self weight
Dead	Frame Dead Load
Services	Frame Service Load
Imposed	Frame Imposed Load

Every other Tekla Structural Designer loadcase that applies in-plane loading to the frame gets exported to Tekla Portal Frame Designer retaining it's original name. Care should be taken in Tekla Structural Designer to ensure that each loadcase name is unique.

If a particular Tekla Structural Designer loadcase does not apply any in-plane load to the frame being exported, the loadcase is not exported and it is also removed from the Tekla Portal Frame Designer load combinations.

If the Tekla Structural Designer model does not contain loadcases named "Dead", "Services" or "Imposed"; the "Frame Dead Load", "Frame Service Load" and "Frame Imposed Load" loadcases are created regardless, (defaulting to the loading values specified in the Tekla Portal Frame Designer Preferences). However, since these loadcases are not included in any load combinations they will have no affect on the design.

If the Tekla Structural Designer model contains loadcases named "Dead", "Services" or "Imposed" which do not apply any in-plane load to the frame being exported; then in the same way as any other loadcase that does not

apply any in-plane load, the loadcase is removed from the Tekla Portal Frame Designer load combinations. (In this situation the "Frame Dead Load", "Frame Service Load" or "Frame Imposed Load" loadcase is still created regardless (defaulting to the loading value specified in the Tekla Portal Frame Designer Preferences), but once again since it does not get included in any load combinations it will have no affect on the design.

Once loadcases that do not apply in-plane load have been removed from a load combination, if the resulting combination is empty, or if it only contains self weight, the combination itself is removed.

Any changes to loads, loadcases, or combinations made in Tekla Portal Frame Designer are not returned to Tekla Structural Designer.

Lateral Restraints

The Tekla Structural Designer lateral restraints are exported, strut restraints are ignored.

Any changes to restraints made in Tekla Portal Frame Designer are not returned to Tekla Structural Designer.

NOTE The below mappings assume default rotations of the rafters and columns. (Changing the rotation by 180 degrees flips the restrained flange in Tekla Portal Frame Designer.)

Rafter Restraints

These are exported to Tekla Portal Frame Designer restraints as follows

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Top Flange	Bottom Flange	
X	X	Torsional
X	-	Outer
-	X	Inner
-	-	No restraint

Column Restraints

These are exported to Tekla Portal Frame Designer restraints as follows

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Face A	Face C	
X	X	Torsional
X	-	Outer
-	X	Inner
-	-	No restraint

NOTE The rafter top flange and column face A can be identified graphically, for details see: 1D member local coordinate system (general case) (page 1310).

Supports

The Fy, Fz and Mx Tekla Structural Designer support properties are used to configure the Tekla Portal Frame Designer base fixities as shown in the following table.

Any changes to base fixities made in Tekla Portal Frame Designer are not returned to Tekla Structural Designer.

Tekla Structural Designer Support	Tekla Portal Frame Designer Base Fixity	
Fy Fixed	Horiz. Stiffness (ULS, SLS, Stability): Restrained	
Fz Fixed	Vertic. Stiffness (ULS, SLS, Stability): Restrained	
Mx Fixed	ULS:	Rot. 100%, Cap. 100%
	SLS:	Restrained
	Stability:	Restrained
Fy Free-Release	Horiz. Stiffness (ULS, SLS, Stability): 0kN/m	
Fz Free-Release	Vertic. Stiffness (ULS, SLS, Stability): 0kN/m	
Mx Free-Release	ULS:	Rot. Free, Cap. 1%
	SLS:	Free
	Stability:	Free
Fy Free-Spring Linear: XkN/m	Horiz. Stiffness (ULS, SLS, Stability): XkN/m	
Fz Free-Spring Linear: XkN/m	Vertic. Stiffness (ULS, SLS, Stability): XkN/m	
Mx Free-Spring Linear: XkNm/rad	ULS:	Rot. XkNm/rad, Cap. 1%
	SLS:	XkNm/rad
	Stability:	XkNm/rad
Fy Free-Spring Non Linear: -ve=XkN/m, +ve YkN/m	Horiz. Stiffness (ULS, SLS, Stability): Max (X, Y) kN/m	
Fz Free-Spring Non Linear: -ve=XkN/m, +ve YkN/m	Vertic. Stiffness (ULS, SLS, Stability): Max (X, Y) kN/m	

Tekla Structural Designer Support	Tekla Portal Frame Designer Base Fixity	
Mx Free-Spring Non Linear: -ve=XkNm/rad, +ve YkNm/rad	ULS:	Rot. Max (X, Y) kNm/rad, Cap. 1%
	SLS:	Max (X, Y) kNm/rad
	Stability:	Max (X, Y) kNm/rad
Mx Free-Nominally Pinned: 10%	ULS:	Rot. Free, Cap. 1%
	SLS:	20%
	Stability:	10%
Mx Free-Nominally Pinned: X%	ULS:	Rot. X%, Cap. 1%
	SLS:	X%
	Stability:	X%
Mx Free-Nominally Fixed: X%	ULS:	Rot. X%, Cap. 100%
	SLS:	X%
	Stability:	X%

To export a single frame

NOTE Export requires a Tekla Portal Frame Designer licence.

1. Right-click the frame in a Scene View.
2. Choose Export *(frame name)* to Tekla Portal Frame Designer from the right-click menu.

   The Tekla Portal Frame Designer application opens to allow the selected frame to be designed.

To export multiple frames

NOTE Export requires a Tekla Portal Frame Designer licence.

1. Select the frames to be exported in a Scene View.
2. On the **Home** tab, click **TPFD Export**.

   The Tekla Portal Frame Designer application opens to allow the selected frames to be designed.

To return revised sections from Tekla Portal Frame Designer

If the section sizes have been revised in Tekla Portal Frame Designer they can be returned to Tekla Structural Designer as follows:

1. In Tekla Portal Frame Designer click **Portal Frame**> **Return sections to model**

Export to Tekla Tedds

Once they have been designed using Tekla Tedds, timber members, precast members, and masonry walls can then be exported to Tedds in order to output the calculations.

NOTE Export requires a Tekla Tedds licence.

Understanding each of the export options

Command	Description
Export to Tekla Tedds > Model	Launches Tekla Tedds and: • Creates a new Tedds Project containing separate documents for each member in the model. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Member	This option only appears for a highlighted member if the option to design using groups is not active. Launches Tekla Tedds and: • Creates a new Tedds Project containing a single document for the member. • The data exported is taken from the last design run (either Member or Selection) associated with the member. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > <*Masonry wall name*>	Launches Tekla Tedds and:

Command	Description
	• Creates a new Tedds Project containing a single document for the wall. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Group	This option only appears for a highlighted member if the option to design using groups is active. Launches Tekla Tedds and: • Creates a new Tedds Project containing a single document for the group. • The data exported is taken from the last design run (either Member, Group, or Selection) associated with the group. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Selection	This option only appears if one or more members are selected. Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member or group in the selection. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > <*Substructure name*>	Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member in the sub structure. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.

To export all timber members, precast members and masonry walls

1. In the **Project Workspace**, click **Structure** tab.

2. In the tree, right-click **Masonry Walls**, or the required timber/precast member type.
3. In the context menu, select **Export using Tekla Tedds > Substructure**

   Tekla Tedds opens to allow the calculations for the all the exported walls/ members to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a single member

1. Right-click the member in a Scene View.
2. Choose **Export to Tekla Tedds > Member** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted member to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a single wall

1. Right-click the member in a Scene View.
2. Choose **Export to Tekla Tedds > <*masonry wall name*>** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted wall to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export multiple members or walls

1. Select the precast and/or timber members to be exported in a Scene View.
2. Right-click and choose **Export to Tekla Tedds > Selection** from the context menu.

   Tekla Tedds opens to allow the calculations for the selected members to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a group

1. Right-click any member of the group in a Scene View.
2. Choose **Export to Tekla Tedds > Group** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted group to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a substructure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to export.
3. In the context menu, select **Export using Tekla Tedds > *<Sub structure name>***

   Tekla Tedds opens to allow the calculations for the substructure to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

See also

Design using Tekla Tedds (page 851)

Check using Tekla Tedds (page 859)

8.4 Export to and import from other applications

You can export data from Tekla Structural Designer to a number of applications and file formats. You can import data from a number of these applications also.

Click the following links to find out more:

- Export to and import from Autodesk Revit (page 1187)
- Export a model to IFC (page 1190)
- Export to and import from Westok Cellbeam (page 1191)
- Export to and import from FBEAM (page 1192)
- Export a model to ADAPT (page 1198)

- Export a model to STAAD (page 1202)
- Export a model to Autodesk Robot Structural Analysis (page 1202)
- Export a model to the cloud (page 1203)
- Export to One Click LCA (page 1204)
- Export to IDEA StatiCa Connection Design (page 1207)

Export to and import from Autodesk Revit

NOTE To import/export models between Tekla Structural Designer and Autodesk® Revit® you will need to have installed the Tekla Structural Designer Integrator for the version of Revit that you want to use. This can be downloaded from the Tekla Structural Designer Downloads page by selecting your Tekla Structural Designer version and setting the File Type to "Integrator for Autodesk Revit".

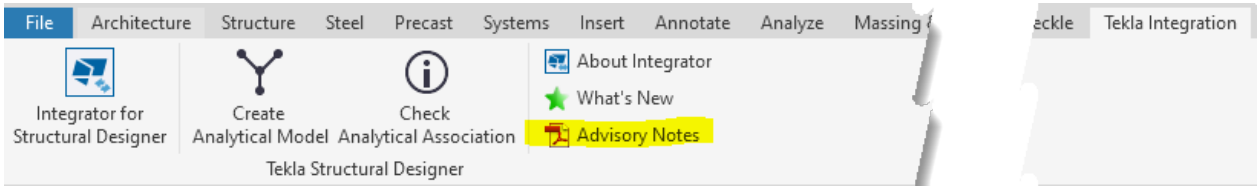
Guidance documentation

An extensive guidance document is available which provides information about workflows, model synchronization, section mapping and much more.

This document is automatically copied to the machine on which the integrator is installed, by default to: 'C:\ProgramData\Tekla\Structural\TSD Integrator \XXXX' folder. (where XXXX is the version installed).

It can also be accessed directly as follows:

1. In Revit, click the **Tekla Integration** tab
2. Click **Advisory Notes**



Export a model to Autodesk Revit

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 1333). This allows individual levels to be:

- only included if the level is a floor

- always included
- never included

---

1. In Tekla Structural Designer, on the **BIM Integration** tab, click **Autodesk Revit Export**.

   The **BIM Integration** wizard opens.
2. Adjust the location and rotation of the model, and click **Next**.
3. Select the items that are included in the model, and click **Next**.
4. Specify the export names of material grades, and click **Next**.
5. Specify the file name and location.
6. Select whether the file is exported for the first time, or whether you want to update an existing model.
7. Click **Finish**.
8. In Revit, on the **Tekla Integration** tab, click **Integrator for Structural Designer**.
9. At the splash screen which provides information about the Integrator, click **Next**.
10. On the next dialog choose the required import into Revit option (either **First time import**, or **Update existing model**, as required), then click **Next**.
11. Select the file (and location) that you wish to import, and click **Next**.
12. Select the structural objects to import, and click **Next**.
13. The **Section and Material Mapping** page indicates the different section families detected in the importing file. If there are any numerical entries in the **Not Recognized** column then there are some mapping discrepancies, these should be investigated before continuing further.
14. Even if the mapping is correct and no problems exist, the actual mapping can be checked by clicking to select the appropriate **Review Mapping** option.
15. Click **Finish**.

Import a model from Autodesk Revit

1. In Revit, on the **Tekla Integration** tab, click **Integrator for Structural Designer**.
2. At the splash screen which provides some information about the Integrator, click **Next**.

3. On the next dialog choose the required export from Revit option (either **First time export**, or **Update existing model**, as required), then click **Next**.
4. Select the structural objects to export, and click **Next**.
5. The **Section and Material Mapping** page indicates the different section families detected in the exporting file. If there are any numerical entries in the **Not Recognized** column then there are some mapping discrepancies, these should be investigated before continuing further.
6. Even if the mapping is correct and no problems exist, the actual mapping can be checked by clicking to select the appropriate **Review Mapping** option.
7. Click **Next**.
8. Specify the name (and location) for the file.
9. Click **Finish**.

   A CXL file is created at the specified location. Once the processing of the CXL file has been completed, a log file will be shown. The log file is simply a text file recording the objects exported, deleted or altered along with any warnings encountered.

   The CXL file can now be imported to Tekla Structural Designer as follows.

10. In Tekla Structural Designer, on the **BIM Integration** tab, click  **Structural BIM Import**.

    The **BIM Integration** wizard opens.

11. Use the **[...]** button to browse and select the appropriate CXL file.

    The remainder of the fields are populated with the settings read directly from the file.

12. Click **Next**.

    The **Import Options** page shows the settings related to the file, it will indicate if this is a first time import or an update, and allows the user to choose Metric or Imperial for the units, a design code and the default level type. The level type can later be adjusted in the construction levels dialog if required.

13. Click **Next** to continue.

    The **Relocate Import Model** page allows for models to be moved to around the datum position in Tekla Structural Designer from a real world co-ordinate. The extent of the model can be seen from the information displayed on the right hand side of the dialog.

14. Click **Next**.

    The **Integration Filter** will be displayed. The five options shown here allow you to verify if grids, levels etc. are to be imported. Please note that

the settings held in the Delete Existing... and four Update... columns should not be used when performing a First Time Import and are only used when synchronizing changes into an existing Tekla Structural Designer model.

15. Click **Next**.

    Mapping of materials to recognized grades will be shown now. Here you can view the different materials recognized in the incoming CXL file and the options to alter materials with a more preferred grade.

16. Click **Next** (if available).

    Following on from Material mapping, the dialog for Decking mapping will be shown (if applicable). Again this allows users to view the decking types being imported and have the option of overwriting the details if required.

17. Click **Finish**.

    After the file has completed processing, the model will be displayed within the Structural 3D view and you can then proceed with validation of the structure before applying any analytical information.

Integration with Autodesk Revit (Playlist)

Revit-TSD Integration (eLearning)

Export a model to IFC

To export your model to IFC, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **IFC Export**.

   The **BIM Integration** wizard opens at the **Relocate Export Model** page.

   The **Relocate Export Model** page allows for models to be moved from the datum position in Tekla Structural Designer to a real world co-ordinate.
3. Move and rotate the export model if required, and click **Next**.
4. On the **Integration Filter** page, select the items that are included in the export model, and click **Next**.
5. On the **IFC Export File** page, select whether the file is exported for the first time, or whether you want to update an existing model. Then specify the file format, name and location.
6. Click **Finish**.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 1333). This allows individual levels to be:

- only included if the level is a floor
- always included
- never included

Export to and import from Westok Cellbeam

To export and import beams to and from Westok Cellbeam, see the following instructions.

Limitations

This following limitations apply:

1. Westok Cellbeam import and export are only available for BS and Eurocode regional codes.
2. The user must carefully review that the slab arrangements, loading, restraints etc of the beam modeled in Tekla Structural Designer match the imported conditions of the beam in Westok Cellbeam. A review of the restraints and bending moment values in both software packages as a minimum is recommended.
3. Loading from columns supported by a beam e.g. in a 'transfer system' are exported as additional load sets for vertical columns only.
   - Note that for the situation of a column which is *supporting* a beam - i.e. the beam is continuous over the column - which will produce a negative reaction on the beam, these reactions are NOT exported. We would note that this is a much less common situation.

Export to Cellbeam

1. On the **BIM Integration** tab, click **Cellbeam Export**.

   The **Export Westok Beams** dialog box opens. Any Westok Cellular beams in the model are listed in the dialog.
2. Select the beams that you want to transfer to Westok Cellbeam, and click **Next**.
3. Select the export format and name the file.
4. Click **Next**.
5. Ensure that you have selected the design combinations whose results you want to use in the Westok beam design.
6. Click **Next**.
7. After reading the warning text, you must check the box to confirm that you have done so before you can proceed with the export.

8. Select the output folder where you want to place the Westok files, and click **Export**.
9. Click **Finish**.

Import from Cellbeam

NOTE In order to import Westok beams into your model, you must have previously created the beams in your Tekla Structural Designer model, and have exported them to Westok design. When you do this, each beam is given a unique identifier. This means that when you import the results from the Westok file, Tekla Structural Designer knows to which beam in your model the imported details apply.

1. On the **BIM Integration** tab, click **Cellbeam Import**.

   The **Import Westok Beams** dialog box opens.
2. Click **Add...**
3. Select the files that you want to import.
4. If necessary, change the file names.
5. Click **Open**.
6. Click **Import**.

   Provided that Tekla Structural Designer recognizes the identifier, the details in the files are imported to the associated beam in your model.

Export to and import from FBEAM

To export to and import from FBEAM, see the following instructions.

TIP Create substructures for your FABSEC® beams to make it easier to review and manage the FBEAM workflow.

Overview

FBEAM is software from Fabsec for the design of composite and non-composite plain and cellular beams.

The workflow for taking FABSEC® beams from a Tekla Structural Designer model and designing then within FBEAM is as follows:

- Select the FABSEC® beams in your Tekla Structural Designer model and click **FBEAM Export**.

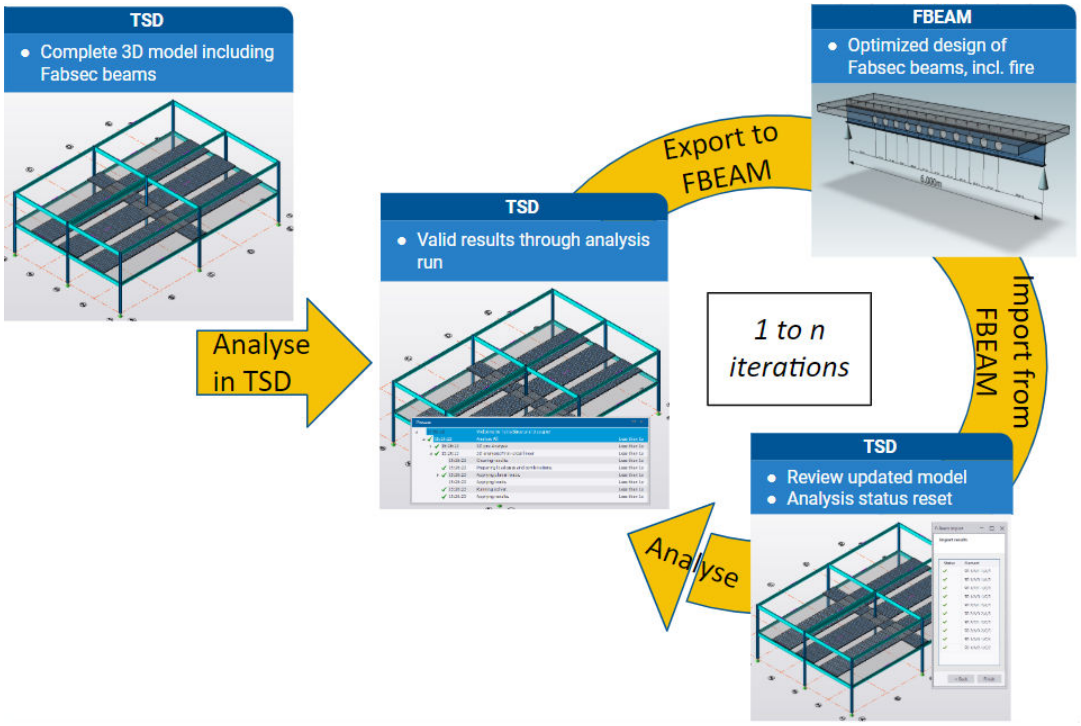
  This generates an XML file containing all the required properties required for each member.

- You can then launch FBEAM and open the project you wish to import the beams into.
- You then click the import button which starts FBEAM's import work flow. This will allow you to load the previously created XML file and select a subset or all of the beams in the file to import.

  Once the process is complete FBEAM will have created all of the members and their associated properties.
- You can then design the FABSEC® beams and specify fire protection coatings.
- Once complete, click the export button in FBEAM.

  This generates another XML file that can be imported back into Tekla Structural Designer.
- In Tekla Structural Designer, (with the model that was used to export the beams open), click the **FBEAM Import** to start the import workflow. This will then update each of the previously exported FABSEC® beams with the properties contained within the FBEAM generated XML file.

The above workflow can be summarized as follows:



Limitations

This following limitations apply to the import/export of FABSEC® beams:

1. Consider the case where multiple combinations are exported and a loadcase of a particular load type e.g Dead appears in one combination

but not all. In FBEAM the analysis will include all loadcases of the same type in each combination. This can overestimate the load significantly.

2. When Wind loadcases are exported to the 'Additional Load Set' for a particular beam, Tekla Structural Designer searches for the Wind loadcase that has the largest positive end force and exports that value to the 'Additional Load Set' and gives it a 'Type', 'Wind' in FBEAM. Similarly for the maximum negative value in Tekla Structural Designer to the 'Type', 'Wind Up' in FBEAM. This loadcase will be used in the design of that beam for all positive wind combinations and all negative wind combinations respectively. Whether a wind combination is considered negative is determined from the start and end values of the distributed (Wind) load applied to that beam (or point loads). If both are negative then the wind combination is negative (uplift).
3. Due to the mismatch between what can be modeled in Tekla Structural Designer and in FBEAM, fixed end moments (FEMs) are unlikely to be consistent when exported. For example, only one set of wind loads is available in FBEAM and so wind left to right and wind right to left cannot be exported simultaneously. Consequently, FEMs and end fixity are not exported.
4. The special loadcase in Tekla Structural Designer 'Self weight - excluding slabs' (i.e. beam self weight) is assumed always to be included in design combinations. Consequently, the setting in FBEAM, 'Include the beam self weight' is always switched on.
5. In any combination if any load type e.g. Dead or Imposed appears more than once with different load factors, only one value of load factor is exported. This situation is unusual but might occur for imposed loads for example when different psi factors might apply.
6. Any load types or direction not supported by FBEAM are not passed through. This includes minor axis loading, axial loading, trapezoidal loads and torsion.
7. FBEAM has a number of load types; “Dead”, “Super Dead”, “Live” etc. In composite design, the FBEAM “Dead” load is applied at the pre-composite stage and “Super Dead” and Live are applied post-composite. In Tekla Structural Designer however the “Dead” load type is applied post-composite, hence there is a discrepancy with FBEAM for this load type. To remedy this, Tekla Structural Designer’s “Dead” load is exported to FBEAM's “Super Dead” load type.
8. Imposed loads are typically entered into the 'protected' Construction Stage load combination in Tekla Structural Designer to represent live loads during construction e.g. due to 'heaping'. These are exported to FBEAM as Load Type 'CL'. They are given a load factor that will be the maximum from all imposed loadcases included in the Construction Stage combination although typically only one is included and with a load factor of 1.5.
9. Beam self weight loads are typically entered into the 'protected' Construction Stage load combination in Tekla Structural Designer.

Normally these would be represented by the Load Type ' D' in FBEAM. However, in order to avoid taking into account all of the dead loads, including those at composite stage, the Construction Stage load combination in FBEAM is populated with the beam self weight as a 'cladding' load, 'C'.

10. The Slab Wet loads exported from Tekla Structural Designer should set the 'Construction Stage Floor type' in FBEAM to 'CL' - typically this will ensure that a load factor of 1.5 is used in the design combination for Construction Stage. For EC this is correct but for BS it should be set to 'D'. This setting is not accessible to the export from Tekla Structural Designer and so must be changed manually if BS 5950 design is to be used.
11. For composite beams when the wet weight of the concrete is included in the Construction Stage combination in Tekla Structural Designer, the results in FBEAM will only be correct in the following circumstances:
    a. Automatic Loading is checked on in the loadcase dialogue when Slab Dry is selected as the load Type,
    b. Automatic Loading is checked on in the loadcase dialogue when Slab Wet is selected as the load Type,
    c. At least on combination includes the Slab Dry loadcase,
    d. The Construction Stage combination includes the Slab Wet loadcase.
12. Beams can be rotated in Tekla Structural Designer but in FBEAM are always assumed to have their web vertical i.e. unrotated. Consequently any beam rotation is not passed through from Tekla Structural Designer to FBEAM and any loads in the global axis system that are applied to a rotated beam are passed through to FBEAM as if they are applied through the vertical web of the beam.
13. When several beams on the same floor are exported from Tekla Structural Designer there can be several instance of 'Floor' in FBEAM. Each of these could be assigned a different size of 'Mesh'. However, only one of these will be imported into Tekla Structural Designer. It is recommended that either only one value of mesh is set for these beams or that any required changes to the slab mesh are made in Tekla Structural Designer and not FBEAM.
14. In composite design, transverse reinforcement whether in the slab or as part of the beam data is exported and imported as an area only. In Tekla Structural Designer the area is derived from the bar size and spacing (or mesh size). In the case of bars, the export transfers the area and in FBEAM the default bar size will be shown with the exported area. For import the area in FBEAM is processed by Tekla Structural Designer to find a bar size and spacing that has an area greater than (or equal to) the area held in FBEAM. Furthermore when 'loose bars' are used in the slab (unusual) these are added to any from the beam data and transferred as one value to FBEAM. Clearly on import these cannot be separated. This approach

has an impact on round tripping that the user must be aware of and make appropriate adjustments post import or export.

15. Partial length shear connector layout with empty segments cannot be created in Tekla Structural Designer. If cases like these are detected when importing from FBEAM the partial shear connector layout will be extended for the full length of the beam in Tekla Structural Designer.
16. Web opening stiffeners applied only to one side of a beam are always set to the right hand side when importing into FBEAM. When importing from FBEAM to Tekla Structural Designer, the existing stiffeners in Tekla Structural Designer keep their side regardless of how they were modeled in FBEAM.
17. Loading from columns supported by a beam e.g. in a 'transfer system' are exported as additional load sets for vertical columns only.
    - Note that for the situation of a column which is *supporting* a beam - i.e. the beam is continuous over the column - which will produce a negative reaction on the beam, these reactions are NOT exported. We would note that this is a much less common situation.
18. Camber values directly set in Tekla Structural Designer are exported to FBEAM* and camber set or adjusted in FBEAM is imported back to Tekla Structural Designer from FBEAM using the "Apply as a Value" setting in Tekla Structural Designer.
    - *Note camber applied in Tekla Structural Designer using the setting "Apply as a proportion of deflection" is NOT exported to FBEAM since the camber value is produced by the design process which is not run for Fabsec beams within Tekla Structural Designer.

Export to FBEAM

Before exporting your FABSEC® beams, you should ensure the analysis has been performed.

1. On the **BIM Integration** tab, click **FBEAM Export**.

   The **FBEAM Export** dialog box opens. Any FABSEC® beams in the model are listed in the dialog.
2. Select the beams that you want to transfer to FBEAM, and click **Next**.
3. After reading the warning text, you must check the box to confirm that you have done so before you can proceed.
4. Select the combinations you want the beams designed for, and click **Next**.
5. Select the location to save to, name the file, and click **Next**.

   The export status of the selected beams is displayed.
6. Click **Finish**.

You can now go in to FBEAM and import the file that you have just created into an existing or new project, so that the beams can be designed.

Import from FBEAM

NOTE In order to import FABSEC® beams, you must have previously created them in your Tekla Structural Designer model, and have exported them to FBEAM. When you do this, each beam is given a unique identifier. After the beams have been designed in FBEAM you must then choose the Tekla Structural Designer XML option to export them to a file. This enables Tekla Structural Designer to know to which beam in your model the imported details apply.

1. On the **BIM Integration** tab, click **FBEAM Import**.

   The **FBEAM Import** dialog box opens. Any FABSEC® beams in the model are listed in the dialog.
2. Select the file that you want to import.
3. Click **Next**

   A list of the FABSEC® beams in the file is displayed from which you can choose the beams to be imported.
4. Select the beams that you want to import.

   You can only select beams to import that correspond to existing beams in the Tekla Structural Designer model.
5. Click **Next**

   The import status of the selected beams is displayed. If any errors have occurred, hovering the cursor over the error displays the reason in a tooltip.
6. Click **Finish**.

   Provided that Tekla Structural Designer recognizes the beams, the details in the file are imported to the associated beam properties, (including in particular the beam UDA properties) in your model.

Review the imported beams

When FABSEC® beams are imported, User Defined Attributes are added to the beam properties to record import status and fire design details. These can be reviewed graphically.

To review the FABSEC® beam UDAs graphically:

1. If necessary, switch the view regime to a **Review View**.

2. On the **Review** tab, click **UDA**.
3. In the **Properties** window, with **[M]ode** set as **Review**, change the **Attribute** to **Import Status**.

   Each FABSEC® beam is color-coded to represent the import status that applies.

Export a model to ADAPT

Before exporting to ADAPT, read the following limitations. Then, you can proceed to follow the detailed instructions to perform the export.

Limitations

The scope of the ADAPT Export is constrained by the fact that some geometries, configurations and properties possible in Tekla Structural Designer are beyond the scope of ADAPT. This imposes some unavoidable limitations on the Export at this time, for example:

- Only concrete and steel materials are exported.
- Only horizontal beams and slabs are exported.
- Roof and wall panels are not supported.
- For concrete walls, the following are not supported (all of which may be in a Tekla Structural Designer); wall openings, non-rectangular walls, wall extension, walls with top ends not at a level.

NOTE Both loads as well as reactions are linked to their reference member/object in Tekla Structural Designer. This means that any loads or reactions applied to a member that are beyond scope of the export will not be exported either.

Details of what gets exported and what is beyond scope are as follows:

Units	• Units are SI in the export file
Levels	• Number of levels Included • Level height Included
Gridlines	• Beyond scope (Adapt import)
Materials	• Concrete: Normal weight Included • Structural Steel Included • Concrete: Lightweight Beyond scope (Adapt import) • Mild Steel (Rebar) Beyond Scope (Tekla Structural Designer export)

	• Timber Beyond Scope (Tekla Structural Designer export) • Prestressing Beyond Scope (Tekla Structural Designer export) • Other / generic Beyond Scope (Tekla Structural Designer export)
Load combinations	• All active combinations are included apart from Seismic Inertia which is beyond scope
Loadcases	• All loadcases are included apart from EHF/NHF which is beyond scope
Beams	• Regular and irregular sections are Included • Beam at level Included • Export by span Included • Horizontal offsets Included • Vertical offsets Included • Tekla Structural Designer Labelling Included • Beam not at level Beyond Scope (Tekla Structural Designer export) • End trimming at column interface Beyond Scope (Tekla Structural Designer export) • Sloping beams Beyond scope (Adapt import) • Axially rotated beams Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Columns	• Regular and irregular sections are Included • Export by stack Included • Axially rotated columns Included • Tekla Structural Designer labeling Included • Tilted columns Included • Multi-stack columns Included

	• Columns with holes Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Walls	• Concrete meshed walls Included • Concrete mid-pier walls Included • Export by stack Included • Tekla Structural Designer labeling Included • No internal wall members Included • Multi-stack walls Included • Bearing walls Included (exc. reactions) • Wall alignments Included • Slopping walls Beyond scope (Adapt import) • Wall openings Beyond scope (Adapt import) • Wall extensions Beyond Scope (Tekla Structural Designer export) • Non-rectangular walls Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Slabs	• Concrete slab on beams Included • Concrete flat slabs Included • Export by panel Included • Tekla Structural Designer labeling Included • Vertical offsets Included • Slab overhangs Included • Non-concrete slabs Beyond Scope (Tekla Structural Designer export) • Slopping slabs Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)

Drop panels	• Concrete slab drops Included • Trimming at slab edges Included • Reinforcement Beyond scope (Adapt import)
Openings	• Slab rectangular openings Included • Slab circular openings Included
Braces	• Beyond Scope (Tekla Structural Designer Export)
Loading	• Self weight Included • User applied Force Included • User applied Moment Included • Level Point loads Included • Horizontal Line loads Included • Level area loads Included • Offsets from reference level Beyond scope (Adapt import) • Line and Area Loads applied to horizontal plane Beyond scope (Adapt import) • Vertical line loads applied as point loads at levels
Releases	• Translational releases Beyond Scope (Tekla Structural Designer Export) • Torsional releases Beyond Scope (Tekla Structural Designer Export)
Stiffness modifiers	• Beyond Scope (Tekla Structural Designer Export)

Instructions

1. On the **BIM Integration** tab, click **ADAPT Export**.

   This shows a dialog which gives the name of the ADAPT import file which Tekla Structural Designer will create. You can change the name and location of the file if necessary.
2. Once the file details are correct click **Save** to create the ADAPT import file.
3. Launch ADAPT and import the file to see your project.

Export a model to STAAD

To export a model to STAAD, see the following limitations and instructions.

Limitations

The scope of the STAAD Export is constrained by the limitations of the STAAD Text File (.std) format.

Note that:

- Elastic extensions cannot be defined in STAAD. Therefore, additional 1D elements are created to preserve model connectivity.
- The exported file will not group elements in any way. 1D elements will not be grouped as members (or columns, beams, etc). In addition, there will not be any grouping of faceted elements from a curved member. 2D elements will not be grouped into panels.
- Section data is exported as analysis properties only. This means that no attempt is made to reference STAAD library sections.
- Material data is exported as analysis properties only. This means that no attempt is made to reference STAAD library materials.
- Objects such as Shear Only Walls have no equivalent native object in STAAD and some of the internal data such as panel geometry and spring properties end up lost during conversion.

Instructions

1. Create the model as usual.
2. On the **BIM Integration** tab, click **STAAD Export**.

   The **Save As** dialog opens.
3. If necessary, change the name and location of the file.
4. Click **Save**.
5. Start STAAD and open the file.

Export a model to Autodesk Robot Structural Analysis

Before you export a Tekla Structural Designer model to Autodesk Robot Structural Analysis, read the limitations of Robot Export in the following paragraphs. Once you have read the limitations, you can proceed to the detailed instructions on how to export models from Tekla Structural Designer to Autodesk Robot Structural Analysis.

Limitations

The scope of the Robot Export is constrained by the limitations of the Robot Text File (STR) Format.

Note that:

- The model exported is the one used for 1st order linear static analysis (see <Linear analysis of structures containing material non-linearity> to remove non-linearity from the exported model.)
- 1D element springs are totally omitted from the export.
- Elastic extensions cannot be defined in Robot. Therefore, additional 1D elements are created to preserve model connectivity.
- The exported file will not group elements in any way. 1D elements will not be grouped as members (or columns, beams, etc). In addition, there will not be any grouping of faceted elements from a curved member. 2D elements will not be grouped into panels.
- Section data is exported as analysis properties only. This means that no attempt is made to reference Robot library sections.
- Material data is exported as analysis properties only. This means that no attempt is made to reference Robot library materials.
- Loads in projection are converted to equivalent loads in Robot.
- Robot will not import the material properties for timber from the STR file. When the file is opened in Robot, you can delete the G value and adjust the values manually after import.
- It is not possible to define part-length distributed torsional loads in the STR file. Therefore, they are converted to equivalent full-length distributed torsional loads.

Instructions

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Robot Export**.

   The **Save As** dialog opens.
3. If necessary, change the name and location of the file.
4. Click **Save**.
5. Start Robot and open the file.

Export a model to the cloud

To export a Tekla Structural Designer model to the cloud, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Cloud Export**.

   The **Save As** dialog opens.
3. If necessary, change the file name and location.

4. Click **Save**.

Export to One Click LCA

 **One Click LCA** is third party software for performing a life cycle assessment of your structure and optimizing its embodied carbon.

From Tekla Structural Designer you can either compile an *offline* report, which is a compact summary of materials in the structure that can be edited if required before being manually imported into One Click LCA, or you can upload the report data directly to One Click LCA *online*.

Overview

Scope

Tekla Structural Designer includes material information in tabular data tables, in general the scope of the exported data matches this. This gives a lot of the most commonly requested material quantity information, but is not considered to be exhaustive:

- Data available in Tekla Structural Designer but not exported
  - Shear studs on composite beams
- Data NOT available in Tekla Structural Designer (and therefore not exported)
  - Reinforcement in any slab other than cast-in-place reinforced concrete slabs
  - Bearing Walls
  - Shear Only Walls
  - Transverse shear reinforcement required by composite beams
  - surface area of timber members, cold rolled members, general material members
  - DELTABEAM® mass - a zero value is exported to help ensure allowance is added manually
  - analysis elements
- Exported Data not otherwise available in Tekla Structural Designer
  - Precast slab screed/topping area
  - Composite slab decking type / area
  - material in gable and parapet posts

- surface area of defined “roof panels” and “wall panels”

The above summary is not guaranteed to be exhaustive, it is important to consider “what is missing”? Are all the significant physical items you defined in Tekla Structural Designer being covered by the export?

Additional information

The following notes may assist with initial use of the export:

- Grouping in One Click LCA

  On import there is a phase where it is suggested that lots of the exported data items can be grouped together on the basis of a shared “IFCMATERIAL”. The sections below give examples where you may consider the value of doing this:

  - composite / non-composite beams

    the mass of these beam types are exported separately.

    you may wish to apply higher carbon factors to composite beams to allow for shear studs and transverse shear reinforcement.

  - steel beams of all different fabrication types

    the mass of steel members of different fabrication types are exported separately.

    e.g. rolled, plated, Westok, FABSEC®, DELTABEAM®

    you may wish to apply different carbon factors to these different types of member

  - surface areas

    the surface areas of slabs and many beams are all exported separately.

    if you have no requirement to use any surface area information then it will be easier to leave these grouped for simpler deletion.

- Slabs

  - The data for each slab is exported separately (a slab may contain many slab-items).

    In simple situations this will mean one group of data for each level, but if multiple slabs are defined in a level you will get separate data for each.

  - Data for different types of slab is also kept separate

    e.g. Cast-in-place reinforced slabs, post-tensioned slabs, precast slabs, composite slabs, various other general material slabs

    For reinforced concrete slabs this allows a more detailed understanding of reinforcement per slab.

- Limitation - as noted above reinforcement data is not available/ exported for other slab types, this will need to be allowed for in one of two ways:
  - Use the surface area information to add an allowance per m2
  - Adjust the carbon factor or concrete/topping in the slabs to allow for reinforcement content

  In either case this will mean avoiding grouping of the relevant data items.
- “roof panels” and “wall panels” surface areas
  - Roof and wall surface area data is included in case it assists with the inclusion of cladding materials.

Feedback

- Please send us feedback if you feel that:
  - Important data is missing
  - Included data should be totalled/presented differently - perhaps there should be options to do things in different ways (possibly at different stages of design?)
  - There should be options to filter out certain data types because you don’t always want to use them (for example surface area info)

Show report

1. On the **BIM Integration** tab, click **One Click LCA**.

   The **One Click LCAReport** dialog opens.
2. Click **Show report** The report opens in Excel, (or whatever software is associated with .xlsx files).

The resulting materials file can then be edited if required before being manually imported into **One Click LCA**, or it can be passed on to a third party and imported into **One Click LCA**.

Show online results

NOTE	Online export requires a One Click LCA license.

1. On the **BIM Integration** tab, click **One Click LCA**.

   The **One Click LCAReport** dialog opens.
2. Enter your One Click LCA user name and password, then click **Log in**.

3. Click **Show online results**. The material data from the Tekla Structural Designer model is uploaded to One Click LCA.
4. Proceed through the One Click LCA wizard:
   a. Create/Choose project
   b. Consider groupings
   c. Review / apply mapping

   On completion the amount of embodied carbon in the model is determined.

Export to IDEA StatiCa Connection Design

The method of export will depend on the version of IDEA StatiCa installed:

- Export to IDEA StatiCa Connection Design (version 22 and later) (page 1207)
- Export to IDEA StatiCa Connection Design (version 21 and earlier) (page 1216)

Export to IDEA StatiCa Connection Design (version 22 and later)

Before exporting connections to IDEA StatiCa, you are advised to be aware of the limitations. Then, you can proceed to follow the detailed instructions to perform the export.

Provided you have IDEA StatiCa version 22 or later installed, connections are exported using the Checkbot link.

The Checkbot link has the following advantages:

- Saves time by exporting multiple connections simultaneously thus reducing errors and speeding up connection design.
- Connections can be designed for all or selected load cases/combinations by user choice.
- You can access both Checkbot and Tekla Structural Designer simultaneously, for reviewing analysis results etc.

For more information about the Checkbot link see the Checkbot info page on IDEA StatiCa's website.

Checkbot link limitations

The Checkbot link is compatible with the following versions of IDEA StatiCa:

- 22 (and later)

The following limitations should be noted.

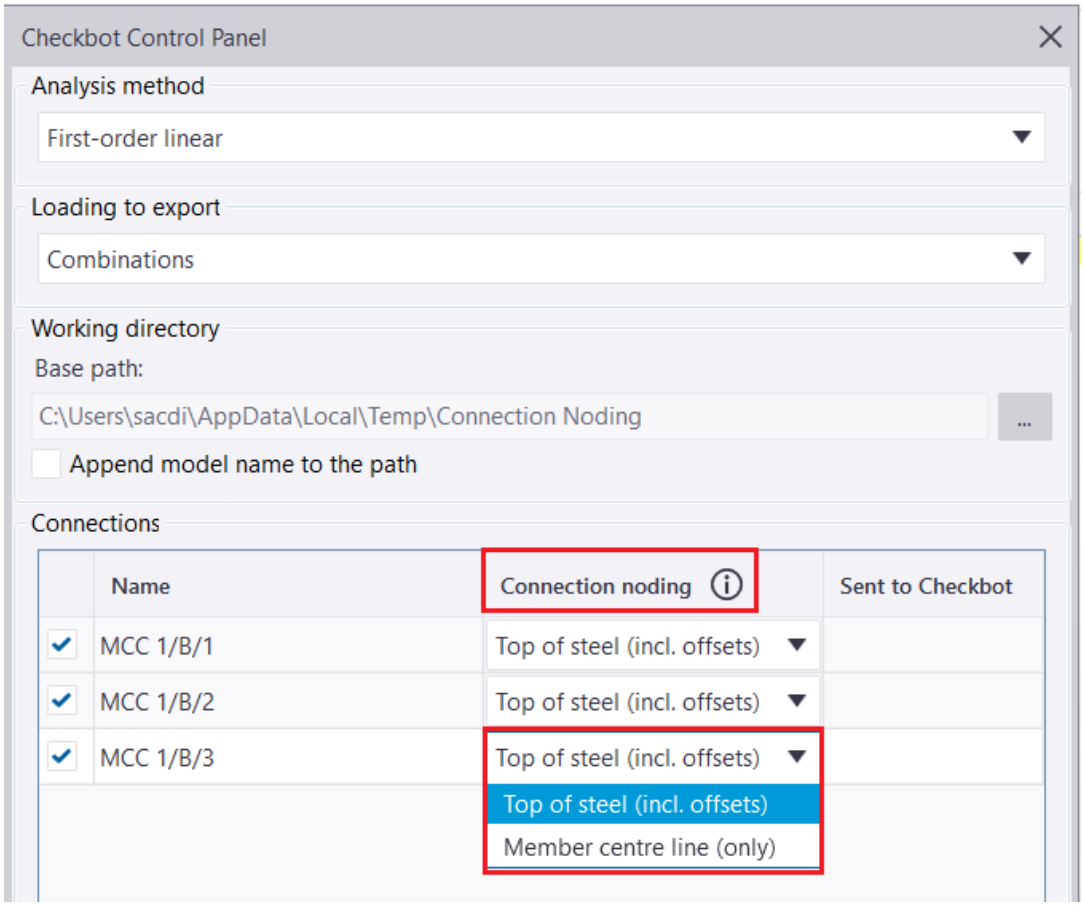
- Regional Codes supported by the link; US, Eurocode, Australia, and BS

NOTE While the link is valid for the BS regional code, connections cannot be designed to BS 5950 in IDEA StatiCa. (They could however be designed to Eurocode).

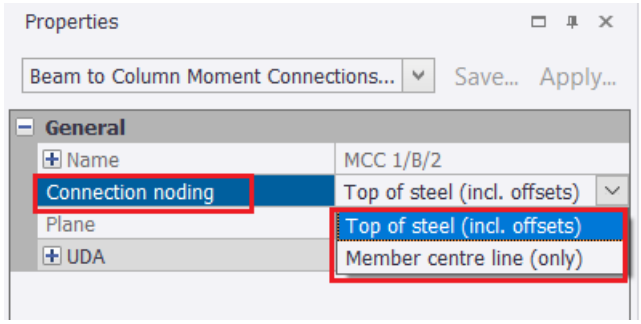
- Column base plates are not exported.
- For connections identified as ‘Moment Connections’ using Update connections in Tekla Structural Designer any out of plane members are not exported.
- For any connection type, haunches are not exported.
- For the 'Top of steel (incl. offsets)' connection noding option only, Tekla Structural Designer uses ToS and the wire model is at this level too. The export to IDEA depicts the ToS so that the 3D graphic looks correct. However, the wire model typically connects at offsets of half the beam depth. This introduces additional forces/moments due to eccentricity of line of action etc. This line of action of the force set can be adjusted in IDEA StatiCa.
- The communication of data is one-way from Tekla Structural Designer to Checkbot. No data (such as section changes or connection details etc) is returned to the Tekla Structural Designer model, so the user is responsible for making any such changes manually.
- The current link cannot handle a huge amount of connection data (e.g. thousands of connections with hundreds of load combinations) in a single instance. For such a case, it is necessary to select an individual or a group of connections to export.
- Multiple connections formed at a single joint in Tekla Structural Designer cannot be exported to IDEA StatiCa in a single instance - it is necessary to export these connections separately.

Connection noding

When using the Checkbot link you must choose the **Connection noding** required for each connection.



Alternatively, after selecting your connections you can select the required option from the **Properties** window.

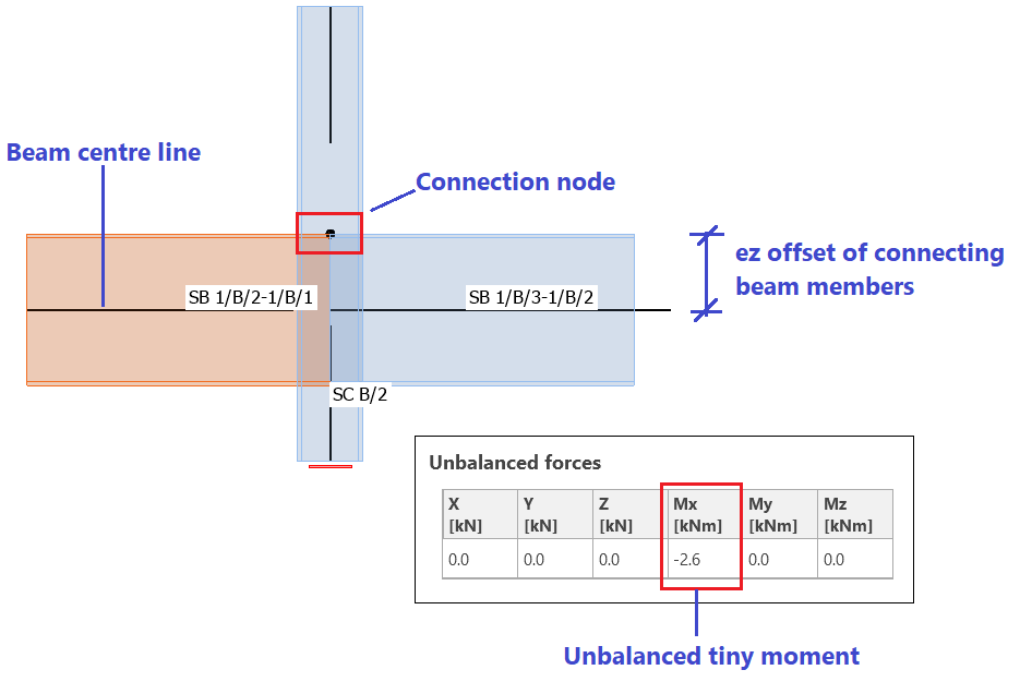


The two options are described below:

Top of steel (incl. offsets)

- This is the default option.

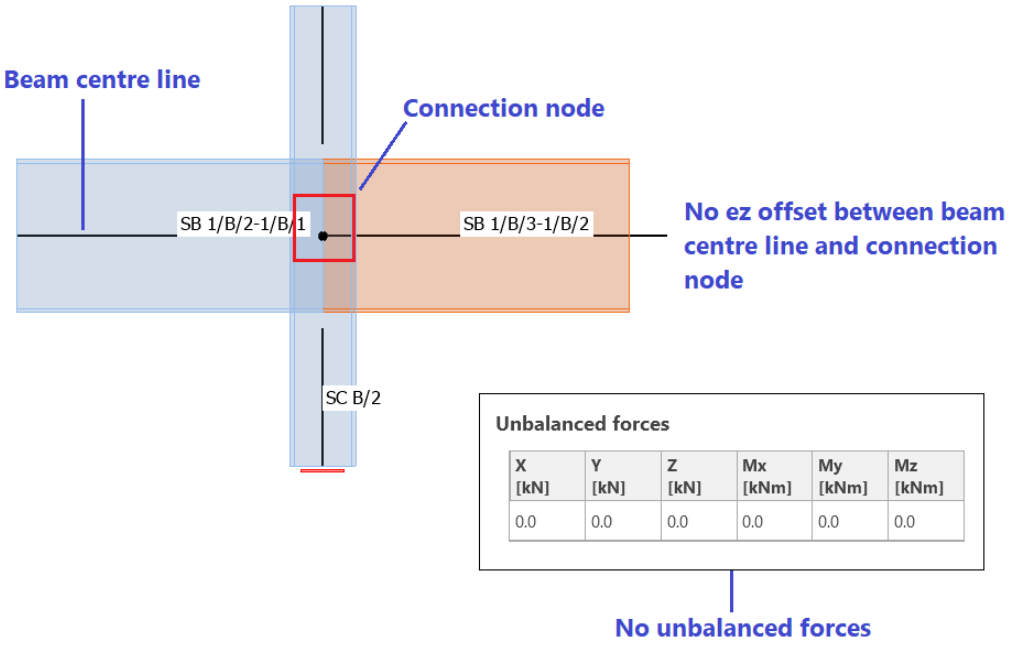
- Connections exported with this option will ensure that, providing there are no offsets, all of the beams connecting at a joint will have the top of the top flange aligned.
- In this case, when exported to IDEA StatiCa, each beam centre line will connect to the node at an offset, ez of half the beam depth.
- Otherwise, any user-defined offsets specified on the 'Alignment' page of the **Properties** window will be retained.
- In the less common case of connecting beam members carrying axial load, small eccentric moments will develop and are reported as 'Unbalanced forces' in IDEA StatiCa. In some cases a connection designer may be uncomfortable with these unbalanced forces, if so, the alternative 'Member centre line (only)' option may be preferred in this situation.



Member centre line (only)

- This option ignores physical member offsets in the Tekla Structural Designer model. This means that when exported to IDEA StatiCa, each beam centre line will connect to the node with ez offset = 0, regardless of any user-defined offsets specified on the 'Alignment' page of the '**Properties** window.

- This option eliminates the occurrence of tiny eccentric moments introduced by connecting beam members carrying axial load. But might not depict the required physical connectivity at the joint.



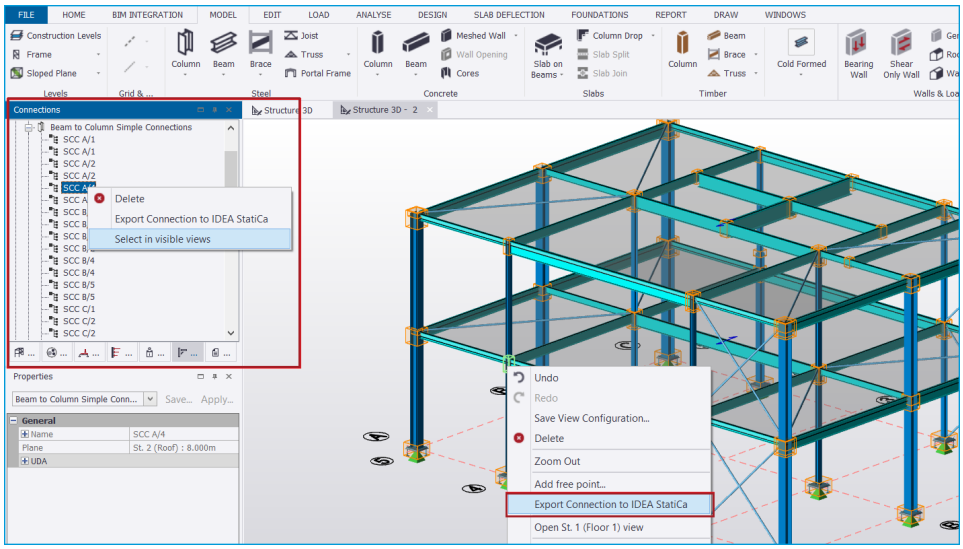
NOTE This option does not change any analysis results of connecting beam members exported to IDEA StatiCa.

Export to IDEA StatiCa using the Checkbot link

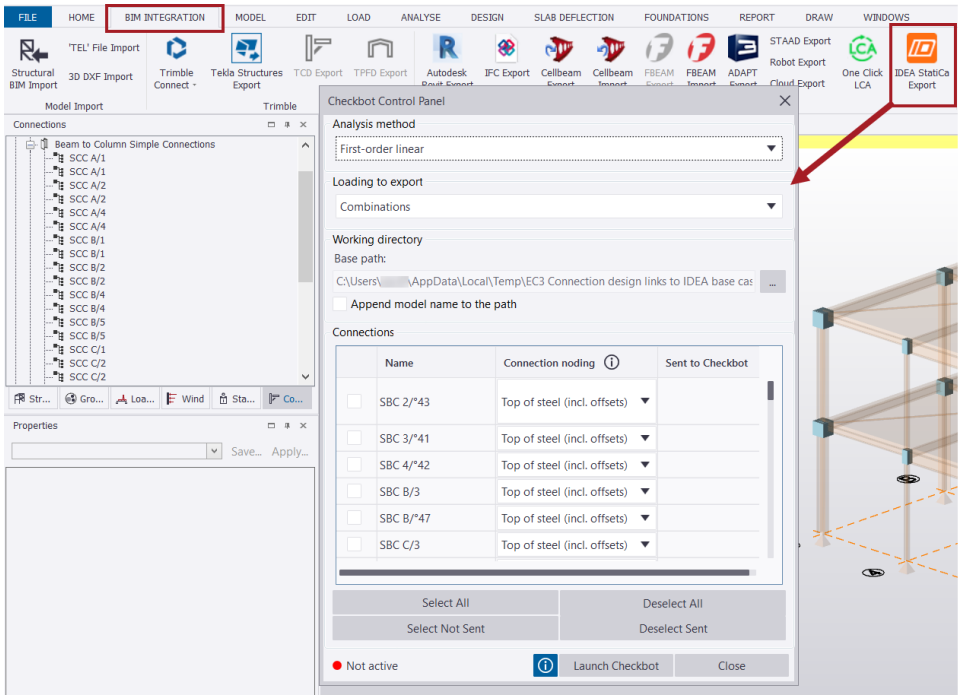
1. Connections can only be exported once the connection objects for them exist. These are created as follows:
   a. Click on the **Connections** tree in the **Project Workspace**
   b. Right-click the **Connections** branch of the tree.
   c. Choose **Update Connections** from the context menu.

   The resulting connections are listed in the **Connections** tree and are also shown by bounding boxes in the Scene Views.
2. Once the connections exist, open the Checkbot Control Panel (page 1531) using one of the following methods:
   a. Right-click a connection in a Scene View and choose **Export Connection to IDEA StatiCa**

b. Or, right-click a connection in the Project Workspace **Connections** tree and choose **Export Connection to IDEA StatiCa**



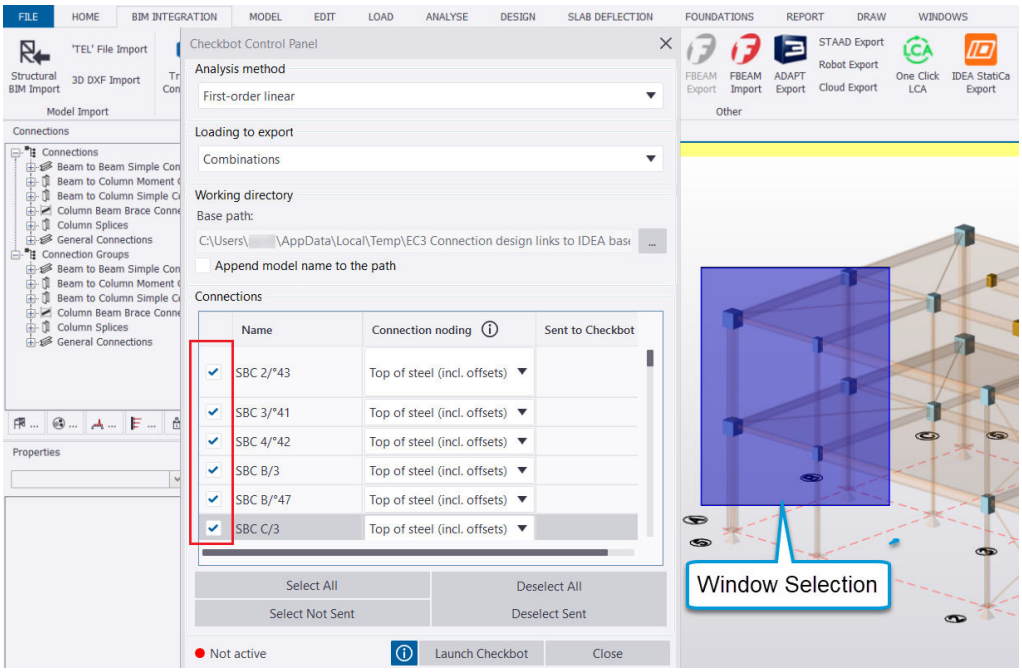
c. Or, from the **BIM Integration** tab, choose **IDEA StatiCa Export**



For either export method, the Checkbot Control Panel (page 1531) is then displayed and populated with the list of connections available for transfer to IDEA StatiCa (as shown above).

3. Select the analysis method.
4. Select the loading to export.
5. Select the connections to export.

While the Checkbot Control Panel is open, selected connections are graphically displayed by coloured boxes - you can make further graphical selections of connections by any of the usual graphical methods such as a window selection, as shown below.

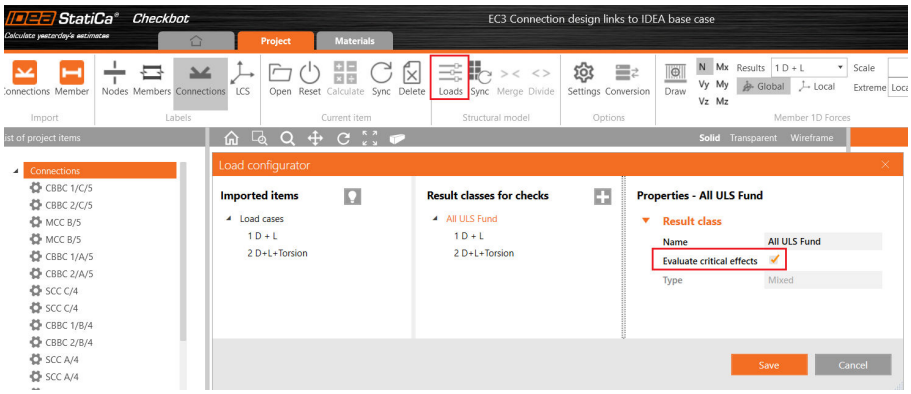


NOTE Connections that have previously been sent to Checkbot will be marked by means of a tick in the column 'Sent to Checkbot'.

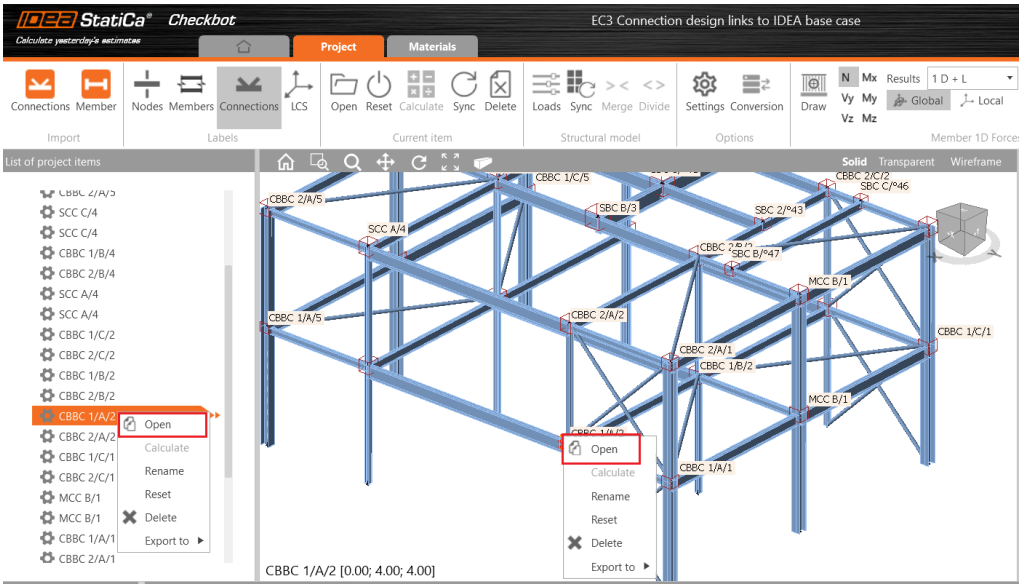
6. Select your required **Connection noding** option, as either **Top of steel (incl. offsets)** (default) or **Member centre line (only)**
7. Once the required connections have been selected and their connection noding options have been set, click **Launch Checkbot**
8. Select the appropriate design code.
9. Select **Create project**
10. In the **Import** group, select **Connections**

NOTE By default, IDEA StatiCa considers critical effect combinations for connection design. Having selected a Result class as shown below,

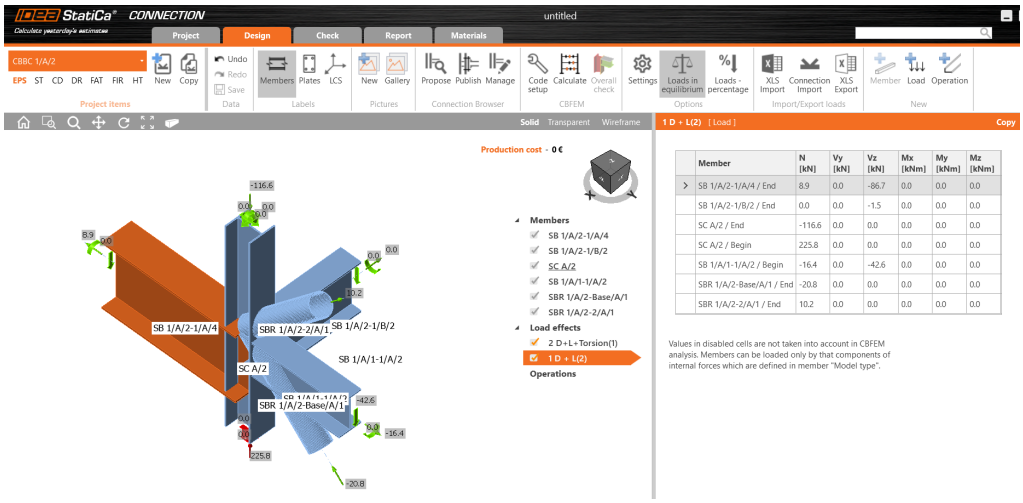
you can clear the option **Evaluate critical effects** to instead consider all the load combinations if required.



11. In Checkbot, select a connection, either from the list of project items or from the graphical view and choose **Open** from the context menu:

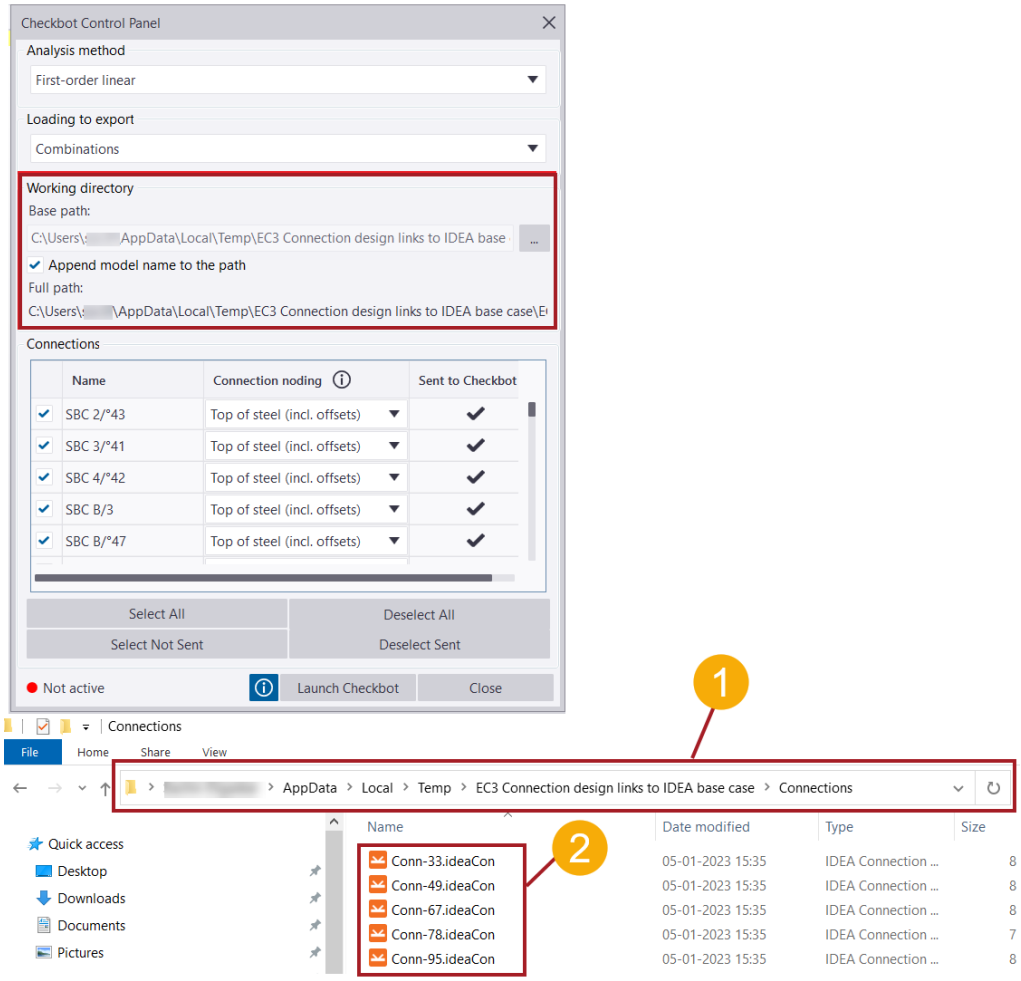


12. The selected connection can then be configured and designed in IDEA StatiCa for selected load cases/ combinations:



13. If you need to edit the connection in a more fundamental manner - such as adjusting the line of action of forces - you must open the exported connection file directly in IDEA StatiCa Connection. In this case full editing of all aspects of the connection is enabled within the scope of IDEA

StatiCa. The connection file is located in the working directory as shown below.



1 - Working directory path

2 - Connections generated in Connections sub-folder

Tekla Structural Designer to IDEA StatiCa Checkbot

Export to IDEA StatiCa Connection Design (version 21 and earlier) (page 1216)

Export to IDEA StatiCa Connection Design (version 21 and earlier)

Before exporting connections to IDEA StatiCa, you are advised to be aware of the limitations. Then, you can proceed to follow the detailed instructions to perform the export.

If you have IDEA StatiCa version 21 or earlier installed, connections are exported one at a time using a simple link.

Link limitations

The link is compatible with the following versions of IDEA StatiCa:

- 9.1
- 10.0
- 10.1 (build 10.1.113 or later)
- 20.1
- 21

The following limitations should be noted.

- Design codes supported; LRFD, ASD, Eurocode and AS (Australia).
  - BS 5950 and IS (India) are not supported.
- Column base plates are not exported.
- For connections identified as 'Moment Connections' using Update connections in Tekla Structural Designer any out of plane members are not exported.
- For any connections identified in Tekla Structural Designer the brace members are not passed through to IDEA StatiCa.
- For any connection type, haunches are not exported.
- For the above versions of IDEA StatiCa, Tekla Structural Designer uses ToS and the wire model is at this level too. The export to IDEA depicts the ToS so that the 3D graphic looks correct. However, the wire model typically connects at offsets of half the beam depth. This introduces additional forces/moments due to eccentricity of line of action etc. This line of action of the force set can be adjusted in IDEA StatiCa.
  - An option to use member centre line noding is available, but to use this you would need to install IDEA StatiCa version 22 onwards (and also be using Tekla Structural Designer version 2022 SP4 onwards).

Export to IDEA StatiCa

1. Right-click the connection in a Scene View.
2. Choose **Export Connection to IDEA StatiCa** from the right-click menu, selecting the analysis results to use at the same time.

3. Specify the file name and location for this connection.

   The IDEA StatiCa application opens to allow the connection to be designed, the following data having been transferred:

   - Connection geometry
   - Section profile & material grade
   - Connection forces for each active solved Tekla Structural Designer combination.

   NOTE The Tekla Structural Designer model remains frozen until you close the IDEA StatiCa connection.

4. Add bolts, stiffeners etc as required and design the connection.

When you close the connection Tekla Structural Designer becomes active once more; the connection file is automatically embedded in the model.

Review of IDEA connections designed in Tekla Structural Designer

1. Within Tekla Structural Designer click **Review > UDA**
2. In the Properties Window set **Mode** to Review and **Attribute** to IDEA StatiCa file

   Joints associated with IDEA StatiCa are shown
3. Use the right-click context menu to open previously exported connections back to IDEA StatiCa.

Integration with IDEA StatiCa Connection design

Export to IDEA StatiCa Connection Design (version 21 and earlier) (page 1216)

9 Manage models

Click the links below to find out more various model management tasks:

- Manage global and model settings (page 1219)
- Manage materials (page 1265)
- Manage properties and property sets (page 1284)
- Manage project revisions and view metrics (page 1289)
- Create and manage user-defined attributes (page 1292)
- Create and manage sub structures (page 1296)
- Working with large models (page 1303)

9.1 Manage global and model settings

Default Global settings (page 1232) exist in Tekla Structural Designer for controlling many aspects of the software operation, such as the analysis, design, report and drawings.

To save editing time on a model-by-model basis, we recommend that the global settings are configured to suit your needs and saved to one or more settings set(s) (page 1220).

For example you might choose to customize the precision of units (page 1225), the reference format (page 1227) used for naming objects, the regional design codes (page 1224) to work to, or any of the many other settings.

When you create a new model its Model settings (page 1243), Analysis settings (page 760), Design settings (page 866), Drawing settings (page 1119), and Footfall assessment settings are loaded from the active settings set. Each of these settings are then specific to the new model only.

If you have changed any of these model specific settings and want to revert them, you can choose to re-load them from the active settings set. Alternatively, if you want to keep them and have them applied to new models

also, you can choose to save them to the active settings set. For details, see: Apply and manage model specific settings (page 1262).

Manage settings sets

The first time Tekla Structural Designer is run after installation you are required to select a country or region. Based on this selection, Tekla Structural Designer creates an initial settings set (the *active* set), containing defaults for design codes, units, and sections, and settings for drawings, schedules, and reports.

You can edit the settings in this set and also create new sets, but only one set can be designated the active settings set.

Each new project takes a copy of the active settings set to form its own independent model settings.

Model settings can then be edited as required without affecting the active settings set.

Import a settings set for a specific region

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Click **Import...**
4. Select the region for which you want to import the settings.
5. Click **OK**.

NOTE To use these settings in your new projects, you will need to make this the **active set**.

Edit the content of a settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set you want to edit.
4. In the **Settings** dialog box, modify the settings according to your needs. For example:
   - Modify the font used to display results in the **Results Viewer**.
   - Modify the appearance of reports.

- Set the units that you want to use, and set the desired precision for the units.
- Set the design codes that you want.
- Set default section sizes for each of the member types.
- Set if confirmations are required for specific actions.
- Set element references.
- Modify the appearance of schedules.
- Modify drawing types and styles.
- Modify the colors used in 2D and 3D views.

5. Click **OK** to save the settings to the selected settings set.

Make a duplicate copy of an existing settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Click **Add Copy**.
4. In the **Settings** dialog box, modify the settings for the new settings sets.
5. Click **OK** to save the settings to the new settings set.

Restore a settings set to 'factory defaults'

After you have edited the contents of a settings set and saved the changes, you might subsequently decide you want to go back to the original 'factory default' values. The easiest way to do this is to re-import the settings set for the same region. The newly imported set is automatically given a new name so that it can exist alongside the original set if required.

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Click **Import...**
4. Select the same region as is used in the set you want to restore to 'factory defaults'.
5. Click **OK**.
6. The newly imported set will adopt a different name to the original set for the same region. For example if the original set was "UK Settings" the new set would be "UK Settings 2"

NOTE To use these settings in your new projects, you will need to make this the **active set**.

If the original set is no longer needed it can be removed. See "Delete a settings set" below.

Change the active settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set required from the drop list.
4. To make the set a default set for new projects, click **>> Active**.
5. Click **OK**.

Delete a settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the set you want to delete.
4. Click **Remove**.

Load settings from the active settings set to the current project

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Ensure that the required settings set is active. If it is not, click **>> Active**.
4. Click **OK**.
5. Open the dialog that contains the settings to be updated, either:

   - On the **Home** ribbon, click  **Model Settings**
   - On the **Home** ribbon, click  **Embodied Carbon Factors**
   - Or, on the **Analyze** ribbon, click  **Settings**

- Or, on the **Design** ribbon, click  **Settings**
- Or, on the **Draw** ribbon, click  **Settings**

6. Click **Load...**

   A confirmation dialog appears.
7. To update the model settings to match the active settings set, click **Yes**.

Save settings from the current project to the active settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Ensure that the settings set is active. If it is not, click **>> Active**.
4. Click **OK**.
5. Open the dialog that contains the settings to be saved.
   - On the **Home** ribbon, click  **Model Settings**
   - Or, on the **Home** ribbon, click  **Embodied Carbon Factors**
   - Or, on the **Analyze** ribbon, click  **Settings**
   - Or, on the **Design** ribbon, click  **Settings**
   - Or, on the **Draw** ribbon, click  **Settings**
6. Click **Save...**

   A confirmation dialog appears.
7. To update the active settings set to match the current settings, click **Yes**.

Copy a settings set from one computer to another

In order to apply consistent settings on a company wide basis, the xml file of the appropriate settings set should be manually copied to other computers.

1. On the computer containing the settings set, go to the **Home** ribbon and click  **Settings**.

   The **Settings** dialog box opens.

2. Go to **Settings Sets**.
3. Click **Open Folder**.
4. Select the xml file of the settings set, and manually transfer it to the **Settings** folder on each destination computer.
5. To view and use the new settings set, restart Tekla Structural Designer on the destination computers.

See also

Settings set settings (page 1234)

Define and modify regional codes and design codes

Tekla Structural Designer allows you to select from a range of international design codes of practice. Each new project initially adopts the codes that have been specified in the active settings set. However, you can also change the codes in the middle of a project.

See also

Design code settings (page 1243)

Define default design codes for new projects

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.
2. Select a suitable settings set, and click **>> Active** to make it active.
3. Go to **Design Codes**.
4. Check that the regional code is set as required.

   The regional code controls which action and resistance codes are available for selection.
5. Set the action and resistance codes according to your needs.
6. Click **OK**.

   Tekla Structural Designer retains the selected codes to apply them for each new project until you decide to modify them.
7. To use the set design codes in a new project, on the **Home** tab, click **New**. A new project opens with its design codes, (and all of its other model settings) copied from the active settings set.

Change design codes in an existing project

WARNING If you change the regional code in an existing project, the following will occur in the model:

- Some materials, steel sections, design section orders, studs, decks and reinforcement may require re-selecting in the model to make them consistent with the new regional code/ unit system.
- Wind loading (if any) and wall/roof panel properties will be deleted. The wind wizard will need rerunning, wall/roof panel properties need resetting and the wind loadcases will need recreating.
- Snow loading (if any) will be deleted. The snow wizard will need re-running.
- Seismic loading (if any) will be deleted. The seismic wizard will need re-running.
- All combinations will be deleted.

1. On the **Home** tab, click **Model Settings**.

   The **Settings** dialog box opens.
2. Go to **Design Codes**.
3. Select the regional code as required.

   The regional code controls which action and resistance codes are available for selection.
4. Set the action and resistance codes according to your needs.
5. Click **OK**.

Define and modify units

Tekla Structural Designer allows you to switch between Metric and US customary units. Furthermore, by following the below instructions you can select the units which you want to use in the selected unit system. For example, if you use US customary units, you can change say, the Force unit to input in either kip or lb, or the Dimension unit to have dimensions measured in either feet, inches, fractional inches, feet and inches, or feet and fractional inches.

Change units and unit precision in an existing project

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog box opens.
2. Go to **Units**.
3. Select the unit system (Metric, or US Customary).

   NOTE If you change the unit system in an existing project, you should also consider the design section orders applied to steel/cold formed members as these may require re-selecting (so that metric or customary size sections are selected aligned with the Units setting).
4. Set the units and unit precision that you want Tekla Structural Designer to use.
5. Click **OK**.

   NOTE The length unit can be set appropriate for the type of dimension being input:

   - **Fine Dimension** units are used for defining stud spacings, section size constraints, and other typically small distances.
   - **Dimension** units are used for defining grid spacings, positioning members, positioning load locations, and so on. They are also used to control the accuracy of any measured dimension lines that you apply to the model.
   - **Deflection** units are used for reporting deflection results.
   - **Distance** units are used for defining large dimensions.

Once you have set the units according to your needs, Tekla Structural Designer will then expect input in the same format.

Example

If the **Dimension** units are set to **ft, in fract.**:

- To input a dimension of two feet, six and one quarter inches, you need to type `2' 6 1/4"`
- To input a series of irregular grid lines in the grid wizard at spacings of fifteen feet, followed by three spacings of twenty feet, six and one half inches, followed by one spacing of fifteen feet, you need to type **`15',3*20' 6 1/2",15'`**

Define the default units and unit precision for new projects

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.
2. Select a suitable settings set, and click **>> Active** to make the set active.
3. Go to **Units**.
4. Select the unit system.
5. Set the units and unit precision that you want Tekla Structural Designer to use.

   NOTE The length unit can be set appropriate for the type of dimension being input:

   - **Fine Dimension** units are used for defining stud spacings, section size constraints, and other typically small distances.
   - **Dimension** units are used for defining grid spacings, positioning members, positioning load locations, and so on. They are also used to control the accuracy of any measured dimension lines that you apply to the model.
   - **Deflection** units are used for reporting deflection results.
   - **Distance** units are used for defining large dimensions.

   ---
6. Click **OK**.
7. To use the set units in a new project, on the **Home** tab, click **New**. A new project opens with its units, (and all of its other model settings) copied from the active settings set.

Once you have set the units according to your needs, Tekla Structural Designer will then expect input in the same format.

Example

If the **Dimension** units are set to **ft, in fract.**:

- To input a dimension of two feet, six and one quarter inches, you need to type `2' 6 1/4"`
- To input a series of irregular grid lines in the grid wizard at spacings of fifteen feet, followed by three spacings of twenty feet, six and one half inches, followed by one spacing of fifteen feet, you need to type **`15',3*20' 6 1/2",15'`**

Manage object references

Tekla Structural Designer contains a flexible object referencing system that is specifically designed considering the use of multiple materials within the same model.

You can define the object references to meet your needs on the **References** page of the **Settings** dialog box.

See also

References settings (page 1245)

Basics of object reference formats

Object reference formats are user definable. Each object type has its own reference format that has been built using individual components. You can customize the reference format by adding or removing components as required.

See some of the typical components in the following table:

Icon	Item	Further information
M	Material	Fully user-definable text. For example: • S for Steel • C for Concrete
C	Characteristic	Fully user-definable text. For example: • B for Beam • C for Column
L1	Start Level Reference	Not applicable
L2	End Level Reference	
P1	Start Point Reference	Grid or construction point reference. The start and end reference points P1 and P2 work from the grids that you define, and from the construction points that have been created
P2	End Point Reference	

Icon	Item	Further information
		automatically when you place members that do not lie between existing points. **NOTE** When reference points are constructed using two grid reference points, you must always use a separator. For example, type `A/11` to avoid confusion with A1 and 1.
D	Direction	Applies for beams only.
X	Count	Tekla Structural Designer keeps a separate count for each level for each object type. **NOTE** Tekla Structural Designer does not keep a count by material. For example, steel beams and concrete beams are included in the same count on a level. If the direction prefix is specified, Tekla Structural Designer keeps separate counts for each direction. Direction 1 is defined as objects falling within ± 45 degrees of the horizontal, and Direction 2 ± 45 degrees of the vertical of the global axis.
Not applicable	Reinforcement > Bar Layer: • T1 • T2 • B1 • B2	• T1 is the top surface reinforcement in the 1st (outer) layer. • T2 is the top surface reinforcement in the 2nd (inner) layer. • B1 is the bottom surface reinforcement in the 1st (outer) layer. • B2 is the bottom surface reinforcement in the 2nd (inner) layer.
	Reinforcement > Bar Size	Not applicable
	Reinforcement > Center Spacing	The center spacing between bars.
	Reinforcement >	An item that is typically added in front of the bar size.

Icon	Item	Further information
	Detailing prefix	
[ ]	Custom text	Fixed text that you can include at any position within the reference. For example: • `Block C`
\ - / [ ] ×	Separators • backslash • dash • slash • space • times	Optional items that you can place between items.

Note that:

- When you create objects of a certain type, the default references from the active settings set are applied.
- You can modify the syntax of the reference format at any time. Objects created after the changes will adopt the new format.
- Once objects have been created, you can edit their references on an object-by-object basis, so that they can be further individualized.
- For members, the object reference does not include the group reference, geometric shape, and section size as part of the reference descriptor. Instead, options are provided in the **Scene Content** settings and the GA drawing's control to show the information.

Modify reference formats and texts in an existing project

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog box opens.
2. Go to **References**.
3. Adjust the way in which the references are applied.
4. Click **OK**.

Modify the reference format syntax of an object type

1. Go to **References** --> **Formats** .
2. Click the **...** button on the right side of the desired reference format.

3. Do one of the following:

To	Do this
Add an extra item to the reference format	• Click **Add...** and select the desired item. The selected item appears at the end of the reference format.
Re-order the items in the reference format	a. Select the item you want to move. b. Drag the item to reposition it.
Add custom text to the reference format	a. Click **Add...** and select **Custom Text**. b. Drag the new component to the desired position. c. Click the **Custom Text** component to edit it. d. Type the desired text in the box. e. Click **Set**.
Remove an item from the reference format	a. Select the item you want to remove. b. Drag the item outside the **Edit Reference Format** dialog box.

4. Click **OK**.

Change the text used for the materials and characteristics in the reference format

1. Go to **References** --> **Texts** .
2. According to your needs, go to the **Characteristics** or **Materials** tab.
3. Modify the desired text according to your needs.
4. Click **OK**.

Renumber members

You can use the **Renumber** command to simultaneously renumber all member types in the model whose reference format includes the count item.

Member types are initially numbered in the order in which they are created. Renumbering makes the members easier to find in the model and on drawings.

By default, the **Renumber** command works from the lowest plane or level upwards. The count starts at 1 and continues sequentially.

NOTE To change the renumbering direction and the starting value, go to **Model Settings** --> **References** --> **General** .

---

1. On the **Structure** tab of **Project Workspace**, right-click the **Members** branch.

2. In the context menu, select **Renumber**.

   Tekla Structural Designer automatically renumbers all members in the model that include a count in their reference format.

Renumber slabs

You can use the **Renumber** command to simultaneously renumber all slab items in the model.

Slab items are initially numbered in the order in which they are created. Renumbering makes them easier to find in the model and on drawings.

By default, the **Renumber** command works from the lowest plane or level upwards. The count starts at 1 and continues sequentially.

NOTE To change the renumbering direction and the starting value, go to **Model Settings** --> **References** --> **General** .

---

1. On the **Structure** tab of **Project Workspace**, right-click the **Slabs** branch.
2. In the context menu, select **Renumber**.

   Tekla Structural Designer automatically renumbers all slab items in the number.

Adjust the default references to be applied to new projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select a suitable settings set in the list, and click **>> Active** to make it active.
4. Go to **References**.
5. Adjust the way in which the references are applied to each of the object types in your projects.
6. Click **OK**.
7. To start a new project using the new references, on the **Home** tab, click **New**.

Global settings

Default global settings exist in Tekla Structural Designer for controlling many aspects of the software operation, such as the analysis, design, report and drawings.

To save editing time on a model-by-model basis, we recommend that the global settings are configured to suit your needs. This way, you do not need to change the model specific settings every time you create a new model.

NOTE When you create a new model file, the global settings are copied to the model file and become the Model settings (page 1243), Analysis settings (page 760), Design settings (page 866), , Footfall assessment settings and Drawing settings (page 1119) specific to that model and that model only.

Click **Home** --> **Settings** to edit the following global settings:

- Settings set settings (page 1234)
- General settings (page 1234)
- Results Viewer settings (page 1236)
- Report settings (page 1086)
- Unit settings (page 1245)
- Design code settings (page 1243)
- Design settings (page 866)
- Analysis settings (page 760)
- Loading settings (page 1247)
- Structure default settings (page 1237)
- Section order default settings (page 1238)
- References settings (page 1245)
- Drawing settings (page 1119)
- Material list settings (page 1253)
- Beam lines settings (page 1253)
- Analysis Model settings (page 1254)
- Solver settings (page 1238)
- User-defined attribute settings (page 1257)
- Scene settings (page 1239)
- Structural BIM settings (page 1259)
-
- Footfall assessment settings
- Embodied carbon settings (page 1262)
- Performance settings (page 1242)

Settings set settings

The **Settings Sets** page in the (page 1583) controls the settings sets which are used to create the model/analysis/design settings in new projects, and which can also be used to replace the model/analysis/design settings in existing projects.

Setting	Description
Available settings sets list	Allows you to select a settings set whose content you can view and modify on the other pages of the **Settings** dialog box.
Add Copy	Creates a new settings set based on the one selected in the **Available settings sets** list. You can then customize the new settings set on the other pages of the **Settings** dialog box.
>> Active	Makes the settings set that is selected in the **Settings** dialog box the active settings set.
Import...	Allows you to import a settings set from another region. The selected settings set appears in the **Settings** dialog box.
Rename	Allows you to rename the settings set that is selected in the **Available settings sets** list.
Remove	Deletes the settings set that is selected in the **Available settings sets** list.
Open Folder	Opens the folder in which the existing settings sets are located.

See also

Manage settings sets (page 1220)

General settings

The **General** page and its subpages in the (page 1583) allow you to configure the language, captions, autosave settings, confirmation messages, and updates in Tekla Structural Designer.

Language	
Select the preferred language list	Allows you to select whether the terms in the interface and the output are in US English or UK English. **NOTE** The language cannot be configured differently in different settings sets.

Appearance	
Show captions on top	Allows you to select whether you want to show captions on the top or the bottom of the **Project Workspace**.

Autorecover	
Autosave	
Enabled	Allows you to select whether Tekla Structural Designer creates an automatic saves of models. Autosave may be useful, for example, when restarting Tekla Structural Designer after a crash. For more information, see: Work with autosave and backups (page 39)
Interval	Allows you to determine the interval at which Tekla Structural Designer creates automatic saves of models.
Backups	
Enabled	Allows you to select whether Tekla Structural Designer creates automatic backups of models. Backup may be useful, for example, if you want to revert to a previous version of the model. For more information, see: Work with autosave and backups (page 39)
Interval	Allows you to determine the interval at which Tekla Structural Designer creates automatic backups of models.
Location	Defines the backup **Location** directory - the default is your Windows user profile Documents directory. Click the link to browse to

Autorecover	
	your preferred location (both local and network locations are allowed).
Maximum number of backups per project	default = 10 (when the limit is reached the older backups start to be automatically deleted).

Confirmations	
Confirm on list	Allows you to select the operations that you want to confirm each time.

Update Service	
Enable Update Service	Allows you to select whether you want Tekla Structural Designer to notify you when new product updates or service packs can be installed, so that you can keep Tekla Structural Designer up to date.
Check for updates when starting Tekla Structural Designer	Allows you to select whether Tekla Structural Designer check if new updates are available each time it starts.
Check for non critical updates every [n] days	Allows you to determine the interval at which Tekla Structural Designer checks if new non-critical updates are available.

Results Viewer settings

The **Results Viewer** page and its subpages in the (page 1583) allow you to configure the appearance of the on-screen results viewer for the settings set selected in the **Select the settings set to edit** list.

NOTE Unlike other settings, changes made to the active settings set on the **Results Viewer** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Setting	Description
General subpage	
Viewer font --> **Change...**	Allows you to modify the font that is displayed on the left side pane of the results viewer.
Styles subpage	

Setting	Description
Report Styles list	Allows you to select the style whose font, color, or other properties you want to modify. The options are: • **Calc Normal**: not used in the current version. • **Table**: controls all the text displayed on the right side pane of the results viewer except for the first row in any table. • **Table Heading**: controls the heading row, or first row, in any table.
Font --> **Change...**	Allows you to change the font used for the currently selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the foreground color of the selected style.
Colors --> **Background** --> **Clear**	Allows you to use a clear background for the selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the background color of the selected style.
Vertical alignment	Allows you to modify the vertical alignment of the selected style.
Indentation	Allows you to determine the indentation of the selected style.
Line spacing	Allows you to determine the line spacing of the selected style.

Structure default settings

The **Structure Defaults** page in the (page 1583) allows you to configure miscellaneous structure default settings that are applied to the settings set selected in the **Select the settings set to edit** list.

Setting	Description
Construction Levels --> **Default Type**	Allows you to define the default type for the new constructions levels you create in the **Construction Levels** dialog box.
Grid & Construction Lines --> **Extension length**	Allows you to define the default extension length of grid and construction lines.

Setting	Description
Pattern loadcases	Allows you to define whether load patterns are applied to loadcases.
Nominal Cover	Allows you to modify the nominal cover of different members. **NOTE** Unlike other settings, changes made to the nominal cover settings are instantly applied to new members.

Section default settings

The **Section Defaults** page in the (page 1583) allows you to specify the default section size for each member type when a new member is created. The default section sizes are applied to the settings set selected in the **Select the settings set to edit** list.

To change the default section size for a member type, click the section size in the **Section** column.

Section order default settings

The **Section Order Defaults** page in the (page 1583) allows you to modify the default section orders for different countries by clicking the desired order in the **Section Order** column.

Solver settings

The **Solver** page in the (page 1583) allows you to specify the solver method that is applied to the settings set selected in the **Select the settings set to edit** list.

Setting	Description
Global Matrix Storage	Allows you to specify the global matrix storage method. The options are **Compressed Sparse Row** and **Skyline**. **NOTE** We strongly recommend that you select the **Compressed Sparse Row** option because it generally reduces the

Setting	Description
	analysis time, particularly in the case of finite element analysis. The Skyline option is provided solely to cater for the possibility of Sparse failing to work for a specific model.

Scene settings

The **Scene** page and its subpages in the (page 1583) allow you to control the color and opacity of each object type in scene views when using the settings set selected in the **Select the settings set to edit** list.

NOTE Unlike other settings, changes made to the active settings set on the **Scene** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Graphics

Setting	Description
Graphics subpage	
Driver	**NOTE** The default settings on the **Graphics** subpage are the recommended settings. Do not adjust them unless the Tekla Support Team instructs you to do so.
Antialiasing	
Transparency	

Colors

Setting	Description
Background color list	Allows you to select whether the table below displays and allows you to adjust the colors for a light or dark background.
Gradient Background	Unselect this option to apply a uniform white or black background to each window, with no gradient applied.

Setting	Description
Reset Colors	Resets all the colors to the default settings.
Color Name	Displays the interface item to which the color applies. **TIP** An arrow is displayed to the left of some of the items, this can be clicked on in order to set the colors for sub-items. **NOTE** The **Gradient Legend** colors are those used when reviewing embodied carbon.
Opacity	Allows you to set the opacity of each color.
Color	Allows you to set the color for each interface item according to your needs.

Fonts

Setting	Description
Reset Fonts	Resets all the fonts to the default settings.
Font Name	Displays the interface item to which the font applies.
Font	Allows you to change the font used for the interface item.
Size	Allows you to adjust the size of the font.
Bold	Allow you to further format the font.
Italic	

Contours

Setting	Description
Lower bound [%]	Displays the lower bound of each FE contour in the results view. **TIP** To adjust the size of the lower bound, modify **Size [%]**.

Setting	Description
Upper bound [%]	Displays the upper bound of each FE contour in the results view. **TIP** To adjust the size of the upper bound, modify **Size [%]**.
Size [%]	Allows you to adjust the size of individual contours according to your needs. Note that the sum of the sizes must equate to 100 %.
Color	Allows you to change the color of a contour.
Split	Divides the contour that is currently selected in the table into two contours that are each half the size of the original contour.
Delete	Deletes the contour that is currently selected in the table.
Reset	Resets the contours to the default configuration that consists of 10 evenly sized contours.

Utilization Ratios

Setting	Description
Minimum value	Allows you to adjust the minimum value of a ratio band. **NOTE** The value of the highest ratio band can be increased above 1.0, if necessary.
Color	Allows you to change the color of a ratio band.
Add	Creates a new ratio band that you can modify to the bottom of the table.
Delete	Deletes the ratio band that is currently selected in the table.
Reset	Resets the ratio bands to the default configuration that consists of 5 evenly sized ratio bands.

Setting	Description
Sort	Rearranges the ratio bands in order of the values (from highest to lowest).

View Settings

Setting	Description
Do not display values of story shear below	Allows you to limit the values of story shear for a new model. This way, you can easily ignore the small values of story shear that might otherwise detract you from the more important story shear values. The story shear values that are less than the limiting value are not displayed in the results view.
Default analysis method in results view	The selected method is used when opening a new graphical view and switching to the results regime, provided that no analysis or design has been run in the current session. If analysis or design has been run the analysis method from that run is selected instead.
Show full pile length	Allows you to select whether Tekla Structural Designer displays the full length of piles or a shorter pile length that you can specify.
2 x scale for steel columns	Allows you to double the scale of steel columns in 2D views in order to simplify viewing the columns and their orientation.

Performance settings

The **Performance** page in the (page 1583) allow you to optimize analysis and design performance according to your preference and your PC's capabilities.

Setting	Description
Design	Options are provided to allow you to take advantage of multi-core processing in the design and chase-down phases. • **Use multi-core processors for design**

Setting	Description
	• **Run chase-downs concurrently**

Model settings

Click **Home** --> **Model Settings** to edit the following settings specific to the current model. After they have been edited, click Save, Load, Cancel, or OK (page 1263) according to your requirements.

- Design code settings (page 1243)
- Unit settings (page 1245)
- References settings (page 1245)
- Loading settings (page 1247)
- Minimum lateral loads settings (page 1249)
- Global imperfections settings (page 1250)
- Live/imposed load reduction settings (page 1251)
- Sway/drift settings (page 1252)
- Grouping settings (page 1252)
- Material list settings (page 1253)
- Beam lines settings (page 1253)
- Analysis Model settings (page 1254)
- Validation settings (page 1256)
- User-defined attribute settings (page 1257)
- Graphics view settings (page 1259)
- Structural BIM settings (page 1259)
- Embodied carbon settings (page 1262)

See also

Analysis settings (page 760)

Design settings (page 866)

Drawing settings (page 1119)

Design code settings

The **Design Codes** page allows you to specify the regional code and the subsequent design codes that are applied

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**

- in *new projects* - when accessed by clicking **Home** --> **Settings**

Item	Description
Select the Regional Code list	Allows you to select the regional code that you want to apply. When you select a regional code in the list, the **Design Codes** table automatically updates. **WARNING** If you change the regional code in an existing project, the following will occur in the model: • Some materials, steel sections, design section orders, studs, decks and reinforcement may require re-selecting in the model to make them consistent with the new regional code/unit system. • Wind loading (if any) and wall/roof panel properties will be deleted. The wind wizard will need rerunning, wall/roof panel properties need resetting and the wind loadcases will need recreating. • Seismic loading (if any) will be deleted. The seismic wizard will need re-running. • Snow loading (if any) will be deleted. The snow wizard will need re-running. • All combinations will be deleted.

Item	Description
Design Codes table	Displays the available action codes and resistance codes, which are dependent on the selected regional code. The lists within the **Design Codes** table can be used to select between available alternatives.

See also

Define and modify regional codes and design codes (page 1224)

Unit settings

The **Units** page allows you to specify the units, format and precision that are applied

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

Item	Description
System list	Allows you to determine whether metric or US customary units are used.
Table of quantities	Displays each quantity, its current unit and its precision. You can modify quantities by selecting them in the table.
Settings list	Displays and allows you to select the units for the currently selected quantity.
Precision list	Displays and allows you to define the precision of the currently selected quantity.
Use for values lower than	Allow you to define the values for which Tekla Structural Designer applies exponential formats.
or greater than or equal to	

See also

Define and modify units (page 1225)

References settings

The **References** page and its subpages allow you to adjust different object reference settings

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

General

Numbering	
Initial value in levels	Allows you to specify the start number (such as 1, 100, 1000) at each construction level for object references that contain the **Count** item.
Renumbering Direction	Allows you to specify renumbering directions that control how the member numbering is applied when you use the **Renumber** command.
Renumber user names	If you have applied user names in order to override the default names assigned to members; when you use the **Renumber** command the user names will be retained unless you select this option.

Grid line and grid point naming	
Ignore letters I & O	Allows you to ignore the letters I and O in grid line names.
Initial number	Allows you to specify the initial number applied to the first grid lines.
Initial letter	Allows you to specify the initial letter applied to the first grid lines.
Naming style list	Allows you to select a desired naming style for grid lines.
Grid point naming style list	Allows you to select a desired naming style for grid points. This directly impacts the Start Point Reference [P1] and End Point Reference [P2] formats typically used for steel member referencing. Note that if the Dir 1/Dir 2 style is selected, the grid line aligned closest to building direction 1 will come first; if the Dir 2/Dir 1 style is selected, the grid line aligned closest to building direction 2 will come first.

Groups

NOTE The **Groups** subpage is only displayed in the **Settings** dialog box not in the **Model Settings** dialog box.

Item	Description
Sub-group Name	Allows you to specify the text used to designate the group labelling. This text forms the stem of the Design Group and Detailing Group names that are displayed on the Groups tab of the Project Workspace. These names are shown in the output reports and drawings when grouped design has been applied.

Formats

Item	Description
Format table	Lists each object type showing its current reference format and allows you to customize the formats. **NOTE** To customize a object reference format, click the click the edit button at the right end of the row.

Texts

Item	Description
Characteristics table	Allows you to specify the text used to designate the characteristic for object references that contain the **Characteristic** item.
Materials table	Allows you to specify the text used to designate the material for object references that contain the **Material** item.

See also

Manage object references (page 1227)

Loading settings

The **Loading** page and its subpages allow you to adjust certain loading settings

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

General

Item	Description
Use load patterning for steel beams	When imposed/live loadcases have been set to be patterned, load patterning is automatically applied to continuous concrete beams. This option allows you to select whether similar patterning is applied to continuous steel beams.
Use patterning of eccentricity moments for steel columns	Allows you to select whether patterned eccentricity moments are considered in the design of steel columns.
Pipework Operating & Testing Content Loadcase Type	Allows you to select whether the ancillary loadcases created for lines of pipework are considered as Dead or Live (Imposed).
Equipment Operating & Testing Content Loadcase Type	Allows you to select whether the equipment loadcases created for operating and testing are considered as Dead or Live (Imposed).

Line/Area Ancillary Loading

Item	Description
Ancillary Type	Ancillary Type.
Dead/Empty Load	The default Dead/Empty Load factors applied for each type.
Live (Imposed)/Content Load	The default Live (Imposed)/Content Load factors applied for each type.
Testing Content Load	The default Testing Content Load factors applied for each type.

Wind Loading

Item	Description
Entity Type	The loadcase names that will be created when specific ancillary types have been specified.
Shape factor C_f	The default C_f factors come from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE
Effective Area$_{xy\ ,\ xz}$	The effective area of entity in xy and xz planes respectively.
Factor$_{xy\ ,\ xz}$	Factor applied to the effective area of entity in xy and xz planes respectively.
Area Per Length$_{xz}$	The area per length.

Minimum lateral loads settings

The **Home** --> **Model Settings** --> **Minimum Lateral Loads** page allows you to set up a minimum lateral load (MLL) on the structure as a percentage of the gravity loading.

Item	Description
Setup Minimum Lateral Loadcases	Choose how many minimum lateral loads to apply. • None • Case A • Case A & B If either 'Case A' or 'Case A & B' are selected, the following further inputs are required.
Gravity Loads	Select the load types used to be used in the calculation. • Dead • Dead & Live (Imposed)
Using Combination	Select the loads to be used in the calculation. • Factored Loads • Unfactored Loads
Direction 1	Set the percentage of the specified gravity load to be applied in Dir 1

Item	Description
Direction 2	Set the percentage of the specified gravity load to be applied in Dir 2

Global imperfections settings

This **Model Settings** page is used to control the percentages of load used for the global imperfection calculations.

The percentages are material independent, which means that the same percentages are used for concrete and steel. The percentages vary depending on the height of the structure and the number of columns in each direction.

EHF Global Imperfections (Eurocode)

Item	Description
Height of the structure	Allows you to specify the effective height of the structure to be used in the EHF calculations.
Set Default	Sets the height of the structure to the highest construction level.
Number of columns in X direction	In the Eurocode Cl 5.2(5) the calculation of the reduction factor a_m depends on the number of contributing members, m. Valid input for m in the X and Y directions is any whole number from 1 to 1000. The default value is 1 which results in a_m = 1.000. If a value of 1000 is entered then a_m would reduce to 0.707.
Number of columns in Y direction	
Global initial sway imperfections	Displays the calculated alpha values and phi percentages for the above input. The override checkboxes can be used to enter your own percentages in place of the calculated values if required.

NL Global Imperfections (US regional code)

Item	Description

Item	Description
Global initial sway imperfections Direction Dir 1, Dir 2	Displays the calculated percentage imperfections to be applied in each direction. The override checkboxes can be used to enter your own percentages in place of the calculated values if required.

NHF Global Imperfections (BS, IS & AS regional codes)

Item	Description
Global initial sway imperfections Direction Dir 1, Dir 2	Displays the calculated percentage imperfections to be applied in each direction. The override checkboxes can be used to enter your own percentages in place of the calculated values if required.

See also

Display notional forces, minimum lateral loads and seismic equivalent lateral forces (page 684)

Live/imposed load reduction settings

The **Home** --> **Model Settings> Live load reduction** (US regional code) or **Imposed load reduction** (other regional codes) page allows you to control the live/imposed load reductions.

US regional code

Item	Description
Limit on live load reductions	Specify the limit on reductions applied to beams. Specify the limit on reductions applied to column stacks and wall panels for the number of floors carried. These apply to all loadcases of type **Live Load**

Item	Description
Limit on roof live load reductions	Specify the limit on reductions applied to beams columns and walls. These apply to all loadcases of type **Roof Live Load**

Other regional codes

Item	Description
Number of floors carried	The number of floors carried by the column or wall.
Reduction percentage	Specify the reduction applied to the column stack or wall panel for the number of floors carried.

See also

Activate reductions in live or imposed loadcases (page 325)

Sway/drift settings

The **Sway/Drift** page displays settings applicable to the sway or drift checks.

By clicking **Home** --> **Model Settings** --> **Sway/Drift** you can display settings applicable to the sway or drift checks for the current project.

Item	Description
Calculate using	Displays the method that will be used for the sway or drift calculations. The option that is offered depends on the code being worked to as follows: • for BS, IS and Aus codes - λ_{crit} (0.5% gravity load approach) • for Eurocode - α_{crit} (0.5% gravity load approach) • for US codes - Indicative stability coeff ($\delta_{2nd}/\delta_{1st}$)

See also

Sway check (page 828)

Drift check (page 823)

Grouping settings

The **Home** --> **Model Settings** --> **Grouping** page allows you to control the tolerance applied when members are grouped in the current project.

Item	Description
Maximum edge length variation	Allows for a tolerance to be applied to the automatic grouping. A member can only be included in an existing group if its span length is within the specified tolerance of the group's (average) span length.
Maximum length variation	Allows for a length tolerance to be applied to the automatic grouping of trusses. A truss can only be included in an existing group if its span length is within the specified tolerance of the group's (average) span length.
Maximum height variation	Allows for a height tolerance to be applied to the automatic grouping of trusses. A truss can only be included in an existing group if its height is within the specified tolerance of the group's (average) height.

Material list settings

The **Material List** page allows you to specify settings affecting material lists

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

Item	Description
Ignore openings with area less than	Allows you to specify the size of opening that can be considered small enough to be ignored when determining the quantity of necessary slab reinforcement.

See also

Apply attribute filters to material lists and reports (page 1295)

Beam lines settings

The **Beam Lines** page allows you to control the parameters used for continuous concrete beam formation

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

NOTE The parameters are only used to control the automatic concrete beam joining that occurs during the design process or when the **Beam Lines** command is run. The parameters are not considered when members are joined manually using the **Join** command.

Item	Description
Join pinned beam end	Allows you to control whether joining should occur or not if a pin is defined at the end the last span of the first beam or the start of the first span of the second beam, the fixity at the end in question changing from pinned to continuous once joined. If the beam is subsequently re-split at the same location, the pin gets reinstated.
Limiting join angle in plan	Specifies the limiting angle in plan to be applied for joining beams. Only beams meeting in plan at an angle less than the specified value can be joined. Where two beams start at the end of the first, the one that has the minimum angle is the one that gets connected.
Limiting join angle in elevation	Specifies the limiting angle in elevation to be applied for joining beams. Only beams meeting in elevation at an angle less than the specified value can be joined. Where two beams start at the end of the first, the one that has the minimum angle is the one that gets connected.
Minimum section overlap	Allows you to apply a tolerance when joining beams if they do not fully overlap in section. You can use it to prevent joining if there is very little physical overlap between the beam cross sections.

See also

Automatically join all concrete beams (page 279)

Analysis Model settings

On the **Analysis Model** page you can adjust certain rigid zone, curved beam, and concrete column centerline settings:

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

Rigid Zones

Design codes allow engineers to assume parts of concrete beams or columns are rigid, leading to more efficient designs.

NOTE Rigid zones should not be confused with rigid offsets, which are used to ensure that the analysis model is properly connected. You can have rigid offsets in the model even if rigid zones are turned off.

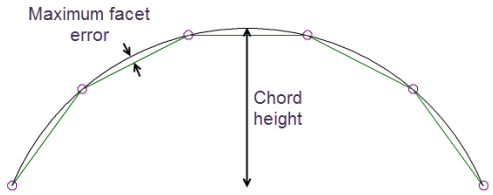
Item	Description
Percentage of rigidity	Allows you to specify the extent of the rigid zone created. The option has an effect only if rigid zones are applied (see below).
Rigid zones not applied	Allows you to switch rigid zones on or off. The option affects where releases are applied in the analysis model and where members start and end for design. When rigid zones are not applied, the design model corresponds to the analysis model, whereas when they are applied, the design model is defined between the ends of the rigid zones.

NOTE There is a significant difference between **Rigid zones not applied** and rigid zones applied with 0% rigidity. The total elastic length of a member will be the same in the two models, but the position of releases and start or end of design members will be different.

Curved Beams

In the solver model, curved beams are replaced by a series of straight line solver elements. The number of solver elements used being controlled by the **Maximum facet error** that has been specified.

Item	Description
Maximum facet error	For any given chord height, reducing this value increases the number of

Item	Description
	solver elements used to represent the curve. 

Column Centerlines

For concrete columns connected to a two-way slab, the column perimeter shape is cut out of the slab 2D element mesh. Stiff analysis 'rigid link' elements then connect the column centerline (where the column 1D elements are located) to the 2D mesh nodes. In some circumstances - for example for L-section columns of certain proportions - very short link elements could result, which could cause validation errors or warnings and analysis solution problems.

The following settings are provided in order to move the column centerline a small distance where needed to avoid these problems.

Item	Description
Automatically move column centerlines when the smallest rigid link is less than the error limit for 1D element length	This option is initially on by default. The error limit used in this check is specified in Validation settings (page 1256)
Automatically move column centerlines when the smallest rigid link is less than the warning limit for 1D element length	This option is initially off by default. The warning limit used in this check is specified in Validation settings (page 1256)

Validation settings

The **Home** --> **Model Settings** --> **Validation** page allows you to control the checks that are applied when the current model is validated.

Item	Description
Error limit for length	Allows you to control when an error is displayed when very short analysis elements are detected.
Warning limit for length	Allows you to control when a warning is displayed when very short analysis elements are detected.

Item	Description
Error limit for quality	Allows you to control when an error is displayed when poor quality 2D elements are detected. **NOTE** 2D element quality depends on two things: skew and aspect ratio. 0% is bad quality: a squashed triangle tends towards bad quality, whereas 100% is perfect quality, so an equilateral triangle is perfect quality.
Warning limit for quality	Allows you to control when a warning is displayed when poor quality 2D elements are detected.
Check for validation warnings	Allows you to specify which model, analysis, and design validation checks are performed. If a box is cleared, the validation check is not performed.

User-defined attribute settings

The **User Defined Attributes** page allows you to control the attributes that are available

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

You can add new attribute definitions, delete them, set a type for each attribute, and adjust the acceptable values.

Item	Description
Name	Displays and allows you to modify the name of the attribute that will be displayed in the **UDA** section of the **Properties** window.
Type list	Allows you to specify a type for the option. The options are text, number, and file.
Source list	Allows you to restrict the allowable input of the attribute, if necessary. The two options are: • **Custom Value**: Does not restrict the allowable input.

Item	Description
	• **Value List**: Restricts the allowable input for an attribute to a pre-set list.
Values	Allows you to specify the allowable attribute values when the source is set to **Value List**.
Linked	This setting has no effect if all levels in the model are unique. If the model contains existing duplicate levels it works as follows: • for UDAs that are linked, a single UDA value is assigned to the entity on the source and the duplicate level. • for UDAs that are not linked, a different UDA value can be assigned to an entity on a duplicate level. **NOTE** In the Construction Levels dialog, when a level is first set to be duplicate of a source level: • initially UDA values from the source level are copied to the duplicate regardless of the **Linked** setting • only from that point on does the linked/not linked functionality apply. • i.e. when set to not linked, the UDA at each level becomes independently editable.
Add	Creates a blank row that allows you to define a new attribute.
Delete	Deletes the currently selected attribute from the table.

Item	Description
Move Up	Move the currently selected attributes up or down in the table and in the **Properties** window.
Move Down	

See also

Create and manage user-defined attributes (page 1292)

Graphics view settings

The **Home** --> **Model Settings** --> **Graphics View Settings** page allows you to control the display of miscellaneous items in 2D and 3D views in the current project.

Setting	Description
Do not display values of story shear below	Allows you to limit the values of story shear for a new model. This way, you can easily ignore the small values of story shear that might otherwise detract you from the more important story shear values. The story shear values that are less than the limiting value are not displayed in the results view.
Default analysis method in results view	The selected method is used when opening a new graphical view and switching to the results regime, provided that no analysis or design has been run in the current session. If analysis or design has been run the analysis method from that run is selected instead.
Show full pile length	Allows you to select whether Tekla Structural Designer displays the full length of piles or a shorter pile length that you can specify.
2x scale for steel columns	Allows you to double the scale of steel columns in 2D views in order to simplify viewing the columns and their orientation.

Structural BIM settings

The **Structural BIM** page and its subpages allow you to control the structural BIM import and export processes

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

Export --> Continuous Objects

Item	Description
Separate objects for each stack	Allows you to select which columns are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export concrete columns as separate objects, no rebar information is exported.
Separate objects for each span	Allows you to select which beams or other members are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export concrete beams as separate objects, no rebar information is exported.
Separate objects for each panel	Allows you to select which wall types are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export shear walls as separate objects, no rebar information is exported.

Export --> Reinforcement Information

Item	Description
Concrete columns	Allows you to select whether bar marks are included in the exported concrete column reinforcement information.
Concrete beams	Allows you to select whether bar marks, link leg count, and region length percentage are included in the

Item	Description
	exported concrete beam reinforcement information.

Export --> Mappings

NOTE The **Export** --> **Mappings** page is only displayed in **Settings**, not in **Model Settings**

Item	Description
Materials --> **Reset mappings for Structural BIM export**	Resets the mappings for Structural BIM export of materials to the default settings.
Decking --> **Reset mappings for Structural BIM export**	Resets the mappings for Structural BIM export of decking to the default settings.

Import --> Settings

Item	Description
Concrete beams --> **Default "Allow automatic join"**	Allows you to control whether new imported concrete beams are automatically joined or not at the end of the import. **NOTE** If the BIM model contains continuous beams as one piece, you might want to clear the option. After the import, you can change the properties back, if necessary.
Steel columns --> **Ignore splice offsets in physical member positions**	Allows you to select whether splice offsets are ignored when importing physical member positions.

Import --> Mappings

NOTE The **Import** --> **Mappings** page is only displayed in **Settings**, not in **Model Settings**

Item	Description
Materials --> **Reset mappings for 'TEL' File import**	Resets the mappings for TEL file import of materials to the default settings.

Item	Description
Materials --> **Reset mappings for Structural BIM import**	Resets the mappings for Structural BIM import of materials to the default settings.
Decking --> **Reset mappings for Structural BIM import**	Resets the mappings for Structural BIM import of decking to the default settings.

See also

Import a project from a Structural BIM Import file (page 1156)

Embodied carbon settings

The **Embodied Carbon** page allows you to control the carbon factors applied

- in the *current* project - when accessed by clicking **Home** --> **Model Settings**
- in *new projects* - when accessed by clicking **Home** --> **Settings**

Item	Description
Carbon factors Carbon Factors defined	**NOTE** The **Carbon Factors defined** button is only displayed in **Settings**, not in **Model Settings** Opens the Embodied Carbon Factors dialog box (page 1542) to allow you to define the embodied carbon factors that will be used in new models.
Steel Connections Fixed-ended allowance	Allows you to define the allowance for fixed-ended connections in steel members as a percentage of the mass of the member.
Steel Connections Pin-ended allowance	Allows you to define the allowance for pin-ended connections in steel members as a percentage of the mass of the member.

Apply and manage model specific settings

To apply and manage the various defaults and settings that apply to the current project, see the following instructions.

Apply and manage model settings

1. On the **Home** ribbon, click  **Model Settings**.
2. Adjust the Model settings (page 1243) according to your needs.
3. After adjusting the settings, do one of the following:
   a. To apply the changes to the current project: Click **OK**
   b. To save the changes back to the active settings set for future use: Click **Save...**
   c. To cancel the changes: Click **Cancel**
   d. To revert to the model settings specified in the active settings set: Click **Load...**

   CAUTION When you save or load, all pages of the Model Settings dialog box are saved/loaded, not just those on the current page.

   Analysis settings (page 760), Design settings (page 866), Drawing settings (page 1119) and have their own dialog boxes and are not affected.

Apply and manage analysis settings

1. On the **Analyze** ribbon, click  **Settings**.
2. Adjust the Analysis settings (page 760) according to your needs.
3. After adjusting the settings, do one of the following:
   a. To apply the changes to the current project: Click **OK**
   b. To save the changes back to the active settings set for future use: Click **Save...**
   c. To cancel the changes: Click **Cancel**
   d. To revert to the model settings specified in the active settings set: Click **Load...**

   CAUTION When you save or load, all pages of the Analysis Settings dialog box are saved/loaded, not just those on the current page.

Apply and manage design settings

1. On the **Design** ribbon, click  **Settings**.
2. Adjust the Design settings (page 866) according to your needs.
3. After adjusting the settings, do one of the following:
   a. To apply the changes to the current project: Click **OK**
   b. To save the changes back to the active settings set for future use: Click **Save...**
   c. To cancel the changes: Click **Cancel**
   d. To revert to the model settings specified in the active settings set: Click **Load...**

   CAUTION When you save or load, all pages of the Design Settings dialog box are saved/loaded, not just those on the current page.

Apply and manage drawing settings

1. On the **Draw** ribbon, click  **Settings**.
2. Adjust the Drawing settings (page 1119) according to your needs.
3. After adjusting the settings, do one of the following:
   a. To apply the changes to the current project: Click **OK**
   b. To save the changes back to the active settings set for future use: Click **Save...**
   c. To cancel the changes: Click **Cancel**
   d. To revert to the model settings specified in the active settings set: Click **Load...**

   CAUTION When you save or load, all pages of the Drawing Settings dialog box are saved/loaded, not just those on the current page.

Apply and manage slab deflection settings

1. On the **Slab Deflection** ribbon, click  **Settings**.
2. Adjust the according to your needs.

3. After adjusting the settings, do one of the following:
   a. To apply the changes to the current project: Click **OK**
   b. To save the changes back to the active settings set for future use: Click **Save...**
   c. To cancel the changes: Click **Cancel**
   d. To revert to the model settings specified in the active settings set: Click **Load...**

CAUTION	When you save or load, all pages of the Slab Deflection Settings dialog box are saved/loaded, not just those on the current page.

9.2 Manage materials

Tekla Structural Designer's material databases contain an extensive range of sections, materials, reinforcement, decking, connectors and more for each regional code and country. You can view and manage these databases from the Materials dialog box (page 1564).

The initial data in the material databases is system data which is protected, so that standard items cannot be accidentally edited or deleted. However, you can add user-defined materials (page 1265) from other sources or suppliers into the databases as and when required. All user-defined properties are fully editable.

Click the links below to find out more:

- Add user-defined materials (page 1265)
- Manage design section orders (page 1272)
- Add simple connection resistances (page 1276)
- Upgrade material databases (page 1282)
- Synchronize user-defined materials in models and databases (page 1282)

Separate databases exist for embodied carbon factors, but these are described elsewhere, for details, see: .

Add user-defined materials

You can add user-defined materials from other sources or suppliers into the material databases as and when required.

WARNING If you add data to a material database, you are responsible for both its accuracy and applicability to the selected design code.

WARNING If you edit a user-defined material which is already used in multiple models, the properties might be different in each - the only way to ensure the properties are synchronised with the current database is to open each model and run Update from Database (page 1284).

Add user-defined sections

You can add user defined steel, timber, cold rolled, or cold formed sections to the material databases.

A user defined steel section can take the form of a custom, or a compound section, a compound section being one that comprises of 2 chords or more connected by battens or lattice or welded.

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**,
3. Select the unit system.
4. In the regional code list, select the region.
5. In the material list, select either steel, timber, coid rolled, or cold formed as required.
6. Click **Manage Sections**.

   The dialog box (page 1581) opens.
7. In the country list, select the database country.
8. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.
9. Click **Add...**
10. Enter values for each of the requested variables.
11. Click **OK**.

The new section size is now displayed in the **Item** pane.

Add user-defined materials

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Material** page.
3. Select the required regional code.
4. Select the required material type.
5. If the required grade is not listed for the selected regional code, click **Add...**

   The **Add Grade** dialog opens.
6. Type in the grade properties.
7. Click **OK** to return the **Materials** dialog box

   The new grade is shown in the list of available grades.
8. To make the new grade the default for the selected regional code in new models, select the grade and click **>> Default**.

Add user-defined masonry using Tedds

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Material** page.
3. Select the required regional code.
4. Select the **Masonry** material type.
5. Click **Add...**

   The **Add Masonry Grade** dialog opens.
6. Click **Define using Tedds**

   A Tedds dialog opens to allow you to select the construction details.
7. Select the material, unit, and bond type.
8. Select the unit strength.
9. Click **Dimensions...**
10. Enter the required dimensions, then click **OK**
11. Click **Mortar Details...**
12. Enter the required details, then click **OK**
13. Click **Finish**
14. Click **OK** to return the **Materials** dialog box The new grade is added to the available grades in the database.

15. (Optional) - to make the new grade the default for the selected regional code in new models, select the grade and click **>> Default**.

NOTE The properties held in the material database are always the isotropic properties for a fully grouted wall.

Add user-defined reinforcement

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Reinforcement** page.
3. Select the required regional code.
4. Select the country to assign the reinforcement to.
5. Select the required type and rib type.
6. If the required class of reinforcement is not listed for the selected regional code, click **Add...** on the right side of the **Available classes** list.

   The **New Reinforcement Class** dialog opens.
7. Type in the reinforcement class properties.
8. Click **OK** to return the **Materials** dialog box

   The new class is shown in the list of available grades.
9. To make the new class the default class for the combination of the selected regional code and country, select the class and click **>> Default**.
10. If the required size of reinforcement is not listed for the selected class, click **Add...** on the right side of the **Available sizes** list.

    The **New Reinforcement Bar Size** dialog opens.
11. Type in the properties of the first bar size.
12. Repeat steps 10 and 11 for all bar sizes that you require.
13. When you are finished, click **Close**.

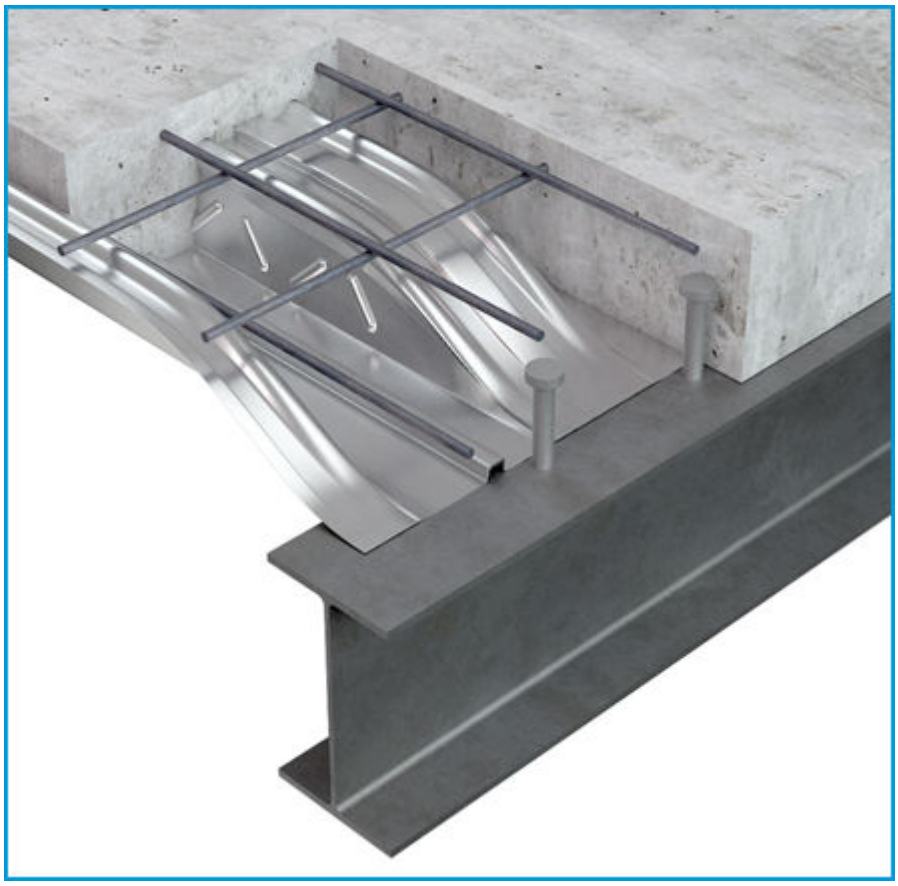
Add user-defined decking

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Decking > Metal Decking** page.
3. Select the required Country.
4. If the required profile is not listed for the selected country, click **Add...** on the right side of the **Available Profiles** list.

   The **Add Metal Profile** dialog opens.

5. Enter the profile properties and details as requested.
    a. Select the Manufacturer, or 'NO NAME' if the manufacturer is not listed.
    b. Enter the Name of the decking
    c. Select the Geometry as trapezoidal or re-entrant
    d. Select the Ends condition as closed or open. (All existing decks in the database are open, with the exception of 'Sigmat Sigdeck 100' which is available for the UK Eurocode only and is closed - as illustrated in the picture below.)



)
    e. Enter the profile details.
6. Click **OK** to return the **Materials** dialog box

    The new profile is shown in the list of available profiles.
7. To make the new profile the default for the selected country, select the profile and click **>> Default**.

8. If the required gauge is not listed for the selected profile, click **Add...** on the right side of the **Available Gauges** list.

   The **Add Metal Gauge** dialog opens.
9. Type in the properties of the first gauge.
10. Repeat steps 8 and 9 for all gauges that you require.
11. When you are finished, click **Close**.

Add user-defined precast concrete planks

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Decking > Precast Concrete Planks** page.
3. Select the required Country.
4. If the required plank is not listed for the selected country, click **Add...** on the right side of the **Available Precast Concrete Planks** list.

   The **Add Precast Concrete Planks** dialog opens.
5. Enter the plank properties as requested.
6. Click **OK** to return the **Materials** dialog box

   The new plank is shown in the list of available planks.
7. To make the new plank the default for the selected country, select the plank and click **>> Default**.
8. If the required depth is not listed for the selected plank, click **Add...** on the right side of the **Available Depths** list.
9. Type in the properties for the first plank depth.
10. Repeat steps 8 and 9 for all depths that you require.
11. When you are finished, click **Close**.

Add user-defined shear connectors

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Shear Connectors** page.
3. Select the required regional code.
4. Select the required country.
5. If the required source of connector is not listed for the selected regional code, click **Add...** on the right side of the **Available sources** list.

   The **Add Source** dialog opens.

6. Type in the name and short name.
7. Click **OK** to return the **Materials** dialog box

   The new source is shown in the list of available sources.
8. If the required diameter of connector is not listed for the selected source, click **Add...** on the right side of the **Available sizes** list.

   The **Add Connector** dialog opens.
9. Type in the properties of the first connector.
10. Repeat steps 8 and 9 for all diameters that you require.
11. When you are finished, click **OK**.

Add user-defined shear connectors

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Shear Connectors** page.
3. Select the required regional code.
4. Select the required country.
5. If the required source of connector is not listed for the selected regional code, click **Add...** on the right side of the **Available sources** list.

   The **Add Source** dialog opens.
6. Type in the name and short name.
7. Click **OK** to return the **Materials** dialog box

   The new source is shown in the list of available sources.
8. If the required diameter of connector is not listed for the selected source, click **Add...** on the right side of the **Available sizes** list.

   The **Add Connector** dialog opens.
9. Type in the properties of the first connector.
10. Repeat steps 8 and 9 for all diameters that you require.
11. When you are finished, click **OK**.

Add user-defined bolts

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Bolts** page.
3. Select the required unit system.

4. Select the required regional code.
5. If the required bolts are not listed for the selected regional code, click **Add...**.

   The **New Bolt** dialog opens.
6. Type in the properties.
7. Click **OK** to return the **Materials** dialog box

   The new bolt is shown in the list of available classes and diameters.

Add user-defined welds

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Welds** page.
3. Select the required unit system.
4. Select the required regional code.
5. If the required leg lengths are not listed for the selected regional code, enter the leg length then click **Add...**.

   The new leg length is shown in the list of available leg lengths.

NOTE If working to the United States regional code, you can add Electrodes in a similar way.

Manage design section orders

A steel or cold formed design section order is a list of sections you wish to consider when performing an autodesign.

The design process commences by starting with the first section in the chosen order file. Any section that fails any of the design conditions is rejected and the design process is then repeated for the next available section in the list.

On completion of the design process, the first satisfactory section from the section designation list is assigned to the member.

View the list of sections in a design section order

You can view the list of sections in a design section order, by following the steps below.

1. Edit the properties of the member.
2. Click the **Design section order** drop list and select **<New\Edit...>**
3. Choose a section order from the available list and then click **Edit...**

The sections contained within the chosen order file appear in the **Sections in use** list on the right of the page.

Specify that a section in the list should not be considered for design

You can control which sections in a design order list are considered as follows:

- Only checked sections within the list are considered during the design process.

  Uncheck a section and it will no longer be considered.

WARNING Limiting the choice of sections by unchecking a section within an order file is a global change that affects ALL projects, (not just the currently open one). It is typically used to eliminate unavailable or non-preferred sections from the design process. If design requirements for an individual member require section sizes to be constrained, (due to, for example depth restrictions), then the choice of sections should be limited instead by using Size Constraints, (as these only affect the current member).

Sort the listed sections by a different property

You can sort a design section order list by following the steps below.

While viewing the list of sections:

1. Click **Criteria...** to open a dialog for selecting the sort critera.
2. Select a property from the **Available Criteria** list and click **Add**.
3. Choose the order for sorting (Ascending, or Descending).
4. Add further criteria as required.
5. Click **OK** to close the dialog.
6. Click **Sort** to re-order by the chosen criteria.
7. Having sorted, if you don't want to subsequently move individual sections up or down the list, check **Keep sorted** to de-activate **Move Up** and **Move Down**.

NOTE Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

Specify that a section is non-preferred

Some sections might be more expensive or difficult to obtain; you might therefore want other sections to be chosen in preference to them, (whilst still keeping them available).

You can achieve this by moving the "non-preferred" sections further down the design order list.

To move a section up or down the list:

1. If **Keep sorted** is checked, you must uncheck it in order to activate **Move Up** and **Move Down**.
2. Highlight the section in the **Sections in use** list and then click **Move Down** or **Move Down** to promote or demote it.

NOTE Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

Reset a design section order back to the original default

If you have made changes to a design order list, the following steps you through resetting the order file to the original default values.

1. Edit the properties of the member.
2. Click the Design section order drop list and select **<New\Edit...>**
3. In the Select a Section Order dialog, highlight the section order that you want to reset.
4. Click **Reset**

The highlighted design section order is reset to its default settings.

NOTE The Reset button is only displayed for the pre-installed section orders. (User defined section orders can be deleted but not reset.)

Change the default design section order

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Section Order Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the page, click the currently displayed section for a member to change it.

   The **Select a Section Order** dialog box opens.
5. Select the required country.

   NOTE If there are no section orders listed for the selected country, they do not yet exist for the current regional code. However, you can still create one by using the **Import...** feature.

6. Highlight an available section order in the list and click **Select**.

   The section order has now been updated.

7. Repeat the steps 4–6 for other members that you want to change.

The new section order will be applied to new models, provided that the settings set that you modified is the active settings set.

Create a new design section order

If you want to create a completely new design section order you can do so as follows:

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.

2. Go to **Sections**,
3. Select the unit system.
4. In the regional code list, select the region.
5. In the material list, select either steel or cold formed as required.
6. Click **Manage Section Orders**.

   The **Select a Section Order** dialog opens.

7. In the country list, select the country.
8. In the **Select a Section Order** dialog, click **Add...**
9. Enter a unique name for the new design section order.
10. Select the the **Section Group** required.
11. Either click **Add All** to add all the available sections, or highlight just the sections you require and click **Add Selected**.

    NOTE When adding selected sections you can use the **Top**, **Bottom**, **Above Selected**, **Below Selected** options to specify where they appear in the list. However, these options are ineffective if you have a sort criteria specified and the **Keep sorted** box is checked.

12. Sort individual sections in the **Sections in use** list, using **Move Up** or **Move Down**, as required.
13. When the list of sections in use is as you want it, click **OK**

The new design section order appears on the list of available section orders.

Import a new design section order

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.

2. Go to the **Section Order Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the page, click the currently displayed section for a member to change it.

   The **Select a Section Order** dialog box opens.
5. Click **Export...** and save the section order as an XML file.
6. Close the **Select a Section Order** dialog box.
7. On the **Section Order Defaults** page, select the settings set that was originally being modified.
8. In the table, click the section order that you want to change.
9. Select the required country.
10. Click **Import...** and select the XML file that you just created.
11. Select the imported file by clicking **Select**.
12. Repeat steps 4–11 for other member types as needed.

Add simple connection resistances

The materials database contains pre-defined connection types with pre-defined resistances for steel beams to Eurocode[1] and US[2] regional codes.

In addition, you can specify user-defined connection types and user-defined resistances for any regional code and save them to the section database. Once defined, these can then be used to across all projects.

[1]The Eurocode resistances are derived from SCI publication P358 and make use of UK NA values for partial safety factors, regardless of the Eurocode Country setting.

[2] The US resistances are derived from AISC publication Steel Construction Manual 14th Ed and are only given when US Customary units are selected. Separate resistances are given for LRFD and ASD.

Pre-defined connection types and resistances

For Eurocodes:

With the Member Type set to Simple Beam, three *pre-defined* connection types are available for S355 and S275 grades:

- Fin Plates,
- Full Depth End Plates
- Partial Depth End Plates

These have connection resistance values defined for UB and UKB section beams. The bolts considered are all size M20, of property class 8.8, and

ordinary or flowdrill type. The plates considered are all S275 grade. Fin Plates and Partial Depth End Plates have resistances defined for beams with 0, 1 or 2 notches, while the resistances for Full Depth End Plates assume no notches. Fin Plates have resistances for 1 or 2 vertical lines of bolts.

For US (ACI/AISC) and US Customary units:

With the Member Type set to Simple Beam, and the steel grade set to 'Any' one *pre-defined* connection type is available:

- Single Plate

Single Plates have connection resistance values defined for W section beams of any Fy value. The bolts considered are sizes 3/4 in, 7/8 in and 1 in, from Group A, with thread condition N, in standard holes. The plates considered are all of Fy 36 ksi, with thickness ranging from 1/4 in to 9/16 in. Single Plates have resistances defined for beams with no coping and top flange coping only. All resistances assume 1 vertical line of bolts only.

In addition, two further Simple Beam connection types are available for the 50 ksi grade:

- All-Bolted Dbl Angle,
- Shear End-Plate

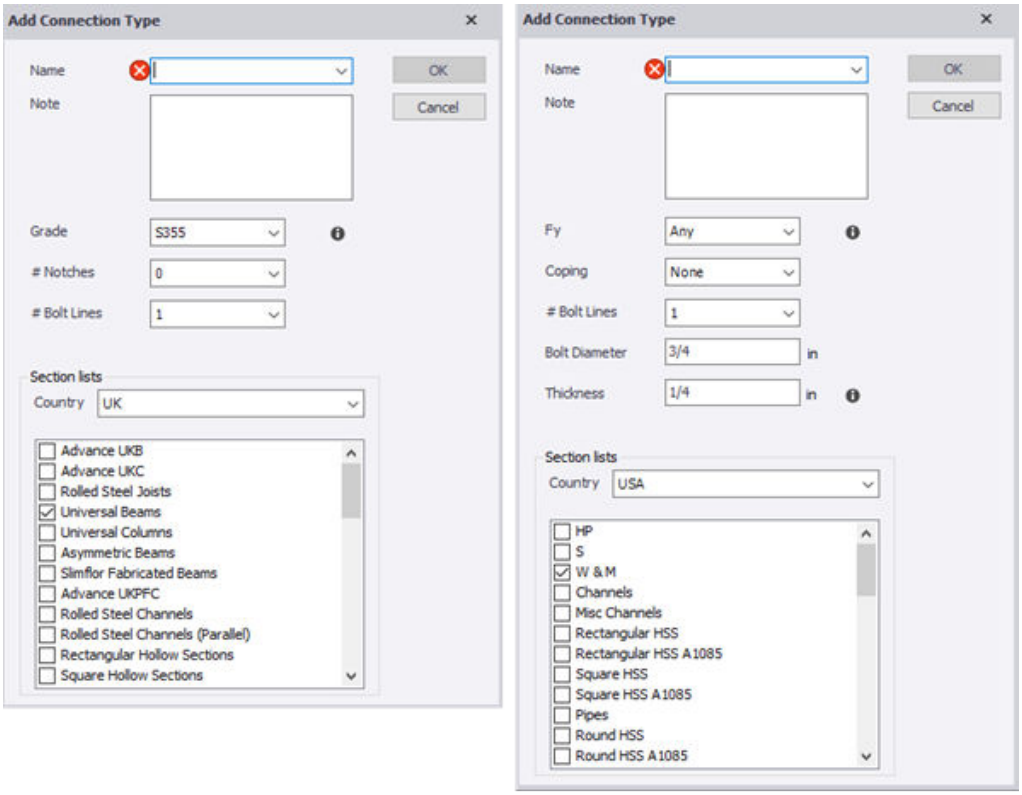
These have connection resistance values defined for W section beams, and for Single Plates with W section beams of F_y 50 ksi. The bolts considered are sizes 3/4 in, 7/8 in and 1 in, from Group A, with thread condition N, in standard holes. The plates and angles considered are all of F_y 36 ksi, with thickness ranging from 1/4 in to 9/16 in. All-Bolted Double Angles have resistances defined for beams with no coping and top flange coping only, while the resistances for Shear End-Plates assume no coping. All resistances assume 1 vertical line of bolts only.

NOTE The pre-defined connection types and resistances can be reviewed in the Connection Resistance dialog box (page 1533).

Add user-defined connection types

1. Open the Connection Resistance dialog box (page 1533)
2. Under **Connection Types**, click **Add...**

An **Add Connection Type** dialog appropriate to the current regional code is displayed.



3. In the Name box enter a name for the Connection Type.

   If you want to create a Connection Type with a number of variations of Grade, Notches or Bolt Lines then it is important to enter exactly the same Name each time - e.g. the database will recognise “CT1” and “CT 1” as separate Types simply because of the space in the second name. This has an effect when viewing results in tabular data or reports, where Tekla Structural Designer carries out an optimisation process to find the first passing resistance based on the name of the Connection Type and the number of Bolt Lines and Bolt Rows assigned to that name

   NOTE It is possible to use the same Name as one of the pre-defined Types and it will be recognised in the optimisation process as essentially being the same Connection Type

4. In the Note box enter any descriptive text you would like to see displayed in the dialog’s Info box.

5. Select a Grade or, with US codes, an F_y value.

   Note that *subgrades* like S355 J0 are ignored e.g. a Connection Type with S355 Grade assigned will have its resistances applied to beams or braces of Grade S355 J0, S355 J2, etc.

6. Select # Notches and # Bolt Lines or, with US codes, select Coping, # Bolt Lines, Bolt Diameter and Thickness (of plate or angle leg).
7. Select the section lists that you will want to define resistances for (this list can be added to later on if required, in the Edit Connection Type dialog).
8. Click **OK**

NOTE **OK** is not active until a Name has been defined. After clicking OK 'Hide undefined values' becomes automatically un-ticked and resistance values can be entered against the required sections. If no resistance values are assigned at this stage for at least one section in a selected Section List, then that List will become de-selected and have to be re-selected later on.

You are now able to **add user-defined connection resistances to the database** against the required sections for this connection type.

Edit user-defined connection types

1. Open the Connection Resistance dialog box (page 1533)
2. Under **Connection Types**, click **Edit...**

   An **Edit Connection Type** dialog appropriate to the current regional code is displayed.
3. Edit the values as required.
4. Click **OK**

Add user-defined connection resistances

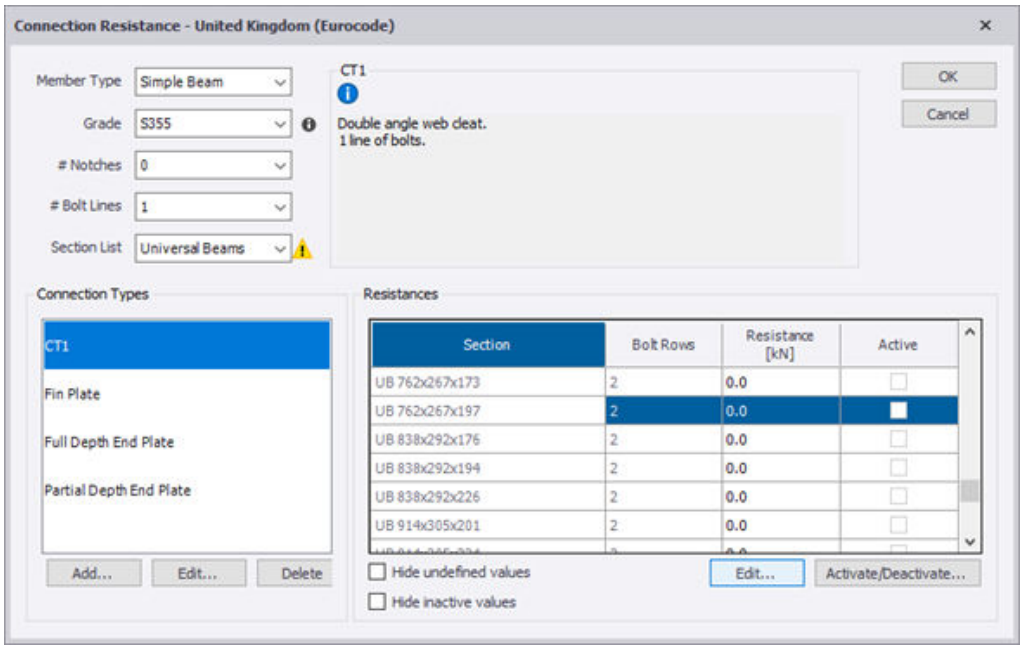
User-defined connection resistances can be added to the database for existing connection types as follows:

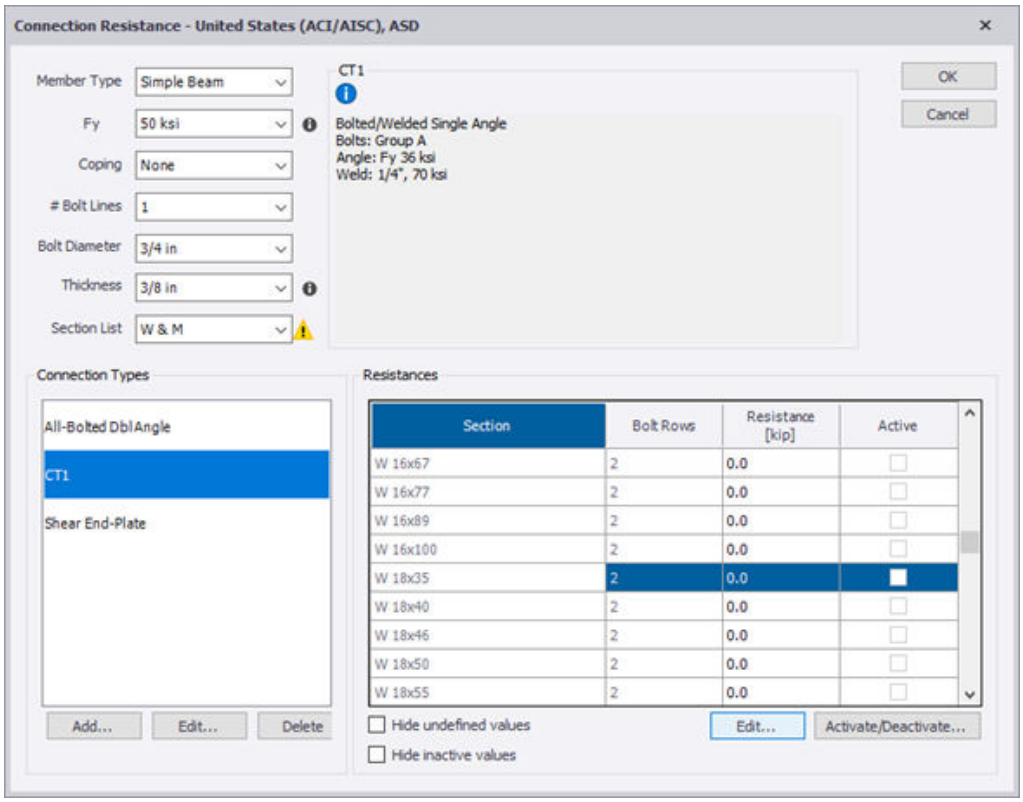
1. If it is not already displayed, open the Connection Resistance dialog box (page 1533)
2. If necessary, use the filters at the top of the dialog until the required connection is displayed in the Connection Types list.
3. Select the connection in the Connection Types list. The sections are listed in the Resistances table, along with the bolt rows and resistance values for the selected connection type.

NOTE If no sections are listed uncheck **Hide undefined values**.

---

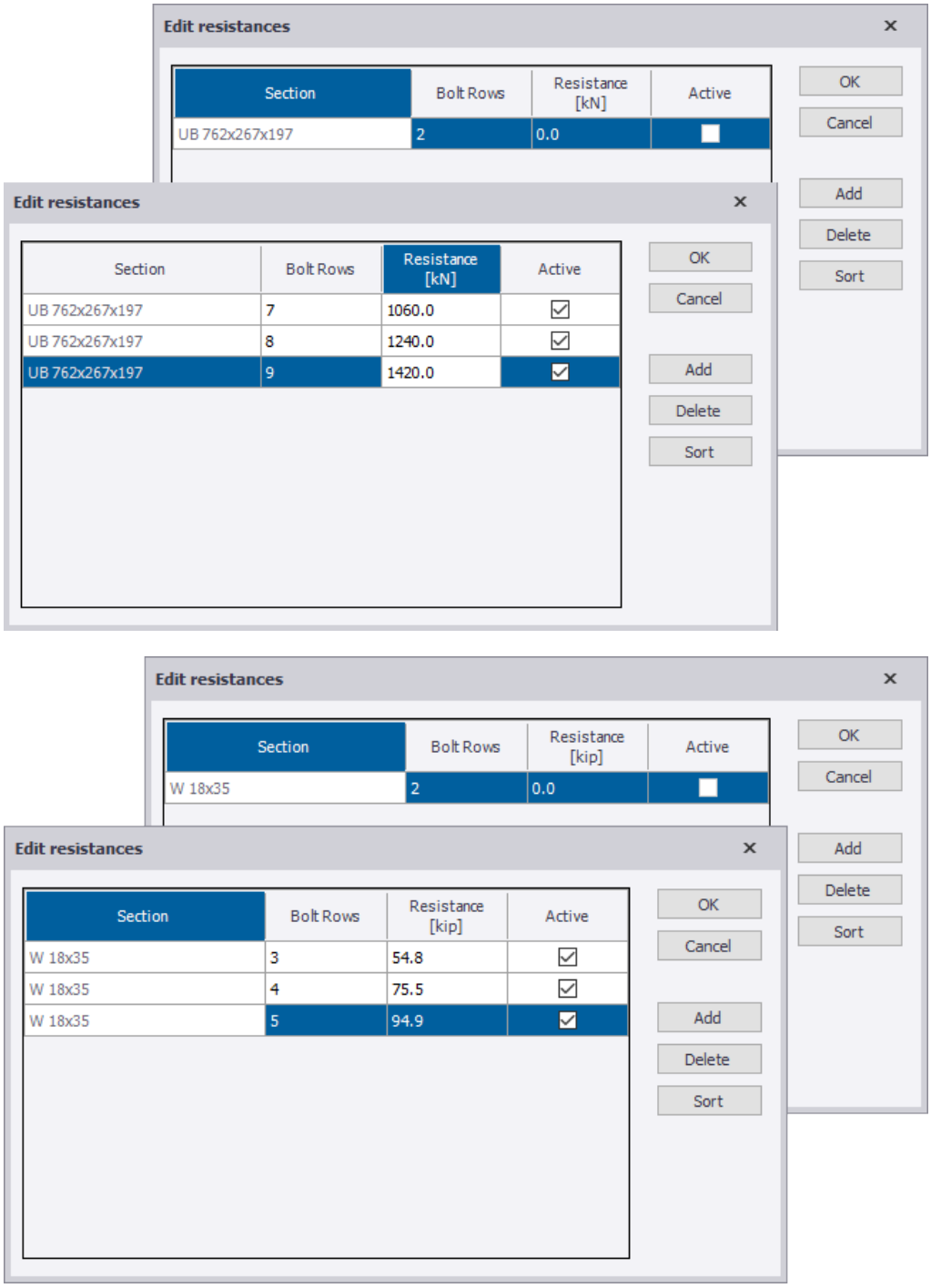
4. Select the section type in the Section List.





5. Resistance values can be entered directly in the table. Alternatively, select a particular section in the table and click **Edit...**. This displays the Edit resistances dialog which allows you to adjust the bolt rows count or to **Add** multiples of the same section size with more than one bolt row count. The bolt row counts can be entered in any order and then use **Sort**

to number from lowest to highest. Use the Edit resistances dialog also to **Delete** data rows if no longer required in your database.



NOTE Connection resistances are made active as the values are entered.

Related video

Predefined connection resistance database for Eurocode and AISC

Upgrade material databases

The installed material databases each have their own database version. You can see the version number separately on each page of the **Materials** dialog box.

The original data in each of the material databases is system data, and it cannot be edited. The version number relates specifically to this system data.

From time to time, new system data may become available through a main release/service pack. When this happens, your local databases will automatically be updated. This process works as follows:

- Providing all databases can be updated automatically without user intervention, when a newly installed release/service pack is first run then the updates are applied and a message is displayed stating "updating to latest database versions". On successful completion the message closes and the program will open.

NOTE When a database version is updated this does not cause you to lose any user-defined material data you may have added to the old version, because the user-defined data is automatically copied back in to the new database.

In certain circumstances, (for example when a model is opened on another computer with a different database version), an inconsistency can arise between the material data in the model and the installed database. The inconsistencies can be identified and resolved on the **Model** page of the **Materials** dialog box, either by updating the model data, or by updating the material databases. For more details, see: Synchronize user-defined materials in models and databases (page 1282)

Synchronize user-defined materials in models and databases

When you define an object, its material properties are initially read from the appropriate material database. Subsequently, the material properties are held separately within the model itself. This means that you can open Tekla Structural Designer on a second computer and run the model, even if the model contains user-defined material properties that don't exist in the material database on the second computer.

In the above scenario, you are not actually required to add the missing properties to the database on the second computer. It may however be

beneficial to do so if you anticipate that you will need to re-use the property in question in new models - see **Add to Database** below for further details.

A second scenario is potentially more problematic. This can occur for example if a user-defined material is applied to a model and a property of the material is subsequently edited in the database after the model has been closed. Because the material properties are held separately within the model, it will continue to use the old property values. In such case, we strongly suggest to manually synchronise the model's user-defined material with the database - see **Update from Database** below for further details.

WARNING Once you edit a user-defined material which is already used in multiple models, the properties might be different in each - the only way to ensure the properties are synchronised with the current database is to open each model and run **Update from Database**.

Add to Database

Because model properties and database properties are held separately, situations can arise where a model that was initially created on another computer is using properties that don't yet exist in the database when it is opened on a different computer.

Consider the following scenario:

1. A user-defined material is added to the Materials database.
2. The user-defined material is applied to a model.
3. The model is saved and the program closed.
4. The same model is reopened on a different computer.

In this situation the properties exist in the model but don’t yet exist in the database on the new computer. Because the properties in the model and the database are kept separate this doesn’t cause a problem - when the model is analyzed the user-defined material properties are available.

If you think you will require the same user-defined material for future models, you can run **Add to Database** as follows:

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Model** page.
3. To locate only those materials that don't yet exist in the database, select **Show only objects not saved in the database**.
4. Select the desired objects in the **Material data objects in model** list.
5. Click **Add to Database**

The selected material data objects are added to the appropriate database.

NOTE You can only use **Add to Database** to add new properties to the database, you can't use it to update an existing object of the same name.

Update from Database

Because model properties and database properties are held separately, situations can arise where the model is using different properties to those currently in the database. The following scenario provides an example of how this can occur:

- A user-defined material is added to the Materials database.
- The user-defined material is applied to a model.
- The model is saved as the program is closed.
- The program is re-started (without reopening the same model).
- The properties of the user-defined material are edited in the Materials database.
- The model is reopened.

NOTE There would not have been a problem if the model had been kept open when the properties in the database were being edited - the issue only arose because the program was restarted and the model was not re-opened before the database edit occured.

WARNING Once you edit a user-defined material which is already used in multiple models, the properties might be different in each - the only way to ensure the properties are synchronised with the current database is to open each model and run **Update from Database**.

To force a model to adopt the properties in the database, you must run **Update from Database** as follows:

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Model** page.
3. Select the desired objects in the **Material data objects in model** list.
4. Click **Update from Database**

The selected material data objects in the model are updated from the appropriate database.

9.3 Manage properties and property sets

When you create a new entity, default properties for it are displayed in the **Properties** window, these can be edited as required before the entity is placed.

Provided the properties are saved to a **property set**, they can be applied to similar entities elsewhere in the current model.

Property sets can also be transferred between models.

In this section we show you how to:

- Save properties to and recall properties from property sets (page 1285)
- Apply property sets to existing entities (page 1286)
- Review where property sets have been applied (page 1287)
- Transfer property sets between models (page 1288)
- Delete property sets (page 1288)

Related information

See also

Modify the properties of entities or model objects (page 111)

Related video

The use of property sets

Save properties to and recall properties from property sets

Saving properties to property sets allows you to re-use them elsewhere.

Save properties from the Properties window to a new property set

NOTE You can only save properties in the **Properties** window to a property set when there are no entities selected. This ensures that each property in the property set has an unique entry.

1. From the **Model**, **Design**, or **Foundations** ribbon tab, click the entity type whose properties you want to save as a property set.

   The list on the top of the **Properties** window now reads **<unsaved set>**.
2. Specify the properties according to your needs.
3. Click the **Save...** button on the top right corner of the **Properties** window.

   The **Add property set** dialog box opens.
4. Name the property set.

5. Click **OK**.

Save the properties of an existing entity to a named property set

1. Hover the mouse pointer over the desired entity until it becomes highlighted.
2. Right-click the entity.
3. In the context menu select **Create property set...** (and if the entity is a beam or column or wall, select the span or stack required).

   The **Add property set** dialog box opens.
4. Name the property set.
5. Click **OK**.

Recall a previously saved property set to the Properties Window

Provided that you have previously saved a property set, you can recall it again later in the **Properties** window, if it is applicable to the current command.

1. From the **Model**, **Design**, or **Foundations** ribbon tab, click the desired entity type.

   The properties applicable to the selected entity type are displayed in the **Properties** window.
2. Click the list on the top of the **Properties** window.
3. In the list, select the desired property set.

Apply property sets to existing entities

You can apply a property set to existing entities in a **Structural View**, but if you want to make a lot of changes property sets can be applied more easily from a **Review View**.

Apply a property set to an individual entity in a Structural View

1. Hover the mouse pointer over the entity until it becomes highlighted.
2. Right-click the entity.
3. In the context menu, select **Apply Property Set...**

   The **Select property set to apply** dialog box opens.
4. Select the desired property set.

5. Click **OK**.

The property set is only applied to the individual entity that you right-click, even if multiple entities were selected. To apply a property set to multiple entities, see the following topic.

Apply a property set to multiple members in a Structural View

1. Select the entities that you want to update.
2. Ensure that the list on top of the **Properties** window shows the correct entity type.
3. On the top right corner of the **Properties** window, click **Apply...**
4. Select the desired property set.
5. Click **OK**.

The property set is now applied to all of the selected entities, provided that they are of the same type.

Apply a property set in a Review View

1. Open a **Review View**
2. On the **Review** tab, click **Property Sets**.
3. In the **Properties** window, do all of the following:
   a. Set **Review/Update** to **Apply property set**.
   b. Select the entity type.

      NOTE Only those entity types that already have property sets saved will be listed.

   c. If the **Member** entity type was selected, you will also need to select the characteristic.
   d. Select the property set that you want to apply.
4. In the **Review View**, click the entities to which you want to apply the property set.

Review where property sets have been applied

To graphically review where property sets have been applied, see the following instructions.

1. Open a **Review View**
2. On the **Review** tab, click **Property Sets**.
3. In the **Properties** window, set **Review/Update** as **Review All**.

4. Still in the **Properties** window, select the entity type that you want to review.

   NOTE Only those entity types that already have property sets saved will be listed.

   Entities of the selected type are color-coded to indicate where property sets have been applied.

Transfer property sets between models

Once property sets have been saved, they can be exported from the current model and then imported into another Tekla Structural Designer model.

Export property sets

1. On the **Home** tab, click  **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to export.
3. Click **Export...**
4. Save the properties as a .tsps file.

Import property sets

1. On the **Home** tab, click  **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to import.
3. Click **Import...**
4. Select the .tsps file that contains the required property sets.
5. Click **Open**.

   The imported property sets are now listed in the **Manage Property Sets** dialog.

Delete property sets

1. On the **Home** tab, click  **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to delete.
3. Click **Delete**.
4. Click **OK**.

9.4 Manage project revisions and view metrics

The **Project Wiki** allows you to view and modify the project summary, view revision history, record revisions and view project metrics.

In the **Project Wiki** dialog box, you can:

- Manage the parameters that are included in the output reports on the **Project Summary** page.
- Track revision history on the **Revisions** page.
- View the time at which each revision was started and last saved on the **Sessions** page.
- View the changes associated with each revision on the **Changes** page.
- View statistics related to the model size on the **Metrics** page and its subpages.
- Manually record notes, to dos etc. on the **Model Info** page.

View and modify project summary

You set the initial project summary as you create a new project. If you want to change the project summary details later, see the following instructions.

1. On the **Home** tab, click  **Project Wiki**.

   The **Project Wiki** dialog box opens.
2. On the **Project Summary** page, modify the project details according to your needs.
3. Click **OK**.

Record revisions

1. Save the project before recording revisions.
2. In the file menu, click **Start Revision**.

   The **Start Revision** dialog box opens.
3. Type the revision ID and add additional notes related to the revision according to your needs.
4. In order to keep a record of the changes made in the revision, select the **Track Changes** option.
5. Click **OK**.
6. Continue to develop the model and save the model under a new name.
7. Repeat steps 2–5 as required.

View revisions

If you have have opted to record revisions you can view the information for each revision in the **Project Wiki** as follows:

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.
2. Select **Revisions**. The changes associated with each revision ID are displayed.

Apply revision ID as an attribute for each modified element

1. On the **Home** tab, click **Project Wiki**.
2. Select **Changes**.
3. Select an appropriate UDA from the Revision Attribute droplist.
4. Either:
   a. Click **Apply automatically** in order for the revision ID to be automatically recorded against the selected UDA on each change, or
   b. Click **Apply** to manually record the revision ID against the selected UDA for the currently elements listed in the dialog.
5. Click **OK**.

View sessions

1. On the **Home** tab, click **Project Wiki**.
2. Select **Sessions** The dates and times of each modeling session are displayed, with the name of the user and the program version. Any model imports are also recorded.

View changes

Changes are recorded for each model revision, provided that when you start the revision you select the **Track Changes** option.

1. On the **Home** tab, click **Project Wiki**.
2. Select **Changes**
3. Select the Revision version. Each changed element is listed, with the type of change, the previous, and new value.

NOTE If required you can add the **Revision ID** as a UDA to each element that has been changed in the current revision. For details, see the **Apply revision ID as an attribute for each modified element** section above.

View metrics

1. On the **Home** tab, click **Project Wiki**.
2. Select **Metrics**.
3. Select an individual metric. The total numbers of objects in the model for the selected metric is displayed.

View and modify model notes, assumptions etc.

You can manually add assumptions, status, to dos and notes via the **Model Info** page in Project Wiki.

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.
2. On the **Model Info** page, enter the information you want to record according to your needs.
3. Click **OK**.

9.5 Create and manage user-defined attributes

You can define user-defined attributes (UDAs) to save miscellaneous data to files, individual members and panels.

User-defined attributes are flexible, and you can use them for a variety of purposes. For example, you can:

- Apply descriptive labels, such as construction phases.
- Record paint specifications.
- Attach office documents, pictures, or other associated files.
- Link to design files from other applications.
- Filter material lists and member design reports for specific attributes.

UDA definitions and values

Example UDA definitions are included in the default settings sets. You can easily modify the definitions according to your needs.

(page 1257) allows you modify the definitions that apply to the current model. From there you can also set UDAs to be **linked** so that the same UDA value gets assigned to all duplicate levels.

You assign specific UDA values to an entity via the **Properties** window. Once a UDA value exists, it can also be assigned to or removed from members graphically by using the **Review View**. For further details, see: Attach UDA values to members and panels (page 1294)

Attaching files as attributes

When a file is attached as an attribute, you can embed it within the Tekla Structural Designer file. When embedded, the attached file is included when the model is transferred to another computer.

NOTE The embedded file only gets attached to the Tekla Structural Designer file when the model is saved using **Save** or **Save As**.

This means that the embedded files are not attached when you use **Save Model Only**. Similarly, if you have to revert to an autosaved version of the model, the embedded files will not be attached.

Transferring attributes

Attributes are transferred when models are exported to Tekla Structures and Revit. In the current release, attributes are not yet included for Tekla Structural Designer drawings.

See also

User-defined attribute settings (page 1257)

Create attribute definitions (page 1293)
Attach UDA values to members and panels (page 1294)
Apply attribute filters to material lists and reports (page 1295)

Create attribute definitions

You can create and modify attribute definitions according to your needs. The attributes are defined either in the **Model Settings** dialog box or in the **Settings** dialog box, depending on whether you want the to the attributes be applied to the current model or new models.

Create attribute definitions in the current model

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog opens.
2. Go to the **User Defined Attributes** page.
3. Click **Add**.
4. Define the following properties:
   - Name
   - Type
   - Source
   - Values (value choices that apply when the source is value list)

NOTE The order in which UDAs are listed in the replicates the order in which the UDAs are listed in the **Model Settings** dialog.

Use the **Move Up** and **Move Down** buttons to reorder the UDAs.

Create attribute definitions for new models

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set to which you want the attributes to apply.
4. Go to the **User Defined Attributes** page.
5. Define the following properties:
   - Name

- Type
- Source
- Values (value choices that apply when the source is value list)

NOTE The order in which UDAs are listed in the replicates the order in which the UDAs are listed in the **Model Settings** dialog.

Use the **Move Up** and **Move Down** buttons to reorder the UDAs.

Attach UDA values to members and panels

You can attach office documents, pictures, or other files to the members and panels of your model. When you attach files to members and panels, they are embedded in the file. The embedded files will also be included in the model if the model is transferred to another computer.

See also

Create attribute definitions (page 1293)

Attach a UDA value using the Properties Window

1. Select the members or panels to which you want to attach attributes.

   NOTE If the selection contains different member types, select the first member type in the list at the top of the **Properties** window.

2. In the **Properties** window, under **UDA**, define the value for each attribute that you want to attach.
3. If the attribute is a file, select the **Embedded** option to save the file inside the model.

   Only embedded files are automatically transferred when the model is copied to another computer.

   TIP For multi-span beams and multi-stack columns or walls, you can attach separate UDAs to individual spans or stacks.

4. Repeat steps 1-3 for additional member types.

Attach an existing UDA value in the Review View

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Set On** or **Toggle**.
3. In the **Properties** window, select the attribute.

4. In the **Properties** window, select a value for the selected attribute.

NOTE You can use some additional settings in the **Properties** window to control which members and panels are displayed in the **Review View**.

- Under **Filter**, you can use the **Show** and **Entity type** to select which attribute values and entities are displayed.
- Select the **Color for other values** box to display members that have an other value of the selected attribute.

5. Click the desired members to add the UDA value.

TIP To remove the UDA value:

- If the **[M]ode** is set to **Set On**, change it to **Set Off**.
- If the **[M]ode** is set to **Toggle**, click the selected members again.

Graphically review the attached UDA values

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Review**.
3. In the **Properties** window, select the attribute that you want to review.

   The members or panels are color-coded to represent the different values of the selected attribute.

Open a file that has been attached as a UDA

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Review**.
3. In the **Properties** window, select the attribute that you want to review.
4. Click the member or panel to which the file is attached.

   Provided that the extension has been associated with an appropriate application, the application opens and displays the file.

Apply attribute filters to material lists and reports

User-defined attributes (UDAs) allow you to filter material lists and member design reports for specific attributes. For more detailed information, see the following paragraphs.

See also

Create attribute definitions (page 1293)

Attach UDA values to members and panels (page 1294)

Material list settings (page 1253)

Apply an attribute filter to material list review data

1. On the **Review** tab, click **Tabular Data**.
2. In the **View Type** list, select **Material List**.
3. In the **Filter** list, select **Selected UDAs**.

   The **Select filter items** dialog box opens.
4. Select the required attribute values.
5. Click **OK**.

Apply an attribute filter to a report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the report.
3. In the **Report Structure** list, right-click the chapter or sub-heading that you want to filter.
4. In the context menu, select **Model Filter** --> **Edit\New...**

   The **Select filter** dialog box opens.
5. Click **Add**.
6. In the **Filter properties** list, select **Selected UDAS**.
7. In the **Selected items** list, select the attribute values.
8. Click **OK**.

9.6 Create and manage sub structures

Sub structures are saved user-defined selection groups of any objects which can be of any size and mix of object types, materials etc. They can be viewed in isolation and used as filters for many areas of program operation. Using sub structures has multiple productivity benefits throughout the Tekla Structural Designer workflow, from model organization and editing through to results review, design and output.

NOTE In Tekla Structural Designer the term *sub structure* is specifically used to refer to a collection of elements, either above or below ground level. It is **not** being used to refer to that part of the structure below ground level (e.g. basements and foundations).

Sub structure characteristics

The basic characteristics of Tekla Structural Designer sub structures are:

- Elements can be a part of more than one sub structure.
- Not every element has to be in a sub structure.
- Deleting a sub structure does not delete the elements within it.
- Sub structures are distinct from, and should not be confused with sub models (page 616).
- Sub structures themselves are never analyzed.
- Sub structure groups are simply collections of other sub structures.

Create a sub structure

1. **To create a sub structure from a Structural View**

1. In the scene view, click or box around the members that you want to include in the sub structure.
2. Right click and from the context menu select **Add to Sub Structure...**
3. In the left hand pane of the dialog, all the selected objects are listed with on/off check boxes
    a. Uncheck any objects that you don't want to be included in the sub structure.
4. In the left hand pane of the dialog, choose an existing or new substructure, and if new, enter the sub structure name.
5. Click **OK**

2. **To create a sub structure from the Project Workspace**

6. In the **Project Workspace** open the **Structure** tree.
7. Expand the **Sub Structures** branch.
8. Right-click the **Sub Structures** sub branch, then in the context menu, select **Create Sub Structure**.
9. Review the sub structure properties in the **Properties** window.
10. In the **Name** field, type the name of the sub structure.

11. Select the color of the sub structure.
12. In the 3D view, click or box around the members that you want to include in the sub structure.
13. Press **[Esc]** when complete.

3. **To create a sub structure from a Review View**

14. On the status bar at the bottom of the window, click  **Review View**.

15. On the **Review** tab, click  **Sub Structures**.
16. Go to the **Properties** window.
17. Set **Review/Update** to **Update Selected**.
18. Set **Update Sub Structure** to **- - New - -**.
19. In the **Name** field, type the name of the sub structure.
20. Select the color of the sub structure.
21. In the review view, click or box around the members that you want to include in the sub structure.
22. Press **[Esc]** when complete.

Edit a sub structure

1. **To edit a sub structure from the Project Workspace**

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to add to, or remove from.
4. in the context menu, select **Edit**.
5. Skip to step 7 below.

2. **To edit a sub structure from a Review View**

6. On the status bar at the bottom of the window, click  **Review View**.

7. On the **Review** tab, click  **Sub Structures**.
8. Go to the **Properties** window.
9. Set **Review/Update** to **Update Selected**.
10. In **Update Sub Structure**, select the sub structure that you want to modify.

11. In **Selection Mode**, select how you want to modify the structure.
12. In the review view, click or box around the members that you want to modify.

Delete a sub structure

NOTE Deleting a sub structure only removes the association between the selected group of members - the members themselves are not deleted.

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to delete.
4. in the context menu, select **Delete**.

Rename a sub structure

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to rename.
4. in the context menu, select **Rename**.
5. Enter the new name as required.

Review sub structures

1. On the status bar at the bottom of the window, click  **Review View**.
2. On the **Review** tab, click  **Sub Structures**.
3. Go to the **Properties** window.
4. Set **Review/Update** to **Review All**.

   Tekla Structural Designer displays each sub structure in a different color.

Create a sub structure group

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch.
3. Right-click the **Sub Structure Groups** sub branch.
4. In the context menu, select **Create Sub Structure Group**.
5. Select the sub structures to be included in the sub structure group.
6. In the **Properties** window.
   a. In the **User name** field, type the name of the sub structure group.
   b. Select the color of the sub structure group.

Open a 3D view of a sub structure

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch.
3. Double-click the sub structure.

   Tekla Structural Designer opens a 3D view of the selected sub structure.

   TIP To open a solver view of the sub structure, right-click the sub structure, and select **Open solver view**.

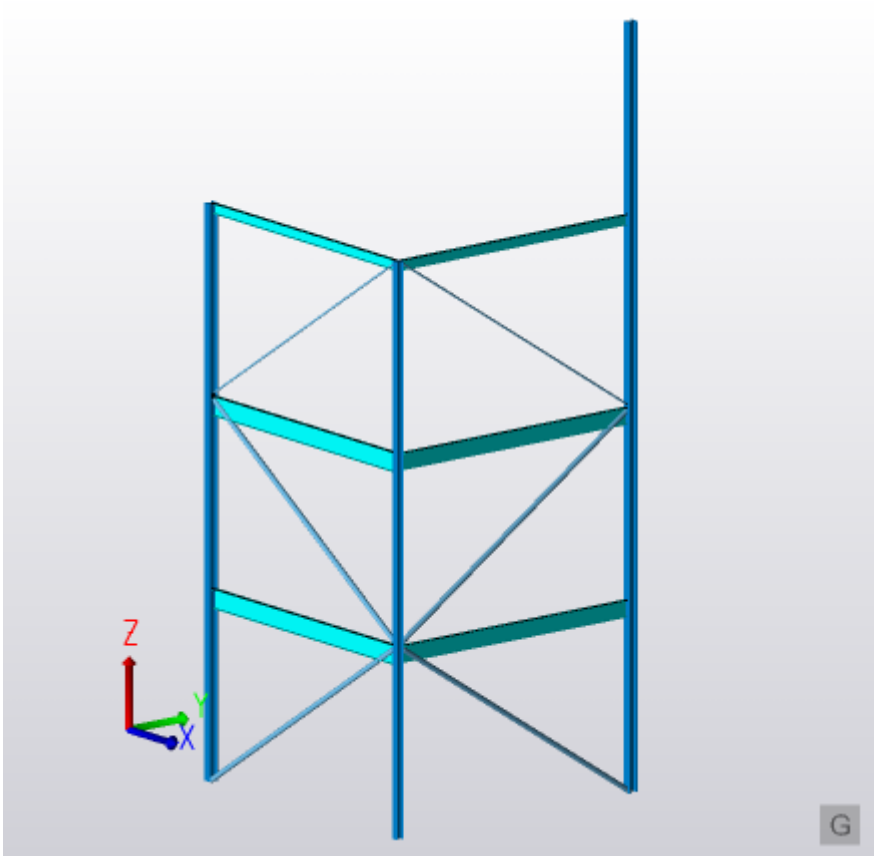
4. Once the view has opened, a G (Ghosted) button is displayed in the bottom right corner. Clicking this button toggles the display to either show just the sub structure, or the sub structure with a 'Ghosted On' view of the rest of the model.

Use Ghosted to see the view in the context of the whole model

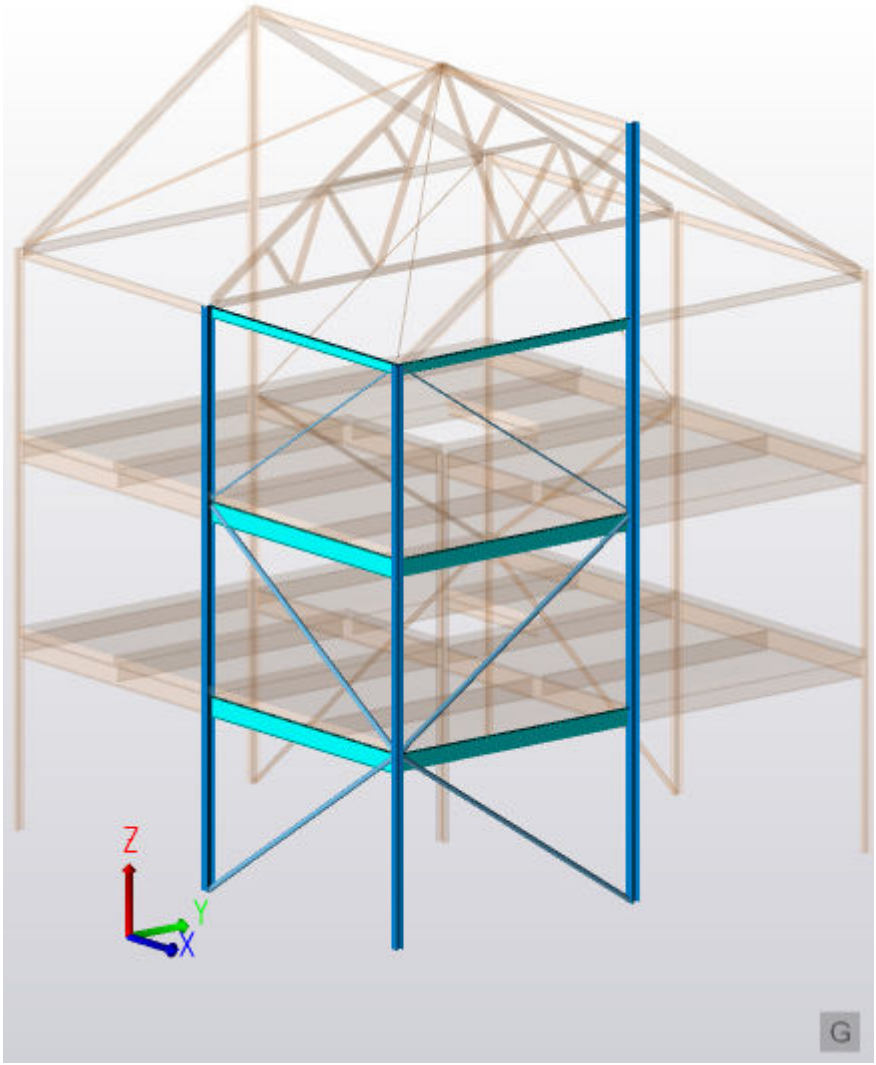
You can toggle **Ghosted** on and off using the G button.

NOTE The G **Ghosted** button is displayed in in the bottom corner of Sub Structure and Sub-model Views. It is also displayed in Level, Frame, and Slope Views when they have been toggled into 3D (via the 2D/3D toggle button).

- **Ghosted off**: the view is displayed in isolation.



- **Ghosted on**: the view is displayed in the context of the whole structure.



Related video

Ghosted Structure view

Related information

Related videos

How do I use Sub structures?

Sub Structure enhancements

See also

Check all members and walls in a sub structure (page 792)

Design all slab items in a sub structure (page 809)

Check all slab items in a sub structure (page 807)

9.7 Working with large models

Although bigger models will typically need more RAM, you should note that it is the volume of model data and results data created during analysis and design that generates the demand for RAM, so by controlling the volume of data you are able to influence the speed of solution.

Your modelling and design choices can affect the volume of data produced; some of the more significant of these choices are discussed below:

Don't mesh concrete slabs in 3D Analysis

Meshing is not necessary in 3D Analysis for traditional Beam and Slab models as these can use FE decomposition instead.

It is also not necessarily required for Flat Slab models, however, it does become required if you have a transfer slab. It is not critical unless the slab is part of the lateral resistance system.

By default the Mesh 2-way slabs in 3D analysis option that controls this is not checked at any level; you should only check it at a given level when you have good reason to do so.

Using coarser shell meshing in large models

You should review and consider adjusting the mesh parameters as the defaults can often be conservative.

- Using coarser mesh parameters has no impact on sway or modal frequencies.
- If you are not concentrating on slab design you can use very coarse meshes.

Ultimately it is your responsibility to be comfortable with the level of mesh refinement applied to the model. However we would definitely recommend using a courser mesh during design development and then perhaps consider refining a bit more at final design stage.

Using coarser semi-rigid diaphragm meshing in large models

If you have employed semi-rigid diaphragms and are experiencing performance issues you should review and consider adjusting the semi-rigid mesh parameters.

It has been observed that using refined semi-rigid meshes generally has negligible impact on results and only has the effect of slowing down analysis and increasing memory requirements. It is suggested that the default mesh parameters may be more refined that is actually necessary.

Limit the number of loadcases and combinations

You can control the number of combinations created when running the Combination Generator.

In particular you should consider limiting the number of wind loadcases and combinations.

Also:

- Don't add wind loading during the initial design development.
- Don't activate pattern loadcases and combinations until you need to, probably only at the final design stage.

Alternative design approach for large models

For large models, rather than running Design All you may be able to save time by running Analyse All instead and then run a selective design, such as:

- Design by level
- Design by frame
- Design by group
- Design by sub-structure
- Design by member

Effective use of Auto Design

Although for the first design run you might choose to use select bars starting from Minima, on subsequent runs it is generally more efficient to use select bars starting from Current; this will run a check on the current steel provision and if inadequate, it will automatically re-select new steel bars to pass the design.

Check Design can also be very effective - you can turn off the Autodesign and then manually deal with any fails.

Design members for FE chasedown analysis results

This is set in the design options for concrete and by default it is checked on for beams, columns and walls. However in a traditional Beam and Slab model it may not actually be necessary; it is generally not critical unless you have some unusual transfer level challenge. In large models you should therefore consider unchecking it.

Re-design columns (or beams) using previous analysis results

If you change the size of a member you can try out its design without being forced to re-analyse.

Basically you can make any edit you want that does not change the number of stacks in a column, (or spans in a beam); although the analysis results will be marked as out of date, you can still do a design for the changed member based on the old analysis results.

So using the list below as examples design can still be done in all the cases noted:

1. Changing column (or beam) size but retaining shape - designs ok (but see 3 and 4).

2. Changing column (or beam) shape - designs ok (but see 3 and 4).
3. Making column (or beam) smaller so that previously attached members no longer attach - this changes number of stacks - design beyond scope.
4. Making column (or beam) larger so that previously un-attached members now attach - this changes number of stacks (spans) - design beyond scope.
5. Adding / editing / deleting beams that attach to a column - design remains possible up to the point that it affects number of stacks - OK
6. Adding / editing / deleting flat slabs that attach to a column - design remains possible up to the point that it affects number of stacks - OK
7. Adding / deleting stacks or levels (or editing level properties) - Adjusting Levels designs ok because number of stacks is the same. If you add or remove stacks then design is beyond scope.

Model organisation

Tekla Structural Designer has a number of features for organising the model than can each be used to increase efficiency:

- Grouping - one design is applied to all members in the group.
- Sub-structures - allow you to focus on specific areas of interest.
- Duplicate levels - generally save modelling time and reduce the volume of data.

When using duplicate levels, you can achieve further efficiency by designing slabs for a fine mesh at one level only, and then check the slabs at duplicates of the level using a courser mesh result.

NOTE Because meshing parameters are "sub model", rather than "level" based, to achieve this you would set coarse mesh parameters in the structure settings but then override them for an individual sub model.

Our general advice for duplicate levels is:

- For preliminary design set a coarse mesh for entire structure
- For final design where there are a lot of duplicate levels
  - possibly refine the mesh used for the entire structure a little
  - but for each set of duplicate levels, select one and adjust the relevant sub-model parameters to get a finer mesh.

Model complexity

Do not model every little architectural detail - especially not things like small holes in slabs and walls.

Design Options

Take control (get it right at the beginning!)

10 Reference

This section contains reference information on properties, commands, settings, options, and dialog boxes.

You can find the following reference information in this section:

- Sign conventions and coordinate systems (page 1306)
- Manage global and model settings (page 1219)
- Properties reference (page 1329)
- Dialog boxes reference (page 1529)

10.1 Sign conventions and coordinate systems

Tekla Structural Designer adopts the standard convention that lower case x, y, and z represent local coordinate systems, whereas upper case X, Y, and Z represent the global coordinate system. For more information on analysis result sign conventions, see the following paragraphs.

Axis systems

The following table presents the axis systems that can be used in Tekla Structural Designer:

Axis system name	Description
Global coordinate system	The global XYZ axis system within which all other systems exist.
Building directions 1 and 2	The principal axes of the building, where dir 1 is rotated at an angle to global X in the horizontal plane.

Axis system name	Description
User coordinate system	A local coordinate system defined by the system or the user.
1D member local coordinate system	The local coordinate system that is applicable to all 1D members, such as beams, columns, and braces.
Mid-pier wall coordinate system	The local coordinate system that is applicable to walls modeled using the mid-pier option.
2D member local coordinate system	The local coordinate system that is applicable to all 2D members, walls, and slabs.
Result line coordinate system	The local coordinate system that is applicable to result lines.
Result strip coordinate system	The local coordinate system that is applicable to result strips.
Foundation reaction coordinate system	The local coordinate system that is applicable to foundations.
Punching shear check axis system	The local coordinate system that is applicable to punching checks.

General information

All global (XYZ) and local (xyz) axis systems follow the right-hand rule, where:

- x axis is the pointing index finger.
- y axis is the crooked middle finger.
- z axis is the extended thumb.

In the directions of positive rotation:

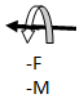
- About x: the y axis moves toward the z axis.
- About y: the z axis moves toward the x axis.
- About z: the x axis moves toward the y axis.

Diagram conventions

All arrows should point in the direction of the force or moment, as the following image illustrates:

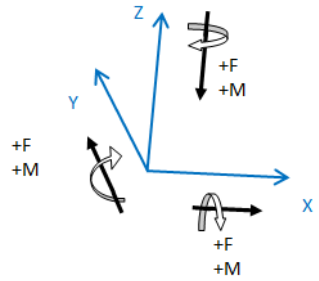


If the arrows are reversed, they become negative forces and moments, as the following image illustrates:

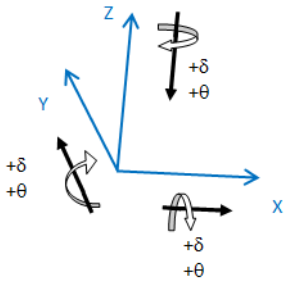


Global coordinate system

The following image illustrates the global axis system and applied load directions.



The resulting deflection directions appears as follows.

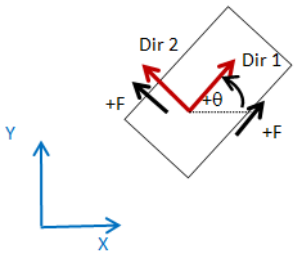


Building directions 1 and 2

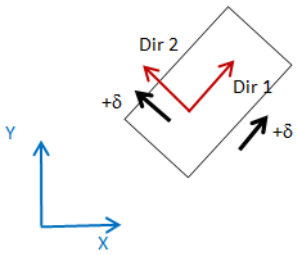
The principal axes of the building (Dir 1 and 2) can be specified at any angle to the global X axis in the global XY plane (positive Z vertically up).

The angle between X and direction 1 is θ, where θ is positive in right-hand rule about Z. Direction 2 is then +90 degrees from direction 1.

The following image illustrates the building directions and applied load directions.



The resulting deflection directions appear as follows.



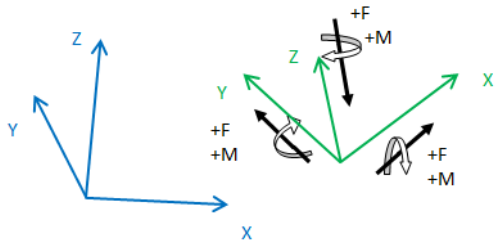
To specify and display the building directions

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.
4. Enter the **Building Direction Rotation** (labeled θ in the above diagram)
5. To have the building direction axes displayed in scene views, select **Show Building Direction Arrows**
6. If you prefer to label the building directions as Dir H/V or Dir X/Y you can change the **Building Direction Labels**.

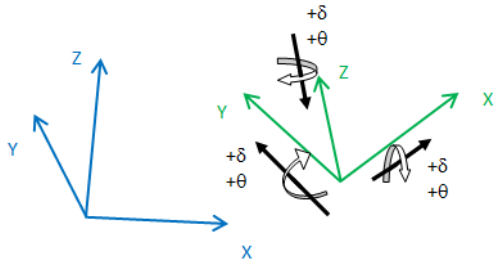
User coordinate system

A user coordinate system can be at any angle to the global coordinate system.

The following image illustrates the axis system of a user coordinate system axis and applied load directions.



The resulting deflection directions appear as follows.



NOTE Every support is given a user coordinate system. Automatically created supports under certain objects default to the following method:

- Support under a single column or wall rotates the foundation forces to align with the y/z-axes of the column or wall
- Support under a mat foundation - uses the global coordinate system.

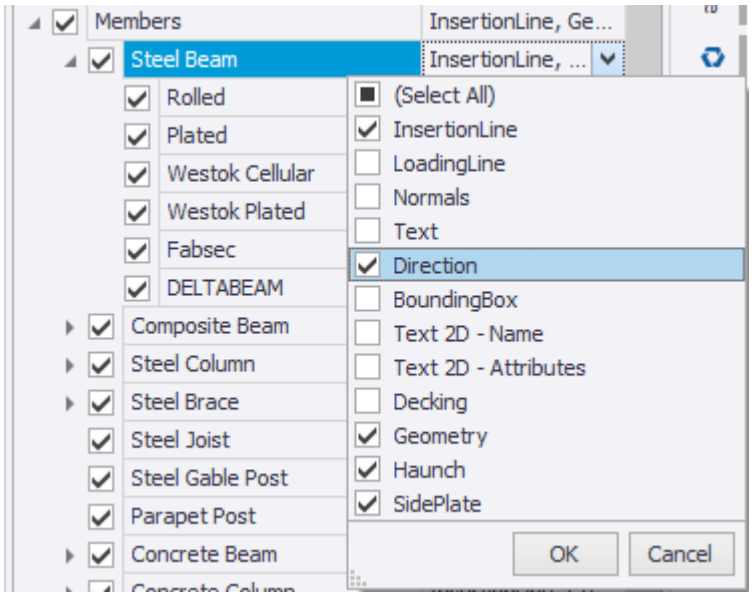
All other supports default to the global coordinate system.

1D member local coordinate system (general case)

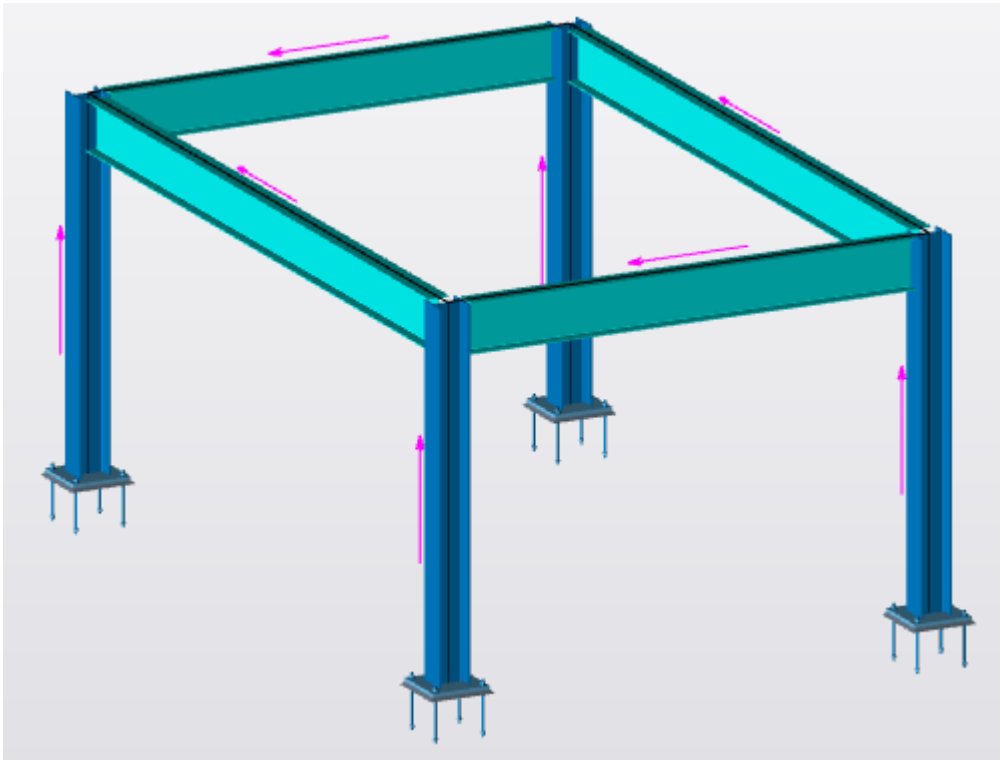
Member orientation

Tekla Structural Designer considers member orientation when displaying analysis results. Therefore, to apply the sign convention correctly, you need to know which is **end 1** and which is **end 2** of the member. For beams you need to be able to identify the top flange, and for columns you also need to be able to identify the four faces: **A, B, C & D**.

To display the direction in a scene view select the **Direction** option for the member type in **Scene Content**.

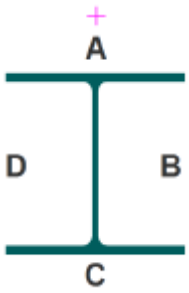


This displays a direction arrow along the member which points from **end 1** to **end 2**.



For beams the arrow is drawn next to the top flange. For columns the direction is always from bottom to the top and the arrow is always drawn adjacent to

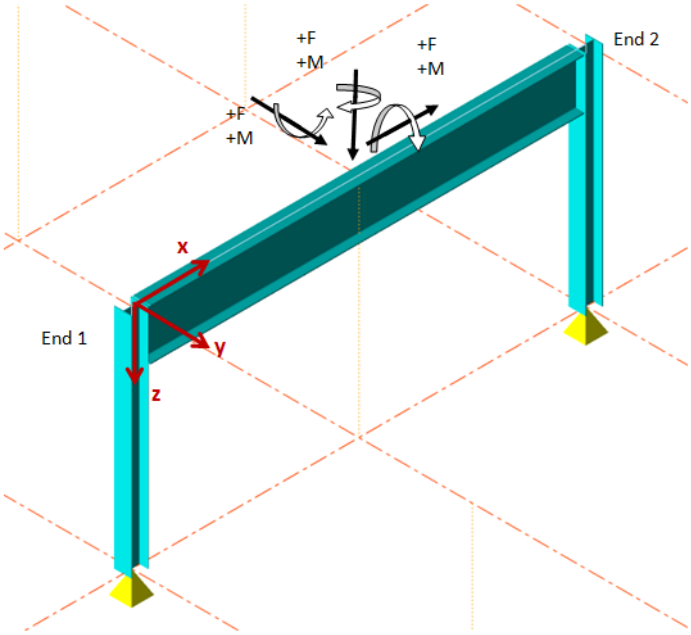
Face A. Looking down from the top of a column, Faces **B**, **C**, and **D** then follow in the clockwise direction.



Local axis system and applied load directions:

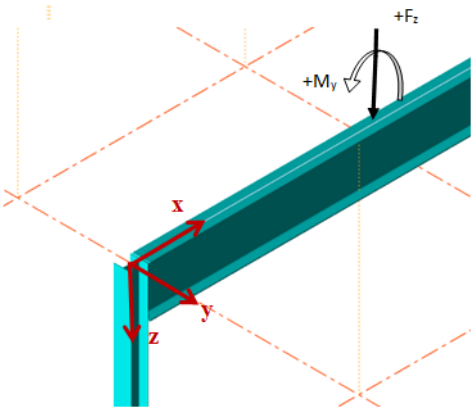
- The local x axis along member starts at **end 1** and ends at **end 2**
- When gamma (γ) = 0:
  - The local z axis lies in the plane created by the local x axis and the global Z axis.
  - The global Z component of the local z axis is always negative.
  - The local y axis follows the right-hand rule.

  gamma (γ) = positive clockwise rotation of y and z axes about the x axis looking towards positive x.

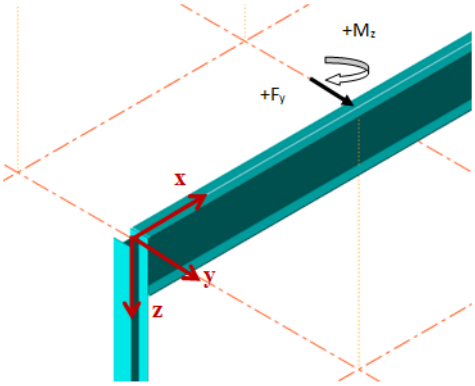


Applied force directions:

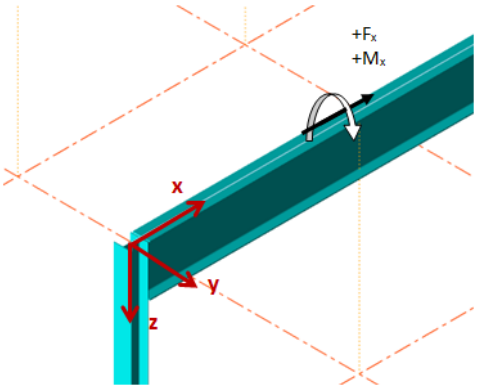
- z = Major (Fz and My):



- y = Minor (Fy and Mz):



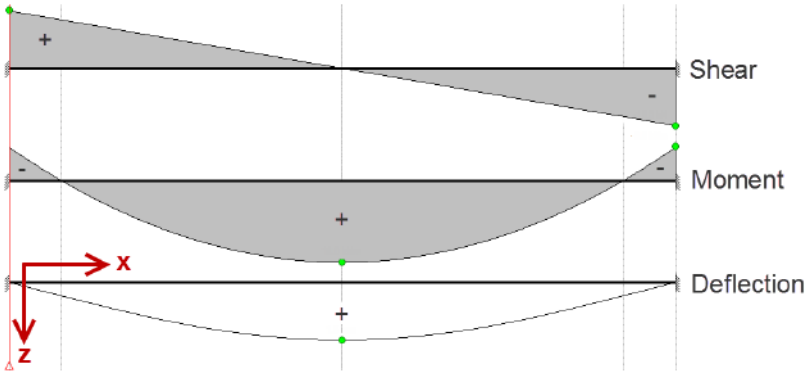
- x = Axial:



Result axis system and directions

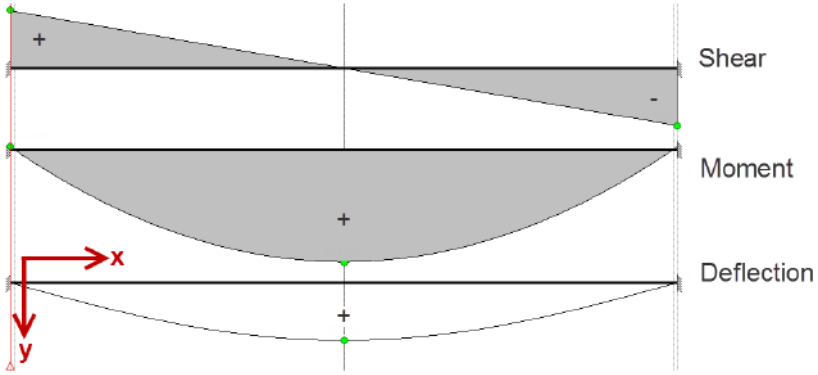
In the major axis:

- Moment major = bending about the y axis
- Shear major = shear along the z axis

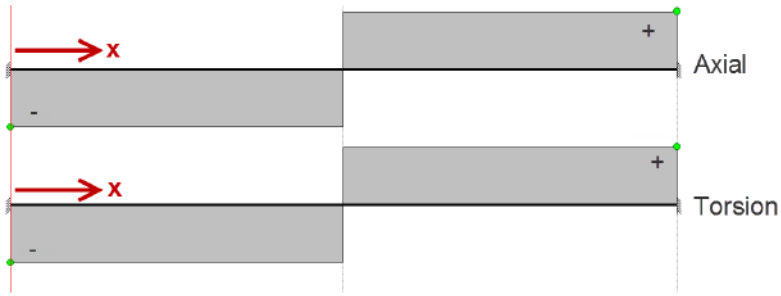


In the minor axis:

- Moment minor = bending about the z axis
- Shear minor = shear along the y axis

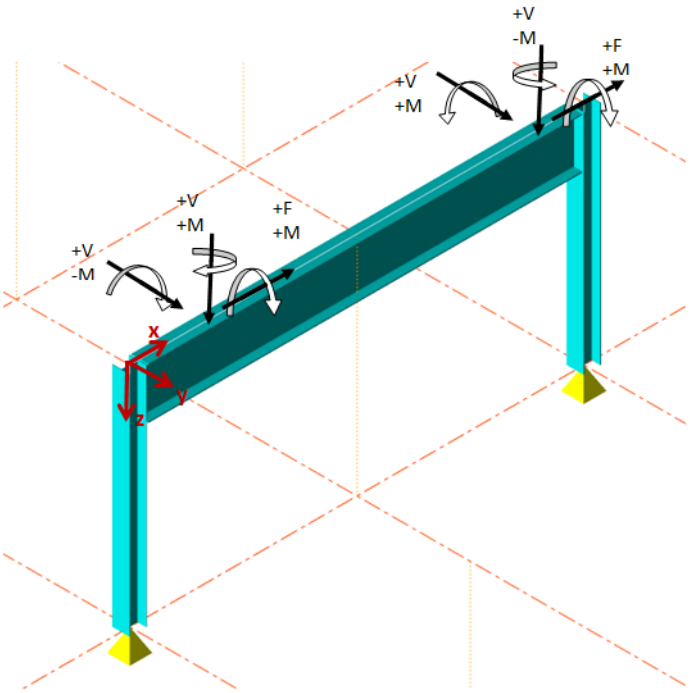


In the axial direction:



Resulting member end forces and directions

Member end forces are the forces applied to the rest of the structure by the member. Based on loading applied above, the forces would be applied as follows:



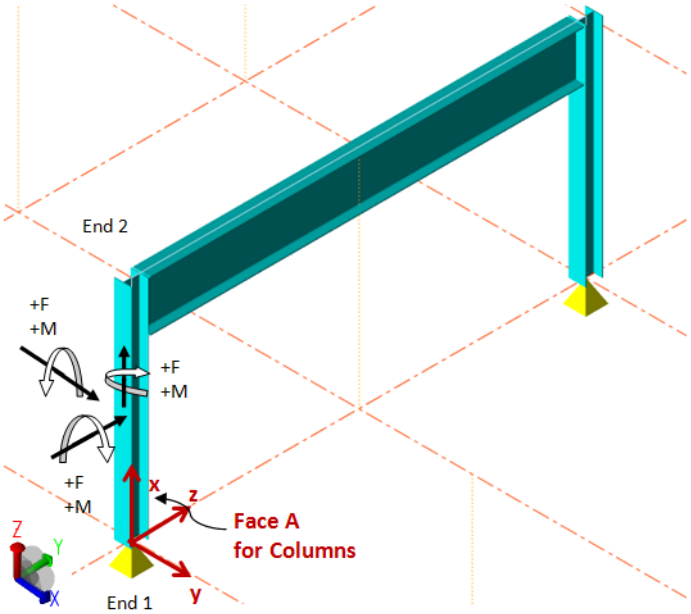
1D member local coordinate system (vertical members)

Local axis system and applied load directions

Local x aligns with global Z (vertical):

- When gamma (γ) = 0:
  - The local y axis aligns with global X.
  - The local z axis according to the right-hand rule.

  gamma (γ) = positive clockwise rotation of y and z about the x axis towards positive x.



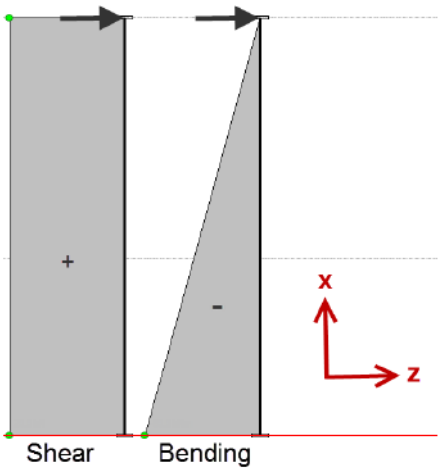
Applied force directions as displayed in the previous image:

- z = Major
- y = Minor
- x = Axial

Result axis system and directions

In the major axis:

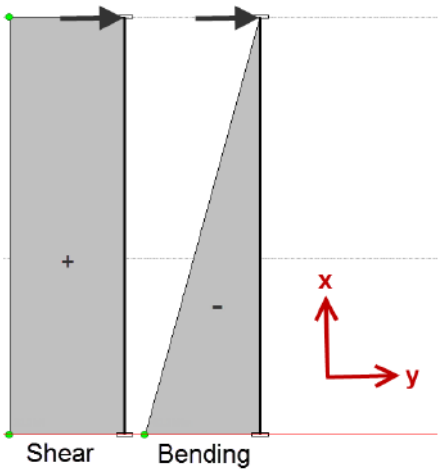
- Moment major = bending about the y axis
- Shear major = shear along the z axis



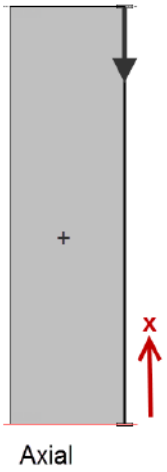
In the minor axis:

- Moment minor = bending about the z axis

- Shear minor = shear along the y axis



In the axial direction:

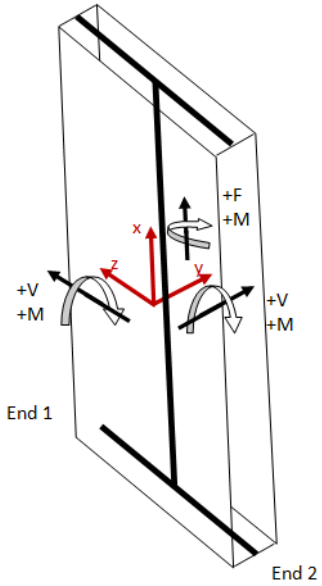


Mid-pier wall coordinate system

Wall axis system and applied load directions

As the following image illustrates, centered on the centroid of the cut section:

- the x axis lies along the stem mid-pier element (positive lowest to highest)
- the z axis lies along the plane of the wall (positive end 2 to end 1)
- the y axis follows the right-hand rule and is normal to the wall.

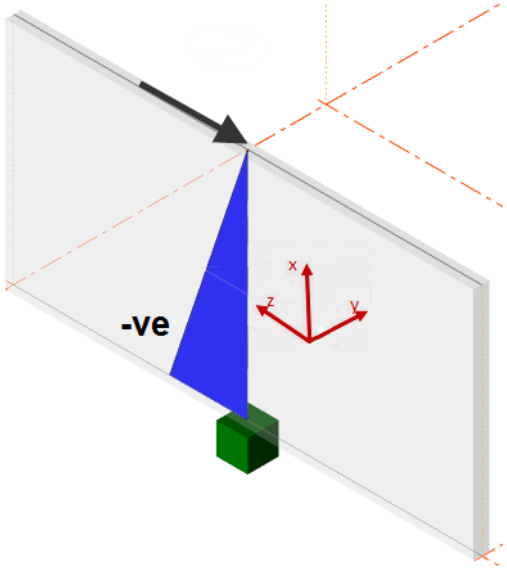


The results from a mid-pier model are in the same axis system as the result line in a meshed wall.

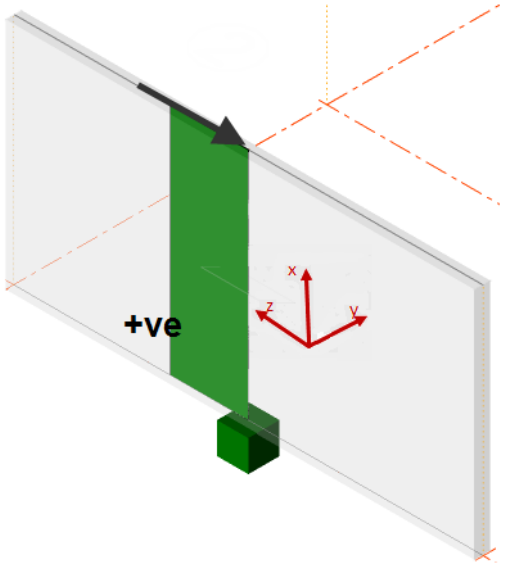
Result axis system and direction

In the major axis:

- Moment major = bending about the y axis:

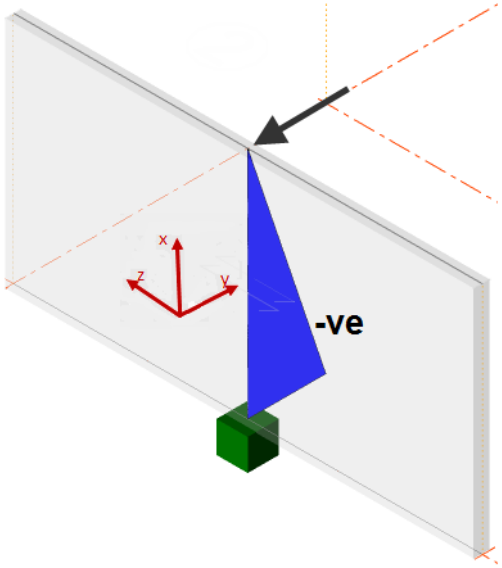


- Shear major = shear along the z axis:

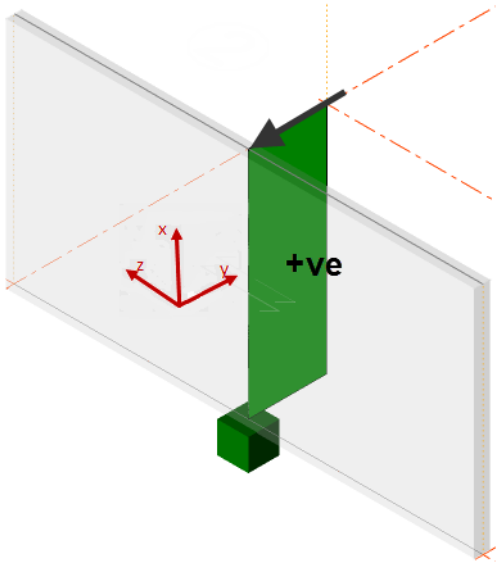


In the minor axis:

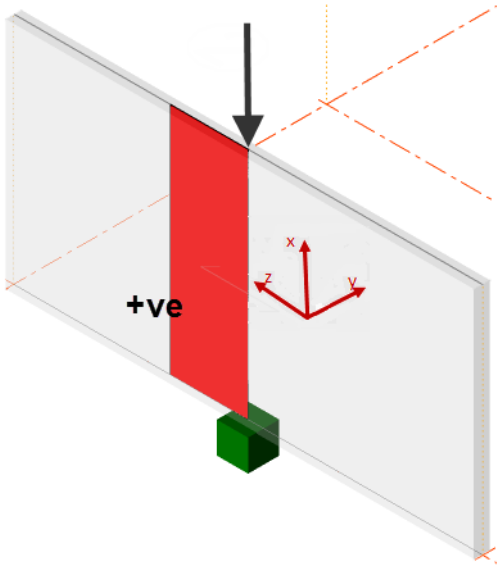
- Moment minor = bending about the z axis:



- Shear minor = shear along the y axis:



In axial and torsion, force is in the x axis and torsion about the x axis:



2D member local coordinate system

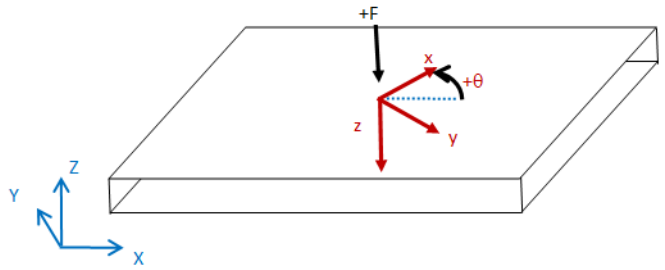
Horizontal panel local axis system and applied load directions

Horizontal panel local axes are the following:

- The local z axis is normal to the plane of the panel
- When θ = 0:
  - The local x axis plane is in the plane of the panel, aligned with the global X axis and positive in the positive global X direction.

- The local y axis is in the plane of the panel and follows the right-hand rule.

θ = positive clockwise rotation of the x and y axis about the z axis looking towards positive z.



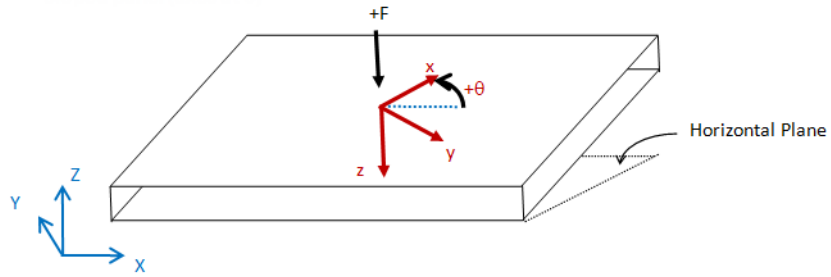
Vertical and sloped panel local axis system and applied load directions

Vertical and sloped panel local axes are the following:

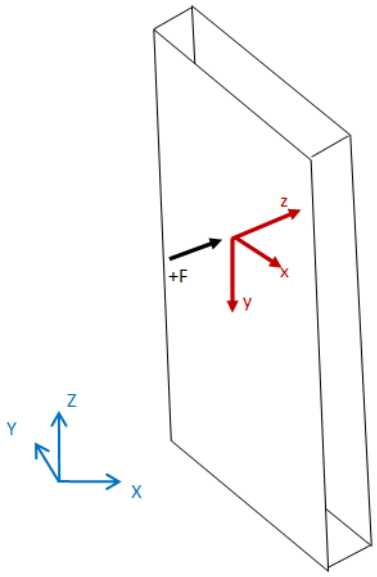
- The local z axis is normal to the plane of the panel.
- When θ = 0:
  - The local x axis plane is in the plane of the panel and in a horizontal plane.
  - The local y axis is in the plane of the panel and follows the line of greatest slope of the plane (positive in the direction of positive global).

  θ = positive clockwise rotation of x and y about z looking towards positive x.

Sloped panel (axes at θ):

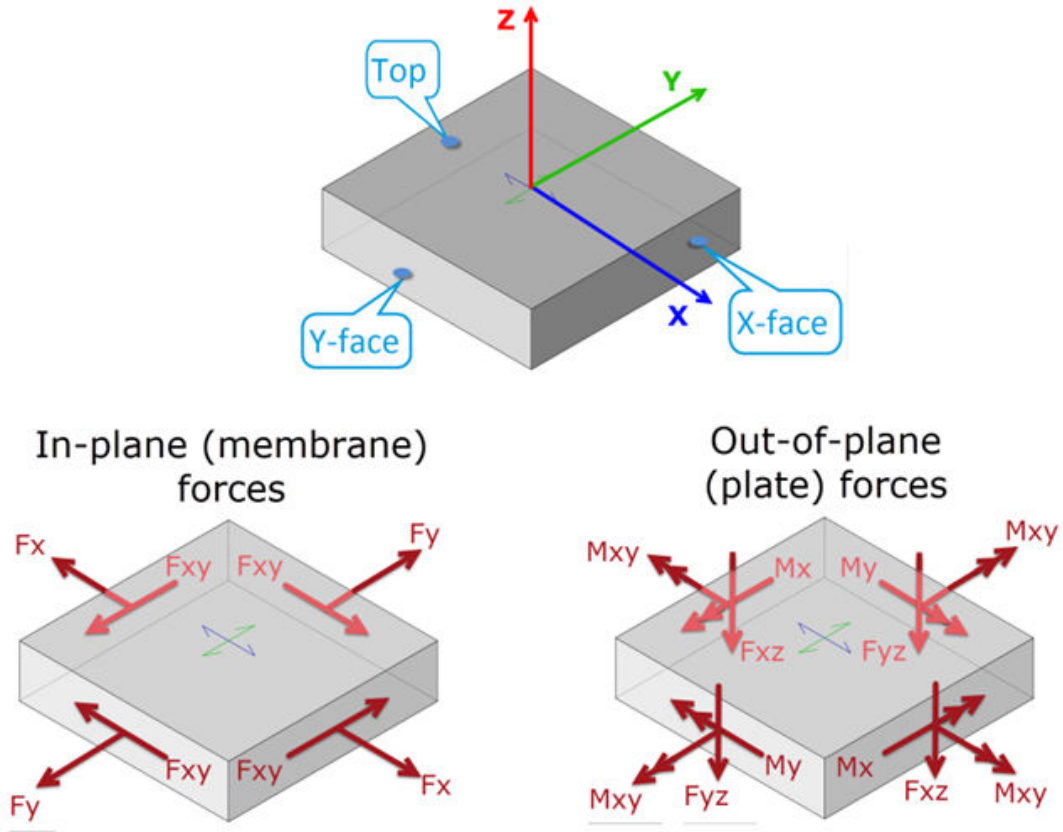


Vertical panel (axes when θ = 0):


z
+F
x
y
Z
Y
X

2D member forces sign convention

The sign convention for 2D member forces is not the same as that of 1D elements. The following diagram illustrates the forces and the panel and 2D element axis system (for results):



- The arrows in the diagram show positive force directions.
  - The double-arrow convention is used for moments: the moment is around the double-arrow, positive being clockwise when looking in the arrow direction.
  - The forces act on a member face cut anywhere in the FE mesh, perpendicular to the force direction.

  Thus, for example, Mx acts on the X face that is perpendicular to the X axis and is moment resulting from spanning in the X direction, Fxz is the out-of-plane shear force acting on the X face, and so on.
- The wood armer design moments (denoted by the *d* suffix) act in the same manner as the unprocessed moments without the *d* suffix. Thus, Mdx acts in the same manner as Mx, and so on.
- The design moments are further classified into top and bottom components for the slab design process.
- The positive Z axis direction (up) follows the right-hand rule and, therefore, is not the same as that for the 2D member local coordinate system. This is

because the 2D member local coordinate system for the applied load directions displays the positive applied load direction convention that, for Z only, is opposite to the convention of the global and 2D element axes.

- A positive moment creates tension in the top surface of the shell. Therefore, the moment over a supporting column is positive, whereas the span moment is negative.
- The conventions for wall results are exactly the same as conventions for columns, so they can be interpreted in the same way.
- The compression of axial loads (Fx and Fy) is negative.
- Out-of-plane shear (Fxz and Fyz) is positive when shear is such that moment is increasing in the positive X or Y direction.

Result line coordinate system

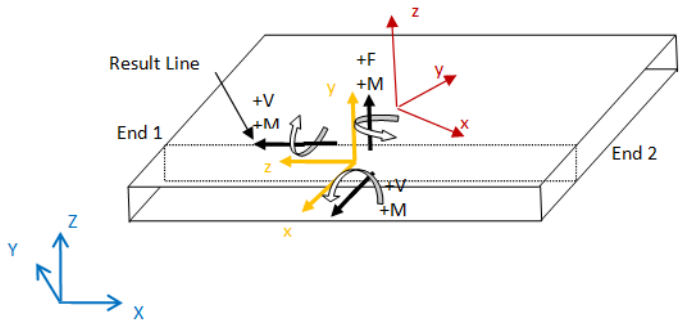
Centered on the centroid of the cut section:

- The z axis lies along the result line (positive end 2 to end 1).
- The y axis normal to plane of mesh (generally positive in the positive Z direction, in special cases positive x towards positive Z).
- The x axis follows the right-hand rule and lies in the mesh, so x is perpendicular to the cut line.

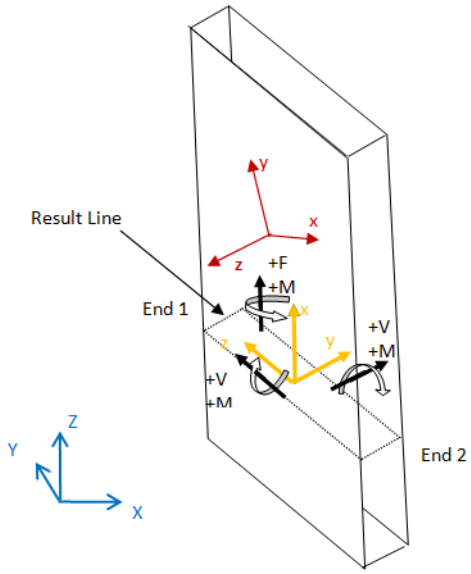
NOTE The results from a result line are exactly like those for a mid-pier model when the cut is horizontal and the cut direction matches the direction required.

- In the major axis: bending about the y axis and shear along the z axis
- In the minor axis: bending about the z axis and shear along the y axis
- Axial and torsion: force in x and bending about the x axis

General case:



Special case:

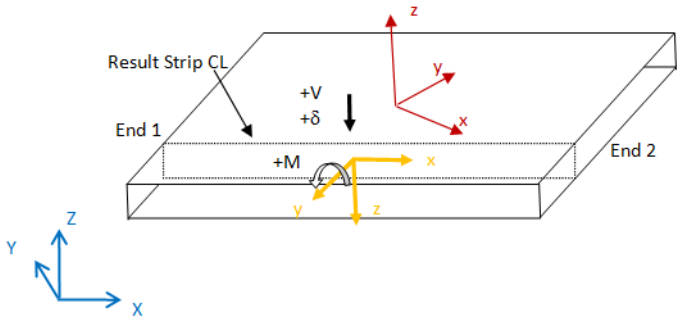


Result strip coordinate system

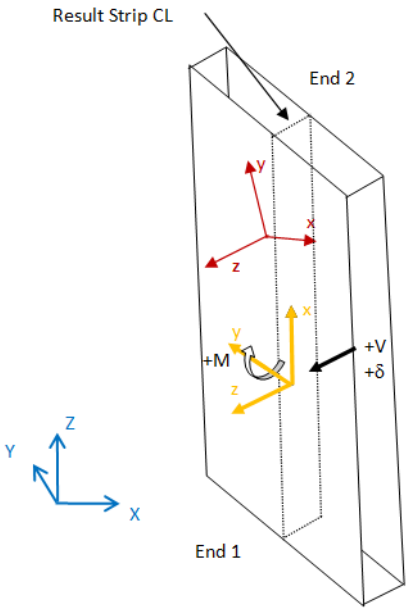
Centered at each station along the strip center line, whether there is a single or several continuous strips:

- The z axis is normal to plane of mesh (generally positive in the negative Z direction, in special cases positive x towards positive Z).
- The x axis lies along the result strip (positive end 1 to end 2)
- The y axis lies along the transverse line to the result strips and follows the right-hand rule, so the y axis is perpendicular to the strip line.
- Deflection in the z direction
- Out of plane moment about the y axis
- Shear in the z direction

General case:

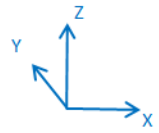


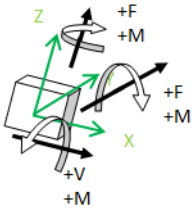
Special case:



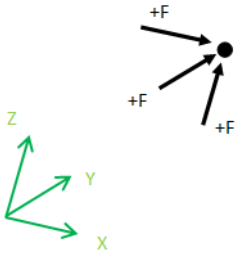
Foundation reaction coordinate system

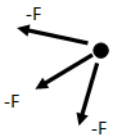
As the following image illustrates, the foundation reaction coordinate system is aligned with the coordinate system for the support node, whether that is the global coordinate system or a user coordinate system.





Reactions are the forces applied to the structure by the foundation. They appear as follows.



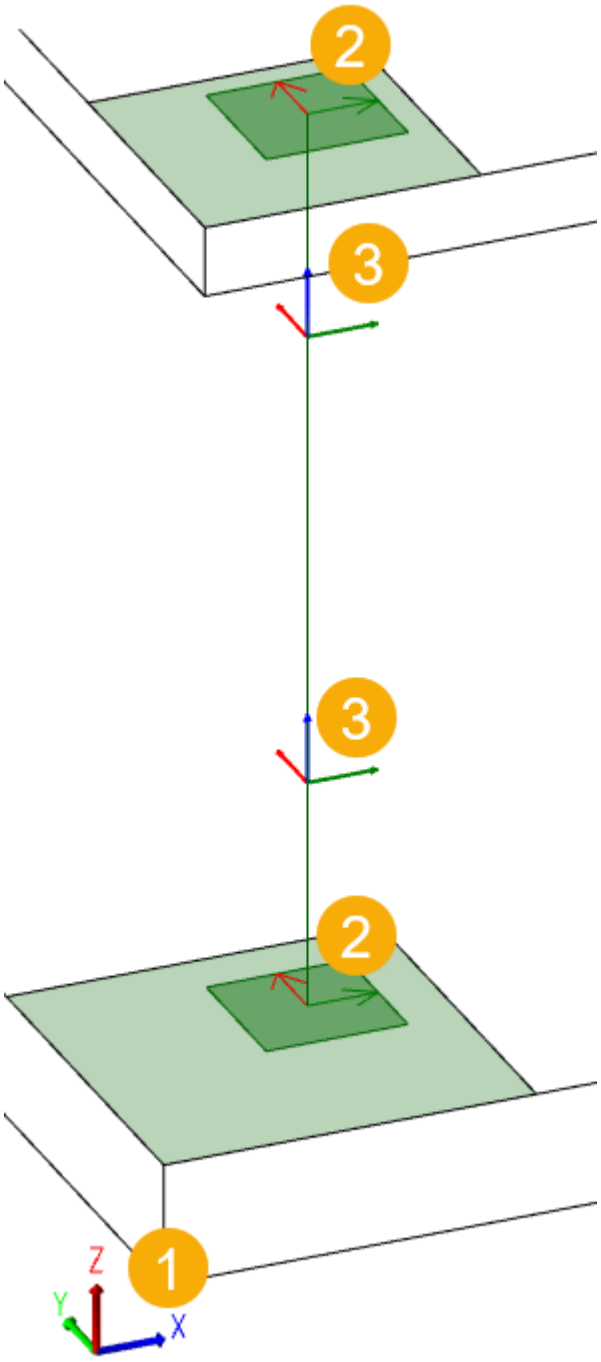


Punching shear check axis system

For all punching checks, the YZ plane is in the slab plane. Using the YZ system in the slab plane coherently assumes that the vertical force is positive upwards following the right hand rule.

- **Punching checks applied to point loads** - by default the check Y and Z axes default to align with global X and Y. The **Loaded Area Orientation** angle can then be used to rotate these in relation to the global Y axis.

  Punching checks applied to columns - as the following image illustrates, the check Y and Z axes are aligned and locked to the local axis system for the column elements so it is easier to relate forces in both objects.



1 - Global coordinate system

2 - Punching shear check axis system	**NOTE** The two axis systems are locked together, so if the column is rotated, the punching check axes also rotate.
3 - Column 1D member local coordinate system	

10.2 Properties reference

This section provides more information about the properties you can modify in Tekla Structural Designer.

Structure geometric and analytical properties

Click the links below to find out more:

- Structure Properties (page 1331)
- Level Properties (page 1333)
- Frame Properties (page 1335)
- Slope Properties (page 1336)
- Sub Model Properties (page 1337)

Members

Click the links below to find out more:

- Beam properties (page 1338)
- Column properties (page 1374)
- Brace properties (page 1402)
- Analysis Element properties (page 1419)
- Base plate properties (page 1422)
- Member characteristic, construction and fabrication properties (page 1427)

Walls and cores

Click the links below to find out more:

- Bearing wall properties (page 1432)
- Concrete meshed and mid-pier wall properties (page 1434)
- Shear only wall properties (page 1450)
- General wall properties (page 1452)
- Concrete core properties (page 1457)

Slabs and mats

Click the links below to find out more:

- Slab item properties (page 1458)
- Slab/Mat overhang properties (page 1479)
- Patch properties (page 1479)
- Foundation mat properties (page 1482)
- Punching check properties (page 1486)
- Result strip properties (page 1491)

Portal frames

Click the links below to find out more:

- Portal frame properties (page 1492)

Isolated foundations

Click the links below to find out more:

- Pad base strip base and pile cap properties (page 1496)

Loading elements

Click the links below to find out more:

- Roof Panel Properties (page 1505)
- Wall Panel Properties (page 1504)
- Line ancillary properties (page 1507)
- Area ancillary properties (page 1510)
- Equipment properties (page 1512)

Solver model and supports

- Support properties (page 1514)
- Solver node properties (page 1517)
- Solver element properties (page 1518)
- Solver element 2D properties (page 1519)

Floor Analysis Models

Click the links below to find out more:

- Floor Analysis Model (FAM) properties (page 1519)

Structure geometric and analytical properties

This section provides more information about some of the structure geometric and analytical properties you can modify in Tekla Structural Designer.

Click the links below to find out more:

- Structure Properties (page 1331)
- Level Properties (page 1333)
- Frame Properties (page 1335)
- Slope Properties (page 1336)
- Sub Model Properties (page 1337)

Structure Properties

Use the **Structure** properties to view or modify the basic properties of the structure.

Property	Description
Building Direction Rotation	Defines the building direction relative to the Global Axis System The default (0 degrees) aligns the building direction 1 arrow with the global X axis and the direction 2 arrow with the global Y axis. Entering a positive value rotates the Building Direction arrows clockwise about positive Global Z, a negative value rotates anti-clockwise. The limiting values are +45 degrees and -45 degrees. (If you enter larger values they will be capped at these limits). The building direction arrows are always at 90 degs to each other.
Show Building Arrows	When shown, building direction arrows are displayed in all 2D and 3D Views.
Building Direction Labels	The labels to be used for the building direction arrows. The options are: • Dir 1/2 • Dir H/V • Dir X/Y
Consequence class	Defines k_{FI} used in strength combinations. • CC3 - k_{FI} = 1.1 • CC2 - k_{FI} = 1.0 • CC1 - k_{FI} = 0.9 **NOTE** Applies to **Finland (Eurocode)** regional code only

Property	Description
Reliability class	Defines γ_d used in strength combinations. • RC3 - γ_{FI} = 1.0 • RC2 - γ_{FI} = 0.91 • RC1 - γ_{FI} = 0.83 **NOTE** Applies to **Sweden (Eurocode)** regional code only
Slab Mesh Size	Defines the mesh size for two way spanning slabs. **NOTE** To optimize solution time consider using a coarser mesh during design development before switching to a more refined mesh at the final design stage.
Slab Uniformity Factor	Defines the mesh uniformity for two way spanning slabs. (100% = maximum uniformity).
Slab Mesh Type	Defines the mesh type for all meshed slabs in the model. The options are: • Quad dominant • Quad only • Tri only
Semi-Rigid Mesh Size	Defines the mesh size for roof panels, and slabs when modeled as semi-rigid diaphragms.
Semi-Rigid Uniformity Factor	Defines the semi-rigid mesh uniformity for roof panels, and slabs when modeled as semi-rigid diaphragms.
Semi-Rigid Mesh Type	Defines the semi-rigid mesh type for roof panels, and slabs when modeled as semi-rigid diaphragms The options are: • QuadDominant • QuadOnly • Triangular
Wall Mesh Horizontal Size	Defines the horizontal mesh size for all meshed walls in the model - but can be overridden in individual wall properties.
Wall Mesh Vertical Size	Defines the vertical mesh size for all meshed walls in the model - but can be overridden in individual wall properties.

Property	Description
Wall Mesh Type	Defines the mesh type for all meshed walls in the model, (but can be overridden in individual wall properties): The options are: • Quad only • Tri only • Quad dominant

Level Properties

Use the **Level** properties to view or modify the basic properties of a level.

General	
Level	The height of the construction level above the base level
Floor	By setting a construction level to be a Floor you are indicating that it is a major level in the building. Floor levels are used to determine items such as your inter story height and positions from which column splices are laid out. If a level is not set to be a floor then no live load reductions will be accounted for in the beams at that level, or in the columns supporting that level. There can certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs, • K Bracing - you require a construction level for the intermediate bracing connection points, • steps in the building floor levels. Where you define a level which is clearly not a floor, then you should not check the floor box.
Reporting Level	By default 'Floor levels' are 'reporting levels', but this can be switched off: • This can be useful where for some modeling reason many closely spaced floor levels have been defined. This option can be unselected in order to combine results with those from the level above when viewing Embodied Carbon totals that are grouped by level. Also, if this option is unselected, the sway/drift, seismic drift and wind drift check results for all column stacks and wall panels at the specific level are excluded from the tabular review data. In addition, the height of structure considered in the Overall Wind Drift check (page 840) is determined according to how this option

General	
	has been set - the height used being the distance between highest level and the lowest level in the model that have **Check for drift** selected. **NOTE** This option is only displayed if the **Floor** option described above is selected.
Type	The level type can be: • T.O.S = Top of Steel • S.S.L = Structural Slab Level • T.O.F = Top of Foundation **NOTE** Slabs are modeled above the level when it is set to T.O.S but below the level when it is set to S.S.L or T.O.F
Short Name	Each construction level should be given a unique reference. Typically this might be a storey number, 1, 2 , 3 etc.
Long Name	Each construction level can also be given a name to further assist identification. 'First Floor', or 'Mezzanine' etc.
Name	Automatically generated from the short and long name. By default this will be used as the name in the **Structure** tree
Mesh slabs in 3D analysis	• On = 2-way slabs (and 1-way slabs with the **Analyze slab** slab item property selected) are meshed in the 3D building analysis and grillage chasedown analysis in addition to the FE chasedown analysis. • Off = 2-way slabs (and 1-way slabs with the **Analyze slab** slab item property selected) are only meshed in the FE chasedown analysis. **NOTE** The Sub Model in which the Level is contained determines the mesh parameters that are applied.
Include in Export	Determine whether Level should be included in CXL or IFC export. • Floor Only - include only if defined as a Floor. • Yes - always include. • No - never include. **NOTE** The lowest Level will always be exported.

General	
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified level • Off = Grid & construction lines are always off for the specified level, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Decomposition> Keep solver model	Controls whether you will be able to review the solver model used for load decomposition between each analysis run: • On = the solver model is retained, you might choose to use this setting if you need to investigate any validation warnings that relate to load decomposition. • Off (default) = the solver model is discarded immediately after analysis has been performed.

Decomposition	
Keep solver model	Controls whether you will be able to review the solver model used for load decomposition between each analysis run: • On = the solver model is retained, you might choose to use this setting if you need to investigate any validation warnings that relate to load decomposition. • Off (default) = the solver model is discarded immediately after analysis has been performed.

Frame Properties

Use the **Frame** properties to view or modify the basic properties of a frame.

Property	Description
Name	Automatically generated, but can be replaced by **User name** if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified frame • Off = Grid & construction lines are always off for the specified frame irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Visible	Controls whether the frame view can be opened or not.

Slope Properties

Use the **Slope** properties to view or modify the basic properties of a frame.

Property	Description
Name	Automatically generated, but can be replaced by **User name** if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting

Property	Description
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified slope • Off = Grid & construction lines are always off for the specified slope irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Visible	Controls whether the slope view can be opened or not.

Sub Model Properties

Use the **Sub Models** properties to view or modify the selected sub model.

Property	Description
Override model's	Select this check box in order to override the Structure meshing properties in the current sub model.
Shell Mesh Size	Defines the shell mesh size for two way spanning slabs in the sub model.
Shell Uniformity Factor	Defines the shell mesh uniformity factor for two way spanning slabs in the sub model.
Slab Mesh Type	Defines the shell mesh type used in the sub model. These options are: • Quad only • Tri only • Quad dominant
Semi-Rigid Mesh Size	Defines the semi-rigid mesh size when slabs are modeled as semi-rigid diaphragms in the sub model.
Semi-Rigid Uniformity Factor	Defines the semi-rigid uniformity factor when slabs are modeled as semi-rigid diaphragms in the sub model.
Semi-Rigid Mesh Type	Defines the semi-rigid mesh type when slabs are modeled as semi-rigid diaphragms in the sub model. The options are: • Quad only

Property	Description
	• Tri only • Quad dominant

Member properties

This section provides more information about some of the member properties you can modify in Tekla Structural Designer.

Click the links below to find out more:

- Beam properties (page 1338)
- Column properties (page 1374)
- Brace properties (page 1402)
- Analysis Element properties (page 1419)
- Base plate properties (page 1422)
- Member characteristic, construction and fabrication properties (page 1427)

Beam properties

When a beam is selected, its properties are displayed in the .

- 'General' properties apply to the whole beam.
- 'All spans' properties are used to rapidly define the same properties for all beam spans.
- Individual 'Span' properties are used to over-ride 'All spans' properties.

The displayed properties are described below for each of the available material types.

Steel beam properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.

Property	Description
Characteristic	Beam
Active *[Only displayed for single span beams]*	Clearing this option makes single span beams inactive in the analysis. See:
Element type	Beam
Material type	Steel
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Attempt non-composite if auto composite solution fails *[Only displayed for composite beams]*	Typically, at the outset you will know which beams are to be non-composite and which are to be composite and you will have specified the construction type accordingly. However, circumstances can arise in which a beam initially intended to be composite proves to be ineffective and the auto-design process fails to find a suitable composite beam solution. Examples might be: • very small beams, • beams with a significant point load close to a support, • beams where the deck is at a shallow angle to the beam, hence the stud spacing is impractical, • beams where, for a variety of reasons, it is not possible to provide an adequate number of studs, Where Tekla Structural Designer is unable to find a section size which works compositely, you can ask for a non-composite design for the same loading. To do this simply select **Attempt non-composite if auto composite solution fails** in the beam properties. You may find that this facility is particularly useful when you right-click a key beam in the model in order to perform an individual member design.
Try composite section size as non-composite *[Only displayed for composite beams]*	Typically, at the outset you will know which beams are to be non-composite and which are to be composite and you will have specified the construction type accordingly. However, there can be instances where the composite beam is sufficiently under utilized that the same size beam would be adequate without the studs i.e. non-compositely. Irrespective of whether a suitable composite design solution exists, selecting this option will try the same

Property	Description
	section size non-compositely and if successful will retain this non-composite solution. **TIP** If working to the US code (AISC 360), ensure that the **Beam design section stepback** option in **Design Settings > Autodesign** is set to a value greater than zero. A value of 3 is recommended.
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Design section order *[Only displayed when 'Autodesign' is selected]*	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Span properties

Property	Description
Section	The section size
Grade	Steel grade
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1254)
Top flange cont. rest.	Define if the top flange is continuously restrained.
Bottom flange cont. rest.	Define if the bottom flange is continuously restrained.
Releases	
Free end 1, end 2	Select this option in order to define a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam end releases and partial fixity (page 568)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed

Property	Description
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	
Apply open structure wind load	Select this option if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	

Property	Description
KLL *[Only displayed for ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Deflection limits	
Apply span \factor	With this option checked, the limit can be defined as a Relative span/factor.
Apply abs. limit	With this option checked, the limit can be defined as an absolute value.
Torsion	
Check for torsion, Apply rotational limit	Used to specify if the member should be checked for torsion, and also to apply a rotational limit if required. See:
Camber	
Apply camber	Used to specify a camber to the beam if required. See:
Floor vibration	
Calculate natural frequency	For composite beams this is fixed to on (for EC and BS regional codes only) and is utilized during autodesign. It can also be switched on for pin ended non-composite beams. The calculated natural frequency is displayed in the results viewer.
Check natural frequency against minimum	For composite beams this is fixed to on (for EC and BS regional codes only). For pin ended non-composite beams it can be used in conjunction with the 'Calculate natural frequency' property to impose an optional 2-step control, ‘calculate’ and ‘check’, which allows for the natural frequency

Property	Description
	value to be calculated (and displayed in Results Viewer) with or without the check. While the US regional code has no requirement for this check, it can still be requested if required. When performed, a simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse particularly for long span beams and does not consider the response side of the behavior i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the designer has the option to perform a 1st Order Modal Analysis. See: • •
Minimum natural frequency	The minimum value against which the natural frequency is checked (default 4Hz).
Include self weight (beam & slab) Include other dead loads Include live *[ACI/ AISC]*, imposed *[Other codes]* loads	The engineer can specify the percentages of each of these loads to be included in the calculation of the maximum static instantaneous deflection, δ
Factor for increased dynamic stiffness of the concrete flange	For composite beams this factor is applied to the beam's short term modular ratio
Treat as composite *[Only displayed for non-composite beams]*	By selecting this option, provided there are composite slab(s) adjacent to the beam it will be treated as composite in footfall analysis but as non-composite in static design. The 'Effective width' and 'Calculate effective width' properties are displayed to allow the concrete flange width above the beam to be calculated.

Property	Description
Effective width Calculate effective width *[Only displayed for composite beams]*	For composite beams the effective width can either be entered directly or calculated from the geometry. See:
Size constraints	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm (6in). If so you would simply enter this value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than this width for the design of this beam. See:
Apply max span/depth ratio Max span/ depth ratio	After setting a max span/depth ratio you can check the 'Apply' button for it to be considered by auto-design. During design, only sections which satisfy the maximum ratio limit will be selected. The setting can also be reviewed and/ or copied via Review View > Show/Alter State. See:
Floor construction *[Only displayed for composite beams]*	
Slab item	If the beam has more than one adjacent slab item, select the slab item to be used when determine the beam's composite properties.
Angle to beam, Condition, Side	These properties are automatically detected. See:
Shear connector type	See:
Welding condition *[Only displayed for EC regional code]*	Whether you are given a choice between 'Site welded' or 'Shop welded' will depend on the type of deck being supported and whether the deck spans parallel or perpendicular to the beam. See:
Override effective width	Allows the automatically calculated effective width to be overridden. See:

Property	Description
Effective width	Select Override effective width if you require to change the automatically calculated value. See:
Calculate Effective width	Click the [...] button to recalculate if required after changing the geometry. See:
Metal deck *[Only displayed for composite beams]*	
Minimum distance to lap	See:
Haunch width above beam for parallel decking, b_0 *[Only displayed for EC regional code where the studs are 'Shop welded' and the supported deck is parallel to the beam]*	If the studs are shop welded, so the decking has to stop and start, you are allowed to increase the haunch width b_0 as per EN 1994-1-1:2004 clause 6.6.4.1. See:
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Seismic *[Only displayed for non-composite beams]*	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Regional Code in the current release.
End 1 / End 2 *[Only displayed for ACI/AISC regional code]*	
Apply SidePlate connection	Select to apply a side plate connection at the specified end(s).
Utilization ratio	

Property	Description
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	
Check for fire resistance *[Only displayed for non-composite, simply supported rolled steel beams to Eurocodes]*	When the check is requested, the additional properties shown below are required. See:
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See: • If 'Check for fire resistance' is also selected, the check is performed using the 'protected' time interval for critical temperature iteration, as specified in Design Settings • When members are not set as 'Protected' and 'Check for fire resistance' is selected, the check is performed using the 'unprotected' time interval for critical temperature iteration, as specified in Design Settings
Exposure	Exposed on three, or four sides
Fire check *[Only displayed for the Eurocode regional code]*	
Use user defined load reduction factor	If selected, the load reduction factor is a user input value (default 0.65). If unselected, the load reduction factor is calculated automatically, as per Cl 2.4.2 of EN 1993-1-2 for each beam based on the applied load.

Property	Description
Required time of fire exposure	R15, R30, R60, R90, or R120
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Concrete beam properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active *[Only displayed for single span beams]*	Clearing this option makes single span beams inactive in the analysis. See:
Element type	Beam
Material type	Concrete
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:

Property	Description
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See:
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Allow automatic join end 1, end 2	When this checkbox is selected - the end in question will be automatically joined to a suitable connecting concrete beam end during design process or when the 'Beam Lines' command is run, (providing the Beam Lines limiting criteria specified in Model Settings are met.)

Span properties

Property	Description
Section	The section size
Concrete type,	While you can apply both normal and lightweight concrete, design using lightweight concrete is currently beyond scope.
Concrete class *[Eurocode]*, Grade *[Other codes]*	Specifies the concrete class *[Eurocode]*, Grade *[Other codes]*.
Concrete density class *[Eurocode only]*	For lightweight concrete only, specifies the density class.
Linearity	• Straight

Property	Description
	• Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1254)
Alignment	
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	
Free end 1, end 2	When this checkbox is selected - defines a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam end releases and partial fixity (page 568)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed

Property	Description
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	

Property	Description
KLL *[Only displayed for ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Analysis & design control	
Structure supporting sensitive finishes *[Only displayed for ACI/AISC and Eurocode Regional Codes]*	Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting this property.
	ACI - The deflection method applied to the beam depends on this setting as follows: • beams not required to support sensitive finishes adopt the simplified method. • beams required to support sensitive finishes adopt the rigorous method.
	Eurocode - The f2 parameter used in the deflection check depends on this parameter as follows: • When selected: f2 is calculated as MIN[1,7/L_{eff}] • When unselected: f2 will be taken as 1.0.
Increase reinforcement if deflection check fails *[Only displayed for Eurocode BS and IS Regional Codes]*	Select in order to increase the reinforcement during the auto-design process if the deflection check fails.
Permissible increase in reinforcement	Specify the max percentage increase in reinforcement that is allowed in order to satisfy the deflection check.

Property	Description
Consider flanges	Select in order to consider flanges in the concrete beam design calculations - once checked additional fields are displayed for specifying an allowance for openings. Flange dimensions can only be calculated by editing the beam once it has been positioned and slabs have been defined. (In this case a 'Calculate flanges' button is also displayed, this can be clicked in order to automatically calculate the flange dimensions based on the adjoining slabs.) See:
Include flanges in analysis *[Only displayed when the 'Consider flanges' option has been selected.]*	Select this checkbox to use flanged beam properties when the analysis is performed. See:
Isolated beam (ACI only) *[Only displayed for ACI code and only when the 'Consider flanges' option has been selected.]*	Select this checkbox in order to apply ACI 318 clause 8.12.4. When the check is performed, if the flange geometry does not meet the requirements the flanges are ignored. See:
User defined flange (left/ right) *[Only displayed when the 'Include flanges in analysis' option has been selected.]*	If you clear this checkbox the flange depth and the effective flange width are determined automatically. If you select the checkbox, two new properties are displayed for defining 'Flange width' and 'Flange depth'. See:
Ignore lateral instability *[Only displayed for Eurocode Regional Code]*	This option allows you to ignore lateral instability for slender spans to EC2 clause 5.9(1). • When selected: the slender span check is excluded from design • When unselected (default): the slender span check is included.

Property	Description
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption. See: Cracked, partially cracked, and uncracked concrete members (page 650)
Design parameters *[Not displayed for ACI/AISC Regional Code]*	
Permanent load ratio option *[Only displayed for Eurocode Regional Code]*	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. See:
Maximum crack width *[Not displayed for ACI/AISC Regional Code]*	The maximum center to center bar spacing for crack control is dependent on the maximum allowable crack width that is specified here.
Nominal cover beam top edge, bottom edge, section side, beam ends	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface. Different values of nominal cover can be specified to the beam edges, sides and ends.
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)

Property	Description
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on three, or four sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)
Reinforcement	
Rib type - longitudinal, Class - longitudinal	Specifies the longitudinal reinforcement properties
Rib type - link, Class - link	Specifies the link properties.
Top longitudinal bar pattern, Bottom longitudinal bar pattern	Choose from standard patterns (which can be setup in Design Options) to control the top and bottom bar arrangement when the beam is auto-designed. See:

Reinforcement

Property	Description
Rib type - longitudinal, Class - longitudinal	Specifies the longitudinal reinforcement properties
Rib type - link, Class - link	Specifies the link properties.

Property	Description
Top longitudinal bar pattern, Bottom longitudinal bar pattern	Choose from standard patterns (which can be setup in Design Options) to control the top and bottom bar arrangement when the beam is auto-designed. See:

Timber beam properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active *[Only displayed for single span beams]*	Clearing this option makes single span beams inactive in the analysis. See:
Element type	Beam
Material type	Timber
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:

Property	Description
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Span properties

Property	Description
Section	The section size
Grade	Timber grade
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1254)
Top flange cont. rest.	Define if the top flange is continuously restrained.
Bottom flange cont. rest.	Define if the bottom flange is continuously restrained.
Releases	
Free end 1, end 2	When this checkbox is selected - defines a cantilever end.

Property	Description
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam end releases and partial fixity (page 568)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)

Property	Description
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
KLL *[Only displayed for ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Deflection limits	
Apply span \factor	With this option checked, the limit can be defined as a Relative span/factor.
Apply abs. limit	With this option checked, the limit can be defined as an absolute value.
Instability factor	

Property	Description
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on three, or four sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

General beam properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active *[Only displayed for single span beams]*	Clearing this option makes single span beams inactive in the analysis. See:
Element type	Beam
Material type	Timber
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)

Property	Description
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).

Span properties

Property	Description
Section	The section size
Grade	The general material grade
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1254)
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	

Property	Description
Free end 1, end 2	When this checkbox is selected - defines a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam end releases and partial fixity (page 568)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	

Property	Description
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
KLL *[Only displayed for ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:

Property	Description
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on three, or four sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Cold Formed beam properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active *[Only displayed for single span beams]*	Clearing this option makes single span beams inactive in the analysis. See:
Element type	Beam
Material type	Cold Formed
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member

Property	Description
	characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Design section order *[Only displayed when 'Autodesign' is selected]*	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Span properties

Property	Description
Section	The section size
Grade	Cold formed steel grade
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.

Property	Description
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1254)
Top flange cont. rest.	Define if the top flange is continuously restrained.
Bottom flange cont. rest.	Define if the bottom flange is continuously restrained.
Releases	
Free end 1, end 2	When this checkbox is selected - defines a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam end releases and partial fixity (page 568)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear

Property	Description
	• Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
KLL *[Only displayed for ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced

Property	Description
	• combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Deflection limits	
Apply span \factor	With this option checked, the limit can be defined as a Relative span/factor.
Apply abs. limit	With this option checked, the limit can be defined as an absolute value.
Torsion	
Check for torsion, Apply rotational limit	Used to specify if the member should be checked for torsion, and also to apply a rotational limit if required. See:
Camber	
Apply camber	Used to specify a camber to the beam if required. See:
Floor vibration	
Calculate natural frequency	Can not be activated for cold formed.
Include self weight (beam & slab) Include other dead loads Include live *[ACI/AISC]*, imposed *[Other codes]* loads	The engineer can specify the percentages of each of these loads to be included in the calculation of the maximum static instantaneous deflection, δ
Factor for increased dynamic stiffness of the concrete flange	For composite beams this factor is applied to the beam's short term modular ratio
Size constraints	
Max depth,	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of

Property	Description
Min depth, Max width, Min width	150mm (6in). If so you would simply enter this value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than this width for the design of this beam. See:
Apply max span/depth ratio Max span/ depth ratio	After setting a max span/depth ratio you can check the 'Apply' button for it to be considered by auto-design. During design, only sections which satisfy the maximum ratio limit will be selected. The setting can also be reviewed and/ or copied via Review View > Show/Alter State. See:
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Regional Code in the current release.
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	

Property	Description
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on three, or four sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Cold rolled beam properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active *[Only displayed for single span beams]*	Clearing this option makes single span beams inactive in the analysis. See:
Element type	Beam
Material type	Cold Rolled
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)

Property	Description
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Span properties

Property	Description
Section	The section size
Grade	Cold rolled grade
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1254)
Top flange cont. rest.	Define if the top flange is continuously restrained.
Bottom flange cont. rest.	Define if the bottom flange is continuously restrained.

Property	Description
Releases	
Free end 1, end 2	When this checkbox is selected - defines a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam end releases and partial fixity (page 568)
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	

Property	Description
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
KLL *[Only displayed for ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:

Property	Description
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on three, or four sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Column properties

When a column is selected, its properties are displayed in the .

- 'General' properties apply to the whole column.
- 'All stacks' properties are used to rapidly define the same properties for all column stacks.
- Individual 'Stack' properties are used to over-ride 'All stacks' properties.
- 'Level' properties are used to specify properties that apply at specific floor levels.

The displayed properties are described below for each of the available material types.

Steel column properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See: ,
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.

Property	Description
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	Steel
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Design section order *[Only displayed when 'Autodesign' is selected]*	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Assume extra floors supported	Enter the number of extra floors supported.
Simple column	See:

Stack properties

Property	Description
Section	The section size
Grade	Steel grade

Property	Description
Face A cont. rest., Face C cont. rest.	Indicates continuous restraint to Face A and/or Face C. See:
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 571)
Cantilever end top, Cantilever end bottom, *[Only displayed for top and bottom individual stacks]*	Cantilever ends can be applied to the top or bottom stacks as required. When applied, all 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) are fixed at both ends of the stack. See: Column releases (page 571)
Axial load release top	An axial (Fx) release is only allowed at the top of the column. See: Column releases (page 571)
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release. See: Column releases (page 571)
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**

Property	Description
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Splice *[Only displayed for individual stacks above stack 1]*	
Splice, Splice offset	Used to indicate a splice at the bottom of a stack, and the offset above the floor level. See:
Size constraints	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have.
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.

Property	Description
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/ shear capacity ratio in Dir 1 Shear demand/ shear capacity ratio in Dir 2 *[Only displayed for ASCE 7]*	When working to the ASCE 7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE 7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	

Property	Description
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Level properties

Property	Description
Live *[ACI/AISC]*, Imposed *[Other codes]* Load reduction	
KLL *[Only displayed for ACI/AISC regional code]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Count the floor as being supported *[Only displayed for Eurocode, BS or IS regional codes]*	If checked, the floor will be treated as supported when calculating the imposed load reductions. See: Overview of live or imposed load reductions (page 326)
Eccentricity	
Apply face A eccentricity, etc.	See:

Concrete column properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.

Property	Description
Group	The name of the group to which the member belongs. See:
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	Concrete
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See:
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Assume extra floors supported	Enter the number of extra floors supported.

Stack properties

Property	Description
Section	The section size

Property	Description
Concrete type,	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Concrete class *[Eurocode]*, Grade *[Other codes]*	Specifies the concrete class *[Eurocode]*, Grade *[Other codes]*.
Concrete density class *[Eurocode only]*	For lightweight concrete only, specifies the density class.
Alignment	
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 571)
Cantilever end top, Cantilever end bottom, *[Only displayed for top and bottom individual stacks]*	Cantilever ends can be applied to the top or bottom stacks as required. When applied, all 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) are fixed at both ends of the stack. See: Column releases (page 571)
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)

Property	Description
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Analysis & design control	
Apply rigid zones	Unless cleared, rigid zones are automatically created at the connection between the column and any connecting beams.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption. See: Cracked, partially cracked, and uncracked concrete members (page 650)
Design parameters *[Only displayed for the Eurocode regional code]*	
Permanent load ratio option Relative humidity (RH) Age of loading	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. Age of loading is the age at which loading is applied. See:
Confinement reinforcement	

Property	Description
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
Slenderness	
Direction 1, Direction 2	• Braced • Bracing
Direction 1 effective length factor, Direction 2 effective length factor	• Calculated • User input value
Stiffness	
Use slab for calculation upper, lower	For the unrestrained length calculation: • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See: Stiffness
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.

Property	Description
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/ shear capacity ratio in Dir 1 Shear demand/ shear capacity ratio in Dir 2 *[Only displayed for the ASCE 7 code]*	When working to the ASCE 7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE 7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.

Property	Description
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)
Reinforcement	
Rib type - vertical, Class - vertical	Specifies the vertical reinforcement properties
Rib type - link/ confinement, Class - link/ confinement	Specifies the link/confinement properties.

Reinforcement

Property	Description
Rib type - vertical, Class - vertical	Specifies the vertical reinforcement properties
Rib type - link/ confinement, Class - link/ confinement	Specifies the link/confinement properties.

Level properties

Property	Description
Live *[ACI/AISC]*, Imposed *[Other codes]* Load reduction	

Property	Description
KLL *[Only displayed for ACI/AISC regional code]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Count the floor as being supported *[Only displayed for Eurocode, BS or IS regional codes]*	If checked, the floor will be treated as supported when calculating the imposed load reductions. See: Overview of live or imposed load reductions (page 326)
Eccentricity *[Only displayed when fabrication is Precast]*	
Apply face A eccentricity, etc.	See:

Timber column properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	Timber
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:

Property	Description
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Assume extra floors supported	Enter the number of extra floors supported.

Stack properties

Property	Description
Section	The section size
Grade	Timber grade
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 571)
Cantilever end top, Cantilever end bottom, *[Only displayed for top and bottom individual stacks]*	Cantilever ends can be applied to the top or bottom stacks as required. When applied, all 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) are fixed at both ends of the stack. See: Column releases (page 571)
Axial load release top	An axial (Fx) release is only allowed at the top of the column. See: Column releases (page 571)

Property	Description
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release. See: Column releases (page 571)
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Splice *[Only displayed for individual stacks above stack 1]*	
Splice, Splice offset	Used to indicate a splice at the bottom of a stack, and the offset above the floor level. See:
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower

Property	Description
	stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Level properties

Property	Description
Live *[ACI/AISC]*, Imposed *[Other codes]* Load reduction	

Property	Description
KLL *[Only displayed for ACI/AISC regional code]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)

General column properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs.
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	Timber
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y

Stack properties

Property	Description
Section	The section size
Grade	General material grade
Alignment	

Property	Description
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 571)
Cantilever end top, Cantilever end bottom, *[Only displayed for top and bottom individual stacks]*	Cantilever ends can be applied to the top or bottom stacks as required. When applied, all 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) are fixed at both ends of the stack. See: Column releases (page 571)
Axial load release top	An axial (Fx) release is only allowed at the top of the column. See: Column releases (page 571)
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release. See: Column releases (page 571)
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	

Property	Description
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Fire proofing	

Property	Description
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Level properties

Property	Description
Live *[ACI/AISC]*, Imposed *[Other codes]* Load reduction	
KLL *[Only displayed for ACI/AISC regional code]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)

Cold Formed column properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	Steel

Property	Description
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Design section order *[Only displayed when 'Autodesign' is selected]*	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Assume extra floors supported	Enter the number of extra floors supported.
Simple column	See:

Stack properties

Property	Description
Section	The section size
Grade	Cold formed steel grade
Face A cont. rest.,	Indicates continuous restraint to Face A and/or Face C. See:

Property	Description
Face C cont. rest. (steel only)	
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 571)
Cantilever end top, Cantilever end bottom, *[Only displayed for top and bottom individual stacks]*	Cantilever ends can be applied to the top or bottom stacks as required. When applied, all 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) are fixed at both ends of the stack. See: Column releases (page 571)
Axial load release top	An axial (Fx) release is only allowed at the top of the column. See: Column releases (page 571)
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release. See: Column releases (page 571)
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.

Property	Description
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Splice *[Only displayed for individual stacks above stack 1]*	
Splice, Splice offset	Used to indicate a splice at the bottom of a stack, and the offset above the floor level. See:
Size constraints	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have.
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.

Property	Description
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/ shear capacity ratio in Dir 1 Shear demand/ shear capacity ratio in Dir 2 *[Only displayed for ASCE 7]*	When working to the ASCE 7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE 7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	

Property	Description
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Level properties

Property	Description
Live *[ACI/AISC]*, Imposed *[Other codes]* Load reduction	
KLL *[Only displayed for ACI/AISC regional code]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Count the floor as being supported *[Only displayed for Eurocode, BS or IS regional codes]*	If checked, the floor will be treated as supported when calculating the imposed load reductions. See: Overview of live or imposed load reductions (page 326)
Eccentricity	
Apply face A eccentricity, etc.	See:

Cold rolled column properties

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs.

Property	Description
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	Timber
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Gravity only	When selected, the member is designed for gravity combinations only. When unselected, the member is designed for gravity and lateral combinations. See:
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Assume extra floors supported	Enter the number of extra floors supported.

Stack properties

Property	Description
Section	The section size
Grade	Cold rolled steel grade
Face A cont. rest., Face C cont. rest. (steel only)	Indicates continuous restraint to Face A and/or Face C. See:

Property	Description
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 571)
Cantilever end top, Cantilever end bottom, *[Only displayed for top and bottom individual stacks]*	Cantilever ends can be applied to the top or bottom stacks as required. When applied, all 6 degrees of freedom (Fx, Fy, Fz, Mx, My, Mz) are fixed at both ends of the stack. See: Column releases (page 571)
Axial load release top	An axial (Fx) release is only allowed at the top of the column. See: Column releases (page 571)
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release. See: Column releases (page 571)
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	

Property	Description
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Instability factor	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the checkbox located under that stack only.
Fire proofing	
Protected	• When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	

Property	Description
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Level properties

Property	Description
Live *[ACI/AISC]*, Imposed *[Other codes]* Load reduction	
KLL *[Only displayed for ACI/AISC regional code]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)

Brace properties

When a brace is selected, its properties are displayed in the .

The displayed properties are described below for each of the available material types.

Steel brace properties

Property	Description
General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Brace
Active	Clearing this option makes braces inactive in the analysis. See:
Element type	Beam
Material type	Steel

Property	Description
General	
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Design section order *[Only displayed when 'Autodesign' is selected]*	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Property	Description
Span	
Section	The section size
Geometry (automatic design) *[Only displayed if Autodesign is selected]*	The cross section type to be designed

Property	Description
Span	
Grade	Steel grade
Connection	• Bolted • Welded
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements
Relaxation factor *[Only displayed if the brace is tension only]*	Default 0.0. Any value entered is not used unless you also check the 'Use relaxation factors for tension only elements' box in Analysis Options. **NOTE** It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis settings (page 760)
Linearity	Straight
Releases	
Fixity end 1, end 2	• Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Include force in eccentricity moment end 1, end 2	Eccentricity moments in steel and precast columns as a result of beam connection eccentricities do not also consider the brace connection eccentricities unless this property is checked for the appropriate end of the brace. • When checked ON the brace axial force is resolved into a vertical force, multiplied by the connection eccentricity, to obtain a connection eccentricity moment.

Property	Description
	Span
	• When checked OFF the brace axial force does not contribute to the total connection eccentricity moment. See: • Steel column connection eccentricity moments - overview
Wind loading	
Apply open structure wind load	Select this option if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live (Imposed) Load Reduction	
KLL *[Only displayed for Regional Code ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for Codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced

Property	Description
Span	
	• combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Compression	
Effectve length factor y-y, z-z	
Tension	
Net area	Specified as an effective net area or a percentage value.
Size constraints	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm (6in). If so you would simply enter this value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than this width for the design of this beam. See:
Apply max span/depth ratio Max span/ depth ratio	After setting a max span/depth ratio you can check the 'Apply' button for it to be considered by auto-design. During design, only sections which satisfy the maximum ratio limit will be selected. The setting can also be reviewed and/ or copied via Review View > Show/Alter State. See:
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Regional Code in the current release.
End 1 / End 2 *[Only displayed for ACI/AISC regional code]*	
Apply SidePlate connection	Select to apply a side plate connection at the specified end(s).

Property	Description
Span	
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).
Fire proofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Timber brace properties

Property	Description
General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:

Property	Description
General	
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Brace
Active	Clearing this option makes braces inactive in the analysis. See:
Element type	Beam
Material type	Timber
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Property	Description
Span	
Section	The section size
Grade	Timber grade
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-

Property	Description
Span	
	linear analysis. If linear analysis is performed they will be treated as linear elements
Relaxation factor *[Only displayed if the brace is tension only]*	Default 0.0. Any value entered is not used unless you also check the 'Use relaxation factors for tension only elements' box in Analysis Options. **NOTE** It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis settings (page 760)
Linearity	Straight
Releases	
Fixity end 1, end 2	Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Wind loading	
Apply open structure wind load	Select this option if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.

Property	Description
Span	
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live (Imposed) Load Reduction	
KLL *[Only displayed for Regional Code ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for Codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Fire proofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

General brace properties

Property	Description
General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Brace
Active	Clearing this option makes braces inactive in the analysis. See:
Element type	Beam
Material type	Timber
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).

Property	Description
Span	
Section	The section size
Grade	The general material grade
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements
Relaxation factor *[Only displayed if the brace is tension only]*	Default 0.0. Any value entered is not used unless you also check the 'Use relaxation factors for tension only elements' box in Analysis Options.

Property	Description
Span	
	NOTE It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis settings (page 760)
Linearity	Straight
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	
Fixity end 1, end 2	Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Wind loading	
Apply open structure wind load	Select this option if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	

Property	Description
Span	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live (Imposed) Load Reduction	
KLL *[Only displayed for Regional Code ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for Codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Fire proofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Property	Description
Span	
Note File	

Cold formed brace properties

Property	Description
General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Brace
Active	Clearing this option makes braces inactive in the analysis. See:
Element type	Beam
Material type	Cold formed
Construction	The available construction options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Fabrication	The available fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 1427)
Autodesign	See:
Design section order *[Only displayed when 'Autodesign' is selected]*	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).

Property	Description
General	
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 194)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

Property	Description
Span	
Section	The section size
Geometry (automatic design) *[Only displayed if Autodesign is selected]*	The cross section type to be designed
Grade	Cold formed grade
Connection	• Bolted • Welded
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements
Relaxation factor *[Only displayed if the brace is tension only]*	Default 0.0. Any value entered is not used unless you also check the 'Use relaxation factors for tension only elements' box in Analysis Options.

Property	Description
Span	
	NOTE It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis settings (page 760)
Linearity	Straight
Releases	
Fixity end 1, end 2	Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Include force in eccentricity moment end 1, end 2	Eccentricity moments in steel and precast columns as a result of beam connection eccentricities do not also consider the brace connection eccentricities unless this property is checked for the appropriate end of the brace. • When checked ON the brace axial force is resolved into a vertical force, multiplied by the connection eccentricity, to obtain a connection eccentricity moment. • When checked OFF the brace axial force does not contribute to the total connection eccentricity moment. See: • Steel column connection eccentricity moments - overview
Wind loading	
Apply open structure wind load	Select this option if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.

Property	Description
Span	
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live (Imposed) Load Reduction	
KLL *[Only displayed for Regional Code ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for Codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Compression	
Effectve length factor y-y, z-z	
Tension	
Net area	Specified as an effective net area or a percentage value.
Size constraints	
Max depth,	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular

Property	Description
Span	
Min depth, Max width, Min width	size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm (6in). If so you would simply enter this value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than this width for the design of this beam. See:
Apply max span/depth ratio Max span/ depth ratio	After setting a max span/depth ratio you can check the 'Apply' button for it to be considered by auto-design. During design, only sections which satisfy the maximum ratio limit will be selected. The setting can also be reviewed and/ or copied via Review View > Show/Alter State. See:
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Regional Code in the current release.
End 1 / End 2 *[Only displayed for ACI/AISC regional code]*	
Apply SidePlate connection	Select to apply a side plate connection at the specified end(s).
Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Property	Description
Span	
Fire proofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Analysis Element properties

When an analysis element is selected, its properties are displayed in the .

- 'General' properties apply to the whole beam.
- 'All spans' properties are used to rapidly define the same properties for all beam spans.
- Individual 'Span' properties are used to over-ride 'All spans' properties.

General properties

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Analysis Element
Active	Clearing this option makes the Analysis Element inactive in the analysis. See:
Element type	• Beam (page 254) • Truss (page 255) • Tension only (page 255)

Property	Description
	• Compression only (page 255) • Linear axial spring (page 255) • Linear torsional spring (page 256) • Non-linear axial spring (page 256) • Non-linear torsional spring (page 256) • Link (page 256) **NOTE** Tension only elements, Compression only elements, and non-linear axil/torsional springs are all non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements.
Material type	• Steel • Concrete • Timber • General • Cold formed • Cold rolled
Rotation	Rotation of the member about its local x axis.

Span properties

Property	Description
Section	The section size
Grade	Timber grade
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements
Relaxation factor *[Only displayed if the brace is tension only]*	Default 0.0. Any value entered is not used unless you also check the 'Use relaxation factors for tension only elements' box in Analysis Options.

Property	Description
	NOTE It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis settings (page 760)
Linearity	Straight
Releases	
Fixity end 1, end 2	Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Wind loading	
Apply open structure wind load	Select this option if you want open structure wind loads to be calculated. See: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Live (Imposed) Load Reduction	

Property	Description
KLL *[Only displayed for Regional Code ACI/AISC]*	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Reduce imposed loads by *[Displayed for Codes other than ACI/AISC]*	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See: Overview of live or imposed load reductions (page 326)
Fire proofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See:
Exposure	Exposed on all sides
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Base plate properties

When a base plate is selected, its properties are displayed in the according to the region that is being worked to.

Base plate (United States ACI/AISC)

Property	Description
General	
Name	The automatically generated name.

Property	Description
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the base plate is placed.
Base Plate	
Autosize	Indicates whether base plate should be autosized when column section changes
Length	Length of base plate. User defined only when **autosize** is off.
Width	Width of base plate. User defined only when **autosize** is off.
Thickness	Thickness of base plate. User defined only when **autosize** is off.
Grade	Grade of base plate.
Prepared for direct contact in bearing (Eurocode only)	Indicates whether the base plate should be considered to be prepared for direct contact in bearing.
Welded Plate Washers	
Use welded plate washers	Indicates whether to take account of welded plate washers in design of the base plate. (Off by default.)
Plate washer (square) side length, L_{pw}	Side length of plate washer. *Initial* size depends on rod diameter. Not updated automatically, nor validated for clashes or overhang of base plate. Linked to list of Widths in Plate Dimensions section of the database.
Plate washer hole diameter, d_{h,pw}	Diameter of hole in plate washer. *Initial* size depends on rod diameter. Not updated automatically.
Plate washer thickness, t_{pw}	Thickness of plate washer. *Initial* size is 0.5 inch. Not updated automatically. Linked to list of Thicknesses in Plate Dimensions section of the database.
Plate washer grade	Grade of plate for the welded washers.
Rod	
Rod Grade	Rod grade.
Rod Size	Rod size.
Embedded depth	Rod embedded depth.
Friction Coefficient	Coefficient of friction between the base plate and the grout layer
Horizontal Shear Transfer	• Friction Alone • Shear on Bolts Alone

Property	Description
	• Friction and Shear on Bolts • Bearing on Shear Lug • Friction and Bearing on Shear Lug
Rod Layout Type	• Symmetric 4 rods • Symmetric
Edge Distance	Distance of bolt from edge of plate
End Distance	Distance of rod from end of plate
Welds	
Type	• Fillet • Butt
Leg Length	Leg length of Fillet weld.
Full Profile	Indicates whether weld is full or partial.
Partial Profile	Length of partial weld, applicable when **Full Profile** is turned off and is only applicable to weld associated with the web.
Concrete base	
Length	Length of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Width	Width of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Depth	Depth of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Concrete	Material grade of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Grout Space	Grout space between base plate and concrete base.
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Base plate (Eurocode)

Property	Description
General	

Property	Description
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the base plate is placed.
Base Plate	
Autosize	Indicates whether base plate should be autosized when column section changes
Length	Length of base plate. User defined only when **autosize** is off.
Width	Width of base plate. User defined only when **autosize** is off.
Thickness	Thickness of base plate. User defined only when **autosize** is off.
Grade	Grade of base plate.
Prepared for direct contact in bearing	Indicates whether the base plate should be considered to be prepared for direct contact in bearing.
Bolts	
Bolt Grade	Bolt grade.
Bolt Size	Bolt size.
Bolt Length	Bolt length.
Projection above base plate	Length the bolt projects above the base plate.
Friction Coefficient	Coefficient of friction between the base plate and the grout layer
Horizontal Shear Transfer	• Friction Alone • Shear on Bolts Alone • Friction and Shear on Bolts
Bolt Layout Type	• Symmetric 4 Bolts • Symmetric • Asymmetric
Edge Distance	Distance of bolt from edge of plate
End Distance	Distance of bolt from end of plate
Anchor Plate	
Type	• Individual • Combined
Width	Anchor plate width.

Property	Description
Thickness	Anchor plate thickness.
Length	Anchor plate length.
Edge Distance	Anchor plate edge distance, applicable when **Type** is Combined.
Welds	
Type	• Fillet • Butt
Leg Length	Leg length of Fillet weld.
Full Profile	Indicates whether weld is full or partial.
Partial Profile	Length of partial weld, applicable when **Full Profile** is turned off and is only applicable to weld associated with the web.
Concrete base	
Length	Length of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Width	Width of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Depth	Depth of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Concrete	Material grade of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Grout Space	Grout space between base plate and concrete base.
Geometric enhancement parameter, α	Geometric enhancement parameter (EN 1992-1-1 Cl. 6.7) automatically calculated from base plate and concrete base dimensions.
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Base plate (Australia, India, British Standards)

Property	Description
General	
Name	The automatically generated name.

Property	Description
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the base plate is placed.
Base Plate	
Autosize	Indicates whether base plate should be autosized when column section changes
Length	Length of base plate. User defined only when **autosize** is off.
Width	Width of base plate. User defined only when **autosize** is off.
Thickness	Thickness of base plate. User defined only when **autosize** is off.
Grade	Grade of base plate.
Prepared for direct contact in bearing (Eurocode only)	Indicates whether the base plate should be considered to be prepared for direct contact in bearing.
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Member characteristic, construction and fabrication properties

The following table lists the available permutations of characteristic, construction and fabrication for each material type.

Material	Characteristic	Construction	Fabrication	Notes
Concrete	Beam	Concrete beam	Cast-in-place	
			Post tensioned	Not designed
			Precast	Not designed
		Coupling beam	Cast-in-place	Coupling beams are the only beams considered by the Assisted core function. If you set the Construction as a Coupling beam when

Material	Characteristic	Construction	Fabrication	Notes
				part of a core, you have made an engineering decision that you are satisfied the section is sufficiently stiff to provide rigid section behavior - Tekla Structural Designer does not check for this.
	Column	Concrete column	Cast-in-place	
			Precast	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
Steel	Beam	Non-composite beam	Rolled	
			Plated	
			Westok cellular	
			Westok plated	
			Fabsec	Not designed
		Composite beam	Rolled	
			Plated	
			Westok cellular	Not designed
			Westok plated	Not designed
			Fabsec	Not designed
	Column	Non-composite column	Rolled	
			Plated	
			Concrete filled	Not designed
			Concrete encased	Not designed
		Composite column	Rolled	Not designed
			Plated	Not designed
	Brace	Steel brace	Rolled	
	Steel joist	Steel joist	Steel joist	Standard, Special or Girder types
	Gable post	Steel gable post	Rolled	
	Parapet post	Steel Parapet post	Rolled	
	Truss member top	Steel Truss member top	Rolled	

Material	Characteristic	Construction	Fabrication	Notes
	Truss member bottom	Steel Truss member bottom	Rolled	
	Truss member side	Steel Truss member side	Rolled	
	Truss member internal	Steel Truss member internal	Rolled	
	Analysis Element	Not applicable	Not applicable	Not designed
Cold formed	Beam	Cold formed beam member	Cold formed	
	Column	Cold formed column member	Cold formed	
	Brace	Cold formed brace member	Cold formed	
	Gable post	Steel gable post	Cold formed	
	Parapet post	Steel Parapet post	Cold formed	
	Truss member top	Steel Truss member top	Cold formed	
	Truss member bottom	Steel Truss member bottom	Cold formed	
	Truss member side	Steel Truss member side	Cold formed	
	Truss member internal	Steel Truss member internal	Cold formed	
	Analysis Element	Not applicable	Not applicable	Not designed

Material	Characteristic	Construction	Fabrication	Notes
Cold rolled	Beam	Cold rolled beam member	Cold rolled	Not designed
	Column	Cold rolled column member	Cold rolled	Not designed
	Eaves beam	Eaves beam	Cold rolled	Not designed
	Purlin	Purlin	Cold rolled	Not designed
	Rail	Rail	Cold rolled	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
Timber	Beam	Timber beam	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Column	Timber column	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Brace	Timber brace	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Gable post	Timber gable post	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member top	Timber Truss member top	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss membe	Timber Truss	Timber	Not designed
			Glulam	Not designed

Material	Characteristic	Construction	Fabrication	Notes
	r bottom	member bottom	Structural composite lumber	Not designed
	Truss member side	Timber Truss member side	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member internal	Timber Truss member internal	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
General	Beam	General material beam	Not applicable	Not designed
	Column	General material column	Not applicable	Not designed
	Brace	General material brace	Not applicable	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed

Wall and core properties

This section provides more information about some of the wall and core properties you can modify in Tekla Structural Designer.

Click the links below to find out more:

- Bearing wall properties (page 1432)
- Concrete meshed and mid-pier wall properties (page 1434)
- Masonry wall properties (page 1443)
- Shear only wall properties (page 1450)
- General wall properties (page 1452)
- Concrete core properties (page 1457)

Bearing wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Bearing Wall **NOTE** Bearing walls must be rectangular in a vertical plane.
Material Type	Choice of: • Concrete • Timber • General
Assume extra floors supported	Enter the number of extra floors supported.
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Concrete type	While you can apply both normal and lightweight concrete, wall design

All panels	
	using lightweight concrete is currently beyond scope.
Grade	The grades available here are set from the Materials button on the Home ribbon. **NOTE** The grade is only required to specify the correct density to be used in the wall self weight calculation.
Thickness	The thickness of the wall. **NOTE** The thickness is only required for the wall self weight calculation.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release

Wall support	
	• Spring Linear • Spring Non-linear

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels) Base level (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Concrete meshed and mid-pier wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.

General	
Wall Type	Meshed Shear Wall Mid-pier Shear Wall **NOTE** Mid-pier shear walls must be rectangular in a vertical plane. Meshed shear walls can be vertical or sloping. See: • FE meshed walls (page 635) • How meshed walls are represented in solver models (page 554) • How mid-pier walls are represented in solver models (page 559)
Fabrication	Choice of: • Cast-in-place • Precast **NOTE** Design of precast members is beyond scope in the current release
Autodesign	Cleared - the specified reinforcement will be checked during the design process. Selected - reinforcement will be designed during the design process. See:
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See:
Assume extra floors supported	Enter the number of extra floors supported.
Rotation (mid-pier only)	0° - wall spans horizontally 90° - wall spans vertically

General	
AutomaticGenerate Support	Cleared - a support will only be created if the Generate support property is selected. Selected - a support will only be created if no members/ slabs capable of providing support exist under the wall.
Generate support	Cleared - no support is created under the wall. Selected - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
KLL (Head Code ACI/ AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Plane	Indicates the grid along which the wall is placed.

All panels	
Concrete type	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Grade	The concrete grades that are available here are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.

All panels	
End 1 extension	The amount the wall is extended or trimmed back from end 1. A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 202)
End 2 extension	The amount the wall is extended or trimmed back from end 2. A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 202)
Reinforcement Layers	Number of layers of reinforcement to be used in the wall, (1 or 2).
Releases	
Minor Top	Fixed Pinned Continuous (incoming members pinned) **NOTE** The 'Continuous' option is only available for FE meshed walls.
Minor Bottom	Fixed Pinned Continuous (incoming members pinned) **NOTE** The 'Continuous' option is only available for FE meshed walls.
Analysis & design control	
Orthotropic	By default, the wall panels are treated as having isotropic material properties. Selecting this option

All panels	
	allows assigning orthotropic material properties. For further details, see: Orthotropic walls (page 644)
Calculate properties	Only available when the Orthotropic option has been selected. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic wall implementation (page 648)
X factor - along the length of the wall and Y factor along the height of the wall.	Only available when the Calculate properties option has been selected. Applies stiffness modification factors in the local X and Y directions
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when the Calculate properties option is cleared (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, (unless the Orthotropic option has been selected), the modification factors specified in Analysis settings (page 760) are applied. When this option is cleared the modification factors are not applied. See also:
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller modification factors are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift

All panels	
	sensitivity calculations are also influenced by this assumption. See: Cracked, partially cracked, and uncracked concrete members (page 650) The Assume cracked option is not available when the Orthotropic option has been selected.
Design parameters	
Permanent load ratio option Relative humidity (RH) Age of loading	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. Age of loading is the age at which loading is applied. See:
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing

All panels	
	the checkbox located under that panel only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the checkbox located under that panel only.
Confinement reinforcement	
Provide support regions	Selected - confinement reinforcement is designed separately in three regions. Cleared - the same confinement reinforcement is designed for the whole stack.
Slenderness	
Major (Minor)	Braced Bracing
Effective length factor direction Major (Minor)	Calculated User input value
Stiffness	
Use slab for calculation (upper major/ minor, lower major/minor)	For the unrestrained length calculation: • • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See:
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest

All panels	
	concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
Seismic	
In a seismic force resisting system	If this is the case, select the checkbox, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/shear capacity ratio in Dir 1 Shear demand/shear capacity ratio in Dir 2 (ASCE 7 code only)	When working to the ASCE 7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE 7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.
Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off

All panels	
	• When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Reinforcement	
Include end zones	Cleared - the wall is designed without end zones of reinforcement Selected - the wall is designed with end zones of reinforcement
Wall zone	
Form, type class	Specifies the wall zone reinforcement properties
End zone	
Form, type class	Specifies the end zone reinforcement properties (if end zones have been requested).

Panel 1, 2, 3 etc.	
	In a multi-panel wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Intermediate levels) Base level (Head Code Eurocode, BS or IS)	If selected, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing (meshed walls only)	
Override Model's	Select this checkbox to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Available mesh types: • Quad only • Tri only • Quad dominant See: How meshed walls are represented in solver models (page 554)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Masonry wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Meshed Shear Wall **NOTE** For masonry walls the wall type can only be a meshed shear wall. See: • FE meshed walls (page 635) • How meshed walls are represented in solver models (page 554)
AutomaticGenerate Support	Cleared - a support will only be created if the Generate support property is selected. Selected - a support will only be created if no members/slabs capable of providing support exist under the wall.
Generate support	Cleared - no support is created under the wall. Selected - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE 7-05/ASCE 7-10. See: Overview of live or imposed load reductions (page 326)
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The masonry grades that are available here are set from the Materials button on the Home ribbon.
Thickness	The actual thickness of the wall (not nominal).
Grout	• Fully grouted • Partially grouted

All panels	
	• Ungrouted
Grout spacing	
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
End 1 extension	The amount the wall is extended or trimmed back from end 1. A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 202)
End 2 extension	The amount the wall is extended or trimmed back from end 2. A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 202)
Releases	
Minor Top	Fixed Pinned Continuous (incoming members pinned)
Minor Bottom	Fixed Pinned Continuous (incoming members pinned)
Analysis & design control	
Orthotropic	By default, the wall panels are treated as having isotropic material

All panels	
	properties. Selecting this option allows assigning orthotropic material properties. For further details, see: Orthotropic walls (page 644)
Calculation	Only available when the Orthotropic option has been selected. • No calculation • Apply X and Y factors - Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic wall implementation (page 648) • Apply section adjustment factors
X factor - along the length of the wall. Y factor - along the height of the wall.	These are only available when the **Apply X and Y factors** Calculation option has been selected. Applies stiffness modification factors in the local X and Y directions
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Material properties, such as Young's moduli, shear moduli, and the Poisson's ratio can be manually set using these fields. They are only editable for **Othotropic** walls when the **No calculation** Calculation option is selected (except □y, which remains read-only). They are not initially displayed when the **Apply section adjustment factors** Calculation option has been selected, but are populated once the analysis is complete. **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, (unless the Orthotropic option has been selected), the

All panels	
	modification factors specified in Analysis settings (page 760) are applied. When this option is cleared the modification factors are not applied. See also:
Assume cracked	Assuming masonry wall sections are cracked has a direct affect on the analysis; smaller modification factors are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption. See: Cracked, partially cracked, and uncracked concrete members (page 650) The Assume cracked option is not available when the Orthotropic option has been selected.
Sway and drift checks	
Drift/Seismic drift checks and consider for seismic torsion	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing the checkbox located under that panel only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this

All panels	
	calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the checkbox located under that panel only.
Slenderness	
Major / Minor	Braced Bracing **NOTE** The bracing and effective length properties of a masonry wall are taken into account when calculating it's effective length, in the same way they are for a reinforced concrete wall (see: . The calculated values are then passed on to Tedds to be used in the design of the wall, where applicable.
Major / Minor effective length factor	Calculated User input value
Stiffness	
Use slab for calculation (upper major/ minor, lower major/minor)	For the unrestrained length calculation: • • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See:

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Panel 1, 2, 3 etc.	
	In a multi-panel wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing	
Override Model's	Select this checkbox to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Available mesh types: • Quad only • Tri only

Meshing	
	• Quad dominant See: How meshed walls are represented in solver models (page 554)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Shear only wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Shear Only Wall **NOTE** Shear only walls must be rectangular in a vertical plane.
Material Type	Choice of: • Concrete • Timber • General
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-

General	
	pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The grades available here are set from the Materials button on the Home ribbon. **NOTE** The grade is only required to specify the correct density to be used in the wall self weight calculation.
Thickness	The thickness of the wall. **NOTE** The thickness is only required for the wall self weight calculation.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Spring stiffness	The stiffness to be used in the analysis. See: How shear only walls are represented in solver models (page 561)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

General wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Meshed Shear Wall
Material type	General
Assume extra floors supported	Enter the number of extra floors supported.
AutomaticGenerate Support	Cleared - a support will only be created if the Generate support property is selected. Checked - a support will only be created if no members/slabs capable of providing support exist under the wall.
Generate support	Cleared - no support is created under the wall. Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The grades that are available for General materials are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller modification factors are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption.
End 1 extension	The amount the wall is extended or trimmed back from end 1. A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 202)
End 2 extension	The amount the wall is extended or trimmed back from end 2. A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 202)
Releases	

All panels	
Minor Top Minor Bottom	Fixed Pinned Continuous (incoming members pinned)
Analysis & design control	
Orthotropic	By default, the slab item is treated as having isotropic material properties. Selecting this option allows assigning orthotropic material properties. For further details, see: Orthotropic walls (page 644)
Calculate properties	Only available when the Orthotropic option has been selected. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic wall implementation (page 648)
X factor - along the length of the wall Y factor along the height of the wall	Only available when the Calculate properties option has been selected. Applies stiffness modification factors in the local X and Y directions
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when the Calculate properties option is cleared (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, (unless the Orthotropic option has been selected), the modification factors specified in Analysis settings (page 760) are applied. When this option is cleared

All panels	
	the modification factors are not applied. See also:
Sway and drift checks	
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing the checkbox located under that panel only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the checkbox located under that panel only.

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.

Wall support	
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Panel 1, 2, 3 etc.	
	In a multi-panel wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Intermediate levels) Base level (Head Code Eurocode, BS or IS)	If selected, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing	
Override Model's	Select this checkbox to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Available mesh types: • Quad only • Tri only

Meshing	
	• Quad dominant See: How meshed walls are represented in solver models (page 554)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Concrete core properties

General	
Name	The automatically generated name for the core.
User Name	Can be used to override the automatically generated name if required.
Rotation	Defines the axis system for reporting the core results: • Dir 1/2 - Main Building Directions • Principal 1/2 - Major and minor local axis • Angle - in Global Coordinate System

Level 1, Level 2 etc.	
Centroid above, Centroid below	Every core will have at least 2 core levels, with a core section for each level. The centroids of the core sections are reported above and below each level.

Slab and mat properties

This section provides more information about some of the slab and mat properties you can modify in Tekla Structural Designer.

- Slab item properties (page 1458)
- Slab/Mat overhang properties (page 1479)
- Patch properties (page 1479)
- Foundation mat properties (page 1482)

- Punching check properties (page 1486)
- Result strip properties (page 1491)

Slab item properties

When a slab item is selected, its properties are displayed in the Properties window (page 61).

- Properties located under 'Slab general' and 'Slab parameters' headings apply to all slab items in the same slab as the selected slab item.
- Properties located under 'General' and all remaining headings apply to the selected slab item only.

The displayed properties are described below for each of the available slab types.

Precast concrete slab item properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	This property specifies: • Span direction for 1-way load decomposition • 2D solver element local axes in the solver model Different angles can be specified for individual slab items within the slab. **NOTE** When SpanDirection is switched on in Scene Content, the rotation angle is represented by a blue arrow. See: and Modify slab/panel span direction (page 231)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Slab type	Precast

Slab general	
Deck type	When the slab type is 'Precast', the deck type is 'Precast concrete plank's
Decomposition	When the slab type is 'Precast', decomposition is 'One-way'
Analyze slab	This option is only displayed for one-way decomposition slabs. When selected, the 1-way slab is enabled for finite element analysis in load decomposition, FE chasedown and optionally for 3D analysis. This option also activates additional **Analysis parameters** in the Slab Item properties to allow orthotropic properties to be specified. These parameters include a Y-factor defaulted to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). For further details, see: Define an orthotropic slab using relative stiffness factors (page 602)

Slab parameters	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class *[Eurocode]*, Grade *[Other codes]*	Specifies the concrete class *[Eurocode]*, Grade *[Other codes]*.
Concrete density class *[Eurocode only]*	For lightweight concrete only, specifies the density class.
Dry density, Dry density per area	Specifies the dry concrete density and dry concrete density per area
Wet density, Wet density per area	Specifies the wet concrete density and wet concrete density per area
Long term elastic modulus = Ecm divided by...	Factor by which the short term elastic modulus is divided to obtain the long term modulus .
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid

Slab parameters	
	• None
Decking properties	
Country, Manufacturer, Reference, Gauge	Defines properties of the precast concrete planks.
Reinforcement in slab for crack control or fire requirements *[Only displayed for precast slab if topping has been specified]*	
Type, Rib type, Mesh type Class, Bar size	Defines reinforcement for crack control or fire requirements
Topping details	
Topping	• None • Non-structural • Structural
Depth	Depth of topping when topping other than 'None' is selected.
Analysis parameters *[Only displayed for precast slab if 'Analyze slab' has been specified]*	
Orthotropic	By default, the slab item is treated as having isotropic material properties. Checking this option allows assigning orthotropic material properties. For further details, see: Orthotropic slabs (page 601)
Calculate properties	Only available when 'Orthotropic' is checked. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic slab implementation (page 605)
X factor and Y factor	Only available when 'Calculate properties' is checked. Applies stiffness modification factors in the local X and Y directions For 1-way slabs the Y-factor defaults to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). **NOTE** Applying the same stiffness modification factors in both directions is an alternative way of specifying stiffness adjustments of isotropic slab items.

Slab parameters	
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when 'Calculate properties' is unchecked (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, the modification factors are applied. Unchecking this option ignores the modification factors for the slab item. See: and Analysis settings (page 760)

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Composite slab item properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	This property specifies: • Span direction for 1-way load decomposition • 2D solver element local axes in the solver model Different angles can be specified for individual slab items within the slab.

General	
	NOTE When SpanDirection is switched on in Scene Content, the rotation angle is represented by a blue arrow. See: and Modify slab/panel span direction (page 231)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all slab items in a slab adopt the same depth. Checking this option allows the selected slab item to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Slab type	Composite slab
Deck type	When the slab type is 'Composite slab', the deck type is 'Profiled metal decking'.
Decomposition	When the slab type is 'Composite slab', decomposition is 'One-way'
Analyze slab	This option is only displayed for one-way decomposition slabs. When selected, the 1-way slab is enabled for finite element analysis in load decomposition, FE chasedown and optionally for 3D analysis. This option also activates additional **Analysis parameters** in the Slab Item properties to allow orthotropic properties to be specified. These parameters include a Y-factor defaulted to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). For further details, see: Define an orthotropic slab using relative stiffness factors (page 602)

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class *[Eurocode]*, Grade *[Other codes]*	Specifies the concrete class *[Eurocode]*, Grade *[Other codes]*.
Concrete density class *[Eurocode only]*	For lightweight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Long term elastic modulus = Ecm divided by... *[Displayed for composite slab and precast only]*	Factor by which the short term elastic modulus is divided to obtain the long term modulus .
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid • None
Ponding allowance option, Ponding allowance value	If required, choose either value or percentage and then specify actual ponding allowance accordingly.
Apply ponding to construction stage	On = ponding allowance applied at construction stage.
Apply ponding to composite stage	On = ponding allowance applied at composite stage
Decking properties (where applicable)	
Country, Manufacturer,	Defines properties of decking (where applicable).

Slab parameters	
Reference, Gauge	
Reinforcement in slab for crack control or fire requirements	
Type, Rib type, Mesh type Class, Bar size	Defines reinforcement for crack control or fire requirements
Analysis parameters *[Only displayed for composite slab if 'Analyze slab' has been specified]*	
Orthotropic	By default, the slab item is treated as having isotropic material properties. Checking this option allows assigning orthotropic material properties. For further details, see: Orthotropic slabs (page 601)
Calculate properties	Only available when 'Orthotropic' is checked. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic slab implementation (page 605)
X factor and Y factor	Only available when 'Calculate properties' is checked. Applies stiffness modification factors in the local X and Y directions For 1-way slabs the Y-factor defaults to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). **NOTE** Applying the same stiffness modification factors in both directions is an alternative way of specifying stiffness adjustments of isotropic slab items.
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when 'Calculate properties' is unchecked (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, the modification factors are applied. Unchecking this option ignores the modification factors for the slab item. See: and Analysis settings (page 760)

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Flat slab properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
X direction	This property specifies: • 2D solver element local axes in the solver model • Bar direction Different angles can be specified for different panels within the slab. **NOTE** When SpanDirection is switched on in Scene Content, the X direction is represented by a blue arrow. See: and Modify slab/panel span direction (page 231)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it.

General	
	NOTE Vertical offsets are not structurally significant
Auto-design	For slab items in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For slab items not in Autodesign mode, the result will be a pass or fail.
Select bars starting from	This option controls the starting point for auto-design procedures.
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Slab type	Flat slab
Deck type	'Reinforced concrete', or 'Post tension'. 'Post tension' slabs can not be designed.
Decomposition	When the slab type is 'Flat slab', decomposition is 'Two-way'

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class *[Eurocode]*, Grade*[Other codes]*	Specifies the concrete class *[Eurocode]*, Grade*[Other codes]*.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density, Dry density per area	Specifies the dry concrete density and dry concrete density per area
Wet density, Wet density per area	Specifies the wet concrete density and wet concrete density per area
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid

Slab parameters	
	• Semi-rigid • None
Analysis parameters	
Orthotropic	By default, the slab item is treated as having isotropic material properties. Checking this option allows assigning orthotropic material properties. For further details, see: Orthotropic slabs (page 601)
Calculate properties	Only available when 'Orthotropic' is checked. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic slab implementation (page 605)
X factor and Y factor	Only available when 'Calculate properties' is checked. Applies stiffness modification factors in the local X and Y directions For 1-way slabs the Y-factor defaults to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). **NOTE** Applying the same stiffness modification factors in both directions is an alternative way of specifying stiffness adjustments of isotropic slab items.
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when 'Calculate properties' is unchecked (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, the modification factors are applied. Unchecking this option ignores the modification factors for the slab item. See: and Analysis settings (page 760)
Design parameters	
Permanent load ratio *[Displayed for Eurocode only]*	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.

Slab parameters	
Maximum crack width *[Not displayed for ACI/AISC Regional Code]*	The maximum center to center bar spacing for crack control is dependent on the maximum allowable crack width that is specified here.
Deflection parameters	
Restraint constant *[Displayed for ACI/AISC Regional Code only]*	See:
Restraint type *[Displayed for Eurocode only]*	See:
Cement class *[Displayed for Eurocode only]*	See:

Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
Reduce live *[ACI/AISC]*, imposed *[Other codes]* loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
Top bars, Bottom bars	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to “none” if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)
Top cover, Bottom cover	Specifies the top and bottom cover to reinforcement.

Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Slab on beams properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle (one-way slabs) or X direction (two way slabs)	This property is used for the following where appropriate: • Span direction for 1-way load decomposition • To determine the 2D solver element local axes in the solver model • Bar direction for Slab on Beam and Flat Slabs. Different angles can be specified for different panels within the slab. **NOTE** When SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow. See: and Modify slab/panel span direction (page 231)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Auto-design	For slab items in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For slab items not in Autodesign mode, the result will be a pass or fail.
Select bars starting from	This option controls the starting point for auto-design procedures.
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the slab item.

Slab general	
User Name	Can be used to override the automatically generated name if required.
Slab type	Slab on beams
Deck type	'Reinforced concrete', or 'Post tension'. 'Post tension' slabs can not be designed.
Decomposition	When the slab type is 'Slab on beams', decomposition can be either 'One-way' or 'Two-way'
Analyze slab	This option is only displayed for one-way decomposition slabs. When selected, the 1-way slab is enabled for finite element analysis in load decomposition, FE chasedown and optionally for 3D analysis. This option also activates additional **Analysis parameters** in the Slab Item properties to allow orthotropic properties to be specified. These parameters include a Y-factor defaulted to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). For further details, see: Define an orthotropic slab using relative stiffness factors (page 602)

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class *[Eurocode]*, Grade*[Other codes]*	Specifies the concrete class *[Eurocode]*, Grade*[Other codes]*.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density, Dry density per area	Specifies the dry concrete density and dry concrete density per area
Wet density, Wet density per area	Specifies the wet concrete density and wet concrete density per area
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid

Slab parameters	
	• Semi-rigid • None
Analysis parameters Only displayed if 'Decomposition' is 'Two-way, (or 'One-way' with 'Analyze slab' also selected).	
Orthotropic	By default, the slab item is treated as having isotropic material properties. Checking this option allows assigning orthotropic material properties. For further details, see: Orthotropic slabs (page 601)
Calculate properties	Only available when 'Orthotropic' is checked. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic slab implementation (page 605)
X factor and Y factor	Only available when 'Calculate properties' is checked. Applies stiffness modification factors in the local X and Y directions For 1-way slabs the Y-factor defaults to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). **NOTE** Applying the same stiffness modification factors in both directions is an alternative way of specifying stiffness adjustments of isotropic slab items.
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when 'Calculate properties' is unchecked (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, the modification factors are applied. Unchecking this option ignores the modification factors for the slab item. See: and Analysis settings (page 760)
Design parameters	

Slab parameters	
Permanent load ratio *[Displayed for Eurocode only]*	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width *[Not displayed for ACI/AISC Regional Code]*	The maximum center to center bar spacing for crack control is dependent on the maximum allowable crack width that is specified here.
Deflection parameters	
Restraint constant *[Displayed for ACI/AISC Regional Code only]*	See:
Restraint type *[Displayed for Eurocode only]*	See:
Cement class *[Displayed for Eurocode only]*	See:

Design parameters	
Adjustment ratio direction X ,Adjustment ratio direction Y (slab on beam only)	These factors are applied to the enclosing lengths in X and Y in order to manually adjust the X and Y direction span in the span-effective depth check. See Slab on beam idealized panels
Enclosing length X, Enclosing length Y (slab on beam only)	The automatically calculated span length in the X direction, and Y direction. See Slab on beam idealized panels
Adjusted length X, Adjusted length X (slab on beam only)	The adjusted span lengths in the X and Y directions. See Slab on beam idealized panels

Design parameters	
Edge category start X, Edge category start Y (slab on beam only)	The assumed support condition at the start of the span in the X direction, and Y direction. See Slab on beam idealized panels
Edge category end X, Edge category end Y (slab on beam only)	The assumed support condition at the end of the span in the X direction, and Y direction. See Slab on beam idealized panels
Average stiffness ratio (Slab on beam ACI/AISC only)	In the current release of Tekla Structural Designer the average stiffness ratio is a user defined value. (Default value = 1.0). The way in which the minimum thickness is calculated directly depends on this value.

Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
Reduce live *[ACI/AISC]*, imposed *[Other codes]* loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
Top bars, Bottom bars	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to “none” if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)

Reinforcement	
Top cover, Bottom cover	Specifies the top and bottom cover to reinforcement.

Utilization ratio	
Apply (to autodesign)	Selected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Unselected - When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	Selected - When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Unselected - When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

General properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle (one-way slabs)	This property is used for the following where appropriate: • Span direction for 1-way load decomposition

General	
or X direction (two way slabs)	• To determine the 2D solver element local axes in the solver model • Bar direction for Slab on Beam and Flat Slabs. Different angles can be specified for different panels within the slab. **NOTE** When SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow. See: and Modify slab/panel span direction (page 231)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Slab type	Slab on beams
Deck type	'General'.
Decomposition	When the slab type is 'General', decomposition can be either 'One-way' or 'Two-way'
Analyze slab	This option is only displayed for one-way decomposition slabs. When selected, the 1-way slab is enabled for finite element analysis in load decomposition, FE chasedown and optionally for 3D analysis. This option also activates additional **Analysis parameters** in the Slab Item properties to allow orthotropic properties to be specified. These parameters include a Y-factor defaulted to 0.01 in order to create a slab that performs in a

Slab general	
	highly 1-way manner (100 times weaker transverse to the intended span direction). For further details, see: Define an orthotropic slab using relative stiffness factors (page 602)

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid • None
Analysis parameters Only displayed if 'Decomposition' is 'Two-way, (or 'One-way' with 'Analyze slab' also selected).	
Orthotropic	By default, the slab item is treated as having isotropic material properties. Checking this option allows assigning orthotropic material properties. For further details, see: Orthotropic slabs (page 601)
Calculate properties	Only available when 'Orthotropic' is checked. Allows setting material properties in two in-plane directions automatically based on the stiffness modification X and Y factors below. For details of how the parameters are calculated, see: Orthotropic slab implementation (page 605)
X factor and Y factor	Only available when 'Calculate properties' is checked. Applies stiffness modification factors in the local X and Y directions For 1-way slabs the Y-factor defaults to 0.01 in order to create a slab that performs in a highly 1-way manner (100 times weaker transverse to the intended span direction). **NOTE** Applying the same stiffness modification factors in both directions is an alternative way of specifying stiffness adjustments of isotropic slab items.

Slab parameters	
Ex, Ey, Gxy, Gxz | Gyz, □x, and □y	Allows manually setting material properties, such as Young's moduli, shear moduli, and the Poisson's ratio. The fields become editable when 'Calculate properties' is unchecked (except □y, which remains read-only). **NOTE** No distinction is made between Gxz and Gyz out-of-plane shear moduli.
Apply modification factors	By default, the modification factors are applied. Unchecking this option ignores the modification factors for the slab item. See: and Analysis settings (page 760)
Material properties	
Grade	Allows you to select the general material from the list of general materials in the materials database.
Override material properties	Select this option to define material properties directly instead of selecting from the database.

Live *[ACI/AISC]*, Imposed *[Other codes]* Load Reduction	
Reduce live *[ACI/AISC]*, imposed *[Other codes]* loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Slab/Mat overhang properties

General	
LengthofBeam	• When checked, the overhang is created by a single click on a supporting beam along the the slab edge. • When unchecked, the overhang is created by clicking two points along the slab edge.
EdgeParallel	• When checked, the overhang is created with a straight edge. • When unchecked, the overhang is created with a curved edge.
Curvature	Specifies the overhang curvature, (when EdgeParallel is unchecked).
Tapered	• When checked, the overhang varies in width. (Width1 to Width2) • When unchecked, the overhang is a constant width (Width1).
Width1	The width at end 1 of the overhang.
Width2	The width at end 2 of the overhang, (when tapered).

Patch properties

General	
Name	The automatically generated name for the patch.
User Name	Can be used to override the automatically generated name if required.
Type	• Column • Beam • Wall
Lx	Specifies the size of the patch in the X direction.

General	
Ly	Specifies the size of the patch in the Y direction.
Associated Slab Panel	Where a patch sits over more than one panel then an automatic choice is made, but you are able to override this if required by selecting one of the alternative panels from a drop-down list.
Align to Panel Reinforcement	When this checkbox is selected, the calculated strip reinforcement is aligned with the background reinforcement in the Associated Slab Panel. **NOTE** This option cannot be activated unless all slab items below the patch are aligned within a tolerance of 10 degrees.
Local X Angle	The angle of the X axis reinforcement is only editable if Align to Panel Reinforcement is cleared.
Surface	Specifies the reinforcement to be associated with and designed by the patch: • Top or Bottom (cannot be both).
Autodesign	This setting applies to all strips in both directions, but reinforcement in either direction can still be set to none - see below. • Cleared - the specified reinforcement will be checked during the design process. • Checked - reinforcement will be designed during the design process.
Select bars starting from	This option controls the starting point for auto-design procedures. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement.
Consider Strips	This setting controls which strips are to be designed by the patch. • X • Y • X and Y
Consider patch surface moments only	Off (default) • patch reinforcement is designed in the surface of the slab at which the patch has been specified AND the slab reinforcement is also checked in the opposite surface. On

General	
	• patch reinforcement is designed in the surface of the slab at which the patch has been specified, but the slab reinforcement in the opposite surface is NOT checked. **NOTE** The only situation when you might need to activate this option is if the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face.(When the other patch is designed additional reinforcement will be provided at that surface, so this check is not required for the original patch).

Strips in X, Strips in Y	
Center, Left and Right Strip	
Width	The left and right strip widths can be specified independently: the center strip width is recalculated accordingly and cannot be edited. (By default the center strip covers half the panel, so that the left and right strips each cover a quarter of the panel.)
Design Force	• Average (of all the FE nodal values within the strip). • Maximum (of all the FE nodal values within the strip).
Reinforcement	
Combine with Panel Reinforcement	When this checkbox is selected, the calculated strip reinforcement takes into account any existing panel reinforcement in the Associated Slab Panel that is in the same alignment as the strip.
Cover as Panel	When this checkbox is selected, the cover is set to be the same as that in the Associated Slab Panel.
Outer Bar Direction as Panel	When this checkbox is selected, the outer bar direction is set to be the same as that in the Associated Slab Panel. When cleared, the outer bar direction can be set in X or Y.

Reinforcement	
Combine with Panel Reinforcement	When this checkbox is selected, the calculated strip reinforcement takes into account any existing panel reinforcement in the Associated Slab Panel that is in the same alignment as the strip.
Cover as Panel	When this checkbox is selected, the cover is set to be the same as that in the Associated Slab Panel.

Reinforcement	
Outer Bar Direction as Panel	When this checkbox is selected, the outer bar direction is set to be the same as that in the Associated Slab Panel. When cleared, the outer bar direction can be set in X or Y.
Reinforcement	This setting is used to specify whether bars or mesh are to be used in each direction. • Mesh • Bars XY • Bars X • Bars Y • None (If Mesh is selected an extra setting then allows you to specify if main bars are in X or Y.)

Reinforcement in X, Reinforcement in Y, or Mesh	
Bar Size, spacing, Mesh type etc.	The actual reinforcement provided in each of the strips is indicated here.

Foundation mat properties

General	
Name	The automatically generated name for the mat item.
User Name	Can be used to override the automatically generated name if required.
Foundation type	Mat
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the mat. See:
Include in diaphragm	If this option is cleared, the mat item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.

General	
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Auto-design	For panels in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Plane	Indicates the level at which the mat is placed.

Slab general	
Name	The parent foundation mat name.
User Name	Can be used to override the automatically generated name if required.
Slab type	***Foundation mat***
Deck type	• Reinforced concrete • Post tension **NOTE** Design of post tensioned slabs is beyond scope in the current release.
Decomposition	Two-way

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Grade/Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab.

Slab parameters	
	• Rigid • Semi-rigid • None
Design parameters	
Permanent load ratio *[Displayed for Eurocode only]*	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width *[Not displayed for ACI/AISC Regional Code]*	The maximum center to center bar spacing for crack control is dependent on the maximum allowable crack width that is specified here.
Deflection parameters	
Restraint constant	See:
Restraint type (Eurocode only)	See:
Cement class (Eurocode only)	See:
Soil parameters	
Use Ground Bearing Springs	Select to specify that the mat is ground bearing.
Allowable Bearing Pressure	The allowable bearing pressure is only required when the 'Use Ground Bearing Springs' option is selected.
Ground Stiffness The following properties are only displayed when the 'Use Ground Bearing Springs' option is selected.	
Type	• Spring Linear • Spring Non-Linear
Stiffness	Only displayed when the Type is set to 'Spring Linear'.
Stiffness -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Tension limit -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Stiffness +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Compression limit +ve	Only displayed when the Type is set to 'Spring Non-Linear'.

Slab parameters	
Horizontal Support	• Fixed • Free • Spring
% of vertical spring stiffness	This field is displayed when the Horizontal Support is set to 'Spring'.

Live/Imposed load reduction	
Reduce live/ imposed loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to "none" if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off

Utilization ratio	
	• When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Punching check properties

General	
Tension Reinforcement	This setting identifies the slab reinforcement to be used in the punching check calculation. • Top • Bottom
Center	The check location (not editable).
Column Drop	Indicates if the check considers a Column Drop (not editable).

General	
Beta - User limit (Head Code Eurocode)	When this checkbox is selected, a minimum value of Beta = 1.15 is applied to all internal columns.
User factor for Vt (Head Code BS)	When this checkbox is selected, the user factor for Vt is applied.
u0 - user reduction	Can be used to manually specify a reduction in the length of the u0 perimeter to account for undefined openings.
u1 - user reduction	Can be used to manually specify a reduction in the length of the u1 perimeter to account for undefined openings.
Check Status and Ratio	Indicates the status of the checks for each calculated perimeter and the overall check ratio.

Loaded Perimeter	
Length	Indicates the length of the u0 Loaded Perimeter (not editable).
Reduced length	The reduced length of u0 after accounting for openings.
BEquiv, DEquiv, BBound, BBound, Bounding Perimeter	Refer to the Concrete Design Reference Guide for the current Head Code for the appropriate definition of these terms.
d Effective Depth	Indicates the average effective depth to the tension reinforcement (not editable). d = (dy + dz) / 2 where dy and dz are the effective depths in the two orthogonal directions. There is a value of d for top steel and a different value for bottom steel. Note this definition changes in the presence of a drop panel. This information is only available if the reinforcement is known in each direction.
Slab Override	When an override is applied the slabs in each direction can be de-activated in the check. In this way the Loaded Perimeter Position can be edited. **NOTE** In the typical case of punching checks around a column, the slab 'y' & 'z', 'positive' & 'negative' are defined by the local axis system of the column. This can be displayed by displaying the Local Axes for 1D Elements in Scene Content.

Control Perimeter	
Length	Indicates the length of the u1 Control Perimeter (not editable).
Reduced length	The reduced length of u1 after accounting for openings.
[-] Design Input (Head Code Eurocode and ACI)	
Use Reinforcement	Select this checkbox in order to apply a default punching reinforcement arrangement that can then either be checked, or used as the starting reinforcement for an auto-design. You would only choose to clear the checkbox when specifying a new check if you want to perform an auto-design but starting from Minima.
Reinforcement type	In the current release only Stud reinforcement is considered.
Arrangement type	The options are: • Orthogonal (default) • Circular
Auto-design	When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.
Select reinforcement starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is 'on'. • Minima - removes the current arrangement and begins with the minimum allowed bar size. • Current - auto-design commences from the current bar arrangement.
Rib type	Specifies the reinforcement rib type.
Grade	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size	The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.
Spacing	Defines the spacing between bars along each rail.
Spacing from column face	Defines the spacing of the first bar in each rail from the column face.
Stud rails spacing in Y direction	Spacing between rails in the local Y direction.

Control Perimeter	
Stud rails spacing in Z direction	Spacing between rails in the local Z direction.
Number of diagonal stud rails on one corner	The number of stud rails adjacent to each corner of the column. (This property is only displayed when the 'Arrangement Type' is Circular)
Number of studs per column face - Y direction	The number of stud rails adjacent to the column face in the local Y direction.
Number of studs per column face - Z direction	The number of stud rails adjacent to the column face in the local Z direction.
Number of studs per rail	Number of studs on each rail.

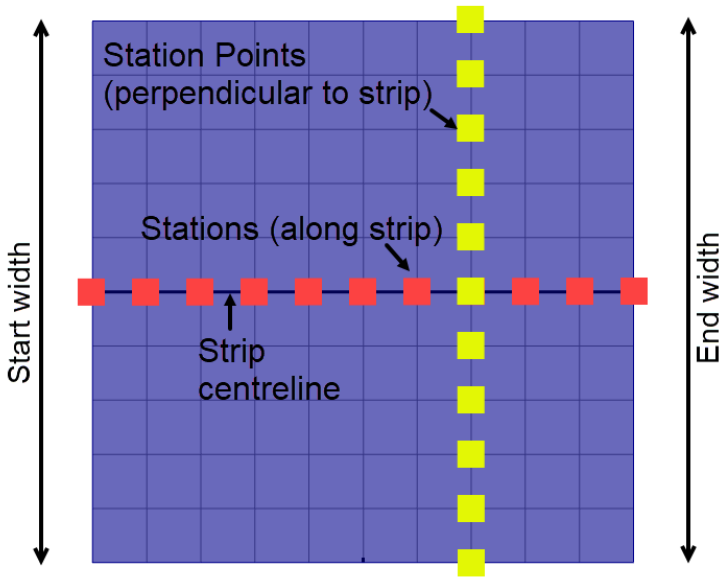
Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 797)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Design Input (Eurocode and ACI only)	
Use Reinforcement	Select this checkbox in order to apply a default punching reinforcement arrangement that can then either be

Design Input (Eurocode and ACI only)	
	checked, or used as the starting reinforcement for an auto-design. You would only choose to clear the checkbox when specifying a new check if you want to perform an auto-design but starting from Minima.
Reinforcement type	In the current release only Stud reinforcement is considered.
Arrangement type	The options are: • Orthogonal (default) • Circular
Auto-design	When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.
Select reinforcement starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is 'on'. • Minima - removes the current arrangement and begins with the minimum allowed bar size. • Current - auto-design commences from the current bar arrangement.
Rib type	Specifies the reinforcement rib type.
Grade	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size	The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.
Spacing	Defines the spacing between bars along each rail.
Spacing from column face	Defines the spacing of the first bar in each rail from the column face.
Stud rails spacing in Y direction	Spacing between rails in the local Y direction.
Stud rails spacing in Z direction	Spacing between rails in the local Z direction.
Number of diagonal stud rails on one corner	The number of stud rails adjacent to each corner of the column. (This property is only displayed when the 'Arrangement Type' is Circular)
Number of studs per	The number of stud rails adjacent to the column face in the local Y direction.

Design Input (Eurocode and ACI only)	
column face - Y direction	
Number of studs per column face - Z direction	The number of stud rails adjacent to the column face in the local Z direction.
Number of studs per rail	Number of studs on each rail.

Result strip properties

General	
Name	The automatically generated name for the strip.
User Name	Can be used to override the automatically generated name if required.
	
Start Width	The total strip width at the first point picked when creating the strip.
End Width	The total strip width at the second point picked when creating the strip.

General	
Result Type	Determines how the strip result is calculated: • **Average** - average of the station point values. • **Maximum** - maximum of: 1. All station point values 2. Any FE solver node found within the station area • **Centreline** - the station point value on the strip centreline. For more details, see result strips Overview (page 693)
Number of Stations	The number of stations per metre along the strip.
Number of Points	The number of points per metre across the strip at each station.
total Strip Area	Automatically calculated from the start and end widths and strip length.

Portal frame properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Autodesign	Cleared - the specified reinforcement will be checked during the design process. Selected - reinforcement will be designed during the design process. See:

All rafters	
Section	The section size
Grade	Steel grade
Design section order	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
UDA	

All rafters	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

All columns	
Section	The section size
Grade	Steel grade
Design section order	The design order file from which a section size will be selected. For details of managing order files, see: Manage design section orders (page 1272)
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing the checkbox located under that panel only.

All columns	
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the checkbox located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the checkbox located under that panel only.
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Support 1, 2, 3 etc.	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the grid along which the portal frame is placed.
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Support 1, 2, 3 etc.	
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Span 1, 2, 3 etc.	
Include end zones	In a multi-span frame rafter and column properties can be entered for a specific span, over-riding those defined at the **All rafters** and **All columns** levels.

Deflection limits (Eurocode only)	
Horizontal deflection limit option	The limit can be expressed as a height over ratio, or absolute limit
Horizontal deflection limit	The limit to apply for the selected option.
Vertical deflection limit option	The limit can be expressed as a height over ratio, or absolute limit
Vertical deflection limit	The limit to apply for the selected option.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Pad base strip base and pile cap properties

Property	Description
General	
Foundation Type	Isolated Pad Base (not editable). Isolated Pile Cap (not editable).
Auto-design depth	• When Autodesign is selected an iterative procedure is used to determine the depth. If the shear design fails the depth is increased until either the check passes, or the maximum depth specified in Design Options has been reached. • When Autodesign is not selected (i.e. check mode), the existing depth is retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base, the depth will be checked or designed according to the auto-design depth setting.
Auto-design size	• When Autodesign is selected an iterative procedure is used to determine the base size. If the bearing check fails the size is increased until either the check passes, or the maximum side length specified in Design Options has been reached. • When Autodesign is not selected (i.e. check mode), the existing size is

Property	Description
	retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base, the size will be checked or designed according to the Auto-design size setting.
Auto-design piles (Pile cap only)	Autodesign piles automatically determines the quantity and size of piles. • Cleared - the specified number of piles at their specified positions will be checked during the design process. • Checked - the number of piles and their positions under the pile cap will automatically determined during the design process. When this checkbox is selected, the actual procedure used will depend on the Pile auto-design method that has been specified in Design Options > Concrete Foundations > Isolated Foundations Piles The **Minimise number of piles** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded the pile with the next smallest load capacity in the catalogue is selected

Property	Description
	and the process starts again. This is repeated until the pile loading check passes. The **Minimise pile capacity** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded one pile is added to the pile group and the process starts again. This is repeated until the pile loading check passes.
Select size/depth starting from	This option only appears when either Autodesign depth or Autodesign size is selected. It sets the autodesign start point for the depth, or size, or both. The options are: • Minima (default) • Current Selecting 'Minima' removes the current depth/size and begins with the minimum allowed depth/size specified in Design Options. For both options the auto-design depth increment is that specified in Design Options> Foundations Isolated Foundations> Foundation Size
Autodesign reinforcement	This setting applies to top and bottom reinforcement, but reinforcement in either location can still be set to none. If enabled, auto-design of reinforcement occurs after the base size and depth have been established. • When Autodesign reinforcement is enabled an iterative procedure is used to determine the reinforcement. If the bending design fails the reinforcement size gets increased and spacing gets decreased until these checks pass. Iterations continue until either a

Property	Description
	satisfactory bar configuration has been achieved, or the maximum bar size and minimum spacing specified in Design Options has been reached. At this point the depth gets increased and the above procedure is repeated. **NOTE** If the smallest bar at largest spacing is insufficient then the next bar size at max spacing is used. If the largest bar is reached the spacing gets reduced and the smallest bar is used. Then the bar size gets increased until the largest bar size reached. • When Autodesign reinforcement is disabled (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their reinforcement auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base the reinforcement is always designed (irrespective of the Autodesign reinforcement setting).

Property	Description
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Autodesign reinforcement is selected. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size specified in Design Options. • Current - the auto design commences from the current bar arrangement.
Foundation Parameters	
Eccentricity in Y direction	Specifies the eccentricity of the base in the Y direction. (For strip bases this is along the wall)
Eccentricity in X direction	Specifies the eccentricity of the base in the X direction. (For strip bases this is across the wall)
Rotation angle	Specifies the rotation of the base: • Pad base: relative to that of the support (which is controlled by the rotation angle specified for column). • Strip base: the angle of the base about global Z.
Shape Pad base and pile cap only)	Specifies the base shape in plan: • Square • Rectangular
Length in Y direction (Pad base and pile cap only)	Specifies the size of the pad in the Y direction.
Length in X direction (Pad base and pile cap only)	Specifies the size of the pad in the X direction.
Width (Strip base only)	Specifies the width of the base
Depth	Specifies the depth of the base.
Concrete class	The concrete grade. The concrete grades that are available are set from the Materials button on the Home ribbon.
Use loaded area (Pad base only)	Selecting this option allows you to specify a user override of the

Property	Description
	punching shear area - defined using the Loaded Area Breadth and Loaded Area Depth properties. This feature can be used in both pad bases and pile caps to, for example, consider unmodeled base plates or pedestals.
Sliding in Y prevented / Sliding in X prevented (Pad base only)	If sliding is prevented by some other means (e.g. if attached to a floor slab in one or other direction) then this can be indicated by selecting the appropriate direction. **TIP** If the direction is not clear make the base non-rectangular so that length in X and hence X-direction is clearer. Selecting this option sets the shear force for the particular direction to zero. This affects the sliding check, but actually affects all checks because there is no additional overturning moment developed due to shear at top of base.
Overturning about X prevented / Overturning about Y prevented (Pad base only)	This option only appears when the Sliding in Y prevented / Sliding in X preventedoption is selected. Selecting this option indicates that overturning is also prevented by some other means for the selected direction, which sets the support moment for that direction to zero. This affects all checks because there is no overturning moment considered.
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4

Property	Description
Piles (Pile cap only)	
Pile type	Pile types that have previously been specified in the Pile Catalogue are available for selection. You can choose <New...> to add additional types.
User defined arrangement	Specifies the eccentricity of the pile cap in the X direction.
Pile arrangement	Opens the Pile arrangement dialog for specifying the number of piles, and spacing, pile type and principal direction.
Number of piles	Specifies the number of piles.
Principal direction	Specifies the principal direction.
Pile spacing	The dimension between pile centers.
Shape	For three piles only, you can choose to specify either a triangular. or rectangular pile cap.
Reinforcement	
Type	• Mesh • Loose Bars • None
Rib type (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type (Head Code ACI)	• Plain • Deformed
Bar type	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size, spacing, Mesh type etc.	The actual reinforcement provided in each of the layers is indicated here.
Top, Bottom, Side cover	Nominal cover to reinforcement.
Soil Parameters	
Soil unit weight	Soil unit weight
Characteristic friction angle	Characteristic friction angle
Presumed bearing resistance (EC Head Code)	The presumed bearing resistance is only displayed and used when the

Property	Description
	'presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)' is selected in Design Options.
Surcharge	
Soil Surcharge depth Other permanent Permanent surcharge load Other variable Variable surcharge load	Drop-list controls allow each surcharge load to be assigned to a specific loadcase. The partial safety factors defined in the design combinations for these cases are then used for these loads in all design calculations. The assigned loadcases are initially set to "None" and must be user-selected. Where no appropriate Dead or Live loadcase exist in the model and so no loadcase can be assigned, zero partial factor values are used in the calculations.
Design shear strength of soil angle (ACI & BS Head Codes)	Design shear strength of soil angle
Allowable bearing capacity (ACI & BS regional codes)	Allowable bearing capacity
Bearing capacity A1 - STR (EC Head Code)	Allowable bearing capacity A1. Only displayed and used when the 'presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)' is unselected in Design Options.
Bearing capacity A2 - GEO (EC Head Code)	Allowable bearing capacity A2. Only displayed and used when the 'presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)' is unselected in Design Options.
UDA	A customizable list of the attributes that can be applied to individual members and panels.

Loading element properties

This section provides more information about some of the loading element properties you can modify in Tekla Structural Designer.

- Roof Panel Properties (page 1505)

- Wall Panel Properties (page 1504)
- Line ancillary properties (page 1507)
- Area ancillary properties (page 1510)
- Equipment properties (page 1512)

Wall Panel Properties

Use the **Wall Panel** properties to view or modify the properties of a wall panel.

The **Wall Panel** command is used to create a wall panel. Once created, the panel properties can then be viewed or modified in the Properties Window:

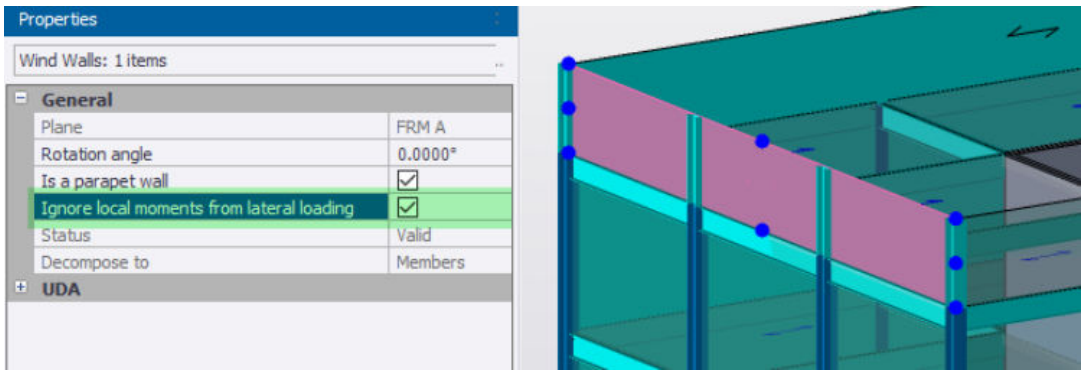
Property	Description
General	
Plane	Describes the plane in which the panel was placed.
Rotation angle	Describes the panel span direction as an angle, 0° is horizontal and 90° is vertical. See: and Modify slab/panel span direction (page 231)
Is a parapet wall	Check to indicate the panel is to be treated as a parapet in the wind analysis. See: Parapet wall panel load decomposition (page 1505)
Ignore local moments from lareral loading	This option is omly available when the “Is a parapet wall” setting is selected. When checked, only a lateral point load is applied to the top of a supporting column. See: Parapet wall panel load decomposition (page 1505)
Gap	Where the funneling gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funneling. A zero gap value explicitly means ignore funneling, for example where this building and the adjacent one are sheltered by upwind buildings.
Solidity	If you indicate that the wall panel is a parapet, then you also need to indicate the Solidity of the parapet. (Walls that are not parapets automatically adopt a solidarity of 1.0).
Status	Indicates whether the panel is valid or not.
Decompose to	Loads can be set to be decomposed to: • Members • Nodes (Default) • Rigid Diaphragms

Property	Description
	NOTE The decompose to nodes option only applies to those loads generated by the wind wizard. It does not apply to those loads applied directly to panels.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels.

Parapet wall panel load decomposition

For the special case of parapet wall panels spanning horizontally to parapet posts supported by columns, the panel load is decomposed as a lateral point load and point moment - which would be produced by a UDL on the post - to the top of the supporting column.

In some circumstances the engineer may wish only the lateral load but not a moment to be applied. This is catered for using the option “Ignore local moments from lateral loading” which is available in the wind wall properties only when the “Is a parapet wall” setting is checked on. When the “Ignore local moments from lateral loading” setting is checked on, only a lateral point load is applied to the top of a supporting column. No moment is applied.



NOTE

- The special decomposition of point loads - with or without a point moment - to supporting columns only occurs when all the following are true; the wind panel spans horizontally to parapet posts, the “Is a parapet wall” setting is checked on for the wall panel, the parapet post is supported by a column.
- In both the following cases a UDL will be decomposed to the members onto which the parapet wind panel directly spans;
  - The member supporting the parapet post is NOT a column.
  - The member onto which the parapet wind panel spans is not a parapet post (e.g. is an extended column or vertical beam).

Roof Panel Properties

Use the **Roof Panel** properties to view or modify the properties of a wall panel.

The **Roof Panel** command is used to create a roof panel. Once created, the panel properties can then be viewed or modified in the Properties Window:

Property	Description
General	
Plane	Describes the level at which the panel was placed.
Rotation angle	This value determines the following where appropriate: • Span direction for 1-way load decomposition • Orientation of semi-rigid 2D elements in the Solver Model See: and Modify slab/panel span direction (page 231)
Include in diaphragm	The options are: • Checked The roof panel is meshed to form a diaphragm of semi-rigid 2D elements. • Unchecked (Default) No diaphragm is defined on the roof panel.
Thickness	Roof panel thickness **NOTE** Only displayed if "Include in diaphragm" is checked
Youngs Modulus	Youngs Modulus **NOTE** Only displayed if "Include in diaphragm" is checked
Shear Modulus	Shear Modulus **NOTE** Only displayed if "Include in diaphragm" is checked
Temperature coefficient	Temperature coefficient. **NOTE** Only displayed if "Include in diaphragm" is checked
Divide Stiffness by	Used to adjust the roof panel stiffness. **NOTE** Only displayed if "Include in diaphragm" is checked
RoofType	The options are: • Default (Default)

Property	Description
	• Flat • Monopitch • Duopitch • Hip Gable • Hip Main • Mansard **NOTE** The default option assigns the roof panel as flat (if the slope < 5 deg) or Monopitch (if the slope is > 5 deg). You can more accurately specify the roof panel by choosing the appropriate option from the list. **NOTE** If the RoofType is changed after the wind model has already been established, you will need run **Update Zones** to reinstate the zoning.
[+] UDA	Customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 1292)

Line ancillary properties

General	
Name	The automatically generated name for the ancillary.
User Name	Can be used to override the automatically generated name if required.
Width, W	Width of ancillary
Type	The following types are available: • Walkways/catwalks[1] • Ladders with cage • Ladders without cage • Access platforms[1] • Operating platforms (storage laydown)[1] • Operating platforms (standard)[1] • Steel stair flight/landing[2]

General	
	• Concrete stair flight/landing[3] • Timber stair flight/landing[3] • Lines of pipework[4] • Lines of cable tray[5] • Other [1] These types expose the **Guardrail** flag below. [2] This type exposes the **Measuring** property and **Guardrail + channel height, H1** value below. [3] These types expose the **Guardrail height, H1** value below. [4] This type exposes the **Category** and **Diameter of largest pipe** values below. [5] This type exposes the **Category** and **Height of Cable Tray** values below.
Default Load Values	If this option is selected, load values for the selected type (with or without guardrail) are obtained from **Model Settings > Loading**, if unselected, load values can be entered directly.
Dead Load	Load to be created in the **Ancillary Dead** loadcase
Live (Imposed) Load	Load to be created in the **Ancillary Live (Imposed)** loadcase
Measuring (Live/Imposed)	For stairs choose whether the live/imposed load is a vertically projected load, or measured along the element.
Live (Imposed) Load loadcase	If working to Eurocodes, some ancillaries may require different Ψ and ϕ factors in which case a new loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Empty Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load loadcase	If working to Eurocodes, some cable trays may require different Ψ and ϕ factors in which case a new operating content loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Testing Content Load	Load to be created in the **Pipework Testing Content** loadcase

General	
Category	For cable trays and pipework, the default load values from **Model Settings > Loading** are those for the selected category.
Guardrail	If this option is selected, the default load values from **Model Settings > Loading** are those for the selected type **with** guardrail.
Guardrail height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Guardrail + channel height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height of cable tray, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Diameter of largest pipe	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. For more information, see: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** **NOTE** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.

General	
Reverse	For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking this flag.

See also

Open structure wind loads (page 417)

Area ancillary properties

General	
Name	The automatically generated name for the ancillary.
User Name	Can be used to override the automatically generated name if required.
Rotation	Defines the span direction.
Type	The following types are available: • Walkways/catwalks[1] • Ladders with cage • Ladders without cage • Access platforms[1] • Operating platforms (storage laydown)[1] • Operating platforms (standard)[1] • Steel stair flight/landing[2] • Concrete stair flight/landing[3] • Timber stair flight/landing[3] • Lines of pipework[4] • Lines of cable tray[5] • Other [1] These types expose the **Guardrail** flag below. [2] This type exposes the **Measuring** property and **Guardrail + channel height, H1** value below. [3] These types expose the **Guardrail height, H1** value below. [4] This type exposes the **Category** and **Diameter of largest pipe** values below. [5] This type exposes the **Category** and **Height of Cable Tray** values below.

General	
Default Load Values	If this option is selected, load values for the selected type (with or without guardrail) are obtained from **Model Settings > Loading**, if unselected, load values can be entered directly.
Dead Load	Load to be created in the **Ancillary Dead** loadcase
Live (Imposed) Load	Load to be created in the **Ancillary Live (Imposed)** loadcase
Measuring (Live/Imposed)	For stairs choose whether the live/imposed load is a vertically projected load, or measured along the element.
Live (Imposed) Load loadcase	If working to Eurocodes, some ancillaries may require different Ψ and ϕ factors in which case a new loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Empty Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load loadcase	If working to Eurocodes, some cable trays may require different Ψ and ϕ factors in which case a new operating content loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Testing Content Load	Load to be created in the **Pipework Testing Content** loadcase
Category	For cable trays and pipework, the default load values from **Model Settings > Loading** are those for the selected category.
Guardrail	If this option is selected, the default load values from **Model Settings > Loading** are those for the selected type **with** guardrail.
Guardrail height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Guardrail + channel height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height of cable tray, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Diameter of largest pipe	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.

General	
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. For more information, see: Open structure wind loads (page 417)
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** **NOTE** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default area per length used in this formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Area per length	The area per length used in the above formula can be edited if required.
Reverse	For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking this flag.

See also

Open structure wind loads (page 417)

Equipment properties

Properties	
Name	The automatically generated name for the equipment.
User Name	Can be used to override the automatically generated name if required.
Type	The following types are available: • Sphere • Vertical Cylinder

Properties		
	• Horizontal Cylinder • Cuboid	
Diameter, D	Diameter of sphere or cylinder. Used to determine the C_f value.	These values are also used in the calculation of wind loads on open structures seen by the equipment. For more information, see: Wind load calculation for equipment
Length, L	Length of cuboid or cylinder. Used to determine the C_f value.	
Breadth, B	Breadth of cuboid. Used to determine the C_f value.	
Height, H	Height of cuboid. Used to determine the C_f value.	
Base position, z	Entering a non-zero value raises/lowers the equipment above/below the loading area.	
Angle to Global X	Can be used to rotate the equipment relative to global X	
CoG offset	Can be used to offset the equipment relative to the global or local axes	
Empty Load		
Apply Load	Load can either be applied at CoG or at Supports. This load is created in the **Equipment Empty** loadcase.	
Load	Enter a single value if at CoG, or individual values at each support.	
Operating Content Load		
Apply Load	Load can either be applied at CoG or at Supports. This load is created in the **Equipment Operating Content** loadcase.	
Load	Enter a single value if at CoG, or individual values at each support.	
Testing Content Load		
Apply Load	Load can either be applied at CoG or at Supports. This load is created in the **Equipment Testing Content** loadcase.	
Load	Enter a single value if at CoG, or individual values at each support.	
Wind loading		

Properties	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. For more information, see: Open structure wind loads (page 417)
Surface	For vertical and horizontal cylinders, specify the surface as either, moderately rough, rough, or very rough. For cylinders the surface and L/D ratio are used to determine the Cf value
End	For horizontal cylinders, specify the end as either flat, or rounded. This is used to determine the Cf value, see ASCE 7 Fig 6-21

See also

Open structure wind loads (page 417)

Solver model and support properties

This section provides more information about some of the Solver model and support properties you can modify in Tekla Structural Designer.

- Support properties (page 1514)
- Solver node properties (page 1517)
- Solver element properties (page 1518)
- Solver element 2D properties (page 1519)

Support properties

Use the Support properties to view or modify the properties of a support.

When a support is first created, it's properties are taken as those displayed in the **Properties** window at that time.

You can either set the properties prior to placing the support, or select a support in the model afterwards to review or modify its properties.

Property	Description
General	

Property	Description
Name	Automatically created name based on the grid point location. **NOTE** This property is only displayed when editing an existing support
User Name	You can enter a user name to replace the automatically created name if required. **NOTE** This property is only displayed when editing an existing support
Plane	Describes the level at which the support was placed. **NOTE** This property is only displayed when editing an existing support
3 Grid Points	The options are: • Checked A user defined coordinate system is applied to the support. (After clicking where you want to create the support, the second click defines the x direction and the third click defines the y direction.) • Unchecked (Default) Support properties are defined in accordance with the global coordinate system. **NOTE** This property is only displayed when creating a new support
Fx, Fy, Fz	The translational degrees of freedom can be set as either Free, or Fixed in each direction.
Mx, My, Mz	The rotational degrees of freedom can be set as either Free, or Fixed in each direction.
Angles	
Inclination, Azimuth and Rotation	When creating new supports, the angles are calculated automatically depending on the placement method (3 Grid Points Checked/Unchecked). When editing existing supports, the angles can be edited in order to redefine the direction in which the support acts. • **Inclination** - inclination about **Global X**

Property	Description
	• **Azimuth** - rotation of inclined support about **Global Z** • **Rotation** - rotation about the **support local z axis** For more information, see: Rotated and inclined supports (page 245)
Translational stiffness x, y, and z	
Type	In order to define a translational spring in a particular direction, the translational degree of freedom in the same direction must first be set to Free. The available types are: • Release - (i.e. zero translational stiffness) • Spring Linear • Spring Non-linear
Stiffness	The options are: • Spring Linear A single stiffness value is entered, which acts in both the positive and negative directions. • Spring Non-linear Two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).
Rotational stiffness x, y, z	
Type	In order to define a rotational spring in a particular direction, the rotational degree of freedom in the same direction must first be set to Free. The available options are: • Release - (i.e. zero rotational stiffness) • Spring Linear • Spring Non-linear • Nominally Pinned • Nominally Free

Property	Description
Stiffness	The options are: • Spring Linear A single stiffness value is entered, which acts in both the positive and negative directions. • Spring Non-linear Two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.
Stiffness Percentage	The options are: • Nominally Pinned - 10% (i.e. 10% * 4 EI/L) • Nominally Fixed - 100% (i.e. 100% * 4 EI/L) **NOTE** In the above equations, L is the length from the base of the column to the level of the next column stack that is denoted as a floor, or it is the distance to the top of the column if shorter.
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).

See also

Create supports (page 242)

Solver node properties

When in a Solver View, solver node properties are displayed in the Properties Window as shown below.

Property	Description
General	
Fx, Fy, Fz	The translational degrees of freedom at the node.
Fx, Fy, Fz	The rotational degrees of freedom at the node.
Coordinate	The node location.

Property	Description
P-Delta	This property cannot be edited.
Exclude from Diaphragm	If checked, the node has been excluded from the diaphragm. For instructions on how to do this, see: Include or remove solver nodes from the diaphragm (page 962)
Diaphragm #	Specifies the diaphragm number to which the node is connected. (Not displayed when viewing the "Working Solver Model").

Solver element properties

When in a Solver View, solver element properties are displayed in the Properties Window as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

Property	Description
General	
Active	When this is set to False the solver element is inactive in the analysis. Only certain member types (braces, analysis elements) can be made inactive.
Type	The type of the solver element
Fabrication	The fabrication type of the solver element.
Construction	The construction type of the solver element.
Material	The solver element material.
Gamma angle	Defines the element orientation about its local x axis. When gamma = 0, the local z lies in the plane created by the local x axis and the global z axis.
Length	The solver element length.
Start Releases	
Fx, Fy, Fz	These define the translational end releases at end 1.
Mx, My, Mz	These define the rotational end releases at end 1.

Property	Description
End Releases	
Fx, Fy, Fz	These define the translational end releases at end 2.
Mx, My, Mz	These define the rotational end releases at end 2.

Solver element 2D properties

When in a Solver View, solver element 2D properties are displayed in the Properties Window as shown below. None of these properties can be directly edited in the Solver View.

Property	Description
Panel	
Type	The 2D element type will be Shell or Semi-rigid depending on the Slab Type, Decomposition and Diaphragm Options that have been set.
Thickness	The 2D element thickness is derived from a different property depending on the slab type: • Composite Slab; Precast Slab; Slab on Beams; Flat Slab - Overall Depth • Steel Deck; Timber Deck - Thickness
Orientation	The 2D element orientation in the solver model follows the rotation angle defined for the slab item to which it belongs.
DivideStiffnessBy	This property applies to semi-rigid elements only. It adjusts the stiffness determined from the material properties in order to control semi-rigid diaphragm flexibility.
CrackedOption	yes/no
Nodes	
Node 1, Node 2, Node 3	The node numbers associated with this element.

Floor Analysis Model (FAM) properties

Edit Note: The below properties are only displayed when the FAM is being edited	
Entity type	Select the entity type to add or remove: • Slab Item • Member Span/Stack • Facade Restraint - only available when **Generate Facade Restraint** is selected • Spandrel Beam - only available when **Generate Spandrel Beams** is selected
Selection Mode	Determines what happens when you click on an entity of the selected type: • Add or Remove • Add Only • Remove Only
Generate Facade Restraint	When selected, facade restraints are created around the external perimeter of slab items in the current floor analysis model (FAM). You can then add or remove individual facade restraints as required.
Generate Spandrel Beams	When selected, any beams around the external perimeter of slab items in the current FAM are marked as spandrel beams. You can then add or remove spandrel beam markings as required.

FAM	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Design Guide	The design guide on which the assessment is to be based. • SCI P354 • CCIP 016 • AISC DG11

FAM	
Settings	The FAM settings set applied to this FAM. Initially there will be a single settings set to choose from, generated from the Footfall Assessment Global Settings for the current design guide. **NOTE** You can configure additional settings sets as required. From the Footfall Analysis ribbon click **Settings**, then on the **Settings Sets Management** page, click **Add**.

FAM > Meshing	
Slab Mesh Size	Defines the slab mesh size. Applies to all 1-way slabs set to be meshed in the FAM Analysis Model Options.
Slab Uniformity Factor	Defines the mesh uniformity. (Default = 50%, where 100% = maximum uniformity). Applies to all 1-way slabs set to be meshed in the FAM Analysis Model Options.
Slab Mesh Type	Defines the mesh type for all meshed slabs in the FAM model. The options are: • Quad dominant • Quad only • Tri only

NOTE The initial properties displayed in the below **Footfall Checks** sections default to values taken from the page in Global Settings.

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
Cut off frequency	20 Hz	1-50	Frequency above which post processing calculations are not performed. Modes with frequencies above this value may be found (see **Total**

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
			number of modes in the **Analysis Settings** section above) but are not considered in the footfall calculation procedures. A warning is issued if this value is greater than the max modal frequency found. For guidance, refer to SCI P354 Table 6.1: *Low frequency floor to high frequency floor cut-off*
Consider all modes	On		When selected all modes below the **Cut off frequency** are considered.
No. of modes	10		This property is only displayed when **Consider all modes** is cleared. The **Cut off frequency** is ignored and the number of modes specified here are considered in the footfall post processing instead.
No. of harmonics	4	1 to 4	Not required for transient response.
Pacing frequency	1.80-2.20Hz@0.01	1 to 5Hz	Can be input as a single value or range and

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
			increment or series of values.
Walker's weight	76 (kg)	50 to 150 (kg)	Walker's weight.
Length of walking path	30 m	0.1 to 100m	Used in the resonance build up factor of resonant response and in VDV calculations for both resonant and transient response. Larger values result in conservative checks. If not known, provide a reasonable max value for best results.
Length of stride	0.75 m	0.1 to 2 m	Required for VDV
Damping ratio	3%	0 to 100%	The design guidelines provide recommended values of the damping ratio if more accurate values cannot be estimated. Refer to SCI P354 Table 4.1: *Critical damping ratios for various floor types* ζ
Axis of vibration perception	Z	X, Y, Z	In local coordinate system of the subject person. Only 'Z' is available in the current release.

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
BS 6841 weighting curve	Wb	Wb, Wd, Wg	The BS 6841 weighting curve used in the calculation of **Weighting Factor, Wh**, in resonant and transient responses.
Target vibration dose value (VDV)	0.2	0.1 to 2.0	The reciprocal of this value gives the number of times in a period that the target VDV is exceeded.
Consider VDV for Pass/Fail	Off		When selected, the pass/fail threshold is determined by the **Target number of events in a period**.
Target number of events in a period	1000	1 to 1000000	Defines the pass/fail threshold when **ConsiderVDV for Pass/Fail** is selected.
Target response factor, Rf	4	0 to 100	Unless **Consider VDV for Pass/Fail** is selected, the pass/fail threshold is determined by this value.
Acceptable hotspots	Off		When selected a **General limit, Rf** value (default 2.0) is set. Regions with Rf values between the general and upper limits will

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
			be given a warning status.

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
Cut off frequency	20 Hz	1-50	Frequency above which post processing calculations are not performed. Modes with frequencies above this value may be found (see **Total number of modes** in the **Analysis Settings** section above) but are not considered in the footfall calculation procedures. A warning is issued if this value is greater than the max modal frequency found.
Consider all modes	On		When selected all modes below the **Cut off frequency** are considered.
No. of modes	10		This property is only displayed when **Consider all modes** is cleared. The **Cut off frequency** is ignored and the number of modes specified here are considered in the footfall post

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
			processing instead.
No. of harmonics	4	1 to 4	Not required for transient response.
Pacing frequency	1.80-2.20Hz@0.01	1 to 5Hz	Can be input as a single value or range and increment or series of values.
Walker's weight	76 (kg)	50 to 150 (kg)	Walker's weight.
Length of walking path	30 m	0.1 to 100m	Used in the resonance build up factor of resonant response and in VDV calculations for both resonant and transient response. Larger values result in conservative checks. If not known, provide a reasonable max value for best results.
Length of stride	0.75 m	0.1 to 2 m	Required for VDV
Damping ratio	3%	0 to 100%	The design guidelines provide recommended values of the damping ratio if more accurate values cannot be estimated. Refer to Appendix A of CCIP-016 for suggested damping values

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
			for different structures.
Axis of vibration perception	Z	X, Y, Z	In local coordinate system of the subject person. Only 'Z' is available in the current release.
Target vibration dose value (VDV)	0.2	0.1 to 2.0	The reciprocal of this value gives the number of times in a period that the target VDV is exceeded.
Consider VDV for Pass/Fail	Off		When selected, the pass/fail threshold is determined by the **Target number of events in a period**.
Target number of events in a period	1000	1 to 1000000	Defines the pass/fail threshold when **ConsiderVDV for Pass/Fail** is selected.
Target response factor, Rf	4	0 to 100	Unless **Consider VDV for Pass/Fail** is selected, the pass/fail threshold is determined by this value.
Acceptable hotspots	Off		When selected a **General limit, Rf** value (default 2.0) is set. Regions with Rf values between the general and upper limits will

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
			be given a warning status.

Footfall Checks (AISC DG11)			
Item	**Default**	**Range**	**Description**
Cut off frequency	20 Hz	1-50	Frequency above which post processing calculations are not performed. Modes with frequencies above this value may be found (see **Total number of modes** in the **Analysis Settings** section above) but are not considered in the footfall calculation procedures. A warning is issued if this value is greater than the max modal frequency found.
Consider all modes	On		When working to AISC DG11 this option can not be cleared.
Walker's weight	0.084 ton (US) 76 kg (metric)	0.055 to 0.165 ton 50 to 150 kg	Walker's weight.
Damping ratio	3%	0 to 100%	The damping ratio can be estimated using the component values in DG1 Table 4-2 if more accurate values are not available.

Footfall Checks (AISC DG11)			
Item	**Default**	**Range**	**Description**
Axis of vibration perception	Z	X, Y, Z	In local coordinate system of the subject person. Only 'Z' is available in the current release.
Fig. 2-1 AISC DG11	Curve A	A, B , C	When determining the pass/fail status at a given node, the calculated peak acceleration at the dominant frequency is compared to the limiting peak acceleration obtained from the selected curve.

Overall Status & Envelopes	
Mass Combination	When displaying the overall status and envelopes choose whether to consider all footfall mass combinations, or an individual combination only.
Response Type	When displaying the overall status choose whether to consider all response types, resonant only, or transient only. Selecting 'None' excludes this FAM from overall footfall status consideration.

10.3 Dialog boxes reference

This section covers some of the important dialog boxes in Tekla Structural Designer and their different options.

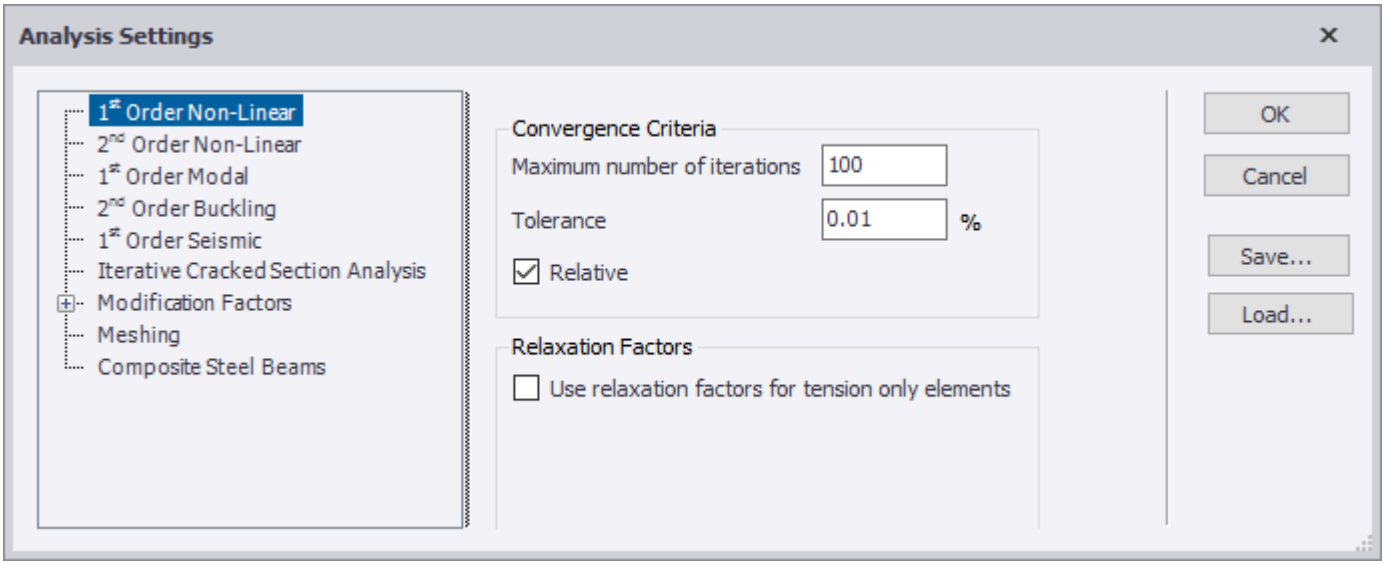
Click the links below to find out more:

- Analysis Settings dialog box (page 1530)
- Checkbot Control Panel (page 1531)
- Connection Resistance dialog box (page 1533)
- Construction Levels dialog box (page 1536)
- Design Settings dialog box (page 1538)

- Drawing Settings dialog box (page 1539)
- Edit stage data dialog box (page 1540)
- Embodied Carbon Factors dialog box (page 1542)
-
-
-
- Load Event Sequences dialog box (page 1550)
- Loading dialog box (page 1557)
- Materials dialog box (page 1564)
- Model Settings dialog box (page 1570)
- Pile catalogue (page 1571)
- Punching shear reinforcement properties dialog box (page 1575)
- Sections dialog box (page 1581)
- Settings dialog box (page 1583)
- Simple Wind Loading dialog box (page 1583)
- Slab Deflection Check Catalogue (page 1592)
- Slab Deflection Settings dialog box (page 1593)
-
-
- Sub Models dialog box (page 1593)

Analysis Settings dialog box

The **Analysis Settings** dialog box and its sub pages allow you to adjust the settings applied to different analyses in the current project.



Location

On the **Analyze** ribbon tab, click **Settings**.

Content

See: Analysis settings (page 760)

Usage

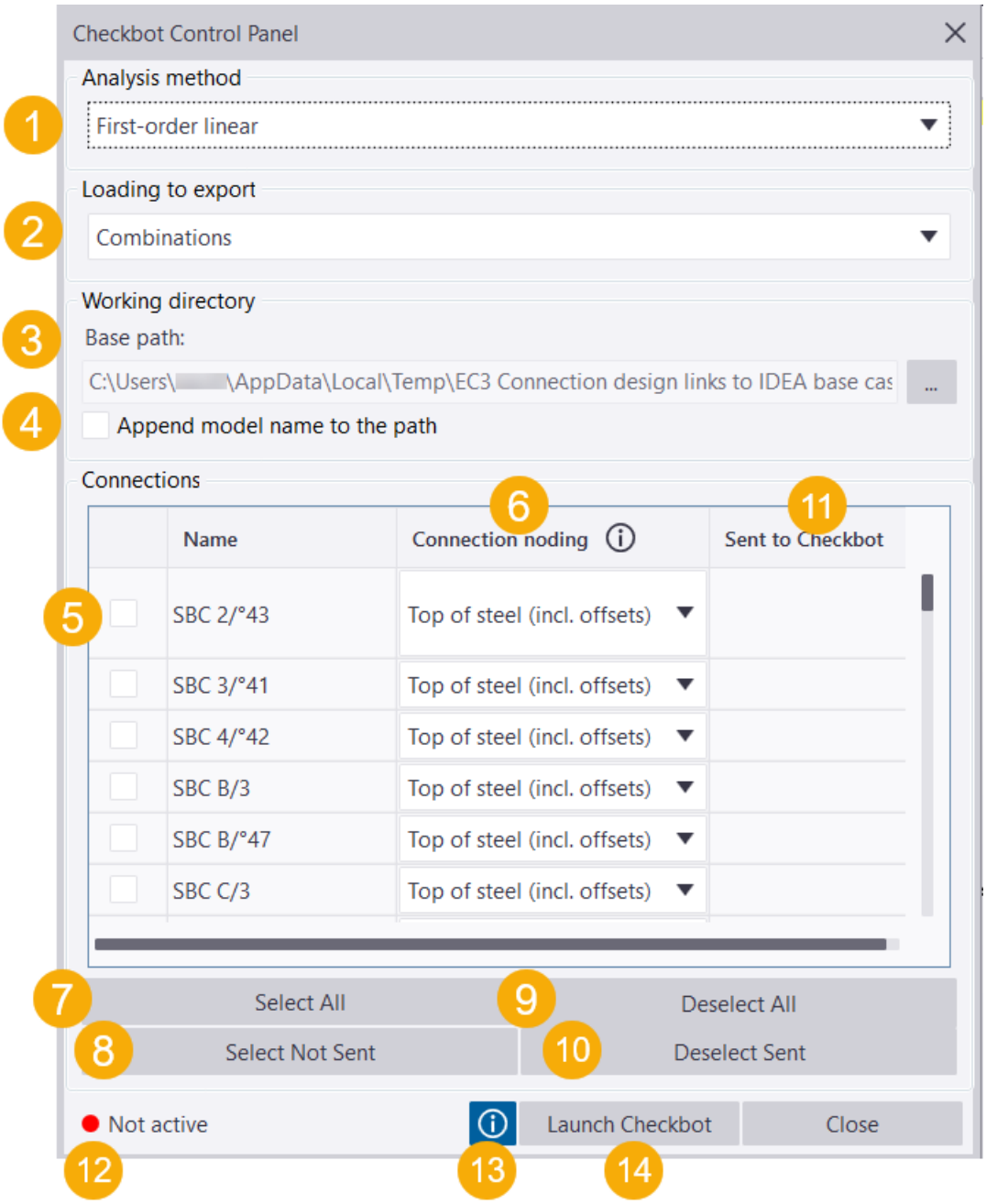
After adjusting the settings, do one of the following:

- To apply the changes to the current project: click **OK**
- To save the changes back to the active settings set for future use: click **Save...**
- To cancel the changes: click **Cancel**
- To revert to the analysis settings specified in the active settings set: click **Load...**

Checkbot Control Panel

The **Checkbot Control Panel** which can be opened by selecting **IDEA StatiCa Export** on the **BIM Integration** ribbon, is used to select connections to be sent to IDEA StatiCa for design.

The dialog box is shown below.



1. **Analysis method** – a drop-down box lists the analysis types for selection.
2. **Loading to export** – This drop-down has two options: 'Combinations' and 'Combinations & Loadcases'.
3. **Working directory** - this allows you to specify the path where the connection data will be stored. You can accept the default path or click the [...] button to browse to and select a different path.

4. **Append model name to the path** – this displays the full file path when checked and creates a folder with the Tekla Structural Designer model name in which exported connections are generated.
5. **Connection selection checkboxes** - connections are selected either by ticking checkboxes, or by graphical selection (by windowing or cursor selection) in the scene view.
6. **Connection noding** - this sets the connection alignment to either Top of steel (incl. offsets), or Member centre line (only). for further details see: Connection noding (page 1208)
7. **Select All** – Selects all listed connections.
8. **Select Not Sent** – Selects connections that have not been exported to IDEA StatiCa previously.
9. **Deselect All** – Deselect all the selected connections in the list.
10. **Deselect Sent** – Deselects connections that have been exported to IDEA StatiCa previously.
11. **Sent to Checkbot** - A tick in this column indicates those connections that have previously been sent to the Checkbot.
12. **Active/Not active** - This flag is used to inform the user when the Checkbot is active.
13. **Information button** – Opens a dialog box of 'IDEA StatiCa Checkbot instructions' for guidance on the Checkbot.
14. **Launch Checkbot** – Opens the Checkbot application page.

Connection Resistance dialog box

The **Connection Resistance** dialog box displays the pre-defined connection resistances for steel beams to Eurocode and US regional codes. It can also be used to add or edit user-defined connection resistances.

To open the dialog box:

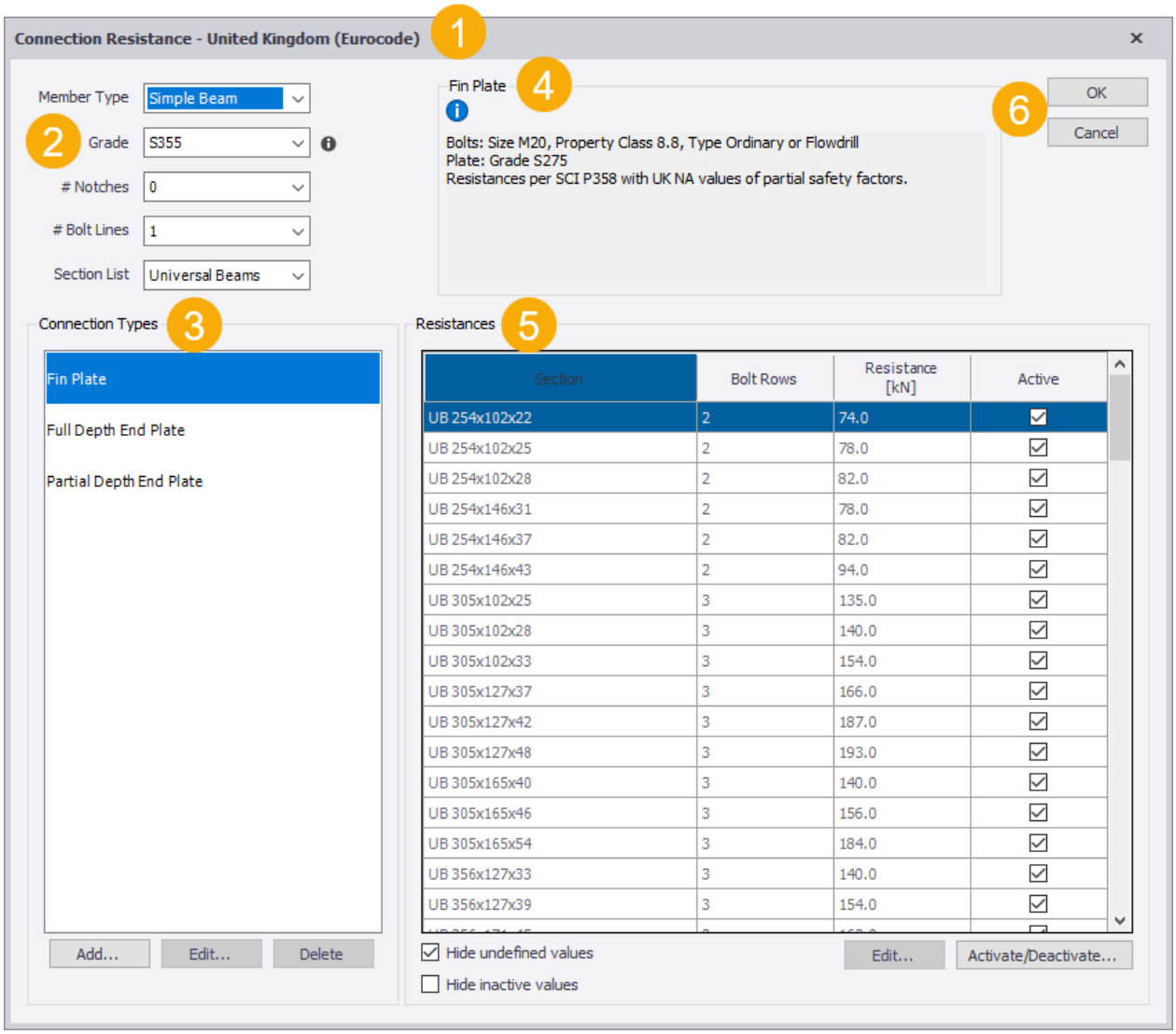
1. Click the **Home** tab.
2. Click  **Materials**.

   The **Materials** dialog is displayed.
3. On the **Sections** page, check that the required regional code and material are displayed.

NOTE Both steel and cold formed materials can have resistances defined, unless the regional code is US, in which case resistances can be defined for steel only.

4. Click **Connection Resistance**

The dialog box content is described below.



1. Title Bar

The title bar displays:

- The currently selected regional code.
- For India this also displays LS or WS, and for US displays LRFD or ASD, according to the setting of the currently open model.

2. Filters

The filter settings determine what is displayed elsewhere in the viewer.

The Member Type filter has options for:

- Simple Beam
- Brace

3. Connection Types

Pre-defined and user-defined Connection Types are listed here according to the Filter settings.

- Click the **Add...** button to add a user-defined Connection Type (page 1277).
- Click the **Edit...** button to edit a user-defined Connection Type.
- Click the **Delete** button to delete a user-defined Connection Type.

NOTE The **Edit...** and **Delete** buttons are only active when a *user-defined* Type is selected.

4. Info box

Displays information related to the currently selected Connection Type. For user-defined Connection Types the information displayed here can be added or edited via the Add... and Edit... buttons in the Connection Types area of the viewer.

5. Resistances

- Resistance values are listed in the window according to the Filter settings and the currently selected Connection Type. The resistance values (for user-defined Connection Types) can be edited directly.
- The 'Active' setting means that, when ticked, the particular resistance will be considered in the check. The Active status can be edited directly in this window for individual resistances but is more quickly edited using Activate/ Deactivate. All pre-defined resistances start out Active.
- If 'Hide undefined values' is un-ticked those resistances which have zero value currently assigned will be displayed along with the other sections. This box is ticked by default when opening the viewer.
- If 'Hide inactive values' is ticked those resistances which have 'Active' un-ticked will be hidden in this window. This box is un-ticked by default when opening the viewer.
- Use the Edit... button to edit the bolt row count or resistance or Active status of a user-defined Connection Type. Note that multiples of the same

section size can be added through the Edit Resistances dialog, so that the same size with different bolt row counts and resistances can be defined.

Click here for an example (page 1279) showing the use of Edit... as described above.

- Use the Activate/Deactivate... button to change the Active setting for multiple sections. Set the filters in the Activate/Deactivate dialog, then set either Activate or Deactivate, and then Apply if you want to change further Active settings or Apply & Close to finish.

  Click here for a Eurocode example or here for a US example showing the use of Activate/Deactivate as described above.

6. OK and Cancel

Button	Description
OK	Commit all changes made in the viewer to the database.
Cancel	Closes the dialog box without saving changes.

Construction Levels dialog box

The **Construction Levels** dialog box allows you to define the levels required in order to construct your model.

Choose from:

- On the **Model** tab, click **Construction Levels**.
- In the **Structure** tree, double-click **Levels**.

Content

Button, command or option	Description
Options	
Ref	Allows you to assign a unique reference to a construction level.
Name	Allows you to assign an optional name to further assist identification. For example, "First floor" or "Mezzanine".
Type	Allows you to select the Type from the drop down menu: • **T.O.S.** = Top of steel

Button, command or option	Description
	• **S.S.L.** = Structural slab level • **T.O.F.** = Top of foundation **NOTE** Slabs are modeled above the level when they are set to T.O.S but below the level when they are set to S.S.L or T.O.F
Level	Allows you to assign the height of the construction level above the base level.
Spacing	Allows you to assign the distance of this construction level to the one immediately below.
Source	Allows you to assign the level as Unique or the same as another level. Unique levels can be edited independently, whereas edits to indentical levels are applied at both levels simultaneously.
Slab Th.	Allows you to set the default thickness for any slab created on the level.
Floor	Allows you to assign the construction level as a major level in the building. Floor levels determine the number of sub models that are created for the chasedown analysis. Floor levels are also used to determine items such as your inter story height. There will certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs • K bracing. You require a construction level for the intermediate bracing connection points • Steps in the building floor levels When you define a level that is clearly not a floor, then you should not check the option. If a floor exists only at some locations in the level (e.g. in a building with stepped floor levels), then you should check the option where applicable.
Buttons	
OK	Allows you to save any changes made.
Cancel	Allows you to discard any changes made.
Insert Above	Allows you to insert a new construction level above the currently highlighted level with the same spacing.
Insert Below	Allows you to insert a new construction level below the currently highlighted level with the same spacing.

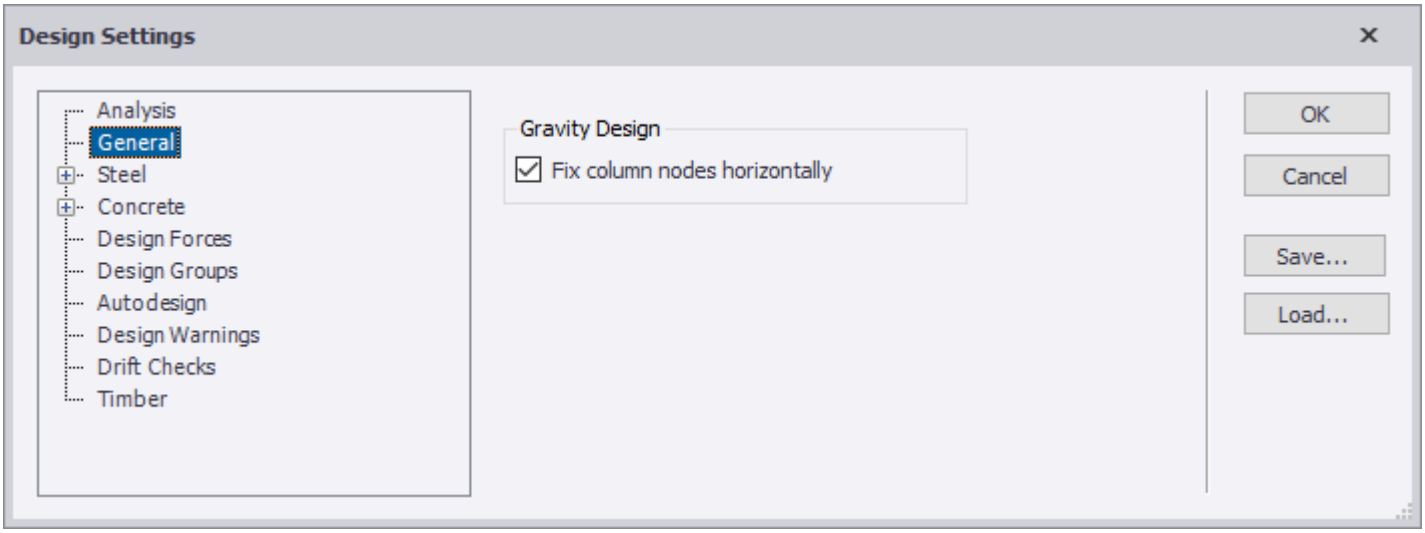
Button, command or option	Description
New on Top	Allows you to insert multiple construction levels above the uppermost level with variable spacings **NOTE** 3*5 inserts three levels, each at 5m spacing 3,4,5 inserts three levels at spacings of 3,4 and 5m respectively.
New at the Bottom	Allows you to insert multiple construction levels below the lowest level with variable spacings
Delete	Allows you to delete the selected level.

See also

Create and manage construction levels (page 121)

Design Settings dialog box

The **Design Settings** dialog box allows you to adjust the design settings applied to the current project.



Location

On the **Design** ribbon tab, click **Settings**.

Content

See: Design settings (page 866)

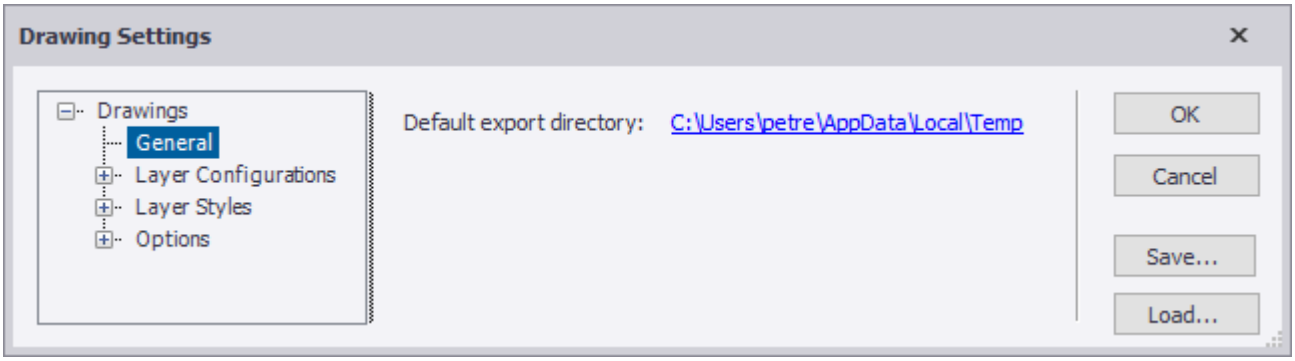
Usage

After adjusting the settings, do one of the following:

- To apply the changes to the current project: click **OK**
- To save the changes back to the active settings set for future use: click **Save...**
- To cancel the changes: click **Cancel**
- To revert to the design settings specified in the active settings set: click **Load...**

Drawing Settings dialog box

The **Drawing Settings** dialog box allows you to adjust the drawing settings applied to the current project.



Location

On the **Draw** ribbon tab, click **Settings**.

Content

See: Drawing settings (page 1119)

Usage

After adjusting the settings, do one of the following:

- To apply the changes to the current project: click **OK**
- To save the changes back to the active settings set for future use: click **Save...**
- To cancel the changes: click **Cancel**
- To revert to the drawing settings specified in the active settings set: click **Load...**

Edit stage data dialog box

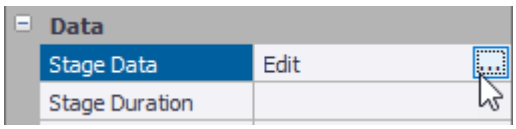
Summary

The **Edit Stage Data** dialog box allows you to define construction stages and their durations when performing a staged construction analysis.

Location

To display the dialog:

1. Switch the view regime for a Structure 3D view to **Review View**.
2. On the **Review** ribbon, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Staged Construction**.
5. In the **Properties** window, click **Stage Data**, then click the **Edit** button



Content

Edit Stage Data

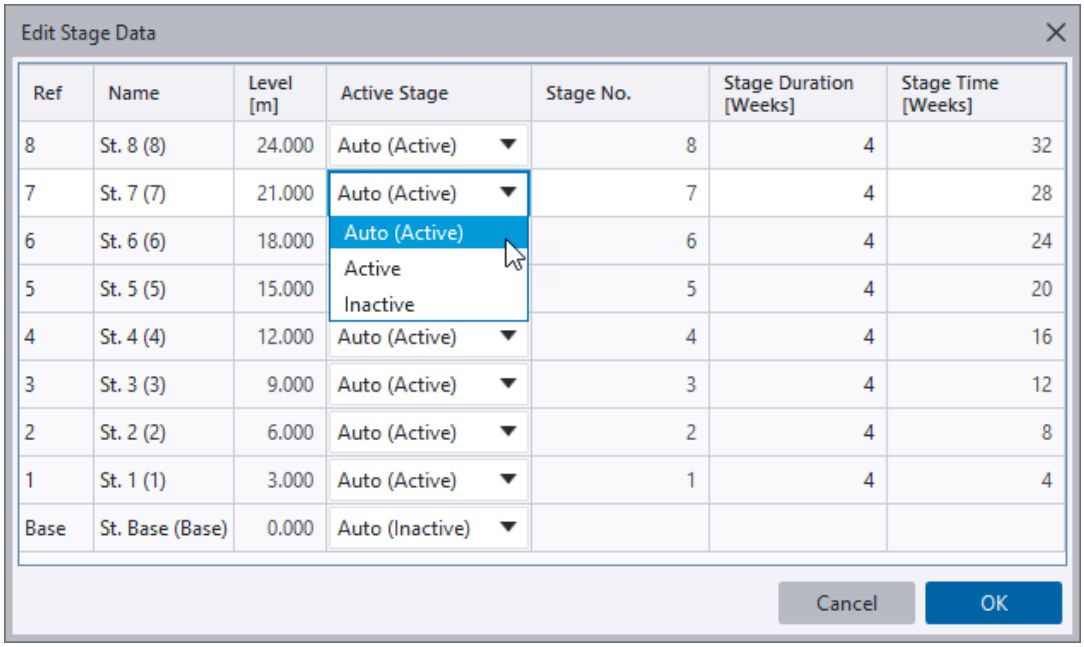
Ref	Name	Level [m]	Active Stage	Stage No.	Stage Duration [Weeks]	Stage Time [Weeks]
8	St. 8 (8)	24.000	Auto (Active)	8	4	32
7	St. 7 (7)	21.000	Auto (Active)	7	4	28
6	St. 6 (6)	18.000	Auto (Active)	6	4	24
5	St. 5 (5)	15.000	Auto (Active)	5	4	20
4	St. 4 (4)	12.000	Auto (Active)	4	4	16
3	St. 3 (3)	9.000	Auto (Active)	3	4	12
2	St. 2 (2)	6.000	Auto (Active)	2	4	8
1	St. 1 (1)	3.000	Auto (Active)	1	4	4
Base	St. Base (Base)	0.000	Auto (Inactive)			

Cancel OK

Each row in the above table corresponds to a level that has been marked as a floor in the **Construction Levels** dialog box.

The number of construction stages can be controlled by adjusting the active/inactive setting in the **Active Stage** column.

- To reduce the number of stages, set **Active Stage** at a particular level to **Inactive**. The members at that level are then moved to the next active stage.



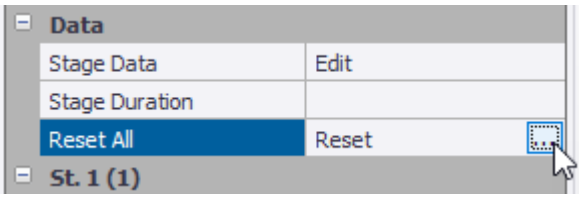
Initially each floor defaults to either Auto (Active), or Auto (Inactive):

- Auto (Active) - the floor level is an active stage because it contains at least one member, or has a member attached to it from below.
- Auto (Inactive) - the floor level is not an active stage because it doesn't contain at least one member, or have a member attached to it from below.
- The auto defaults can be overridden by manually setting a level to be an active or an inactive stage.

Stage durations can be edited in the above table, or via the Properties window.

The Stage time (which can't be edited) is the sum of durations of current and previous stages; it only affects concrete creep allowance during the construction stages.

If you have made edits to stages and/or durations but then change your mind, you can revert back to the original defaults by clicking the **Reset All** button in the **Properties** window.



Note that if you do click **Reset All**:

- All stages are reset to their original Auto (Active)/(Inactive) setting
- All members are reassigned to their default stage.
- All stage durations are reset to default.

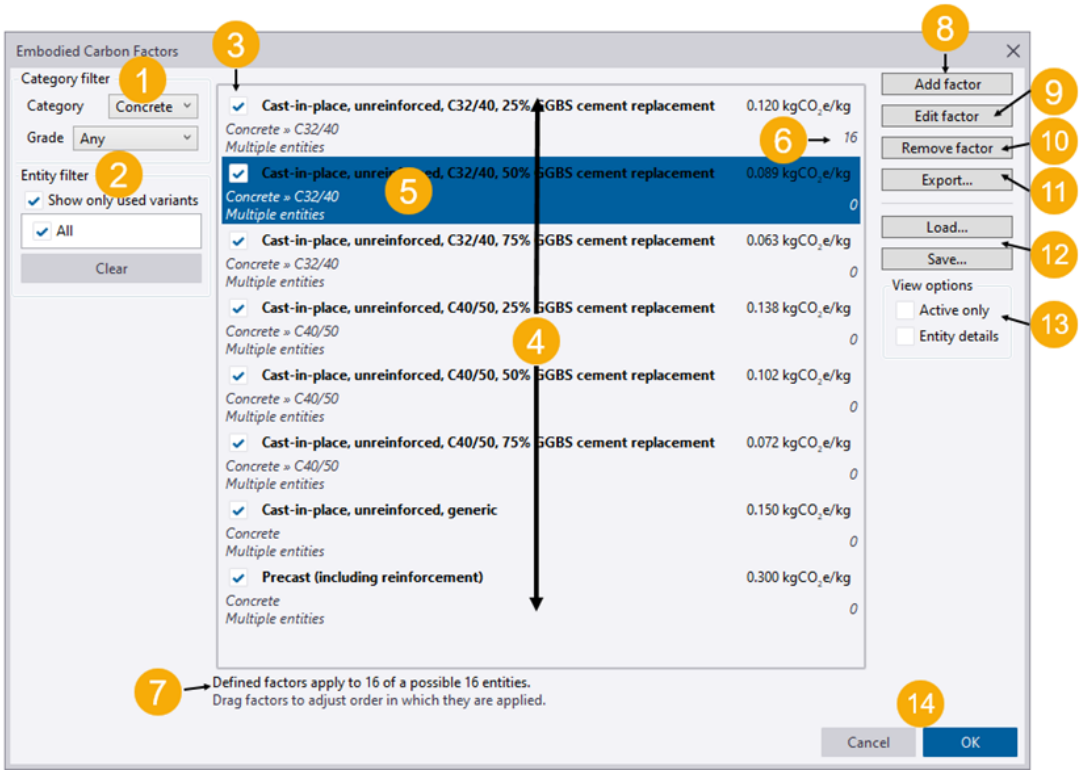
Embodied Carbon Factors dialog box

The **Embodied Carbon Factors** dialog box on the **Home** ribbon is used to add and to manage embodied carbon factors (ECFs). The order in which factors are arranged in the dialog determines the priority in which they are assigned to models.

Filters can be applied to make it easier to interrogate the list, these do not affect how the factors are subsequently applied to the model.

NOTE Any changes made in the dialog will only apply to the current model until the factors are saved to the active global settings set.

The dialog box is shown below.



1. Category filter

The category filter contains a list of all item categories that it is possible to apply embodied carbon factors to.

Select a category in order to see the factors that have been previously defined in that category.

Some of these such as 'Cladding' are optional items and might not have any factors defined.

You cannot add new categories.

Additional filters are also provided which vary depending on the category selected. For example, when the Steel category is selected the filters are *Grade* and *Section*.

NOTE For the UK Settings Set, some initial defaults have been provided for the Concrete, Metal Deck, Reinforcement, Steel, and Timber categories.

---

2. Entity filter

Each factor is associated with one or more variants within a specified category.

As there are potentially hundreds of factors that could be defined in any one category, the **Entity filter** provides a means of filtering the list to make it easier to interrogate a specific variant.

The **Entity filter** defaults to displaying all factors applicable to any variant. If this results in a long list that requires a lot of scrolling, you might choose to filter the list as follows:

1. Unselect **All** in order to see a choice of variants that apply to the current category.
    - Click **Show only used variants** if you want to have the choice of variants reduced further to just those that exist in the currently open model.
2. Select one or more variants in order to filter the main list accordingly.
    - Click **Clear** if you want to clear the selected variants and return to the default **All** setting.

3. Active flag



If the **Active** flag is unselected, the factor will not be applied to model.

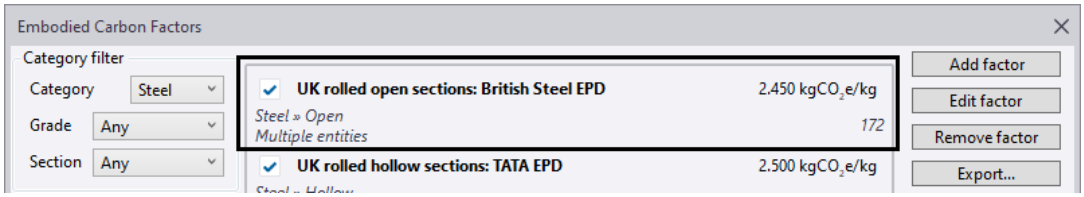
4. Filtered list of embodied carbon factors

The list is displayed in accordance with the **Category filter** and **Entity filter** criteria. It also takes account of the specified **View options**.

The list can be reordered by selecting individual items and dragging them up or down. The order is very important because each entity in the model will adopt the first applicable factor it finds in the list.

The number of entities to which the factor currently applies is shown to the right of the item. When the list is reordered these numbers are instantly updated.

5. Embodied carbon factor item



When a factor is added its name and value appear in the list as shown.

Beneath the item to the left are its category and entity criteria, and on the next line, the entities to which it applies.

Beneath the item to the right the number of entities to which the factor currently applies is shown.

6. Number of entities the factor currently applies to



The number of entities to which a factor currently applies is shown to the right of the item. When the list is reordered this number is instantly updated.

7. Defined factors applied to possible entities message

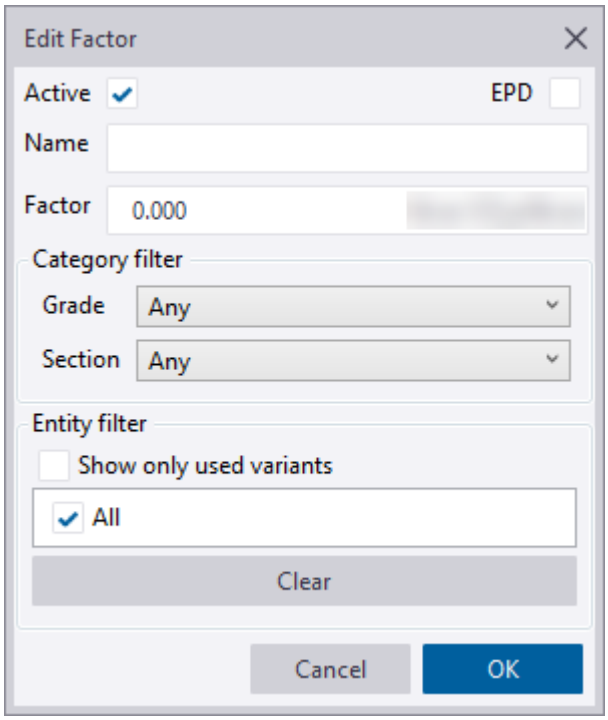
Located underneath the list of factors, this message indicates for the current filtering criteria, the number of entities that have a factor applied, and the total number of entities in the model meeting the criteria.

NOTE If the numbers don't match this indicates that the total embodied carbon will be under-reported.

---

8. Add factor

Add factor is used to add new factors within the category currently shown in the **Category filter**. Initially these would be available for the current model only.



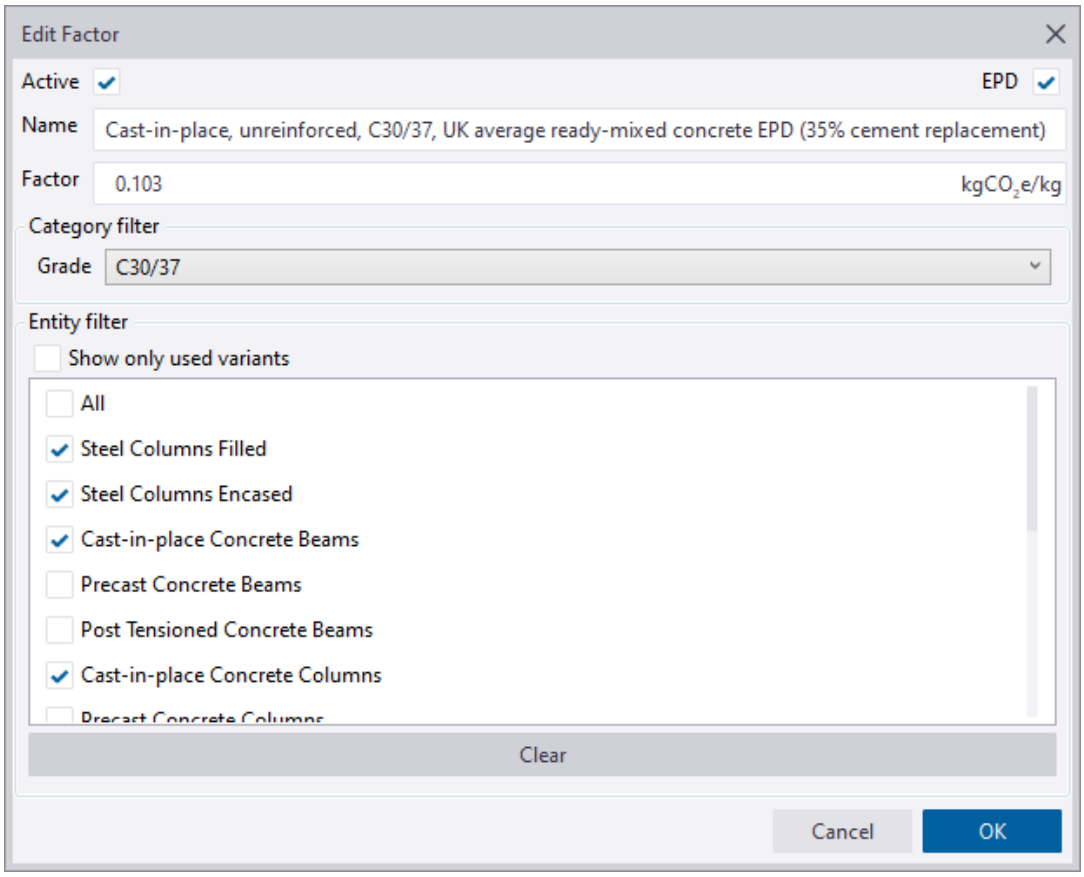
Item	Description
Active	• selected - the factor will initially be active when added • unselected - the factor will initially be inactive when added
EPD	This flag is used to indicate that the factor has come from an EPD. To make this clear it is suggested that you include 'EPD' in the name. **NOTE** Flagging a factor as from an EPD in this way does not make it uneditable. (Users might wish to update factors when EPDs get updated due to new manufacturing practices, and they might wish to change the name, or the filter determining which entities the factor applies to.)
Name	The name that you assign to a factor should be an exclusive identifier, so that it can be clearly differentiated from other factors that exist in the same category.
Factor	• defined in kgCO2/kg (metric units) • defined in lb av CO2 e/ib av (US customary units)

Item	Description
Category filter	The filters shown here will vary depending on the category. You can apply a filter in order to limit the range of applicability of the factor. **NOTE** The main category cannot be set from here. It has to be preselected in the **Category filter** on the **Embodied Carbon Factors dialog** before clicking **Add factor**.
Entity filter	You can apply an entity filter in order to further limit the range of applicability of the factor. The variants available in the entity filter depend on the applied category filter. Accepting the default setting **All** would allow the factor to be applied to all variants within the current category filter. Alternatively, unselect **All** in order to see a list of all variants to which the factor could potentially be applied. From this list you can then select one or more variants to which the factor will be exclusively applied. • Click **Clear** if you want to clear the selected variants and reselect. • Click **Show only used variants** to have the list only show those variants that have been used in the currently open model.
OK / Cancel	Click OK to add the factor to the list.

NOTE Once factors have been added, the updated list should be saved to the currently active global settings in order to make it available to new models also.

9. Edit factor

Edit factor is used to edit existing factors within the category currently shown in the **Category filter**



Item	Description
Active	• selected - the factor will initially be active after editing • unselected - the factor will initially be inactive after editing
EPD	This flag is used to indicate that the factor has come from an EPD. **NOTE** Flagging a factor as from an EPD in this way does not make it uneditable. (Users might wish to update factors when EPDs get updated due to new manufacturing practices, and they might wish to change the name, or the filter determining which entities the factor applies to.)
Name	The name that you assign to a factor should be an exclusive identifier, so that it can be clearly differentiated from other factors that exist in the same category.
Factor	• defined in kgCO2/kg (metric units), or • lb av CO2 e/ib av (US customary units)

Item	Description
Category filter	The filters shown here will vary depending on the category. You can apply a filter in order to limit the range of applicability of the factor.
Entity filter	You can apply an entity filter in order to further limit the range of applicability of the factor. The variants available in the entity filter depend on the applied category filter. Accepting the default setting **All** would allow the factor to be applied to all variants within the current category filter. Alternatively, unselect **All** in order to see a list of all variants to which the factor could potentially be applied. From this list you can then select one or more variants to which the factor will be exclusively applied. • Click **Clear** if you want to clear the selected variants and reselect. • Click **Show only used variants** to have the list only show those variants that have been used in the currently open model.
OK / Cancel	Click OK to add the factor to the list.

NOTE Initially any edits would apply to the current model only, but they can be saved to the currently active global settings in order to make them applicable to new models also.

10. Remove factor

If you select a factor in the main list and then click **Remove factor** it is deleted.

11. Export

Exports factors to a spreadsheet. You are given the choice to export:

- All factors
- Active factors
- Used factors

12. Load and Save

Button	Description
Load	Load carbon factors from the currently active global settings.

Button	Description
	WARNING Any existing factors will be discarded.
Save	Save carbon factors to the currently active global settings. **WARNING** Any existing factors in the currently active global settings will be discarded.

13. View options

View options apply to the display of the main list. When selected these apply as follows:

- Active only - only those factors with the Active flag selected are displayed in the main list.
- Entity details - the *multiple entities* text is expanded to list the entity details instead.

14. OK and Cancel

Button	Description
OK	Commit all changes made in the viewer to the database.
Cancel	Closes the dialog box without saving changes.

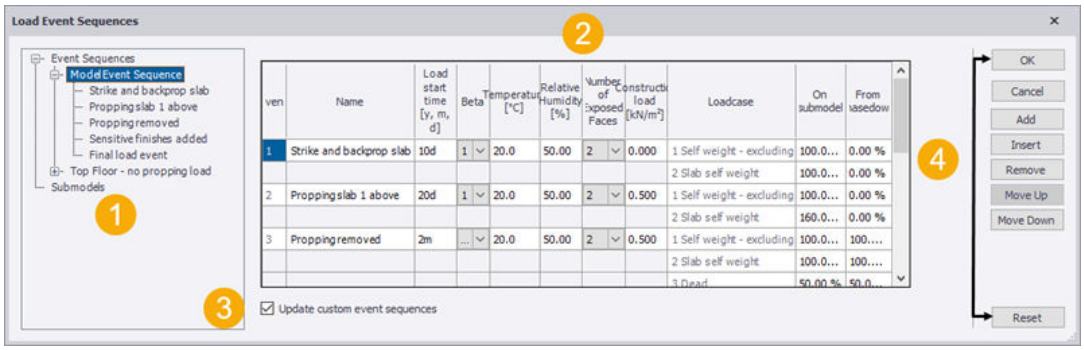
Load Event Sequences dialog box

The **Load Event Sequences** dialog box is used to define all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

To display the dialog box:

- On the **Slab Deflection** toolbar, click **Event Sequences**

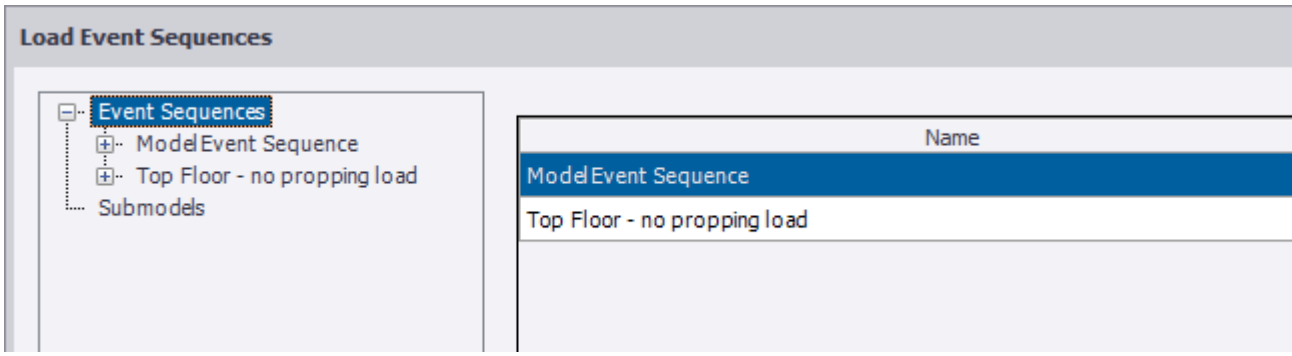
The dialog box content is described below.



1. Event sequences and submodels pane

Event Sequences

Select **Event Sequences** to show a summary of the model event sequence and any custom event sequences that have been defined.

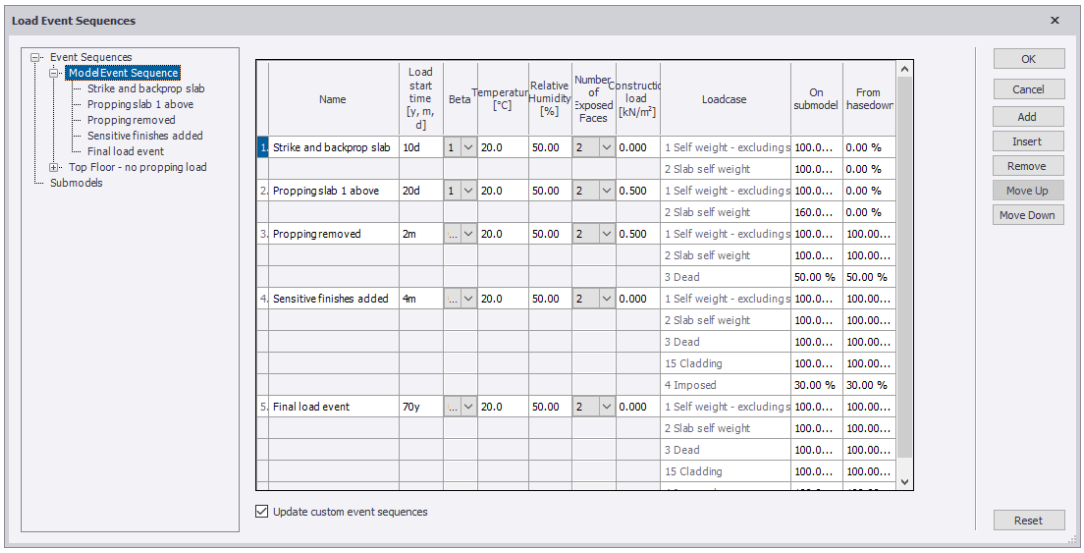


When this page is active you can:

- **Add** a Custom Load Event Sequence.
- **Remove** a selected Custom Load Event Sequence.
- Review which Event Sequences are used in submodels

Model Event Sequence

Select **Model Event Sequence** to display the various event sequence parameters for editing.

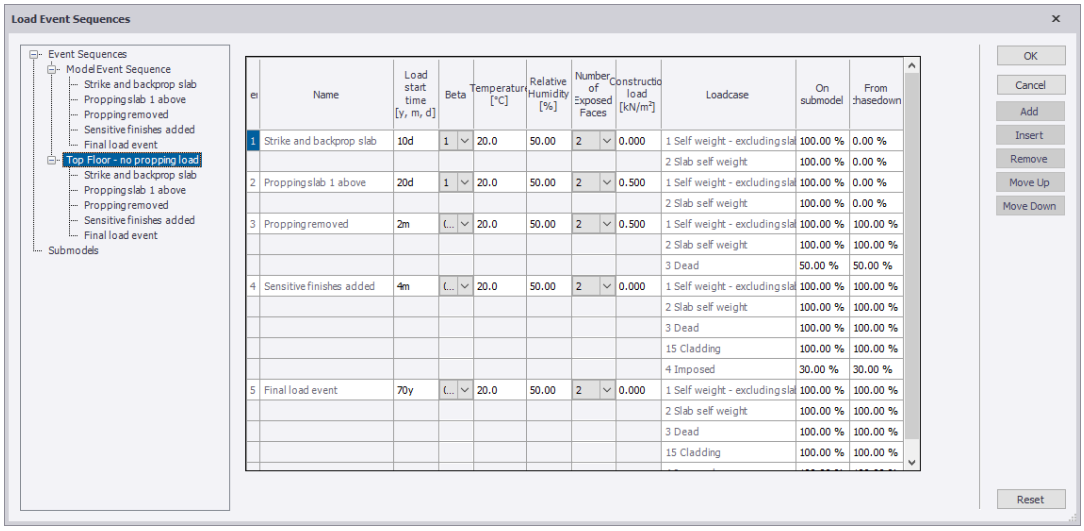


When this page is active you can:

- Review the existing Load Events and edit all the quantities (except included loadcases)
- **Add** an event to the end of the sequence
- **Insert** an event above the selected event in the sequence
- **Remove** events
- **Move Up** or **Move Down** events to re-order the sequence
- **Reset** the Model Event Sequence to the default Event Sequence in the active Settings Set

Custom Event Sequences

Pages for custom event sequences are only displayed if they have been added from the **Event Sequences** page.



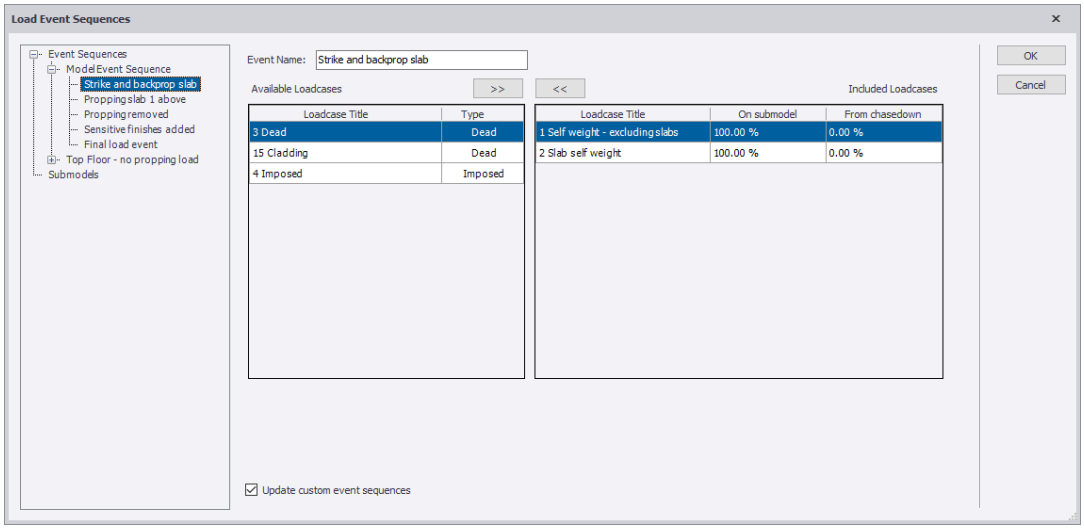
When a custom event sequence page is active you can:

- Review the existing Load Events and edit all the quantities (except included loadcases)
- **Add** an event to the end of the sequence
- **Insert** an event above the selected event in the sequence
- **Remove** events
- **Move Up** or **Move Down** events to re-order the sequence
- **Reset** the custom event sequence to be the same as the Model Event Sequence

NOTE Model Event Sequences and Custom Event Sequences do not behave differently - they are both just Event Sequences.

Load Event subpages

Load Event subpages are displayed under the Model Event Sequence, and any custom Event Sequences pages. Each Load Event has a separate subpage which is used to define the loadcases included in the Load Event.



The [>>] and [<<] buttons are used to add or remove loadcases from the Included Loadcases list.

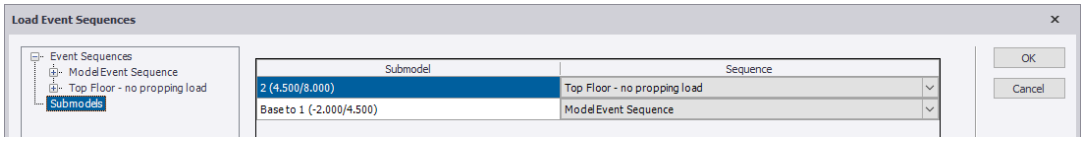
On submodel specifies the percentage of each included loadcase on the submodel.

From chasedown specifies the percentage of each included loadcase from the chasedown.

NOTE For propping loadcases - “From Chasedown” should be 0%.

Submodels

Select **Submodels** to assign a custom event sequence to a submodel if required.



- With this row selected you can edit the section size and grade for all stacks simultaneously.

2a. Event sequence parameters table (Eurocode)

Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Construction load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	1	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	1	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	2m	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	30.00 %	30.00 %
5	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

The input parameters required for each event are:

- **Event Number** - Automatic
- **Event Name** - A user defined name to help explain the event
- **Load Start time** - The time at which the event takes place
- **Beta** - a coefficient to take account of the influence of the duration of loading. (See: EC2 Clause 7.4.3) This coefficient is used to account for the fact that tension stiffening effects reduce over time.

  Tension stiffening is the phenomenon that when the concrete is not fully cracked, there is still concrete in the tension zone that transfers some tensile forces, so the stiffness is greater than that of the fully cracked stiffness (and less than the uncracked stiffness). Since this effect reduces over time, to model loads with a longer duration, you want to model a lower stiffness, which means setting beta = 0.5.

In Tekla Structural Designer, Beta defaults to 1.0 where the start event time is ≤ 30 days and 0.5 if >30 days, but may be changed for any event.

Tension stiffening reduces over time because of increased cracking and bond failure between the steel reinforcement and concrete.

These phenomena cannot be reversed, so reduced tension stiffening cannot be recovered.

For this reason, if Beta is set to 0.5, and then in a later event set to 1.0, a warning flag is shown.

- **Temperature** - Used in the calculation of the Composite Modulus. The effective age of concrete is adjusted to account for the defined temperature.
- **Relative Humidity** - Used in the calculation of creep.
- **Number of Exposed Faces** - Used for the calculation of creep.
- **Construction load** - The construction load you wish to allow for at the chosen load event.
- **Loadcase** - You select the loadcases you wish to be included in the event.
- **On submodel** - The % of load to apply to the slab, applied directly to the sub-model.
- **From chasedown** - The % of load to apply to the slab, from the reactions established in analysis of the sub-models above.

2b. Event sequence parameters table (ACI)

Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Start event	10d	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
2	Finishes added	3m	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	33.00 %	33.00 %
3	Final load event	5y	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	100.00 %	100.00 %
4	Live load only	5y	2.350	0.800	0	0.0	3 Live	100.00 %	100.00 %

The input parameters required for each event are:

- **Event Number** - Automatic
- **Event Name** - A user defined name to help explain the event
- **Load Start time** - The time at which the event takes place

- **Ultimate Creep Coefficient**
- **Aging Coefficient**
- **Number of Exposed Faces** - Used for the calculation of creep.
- **Construction load** - The construction load you wish to allow for at the chosen load event.
- **Loadcase** - You select the loadcases you wish to be included in the event.
- **On submodel** - The % of load to apply to the slab, applied directly to the sub-model.
- **From chasedown** - The % of load to apply to the slab, from the reactions established in analysis of the sub-models above.

3. Update custom event sequences

If you select the **Update custom event sequences** option, changes made to the Event in the Model Event Sequence are also replicated in custom event sequences.

4. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Add	Depends on the page selected in the Event sequences and submodels pane.
Insert	Depends on the page selected in the Event sequences and submodels pane.
Remove	Depends on the page selected in the Event sequences and submodels pane.
Move Up	Depends on the page selected in the Event sequences and submodels pane.
Move Down	Depends on the page selected in the Event sequences and submodels pane.
Reset	Depends on the page selected in the Event sequences and submodels pane.

Loading dialog box

Summary

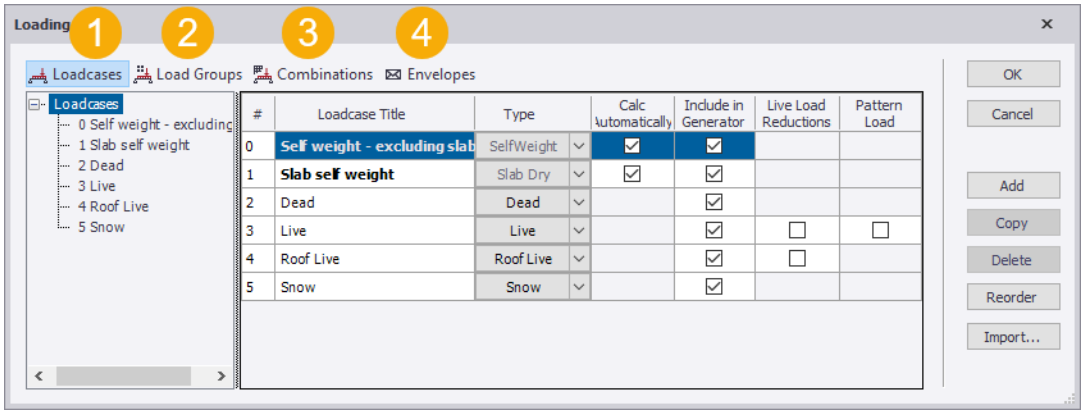
The **Loading** dialog box is used to define loadcases, load groups, combinations and envelopes.

Location

On the **Load** tab, click:

- **Loadcases** to open the dialog on the **Loadcases** page.
- **Load Groups** to open the dialog on the **Load Groups** page.
- **Combination** to open the dialog on the **Combinations** page.
- **Envelope** to open the dialog on the **Envelopes** page.

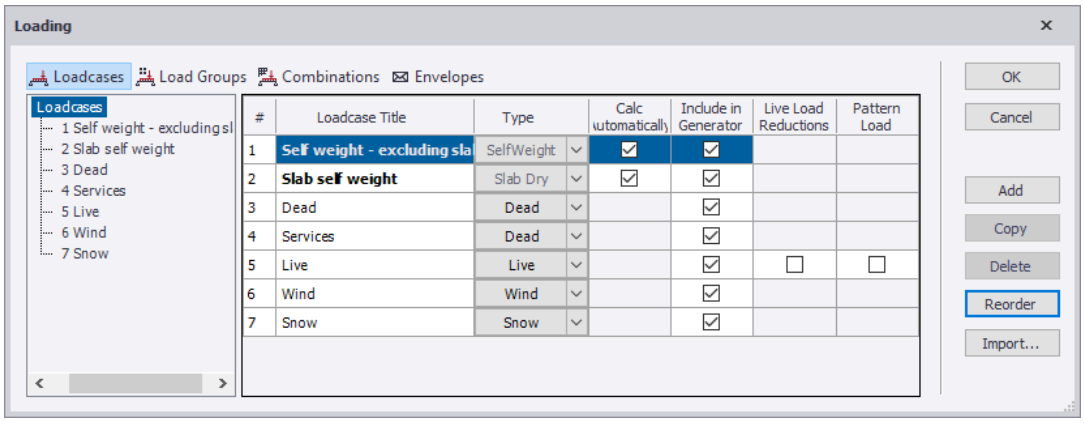
Content



1. Loadcases

Loadcases table

Select **Loadcases** in the left hand pane to show a table of loadcases that have been defined.



You can click within the table to:

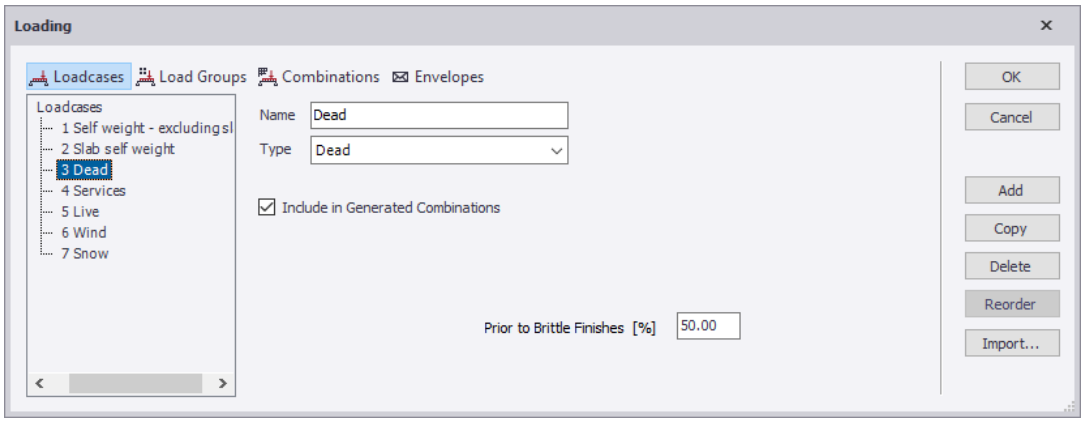
- Rename a Loadcase Title.
- Change the type of a loadcase.
- Choose whether the "Self weight excluding slabs" and Slab self weight" cases are to be calculated automatically, or defined manually.
- Choose which cases are to be included when generating the load combinations.
- Apply reductions to live/imposed cases.
- Apply pattern loading to live/imposed cases.

Using the buttons you can:

- a new loadcase to the table. (page 325)
- **Copy** an existing loadcase.
- **Delete** an existing loadcase.
- to move loadcases up or down the table. (page 329)
- loadcases from a spreadsheet. (page 343)

Loadcase parameters

Select an individual loadcase in the left hand pane to show parameters specifically applicable to that loadcase.



Certain values can only be specified by selecting individual loadcases, for example:

- For dead loadcases: **Prior to Brittle Finishes %** .
- For live/imposed loadcases: **Long Term %** .

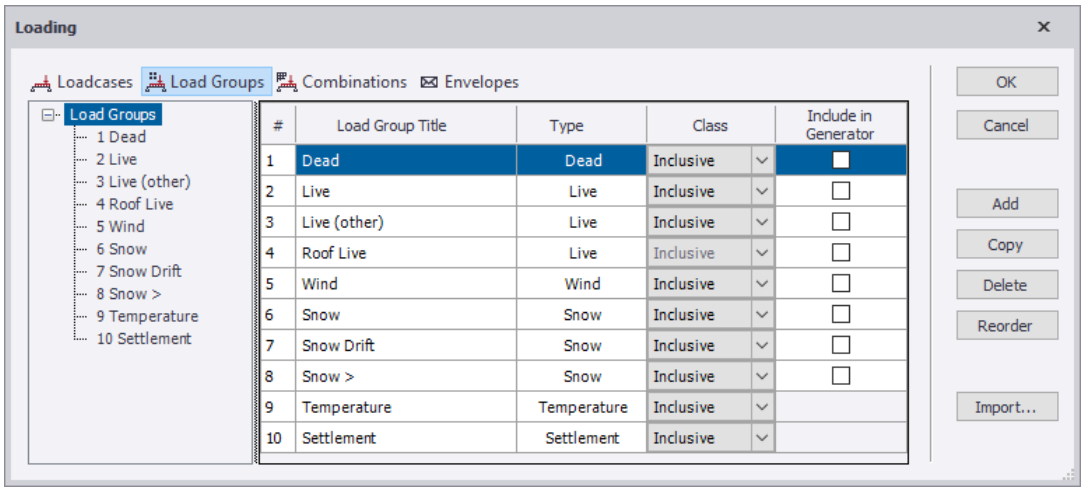
See also

Create and manage loadcases (page 325)

2. Load Groups

Load Groups table

Select **Load Groups** in the left hand pane to show a table of any load groups that have been defined.



Click within the table to:

- Rename a load group title.
- Set the load group class as Inclusive, or Exclusive.

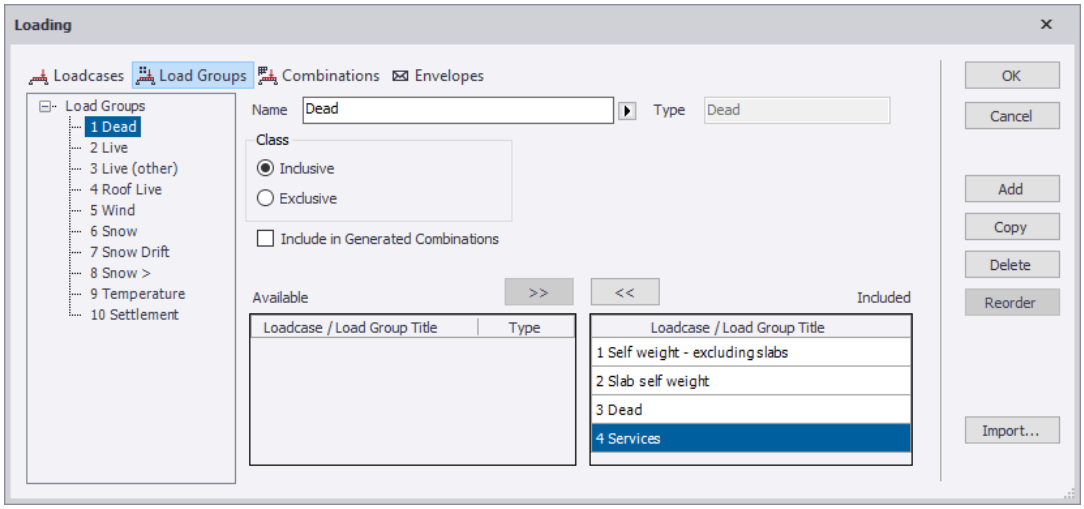
- Choose whether the load group is to be included in the generator.

Using the buttons you can:

- **Add** a new load group in the table.
- **Copy** an existing load group.
- **Delete** an existing load group.
- **Reorder** to move load groups up or down the table.
- **Import...** load groups from a spreadsheet.

Load group parameters

Select an individual load group in the left hand pane to set up the load group content.



The load group class is set as Inclusive, or Exclusive.

The loadcases/load groups currently in the load group are shown in the Included list.

Use the [>>] and [<<] buttons to add or remove loadcases/load groups from the Available list to the Included list.

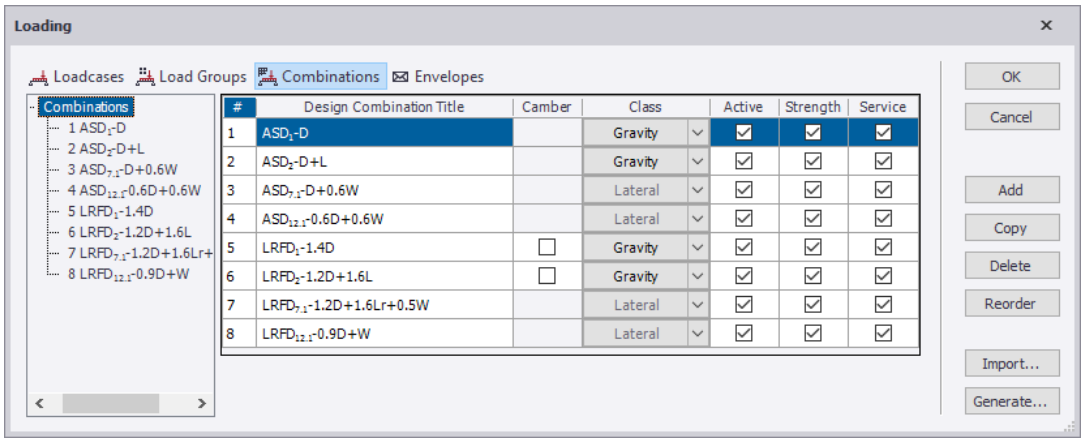
See also

Create load groups (page 329)

3. Combinations

Combinations table

Select **Combinations** in the left hand pane to show a table of combinations that have already been defined or generated.



You can click within the table to:

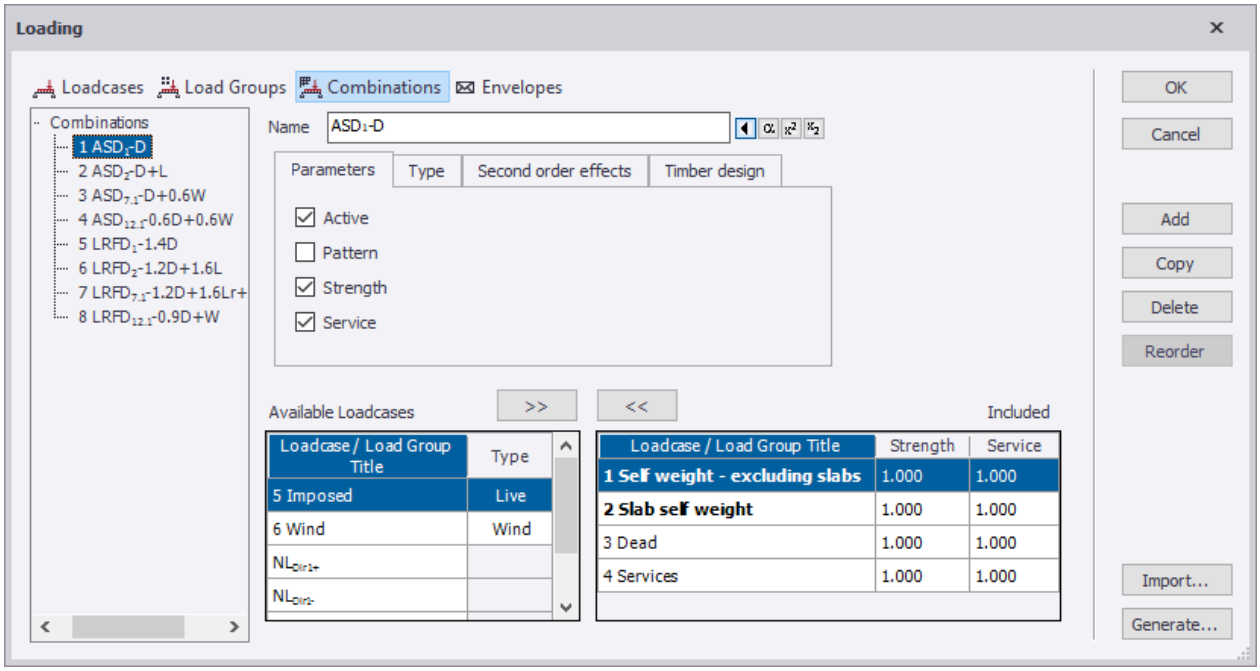
- Rename a combination title.
- Change the class (page 339) of a combination (Construction Stage, Gravity, Seismic, Seismic RSA, Modal Mass).
- Choose the gravity combination to be used for any steel beam camber calculations.
- Choose which combinations are active for analysis and design.
- Choose which active combinations are to be assessed for strength.
- Choose which active combinations are to be assessed for service.

Using the buttons you can:

- a new combination in the table. (page 341)
- **Copy** an existing combination.
- **Delete** an existing combination.
- to move combinations up or down the table. (page 347)
- combinations from a spreadsheet. (page 343)
- load combinations automatically. (page 340)

Combination parameters, type and other properties

Select an individual combination in the left hand pane to show parameters, type and other properties specifically applicable to that combination only.



The tabbed boxes are used to set:

- **Parameters**: (Active, Pattern, Strength and Service).
- **Type**: Head code dependent type settings:
  - US regional code - ACI/LRFD or ASD.
  - Eurocode - Formula type (STR, STR 6.10 a&b, GEO, ACC, Seismic, EQU)
  - India - Limit State or Working Stress
  - BS & Australia - not applicable
- **Second order effects**: the amplification factor, its direction, and whether it is applied to all loads or lateral loads only.
- **Timber design**: (US and Eurocodes only) the load duration/time effect factor.

Available and included loadcase lists

The **Available Loadcases** list displays loadcases that can be included in the selected combination.

The **Included Loadcases** list displays loadcases already included in the selected combination.

- Use the [>>] and [<<] buttons to move loadcases between the Available and Included Loadcases lists, then:
  - In the **Strength** column specify the strength factors for included loadcases.

- In the **Service** column specify the service factors for included loadcases.

- For Eurocode gravity combinations only:
  - In the **SLS Quasi** column specify the quasi service factors for included loadcases.

See also

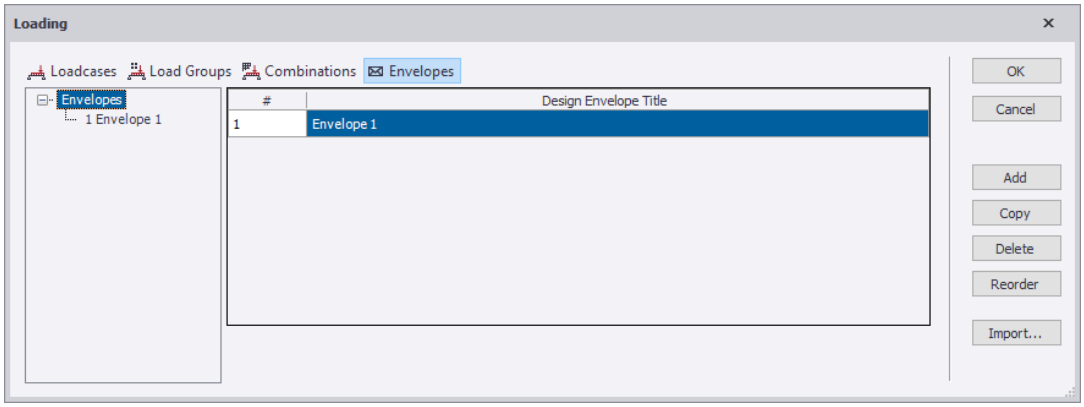
Create and manage load combinations (page 338)

Combinations (Indian Standards)

6. Envelopes

Envelopes table

Select **Envelopes** in the left hand pane to show a table of any envelopes that have been defined.

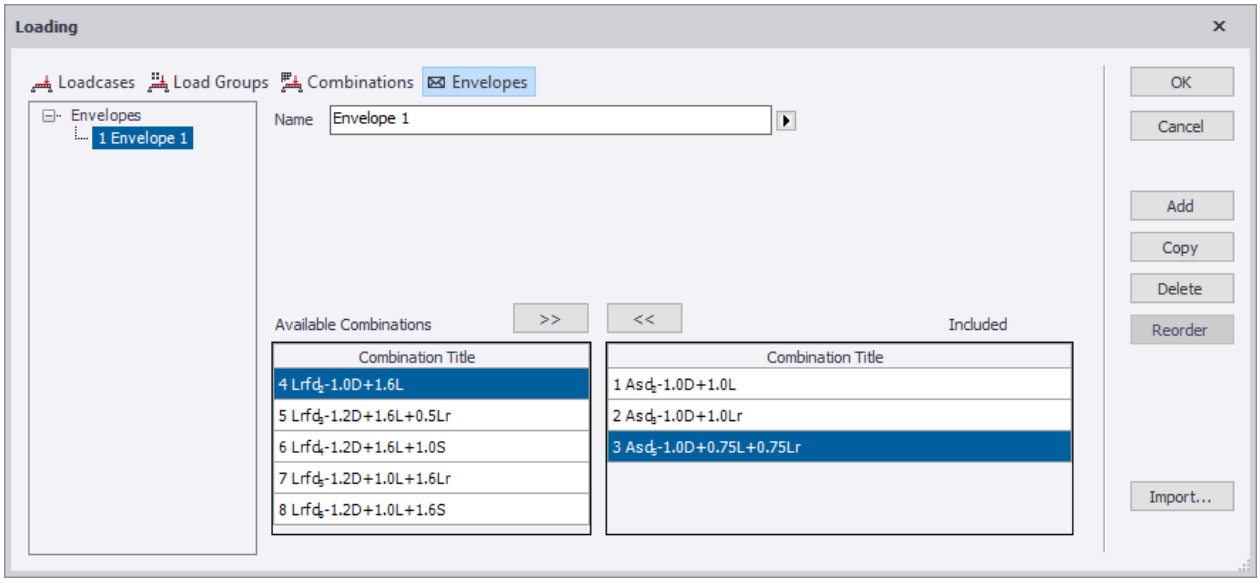


Using the buttons you can:

- **Add** a new envelope in the table.
- **Copy** an existing highlighted envelope.
- **Delete** an existing highlighted envelope.
- **Reorder** to move envelopes up or down the table.
- **Import...** to import envelopes from a spreadsheet.

Envelope parameters

Select an individual envelope in the left hand pane to set up the envelope content.



The combinations currently in the envelope are shown in the Included list.

Use the [>>] and [<<] buttons to add or remove combinations from the Available Combinations list to the Included list.

See also

Create envelopes (page 348)

Materials dialog box

The **Materials** dialog box allows you to view material database properties and manage material databases. The databases contain an extensive range of sections, materials, reinforcement, decking and connectors for each regional code and country.

NOTE Although the Materials dialog can be used to view the properties for any of the regional codes, only the properties for the currently selected regional code in **Model Settings** can be applied to the model.

To display the dialog box:

1. Click the **Home** tab.
2. Click **Materials**.

The dialog box content is described below.

Sections settings

The **Sections** page of the **Materials** dialog box allows you to view the available steel, cold formed, cold rolled, and timber sections for each regional code. You can also add new user-defined sections.

Field or button	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Unit System	Allows you to select which units are used for sections.
Region	Allows you to select the regional code whose material database properties you want to view.
Material	Allows you to select the material whose sections you want to view.
Manage Sections	Allows you to add, modify, and delete user-defined sections.
Manage Section Orders	Allows you to adjust section orders according to your needs.
Connection Resistance	Allows you to add connection resistance information for a section.
Steel Joists	Allows you to add steel joist information for a section.
Plate Dimensions	Allows you to adjust plate widths and thicknesses.

Material settings

The **Material** page of the **Materials** dialog box allows you to view properties of each grade of each material for each regional code. You can also add new user-defined sections.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Region	Allows you to select the regional code whose material database properties you want to view.
Material Type	Allows you to select the material type whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined material grades to the **Available Grades** list.

Button or field	Description
View...	Allows you to view the properties of pre-defined material grades.
Delete	Allows you to permanently delete user-defined material grades.
>> Default	Allows you to make the currently highlighted material grade a default material grade, so that it appears in the **Default Grade** field.

Reinforcement settings

The **Reinforcement** page of the **Materials** dialog box allows you to view the properties of reinforcement classes and bar sizes.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Region	Allows you to select the regional code whose material database properties you want to view.
Country	Allows you to select the country whose material database properties you want to view.
Type	Allows you to select the type whose material database properties you want to view.
Rib Type	Allows you to select the rib type whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined reinforcement classes or bar sizes to the appropriate list.
View...	Allows you to view the properties of pre-defined reinforcement classes or bar sizes.
Delete	Allows you to permanently delete user-defined reinforcement classes or bar sizes.
>> Default	Allows you to make the currently highlighted reinforcement class a default class, so that it appears in the **Default Class** field.

Decking settings

The **Decking** page of the **Materials** dialog box contains different sub pages: **Metal Decking** and **Precast Concrete Decking**. The **Metal Decking** page allows you to view the properties of different profiles and gauges, and the **Precast Concrete Decking** page allows you to view the properties of different precast concrete planks and depths. The sub pages contain the same commands.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Country	Allows you to select the country whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined profiles, gauges, planks, or depths to the appropriate list.
View...	Allows you to view the properties of pre-defined profiles, gauges, planks, or depths.
Delete	Allows you to permanently delete user-defined profiles, gauges, planks, or depths.
>> Default	Allows you to make the currently highlighted profile, gauge, plank, or depth a default option, so that it appears in the appropriate **Default** field.

Shear Connectors settings

The **Shear Connectors** page of the **Materials** dialog box allows you to view the properties of different connectors.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Region	Allows you to select the regional code whose material database properties you want to view.

Button or field	Description
Country	Allows you to select the country whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined sources or connectors to the appropriate list.
View...	Allows you to view the properties of pre-defined sources or connectors.
Delete	Allows you to permanently delete user-defined sources or connectors.
>> Def. Metal	Allows you to make the currently highlighted option the default option for metal, so that it appears in the **Default for Metal** field.
>> Def. Concrete	Allows you to make the currently highlighted option the default option for concrete, so that it appears in the **Default for Concrete** field.

Bolts (Rods in US) settings

The **Bolts**/**Rods** page of the **Materials** dialog box allows you to view the properties of bolt/rod classes and sizes.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the bolts/rods database to a newer version.
Unit System	Allows you to select which units are used for bolts/rods.
Region	Allows you to select the regional code whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined bolts/rods.
Copy...	Allows you to copy an existing bolt/rod.
View...	Allows you to view the properties of the selected bolt/rod.
Delete	Allows you to permanently delete user-defined bolts/rods.

Button or field	Description
>> Default	Allows you to make the currently highlighted bolt a default, so that it appears in the Default Bolt/Rod field.

Welds settings

The **Welds** page of the **Materials** dialog box allows you to view the properties of welds leg lengths and electrodes.

NOTE Electrodes are US only

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the welds database to a newer version.
Unit System	Allows you to select which units are used for welds.
Region	Allows you to select the regional code whose material database properties you want to view.
Leg Lengths	Allows you to view the Leg Lengths currently in the welds database.
Defaults...	Allows you to set and view the default weld leg lengths
Reset	Allows you to reset the default weld leg lengths and delete all user defined welds.
Add	If necessary, allows you to add user-defined leg lengths to the database.
Electrodes	
Electrodes	Allows you to view the list of electrodes in the welds database.
Add	If necessary, allows you to add user-defined electrodes to the database.
View	Allows you to view the properties of pre-defined electrodes.
Delete	Allows you to permanently delete user-defined electrodes.
>> Default	Allows you to make the currently highlighted option the default electrode, so that it appears in the **Default Electrode** field.

Model settings

The **Model** page of the **Materials** dialog box allows you to both update the material databases with new properties from the model and update material properties in the model with new values from the material databases.

Button, field, or column	Description
In Database	Uses **?** to display if there are inconsistencies between the material data in the model and the material databases.
Add to Database	Allows you to update the material databases with values from the model.
Update from Database	Allows you to update the material properties in the model with values from the material databases.
Show only objects not saved in the database	Hides the model properties that are consistent with the material databases.

See also

Manage materials (page 1265)

Model Settings dialog box

The majority of settings for the current project are accessed from the **Model Settings** dialog box.

Location

On the **Home** ribbon tab, click  **Model Settings**.

Content

See: Model settings (page 1243)

After adjusting the settings, do one of the following:

- To apply the changes to the current project: click **OK**
- To save the changes back to the active settings set for future use: click **Save...**
- To cancel the changes: click **Cancel**
- To revert to the model settings specified in the active settings set: click **Load...**

See also

Apply and manage model settings (page 1263)

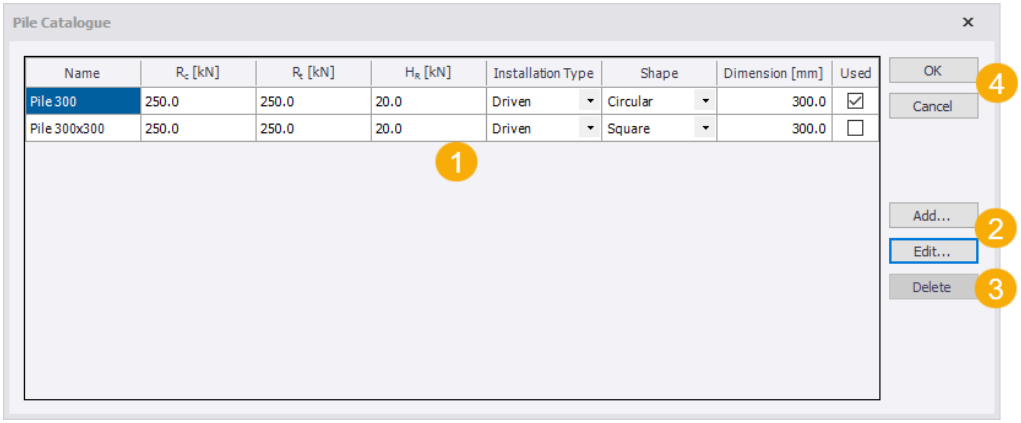
Pile catalogue

The **Pile Catalogue** is used to add or edit pile dimensions and capacities.

To open the dialog:

1. Click the **Foundations** tab.
2. Click **Pile Catalogue**.

   The **Pile Catalogue** dialog is displayed.



1. Pile data

If working to Eurocodes the following data is required.

Title	Description
Name	Pile name
$R_{c,d,1}$	Axial compressive resistance GEO
$R_{t,d,1}$	Axial tensile resistance GEO
$R_{c,d,2}$	Axial compressive resistance STR
$R_{t,d,2}$	Axial tensile resistance STR
$H_{R,1}$	Lateral resistance GEO
$H_{R,2}$	Lateral resistance STR
Installation type	• Driven • Bored • CFA

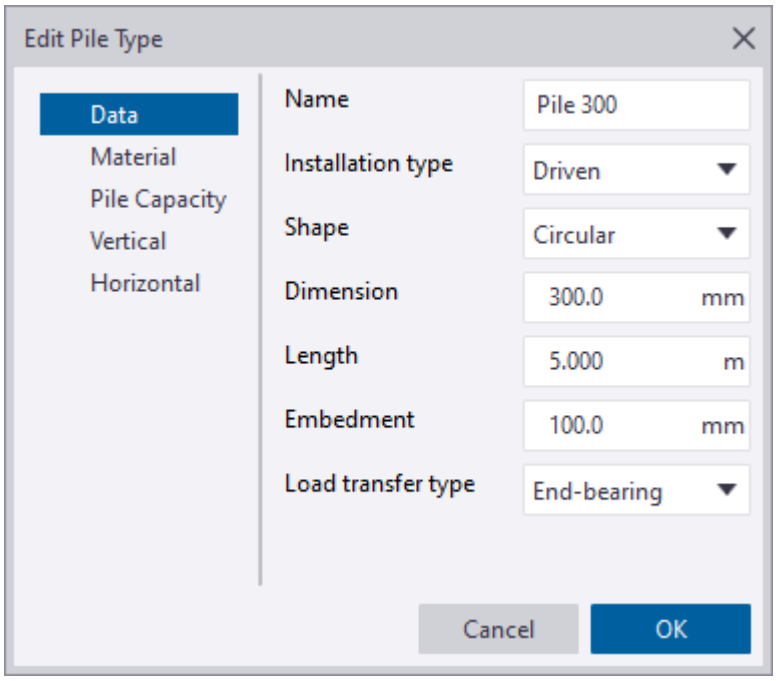
Title	Description
Shape	• Circular • Square
Dimension	Diameter, or length of side
Used	Automatically checked when the pile has been assigned in the model

If working to other regional codes the following data is required.

Title	Description
Name	Pile name
R_c	Axial compressive resistance
R_t	Axial tensile resistance
H_R	Lateral resistance
Installation type	• Driven • Bored • CFA
Shape	• Circular • Square
Dimension	Diameter, or length of side
Used	Automatically checked when the pile has been assigned in the model

2. Add/Edit

Opens the below dialog for adding/editing piles.



Data	
Name	Pile name
Installation type	• Driven • Bored • CFA
Shape	• Circular • Square
Dimension	Diameter, or length of side
Length	Length of pile
Embedment	Length of embedment
Load transfer type	• End-bearing • Friction

Material
NOTE Material information defined here has no design significance. It is assigned solely to allow embodied carbon to be calculated and reported for the 'Pile' construction type.

Material	
Material type	• Steel • Concrete • Timber • General • Cold Formed • Cold Rolled
Material grade	Grade of selected material.
Volume adjustment	There are many and varied piling systems. Although a theoretical diameter and length are input, in practice these can be minimums or approximations. A volume adjustment can therefore be applied to allow different systems to be catered for. Example - an in-situ under-reamed piling system might be given an adjustment >> 100%
Reinforcement grade	For concrete, the reinforcement grade is selected by clicking [...]
Reinforcement Volume %	If you set a percentage > 0% the calculated mass of reinforcement will be able to be used when calculating reinforcement ECFs. See the below note for further details.
Reinforcement Mass	Calculated using the above volume %.

NOTE Reported carbon for cast-in-place concrete piles

There are two options available:

- Option 1 - Simplest
  - Treat in the same was as precast - apply a single factor that allows for both concrete and rebar.
  - Can use the existing “precast” ECF, or more likely create one specifically for piles.
- Option 2 - Treat concrete and rebar separately:
  - Edit appropriate ECFs in the concrete category to activate the **Pile** variant in the Entity filter.
  - In the **Pile Catalogue** set a reinforcement % for each pile. The resulting reinforcement mass will then be used to calculate the

embodied carbon mass of the pile reinforcement. This can then be picked up by appropriate reinforcement ECFs in the reinforcement category.

Pile Capacity	
Axial compressive resistance	Axial compressive resistance
Axial tensile resistance	Axial tensile resistance
Lateral resistance	Lateral resistance

Vertical	
Linearity	• Spring linear • Spring non-linear
Vertical spring stiffness	Linear springs only.
Vertical compression spring stiffness	Non-linear springs only.
Vertical compression limit	Non-linear springs only. The spring compression capacity.
Vertical tension spring stiffness	Non-linear springs only.
Vertical tension limit	Non-linear springs only. The spring tension capacity.

Horizontal	
Horizontal restraint	• Fixed • Spring linear
Horizontal spring stiffness	Linear springs only.

3. Delete

Deletes the selected pile data.

4. OK/Cancel

Click **OK** to save, or **Cancel** to close without saving.

Punching shear reinforcement properties dialog box

Summary

The **Punching Shear Reinforcement Properties** dialog box allows you to modify the reinforcement used in a punching shear check. The graphic in the

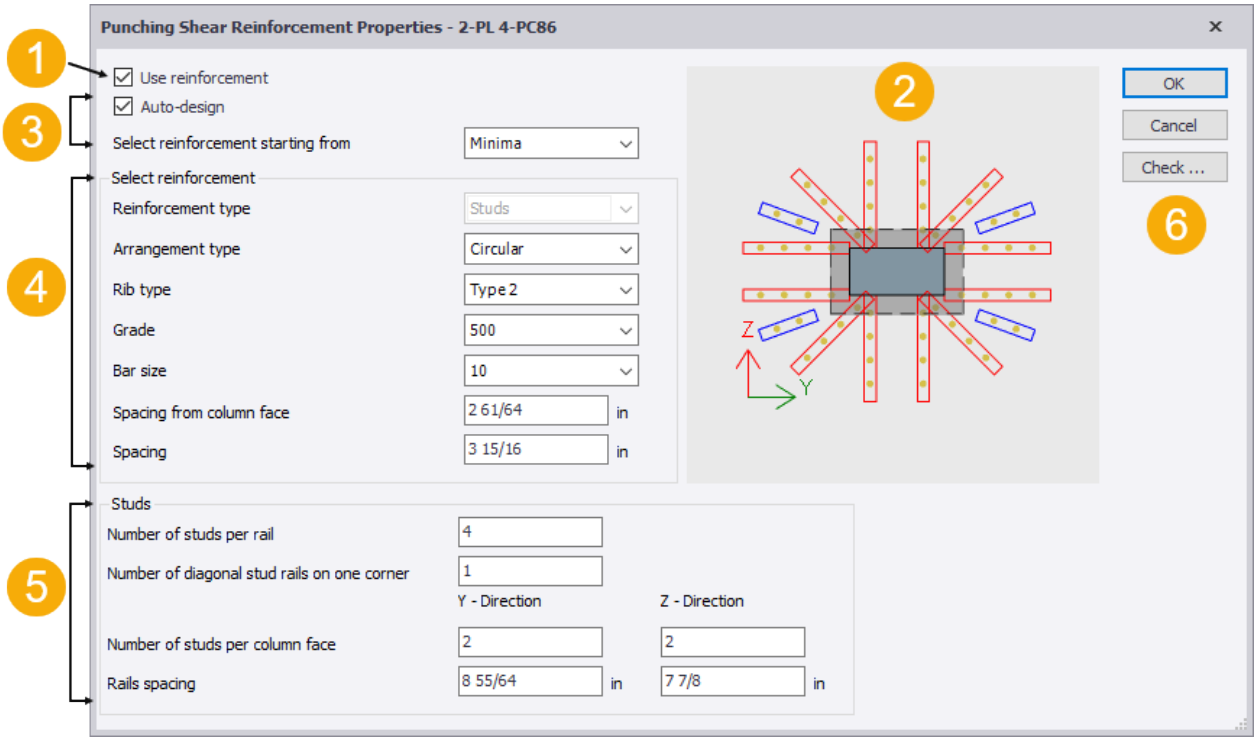
dialog box previews the reinforcement and updates to match the changes that you make. The graphic also indicates other items that are specific to the check location.

Location

To display the dialog:

1. Right click on an existing punching check item.
2. In the context menu, select **Edit Reinforcement**.

Content



1. Use reinforcement

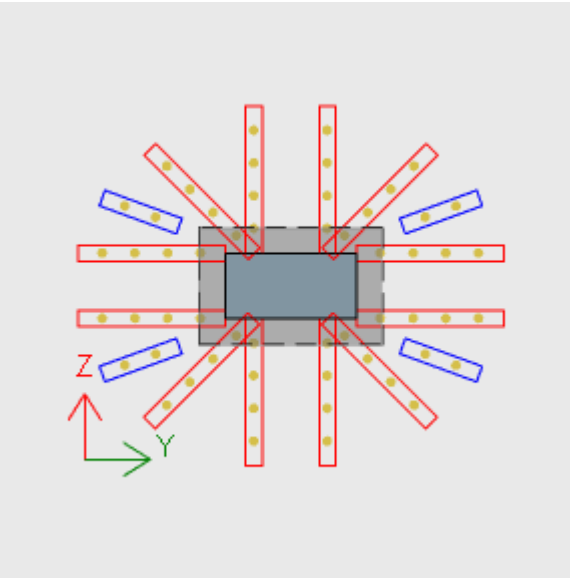
Selecting the **Use reinforcement** option allows you to apply a default punching reinforcement arrangement that can either be checked or used as the starting reinforcement for an auto design.

2. Preview graphic

The components of the preview graphic are as follows:

Stud rail reinforcement

Stud rail reinforcement is only shown if you have selected the **Use reinforcement** option.



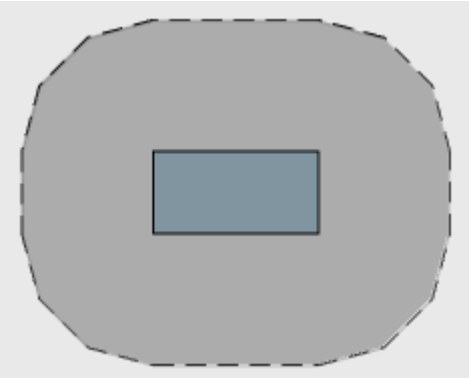
The red design rails signify rails considered in design.

The blue detailing rails are only considered for detailing purposes.

Perimeters

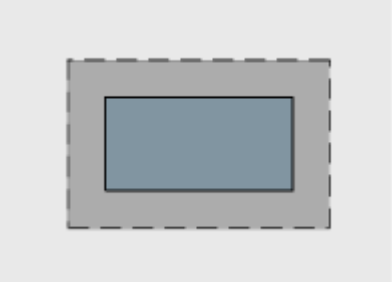
Perimeters are displayed as dashed lines around the column section and will have different shapes and positioning depending upon the regional code being worked to.

Control Perimeter (EC)



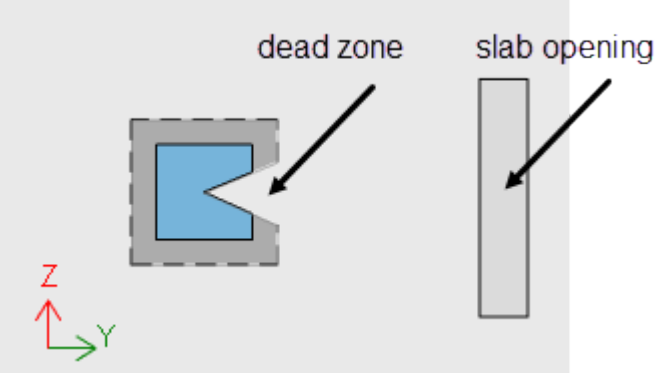
The control perimeter is the perimeter of the dark grey area as shown above. This will vary depending on existing slab edges and openings.

Critical Perimeter (ACI)



The critical perimeter is the perimeter of the dark grey area as shown above. This will vary depending on existing slab edges and openings.

Slab opening dead zones



Slab opening dead zones are displayed as interruptions in perimeters and reinforcement.

Punching shear check local axis



Using the YZ system orients the X axis upwards following the right-hand rule. This is also the local axis system for the column elements, so it is easier to relate.

Additional Perimeters

If reinforcement is found to be required, additional perimeters will be displayed representing positions beyond the critical/control perimeter where the reinforcement requirement is checked.

3. Auto-design

If you select the **Auto-design** option, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. You can then select the starting point in the **Select reinforcement starting from** list.

The **Select reinforcement starting from** list allows you to select the staring point for auto-design procedures. The options are:

- **Minima**: removes the current arrangement and starts the reinforcement with the minimum allowed bar size.
- **Current**: auto design starts from the current bar arrangement. The **Current** option is only available if you have selected **Use reinforcement** in the **Properties** window.

4. Select reinforcement parameters

Reinforcement type

In the current release, only stud reinforcement is available.

Arrangement type

Allows you to define whether the reinforcement arrangement is orthogonal or circular. For more information, see: .

Rib type

Allows you to specify the reinforcement rib type.

Grade

The reinforcement grades that are available here are set in the **Materials** dialog box.

Bar size

The reinforcement bar sizes that are available here are in the **Materials** dialog box.

Spacing from column face

Defines the spacing of the first bar in each rail from the column face.

NOTE The option is only available if the **Use reinforcement** option has been selected.

Spacing

Defines the spacing between bars along each rail.

NOTE The option is only available if the **Use reinforcement** option has been selected.

5. Studs parameters

Number of studs per rail

Allows you to define the number of studs on each rail.

- Minimum number of studs = 2
- Maximum number of studs = 20
- The actual number of studs per rail depends on the design, user editing and punching check characteristics (position, presence of openings, spacing of rails)

Number of diagonal stud rails on one corner

Allows you to define the number of stud rails adjacent to each corner of the column.

NOTE The option is only displayed when **Arrangement type** is set to **Circular**.

- Positioned at corners and at an angle with the check local axis
- Placed at equal angles from each other and the two orthogonal directions starting at 45 degrees for a single corner rail
- Angles are measured with center at the origin of the two closest face rails
- The minimum number of rails is 0
- The maximum number of rails is 10

Number of studs per column face

Allows you to define the number of stud rails adjacent to the column face in the local y and z directions.

- Positioned between column bounding rectangle projected boundaries and aligned with the check local axis
- Arranged symmetrically in relation to the section bounding rectangle centerline, except at edges where the edge rail can be offset to the inside of the slab by a minimum value equal to the slab concrete cover.
- The minimum number of rails is 1
- The maximum number of rails is 20

Rails spacing

Allows you to define the spacing between rails in the local y and z directions.

6. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.

See also

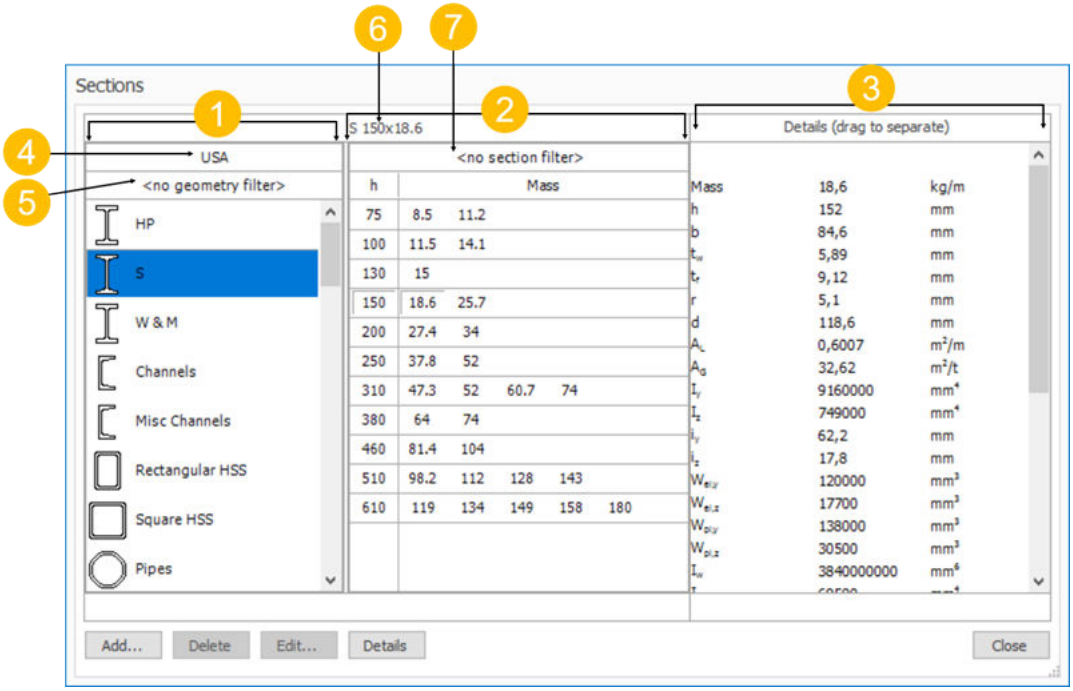
Create punching checks (page 812)

Design punching shear (page 815)

Sections dialog box

The **Sections** dialog box is used to select sections to apply to members and also to manage the sections stored in the material database.

The dialog box content is described below.

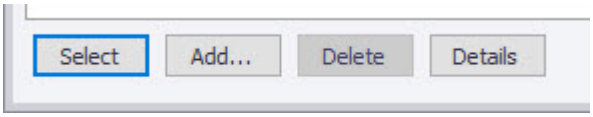


1. **Page** pane: views the different section geometries.

2. **Item** pane: views the different sizes of the selected section when you select a section geometry **Page** pane. Any user-defined sections are marked with ***.
3. **Details** pane: views the details of the selected section when you click the **Details** button.
4. **Country** droplist: allows you to display the sections of each country.
5. **Geometry filter**: allows you to filter sections according to their geometry.
6. **Selected section**: displays the details of the currently selected section.
7. **Section filter**: allows you to filter sections according to their details.

Selecting a section to apply to member

In the **Properties** window or **Properties** dialog box, by selecting **<New\Edit...>** in the **Sections** list, the **Sections** dialog box opens with the following buttons displayed:

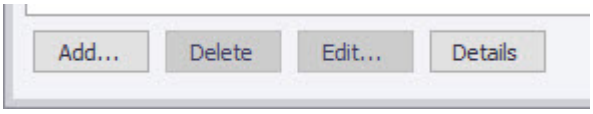


- **Select** allows you select a section to be applied to a member.
- **Add...** allows you to add a user-defined section to the Sections database.

  For further details, see: Add user-defined sections (page 1266)
- **Delete** allows you to delete a user-defined section from the Sections database. (This button is only active when a user-defined section is picked.)
- **Details** opens the **Details** pane and displays the properties of the currently selected section.

Managing sections

On the **Home** ribbon, clicking **Materials** --> **Manage Sections** opens the **Sections** dialog box with the following buttons displayed:



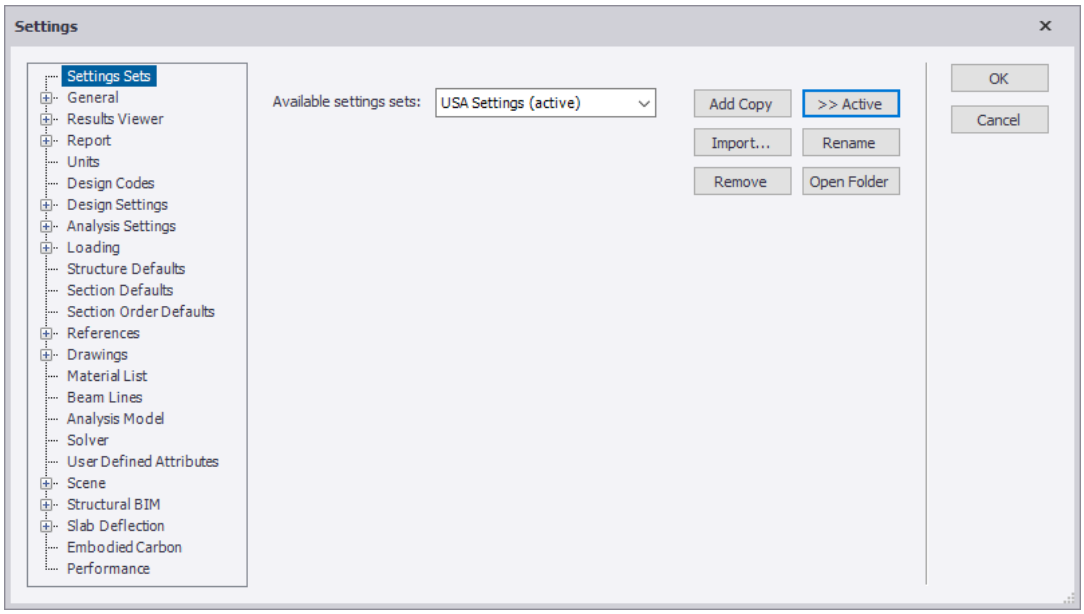
- **Add...** allows you to add a user-defined section to the Sections database.

  For further details, see: Add user-defined sections (page 1266)
- **Delete** allows you to delete a user-defined section from the Sections database. (This button is only active when a user-defined section is picked.)
- **Edit...** allows you to modify the properties of a user-defined section. (This button is only active when a user-defined section is picked.)

- **Details** opens the **Details** pane and displays the properties of the currently selected section.

Settings dialog box

The **Settings** dialog box allows you to manage defaults, collectively referred to as a settings set, that are used in future projects. In addition, you can manage general and display settings that are applied instantly to the current work session.



Location

On the **Home** ribbon tab, click **Settings**.

Content

See: Global settings (page 1232)

See also

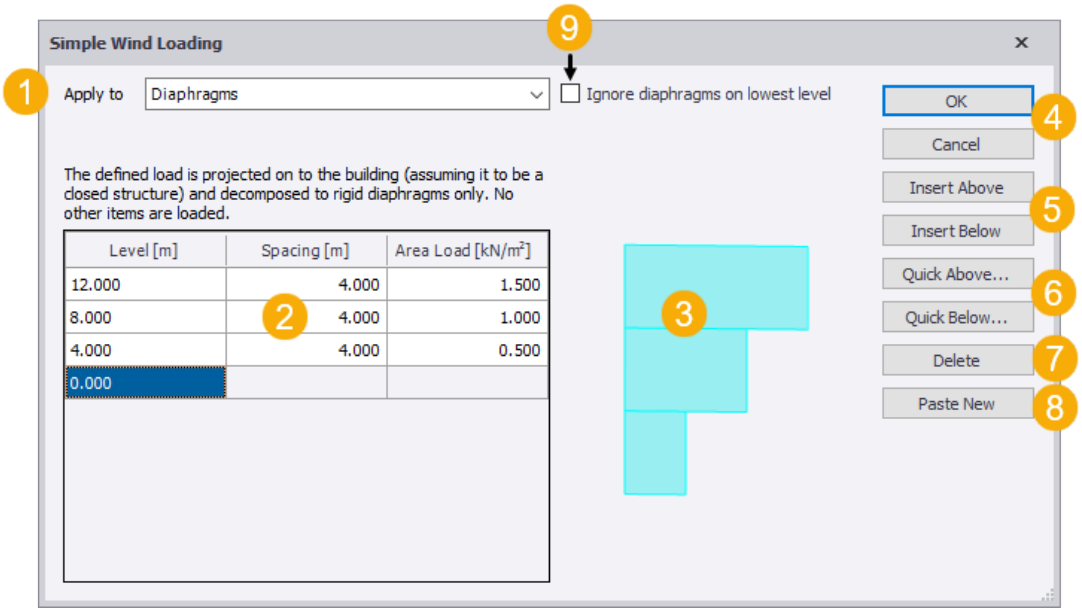
Manage settings sets (page 1220)

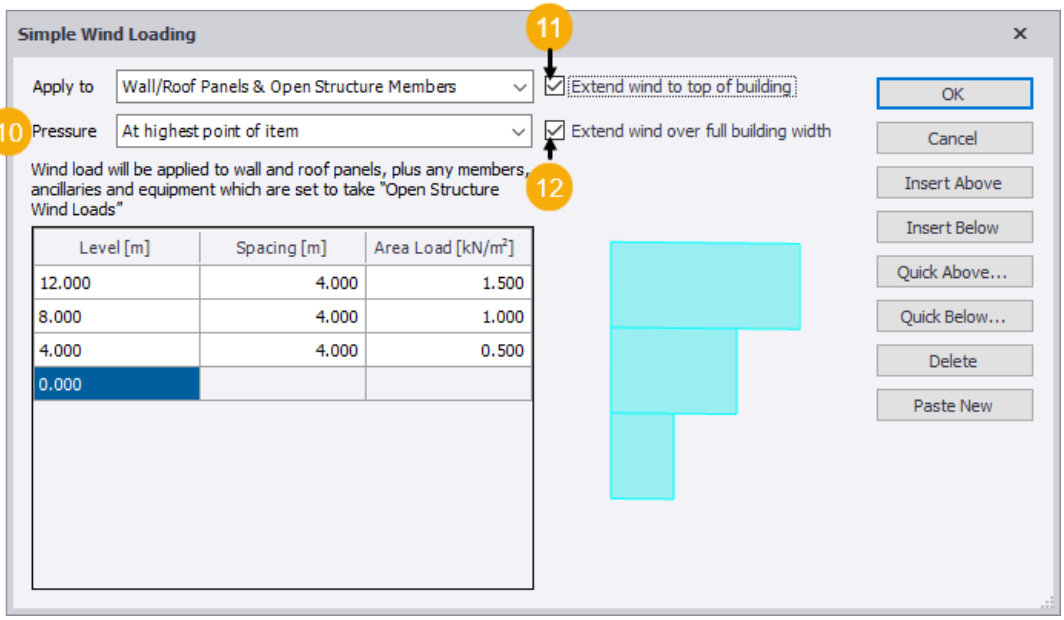
Simple Wind Loading dialog box

The **Simple Wind Loading** dialog box is accessed by clicking **Simple Wind** on the **Load** ribbon. It is used apply wind load to diaphragms, or to wall/roof panels and open structure members; it doesn't require a wind model to be generated.

NOTE you must select a manually defined wind loadcase in the **Loading** list to access **Simple Wind**

The dialog box is shown below as it appears when applied to diaphragms (top) and wall/roof panels & open structure members (bottom).





See also

Instructions for applying simple wind loads to diaphragms (page 398)

Instructions for applying simple wind loads to wall/roof panels and open structure members (page 411)

Apply wind loading (page 372)

1. Apply to (droplist)

From the droplist choose whether you want the simple wind load to be applied to:

- **Diaphragms** - The load from lowest to highest level will be decomposed as point loads on to those rigid diaphragms that exist between the lowest and highest levels defined.

  NOTE If the **Ignore diaphragms on lowest level** option is checked the lowest level diaphragms are not considered in the decomposition process.

- **Wall/Roof Panels & Open Structure Members** - The load is decomposed to wall and roof panels, plus any members, ancillaries and equipment which are set to take 'Open Structure Wind Loads'.

2. Load table

This table is used to define the area load acting between the lowest and highest level specified.

To create a stepped loading profile up the height of the building, use the **Insert Above**, **Insert Below**, **Quick Above** and **Quick Below...** buttons.

TIP You are not restricted to specifying the highest level to align with the topmost diaphragm. The flexibility this provides can be useful for example when you have parapet walls that haven't been modeled - by setting the level to be the top of the parapets you can still get the wind pressure into the building.

3. Load profile

The information defined in the Load table is drawn as a load profile.

4. OK and Cancel

Button	Description
OK	Saves the simple wind load.
Cancel	Closes the dialog box without saving changes.

5. Insert Above/Below

Button	Description
Insert Above	Inserts a single level into the Load table above the selected level.
Insert Below	Inserts a single level into the Load table below the selected level.

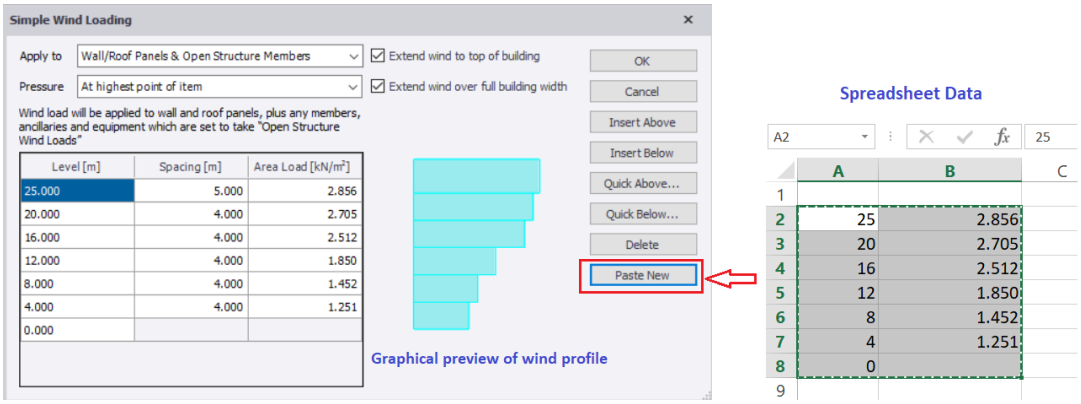
6. Quick Above/Below

Button	Description
Quick Above	Inserts multiple levels into the Load table above the selected level. **NOTE** Levels are input by defining spacings between levels separated by commas. If you have a number of spacings that are at the same centers, then you can specify these as number x spacing. e.g. to specify one spacing of 4m, followed by 3 spacings of 6m, enter `4,3x6`
Quick Below	Inserts multiple levels into the Load table above the selected level.

7. Delete

Deletes the selected row from the Load table.

8. Paste New



Pastes loading information directly into the Load table from the clipboard. Any existing data in the table is deleted.

To use this method, data should first be copied to the clipboard from a spreadsheet. Each row should contain two columns in the following format.

1. Level
2. Wind pressure

9. Ignore diaphragms on lowest level option

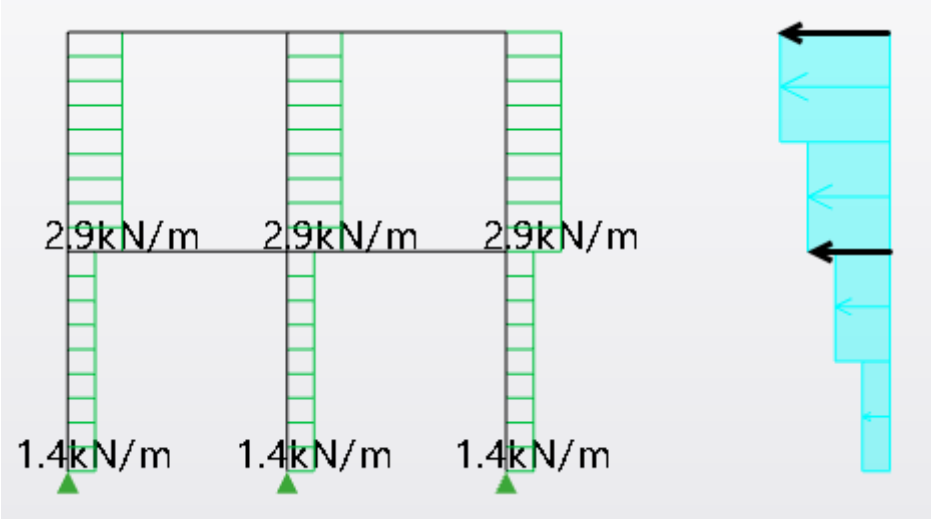
Click this option if you don't want any of the load to be decomposed on to the lowest diaphragm within the range of levels defined in the Loads table.

NOTE If for some reason there are diaphragms at the ground level, then you may decide to check **Ignore diaphragms on lowest level** in order to ensure no load is lost directly to the foundations.

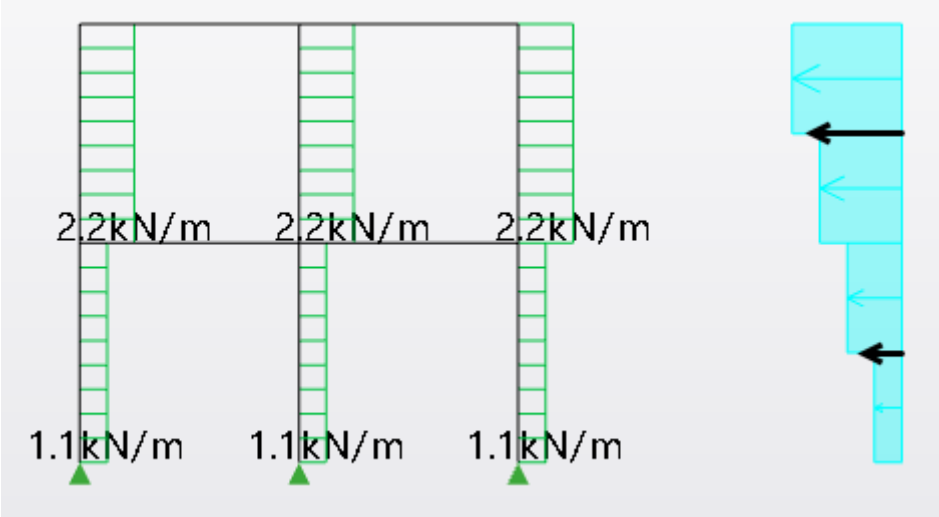
10. Pressure (droplist)

From the droplist choose whether you want the pressure to be applied:

- **At highest point of item** - wall/roof panels & open structure members are loaded using the pressure at the top of the item.



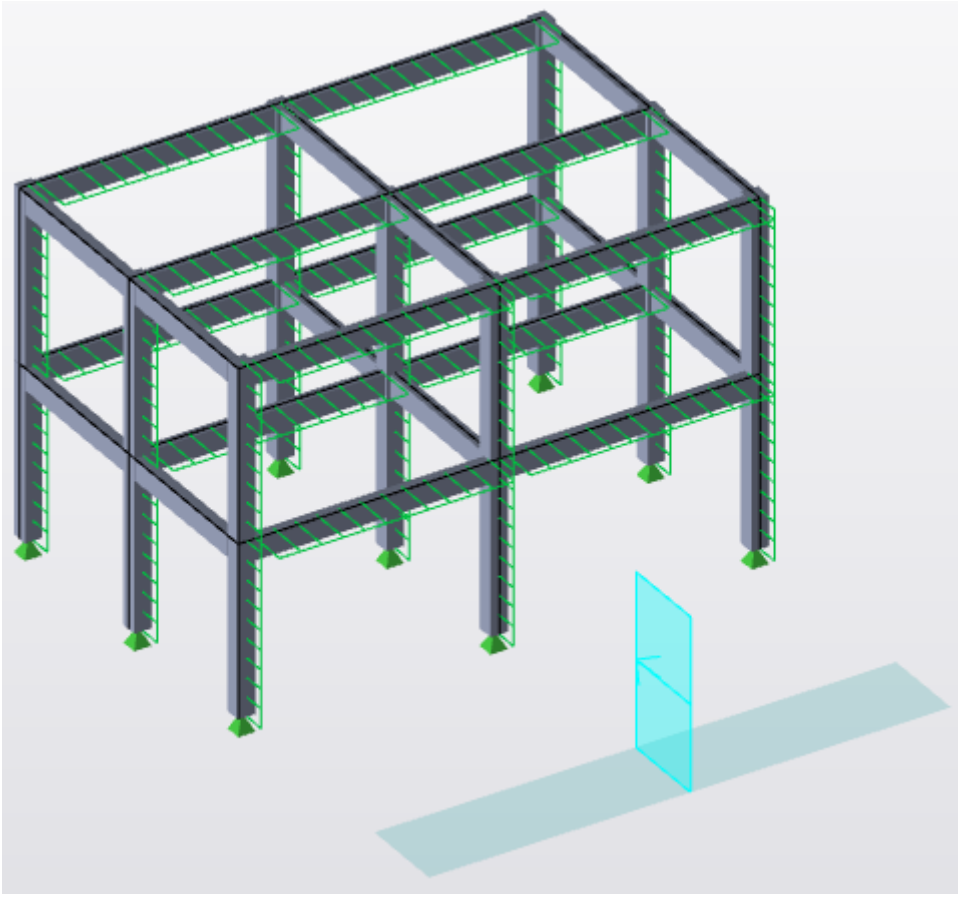
- **Average between highest and lowest point** - wall/roof panels & open structure members are loaded using the average pressure between the top and bottom of the item.



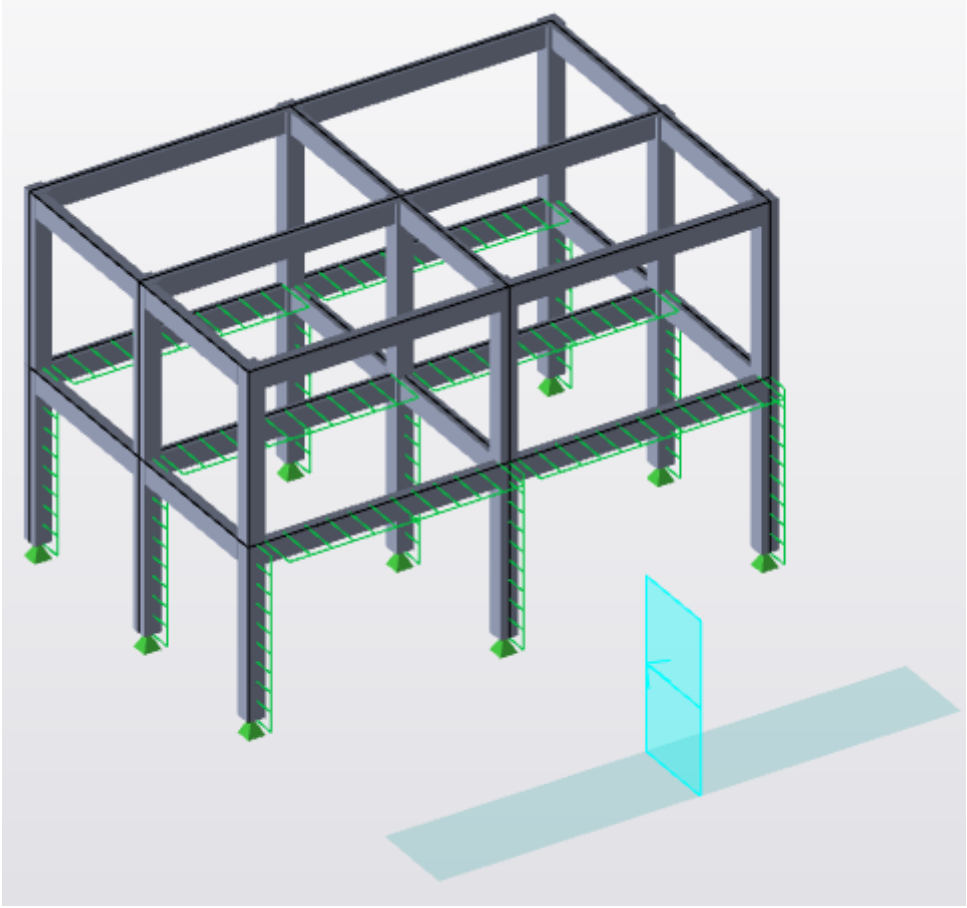
11. Extend wind to top of building option

This option is used to control the extent of wind vertically.

- When selected - wall/roof panels & open structure members see the wind load, irrespective of whether they lie within, or outside the load height.



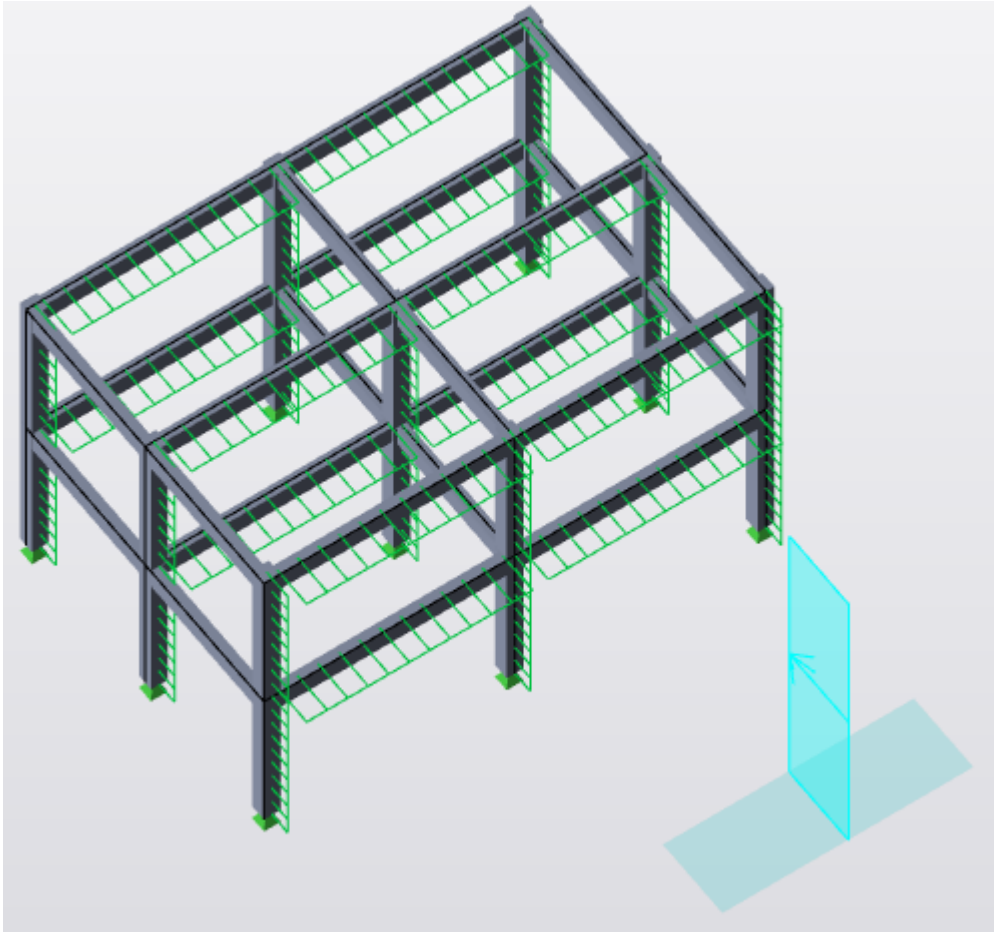
- When unselected - only those wall/roof panels & open structure members that lie within the load height see the wind load.



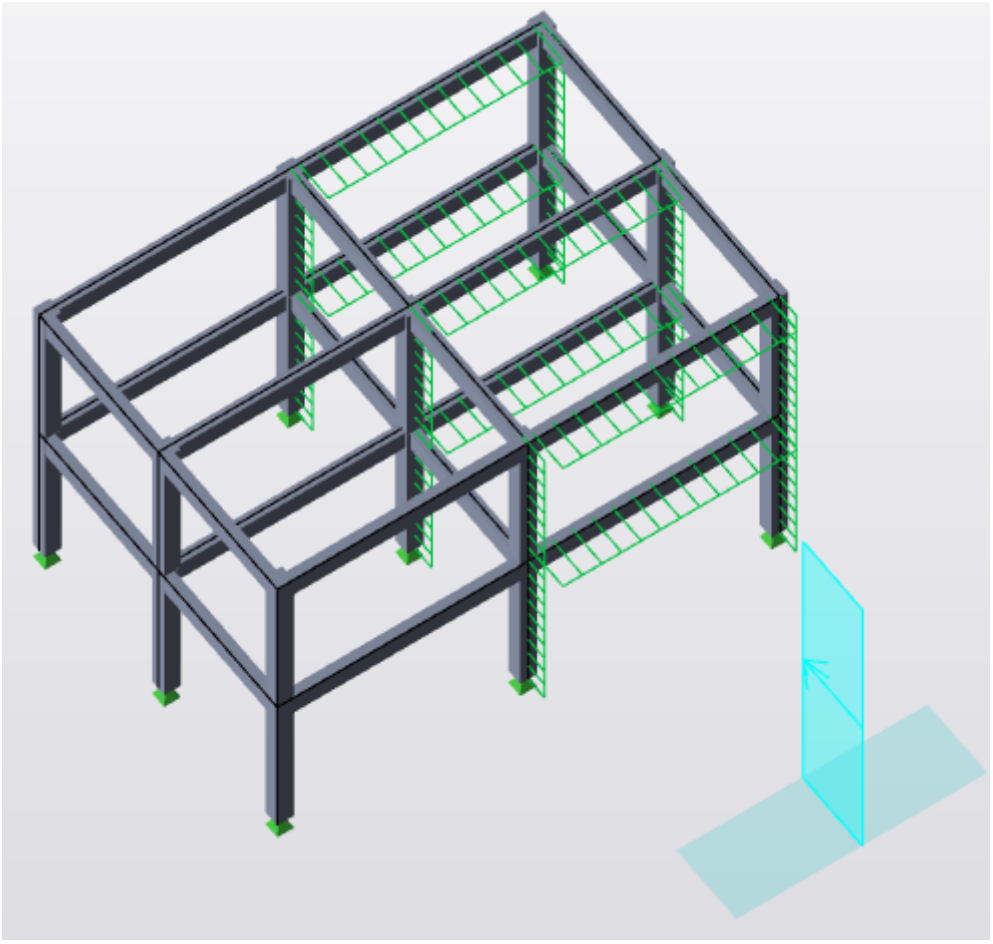
12. Extend wind over full building width option

This option is used to control the extent of wind horizontally.

- When selected - wall/roof panels & open structure members see the wind load, irrespective of whether they lie within, or outside the load width.



- When unselected - only those wall/roof panels & open structure members that lie within the load width see the wind load.



Slab Deflection Check Catalogue

The **Slab Deflection Check Catalogue** is used to define the deflection checks that are applied to check lines.

Location

On the **Slab Deflection** tab, click **Deflection Checks**.

Content

Each deflection check has a unique name and can either be defined as a total or instantaneous check for a specific event, or a differential check between load events. A deflection limit is set and you can specify if the check is to be applied to each new check line as it is defined.

Field or button	Description
Name	Allows you to modify the check name.

Field or button	Description
Type	Allows you to select the deflection check type, see .
Start Event	Allows you to select the start event for a differential check.
Event	Allows you to select the event to which the check applies.
Deflection Limit	Allows you to specify the deflection check limit.
Use in new Check Lines	Check the box only to be automatically apply the check to new check lines as they are created.
Add	Allows you to add a new row in the table for defining a new check.
Remove	Removes the selected check from the table.

Slab Deflection Settings dialog box

The **Slab Deflection Settings** dialog box and its sub pages allow you to adjust the settings applied to the different analyses.

Location

On the **Slab Deflection** ribbon tab, click **Settings**.

Content

See:

After adjusting the settings, do one of the following:

- To apply the changes to the current project: click **OK**
- To save the changes back to the active settings set for future use: click **Save...**
- To cancel the changes: click **Cancel**
- To revert to the slab deflection settings specified in the active settings set: click **Load...**

Sub Models dialog box

The **Sub Models** dialog box allows you to split the structure into a continuous series of sub models, working from the top of the building down to, and including the foundations.

Location

In the **Structure** tree, double-click **Sub Models**.

Content

Button, Command or Option	Description
Options	
Level	Before any analysis has been performed, only two levels are displayed, one at a distance (2m) above the highest construction level and a second at the same distance below the lowest level. These cannot be changed. Hence, at this point, there is a single sub-model comprising the whole structure.
Active	Only active levels act to divide the structure into sub-models. The top and bottom levels must always remain active, as there must always be at least one sub model. Once intermediate levels have been inserted, you can choose to inactivate them if required. When this occurs the sub-models immediately above and below the level in question are merged into a single sub-model.
Auto Generate	If any adjustments to the cutting planes or cutting plane levels, or any of them are set as inactive you must also clear Auto-Generate. Otherwise the changes you have made will be lost the next time the sub models are generated.
Buttons	
OK	Allows you to save any changes made.
Cancel	Allows you to discard any changes made.
Insert Above	Allows you to insert a new sub model between the selected level and the one above. It defaults to being located half way between the two, however this can be edited manually, provided it remains between the two reference levels.
Insert Below	Allows you to insert a new sub model between the selected level and the one below. It defaults to being located half way between the two, however this can be edited manually, provided it remains between the two reference levels.
Delete	
Generate	Allows you to auto generate default sub models or not • Checked Auto Generate creates default sub models for every level specified as a floor in the Construction Levels dialog. It is optional as default sub models are automatically generated for you when you run the analysis, provided the Auto Generate box on the

Button, Command or Option	Description
	dialog is checked. You may choose to do this if you want to review the sub models prior to the first run of the analysis. • Unchecked Does not Auto Generate sub models prior to the first run of the analysis. **NOTE** The Generate button can also be used to revert back to the default sub models at any time.

See also

Sub models (page 616)

Create sub models (page 617)

Index

.....842

.....436,1504

..... 1101
Activate reductions in live or imposed loadcases 325
Activate rigid diaphragm option within a slab 623
Activate semi-rigid diaphragm action within a slab 623
Add haunches to steel beams.....190
Add or remove elements in sub structures 1298
Adjust and apply drawing settings 1093
Adjust global slab mesh properties 586
Adjust wall mesh parameters in structure properties 639
Align a column to a specific angle or an angled grid line 156
Analysis Element properties.....1419
Analysis Model settings 1254
Analysis Settings dialog box..... 1530
Apply an architectural grid to a specific level 139
Apply and modify design options 777
Apply cantilever ends 978
Apply curved edges to existing slab items 227
Apply different mesh properties at different levels 586
Apply different mesh properties to an individual wall 640
Apply loads to loadcases on completion of the snow wizard..... 431
Apply member loads 357
Apply patterning to load combinations .. 351
Apply rotational stiffness 997
Apply structure loads 361
Automatically join all beams 279
Beam line settings 1253
BIM Integration 1156
Check selected members and walls..... 790
Check using Tekla Tedds..... 859
Connection Resistance dialog box.....1533
Construction levels dialog box..... 1536
Copy and rotate objects 271
Copy material grades 966
Copy or modify slab and foundation reinforcement 964
Copy or modify user-defined attributes ..970
Copy properties 967
Copy quick connector layout.....989
Copy section sizes 966
Copy shear connectors 999
Copy transverse reinforcement 1005
Copy web openings 1009
Create a new template 38
Create a pile type catalogue 262
Create a sub structure 1297
Create a sub structure group 1299
Create and manage loadcases 325
Create and manage loadcases, groups, combinations, envelopes and patterns....324
Create and manage user-defined attributes 1292
Create and modify patches 800
Create and modify reports 1062
Create column drops 227
Create concrete cores.....205
Create concrete member schedule drawings 1110
Create continuous beams 171
Create curved beams 172
Create DELTABEAMS 183,184,185
Create diaphragm loads 362
Create door or window openings 204
Create drawing scales 1094
Create drawings 1090
Create drawings in batches 1113
Create FABSEC beams 182

Create foundation drawings 1108
Create free form trusses 233
Create full UDLs 358
Create gable posts or parapet posts 155
Create general arrangement drawings . 1098
Create load combinations manually 341
Create loadcases 325
Create loading plan drawings 1102
Create member detail drawings 1103
Create modal mass combinations 343
Create models 97
Create nodal loads 369
Create partial-length torsional UDLs and VDLS 361
Create partial-length UDLs or VDLs 358
Create point loads and moment loads 360
Create punching checks 812
Create reports 1065
Create reports sand drawings 1062
Create slab and mat drawings 1107
Create slab or mat openings 223
Create strip bases 262
Create sub models 617
Create temperature loads 369
Create torsion full UDLs 360
Create Westok Cellular or Westok Plated beams 182
Create, modify, or delete layer configurations 1094
Create, modify, or delete layer styles 1096
Delete construction levels 123
Delete panel, member, and structure loads 370
Delete sub models 618
Delete sub structure 1299
Delete the snow model 434
Design review filters 951
Design selected members and walls.....792
Design settings.....877,910,926,940
Design settings 876
Design Settings - Minimum Design Forces Examples.....915
Design Settings dialog box..... 1538
Design using Tekla Tedds.....851
Dialog boxes 1529
Diaphragm loads and diaphragm load tables 362
Display notional forces, minimum lateral loads and seismic equivalent lateral forces 684
Drawing categories 1091
Drawing Settings dialog box.....1539
Edit Reinforcement dialog box.....1575
Embodied Carbon Factors dialog 1542
Embodied carbon settings 1262
Example reports 1076
Exclude individual nodes from a rigid diaphragm 625
Exclude slab items from a diaphragm 625
Export a model to Autodesk Robot Structural Analysis 1202
Export a model to STAAD 1202
Export a model to Tekla Structures 1176
Export a model to the cloud 1203
Export reports 1075
Export tabular review data to Excel.....1059
Export to and import from other applications 1186
Export to Trimble applications.....1176
Filter reports 1069
Fire proofing 981
Further orthotropic slab modeling considerations.....613
Grouping model settings 1252
How bearing walls are represented in solver models.....565
How mid-pier walls are represented in solver models.....559
Identify the nodes constrained by rigid diaphragms 625
Import loadcases and combinations from a spreadsheet.....343
Import model data 1156
Live/imposed load reduction settings ...1251
Load groups.....348
Load Event Sequences dialog box..... 1550
Loading settings 1247
Make a level an identical copy of another level 122
Make a level an independent copy of another level 122
Manage and view result strips 693
Manage display and design result lines... 698
Manage drawings in batches 1112
Manage FE meshed slabs..... 584

Manage material databases 1265
Manage properties and property sets .. 1284
Manage schedule drawings in batches .1117
Material list settings 1253
Material lists for cold formed.....1051
Material lists for concrete.....1039
Material lists for general materials.....1052
Material lists for steel.....1034
Material lists for timber..... 1050
Merge planes 300
Mesh type and size guidelines for FE walls....
635
Mirror objects to new locations 272
Model Settings dialog box..... 1570
Model steel beams and cold formed beams
.....176
Modify assumed cracked settings 972
Modify auto design settings 961,971
Modify BIM status 964
Modify concrete column alignment or
specify offsets 165
Modify end fixity 963
Modify gravity only settings 983
Modify panel, member, and structure loads
.....370
Modify punching shear check position ... 988
Modify SidePlates 999
Modify slenderness settings 975
Modify stud auto layout 1004
Modify the position of beams 173
Modify the properties of a construction level
.....122
Modify wind loading 1011
Move an rotate objects 271
Navigate reports 1073
Nominate a gravity combination to be used
for minimum lateral load calculations..... 342
Object reference settings 1245
Open a 3D view of a sub structure 1300
Open a 3D view of a sub model 618
Open a project.....36
Orthotropic slabs examples..... 607
Override effective width 988
Pile catalogue 1571
Print reports 1076
Reference 1306
Rename sub structure 1299
Renumber all load combinations 347
Renumber all loadcases 329
Report terminology 1063
Reset reinforcement marks in concrete
detail drawings 1115
Review and apply property sets 969
Review and copy size constraints 1000
Review and modify diaphragm settings ..962
Review and modify drift checks 981
Review and modify member filters 968
Review and modify restraints 990
Review and modify seismic drift checks ..998
Review and modify SFRS type and direction
settings 998
Review and modify sway checks 1005
Review and modify user defined utilization
ratios 1006
Review and modify wind drift checks 1010
Review and set imposed load reduction .986
Review and set camber 976
Review and set live load reduction 984
Review carbon factors 979
Review concrete beam flanges 969
Review drawings 1116
Review drift check tabular results 1019
Review embodied carbon detail 1054
Review embodied carbon overview 1055
Review floored area tabular results.....1057
Review foundation and pile design 945
Review kamp results 1017
Review material list tabular results 1029
Review member design 943
Review seismic drift check tabular results ....
1022
Review slab and mat design 946
Review sub structures 1299
Review sub structures 968
Review sway check tabular results 1013
Review the slab mesh before the analysis ....
587
Review the wall mesh before the analysis ....
640
Review utilization and embodied carbon ....
1008
Review where property sets have been
applied 1287
Roof Panel Properties 1505
RSA seismic results 701
Run a 1st order modal analysis 514

Run a 2nd order buckling analysis 496
Run a seismic analysis 515
Run combined analysis and member design 786
Run the snow wizard..... 429
Set up global imperfections 328
Set up minimum lateral loadcases 328
Setting out steel and cold formed columns 157
Shear only walls overview.....561
Sign conventions and coordinate systems 1306
Simple Wind Loading dialog box.....1583
Slab Deflection Settings dialog box..... 1593
Snow loading 426
Sort and filter tabular review data 1058
Specify a column splice 163
Specify concrete column alignment relative to the grid 164
Specify the drawing layout 1114
Specify the loading for load-dependent drawings 1114
Start a new project 36
Structural BIM settings 1259
Sub Model Properties 1337
Sub models 616
Sub Models 1593
Support properties 1514
The Materials dialog box 1564
The Sections dialog box 1581
The Slab Deflection Check Catalogue 1592
Unit settings 1245
Update load patterns 351
Update snow loads 433
Upgrade material databases 1282
Use templates in new projects 37
User-defined attribute settings 1257
Validation settings 1256
View and modify wind properties 58
View drawings 1115
View load status 57
View mode shapes 701
View status.....59
View the revision history of drawings ... 1116
What is a solver model?..... 537
Wind loading 372
Work with autosave and backups.....39
Working with member forces table options 723
Working with model filters.....725

1

1st order nonlinear - Nonlinear Supports.... 526
1st order linear - 3D truss.....524
1st order linear - Simple cantilever.....520
1st order linear - Simply supported orthotropic square slab.....522
1st order linear - Simply supported square slab..... 520
1st order linear - Thermal load on simply supported beam..... 525
1st order modal - Bathe and Wilson eignenvalue problem..... 533
1st order modal - Simply supported beam.... 533
1st order nonlinear - Displacement loading of a plane frame.....527
1st Order Nonlinear - Simple cantilever...525

2

2nd order buckling - Euler strut buckling 534
2nd order buckling - Plane frame..... 535
2nd order linear - simple cantilever..... 528
2nd order linear - Simply supported beam.... 529
2nd order nonlinear - Compression only element..... 532
2nd order nonlinear - Tension only cross brace.....531

A

Add a span into an existing beam..... 174
Add overhangs to existing slab or mat edges 225
Add user-defined materials.....1265
Alternative wind load decomposition method for complex models..... 384
Analysis limitations and assumptions.....516
Analysis model choices..... 568

Analysis results....662
Analysis settings....760
Analysis types....482
Analysis verification examples....519
Analyze All (Static)....497
Analyze models....482
Apply and manage model settings....1262
Apply attribute filters to material lists and reports....1295
Apply loading....324
Apply panel loads....352
Apply panel, member, and structure loads....351
Apply property sets to existing entities..1286
Apply simple wind loads to diaphragms..398
Apply simple wind loads to wall/roof panels and open structure members....411
Area ancillary properties....1510
ASCE 7 Horizontal Design Spectrum....468
ASCE 7/UBC Horizontal Design Spectrum Taiwan....476
ASCE 7/UBC Horizontal Design Spectrum Thailand....477
Attach UDA values to members and panels....1294
Autodesign versus check design....778

B

Base plate properties....1422
Beam end releases and partial fixity....568
Beam properties....1338
Bearing wall properties....1432
Brace properties....1402

C

Change result diagram scale settings....706
Charts....1059
Check drift, sway, seismic drift, wind drift, and overall displacements....822
Check floor vibration....819
Check the model....323
Checkbot Control Panel....1531
Clear Tekla Tedds Data....865
Code spectra and site specific spectra....465
Column properties....1374
Column releases....571
Concrete core properties....1457
Concrete member cracked or uncracked status....650
Concrete wall properties....1434
Construction levels....122
construction lines....139
Copy loads....274
Copy Westok openings....1009
Create supports....242
Create a new project based on a template....38
Create a truss....232
Create a wind model and wind loads....387
Create analysis elements....253
Create and manage construction levels...121
Create and manage construction lines....139
Create and manage free points....301
Create and manage load combinations..338
Create and manage wind loadcases....391
Create and modify scene view tab groups.73
Create attribute definitions....1293
Create beam end force drawings....1099
Create beams....167
Create beams, columns and braces....148
Create bearing walls....208
Create braces....190
Create column splice load drawings....1100
Create columns....149,152
Create cutting plane....281
Create frames and slopes....145
Create general walls....213
Create grid lines....123
Create inclined columns and cranked columns....154
Create infill members....299
Create isolated foundations....260
Create mat foundations....265
Create mats....266
Create members with cold rolled sections....239
Create meshed or midpier concrete walls....200
Create pad bases....261
Create pile caps....263
Create planar drawings....1097
Create plated and compound section steel beams....177

Create plated or compound section steel columns....158
Create portal frames.... 236
Create settlement loads 369
Create shear only walls.... 211
Create single-span beams.... 170
Create slab items.... 221
Create slabs.... 216
Create space trusses 233
Create the model.... 120
Create trapezoidal loads 359
Create trusses.... 231
Create trusses and joists.... 231
Create wall and roof panels.... 240
Create walls.... 198
Create walls, cores, and bearing walls.... 197
Create web openings.... 186
Customize the display of 2D contours.... 705

D

Decompose panel loads.... 370
Default spectra.... 466
Define whether two-way slabs are meshed for 3D building analysis and grillage chasedown analysis.... 585
Delete entities.... 277
Delete property sets.... 1288
Design settings.... 866
Design and check isolated foundations .. 818
Design and check mat foundations 809
Design and check patches.... 810
Design and check slabs.... 806
Design code settings 1243
Design models.... 776
Design settings.... 874,875,927,938,939
Design Settings 866,878,907,932,936
Design slabs and mats, run punching shear checks.... 800
Design steel members and cast-in-place concrete beams, columns and walls.... 776
Design timber members, precast members, and masonry walls using Tekla Tedds.... 849
Dimensions and measurements.... 256
Display 1D deflections.... 667
Display 1D results.... 667
Display 2D deflections.... 691
Display 2D view in isometric.... 707
Display analysis results in a Results View 662
Display AsReq contours.... 692
Display core lines.... 692
Display deflections.... 668
Display forces in 2D elements 685
Display loading summaries in the Project Workspace.... 758
Display member analysis results in a Load Analysis View.... 748
Display reactions 665
Display stresses in 2D elements 687
Display sway, drift, and story shear 668
Display tabular analysis results in a Solver Model Data view.... 707
Display wall lines.... 692
Drawing settings 1119
Drift check.... 823

E

Edit stage data dialog box.... 1540
Edit the model.... 270
Element types.... 254
EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA.... 470
EN 1998-1 Horizontal Design Spectrum (Malaysia NA).... 472
Equipment properties 1512
Export a model to ADAPT.... 1198
Export a model to IFC 1190
Export to and import from FBEAM.... 1192
Export to and import from Autodesk Revit.... 1187
Export to and import from Westok Cellbeam 1191
Export to IDEA StatiCa Connection Design.... 1207
Export to IDEA StatiCa Connection Design.... 1207,1216
Export to One Click LCA.... 1204
Export to Tekla Connection Designer.... 1177
Export to Tekla Portal Frame Designer.. 1178
Export to Tekla Tedds.... 1183
Exporting wind tunnel data workflow.... 424
Extend, move, or rotate construction lines and arcs.... 144
Extend, move, or rotate grid lines and arcs 136

F

FE meshed walls.....635
Features of 3D analysis and chasedown analysis.....506
Floor Analysis Model(FAM) properties .. 1519
Format reports.....1070
Foundation mat properties 1482
Frame Properties 1335,1336

G

General settings 1234
General wall properties 1452
Generate load combinations automatically 340
Get to know Tekla Structural Designer basic working methods..... 97
Global Imperfections settings..... 1250
Global Settings..... 1232
Graphics view settings 1259

H

Hide, re-display and move windows.....87
How meshed walls are represented in solver models 554
How slab properties and features impact on meshing.....587
How to save view configurations..... 67
How to use the Quick Access window..... 84

I

Import 3D reference models.....1165
Import a project from a Structural BIM Import file 1156
Import a project from a TEL file..... 1158
Import data from a 3D DXF file..... 1164
Import grids or a shadow from a DXF file 134
Inclined and rotated supports.....245
interface components..... 42
interface components..... 592
Introducing Tekla Structural Designer..... 29
Introducing Tekla Structural Designer..... 29
IS893 (Part 1) Horizontal Design Spectrum.... 474

J

Join and split objects.....278

K

Keyboard functions and shortcuts.....88

L

Level Properties 1333
Limitations of wind decomposition to diaphragms.....402
Line ancillary properties 1507
Linear, non-linear and buckling analyses 494
Load combination classes 339
Load groups.....329
Loading dialog box..... 1557

M

Manage envelopes.....348
Manage groups..... 54
Manage models 1219
Manage object references..... 1227
Manage project revisions and view metrics 1289
Manage sub structures..... 1296
Managing diaphragm action in roof panels and slabs..... 622
Manual wind loads 423
Masonry wall properties 1443
Member characteristic, construction and fabrication properties..... 1427
Member design stage 785
Member global offsets..... 194
Meshed wall openings analysis model.....641
Minimum Lateral Loads settings 1249
Modal analysis.....508
Model Settings..... 1243
Model steel joists..... 234
Modeling.....173,174

Modeling diaphragm action in roof panels and slabs.....619
Modify column splice positions 969
Modify model properties..... 48
Modify slab/panel span direction..... 231
Modify the geometry of steel trusses 234
Modify the properties of existing trusses 234
Modify wall supports.....203
Modify wind zones of multibay structures.... 389
Move the model or the DXF shadow 297

N

Number and renumber grids.....137

O

One-way and two-way load decomposition 626
Open and close views.....64
Open structure wind loads..... 417
Orthotropic slab implementation.....605
Orthotropic slabs..... 601
Orthotropic wall implementation..... 648
Orthotropic walls..... 644
Overall displacement 842
Overall wind drift check 840
Overview of combined analysis and member design.....779
Overview of Planarize.....302
Overview of the concrete wall model 198
Overview of the Project Workspace 46
Overview of the slab model 217

P

Pad strip base and pile cap properties . 1496
Parapet wall panel load decomposition 1505
Partial fixity of column bases..... 252
Patch properties 1479
Performance settings 1242
Place piles and pile arrays in mats.....267
Planarize Example..... 312
Portal frame haunch geometry.....238
Portal frame properties..... 1492
Properties reference.....1329
Property editing methods.....111
Punching check properties 1486

R

Rationalize the model.....298
Re-position entities by moving nodes or edges..... 112
Report settings 1086
Result strip properties 1491
Results Viewer settings 1236
Reverse member axes and panel faces... 280
Review and copy deflection limits 980
Review and modify Staged Construction stages..... 1001
Review design summary tabular results1012
Review designs..... 943
Review models 942
Review portal frame deflection..... 1058
Review wind drift check tabular results 1025
Rigid offsets and rigid zones in concrete members.....572
Rigid offsets examples..... 574
Rigid zones examples.....578
Roof..... 382
RoofType 427
Run 3D only (Static).....498
Run a 1st order linear or non-linear analysis 494
Run a 2nd order linear or non-linear analysis.....495
Run FE chasedown or grillage chasedown analysis 499

S

Save properties to property sets..... 1285
Scene settings 1239
Section default settings 1238
Section order default settings 1238
Seismic loadcases..... 478
Seismic drift check.....832
Seismic loading..... 434
Select entities..... 100

Select a section in the Sections dialog box 109
Set the analysis type and loading for viewing analysis results....664
Settings and options....1219
Settings dialog box.... 1583
Settings set settings 1234
Shear only wall properties 1450
Show and alter state 958,970
Simple wind load example (diaphragms..400
Simple wind load example (panels....413
Simple wind load workflow for open structures 416
Simple wind pressure loading....396
Slab item properties 1458
Slab/Mat overhang properties 1479
Solver Element (1D) Types.... 551
Solver Element (2D) Types.... 553
Solver element 2D properties.... 1519
Solver element properties....1518
Solver model types.... 540
Solver models....537
Solver node properties.... 1517
Solver settings 1238
Solver View: Open....549
Specify extensions and releases for concrete walls....202
Specify the beam type and section size... 168
Specify the column type and section size 149
Specify the material for general slab types.... 228
Split and join slabs and mats....229
Start Tekla Structural Designer....34
Structure default settings 1237
Structure Properties 1331
Sway/drift settings 1252
Synchronize user-defined materials in models and databases....1282

T

Tabular review data 1011
Tekla Structural Designer configurations...31
The Direct Analysis Method....655
The sway check 828
Tips for basic tasks.... 118
Transfer property sets between models1288

U

Understanding story shears and story forces....671
Use regional codes and design codes....1224
Use settings sets.... 1220
Use units.... 1225
Using imported wind tunnel information 426
Using the ASCE 7 seismic wizard....436
Using the Eurocode EN1998-1:2004 seismic wizard....452
Using the IS1893 seismic wizard....460
Using the UBC 1997 seismic wizard....445
Utilization ratio....797

V

Validate the model for design issues.... 799

W

Why chasedown analyses are included in Analyze All (Static)....499
Wind drift check 836
Wind model load decomposition....378
Wind model loadcases.... 377
Wind panel decomposition....378
Wind tunnel testing overview....423
Wind wizard pressure loading....373
Wind wizard workflow....374
Wind wizard workflow for open structures 392
Working collaboratively with Trimble Connect....1167
Working with foundation reactions axis systems.... 726
Working with large models 1303

Z

Zoom, pan, rotate and walk through scene views.... 98