

Tekla Structural Designer 2024

Engineers Handbooks

March 2024

©2024 Trimble Inc. and affiliates

Contents

1 Steel design handbook 19

1.1 Combined analysis and design choices for steel structures 19

Gravity design 19
Static design 20
Designing individual members for gravity only 21

1.2 Steel member autodesign 21

Size constraints 22
Applying a size constraint to an element 22
Steel and composite beam - autodesign optimization 23
Beam design section step back 23
Prefer fewer connectors over section weight 24

1.3 Steel member design groups 24

Why use steel design groups? 24
What happens in the group design process? 25
Steel design group requirements 25
Group management 26

1.4 Steel member orientation 27

1.5 Steel beam design 28

Overview 29
Fabrication 29
Restraints 30
Deflection limits 31
Torsion 31
Camber 32
Floor vibration 33
Instability factor 33
Web openings 34
Fire check (Eurocode only) 35
Plated steel beams 35
Plated beams - AISC regional code 35
Plated beams - Eurocode regional code 37
Plated beams - BS regional code 38
Plated beams - Indian regional code 39
Plated beams - Australian regional code 40

1.6 Composite beam design 41

Overview 41
Construction Stage loading 42
Composite stage loading 43
Allow non-composite design 43
Fabrication 45
Restraints 45
Deflection limits 46
Camber 46
Floor vibration 47

Floor construction..... 47
Metal deck..... 52
Stud strength..... 52
Connector layout..... 53
Transverse reinforcement..... 61
Instability factor..... 62
Web openings..... 63
Precast concrete planks (Eurocode only)..... 63
General limitations and assumptions..... 63
Concrete properties..... 65
Loading..... 65
Shear Connectors..... 65
Longitudinal Shear..... 66
Composite Moment of Inertia..... 66
1.7 Steel column design..... 66
Overview..... 67
Simple columns..... 67
Fabrication..... 68
Restraints..... 71
Splice and splice offset..... 72
Web openings..... 73
Instability factor..... 73
Steel column connection eccentricity moments..... 74
Overview..... 74
Define connection eccentricity values..... 77
Pattern eccentricity moments for live loadcases..... 78
Review connection eccentricity moments..... 78
1.8 Column base plate design..... 80
Column base plate design workflow..... 80
Data sources..... 80
Generate base plates..... 81
Design columns..... 81
Edit base plate properties..... 81
To copy, move or delete base plates..... 82
Check base plates..... 82
Review check status and utilization..... 83
View results and print calculations..... 83
Create drawings..... 84
1.9 Steel brace design..... 84
Steel brace overview..... 84
Input method for A and V Braces..... 85
Steel brace in compression..... 85
Steel brace in tension..... 86
1.10 Steel joist design..... 86
Steel joist design overview..... 86
Assumptions and limitations..... 87
Loading..... 89
Joist member reports..... 91
1.11 Steel truss design..... 91
Steel truss design overview..... 92
Assumptions and Limitations..... 94
1.12 Portal frame design..... 95

2 Steel connections handbook....96
2.1 Check simple connection resistance.... 96
Overview.... 96
Specify 'active' connection resistances (Eurocodes)....97
Specify 'active' connection resistances (US)....99
Run resistance checks....101
The connection optimization process....102
Display connection resistance checks in a review data table....103
Create and display a connection resistance report....104
Related video....105
2.2 Create and size SidePlate connections....105
SidePlate connections theory.... 105
Overview of SidePlate connections....105
Permitted SidePlate connections in Tekla Structural Designer.... 106
SidePlate workflow in Tekla Structural Designer....107
SidePlate 'joint' and 'connection' terminology....108
SidePlate validation....109
Initial sizing of the SidePlates.... 109
Analysis model adjustment....109
Design/Analysis processes and recommended workflows....110
Effects of SidePlate on steel member design.... 112
Create SidePlate connections.... 112
Beam properties - SidePlate....113
2.3 Create and check column base plates....114
Create column base plates....114
Check column base plates....115
2.4 Create and design other connections....115
Overview....115
Update connections.... 117
Grouping of connections.... 118
Design connections.... 118
Steel connection formation rules 119
Recommended workflows for specific connection types....119
Portal frame connections....119
Base plate connections (not in portal frames)....120
Other connections (not in portal frames)....120
Limitations when using Tekla Connection Designer with Tekla Structural Designer . 121
General.... 121
Base plate connections 121
Moment connections....122
Simple connections....122
Hollow section connections.... 122
Column splice connections.... 122
2.5 Export connections to another application for design 123
2.6 Manage connections in the Project Workspace.... 123
Update connections.... 124
Edit connection.... 124
Select connection in visible views....124
Design connection....124
Export connection to Tekla Connection Designer.... 125
3 Concrete design handbook 126

3.1 Concrete member design workflow....127
Set up pattern loading....127
Set all beams columns and walls into autodesign mode....128
Review beam and column design groups....128
Review beam, column and wall design parameters and reinforcement settings....129
Perform the concrete design....129
Review stability issues....131
Review the design status and ratios....132
Create reports....132
Create drawings and schedules....137
3.2 Concrete member autodesign....143
Autodesign (concrete beam)....144
Autodesign (concrete column)....144
Autodesign (concrete wall)....145
Select bars starting from....145
Specify the allowed bar size range....145
3.3 Concrete beam and column groups....146
Why use design and detailing groups?....146
What happens in the group design process?....147
Design group requirements....147
Detailing group requirements....148
Group management....150
3.4 Concrete beam design fundamentals....151
Beam design settings....151
Concrete type....151
Deflection control (ACI/AISC)....152
Deflection control (AS 3600)....153
Deflection control (Eurocode BS and IS)....154
Ignore lateral instability (Eurocode)....155
Consider flanges....155
Design parameters (Eurocode only)....155
Nominal cover....156
Reinforcement - longitudinal bar patterns....156
Flanged concrete beams....159
Modeling requirements....159
Include flanges in analysis....160
Isolated beam (ACI/AISC)....160
Effective Width of flanges (ACI/AISC)....160
Effective Width of flanges (Eurocode)....161
3.5 Concrete column design fundamentals....162
Column design settings....162
Concrete type....162
Apply rigid zones....163
Design parameters (Eurocode only)....163
Confinement reinforcement....164
Slenderness....164
Stiffness....165
Sway/Drift Checks....166
Nominal cover....166
Reinforcement....166
3.6 Concrete wall design fundamentals....169
Wall design settings....169
Concrete type....169

End 1 and End 2 extensions....169
Reinforcement layers....170
Design parameters (Eurocode only)....170
Sway/Drift Checks....171
Confinement reinforcement....171
Slenderness....172
Stiffness....172
Nominal cover....173
Reinforcement....173
3.7 Interactive concrete member design....174
Interactive concrete beam design....174
Opening the Interactive Beam Design Dialog....174
Changing the bar pattern used in the Interactive Beam Design Dialog....174
Interactive Beam Design dialog box....175
Interactive concrete column design....178
Opening the Interactive Column Design Dialog....178
Arranging bars in the Interactive Column Design Dialog....178
Using a 2nd layer of bars....181
Column interaction diagrams (US customary units)....182
Column interaction diagrams (metric units)....186
Defining additional column design cases for user defined forces....190
Interactive Column Design dialog box....190
Interactive concrete wall design....198
Opening the Interactive Wall Design Dialog....198
Wall interaction diagrams (US customary units)....198
Wall interaction diagrams (metric units)....202
Defining additional wall design cases for user defined forces....206
Using result lines for local section design around openings....206
....206
Interactive Wall Design dialog box....207
3.8 Flat slab design workflow....215
Set up pattern loading....217
Establish slab design moments....218
Consider simple (linear) deflection (US customary units)....218
Consider simple (linear) deflection (metric units)....221
Select a level, or sub-model....224
Add patches....224
Design panels....225
Review/optimize panel design....226
Design patches....227
Review/optimize patch design....228
Add and run punching checks....228
Rigorous deflection check....230
Create reports....230
Create drawings and quantity estimations....235
3.9 Slab on beams design workflow....239
Set up pattern loading....240
Establish slab design moments....241
Select a level, or sub-model....241
Add beam and wall top patches....242
Design panels....243
Review/optimize panel design....244
Design beam and wall patches....245
Review/optimize patch design....245

Create reports.... 246
Create drawings and quantity estimations.... 250
3.10 Concrete slab design fundamentals.... 255
Slab panels.... 255
Slab patches.... 255
Punching.... 256
Concrete type.... 256
How the load decomposition method affects slab design.... 256
Two-way spanning slab panel design moments.... 257
Reinforcement design settings.... 257
Slab panel reinforcement.... 258
Alignment of slab panel reinforcement.... 258
Slab patch reinforcement.... 258
Span effective depth checks for irregular shaped panels.... 261
Slab on beam idealized panels.... 261
Rotation angle for slab items and panels.... 263
3.11 Punching shear check fundamentals.... 264
Overview.... 265
Applicability.... 265
Punching shear check axis system.... 266
Punching shear perimeters.... 269
Reinforcement type and properties.... 271
Reinforcement design settings.... 272
Reinforcement auto-design.... 272
Reinforcement layout.... 273
Use of property sets to standardize checks and reinforcement 279
3.12 Pad base design workflow.... 281
Apply pad bases under supported columns.... 281
Auto-size pad bases individually for loads carried.... 282
Apply grouping to rationalize pad base sizes 284
Review/optimize base design 286
Create drawings and quantity estimations.... 286
Print calculations.... 287
Ignoring small shear forces in pad base design checks.... 287
Activating the option to ignore small shear forces.... 288
Impact on the bearing capacity checks.... 288
Impact on the bending capacity, shear capacity, punching capacity, and sliding checks.... 290
3.13 Pile cap design workflow.... 291
Apply pile caps under supported columns.... 292
Auto-size pile caps individually for loads carried.... 293
Apply grouping to rationalize pile cap sizes 294
Review/optimize pile cap and pile design 295
Create drawings and quantity estimations.... 296
Print calculations.... 296
3.14 Pad base, strip base and pile cap design forces.... 296
3.15 Mat foundation design workflow (metric units).... 297
Design the structure before supporting it on the mat.... 299
Determine the soil parameters.... 299
Determine the remaining mat properties.... 301
Create the mat.... 301
Enable soil structure interaction.... 302
Model validation.... 303

Perform the model analysis 303
Check foundation bearing pressure and deformations 304
Re-perform member design..... 305
Open an appropriate view in which to design the mat..... 305
Add patches..... 306
Design mats..... 306
Review/optimize mat design..... 307
Design patches..... 308
Review/optimize patch design..... 308
Add and run punching checks..... 309
Create drawings and quantity estimations..... 311
Print calculations..... 311
3.16 Mat foundation design workflow (US customary units)..... 312
Design the structure before supporting it on the mat..... 313
Determine the soil parameters..... 313
Determine the remaining mat properties..... 315
Create the mat..... 316
Enable soil structure interaction..... 316
Model validation..... 317
Perform the model analysis 317
Check foundation bearing pressure and deformations 318
Re-perform member design..... 319
Open an appropriate view in which to design the mat..... 320
Add patches..... 320
Design mats 321
Review/optimize mat design..... 321
Design patches..... 322
Review/optimize patch design..... 323
Add and run punching checks..... 323
Create drawings and quantity estimations..... 325
Print calculations..... 325
3.17 Piled mat foundation design workflow (US customary units)..... 326
Design the structure before supporting it on the mat..... 327
Create the mat..... 327
Define the pile catalogue..... 328
Add piles to the mat..... 328
Remove existing column and wall supports..... 330
Model validation..... 331
Perform the model analysis 331
Perform the pile design..... 332
Review the pile design status and ratios..... 332
Add and run pile punching checks..... 333
Perform the mat design..... 336
3.18 Piled mat foundation design workflow (metric units)..... 336
Design the structure before supporting it on the mat..... 337
Create the mat..... 337
Define the pile catalogue..... 338
Add piles to the mat..... 338
Remove existing column and wall supports..... 340
Model validation..... 341
Perform the model analysis 341
Perform the pile design..... 342
Review the pile design status and ratios..... 342
Add and run pile punching checks..... 343

Perform the mat design....345

4 Precast member design handbook 346

4.1 Precast member design limitations and assumptions....346

4.2 Precast member design workflow....347

Configure precast beam and column design settings....347
Define and place precast members....348
Configure precast groups....349
Set the Tedds results output level....349
Establish design forces by running the analysis....350
Design using Tekla Tedds....350
Check the design after changes....352
Output the calculations....353

4.3 Precast member design groups....355

Why use precast design groups?....355
Activating precast member design groups....356
Group management....356
Precast design group requirements....357

4.4 Precast beam design....357

Section shapes....357
Beam arrangement....358
Concrete type....358
Nominal cover....358
Reinforcement - longitudinal bar patterns....358
Design sections....366
Default reinforcement in the Tekla Tedds calculation....369
Lifting checks....370
Analysis forces transferred from Tekla Structural Designer....370
Other precast beam properties....371

4.5 Precast column design....371

Section shapes....371
Concrete type....371
Nominal cover....371
Reinforcement....372
Lifting Checks and Splice Design....374
Analysis forces transferred from Tekla Structural Designer....374
Other precast column properties....375

4.6 Precast column connection eccentricity moments....375

Overview....375
Define connection eccentricity values....377
Pattern eccentricity moments for live loadcases....378
Review connection eccentricity moments....378

5 Timber member design handbook 381

5.1 Timber member design limitations and assumptions....381

5.2 Timber member design workflow....383

Set the timber design code....383
Configure timber design settings....384
Define and place timber members....385
Create load combinations and review load duration/time effect factors....386
Generate timber design envelopes 388
Configure timber groups....388

Set the Tedds results output level..... 389
Run analysis..... 389
Design using Tekla Tedds..... 389
Check the design after changes..... 391
Output the calculations..... 391
Design timber members using Tekla Tedds..... 393
Design a selection..... 393
Design a group..... 393
Design a grouped member..... 395
Design an ungrouped member..... 396
Design model..... 396
Check timber members using Tekla Tedds..... 396
Check model..... 396
Check a selection..... 396
Check a member..... 397
Check a group..... 397
5.3 Timber member design groups..... 397
Why use timber design groups?..... 397
Activating timber member design groups..... 398
Group management..... 399
Timber design group requirements..... 400
5.4 Timber combination sets and strength design cases..... 401
Timber combination sets..... 401
Strength design cases..... 402
5.5 Timber beam deflection checks - US..... 403
Specify deflection checks..... 403
Deflection design cases..... 404
Long term deflections due to creep effects..... 406
Automatic adjustment of design deflection values..... 408
5.6 Timber beam deflection checks - EC..... 408
Specify deflection checks..... 408
Deflection design cases..... 409
Long term deflections due to creep effects..... 411
Automatic adjustment of design deflection values..... 412
5.7 Timber column stacks and lifts..... 413
Design of a single stack lift..... 413
Design of a multi-stack lift..... 414
6 Masonry wall design handbook 418
6.1 Masonry wall design - introduction..... 418
6.2 Masonry wall limitations and assumptions..... 419
6.3 Masonry wall design workflow..... 421
Download and open the tutorial model..... 421
Choose the masonry design code..... 422
View the default masonry wall material grades..... 423
Define masonry wall properties..... 424
Place a wall..... 427
Review the masonry modification factors..... 429
Create a wall opening..... 429
Create result lines around an opening..... 433
Create load combinations..... 436
Establish design forces by running the analysis..... 437

Check all the walls in the model..... 437
Design failing wall W2..... 439
Review the design of W3 and associated result lines..... 443
Design failing result line RL 1..... 445
Output the calculations..... 450
6.4 Masonry wall design cases..... 451
7 Staged construction handbook 454
7.1 Introduction to staged construction analysis..... 454
What is staged construction analysis?..... 454
Benefits of staged construction analysis..... 455
Scope in Tekla Structural Designer..... 455
Implementation in Tekla Structural Designer..... 456
7.2 Overview of staged construction analysis..... 457
Create and load the model as normal..... 457
Review construction stages and stage durations..... 458
Reassign members between construction stages..... 461
Create a staged construction load combination..... 462
Review concrete modification factors..... 466
Activate staged construction analysis and select other results sets..... 467
Run the design..... 468
Review the results..... 468
Create output..... 469
Construction stages and design stages..... 469
Staged construction modification factors 471
Concrete creep allowance during construction stages 476
7.3 Staged construction limitations and assumptions..... 480
Limitations..... 481
7.4 Design comparison - with and without staged construction..... 481
Design force envelope without staged construction..... 482
Design force envelope with staged construction..... 485
Conclusions 489
7.5 Staged construction workflow example - Eurocodes..... 490
Download and open the tutorial model..... 490
Define construction stages and their stage durations..... 491
Assign members to construction stages..... 496
Create a Staged Construction load combination..... 499
Review the modification factors..... 501
Configure the analyses..... 502
Run the design and review the design results..... 503
Output the calculations..... 507
Display staged construction analysis results..... 507
Display staged construction solver models 509
7.6 Staged construction workflow example - United States (ACI/AISC)..... 512
Download and open the tutorial model..... 512
Define construction stages and their stage durations..... 513
Assign members to construction stages..... 519
Create a Staged Construction load combination..... 522
Review the modification factors..... 524
Configure the analyses..... 525
Run the design and review the design results..... 526
Output the calculations..... 530

Display staged construction analysis results..... 530
Display staged construction solver models 532

8 Seismic analysis and design handbook 535

8.1 Introduction to seismic analysis and design..... 535
Definitions..... 535
Overview..... 538
Seismic Wizard..... 539
Vertical and Horizontal Irregularities..... 540
Vertical Loads..... 540
Torsion..... 544
Modal Analysis..... 546
% of Gravity Load Method 546
Equivalent Lateral Force Method..... 546
Response Spectrum Analysis Method..... 546
Response Spectrum Analysis Processes..... 547
Seismic Drift..... 548
Design Coefficients and Factors (ASCE 7/UBC)..... 548
Workflow to check for torsional irregularity and amplification..... 549

8.2 Limitations of Seismic Design..... 550

8.3 Seismic force resisting systems..... 551
Available SFRS types..... 551
Members allowed in the SFRS..... 552
Assigning members to the SFRS..... 553
Special Moment Frames - assigning connection types at steel beam ends..... 553
Validation of the SFRS..... 553
Auto design of SFRS members..... 554

8.4 Seismic design methods..... 554
Seismic analysis and conventional design..... 555
ELF seismic analysis and conventional design..... 555
RSA seismic analysis and conventional design..... 555
Seismic analysis and seismic design..... 556
ELF seismic analysis and seismic design..... 556
RSA seismic analysis and seismic design..... 557

9 Footfall assessment handbook 559

9.1 Footfall assessment - introduction..... 559

9.2 Footfall analysis general procedure..... 560
Design guide recommendations..... 561
Floor analysis model preparation..... 562
Load to mass conversion..... 564
Modal analysis..... 565
Excitation and response locations..... 565
Analysis post-processing - SCI P354 and CCIP-016..... 566
Analysis post-processing - AISC DG11..... 567

9.3 Footfall analysis limitations and assumptions..... 570

9.4 Footfall analysis basic workflow - SCI P354..... 572
Download and open the tutorial model..... 573
Open an appropriate view..... 573
Confirm the floor analysis model settings..... 574
Create a floor analysis model..... 575
Create a Footfall mass combination..... 579

Analyse the model and view the mode shapes.... 580
Configure the footfall checks.... 582
View the footfall check results.... 583
Perform a VDV assessment.... 588
Output the calculations.... 589
Additional display and review options.... 590
Create additional floor analysis models, edit or delete models.... 595
9.5 Footfall analysis basic workflow - CCIP 016.... 597
Download and open the tutorial model.... 597
Open a footfall analysis view.... 597
Confirm the floor analysis model settings.... 598
Create a floor analysis model.... 601
Create a footfall mass combination.... 606
Analyse the model and view the results.... 607
Configure the footfall checks.... 610
View the footfall check results.... 611
Perform a VDV assessment.... 617
Output the calculations.... 620
Additional display and review options.... 621
Create additional floor analysis models, edit or delete models.... 630
9.6 Footfall analysis basic workflow - AISC DG11.... 632
Download and open the tutorial model.... 632
Open a footfall analysis view.... 633
Confirm the floor analysis model settings.... 634
Create a floor analysis model.... 637
Create a footfall mass combination.... 649
Analyse the model and view the results.... 651
Configure the footfall checks.... 654
View the footfall check overall results.... 655
View resonant response results.... 655
View transient response results.... 662
View enveloped response results.... 670
Edit the FAM and add a facade restraint.... 672
Reanalyse and view the updated results.... 675
Output the calculations.... 676
Additional display and review options.... 678
Manage FAMs in a Structure 3D view.... 680
9.7 Footfall assessment settings.... 683
Settings Set Management.... 684
Analysis Settings.... 684
Analysis Model Options.... 687
Concrete Modification Factors.... 688
Footfall Checks (SCI P354).... 690
Footfall Checks (CCIP-016).... 693
Footfall Checks (AISC DG11).... 697
9.8 Footfall assessment references.... 698
10 Slab deflections handbook.... 700
10.1 Get started with slab deflection analysis.... 700
10.2 Open a Slab Deflections View.... 701
10.3 Work with event sequences.... 702
Add an event to the end of the event sequence.... 702

Insert an event within the event sequence..... 702
Re-order events in the event sequence..... 702
Remove an event from the event sequence.....703
Edit event parameters.....703
Edit event loadcases.....703
Create a custom event sequence..... 704
Apply a custom event sequence to a submodel.....704
10.4 Work with check lines.....704
Create the deflection checks to be applied to check lines..... 705
Create a check line.....705
Delete a check line.....705
10.5 Run a slab deflection analysis..... 706
Run a slab deflection analysis for the current sub model.....706
Run a slab deflection analysis for all sub models.....706
Run a slab deflection analysis for selected sub models..... 707
10.6 Slab deflection results and reports.....707
Display slab deflection analysis results.....707
Display deflection contours..... 707
Display extent of cracking..... 708
Display relative stiffness.....708
Display effective reinforcement..... 709
Display check line results.....709
Display deflections along all check lines..... 709
Display detailed deflections and average slopes along an individual check Line. 709
Display check line status and utilization.....710
Display slab deflection status and utilization.....710
Display slab deflection status..... 710
Display slab deflection utilization.....711
Slab deflection optimization 712
View slab deflection reports.....712
View an individual check line report..... 713
View all/multiple check line reports..... 713
View an effective modulus report..... 713
10.7 Slab deflection concepts, workflows and examples 713
Slab deflection methods.....714
Deemed-to-Satisfy Checks..... 714
Rigorous theoretical deflection estimation.....715
Rigorous slab deflection workflow 716
Factors that affect rigorous slab deflection estimates.....716
Quasi-permanent load factors (EC2).....717
Beta coefficient (EC2).....718
Restraint type (EC2).....719
Restraint constant (ACI)..... 721
Concrete Properties (Eurocode)..... 722
Concrete Properties (ACI)..... 723
Stiffness Adjustments.....724
Shrinkage.....724
Event sequences 724
Construction stage events 725
A typical model event sequence 726
Custom event sequences 733
Understanding event sequence deflections 736
Slab deflection analysis sequence 737
Total, differential, and instantaneous deflection types 738

Slab deflection calculations in depth..... 739
Interrogating slab deflection calculations..... 739
Composite creep..... 740
Extent of cracking..... 742
Relative stiffness..... 744
Effective reinforcement..... 745
Shrinkage allowance..... 747
Check lines..... 751
Setting up the checks in advance (via the slab deflection check catalogue)..... 751
Application of check lines..... 751
Displaying check line results..... 752
Check line reports..... 753
Slab deflection status and utilization..... 753
Slab deflection example (Eurocode)..... 755
Deemed to satisfy slab deflection checks example (Eurocode)..... 756
Rigorous slab deflection analysis example (Eurocode)..... 759
Slab deflection example (ACI)..... 789
Deemed to satisfy slab deflection checks example (ACI)..... 789
Rigorous slab deflection analysis examples (ACI)..... 791
10.8 Slab deflection settings..... 848
11 Stability and imperfections handbook..... 853
11.1 Overview of stability requirements..... 853
Global second-order (P-Δ) effects..... 854
Member second-order (P-δ) effects..... 855
When must global and member second order effects be considered?..... 856
Global imperfections..... 856
Member imperfections..... 857
11.2 Allowing for global second-order effects..... 857
Choice of analysis type (ACI/AISC)..... 858
When should a concrete building be classed as non-sway?..... 858
Choice of analysis type (BS)..... 859
First-order (Elastic) analysis..... 859
Amplified forces (k_{amp}) method..... 859
Second-order analysis..... 860
A practical approach to setting the analysis type..... 860
How do I assess the worst elastic critical load factor for the building?..... 861
How is the elastic critical load factor calculated?..... 862
What are the twist results?..... 862
Choice of analysis type (Eurocode)..... 863
First-order (Elastic) analysis..... 863
Amplified forces (k_{amp}) method..... 863
Validity of the amplified forces method..... 864
Second-order analysis..... 864
A practical approach to setting the analysis type..... 864
When should a building be classed as sway sensitive?..... 866
How do I assess the worst elastic critical load factor for the building?..... 866
How is the elastic critical load factor calculated?..... 866
What are the twist results?..... 867
Derivation of the k_{amp} formula for concrete structures..... 868
Use of modification factors..... 870
11.3 Allowing for global imperfections..... 871
Allowing for global imperfections (ACI/AISC)..... 871

Allowing for global imperfections (Eurocode)....871
Allowing for global imperfections (BS)....872
11.4 Allowing for member imperfections....872
Allowing for member imperfections (ACI/AISC)....873
Allowing for member imperfections (Eurocode)....873
Allowing for member imperfections (BS)....873

12 Industrial structures handbook....874
12.1 Ancillaries....875
What are ancillaries used for?....875
Ancillary load default values....877
Ancillary loadcases....878
Ancillary load decomposition....879
Create line ancillary loads....881
Create area ancillary loads....882
Generate ancillary combinations....883
Create an ancillary loads report....886
12.2 Equipment....887
Overview....888
Equipment loadcases....891
Equipment load decomposition....892
Create equipment and equipment loads....895
Create additional equipment loads in other loadcases....895
Generate equipment combinations....896
Create an equipment loads report....899
12.3 Inactive members....900
Which members can be made inactive?....900
To make a member inactive....900
Inactive member load decomposition....901
Typical usage cases for inactive members....902
12.4 Fire proofing....907
How to apply fire proofing....907
Fire protection material details....907

13 Embodied carbon handbook....909
13.1 Measuring the carbon impact of a structure....909
Global impact of construction industry....909
Typical emissions at each stage of the structure's life....909
Measuring Product Stage Carbon....910
Reporting and export of embodied carbon data....911
13.2 Add and manage embodied carbon factors....911
Set up a global set of factors....912
Set up and edit the local set of factors....912
Add factor....912
Edit factor....913
Remove factor....914
Reorder the list of factors....915
Set a factor as active or inactive....915
Export factors to a spreadsheet....915
13.3 Embodied carbon workflow....915
Set up embodied carbon factors....916
Manage how factors are applied to the model....920

Review how factors have been applied....922
Examine embodied carbon overview....922
Display the overview data as a chart....929
Examine embodied carbon details 931
Review utilization and embodied carbon....932
Create reports....933

1 Steel design handbook

This handbook contains information relating to the design of steel members in Tekla Structural Designer.

- Combined analysis and design choices for steel structures (page 19)
- Steel member autodesign (page 21)
- Steel member design groups (page 24)
- Steel member orientation (page 26)
- Steel beam design (page 28)
- Composite beam design (page 41)
- Steel column design (page 66)
- Column base plate design (page 80)
- Steel brace design (page 84)
- Steel joist design (page 86)
- Steel truss design (page 91)
- Portal frame design (page 95)

See also

Steel connections handbook (page 96)

1.1 Combined analysis and design choices for steel structures

Gravity design

For many steel structures a separate gravity design is not necessary and you can simply proceed directly to a **Static design**.

In some circumstances however, you may find it preferable to adopt a two-stage workflow in which gravity members are sized prior to the full design.

The benefits of this are:

- For larger models it can significantly speed up the analysis and design time
- If lateral resisting systems are not modeled, then mechanisms can result and the analysis may not find a solution. Since these lateral resisting systems are perhaps unknown early in the project (or insufficient systems have been provided), running a gravity design has the added benefit of automatically fixing column nodes (not attached to a diaphragm) horizontally, which stabilizes the model.

The gravity design stage enables you to design all members (and hence size those members with the Autodesign property checked), for only the active gravity combinations (this will include the Construction Stage combination). As an example, beams within a rigid floor diaphragm are unaffected by lateral load, and hence they can be sized using only the gravity combinations.

Gravity design is initiated by clicking **Design Steel (Gravity)**.

After the gravity design has been completed, by default all steel members are reset to check design mode.

You may then decide to reset certain members to auto design e.g. columns and beams in 'moment frames', columns and braces in vertical or horizontal braced bays. In such cases, when the full static design is performed member 'pre-sizing' will take place and for members resisting lateral loads the new section size will be used if it is larger than that which resulted from the gravity design.

Static design

Full static design is initiated by clicking **Design Steel (Static)** or **Design All (Static)** on the **Design** tab.

All beams, columns and braces that do not have the **Gravity only** property checked, are designed or checked for all active combinations; Gravity only design members are designed or checked for the active gravity combinations only.

As part of the full design process a 3D Analysis is performed, for which you must select (via Design Options) the analysis type. The choice of analysis type will depend on the code being designed to.

Designing 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

Setting columns that do not help resist lateral loads to be designed for gravity loads only, will reduce the overall design time.

Engineering judgement 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, they are only designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered. Hence a warning is provided, if the ignored loads exceed a preset limit.

1.2 Steel member autodesign

The design mode for each member is specified in its properties.

- When **Autodesign** is not selected (i.e. check mode), you assign your desired section size to the member and Tekla Structural Designer determines if the section is sufficient.
- When **Autodesign** is selected the section type to be used is specified from a Design Section Order and Tekla Structural Designer attempts to automatically determine a suitable section.

  The following controls can be applied to further limit the sections considered:

  - Size constraints (page 22)
  - Steel and composite beam - autodesign optimization (page 23) (US Regional code only)

- Design section orders

NOTE If a member type has been set to be designed using Design groups, then if at least one member of the group is set to autodesign, the whole group will be automatically designed.

Size constraints

When undertaking an Autodesign, the Size Constraints property allows you to ensure that the sections proposed by Tekla Structural Designer match any particular size constraints you may have.

For instance:

- Minimum width - you may want to ensure a minimum flange width to satisfy bearing requirements for any supported beams. To achieve this, you would enter a value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than the specified minimum width for the design of the beam.
- Maximum depth - you may want to ensure that the maximum beam depth is specified to ensure that it fits within a floor zone. To achieve this, you would enter a value as the Maximum depth and Tekla Structural Designer would not consider sections with depths greater than the specified value for the design of the beam.
- Max span/depth ratio (beams only) - you may want to set a max span/depth ratio limit above which autodesign will reject sections that are of insufficient depth.
  - The depth is taken as the depth from the section properties - for single and double angles this is the vertical leg length.
  - Limit is not applied to any member which has rotation.

NOTE Size constraints only work when undertaking an **Autodesign**

Applying a size constraint to an element

You can control which sections in a design order list are considered, by following the procedure below.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window

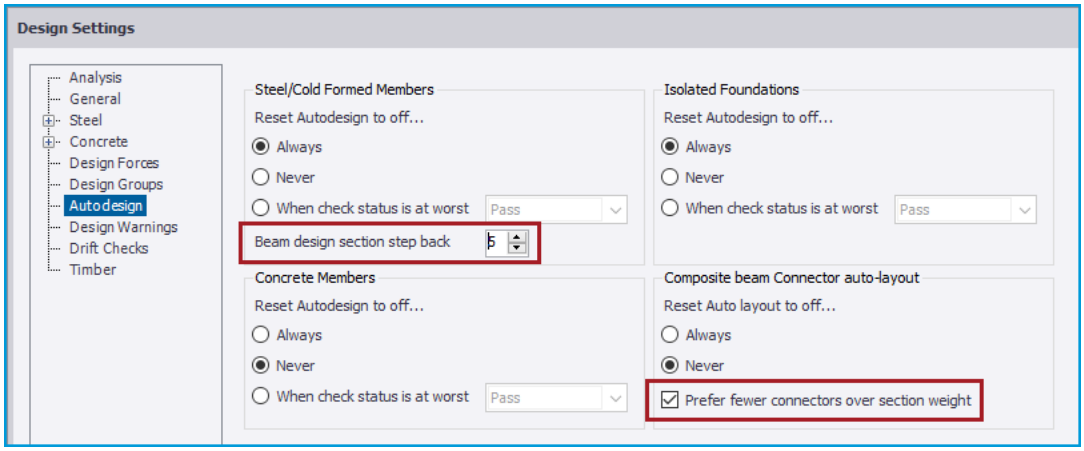
2. Expand All spans/stacks or the individual span/stack as required to reveal **Size Constraints**
3. Expand **Size Constraints** and set the **Max depth**, **Min depth**, **Max width**, **Min width**, or **Apply max/span/depth ratio** as required.

When an **Autodesign** is performed, the section selected from the section order list will fall within any size constraint limits set.

Steel and composite beam - autodesign optimization

NOTE The autodesign optimization features described here are only available for the United States Regional code.

In some circumstances (auto) Design of an individual beam following analysis could select a more optimized section than a full Steel Design process with the autodesign property enabled. This can occur for both composite and non-composite beams potentially, generally to do with the influence of the self weight of presize sections. Additional autodesign settings for the United States can be used to improve section optimization in such cases - the settings are located in Design Settings > Autodesign as shown in the picture below.



Beam design section step back

This relates to the design of steel beams in general and adds further control of the autodesign iterative process. The steel beam autodesign process sequentially loops through the steel section order list (from top to bottom) to identify a passing section together with analysis results to update the beam self weight each time there is a section change. For subsequent analysis and design iterations, looping through the section list starts by default from the most efficient passing section identified in the previous iteration which, depending on the change in self weight, might not always be the most optimum.

- The option allows you to specify where the iterative looping starts in the section list in relation to the optimum section from the prior iteration - either at the same section (a value of 0) or up to 5 sections before it in the list. Hence it is termed the section 'step back' value. Occasionally using this

with a value 1-5 can improve design efficiency, however with a potential to impact design times particularly for larger models.

Prefer fewer connectors over section weight

This relates specifically to steel composite beam design and the influence of studs (connectors) and section size on optimization - for a similar moment capacity you can potentially have a solution of more studs with a lighter section, or less studs with a heavier section. A designer may consider either of these solutions the 'optimum' in terms of overall economy - hence this setting which allows the designer to specify their preferred criteria:

- When enabled (default) - autodesign seeks the smallest possible number of studs in lieu of a potentially heavier section.
- When disabled - autodesign seeks the lightest possible section potentially requiring more studs.

1.3 Steel member design groups

Why use steel design groups?

Steel members are not automatically placed into design groups. They are however, automatically put into groups, primarily for editing purposes. In this way, individual groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.

If required, grouping can also (optionally) be utilized in order to design steel member types according to their groups.

In order to use grouping for this second purpose, you should first ensure that your groups are configured so that each group only contains those members that you intend to eventually have the same section size applied.

NOTE Grouped design is optional and can be activated if required, via Design Options> Design Groups

NOTE A fixed set of rules are initially used to determine the **automatic member grouping**: for example beams are designated as either direction 1 or direction 2, are of a similar span, material grade and construction type, columns have the same number of stacks etc. **You can manually edit groups.**

What happens in the group design process?

When the option to design a specific steel member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the section size so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the section size is adequate for each and if this proves not to be the case, it is increased as necessary and the revised section size is copied to all members in the group.
- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilizations are established.

NOTE When performing the design of a group of composite beams the changes in section size may mean that for some beams the existing stud layout will no longer remain adequate. To ensure the safe and rational design of the studs, the stud design routine is re-run at the end of the group design for all beams with the stud auto-layout active in the beam properties.

Steel design group requirements

Steel member design groups are formed according to the following rules:

Member type	Design group rules

Steel beam	• A beam element may only be in **one** design group
	• All beam elements in the group must have the same Construction/Fabrication i.e. Non-composite Rolled,
	• All beam elements in the group must have the same material properties
	• All beam elements in the group are of a similar span.*
	• All beam elements in the group must be aligned in the same direction.
	  • Groups that align with Dir 1 are automatically placed under the 'Dir1' main group heading
	  • Groups that align with Dir 2 are automatically placed under the 'Dir2' main group heading

Member type	Design group rules
	• Groups that don't align with either Dir 1, or Dir 2 are automatically placed under the 'Other' group heading
Steel column	• A column element may only be in **one** design group • All column elements in the group must have the same Construction/Fabrication i.e. Non-composite Rolled, • All column elements in the group must have the same material properties • All column elements in the group are of a similar length.* • All column elements in the group must have the same number of stacks

* In Model Settings>Grouping the user defined Maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes

Group management

Automatic Grouping

Steel members are grouped automatically according to the above group requirements.

Manual/Interactive Grouping

To manually reassign a member to a different group, locate it in the **Project Workspace**,**Groups tab** and drag it to a different group

NOTE A cross will appear if the element does not meet the group requirements.

NOTE A member can only be placed into a single design group.

Regroup ALL Model Members

If you have made changes in Design Options that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

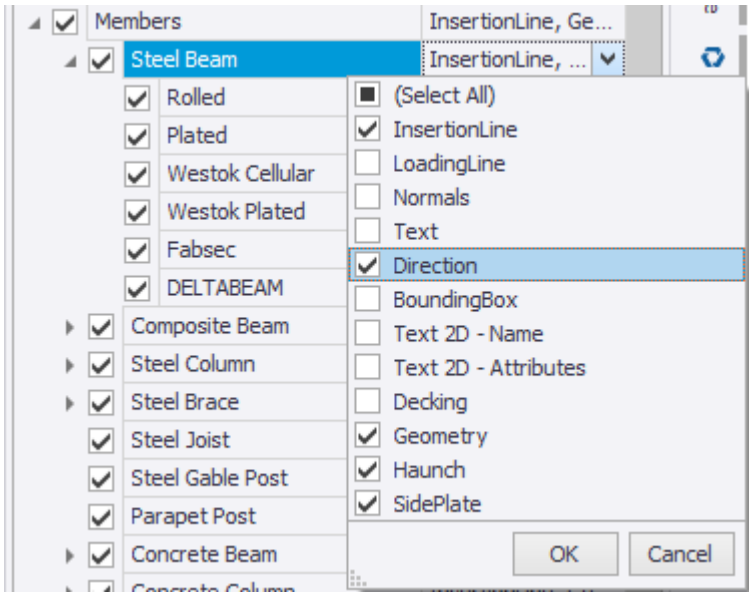
NOTE Any manually applied grouping will be lost if you elect to re-group!

1.4 Steel member orientation

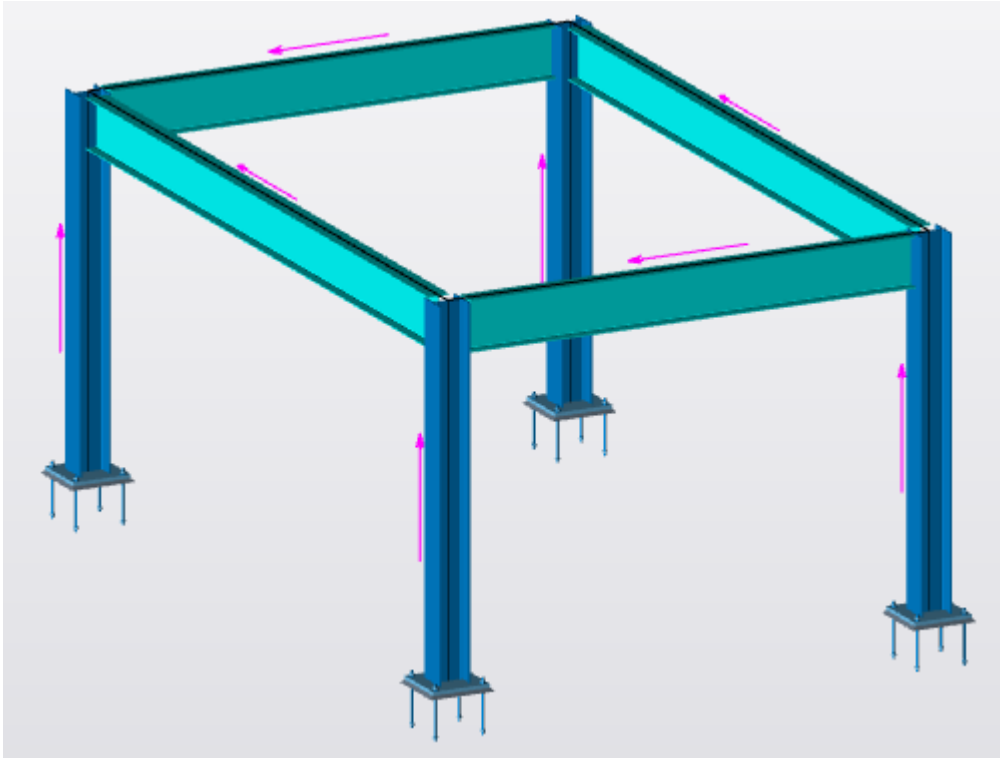
Tekla Structural Designer considers member orientation when displaying analysis results. Therefore, to apply the sign convention correctly, you need to know which is **end 1** and which is **end 2** of the member. For columns you also need to be able to identify the four faces: **A, B, C & D**.

To determine each of the above, you firstly need to display a direction arrow along each of the members, which can be done as follows:

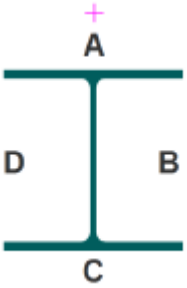
1. Open the **Scene Content** side pane.
2. Expand the Members group and then select the **Direction** option for the required member type(s).



In the active scene view a direction arrow is now displayed along the member which points from **end 1** to **end 2**.



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.



1.5 Steel beam design

Overview

Tekla Structural Designer will design steel beams for an international range of doubly symmetric I-sections, C-sections, rectangular and square hollow sections, single angles, double angles and tees for many different countries and also for many specific manufacturers. Plated beam design is also available for some regional codes.

Steel beams are designed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

In addition to major axis bending, minor axis bending and axial loads are also considered.

In its simplest form a steel non-composite beam can be a cantilever, or a single member between supports to which it is pinned.

It can also be a continuous beam consisting of multiple members that do not, with the exception of the remote ends, transfer moment to the rest of the structure. Each span of a continuous beam can be of different section size, type and grade.

At the remote ends of the beam there are a number of options for the end fixity depending upon to what the end of the beam is connected. These are:

- Free end
- Moment connection
- Pin connection
- Fully fixed
- Partially fixed (% of 4EI/L)
- Partially fixed (linear spring)

The beam may have incoming beams providing restraint and may or may not be continuously restrained over any length between restraints.

Fabrication

The **Fabrication** type is specified in the beam properties.

Fabrication types that can be designed as non-composite steel beams in Tekla Structural Designer are dependent on the design code.

Refer to the following table for details.

Head code	Rolled	Plated	Westok cellular	Westok plated	FAB SEC	DELTA BEAM
AISC	Yes	No	No	No	No	No
EC	Yes	Yes	Yes	Yes	No	No
BS	Yes	Yes	Yes	Yes	No	No

Head code	Rolled	Plated	Westok cellular	Westok plated	FAB SEC	DELTA BEAM
IS	Yes	Yes	No	No	No	No
AS	Yes	Yes	No	No	No	No

For guidance in relation to design of plated steel beams to a specific regional code, see: Plated steel beams (page 35)

Restraints

Each beam may have incoming beams providing restraint and may or may not be continuously restrained over any length between restraints.

Conditions of restraint can be defined in- and out-of-plane for compression (strut) buckling and top and bottom flange for lateral torsional buckling (LTB). The effective lengths used in the buckling checks depend on the type of restraint, particularly at supports.

In all cases, the program sets the default effective length to 1.0L, it does not attempt to adjust the effective length (between supports for example) in any way. You are expected to adjust the effective length factor (up or down) as necessary. Any compression or LTB effective length can take the type “Continuous” to indicate that it is continuously restrained over that length.

LTB and Compression restraints

LTB and Compression restraints are determined from the incoming members described within the Tekla Structural Designer model.

The default assumption is that an incoming member connecting to either side of the beam provides lateral restraint to the top flange only, and compression restraint about the minor axis. An incoming member connecting from above or below doesn't provide lateral restraint but provides compression restraint about the major axis.

By right-clicking a member to edit its properties in the **Properties** dialog box, you are then able to edit the restraints. You can indicate also continuously restrained sub-beams and edit length factors.

Note that the same level of restraint editing is not provided in the **Properties** window (although it does allow you to independently set both the top and bottom flanges as continuously restrained for the entire member length via the Top/Bottom flange cont. rest. properties).

TIP As an alternative to using the Steel Beam **Properties** dialog box, restraint settings for steel beams can be easily reviewed and adjusted graphically via **Review View > Show/ Alter State**. For more details see: Review and modify restraints

Torsion restraints

Torsion restraints are currently displayed in the **Properties** dialog box for information only. In design the beam is treated as unrestrained against warping and restrained against torsion.

Continuously Restrained Flanges

In the **Properties** window you can independently set both the top and bottom flanges as continuously restrained. By setting **Top flange cont. rest.** and/or **Bottom flange cont. rest.** to "Yes" the relevant buckling checks are not performed during the design process.

Deflection limits

Serviceability criteria often control the design of normal composite beams. This is because they are usually designed to be as shallow as possible for a given span.

Deflection Limits can be specified in the beam properties. These allow you to control the amount of deflection in both composite beams and steel beams by applying either a relative or absolute limit to the deflection under different loading conditions.

A typical application of these settings might be:

- not to apply any deflection limit to the slab loads, as this deflection can be handled through camber,
- to apply the relative span/over limit for live load deflection, to meet code requirements,
- possibly, to apply an absolute limit to the post composite deflection to ensure the overall deflection is not too large.

Torsion

A **Check for torsion** can be requested in the beam properties.

Torsion can only be checked to the AISC or Eurocode regional code and only under the following conditions:

- Section geometry
  - doubly symmetric rolled I-sections
  - closed RHS/SHS/CHS sections (steel and cold formed)
  - closed HSS sections
- pinned
  - single span beams
  - applied torsion moment
  - no web openings

If any of the above conditions are contravened the check is beyond scope.

Only a check design is performed, (no auto-design for torsion).

The procedure and scope are different for Open sections (I's) vs closed sections (HSS's)

- Open Sections:
  - Torsion design and angle rotation check will be carried out for applied torsion forces only. "Inherent" torsion is not checked
- Closed Sections
  - "Inherent" torsion and/or physically applied torsion loads checked
  - Angle of rotation check carried out for applied forces only

Camber

You can introduce camber by selecting **Apply camber** in the beam properties.

Camber is primarily used to counteract the effects of dead load on the deflection of a beam. This is particularly useful in long span composite construction where the self-weight of the concrete is cambered out. It also ensures little, if any, concrete over pour occurs when placing the concrete.

The amount of camber can be specified either:

- As a value
- As a proportion of span
- As a proportion of deflection
  - If this option is selected, the engineer should identify the combination to be used for the calculations by selecting Camber adjacent to the appropriate gravity combination on the Loading combination dialog page.

In the latter case, if 100% of the dead load deflection is cambered out, it is also possible to include a proportion of the live load deflection if required.

Lower limits can be set for span length and calculated camber below which camber is not applied. This ensures that unwanted or impractical levels of camber are not specified.

Limits to the maximum allowable camber as well as to the minimum section web thickness to avoid web crippling can also be defined. These will be accounted for in the automatic design process affecting the section size selection.

See also: Support article: Composite Beam deflections including camber

Floor vibration

Natural frequency

For steel beams the natural frequency calculation can optionally be requested by selecting **Calculate natural frequency** in the beam properties. When requested it can optionally be checked against a minimum frequency if required.

When the calculation is activated, 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 ALSO

-
-

Treat as composite

The general expectation is that any steel beam with a minimum nominal composite behavior should be considered as composite for footfall analysis, even if for design purposes it's still a non-composite beam.

This can catered for by selecting **Treat as composite** in the beam properties. Enabling this option allows the concrete flange width above the beam to be calculated taking into account the adjacent composite slab(s) in the same way as it would be for a beam that is defined as composite. (The beam's composite properties are only used in the footfall analysis).

Instability factor

Long members in a model that have axial force in them can be unstable during second-order analysis because their individual elastic critical buckling load factor is lower than the elastic critical buckling load factor of the building as a whole and is less than 1.0.

However, often such members, for example the rafters in a portal frame, are stable in design because, for example, there are many smaller members or sheeting, that restrain the member in reality. They fail in the analysis because it is too resource intensive to model all the individual restraining members in the model which would also add unwanted clutter.

To prevent or to reduce the incidence of such failures during the analysis a multiplier can be applied to the minor axis inertia of these members which caters for the effect of the restraining members.

This multiplier is defined by selecting **Prevent out of plane instability** in the member properties and then entering a suitable value in the **Instability factor** field.

When activated, the minor axis inertia of the member is multiplied by the factor, thus increasing its minor axis (theoretical elastic) buckling capacity. This only occurs in the **2nd order analysis** solver model. The default value for the factor is 20. This is just a sensible initial value, not a 'magic' number or one that will work in all circumstances. This behaviour also applies to the pre-size sections assigned during steel auto-design using 2nd order analysis. Hence the setting can be used to assist when buckling is occurring in the 2nd order analysis that is part of steel auto-design.

Since the setting adds ‘artificial’ stiffness to the model, the engineer should take care to ensure that it does not inappropriately increase the lateral stiffness of the structure as a whole leading to inaccurate results.

NOTE This multiplier is applied to prevent unwanted behaviour in the analysis model. While the analysis results may be affected by this adjustment, the increased minor axis inertia is NOT used in design checks.

Web openings

You cannot currently automatically design sections with web openings, you must perform the design first to get a section size, and then add and check the openings. This gives you complete control of the design process, since you can add appropriate and cost effective levels of stiffening if required, or can choose a different beam with a stronger web in order to reduce or remove any stiffening requirement.

When openings are added they can be defined as rectangular or circular and can be stiffened on one, or on both sides.

Openings cannot be defined from the **Properties** window, they can only be defined from the **Properties** dialog box, (by right-clicking on the member and selecting **Edit...**)

For guidance in relation to a specific regional code, see:

-
-
-

Fire check (Eurocode only)

The mechanical resistance of a steel beam in case of fire can be checked in accordance with EN 1993 & national annex for the UK, Ireland, Singapore, Malaysia, Sweden, Norway, Finland or the recommended Eurocode values.

The fire check is activated by selecting **Check for fire resistance** located under **Fire proofing** in beam properties, after which the user is required to specify:

- Load reduction factor for fire
- Required time of fire exposure
- Exposure
- Protected, or unprotected
    - If the protected option is selected, the engineer is then required to specify the fire protection material details.

The time interval for critical temperature iteration is specified in Design Settings, for both 'unprotected' and 'protected' situations.

The check compares the design shear force against the design resistance at the required time of fire exposure.

For the scope of the check and the limitations that apply, see:

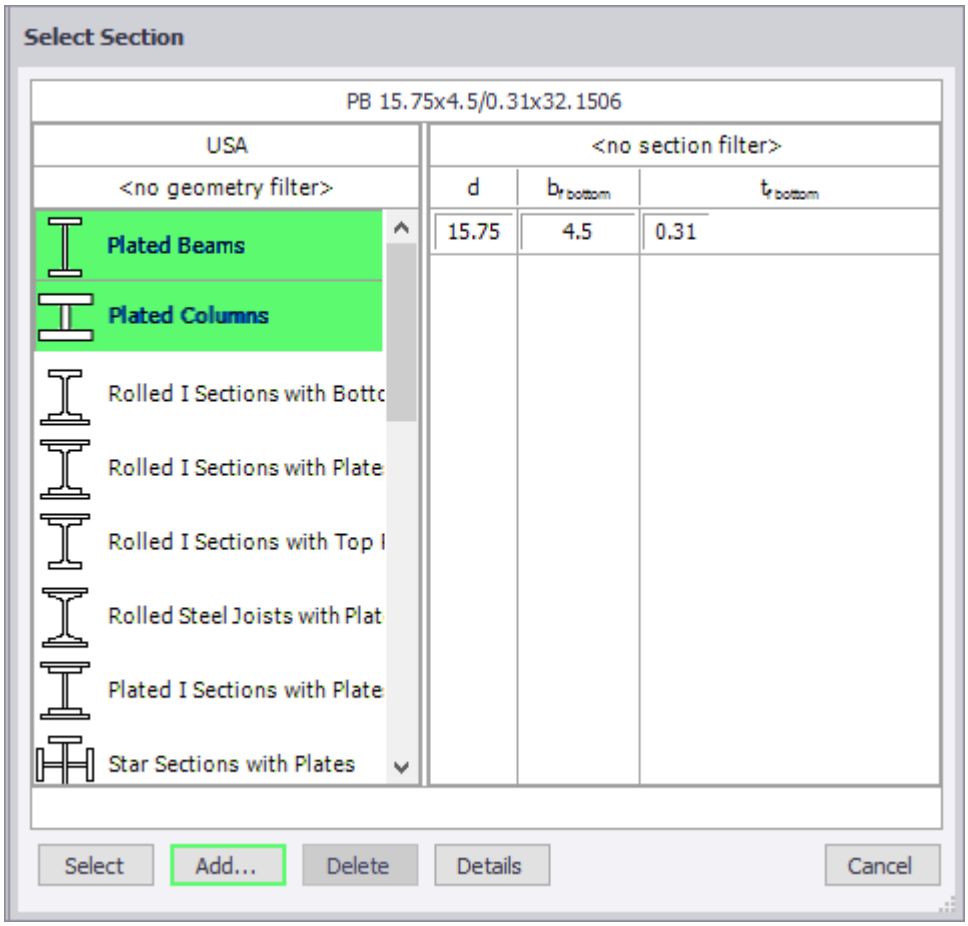
Plated steel beams

Plated beams - AISC regional code

Steel beams with the Fabrication type (page 29) set to **Plated** can be autodesigned or checked to AISC 360-05, 360-10 and 360-16, provided the section is selected from the lists of **Plated Beams** or **Plated Columns** highlighted below.

If the plated section were to be selected from any of the other lists of Compound section types (e.g. Rolled I-sections with Plates), design or checking would be beyond scope.

While one plated beam section has been provided as a default, you can define (and save) your own plated sections to the database by clicking the **Add...** button shown below.



Autodesign is only possible once you have added a range of sections to choose from. It would also require a **Design section order** to be specified, for which a new design section order would have to be created specifically for the plated sections.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

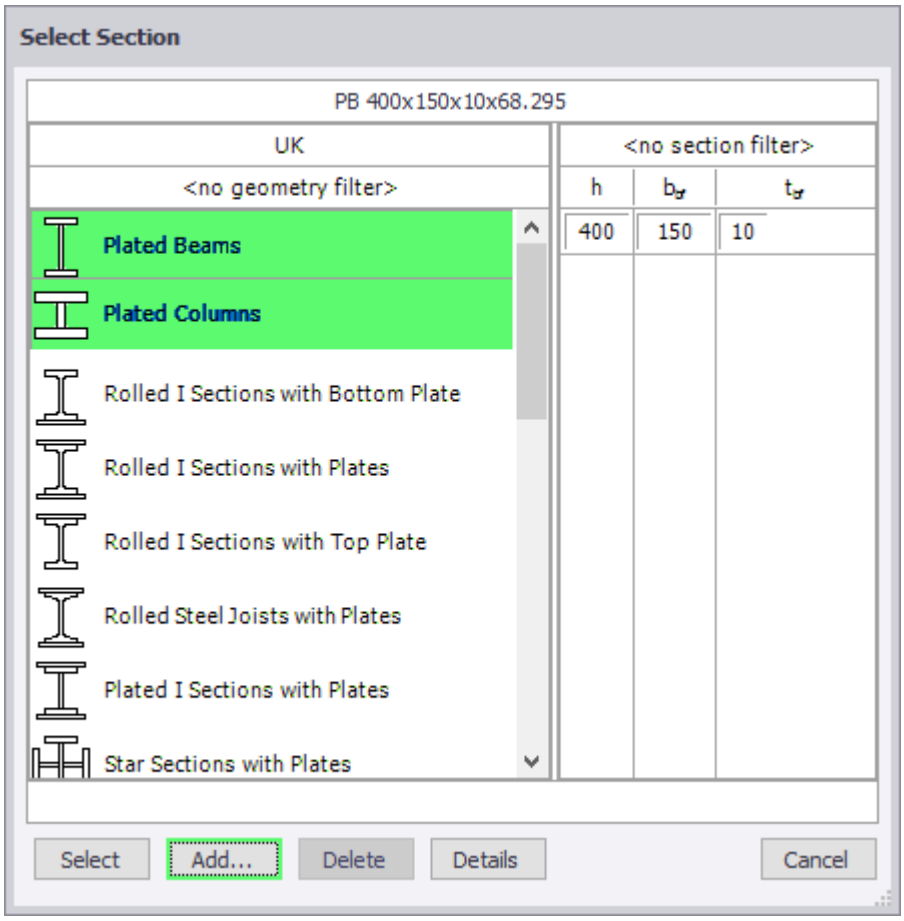
- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- Only parallel flange sections can be used.

Plated beams - Eurocode regional code

Steel beams with the Fabrication type (page 29) set to **Plated** can be autodesigned or checked to Eurocodes provided the section is selected from the lists of **Plated Beams** or **Plated Columns** highlighted below.

If the plated section were to be selected from any of the other lists of Compound section types (e.g. Rolled I-sections with Plates), design or checking would be beyond scope.

While one plated beam section has been provided as a default, you can define (and save) your own plated sections to the database by clicking the **Add...** button shown below.



Autodesign is only possible once you have added a range of sections to choose from. It would also require a **Design section order** to be specified, for which a new design section order would have to be created specifically for the plated sections.

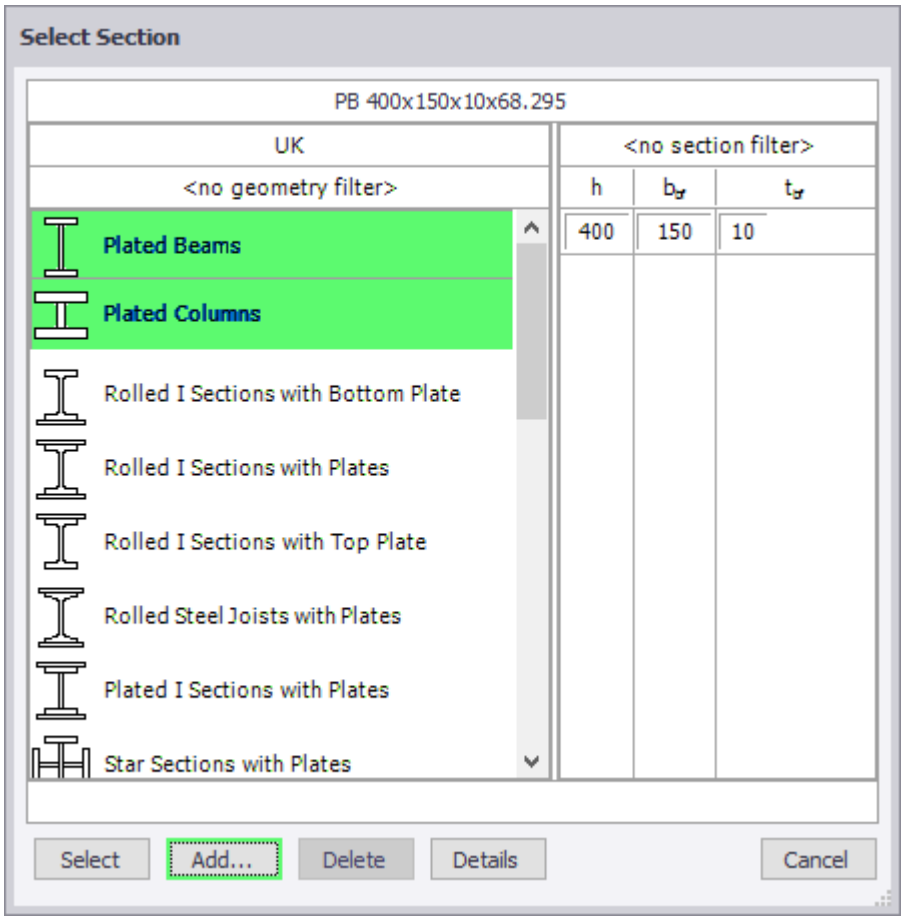
In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - BS regional code

Steel beams with the Fabrication type (page 29) set to **Plated** can be autodesigned or checked to BS 5950 provided the section is selected from the lists of **Plated Beams** or **Plated Columns** highlighted below.

If the plated section were to be selected from any of the other lists of Compound section types (e.g. Rolled I-sections with Plates), design or checking would be beyond scope.

While one plated beam section has been provided as a default, you can define (and save) your own plated sections to the database by clicking the **Add...** button shown below.

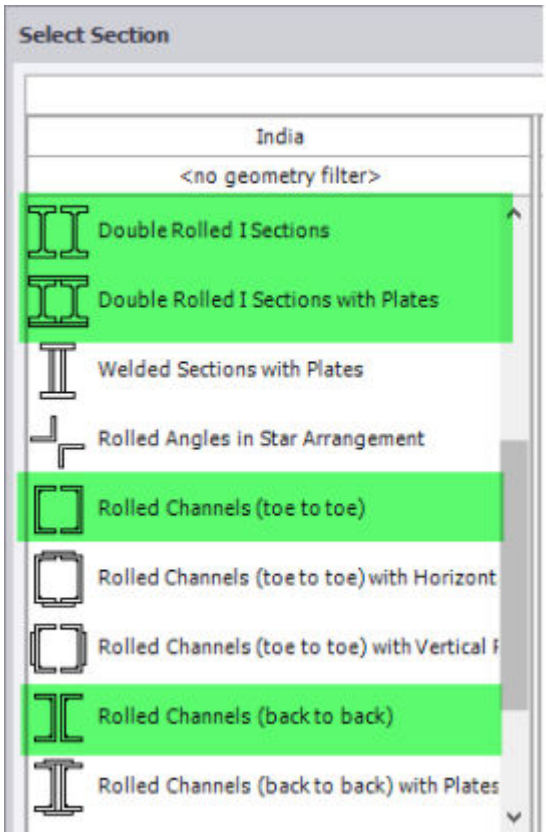


Autodesign is only possible once you have added a range of sections to choose from. It would also require a **Design section order** to be specified, for which a new design section order would have to be created specifically for the plated sections.

In general, the scope of strength checks undertaken is the same as for rolled sections.

Plated beams - Indian regional code

When the steel beam Fabrication type (page 29) is set to **Plated** the range of compound sections highlighted below can be autodesigned or checked to the Indian design code.



The section shapes supported are:

- Rolled Channels back to back
- Rolled Channels toe to toe
- Doubled rolled I sections
- Doubled rolled I sections with plates

The scope of design of these compound sections includes both beams and columns and autodesign.

Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for compound sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.

- High Shear case with minor axis moment is beyond scope.
- Design of the lacing or battening system is beyond scope.
- Only parallel flange sections can be used.

Plated beams - Australian regional code

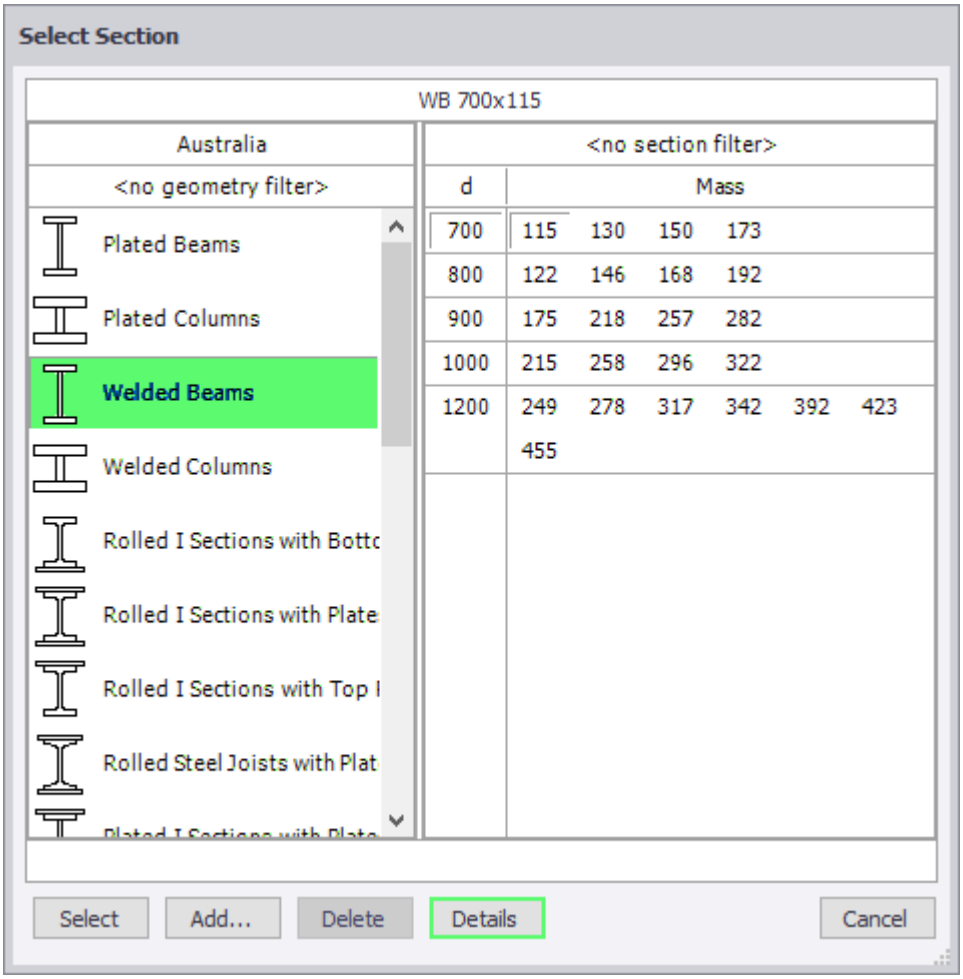
When the Fabrication type (page 29) is set to **Plated**, only welded doubly symmetric I-section beams (as highlighted below) can be autodesigned or checked to the Australian code.

A pre-defined range of these is available from the **Welded Beams** page of the Select Section dialog - these can be both checked and autodesigned.

Autodesign requires a **Design section order** to be specified, the Welded Beams design section order is provided for this purpose.

While you can add to the range of welded beams the Select Section dialog by clicking the **Add...** button, any such user-defined beams can only be analyzed, but are not designed.

Plated beams available on other pages of the Select Section dialog can be analyzed, but are not designed.



1.6 Composite beam design

Overview

Composite beams are designed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

The beams must be simply supported, single span unpropped structural steel beams.

The following are beyond scope:

- continuous or fixed ended composite beams,
- composite sections formed from hollow rolled sections,

- composite sections where the concrete slab bears on the bottom flange,
- the use of fiber reinforcement.

Beams are designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered.

NOTE If either minor axis bending or axial loads exist which exceed a limit below which they can be ignored, a warning is given in the beam design summary.

Profiled steel sheeting can be perpendicular, parallel and at any angle in between relative to the supporting beam web.

Tekla Structural Designer will determine the size of beam which:

- acting alone is able to carry the forces and moments resulting from the construction stage,
- acting compositely with the slab using profile steel decking (with full or partial interaction) is able to carry the forces and moments at the composite stage,
- acting compositely with the slab using profile steel decking (with full or partial interaction) is able to provide acceptable deflections, service stresses and natural frequency results.

Alternatively you may give the size of a beam and Tekla Structural Designer will then determine whether it is able to carry the previously mentioned forces and moments and satisfy the Serviceability requirements.

An auto-layout feature can be used for stud placement which caters for both uniform and non-uniform layouts.

Construction Stage loading

You define these loads into one or more loadcases as required.

The loadcase defined for construction stage slab wet concrete has a Slab wet loadcase type specifically reserved it. Clicking the Calc Automatically checkbox enables this to be automatically calculated based on the wet density of concrete and the area of slab supported. An allowance for ponding can optionally be included by specifying it directly in the composite slab properties.

If you uncheck Automatic Loading, this loadcase is initially empty - it is therefore important that you edit this loadcase and define directly the load in the beam due to the self weight of the wet concrete. If you do not do this then you effectively would be designing the beam on the assumption that it is propped at the construction stage.

It is usual to define a loadcase for Live construction loads in order to account for heaping of the wet concrete etc.

Having created the loadcases to be used at construction stage, you then include them, together with the appropriate factors in the dedicated

construction stage design combination. You can include or exclude the self-weight of the beam from this combination and you can define the load factors that apply to the self weight and to each loadcase in the combination.

NOTE You should include the construction stage slab wet concrete loadcase in the construction stage combination, it cannot be placed in any other combination since it's loads relate to the slab in its wet state. Conversely, you cannot include the Slab self weight loadcase in the construction stage combination, since it's loads relate to the slab in its dry state. The loads in the construction stage combination should relate to the slab in its wet state and any other loads that may be imposed during construction.

NOTE TIP: If you give any additional construction stage loadcases a suitable title you will be able to identify them easily when you are creating the construction stage combination.

Composite stage loading

You define the composite stage loads into one or more loadcases which you then include, together with the appropriate factors in the design combinations you create. You can include or exclude the self-weight of the steel beam from any combination and you can define the load factors that apply to the beam self weight and to each loadcase in the combination.

The Slab self weight loadcase is reserved for the self weight of the dry concrete in the slab. Clicking the Automatic Loading checkbox enables this to be automatically calculated based on the dry density of concrete and the area of slab supported. An allowance for ponding can optionally be included by specifying it directly in the composite slab properties.

If you uncheck Automatic Loading, the Slab self weight loadcase is initially empty - it is therefore important that you edit this loadcase and define directly the load in the beam due to the self weight of the dry concrete. For each other loadcase you create you specify the type of loads it contains – Dead, Live or Wind.

For each load that you add to an Live loadcase you can specify the percentage of the load which is to be considered as acting long-term (and by inference that which acts only on a short-term basis).

All loads in Dead loadcases are considered to be entirely long-term while those in Wind loadcases are considered entirely short-term.

Allow non-composite design

The following controls which are located under the **General** heading in the beam properties can be selected to enable non-composite design in specific circumstances.

- Attempt non-composite if auto composite solution fails
- Try composite section size as non-composite

Attempt non-composite if auto composite solution fails

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 with negative bending moments,
- 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.

For the non-composite attempt the properties of the composite beam, including the stud count, are retained.

Try composite section size as non-composite

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 section size non-compositely and if successful will retain this non-composite solution, discarding the studs.

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 **Fabrication** type is specified in the beam properties.

The fabrication types that can be designed as composite steel beams in Tekla Structural Designer are dependent on the design code.

Refer to the following table for details.

Regional code	Rolled	Plated	Westok cellular	Westok plated	FAB SEC	DELTA BEAM
AISC	Yes	No	No	No	No	No
EC	Yes	Yes	Yes	Yes	No	No
BS	Yes	Yes	Yes	Yes	No	No
IS	N/A - Design of composite beams to IS codes is currently beyond scope					
AS	N/A - Design of composite beams to AS codes is currently beyond scope					

For guidance in relation to design of plated composite beams to Eurocodes or the BS regional code, see: Plated steel beams (page 35)

Restraints

You can independently set both the top and bottom flanges of a composite beam as continuously restrained in the **Properties** window.

When the beam is initially created the decking direction is unknown until the beam is actually placed and the floor slab and direction are also created. Hence defaults are provided for each eventuality.

The defaults are:

- for perpendicular decks the deck restrains the beam top flange
- for parallel decks the deck does NOT restrain the beam top flange
- for precast decks the deck restrains the beam top flange
- for all decks the deck does NOT restrain the beam bottom flange

By setting the top flange as continuously restrained and/or the bottom flange as continuously restrained the relevant buckling checks are not performed during the design process.

When not continuously restrained, LTB and Compression restraints are determined from the incoming members described within the Tekla Structural Designer model. The buckling checks are based on these.

By right-clicking a member to edit its properties in the **Properties** dialog box, you are then able to edit the restraints. You can indicate continuously restrained sub-beams and also edit length factors.

For composite beams the buckling checks are only performed at construction stage as at composite stage they are always assumed to be fully restrained.

TIP Restraint settings for composite beams can be easily reviewed and adjusted graphically via **Review View > Show/ Alter State**. For more details see: Review and modify restraints

Deflection limits

Serviceability criteria often control the design of normal composite beams. This is because they are usually designed to be as shallow as possible for a given span.

Deflection Limits can be specified in the beam properties. These allow you to control the amount of deflection in both composite beams and steel beams by applying either a relative or absolute limit to the deflection under different loading conditions.

A typical application of these settings might be:

- not to apply any deflection limit to the slab loads, as this deflection can be handled through camber,
- to apply the relative span/over limit for live load deflection, to meet code requirements,
- possibly, to apply an absolute limit to the post composite deflection to ensure the overall deflection is not too large.

Camber

You can introduce camber by selecting **Apply camber** in the beam properties.

Camber is primarily used to counteract the effects of dead load on the deflection of a beam. This is particularly useful in long span composite construction where the self-weight of the concrete is cambered out. It also ensures little, if any, concrete over pour occurs when placing the concrete.

The amount of camber can be specified either:

- As a value
- As a proportion of span
- As a proportion of deflection
    - If this option is selected, the engineer should identify the combination to be used for the calculations by selecting Camber adjacent to the appropriate gravity combination on the Loading combination dialog page.

In the latter case, if 100% of the dead load deflection is cambered out, it is also possible to include a proportion of the live load deflection if required.

Lower limits can be set for span length and calculated camber below which camber is not applied. This ensures that unwanted or impractical levels of camber are not specified.

Limits to the maximum allowable camber as well as to the minimum section web thickness to avoid web crippling can also be defined. These will be accounted for in the automatic design process affecting the section size selection.

See also: Support article: Composite Beam deflections including camber

Floor vibration

Whether a natural frequency calculation is performed or not is controlled by the **Calculate natural frequency** setting in the beam properties.

For EC and BS regional codes the composite beam natural frequency is always calculated and checked against a minimum frequency (with results displayed in Results Viewer).

For the US regional code the natural frequency calculation can optionally be requested, and when requested can optionally be checked against a minimum frequency if required. Note that the US regional code has no requirement for this check.

When the calculation is activated, 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 ALSO

-
-

Floor construction

Deck Type, Angle and Condition

The deck type and angle used in the beam design are determined from the properties of an adjacent slab item, (provided the slab item type has been set to Composite slab). If there are multiple adjacent slab items with different properties, it is the users responsibility to indicate which one governs.

Note the following with regard to the slab item properties:

- You will find that a wide range of profiled metal decks have been included for manufacturers from many countries. Precast concrete planks (page 63) are also available, but only for the EC regional code.

- The slab item's rotation angle relative to the global X axis is used to set the profiled metal deck as spanning at any angle between 0° (parallel) and 90° (perpendicular) to the direction of span of the steel beam.
- The concrete type can be set to either normal or lightweight, however design using lightweight slabs is only available for the Eurocode.
- When designing to the Eurocode, warnings are issued in the design if the concrete slab does not comply with the following constraints:
  - Normal weight concrete range C20/25 - C60/75 - See EN 1994-1-1:2004 Clause 3.1(2),
  - Lightweight concrete range LC20/22 - LC60/66 - See EN 1994-1-1:2004 Clause 3.1(2),
  - Minimum density for lightweight concrete 1750 kg/m3 - see EN 1994-1-1:2004 Clause 6.6.3.1(1).

The beam's "condition" is:

- restricted to internal if it has composite slabs attached along its full length on both sides,
- restricted to edge if it has no composite slabs on one side,
- defaulted to edge (but editable) if it has composite slabs on both sides but not along the full length.

Shear connector type

The shear connection between the concrete slab and the steel beam is achieved by using shear studs.

NOTE The use of channel connectors or Hilti™ connectors is currently beyond the program scope.

Regional Code: US

3/4 in diameter studs with 3 3/8, 3 7/8, 4 3/8, 4 7/8, 5 3/8, 5 7/8 nominal height are offered.

Regional Code: Eurocode

- 19mm diameter studs with 100, 125, and 150mm nominal height (95, 120 and 145mm as welded height).
- 22mm diameter studs are also offered with 125 and 150 nominal height. These can only be used for precast plank decks and profiled metal decks with 'closed' ends. (e.g. Sigmat).

Regional Code: BS

19mm diameter studs with 100 and 125 nominal height (95 and 120 as welded height) are offered. 22mm diameter studs are also offered but only for precast plank decks. Studs do not have a given capacity as their resistance is derived.

Welding condition (Eurocode only)

The stud welding types that are available depend on the decking being supported. Changing the welding condition can be used to control whether the decking is treated as continuous or discontinuous.

Decking	Welding type	Effect
Precast concrete planks	'Site welded', or 'Shop welded'	No effect.
Closed ends decking (e.g. Sigmat)	'Shop welded'	Enables additional functionality for closed ends decking.
Other profiled metal decking - perpendicular	'Site welded'	No effect.
Other profiled metal decking - parallel	'Site welded', or 'Shop welded'	If 'Site welded' is selected the deck is treated as discontinuous. This enables the additional setting of Haunch width, b_0. See:

Effective width

The effective width is used in strength and serviceability calculations. It can be entered manually, or be calculated automatically, either as a one time process or each time a design is performed.

To manually enter the effective width:

1. Specify the **Floor construction > Effective width** value you wish to use.

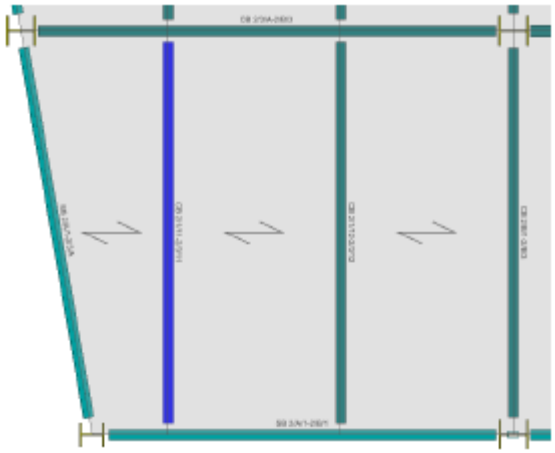
> **NOTE** If any amendments are made to the geometry of the model then you would need to amend the effective width value as necessary.
>
> The 'Calculate' button shown below the effective width property can be used as a helper to determine the code based values.

To calculate the effective width automatically as a one time process:

1. First select **Calculate effective width** in the properties.
2. Then click the [...] box that appears next to **Calculate**.
3. The calculated value is reported in the effective width property.

Although the program calculates the effective width, it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgment may sometimes be required.

EXAMPLE Consider the beam highlighted below:



The program calculates the effective width as the sum of:

- to the right of the beam, b_e(right) = beam span/8
- to the left of the beam, b_e(left) = one half of the shortest distance to the center line of the adjacent diagonal beam

To the left of the beam, some engineers might prefer to use one half of the mean distance to the adjacent beam. To do so you would need to manually adjust the calculated value via the Floor construction page of the Beam Properties.

NOTE If any amendments are made to the geometry of the model then you would need to use Calculate again or use the procedure below.

To automatically calculate the effective width each time a design is performed:

To combat the possibility of model edits and the need to recalculate the effective width for individual beams it is poss'ible to automatically perform the 'calculate' process discussed above automatically to all beams prior to a 'Design' command.

1. Ensure **Update effective width prior to design check** is checked on in **Design > Settings > Steel > Composite beam**
2. Each composite beam's floor construction properties will then display an **Override effective width** option.
    a. With **Override effective width** unchecked, the effective width will be determined when a 'Design' command is performed and the value is reported to you in the effective width property which is not editable.
    b. With **Override effective width** checked, you can override the effective width value and the value you input into the effective width property is used for design. The 'Calculate' button can still be used to advise the code based values.

Regional code based values: US

The 'Calculate' button determines the effective width of the compression flange, b_e for each composite beam as per Section I3.1a (360-05/-10).

For each side of the beam, it is taken as the smaller of:

- beam span/8 – span taken as the center to center of supports
- one half of the distance to the center line of the adjacent beam
- the distance to the edge of the slab

Regional code based values: EC

The 'Calculate' button determines the effective width of the compression flange, b_{eff}, for each composite beam as per Clause 5.4.1.2 of EC4.

Unless hollowcore units are used, it is taken as the smaller of:

- Secondary beams: the spacing of the beams, or beam span/4
- Primary beams (conservatively): 80% of the spacing of beams, or beam span/4
- Edge beams: half of above values, as appropriate, plus any projection of the slab beyond the centerline of the beam.

For hollowcore precast plank units only, we calculate the effective width for each side of the beam as the minimum of:

- Assumed concrete fill (500 mm) + recommended gap
- beam span/8 – span taken as the center to center of supports
- one half of the distance to the center line of the adjacent beam
- the distance to the edge of the slab

Regional code based values: BS

The 'Calculate' button determines the effective width of the compression flange, b_e, for each composite beam as per section 4.6 of BS 5950 : Part 3 : Section 3.1 : 1990.

For each side of the beam, it is taken as the smaller of:

- beam span/8 – span taken as the center to center of supports
- one half of the distance to the center line of the adjacent beam (for slabs spanning perpendicular)
- 40% of the distance to the center line of the adjacent beam (for slabs spanning parallel)
- the distance to the edge of the slab

To override the effective width in a Review View:

The Show/Alter State **Override Effective Width** attribute allows you to graphically set the composite beams to which an effective width override applies. For details, see:

Metal deck

Provided the adjacent slab item specified in 'Floor Construction' is a composite slab, the following 'Metal Deck' values can be specified in the beam properties.

Minimum lap distance

The position and attachment of the decking is taken into account in the longitudinal shear resistance calculations.

The applied longitudinal shear force is calculated at the center-line of the beam, and at the position of the lap (if known). If the position of the lap is not known, then the default value of 0mm should be used (that is the lap is at the center-line of the beam) as this is the worst case scenario.

Haunch width above beam for parallel decking, b_0 - (Eurocode only)

This property is 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:

Stud strength

The stud properties you can choose from are appropriate to the stud source. For additional Eurocode specific information, see:

All types of stud may be positioned in a range of patterns.

Stud groups - Regional Code ACI/AISC

You can allow group sizes of 1, 2, 3 or 4 studs - any group sizes that you don't want to be considered can be excluded.

For example, if you do not set up groups with 3 or 4 studs, then in auto-design the program will only try to achieve a successful design with a maximum of 2 studs in a group.

For each group that you allow you must enter the 'Set distance emid to' - as either >= 2in or < 2in and you also specify the pattern to be adopted (e.g. along the beam, across the beam, or staggered).

Stud groups - Regional Code Eurocode or BS

You can allow group sizes of 1 or 2 studs - any group sizes that you don't want to be considered can be excluded.

For example, if you do not set up groups with 2 studs, then in auto-design the program will only try to achieve a successful design with a maximum of 1 stud in a group.

For groups with 2 studs you must specify the pattern to be adopted (e.g. along the beam, across the beam, or staggered).

NOTE It is up to you to check that a particular pattern fits within the confines of the rib and beam flange since Tekla Structural Designer will draw it (and use it in design) anyway.

Optimize shear interaction

If you choose the option to optimize the shear interaction, then Tekla Structural Designer will progressively reduce the number of studs either until the minimum number of studs to resist the applied moment is found, until the minimum allowable interaction ratio is reached or until the minimum spacing requirements are reached. This results in partial shear connection.

Connector layout

When running in Auto-design mode you may not want to specify the stud layout at the start of the design process. To work in this way check **Auto-layout** to have the program automatically control how the stud design will proceed. When the beam is subsequently designed Auto-layout invokes an automatic calculation of the required number of studs, which is optimized to provide an efficient design.

NOTE 'Auto layout' can actually be checked regardless of whether you are auto designing the beam size or checking it. The combination of 'Check' design with 'Auto layout' of studs can be used to assist you to rationalize your designs e.g. to force a beam to be the same size as others in the building but have Tekla Structural Designer determine the most efficient layout of studs.

You may choose to perform the initial design with Auto-layout checked and then refine the spacing with Auto-layout cleared if the spacing is not exactly as you require. This may arise if for instance the theoretical design needs to be marginally adjusted for practical reasons on site.

Auto-layout for perpendicular decks

For perpendicular decks, the Auto-layout dialog provides two options for laying out the studs:

- Uniform
- Non-uniform
- Near uniform (US design only)

Uniform

The Uniform option forces placement in ribs at the same uniform spacing along the whole length of the beam.

Whether the stud groups are placed in every rib (as shown above), alternate ribs, or every third rib etc. can be controlled by adjusting the limits you set for Minimum group spacing ( ) x rib and Maximum group spacing ( ) x rib.

The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the Stud strength page.

EXAMPLE If you set Minimum group spacing 2 x rib and Maximum group spacing 3 x rib, then the program will only attempt to achieve a solution with studs placed in alternate ribs, or studs placed in every third rib. It will not consider a solution in which studs are placed in every rib.

Additionally, if on the Studs - Strength page, you have allowed groups of 1 stud and 2 studs; then if 1 stud per group proves to be insufficient the program will then consider 2 studs per group.

Non-uniform

If optimization has been checked (see **Optimize shear interaction** in the preceding **Stud Strength** topic) studs are placed at suitable rib intervals (every rib, alternate ribs, every third rib etc.), in order to achieve sufficient interaction without falling below the minimum allowed by the code.

NOTE The optimum and minimum amounts of shear connection are defaulted to 50% and 25% respectively. These can be adjusted if required.

If optimization has not been checked, studs are placed at suitable rib intervals in order to achieve 100% interaction.

Knowing the number of studs necessary to achieve the required level of interaction, it is possible that placement at a given rib interval could result in a shortfall; the program will attempt to accommodate this by working in from the ends, (as shown in the example below). If every rib is occupied and there is still a shortfall, the remainder are 'doubled-up', by working in from the ends once more.

In this example the point of maximum moment occurs one third of the way along the span, this results in an asymmetric layout. If you prefer to avoid such arrangements you can select **Adjust layout to ensure symmetrical about**

center line. A redesign would then result in the symmetric layout shown below.

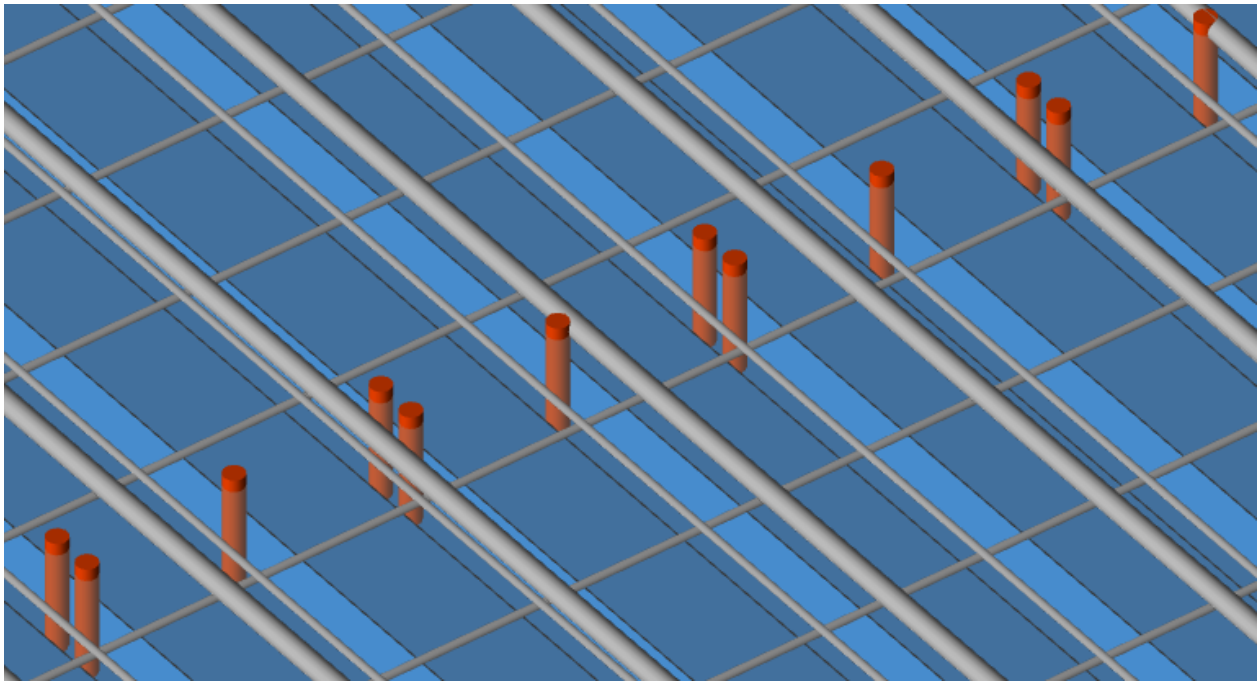


For both Uniform and Non-uniform layouts, if the minimum level of interaction cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

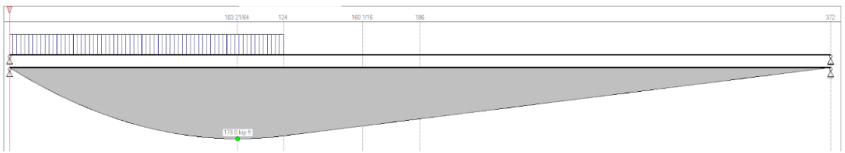
Near uniform (US design only)

The Near uniform layout option can produce a design which is both more efficient - using less studs - and more practical for construction than Uniform or Non-uniform layouts.

To illustrate, for a case where the optimum solution falls between 1 and 2 studs per rib, the Near uniform auto-layout method will try a [2,1] alternating pattern along the span of the beam saving 25% on studs compared to 2 per rib and producing a simple pattern for construction: 1 stud per rib, then add 1 more to alternate ribs.



EXAMPLE



Consider the example above where the bending moment diagram is skewed to one side. The 'near uniform' setting requires studs to be placed at the same spacing along the whole length of the beam but the critical spacing is set by the shorter length to the maximum moment, in this case by the left hand portion. This example requires 11 studs to the maximum moment position and there are 8 ribs to this position.

Firstly the 'base' layout of studs is calculated as (no. of studs)/(no. of ribs) = 11/8 = 1.375 and thus base_no = 1. That is 1 stud per rib to give 8 studs but 11 are required so 'remainder' = 3.

Secondly the 'remainder' is evenly spread over the 8 ribs and so some of the ribs must be given an extra stud. In this case this is achieved by doubling up in ribs 1, 3, 5, & 7.

The final layout of studs to the left of the maximum moment is 2,1,2,1,2,1,2,1 = 12 which is > the 11 required and hence OK.

For the other portion of the beam, in this case to the right of the maximum moment, the process starts with rib 'n' i.e. the last rib on the beam and then works back towards the maximum moment position using the same procedure viz.

1 stud is added per every rib to give the 'base' layout. Then the 'remainder' is distributed by adding 1 more to rib 'n'. Step = 2 as determined by the shorter length so studs are doubled up in rib n-2, then n-4, n-6, n-8, etc.

The final layout of studs to the right of the maximum moment position (starting from end 2) is 2,1,2,1,2,1,2,1,2,1,2,1,2,1,2,1,2,1,2,1,2,1,2,1,2,1,2 = 35.

Thus the total number of studs = 47 and these are laid out (near) uniform.

For comparison a 'non-uniform' layout would give 11 studs to the left of the maximum moment position and 12 to the right (per alternate ribs) = 23 in total. For the 'uniform' setting, 1 stud per rib would be inadequate and so 2 per rib would be found to give 62 in total.

Auto-layout for parallel decks

For parallel decks, the Auto-layout again provides Uniform and Non-uniform layout options, but the way these work is slightly different.

Uniform

The Uniform option forces placement at a uniform spacing along the whole length of the beam. The spacing adopted will be within the limits you set for Minimum group spacing distance and Maximum group spacing distance. If the point of maximum moment does not occur at mid span, the resulting layout will still be symmetric.



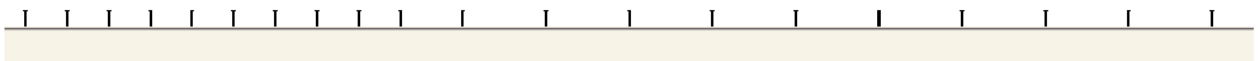
The number of studs in each group will be the same along the whole length of the beam, this number can be controlled by adjusting the limits you set on the Stud strength page.

Non-uniform

If optimization has been checked (see **Optimize shear interaction** in the preceding **Stud Strength** topic) studs are placed at a suitable spacing in order to achieve sufficient interaction without falling below the minimum allowed by the code.

If optimization has not been checked, studs are placed at a suitable spacing in order to achieve 100% interaction.

If the point of maximum moment does not occur at mid span, the resulting non-uniform layout can be asymmetric as shown below.



NOTE For US design only, where the Auto-layout option is set to Non-uniform and the beam supports more than five secondary beams, the layout reverts to uniform, producing a more practical design solution.

For both Uniform and Non-uniform layouts, if the minimum level of interaction cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

Manual Stud Layout

You may prefer to manually define/adjust the group spacing along the beam. This can be achieved by unchecking Auto layout.

NOTE If you specify the stud spacing manually, then it is most important to note:

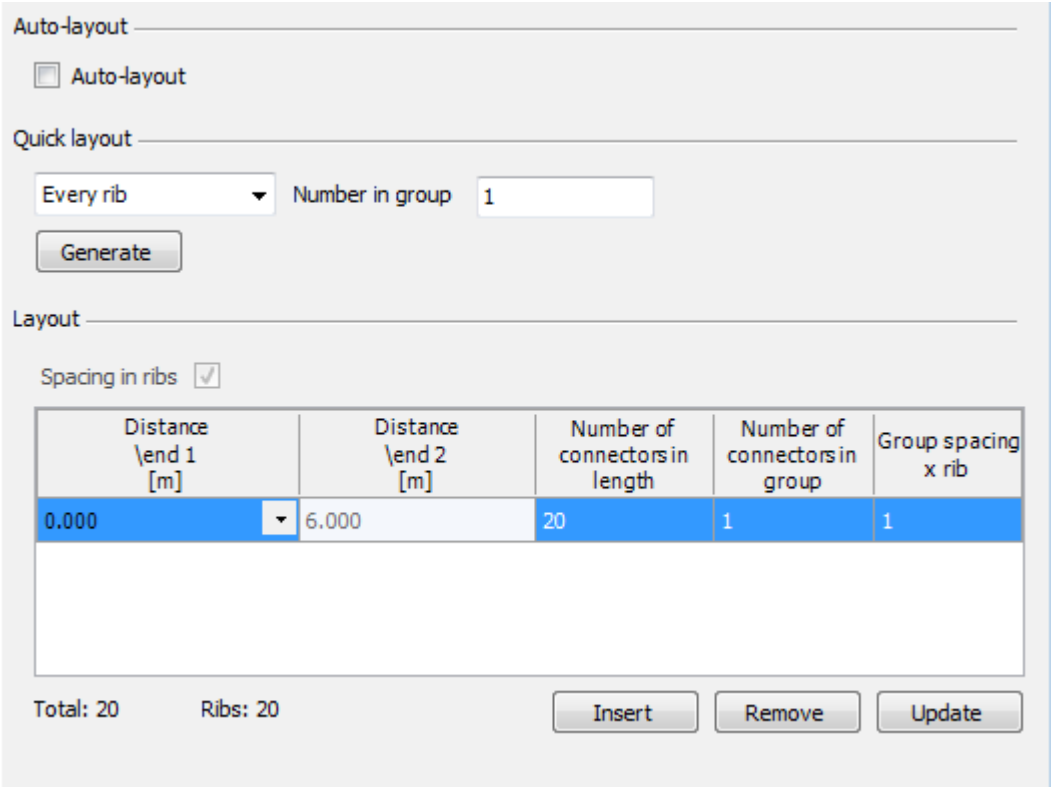
- the resulting design may not be the optimal design possible for the beam, or

- composite design may not be possible for the stud spacing which you have specified.

To generate groups of studs at regular intervals along the whole beam use the Quick layout facility. Alternatively, if you require to explicitly define the stud layout to be adopted for discrete lengths along the beam use the Layout table.

Manual layout for Perpendicular decks

For perpendicular decks, the dialog for manual layouts is as shown:

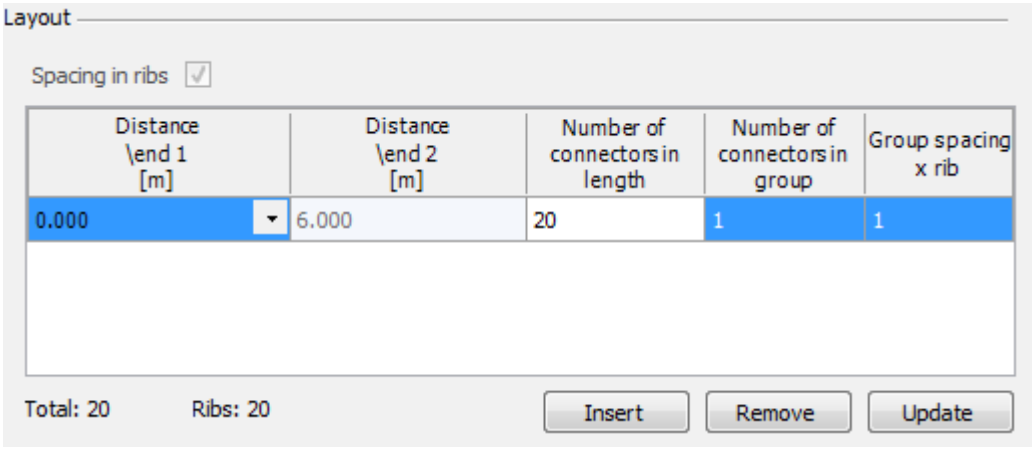


To use Quick layout, proceed in one of two ways:

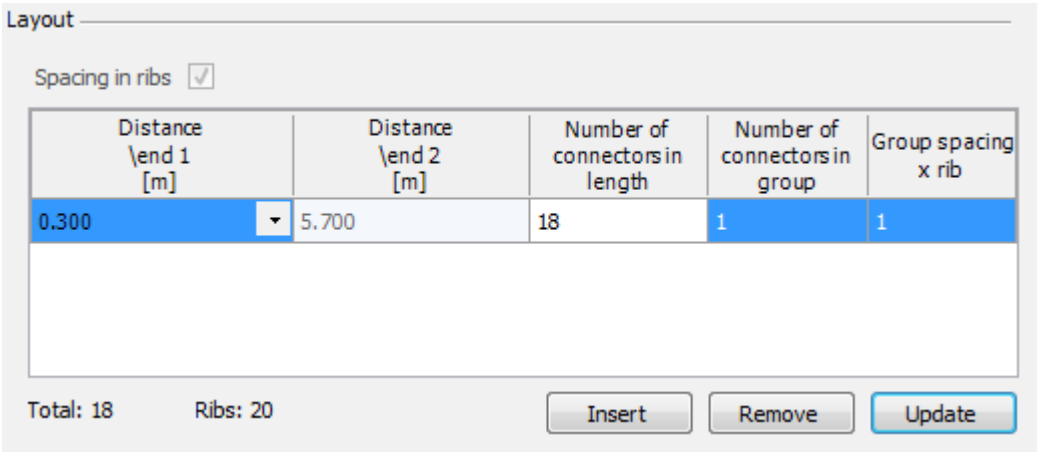
- Choose to position groups in either every rib, or alternate ribs, then specify the number of studs required in the group and click Generate.
- Alternatively: specify the total number of studs, then when you generate, if the number specified is greater than the number of ribs, one will be placed in every rib and the remainder will be 'doubled-up' in the ribs at each end starting from the supports. Similarly if the number specified is less than the number of ribs, but greater than the number of alternate ribs, one will be placed in every alternate rib and the remainder will be placed in the empty ribs. Limits of 600mm or 4 x overall slab depth, (whichever is less), are considered.

To use the Layout table:

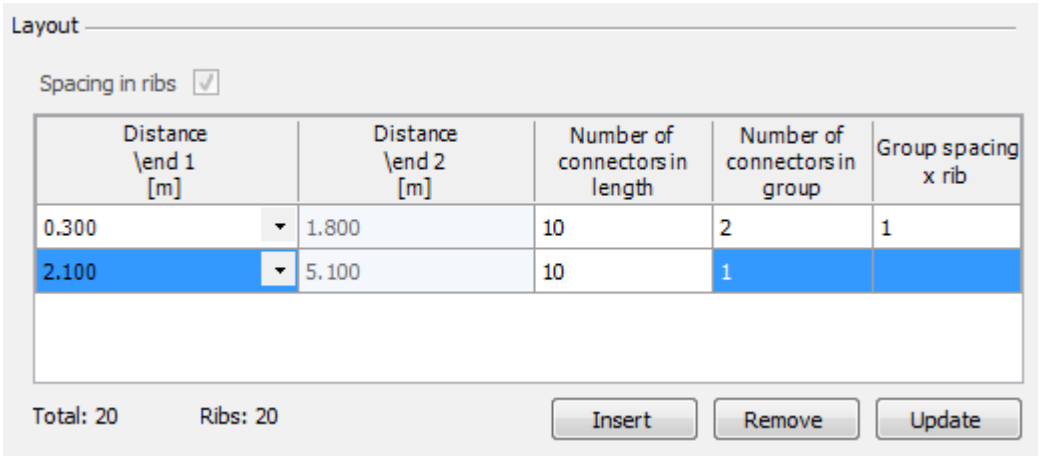
- For each segment you should define the following parameters: No. of connectors in length and No. of connectors in group; Group spacing x rib.



Your input for these parameters is used to automatically determine Distance end 2 - this latter parameter cannot be adjusted directly, hence it is dimmed.

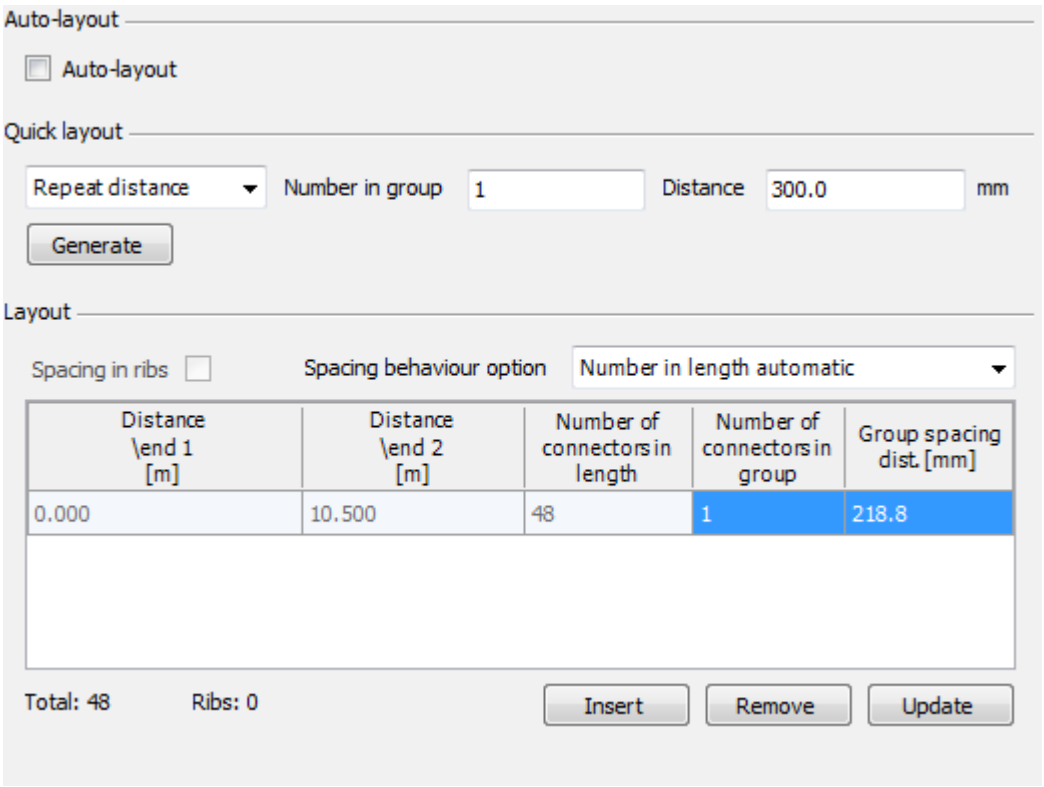


- If required click Insert to divide the beam into additional segments. (Similarly Delete will remove segments). You can then specify a different stud layout for each segment.
- We would advise that having entered No. of studs in length, group and spacing and ignoring Distance ends 1 and 2 you click Update, this will automatically fill in the missing fields.



Manual layout for Parallel decks

For parallel decks, the dialog for manual layouts is as shown:



To use Quick layout, proceed in one of two ways:

- Choose to position groups at a set repeat distance, then specify the number of studs required in the group and click Generate.

- Alternatively: specify the total number of studs, then click Generate - the program calculates the repeat distance automatically, subject to the code limits.

To use the Layout table:

- The preferred method is to choose the option Spacing distance automatic, in which case you can adjust the No. of connectors in length and No. of connectors in group. Alternatively you could choose the option Number in length automatic and then adjust No. of connectors in group and Group spacing dist.
- If required click Insert to divide the beam into additional segments. (Similarly Delete will remove segments). You can then specify Distance end 1 for each new segment and it's own stud layout.

Transverse reinforcement

This reinforcement is be provided specifically to resist longitudinal shear.

Since the profile metal decking can be perpendicular, parallel or at any other angle to the supporting beam the following assumptions have been made:

- if you use single bars they are always assumed to be at 90° to the span of the beam,
- if you use mesh then it is assumed to be laid so that the main bars are at 90° to the span of the beam.

The reinforcement you specify is assumed to be placed at a position in the depth of the slab where it is able to contribute to the longitudinal shear resistance

Automatic transverse shear reinforcement design

It is possible to automatically design the amount of transverse shear reinforcement for each beam. This is achieved in Tekla Structural Designer by checking the Auto-select option on the Transverse reinforcement tab of the Composite Beam Properties.

NOTE The Auto-select option for designing transverse shear reinforcement is only available when the beam is in auto-design mode.

If you are checking a beam, then you must specify the transverse shear reinforcement that you will provide, and then check out this arrangement.

The auto-selected bars can be tied into the stud group spacing by checking the Bar spacing as a multiple of stud spacing option. Alternatively, the spacing can be controlled directly by the user.

Bar spacing as a multiple of stud spacing

When the option Bar spacing as a multiple of stud spacing is checked, the Transverse Reinforcement tab provides the user with controls on the bar size and the multiples of stud spacing.

These can be used to achieve a selection of say, 12mm diameter bars at 2 times the stud spacing, with a slightly greater area than a less preferable 16mm diameter bars at 4 times the stud spacing.

Controlling the bar spacing directly

When the option Bar spacing as a multiple of stud spacing is not checked, the Transverse Reinforcement tab provides the user with direct control on the bar size and the bar spacing.

Instability factor

Long members in a model that have axial force in them can be unstable during second-order analysis because their individual elastic critical buckling load factor is lower than the elastic critical buckling load factor of the building as a whole and is less than 1.0.

However, often such members, for example the rafters in a portal frame, are stable in design because, for example, there are many smaller members or sheeting, that restrain the member in reality. They fail in the analysis because it is too resource intensive to model all the individual restraining members in the model which would also add unwanted clutter.

To prevent or to reduce the incidence of such failures during the analysis a multiplier can be applied to the minor axis inertia of these members which caters for the effect of the restraining members.

This multiplier is defined by selecting **Prevent out of plane instability** in the beam properties and then entering a suitable value in the **Instability factor** field.

When activated, the minor axis inertia of the member is multiplied by the factor, thus increasing its minor axis (theoretical elastic) buckling capacity. This only occurs in the **2nd order analysis** solver model. The default value for the factor is 20. This is just a sensible initial value, not a 'magic' number or one that will work in all circumstances. This behaviour also applies to the pre-size sections assigned during steel auto-design using 2nd order analysis. Hence the setting can be used to assist when buckling is occurring in the 2nd order analysis that is part of steel auto-design.

Since the setting adds ‘artificial’ stiffness to the model, the engineer should take care to ensure that it does not inappropriately increase the lateral stiffness of the structure as a whole leading to inaccurate results.

NOTE This multiplier is applied to prevent unwanted behaviour in the analysis model. While the analysis results may be affected by this adjustment, the increased minor axis inertia is NOT used in design checks.

Web openings

You cannot currently automatically design sections with web openings, you must perform the design first to get a section size, and then add and check the openings. This gives you complete control of the design process, since you can add appropriate and cost effective levels of stiffening if required, or can choose a different beam with a stronger web in order to reduce or remove any stiffening requirement.

When openings are added they can be defined as rectangular or circular and can be stiffened on one, or on both sides.

Openings cannot be defined from the **Properties** window, they can only be defined from the **Properties** dialog box, (by right-clicking on the member and selecting **Edit...**)

For guidance in relation to a specific regional code, see:

-
-
-

Precast concrete planks (Eurocode only)

NOTE The design of composite beams with precast concrete planks is only available for Eurocodes. It is not currently supported for other regional codes.

General limitations and assumptions

The following limitations and assumptions apply to the use of precast concrete planks:

Cross-section classification is restricted to Classes 1 & 2.

As per normal composite beams there is no requirement to check for transverse force as it is assumed there are no loads or support conditions that would necessitate this.

A balanced condition is assumed during the construction stage and the top flange of the beam is treated as laterally restrained in construction. This condition should be evaluated against the particular application, if it is not suitable then it should be cleared.

Both hollow core units and solid planks are assumed to act compositely only with perpendicular secondary beams and not primary beams parallel to the span of the PC units. Beams neither parallel nor perpendicular to the PC slab are termed angled and are also designed non-compositely.

Precast unit

Design of the precast units themselves is not carried out. It is assumed that the application and loading conditions of the particular precast unit is justified before design of the composite beam is carried out.

The ability to model slab openings is not restricted. The effect an opening has on the behaviour of the precast plank is however, not taken into account and the engineer should verify this to be safe.

P401 restricts the design of composite beams with precast concrete units to the following:

- Hollow core units with circular or circular elongated openings along their length (150mm – 260mm deep). It is assumed all hollow core units modelled will have circular or circular elongated cores. Cores with other cross-sectional geometries may need additional design and verification, this is beyond the scope of SCI P401
- Solid precast planks (75mm – 100mm deep)
- Downstand beams

Deeper units can be chosen than the sizes stated above, however design will not be carried out.

It is assumed that the concrete infill does not contribute to the overall weight of the slab.

If a solid slab is chosen, the contribution of the precast slab is ignored in resistance and stiffness calculations.

If a hollow core unit is chosen, the contribution of the concrete topping is ignored in resistance and stiffness calculations.

Steel Beam

The following applies to steel beam sections:

- Minimum flange width
    - Internal beam - 220mm for shop-welded and 235 for site welded shear connectors.
    - Edge beam – 2 * (6 * stud diameter)

  These recommendations can be reduced by decreasing the bearing - special provisions must be made after consultation with both the precast manufacturer and the steelwork providers*
- Web openings are ignored in design*
- No significant point loads are applied to the composite beam*
- Beam must not behave as a cantilever

- For solid precast units only the topping is to be included as the joints between the units may not be in good contact
- For hollow core units only the precast plank will be taken into account during design
- A warning is issued in this case and subsequent design is carried out assuming that the engineer has justified the particular condition as safe.

Bearing

A default bearing of 75mm minimum is used. This can be reduced but the engineer must consult the precast manufacturer and steelwork provider. The minimum flange width is therefore also reduced.

Concrete properties

Overall properties of the slab should be specified. It is up to the engineer to decide those that govern the overall slab (PC plank + topping). It is these properties that are used to carry out design calculations and slab self-weight.

Loading

Slab self-weight

The concrete infill in the hollow cores is not taken into account in the calculation of the overall weight of the slab.

Where either no topping or structural topping is used, both dry and wet overall self-weight is calculated from the self-weight of the precast unit plus any topping. Where a non-structural topping is used, the engineer is expected to input the ADDITIONAL self-weight of the TOPPING themselves.

Significant Point Loads

Significant point loads are beyond the scope of design in SCI P401. A warning is issued if a significant point load is present on the composite beam. Subsequent design calculations are carried out assuming the point load has no effect on the composite behaviour of the beam. The engineer must carry out additional hand calculations to justify this assumption is safe.

Shear Connectors

19mm and 22mm diameter shear studs are allowed in composite design with precast planks.

Should the engineer choose to place shear connectors in pairs, no dimensional check is carried out. It is assumed that the engineer has justified their use in pairs.

Longitudinal Shear

It is assumed the shear force is divided equally between the two sides of the beam flange.

The factors that influence the longitudinal shear capacity of your composite beam are:

- Concrete strength, slab depth and slab width – you cannot change these independently for the longitudinal shear check, since they apply equally to the entire composite beam design,
- The areas of Transverse and Other reinforcement which you provide in your beam

Transverse

Transverse reinforcement is designed to ensure VEd ≤ VRd. Additional reinforcement to that detailed in design may be required for other purposes.

Refer to SCI P401 for recommended minimum bar sizes and spacing of transverse reinforcement. In the case of a solid slab, the additional mesh is ignored in transverse reinforcement calculations as only either mesh or loose bars can be chosen. Additional mesh, however, can be applied to the slab reinforcement – see “Other”.

It is possible to increase the maximum spacing of transverse reinforcement from that shown in SCI P401 Table 3.1, however it must be noted that this is being done under the engineer’s own judgement.

EN 1992-1-1, 6.2.4 is used to determine the design resistance VRd to the longitudinal shear at the potential failure surface a-a (shown in Figure 4.7 in SCI P401). Failure surface b-b however is not checked in design.

Refer to SCI P401 for recommendations on the detailing of transverse reinforcement and minimum bar length.

Other

Any “other” slab reinforcement in the topping applied to a hollow core unit is ignored in design.

Composite Moment of Inertia

When determining the moment of inertia of a composite section with a hollow core unit, the section is taken as a solid slab (i.e. hollow cores aren’t taken into account).

For stiffness calculations the concrete below the neutral axis is considered as it will contribute some stiffness. However when carrying out resistance calculations, this concrete is ignored.

1.7 Steel column design

See also:

Steel column connection eccentricity moments (page 74)

Overview

Tekla Structural Designer allows you to analyse and design a structural steel column which can have moment or simple connections with incoming members, and which can have fixity applied at the base. The column can have incoming beams which may also be capable of providing restraint, and may have splices along its length at which the section size may vary. You are responsible for designing the splices appropriately.

In its simplest form a steel column can be a single pin ended member between construction levels that are designated as floors.

More typically it will be continuous past one or more floor levels, the whole forming one single entity typically from base to roof.

Steel columns that share moments with steel beams form part of a rigid moment resisting frame.

In all cases you are responsible for setting the effective lengths to be used appropriate to the provided restraint conditions. All defaults are set to 1.0L.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not considered.)

If working to either AISC or Eurocodes, the design will take into consideration additional moments resulting from the eccentricity of pinned beam end connections. If working to other regional codes, while the eccentricity moments are calculated, they are not used in the design.

Simple columns

A steel column can be designated as a **Simple column** in the column properties - in which case specific design rules are required.

A simple column should not have any applied loading in its length.

Simple columns are subject to axial forces and moments due to eccentricity of beam reactions.

In order to prevent end fixity moments you would have to manually pin the ends of the column.

NOTE The simple column design rules have not yet been implemented in Tekla Structural Designer: such columns are thus classed as “beyond scope” when designed.

Fabrication

The **Fabrication** type is specified in the column properties.

The steel column fabrication types that can currently be designed in Tekla Structural Designer are dependent on the design code, and also the construction type specified in the column properties - refer to the following table for details.

Construction: Non-composite column

Regional code	Rolled	Plated	Concrete filled	Concrete encased
AISC	Yes	No	No	No
EC	Yes	No	No	No
BS	Yes	No	No	No
IS	Yes	Yes	No	No
AS	Yes	Yes	No	No

Construction: Composite column

Regional code	Rolled	Plated
AISC	No	No
EC	No	No
BS	No	No
IS	No	No
AS	No	No

Composite, concrete filled, and concrete encased steel columns

Whilst composite columns, concrete filled hollow sections and concrete encased sections can be specified in Tekla Structural Designer they are not designed.

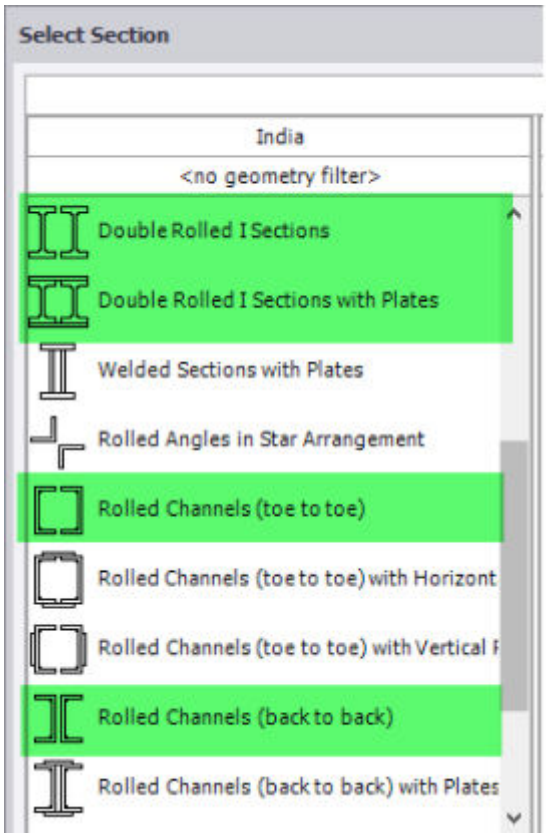
For composite columns and concrete filled hollow sections, the analysis uses the bare steel inertia and not the ‘effective composite inertia’. This is conservative as the lower stiffness of the bare section will promote more second-order effects. On the other hand for concrete encased columns Tekla Structural Designer takes into account the encasement based on the size specified by the user.

When working to the US regional code: AISC 360-16, sub-section I1.5 deals with the calculation of stiffness for concrete filled hollow sections and concrete encased sections used in the Direct Analysis Method (DAM). In accordance with C2.3, Tekla Structural Designer takes account of the standard stiffness adjustment factor of 0.8 with τb set to 1.0 given the additional notional load of 0.1% is also applied. I1.5 requires these types of member to take τb as 0.8 when considering the flexural stiffness. The user can achieve this for concrete

encased columns by adjusting the Modification Factors in the . (For composite columns and concrete filled hollow sections which both use the bare steel stiffness, no adjustment is required).

Plated columns - Indian regional code

When the Fabrication type is set to Plated, a range of steel compound sections can be designed to the Indian design code.



The section shapes supported are:

- Rolled Channels back to back
- Rolled Channels toe to toe
- Doubled rolled I sections
- Doubled rolled I sections with plates

The scope of design of these compound sections includes both beams and columns and autodesign.

NOTE Autodesign requires a **Design section order** to be specified, a new design section order can be created specifically for compound sections once these have been added to the sections database.

In general, the scope of strength checks undertaken is the same as for rolled sections with the following current limitations:

- The design is for non-composite sections only.
- Slender Sections are beyond scope.
- High Shear case with minor axis moment is beyond scope.
- Design of the lacing or battening system is beyond scope.
- Only parallel flange sections can be used.

Plated columns - Australian regional code

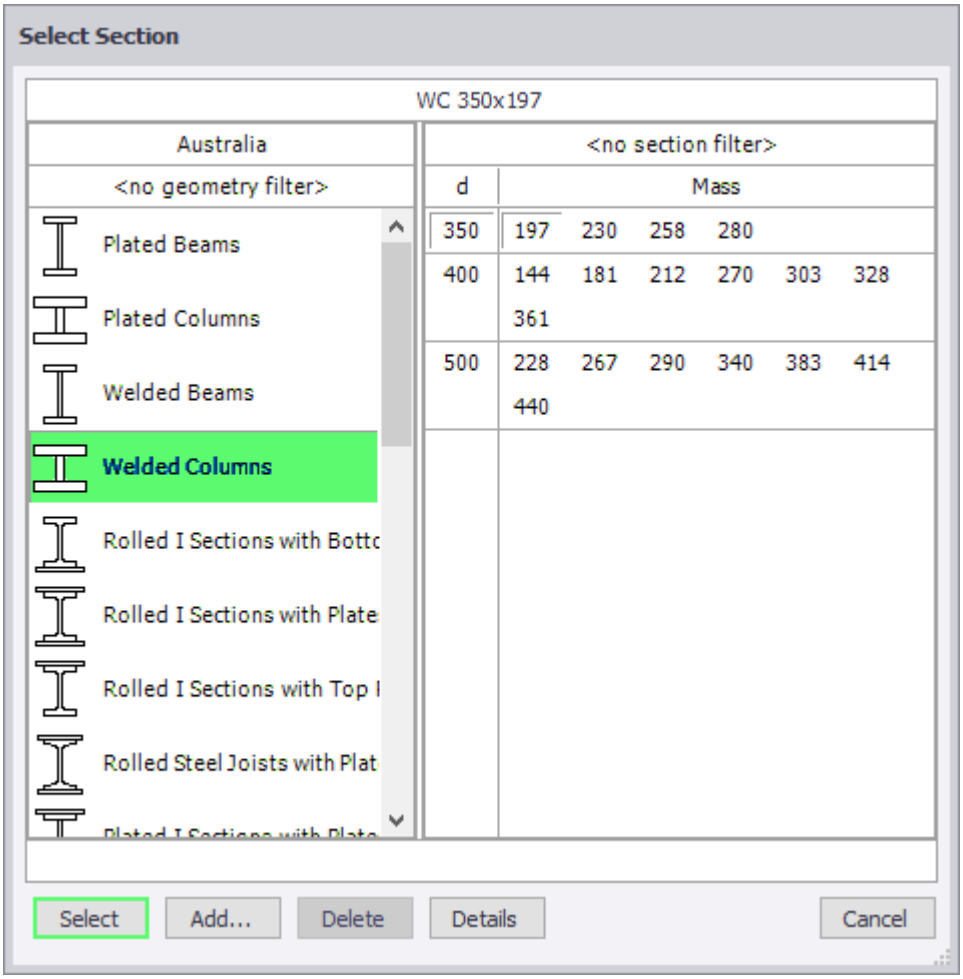
When the Fabrication type is set to Plated, only welded doubly symmetric I-section columns can be designed to the Australian code.

A pre-defined range of these is available from the **Welded Columns** page of the Select Section dialog - these can be both checked and autodesigned.

NOTE Autodesign requires a **Design section order** to be specified, the Welded Columns design section order is provided for this purpose.

While you can add to the range of welded columns in the Select Section dialog by clicking the **Add...** button, any such user-defined columns can only be analyzed, but are not designed.

Plated columns available on other pages of the Select Section dialog can be analyzed, but are not designed.



Restraints

Restraints to flexural and torsional buckling are determined from the incoming members that connect to the column. The buckling checks are based on these restraints.

Restraints are considered effective on a particular plane providing they are within ±45° to the local coordinate axis system.

In all cases Tekla Structural Designer sets the default unrestrained length factor between restraints to 1.0.

You have the control to set any unrestrained length to be continuously restrained over that length - when set in this way the relevant buckling check is not performed during the design process.

NOTE The Steel Column **Properties** window only allows you to set entire stacks as either continuously restrained or unrestrained. In order to

specify restraints between the incoming members within each stack it is necessary to open the Steel Column **Properties** dialog box instead.

TIP As an alternative to using the Steel Column **Properties** dialog box, restraint settings for steel columns can be easily reviewed and adjusted graphically via **Review View > Show/ Alter State**. For more details see: Review and modify restraints

Lateral torsional buckling (LTB):

- Members framing into either Face B or D are by default assumed to provide full LTB restraint (i.e. Face A and Face C are both set as restrained). You therefore need to consider whether or not your particular configuration of incoming members is capable of providing this level of LTB restraint. If necessary you can edit the default restraint provision by selecting the Face A override and/or Face C override checkboxes as appropriate.

  NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 26)

Compression/Strut buckling:

- Members framing into either Face A or C will by default provide restraint against major axis compression buckling. Members framing into either Face B or D will by default provide restraint against minor axis compression buckling. You can remove these default restraints if required by selecting the Major override and/or Minor override checkboxes.

Splice and splice offset

Splices can be specified by checking **Splice** under the appropriate **Stack** heading (apart from Stack 1) in the column properties and then specifying the **Splice offset** from the lower end of the stack.

Splices are allowed at floor levels only and must be placed at changes of angle between two adjacent stacks and at changes of section size or type. A validation error will result if this is not the case. The splice offset from the floor level defaults to 500mm and is considered not to be structurally significant.

You must detail the splice to resist the applied forces and moments. The detail should provide continuity of stiffness and strength. Splices given considerable offset should be designed to take account of the P-d moment (also known as moment induced by strut action) at the position as well as the forces from the analysis.

NOTE To remind you of the above requirement the following warning is issued when splice loads are exported to BIM as part of the analysis

results: *"The loads given do not include any additional moment due to member buckling."* A similar warning is also issued when splice loads are included in reports. This is an explicit requirement in the British Standards (Clause 6.1.8.2 of BS 5950-1), although as the same engineering principle applies the world over, we issue it for all regional codes.

Each lift (length between splices) of a general column can be of different section size and grade. Different section types within the same column are not allowed due to the particularly complex design routines that general columns require. You are responsible for guaranteeing that the splice detail ensures that the assumptions in the analysis model are achieved and that any difference in the size of section between lifts can be accommodated practically.

Web openings

NOTE In the current release of the program the design or checking of columns with web openings is “Beyond Scope”

You can define rectangular or circular openings and these can be stiffened on one, or on both sides.

Openings cannot be defined from the **Properties** window, they can only be defined from the **Properties** dialog box, (by right-clicking on the member and selecting **Edit...**)

Web openings can be added to a column by a 'Quick-layout' process or manually.

The 'Quick-layout' process, which is activated using the checkbox on the Web openings dialog page, adds web openings which meet certain geometric and proximity recommendations (taken from Table 2.1 of SCI Publication P355 if the Regional Code is set to Eurocode, or taken from SCI Publication P068 if the Regional Code is set to BS). The openings so created are the maximum depth spaced at the minimum centers recommended for the section size.

Web openings can also be defined manually. With Quick-layout cleared, the `Add' button adds a new line to the web openings grid to allow the geometric properties of the web opening to be defined.

Instability factor

Long members in a model that have axial force in them can be unstable during second-order analysis because their individual elastic critical buckling load

factor is lower than the elastic critical buckling load factor of the building as a whole and is less than 1.0.

However, often such members, for example the rafters in a portal frame, are stable in design because, for example, there are many smaller members or sheeting, that restrain the member in reality. They fail in the analysis because it is too resource intensive to model all the individual restraining members in the model which would also add unwanted clutter.

To prevent or to reduce the incidence of such failures during the analysis a multiplier can be applied to the minor axis inertia of these members which caters for the effect of the restraining members.

This multiplier is defined by selecting **Prevent out of plane instability** in the column properties and then entering a suitable value in the Instability factor field.

When activated, the minor axis inertia of the member is multiplied by the factor, thus increasing its minor axis (theoretical elastic) buckling capacity. This only occurs in the **2nd order analysis** solver model. The default value for the factor is 20. This is just a sensible initial value, not a 'magic' number or one that will work in all circumstances. This behavior also applies to the pre-size sections assigned during steel auto-design using 2nd order analysis. Hence the setting can be used to assist when buckling is occurring in the 2nd order analysis that is part of steel auto-design.

Since the setting adds ‘artificial’ stiffness to the model, the engineer should take care to ensure that it does not inappropriately increase the lateral stiffness of the structure as a whole leading to inaccurate results.

NOTE This multiplier is applied to prevent unwanted behavior in the analysis model. While the analysis results may be affected by this adjustment, the increased minor axis inertia is NOT used in design checks.

Steel column connection eccentricity moments

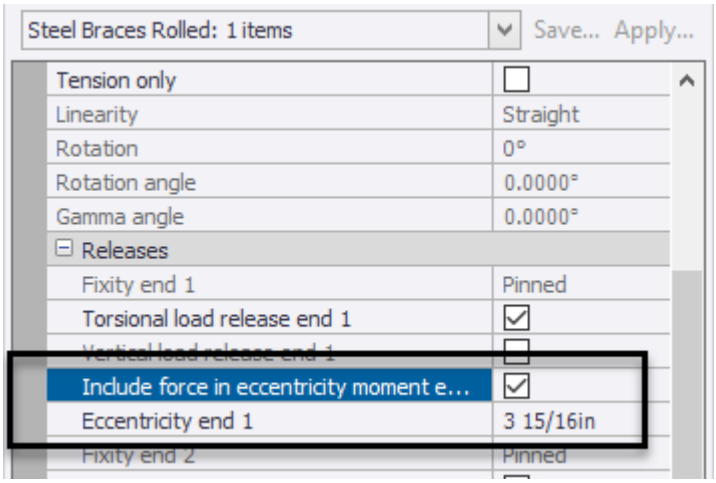
Overview

Nominal eccentricity moments that arise from beam end reactions are considered in Tekla Structural Designer's steel column design.

These moments do not come directly from the global analysis but instead are calculated at the ‘load analysis’ post-processing stage as follows:

- At each level the eccentricity of each connection is taken as half the depth of the supporting column, plus an additional user defined offset from the column face.
- At each level the pinned beam end reactions connecting to the column at each face are determined.

- If braces also connect to the same face, the force in the brace will also be taken into consideration if the "Include force in eccentricity moment" brace release property is checked for the appropriate end of the brace.



- Taking the beam end reactions (and brace forces if included) on opposite faces multiplied by their connection eccentricities, resultant eccentricity moments are determined.
- These moments are then distributed above and below the level based on the column stiffnesses.

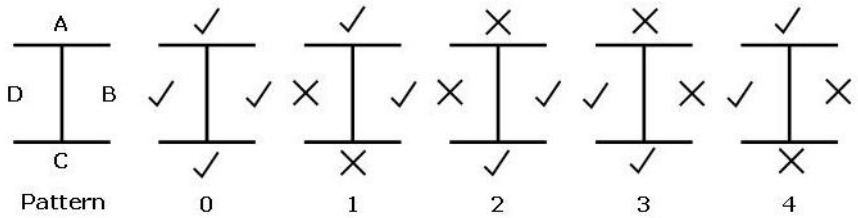
NOTE The eccentricity moments are typically assumed not to be transferred beyond the level at which they are applied.

Patterning of eccentricity moments

The eccentricity moments resulting from live loads can be patterned if required to account for the likelihood that the load is not present on all spans simultaneously.

When eccentricity moment patterning is enabled you must then indicate which of the live cases are to be patterned, (you may for example decide not to pattern storage loads.)

For those live cases with patterning enabled, five patterns are considered. These are:



Pattern 0 is for the full live load at all positions i.e. no patterning - this gives the maximum axial force in any one stack with (usually) lower eccentricity moment.

Patterns 1 to 4 are ‘true’ patterns switching live load ‘on’ and ‘off at each pair of positions around the column in order to generate the maximum live eccentricity moments about the major and minor axes of the column.

NOTE The same pattern is applied at the top and bottom of the stack, so for example it is not possible to have P1 at the top and P4 at the bottom.

Design

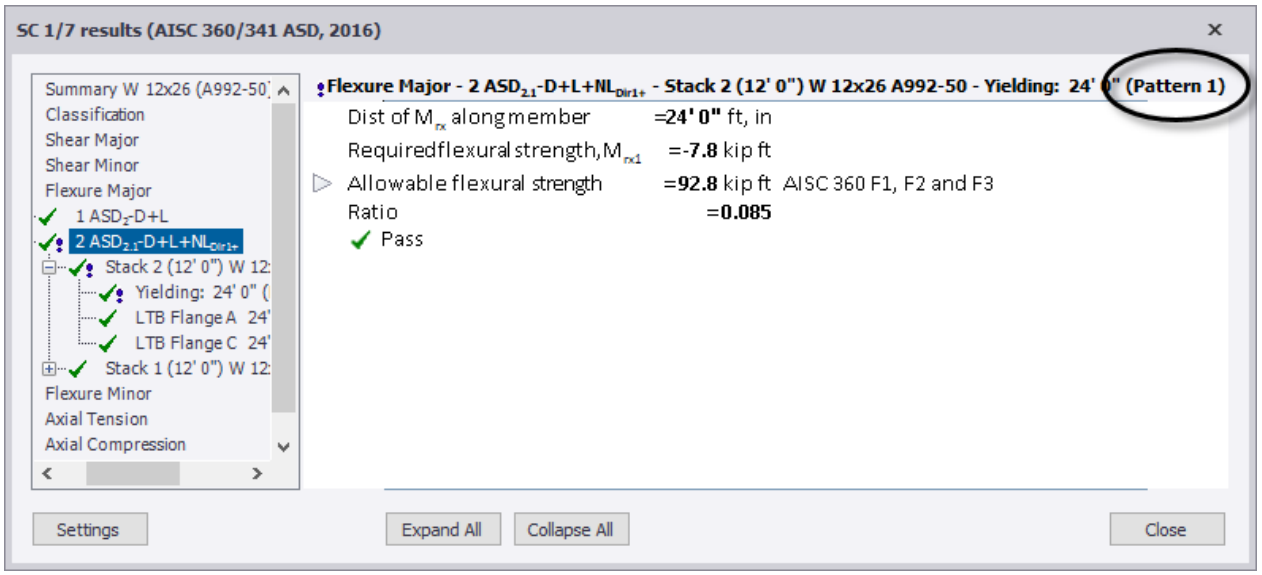
NOTE Patterned eccentricity moments are only considered in the design for the **AISC** or **Eurocode** regional codes. If working to other regional codes, while the patterned eccentricity moments are calculated, only the fully loaded pattern (P0) is used in the design.

In general, eccentricity moments are only added to the 'real' moments at the ends of each stack and are only added if they make the design worse.

If you have elected to pattern live eccentricity moments these are considered in conjunction with the eccentricity moments from other types of load, and with the 'real' moments.

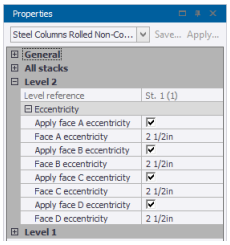
- As the eccentricity moments are considered localised to each floor the full axial force from other floors is maintained. The axial force at the level under consideration will be slightly reduced with patterning enabled as the live floor loading will not be present on all sides simultaneously.
- Since it is not known whether a reduced axial force with more eccentricity moment is a worse case than full axial and a lower (or even zero, in the balanced case) eccentricity moment, the design loops through all patterns in order to consider each eventuality.
- The patterned eccentricity moments are considered in all design checks apart from 'Shear', (which is unaffected).
- To keep the design details to a manageable level, results for every pattern are not listed; the pattern which produces the governing design forces is

listed in the check combination and location tree and details heading, (as shown below).



Define connection eccentricity values

The eccentricities at each level are defined in the column properties and a different eccentricity can be applied to each face.



As long as the option to apply eccentricity at a face is selected, the total eccentricity at that face is taken as half the dimension of the supporting column, plus the additional eccentricity from the face as specified in the **Properties** window shown above.

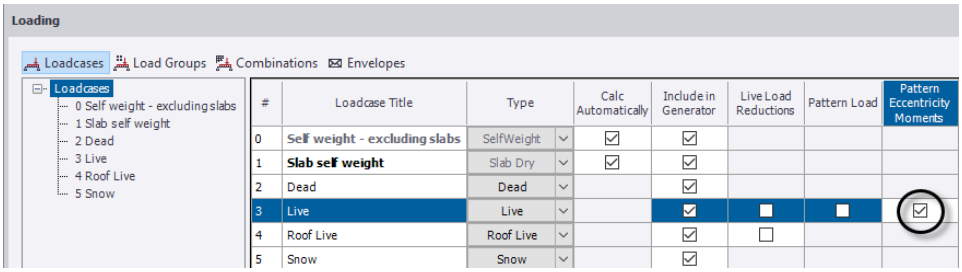
If you clear the option to apply eccentricity at a face the end reaction on that face is applied axially.

NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 26)

Pattern eccentricity moments for live loadcases

Patterning can be switched on for specific live loadcases in a two-step process as follows:

1. From the **Home** ribbon:
   a. Click **Model Settings > Loading > General**
   b. Select **Use patterning of eccentricity moments for steel columns**
   c. Click **OK**
2. From the **Loadcases** page of the **Loading dialog**:
   a. Select a live loadcase that you want to be patterned
   b. Select **Pattern Eccentricity Moments for Steel Columns**
   c. When patterning has been selected for each of the required loadcases, click **OK**

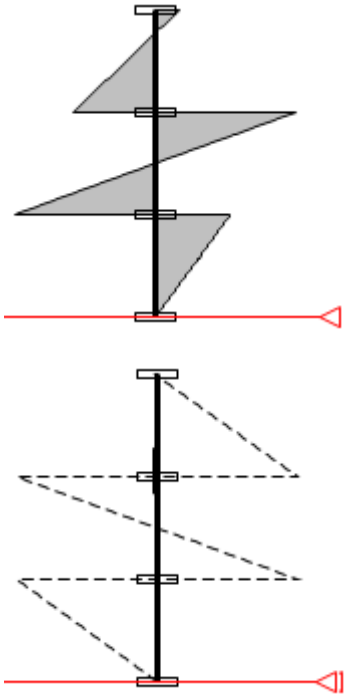


Review connection eccentricity moments

Because eccentricity moments do not come directly from the global analysis they cannot be displayed graphically in a **Results View**, they can only be displayed on a column by column basis by opening a Load Analysis View.

With a **Load Analysis View** open and the required loadcase or combination selected in the **Loading** list, you then select the **Major**, or **Minor** direction in the **Loading Analysis** ribbon.

The 'real' moments are displayed as a shaded diagram using solid lines, the eccentricity moments as an unshaded using dashed lines:

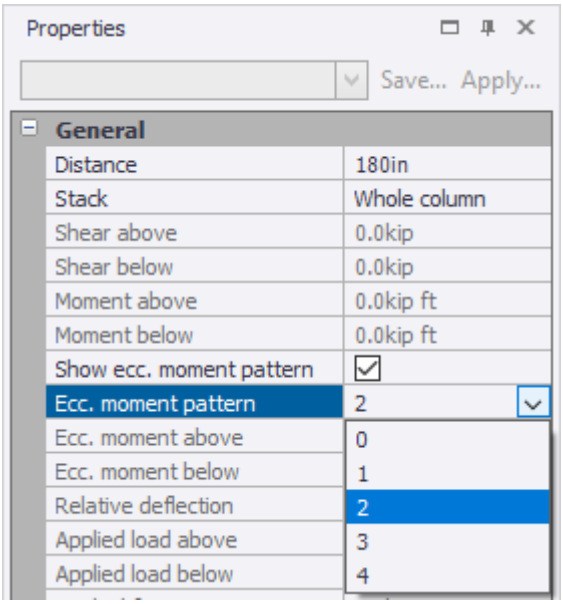


The red marker line can be set to a specified distance in the **Properties** window to allow the real and ecc. moment values above and below the line to be displayed.

Displaying patterned eccentricity moments

When you select a patterned live loadcase a **Show ecc. moment pattern** box will become available in the **Properties** window.

After selecting **Show ecc. moment pattern** you can then click **Ecc. moment pattern** in order to select the pattern to display from the droplist.



1.8 Column base plate design

The definition and check of column base plates is an intrinsic part of Tekla Structural Designer - all data associated with a particular column base plate is held within the model. Only simple column base design checks under Eurocode and US regional code are supported in the current release.

Click the following links to find out more:

- Column base plate design workflow (page 80)

When working to British Standards, while base plates cannot be checked directly in Tekla Structural Designer, they can be transferred to Tekla Connection Designer for design. For details, see: Create and design other connections (page 115).

Related information

Related video

Simple column base plate workflow and design

Column base plate design workflow

NOTE To find out more about the theory, assumptions and checks performed for the regional code being worked to, see: or .

Related information

Related video

Simple column base plate workflow and design

Data sources

Whilst all column base plate data is held in the model, the source of such data is several fold. This includes:

- Default data - when the column base plates within the model are set up by Tekla Structural Designer, intelligent defaults are used that can establish a part or full solution to the configuration.
- Derived data - the model already holds such items as the section size and grade of the column and concrete foundation, plus the design forces.
- Property Sets - certain data can be set to be used to modify existing properties of the column base plates e.g. base plate steel grade, bolt size etc. For more information, see:

NOTE If a concrete base is already modeled below a base plate then the concrete base properties Length, Width, Depth, and Concrete (grade) will not be modified when applying a base plate property set.

- Added data - any individual column base plate can be edited to improve or add to the configuration e.g. additional bolts.

Generate base plates

When a steel column is created or copied a base plate is automatically generated at the lower end of the column if a support is required.

Base plates can also be created by clicking the (Add) Base Plate button on the Design ribbon and then selecting a steel column in the model. Note that moving a steel column that already has a base plate attached will automatically move the attached base plate with it.

Design columns

Base plates have the capability of *automatically sizing* when the column size changes.

This **Autosize** option, which is on as default, will update the plate size and the bolt layout when the column size changes to avoid an invalid base plate configuration. This is not to be confused with automatic design of the base plate which is beyond the scope of Tekla Structural Designer.

Edit base plate properties

Once the associated columns are sized it's possible to turn off **Autosize** and start to finalize the base plate configuration.

This is easily done based on the column size by navigating to **Structure Tree > Members > Base Plates** and selecting the column size.

You can then adjust the column base plate properties in the **Properties** window.

Alternatively, and also for more complex bolt/rod layouts, you can right click an individual base plate and select Edit from the context menu, which opens an Edit dialog for all of the base plate properties.

You can see whether your base plate configuration looks sensible by display in the Structure 3D view.

To copy, move or delete base plates

- Base plates can be moved/copied/deleted independently of the column they are associated to as long as the target location is the bottom of a valid column.
- Base plates can be moved/copied/deleted with the associated column.
    - By simply selecting the column to move/copy/delete the base plate, as default, will be selected too.
    - You can chose to only move/copy/delete the column by de-selecting the base plate from the usual dialog.

Check base plates

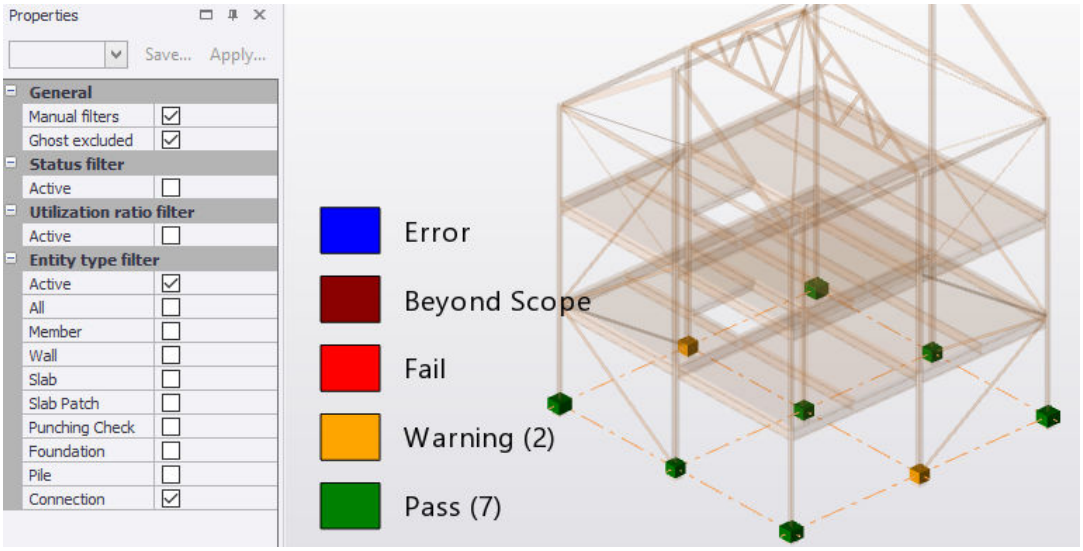
Having established and rationalized the base plate configurations it is now logical to carry out a design check.

You can check an individual base plate, or a selection of base plates and adjust them as you proceed or, once you are content with the layout, you can check them all in a single operation:

- To check an individual base plate - select it in the model (or in the Structure Tree), right-click to display the context menu and select **Check Base Plate**
- To check a selection of base plates - navigate to the **Structure Tree > Members > Base Plates**, select the column size to check, right-click and select **Check Base Plates**
- To check all base plates in the model - go to the **Design** tab and select **Check Base Plates**

Review check status and utilization

You can then use Review View to visually check which column base plates have passed and which have not, and their associated utilization ratios.



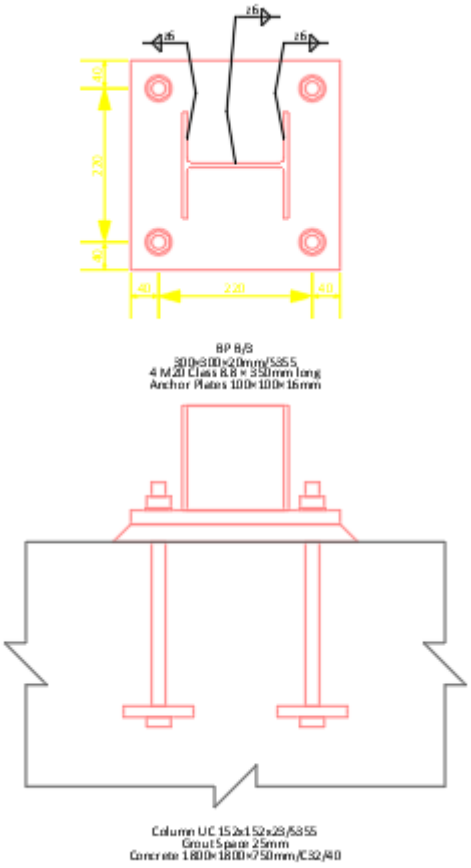
View results and print calculations

Results of your column base plate checks can be viewed on the screen by right clicking on a base plate and selecting **Check Base Plate** from the context menu.

When you are ready to create a report:

- if you require the input, diagrams, and design results you should incorporate the **Base Plates** chapter,
- if you only require a summary of base plate design check calculations you should include the **Base Plate Design Summary** chapter. (The default includes this along with design calculations for other member types in the model).
- if you require a material list containing base plate components, for example rods (US), or bolts (Eurocode), and anchor plates, these are listed in the Material Listing report, as well as the plates themselves, but not the welds.

Create drawings



Base plate detail drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

1.9 Steel brace design

Steel brace overview

Tekla Structural Designer allows you to analyze and design a steel member with pinned end connections for axial compression and tension.

Steel braces can be specified as rolled I-sections, C-sections, T-sections, rectangular, square and circular hollow sections, angles, double angles, and flat sections.

NOTE If working to the US head code, a brace specified as a flat section can only be analyzed, but design is beyond scope.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

Applied loading

The following points should be noted:

- Loads for the brace are derived from the building model.
- Element loads cannot be applied directly to the brace itself.
- Live and Roof Live load reductions are not applied.
- Moments due to self weight loading are ignored.

Design Forces

The design forces for strength checks are obtained from an analysis of the entire structure. Braces can be subject to axial compression or tension, but will not be subject to major and minor axis bending.

Input method for A and V Braces

A and V Braces should be modeled using special tools which can be found on the 'Steel Brace' drop list in the 'Steel' section on the 'Model' tab.

Although it is also possible to model the exact same brace arrangement using individual elements created using the simple 'Steel Brace' command, it is important to note that whilst the Notional Loads \ EHFs (Equivalent Horizontal Forces) calculated for models built using the A or V Brace tools are correct, this is not the case when the A or V braces are built up out of individual brace members. In this latter case, elements of the vertical loads that are supported by the bracing system are 'lost' and are not included in the Notional Load \ EHF calculations with the result that the calculated Notional Loads \ EHFs are not correct.

Steel brace in compression

Effective length factors are defined for each axis of buckling.

- Effective length factor y-y
- Effective length factor z-z

NOTE If designing to British Standards, see Steel brace design to BS 5950 - Axial compression

Steel brace in tension

The net area of the section is required for tension checks. This can be specified either as:

- Percentage value
- Effective net area

NOTE If designing to British Standards, see Steel brace design to BS 5950 - Axial tension

1.10 Steel joist design

Steel joist design overview

Steel joists, (or bar joists), are a specific type of members used in the United States. They are simply supported secondary members that do not support any other members - they only support loaded areas, i.e. 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.

In Tekla Structural Designer steel joist design is performed in accordance with the 44th Edition of the Steel Joist Institute (SJI) specification, which uses a similar approach to that embodied in AISC 360-05/10/16 LRFD and ASD.

Standard types

Steel joists are constrained to standard types specified by the Steel Joist Institute. They are standardized in terms of span, depth and load carrying capacity. There are four standard types of steel joist available in Tekla Structural Designer.

- K series joists - parallel chord steel joists (2 variations; rod or angle webs) - depths 8" to 30" with spans up to 60ft.
    - Including 2.5 K series joist substitutes - a depth of 2.5in, intended to be used for spans up to 10ft.
- KCS series joists - K series adapted and specially designed for constant moment/shear along length (position of point loads become irrelevant).
- LH series joists - long span joists - depths 18" to 48" for clear spans up to 96ft.
- DLH series joists - deep long span joists - depths 52" to 120" for clear spans up to 240ft.

Special Joists

"SP" suffixes can be added to K, LH and DLH Series joists. Special Joists can handle 'non-uniform' loading situations. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Joist Girders

These are provided as an option to support steel joists. They will attract loads and participate in the 3D structural analysis, but they cannot be checked or designed.

Assumptions and limitations

The following assumptions and limitations currently apply to the design of steel joists in Tekla Structural Designer:

- All steel joists are considered as simply supported members.
- Steel joists cannot be released axially.
- Cantilevered joists cannot be defined.
- Joist Girders are able to support other joists and brace members, other joists are only able to support brace members.
- Joist inertia and area values are taken directly from the Steel Joist Institute tables, with the exception of Joist Girders for which you must provide the relevant data.
- For all joist types. any resulting load, other then those in the major axis, that exceeds the user specified Ignore Forces Below setting on Design Options is reported as a Warning in the results viewer along with the type and value of force detected.
- Design is currently beyond scope for SP joists and Joist Girders.
- For all steel joists, it is assumed that the top chord is sufficiently braced against lateral torsional buckling.
- There is no restriction on the minimum span for which a joist can be defined even though due to their open web nature joists can be almost impossible to fabricate for very small spans. The user should check the suitability of using such a joist in these situations.
- For steel joists which support a generic concrete slab, it is assumed that the minimum concrete slab thickness of 2 inches (50mm) is present. (SJI Steel Joist Specification 5.9.2). This is not checked by Tekla Structural Designer.
- In design, the user is expected to refer to the bridging requirements in the SJI Specification and decide the appropriate details for the relevant scenario. This is not checked by Tekla Structural Designer.
- ‘Non uniform’ loads are accommodated by KCS joists. E.g. parapet snow drift load, partition walls. If no KCS joists can be found then SP joists can be used for these loads but these are not designed/checked.

- Top chord extensions used as eaves and awnings are not designed.
- Camber of the joists is not shown in the graphics nor handled by Tekla Structural Designer.
- The design and specification of the joist seats (regular or sloping) is not handled by Tekla Structural Designer.
- Double joist configurations cannot be defined in Tekla Structural Designer.
- The design does not consider the minimum bearing requirements for K, KCS, LH or DLH joists – these are the responsibility of the designer/ engineer.
- Customisation of KCS joists to accommodate any applied concentrated loads is not considered byTekla Structural Designer.
- Loadings for accessories to the joist are not included in the standard load tables. An allowance for this should be included by the designer in the loading of the model.
- Steel joists are not designed for composite action and when supported by conventional composite a validation warning is issued. Composite Joists, CJ Series, cannot be defined in Tekla Structural Designer.
- Moment connections at steel joist ends are not allowed.
- Duct openings are stated in the standard Steel Joist Tables for standard panel sizes. Actual spacing and layout of the duct arrangements are beyond the scope of Tekla Structural Designer.
- Fire resistance requirements are not designed nor checked in Tekla Structural Designer, the designer must ensure suitable compliance.
- There is no design for net uplift currently although uplift can exist in a combination provided it is overcome by other gravity loads. The designer should ensure that any uplift due to Wind loads is accurate as the current version of the Wind Wizard does not determine wind forces for ‘Components and Cladding’.
- Sloping joists are permitted providing the loading is normal to the joist and the span will be taken as the sloped length. However, joists with sloping top chords are not allowed.
- SJI allows grades other than 50 ksi [345 MPa] to be used but the Safe Load Table values are based on 50 ksi [345 MPa]. Tekla Structural Designer **defaults to Grade A992 – 50**, it is the user's responsibility to check the suitability of this or any other grade they wish to use. No adjustment is made for higher or lower grades and this is entirely the designer’s call.

Loading

The loads on the joist are from 'one-way' load decomposition. The joist is analyzed as a pin ended beam, only loads in the plane of the web are considered.

Joist Girders should only be used to support steel joists, and should therefore have regularly spaced point loads of similar magnitude along their length. As they are not designed or checked various types of loading could be present, so the user should verify that they have been used appropriately.

Steel Joists are essentially used to support full length uniformly distributed loads, however it is acceptable for the joist to be designed to support other load configurations. This requires the loading pattern on the joist to be assessed and classified into one of the following loading types: Uniform (or near uniform), Equivalent Uniform, or Non-Uniform.

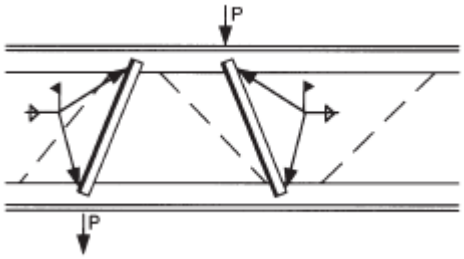
The classification of loading type is made in accordance with the following methods:

Loading Type	Method
Uniform	• No concentrated (point) loads are allowed. • Loads must be (member) Full UDL, UDL, VDL or Trapezoidal Load acting in the Major or Global Z directions. • Loads must be applied over the full length of the beam. • A percentage load tolerance is calculated and provided this is less than the **uniform load tolerance** specified in Design Settings, then the load combination is classified as 'Uniform'.
Equivalent Uniform	• Determine the position of the point of zero shear relative to the centre span point of the joist. • If the point of zero shear is located outside the **maximum eccentricity of zero shear** limit specified in Design Settings, then the procedure is ended and the load combination is classified as 'Non-uniform'. • If the point of zero shear is located within the maximum eccentricity of zero shear limit then an 'Equivalent Uniform' load is established. • The equivalent uniform load is then used to calculate maximum shear force, bending moment and deflection. The percentage variation of these values to the actual values from the beam analysis is then calculated and compared with the **equivalent load tolerance** limit specified in Design Settings.

Loading Type	Method
Non-Uniform	• All loads not qualifying as 'Uniform' or 'Equivalent Uniform' to the above methods are considered as 'Non-uniform'. **NOTE** It is possible for an individual loadcase e.g. Live to be classified as 'Non-uniform' which when combined with other loads becomes 'Equivalent uniform'. Thus, when designing for strength (comparing with the 'black value' in the SJI Tables) the loading is valid whereas when checking the deflection (comparing with the 'red value' in the SJI tables) the service design could be invalid (Fail).

Concentrated Loads

All joists supporting concentrated loads require special treatment by the manufacturer/fabricator even if the loading configuration can be configured as 'Uniform' or 'Equivalent Uniform' since the joist design usually presumes that all concentrated loads are applied at panel points. It is common practice for "field installed members" to be located at all concentrated loads not occurring at panel points as illustrated below.



In Tekla Structural Designer concentrated loads on K, LH and DLH joists are limited to the **maximum sum of concentrated loads** limit specified in Design Settings. This value is unfactored and in the event that the sum of all **unfactored** point loads in all loadcases within a combination exceeds this value, the relevant load combination is classified as 'Non-uniform'.

Uplift

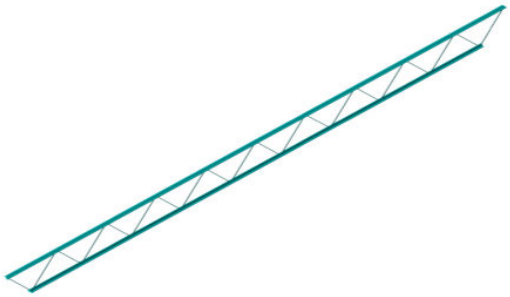
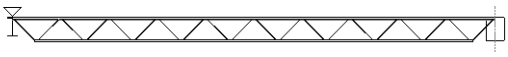
If (net) uplift is detected for a combination no design is performed for it and a warning message is displayed.

NOTE If uplift exists in a loadcase but is 'overcome' by positive loading in another loadcase then the design is valid and no warning is displayed.

Joist member reports

Joist design results can be viewed on the screen and incorporated into member design reports. Joists are also included in material listing reports.

The member design report for steel joists is configurable, but limited to the following chapters:

Chapter	Content
Picture	
Drawing	
Loading	**Loading** Loadcase | Source | Direction | In Proj. | Span/Stack | Type | Q_k [klf] | Pos [ft, in] | Length [ft, in] 0 Self weight - excluding slabs | User | Global Z | | 1 | Full UDL | 0.01 | | 26' 9 3/8" 2 Dead | Decomposition 1-way | Global Z | | 1 | UDL | 0.10 | 0" | 26' 9 3/8" 4 Live | Decomposition 1-way | Global Z | | 1 | UDL | 0.08 | 0" | 26' 9 3/8" **Load Reductions** Loadcase | Imposed Load Reductions | Span | Reduction Factor | Tributary Area [ft²] | K_{LL} Factor 4 Live | No
Design Summary	Design Condition | # | Design Value | Design Capacity | Units | U.R. | Status Min. Joist Depth | - | 16 | 13 25/64 | in | 0.837 | ✓ Pass Strength | 62 | 0.19 | 0.23 | klf | 0.826 | ✓ Pass Deflection Dead | 62 | 0.11 | - | klf | - | - Deflection Live | 62 | 0.08 | 0.14 | klf | 0.593 | ✓ Pass Deflection Total | 62 | 0.19 | 0.23 | klf | 0.826 | ✓ Pass Head code: United States (ACI/AISC), design code: AISC 360/341 ASD (2010)
Design Calculations	Min. Joist Depth Span =26' 9 3/8" ft, in Minimum depth =1' 1 3/8" ft, in Joist depth =16 in Ratio =0.837 ✓ Pass Strength 62 ASD_2-D+L - Critical Required total load =0.19 klf Design total load =0.23 klf Ratio =0.826 ✓ Pass

1.11 Steel truss design

Steel truss design overview

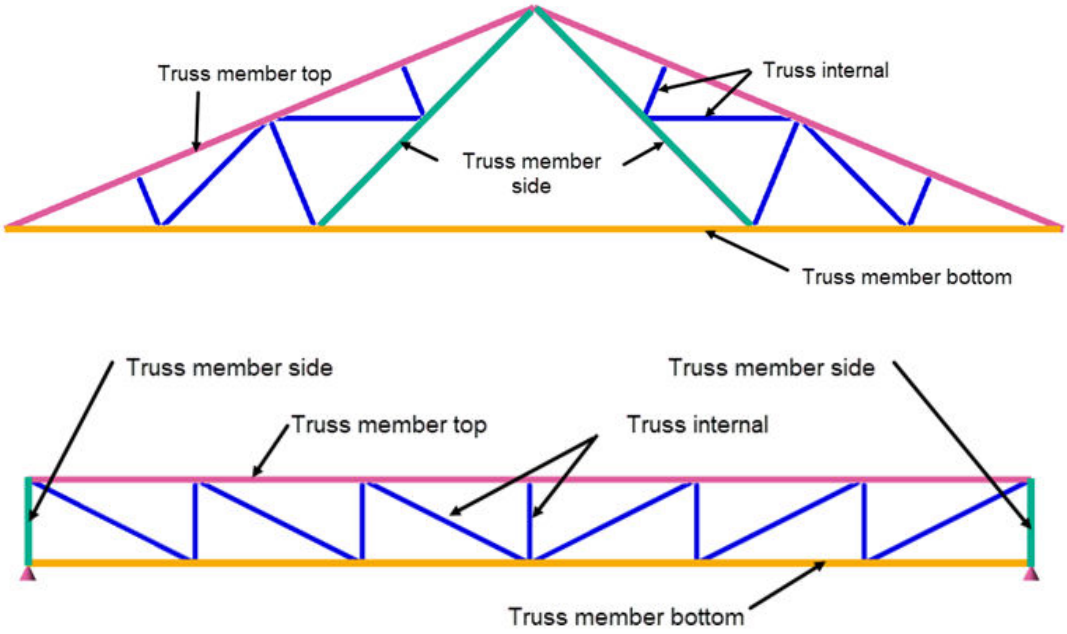
In Tekla Structural Designer although trusses can be defined in any material, design is restricted to steel and cold formed truss members only.

The Truss Wizard provides a wide range of standard truss shapes and configurations. This covers most practical layouts of truss. However, you can also modify a standard truss to produce one of slightly different configuration or manually 'stick build' your own truss from truss members if it is an unusual layout.

Trusses created by the Truss Wizard will comprise a mix of the following truss member types:

- Truss member top
- Truss member bottom
- Truss member side
- Truss internal

These are shown in two of the standard configurations below:



Solver elements 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. Consequently, all intersecting members 'node' at the same point – i.e. all internals meet along the set out line of the chords; this assumes that set out lines are coincident with the centroidal axes. Therefore, no in–plane eccentricities are considered in the analysis and design of the trusses.

A wide range of section types can be defined that includes all the common rolled sections. There is no restriction on the type and size of sections that can be connected within the truss. The practicality and efficiency of connections between members is your responsibility. A 'beyond scope' status is issued if a section type has been applied that cannot be checked for a given design condition.

Design is performed using a set of design forces obtained from 3D Analysis. (Grillage Chasedown Analysis and FE Chasedown Analysis results are not required.)

The design checks performed depend on the truss member types as follows:

- **Truss member top and bottom** - these are continuous members with axial force and bending principally in the plane of the truss. They are designed as beams (page 28) for all internal forces (axial, major and minor axis bending and shear) depending upon the section type used. Where tension exists in a chord member, the tension capacity is based on the effective net area.
- **Truss internal** - these are restricted to be pin ended, and are designed to the appropriate clauses for tension and compression members. Primary bending moments due to self weight and secondary moments due to eccentricity of their connections are ignored. Effective lengths for compression and effective net area for bolted and welded connections can be taken into account via the properties of the truss members.
- **Truss member side** - depending on the truss type selected in the Truss Wizard these will either default to pin ended or fully fixed. If pin ended, the design checks are the same as for Truss internal. If fully fixed at both ends (as in the case of a Vierendeel truss), the design checks are the same as for Truss member top and bottom.

  NOTE If a (pin ended) side is intersected by another truss member then it is still designed for axial force only, but a warning is issued indicating the forces that have been ignored.

For top and bottom chords, conditions of restraint can be defined in and out-of-plane for strut buckling and, top and bottom flange for lateral torsional buckling (LTB). It is upon these that the buckling checks are based. Incoming members are identified by the program and sensible default values for whether these provide restraint or not are set up (see **Assumptions and Limitations** section below). Restraint cannot be added where no incoming member exists but full control of the effective length factors is provided. In all cases Tekla Structural Designer sets the default effective length to 1.0L, it does not attempt to adjust the effective length (between supports for example) in any way. You are expected to adjust the effective length factor (up or down) as necessary. You can also indicate chord sub-beams to be continuously restrained over their length where appropriate.

NOTE Effective length factors and continuous sub-beam restraints are edited by right clicking on individual chords and selecting Edit *chord reference* from the context menu - these can then be specified from the respective lateral and strut restraint pages of the Properties dialog. Continuous sub-beam restraints can also be edited in Show/Alter State > Restraints, as can restraint of internals to chord (defaults to in-plane only for strut and unrestrained for LTB).

Results of your truss design can be viewed on the screen and incorporated into a report. Truss members are listed as a separate type in the Material Listing report.

Assumptions and Limitations

Limitations

The following limitations apply:

- Web openings, plated sections including Fabsec beams (with or without openings) and Westok beams cannot be used as truss members
- Chord members cannot be placed vertically
- The arch member of a bowstring truss is drawn and designed as a series of facets and not as one continuous curved member
- Truss internals cannot be loaded directly and no loads from floors and roofs are decomposed to them - sides can be loaded, but forces other than axial are then ignored in the design if the pinned ends are not removed.
- Truss chords are not by default excluded from diaphragm action within a floor slab, but they can be deliberately excluded if required. See: https://teklastructuraldesigner.support.tekla.com/support-article/2816265

Assumptions - Restraints

- In both top and bottom chords the node points are assumed not to have incoming out of plane members unless you define such members in the model. Hence, at these positions only in-plane strut buckling restraint is assumed as a default. You can of course change these.
- In a top chord any incoming members not at node points are assumed to provide the following LTB and strut restraints,
- - incoming members at 90 degrees (± 45 degrees) to the plane of the truss i.e. horizontal for a truss in the vertical plane, top and bottom flange restraint for LTB and out-of plane strut buckling restraint,
  - incoming members at 0 degrees (± 45 degrees) i.e. vertical for a truss in the vertical plane, no LTB restraint and in-plane strut buckling restraint.
- In a bottom chord any incoming members not at node points are assumed to provide the following LTB and strut restraints,

    - incoming members at 90 degrees (± 45 degrees) to the plane of the truss i.e. horizontal for a truss in the vertical plane, top and bottom flange restraint for LTB and out of-plane strut buckling restraint,
    - incoming members at 0 degrees (± 45 degrees) i.e. vertical for a truss in the vertical plane, no LTB restraint and in-plane strut buckling restraint.
- Lateral restraints to the top or bottom flange of a chord are assumed to be capable of resisting restraint forces not less than those specified in the relevent section(s) of the design code.
- In all cases, for all member characteristics it is assumed that you will make a rational and 'correct' choice for the effective lengths between restraints *The default value for the effective length factor of 1.0 may be neither correct nor safe.*

1.12 Portal frame design

If portal frames are modelled and designed in Tekla Structural Designer they will be designed elastically for the forces determined from the 3D analysis, in the same manner as other steel beams and columns.

A more economic design can be obtained by exporting individual frames to **Tekla Portal Frame Designer**. This industry leading portal frame software performs an elastic-plastic analysis and design and undertakes member stability checks to EC3 or BS5950. Once designed, the resulting sections can then be returned to the Tekla Structural Designer model.

NOTE Tekla Portal Frame Designer is a separately purchasable product.

2 Steel connections handbook

- Check simple connection resistance (page 96)
- Create and size SidePlate connections (page 105)
- Create and check column base plates (page 114)
- Create and design other connections (page 115)
- Export connections to another application for design (page 122)
- Manage connections in the Project Workspace (page 123)

2.1 Check simple connection resistance

Tekla Structural Designer provides a quick and simple way of checking the shear capacity of simple beam connections, and the axial capacity of brace connections.

Overview

Tekla Structural Designer has a Connection Resistance database for simple beams and braces. For simple beams the database contains pre-defined types with pre-defined resistances to Eurocode and US regional codes.

You can add user-defined connection types and user-defined resistances to the database for any regional code, for steel and cold formed materials. Once defined, these can then be used across all projects.

Each resistance value in the database has an 'active' flag which must be 'on' for the resistance to be considered. (Simple beams and braces in the model that have no active resistances are not checked.)

Having configured the active resisistances in the database as required, the checks are then performed in the model as follows:

- For each simple beam the active shear resistances for each available connection type are read from the database.

- These resistances are compared to the maximum shear force at the connection (determined from a 3D analysis)
- The connection configuration is optimized accordingly for each connection type.
- A similar process is then followed for each brace, by comparing the active axial resistances to the maximum applied axial force.

Results can be displayed in a review data table, or output via a connection resistance report.

Specify 'active' connection resistances (Eurocodes)

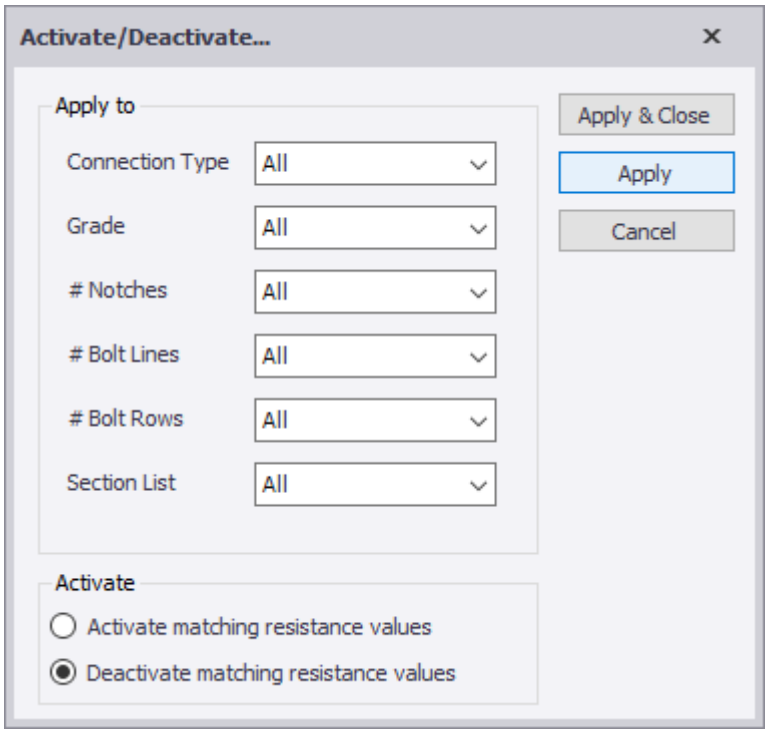
Since only active resistances are considered in the check, you can deactivate resistances for those connection configurations you do not want to be considered.

The active status of resistances can be set on or off directly (by ticking or unticking a checkbox in the Active column) but a quicker way of changing the active status is with the Activate/Deactivate dialog.

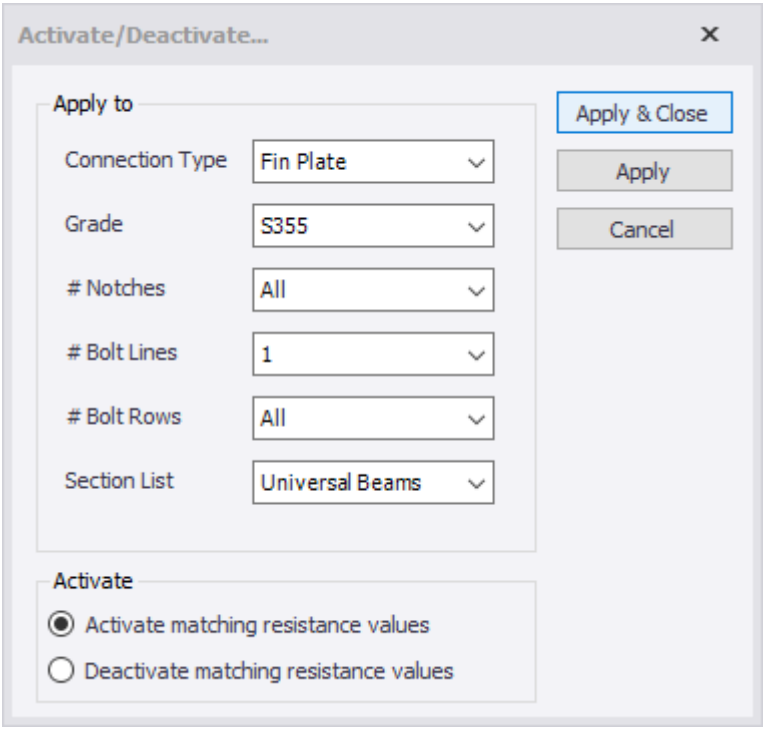
1. In the , click **Activate/Deactivate...**
2. Set the filters in the dialog, then set either Activate or Deactivate, as required.
3. Click **Apply** if you want to change further Active settings, or click **Apply & Close** to finish.

Example: From the pre-defined connection types you want only Fin Plates to be active, only for S355 Universal Beams, and only those Fin Plates with 1 line of bolts. There are a number of ways to achieve this using **Activate/ Deactivate** but one way would be as follows:

1. Click **Activate/Deactivate...**
2. Deactivate everything - select 'All' in each of the Activate/Deactivate filters, select 'Deactivate matching resistance values', then **Apply**



3. To activate only Fin Plates for S355 Universal Beams with 1 line of bolts
   a. select Fin Plate in the Connection Type filter,
   b. select S355 in the Grade filter,
   c. select 1 in the # Bolt Lines filter,
   d. select Universal Beams in the Section List filter,
   e. select 'All' in the other filters (# Notches, # Bolt Rows),
   f. select 'Activate matching resistance values',
   g. click **Apply & Close**



Following step 3, you will find ticks in the Active column only for Fin Plates with S355 Universal Beams with 1 line of bolts, *but note these changes are not saved to the database, and do not become active in Reports or in Tabular Data, until clicking OK in the main dialog.*

Specify 'active' connection resistances (US)

Since only active resistances are considered in the check, you can deactivate resistances for those connection configurations you do not want to be considered.

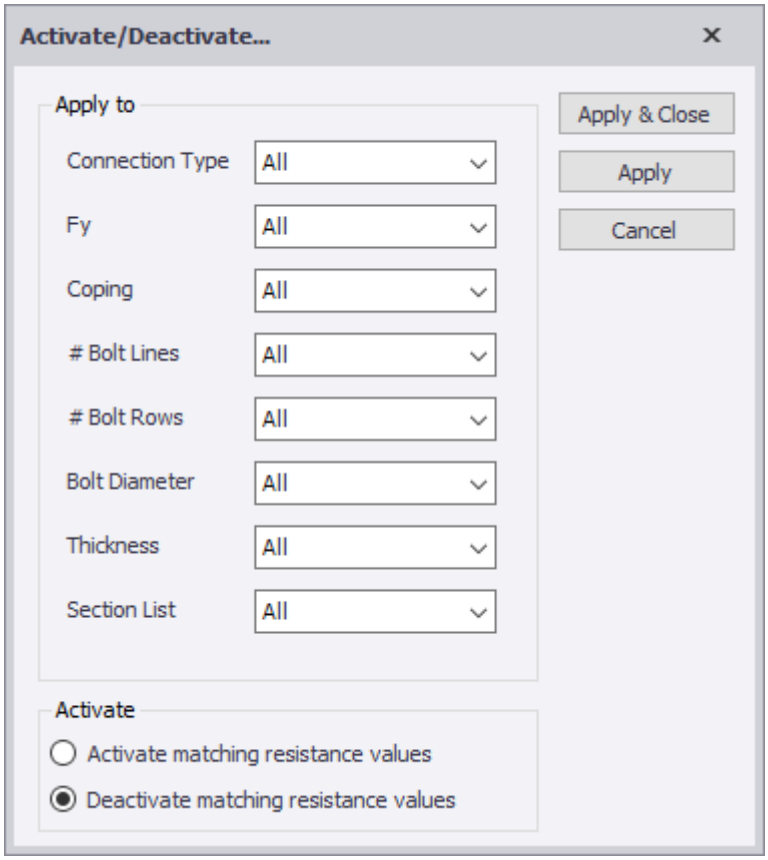
The active status of resistances can be set on or off directly (by ticking or unticking a checkbox in the Active column) but a quicker way of changing the active status is with the Activate/Deactivate dialog.

1. In the , click **Activate/Deactivate...**
2. Set the filters in the dialog, then set either Activate or Deactivate, as required.
3. Click **Apply** if you want to change further Active settings, or click **Apply & Close** to finish.

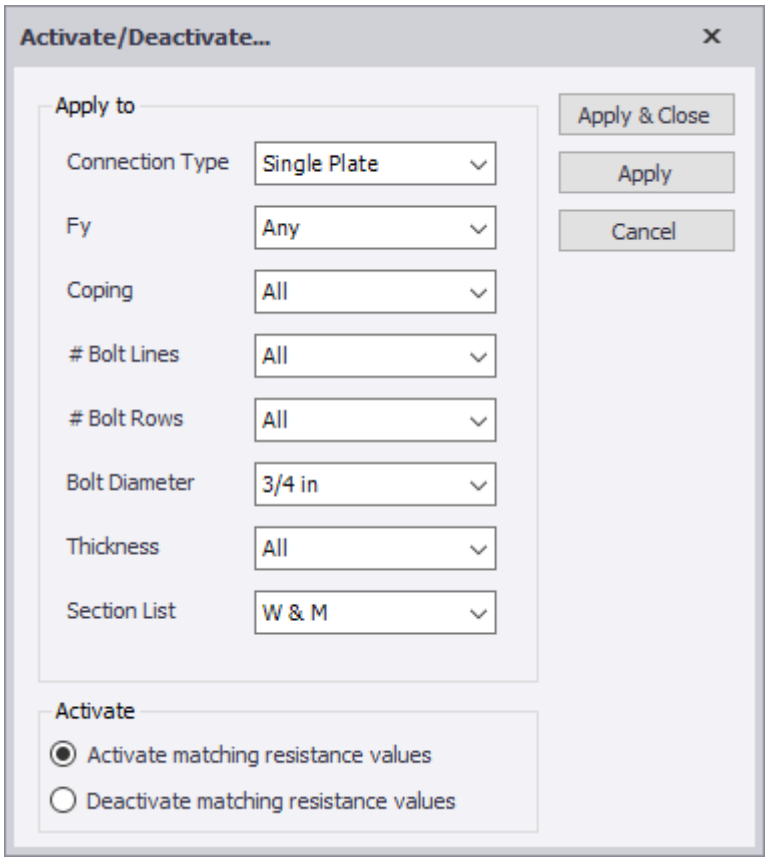
Example: From the pre-defined connection types you want only Single Plates to be active, for W & M beams of 'Any' Fy, only for 3/4 in bolts, but for all

defined thicknesses of plate. There are a number of ways to achieve this using **Activate/Deactivate** but one way would be as follows:

1. Click **Activate/Deactivate...**
2. Deactivate everything - select 'All' in each of the Activate/Deactivate filters, select 'Deactivate matching resistance values', then **Apply**



3. To activate only Single Plates for W & M Beams of 'Any' Fy, with 3/4 in diameter bolts and all plate thickness
   a. select Single Plate in the Connection Type filter,
   b. select Any in the Fy filter, select 3/4 in in the Bolt Diameter filter,
   c. select W & M in the Section List filter,
   d. select 'All' in the other filters (Coping, # Bolt Lines, # Bolt Rows, Thickness),
   e. select 'Activate matching resistance values',
   f. click **Apply & Close**



Following step 3, you will find ticks in the Active column only for Single Plates, with Any Fy selected, and W & M Beams with 3/4 in diameter bolts, *but note these changes are not saved to the database, and do not become active in Reports or in Tabular Data, until clicking OK in the main dialog.*

Run resistance checks

Provided you have an appropriate set of 'active' resistances, the checks are performed automatically, using the current analysis results - you can proceed directly to a review of the check results.

- If necessary, analyse the model to generate an up-to-date set of analysis results,
- Review the check results, see: ***Display connection resistance checks in a review data table*** below.

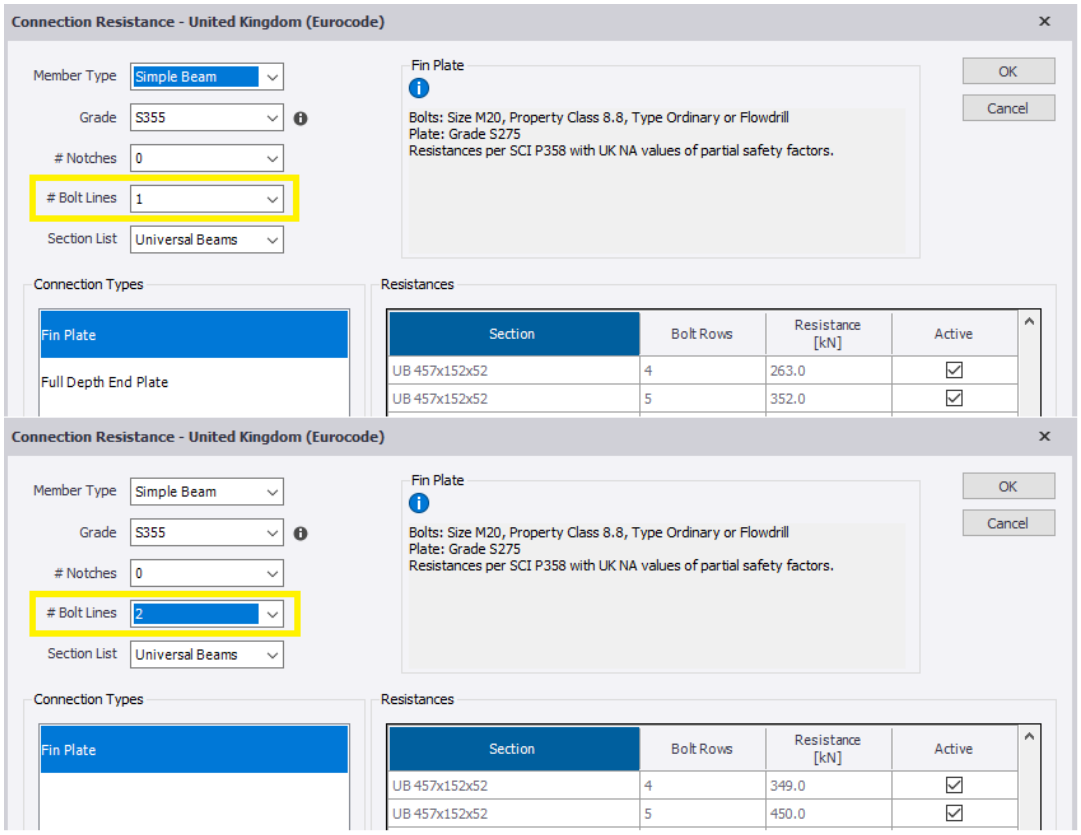
The connection optimization process

Tekla Structural Designer carries out an optimization process to find the first passing Active resistance based on the name of the Connection Type and the number of Bolt Lines and Bolt Rows assigned to that name.

For Eurocodes, separate check results are reported for each 'active' connection type and each notch variation.

For US codes, separate check results are reported for each 'active' connection type and each coping variation.

Example: to Eurocode, a Fin Plate for an S355 UB 457x152x52 with 1 and then 2 lines of bolts has *pre-defined* resistances as shown below. All resistances are indicated as Active.



If an S355 UB 457x152x52 in a model is found to have an applied major shear force of 250 kN then the connection resistance check will report that a Fin Plate with 1 bolt line and 4 bolt rows (resistance 263 kN) is adequate.

If the applied major shear force is 300 kN then the connection resistance check will report that a Fin Plate with 1 bolt line and 5 bolt rows (resistance 352 kN) is adequate.

If the applied major shear force is 355 kN then the connection resistance check will report that a Fin Plate with 2 bolt lines and 5 bolt rows (resistance 450 kN) is adequate.

For each of the applied major shear forces given above only the *first passing* resistance is reported, by looping through the *Active* bolt rows for 1 bolt line followed, where necessary, by looping through the *Active* bolt rows for 2 bolt lines.

If Tekla Structural Designer gets to the end of these loops and still doesn't find a passing resistance then it reports the last resistance found and a fail status e.g. if the applied major shear force is 455 kN then the connection resistance check will report that a Fin Plate with 2 bolt lines and 5 bolt rows (resistance 450 kN) fails.

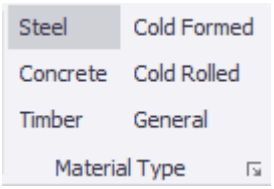
Now suppose that the UB 457x152x52 with 1 bolt line and 5 bolt rows (resistance 352 kN) is *not* Active. The first shear check above (applied force 250 kN) would be unchanged but the second check (applied force 300 kN) would now report that a Fin Plate with 2 bolt lines and 4 bolt rows (resistance 349 kN) is adequate. This illustrates the interplay of the resistance Active status with the optimization process.

Display connection resistance checks in a review data table

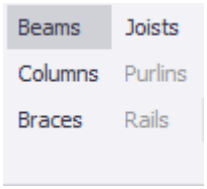
To create a tabular connection resistance summary:

1. If necessary, switch the view regime to a **Review View**.
2. On the **Review** ribbon tab, click **Tabular Data**.

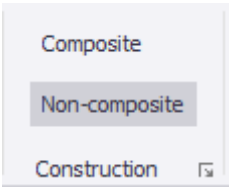
   A **Review Data** tab opens on the ribbon and a **Review Data View** is displayed.
3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Connection Resistance**.
4. In the **Material Type** group, select the material (either **Steel**, or **Cold Formed**).



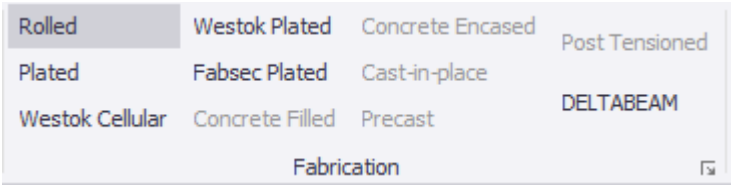
   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.
5. In the **Characteristic** group, select the required characteristic (either **Beams**, or **Braces**).



6. In the **Construction** group, select **Composite**, **Non-composite**, or both.



7. In the **Fabrication** group, select the required fabrication types.



A resistance check is provided for the filtered selection for every section where a resistance has been specified in the database.

NOTE A resistance check is not provided for beams that do not have simple connections.

NOTE If there is uplift in any load combination then the maximum uplift shear force will be displayed, with a warning that connection resistance is not checked for uplift condition.

NOTE Each time you re-analyse the building, if you have the Connection Resistance data table open you must close and then re-open it to update the results.

Create and display a connection resistance report

1. On the **Report** tab, use the droplist to select the existing **Connection Resistance** report.
2. Click **Model Report...**

3. Review the report structure and adjust as required.
4. Click **OK** to save the report.
5. Click **Show Report**

Related video

Predefined connection resistance database for Eurocode and AISC

2.2 Create and size SidePlate connections

SidePlates are a US proprietary connection system that can be applied in Tekla Structural Designer but are considered separately to other connection types.

Click the following links to find out more:

- For background to their usage in Tekla Structural Designer, see
  SidePlate connections theory (page 105)
- For modeling instructions, see
  Create SidePlate connections (page 112)

SidePlate connections theory

Overview of SidePlate connections

SidePlate® is a type of moment connection for connecting I or HSS section beams to I, HSS, built up box or built up WF (cruciform) section columns.

SidePlates are welded to the column section in the fabrication shop. Plates or angles are similarly welded to beam ends.

The column and beam units are transferred to site where the columns are erected and the beams are lifted into place and either bolted or welded to the columns. SidePlate connections are used on buildings of 1 to 30 stories tall.

Within the structure, the beam end is held in position by SidePlates. The net result is moment connection with a very stiff section of column and a stiffened beam end.

SidePlate connection types can be:

- Non-Seismic moment connections
  - A beam/column moment connection anywhere in the structure

- Seismic moment connections
  - A seismic beam/column moment connection within any of the following seismic force resisting systems Ordinary Moment Frame (OMF), Intermediate Moment Frame (IMF) or Special Moment Frame (SMF).

SidePlate connections work for all WF and HSS sections in the AISC steel book, and for all UB and UKB sections. However it is worth noting the SidePlate datafile may NOT have a full set of values for all these sections; if this is the case then no SidePlate connection can be applied at the joint.

Tekla Structural Designer can perform an initial sizing for the SidePlate connection, but the final detailed design of the SidePlate connection has to be undertaken in SidePlate® software.

Permitted SidePlate connections in Tekla Structural Designer

RESTRICTION In Tekla Structural Designer, SidePlate connections are currently only permitted for US Customary sections and not for metric sections.

Column		SidePlate Permitted Connections				Tekla Structural Designer Status
		Non-Seismic Moment Connections		Seismic Moment Connections		
		Beam		Beam		
		I	HSS	I	HSS	
 W	Flange	yes	yes	yes	yes	Modeled
	Web	yes	yes	Warning	Warning	Analyzed Designed
HSS 	Flange	yes	yes	yes	yes	Modeled
	Web	yes	yes	yes	yes	Analyzed Designed
Plated box 	Flange	yes	yes	yes	yes	Modeled
	Web	yes	yes	yes	yes	Analyzed **Not Designed**

Column		SidePlate Permitted Connections				Tekla Structural Designer Status
		Non-Seismic Moment Connections		Seismic Moment Connections		
		Beam		Beam		
		I	HSS	I	HSS	
W + 2xWT 	Flange	yes	yes	yes	yes	**Not Modeled** **Not Analyzed** **Not Designed**
	Web	yes	yes	yes	yes	
W + 1xWT 	Flange	yes	yes	yes	yes	**Not Modeled** **Not Analyzed** **Not Designed**
	Web	yes	yes	Warning	Warning	

SidePlate workflow in Tekla Structural Designer

A brief description of the SidePlate workflow in Tekla Structural Designer is given here with more detail available in the subsequent topics.

- **Modeling**: You define SidePlate connections by changing properties of the relevant beam or beams.

  NOTE The **Update Connections** process used for defining other connection types is **not** applicable for SidePlate.

  Connection objects are not created for any point on a steel column where *any* of the beams attached at that point have a SidePlate connection.

- **Visualisation**: Each connection is initially shown in the graphics as a simple box. After sizing has been performed it is shown as a pair of plates.
- **Validation**: Some preliminary checks are carried out in Model validation, (additional checks are done subsequently in **Initial sizing** and **Analysis model adjustment**). Any problems do not stop the analysis and design process but mean the relevant beam and or column will not benefit from the enhanced properties.
- **Initial sizing**: The initial SidePlate dimensions are determined during the Design Steel (Static & RSA) and Design All (Static & RSA) processes, using data provided by SidePlate and incorporated into Tekla Structural Designer.

NOTE After the initial sizing process, any changes to loading, (individual loads, loadcases and combinations), will not trigger change control for the SidePlates, i.e. they will keep those initial sizes. You can reset the status using Review mode if you decide that the sizing process needs to be run again.

- **Analysis model adjustment**: Where valid SidePlate dimensions exist, the analysis model passed to the solver is adjusted to reflect the increased stiffness of the SidePlate connections. This is done during stand-alone analysis as well as Design.
- **Steel member design**: If SidePlate connections exist, they are taken into account in the member design for steel beams and columns. The changes affect both static and seismic design.
- **Review**: The SidePlate connection status can be seen easily in a Review view, and tooltips give more information.
- **Reports**: A “SidePlate Connection Report” can be generated.
- **Final design**: SidePlate developers can use the **Tekla Structural Designer API** in order to fully design a SidePlate connection defined and initially sized in Tekla Structural Designer

NOTE Design of SidePlate connections is beyond scope for both Tekla Connection Designer and export to the IDEA StatiCa Connection Design program.

SidePlate 'joint' and 'connection' terminology

NOTE
- A SidePlate **joint** exists at a column node and consists of 1 to 4 valid connections.
- Every suitable beam framing into the column node can create a SidePlate **connection**.

A SidePlate connection exists at End 1 and / or End 2 of a steel beam where **Apply SidePlate Connection** is true. It will still exist even if other data makes it invalid. The connection position is the reference point at the relevant end of the beam.

All connections sharing the same position on a column are considered as a single SidePlate Joint. No attempt is made to merge separate SidePlate joints even where the relevant levels are close.

Various checks are performed on each connection and each joint, and the analysis model is only be adjusted if they all pass. In particular, if the depths of two SidePlate joints overlap then a warning is generated and the analysis model is not be adjusted.

SidePlate validation

SidePlate validation is performed during model validation.

If warnings are issued, this will typically result in the Connection Status being set to "Not Applied" in the **Properties** window.

NOTE Until the “Not Applied” Connection Status is resolved, the connection will not be considered for the initial sizing process.

Initial sizing of the SidePlates

The initial SidePlate dimensions are determined during the Design Steel (Static & RSA) and Design All (Static & RSA) processes.

The initial sizing process is only performed where the Sizing Status is “Not Performed” and the Connection Status is **not** “Not Applied”.

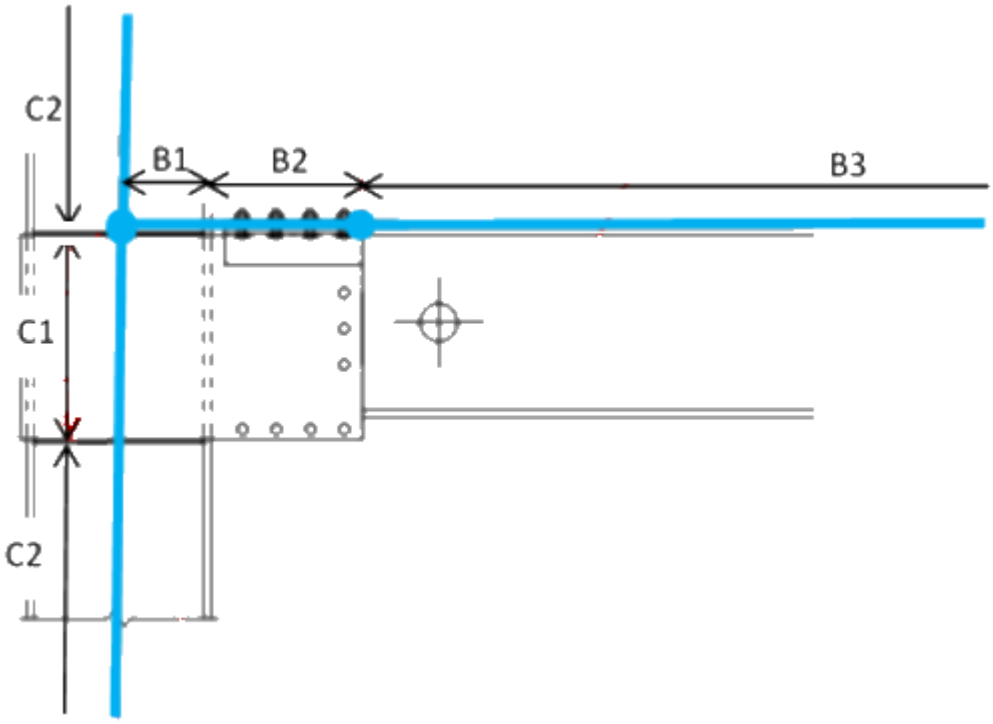
For each connection that is considered, a number of checks are performed in order to verify that SidePlate can be applied. If any of the checks fail, the Connection Status is set to "Not Applied".

NOTE After the initial sizing process, any changes to loading, (individual loads, load cases and combinations), will not trigger change control for the SidePlates, i.e. they will keep those initial sizes. You can reset the status using Review mode if you decide that the sizing process needs to be run again.

Analysis model adjustment

Analysis model adjustment is only performed for valid SidePlate connections that have successfully completed the initial sizing process, i.e. where the Sizing Status is **not** “Not Performed”.

Both the geometry and the properties of the Solver model are dependent upon the size of the SidePlate connection.



For the beam

- B2 – a new 1d element on the beam, with stiffened beam properties.
- B1 – a rigid offset on the end of B2
- B3 – normal bare beam properties

For the Column

- C1 – a rigid offset at the top of the column 1d element
- C2 – normal column properties

Design/Analysis processes and recommended workflows

Tekla Structural Designer can automatically size members throughout a structure – we call this Autodesign. The process is complex and involves sophisticated initial sizing, multiple analysis runs with repeated design cycles. The interweaving of initial sizing for SidePlate connections into the Autodesign routines is a highly complex process and for the first implementation remains beyond scope. Instead you have to manually reduce section sizes to take advantage of the SidePlate advantages.

Any beams and columns not affected by SidePlate connections can be AutoDesigned as normal during this process.

The following examples show the results for 2 different workflows where each step is carried out in the exact order specified and assuming that there are no model issues.

Recommended Workflow

- Full model created including loadcases and combinations - AutoDesign left checked for all columns, beams etc.
- Check “Apply SidePlate Connection” for selected beams - also set Utilization Ratio to 1.25 for those beams.
- Set Utilization Ratio to 1.1 for selected columns
- Run 1st order linear analysis
    - Validation will generate SidePlate AutoDesign warnings where relevant
    - SidePlate Initial Sizing will not be performed because not Static Design
    - Adjust Solver Model will not be performed because no Connections will have Sizing Status not “Not Performed”.
- Run Gravity Design Steel
    - Validation will generate SidePlate AutoDesign warnings where relevant
    - SidePlate Initial Sizing will not be performed because not Static Design
    - Adjust Solver Model will not be performed because no connections will have Sizing Status not “Not Performed”.
- Reset Utilization Ratio to 1.0 for relevant beams and columns
- Run Static Design Steel
    - No SidePlate related validation warnings
    - SidePlate Initial Sizing is performed for all connections
    - Adjust Solver Model will be performed for subsequent analysis
- Manually adjust section sizes to achieve further size savings with SidePlate connections
    - SidePlate Connection Status will be reset where necessary
- Run Static Design Steel
    - No SidePlate related validation warnings
    - SidePlate Initial Sizing is performed only where section sizes have been changed
    - Adjust Solver Model will be performed

At the end of this workflow, any subsequent stand-alone analysis will also use the adjusted solver model.

Other Workflow

- Full model created including loadcases and combinations - AutoDesign left checked for all columns, beams etc.

- Check “Apply SidePlate Connection” for selected beams.
- Run Static Design Steel
  - Validation will generate SidePlate AutoDesign warnings where relevant
  - SidePlate Initial Sizing will not be performed because Connection Status will be set to Warning
  - Adjust Solver Model will not be performed because no Connections will have Sizing Status not “Not Performed”.
- Run Static Design Steel
  - No SidePlate related validation warnings
  - SidePlate Initial Sizing is performed for all Connections
  - Adjust Solver Model will be performed for subsequent analysis

Effects of SidePlate on steel member design

Any beams and columns not affected by SidePlate connections can be AutoDesigned as normal.

Beams and columns affected by SidePlate connections cannot be AutoDesigned.

Where SidePlate connections exist, changes are required for both static and seismic member design.

For static design

- Beam and column buckling lengths are affected.
- Beam and column design checks are adjusted so that they are applied outside the physical length of any SidePlate connection

For seismic design

- OMF frames - no change
- IMF frames – various changes
- SMF frames – various changes

Create SidePlate connections

To add SidePlate connections:

1. Select the steel beam(s) to which you want to apply SidePlate connections.
2. In the **Properties** window, expand the End 1, or End 2 SidePlate heading and select **Apply SidePlate connection** at one, or both ends as required.

3. In the **Properties** window, select the SidePlate **Connection type** that is appropriate to the SFRS type that has been specified, or, if not in a seismic frame, set the type as **SidePlate MF** (moment frame).

   Each connection is initially shown in the graphics as a simple box. After sizing has been performed it will be shown as a pair of plates.

See also

Beam properties - SidePlate (page 113)

Beam properties - SidePlate

For steel beams the **Properties** window has a SidePlate group within which an **Apply SidePlate Connection** checkbox can be selected for End 1 and End 2 of the beam.

When selected the following properties are displayed:

- Seismic Provision - depending on the regional code, one of:
  - Non-Seismic (ACI/AISC only)
  - Seismic OMF
  - Seismic IMF (ACI/AISC only)
  - Seismic SMF
- Field Connection, (user option if Connection Type is Seismic SMF, otherwise fixed to Bolted)
  - Bolted (default)
  - Welded
- Datafile look up criteria (readonly)
  - Use %Mp (if Connection Type is Non-Seismic)
  - Use clear span (otherwise)
- Connection Status - one of
  - Unknown (default)
  - Pass
  - Warning
  - Not Applied
- Failure Reason - one of
  - Unknown (default)
  - Datafile Look Up Failed
  - Geometric Compatibility

    - Beam Element too short for Rigid Offset
    - Column Element too short for Rigid Offset
- Sizing Status - one of
    - Not Performed (default)
    - Initial
    - Final
- SidePlate size and other data set during the Initial Sizing Process, (Invisible if Sizing Status is Not Performed)
    - DimA – the outstand distance required for the SidePlate connection
    - DimB – the depth required for the SidePlate connection
    - Tsp – the thickness of the SidePlates
    - DimH – the length of the SidePlates
    - $\%M_p$- if calculated during initial sizing
    - Clear span - if calculated during initial sizing

NOTE These properties are for the start and end of the whole beam. Where a steel beam is multi-span it is not possible to apply SidePlate connections to the internal points.

2.3 Create and check column base plates

Base plates can only be created under steel columns. How these can be checked depends on the regional code being worked to.

Create column base plates

Tekla Structural Designer automatically adds a base plate on creation of a new steel column if a support is required.

NOTE If an existing steel column is missing a base plate you can add one by clicking the (Add) Base Plate button on the Design ribbon and then select the column.

Check column base plates

How base plates are checked depends on the head code being worked to.

- **AISC and Eurocode**

  Base plates can be checked directly in Tekla Structural Designer, but cannot be designed in Tekla Connection Designer. For details, see: Column base plate design (page 80) in the Steel Design Handbook

- **British Standards**

  Base plates **cannot** be checked directly in Tekla Structural Designer, but **can** be designed in Tekla Connection Designer. For details, see: Create and design other connections (page 115)

- **Australian & Indian Standards**

  Base plates **cannot** be checked in Tekla Structural Designer, or designed in Tekla Connection Designer.

2.4 Create and design other connections

NOTE These topics relate to the *design* of connections using either Tekla Connection Designer, or IDEA StatiCa; they do not cover checking of simple connection resistance (page 96), or checking of AISC/Eurocode base plates (page 80), which are separate topics.

Overview

Before connections can be designed in Tekla Structural Designer valid connection objects have to be created in the model. This is achieved by running **Update connections** from the Project Workspace. This applies a set of rules (page 118) to create and organize connections into the following types:

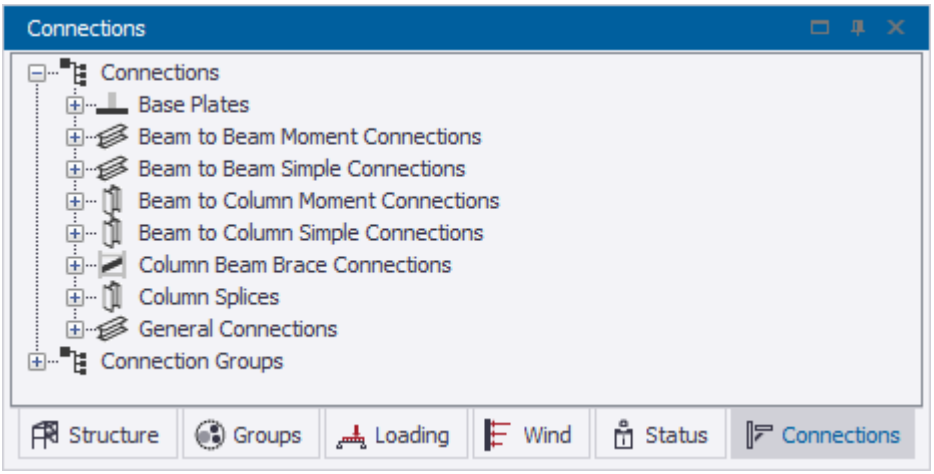
- Base plate connections - BS regional code only

  NOTE Base plates to AISC or Eurocode regional codes are handled differently, see: Column base plate design (page 80)

- Beam to beam moment connections
- Beam to beam simple connections
- Beam to column moment connections
- Beam to column simple connections
- Column beam brace connections
- Column splices

- General connections

Valid connections are then displayed in the scene views and are also listed in the **Connections** tree.



Provided a license of Tekla Connection Designer is available, some of these can then be designed or exported to Tekla Connection Designer. If you have a license of IDEA StatiCa you can also export some types to that software for design.

Type	Connection Design Options
Base plate connections	• Design Connection (BS regional code only) • Export to Tekla Connection Designer (BS regional code only)
Beam to beam moment connections	• Design Connection (Eurocode and BS regional codes only) • Export to Tekla Connection Designer (Eurocode and BS regional codes only) • Export to IDEA StatiCa
Beam to beam simple connections	• Export to IDEA StatiCa
Beam to column moment connections	• Design Connection (Eurocode and BS regional codes only) • Export to Tekla Connection Designer (Eurocode and BS regional codes only) • Export to IDEA StatiCa
Beam to column simple connections	• Export to IDEA StatiCa

Type	Connection Design Options
Column beam brace connections (beam/ column junctions to which one or more brace members also connect)	• Export to IDEA StatiCa
Column splices	• Design Connection (Eurocode and BS regional codes only) • Export to Tekla Connection Designer (Eurocode and BS regional codes only) • Export to IDEA StatiCa
General Connections	• Export to IDEA StatiCa

See also

- Recommended workflows for specific connection types (page 119)
- Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 121)
-

Update connections

To create connection objects for the first time, or to recreate connections after changes to the model:

1. Click on the **Connections** tree in the **Project Workspace**
2. Right-click the Connections branch.
3. Choose **Update Connections** from the context menu.

   This applies a set of rules (page 118) to determine all valid steel connections in the model. The resulting connections are listed in the Connections Tree and are also shown by bounding boxes in the Scene Views.

   NOTE If the model has been changed so that new connections exist, these are not created automatically; you are required to update connections as required.

Grouping of connections

Connection objects are grouped automatically by their associated member section size, the resulting groups being displayed in the Project Workspace > Connection tree.

The following rules are applied in automatic connection grouping:

- Beam to Beam Simple Connections - are grouped by the associated beam section size. Where there are two different sizes of beams associated with connection then grouping is based on the higher section size of the beam.
- Beam to Column Moment Connections - are grouped by the associated column section size.
- Beam to Column Simple Connections - are grouped by the associated column section size.
- Column beam brace connections - are grouped by the largest associated brace section size.
- Column Splices - are grouped by the associated column stack below section size.
- General Connections - where a general connection is associated with a column then it will be grouped by the column stack below section size, otherwise it will be grouped by the associated beam section size.

Design connections

Design using Tekla Connection Designer is initiated in one of two ways:

1. From the Project Workspace:
   - Expand the appropriate branch in the **Connections** tree
   - Right click on the required connection reference and choose **Design connection** from the context menu.
2. From a scene view:
   - Right click on the connection object and choose **Design connection** from the context menu.

See also

Recommended workflows for specific connection types (page 119)

Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 121)

Export connections to another application for design (page 122)

Steel connection formation rules

The following rules relate to connection formation in Tekla Structural Designer.

1. beam to column connection
    a. beam connections to the column base are ignored
    b. column must be a symmetric section
    c. more then 1 beam attached to the same column face prevents any connection
    d. a continuous beam connected to the column flanges prevents any connection
    e. a beam with a fully fixed or moment end condition connected to the column web prevents any connection
2. beam to beam - for a particular construction point (node)
    a. any column at the node prevents the connection
    b. only two beam ends can be connected, a 3rd beam at the same node prevents any connection.
    c. connections cannot be formed at beam internal nodes
    d. the beams being connected must be symmetric and of the same section size
    e. connected beam ends cannot be free or pinned
    f. top flange haunches prevent the connection
    g. beam ends being connected must have no end plates specified, or both have the same end plate
    h. beam ends being connected must have no bottom haunch specified, or both have the same bottom haunch
3. any non-steel member at the node prevents any connection
4. braces are ignored in the procedure

Recommended workflows for specific connection types

NOTE Connection design requires a Tekla Connection Designer licence.

Portal frame connections

Although it is possible to design portal frame connections in Tekla Structural Designer - i.e. Base Plate, Eaves, and Apex moment connections; it should be

noted that the design forces used for connection design within Tekla Structural Designer are those from its own elastic analysis and may not be appropriate.

A more correct approach would be to derive the connection forces from a Tekla Portal Frame Designer plastic analysis. To do this you would need to export the portal frame to Tekla Portal Frame Designer and then design both the frame and its connections within Tekla Portal Frame Designer.

Note that it in addition to the connection forces (usually) being fundamentally different between Tekla Portal Frame Designer and Tekla Structural Designer there are other potential differences also:

- for eaves connections - in Tekla Portal Frame Designer the connection design forces are taken at the column face (rather than the rafter/column intersection),
- for column bases - in Tekla Portal Frame Designer they can be pinned at ULS but % fixity can be taken for SLS, while Tekla Structural Designer will always give an analysis moment for % fixity regardless of ULS/SLS and this moment would be designed for.

NOTE Connection design via Tekla Portal Frame Designer requires both a Tekla Portal Frame Designer and a Tekla Connection Designer licence.

Base plate connections (not in portal frames)

Base plate connections can be edited directly in Tekla Structural Designer.

Provided they are in elastically analyzed frames they can be also be designed using the design forces obtained from Tekla Structural Designer.

NOTE The analysis results used for the connection design follow the selection (1st order or 2nd order) made on the Analysis page of the Tekla Structural Designer Design Options.

Base plate connections can be exported to Tekla Connection Designer if you require, in which case if you edit the connection geometry while in Tekla Connection Designer you should use the “Return Connection to Tekla Structural Designer” command (under the “Connection” menu) to return the edited connection data back to Tekla Structural Designer.

Other connections (not in portal frames)

The recommended workflow for connection types other than Base Plate Connections is:

- Select the connection(s) to be designed and export to Tekla Connection Designer always.
- Design in Tekla Connection Designer and print the reports.
- While the “Return Connection to Tekla Structural Designer” command can be used, this does not currently return all connection data to Tekla

Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/ or design processes run and the connection exported again for checking, the engineer should review the connection data and may need to re-enter some of the settings previously made.

NOTE While the "Return Connection to Tekla Structural Designer" command can be used In Tekla Connection Designer, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/or design processes are re-run; if the connection is exported again for checking, the engineer would need to review the connection data as it may be necessary to re-enter some of the settings previously made.

See also

Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 121)

Limitations when using Tekla Connection Designer with Tekla Structural Designer

General

- The analysis results used for the connection design follow the selection (1st order or 2nd order) made on the Analysis page of the Tekla Structural Designer Design Options.
- It is always necessary to export all connection types to Tekla Connection Designer for reporting purposes.

Base plate connections

- When working to either AISC or Eurocode, base plates cannot be transferred to Tekla Connection Designer for design, but can be checked directly in Tekla Structural Designer. For details, see: Column base plate design (page 80) in the Steel Design Handbook.
- When working to British Standards, base plates cannot be checked directly in Tekla Structural Designer, but can be transferred to Tekla Connection Designer for design. For details, see: Create and design other connections (page 115).
- Base plate connections only deal with major axis shear. Hence when transferred to Tekla Connection Designer only one value for shear is available in the Combinations grid. (This is a limitation in the Green Book also.)

- When working to Australian or Indian Standards, base plates cannot be checked in Tekla Structural Designer, or transferred to Tekla Connection Designer for design.

Moment connections

- While the “Return Connection to TSD” command can be used, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/ or design processes run and the connection exported again for checking, the engineer should review the connection data and may need to re-enter some of the settings previously made.
- For Beam to Column moment connections, column forces are exported and displayed on the "Column" tab of the Combinations page.
    - When designed to Eurocodes, there is a reduction factor for column web in compression to allow for coincident forces in the column itself i.e. moment and axial force. This is called kwc and in most cases is 1.0 i.e. no reduction. But where the stress due to the column forces is high (> 0.7 fy) then kwc is < 1.0. See Clause 6.2.6.2 (2).
    - These forces are NOT populated by the TCD integration link and must be entered manually by the engineer.

Simple connections

- It is not currently possible to transfer Simple Connections from Tekla Structural Designer to Tekla Connection Designer.

Hollow section connections

- It is not currently possible to transfer Simple Connections from Tekla Structural Designer to Tekla Connection Designer.

Column splice connections

- Splice connections with significant ‘real’ moments (other than eccentricity moments) cannot currently be transferred from Tekla Structural Designer to Tekla Connection Designer.

NOTE Significant 'real' moments being greater than the limit in Design Settings.

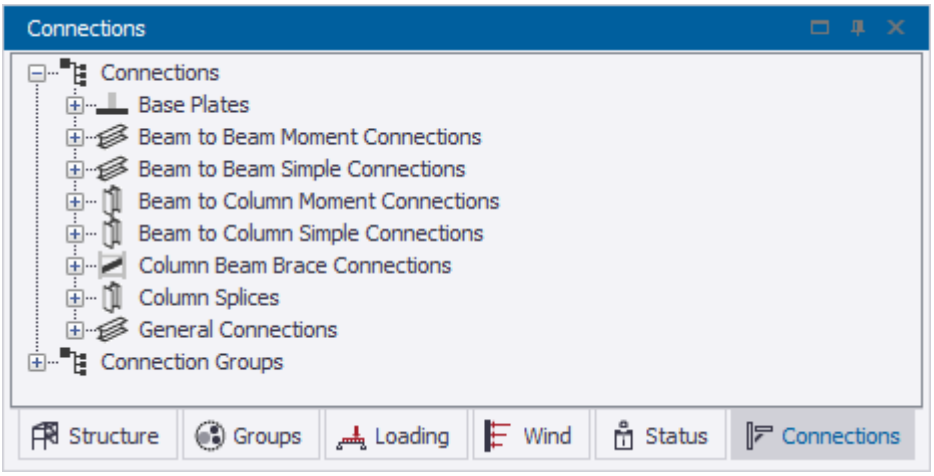
2.5 Export connections to another application for design

Click the following links to find out more about exporting specific connections into another application for design:

- Recommended workflow for TCD connection types (page 119)
-
-
-

2.6 Manage connections in the Project Workspace

The **Connections** tree organizes steel connections in a hierarchical way, by Type (i.e. Type/Reference), and by Group (i.e. Type/Section size/Reference).



When opened for the first time, or if the model has been edited in a way that affects the connection configurations within it, you must run **Update Connections** in order to determine the valid connections.

This applies a set of rules (page 118) to create and organize connections into the following Types:

- Base plate connections
- Beam to Beam Moment Connections
- Beam to Beam Simple Connections
- Beam to Column Moment Connections
- General Connections

Valid connections are displayed in the scene views and are also listed in the **Connections** tree

NOTE Simple connection resistance (page 96) and SidePlate sizing (page 105) are independent processes that do not require valid connections to be formed in the above manner. Consequently no information is displayed for them in the **Connections** tree.

Update connections

To create connection objects for the first time, or to recreate connections after changes to the model:

1. Click on the **Connections** tree in the **Project Workspace**
2. Right-click the Connections branch.
3. Choose **Update Connections** from the context menu.

   This applies a set of rules (page 118) to determine all valid steel connections in the model. The resulting connections are listed in the Connections Tree and are also shown by bounding boxes in the Scene Views.

   NOTE If the model has been changed so that new connections exist, these are not created automatically; you are required to update connections as required.

Edit connection

- In the **Connections** tree, right click on the appropriate connection reference and choose **Edit...** from the context menu. A dialog opens allowing the connection to be edited.

Select connection in visible views

- In the **Connections** tree, right click on the appropriate connection reference and choose **Select in visible views** from the context menu. The connection is selected if it exists in the active scene view.

Design connection

- In the **Connections** tree, right click on the appropriate connection reference and choose **Design connection** from the context menu. The connection is designed using Tekla Connection Designer.

Export connection to Tekla Connection Designer

- In the **Connections** tree, right click on the appropriate connection reference and choose **Design connection** from the context menu. The connection is designed using Tekla Connection Designer.

See also

Create and check column base plates (page 114)

Create and design other connections (page 115)

Export connections to another application for design (page 122)

3 Concrete design handbook

This handbook contains information relating to cast-in-place concrete design in Tekla Structural Designer.

Cast-in-place concrete members and walls

These can be batch designed directly from the **Design** tab, the necessary analysis being automatically performed as part of the batch design process. As an alternative to batch design, once analysis has been performed you can choose to adopt selective, or, interactive design. For further details, see:

- Concrete member design workflow (page 127)
- Concrete member autodesign (page 143)
- Concrete beam and column groups (page 146)
- Concrete beam design fundamentals (page 150)
- Concrete column design fundamentals (page 162)
- Concrete wall design fundamentals (page 169)
- Interactive concrete member design (page 173)

Cast-in-place concrete slabs

These can be designed directly from the **Design** tab, but a certain amount of user interaction is required. For further details, see:

- Flat slab design workflow (page 215)
- Slab on beams design workflow (page 239)
- Concrete slab design fundamentals (page 255)
- Punching shear check fundamentals (page 264)

Cast-in-place concrete foundations

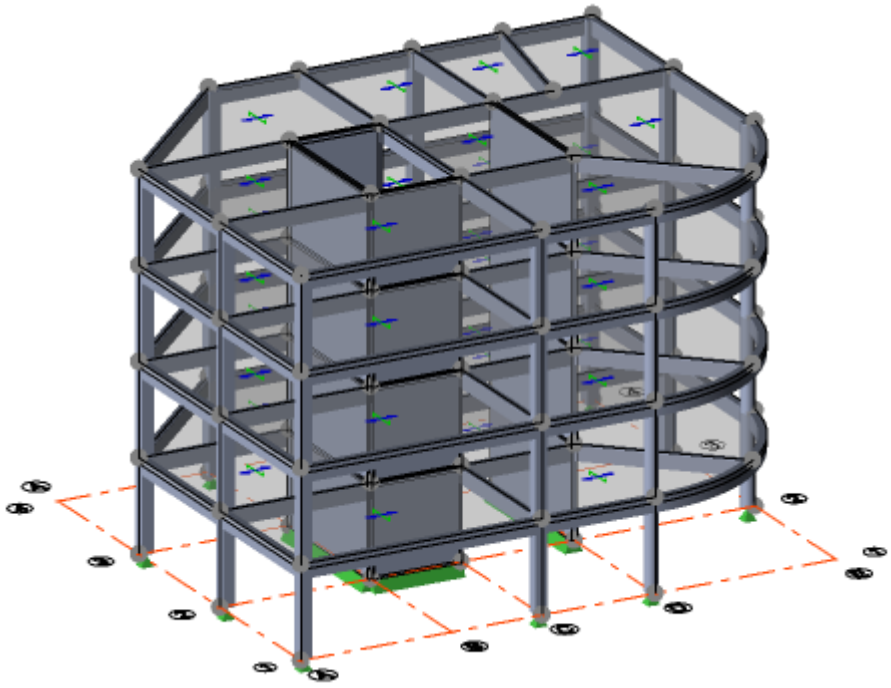
Pad bases, pile caps, strip bases, mats and piled mats can be designed from the **Foundations** tab. For further details, see:

- Pad base design workflow (page 281)
- Pile cap design workflow (page 291)

- Pad base, strip base and pile cap design forces (page 296)
- Mat foundation design workflow (US customary units) (page 311)
- Mat foundation design workflow (metric units) (page 297)
- Piled mat foundation design workflow (US customary units) (page 326)
- Piled mat foundation design workflow (metric units) (page 336)
- Concrete slab design fundamentals (page 255)
- Punching shear check fundamentals (page 264)

3.1 Concrete member design workflow

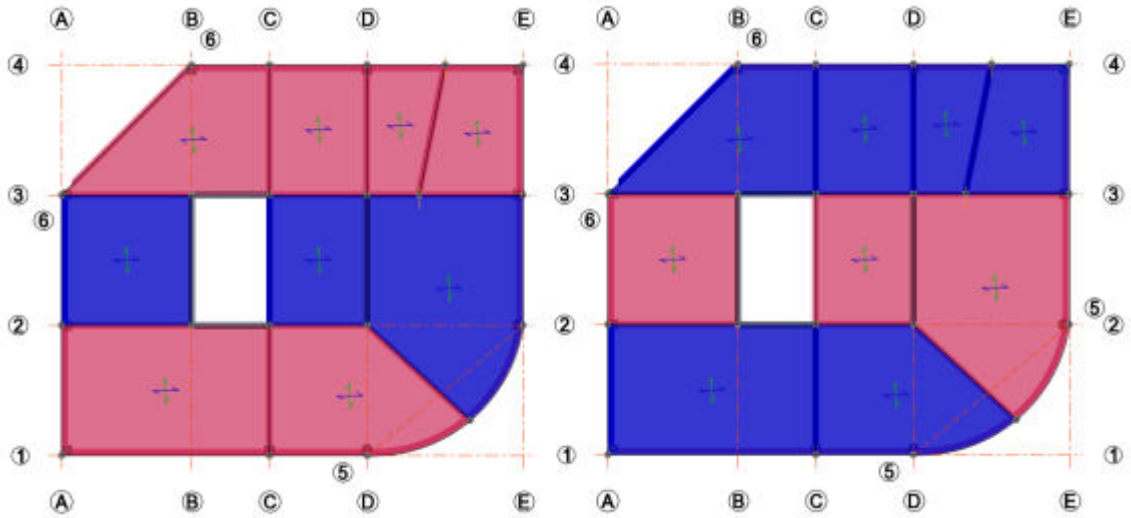
The following example illustrates the typical process to analyze and design all the beams, columns and walls in a concrete structure.



Set up pattern loading

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each

pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Set all beams columns and walls into autodesign mode

For the first pass, in order to get an efficient design at the outset, it is suggested that you set all members to "autodesign" with the option to select bars starting from Minima.

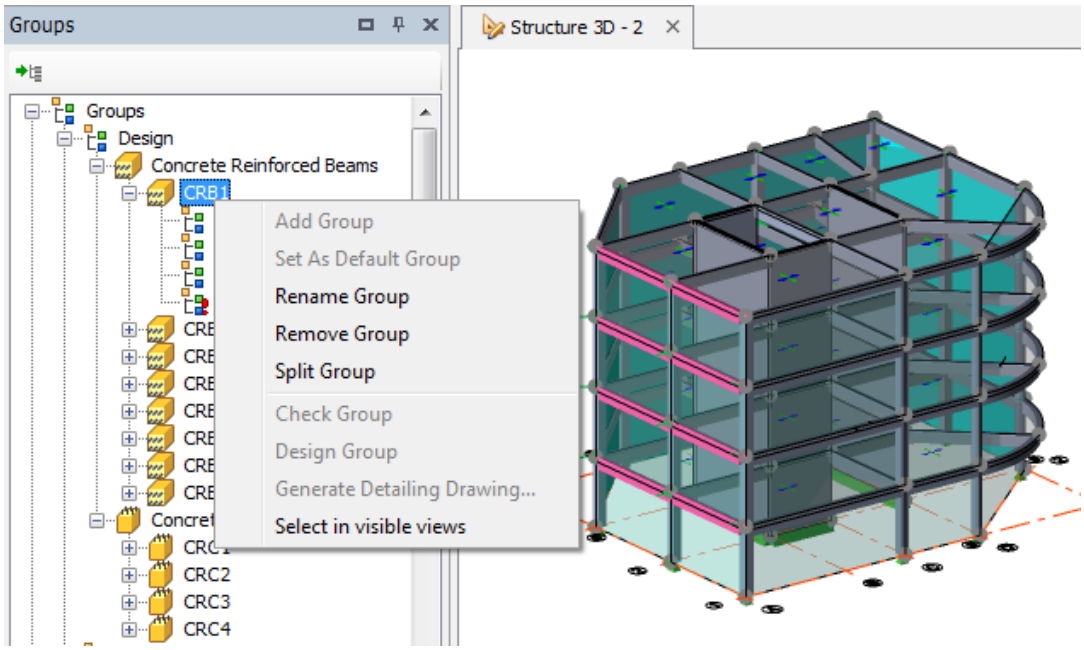
Related concept

The autodesign process for concrete members (page 143)

Review beam and column design groups

Provided that the concrete beam and column options are checked in Design Settings > Design Groups, the design groups shown in the Groups tab of the

Project Workspace will be applied in the beam and column autodesign processes.



Groups will initially have been established for members sharing the same geometry, but you should consider reviewing and amending them if required.

Review beam, column and wall design parameters and reinforcement settings

The member design parameters and reinforcement settings should be carefully considered prior to running the design.

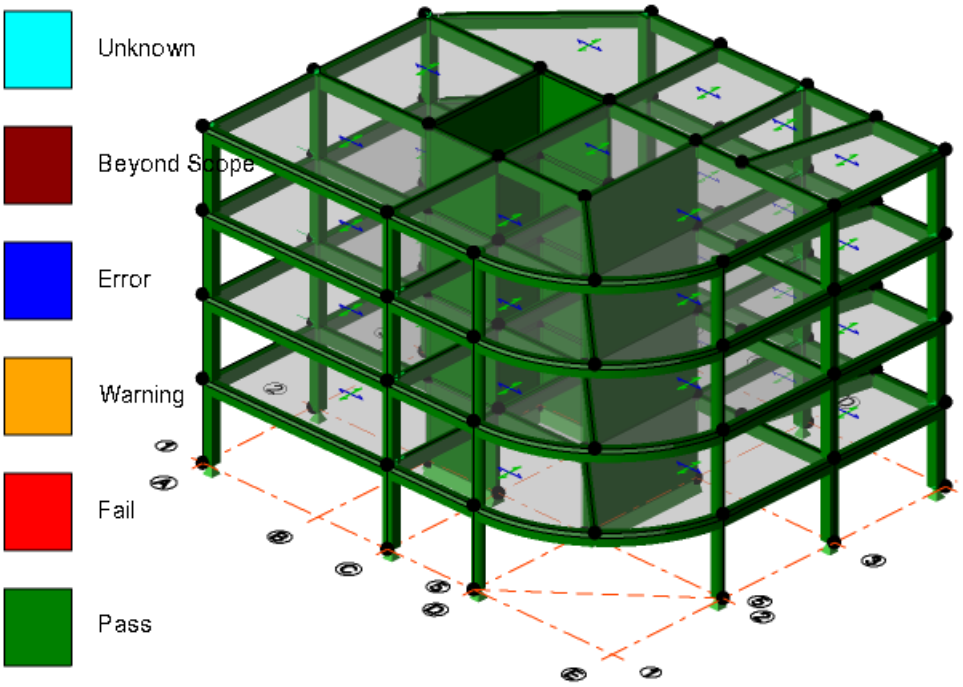
Perform the concrete design

By running Design Concrete (Static) from the Design ribbon, you effectively combine analysis and design (with the exception of slab design) into a single automated process.

Up to three separate analyses are automatically performed in order to generate the design forces required for the concrete beam, column, and wall design:

- 3D Analysis
- Grillage chasedown analysis
- FE chasedown analysis

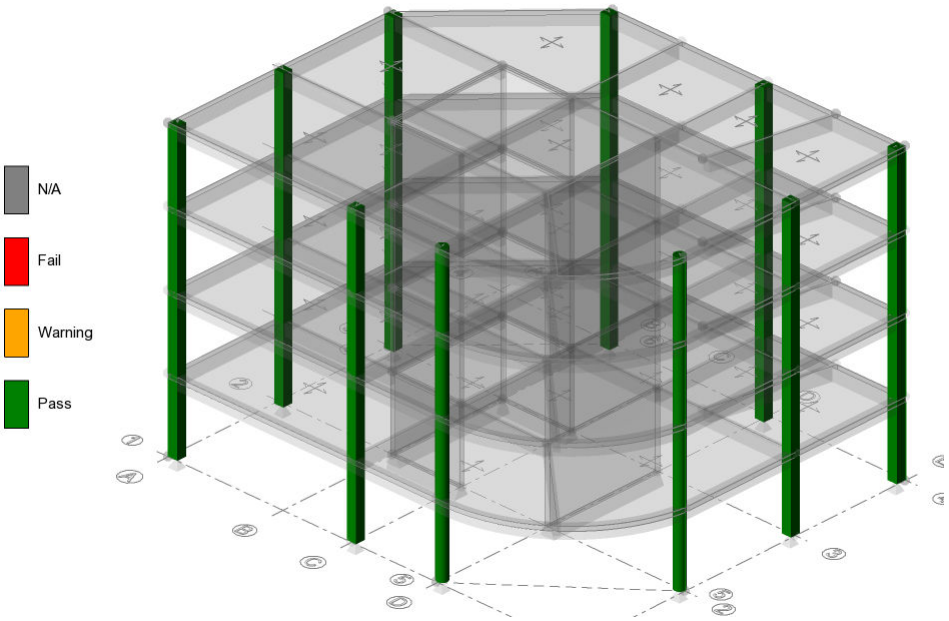
NOTE The sets of forces established from the FE chasedown analysis are considered by default for the design of each concrete member type. They can however be switched off should you decide that they are not required. The control for doing this for beams is located in Design Settings > Concrete > Beam > General Parameters. A similar control is provided for columns and walls also.



NOTE Reinforcement is designed, but member sizes are not changed during the design process.

Review stability issues

Issues relating to stability will be flagged in the Design branch of the Project Workspace Status tab. They can also be review graphically from Show/Alter State in the Review View.



Tabular results can be investigated from the Review View by clicking Tabular Data in the ribbon and then selecting the required View Type.

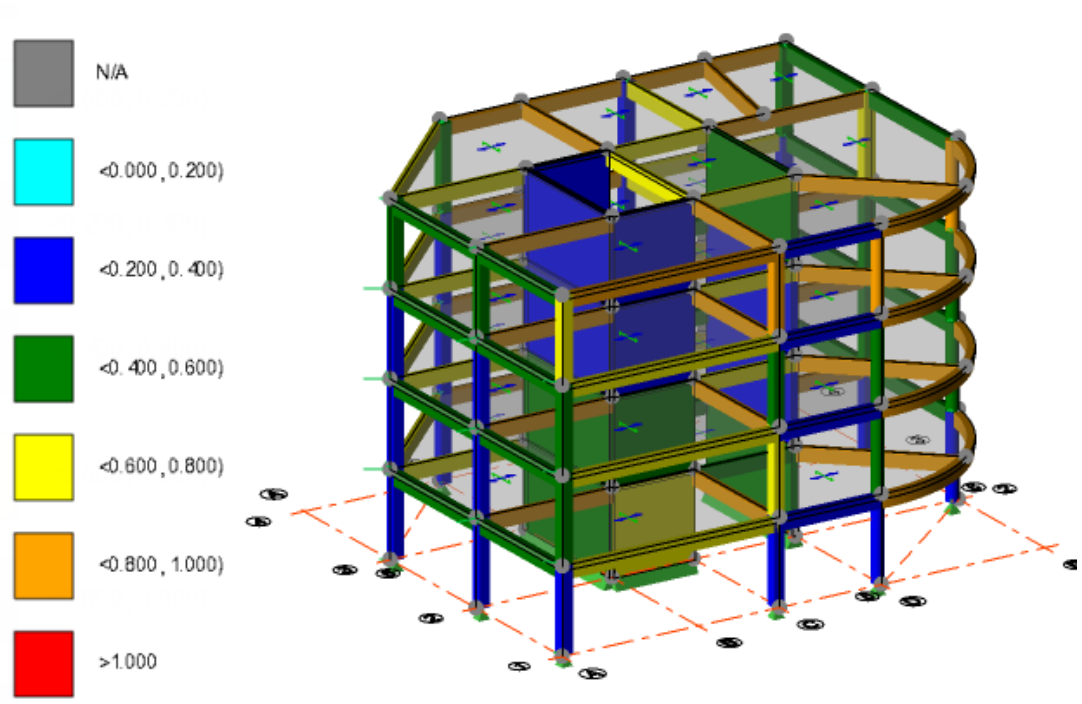
Structure 3D - 2 | Review Data | St. 1 (1) 2D

Wind Drift

Reference	Stack No.	Deflection Dir 1	Deflection Dir 2	Drift Dir 1	Drift Dir 2	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2
C1	4	2.9	-0.9	0.7	-0.2	4306.679	-17968.546	✓ Pass	✓ Pass
C1	3	2.2	-0.7	0.8	-0.2	3784.083	-13780.301	✓ Pass	✓ Pass
C1	2	1.4	-0.5	0.8	-0.2	3833.286	-12311.811	✓ Pass	✓ Pass
C1	1	0.6	-0.2	0.6	-0.2	5189.121	-12372.690	✓ Pass	✓ Pass
C2	4	2.3	-0.9	0.6	-0.2	5033.645	-17968.546	✓ Pass	✓ Pass
C2	3	1.8	-0.7	0.7	-0.2	4518.169	-13780.301	✓ Pass	✓ Pass

Review the design status and ratios

You can display the Design Status and Ratios from the Review View in order to determine if any remodelling is required.



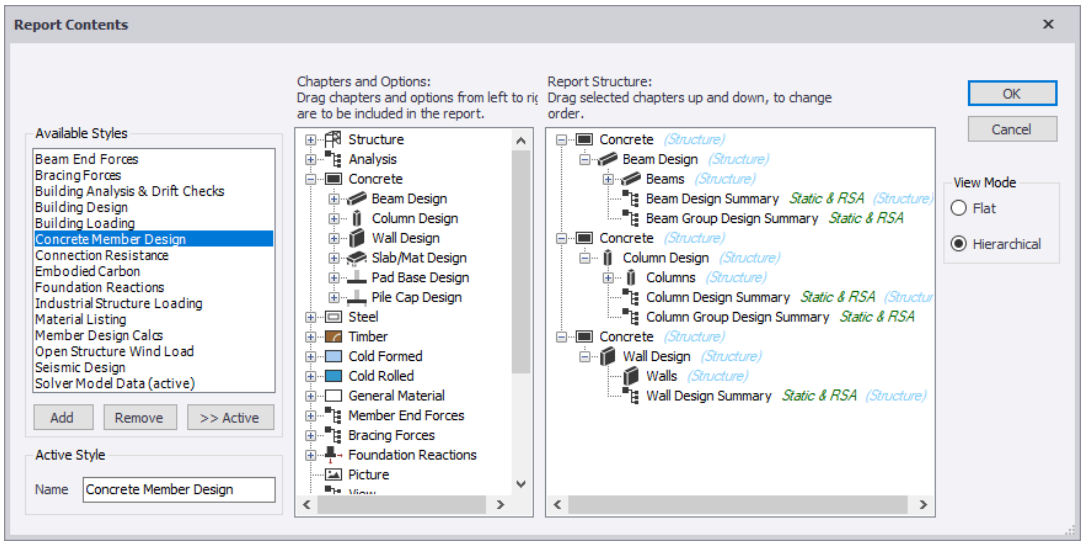
In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

If you make any changes, to see their effect simply re-run Design Concrete (Static) once more.

Create reports

To add concrete member design calculations to any model report, in the Report Contents dialog box, select the report style, then open the **Concrete**

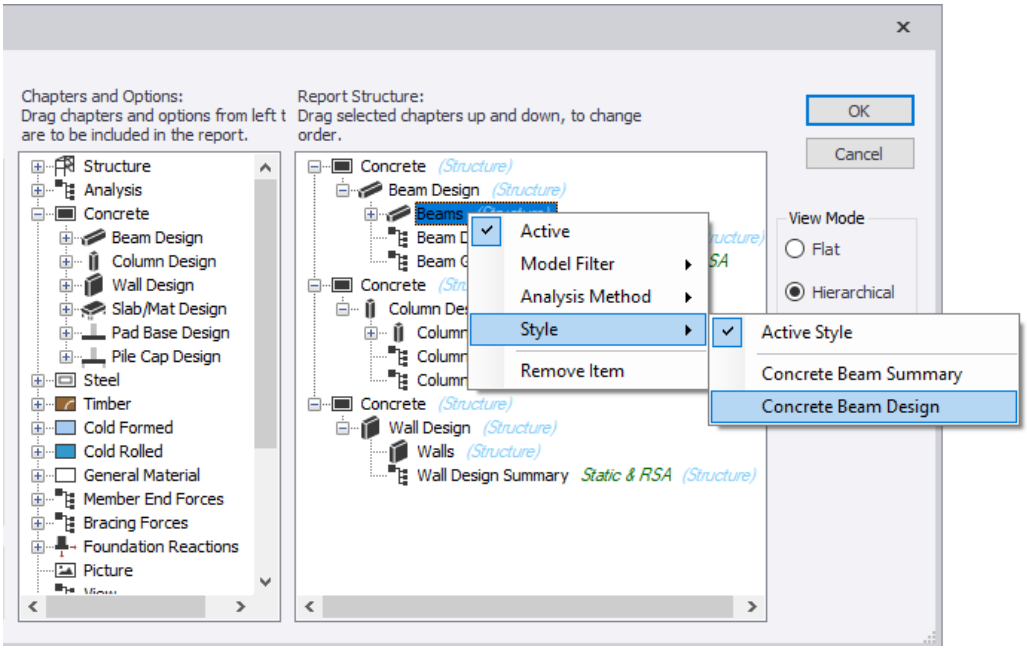
chapter and drag the **Beam Design**, **Column Design** and **Wall Design** chapters in the report structure.



If you expand the included chapters you will see the following sub-chapters:

- **Beams**, **Columns** and **Walls** - these sub-chapters are for reporting the individual member designs. They each have a member report style assigned which controls the level of output produced.

  The style can be selected from the right click context menu as shown below.



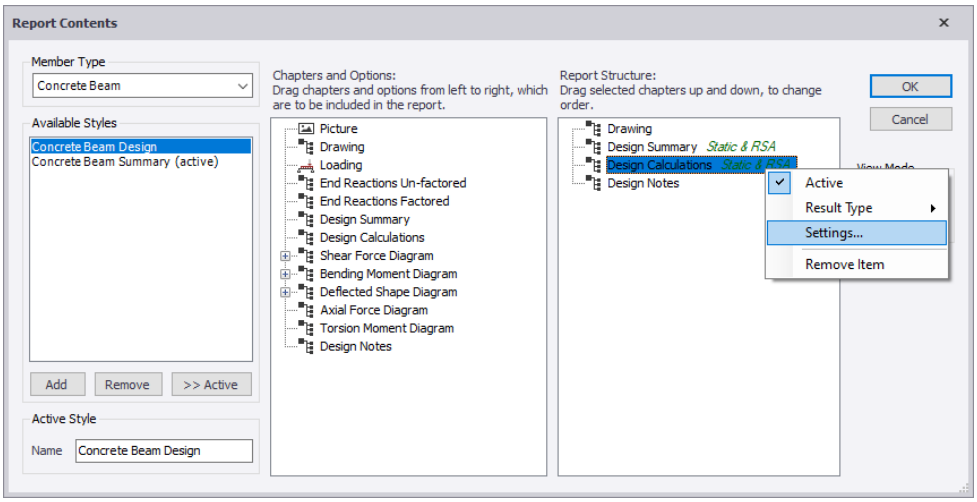
NOTE You can either choose a named style, or you can choose "Active Style" (in which case when you come to display the model report it will adopt whichever member report style is active at that time).

Each named style can be configured to include more or less detail, which can be done in a separate operation as follows:

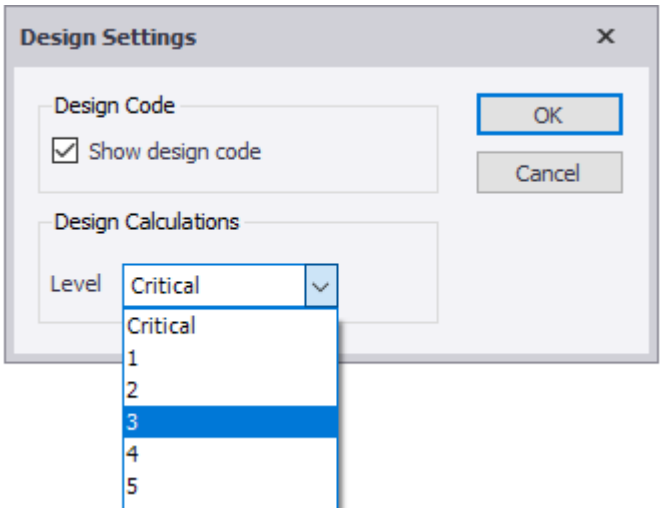
1. Close the above dialog box, and open the **Member Reports** dialog box.
2. Choose the member types in turn (e.g. Concrete Beam, Concrete Column and Concrete Wall) and select an existing style, (or add an new one if required).

   NOTE This dialog box is also used to review which member report style has been nominated as the "Active Style". You can switch this to the currently selected style by clicking the **>> Active** button.

3. Drag chapters and options into the report structure as required.
4. Configure the level of calculation detail, by right clicking on the Design Calculations chapter and selecting **Settings** from the context menu.



   - Choose the level of detail that the calculations are to be expanded to in the report, (critical, with no levels expanded being the minimum), then click OK to close the dialog box.



5. Click OK to close the **Member Reports** dialog box.

- **Beam Design Summary**, **Column Design Summary** and **Wall Design Summary** - these sub-chapters will add tabular summaries of the designs into the report which can be filtered as required.

Beam Design Summary

Static

Member Reference	Group Ref.	Span	Section	Grade	Length [m]	Utilization	Status
CB 1 V1	CRB12-D2	1	250x500	C32/40	5.000	0.681	Pass
CB 1 V1	CRB12-D2	2	250x500	C32/40	5.000	0.681	Pass
CB 4 V1	CRB12-D3	1	250x500	C32/40	5.000	0.700	Pass
CB 4 V1	CRB12-D3	2	250x500	C32/40	5.000	0.699	Pass
CB 3 V1	CRB12-D2	1	250x500	C32/40	5.000	0.674	Pass
CB 3 V1	CRB12-D2	2	250x500	C32/40	5.000	0.678	Pass
CB 2 V1	CRB12-D2	1	250x500	C32/40	5.000	0.673	Pass
CB 2 V1	CRB12-D2	2	250x500	C32/40	5.000	0.677	Pass
CB 1 V2	CRB13-D1	1	250x500	C32/40	7.071	0.831	Pass

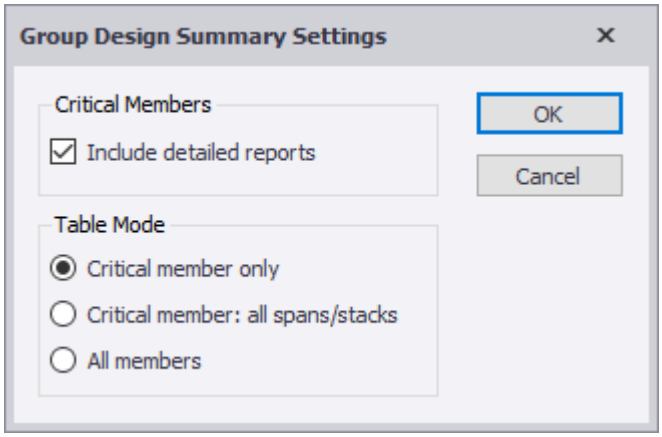
- **Beam Group Design Summary** and **Column Group Design Summary**- these sub-chapters will add tabular group design summaries into the report.

Beam Group Design Summary

Static

Reference	No. in Group	Critical Member	Spans	Critical Span	Section	Grade	Max. UR	Status
CRB1	4	CB 3 H9	3	3	250x500	C32/40	0.946	Pass
CRB12	4	CB 4 V1	2	1	250x500	C32/40	0.700	Pass
CRB13	4	CB 3 V2	1	1	250x500	C32/40	0.850	Pass
CRB14	4	CB 3 H1	2	1	250x500	C32/40	0.958	Pass
CRB15	16	CB 4 H4	1	1	250x500	C32/40	0.877	Pass
CRB16	8	CB 2 V4	3	3	250x500	C32/40	0.796	Pass
CRB17	8	CB 3 H10	2	2	250x500	C32/40	0.932	Pass

The group summary tables can optionally be followed by member design reports for either the critical member only, all spans/stacks of the critical member, or all members. These options are set by right clicking on the group design summary sub-chapter and selecting **Settings** from the context menu.



NOTE If detailed reports are included in the group design summary these adopt the active member report style.

To print the model report: on the ribbon, go to the **Report** tab, select the report style name and then click **Show Report**.

Concrete

Beam Design

Beams

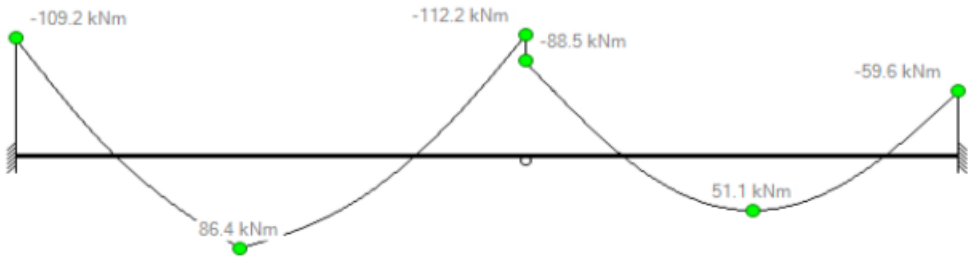
Cast-in-place Concrete

Head code: United Kingdom (Eurocode), design code: BS EN 1992-1-1 + UK NA (2004)

CB 1 H1

Bending Moment Diagram, First-order linear, Strength Factors

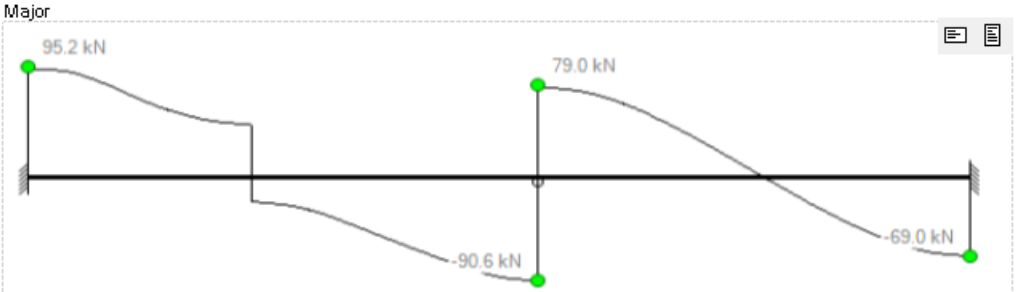
Major



Combination: 3 Str_1 - $\gamma_{G,sup}D+\gamma_Q I+\gamma_Q RI$, Major Moment for CB 1 H1

Shear Force Diagram, First-order linear, Strength Factors

Major



Combination: 3 Str_1 - $\gamma_{G,sup}D+\gamma_Q I+\gamma_Q RI$, Major Shear for CB 1 H1

Static

CB 1 H1 - 1 250x500 - Critical

Design summary bending top

Region	1	2	3
Analysis	Grillage Chase Down	3D Building Analysis	3D Building Analysis
Combination	3	3	3
M_{Ed}	**109.4** kNm	**0.0** kNm	**112.2** kNm
d	**452.0** mm	**454.0** mm	**449.5** mm
d_2	**48.0** mm	**48.0** mm	**48.0** mm
K / K'	**0.32**	**0.00**	**0.34**
z	**423.5** mm	**0.0** mm	**420.0** mm
$A_{s,req}$	**594** mm²	**0** mm²	**614** mm²

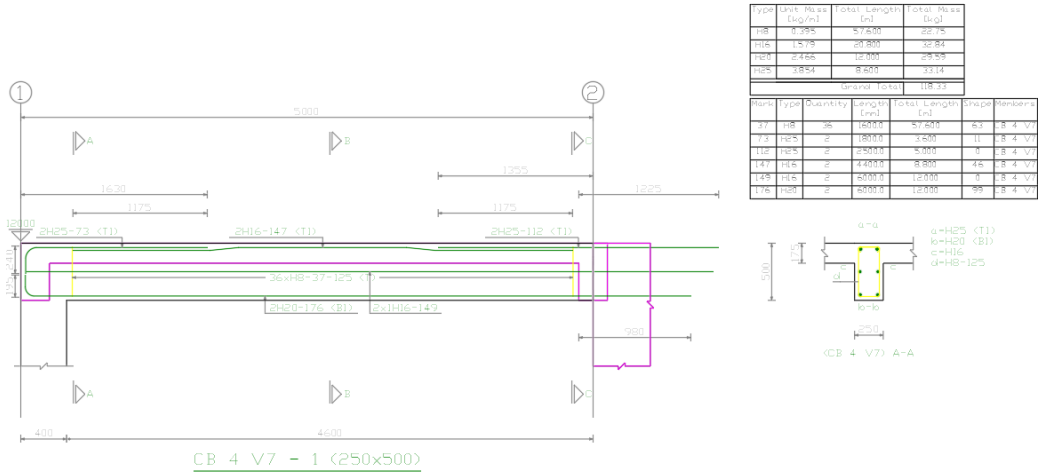
Create drawings and schedules

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of

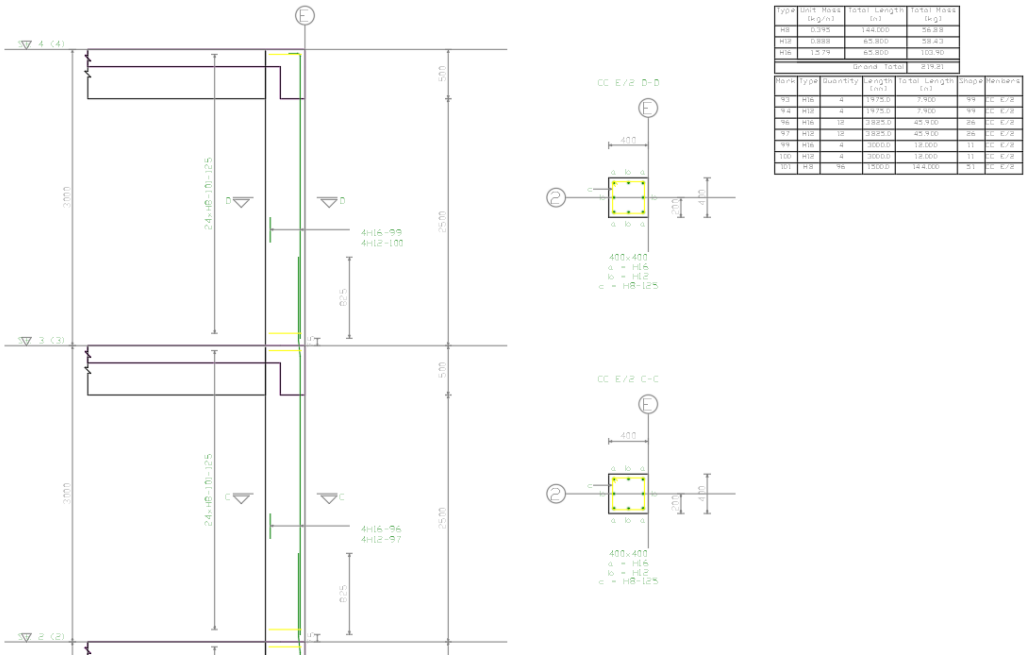
communication between the engineer and the detailer. Member detail drawings and member schedules are both available.

The member details drawings have separate variants for concrete beams, columns and walls:

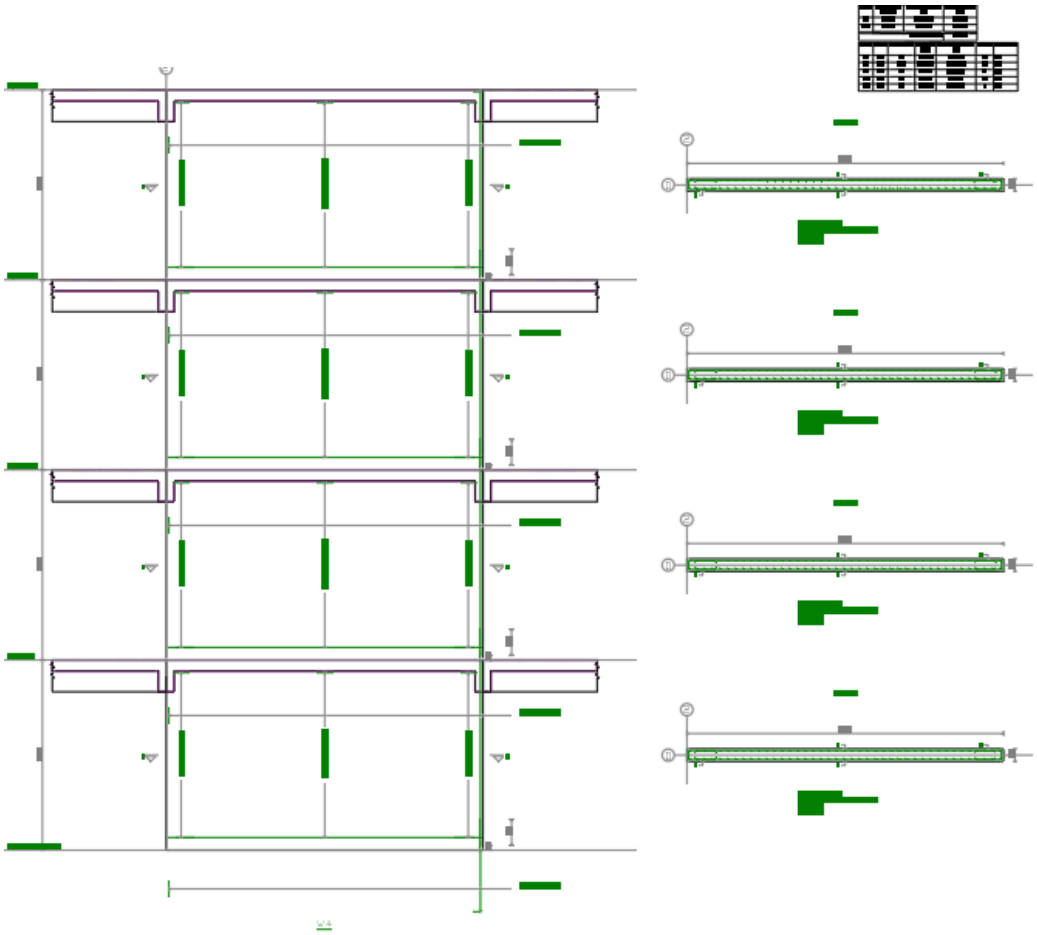
- **Concrete Beam Detail** - this drawing variant can be created by right-clicking on the required beam and selecting **Generate Drawing** from the context menu. The drawing displays the reinforcement in the beam elevation and section and includes a table of bar quantities.



- **Concrete Column Detail** - this drawing variant can be created by right-clicking on the required column and selecting **Generate Drawing** from the context menu. The drawing displays the reinforcement in the column elevation and section and includes a table of bar quantities.

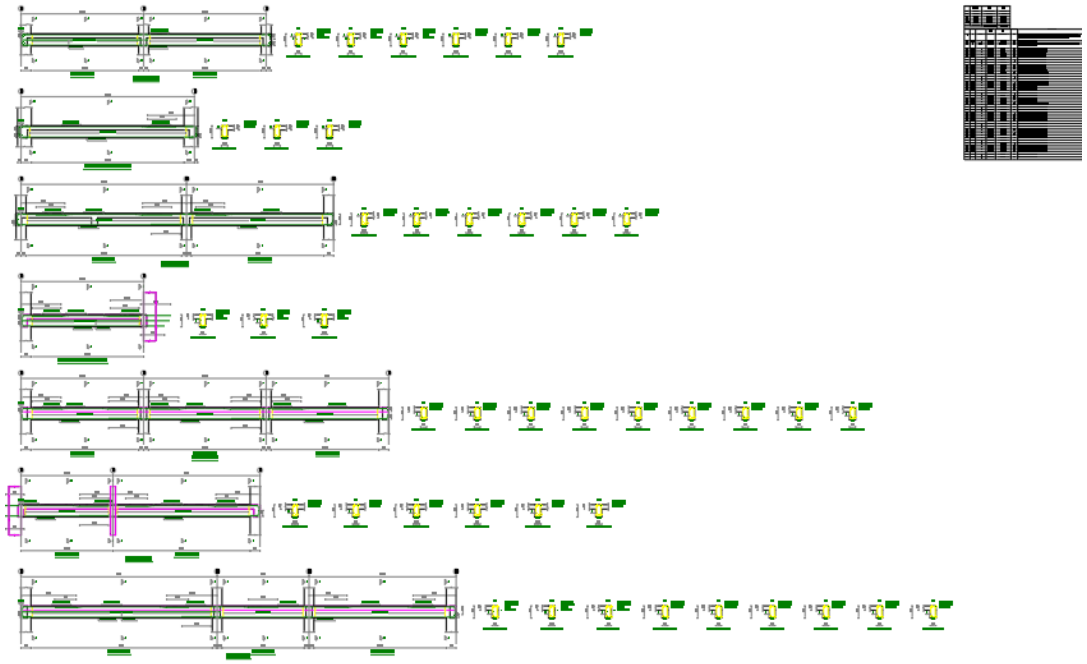


- **Concrete Wall Detail** - this drawing variant can be created by right-clicking on the required wall and selecting **Generate Drawing** from the context menu. The drawing displays the reinforcement in the wall elevation and section and includes a table of bar quantities.



The quickest way to create multiple beam, column or wall details is to use the .

A beam batch drawing showing all the beam details at a particular level is shown below.



The member schedule drawings have separate variants for concrete beams, columns and walls:

- **Beam Schedule** - to create this drawing variant, first open a 2D scene view of the level for which you want it to be created, then click **Beam Schedule**

on the **Draw** tab of the ribbon. Alternatively, to create beam schedules for multiple levels, use the .

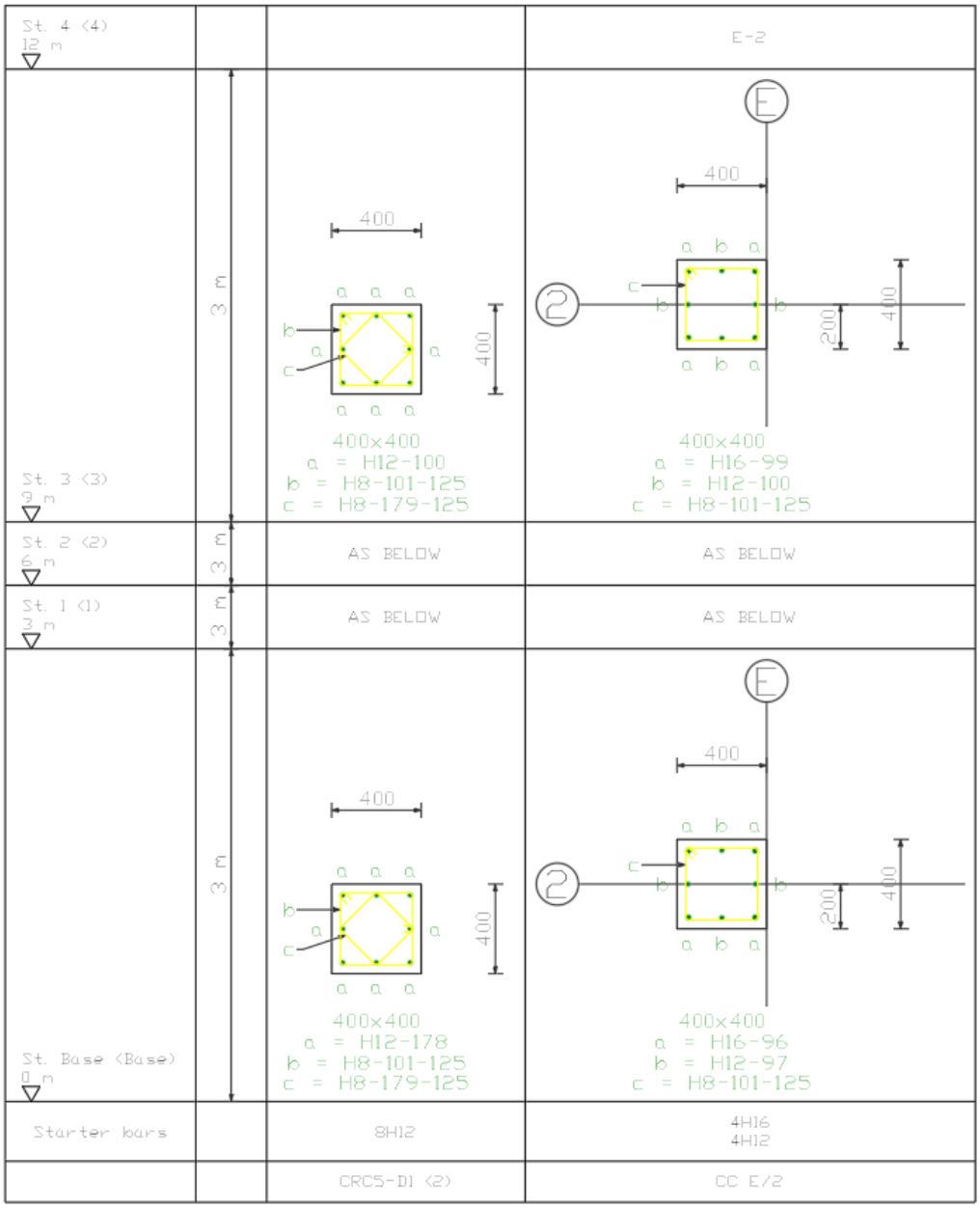
Concrete Beam Schedule - St_1 (1)

Mark	Size		Bottom Bars			Top Bars			Links				Side Bars	Revision / Notes
	Width	Height	Left	Centre	Right	Left	Centre	Right	Type	Left	Centre	Right		
CRB1-1	250	500		2H25-35		2H25-16 (25.00%)	2H12-7				H8-125 (2 legs, 100.00%)		2H16-62	First Span 4 beams in group 1 beam(s) in schedule Torsion links required
CRB1-2	250	500		2H16-38		2H25-15 (50.00%)					H8-300 (2 legs, 100.00%)			Internal Span 4 beams in group 1 beam(s) in schedule
CRB1-3	250	500		2H16-36		2H16-15 (25.00%)	2H12-7	2H20-16 (25.00%)			H8-300 (2 legs, 100.00%)			End Span 4 beams in group 1 beam(s) in schedule
CRB2-1	250	500		2H16-35		2H16-16 (25.00%)	2H16-7				H8-125 (2 legs, 100.00%)		2H16-62	First Span 4 beams in group 1 beam(s) in schedule Torsion links required
CRB2-2	250	500		2H16-36		2H16-1 (25.00%)	2H16-7	2H16-16 (25.00%)			H8-300 (2 legs, 100.00%)			End Span 4 beams in group 1 beam(s) in schedule
CRB13	250	500		2H16-31		2H16-16 (25.00%)	2H16-7	2H20-16 (25.00%)			H8-300 (2 legs, 100.00%)			Single Span 4 beams in group 1 beam(s) in schedule
CRB14-1	250	500		2H20-35		2H20-16 (25.00%)	2H16-7				H8-300 (2 legs, 100.00%)			First Span 4 beams in group 1 beam(s) in schedule
CRB14-2	250	500		2H16-36		2H25-1 (25.00%)	2H16-7	2H16-16 (25.00%)			H8-300 (2 legs, 100.00%)			End Span 4 beams in group 1 beam(s) in schedule
CRB15	250	500		2H20-37		2H25-16 (25.00%)	2H16-7	2H25-3 (25.00%)			H8-125 (2 legs, 100.00%)		2H16-61	Single Span 16 beams in group 4 beam(s) in schedule Torsion links required
CRB16-1	250	500		2H20-35		2H20-16 (25.00%)	2H16-7				H8-300 (2 legs, 100.00%)			First Span 8 beams in group 2 beam(s) in schedule
CRB16-2	250	500		2H16-38		2H20-1 (25.00%)	2H16-7				H8-300 (2 legs, 100.00%)			Internal Span 8 beams in group 2 beam(s) in schedule
CRB16-3	250	500		2H20-36		2H25-1 (25.00%)	2H16-7	2H16-16 (25.00%)			H8-300 (2 legs, 100.00%)			End Span 8 beams in group 2 beam(s) in schedule
CRB17-1	250	500		2H16-39		2H16-3 (25.00%)	2H16-7				H8-300 (2 legs, 100.00%)			First Span 8 beams in group 2 beam(s) in schedule
CRB17-2	250	500		2H25-36		2H25-1 (25.00%)	2H12-7	2H25-16 (25.00%)			H8-300 (2 legs, 100.00%)			End Span 8 beams in group 2 beam(s) in schedule

Bar bending details

Reference	Clear Span	Beam Column Wall	Clear Span	Beam Column Wall	Clear Span
1	Note 1		Note 1		
3	Note 2		Note 1		
7		Note 3	Note 3		
15	Note 1		Note 4		
16		Note 1			
31					
35			Note 5		
36				Note 6	
37				Note 2	
38	Note 6		Note 5		
39	Note 2		Note 5		
61		Note 7		Note 7	
62		Note 2		Note 7	

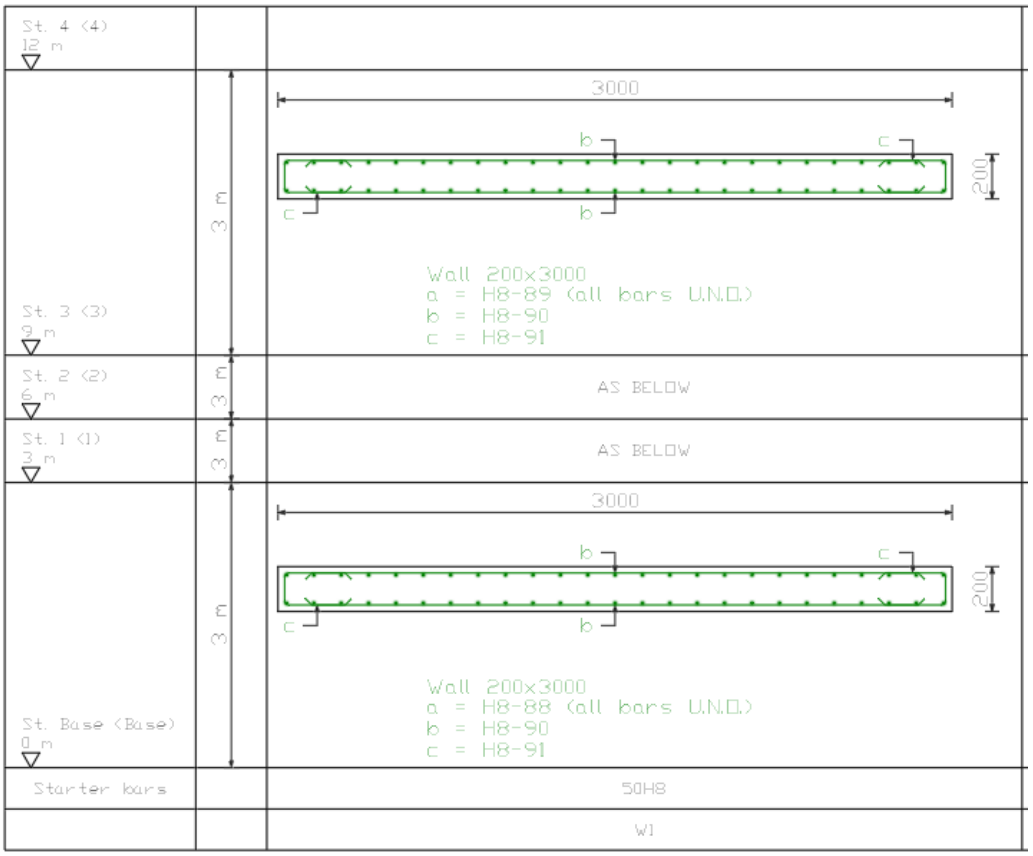
- **Concrete Schedule** - to create this drawing variant click **Column Schedule** on the **Draw** tab of the ribbon, then select the columns you want to include.



Type	Unit Mass [kg/m]	Total Length [m]	Total Mass [kg]
H8	0.395	652.800	257.86
H12	0.888	316.200	280.79
H16	1.579	65.800	103.90
		Grand Total	642.54

Mark	Type	Quantity	Length [mm]	Total Length [m]	Shape	Members
93	H16	4	1975.0	7.900	99	CC E/2
94	H12	4	1975.0	7.900	99	CC E/2
96	H16	12	3825.0	45.900	26	CC E/2
97	H12	12	3825.0	45.900	26	CC E/2
99	H16	4	3000.0	12.000	11	CC E/2
100	H12	20	3000.0	60.000	11	CC D/1, CC E/1, CC E/2
101	H8	288	1500.0	432.000	51	CC D/1, CC E/1, CC E/2
[illegible]		[illegible]	[illegible]	[illegible]	34	[illegible]
178	H12	48	3625.0	174.000	26	CC D/1, CC E/1
179	H8	192	1150.0	220.800	51	CC D/1, CC E/1

- **Wall Schedule** - to create this drawing variant click **Wall Schedule** on the **Draw** tab of the ribbon then select the walls you want to include.



Type	Unit Mass [kg/m]	Total Length [m]	Total Mass [kg]
H8	0.395	4244.400	1676.54
H12	0.888	952.000	845.38
		Grand Total	2521.91

Mark	Type	Quantity	Length [mm]	Total Length [m]	Shape	Members
87	H8	264	1575.0	415.800	34	W1, W2, W3, W4
88	H8	792	3425.0	2712.600	26	W1, W2, W3, W4
89	H8	264	3000.0	792.000	11	W1, W2, W3, W4
90	H8	96	2700.0	259.200	0	W1
91	H8	96	675.0	64.800	21	W1
102	H12	64	2700.0	172.800	0	W3
103	H12	192	925.0	177.600	21	W2, W3, W4
104	H12	128	4700.0	601.600	0	W2, W4

3.2 Concrete member autodesign

The design mode for each member is specified in its properties.

Autodesign (concrete beam)

- When Autodesign is selected an iterative procedure is used to select longitudinal bars for each bending design region on the beam, both top and bottom. Similarly an iterative procedure is used to select links/stirrups for each shear design region on the beam.
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

NOTE If concrete beams have been set to be designed using Design and detailing groups (concrete) (page 146), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Rationalization of Reinforcement

The Auto-design process returns a set of information about the reinforcement to be provided in each design region of the beam. The number and size of the longitudinal bars in the top and bottom of the beam is given as well as the size, number and spacing of the shear links/stirrups.

This information is then "rationalized" to give an arrangement of longitudinal reinforcement that provides a solution for the beam as a whole whilst still meeting the requirements of the individual design regions.

The rationalization process is carried out separately for the longitudinal bars in the top of the beam and those in the bottom of the beam.

The arrangement of shear links/stirrups is not rationalized. These can vary in size, spacing and number from region to region without having any impact on adjoining regions.

Autodesign (concrete column)

- When Autodesign is selected an iterative procedure is used to design both the longitudinal bars and links. This applies the spacing maximisation method which attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link/tie spacing.
- For rectangular, circular and parallelogram sections, if a single layer of reinforcement is not sufficient, autodesign will attempt a two layer solution. For circular sections, if a second layer is required you can control the minimum layer spacing via Design Settings (a larger spacing will be used if needed).
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

NOTE If concrete columns have been set to be designed using Design and detailing groups (concrete) (page 146), then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Autodesign (concrete wall)

- When Autodesign is selected an iterative procedure is used to determine the reinforcement. A spacing maximisation method is applied for both longitudinal bars and links/ties. This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link/tie spacing.
- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

Select bars starting from

This option appears for concrete members when Autodesign is selected. It sets the autodesign start point for both longitudinal bars and links/stirrups/ties.

The options are:

- Minima (default)
- Current bars

Selecting Minima removes the current arrangement and begins with the minimum allowed bar size from the selection order.

NOTE When a member is in check mode, it can still be autodesigned "on the fly" by choosing Design Member from the right-click menu. In this case it uses the **Select bars starting from** option currently assigned to the member for autodesign. (This property is hidden when in check mode, but can be confirmed if required by switching the member back to Autodesign mode.)

Specify the allowed bar size range

You can specify the minimum and maximum bar sizes that should apply to the different concrete members when they are auto-designed.

1. On the **Design** tab, click **Settings**.
2. Go to **Concrete** --> **Cast-in-place**
3. Locate the appropriate concrete member type page, **Beam**, **Column**, **Wall** etc.

4. Select the minimum and maximum bar sizes that apply to auto design.
5. Click **OK**

The new design options now apply to the current project. If you want them to apply to new projects as well, save the changes back to the active set.

3.3 Concrete beam and column groups

Why use design and detailing groups?

Concrete beams and columns and isolated foundations are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design and detailing purposes - to standardise designs, reduce processing time, and reduce the volume of output created.

In a manual process, the Engineer might select a number of sufficiently similar members to form a "design group" to carry out a single design that is sufficient for all members in the group. Using this single set of design results, they would then create sub-groups of the members in the design group to produce a set of output details for each of these sub-groups.

In Tekla Structural Designer, concrete design groups are analogous to the manually created design groups described above. Concrete detailing groups are analogous to the sub-groups.

A fixed set of rules are used to automatically determine member groups: for example beams must be of similar spans, columns must have the same number of stacks, bases must be of similar lengths in X and Y, and similar depths etc. The same rules also constrain manual group editing.

NOTE Grouped design and detailing is optional and can be deactivated if required:

From the **Design** tab, click Settings> Design Groups, then select or unselect the member types to be designed in groups.

What happens in the group design process?

When the option to design a specific concrete member type using groups is checked, for that member type:

- In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the reinforcement so obtained is copied to all other members in the group.
- These other members are then checked one by one to verify that the reinforcement is adequate for each and if this proves not to be the case, it is increased as necessary and the revised reinforcement is copied to all members in the group.
- This process continues until all members in the group have been satisfactorily checked.
- A final check design is then carried out on each group member. During this process peak and individual utilizations are established.

Design group requirements

Concrete member design groups are formed according to the following rules:

Member type	Design group rules
Concrete beam	• A beam element may be in only **one** design group. • Design groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length. • All beam elements in the group must have identical material properties and nominal cover. • All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.
Concrete column	• A column element may be in only **one** design group. • All column elements in the group must have an identical number of stacks. • For each individual stack all column elements in the group must have an identical cross section, and stack length. • All column elements in the group must have identical material properties and nominal cover.

Member type	Design group rules
Pad base	• A pad base may be in only **one** design group. • Each base in the group must have an identical cross section and depth. • Each base in the group must have identical eccentricities in X and Y. • Each base in the group must have identical material properties and nominal cover.
Pile cap	• A pile cap may be in only **one** design group. • Each pile cap in the group must have an identical cross section and depth. • Each pile cap in the group must have identical eccentricities in X and Y. • Each pile cap in the group must have identical material properties and nominal cover

Detailing group requirements

Each parent design group is sub-divided into one or more detailing groups.

Although there can be a "1 to 1" relationship between a design group and a detailing group, in practice there will often be a "1 to many" relationship as each design group is likely to require several detailing groups to allow for differences in the connected geometry.

Detailing groups are formed for the different concrete member types based on the following rules:

Member type	Design group rules
Concrete beam	• A detailing group may be associated with only **one** parent design group. • A beam element may be in only **one** detailing group. • Detailing groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • The cross section, including flange width where appropriate, span length and material properties in span • *"i"* of **all** beam elements in the group must be identical .

Member type	Design group rules
	• All beam elements in the group must have identical plan offsets. • All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans. • All beam elements in the group must have identical inclinations. • The support types and sizes, including the attached structure above and below the beam element, must be identical in all beam elements in the group however different support types and sizes in individual multi-span continuous beams are acceptable i.e. support *i* in beam element *j* must be identical to support *i* in all other beam elements in the group but supports *i* and *i+1* in beam element *j* may be different.
Concrete column	• A detailing group may be associated with only **one** parent design group, • A column may only be in **one** detailing group, • All columns in the detailing group must have an identical number of stacks, • All columns in the group must have an identical cross-section, rotation and alignment/snap levels/offsets in stack 'i'. In a multi-stack column, the cross-section may be different in each stack, i.e. the cross-section in span 'i' may be different to that in span 'j'. • Stack 'i' and stack 'i+1' must be co-linear for **all** columns, OR must be non-co-linear with an identical degree of non-co-linearity for all columns. The exact inclination must be the same for stack 'i' in all columns. • At every level each column is considered to be either "internal" or "external" (depending on if it has beams framing into it on all four sides, or not). These settings do not have to be identical for columns to be in the same group, but only if you have selected the option: Provide ties through floor depth for internal columns in Design Options > Concrete > Column > Detailing Options.
Pad base	• A detailing group may be associated with only **one** parent design group. • A base may be in only **one** detailing group.

Member type	Design group rules
	• The attached column cross-section above the base must be identical for all bases in the group however different support types are acceptable.
Pile cap	• A detailing group may be associated with only **one** parent design group. • A pile cap may be in only **one** detailing group. • The attached column cross-section above the base must be identical for all pile caps in the group however different support types are acceptable.

Group management

Automatic Grouping

Concrete beams and columns are grouped automatically.

In **Model Settings** > **Grouping** the user defined Maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

Detailing groups cannot be edited manually.

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup ALL Model Members

If you have made changes in Design Settings that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Model Editing and Group Validity Checks

When new beam elements are created when a "split" or "join" command is run the resulting beam elements are automatically placed in existing design and detailing groups [or new groups created].

3.4 Concrete beam design fundamentals

Beam design settings

Prior to running beam design you should review the design settings by selecting **Design > Settings > Concrete > Beam**.

These settings are used to control:

- Reinforcement settings, e.g.
  - Minimum and maximum bar sizes
  - Minimum and maximum spacing values
  - Maximum short span length
- Detailing settings
- Bar patterns
- Link settings
- General parameters (depending on the regional code being designed to)

For further details, see: Beam design settings

Concrete type

While you can apply both normal and lightweight (LW) concrete in the beam properties, beam design using lightweight concrete is only available for Eurocodes.

When using other regional codes beams can only be designed using normal weight concrete.

LW density classes and grades

6 Density classes (1.0, 1.2 ..... 2.0) are available and 15 default grades are provided; 5 in each of the density classes: 1.6, 1.8 and 2.0.

- For example the grade name “LWAC30/37-DC1.8” denotes; LWAC = Lightweight aggregate concrete; 30/37 = Strength class; DC1.8 = density class.
- Custom LW grades can be added for which note that new LW-specific property η_1 must be specified.

NOTE LW grades can be reviewed, edited and applied via Review View > Show/Alter State Material Grade Attribute.

Deflection control (ACI/AISC)

Tekla Structural Designer implements both a simplified, and also a more rigorous method for controlling deflection.

The actual method applied to a specific beam will depend upon whether it is required to support sensitive finishes.

When the rigorous method is applied, immediate short-term deflections, and also long-term deflections resulting from creep and shrinkage of flexural members are considered.

Structure supporting sensitive finishes

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 Structure supporting sensitive finishes under the Design Control heading in the beam properties.

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.

The simplified method

For those beams that have not being chosen to support sensitive finishes, deflection is controlled by the simplified method of limiting the span to depth ratio.

This check can be satisfied by providing a total beam depth which exceeds h_{min}, (where h_{min} depends on the clear span and beam end conditions).

If this check is not satisfied the beam is then rechecked by the rigorous method.

The rigorous method

For beams that do not meet minimum thickness requirements determined by the simplified method, or that support sensitive finishes, deflections should be calculated by rigorous method.

The parameters listed below directly affect the rigorous calculation method and hence require consideration:

1. Does the beam support sensitive finishes?
    - If the Structure supporting sensitive finishes beam property (located under the Design Control heading) is cleared the simplified method may suffice in which case rigorous calculations are not required.
2. Can the beam flanges be taken into account?
    - Selecting Consider flanges (located under the Design Control heading) can assist in reducing the calculated deflections.
3. Does the beam contain compression steel?

- In the current release of the program it has been assumed that the beam always contains compression steel for the rigorous method deflection calculations.

4. 4. Are the deflection limits appropriate?
   - These limits are editable in the beam properties, the default ratios being span/360 for live load, and span/480 for total load affecting sensitive finishes.
   - The default span\over limits tend to produce conservative results.
   - You are also given the flexibility to specify absolute deflection limits as an alternative, or in addition to the above span\over limits.
5. Is the long term deflection period correctly specified?
   - The default value is 5 years, but this can be edited via Design Options > Beam > General Parameters
6. Is the time at which brittle finishes are introduced correctly specified?
   - The default value is 1 month, but this can be edited via Design Options > Beam > General Parameters
7. Are the percentages of dead load applied prior to brittle finishes and long term live load correctly specified?
   - The deflection check takes account of these percentages, (defaulting to 50% of dead load applied prior to brittle finishes and 33% of long term live load). You are able to adjust each of these by selecting the loadcase name from the left hand side of the Loadcases dialog and then adjusting the value.

Deflection control (AS 3600)

Tekla Structural Designer controls deflections either by limiting span to depth ratios, or by applying the simplified method. The choice of method being set via Design Options > Concrete > Beam > General Parameters.

The 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 (Ecj) and an effective second moment of area of member (I_{ef}).

When the simplified method is applied, the shrinkage parameters that are required are specified on the Design Options > Concrete > Beam > General Parameters page.

Limiting span to depth ratios method

The deflection of reinforced concrete beams is not directly calculated and the serviceability of the beam is measured by comparing the calculated limiting effective span/effective depth ratio (L_{ef}/d)

Structure supporting sensitive finishes

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 **Calculate deflection after installation of finishes** under the Design Limits heading in the beam properties.

When this option has been selected an additional span\over or absolute limit can be specified and checked against in the beam properties.

• beams not required to support sensitive finishes adopt the simplified method.

• beams required to support sensitive finishes adopt the rigorous method.

Parameters affecting deflection

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. 1. Increase reinforcement if deflection check fails
    - If this beam property (located under the Design Control heading) is cleared and the check fails, then the failure is simply recorded in the results
    - If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.
2. Consider flanges
    - Checking this beam property (located under the Design Control heading) can assist in satisfying the deflection check.

Deflection control (Eurocode BS and IS)

Tekla Structural Designer controls deflection by comparing the calculated limiting span/effective depth ratio L/d to the maximum allowable value $(L/d)_{max}$

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. Increase reinforcement if deflection check fails
    - If this beam property (located under the Design Control heading) is cleared and the check fails, then the failure is simply recorded in the results

    If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.
2. Structure supporting sensitive finishes (Eurocode only)

- This beam property (located under the Design Control heading, and on by default) is used to control the value of the f2 parameter used in the deflection check. When unchecked f2 will be taken as 1.0.

3. Consider flanges
   - Checking this beam property (located under the Design Control heading) can assist in satisfying the deflection check.

Ignore lateral instability (Eurocode)

This option (located under the Design Control heading) allows you to consider or ignore lateral instability for slender spans to EC2 clause 5.9(1) (off by default). When the option is checked on the slender span check is excluded from design.

Consider flanges

Flanged beam properties are specified by selecting **Consider flanges**, located under **Analysis & design control** in the **Properties** window.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "Γ" shaped with a slab on only one side of the beam.

The characteristic behavior of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Related concept

Flanged concrete beams (page 159)

Design parameters (Eurocode only)

Located under the Design parameters heading in the beam properties, the following parameters relating to shrinkage and creep can be specified for individual members.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine

the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + Ψ_2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and Ψ_2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

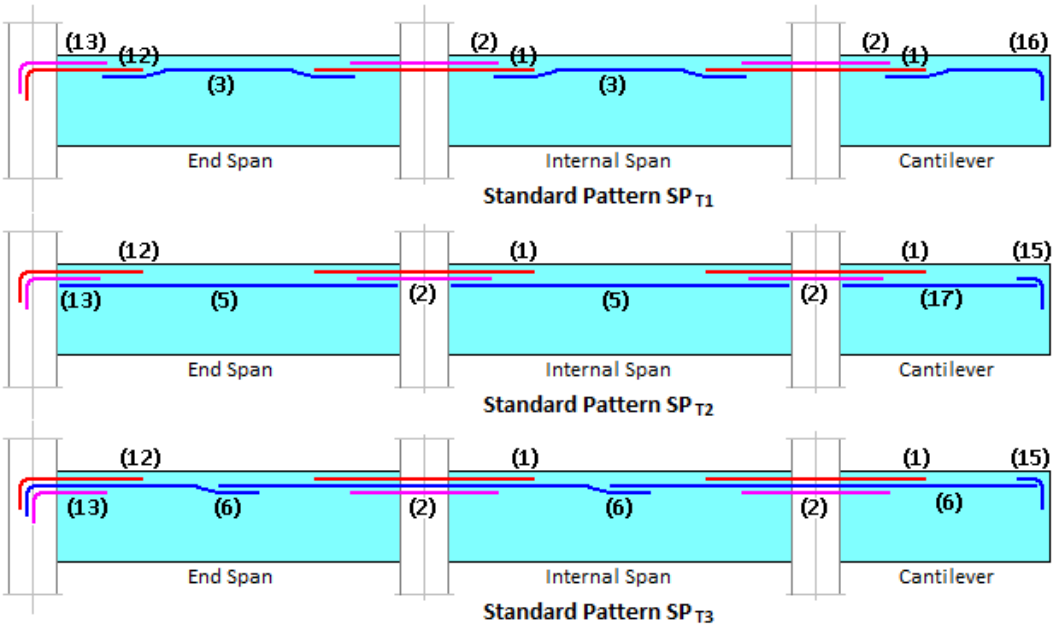
Nominal cover

In the beam properties, 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.

Reinforcement - longitudinal bar patterns

Under the reinforcement heading in the beam properties, there are three Standard Patterns available for top reinforcement, SP_{T1}, SP_{T2} and SP_{T3} and two Standard Patterns available for bottom reinforcement, SP_{B1} & SP_{B2} as illustrated in the figures below.

Standard Patterns of Top Reinforcement



The bars used in the Standard Top Patterns are:

(1) Straight bar extending to approximately 25% or 33% of each span (end points of this bar are determined by the design region settings)

(2) Straight bar extending to approximately 10% of each span (end points of this bar are determined by the design region settings) - if required by the design

(3) Double cranked bar lapped with bar (1)

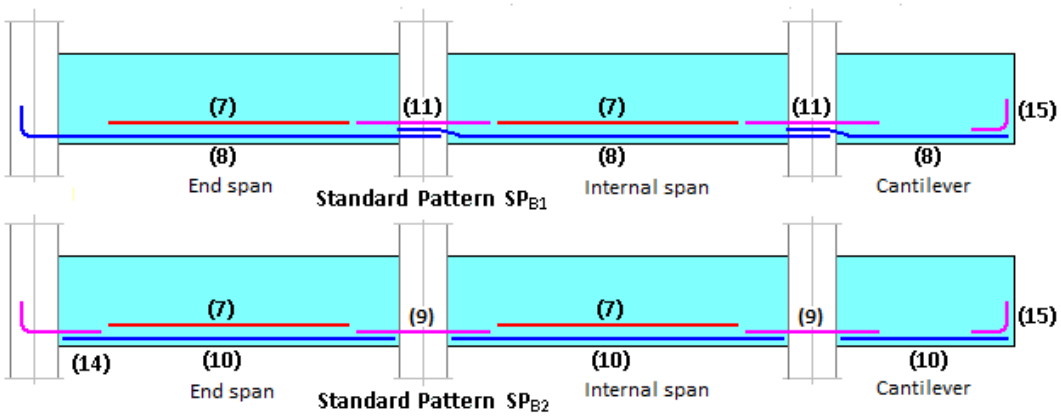
(5) Straight bar running approximately from face to face of beam supports

(6) Single cranked bar running from center span to center span with the option to merge bars if they are the same size and number to extend the bar over several spans

(12) Bob bar

(13) Bob bar

Standard Patterns of Bottom Reinforcement



The bars used in the Standard Bottom Patterns are:

(4) Bar with a bob at each end

(7) Straight bar with a length approximately 70% of span - if required by the design

(8) Single cranked bar extending over several spans or over one span only and lapped within a support - with bob if it continues over an end span.

(9) Straight bar

(10) Straight bar running approximately from face to face of beam supports

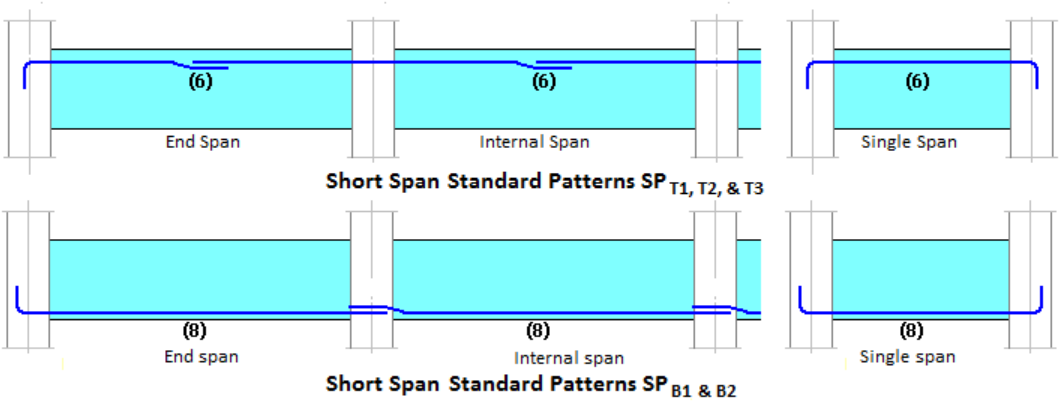
(11) Straight bar

(14) Bob bar

Modified versions of the above standard patterns are applied for use in single spans and in cantilever spans where no backspan beam is present.

For short span beams, it becomes uneconomic and impractical to lap bars in beams. These facts coupled with the anchorage lengths that are required make the use of multiple design regions for the longitudinal reinforcement unnecessary. To cater for this a short span beam length can be defined in **Design Options > Beam > Reinforcement Settings** and the bar patterns adopted for such short spans are as shown below:

Standard Patterns of Reinforcement for Short Span Beams



Flanged concrete beams

Flanged beam properties are specified by selecting **Consider flanges**, located under **Analysis & design control** in the **Properties** window.

Typically, flanged beams can be either "T" shaped with a slab on both sides of the beam or "Γ" shaped with a slab on only one side of the beam.

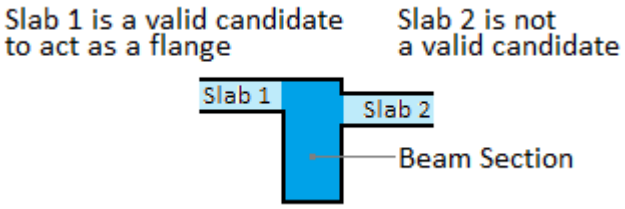
The characteristic behavior of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

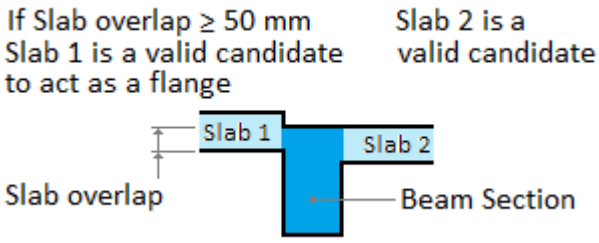
This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Modeling requirements

If a slab is present (and provided that a user defined flange has not been specified), the program automatically validates the slab as a potential candidate for being a beam flange using a number of criteria, the main ones being;

- the slab can be on one or both sides of the beam but
  - it must extend for a distance ≥ the slab depth from the vertical face of the beam and
  - it must extend for the full span length of the beam
- the slab must be a reinforced concrete slab
- if there are slabs on both sides of the beam, they may be of different depths and these depths may vary along the length of the beam
- the slab(s) must have a top surface coincident with the top surface of the beam or there must be a slab overlap ≥ 50 mm (2 inches) as shown in the following illustrations.





The effective width of any valid slab on each side of the beam, $b_{eff,i}$, is calculated and the **Flange width** results that are appropriate at the mid-span length point are displayed along with the **Flange depth**.

Include flanges in analysis

When **Consider flanges** is selected, an **Include flanges in analysis** checkbox becomes available.

Selecting this option allows the flanged beam section properties to be considered in the analysis, stiffening the beam and reducing the deflection.

Isolated beam (ACI/AISC)

ACI 318 clause 8.12.4 states:

"Isolated beams, in which the T-shape is used to provide a flange for additional compression area shall have a flange thickness not less than one-half the width of web and an effective flange width no more than four times the width of web."

When **Consider flanges** is selected, an **Isolated Beam** checkbox is displayed to control whether or not the above code limit is applied. When the check is performed, if the flange geometry does not meet the above requirements the flanges are ignored.

NOTE Our understanding is that while this limit usually applies to precast beams, it is not usually applied to in-situ construction. Therefore, by the default the **Isolated Beam** checkbox is cleared, which means that the above check is not performed.

Effective Width of flanges (ACI/AISC)

For ACI 318-08 and ACI 318-11

For "T" shaped flanged beams the effective flange width, b_{eff}, is given by :

$b_{eff} = MIN(L/4, 16*h_f + b_w, b_1 + b_2 + b_w) - O_{wi}$

For beams with slab one side only, the effective flange width, b_{eff} is given by :

b_{eff} = MIN(L/12+ b_w, 6*h_f + b_w, b_i + b_w) - O_{wi}

where

L = span length

IF construction is continuous:

= distance of center-to-center of supports

ELSE

= MAX(clear span + h, distance between centers of supports)

b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centerline of any supporting steel beam

O_{wi} = the user specified allowance for an opening

For ACI 318-14

All limits on the flange width apply to the overhangs on each side of the beam. It is also clarified in this version that the clear span should be used in these calculations.

$b_{eff,i}$ = MIN(l_n/8, 8*h_f , b_i) - O_{wi}

Effective Width of flanges (Eurocode)

The effective width of the compression flange is based on L_0, the distance between points of zero bending moment.

For flanged beams the following values of L_0 are to be used;

For a simply supported beam L_0 = L

For a continuous beam, the value of L_0 may be obtained using the following simplified rules;

End span of a continuous beam with a pinned end support L_0 = 0.85*L

End span of a continuous beam with a fixed end support L_0 = 0.70*L

Internal span of a continuous beam L_0 = 0.70*L

where

L = the clear length of the span under consideration

The effective flange width, b_{eff}, is given by;

b_{eff} = b_w + ∑$b_{eff,i}$

where

$b_{eff,i}$ = the effective width of the flange on side i of the beam

= MIN[0.2*L_0, b_i,(0.2*b_i + 0.1*L_0)] - $O_{w\,i}$

where

L_0 = the distance between points of zero moment as defined above

b_i = 0.5 * the clear distance between the vertical faces of the supports for the valid concrete slab on side i of the beam or from the vertical face of the beam to the centreline of any supporting steel beam

b_w = the width of the beam

O_{wi} = the user specified allowance for an opening

NOTE If the slab thickness varies on each side of the beam, the thinner value is used in calculating the beam properties.

The above calculation for b_{eff} is also used for "Γ" beams with a slab on only one side although in this case, b_1 or b_2 as appropriate is = 0.

3.5 Concrete column design fundamentals

Column design settings

Prior to running column design you should review the design settings by selecting **Design > Settings > Concrete > Column**.

These settings are used to control:

- Reinforcement layout, e.g.
  - Minimum and maximum bar sizes
  - Minimum and maximum spacing values
  - Confinement reinforcement
- Detailing settings
- General parameters (depending on the regional code being designed to)

For further details, see: Column design settings

Concrete type

While you can apply both normal and lightweight concrete in the column properties, column design using lightweight concrete is currently beyond scope.

Apply rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections.

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 positions of releases in analysis model are affected, and member start and end points for design are also adjusted.

The Apply rigid zones setting is located under the Design control heading in the column properties.

Design parameters (Eurocode only)

Located under the Design parameters heading in the column properties, the following parameters relating to shrinkage and creep can be specified for individual members.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + Ψ_2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and Ψ_2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Confinement reinforcement

Located under the Confinement reinforcement heading in the column properties, the **Provide support regions** setting determines the way each stack is divided into regions for the purpose of designing the confinement reinforcement.

- Checked - confinement reinforcement is designed separately in three regions.
- Cleared - the same confinement reinforcement is designed for the whole stack.

Slenderness

Located under the Slenderness heading in the column properties, the significant parameter within the slenderness criteria is a choice of how the column is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.

- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the **User input value** option, or it is calculated in accordance with the requirements of the selected design code.

Stiffness

The stiffness settings affect the calculation of clear height, also referred to as the unsupported or unrestrained length (depending on the regional code being worked to) which is the clear dimension between the restraining beams at the bottom of the stack and the restraining beams at the top of the stack. The unsupported length may be different in each direction.

Effective concrete beams

An effective concrete beam is one which provides stiffness at a restraint position. A concrete beam is only considered effective if it is "fixed" at the position where it joins to the column. Concrete beams are only effective in a direction if they are within 45° of that direction, and therefore no concrete beam can be effective in both directions. A concrete beam is only effective if its angle to the horizontal is 45° or less.

A concrete beam only restrains the end of the stack if it is within the depth of the stack section from the end of the stack, and if its centre is nearer to this end of the stack than the far end. Therefore, at a node at a stack join, if the top of the beam is below the node by a dimension greater than the depth of the stack below the node, it is not considered. Similarly, if the bottom of the beam is above the node by a dimension greater than the depth of the stack above the node, it is not considered.

Effective flat slabs/other types of slab

When determining the unsupported length, if no effective beams are found at the end of a stack, Tekla Structural Designer considers whether there is a flat slab restraining the stack at that end. The Use slab for calculation... upper/lower, major/minor options, (which are located under the Stiffness heading in the column properties), are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for any other type of slab panel at that end. If a panel is found, then provided it has the Include in diaphragm property selected, it acts as a restraint at the position, in the same way as a flat slab.

A flat slab or any other type of slab panel only restrains the end of the stack if it is within half the slab depth from the end of the stack, and if its centre is nearer to this end of the stack than the far end.

If, at an end of the stack, no effective beams, flat slab or other slab panel that acts as a restraint is found, then the unsupported length includes the stack beyond this restraint, and the same rules apply for finding the end of the unsupported length at the end of the next stack (and so on). If there is no

stack beyond this restraint (i.e. this is the end of the column), the unsupported length ends at the end of the column.

Sway/Drift Checks

By default all stacks of all columns are taken into account in the sway/drift, wind drift and seismic drift checks.

Located under the Sway and drift checks heading in the column properties, these parameters 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.

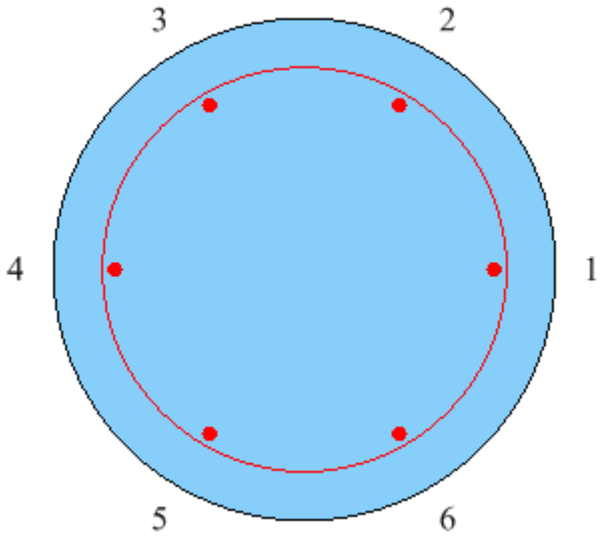
Nominal cover

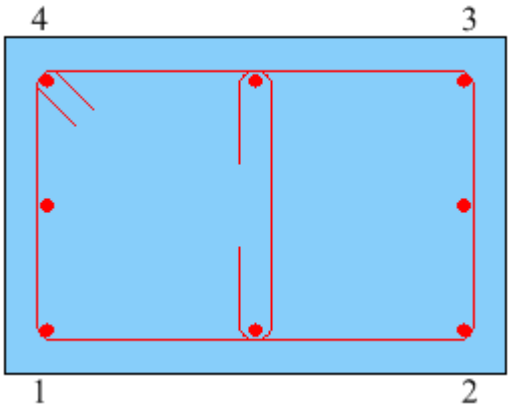
In the column properties, the nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement

The type and grade of vertical and confinement reinforcement to be considered for the design are specified under the Reinforcement heading in the column properties.

Where possible bars are arranged in a single layer:

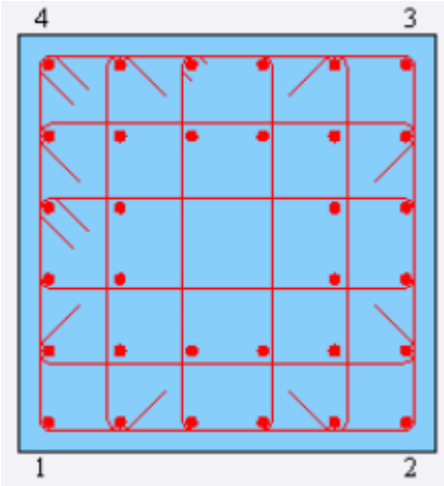


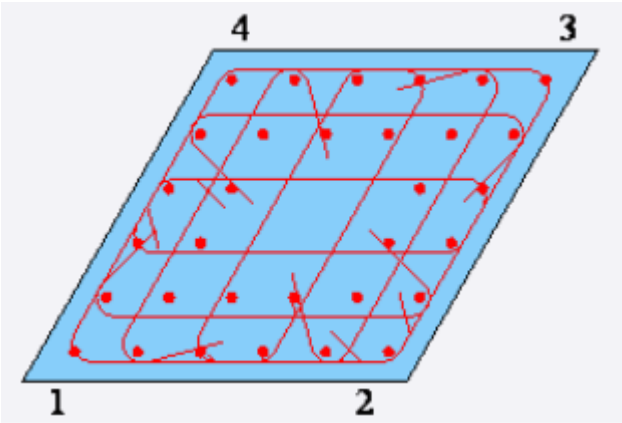


Using a 2nd layer of reinforcement

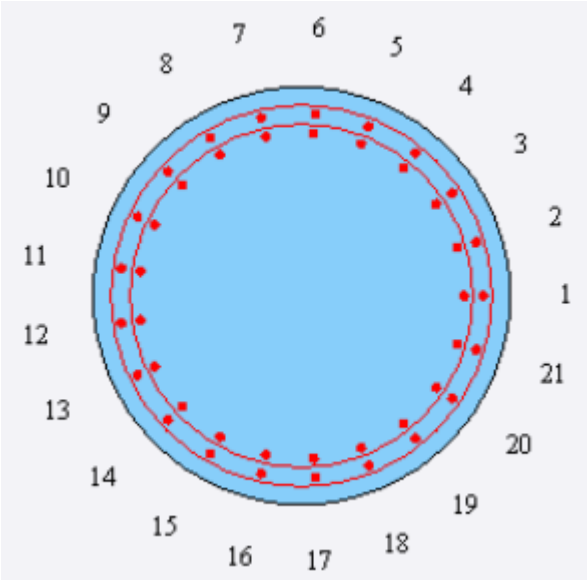
- In Autodesign: for rectangular, circular and parallelogram sections, if a single layer of reinforcement is not sufficient, design will attempt a two layer solution.
  - Rectangular and parallelogram sections: a second layer is added in line with the first bar along the adjacent length.

  NOTE At least 2 intermediate bars are needed in each direction for a 2nd layer to be added.





- Circular sections: if a second layer is required you can control the minimum layer spacing via Design Settings (a larger spacing will be used if needed).



- A link is added along the second layer for all 3 section shapes.
- In Interactive Design: you can choose to toggle the second layer on or off in the Interactive Design dialog.
- When using a 2nd layer, design checks are unchanged, except for the addition of a layer spacing check for circular sections.

3.6 Concrete wall design fundamentals

Wall design settings

Prior to running wall design you should review the design settings by selecting **Design > Settings > Concrete > Wall**.

These settings are used to control:

- Reinforcement layout, e.g.
  - Minimum and maximum bar sizes
  - Minimum and maximum spacing values
  - End zones
- Detailing settings
- General parameters (depending on the regional code being designed to)

For further details, see: Wall design settings

Concrete type

While you can apply both normal and lightweight concrete in the wall properties, wall design using lightweight concrete is currently beyond scope.

End 1 and End 2 extensions

Wall extensions (End 1/End 2) can be applied in the wall properties in order to remove physical overlaps with adjoining walls and columns without compromising the integrity of the underlying analysis model.

Negative extensions can be created automatically where appropriate. Extensions can also be defined manually if required, in which case they can be input with either positive or negative values:

- A positive extension extends the wall length beyond its insertion point.

- A negative extension trims the wall back from the insertion point.

Although the length of the wall used in the analysis model (L_{wall}) is unchanged, the wall length that is used in the design, quantity reporting and drawings changes to $L_{wall,d}$

Reinforcement layers

Either one or two layers of reinforcement can be specified in the wall properties.

Design parameters (Eurocode only)

Located under the Design parameters heading in the wall properties, the following parameters relating to shrinkage and creep can be specified for individual walls.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + Ψ_2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and Ψ_2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Sway/Drift Checks

By default all panels of all walls are taken into account in the sway/drift , wind drift and seismic drift checks.

Located under the Sway and drift checks heading in the wall properties, these parameters provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small stack 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.

Confinement reinforcement

Located under the Confinement reinforcement heading in the wall properties, the **Provide support regions** setting determines the way each panel is divided into regions for the purpose of designing the confinement reinforcement.

- Checked - confinement reinforcement is designed separately in three regions.
- Cleared - the same confinement reinforcement is designed for the whole panel.

Slenderness

Located under the Slenderness heading in the wall properties, the significant parameter within the slenderness criteria is a choice of how the wall is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.
- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the **User input value** option, or it is calculated in accordance with the requirements of the selected design code.

Stiffness

The stiffness settings affect the calculation of clear height, also referred to as the unsupported or unrestrained length (depending on the regional code being worked to) which is the clear dimension between the restraining beams at the bottom of the panel and the restraining beams at the top of the panel. The unsupported length may be different in each direction.

Effective Concrete Beams

An effective concrete beam is one which provides stiffness at a restraint position. A concrete beam is only considered effective if it is "fixed" at the position where it joins to the wall. Concrete beams are only effective in a direction if they are within 45° of that direction, and therefore no concrete beam can be effective in both directions. A concrete beam is only effective if its angle to the horizontal is 45° or less.

A concrete beam only restrains the end of the panel if it is within the depth of the panel section from the end of the stack, and if its centre is nearer to this end of the panel than the far end. Therefore, at a node at a panel join, if the top of the beam is below the node by a dimension greater than the depth of the panel below the node, it is not considered. Similarly, if the bottom of the beam is above the node by a dimension greater than the depth of the panel above the node, it is not considered.

Effective flat slabs/other types of slab

When determining the unsupported length, if no effective beams are found at the end of a panel, Tekla Structural Designer considers whether there is a flat slab restraining the panel at that end. The Use slab for calculation... upper/lower, major/minor options, (which are located under the Stiffness heading in the wall properties), are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for any other type of slab panel at that end. If a panel is found, then provided it has the Include in diaphragm property selected, it acts as a restraint at the position, in the same way as a flat slab.

A flat slab or any other type of slab panel only restrains the end of the panel if it is within half the slab depth from the end of the panel, and if its centre is nearer to this end of the panel than the far end.

If, at an end of the panel, no effective beams, flat slab or other slab panel that acts as a restraint is found, then the unsupported length includes the panel beyond this restraint, and the same rules apply for finding the end of the unsupported length at the end of the next panel (and so on). If there is no panel beyond this restraint (i.e. this is the end of the wall), the unsupported length ends at the end of the wall.

Nominal cover

Nominal concrete cover is specified in the wall properties.

For walls, it is measured as follows:

- For 1 layer of reinforcement, the vertical bar is on the centre-line of the wall thickness, the face of the horizontal bar is closest to the critical concrete face.
- For 2 layers of reinforcement, the horizontal bars are placed outside the vertical bars at each face.

The nominal concrete cover is measured to the face of the horizontal bar or any link/confinement transverse reinforcement that may be present.

Reinforcement

Under the Reinforcement heading in the wall properties, the Reinforcement layers, Form and Include end zones properties can be combined as required in order to obtain a range of reinforcement patterns, e.g:

- Single layer, using mesh reinforcement
- Two layers, using mesh reinforcement
- Single layer, using loose bars
- Two layers, using loose bars
- End zones, with a single layer of mesh in the mid zone
- End zones, with two layers of mesh in the mid zone
- End zones, with a single layer of loose bars in the mid zone
- End zones, with two layers of loose bars in the mid zone

3.7 Interactive concrete member design

The combined analysis and design processes, **Design Concrete (Static)**, **Design All (Static)** etc. are complemented by the program's interactive member design facilities. These allow you to interact with and override the initial concrete member designs.

The following interactive member designs are provided:

- Interactive concrete beam design (page 174)
- Interactive concrete column design (page 178)
- Interactive concrete wall design (page 198)

Generally you are advised to perform interactive member designs only after the combined analysis and design has been carried out. In this way multiple analysis models will have been considered to arrive at the forces being designed for.

The Interactive design dialog boxes display a limited selection of the relevant critical design results including bar details and allow you to make changes to the number, size and spacing (for links/ties only) of the selected bars.

After making changes, you are able to see the effect on the displayed results - you then have the option of canceling or accepting the changes.

Interactive concrete beam design

Opening the Interactive Beam Design Dialog

The Interactive Beam Design dialog box (page 175) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the member you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Beam Design dialog is displayed.
3. Interactively adjust the reinforcement as required until the design is satisfactory.

Changing the bar pattern used in the Interactive Beam Design Dialog

1. If the Interactive Beam Design dialog box is open, click Cancel to close it.
2. If necessary, re-select the beam to be designed.
3. Change the Top and Bottom longitudinal bar pattern in the Properties Window as required.
4. Hover the cursor over the beam until its outline is highlighted, then right-click.

5. From the context menu select Interactive Design...

The Interactive Beam Design dialog box (page 175) opens and reinforcement is automatically re-selected for the beam based on the new bar pattern.

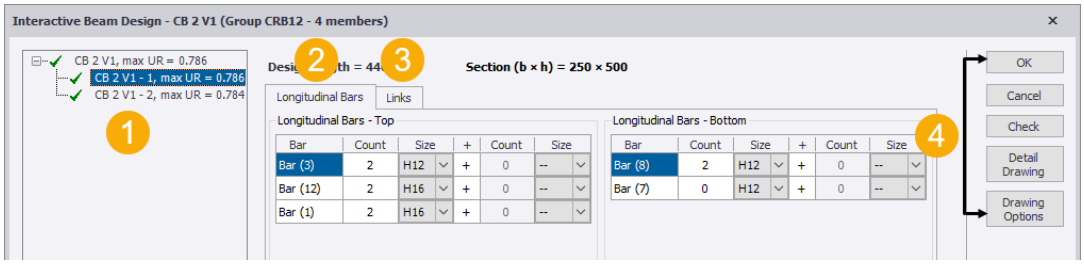
Interactive Beam Design dialog box

The **Interactive Beam Design** dialog box shows the current reinforcement and check results for each beam span in the selected beam line. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

To display the dialog box:

1. Right click on an existing concrete beam.
2. In the context menu, select **Interactive Design...**.

The dialog box content is described below.



1. Beam/span summary pane

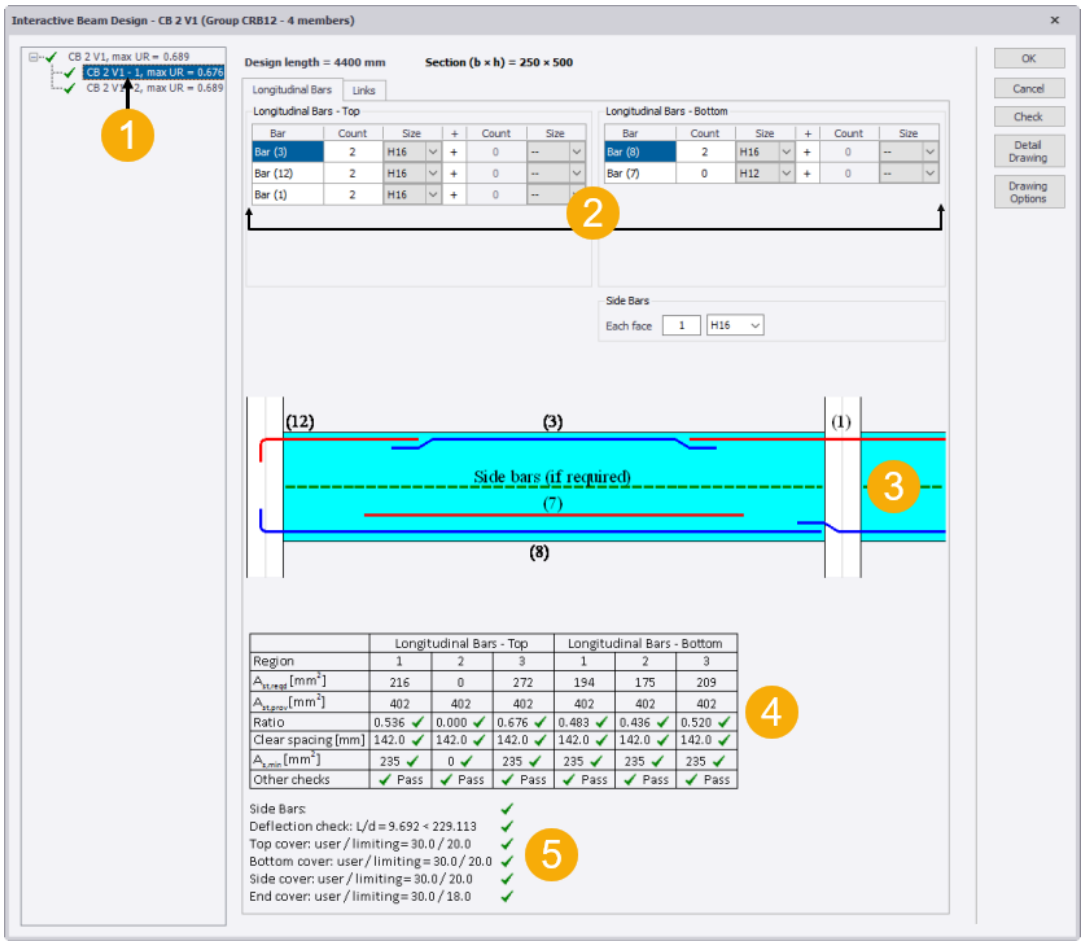


The top row in this pane shows the beam line summary, consisting of the overall utilization ratio and design status.

Subsequent rows show the design status of each span and associated utilization ratio.

- To design a particular span, click on the corresponding row for that span in the summary pane.

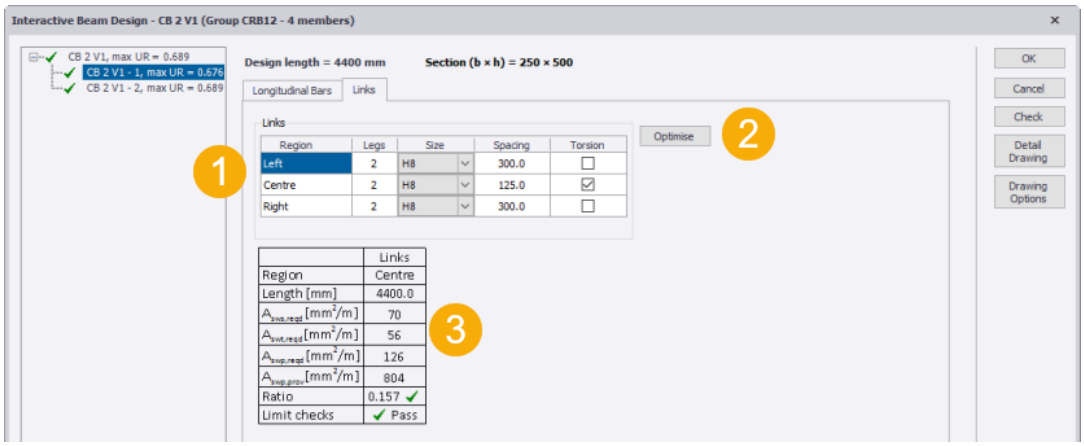
2. Longitudinal Bars tab



1. **Span Summary**: Displays the design status of the selected span and the associated utilization ratio.
2. **Bar Selection Tables**: Used for editing the longitudinal bars into the beam.
    - Each row in the table is labeled with a specific “bar number” (taken from the standard patterns applied to the beam in the Properties Window); these represent bar locations within the beam.
    - Two different bar sizes can be defined in each row, the only restriction being that the second bar must always be smaller than the first.
    - The number of bars of each size is defined using the **Count** field.
    - When bars are joined to the adjacent span, changing those bars within this span has the effect of changing those bars in the adjacent span, as they are effectively the same bar. (This is only done when the spans are "matching" in terms of their alignment and dimensions.)
3. **Bar Pattern Layout**: a schematic diagram representing the top and bottom patterns assigned to the beam.

4. **Design Summary Table**: Displays critical results for each of the design regions from all combinations:
    - Area of reinforcement required, $A_{s,reqd}$
    - Area of reinforcement provided, $A_{s,prov}$
    - Reinforcement area utilization ratio
    - Smallest clear spacing between bars
    - Minimum required reinforcement area, $A_{s,min}$
5. **Additional checks**: Side bar, deflection and cover check results are displayed below the design summary table.

3. Links/Stirrups tab



1. **Link/Stirrup Selection Table**: Specifies the number of link/stirrup legs, size and spacing in each of the regions.
2. **Optimise Button** : This calculates the optimum length of the central region given the reinforcement that you have selected. The button is not be visible when the beam is in a design group with other beams, and is also not visible when the span is a cantilever.
3. **Link/Stirrup Design Summary Table**: Displays the most critical result from all combinations:
    - Region length
    - Link/Stirrup area over spacing required for shear, $A_{sw,reqd/s}$
    - Link/Stirrup area over spacing required for torsion, $A_{swt,reqd/s}$
    - Link/Stirrup area provided, $A_{sw,prov}$
    - Link/Stirrup utilization ratio
4. **Buttons**: (See separate section below.)

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.
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog

See also

Interactive concrete member design (page 173)

Interactive concrete column design

Opening the Interactive Column Design Dialog

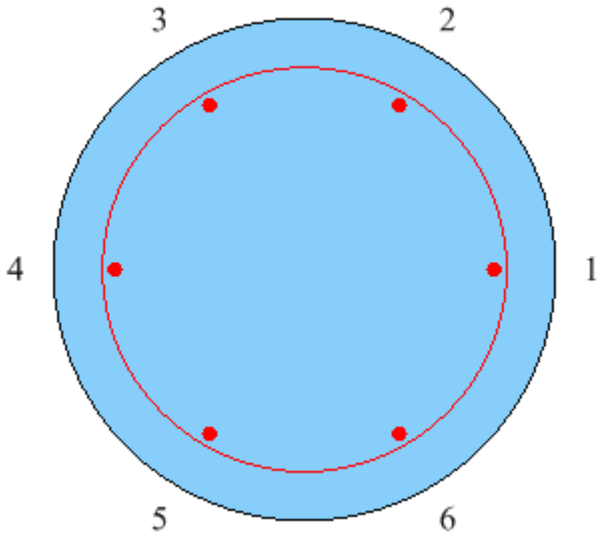
The Interactive Column Design dialog box (page 190) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the member you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Column Design dialog is displayed.
3. Click on an individual stack in the Column/stack summary pane (page 191).
4. Interactively adjust the reinforcement as required for the chosen stack until the design is satisfactory.

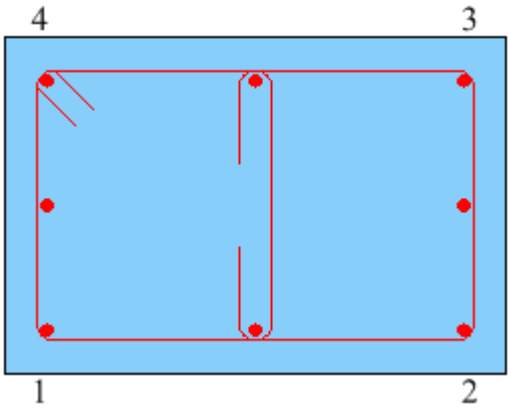
Arranging bars in the Interactive Column Design Dialog

The way in which bars are arranged depends on the column shape.

In circular columns, bars are arranged simply by modifying the bar size and count fields.



In rectangular columns bars are arranged as described below.



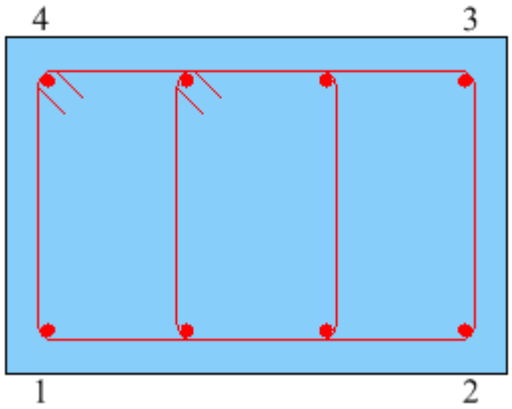
Principal bars exist at fixed locations; they are labeled with numbers in the cross-section, as shown above. You can only change the principal bar sizes, not their locations.

Intermediate bars are the unnumbered bars in the cross-section. You can change both their size and number. They are defined in the bar location table by reference to the principal bars between which they lie.

Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]	
1-2	1	249.0	✓	3-4	1	249.0	✓
2-3	1	149.0	✓	4-5	1	149.0	✓

A count of "1" for each intermediate length in the bar location table indicates that a single intermediate bar is positioned between each of the principal bars.

If the count is increased to "2" for Int. length 1-2, but reduced to "0" for Int. length 2-3, the following arrangement is achieved. (Two intermediate bars are positioned between principal bars 1 and 2, but there are now no intermediate bars between principal bars 2 and 3.)



Note that Int. lengths 3-4 and 4-5 are adjusted automatically in the table to match.

Int. length	Count	Ctr spacing [mm]		Int. length	Count	Ctr spacing [mm]	
1-2	2	166.0	✓	3-4	2	166.0	✓
2-3	0	298.0	✓	4-5	0	298.0	✓

Link / Tie arrangements in rectangular and parallelogram sections have the following basic options:

- Single links/ties,
- Double links/ties,
- Triple links/ties,
- Cross links/ties.

Tie bars are used with these arrangements. Link/Tie arrangements in other section shapes use standard link/tie positions with additional tie bars where required.

Using a 2nd layer of bars

It is sometimes not possible to find a solution using a single-layer of reinforcement - (a problem more common in hi-rise structures with large columns in lower stacks).

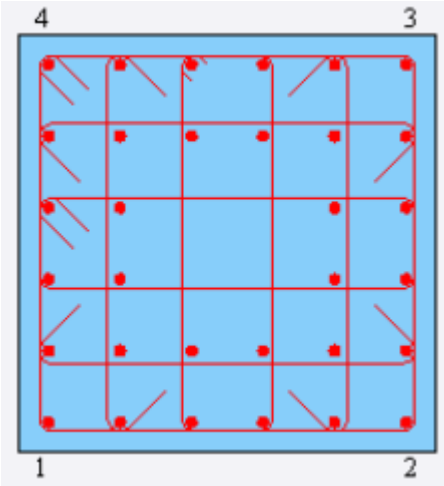
Selecting the **Use 2nd layer** option in the interactive design provides you with more flexibility to find a solution in these cases.

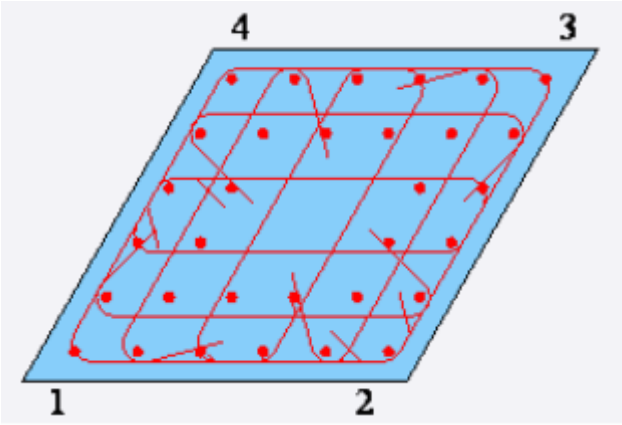
A 2nd layer can be applied to rectangular/parallelogram and circular sections.

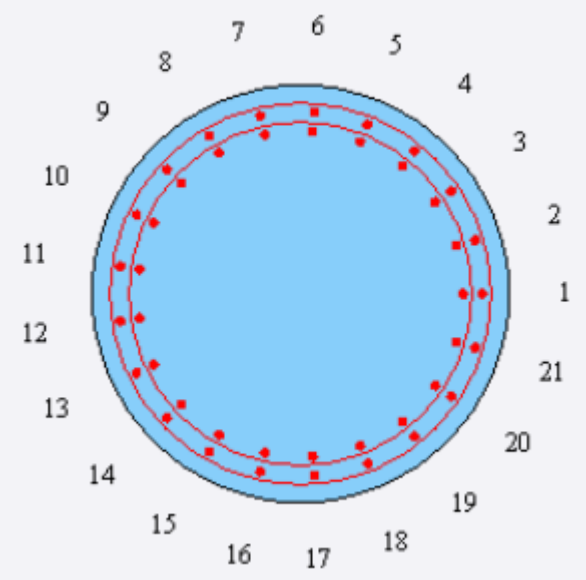
Rectangular/parallelogram: a second layer in line with the first bar along the adjacent length

- At least 2 intermediate bars are needed in each direction

A link is added along the second layer for all 3 section shapes.







When using a 2nd layer, design checks are unchanged, except for the addition of a layer spacing check for circular sections.

Column interaction diagrams (US customary units)

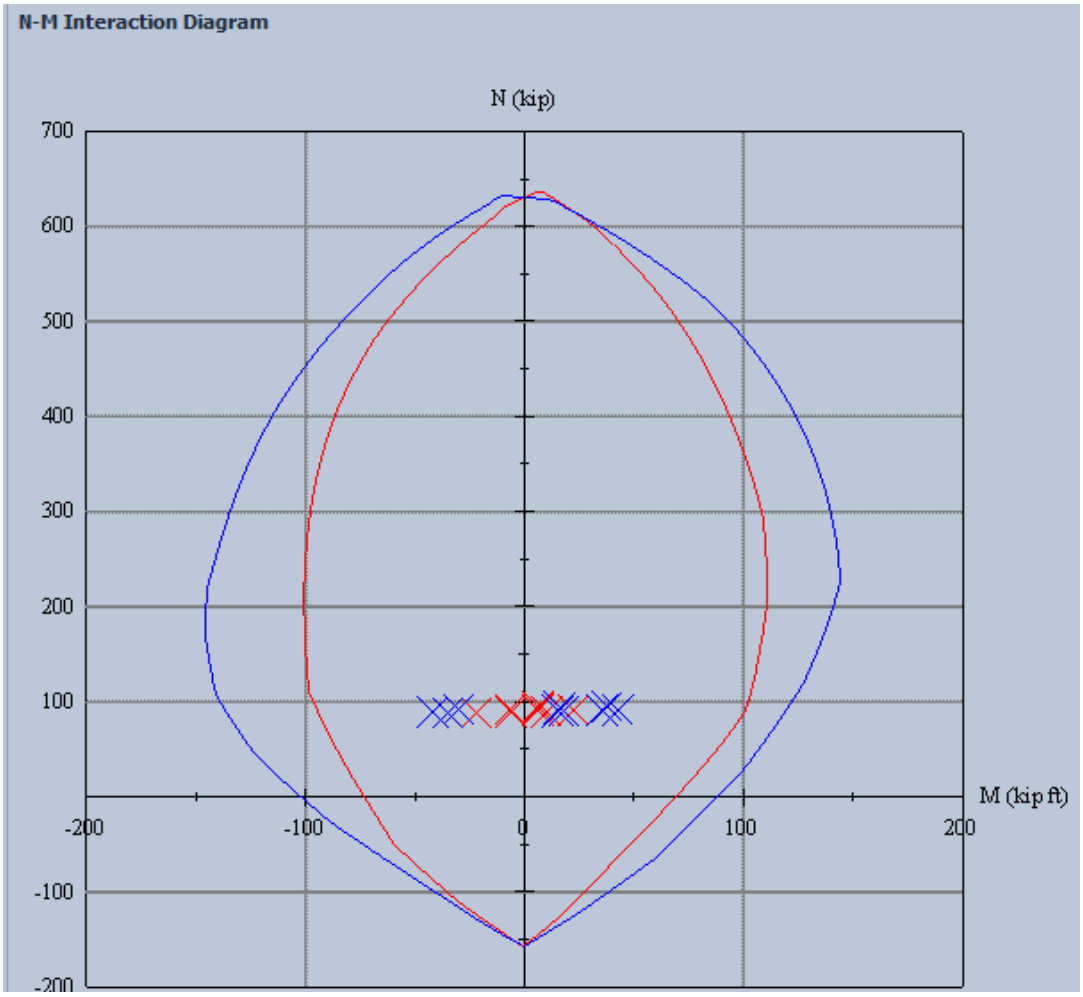
To visually observe the utilization of the design, interaction diagrams can be drawn for individual columns by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Column axial force-moment interaction diagram

The column axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.

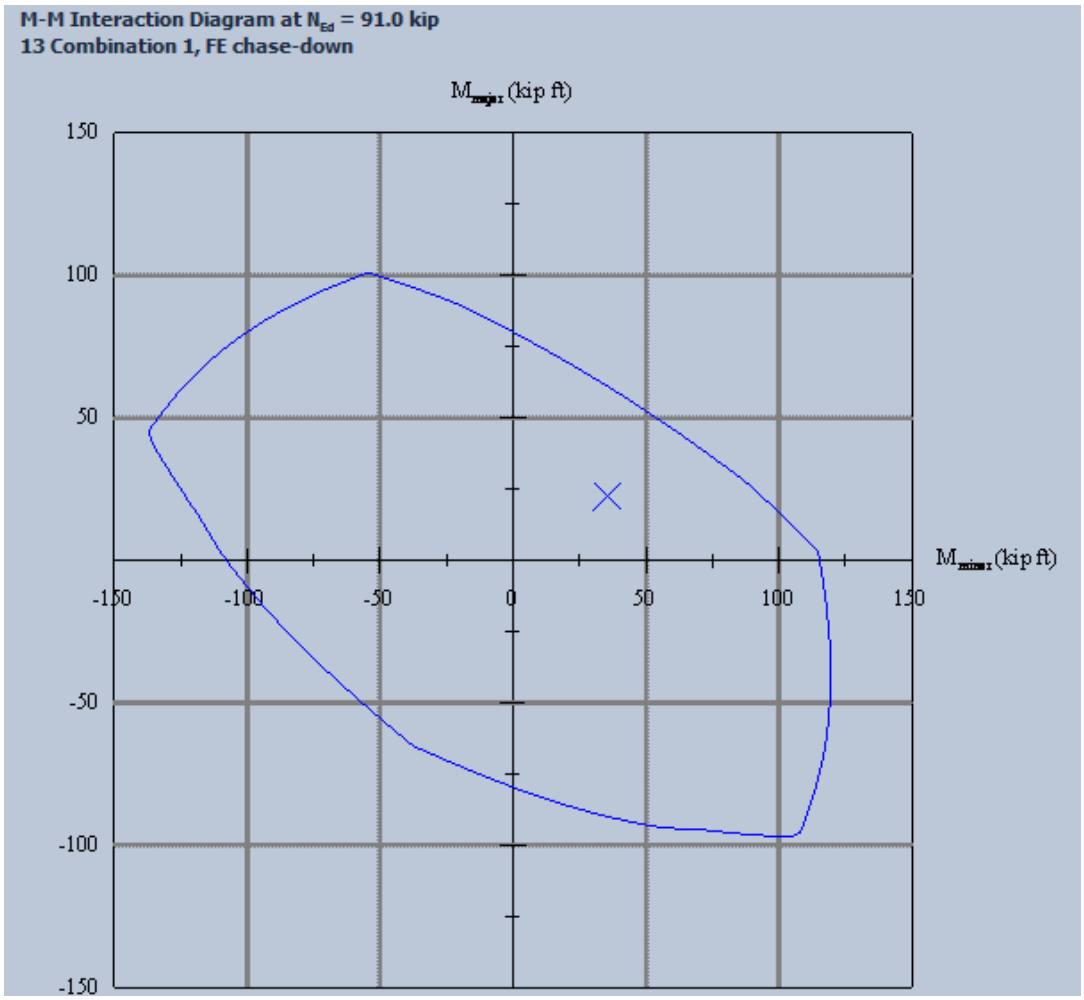


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

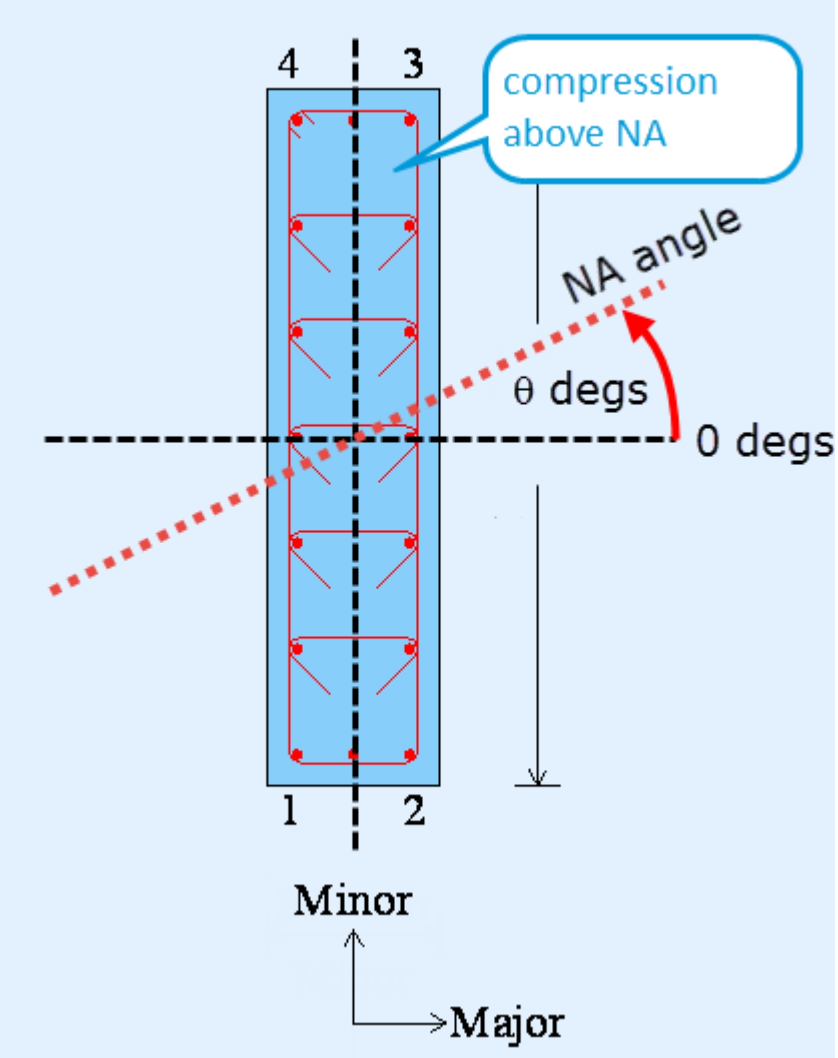
In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

Column moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a column.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force. The design process for biaxial bending is as follows:

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the

ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - meaning the corner of the column near bar 4 is at the top and the point near bar 2 is at the bottom. The linear strain distribution between these points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Column interaction diagrams (metric units)

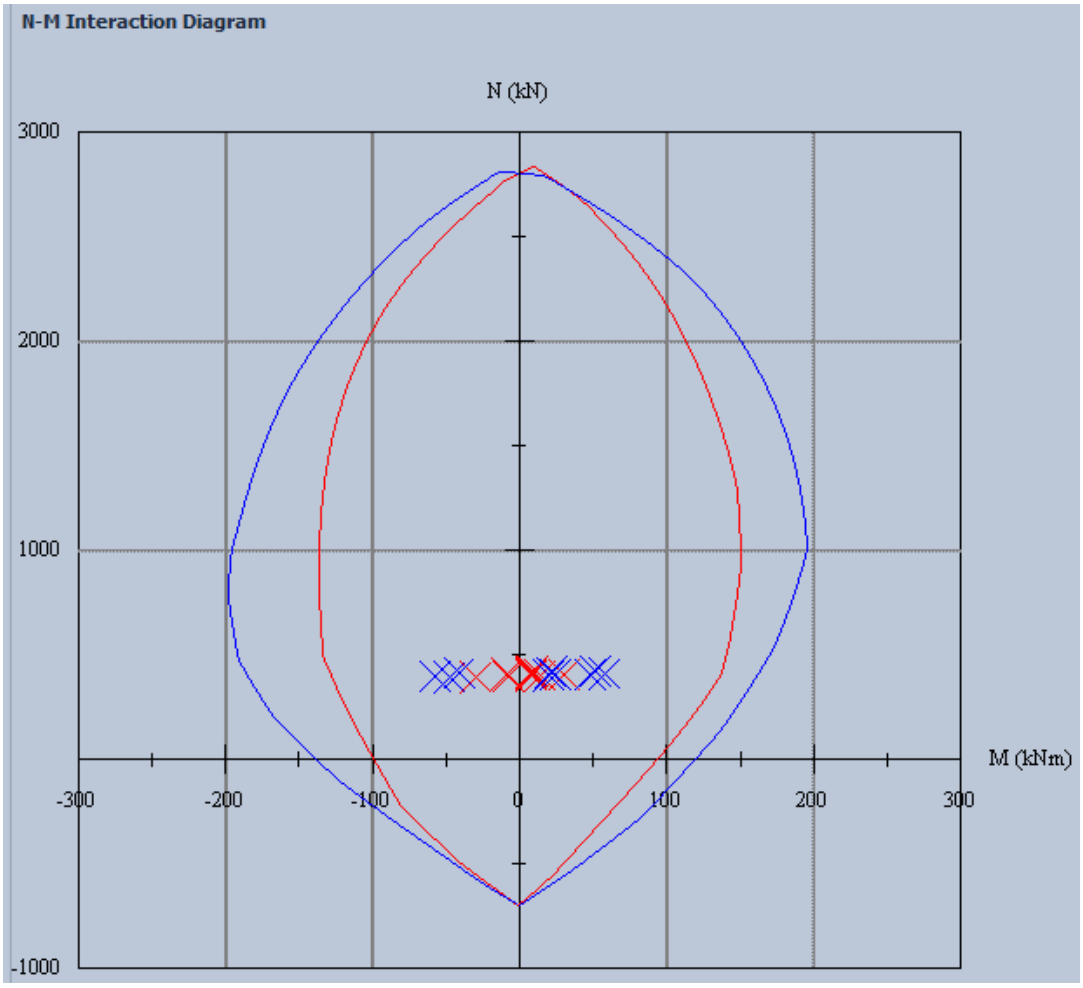
To visually observe the utilization of the design, interaction diagrams can be drawn for individual columns by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Column axial force-moment interaction diagram

The column axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.

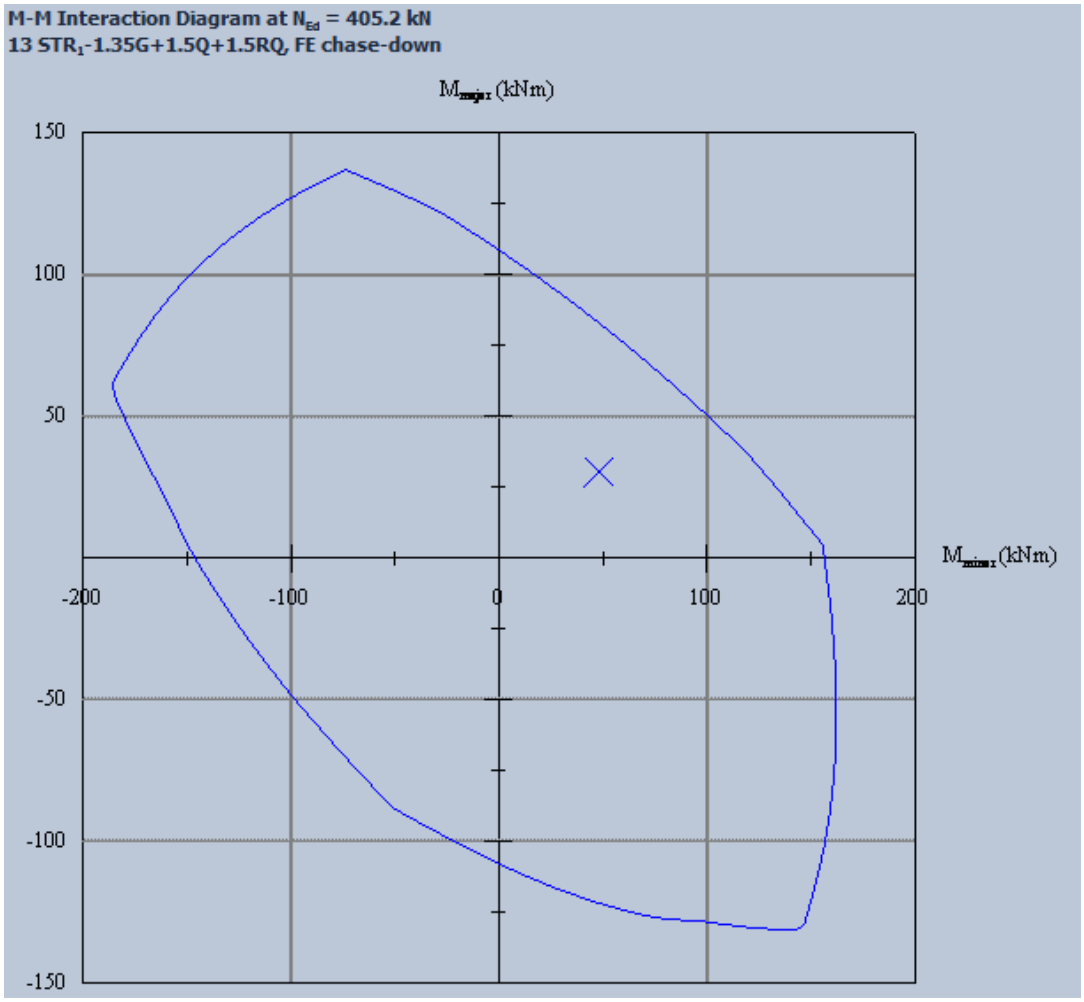


The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

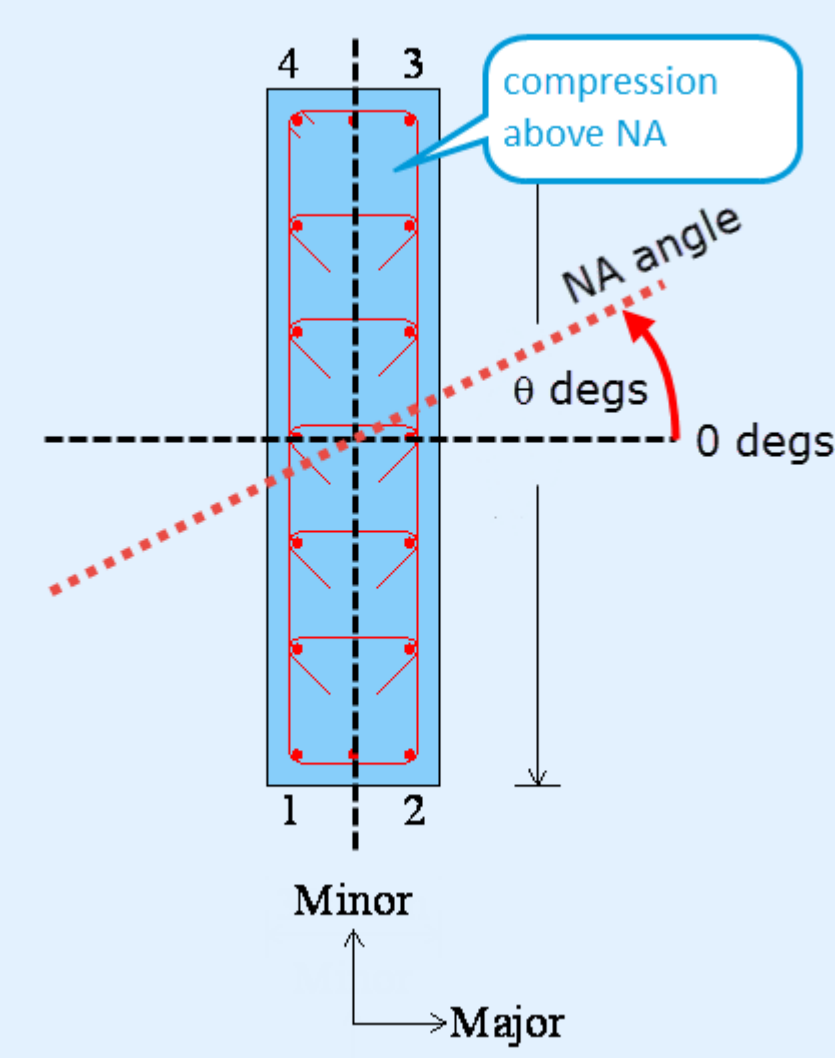
In general, the envelope will only be symmetrical for symmetrically reinforced rectangular and circular sections.

Column moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a column.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force. The design process for biaxial bending is as follows:

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the

ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - meaning the corner of the column near bar 4 is at the top and the point near bar 2 is at the bottom. The linear strain distribution between these points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Defining additional column design cases for user defined forces

Additional design cases can be specified in order to for example design for results from Post Tensioning analysis programs. These additional forces are entered per selected stack on the Additional Design Cases page of the dialog. Any number of design cases can be added and are checked alongside regular combinations.

1. In the Interactive Column Dialog, select Additional Design Cases (page 195) tab.
2. Click Design Cases to open a dialog in which to add the cases (these belong to the model, so appear for all column stacks and wall panels).
3. Click OK to close the Additional Design Cases dialog.
4. Make relevant cases Active in the current stack.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all column stacks where appropriate.

The additional loading cases are now always checked whenever the regular combinations are checked.

Interactive Column Design dialog box

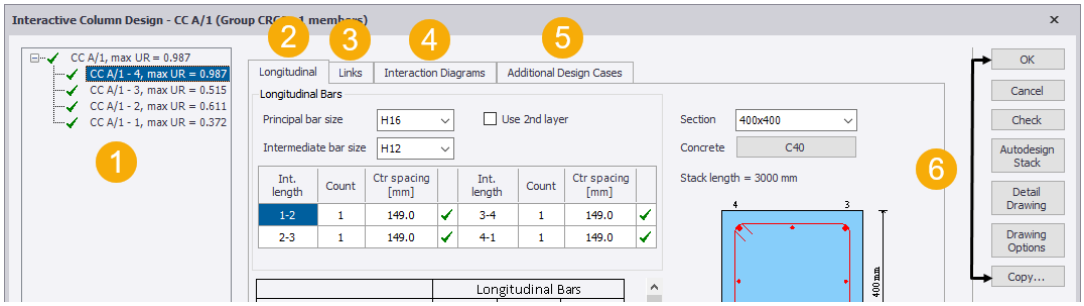
The **Interactive Column Design** dialog box shows the current reinforcement and check results for each stack in the selected column. When any of the editable fields are changed, the checks are re-run and the results are updated;

enabling you to quickly see the effect of each change you make to the reinforcement.

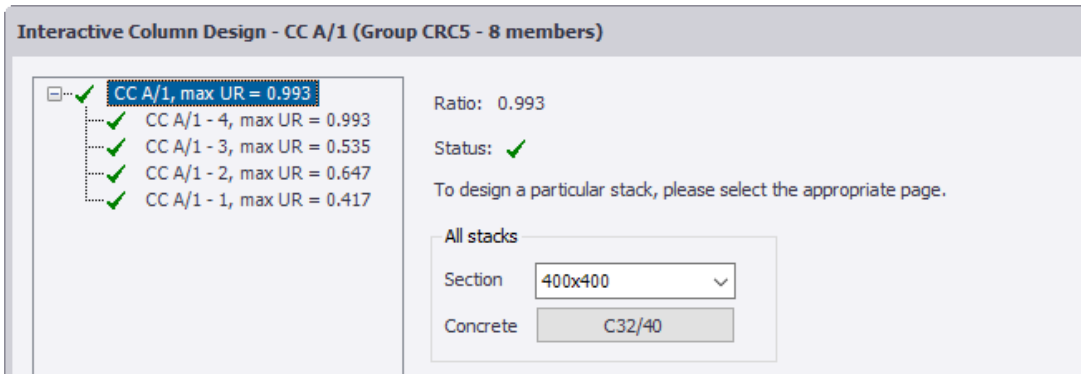
To display the dialog box:

1. Right click on an existing concrete column.
2. In the context menu, select **Interactive Design...**.

The dialog box content is described below.



1. Column/stack summary pane



The top row in this pane shows the column summary, consisting of the overall utilization ratio and design status.

- With this row selected you can edit the section size and grade for all stacks simultaneously.

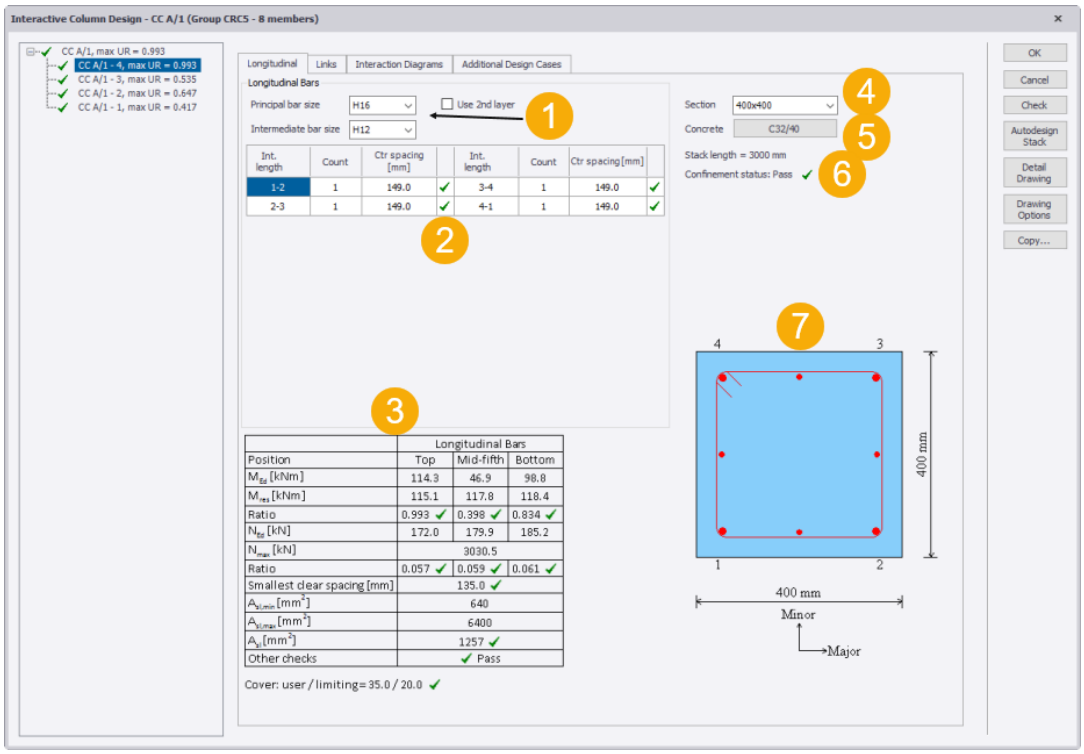
Subsequent rows show the design status of each stack and associated utilization ratio.

- To design a particular stack, click on the corresponding row in the summary pane.

2. Longitudinal tab

All straight-edged cross sections have “Principal” bars located at shear tie corners. Between these, evenly spaced identical “Intermediate” bars can be

located. Circular sections have 6 or more evenly spaced bars around the edge of the section.



1. **Longitudinal Bars**:
   - **Principal bar size**: Used to change the size of **all** principal bars.
     - **Use 2nd layer**: Check the box to allow a second bar layer if required.
       - **Spacing**: For circular sections the user can control the layer spacing.
   - **Intermediate bar size**: (Not displayed for circular columns) Used to change the size of **all** intermediate bars.
2. **Bar Location Table**: Used for adding intermediate bars into the cross-section:
   - Int. length - identifies the edge along which the bars are positioned
   - Count - for changing the number of intermediate bars along the length
   - Ctr spacing - the centerline spacing for the current number of bars along the length
   - Status - indicates when the maximum bar spacing limit has been exceeded. (When the minimum bar spacing limit is exceeded this is displayed elsewhere in the Design Summary Table).
3. **Design Summary Table**: Displays the most critical result from all combinations:

- Design and Resistance Moments and Moment Ratios
- Axial Force, Axial Resistance and Axial Ratios
- Smallest clear bar spacing
- Minimum area of steel
- Area of steel provided

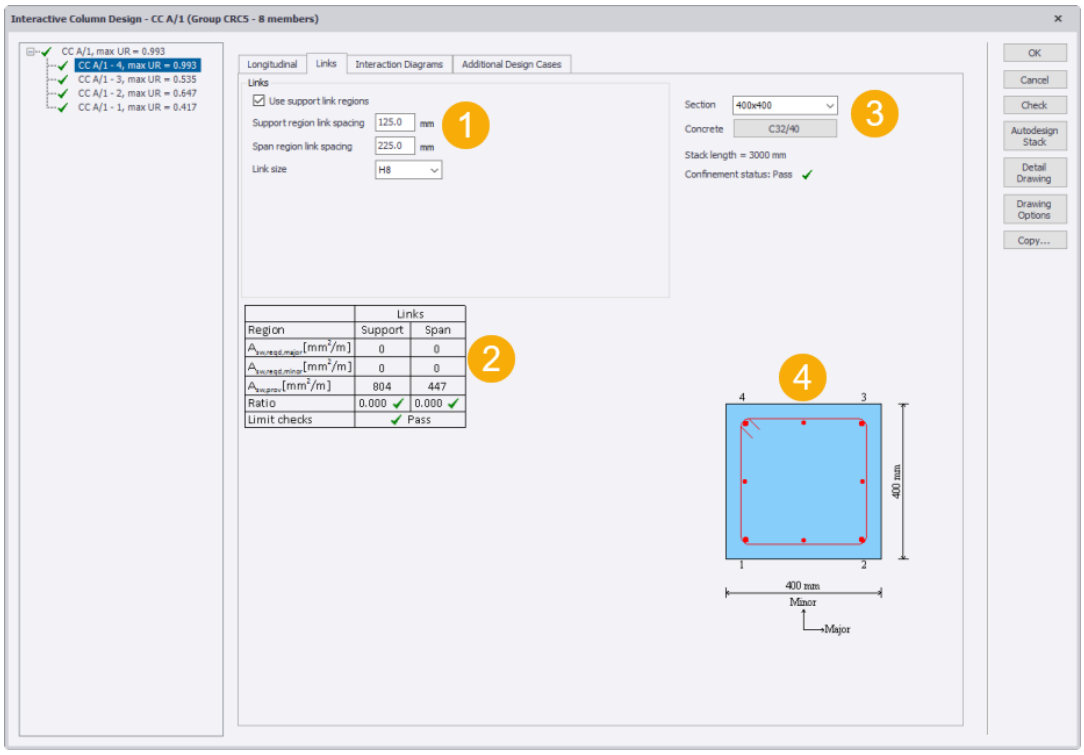
4. **Section Droplist**: Used for changing the section size for the current stack.

NOTE If the droplist is used to change the section shape an autodesign is performed.

---

5. **Concrete Droplist**: Used for changing the concrete grade for the current stack.
6. **Confinement status** : This status is determined based on the requirements for bars being tied.
7. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Tie locations
   - Section dimensions
   - Principal bar labels

3. Links/Ties tab



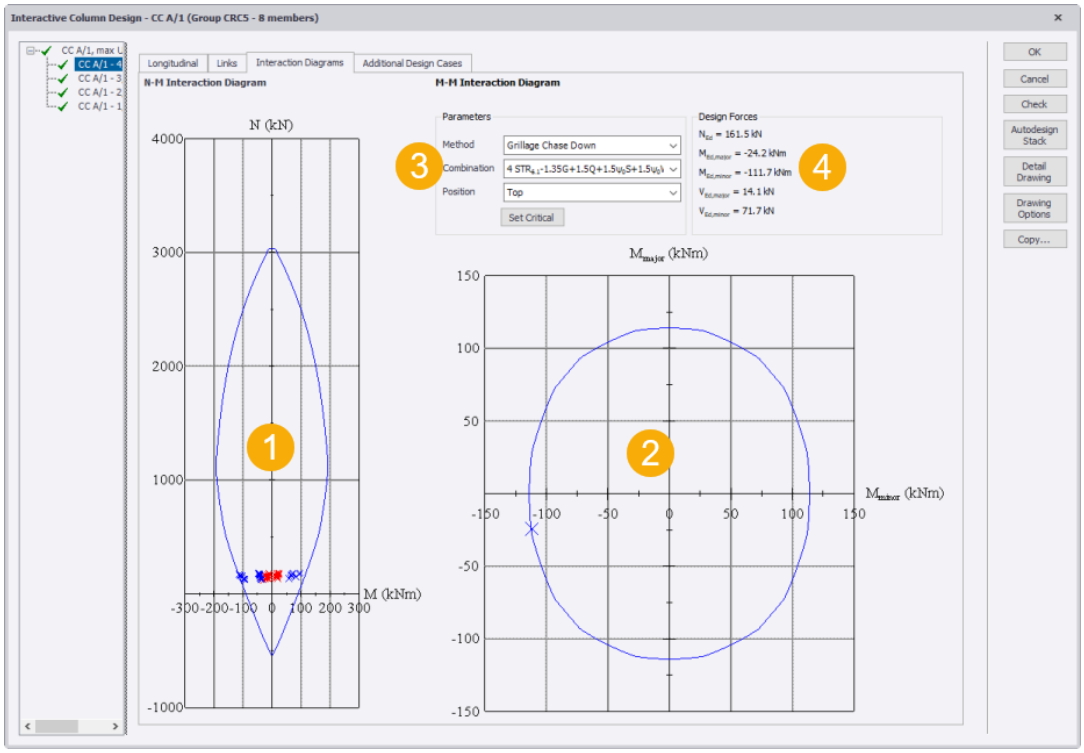
1. **Links**:
   - **Use support region links/ties** : Select to design support regions for the links/ties.
   - **Link/Tie spacing** : Specifies the link/tie spacing (if support regions are applied two different spacings can be specified)
   - **Link/Tie size**: Used to change the size of link/tiebars (all must have the same size).
2. **Link/Tie Design Summary Table**: Displays the most critical result from all combinations:
   - Region length
   - Link/Stirrup area over spacing required, major
   - Link/Stirrup area over spacing required, minor
   - Link/Tie area over spacing provided
   - Link/Tie utilization ratio
3. **Section Droplist**: Used for changing the section size for the current stack.

   NOTE If the droplist is used to change the section shape an autodesign is performed.

4. **Cross-section** : The drawing displays:

- Exact bar positions (drawn to scale)
- Link/Tie locations
- Section dimensions
- Principal bar labels

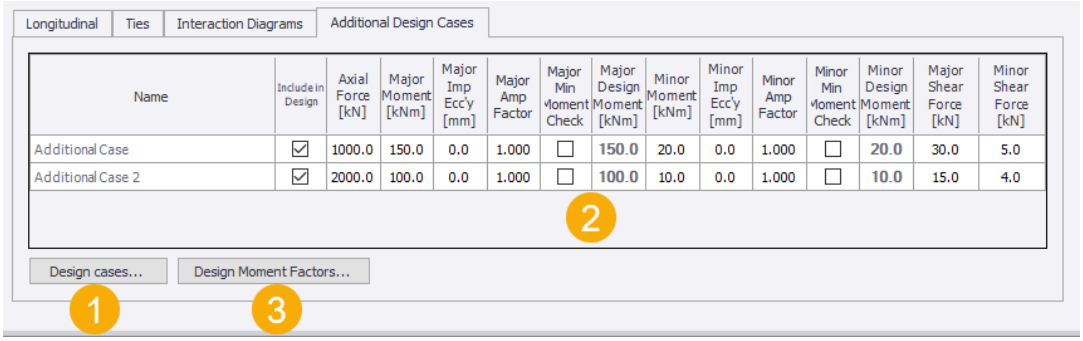
4. Interaction Diagram tab



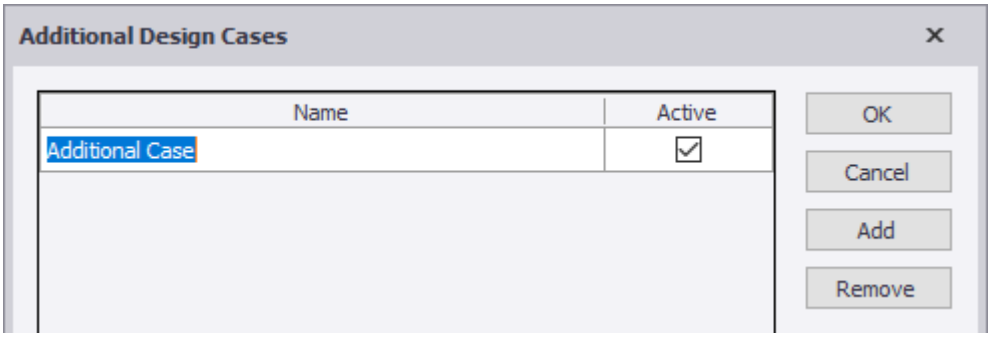
1. **N-M Interaction diagram** : Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
   - The curves for bending about the major axis are shown in red
   - The curves for bending about the minor axis are shown in blue
2. **M-M Interaction diagram** : The diagram is different for each value of axial force. Initially the diagram at the axial force of the critical combination is drawn - this is the combination with the highest M_{Ed} / M_{res} ratio.
3. **Parameters**: Select the analysis method, combination, and position for which the diagrams are displayed.
   - **Set Critical** button: If you have changed the parameters for which the diagrams are displayed, you can click this button in order to revert back to the critical parameters.
4. **Design Forces**: The design forces applicable for the selected parameters.

5. Additional Design Cases tab

This tab can be used for example to design for results from Post Tensioning analysis programs.



1. **Design cases...** button: Click to open a dialog in which to add any additional design cases.



   The design case names defined here are available to all columns and walls in the model.

   The **Active** box in this dialog is used to control which design cases are considered (for the entire model).

   After clicking OK you are then able to specify the design case design forces in the **Design cases table**.

2. **Design cases table**: Each design case added via the **Design cases...** button appears in the table, but only those that were marked as active (for the entire model) are then available to have design forces applied.

   In order to apply the design case design forces to a given stack you must:

   - Select the required stack in the stack summary
   - Select **Include in Design** for the design case
   - Enter the design forces

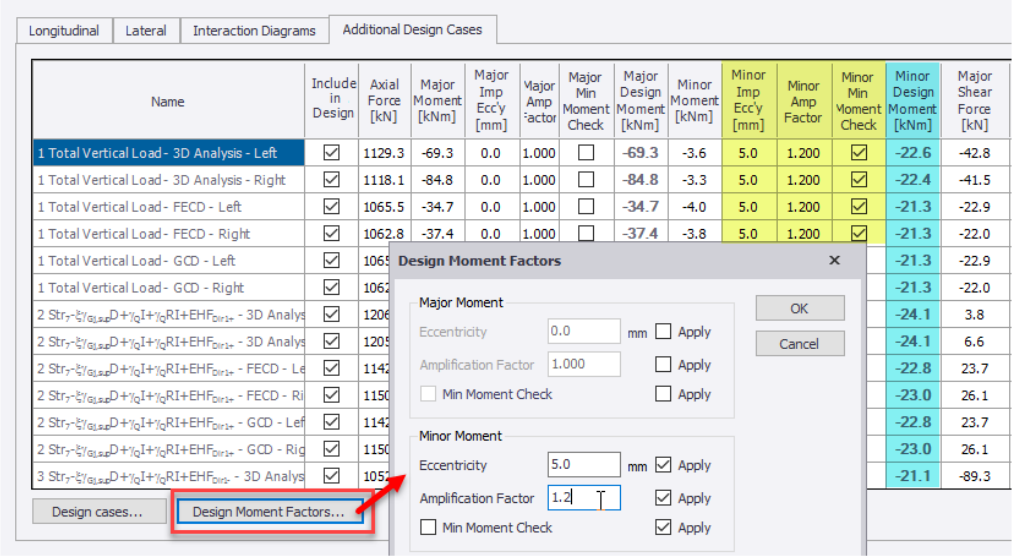
3. **Design Moment Factors...** button: Offers three potential “Design Moment” adjustments for each direction:

   - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.

- Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety).
- Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension “h” in the direction considered).

When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.

The adjustment values and options can be applied to individual Cases and also quickly in a single operation to all Active Cases (those with “Active” option checked on) as shown below:



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.
Autodesign Stack	Performs an autodesign reselecting bars for the current stack
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog

Button	Description
Copy	Opens a new dialog enabling the selective coping of section/concrete grade/reinforcement from the selected stack to other stacks in the same column.

See also

Interactive concrete member design (page 173)

Interactive concrete wall design

Opening the Interactive Wall Design Dialog

The Interactive Wall Design dialog box (page 207) can be opened from any of the 2D or 3D Views as follows:

1. Right-click the wall you want to design interactively.
2. Select **Interactive Design...** (Static or RSA as required). The Interactive Wall Design dialog is displayed.
3. Click on an individual panel in the wall/panel summary pane (page 207).
4. Interactively adjust the reinforcement as required until the panel design is satisfactory.

Wall interaction diagrams (US customary units)

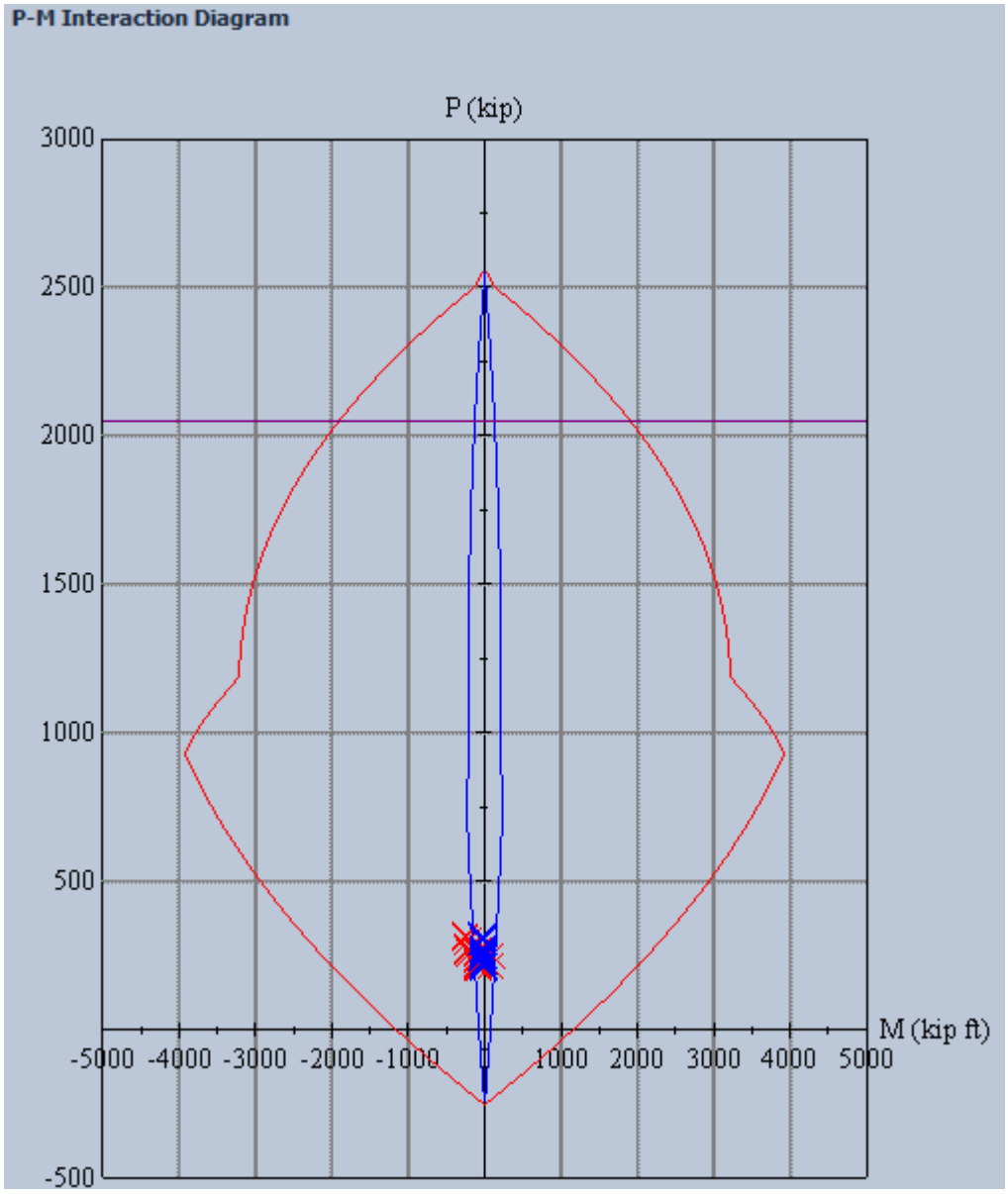
To visually observe the utilization of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Wall axial force-moment interaction diagram

The wall axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

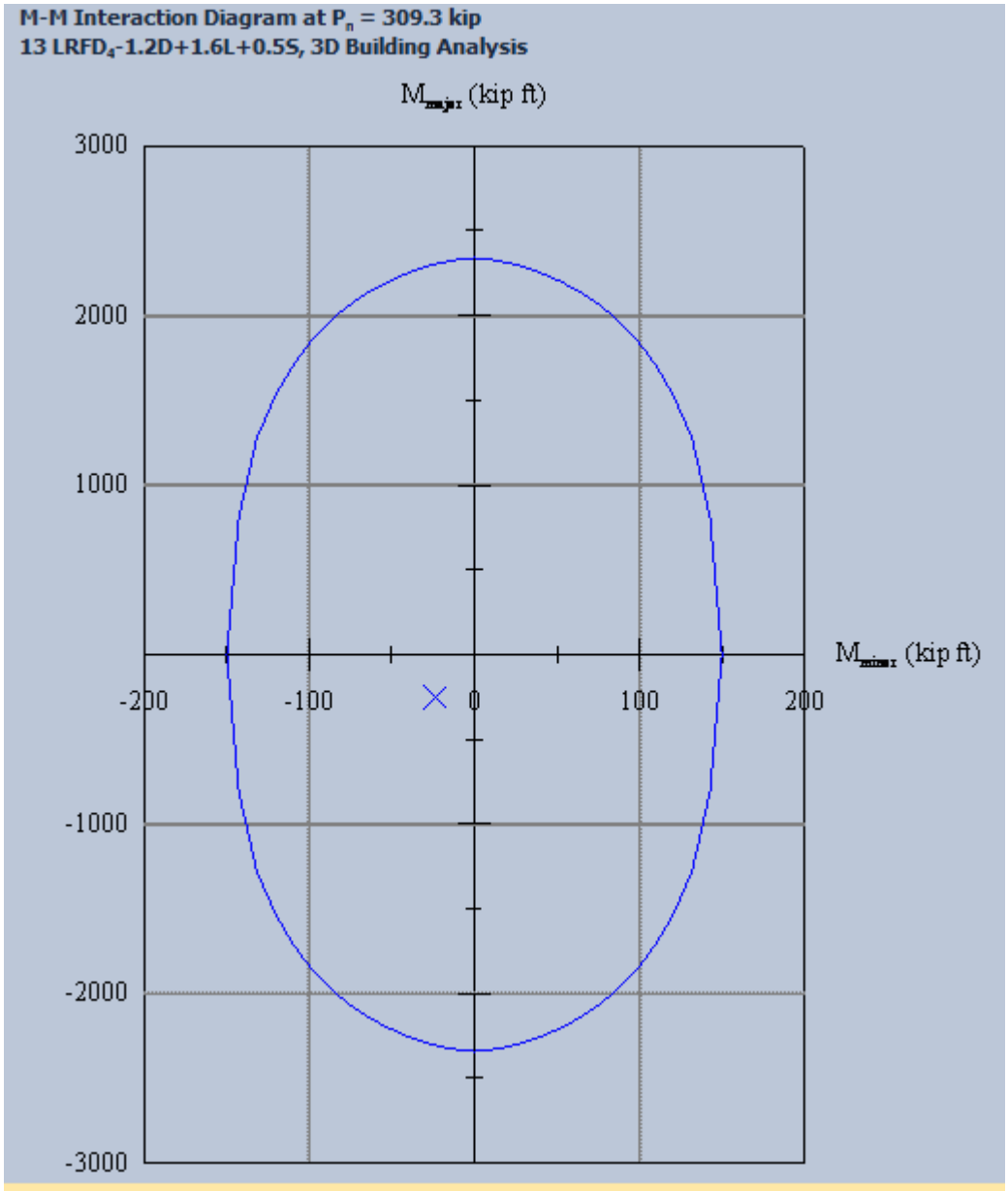
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.



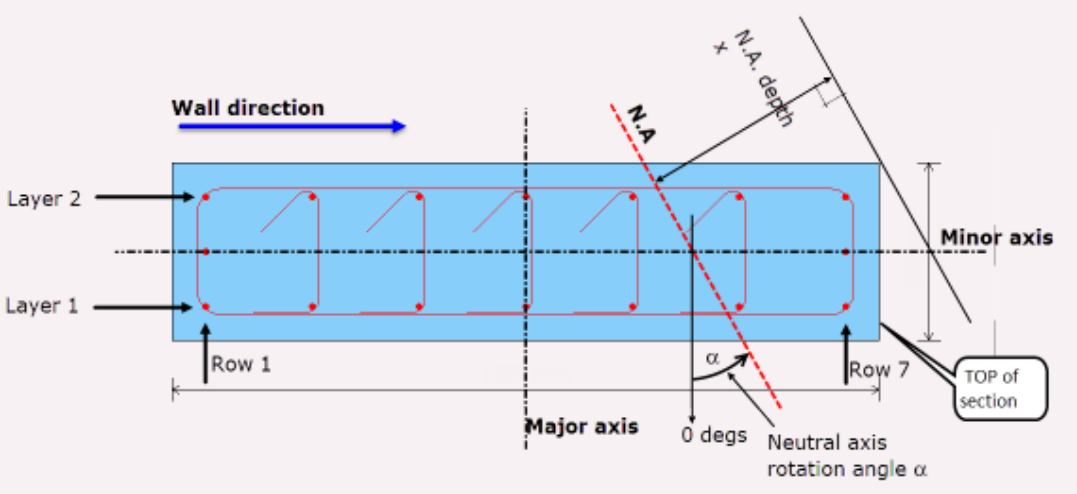
The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

Wall moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a wall. The wall major and minor axes follow the same convention as columns - the major axis is perpendicular to the length (on plan) of the wall as shown.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force.

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - The linear strain distribution between the top and bottom points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Wall interaction diagrams (metric units)

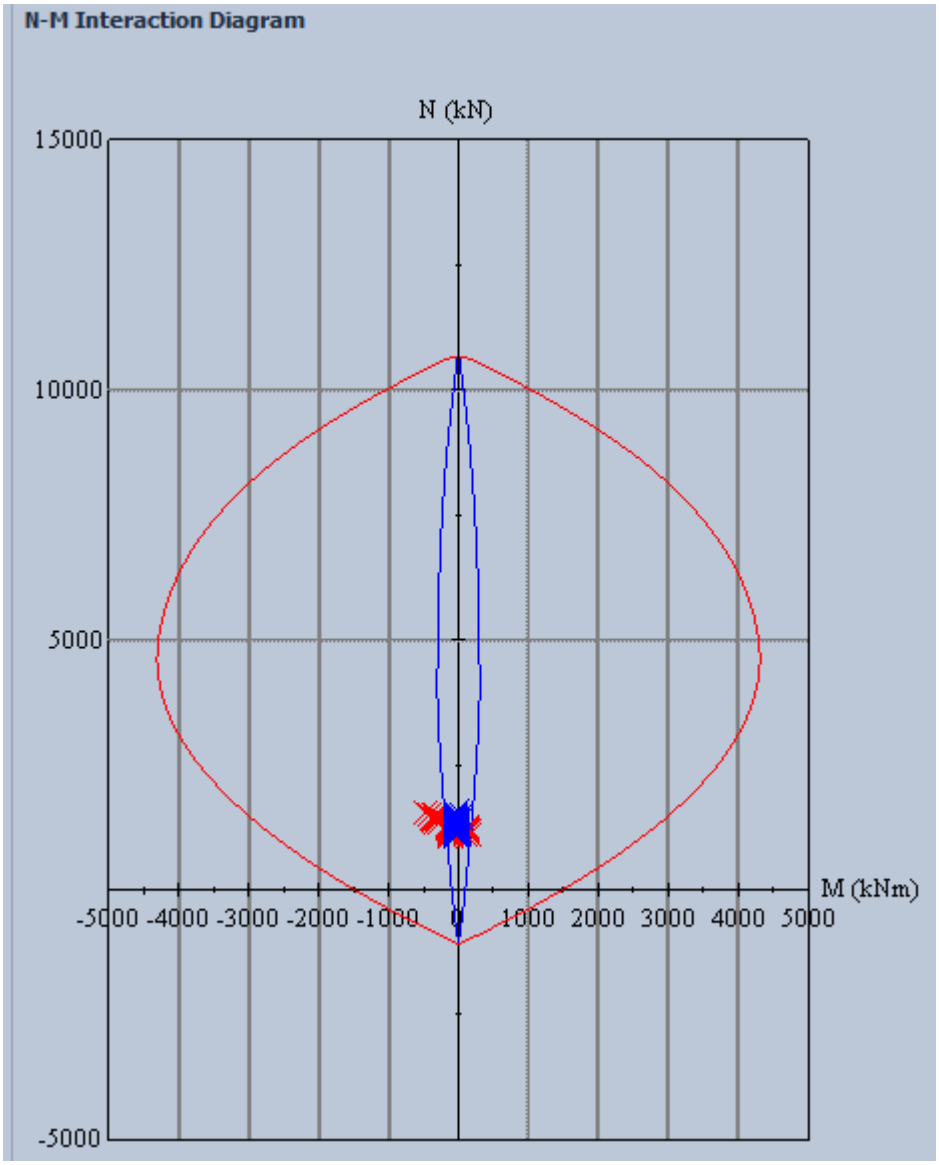
To visually observe the utilization of the design, interaction diagrams can be drawn for individual walls by accessing the interactive design. There are two types of interaction diagrams: "axial force-moment interaction diagrams" and "moment interaction diagrams".

When the dialog is first opened the moment interaction diagram is displayed for the critical parameters, but you can then view the diagram for any analysis method, combination and position. The "Set Critical" button can be used to return to the critical parameters.

Wall axial force-moment interaction diagram

The wall axial force - moment interaction diagram is created by fixing the neutral axis rotation so that the neutral axis is perpendicular to the desired direction of moment resistance and varying the depth of the neutral axis. This is done about four orientations, separated by 90°: positive moment resistance in the y-direction; positive moment resistance in the z-direction; negative moment resistance in the y-direction; negative moment resistance in the z-direction.

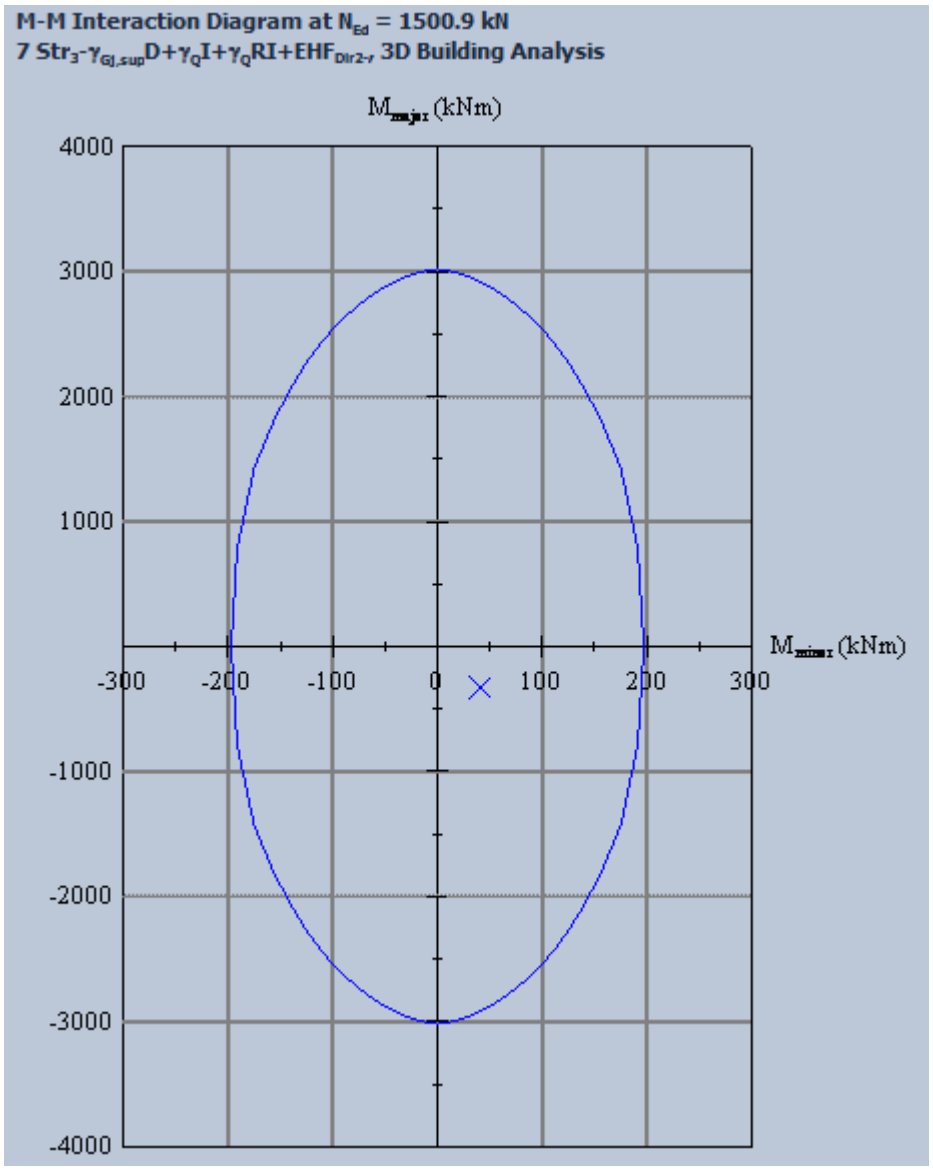
This method creates a list of points plotted on the interaction diagram for each rotation. Each direction having two lists: one for positive moment resistance and one for negative moment resistance. The failure envelopes created from two lists referring to moments in the same direction (for positive and negative in that direction) are shown in the same colour, a different colour being used for each of the two directions.



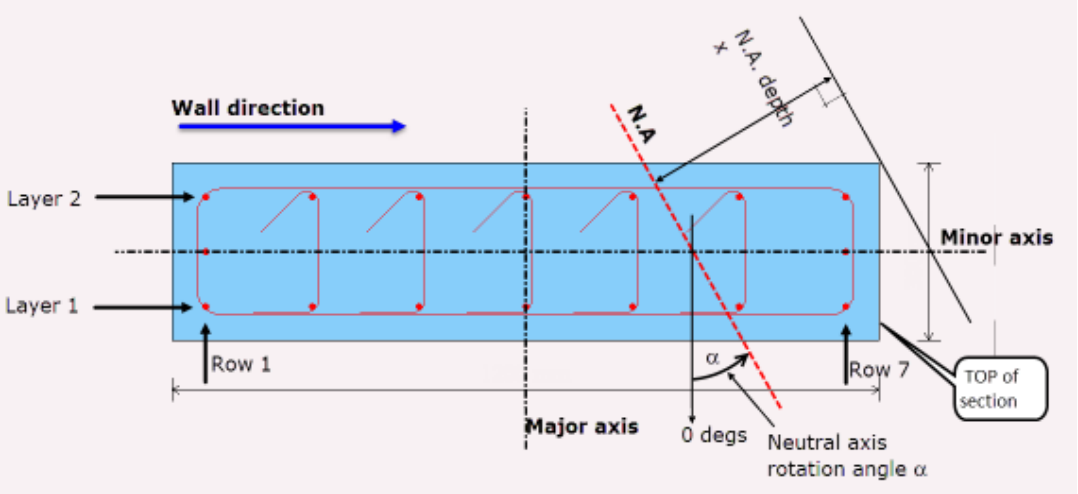
The applied values are plotted for all combinations on the same diagram. The points referring to moments in each of the directions are in the same colour as the failure envelope for that direction. Points for both directions are plotted on the same diagram.

Wall moment interaction diagram

The moment interaction diagram is created for a specific axial force, and is used to show the interaction of the applied moments compared to the moment interaction failure envelope. The failure envelope is created by taking many rotations of neutral axis, and calculating the depth for that rotation and axial force. Moments are then taken about the plastic centroid to calculate the moment resistance in each direction.



The moment interaction diagram is calculated for each combination individually, as it is calculated based on the applied axial force which is typically different for each combination.



The resultant moment (of resistance) angle is zero when creating positive moments about the major axis and no moment about the minor axis. For consistency, the neutral axis rotation is zero when it lies parallel with the major axis with compression above the major axis. The resultant moment angle (both applied and resistance) and neutral axis angle are then measured anti-clockwise from this zero position.

The method and convention is the same for columns and walls. The convention is shown graphically above for a wall. The wall major and minor axes follow the same convention as columns - the major axis is perpendicular to the length (on plan) of the wall as shown.

The N.A. rotation is that for the calculated NA depth and at which the ratio of moments of resistance equals that of the applied moments for the applied axial force.

For biaxial bending design the method of checking whether the reinforcement in the section is sufficient is to check whether the bending resistance of the section is larger than the applied moment for a given axial force. Tekla Structural Designer therefore calculates the neutral axis position (rotation and depth) at which the ratio of the moment limits in each direction is equal to the ratio of the applied moments and the resultant axial resistance of the section is equal to the applied axial force.

This is done by calculating the neutral axis depth at which the applied axial force would equal the ultimate axial resistance of the section, and calculating the ultimate moment resistance in each direction for this neutral axis depth.

For this calculation the cross-section shown above is effectively rotated so that the neutral axis depth is horizontal - The linear strain distribution between the top and bottom points is then used to calculate the stress in each bar.

If the ratio of the ultimate moment resistance in each direction is not equal to the applied ratio, then Tekla Structural Designer iterates to find the next neutral axis angle "guess" and re-runs the process.

When the final neutral axis angle has been found, the program compares the resultant applied moment with the resultant moment resistance to find the moment utilization ratio for the applied force and moment combination.

The moments of resistance about each axis are given in the output below the respective tables which calculate the moment resistance contribution of each bar for that direction.

Defining additional wall design cases for user defined forces

Additional design cases can be specified typically in order to for example design for results from Post Tensioning analysis programs. These additional forces are entered per selected panel on the Additional Design Cases page of the dialog. Any number of design cases can be added and are checked alongside regular combinations.

1. In the Interactive Wall Dialog, select Additional Design Cases (page 213) tab.
2. Click Design Cases to open a dialog in which to add the cases (these belong to the model, so appear for all column stacks and wall panels).
3. Click OK to close the Additional Design Cases dialog.
4. Make relevant cases Active in the current stack.
5. Enter the loading for the Active cases.
6. Repeat 4 and 5 for all wall panels where appropriate.

The additional loading cases are always checked whenever the regular combinations are checked.

NOTE Additional design cases can also be set up directly from **Result Lines** in order to facilitate local section design around openings - see the following topic.

Using result lines for local section design around openings

Interactive design of local sections around/between openings is possible using **Result Lines**.

The design is performed either as a column or wall section (as specified by the user), using those design forces determined along the lengths cut by the result lines. Engineering judgement is therefore required when positioning the lines to ensure suitable design forces are obtained.

For further details of the process, see:

Related video

Interactive design using Result lines

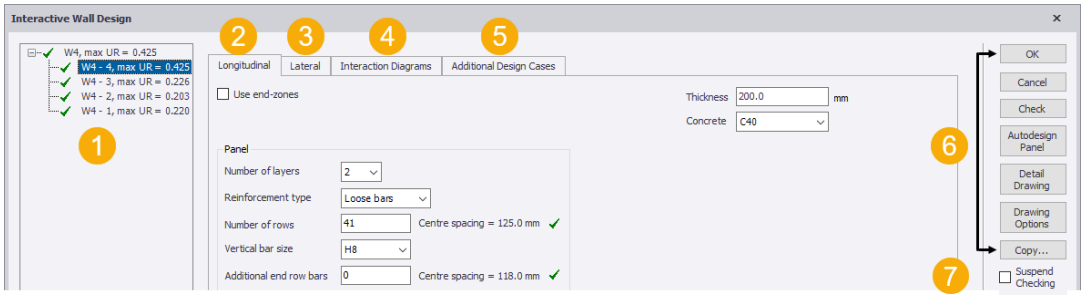
Interactive Wall Design dialog box

The **Interactive Wall Design** dialog box shows the current reinforcement and check results for each panel in the selected wall. When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of each change you make to the reinforcement.

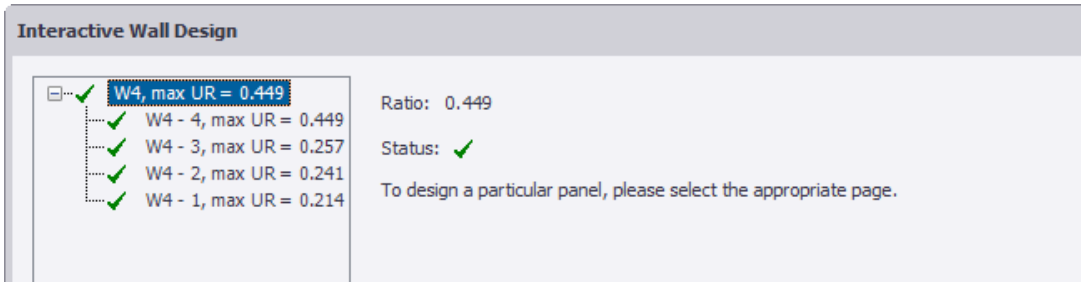
To display the dialog box:

1. Right click on an existing concrete wall.
2. In the context menu, select **Interactive Design...**.

The dialog box content is described below.



1. Wall/panel summary pane



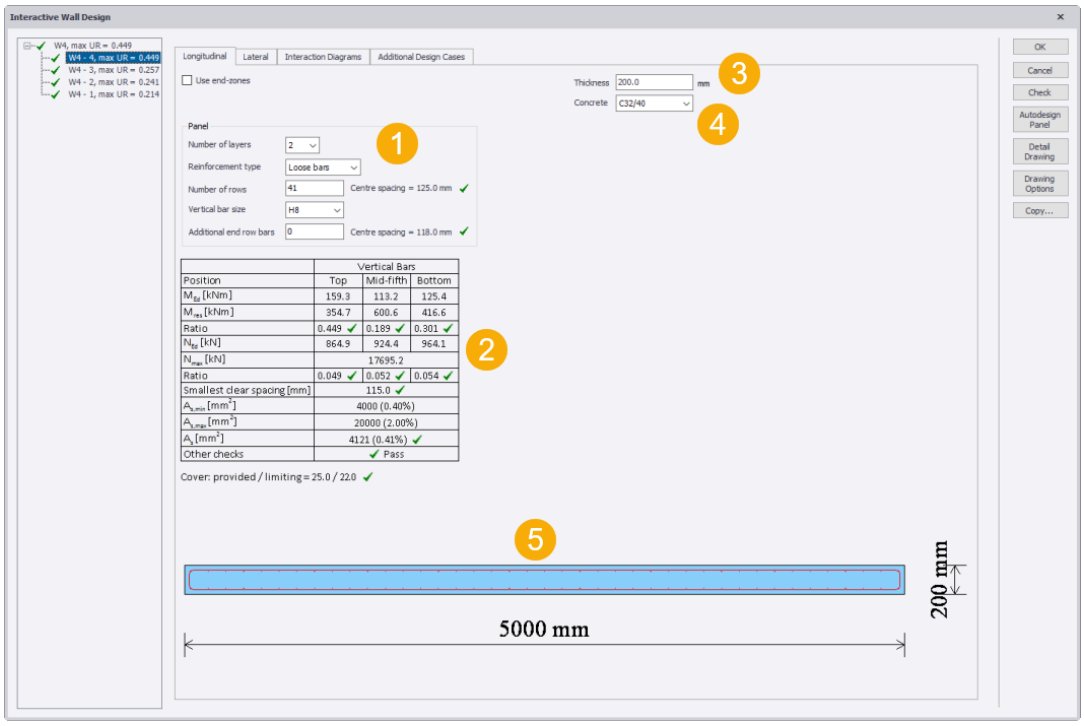
The top row in this pane shows the Wall summary, consisting of the overall utilization ratio and design status.

- With this row selected you can edit the thickness and grade for all stacks simultaneously.

Subsequent rows show the design status of each panel and associated utilization ratio.

- To design a particular panel, click on the corresponding row for the panel in the summary pane.

2. Longitudinal tab (no end-zones)

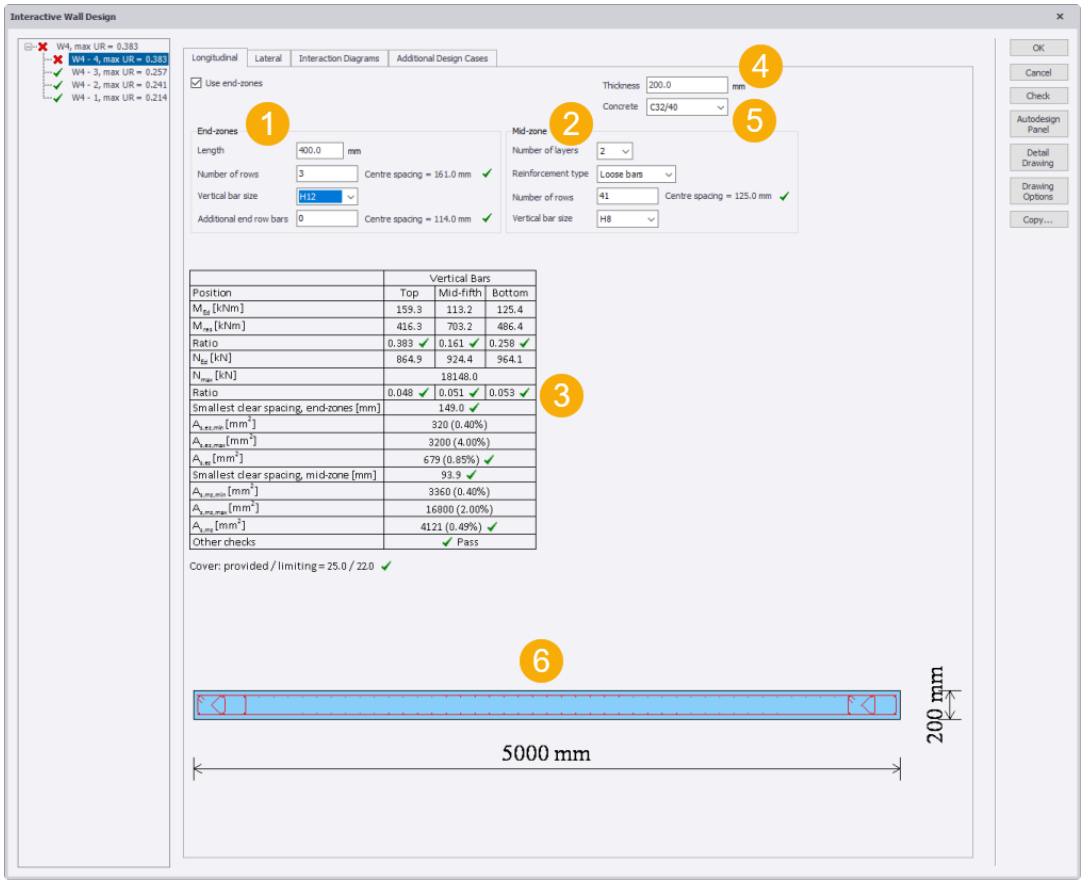


Use end-zones : With this box unchecked end-zones are not used.

1. **Panel**: Used for adding either one or two layers of bars in the panel.
   - Number of layers - (1, or 2)
   - If Reinforcement type = loose bars
     - Number of rows - the number of bars in each layer
     - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
     - Vertical bar size - specifies the size to be checked
     - Additional end row bars - specifies the number of additional bars in the end row
     - Centre spacing (end rows) - the spacing between layers (measured centre to centre)
   - If Reinforcement type = mesh
     - Mesh size - specifies the mesh size to be checked
     - End row vertical bar size - specifies the vertical bar size at the ends of the mesh
   - Additional end row bars - specifies the number of additional bars in the end row
   - Centre spacing (end rows) - the spacing between layers (measured centre to centre)

2. **Design Summary Table**: Displays the most critical result from all combinations:
    - Design and Resistance Moments and Moment Ratios
    - Axial Force, Axial Resistance and Axial Ratios
    - Smallest clear bar spacing
    - Minimum area of steel
    - Area of steel provided
3. **Thickness**: Used for changing the thickness of the current panel.
4. **Concrete Droplist**: Used for changing the concrete grade for the current panel.
5. **Cross-section** : The drawing displays:
    - Exact bar positions (drawn to scale)
    - Link/Tie locations
    - Section dimensions

2. Longitudinal tab (with end-zones)

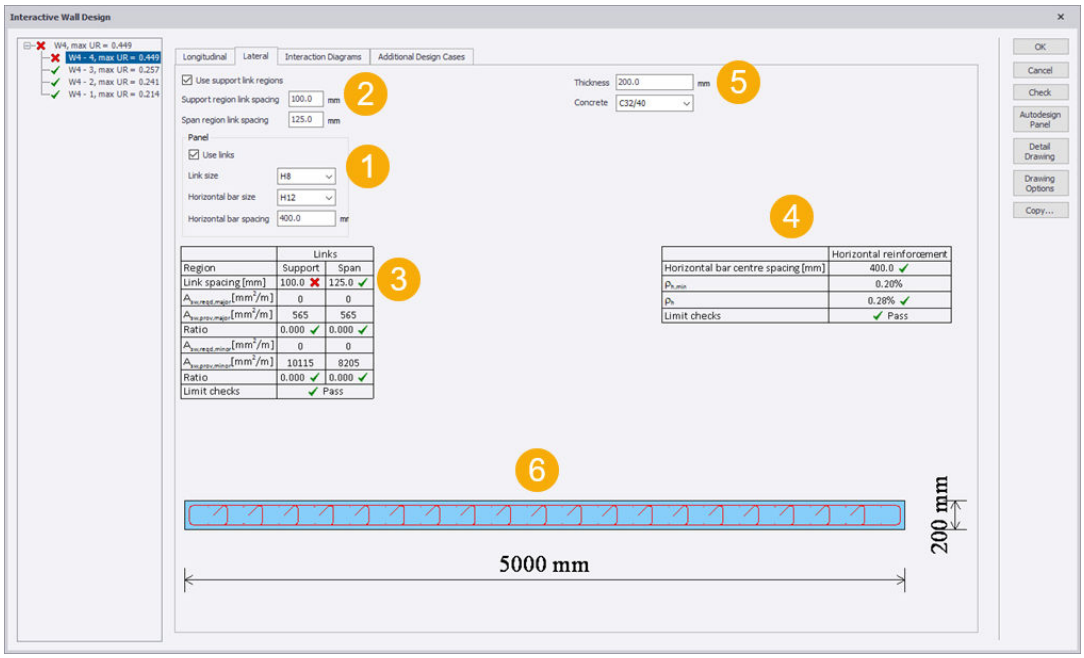


Use end-zones : With this box checked end-zones are used.

1. **End-zones**: Used for adding bars in the end-zones.
    - Length - length of each end-zone
    - Number of rows - the number of loose bars in each end-zone
    - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
    - Vertical bar size - specifies the size to be checked
    - Additional end row bars - specifies the number of additional bars in the end row
    - Centre spacing (end rows) - the spacing between layers in the end-zone (measured centre to centre)
2. **Mid-zone**: Used for adding either one or two layers of bars in the panel between the end-zones.
    - Number of layers - (1, or 2)
    - If Reinforcement type = loose bars
        - Number of rows - the number of bars in each layer
        - Centre spacing - the spacing between adjacent bars in a layer (measured centre to centre)
        - Vertical bar size - specifies the size to be checked
    - If Reinforcement type = mesh
        - Mesh size - specifies the mesh size to be checked
        - End row vertical bar size - specifies the vertical bar size at the ends of the mesh
3. **Design Summary Table**: Displays the most critical result from all combinations:
    - Design and Resistance Moments and Moment Ratios
    - Axial Force, Axial Resistance and Axial Ratios
    - Smallest clear bar spacing
    - Minimum area of steel
    - Area of steel provided
4. **Thickness**: Used for changing the thickness of the current panel.
5. **Concrete Droplist**: Used for changing the concrete grade for the current panel.
6. **Cross-section** : The drawing displays:
    - Exact bar positions (drawn to scale)
    - Link/Tie locations

- Section dimensions

3. Lateral tab



1. **Panel**:
   - **Use links/ties** : Select to specify links/ties.
     - **Link/Tie size**: Used to change the size of link/ties (all must have the same size).
   - **Horizontal bar size**: Used to specify the size of horizontal bars.
   - **Horizontal bar spacing**: Used to specify the vertical spacing of horizontal bars.
2. **Use support region links/ties**: Select to design support regions for the links/ties.

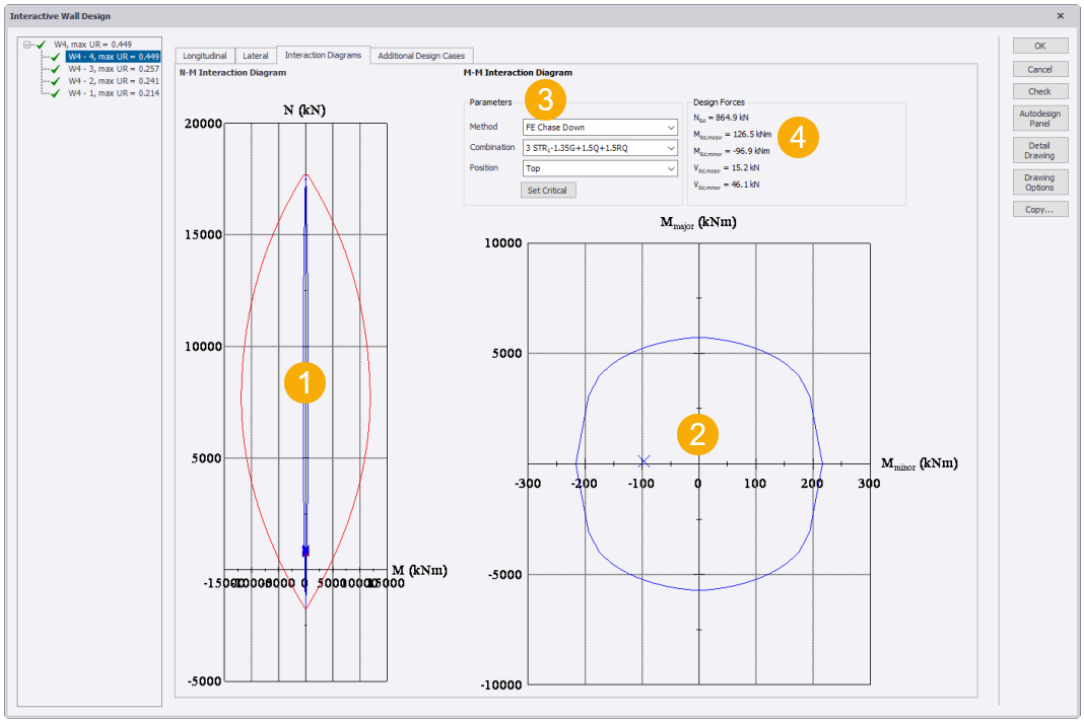
   NOTE Only displayed provided Use links/ties box is also checked.

   - **Link/Tie spacing** : Specifies the link/tie spacing (if support regions are applied two different spacings can be specified)
3. **Link/Tie Design Summary Table**: Displays the most critical result from all combinations:
   - Link/Tie area over spacing required, major
   - Link/Tie area over spacing provided, major
   - Link/Tie utilization ratio, majo
   - Link/Tie area over spacing required, mino

- Link/Tie area over spacing required, mino
- Link/Tie utilization ratio, minor

4. **Horizontal Reinforcement Summary Table**: The table displays the horizontal bar spacing and reinforcement ratios.
5. **Thickness**: Used for changing the thickness of the current panel.
6. **Cross-section** : The drawing displays:
   - Exact bar positions (drawn to scale)
   - Link/Tie locations
   - Section dimensions

4. Interaction Diagrams tab

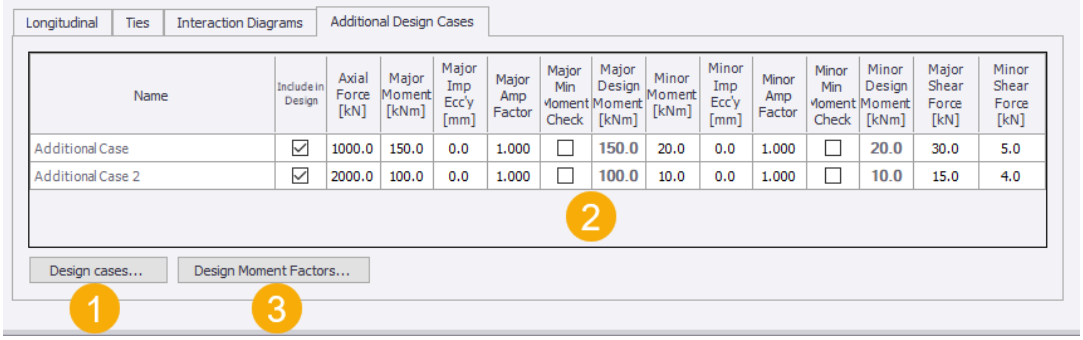


1. **N-M Interaction diagram** : Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
   - The curves for bending about the major axis are shown in red
   - The curves for bending about the minor axis are shown in blue
2. **M-M Interaction diagram** : The diagram is different for each value of axial force. Initially the diagram at the axial force of the critical combination is drawn - this is the combination with the highest M_{Ed} / M_{res} ratio.

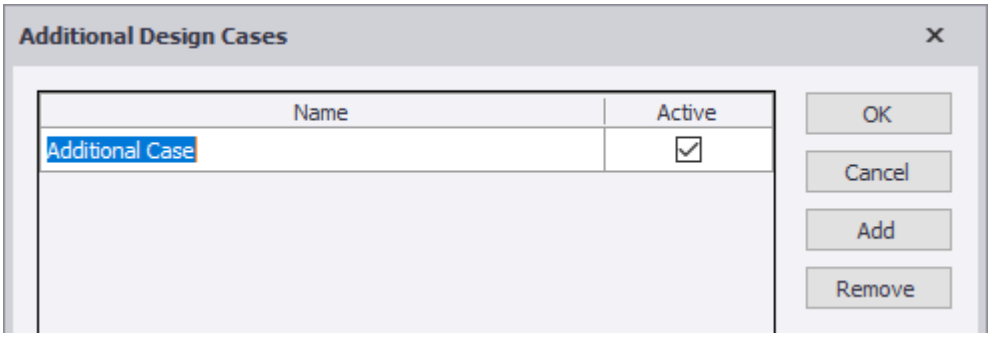
3. **Parameters**: Select the analysis method, combination, and position for which the diagrams are displayed.
   - **Set Critical** button: If you have changed the parameters for which the diagrams are displayed, you can click this button in order to revert back to the critical parameters.
4. **Design Forces**: The design forces applicable for the selected parameters.

5. Additional Design Cases tab

This tab can be used for example to design for results from Post Tensioning analysis programs.



1. **Design cases...** button: Click to open a dialog in which to add any additional design cases.



   The design case names defined here are available to all columns and walls in the model.

   The **Active** box in this dialog is used to control which design cases are considered (for the entire model).

   After clicking OK you are then able to specify the design case design forces in the **Design cases table**.

2. **Design cases table**: Each design case added via the **Design cases...** button appears in the table, but only those that were marked as active (for the entire model) are then available to have design forces applied.

   In order to apply the design case design forces to a given panel you must:

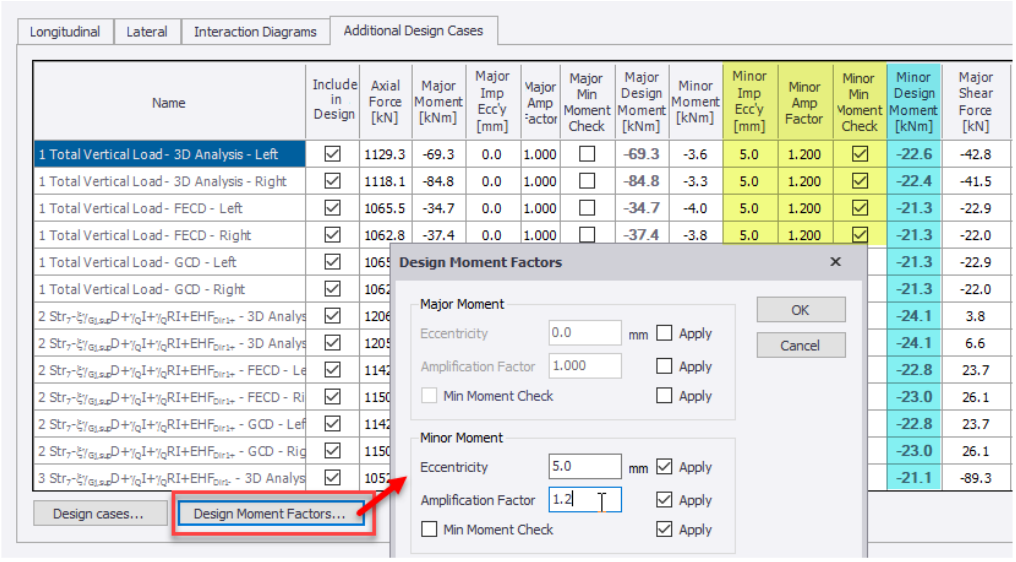
   - Select the required panel in the panel summary

- Select **Include in Design** for the design case
- Enter the design forces

3. **Design Moment Factors...** button: Offers three potential "Design Moment" adjustments for each direction:
   - Set an imperfection eccentricity allowance (Eurocode only). This is added to the analysis moment.
   - Apply an amplification factor to allow for Second Order Effects (could also be considered as a way to introduce an extra factor of safety).
   - Apply a minimum moment check in one or both directions (the calculation of this is specific to the Head Code set and is a function of the section dimension "h" in the direction considered).

   When applied, the resulting adjusted design moment is automatically calculated and displayed in the dialog.

   The adjustment values and options can be applied to individual Cases and also quickly in a single operation to all Active Cases (those with "Active" option checked on) as shown below:



6. Buttons

Button/option	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.

Button/option	Description
Autodesign Panel	Performs an autodesign reselecting bars for the current panel
Detail Drawing	Creates a detail drawing for the selected member
Drawing Options	Opens the DXF Export Preferences dialog
Copy	Opens a new dialog enabling the selective coping of wall thickness/ concrete grade/reinforcement from the selected panel to other panels in the same wall.

7. Suspend Checking option

By default this option is unselected and checks are made after every edit. For large structures with many combinations interactive checking takes time which can be frustrating if you want to make several changes.

When this option is selected, checking is suspended. (The option also changes to red to highlight that this is the case).

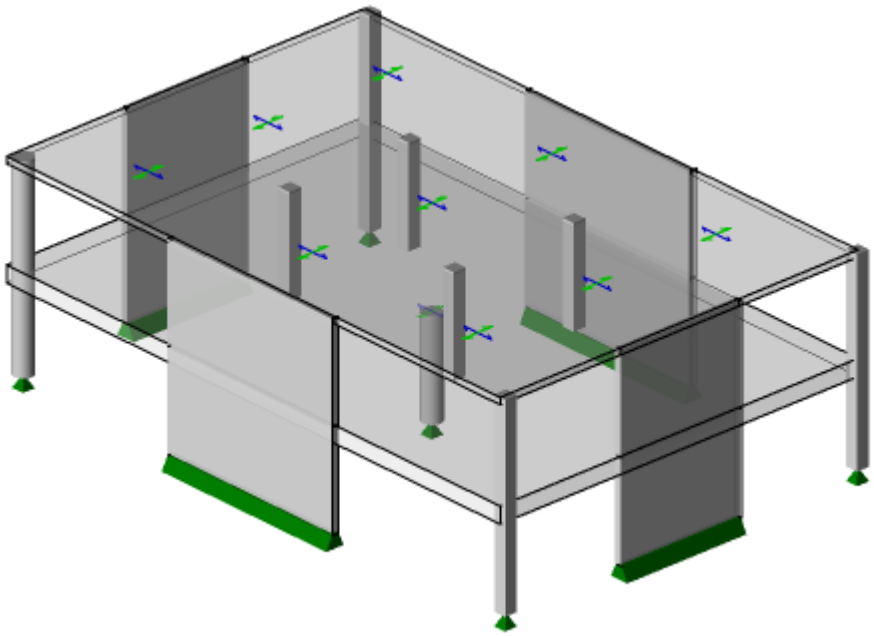
Checks are only carried out and results updated when the option is unselected once more.

See also

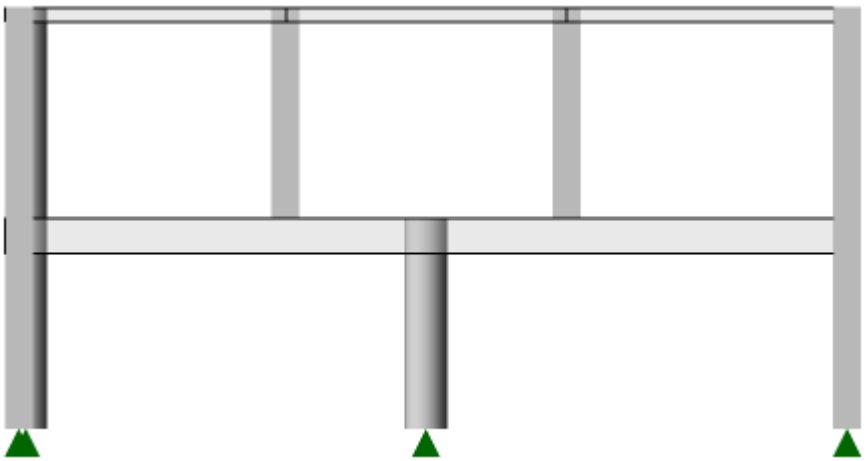
Interactive concrete member design (page 173)

3.8 Flat slab design workflow

A simple flat slab model as shown below is used in order to demonstrate the techniques involved in the slab design process.

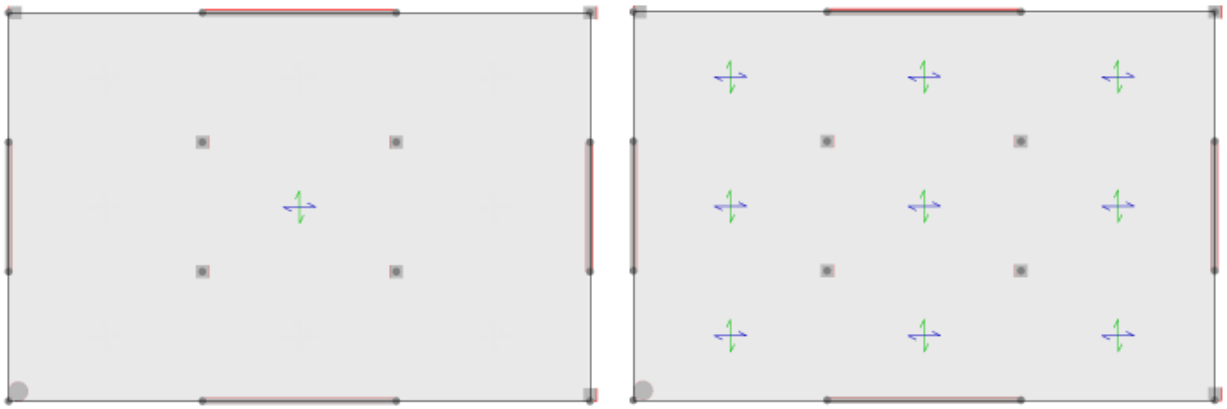


Note that there is a transfer level at the first floor:



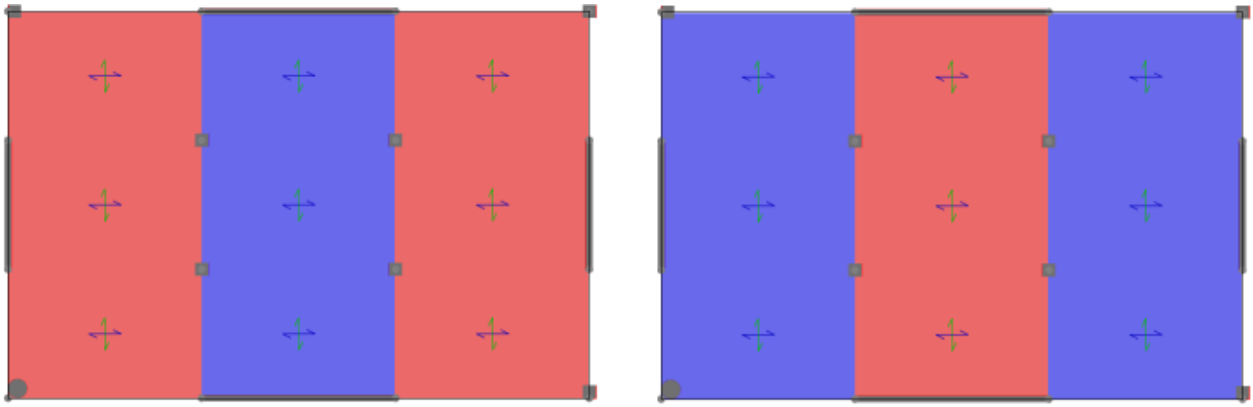
Set up pattern loading

If necessary you should consider manually splitting and joining slab panels to facilitate management of the pattern loading process.



By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by: toggling the loading status via **Update Patterns** on the **Load** tab on the ribbon.

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via **Update Patterns** on the **Load** tab on the ribbon.



For further details, see: .

Establish slab design moments

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Analyze All**, **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE chasedown model results - as each floor is analyzed individually this method mimics the traditional design approach.

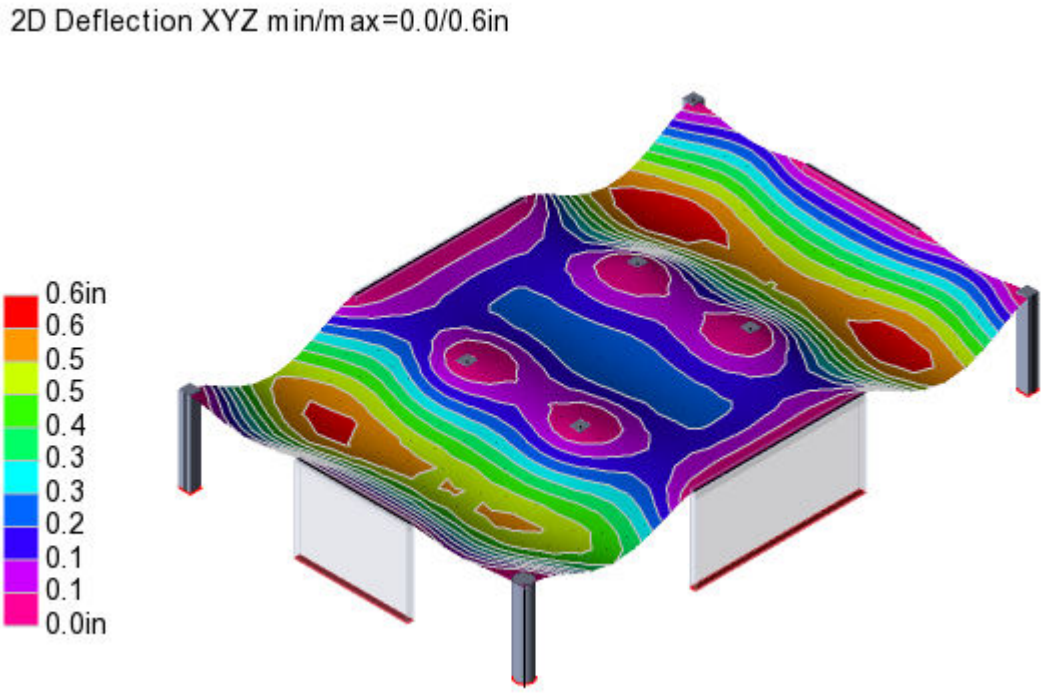
If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

NOTE It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however: - you may choose to mesh them to cater for the possibility of un-braced flat slab design. - more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Consider simple (linear) deflection (US customary units)

Approximate slab deflections can be obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View, (typically by opening a separate Level view of each floor).

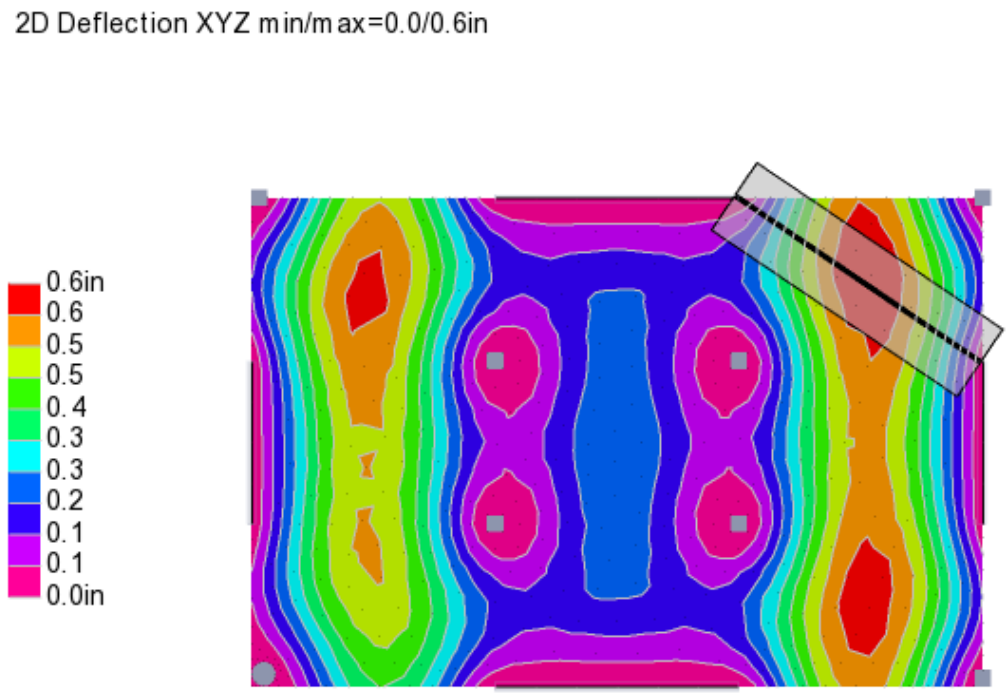
NOTE Deflection results for combinations should be viewed based on "service" rather than "strength" factors - in this way the applied stiffness adjustments do not need to account for load factors.



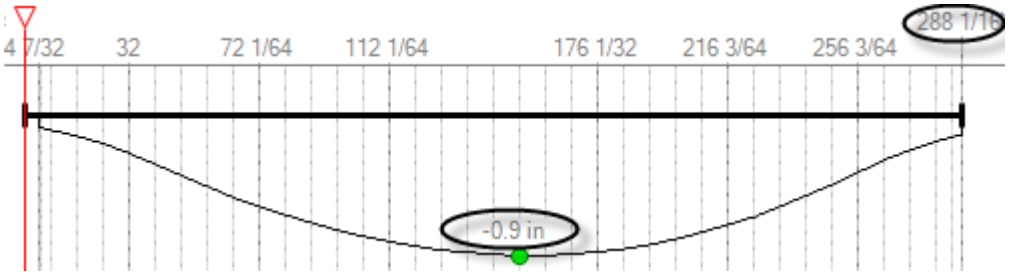
NOTE For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

If the Head Code you are working to prevents you from running a **Rigorous Deflection Check** you can still use a "Deemed-to-Satisfy" method to assess deflections utilizing slab strips as outlined below:

1. With the level view displayed in 2D, click Create Strip and create a strip between supports.



2. To display the strip results right-click the strip and choose Open Load Analysis View. The maximum deflection and span of the strip are both reported.



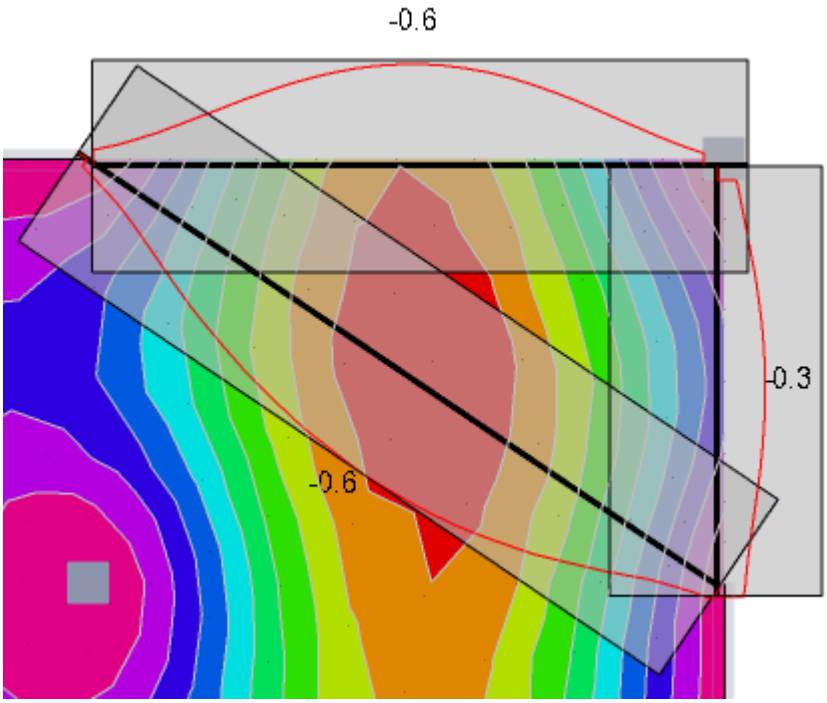
In this example there is a requirement to limit total deflection to span/240

Taking the span as the length of the slab strip: 288in

Deflection limit: 288/240 = 1.2 in

Actual deflection: 0.9 in - the check passes

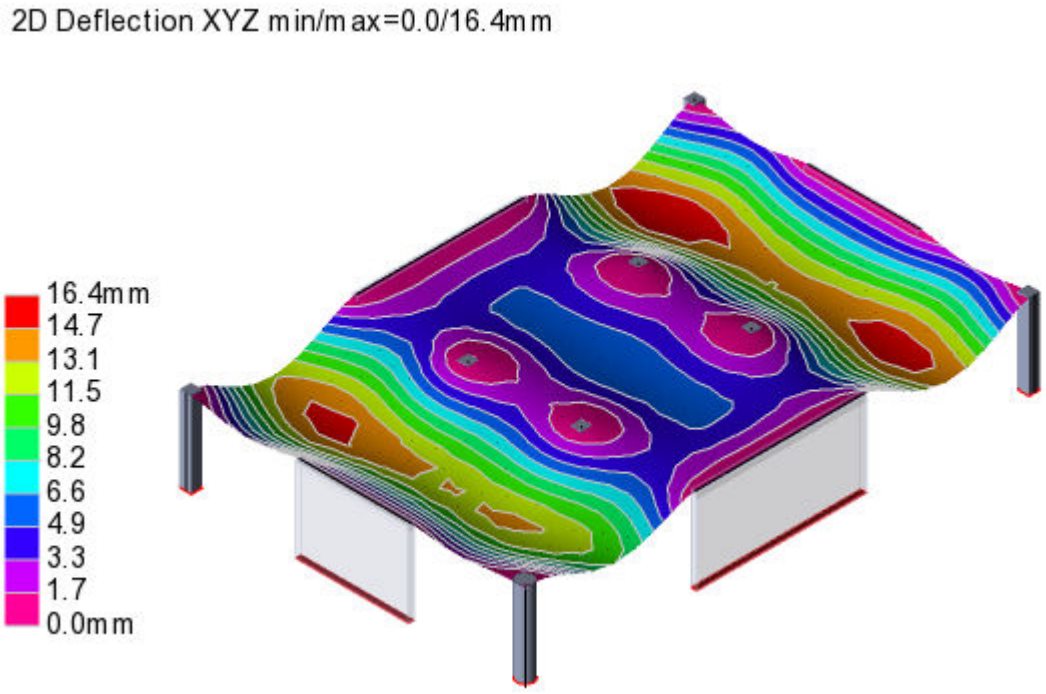
The initial check was performed taking the diagonal across a slab panel. Checks should also be made between the horizontal and vertical spans. More generally, check the max deflection occurring along a straight line between any two support points.



Consider simple (linear) deflection (metric units)

Approximate slab deflections can be obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View, (typically by opening a separate Level view of each floor).

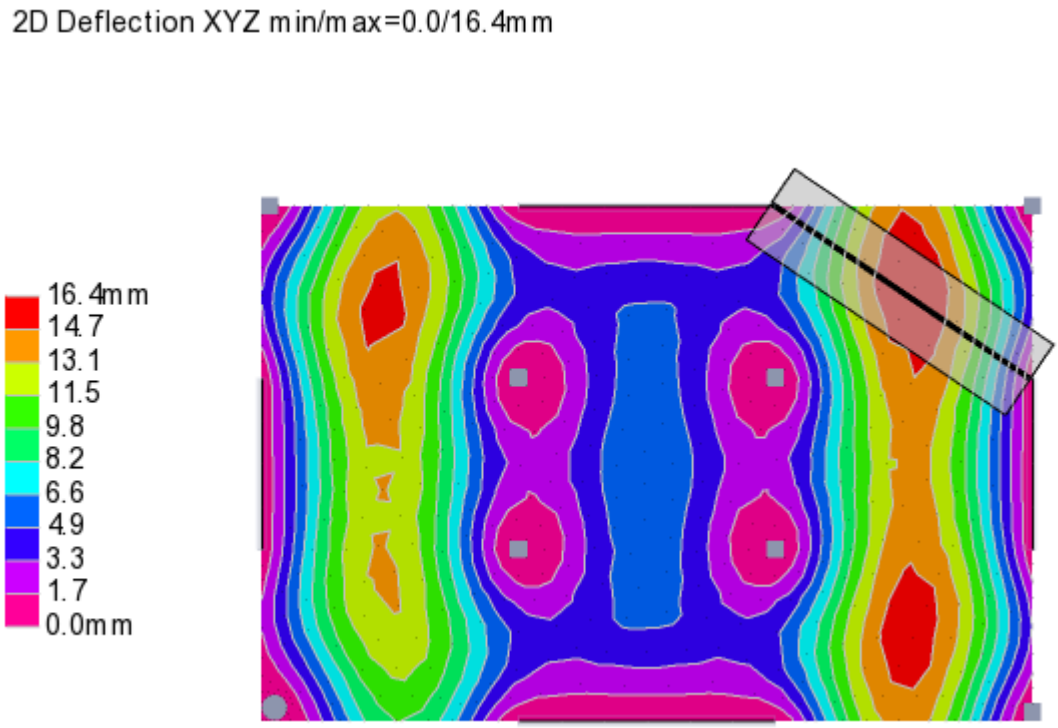
NOTE Deflection results for combinations should be viewed based on "service" rather than "strength" factors - in this way the applied stiffness adjustments do not need to account for load factors.



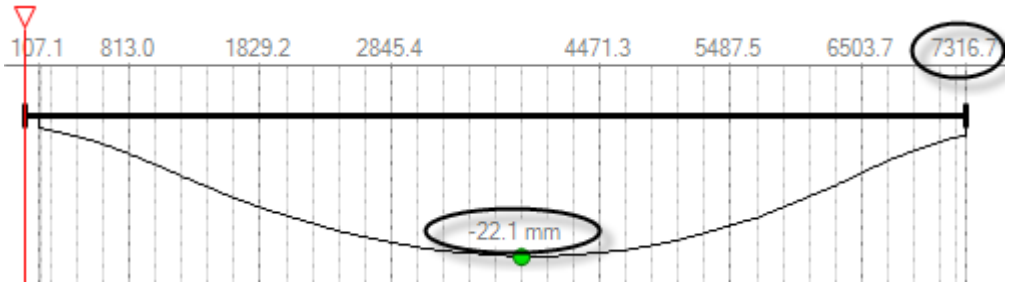
NOTE For Flat Slab panels - Deflection checks are not performed automatically. In practice flat slab models are often irregular, so some degree of engineering judgement will be required.

If the Head Code you are working to prevents you from running a **Rigorous Deflection Check** you can still use a "Deemed-to-Satisfy" method to assess deflections utilizing slab strips as outlined below:

1. With the level view displayed in 2D, click Create Strip and create a strip between supports.



2. To display the strip results right-click the strip and choose Open Load Analysis View. The maximum deflection and span of the strip are both reported.



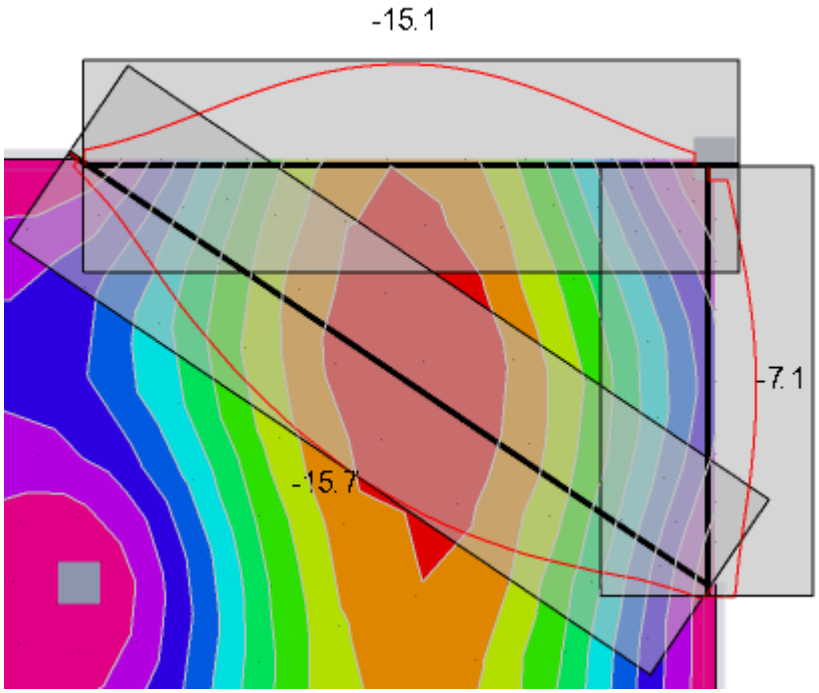
In this example there is a requirement to limit total deflection to span/250

Taking the span as the length of the slab strip: 7317mm

Deflection limit: 7317/250 = 29mm

Actual deflection: 22mm - the check passes

The initial check was performed taking the diagonal across a slab panel. Checks should also be made between the horizontal and vertical spans. More generally, check the max deflection occurring along a straight line between any two support points.



Select a level, or sub-model

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered. Running **Design Slabs** or **Design Patches** from the **Design** tab on the ribbon will take longer as it considers all the slabs and patches in the model.

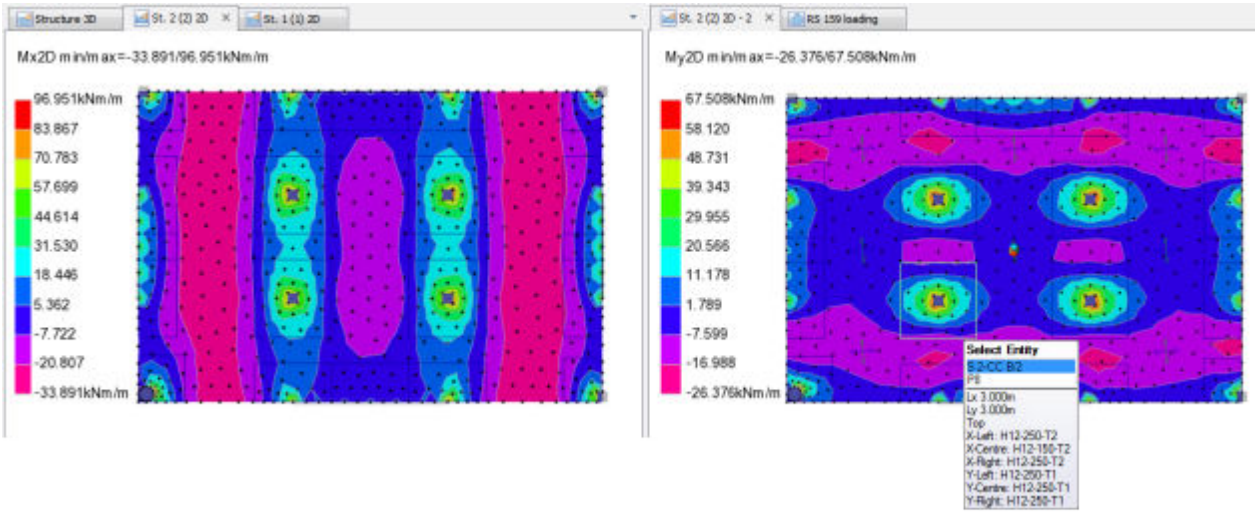
Add patches

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE

chasedown result type) have Mx moments on in one view on the left and My moments in a second view on the right, as below:



By doing this, it is possible to see how patches extend over the peaks.

Typically, at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimization stage.

In a "slab on beam" situation, you may want to add beam and wall patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to none and the panel design should still pass.

Design panels

NOTE Panel design is dependent on the areas of patches (patch areas which are excluded from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

- From the **Design** tab on the ribbon run **Design Slabs** in order to design or check all the panels in the model - by default newly created panels will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

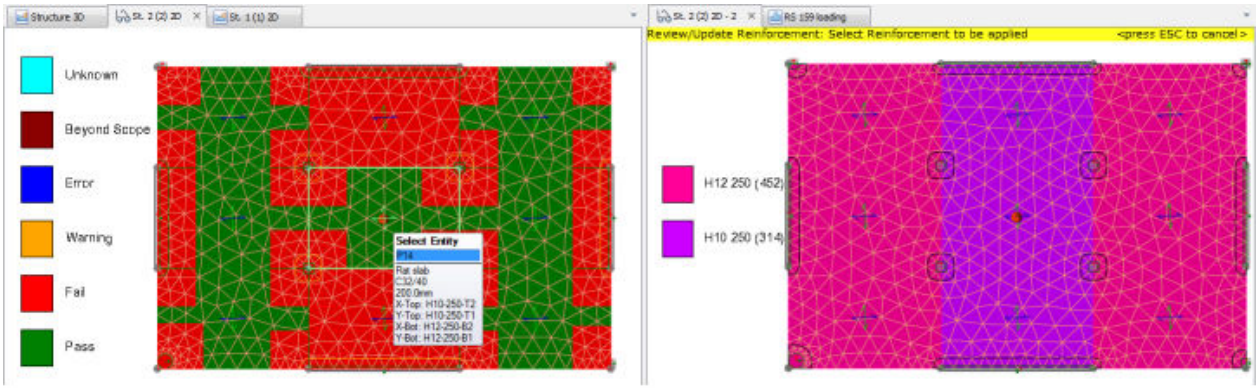
- In the 2D View of the floor which you want to design right-click and choose either **Design Slabs** or **Check Slabs**. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

NOTE When accessed from the right-click menu, these commands operate as follows:

- **Design Slabs** will re-design slabs (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Slabs** will check the current reinforcement in slabs regardless of the current autodesign setting.

Review/optimize panel design

Once again it is suggested that you use split Review Views to examine the results as indicated below.



In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

The view on the left shows Slab Design Status, the view on the right shows Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a flat slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to).

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the

patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Design tab on the ribbon run Design Patches in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

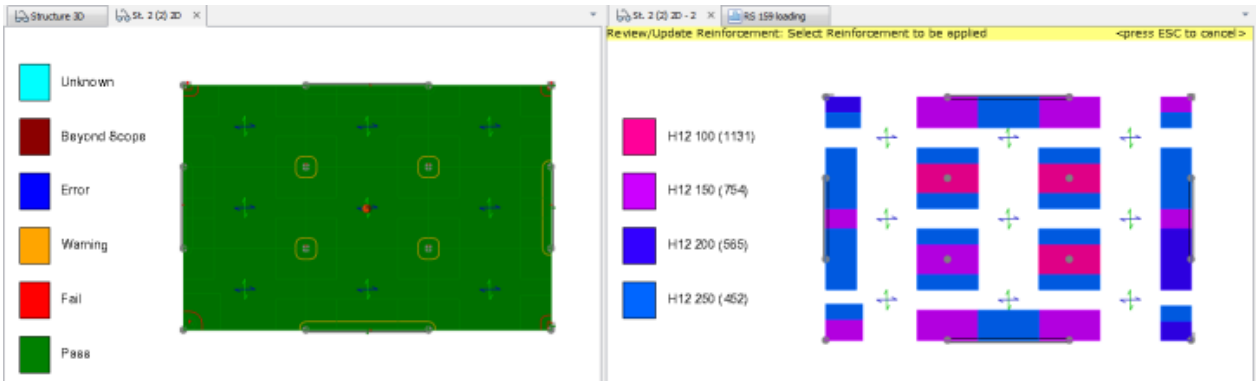
- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimized)
- Wall patches - can the width be adjusted (minimized)
- Column patches - Is the size reasonable - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the slab reinforcement to be matched (if the slab has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

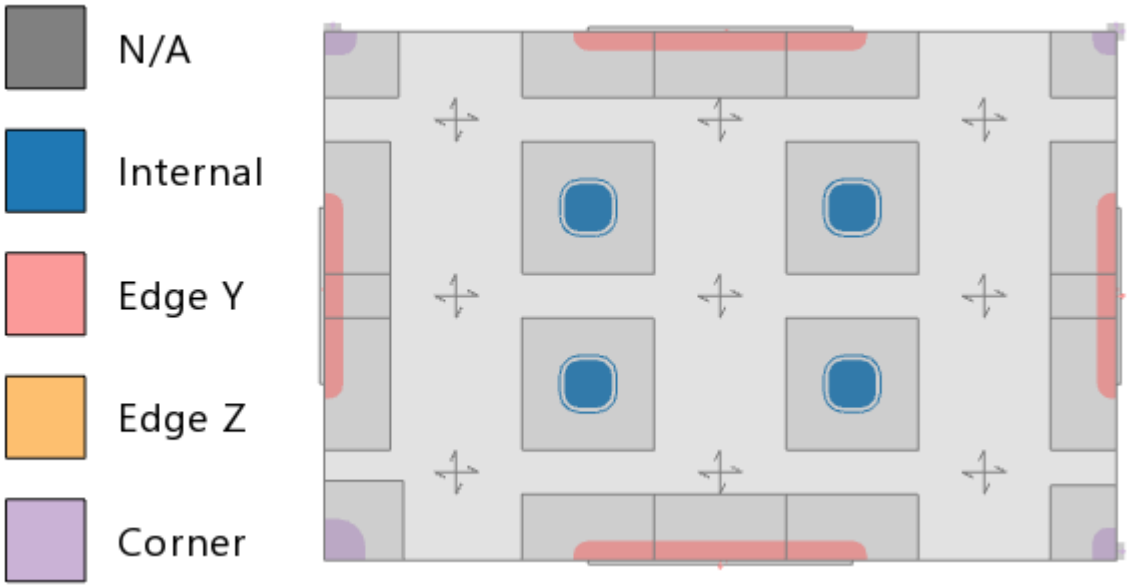


Add and run punching checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added at individual locations or can be apply to multiple locations by windowing. For further details, see:

You can then select any check and review the properties assigned to it. Internal/edge/corner locations are automatically determined (with a user override if you require).

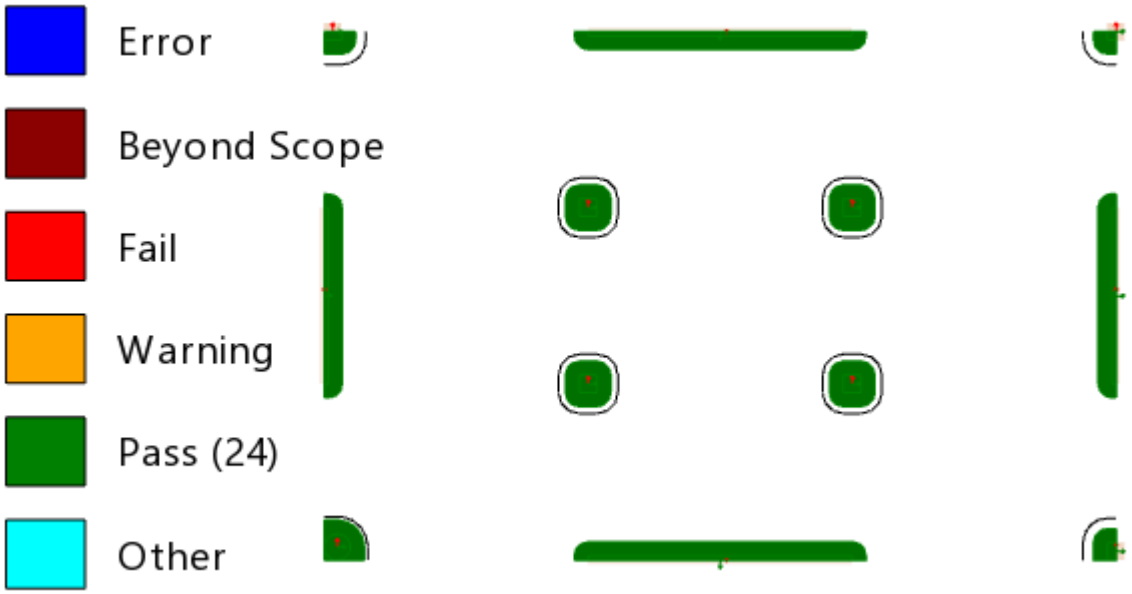


For further details, see: .

Once the punching checks have been created you can proceed to design or check them. For further details, see: .

After design/checking the status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the slab



NOTE In order to see the punching check status in the Review View as shown above, you might need to first switch off Slab Items and Slab Patches in Scene Content.

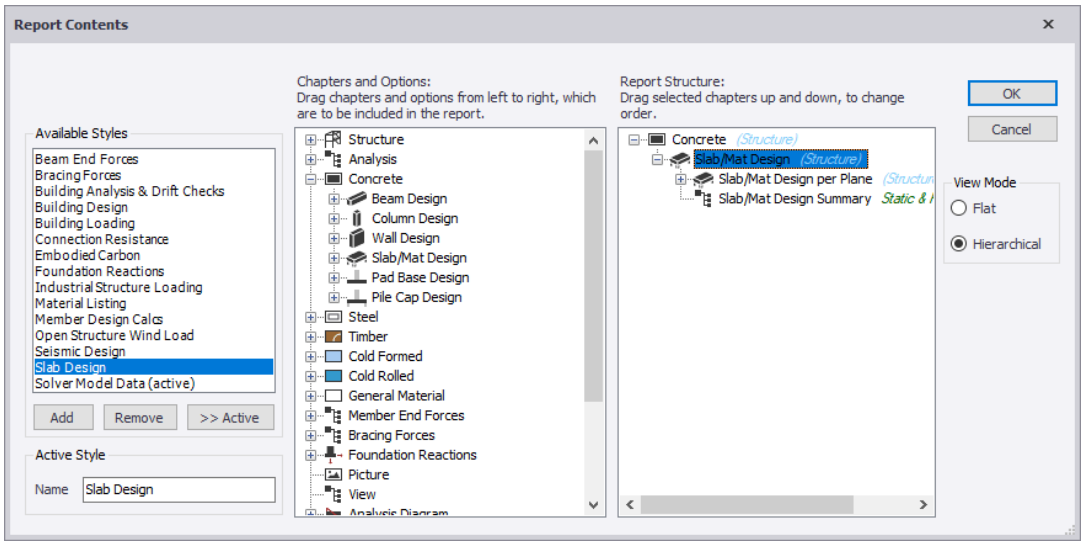
Rigorous deflection check

In Tekla Structural Designer you can choose to adopt a rigorous approach to slab deflection calculation using iterative cracked section analysis. For further details, see: Slab deflections handbook (page 700).

Create reports

To add slab design and punching shear check calculations to any model report, in the Report Contents dialog box, select the report style, then open the

Concrete chapter and drag the **Slab/Mat Design** chapter in the report structure.



This chapter includes two sub-chapters:

- **Slab/Mat Design Summary** - by default this chapter summarizes all the slab panel and patch designs, and punching shear checks in the model. It can be filtered as required.

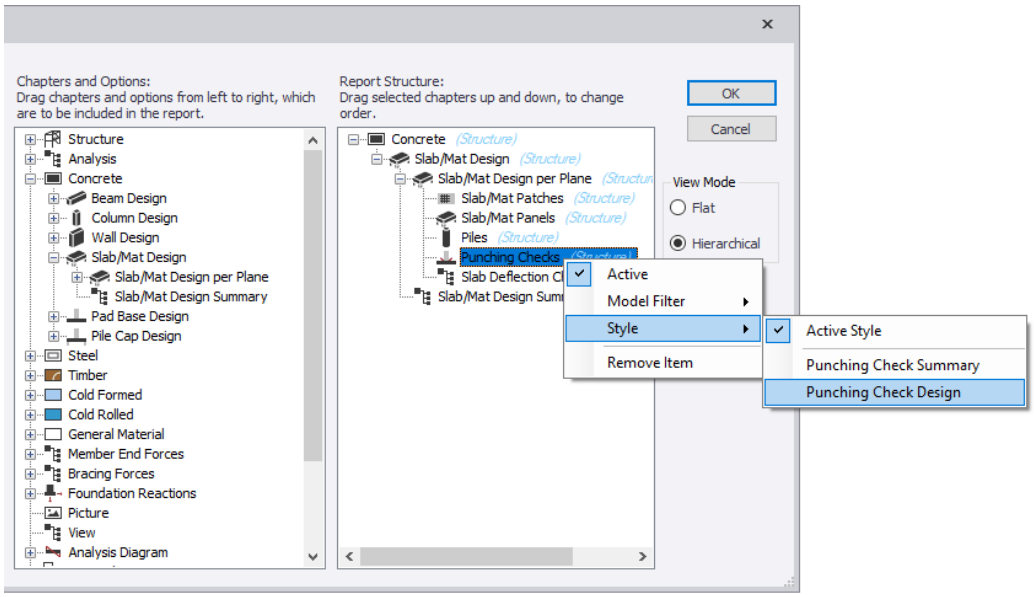
Slab/Mat Design Summary

Static & RSA

Level	Slab	Reference	Type	Thickness [mm]	Grade	Utilization	Status
St. 1 (1) : 3.000m	S 1	P1	Flat slab	500.0	C32/40	0.800	Pass
St. 1 (1) : 3.000m	S 1	S 1-CC A/1	Column Patch	500.0	C32/40	0.877	Pass
St. 1 (1) : 3.000m	S 1	S 1-CC C/3	Column Patch	500.0	C32/40	0.804	Pass
St. 1 (1) : 3.000m	S 1	S 1-CC C/2	Column Patch	500.0	C32/40	0.801	Pass
St. 1 (1) : 3.000m	S 1	S 1-CC B/2	Column Patch	500.0	C32/40	0.798	Pass
St. 1 (1) : 3.000m	S 1	S 1-CC B/3	Column Patch	500.0	C32/40	0.791	Pass
St. 1 (1) : 3.000m	S 1	S 1-CC F/8	Column Patch	500.0	C32/40	0.784	Pass
St. 1 (1) : 3.000m	S 1	S 1-W2-DP35	Wall Patch	500.0	C32/40	0.694	Pass

- **Slab/Mat Design per Plane** - this chapter contains sub-chapters for the individual slab panel and patch designs, and punching shear checks. Each of these have a member report style assigned which controls the level of output produced.

  The style can be selected for each sub-chapter from the right click context menu as shown below.



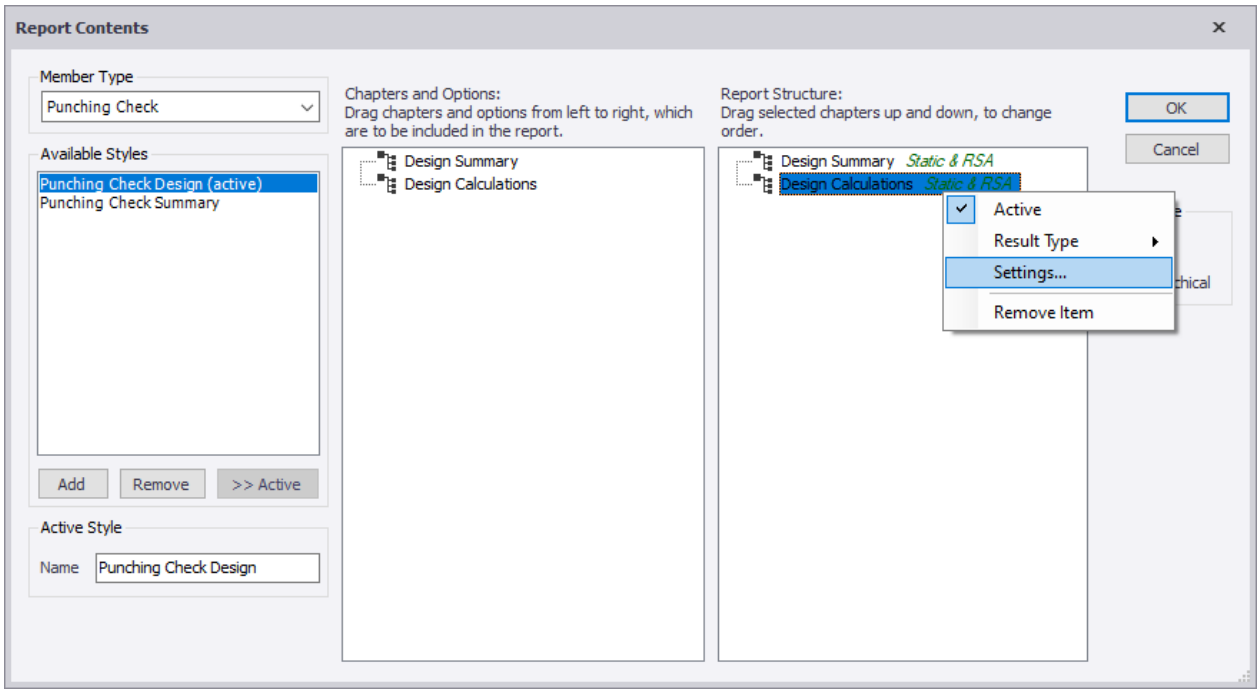
NOTE You can either choose a named style, or you can choose "Active Style" (in which case when you come to display the model report it will adopt whichever member report style is active at that time).

Each named style can be configured to include more or less detail, which can be done in a separate operation as follows:

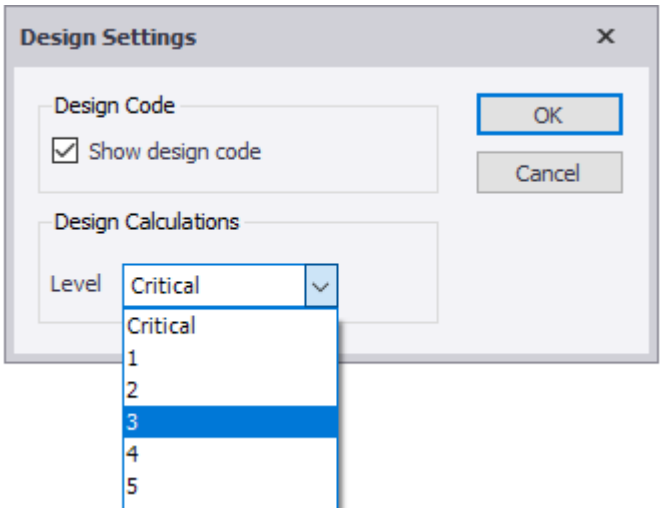
1. Close the above dialog box, and open the **Member Reports** dialog box.
2. Choose the member types in turn (e.g. Punching Check, Slab/Mat Panel, Slab/Mat Patch) and select an existing style, (or add an new one if required).
3. Drag chapters and options into the report structure as required.

NOTE This dialog box is also used to review which member report style has been nominated as the "Active Style". You can switch this to the currently selected style by clicking the **>> Active** button.

4. Configure the level of calculation detail, by right clicking on the Design Calculations chapter and selecting **Settings** from the context menu.



- Choose the level of detail that the calculations are to be expanded to in the report, (critical, with no levels expanded being the minimum), then click OK to close the dialog box.



5. Click OK to close the **Member Reports** dialog box.

To print the model report: on the ribbon, go to the **Report** tab, select the report style name and then click **Show Report**.

Concrete

Slab/Mat Design

Slab/Mat Design per Plane

St. 2 (2)

Slab/Mat Patches

S 2-CCA/1

Static

Overall Summary

Strip	Reinforcement Provided [mm²/m]			Area Required	Utilization
	From Slab	Added In Strip	Total Area Provided		
Strip X-Left	H12-250.0-T2	H12-250.0-T2	905	271	0.300
Strip X-Centre	H12-250.0-T2	H12-150.0-T2	1206	1038	0.860
Strip X-Right					
Strip Y-Left					
Strip Y-Centre	H12-250.0-T1	H12-250.0-T1	905	600	0.663
Strip Y-Right	H12-250.0-T1	H12-250.0-T1	905	140	0.155

Slab/Mat Panels

Slab: S 2, Panel: P8

Static

Reinforcement Design Summary

Reinforcement Layer	Reinforcement Provided	Area Provided [mm²/m]	Area Required [mm²/m]	Utilization
Bottom-X	H12-250.0-B2	452	307	0.678
Bottom-Y	H12-250.0-B1	452	279	0.617
Top-X	H12-250.0-T2	452	228	0.504
Top-Y	H12-250.0-T1	452	65	0.144

Punching Checks

PC 17

Static

Summary

	Section	Perimeter	u_0 / u_1 / u_2 [mm]	v_{Ed} [N/mm²]	v_{Rd} [N/mm²]	Ratio	Status	A_{sw} / s_r [mm²/m]	u_{out} [mm]	Studs Provided
Column	400.0x400.0	Loaded	600.0	1.562	5.581	0.280	✔ Pass			
		Control	1096.4	0.809	2.039	0.397	✔ Pass	821	1374.4	5 x H10-140-4-450

Static

1 Combination - Critical

FE Chase Down - Critical

Shear force	V_{Ed} = **98.7** kN	
Total adjustment	ΔV_{Ed} = L + Swt + R = **5.3** kN	
Adjusted perimeter shear force	$V_{Ed,red} = V_{Ed} - \Delta V_{Ed}$ = **93.4** kN	
Moment about major axis	M_{major} = **-47.9** kNm	
Moment about minor axis	M_{minor} = **68.9** kNm	
Bounding rectangle breadth	B_{bound} = **400.0** mm	
Bounding rectangle depth	D_{bound} = **400.0** mm	
Equivalent rectangle breadth	B_{equiv} = **400.0** mm	
Equivalent rectangle depth	D_{equiv} = **400.0** mm	
Effective depth to tension reinforcement in slab	d = **158.0** mm	
Tension reinforcement ratio in slab resisting bending about major axis	$\rho_{l,major}$ = **0.55** %	
Tension reinforcement ratio in slab resisting bending about minor axis	$\rho_{l,minor}$ = **0.67** %	
Tension reinforcement ratio in slab	$\rho_l = MIN[\sqrt{\rho_{l,major} \times \rho_{l,minor}}, 2\%]$ = **0.61** %	EN 1992-1-1:2004 Section 6.2.2(1)
Loaded perimeter	u_0 = **600.0** mm	
First basic control perimeter	u_1 = **1096.4** mm	
Unreinforced punching shear resistance	$v_{Rd,c} = MAX[C_{Rd,c} \times k \times (100 \times \rho_l \times f_{ck})^{1/3}, v_{min}]$ = **0.645** N/mm²	EN 1992-1-1:2004 Section 6.4.4(1)
Magnification factor	$\beta = MIN[\beta_A, \beta_B]$ = **1.500**	
Shear stress at face of loaded perimeter	$v_{Ed,0} = \beta \times V_{Ed} / (u_0 \times d)$ = **1.562** N/mm²	
Maximum punching shear resistance	$v_{Rd,max} = 0.5 \times \nu \times f_{cd}$ = **5.581** N/mm²	EN 1992-1-1:2004 Section 6.4.5(3)
Shear stress utilization ratio	$v_{Ed,0} / v_{Rd,max}$ = **0.280**	

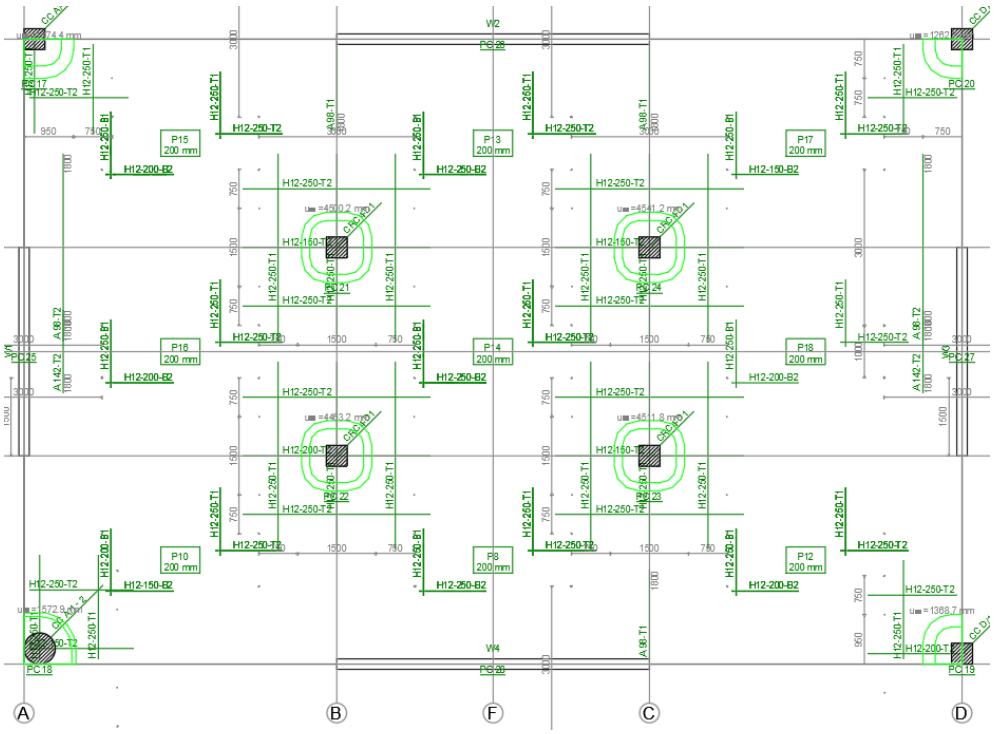
✔ Pass

Create drawings and quantity estimations

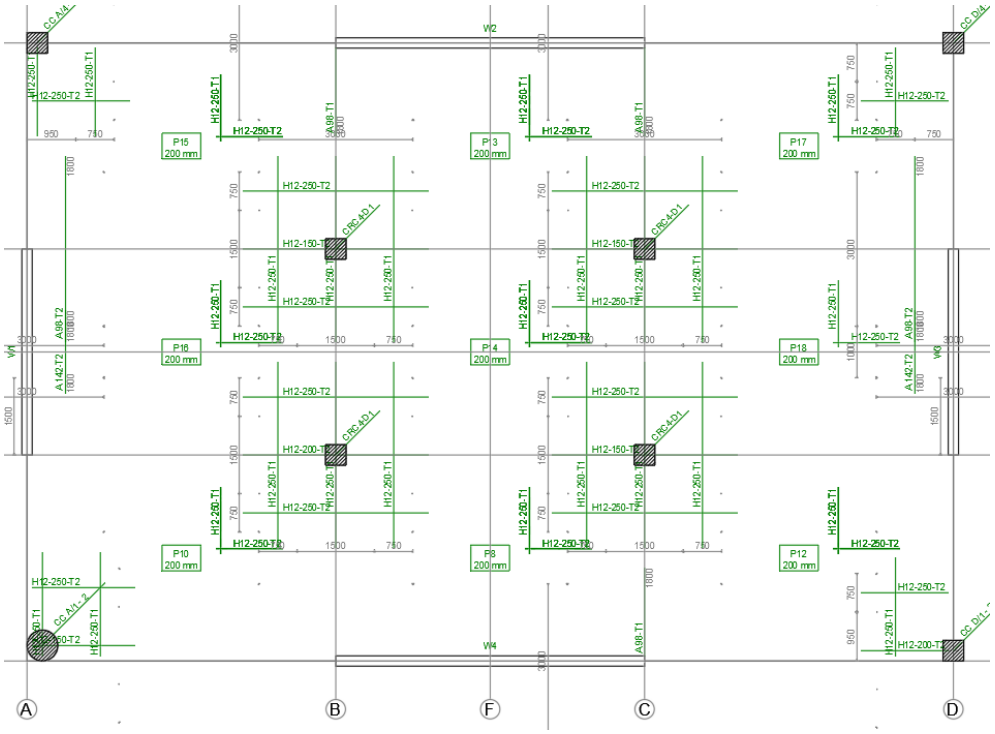
Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.

Two drawing variants are available for slabs and mats:

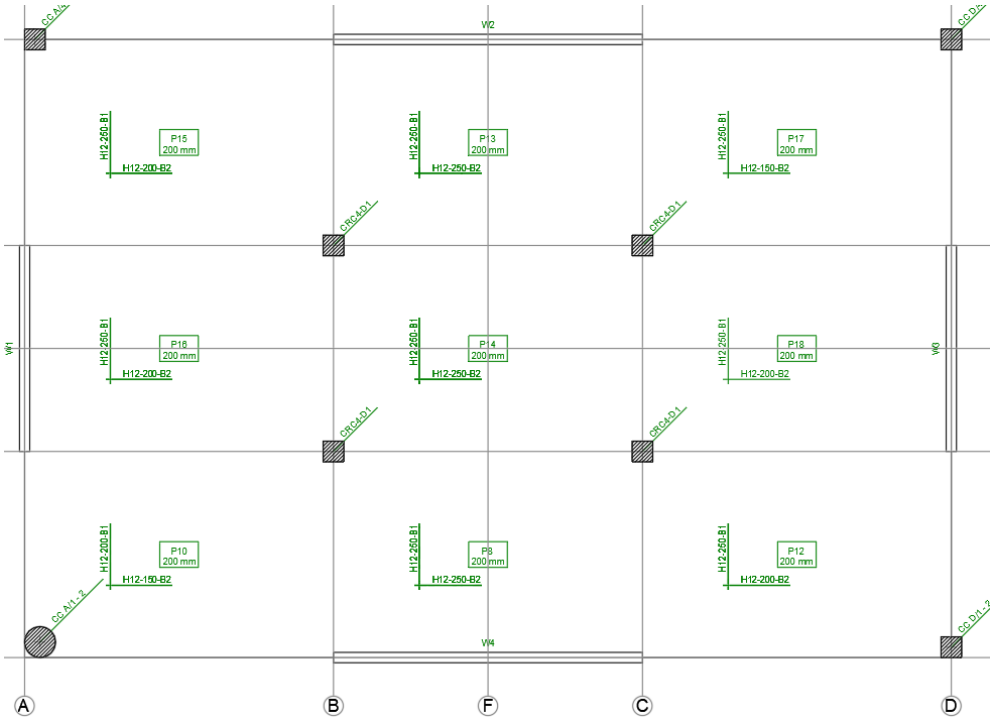
- **slab/mat layout** - created from the Draw tab of the ribbon by selecting **Slab/Mat Detailing**. This variant has pre-configured layer configurations available to limit the reinforcement displayed:
  - **Slab/Mat Detail (All)** - this configuration displays top and bottom reinforcement, and also displays punching perimeters



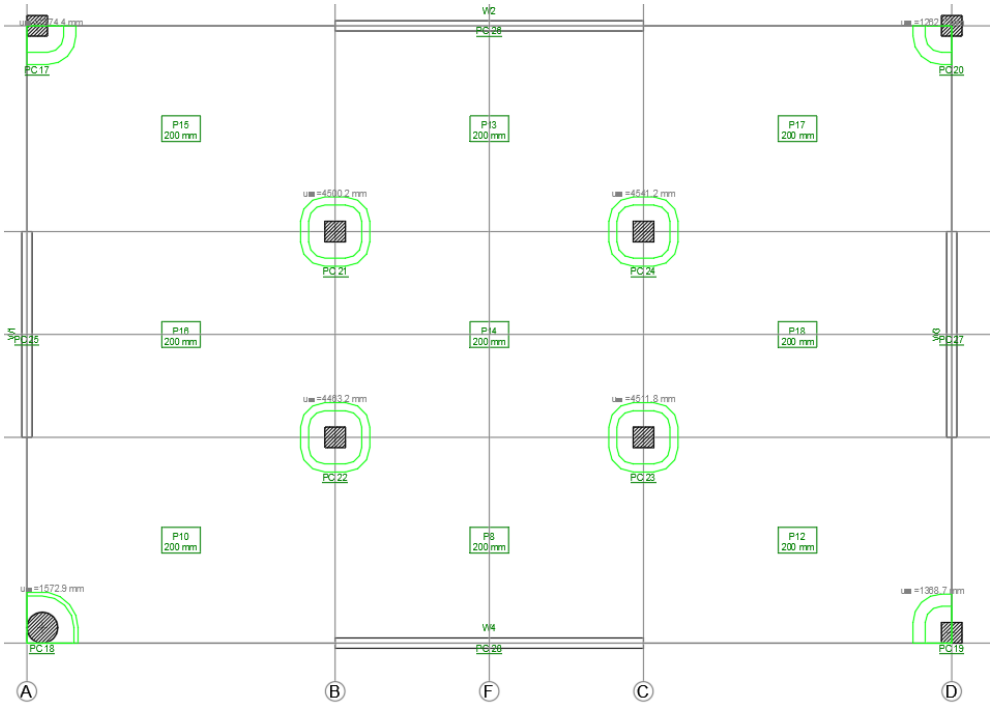
- **Slab/Mat Detail (Top Reinforcement)** - this configuration displays top reinforcement only



- **Slab/Mat Detail (Bottom Reinforcement)** - this configuration displays bottom reinforcement only

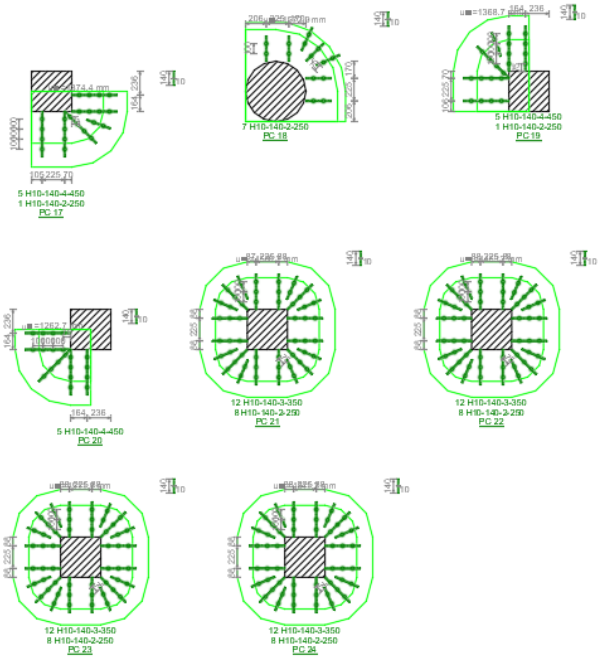


- **Slab/Mat Detail (Punching)** - this configuration displays the punching perimeters on the layout, and also includes separate details of any stud rail reinforcement that has been designed.



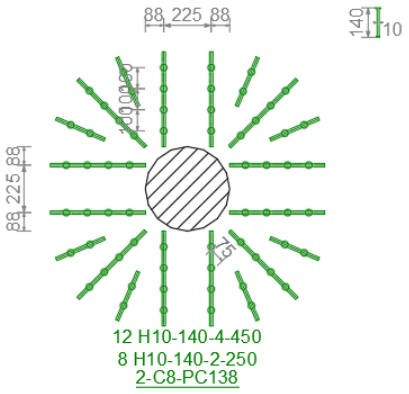
Slab Top and Bottom Reinforcement			
Type	Unit Mass [kg/m]	Total Length [m]	Total Mass [kg]
H12	0.888	4732.220	4202.21
		Total	4202.21
		Total + 10.00 % slab detailing allowance	4622.43
Type	Unit Mass [kg/m²]	Total Area [m²]	Total Mass [kg]
A98	1.54	36.0	55.39
A142	2.23	6.0	13.32
		Total	68.71
		Total + 10.00 % slab detailing allowance	75.58
		Grand Total	4698.01

Stud Rails						
Stud Rail Ref.	Stud Diameter [mm]	Stud Height [mm]	Studs per Rail	Stud Spacing [mm]	Rail Length [mm]	Total No. of Rails
H10-140-3-350	10.0	140.0	3	100.0	350.0	48
H10-140-2-250	10.0	140.0	2	100.0	250.0	41
H10-140-4-450	10.0	140.0	4	100.0	450.0	15



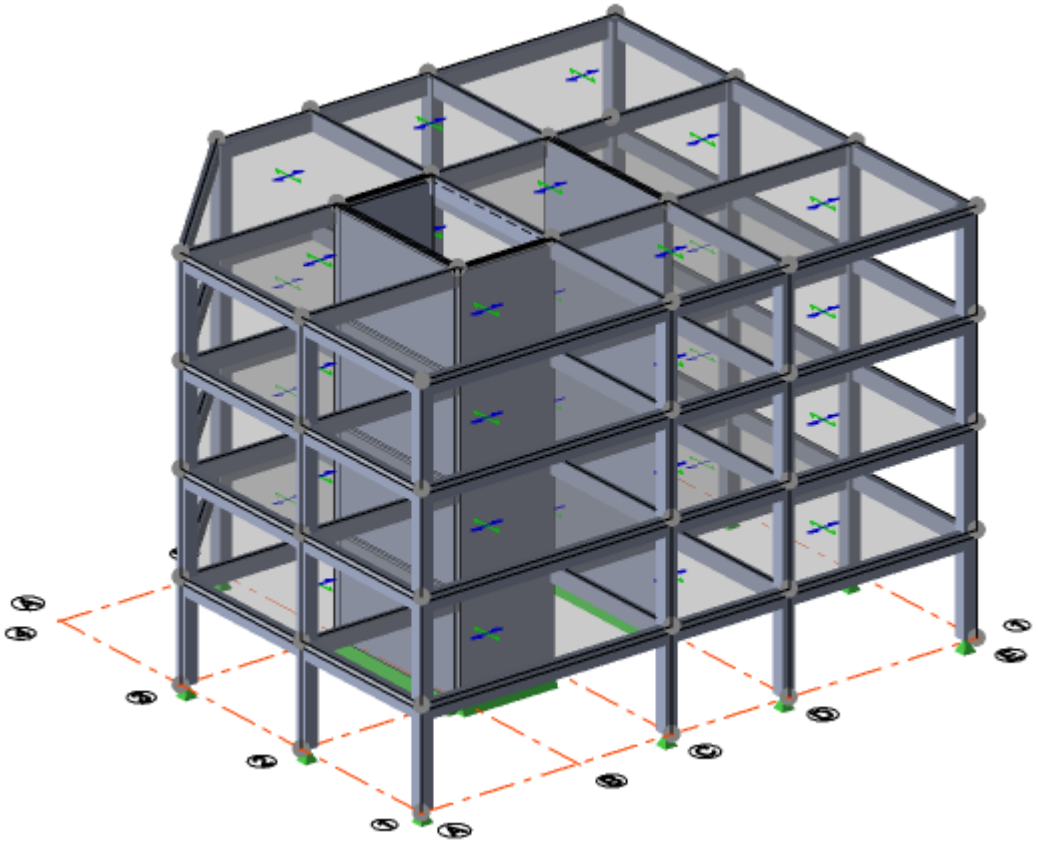
- **punching check detail** - this variant can be created by right clicking on an individual punching check item in a scene view and selecting **Generate Drawing** from the context menu.

Stud Rails						
Stud Rail Ref.	Stud Diameter [mm]	Stud Height [mm]	Studs per Rail	Stud Spacing [mm]	Rail Length [mm]	Total No. of Rails
H10-140-4-450	10.0	140.0	4	100.0	450.0	12
H10-140-2-250	10.0	140.0	2	100.0	250.0	8



3.9 Slab on beams design workflow

A simple slab on beam model as shown below is used in order to demonstrate the techniques involved in the slab design process.

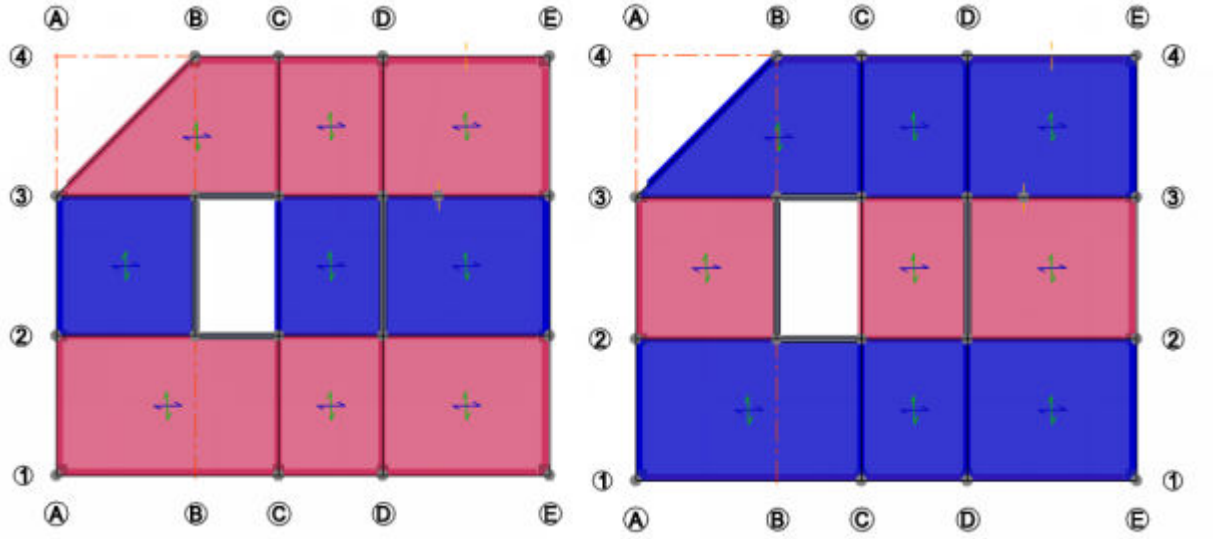


Note that not all the slab panels are rectangular.

Set up pattern loading

By default, only beam loads, and slab loads that have been decomposed on to beams are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each

pattern load by toggling the loading status via Update Load Patterns on the Load ribbon.



Establish slab design moments

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Analyze All**, **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE chasedown model results - as each floor is analyzed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis model results will also be considered.

NOTE It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however: - you may choose to mesh them to cater for the possibility of un-braced flat slab design. - more likely, you may do so to deal with significant transfer slabs - e.g. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Select a level, or sub-model

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the "3D geometry" of a model

may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered. Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

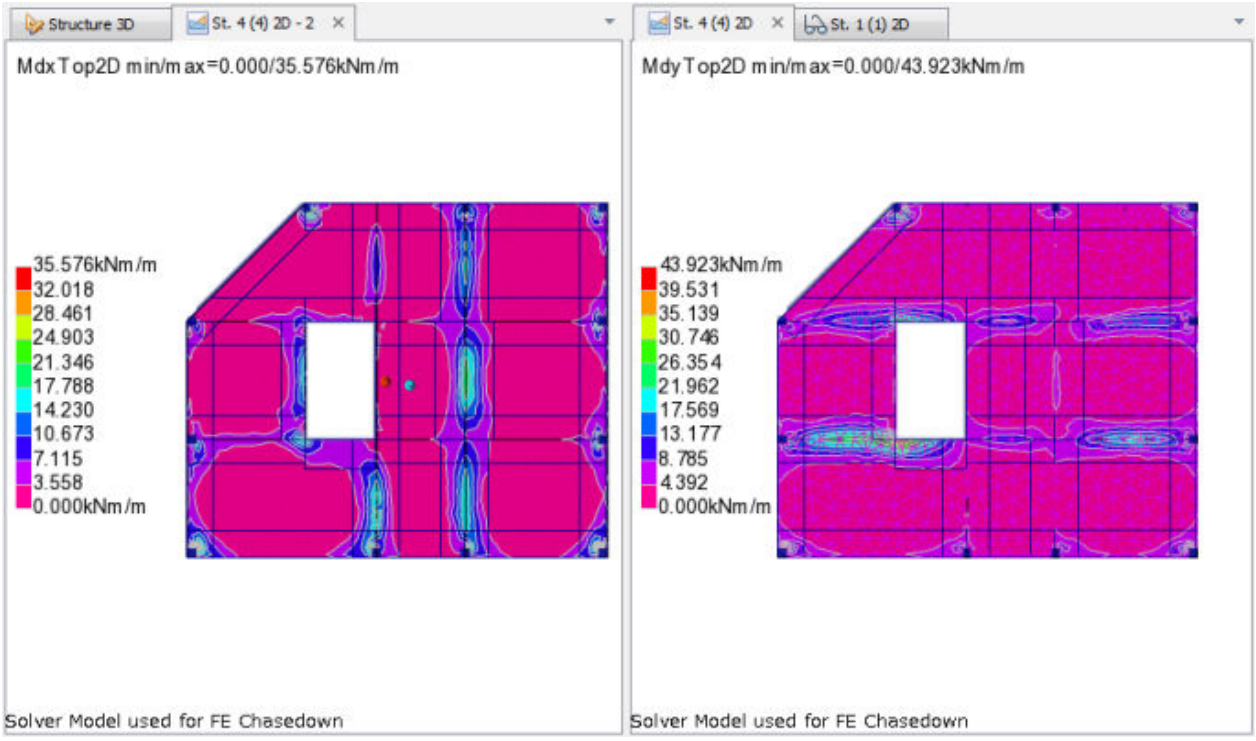
Add beam and wall top patches

You may optionally want to add beam and wall "top surface" patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to none and the panel design should still pass.

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating beam patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mdx top moments on in one view on the left and Mdy top moments in a second view on the right, as below



By doing this, it is possible to see how patches extend over the moment contours.

It is suggested that at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimization stage.

Design panels

NOTE Slab on beams panel design takes account of any beam or wall patches (by excluding the patch areas from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

- From the Design ribbon run Design Slabs in order to design or check all the panels in the model - by default newly created panels will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Slabs or Check Slabs. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

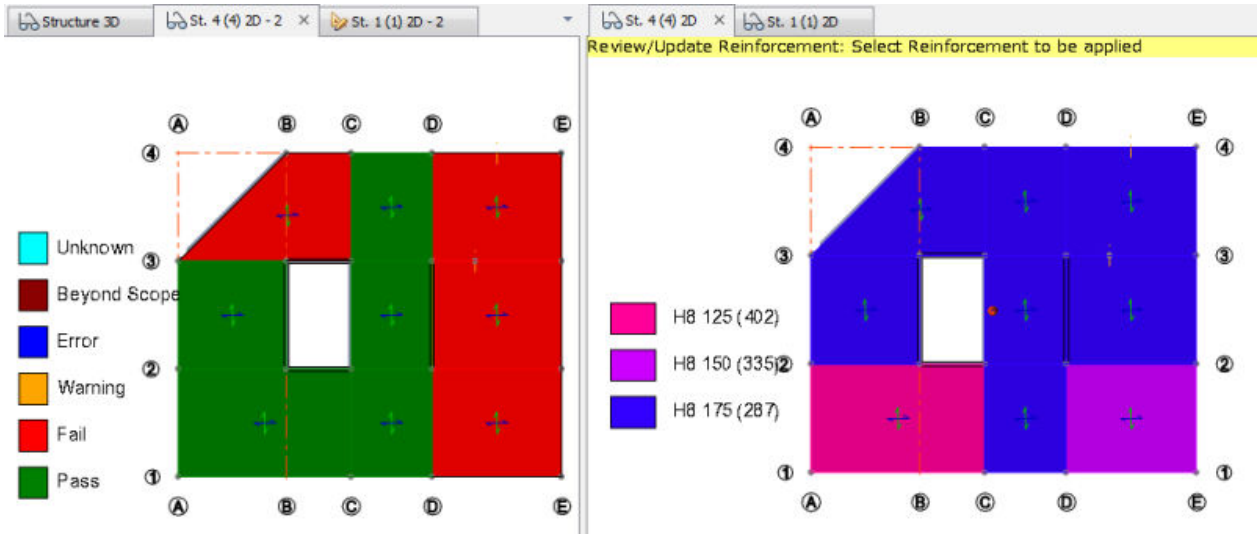
NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Slabs will re-design slabs (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Slabs will check the current reinforcement in slabs regardless of the current autodesign setting.

Review/optimize panel design

It is suggested that you use split Review Views to examine the results as indicated below.



The view on left shows Slab Design Status, (with slab patches turned off in Scene Content to assist clarity), the view on right shows Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to).

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.
- If the span-effective depth check is failing, review the panel properties to confirm that edge categories are set correctly, (and if the panel shape is irregular consider whether it is being suitably idealized as a rectangular panel). See: Slab on beam idealized panels (page 261)

Design beam and wall patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design any beam or wall patches that you may have defined.

To do this, either:

- From the Design ribbon run Design Patches in order to design or check all the patches in the model - by default newly created patches will all be in "auto-design" mode - so reinforcement is selected automatically.

  or

- In the 2D View of the floor which you want to design right-click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

- Design Patches will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

- Check Patches will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

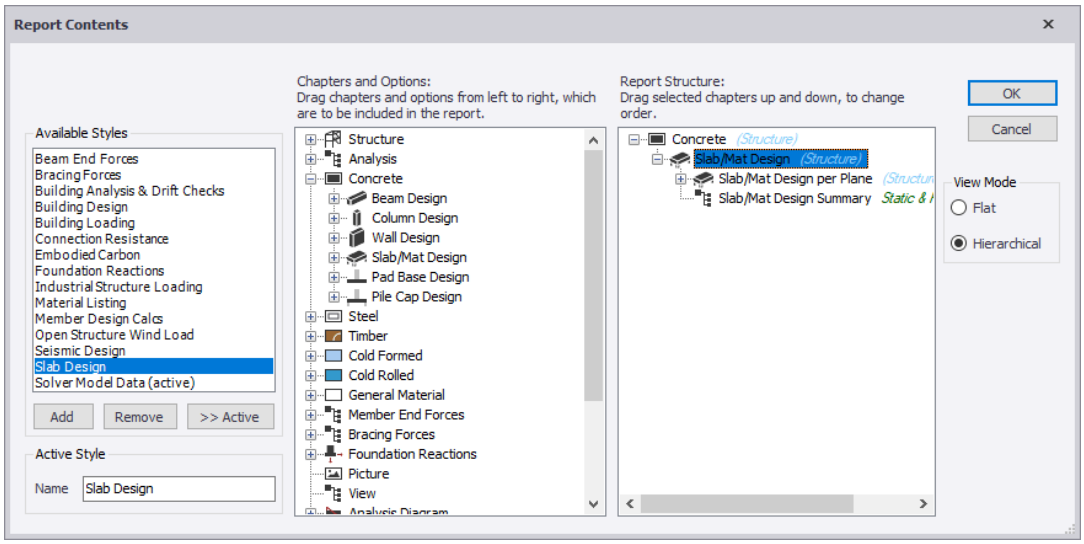
At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimized)
- Wall patches - can the width be adjusted (minimized)
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.

- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement.

Create reports

To add slab design and punching shear check calculations to any model report, in the Report Contents dialog box, select the report style, then open the **Concrete** chapter and drag the **Slab/Mat Design** chapter in the report structure.



This chapter includes two sub-chapters:

- **Slab/Mat Design Summary** - by default this chapter summarizes all the slab panel and patch designs, and punching shear checks in the model. It can be filtered as required.

Concrete

Slab/Mat Design

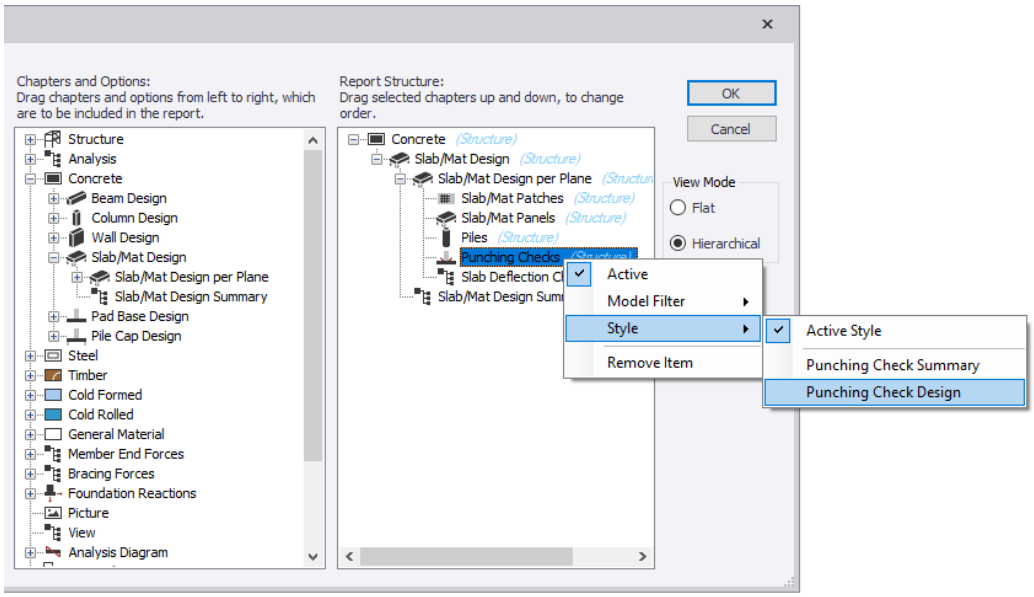
Slab/Mat Design Summary

Static & RSA

Level	Slab	Reference	Type	Thickness [mm]	Grade	Utilization	Status
St. 4 (4) : 12.000m	S 2	SI 46	Slab on beams	175.0	C32/40	0.783	Pass
St. 4 (4) : 12.000m	S 2	SI 30	Slab on beams	175.0	C32/40	0.771	Pass
St. 4 (4) : 12.000m	S 2	SI 50	Slab on beams	175.0	C32/40	0.686	Pass
St. 4 (4) : 12.000m	S 2	SI 6	Slab on beams	175.0	C32/40	0.613	Pass
St. 4 (4) : 12.000m	S 2	SI 2	Slab on beams	175.0	C32/40	0.602	Pass
St. 4 (4) : 12.000m	S 2	SI 22	Slab on beams	175.0	C32/40	0.533	Pass
St. 4 (4) : 12.000m	S 2	SI 18	Slab on beams	175.0	C32/40	0.463	Pass
St. 4 (4) : 12.000m	S 2	SI 42	Slab on beams	175.0	C32/40	0.360	Pass
St. 4 (4) : 12.000m	S 2	S 2-CB 4 V1 - 1	Beam Patch	175.0	C32/40	0.995	Pass
St. 4 (4) : 12.000m	S 2	S 2-CB 4 V5 - 1	Beam Patch	175.0	C32/40	0.981	Pass
St. 4 (4) : 12.000m	S 2	S 2-CB 4 V2 - 1	Beam Patch	175.0	C32/40	0.980	Pass
St. 4 (4) : 12.000m	S 2	S 2-CB 4 V4 - 3	Beam Patch	175.0	C32/40	0.972	Pass
St. 4 (4) : 12.000m	S 2	S 2-CB 4 H1 - 2	Beam Patch	175.0	C32/40	0.966	Pass
St. 4 (4) : 12.000m	S 2	S 2-CB 4 V8 - 3	Beam Patch	175.0	C32/40	0.966	Pass
St. 4 (4) : 12.000m	S 2	S 2-W3-DP26	Wall Patch	175.0	C32/40	0.963	Pass

- **Slab/Mat Design per Plane** - this chapter contains sub-chapters for the individual slab panel and patch designs, and punching shear checks. Each of these have a member report style assigned which controls the level of output produced.

  The style can be selected for each sub-chapter from the right click context menu as shown below.



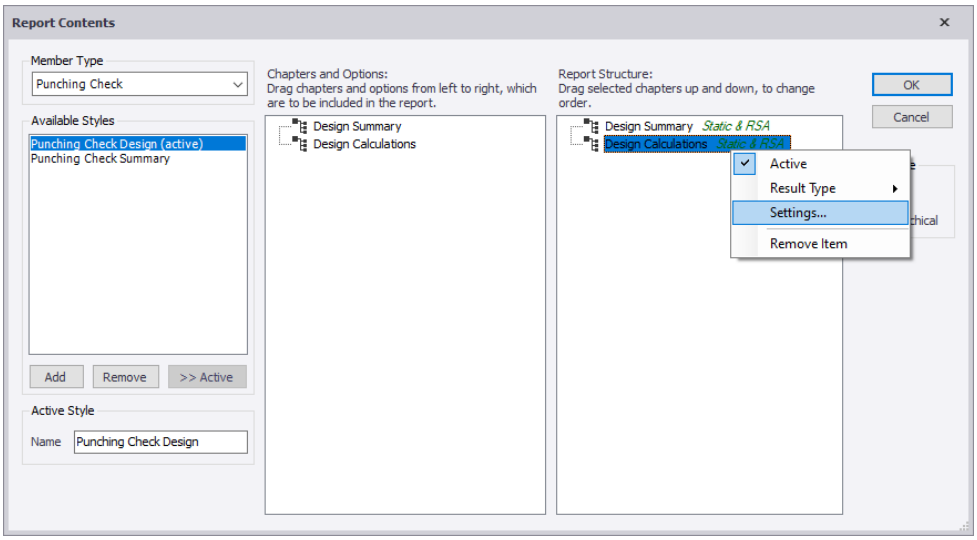
NOTE You can either choose a named style, or you can choose "Active Style" (in which case when you come to display the model report it will adopt whichever member report style is active at that time).

Each named style can be configured to include more or less detail, which can be done in a separate operation as follows:

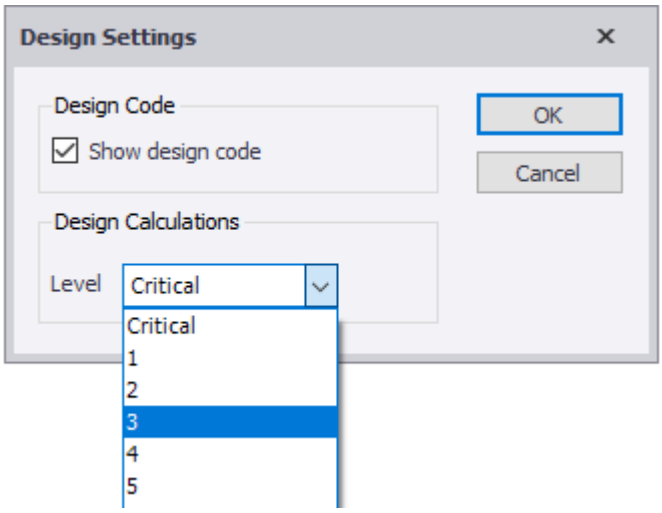
1. Close the above dialog box, and open the **Member Reports** dialog box.
2. Choose the member types in turn (e.g. Punching Check, Slab/Mat Panel, Slab/Mat Patch) and select an existing style, (or add an new one if required).
3. Drag chapters and options into the report structure as required.

NOTE This dialog box is also used to review which member report style has been nominated as the "Active Style". You can switch this to the currently selected style by clicking the **>> Active** button.

4. Configure the level of calculation detail, by right clicking on the Design Calculations chapter and selecting **Settings** from the context menu.



- Choose the level of detail that the calculations are to be expanded to in the report, (critical, with no levels expanded being the minimum), then click OK to close the dialog box.



5. Click OK to close the **Member Reports** dialog box.

To print the model report: on the ribbon, go to the **Report** tab, select the report style name and then click **Show Report**.

Concrete

Slab/Mat Design

Slab/Mat Design per Plane

St. 4 (4)

Slab/Mat Patches

S 2-CB 4 H1 - 1

Static

Overall Summary

Strip	Reinforcement Provided [mm²/m]			Area Required	Utilization
	From Slab	Added In Strip	Total Area Provided		
Strip X-Left	NA	NA			
Strip X-Centre	None	H8-125.0-T1	402	381	0.949
Strip X-Right	NA	NA			
Strip Y-Left	None	H10-125.0-T2	628	540	0.859
Strip Y-Centre	None	H10-125.0-T2	628	39	0.062
Strip Y-Right	None	H10-125.0-T2	628	153	0.244

Slab/Mat Panels

Slab: S 2, Panel: SI 2

Static

Reinforcement Design Summary

Reinforcement Layer	Reinforcement Provided	Area Provided [mm²/m]	Area Required [mm²/m]	Utilization
Bottom-X	H8-175.0-B2	287	170	0.593
Bottom-Y	H8-125.0-B1	402	242	0.602
Top-X	None	0	0	0.000
Top-Y	None	0	0	0.000

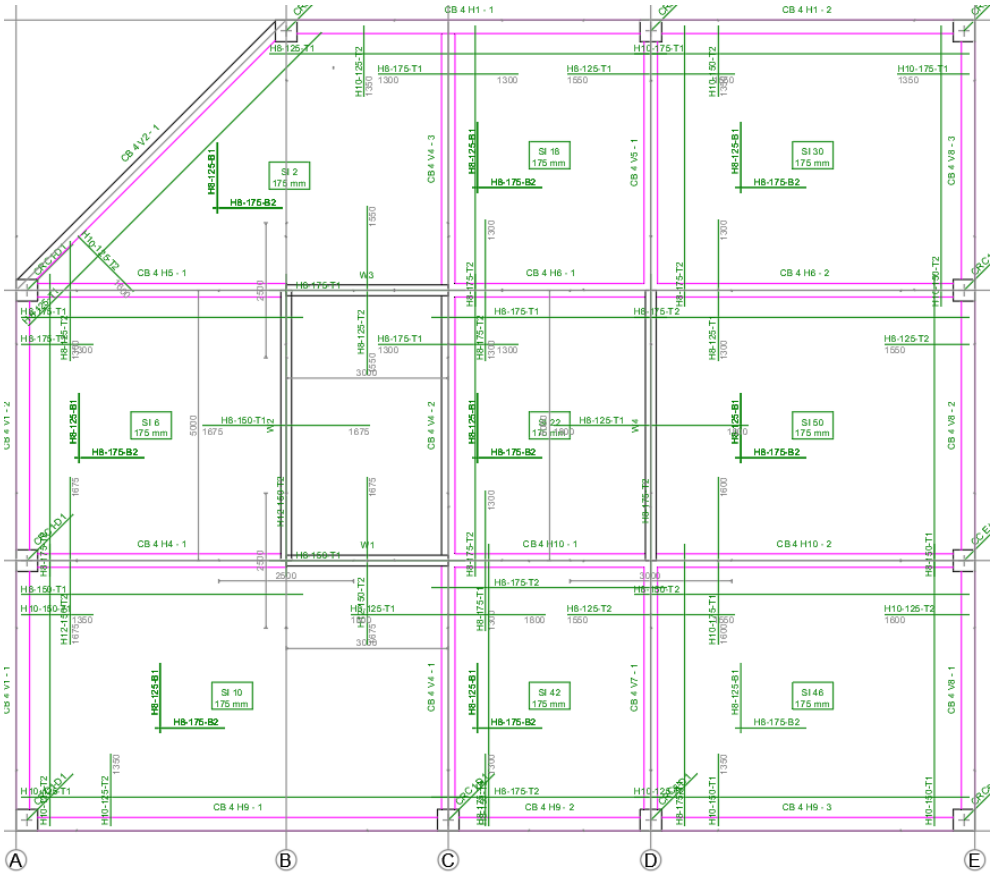
Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

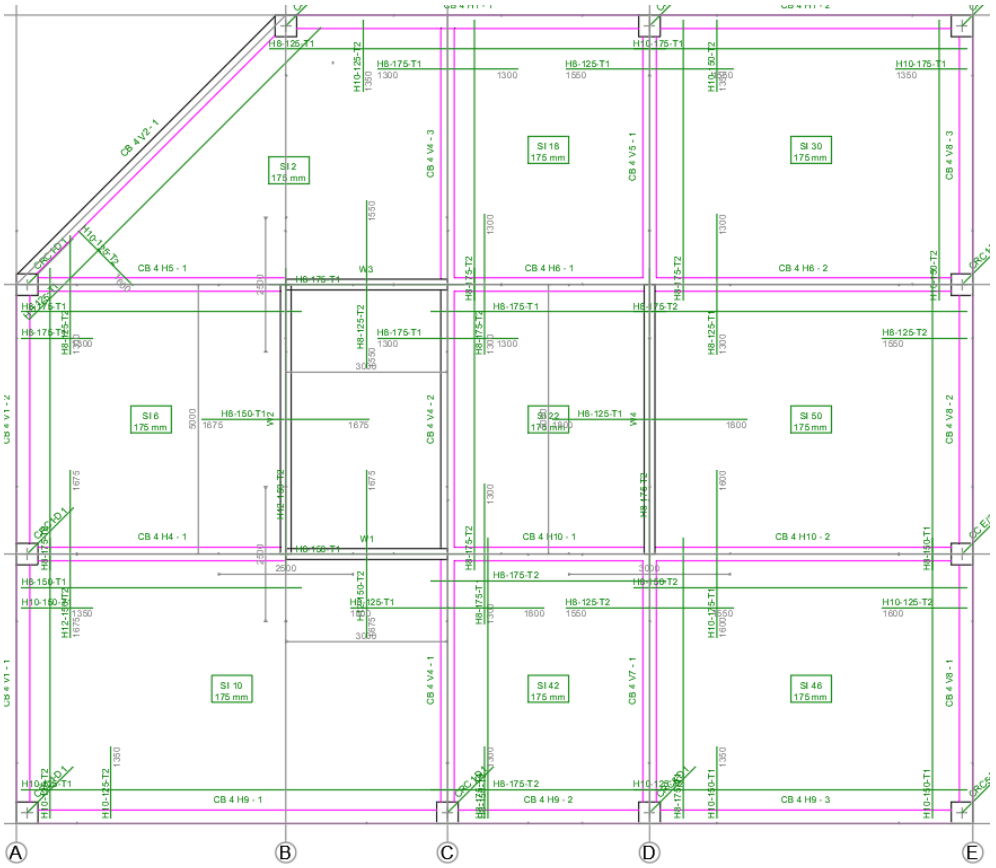
Two drawing variants are available for slabs and mats:

- **slab/mat layout** - created from the Draw tab of the ribbon by selecting **Slab/Mat Detailing**. This variant has pre-configured layer configurations available to limit the reinforcement displayed:

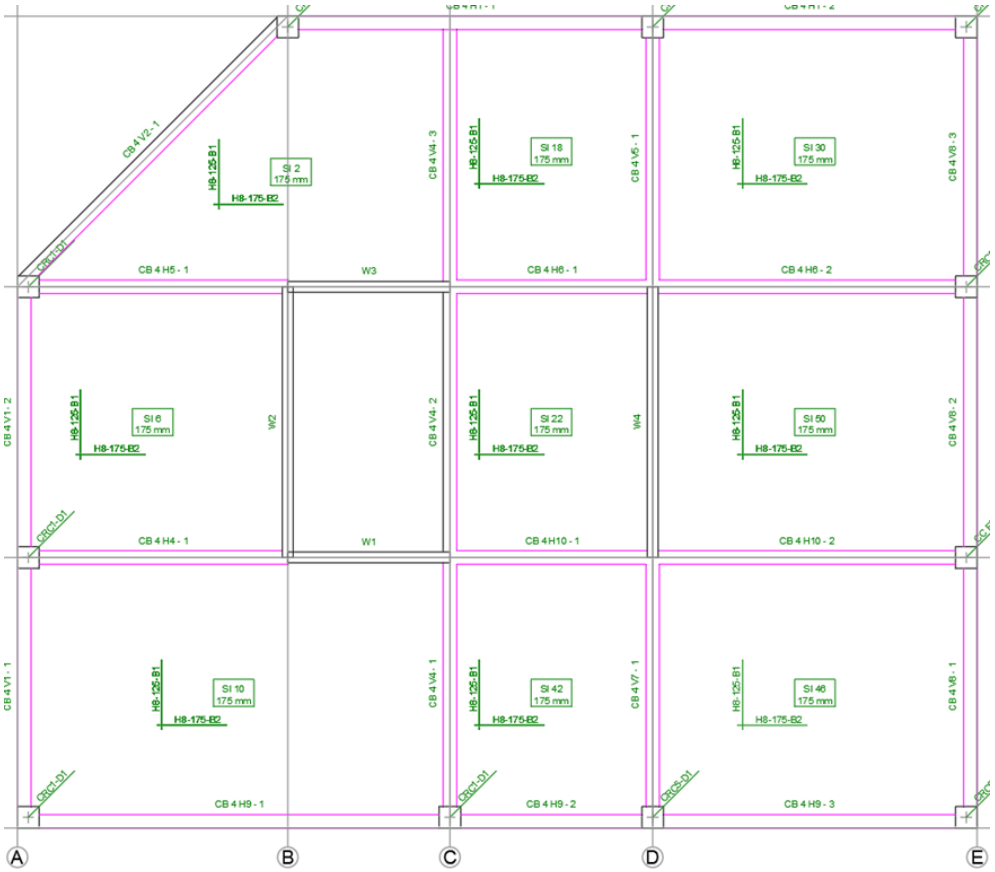
- **Slab/Mat Detail (All)** - this configuration displays top and bottom reinforcement, and also displays any punching perimeters



- **Slab/Mat Detail (Top Reinforcement)** - this configuration displays top reinforcement only



- **Slab/Mat Detail (Bottom Reinforcement)** - this configuration displays bottom reinforcement only

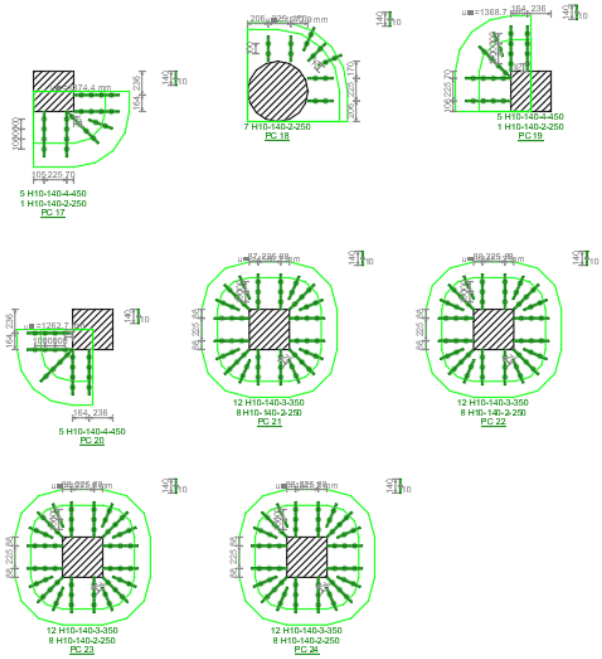


- **Slab/Mat Detail (Punching)** - there were no punching checks specified in this example, but if there were they could be displayed on the layout using this configuration. If any stud rail reinforcement had been

designed this would also be included to the side of the layout, as shown below.

Slab Top and Bottom Reinforcement			
Type	Unit Mass [kg/m]	Total Length [m]	Total Mass [kg]
H12	0.888	4732.220	4202.21
		Total	4202.21
		Total + 10.00 % slab detailing allowance	4622.43
Type	Unit Mass [kg/m²]	Total Area [m²]	Total Mass [kg]
A98	1.54	36.0	55.39
A142	2.23	6.0	13.32
		Total	68.71
		Total + 10.00 % slab detailing allowance	75.58
		Grand Total	4698.01

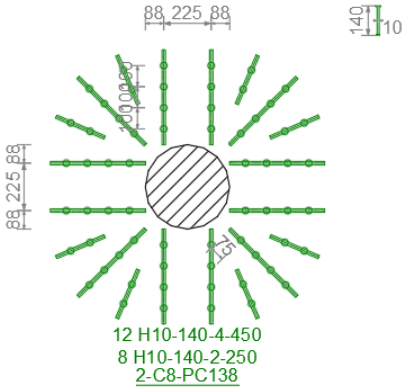
Stud Rails						
Stud Rail Ref.	Stud Diameter [mm]	Stud Height [mm]	Studs per Rail	Stud Spacing [mm]	Rail Length [mm]	Total No. of Rails
H10-140-3-350	10.0	140.0	3	100.0	350.0	48
H10-140-2-250	10.0	140.0	2	100.0	250.0	41
H10-140-4-450	10.0	140.0	4	100.0	450.0	15



- **punching check detail** - if punching checks had been specified in this example, this variant could be used to display the details of any stud rail

reinforcement by right clicking on an individual punching check item in a scene view and selecting **Generate Drawing** from the context menu.

Stud Rails						
Stud Rail Ref.	Stud Diameter [mm]	Stud Height [mm]	Studs per Rail	Stud Spacing [mm]	Rail Length [mm]	Total No. of Rails
H10-140-4-450	10.0	140.0	4	100.0	450.0	12
H10-140-2-250	10.0	140.0	2	100.0	250.0	8



3.10 Concrete slab design fundamentals

Slab panels

Following analysis, the area of steel required for a given panel is determined from the analysis results.

For each panel the basic design check is performed in all areas where no patches exist:

- Is $A_{s,prov} > A_{s,reqd}$?

Slab patches

Patches are used during the design of concrete slabs as a way of managing the physical and design data. Each patch defines a rectangular area of slab within which FE analysis results are collected, enabling Tekla Structural Designer to perform the patch design process. Design moments are calculated along result strips, embedded within each patch. Depending on the patch type, a patch can contain up to 6 result strips, catering for up to 3 strips of reinforcement in each of two perpendicular directions.

Punching

Punching checks can be requested at any

- column stack/slab (or foundation mat) intersection
- wall stack/slab (or foundation mat) intersection
- applied slab point load

For further details, see: Punching shear check fundamentals (page 264)

Concrete type

While you can apply both normal and lightweight (LW) concrete in the slab properties, slab design using lightweight concrete is only available for Eurocodes.

When using other regional codes slabs can only be designed using normal weight concrete.

LW density classes and grades

6 Density classes (1.0, 1.2 ..... 2.0) are available and 15 default grades are provided; 5 in each of the density classes: 1.6, 1.8 and 2.0.

- For example the grade name “LWAC30/37-DC1.8” denotes; LWAC = Lightweight aggregate concrete; 30/37 = Strength class; DC1.8 = density class.
- Custom LW grades can be added for which note that new LW-specific property η_1 must be specified.

NOTE LW grades can be reviewed, edited and applied via Review View > Show/Alter State Material Grade Attribute.

How the load decomposition method affects slab design

Only slabs that have been specified with two-way load decomposition are designed.

One-way load decomposition in Tekla Structural Designer is a simple procedure that does not determine slab design forces. When a slab's decomposition is set as one-way it is assumed that it is some form of precast slab (presumably designed by safe load tables).

- A flat slab panel always uses two-way load decomposition.
- A slab on beams panel can either be specified to use one-way or two-way decomposition - however if it is specified as one-way it cannot then be designed in Tekla Structural Designer.

It should be noted that any in-situ slab is capable of two-way decomposition:

- When a slab is set as two-way it will only effectively span in 2 directions if its proportions and support conditions mean that there will be a two-way effect.
- For example - If a slab that has a span of 6 units in one direction and 50 units in the other is set to two-way decomposition, then although it is two-way the FE analysis will still inherently take the load one-way.

Two-way spanning slab panel design moments

When a slab panel is specified with two-way decomposition, a general FE based approach is used to determine the design moments. If you have elected to mesh 2-way slabs in the analysis, the 3D Analysis results are also considered:

- The worst design moment (per unit width) is found in each direction of the slab - if the design moment is zero in one of the directions then the analysis has shown that the slab is effectively spanning one-way and the supplied reinforcement in this secondary direction will be selected to suit the minimum requirements of secondary reinforcement.
- Note that this FE based approach inherently caters for point loads, line loads, openings, etc and for the possibility of variable adjacent span lengths in a continuous "1-way" slab (and of course it can still be applied to the simple case of a "1-way" slab with a uniform UDL applied and uniform span lengths).

Reinforcement design settings

Prior to running slab design you should review the design settings by selecting **Design > Settings > Concrete > Slab**.

These settings are used to control:

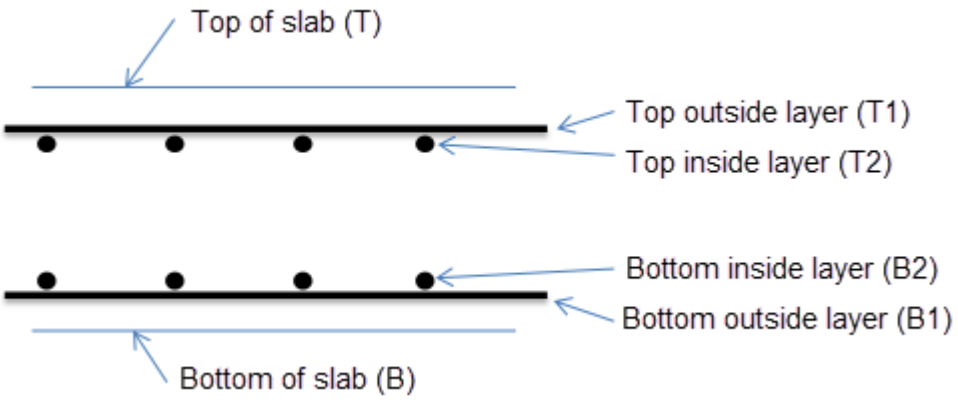
- Reinforcement layout, e.g.
  - Minimum and maximum user bar sizes
  - Minimum and maximum user spacing values
  - Maximum bar lengths
  - Auto selection of outer bars
  - Use of mesh
- Patches
  - Default patch size
  - Use of mesh

- General parameters (depending on the regional code being designed to)

For further details, see: Slab design settings

Slab panel reinforcement

In a slab you have two surfaces: Top (T) and Bottom (B).



In each surface you have two layers of steel in orthogonal directions - X direction steel and Y direction steel. Layer 1 is the outside layer - the one closest to the surface. Layer 2 is the inside layer. Which direction is the outer layer is controlled by this slab panel setting: Outside layer in X direction.

Any of the four layers can be set to "none" if required.

Alignment of slab panel reinforcement

X direction reinforcement is aligned to the panel span, which is determined by the slab's **rotation angle** property. Y direction reinforcement is always perpendicular to the X direction reinforcement.

For further details, see: Rotation angle for slab items and panels (page 263)

Slab patch reinforcement

Tekla Structural Designer contains the following patch types:

- **Column patch**: can be placed at column stack heads
- **Beam patch**: can be placed along beams
- **Wall patch**: can be placed along walls
- **Panel patch**: can be placed at a specified position within the panel boundary

- not restricted to a centralized position or to existing within one panel
- can be positioned under loads

Patches can be either on the top or the bottom of the slab and may or may not have reinforcement defined in them. If you have not defined any reinforcement, Tekla Structural Designer uses the background reinforcement.

If you have defined reinforcement, then it will either be used on its own, or if you select the "Combine with Panel Reinforcement" option, the sum of the background + patch reinforcement will be used. Note that this option is only selectable when the "Align to Panel Reinforcement" option is also selected since combining in this way would only be valid provided the reinforcement is reasonably aligned. Choosing the reinforcement to be combined also forces the "Cover as Panel" option to be selected as the program assumes the reinforcement to be in a single layer. If the reinforcement is not combined you can specify the cover to the patch reinforcement by turning off the "Cover as Panel" option.

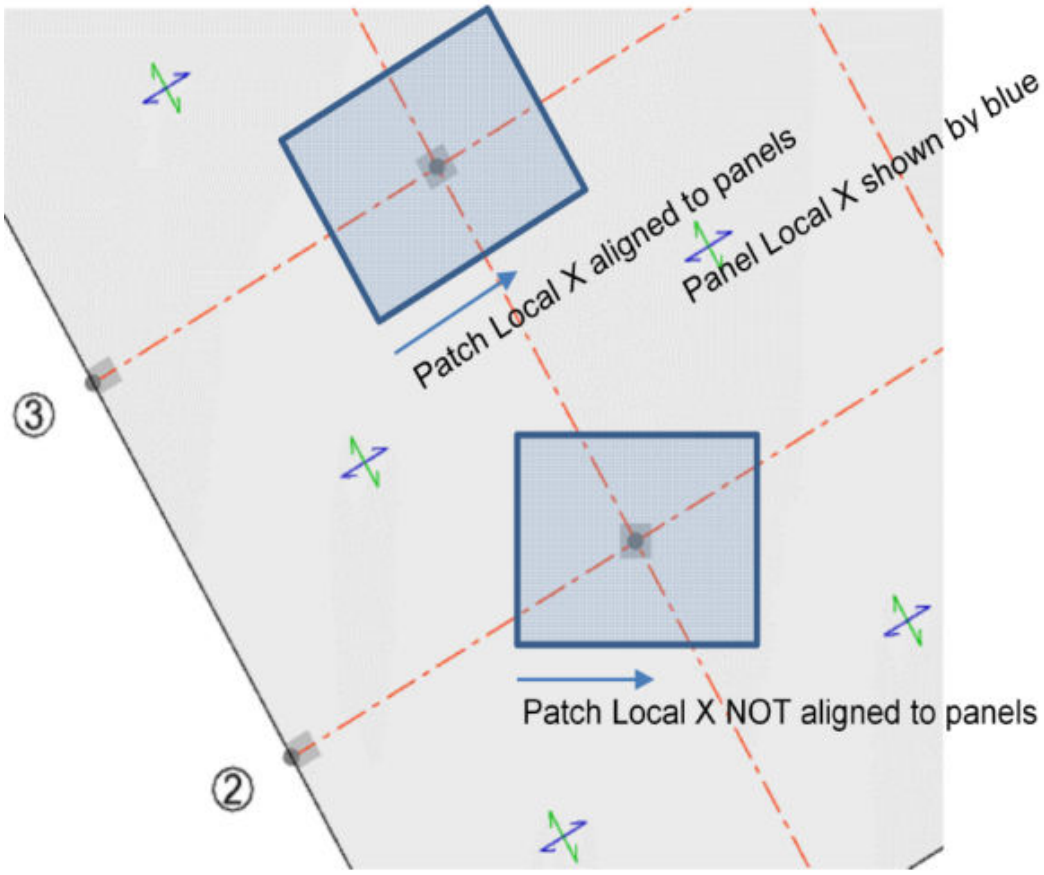
Note that patches may overlap on the plan view, and there is no restriction on this, even patches relating to the same layer of reinforcement are allowed to overlap. This situation is handled conservatively during design by simply ignoring the overlap.

Each patch manages reinforcement and the reinforcement design using a number of slab design strips. Some key points to bear in mind considering patch design are:

- A patch only manages data in the top or the bottom layers of a slab, not both.
- There can be up to 3 design strips in each direction of a patch
- There is no requirement to have design strips in both directions - there can be one design strip in one direction and none in the other.
- Within any strip there might be patch reinforcement to consider but note that:
  - The underlying panel reinforcement can be none
  - The added patch reinforcement can be none.
- If there is patch reinforcement to consider this can be considered instead of, or in addition to, relevant slab reinforcement

Slab patch strip design

For the strip designs within each patch it is necessary to establish which bar layer is to be designed and work out if and how the patch reinforcement combines with the panel reinforcement.



As shown above there are 2 distinct options:

1. Patch aligned to panel
2. Patch NOT aligned to panel:

In both cases the reinforcement that is determined for use in the design checks is some combination of the panel and patch reinforcement. Expanding upon this the cases considered are:

Patch aligned to panel:

1. Patch reinforcement type = NONE then the panel reinforcement is used for all layers
2. Combine with Panel Reinforcement = No Then the patch reinforcement is used in the patch surface and the panel reinforcement is used in the opposite surface.
3. Combine with Panel Reinforcement = YES Then the patch reinforcement is combined with panel reinforcement and used in the patch surface and the panel reinforcement is used in the opposite surface.

Patch NOT aligned to panel:

Patch reinforcement is used in the surface to which the patch applies. Reinforcement in the other surface is taken from the panel using the most aligned possibility.

Patches to both surfaces

As stated above, patch reinforcement is only ever used in the surface to which a patch applies, reinforcement in the other surface is taken from the panel.

In rare situations you may have separate patches on both surfaces; in which case you would want the patch reinforcement from the top patch to be considered on the top surface and patch reinforcement from the bottom patch to be considered on the bottom surface.

In this specific situation, for both patches the other surface doesn't necessarily need to be designed to only consider reinforcement in the panel; you can avoid this by selecting the patch property **Consider patch surface moments only**.

Span effective depth checks for irregular shaped panels

Only slabs that have been specified with two-way load decomposition are designed.

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

For further details, see: Slab on beam idealized panels (page 261)

Slab on beam idealized panels

Only slabs that have been specified with two-way load decomposition are designed.

In order to perform span - effective depth checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

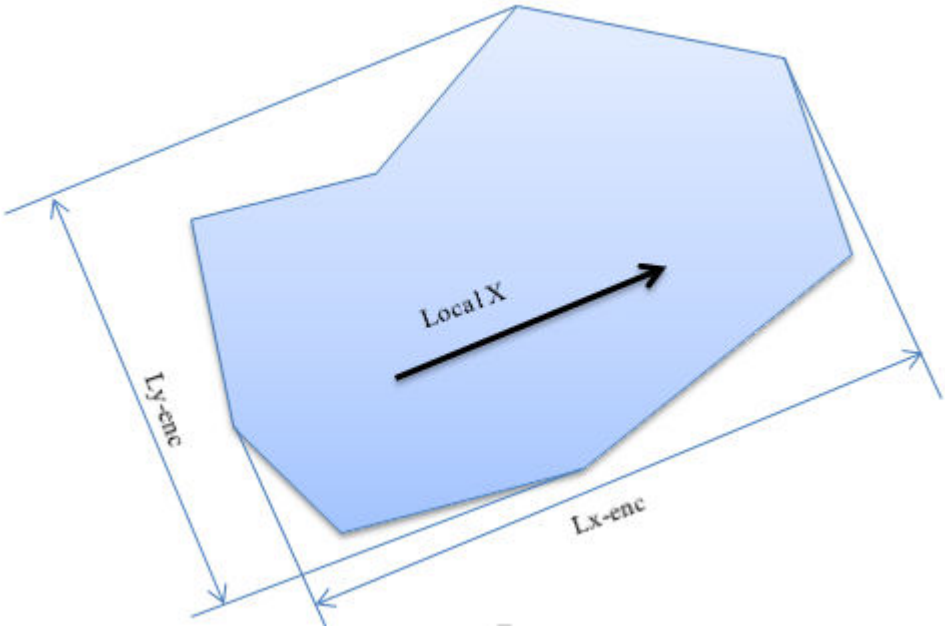
To do this, the enclosing lengths of the panel in X and Y are first determined, (local X being defined by the panel rotation angle):

Lx-enc = maximum overall length of the panel measured parallel to local X

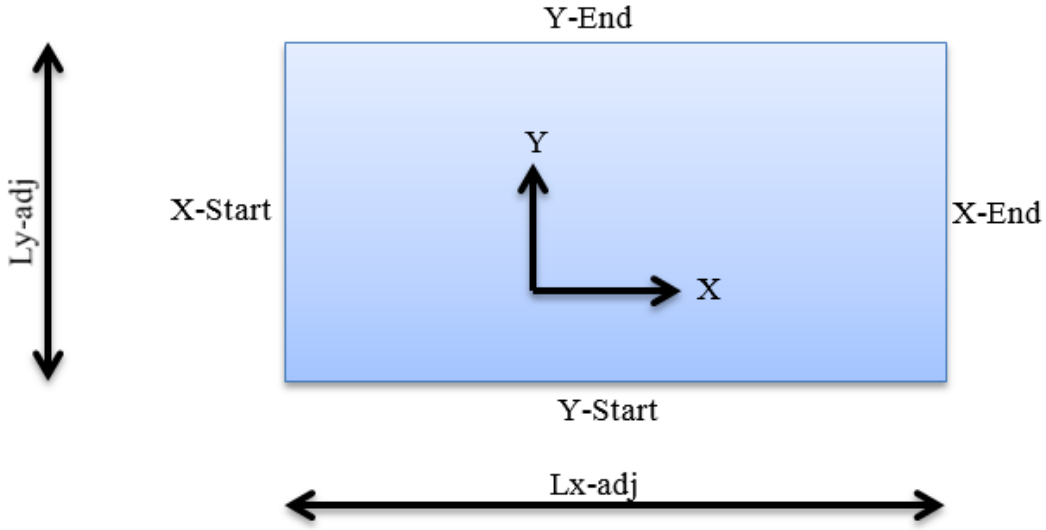
Ly-enc = maximum overall length of the panel measured perpendicular to local X

A user specified adjustment ratio is then applied to these lengths to determine the adjusted lengths. Conservatively the adjustment ratio defaults to 1.0 in both directions.

In situations where the panel does not have 4 sides, (such as the one shown below), some engineering judgement might be required when deciding on appropriate values of the adjustment ratios in each direction.



The resulting idealized panel with dimensions in X and Y is illustrated below:.



Edge Category

For the span-effective depth check, the edge categories in each direction have to be manually assigned to the idealized slab panel. The three edge categories being:

- Unsupported
- Continuous Support

- Dis-continuous Support (default)

Rotation angle for slab items and panels

The rotation angle in the slab item/panel properties is used to control the span direction. For reinforced slabs the X direction reinforcement is aligned to the span.

NOTE When SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow.

Slab items and panels defined by a level

When the rotation angle/X direction = 0

- Span direction aligns with global X

When the rotation angle/X direction = 90

- Span direction aligns with global Y

Panels defined by a frame

When the rotation angle = 0

- Span direction is horizontal

When the rotation angle = 90

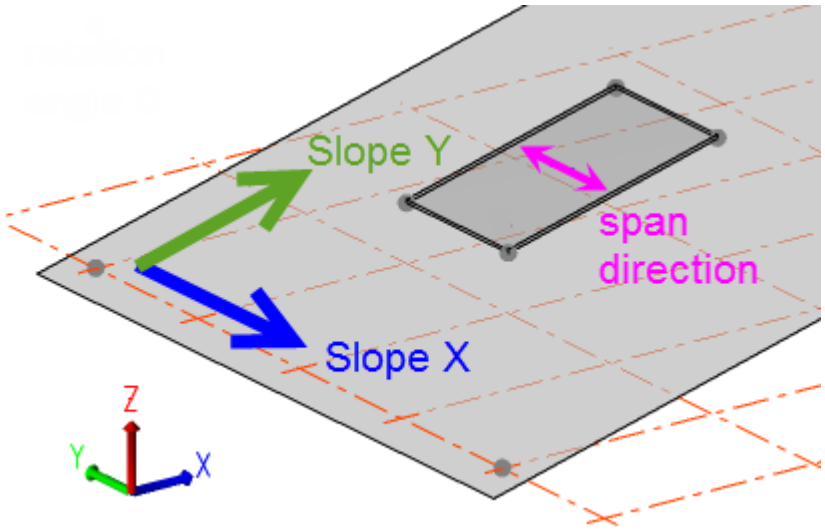
- Span direction is vertical

Slab items and panels defined by a sloped plane

NOTE In a sloped plane positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

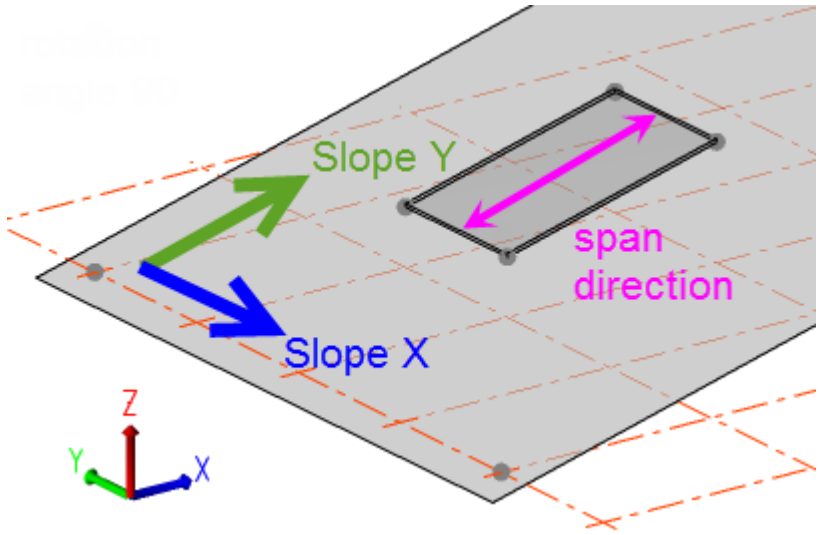
When the rotation angle/X direction = 0

- Span direction aligns with X direction of the slope (i.e. perpendicular to the slope, as shown below)



When the rotation angle/X direction = 90

- Span direction aligns with Y direction of slope (i.e. parallel to the slope, as shown below)



3.11 Punching shear check fundamentals

Before you check punching shear in Tekla Structural Designer you should familiarize yourself with some concepts that apply to the punching checks generally and others that apply when the checks are being used to design shear rail reinforcement.

Overview

When punching check items are designed/checked an applied load on the slab/mat foundation is calculated accounting for the difference in column/wall axial load and bending. The slab shear stresses are then checked, accounting for the reinforcement present (background and column/general patches).

All columns and walls that are not perpendicular to slabs are treated as if they are for the punching areas developed. The relevant load is resolved to act perpendicular to the slab for punching checks. This is conservative as the punching area/perimeter will be smaller than that for the angled column or wall.

Column head drops and the presence of openings within a certain distance of the punching shear boundary are taken into account in the punching shear calculations.

Where punching checks are closely spaced and the perimeters overlap the checks are beyond scope.

If working to the Eurocode or ACI head code, the punching reinforcement can optionally be designed at certain locations as an orthogonal or circular arrangement of stud rails. For details of the locations, see: Applicability (page 265).

NOTE View punching checks applied to walls with caution. Their applicability to long walls is particularly questionable, as the check does not consider the potential for stress concentrations at the ends of the wall.

Applicability

Punching shear can be **checked** at the following locations:

- **In flat slabs**
  - Concrete columns supporting a flat slab with and without drops
  - Concrete columns supported by a flat slab
  - Concrete columns through a flat slab with or without drops
  - Concrete walls supporting a flat slab
  - Concrete walls supported by a flat slab
  - Concrete walls through a flat slab
  - Columns in other materials supported by a flat slab
  - Point loads at either face of a flat slab
- **In foundation mats**

    - Concrete columns supported by a foundation mat
    - Concrete walls supported by a foundation mat
    - Columns in other materials supported by a foundation mat
    - Point loads on a foundation mat
    - individual piles supporting a foundation mat (but only if designing to **Eurocodes**, or **ACI 318**)

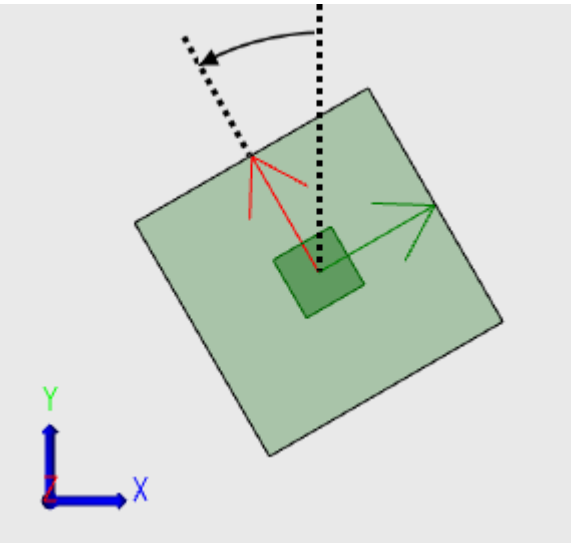
If designing to **Eurocodes**, or **ACI 318**, punching shear **reinforcement can be designed** at the following locations:

- **In flat slabs**
    - Concrete columns supporting a flat slab without drops
    - Concrete columns supported by a flat slab
    - Concrete columns through a flat slab without drops
    - Columns in other materials supported by a flat slab
    - Point loads at either face of a flat slab
- **In foundation mats**
    - Concrete columns supported by a foundation mat
    - Columns in other materials supported by a foundation mat
    - Point loads on a foundation mat

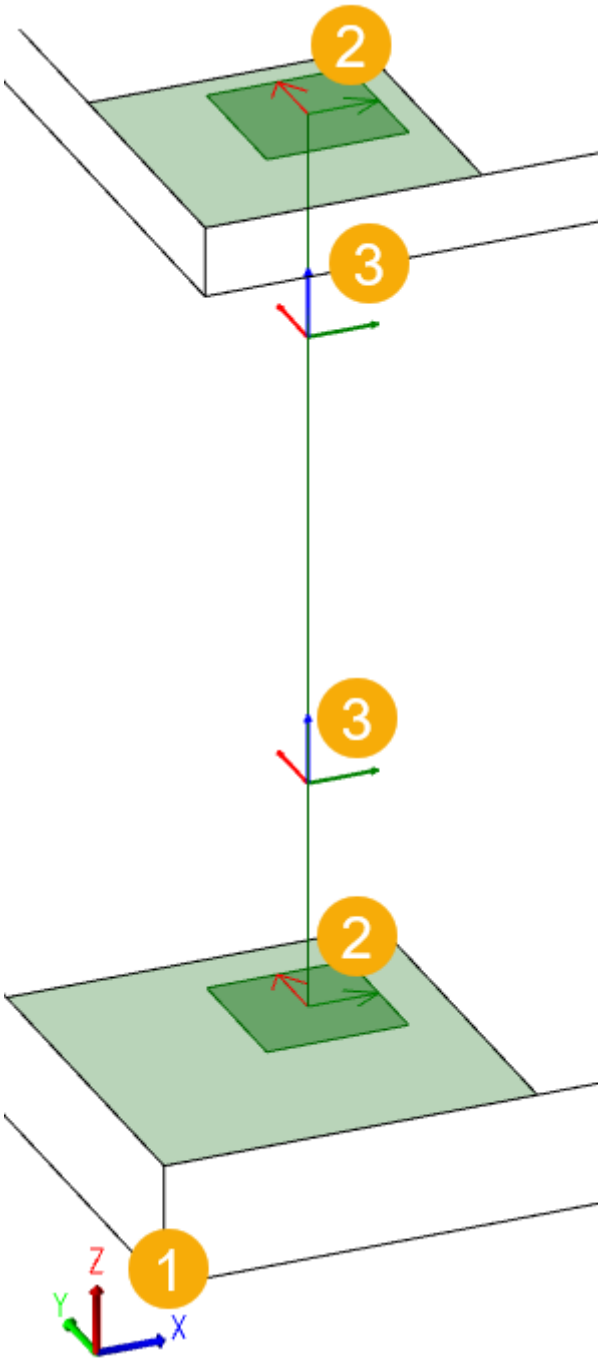
Punching shear check axis system

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.


Y
X

- **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 punching shear check axis system and the column 1D member local coordinate axis system are locked together, so if the column is rotated, the punching check axes also rotate.
3 Column 1D member local coordinate system	

Punching shear perimeters

Punching shear perimeters are peripheral lines around the section of the column/wall/point load being checked for punching shear.

- Can have different shapes based on the check location, column/wall/point load section and/or type of reinforcement if applicable
- Can be interrupted by openings or slab edges
- Length can be adjusted by the user through the punching check properties grid

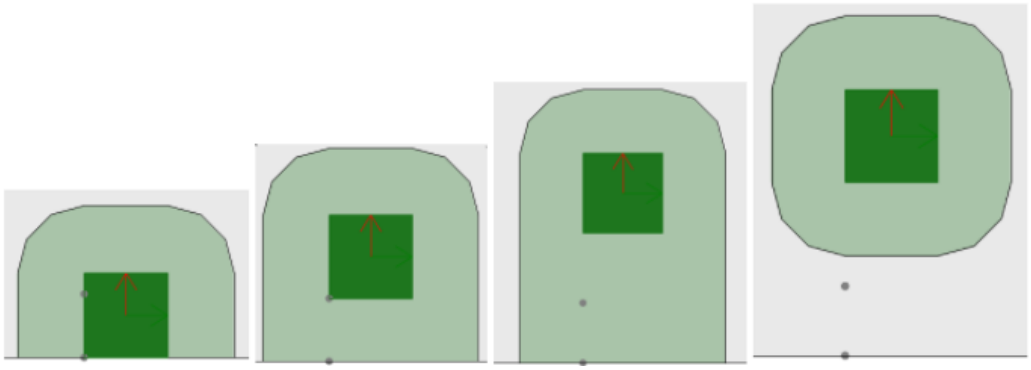
Every punching shear perimeter is positioned at a distance from column face which is a multiple of the effective depth found at the first punching shear perimeter (control/critical). This results in evenly spaced perimeters either inside and outside slab patches where reinforcement can vary.

The types of perimeter vary according to the design code being worked to. For more information, see:

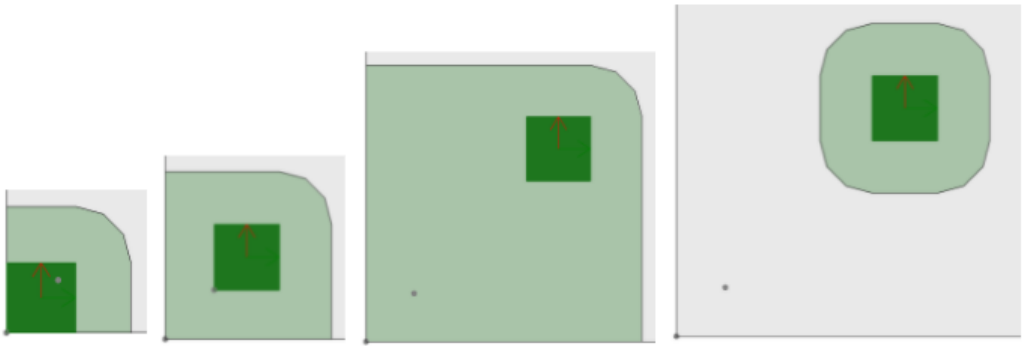
- (ACI 318)
- (Eurocode)

The effect of slab edges on punching shear perimeters

- **Edges**: perimeters have three sides or less

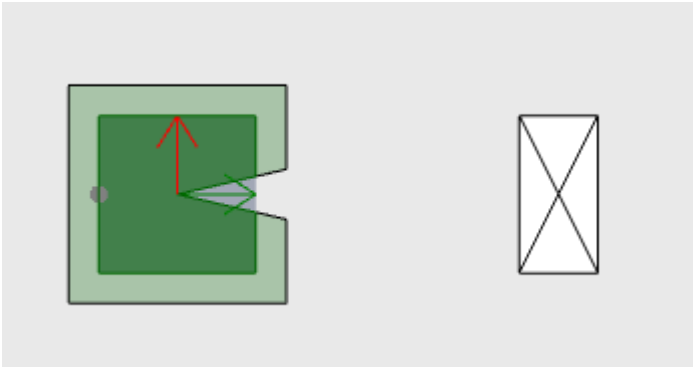


- **Corners**: have two sides or less



- Perimeters use the slab edges as a boundary until the length of the full internal perimeter is found to be smaller than the edge/corner perimeter

The effect of openings on punching shear perimeters



A dead-zone inside two tangents drawn from column center to opening outline is created where perimeters are considered ineffective (total perimeter length gets reduced) if an opening edge is found at less than a specified distance from the loaded area. The distance varies according to the design code being worked to. For more information, see:

- (ACI 318)
- (Eurocode)

User perimeter reductions

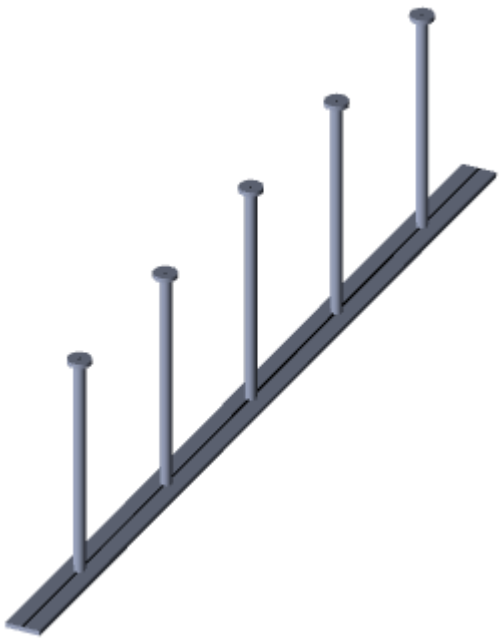
If you want to consider the effect of openings, but do not want to place them in the slab, this can be done by specifying a **user reduction** in the punching check **Properties** window. When applied, the length of the perimeter is reduced by the specified amount.

User perimeter reductions can also be applied in order to adjust the length of perimeters of irregular section shapes.

NOTE if designing to Eurocodes, in earlier Tekla Structural Designer versions you were advised to specify user perimeter reductions in order to obtain the reduced loaded perimeter for edge and corner cases as suggested by EN1992-1-1 clause 6.4.5(3). From Tekla Structural Designer version 2023 onwards this is no longer required as column punching shear checks for slabs, pad bases and pile caps have been revised to automatically take account of the upper limit on column loaded perimeter length for the cases of edge and corner columns, as prescribed in Clause 6.4.5(3). The new limit is imposed before any further user reductions are considered.

Reinforcement type and properties

If designing slabs or foundation mats to Eurocodes, or ACI 318, punching shear reinforcement is allowed in the form of stud rails. No punching reinforcement is allowed if designing isolated foundations.



Stud reinforcement uses the same material database as ordinary concrete reinforcement. New steel grades and bar/stud sizes can be defined by clicking **Home > Materials > Reinforcement**

NOTE **Type** and **Rib Type** are not used in design but are relevant to filter the classes and sizes

Reinforcement design settings

If you are using the Eurocode or ACI regional code, and have specified stud rail reinforcement when creating the punching checks, you should review the reinforcement design settings by selecting **Design > Settings > Concrete > Punching Shear > Reinforcement Layout**.

These settings are used to control:

- Minimum and maximum bar sizes
- Minimum and maximum spacing values
- Auto-design method, and the “Starting number of studs per face” in auto-design

These settings only apply to slab punching checks and not the punching checks applied to isolated foundations.

For further details, see:Punching shear design settings

Reinforcement auto-design

When auto-design is selected you are can choose to select reinforcement from **Minima** or from the **Current** reinforcement layout.

Depending on how you have configured your Punching shear design settings auto-design will attempt to either:

- **Minimize Number** of studs on the first reinforcement line used in the design (equivalent to number of design rails), or
- **Minimize Size** of the studs used in design

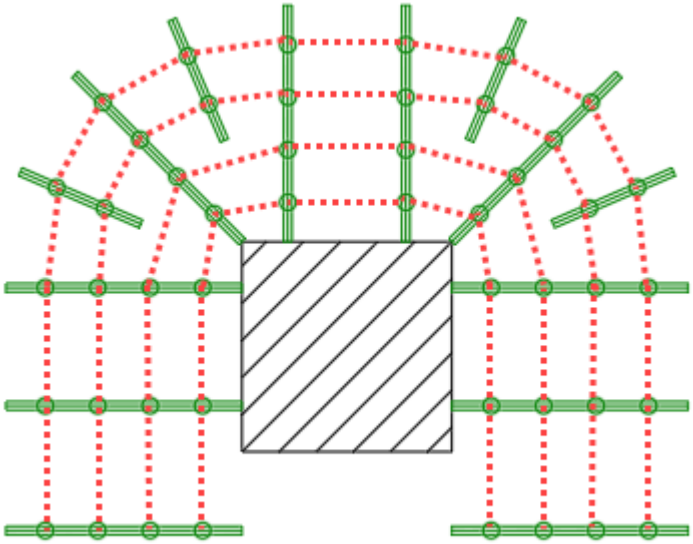
When starting auto-design from minima:

- Number of stud rails per column face starts at 1;
- Number of studs per rail starts at 2;
- Number of rails or stud size - depending on the design option - increases along a peripheral line until a pass is obtained;
- Number of studs per rail increases until reinforcement is no longer required.

Reinforcement layout

NOTE Punching reinforcement can only be specified when designing to Eurocodes, or ACI 318, and it only applies to slab punching checks - i.e., no punching reinforcement is allowed in isolated foundations.

- **Reinforcement arrangements** - the following arrangements are available:
  - **Circular arrangement** (predominantly used in EC design)
  - **Orthogonal arrangement** (predominantly used in ACI design)
- **Reinforcement lines** are:
  - peripheral lines around the loaded area passing through stud positions of the same index along all "design rails"
  - reinforcement lines are not shear perimeters

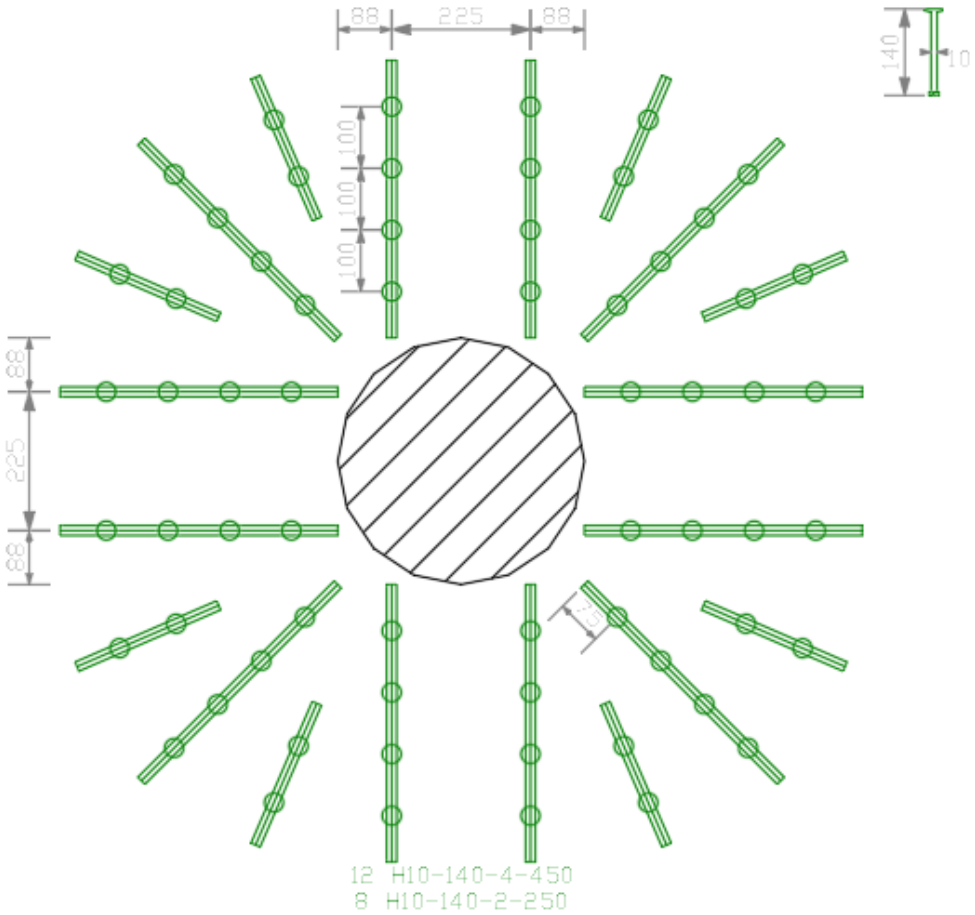


- **Rail positions** - Within the circular and orthogonal arrangements the rail positions are:
  - **Face rails**
    - Positioned between column bounding rectangle projected boundaries and aligned with the check local axis
    - Arranged symmetrically in relation to the section bounding rectangle centerline, 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

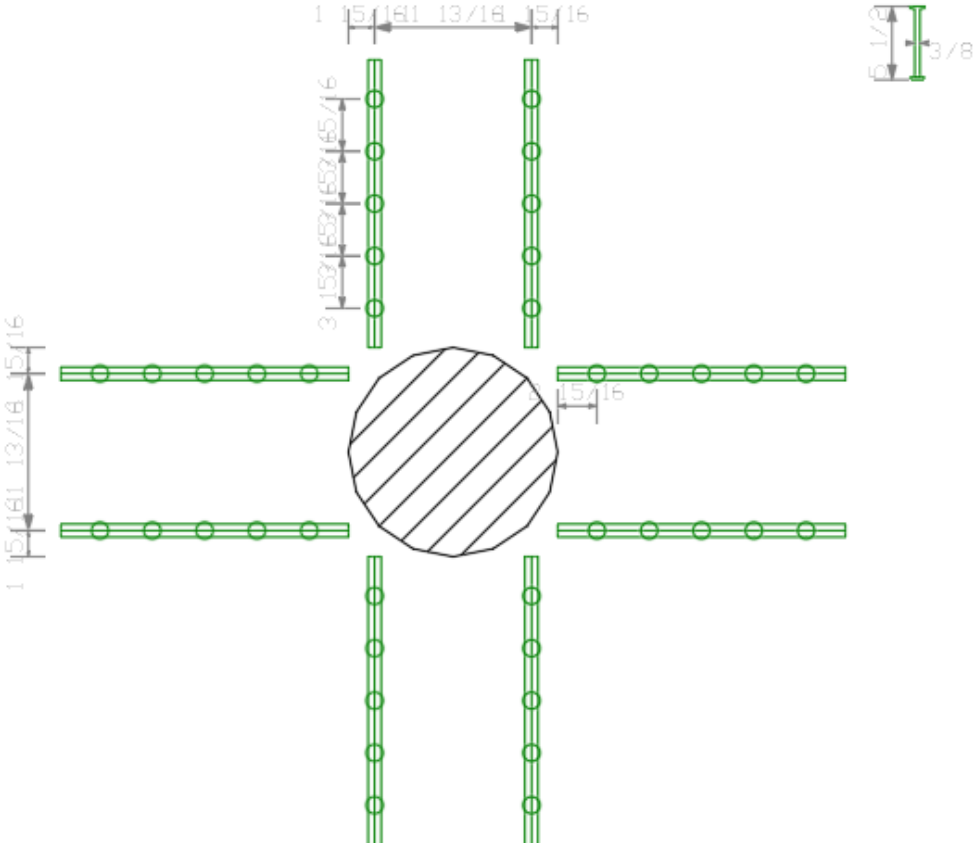
- **Diagonal rails**
  - Positioned at corners and at an angle with the check local axis
  - Placed at equal angles from each other and the two orthogonal directions starting at 45 degrees for a single corner rail
  - Angles are measured with center at the origin of the two closest face rails
  - The minimum number of rails is 0
  - The maximum number of rails is 10

- **Rail positions within arrangements**
  - **Circular arrangement**

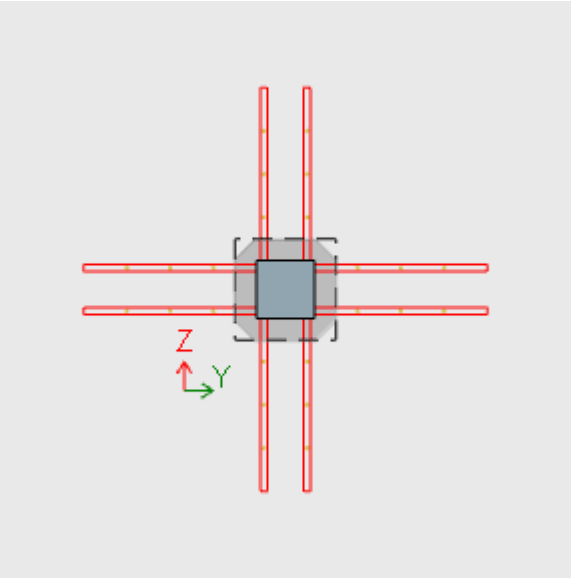


  - Face rails and corner rails
  - Can have edge rails as well
  - Provided uniformly around the shear perimeter
  - Shorter rails might be used to fulfill spacing requirements

- **Orthogonal arrangement**

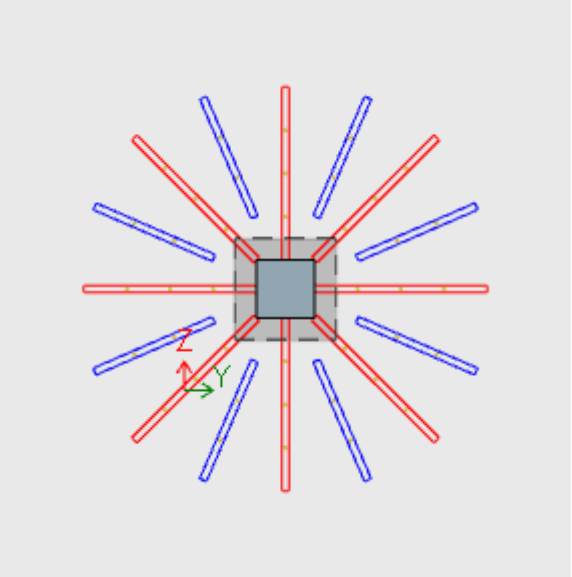


  - Face rails but no corner rails
  - Can have edge rails as well
  - Non-uniformly around the shear perimeter
- **Number of studs per rail**
  - Minimum number of studs = 2
  - Maximum number of studs = 20
  - The actual number of studs per rail depends on the design, user editing and punching check characteristics (position, presence of openings, spacing of rails)
- **Rail types**
  - **Design rails**
    - Identified in red in user interface



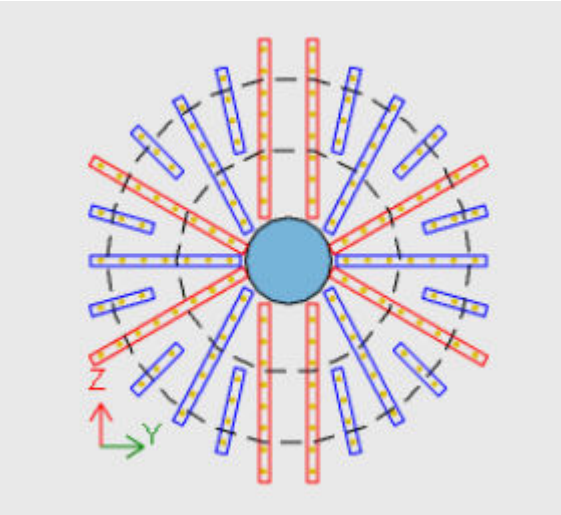
- Full length rails
- Rail properties are user editable
- Considered in the design

- **Detailing rails**
  - Identified in blue in the user interface

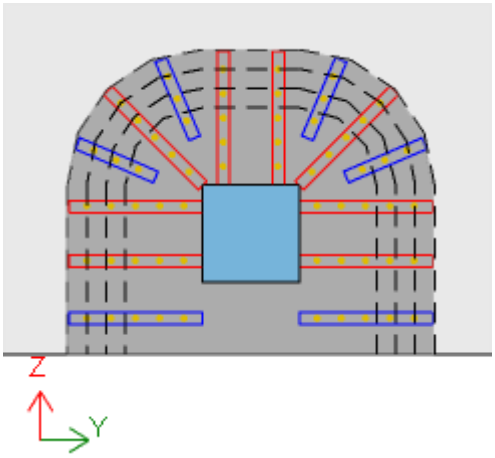


- Used to fulfill spacing requirements (circular arrangements only)
- Used to fill distances to edges
- Shorter rails interrupted by opening dead-zones

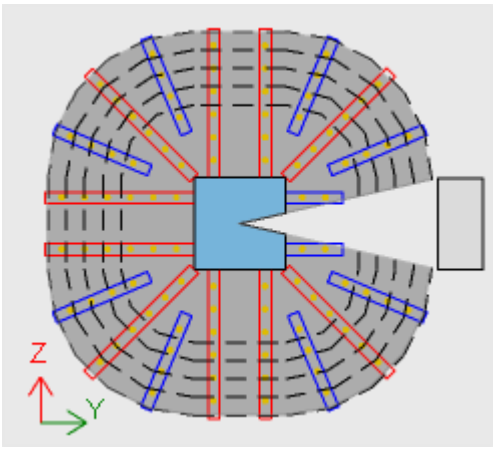
    - Shorter rails interrupted by edges
    - User has no direct control over these but they respond to changes to design rails
    - Not considered in the design
- **Types of detailing rail**
    - Rails used to fulfill spacing requirements (circular arrangement)
        - Can be shorter or have similar size to “design rails”
        - Will occur in circular arrangements only
        - Automatically added if the spacing of the “design rails” is larger than the maximum allowed along the first reinforcement line
        - Automatically added if the spacing of the “design rails” is larger than the maximum allowed at reinforcement perimeters beyond the first reinforcement line (shorter rails).



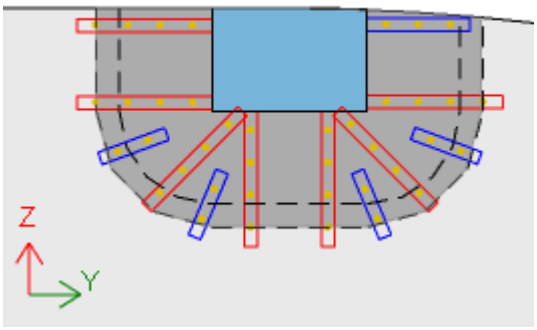
    - Rails used to fill distances to edges
        - Will in most usual cases have the same length as “design rails”
        - Can have different length if affected by opening dead-zones or angled edges
        - Can occur in both circular and orthogonal arrangements
        - Placed between the column section boundary and the slab edge
        - Only get used if “Design rails” are added
        - Automatically added if the column is eccentric to the slab edge by more than the maximum allowed rail spacing
        - Spacing between edge “detailing” rails is equal to the maximum allowed rail spacing in the given direction



- Shorter rails due to interruption by an opening dead-zone
  - Will be shorter than “design rails”
  - Can occur with “design rails” and “detailing rails”
  - Can occur in both circular or orthogonal arrangements



- Shorter rails due to interruption by edges
  - Will be shorter than “design rails”
  - Can occur with “design rails” and “detailing rails”
  - Can occur in both circular or orthogonal arrangements



Use of property sets to standardize checks and reinforcement

The benefits of using property sets with punching checks are:

- Quick reinforcement standardization - allows to have a standard reinforcement arrangement (rails/spacings) checked at various positions throughout the model
- Ensuring the same user property overrides are used at identical positions
- Applying point load loaded area properties quickly to any point loads

Punching check property sets can be created in two ways:

- from the ribbon,
  - by clicking **Home > Manage Property Sets > New... > Punching Check**
- from a scene view,
  - by right clicking on an existing punching check item and selecting **Create property set**

Creating a punching check property set will save the following properties:

- Tension reinforcement surface (top/bottom);
- Point Load Breadth;
- Point Load Depth;
- Point Load Orientation;
- Beta - user limit for EC;
- User factor for Vt for BS design;
- u_0 user reduction for EC and BS design;
- u_1 user reduction for EC and BS design;
- b_o user reduction for ACI design;

- Slab override and the associated flags for slab in y+/y-/z+/z-;
- Slab override (drop) and the associated flags for slab in y+/y-/z+/z-;
- All reinforcement design data (everything under "Design Input" in the properties window)

Once created, a punching check property set can then be applied to other punching checks from the **Properties** window.

Applying a punching check property set will update:

- Tension reinforcement surface (top/bottom);
- Point Load Breadth - point loads only;
- Point Load Depth - point loads only;
- Point Load Orientation - point loads only;
- Beta - user limit in EC design;
- User factor for Vt in BS design;
- u_0 user reduction in EC and BS design;
- u_1 user reduction in EC and BS design;
- b_o user reduction in ACI design;
- Slab override and the associated flags for slab in y+/y-/z+/z-. If not overridden the slab presence flag statuses are not passed;
- Slab override (drop) and the associated flags for slab in y+/y-/z+/z-. If not overridden the slab presence flag statuses are not passed;
- Reinforcement design data - Will apply only where reinforcement arrangements can be designed.

Note the following:

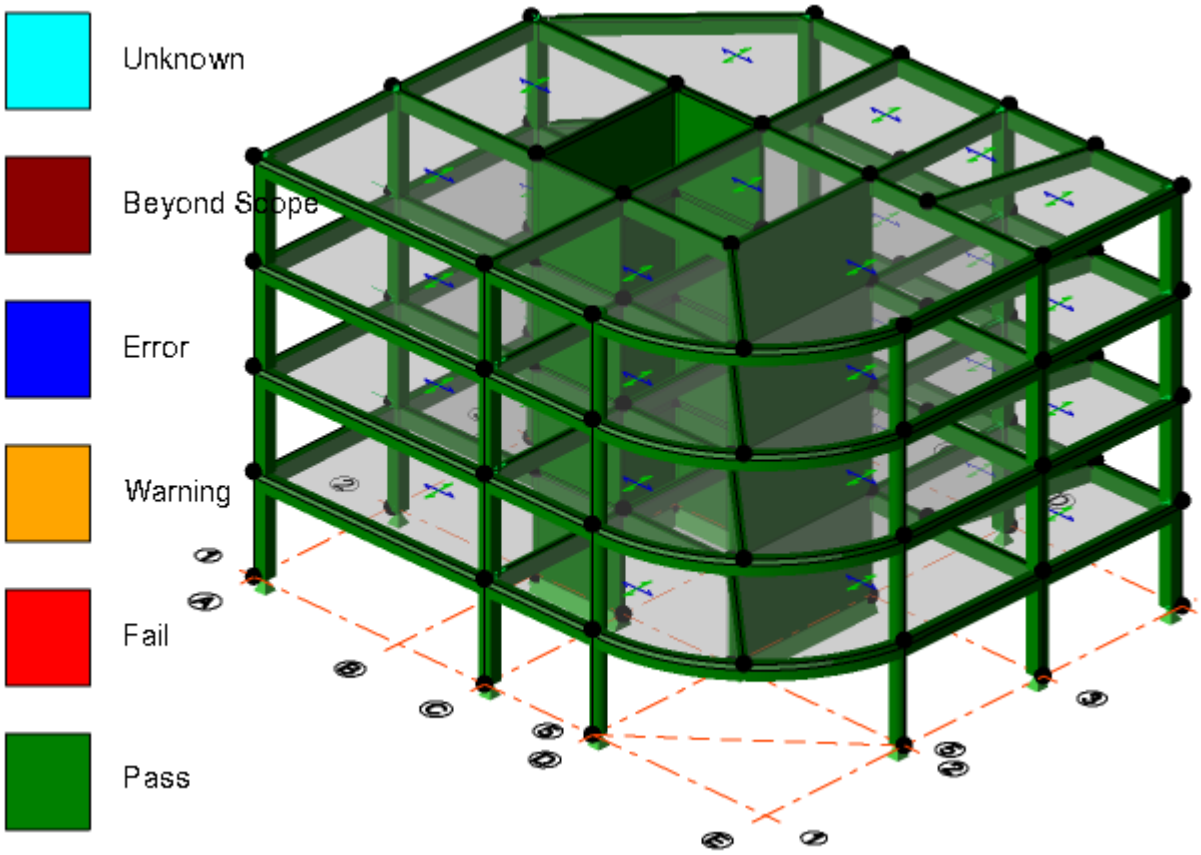
- Punching check property sets can be edited at any stage between creation and application;
- Only property fields available in the destination check get populated by the property set;
- It is not a reinforcement copy feature. As an example, applying property sets from edge or corner checks to internal checks will result in a different number of rails with the same properties being used;
- Adding a column drop hides but does not delete the reinforcement data if it exists and this will reflect in any property sets saved at this position afterwards;
- Inconsistent slab presence flag statuses may occur between slab and drop when a property set from a punching check without a drop is applied to another at a different position and with a slab drop;

See also

Flat slab design workflow (page 215)

3.12 Pad base design workflow

The small concrete building model shown below will be used to demonstrate the pad base design process.



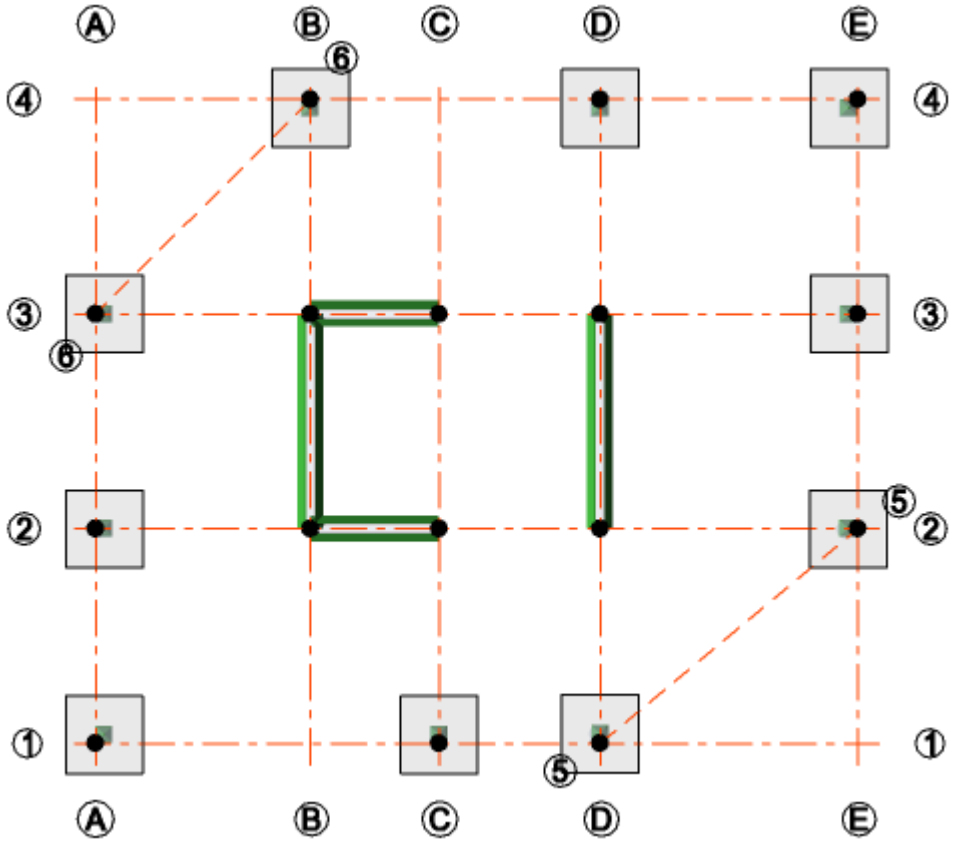
The model has already been designed prior to placing the bases.

Apply pad bases under supported columns

Pad bases can only be placed under, and be loaded by supported columns; strip bases can only be placed under and be loaded by supported walls.

NOTE If a ground beam is attached to the same support, loading from the beam will also be considered in the base design.

At this stage, as you are not aware of the individual base size and depth requirements; you can simply choose to place the bases where required, accepting the default size/depth offered.



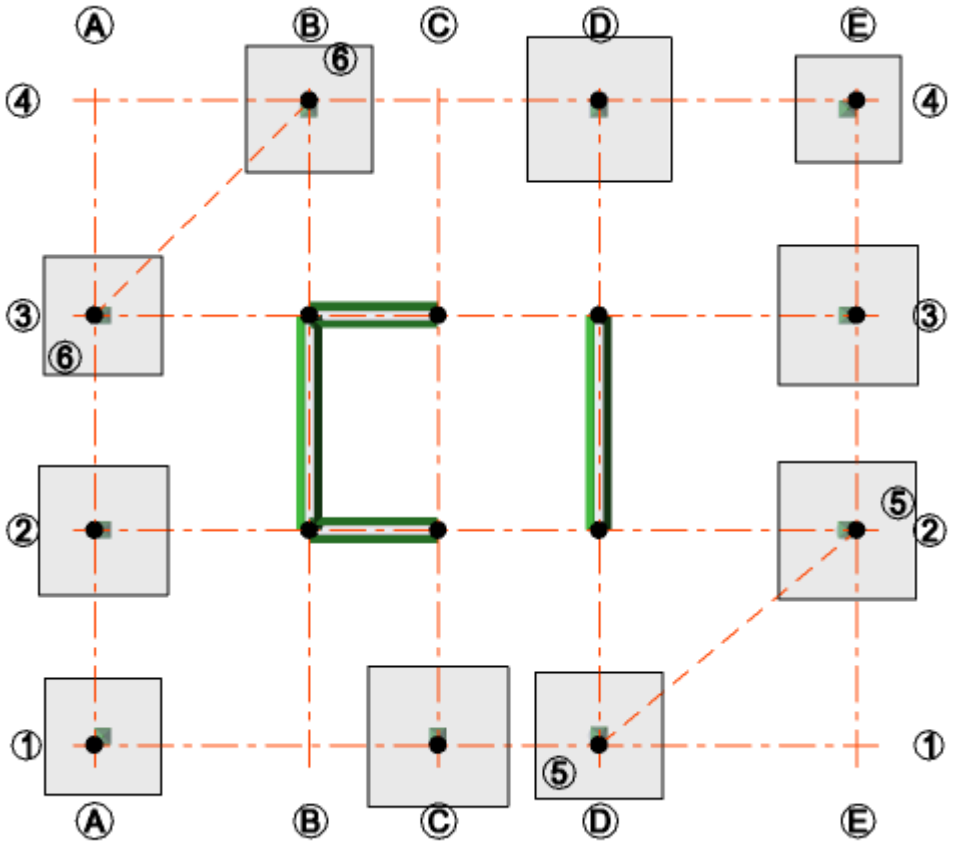
Auto-size pad bases individually for loads carried

To obtain an idea of the range of potential sizes, bases should initially be designed individually for their respective loads, as follows:

1. Access **Design Groups** page of the **Design Settings** dialog box to ensure that group design is turned off for Isolated Foundations.
2. Select the bases to be auto-sized and in the **Properties** window and choose to auto-design both the size and depth; In this way the program establishes suitable base dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.
3. From the **Foundations** tab click **Design Pad Bases**.

   Each base will be sized accordingly (any that are not in auto-design mode will simply be checked).



4. With the auto-design options cleared, you can then adjust individual base dimensions and re-check if required (by right-clicking the base that has been edited and choosing Check Member).

   The site boundary may impose restrictions on the positioning of an isolated foundation relative to the column/wall it supports. This restriction may result in a requirement for an offset base, this can be achieved by specifying the eccentricity required in the base properties.

NOTE The overall auto-design procedure is summarized as follows:

1. Bearing Design: - increase size
2. Bending Design: - increase reinforcement - If max allowable reinforcement is reached then increase depth, set reinforcement back to start point, and go back to step 1.
3. Shear Design - increase depth, set reinforcement back to start point, and go back to step 1
4. Punching Shear Design - increase depth, set reinforcement back to start point, and go back to step 1
5. Sliding Checks - increase depth, set reinforcement back to start point, and go back to step 1

6. Uplift Checks - increase size, set reinforcement back to start point, and go back to step 1

At every stage, if the max allowable depth is reached the design fails.

Apply grouping to rationalize pad base sizes

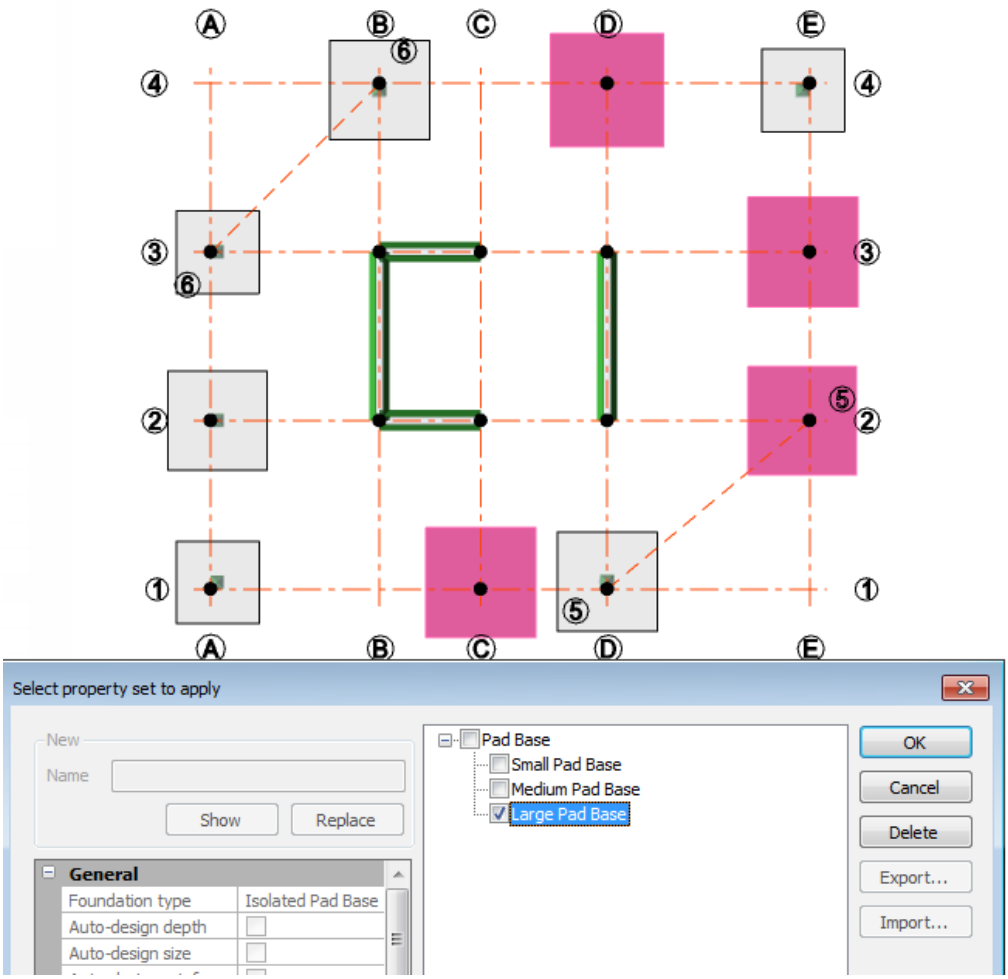
NOTE Grouping can only be applied to pad bases - not to strip bases.

Once pad bases have been sized individually, the designs can be rationalized by activating grouping, in order to obtain one design per group sufficient for all bases within the group.

This is done as follows:

1. Select a base that you want to be in a particular group.
2. In the **Properties** window, ensure it is set to auto-design.
3. Right-click the same base and from the context menu choose Create Property Set...
4. Select all the other bases that you want to be in the same group.

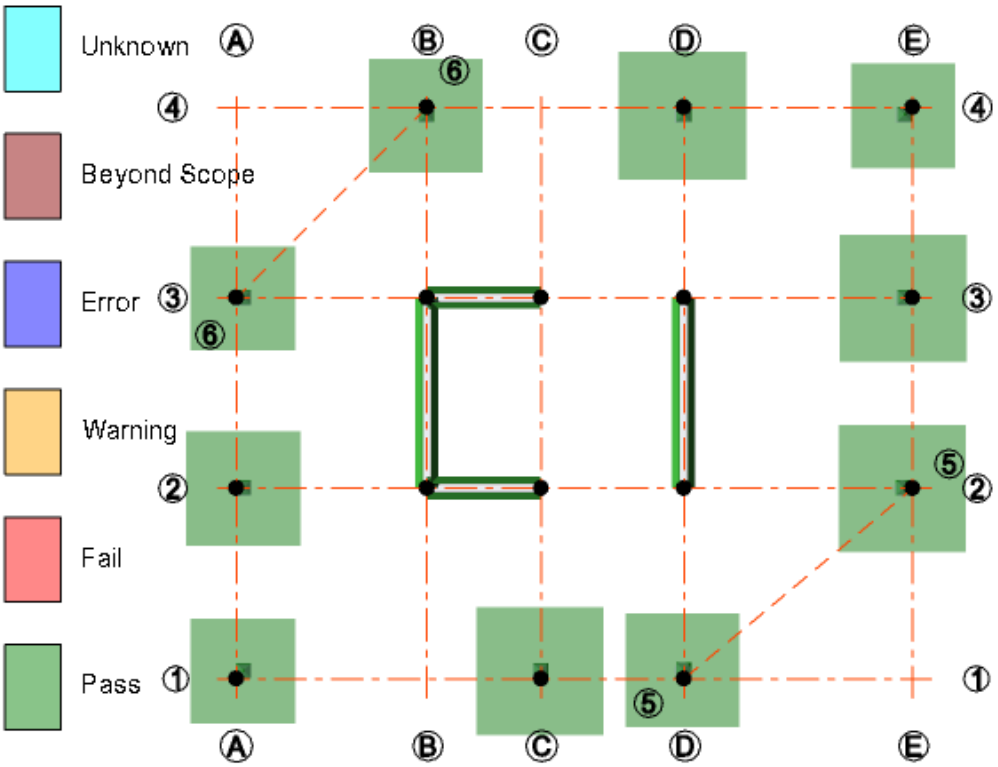
5. In the **Properties** window, click **Apply...** to apply the property set you have just created to the selected bases.



NOTE Ensure you click **Apply...** from the **Properties** window and not from the right-click menu, otherwise the properties will only be applied to the last base selected.

6. From the Groups page of the Project Workspace, right-click Pad Bases (under the Design branch) and choose Regroup Members - this will put those bases that share similar properties into the same group.
7. Open **Design Settings** dialog box, and from the **Design Groups** page select the option to design isolated foundations using groups.

8. Click **Design Pad Bases** - the results obtained will reflect the grouping that has been applied.



Review/optimize base design

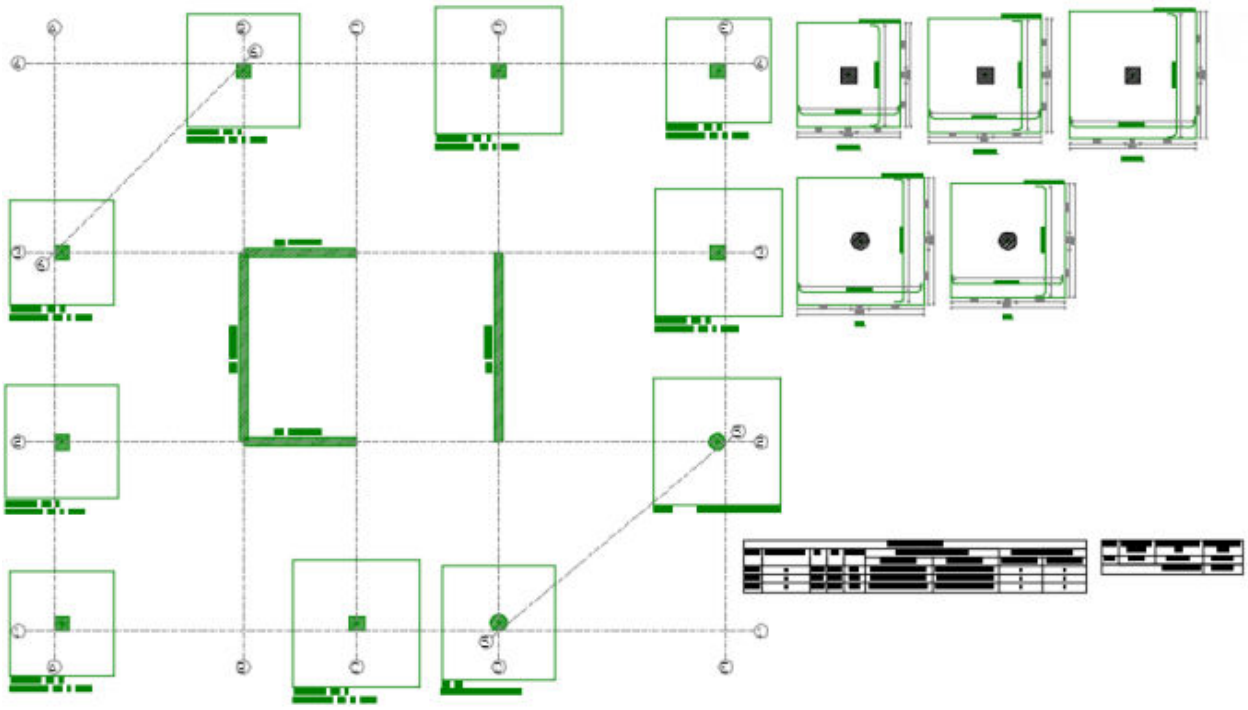
In the Review View you can examine the design efficiency by switching from Foundations Status to Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings pages in the **Design Settings** dialog box.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to

eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer



Print calculations

Create a model report that includes the concrete pad base design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Ignoring small shear forces in pad base design checks

3D analysis will always develop small shear forces at pinned bases – in two directions - which can over-complicate the base design procedure and results. In many circumstances these forces would traditionally be ignored by engineers or not even evaluated by hand methods.

In Tekla Structural Designer an option is provided to allow these forces to be ignored. This allows the engineer, at their discretion, to define an ignorable %

of passive pressure value for pinned bases. (The default for this being 1% of the passive resistance.)

Setting the level of ignorable force is a matter of judgment; it is not defined in any code, and logically it should relate in some way to the size of the foundation.

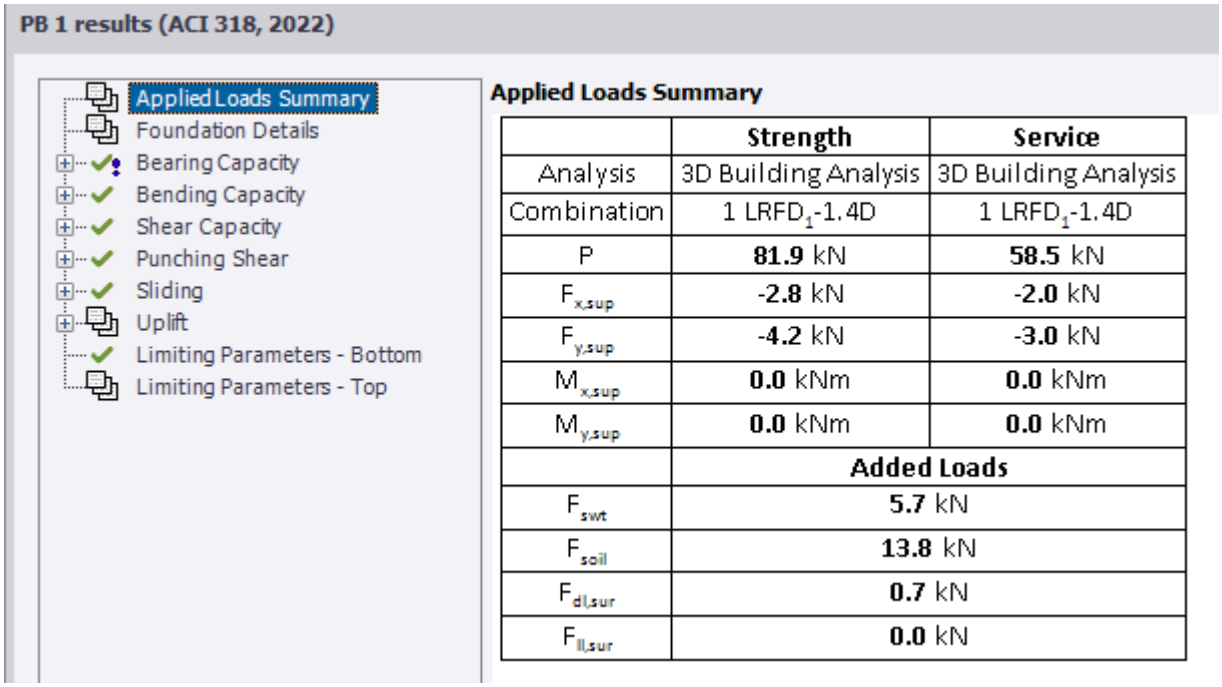
Activating the option to ignore small shear forces

This option can be activated as follows:

1. From the **Design** tab, select **Settings > Concrete > Foundations > Isolated Foundations > General Parameters**
2. Under **Passive Resistance** select, **Consider passive resistance of soil for sliding check**
3. Specify the required **Ignorable percentage of passive pressure**

Impact on the bearing capacity checks

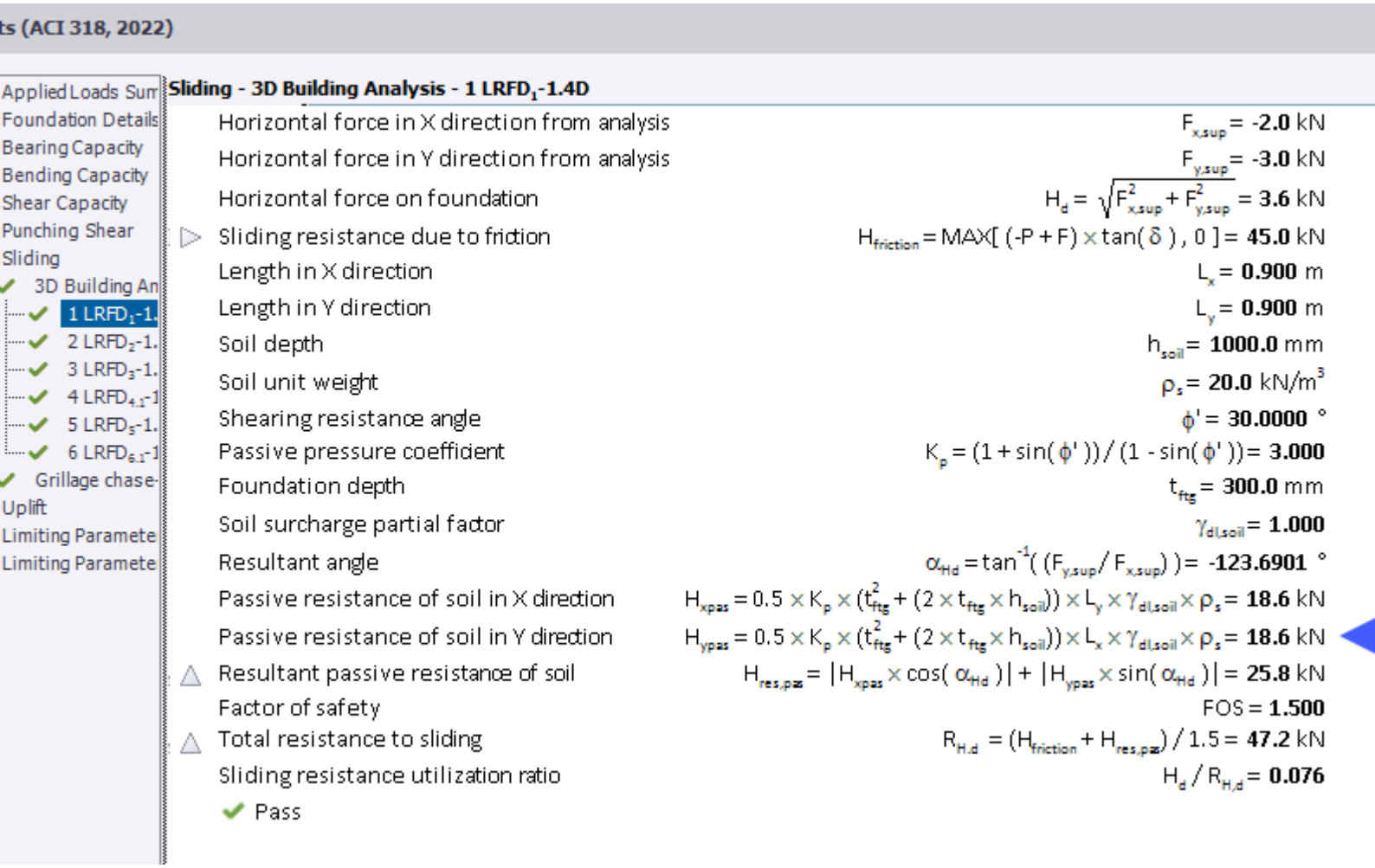
When an ignorable percentage of passive pressure (IPPP) greater than 0% has been specified, the analysis shear reactions Fx,sup and Fy,sup on the applied loads summary are unchanged.



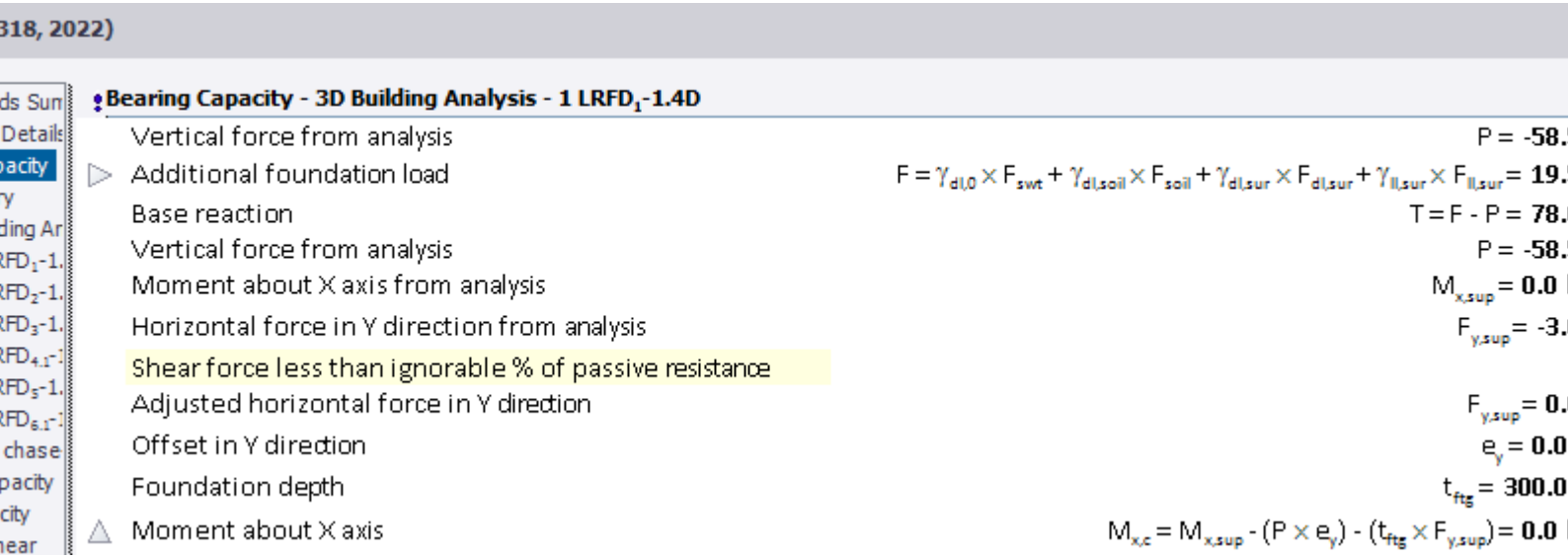
	Strength	Service
Analysis	3D Building Analysis	3D Building Analysis
Combination	1 LRFD$_1$-1.4D	1 LRFD$_1$-1.4D
P	**81.9** kN	**58.5** kN
$F_{x,sup}$	**-2.8** kN	**-2.0** kN
$F_{y,sup}$	**-4.2** kN	**-3.0** kN
$M_{x,sup}$	**0.0** kNm	**0.0** kNm
$M_{y,sup}$	**0.0** kNm	**0.0** kNm
	Added Loads	
F_{swt}	**5.7** kN	
F_{soil}	**13.8** kN	
$F_{dl,sur}$	**0.7** kN	
$F_{ll,sur}$	**0.0** kN	

For Moment about X axis:

- If Mx,sup = 0
  - The Passive Resistance of Soil in Y-Direction (Hy,pas) is calculated (this can be viewed in sliding resistance check shown below)



- If Fy,sup <= IPPR of Hy,pas
  - In the Bearing Capacity check, because the shear force is less than the ignorable % of passive resistance, the horizontal force in the Y direction is adjusted to zero and the bearing pressure checks continue using the replacement zero value.



- Else, If Fy,sup > IPPR of Hy,pas
  - The bearing pressure checks continue using the the original Fy,sup value

For Moment about Y axis:

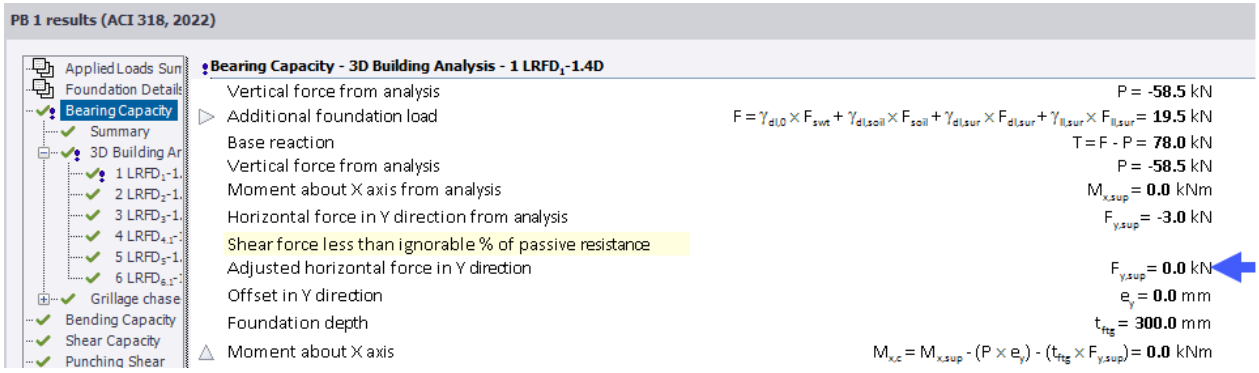
- The process is repeated as above reversing X and Y

Impact on the bending capacity, shear capacity, punching capacity, and sliding checks

When an ignorable percentage of passive pressure (IPPP) greater than 0% has been specified, it impacts the remaining checks as follows.

Bearing Capacity Checks:

- For Moment about X axis:
  - If Fy,sup has been determined as ignorable for bearing (see above) then it is ignored here also.



- For Moment about Y axis:

  The process is repeated reversing X and Y

Shear Capacity Checks:

The same adjustments are made as in bending capacity checks above.

Punching Capacity Checks:

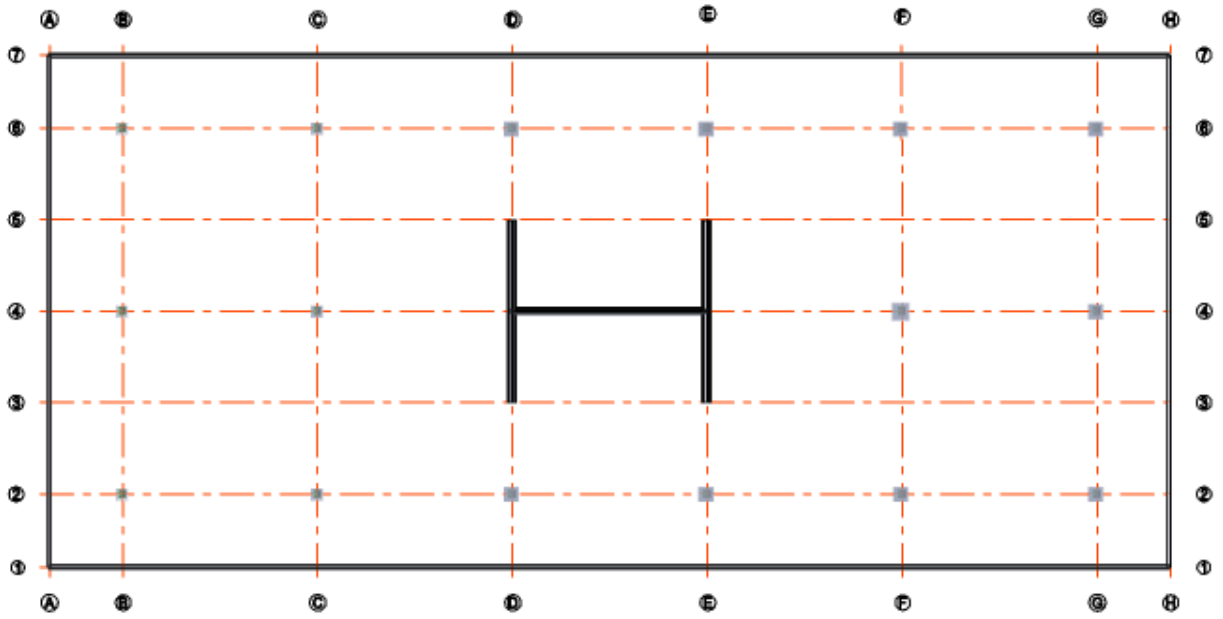
The same adjustments are made as in bending capacity checks above.

Sliding Checks:

The same adjustments are made as in bending capacity checks above.

3.13 Pile cap design workflow

The small concrete building model shown below will be used to demonstrate the pile cap design process.



The model has already been designed prior to placing the pile caps.

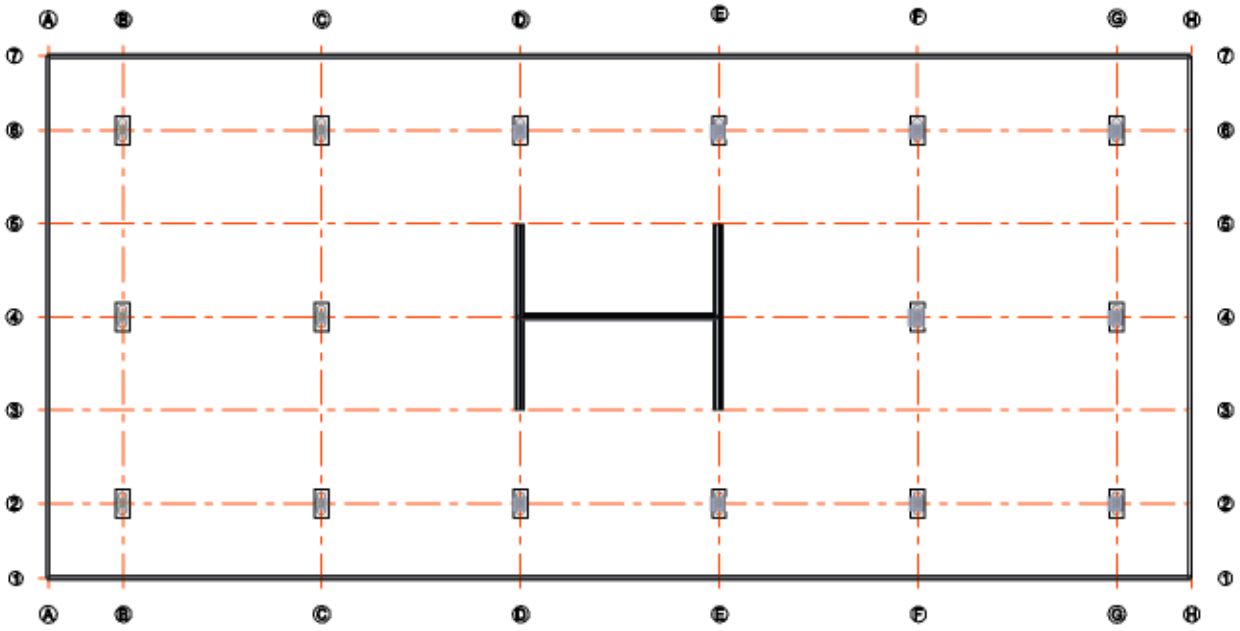
Apply pile caps under supported columns

Before a pile cap can be placed the **Pile Type Catalogue** must contain at least one pile type.

Pile caps can only be placed under and be loaded by supported columns.

NOTE If a ground beam is attached to the same support, loading from the beam will also be considered in the pile cap design.

At this stage, as you are not aware of the individual pile cap size and depth requirements; you can simply choose to place pile caps where required, accepting the default size/depth offered.



Auto-size pile caps individually for loads carried

To obtain an idea of the range of potential sizes, pile caps should initially be designed individually for their respective loads

Note that when piles are auto-designed the outcome will depend on the auto-design method selected; the pile cap size will either be based on the minimum number of piles required, or on the minimum pile capacity.

To individually size the pile caps:

1. In the **Design Settings** dialog box got to **Concrete > Cast-in-place > Foundations > Isolated Foundations > Piles** to choose the pile auto-design method required: (minimize pile capacity, or minimize number of piles).
2. Access **Design Groups** page of the **Design Settings** dialog box to ensure that group design is turned off for Isolated Foundations.
3. Select the pile caps to be auto-sized and then in the **Properties** window choose to auto-design both the piles and depth; In this way the program will establish suitable pile cap dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.
4. From the **Foundations** tab, click **Design Pile Caps** and all the pile caps set in auto-design mode will be sized accordingly. (Those not in auto-design mode will simply be checked). Similarly the piles beneath the pile caps will either be designed (if pile auto-design mode is enabled) or checked.

At any point you can switch to a user defined arrangement, modify the pile cap configuration and have the design re-checked.

One example where you might choose a user defined arrangement is where the site boundary imposes restrictions on the positioning of the pile cap relative to the column/wall it supports. Switching to a user defined arrangement allows you to specify an eccentricity and create an offset pile cap.

Apply grouping to rationalize pile cap sizes

Once pile caps have been sized individually, the designs can be rationalized by activating grouping, in order to obtain one design per group sufficient for all pile caps within the group.

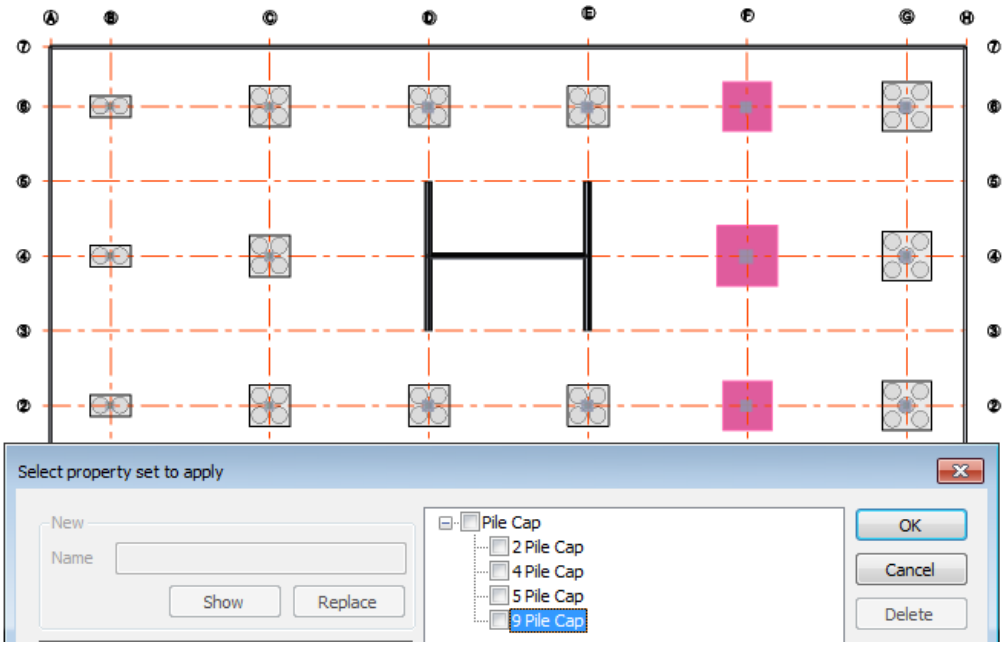
This is done as follows:

1. Select a pile cap that you want to be in a particular group.
2. In the **Properties** window, ensure it is set to auto-design.
3. Right-click the same pile cap and from the context menu choose Create Property Set...
4. Select all the other pile caps that you want to be in the same group.

NOTE When applied moments are significant, be cautious when grouping pile caps where auto-design has initially determined different principal directions.

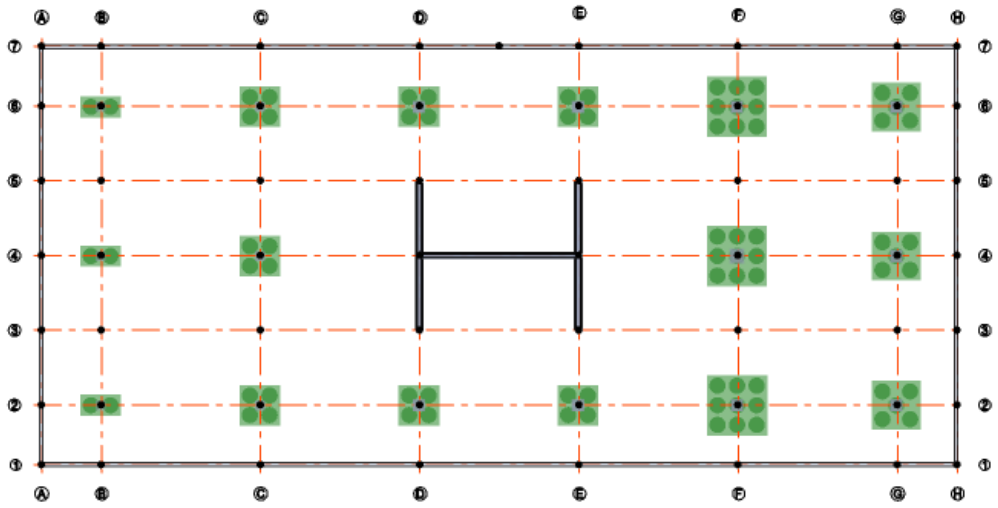
---

5. In the **Properties** window, click **Apply...** to apply the property set you have just created to the selected pile caps.



NOTE Ensure you click **Apply...** from the **Properties** window and not from the right-click menu, otherwise the properties will only be applied to the last pile cap selected.

6. From the Groups page of the Project Workspace, right-click **Pile Caps** (under the Design branch) and choose Regroup Members - this will put those pile caps that share similar properties into the same group.
7. Open the **Design Settings** dialog box, and from the Design Groups page select the option to design isolated foundations using groups.
8. Click **Design Pile Caps** - the results obtained will reflect the grouping that has been applied.



Review/optimize pile cap and pile design

In the Review View you can graphically examine the pile cap design status & efficiency by switching between Foundations Status and Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

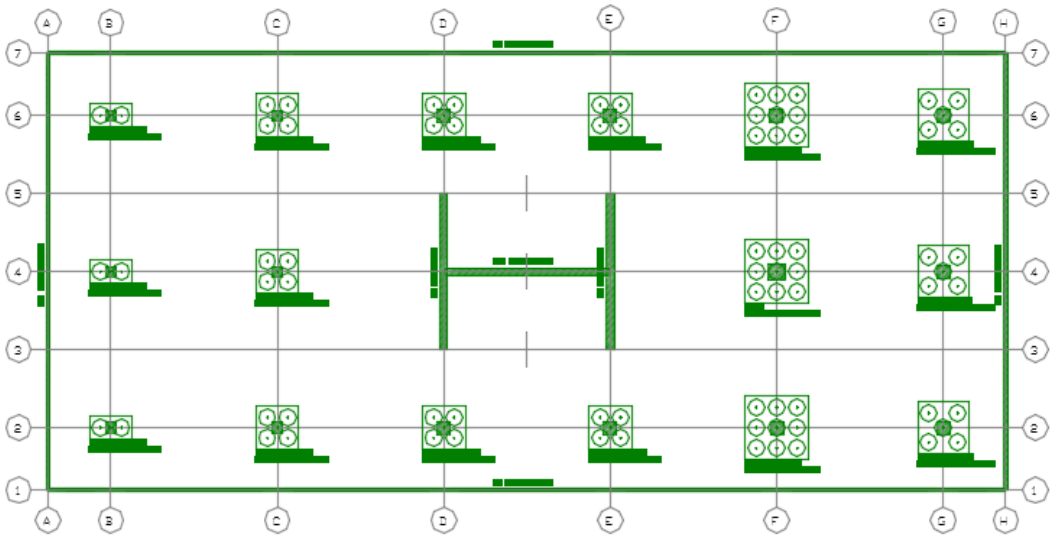
Similarly you can graphically examine the pile design status & efficiency by switching between Piles Status and Piles Ratio.

If the selections are unacceptable you may need to review the settings in **Design Settings > Concrete > Cast-in-place > Foundations**

Design results can be examined in more detail by selecting Tabular Data and setting the View Type to Design Summary. By selecting the Pile Caps or Piles as the Characteristic (and the appropriate material) the full results for each pile cap or pile can be examined.

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer.



Print calculations

Create a model report that includes the concrete pile cap design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

3.14 Pad base, strip base and pile cap design forces

Forces acting on supports

The following forces and moments on the supports are determined from the analysis of the active load combinations:

- Vertical force in direction Z
- Horizontal forces in directions Y and X
- Moments around X and Y axis

NOTE A torsional moment around the Z axis is also determined, but the base/pile cap is not designed for this in the current release.

Foundation self weight

The foundation self-weight is automatically calculated and applied as an added load, F_{swt}

Soil self weight

The surcharge depth and soil unit weight that have been specified in the base/pile cap properties are used to determine the soil self weight. This is applied as an added load, F_{soil}

NOTE In the current release, horizontal pressure caused by soil is not considered.

Additional surcharge loads

For isolated foundations user can apply additional surcharge loads: acting in the global Z direction.

- Permanent (dead) surcharge load
- Variable (live) surcharge load

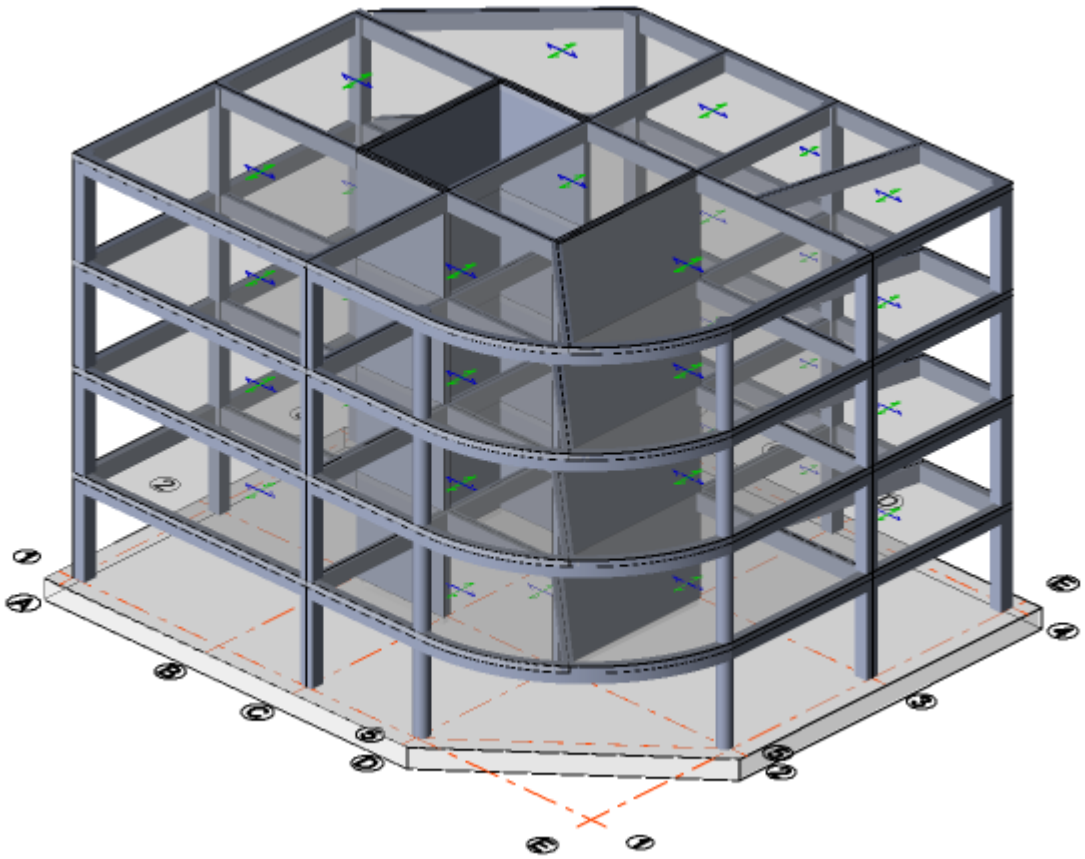
Design Forces

To obtain the design forces, the loads due to foundation self weight, soil self weight and additional surcharge are added to the forces acting on the supports.

These design forces (axial load and bi-axial shear and moment) are then applied to the base/pile cap at the foundation level.

3.15 Mat foundation design workflow (metric units)

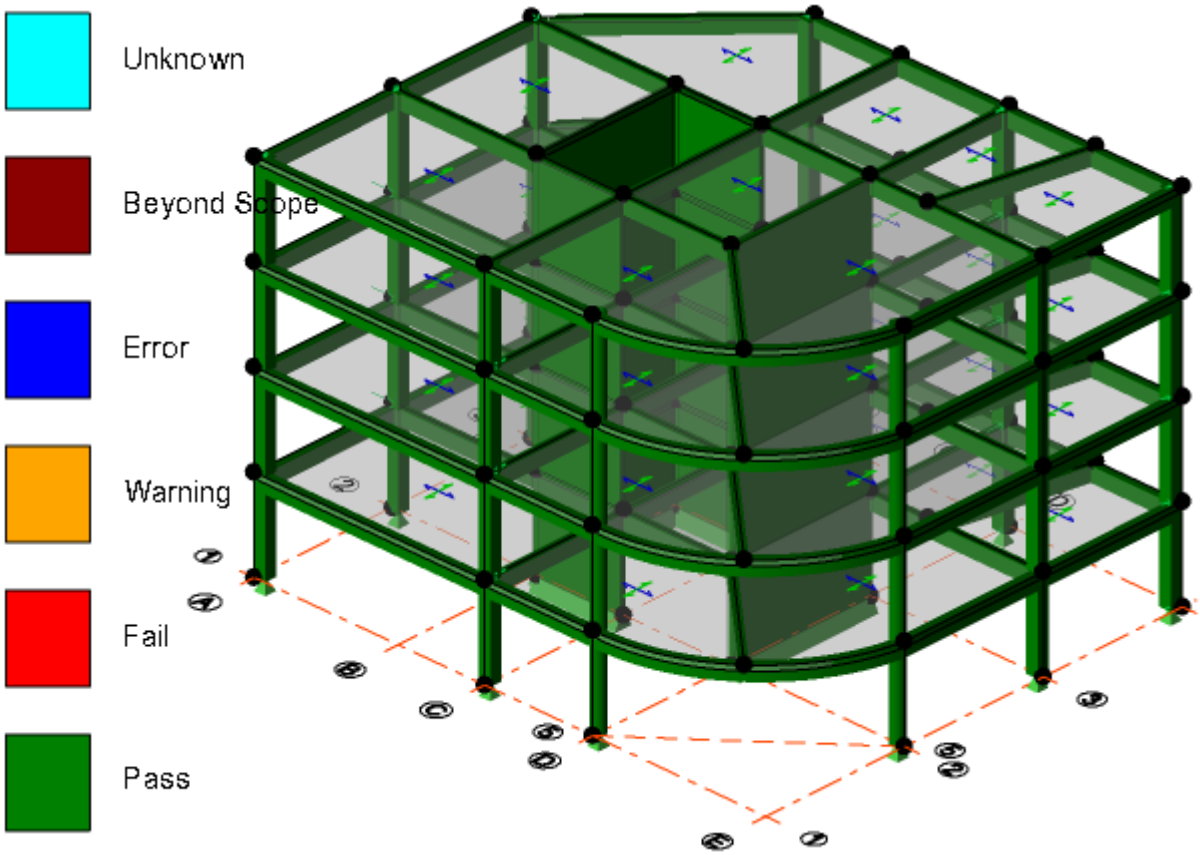
The small concrete building model shown below will be used to demonstrate the process for modeling and designing a mat foundation.



For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



Determine the soil parameters

Unless you are going to define discreet piled supports, the mat will need to be supported on ground bearing springs. These are activated by selecting **Use Ground Bearing Springs** (located under the **Slab parameters > Soil Parameters** sub-heading in the mat properties).

When these are activated you have then to also specify:

- Allowable bearing pressures
- Ground stiffness type (Linear, or non linear spring)
  - Linear ground stiffness, or,
  - Non-linear ground stiffness (+/-) and tension/compression limits
- Horizontal support

Allowable bearing pressures

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground stiffness type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear spring are applied:

- Allowable bearing pressures are checked
- Uplift (tension) is checked
- If no problems then linear springs are sufficient

When non-linear springs are applied:

- You can have compression only
- And also capped compression
- Either way analysis takes longer

NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Ground stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below[1] illustrates the potential range of values that might be considered.

Material	Lower Limit (kN/m^3)	Upper Limit (kN/m^3)
Loose Sand	4,800	16,000
Medium Dense Sand	9,600	80,000
Dense Sand	64,000	128,000
Clayey Medium Dense Sand	32,000	80,000
Silty Medium Dense Sand	24,000	48,000
Clayey Soil (qa<200kPa)	12,000	24,000
Clayey Soil (200<qa<800kPa)	24,000	48,000
Clayey Soil (qa>800kPa)	48,000	200,000

1. Reference: "Foundation Analysis and Design", Joseph E. Bowles, 1995 (Table 9-1)

Horizontal support

The degree of horizontal support provided by the ground springs can be set as:

- Fixed
- Free
- Spring (default 20% of vertical spring stiffness)

If set to "Free" a mechanism will result unless you provide additional discreet supports.

Determine the remaining mat properties

You are required to manually specify the **Reduce live loads by** (US regional code), or **Reduce imposed load by** (other regional codes) mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an "area" method of mat creation you will also need to specify the amount of slab overhang.

The remaining properties are then similar to those used to define a typical concrete flat slab.

In this example the minimum area method is used to create a mat with the following properties:

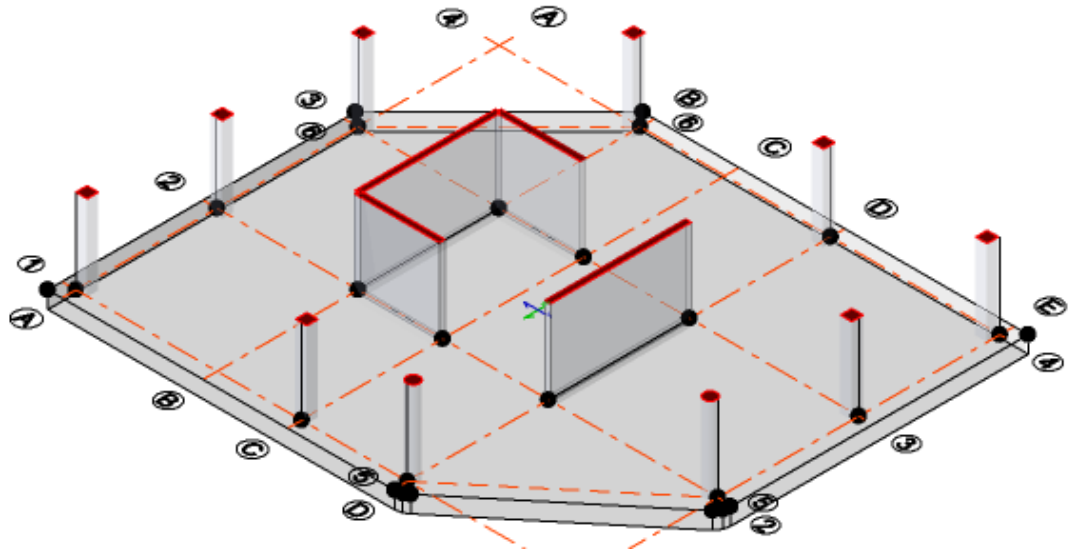
- Imposed/live loads reduced by 30%
- Default overhang
- Mat thickness 600mm
- Ground bearing springs used
- Default allowable bearing pressures
- Default linear spring properties

Create the mat

To create the mat:

1. Choose the method, (e.g. Minimum Area)
2. Enter the mat properties, (see above)
3. Click on those columns (or walls) that define its perimeter,

4. Either press <Enter>, or re-click on one of the selected columns to complete the mat definition.



NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

Enable soil structure interaction

When not supported by a mat, columns and walls typically have supports at their bases.

When a mat is introduced these supports must be removed - as the mat now supports the whole building on ground bearing springs. Consequently adding a mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

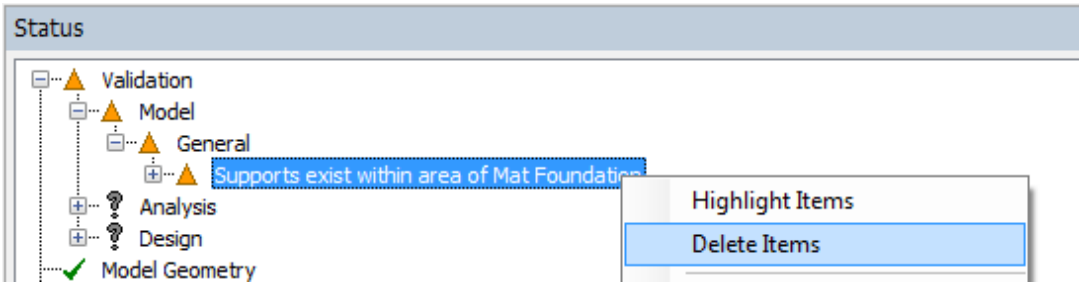
If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed from the Design ribbon by running **Design All (Static)** and any seismic RSA combinations by running **Design All (RSA)**. These processes will also recheck all the member designs taking account of the effects of soil structure interaction.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs supported on ground bearing springs.

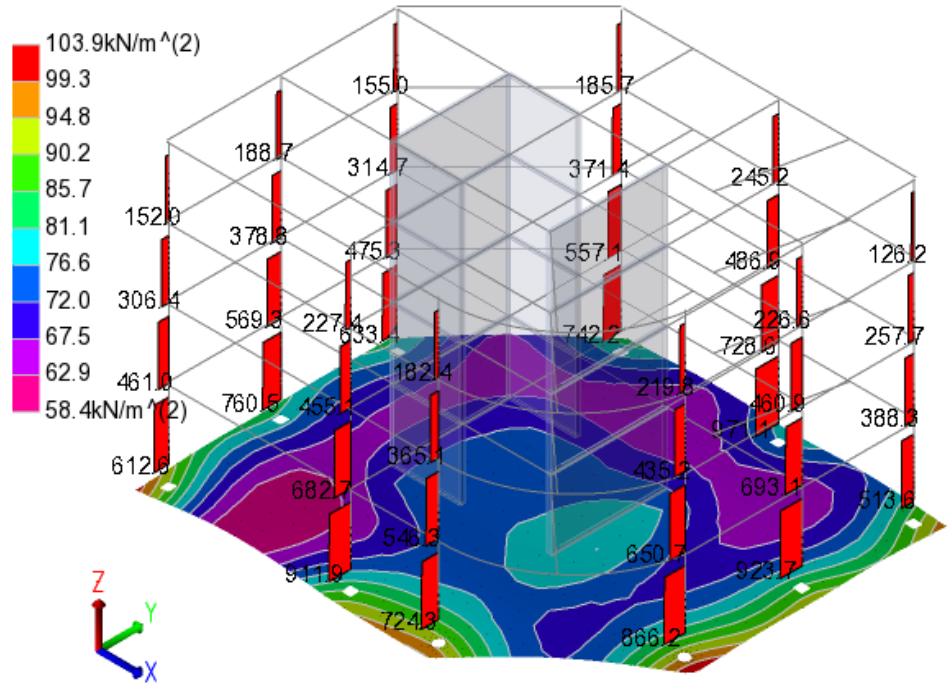
In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

NOTE In a large model, if you anticipate several analysis might be required to determine the size of mat required, you might find it more efficient to just run the analyses that are required from the Analyze ribbon then re-run the member design at a later stage.

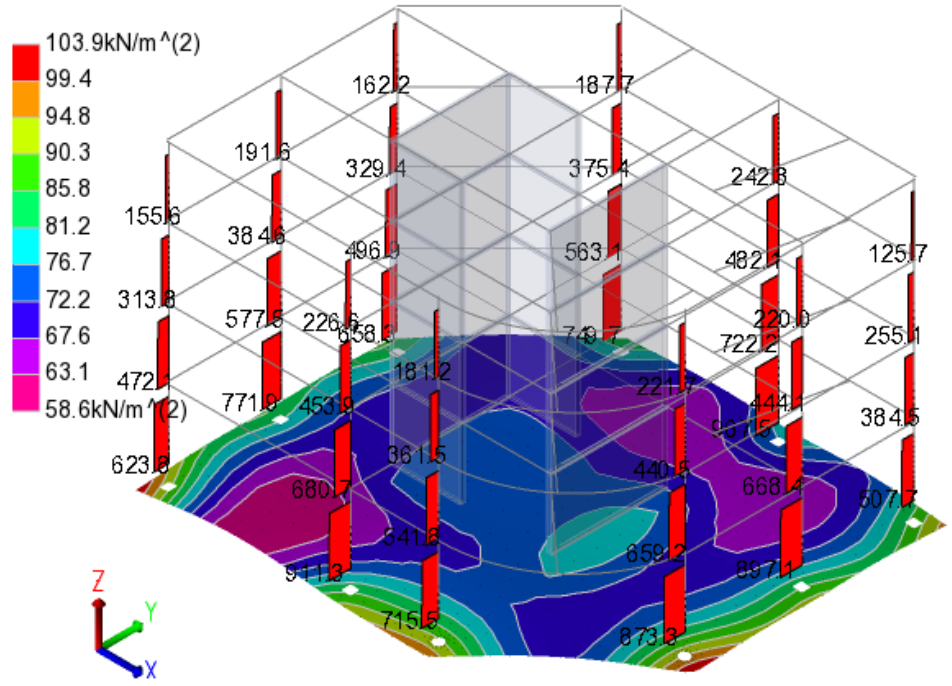
NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Check foundation bearing pressure and deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.



Solver Model used for 1st Order Linear



Solver Model used for Grillage Chasedown

By viewing the 2D deflection results for combinations based on "service" rather than "strength" factors the stiffness adjustments that you apply (via Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

The default stiffness adjustments are dependent on the design code. For design to EC2 the default adjustment factor applied is 0.2. For design to ACI the default adjustment factor applied is 0.25.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be "Valid" or "Out of Date".

Unless the status is "Valid" you should recheck the member designs (taking into account the effects of soil structure interaction) by running **Design All (Static)** from the Design ribbon.

NOTE Similarly if an RSA design has previously been performed, but is now out of date **Design All (RSA)** should be re-run.

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring Select bars starting from is set to Current rather than Minima) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

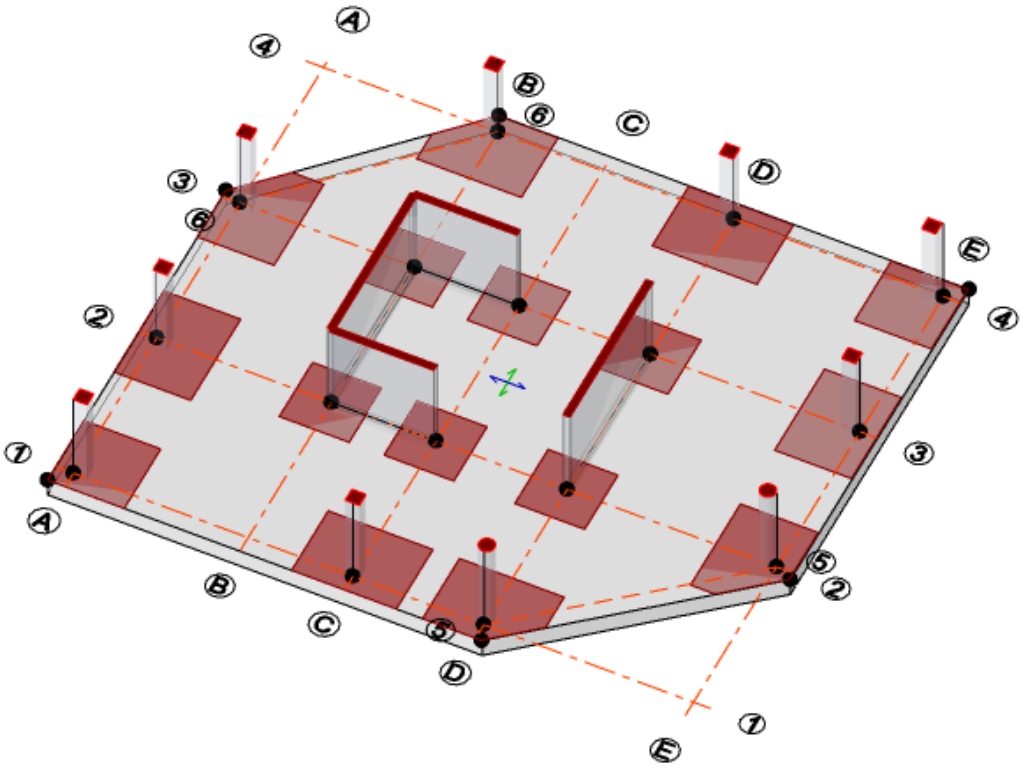
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered. Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgment.



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimization stage.

NOTE Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design mats

NOTE Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

- From the Foundations ribbon run **Design Mats** in order to design or check all the mats in the model (each according to their own autodesign setting), or,
- If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either **Design Slabs** or **Check Slabs**.

NOTE When accessed from the right-click menu, these commands operate as follows:

- **Design Slabs** will re-design slabs and mats (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Slabs** will check the current reinforcement in slabs and mats regardless of the current autodesign setting.

Review/optimize mat design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 150mm.

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.

- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Foundations ribbon run **Design Patches** in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,
- In the 2D View of the level which you want to design right-click and choose either **Design Patches** or **Check Patches**. Working in this way restricts the design or checking to the patches in the current view.

NOTE When accessed from the right-click menu, these commands operate as follows:

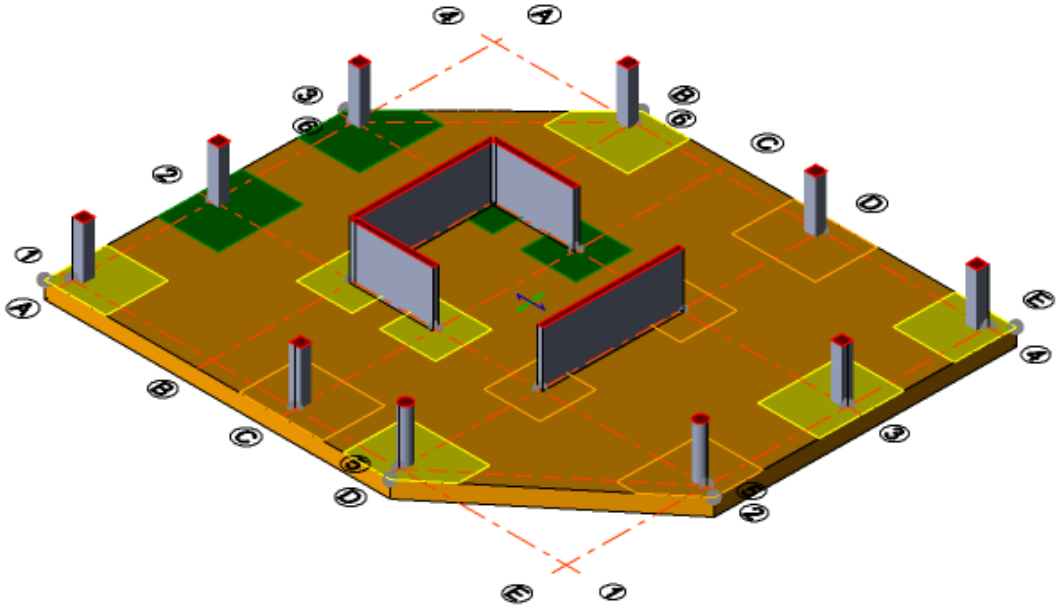
- **Design Patches** will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Patches** will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimized)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include

forcing the spacing of the mat reinforcement to be matched (if the mat has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

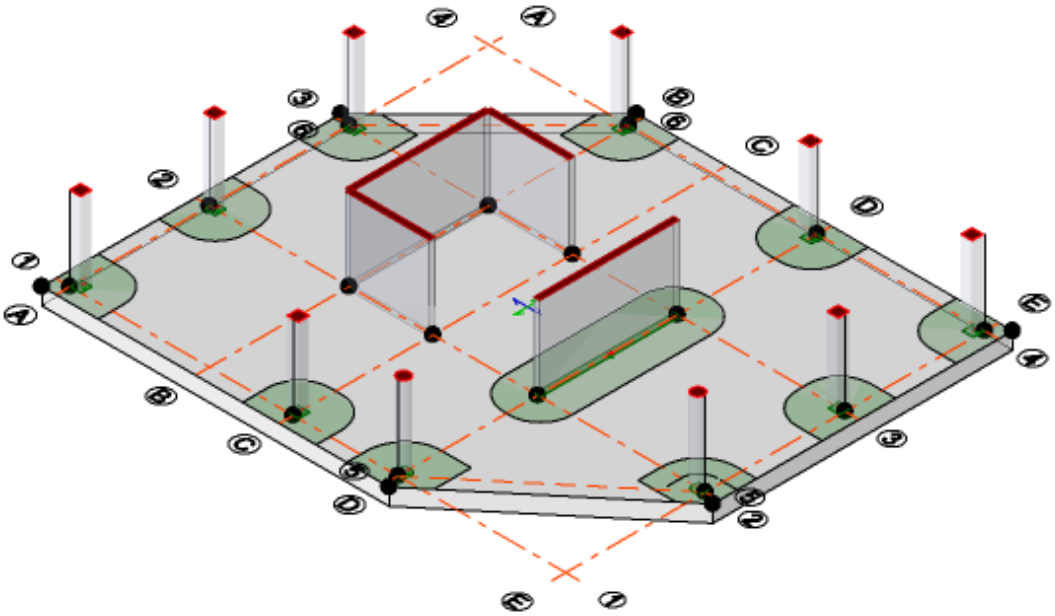


Add and run punching checks

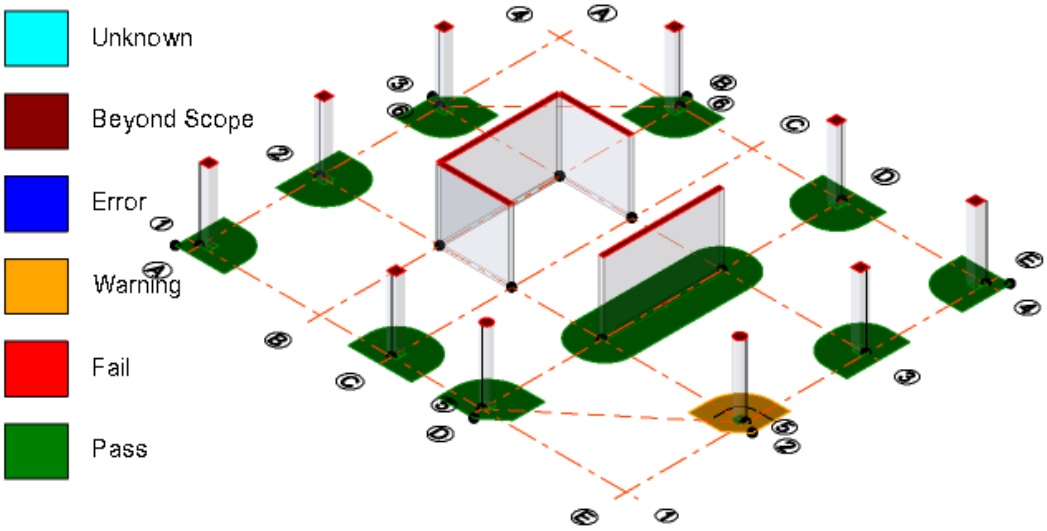
Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.



Once added click **Design Punching Shear**. See:

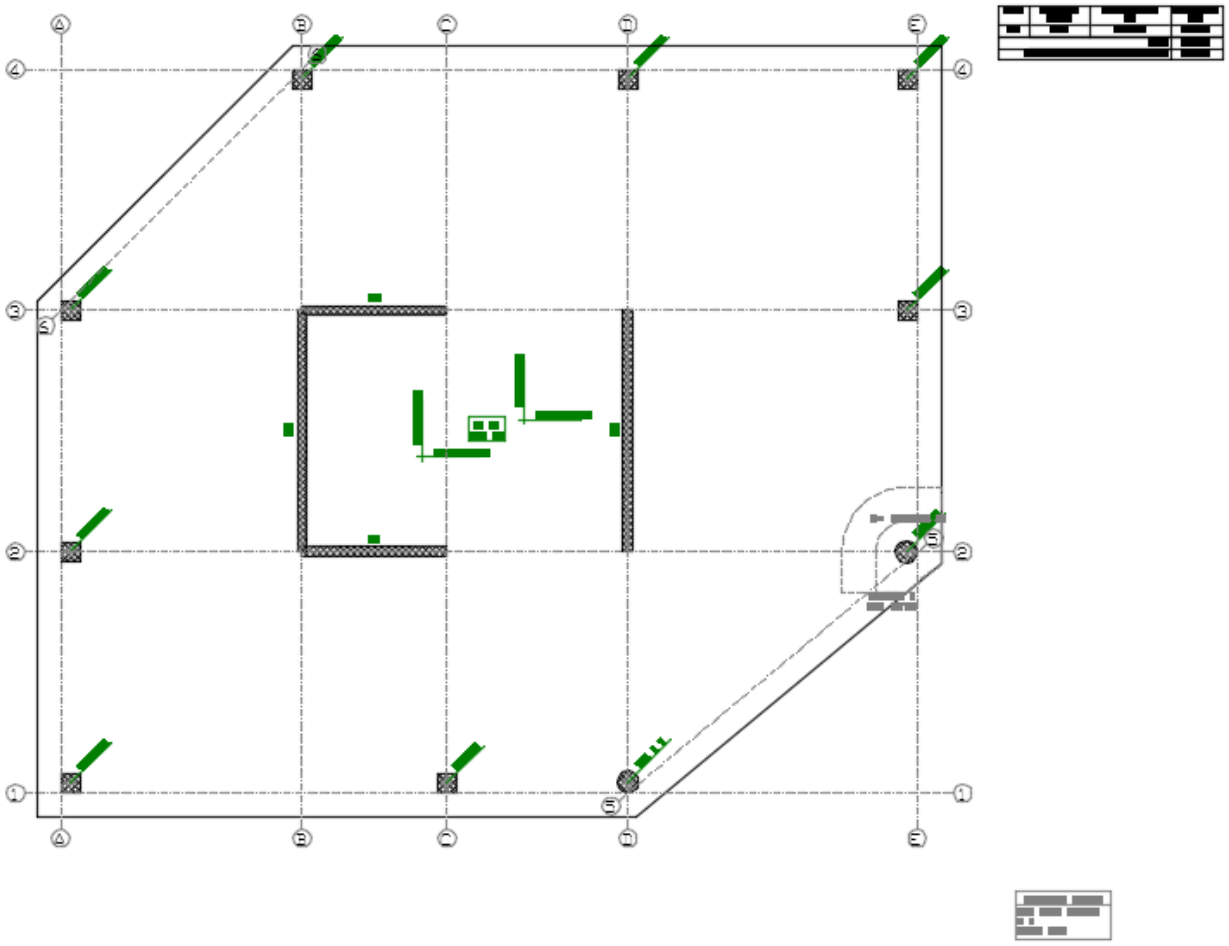


The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer

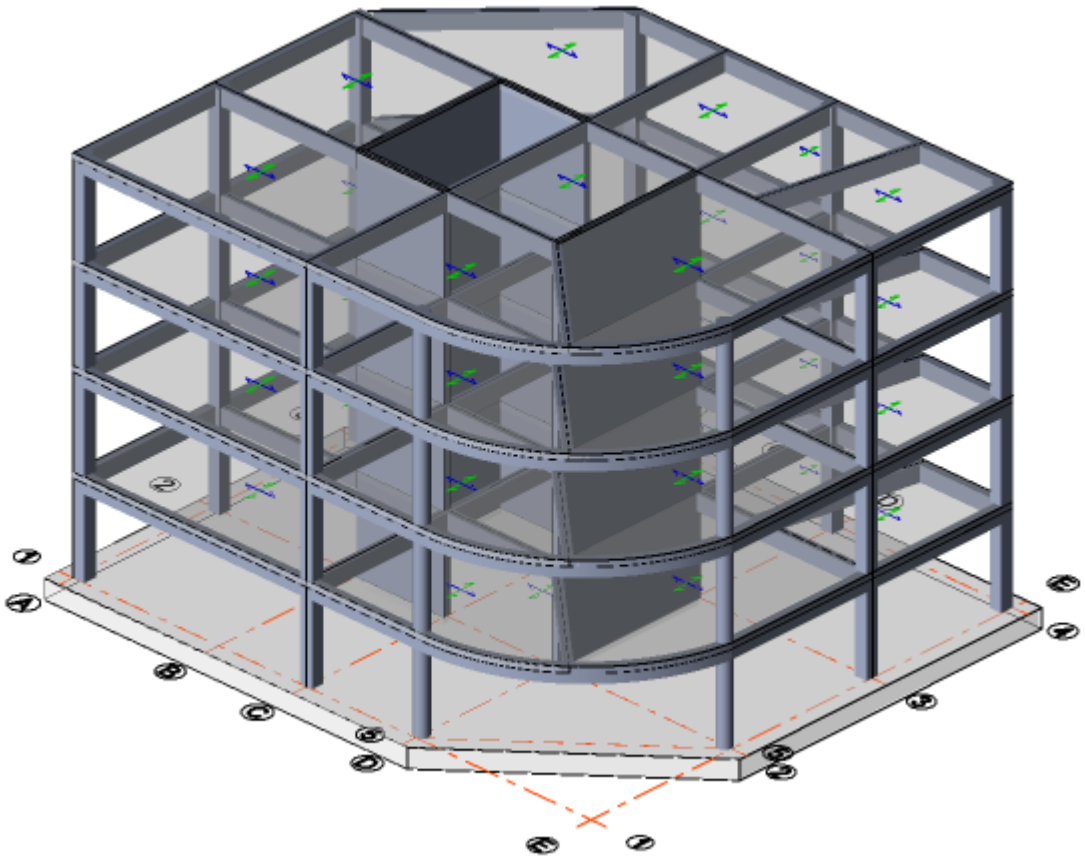


Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

3.16 Mat foundation design workflow (US customary units)

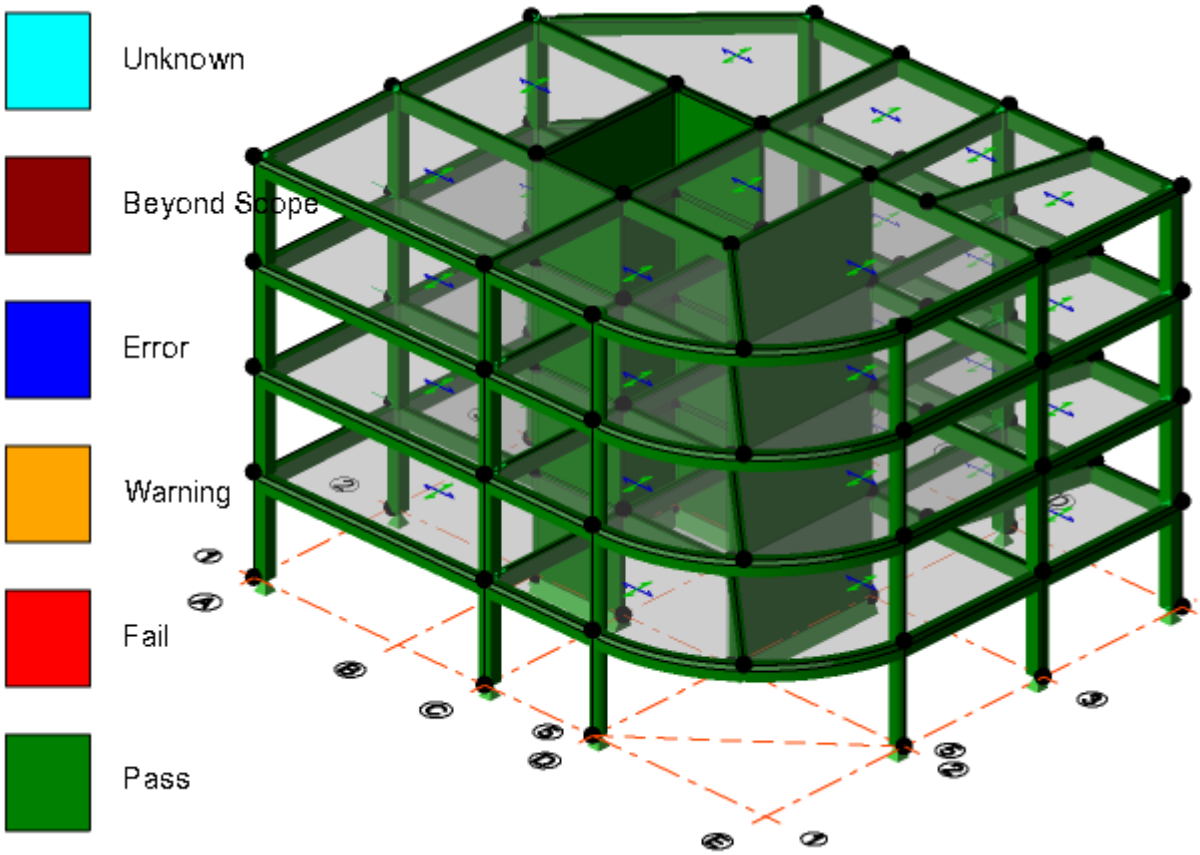
The small concrete building model shown below will be used to demonstrate the process for modeling and designing a mat foundation.



For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



Determine the soil parameters

Unless you are going to define discreet piled supports, the mat will need to be supported on ground bearing springs. These are activated by selecting **Use Ground Bearing Springs** (located under the **Slab parameters > Soil Parameters** sub-heading in the mat properties).

When these are activated you have then to also specify:

- Allowable bearing pressures
- Ground stiffness type (Linear, or non linear spring)
  - Linear ground stiffness, or,
  - Non-linear ground stiffness (+/-) and tension/compression limits
- Horizontal support

Allowable bearing pressures

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

Ground stiffness type

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear springs are applied:

- Allowable bearing pressures are checked
- Uplift (tension) is checked
- If no problems then linear springs are sufficient

When non-linear springs are applied:

- You can have compression only
- And also capped compression
- Either way analysis takes longer

NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Ground stiffness

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below[1] illustrates the potential range of values that might be considered.

Material	Lower Limit		Upper Limit	
	(kN/m^3)	**(kip/ft^2/ft) approx.**	**(kN/m^3)**	**(kip/ft^2/ft) approx.**
Loose Sand	4,800	31	16,000	102
Medium Dense Sand	9,600	61	80,000	509
Dense Sand	64,000	407	128,000	815
Clayey Medium Dense Sand	32,000	204	80,000	509
Silty Medium Dense Sand	24,000	153	48,000	306

Material	Lower Limit		Upper Limit	
	(kN/m^3)	(kip/ft^2/ft) approx.	(kN/m^3)	(kip/ft^2/ft) approx.
Clayey Soil (qa<200kPa)	12,000	76	24,000	153
Clayey Soil (200<qa<800kPa)	24,000	153	48,000	306
Clayey Soil (qa>800kPa)	48,000	306	200,000	1273

1. Reference: "Foundation Analysis and Design", Joseph E. Bowles, 1995 (Table 9-1)

Horizontal support

The degree of horizontal support provided by the ground springs can be set as:

- Fixed
- Free
- Spring (default 20% of vertical spring stiffness)

If set to "Free" a mechanism will result unless you provide additional discreet supports.

Determine the remaining mat properties

You are required to manually specify the **Reduce live loads by** mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an "area" method of mat creation you will also need to specify the amount of slab overhang.

The remaining properties are then similar to those used to define a typical concrete flat slab.

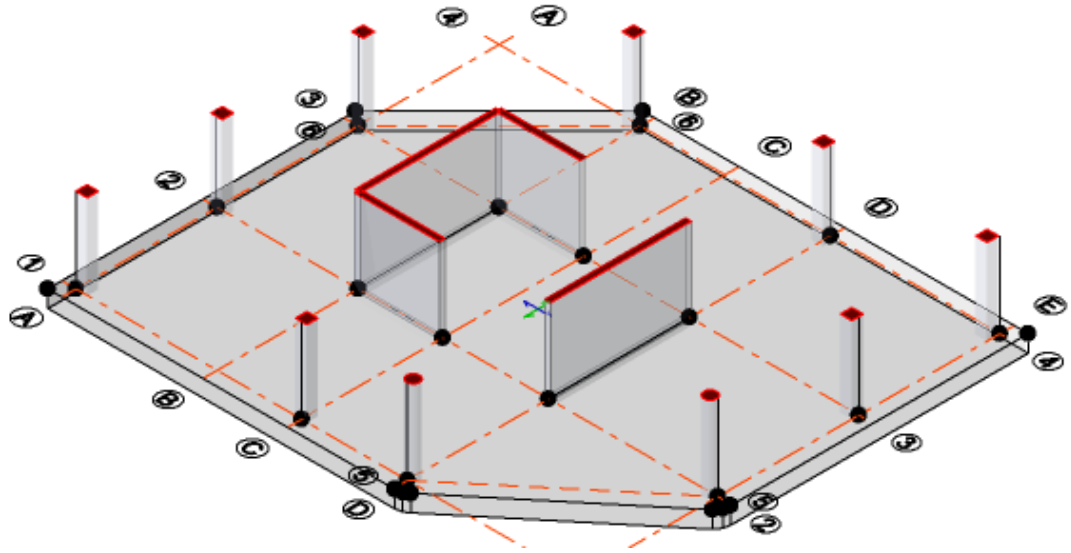
In this example the minimum area method is used to create a mat with the following properties:

- Live loads reduced by 30%
- Overhang, 20in.
- Mat thickness, 24in.
- Ground springs used
- Default allowable bearing pressures
- Default linear spring properties

Create the mat

To create the mat:

1. Choose the method, (e.g. Minimum Area)
2. Enter the mat properties, (see above)
3. Click on those columns (or walls) that define its perimeter,
4. Either press <Enter>, or re-click on one of the selected columns to complete the mat definition.



NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

Enable soil structure interaction

When not supported by a mat, columns and walls typically have supports at their bases.

When a mat is introduced these supports must be removed - as the mat now supports the whole building on ground bearing springs. Consequently adding a mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of "soil structure interaction" (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

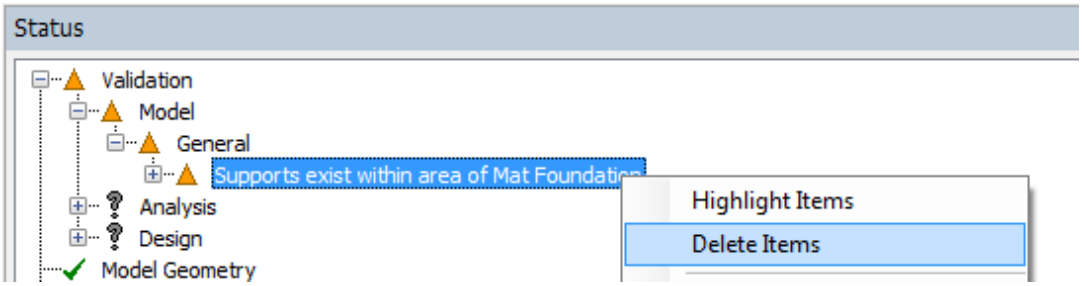
If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed from the Design ribbon by running **Design All (Static)** and any seismic RSA combinations by running **Design All (RSA)**. These processes will also recheck all the member designs taking account of the effects of soil structure interaction.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs supported on ground bearing springs.

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

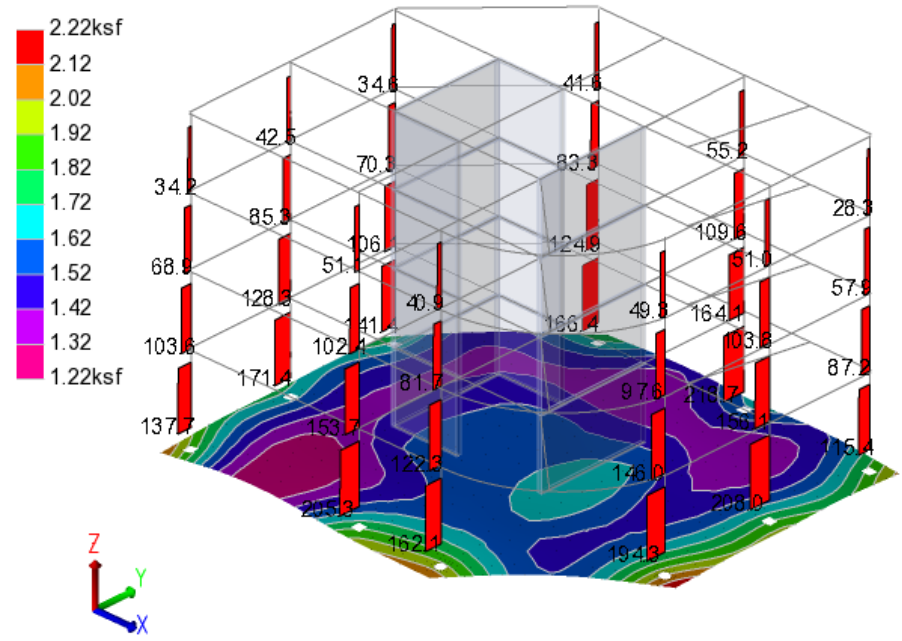
NOTE In a large model, if you anticipate several analysis might be required to determine the size of mat required, you might find it more efficient to just run

the analyses that are required from the Analyze ribbon then re-run the member design at a later stage.

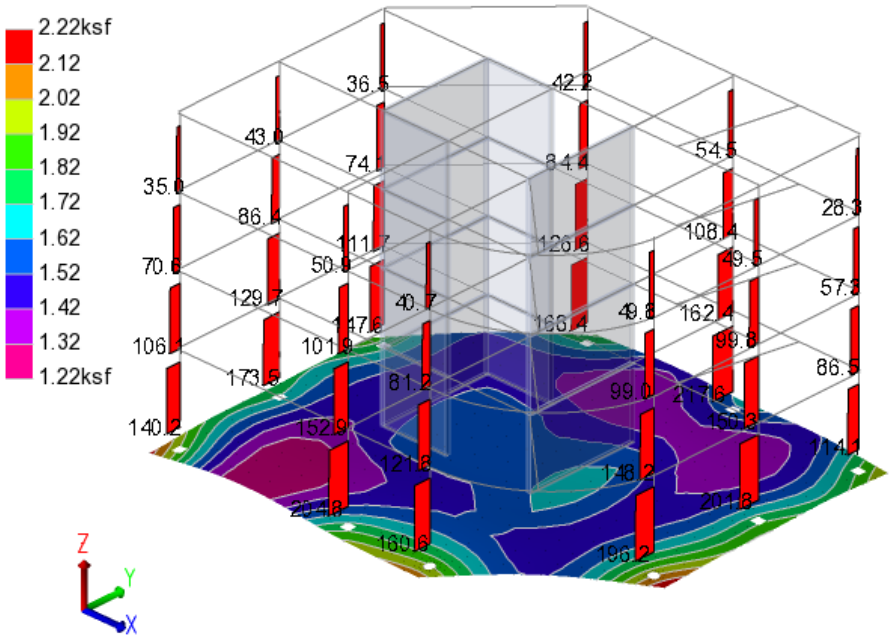
NOTE For compression-only supports (-ve stiffness set to zero) a loadcase of lateral load only will never solve for Non-linear analysis. What happens in the solver is what would happen in reality if Gravity suddenly turned off! Structures stabilized by gravity load would flip over in the presence of lateral load - i.e. no static equilibrium position exists to be found.

Check foundation bearing pressure and deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.



Solver Model used for 1st Order Linear



Solver Model used for Grillage Chasedown

By viewing the 2D deflection results for combinations based on "service" rather than "strength" factors the stiffness adjustments that you apply (via Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

For the above analysis Tekla Structural Designer default stiffness adjustment factors were used, the default factor for foundation mats being 0.2 for both ACI and EC2 design codes.

Re-perform member design

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be "Valid" or "Out of Date".

Unless the status is "Valid" you should recheck the member designs (taking into account the effects of soil structure interaction) by by running **Design All (Static)** from the Design ribbon.

NOTE Similarly if an RSA design has previously been performed, but is now out of date **Design All (RSA)** should be re-run.

If any concrete members now fail, you can switch them back into Autodesign mode, (ensuring Select bars starting from is set to Current rather than Minima) and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

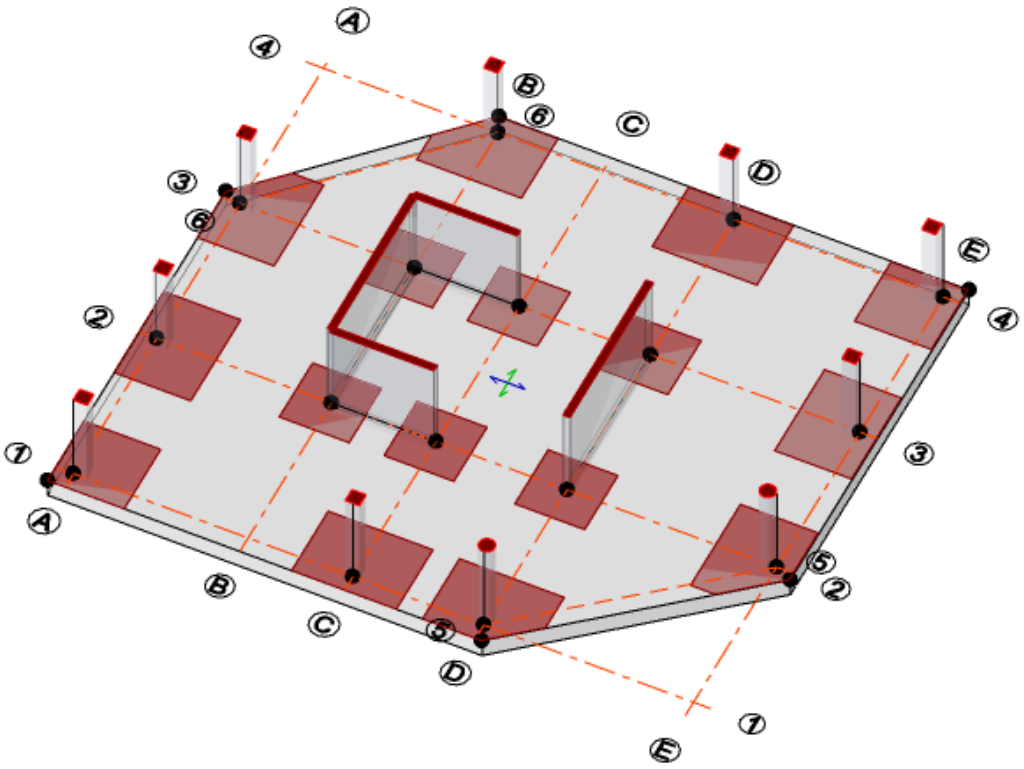
Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the "3D geometry" of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

NOTE When working in a 2D View use the right-click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered. Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add patches

This is an interactive process - requiring a certain amount of engineering judgment.



When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimization stage.

NOTE Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design mats

NOTE Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

- From the Foundations ribbon run **Design Mats** in order to design or check all the mats in the model (each according to their own autodesign setting), or,
- If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right-click and choose either **Design Slabs** or **Check Slabs**.

  NOTE When accessed from the right-click menu, these commands operate as follows:

  - **Design Slabs** will re-design slabs and mats (potentially picking new reinforcement) regardless of the current autodesign setting.
  - **Check Slabs** will check the current reinforcement in slabs and mats regardless of the current autodesign setting.

Review/optimize mat design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 3 size bars are selected and you would just never use anything less than a 4 size bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right-click option to design panels unless you really want to.)

NOTE Auto-design will select the smallest allowable bars at the minimum centers necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a "minimum spacing (slab auto design)" = 6".

After using the Review View update mode to standardize reinforcement you can then run Check Slabs from the right-click menu to check the revised reinforcement.

Remember:

- Consider swapping between Status and Ratio views - if utilizations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

Design patches

Having established and rationalized the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

- From the Foundations ribbon run **Design Patches** in order to design or check all the foundation patches in the model - by default newly created patches will all be in auto-design mode - so reinforcement is selected automatically, or,
- In the 2D View of the level which you want to design right-click and choose either **Design Patches** or **Check Patches**. Working in this way restricts the design or checking to the patches in the current view.

  NOTE When accessed from the right-click menu, these commands operate as follows:

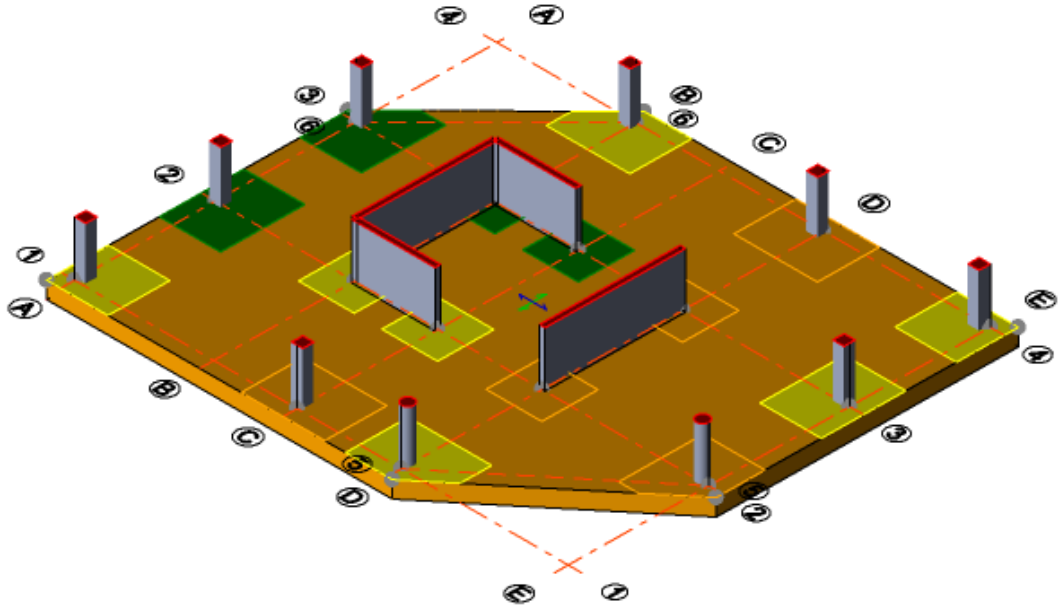
  - **Design Patches** will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.

- **Check Patches** will check the current reinforcement in the patches regardless of the current autodesign setting.

Review/optimize patch design

At this stage the patch sizes can be reviewed:

- Wall patches - can the width be adjusted (minimized)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.
- Now click Slab Reinforcement in the Review View to review and standardize the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has size 4 at 8in, then in the patch don't add size 4 at 5in, swap to size 5 at 8in - then there will be alternate bars at 4in crs in this strip of the patch.

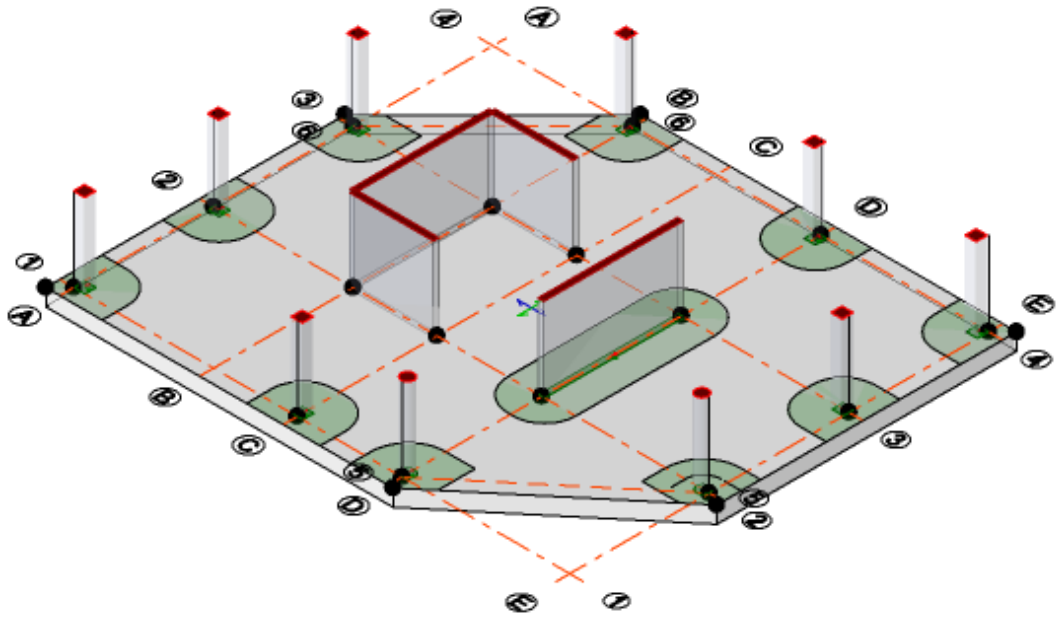


Add and run punching checks

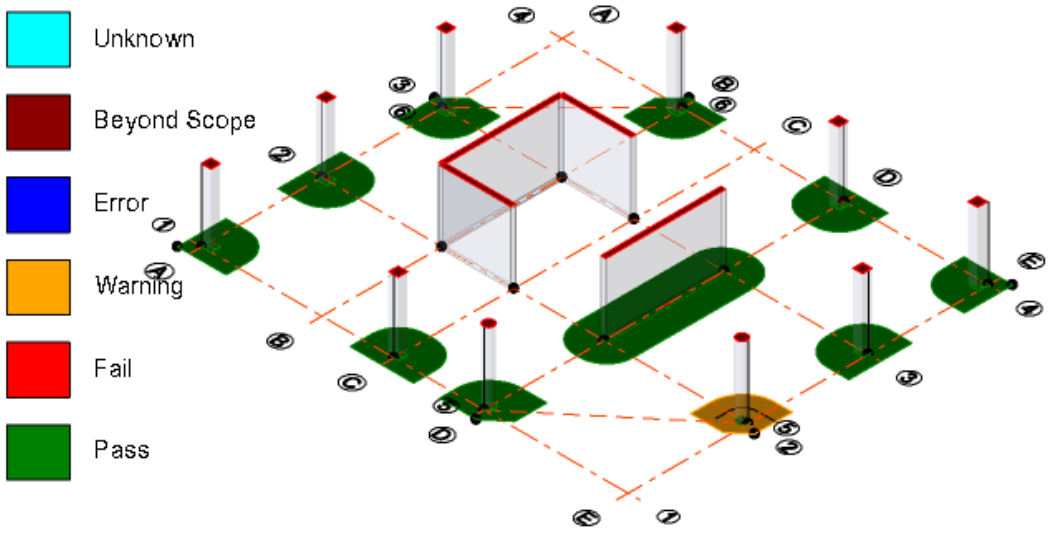
Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.



Once added click **Design Punching Shear**. See:



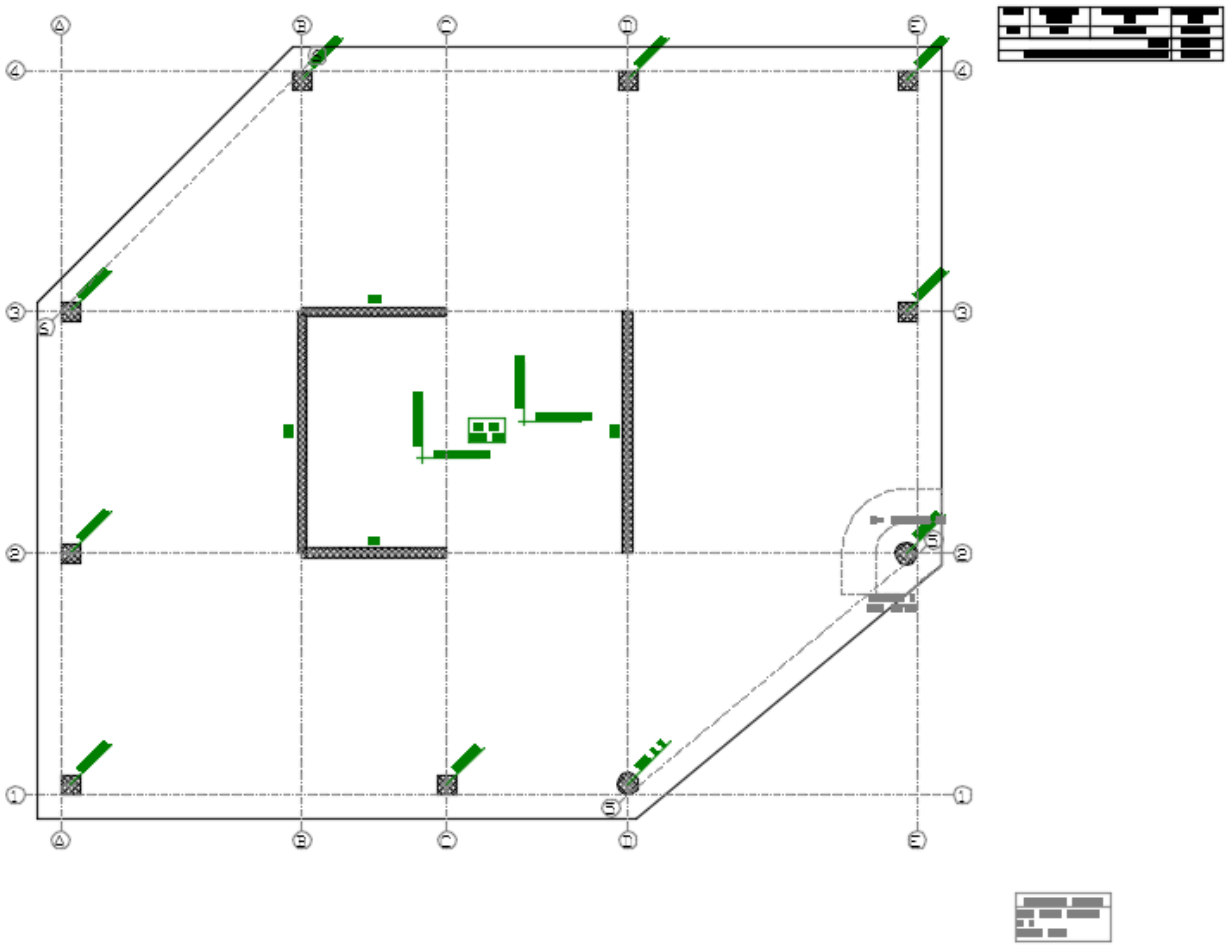
The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet

- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Create drawings and quantity estimations

Drawings that convey the structural intent are easy to create. It should be borne in mind that these are NOT the final detail drawings, their purpose is to eliminate the need for manual mark-up drawings as a means of communication between the engineer and the detailer



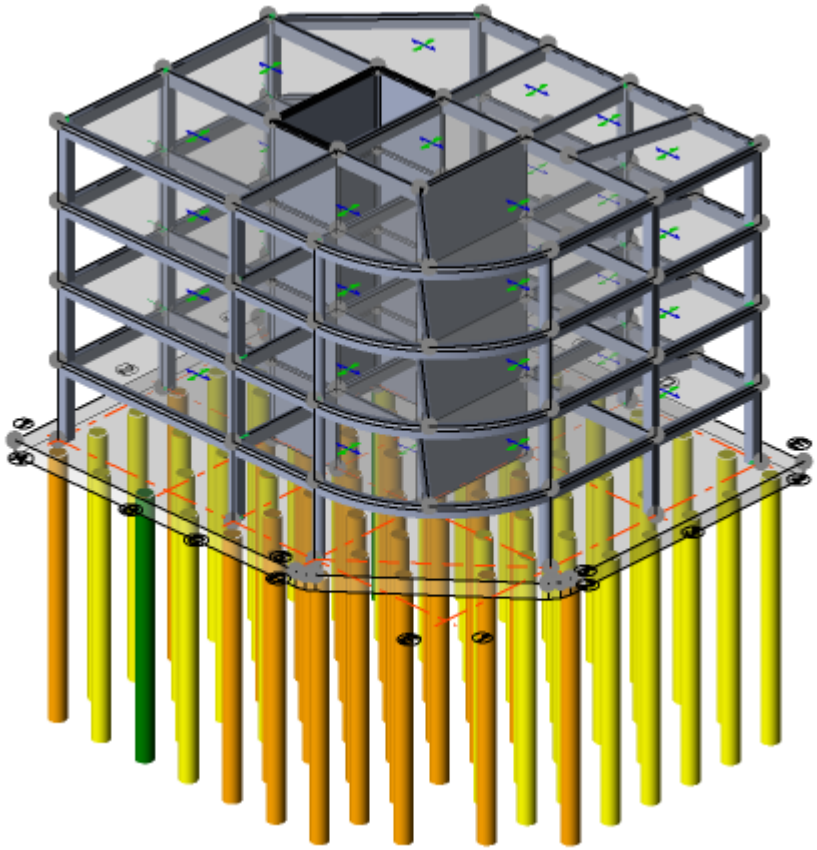
Print calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report

includes these along with design calculations for other member types in the model).

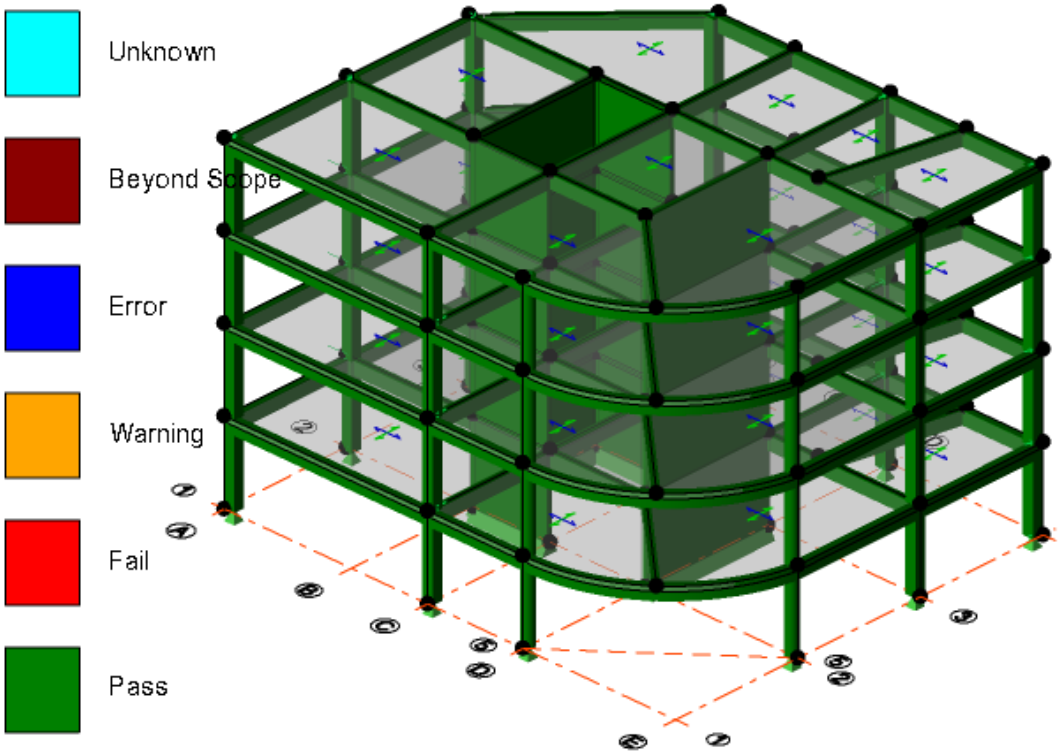
3.17 Piled mat foundation design workflow (US customary units)

The following example illustrates the typical process to model and design piles in a piled mat foundation.



Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to "check and increase" the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from current.)

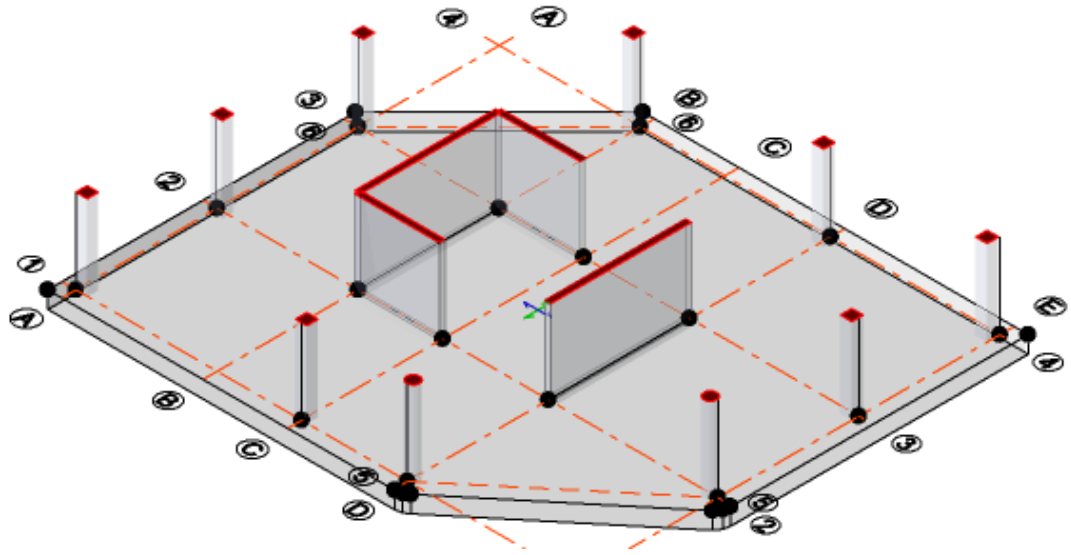
Create the mat

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally clear Use Ground Bearing Springs (under Soil Parameters in the mat properties.)

NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- An overhang of 40in.
- Mat thickness 24in.
- The **Use Ground Bearing Springs** option cleared



Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

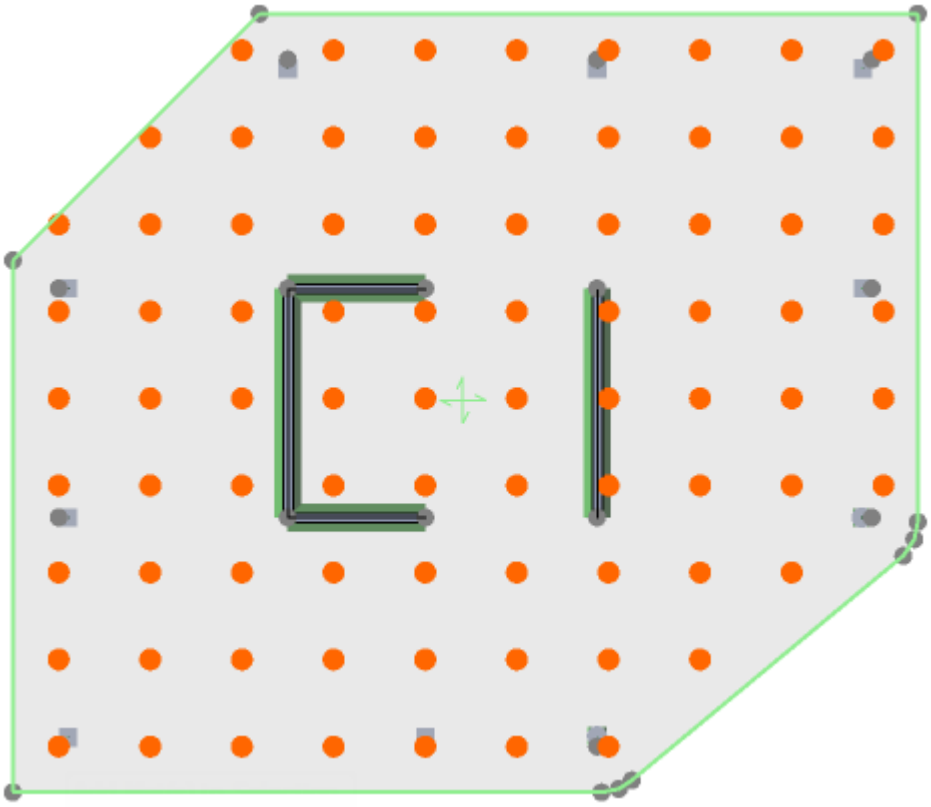
In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as Fixed.

Add piles to the mat

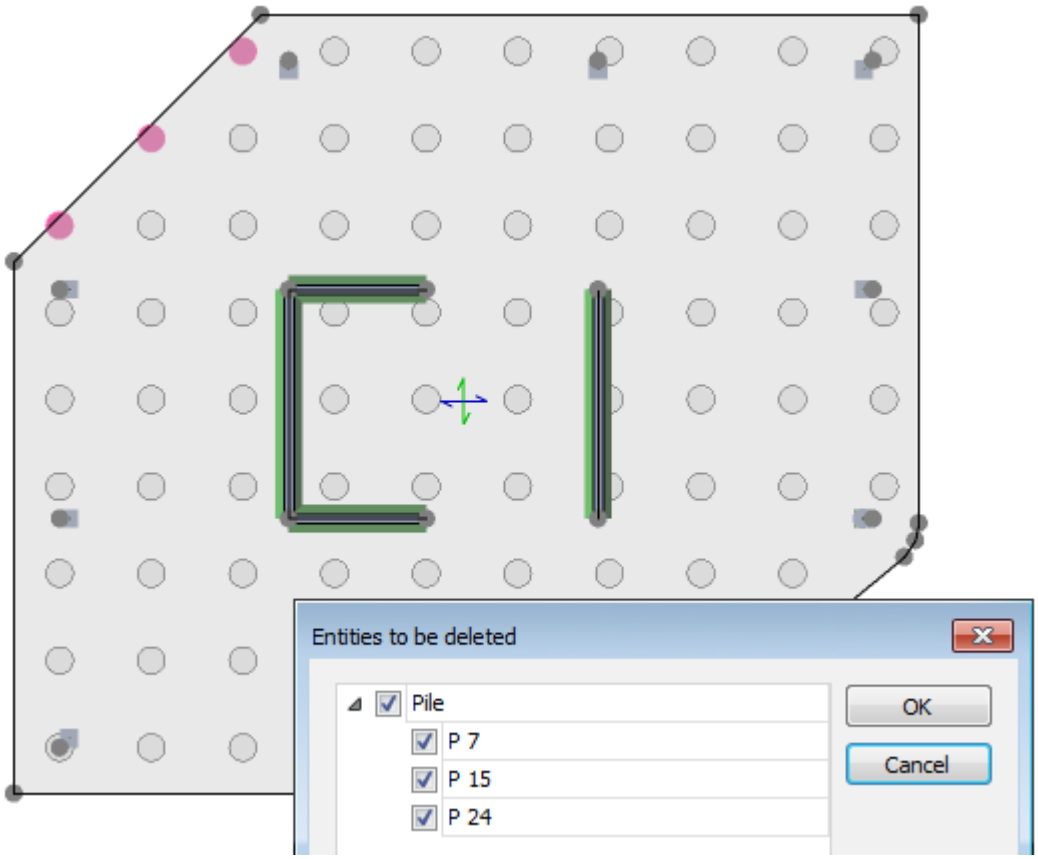
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.



After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.



Remove existing column and wall supports

When not supported by a mat, columns and walls typically have supports at their bases.

When a piled mat is introduced these supports must be removed - as the mat now supports the whole building on pile springs. Consequently adding a piled mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of support settlement. In the past this has often been ignored, even though design codes suggest that it should be considered.

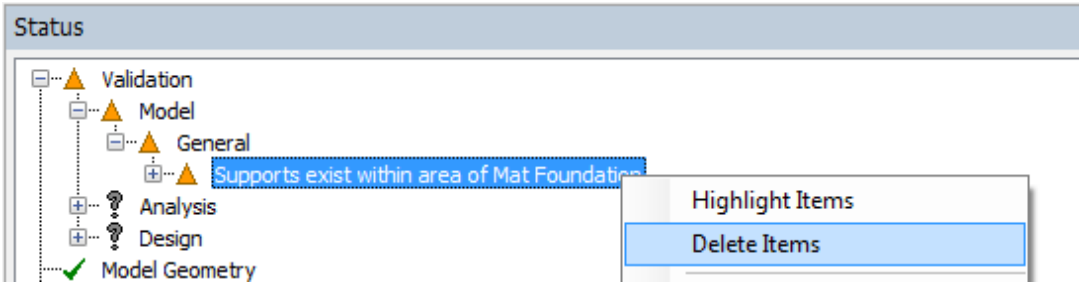
Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Analyze All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, (or a combination of both).

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

NOTE Analyze All is run in preference to Design All at this stage because member design is influenced by, and should therefore follow after the piled raft design.

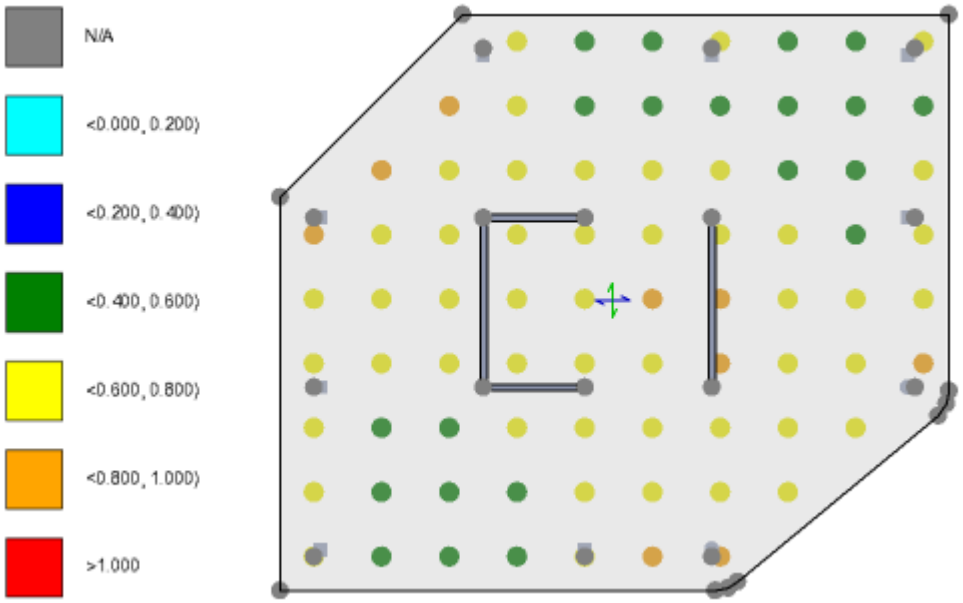
Perform the pile design

The piles are checked (and the mat is designed) by running **Design Mats** from the Foundations ribbon.

NOTE The pile types/sizes are not changed during this process.

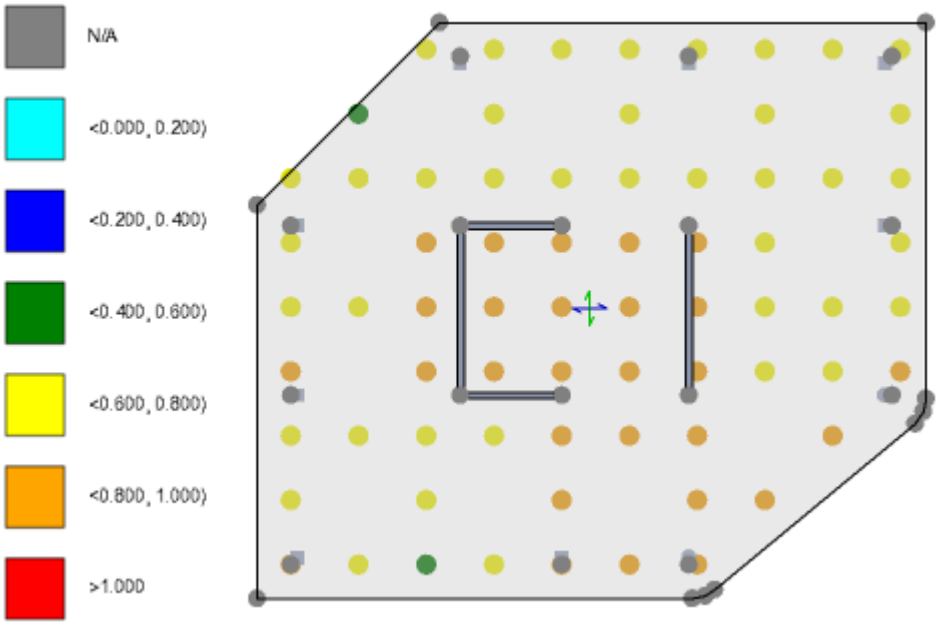
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the Review View in order to determine if any remodeling of piles is required.



In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run Analyze All followed by Design Mats once more.



At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.

Design results can be examined in more detail by selecting Tabular Data from the ribbon and setting the View Type to Design Summary. By selecting Piles as the Characteristic (and the appropriate material) the full results for each pile can be examined.

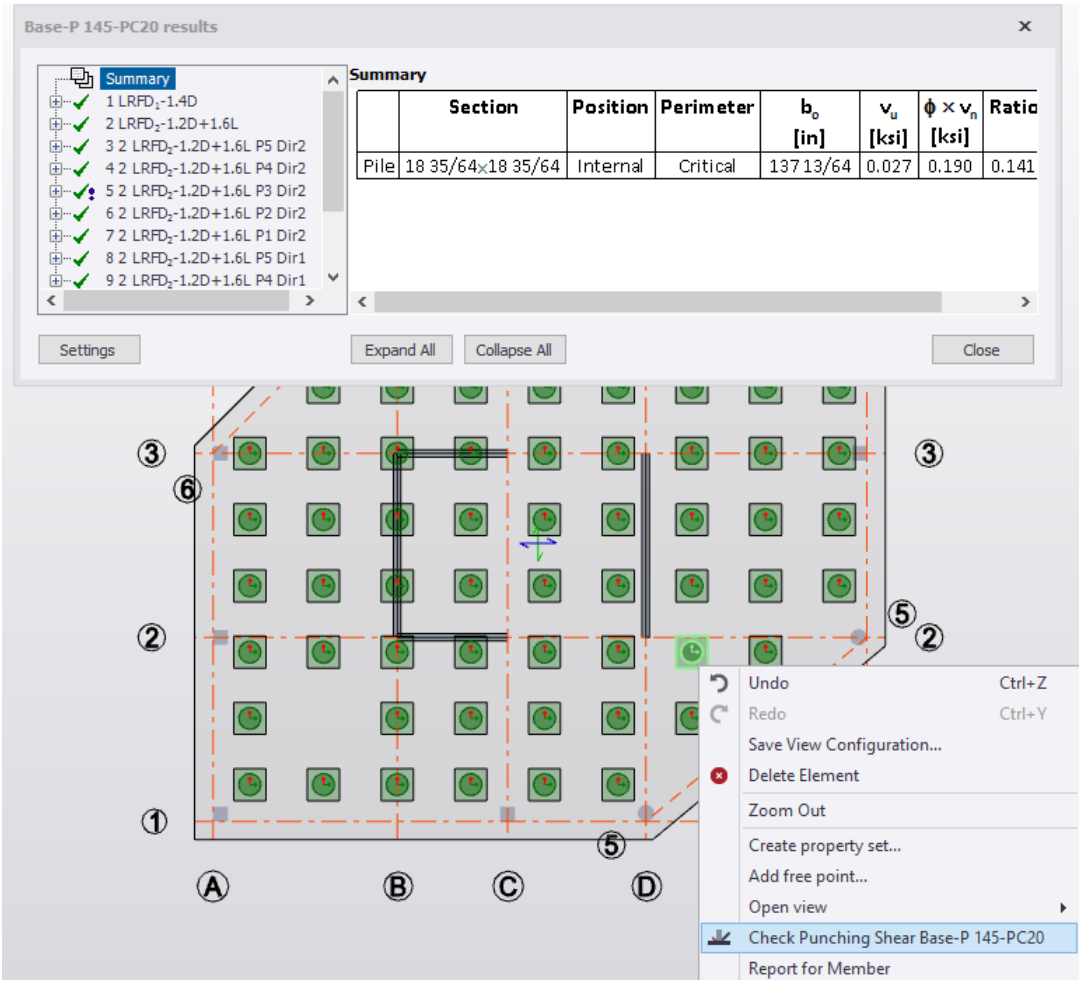
Add and run pile punching checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

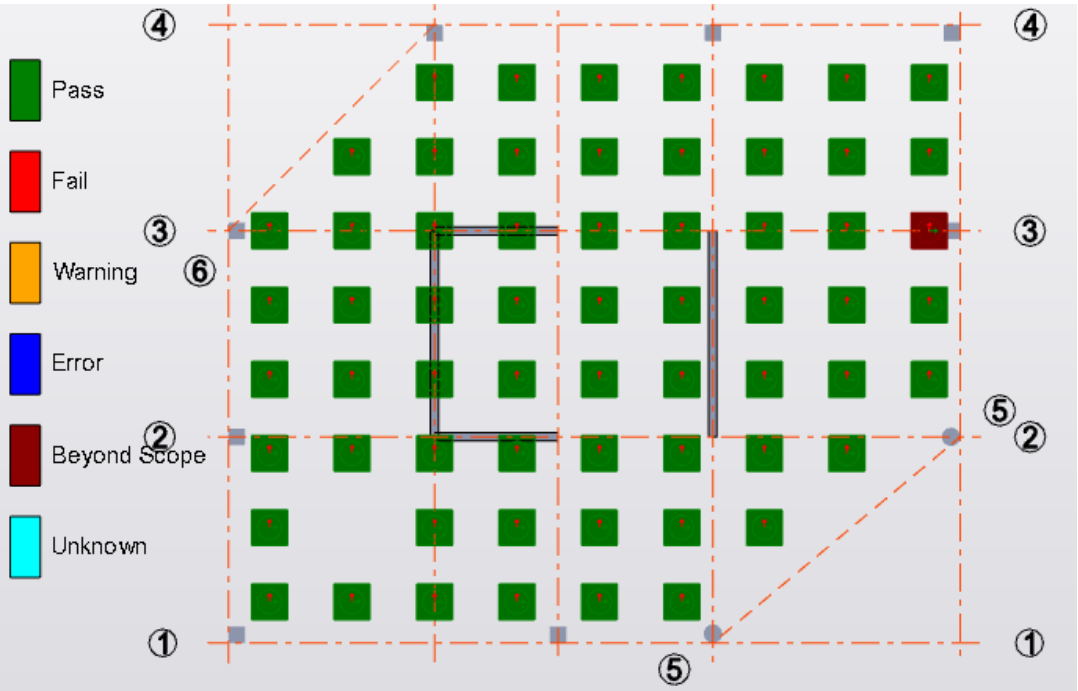
Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.

Once added you can then design and check them individually, if required.



Or you can run all the checks in one go from the Foundations ribbon, by clicking **Design Punching Shear**. See:



The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed
- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Key aspects of the check performed are:

- Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
- The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.
- The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).
- Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.

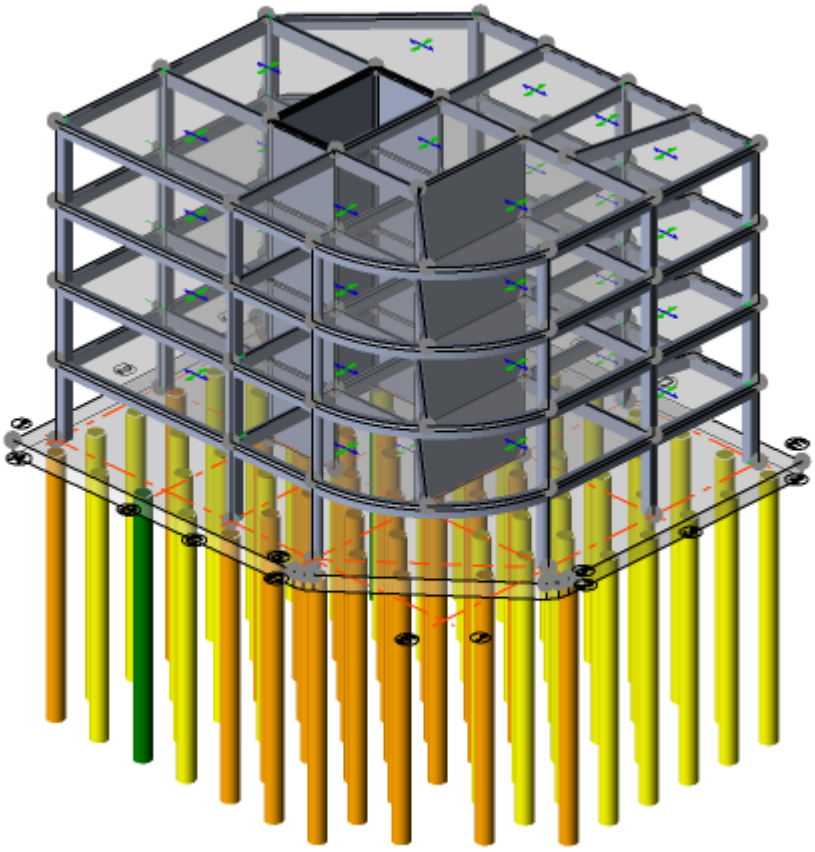
- There is an additional pile-specific Design setting to use the pile capacity in the punching check in **Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles** (default Off).

Perform the mat design

The Design of the mat itself is described in the Mat foundation design workflow (US customary units) (page 311) topic.

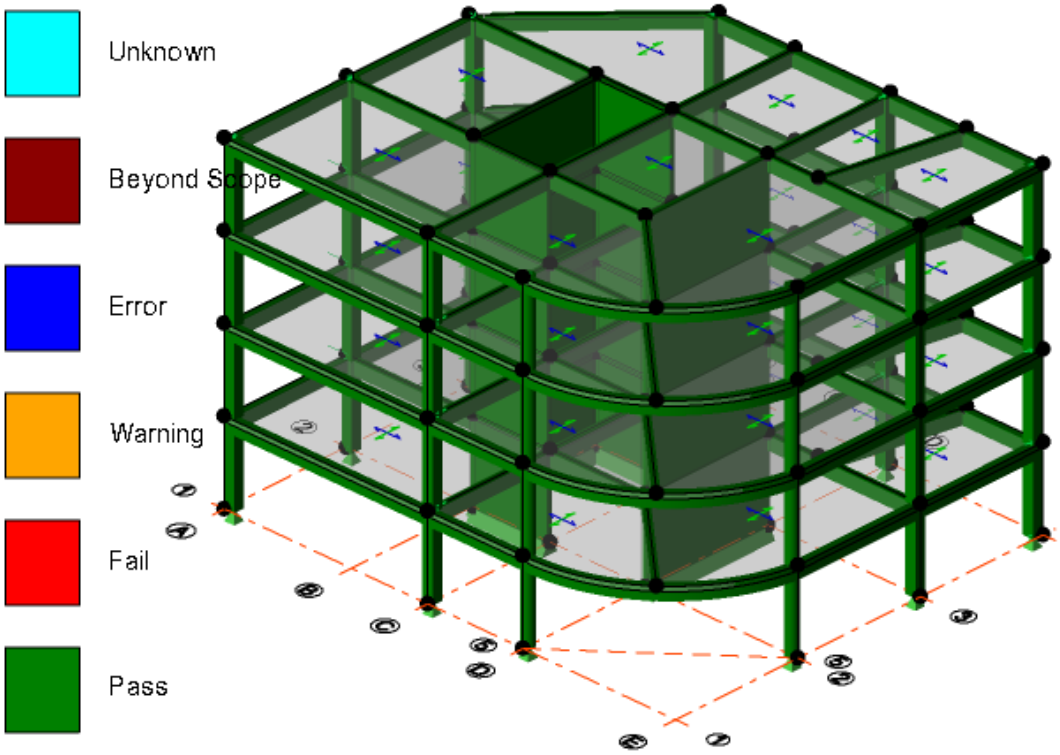
3.18 Piled mat foundation design workflow (metric units)

The following example illustrates the typical process to model and design piles in a piled mat foundation.



Design the structure before supporting it on the mat

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.



In order to retain the existing reinforcement design all members should be set to check mode. Alternatively you might choose to "check and increase" the reinforcement instead, (by leaving members in autodesign mode with the option to select bars starting from current.)

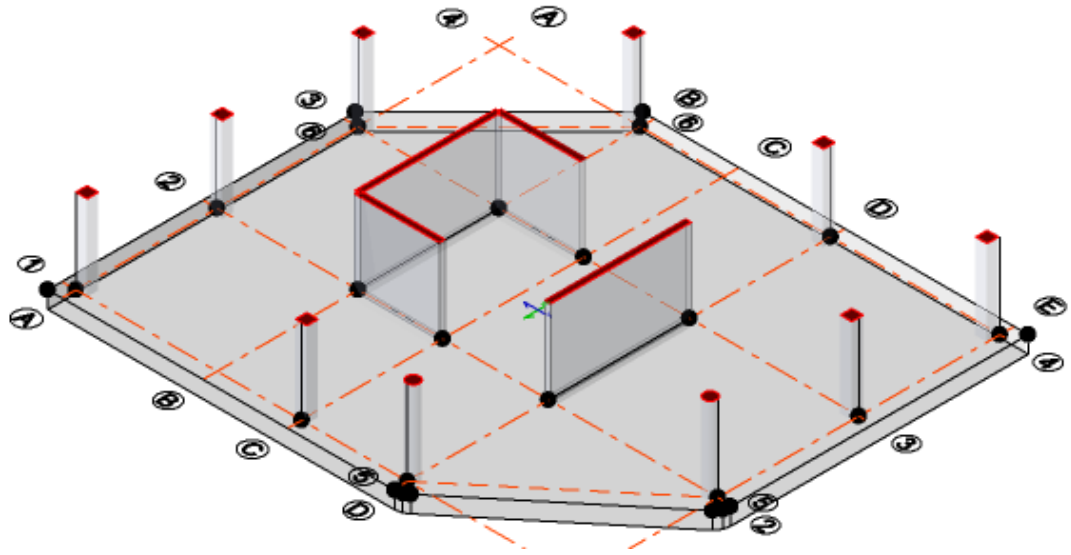
Create the mat

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally clear Use Ground Bearing Springs (under Soil Parameters in the mat properties.)

NOTE The "Mesh 2-way slabs in 3D Analysis" option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- An overhang of 1.0m
- Mat thickness 600mm
- The **Use Ground Bearing Springs** option cleared



Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

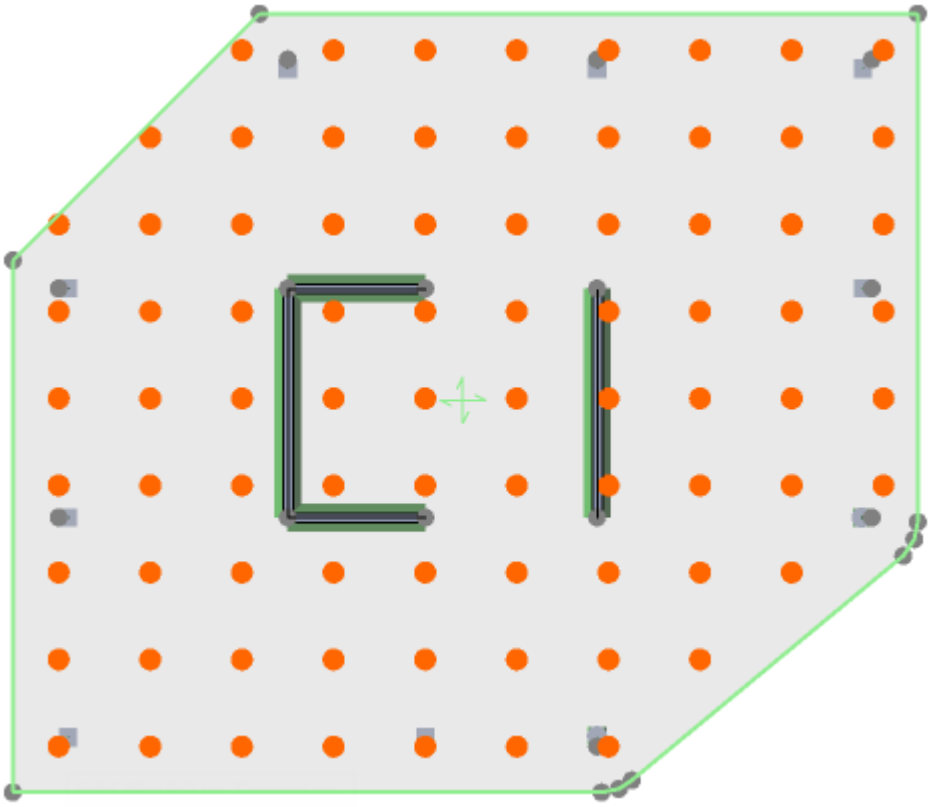
In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as Fixed.

Add piles to the mat

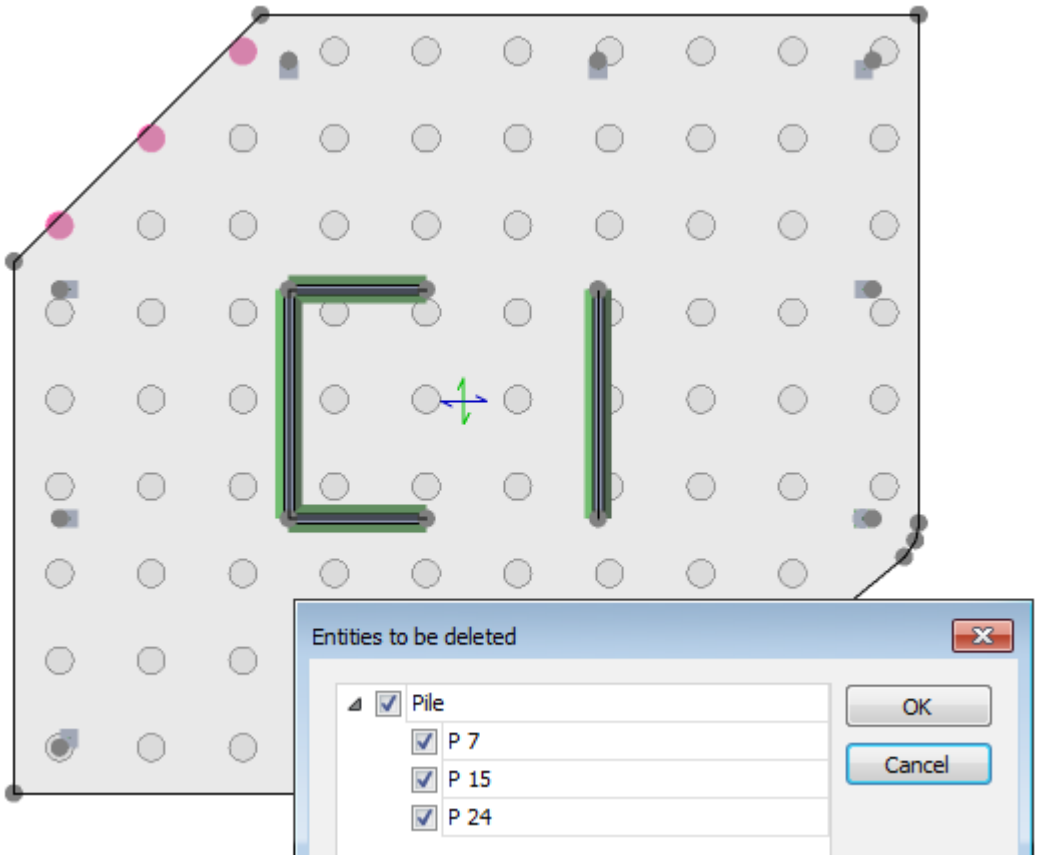
Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.



After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.



Remove existing column and wall supports

When not supported by a mat, columns and walls typically have supports at their bases.

When a piled mat is introduced these supports must be removed - as the mat now supports the whole building on pile springs. Consequently adding a piled mat means re-analysis and hence re-design of the whole building.

NOTE A simple technique for detecting and removing the supports is described in the next section "Model Validation".

Inherent in the re-design is the inclusion of support settlement. In the past this has often been ignored, even though design codes suggest that it should be considered.

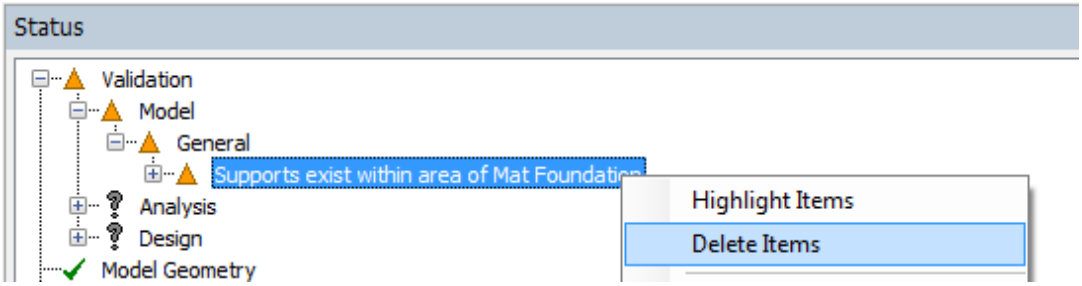
Note however that soil structure interaction only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A "Supports exist within area of Mat Foundation" warning is issued if member supports conflict with ground springs. (This can be remedied by right-clicking on the warning and choosing Delete Items).



Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analyzed by running Analyze All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

In each of the 3D, FE chasedown, and grillage chasedown analyses that get performed mats are modeled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, (or a combination of both).

In the FE chasedown and grillage chasedown models the mat and first level above the mat are always combined in a single sub-model.

NOTE Analyze All is run in preference to Design All at this stage because member design is influenced by, and should therefore follow after the piled raft design.

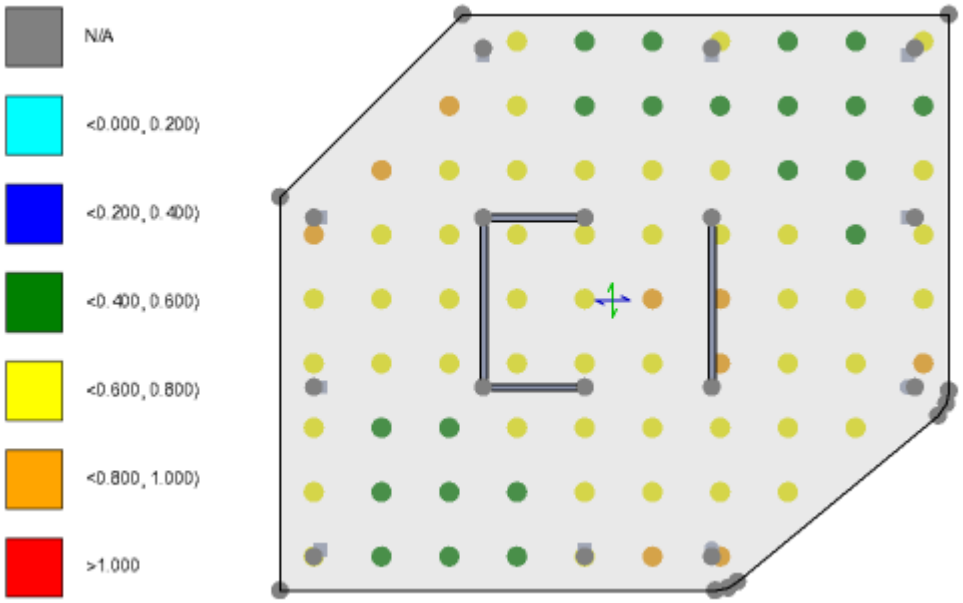
Perform the pile design

The piles are checked (and the mat is designed) by running **Design Mats** from the Foundations ribbon.

NOTE The pile types/sizes are not changed during this process.

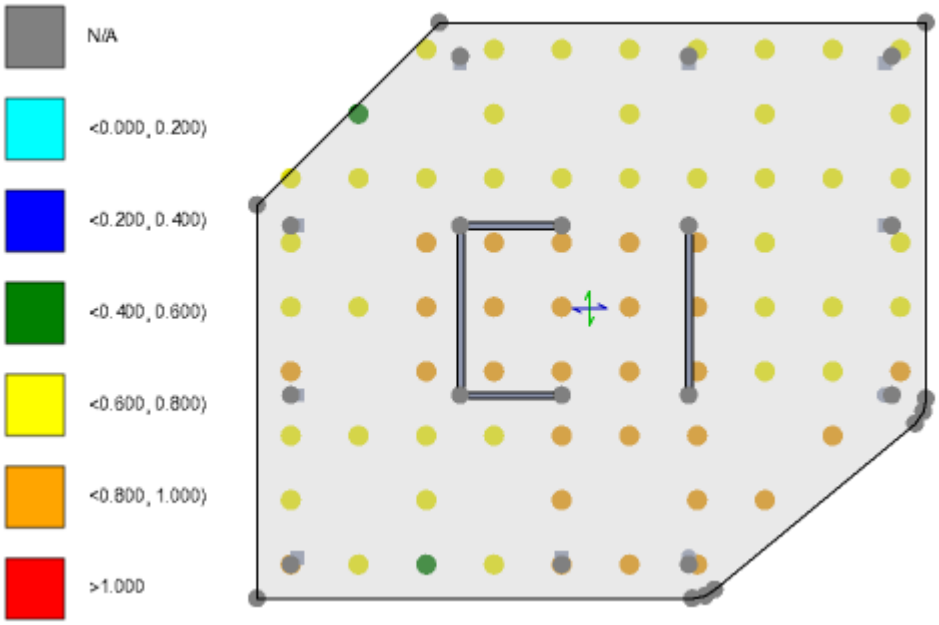
Review the pile design status and ratios

You can display the Pile Status and Pile Ratios from the Review View in order to determine if any remodeling of piles is required.



In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run Analyse All followed by Design Mats once more.



At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.

Design results can be examined in more detail by selecting Tabular Data from the ribbon and setting the View Type to Design Summary. By selecting Piles as the Characteristic (and the appropriate material) the full results for each pile can be examined.

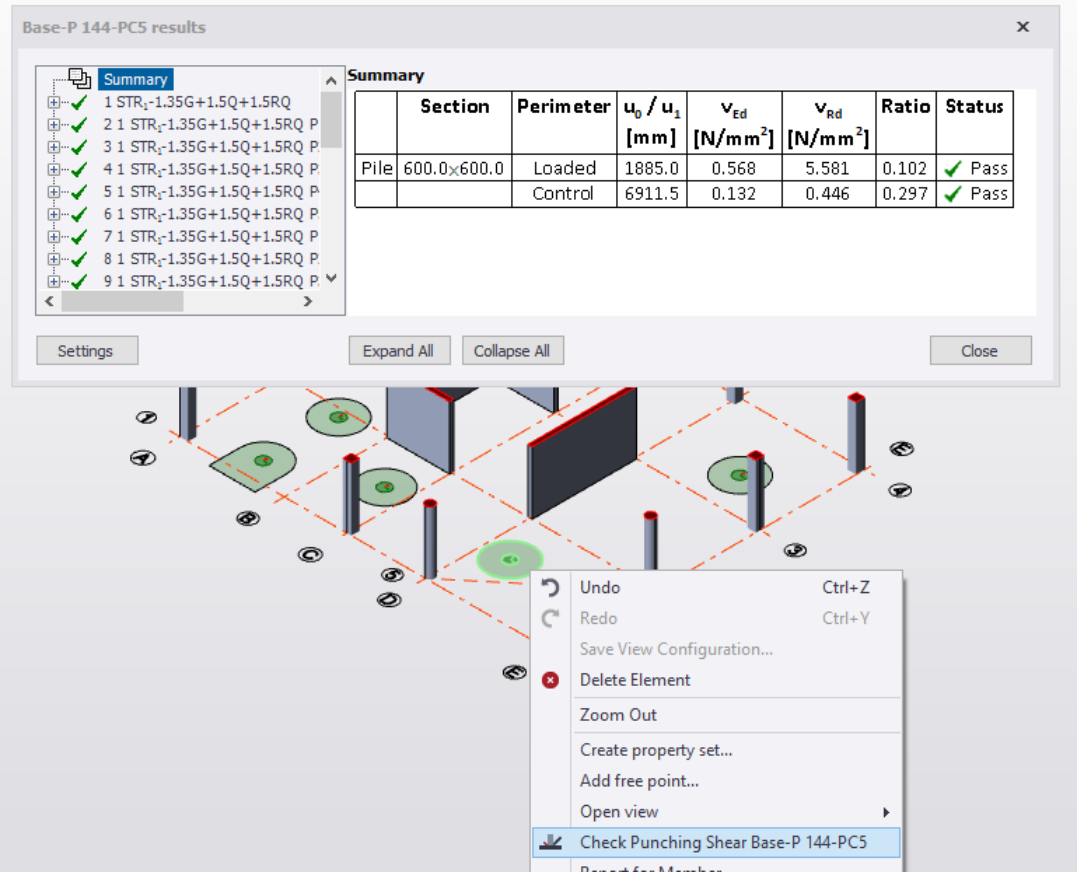
Add and run pile punching checks

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

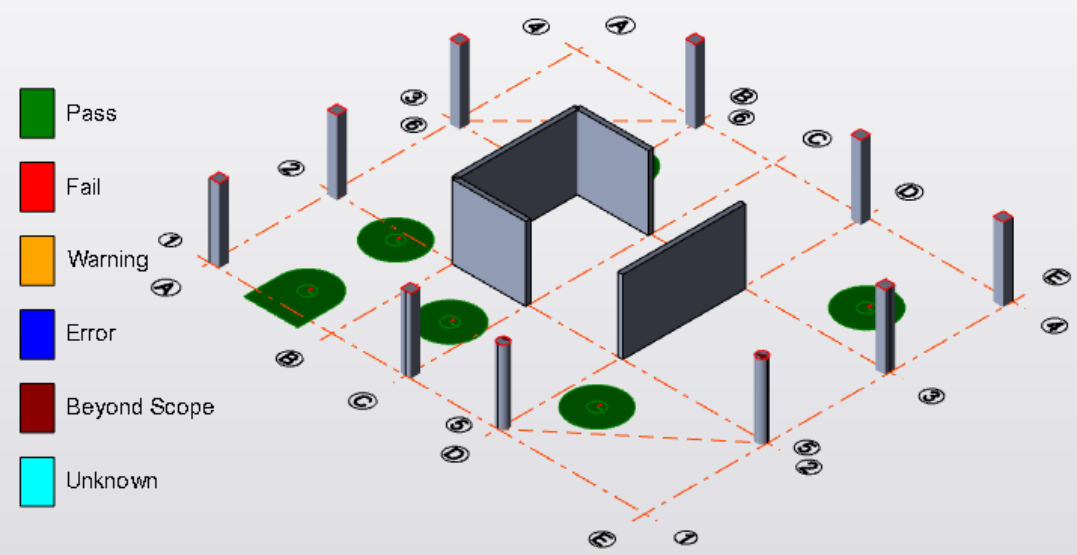
Punching checks can be added individually, or by windowing over an area. See:

You can then select any check and review the properties assigned to it.

Once added you can then design and check them individually, if required.



Or you can run all the checks in one go from the Foundations ribbon, by clicking **Design Punching Shear**. See:



The checks are done and status is shown as:

- Pass - if no shear reinforcement is needed
- Warning - if shear reinforcement is needed

- Fail - if it is impossible to achieve required capacity by adding share reinforcement
- Unknown - if check not run yet
- Beyond scope or error - if for example the centroid of the column/wall lies outside the mat

Key aspects of the check performed are:

- Punching shear resistance is assumed to be provided by the concrete alone - there is no option to add specific punching shear reinforcement in the form of studs and rails (as there is for column punching checks, including those supported by mats).
- The check considers 3D Building Analysis, FE Chase-Down and Grillage Chase-Down results for all active gravity, wind, seismic and RSA load combinations.
- The check considers only a single pile - not a pair - and only vertical shear not moment (as piles are modeled a pinned spring supports without moment fixity).
- Just as for punching checks of columns supported by piled mats, all loading and reactions (from ground bearing springs) within the punching perimeter are considered.
- There is an additional pile-specific Design setting to use the pile capacity in the punching check in **Design Settings > Concrete > Cast-in-place > Foundations > Mat Foundations > Piles** (default Off).

Perform the mat design

The Design of the mat itself is described in the Mat foundation design workflow (metric units) (page 297) topic.

4 Precast member design handbook

Precast structures can be modeled and analyzed in Tekla Structural Designer. Precast beams and columns can then be designed (to Eurocodes only), provided that you have access to a license of Tekla Tedds.

The following topics are covered in this handbook:

- Precast member design limitations and assumptions (page 346)
- Precast member design workflow (page 347)
- Precast member design groups (page 355)
- Precast beam design (page 357)
- Precast column design (page 371)
- Precast column connection eccentricity moments (page 375)

Precast modeling, analysis and design using Tekla Tedds

4.1 Precast member design limitations and assumptions

The following limitations and assumptions apply to the design of Precast beams and columns in Tekla Structural Designer:

- A compatible version of Tekla Tedds must be installed and licensed.
- The locale must be set correctly in Tekla Tedds before attempting to run the design from Tekla Structural Designer.
- Only design of normal weight precast concrete beams and columns to Eurocodes is supported.
- Design of precast concrete slabs and walls is beyond scope for all regional codes.

- The process is only applicable to precast structures that don't make use of in-situ structural toppings.
- Precast construction which adopts a more hybrid construction involving the use of in-situ toppings exceeds the limitations of the Tekla Tedds calculations and should therefore be considered beyond scope.

NOTE For further information relating to the scope, assumptions and limitations, general notes and references specific to the Tekla Tedds calculation itself, please refer to the respective calculation documentation in the Tedds calculation library.

4.2 Precast member design workflow

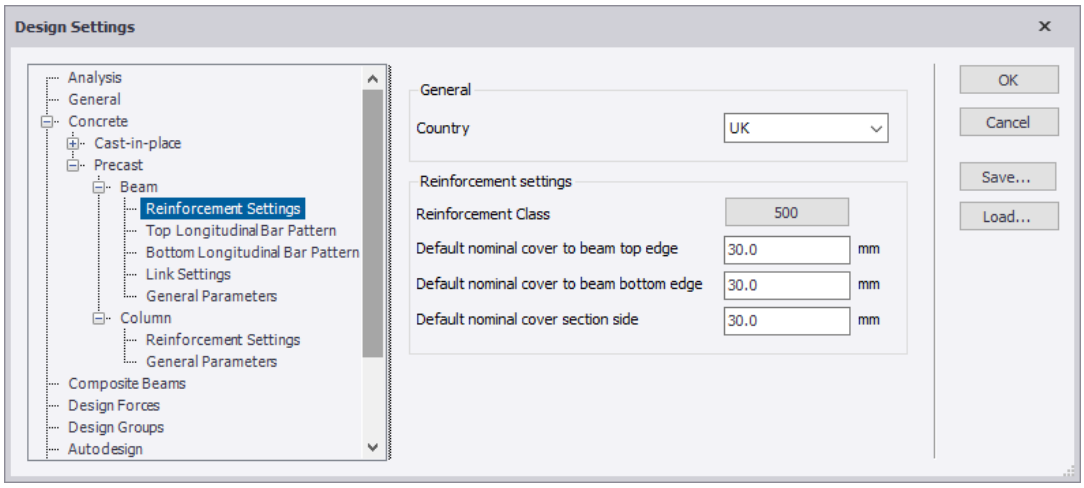
The basic workflow for precast design in Tekla Structural Designer is described in the sections below.

Precast modeling, analysis and design using Tekla Tedds

Configure precast beam and column design settings

Specific precast beam design settings and precast column design settings that apply to all the beams and columns in the model should be specified prior to running the designs.

These are set on the respective **Concrete>Precast** pages of the **Design Settings** dialog.

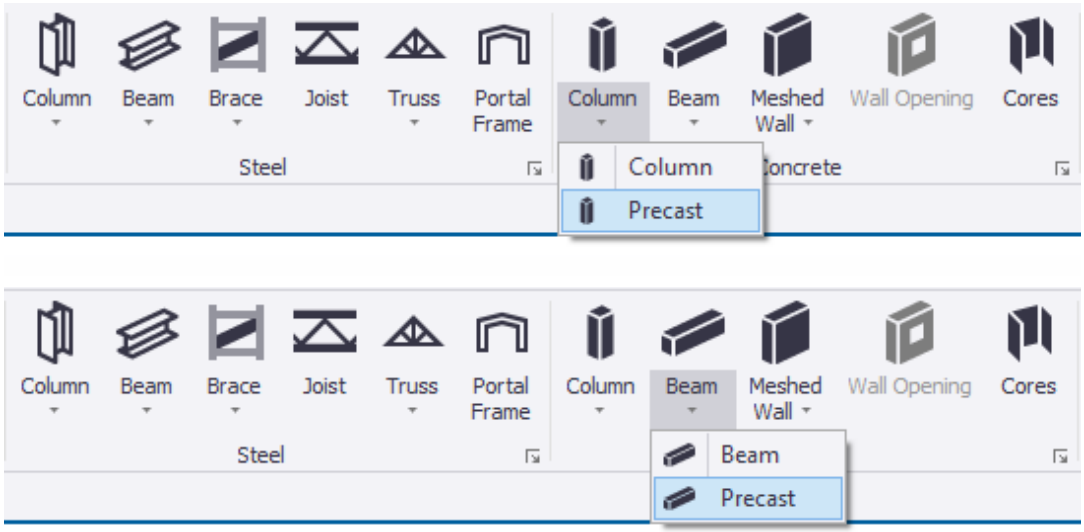


By ensuring the defaults are set correctly you can avoid having to manually set the values in each Tedds precast calculation as it is run.

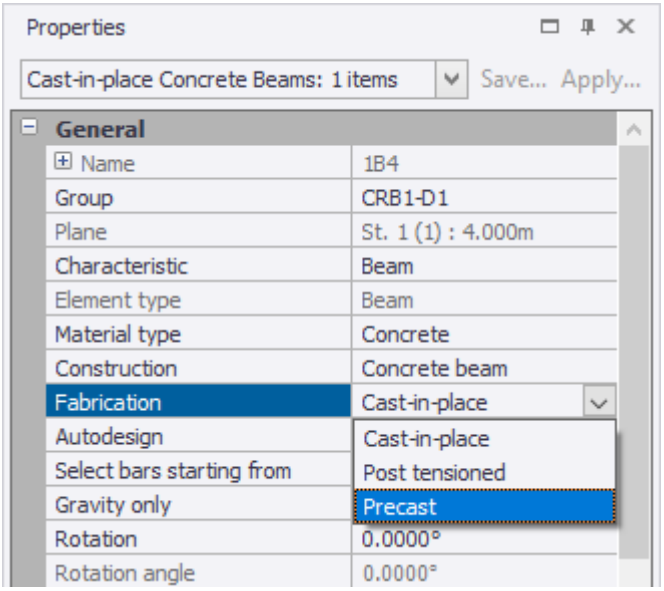
You still have control within individual calculations to adjust these settings on a member by member or group basis.

Define and place precast members

To place precast members, simply use the **Concrete Column > Precast**, or **Concrete Beam > Precast** commands on the Model tab.

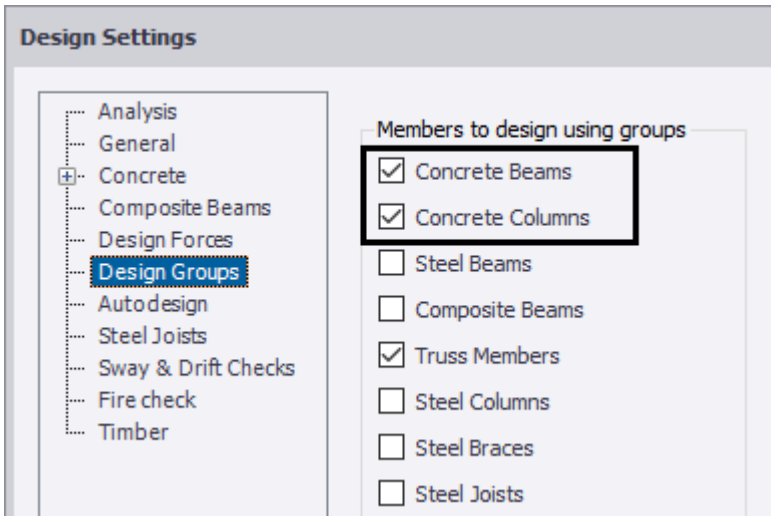


An alternative to this would be to use the **Concrete Column > Column**, or **Concrete Beam > Beam** commands and to subsequently change the Fabrication parameter to be Precast.



Configure precast groups

Customizable member groups are created automatically. You can choose whether to utilize them for design purposes via the **Design Groups** page of the **Design Settings** dialog.



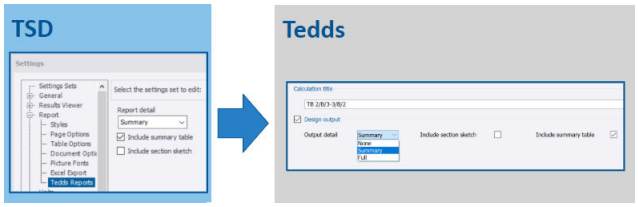
Using precast member design groups (page 355) can help speed the overall design process, particularly so in the case of medium to large size models.

- An envelope of group design forces is passed to a single Tedds calculation.
- User can manipulate design options, member size, material etc.
- Any changes are applied to each member in group.
- Each individual member of group is checked against individual loads without any further user interaction.

The initial precast member design groups can be reviewed from the **Groups** tab of the **Project Workspace** - you can move members into new groups as required.

Set the Tedds results output level

You can choose the output level for the Tedds precast calculations in advance by clicking **Home > Settings > Report > Tedds Reports**



By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds precast calculation as it is run.

The setting applies to all Tedds linked calculations (precast and timber).

Establish design forces by running the analysis

Precast members can only be designed provided a set of analysis results exist for the appropriate load combinations. These results can be generated from the **Analyze** ribbon by running .

Once load combinations have been analyzed **Design using Tekla Tedds** becomes available. When this is run (as described in the next section), a design force envelope is established and all critical load combinations are considered in a single Tedds calculation.

Design using Tekla Tedds

NOTE We recommend that the Tekla Structural Designer model is developed as much as possible prior to considering the design of the precast members. This should ensure that the correct distribution of forces is carried through into the Tekla Tedds calculations.

Design a Selection

To design several precast members or groups in one go:

1. Select the precast members you want to design.
2. Right click and select Design using Tekla Tedds> Selection
3. Design the selected members.

   NOTE If grouped design is active, a single grouped design is performed for each group included in the selection using critical design forces established from all members in the group (irrespective of whether or not they were included in the selection). At the end of the process all members in each designed group are checked, (irrespective of whether or not they were included in the selection).

4. When changes are made to the model, you can check if the existing sections are still sufficient by running
5. Check In Tedds from the **Design** tab.

   NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you

want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data from the right-click context menu.

Design a group

If you have activated concrete member design groups, each group can be designed as follows:

1. Highlight any member in the group, right click and select Design using Tekla Tedds> Group

   This design gathers the analysis results for all members in the group and collates them in to one design. (In effect assuming that all the worst case loads are happening on one member simultaneously.)

2. Design the selected member for these loads.
3. Click **Finish**.

   If the section size was changed during the design, all members of the group will be updated to the new section. If the reinforcement data was changed this is also copied to all group members.

   Irrespective of whether the section or reinforcement has been changed or not, all members in the group are then automatically checked against only the loads that they see individually. A pass fail status and utilization ratio is calculated accordingly for each one.

4. In the Review View, review the utilization ratios for all members in the group - if these all indicate an efficient design, the process can stop at this point.

   NOTE For columns in particular, the envelope of design forces applied to members of the group can be overly conservative - e.g. if some columns are loaded about one axis, and some loaded about the other, they are all designed as if loaded about *both* axes.

   ---

5. If a group member has a lower than desired utilization, right click on it and select Design using Tekla Tedds> Member
6. In the Review View, review the resulting utilizations for the group members - note that some of might now fail.
7. Using the new utilizations to better inform you choice, add extra groups as necessary and re-allocate the members between the groups.
8. Run Design using Tekla Tedds> Group for one member in each of the new groups.
9. Iterate the process, starting from step 5 above.
10. When changes are made to the model, you can check if the existing designs are still sufficient by running Check In Tedds from the Design tab.

Alternatively you can check groups individually by running Check using Tekla Tedds> Group for one member in each group.

NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data from the right-click context menu.

Ungrouped Design

If you have elected not to make use of design groups, each member can be designed individually as follows:

1. Highlight the member, right click and select Design using Tekla Tedds> Member
2. Design the selected member.
3. Continue to design additional members in the same way as required.
4. When changes are made to the model, you can check if the existing sections are still sufficient by running Check In Tedds from the Design tab.

NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. If you want to manually clear the Tedds calculation data from a previous design run you can do so by running Clear Tekla Tedds Data from the right-click context menu.

Design model

If you want to design every precast (and timber) member in the model in one go:

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**

   At the end of the process the status and utilization of each member is displayed in a Review View.

Check the design after changes

If changes are made to the model you can run a 'check' design to determine if the existing sections are still sufficient. A check is quicker to perform than a

design because the Tedds calculation runs in the background without having to display the Tedds calculation dialog.

The updated utilizations can then be reviewed in a Review View.

Check the whole model

To check all the existing Tedds member designs,

1. Click Check In Tedds from the **Design** tab.

   This reruns all the Tedds calculations in the background using the latest analysis results.

Check a selection

To check several members or groups in one go,

1. Select the members or groups you want to check.
2. Right click and select **Check using Tekla Tedds > Selection**

   The Tedds calculations for the selected members or groups run in the background using the latest analysis results.

Check a member

To check a single member,

1. Highlight the member you want to check.
2. Right click and select **Check using Tekla Tedds > Member**

   The Tedds calculations for the selected member runs in the background using the latest analysis results.

   The updated status and utilization are displayed in the member tooltip.

Check a group

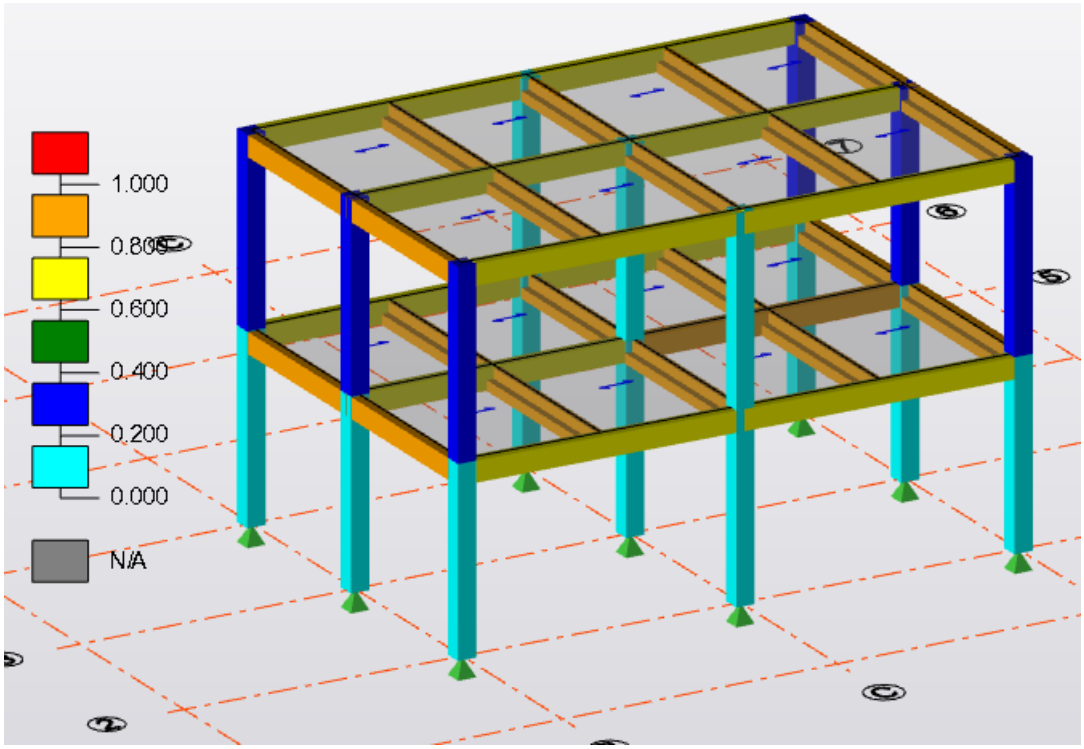
To check a member group,

1. Highlight a member in the group you want to check.
2. Right click and select **Check using Tekla Tedds > Group**
   - The Tedds calculations for the selected group run in the background using the group critical design forces.
   - All members in the group are then checked against their individual design forces.

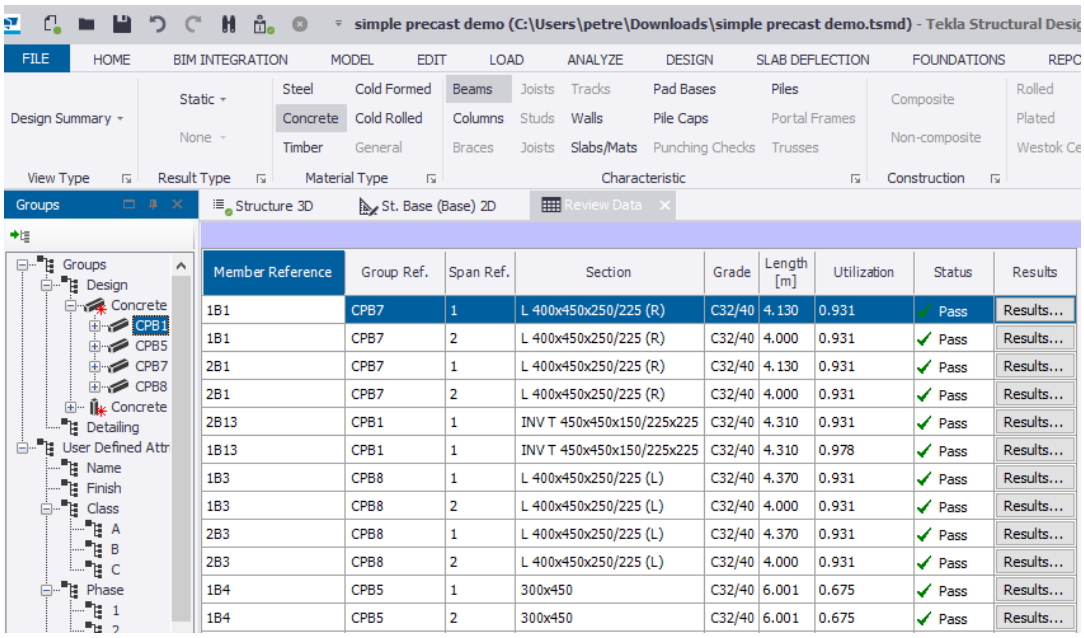
Output the calculations

The Tekla Tedds design results are returned to the Tekla Structural Designer model.

At this stage you can display member design status and utilization ratios from a Review View.

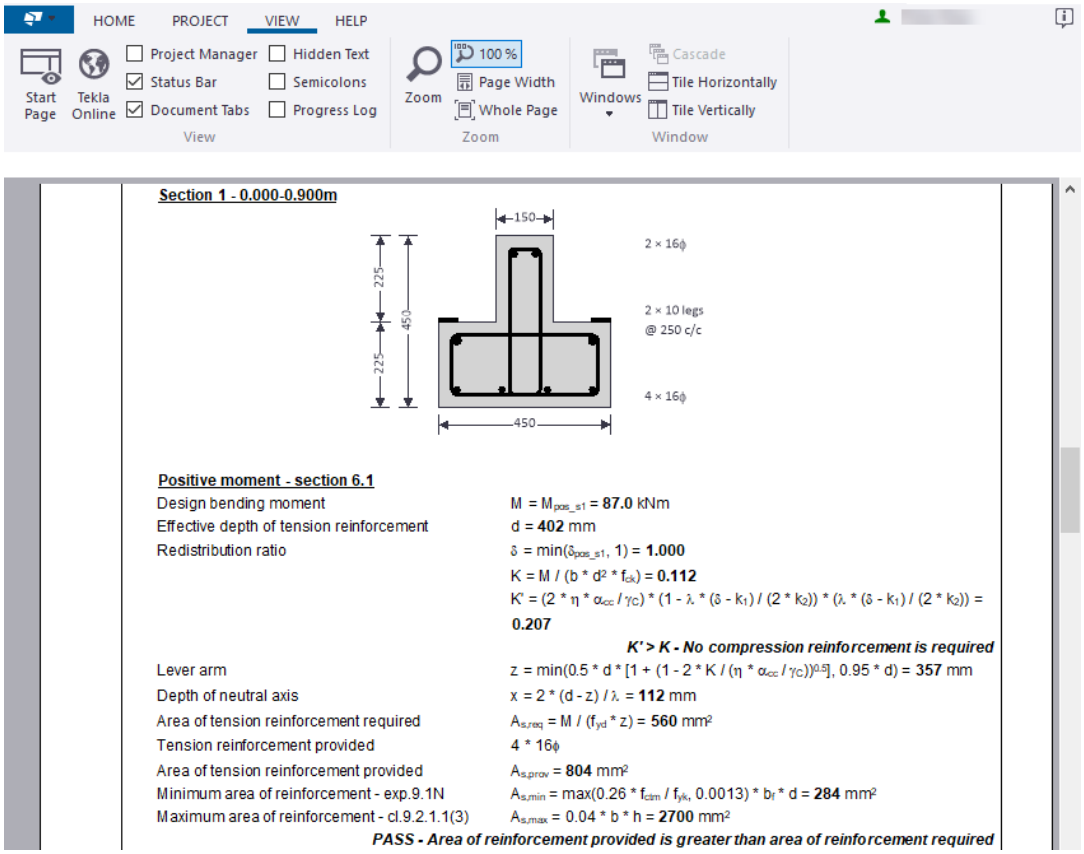


You can also display the design status tabular results.



Member Reference	Group Ref.	Span Ref.	Section	Grade	Length [m]	Utilization	Status	Results
1B1	CPB7	1	L 400x450x250/225 (R)	C32/40	4.130	0.931	Pass	Results...
1B1	CPB7	2	L 400x450x250/225 (R)	C32/40	4.000	0.931	Pass	Results...
2B1	CPB7	1	L 400x450x250/225 (R)	C32/40	4.130	0.931	Pass	Results...
2B1	CPB7	2	L 400x450x250/225 (R)	C32/40	4.000	0.931	Pass	Results...
2B13	CPB1	1	INV T 450x450x150/225x225	C32/40	4.310	0.931	Pass	Results...
1B13	CPB1	1	INV T 450x450x150/225x225	C32/40	4.310	0.978	Pass	Results...
1B3	CPB8	1	L 400x450x250/225 (L)	C32/40	4.370	0.931	Pass	Results...
1B3	CPB8	2	L 400x450x250/225 (L)	C32/40	4.000	0.931	Pass	Results...
2B3	CPB8	1	L 400x450x250/225 (L)	C32/40	4.370	0.931	Pass	Results...
2B3	CPB8	2	L 400x450x250/225 (L)	C32/40	4.000	0.931	Pass	Results...
1B4	CPB5	1	300x450	C32/40	6.001	0.675	Pass	Results...
1B4	CPB5	2	300x450	C32/40	6.001	0.675	Pass	Results...

Detailed Tekla Tedds calculations for precast members do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds.



4.3 Precast member design groups

NOTE The use of suitable design groups for precast members is recommended. Using design groups will help speed the overall design process and allow for identical reinforcement amounts to be provided for each element within the group.

Why use precast design groups?

Precast beams and columns are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design - to standardise designs, reduce processing time, and reduce the volume of output created.

A fixed set of rules are used to automatically determine member groups: for example beams must be of similar spans, columns must have the same

number of stacks. Members in all member groups must be of similar lengths and cross-section.

NOTE Although precast members are automatically grouped for ease of editing, the use of groups for design is optional and can be deactivated if required: From the Design tab, click Settings> Design Groups, then select or unselect concrete beams and/or concrete columns as required.

Activating precast member design groups

Precast beam and column design groups are activated as follows:

1. From the **Design** tab, click Settings> Design Groups
2. Select concrete beams and/or concrete columns to activate precast beam and/or precast column design groups.

Group management

Automatic Grouping

Precast beams and columns are grouped automatically according to a fixed set of requirements.

In Model Settings > Grouping the user defined maximum length variation is used to control whether elements are sufficiently similar to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

See:

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup ALL Model Members

If you have made changes in Design Settings that affect grouping, you can update the groups accordingly from the Groups tab of the Project Workspace by clicking Re-group ALL Model Members.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Precast design group requirements

Precast member design groups are formed according to the following rules:

Member type	Design group rules
Precast beam	• A beam element may be in only **one** design group. • Design groups may be formed from single span or multi-span continuous beams. • All beam elements in the group must have an identical number of spans. • For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length. • All beam elements in the group must have identical material properties.
Precast column	• A column element may be in only **one** design group. • All column elements in the group must have an identical number of stacks. • For each individual stack all column elements in the group must have an identical cross section, and stack length. • All column elements in the group must have identical material properties.

4.4 Precast beam design

Specific aspects of the precast beam design workflow in Tekla Structural Designer are described below:

Section shapes

The following section shapes can be used within the Tekla Structural Designer - Tekla Tedds workflow.

- Rectangular
- L-Section
- Inverted L-Section
- T-Section

- Inverted T-Section

Beam arrangement

The Tekla Tedds calculations have no concept of the model geometry within Tekla Structural Designer and so there are no limitations on the design of curved beams in both major and minor axes. Continuous beams, cantilevered beams etc. are similarly supported by the workflow.

Concrete type

While you can apply both normal and lightweight concrete in the beam properties, precast beam design using lightweight concrete is currently beyond scope.

Nominal cover

The default nominal top, bottom and side cover values set in **Design Settings> Precast> Beam> Reinforcement Settings** are automatically passed through to the Tekla Tedds calculation. The nominal concrete cover is the distance between the surface of the reinforcement closest to the concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

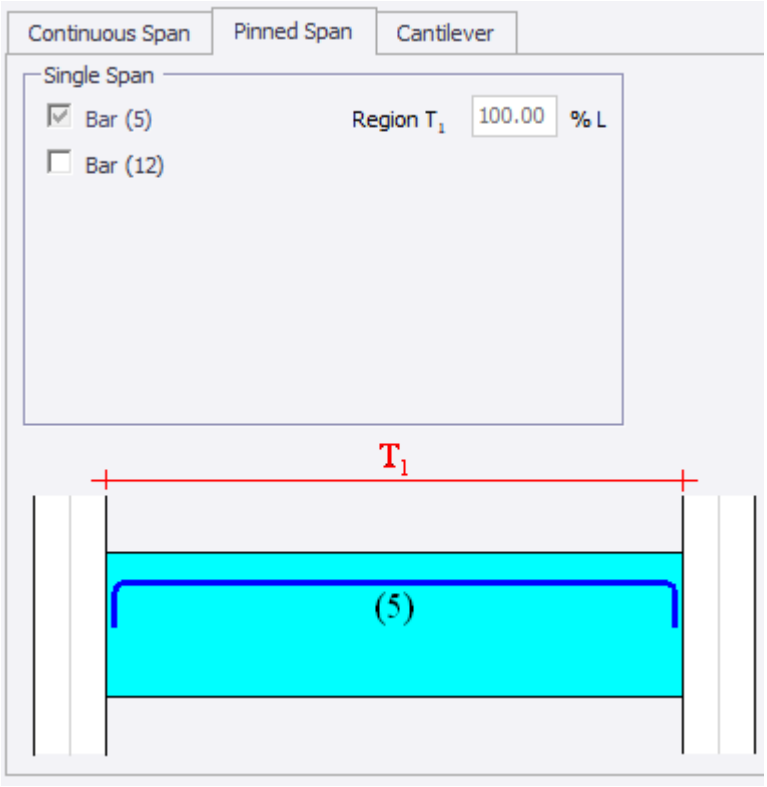
Reinforcement - longitudinal bar patterns

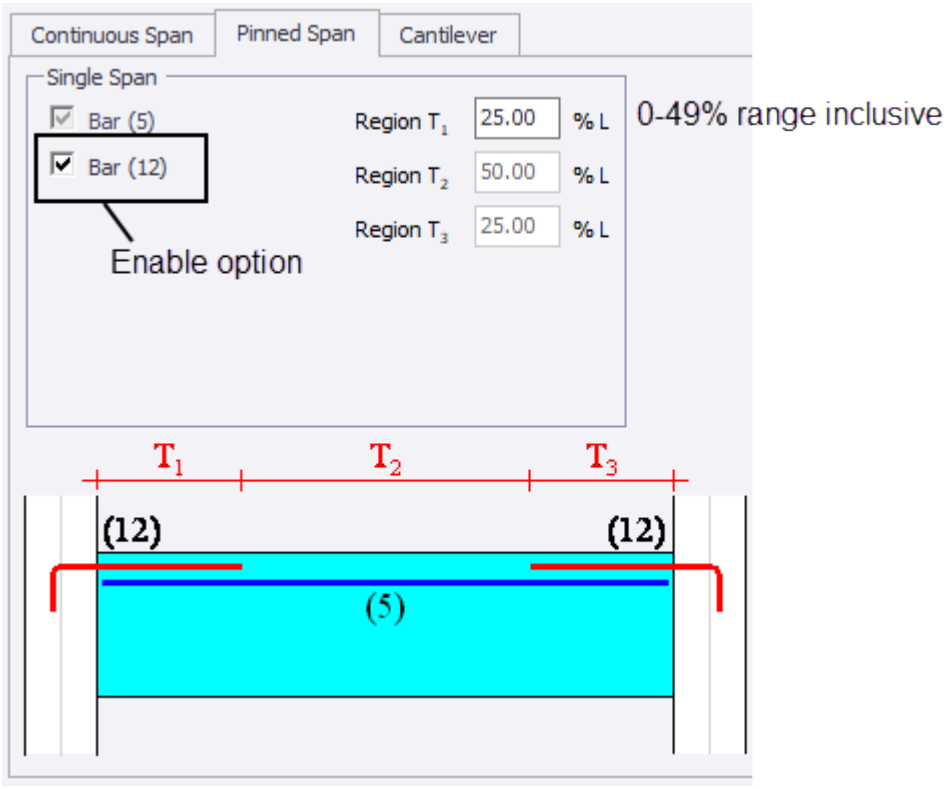
NOTE The reinforcement regions are considered to be the extent of the effective reinforcement. Anchorage and anchorage lengths are not considered in precast beam design within Tekla Tedds.

In **Design Settings> Precast> Beam** there is a one Standard Pattern available for top longitudunal reinforcement, Precast Top 1, and one Standard Pattern available for bottom longitudunal reinforcement, Precast Bottom 1 as illustrated in the figures below.

Standard Pattern of Top Reinforcement - Precast Top 1

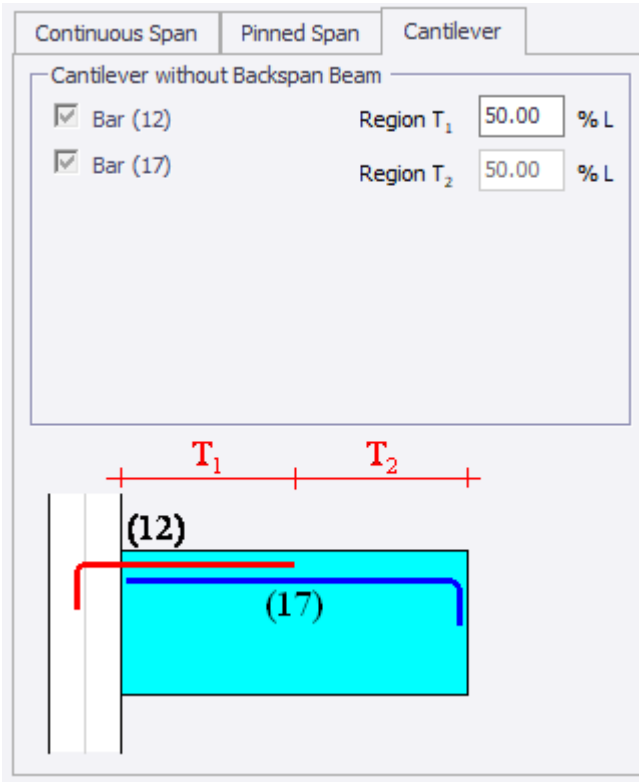
Pinned span settings are applied to all spans with Fixity end 1 and 2 set to Pin.


Continuous Span
Pinned Span
Cantilever
Single Span
Bar (5)
Bar (12)
Region T1
100.00
% L
T1
(5)

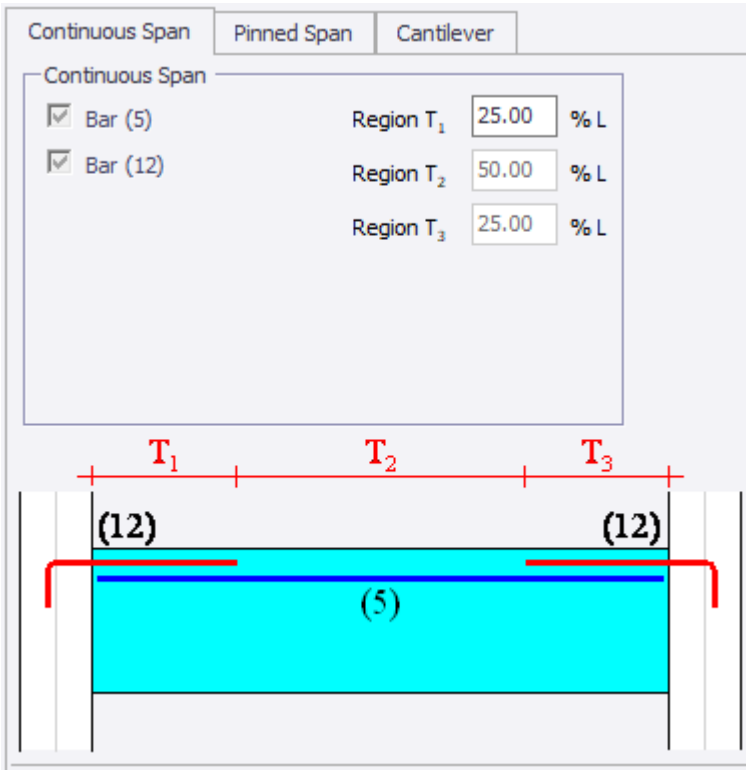


NOTE As shown above, the concept of continuity bars in pinned spans can be catered for by enabling Bar (12). When enabled, the end region length is allowed to be 0%.

Cantilever span settings are applied to all spans with cantilever selected.

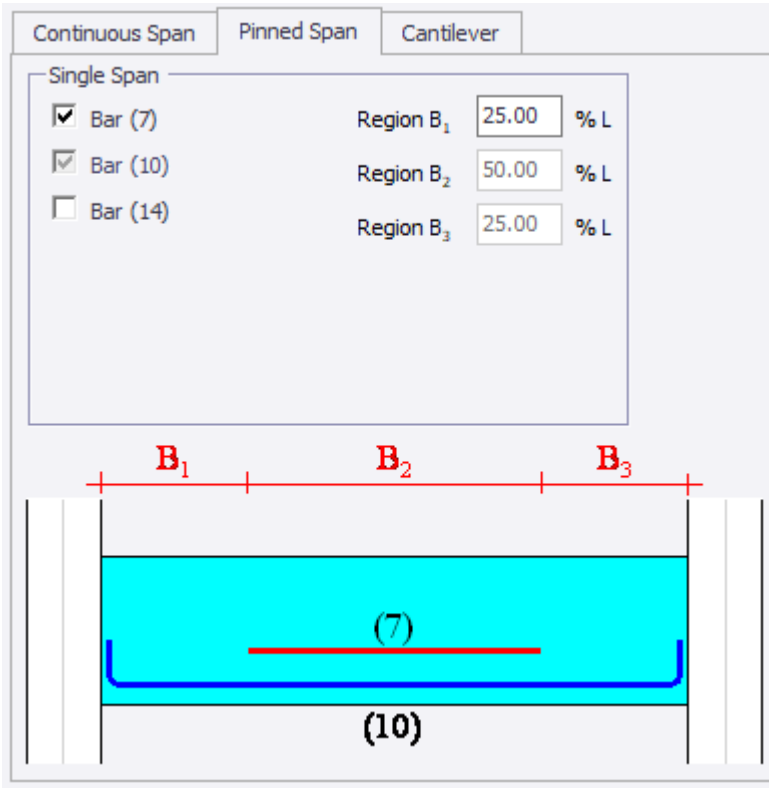


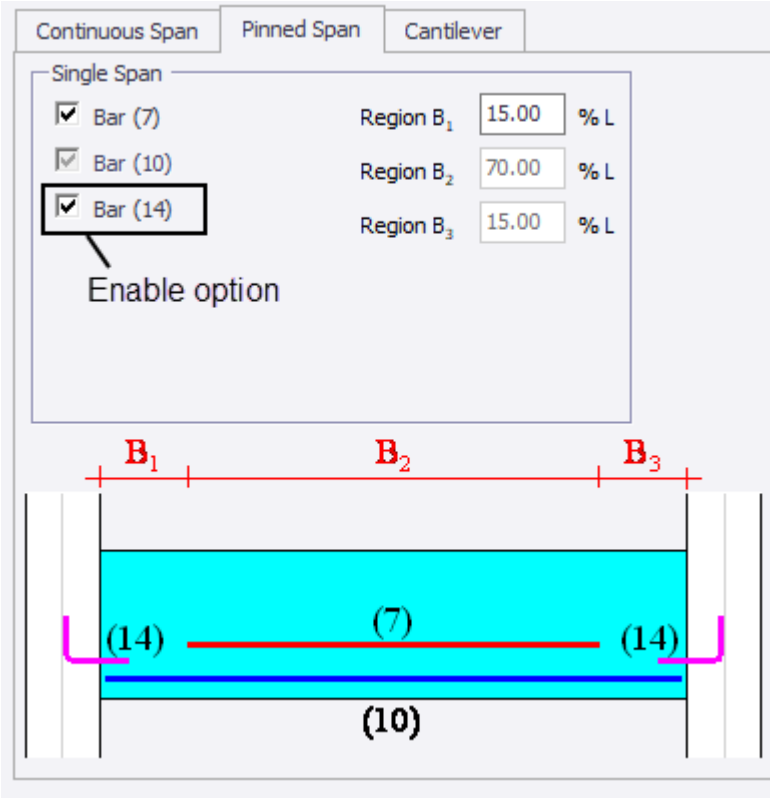
Continuous span settings apply to all other spans.



Standard Pattern of Bottom Reinforcement - Precast Bottom 1

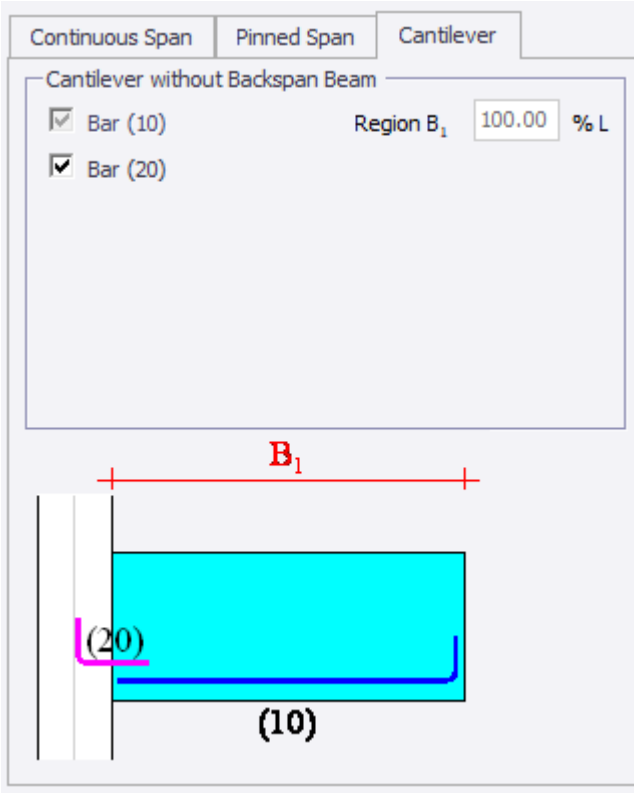
Pinned span settings are applied to all spans with Fixity end 1 and 2 set to Pin.


Continuous Span
Pinned Span
Cantilever
Single Span
Bar (7)
Bar (10)
Bar (14)
Region B1
25.00
% L
Region B2
50.00
% L
Region B3
25.00
% L
B1
B2
B3
(7)
(10)

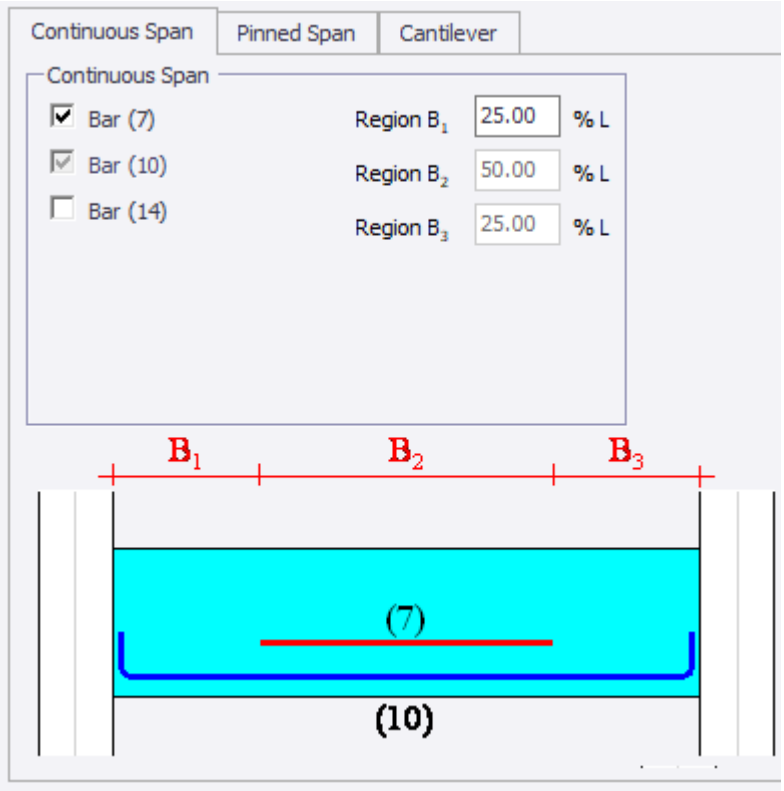


NOTE As shown above, the concept of continuity bars in pinned spans can be catered for by enabling Bar (14).

Cantilever span settings are applied to all spans with cantilever selected.



Continuous span settings apply to all other spans.



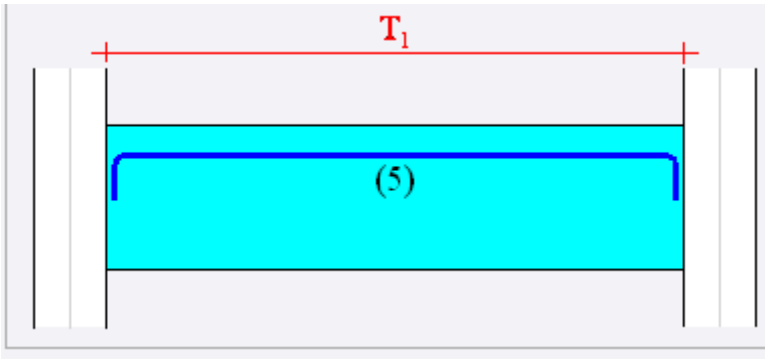
Design sections

An individual design section is created for each length of beam with similar reinforcement definitions. Each design section is then designed for the maximum forces in that region of similar reinforcement.

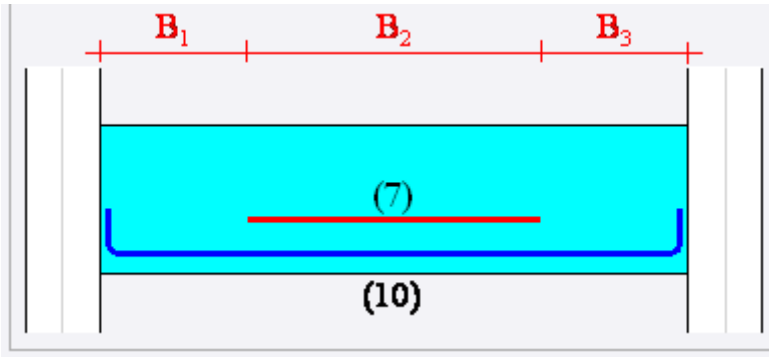
The numbers of design sections and their locations are determined by aggregating the different regions specified in the **Design Settings> Precast> Beam**: **Top Longitudinal Bar Pattern**, **Bottom Longitudinal Bar Pattern** and **Link Settings**.

Example: The default **Precast Top 1** and **Precast Bottom 1** longitudinal bar patterns for a Pinned Span are adopted without any continuity bars added as follows:

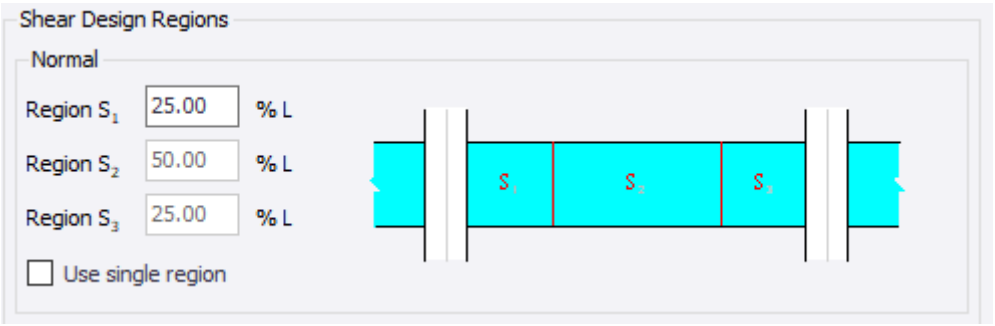
- **Precast Top 1**: single region 100% of the beam span.



- **Precast Bottom 1**: B1 and B3 regions set as 15% of the beam span.

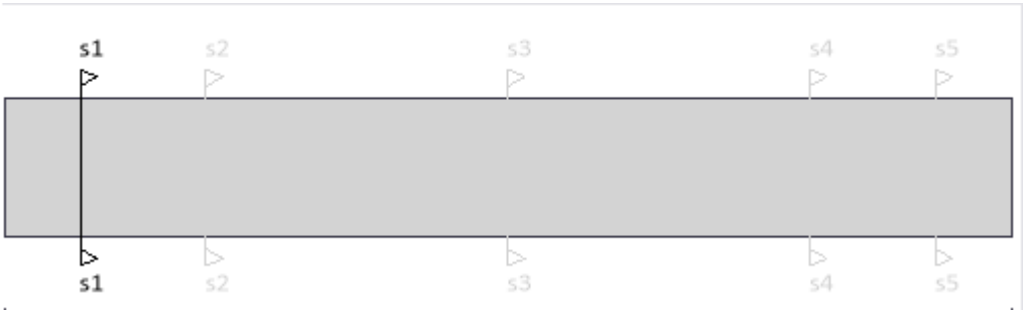


- Default **Link Settings** are also adopted: S1 and S3 regions set as 25% of the beam span.



For a pinned single span precast beam, different region boundaries would therefore exist at 15%, 25%, 75% and 85% of the span.

Consequently, when the span is designed in Tekla Tedds, 5 design sections would be created:



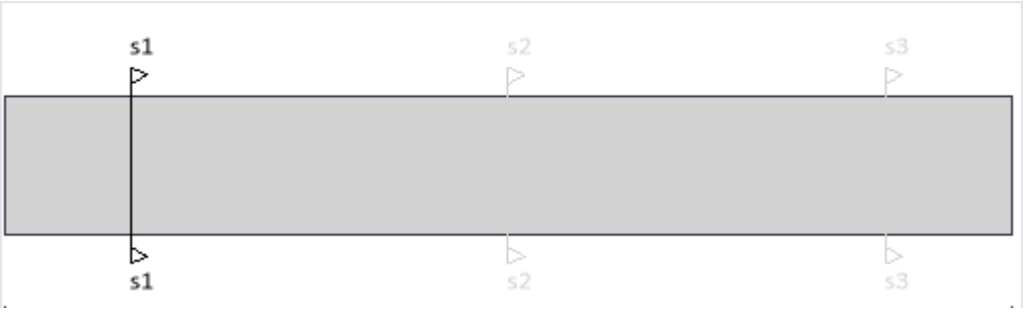
- s1 - positioned half way along the first bottom span region (7.5% of the span)
- s2 - positioned half way between the first and second region boundaries (20% of the span)
- s3 - positioned half way along the second and third region boundaries (50% of the span)
- s4 - positioned half way between the third and forth region boundaries (80% of the span)
- s5 - positioned half way along the last bottom span region (92.5% of the span)

NOTE In this example the first design section lies in the B1 bottom span region, so the positive design moment at s1 would be the maximum that occurs within B1; the second, third and forth design sections all lie in B2, so the positive design moment at s2, s3 and s4 would be the maximum that occurs within B2.

The engineer can of course adjust the standard bar patterns in the Tekla Structural Designer design settings to suit their requirements.

For instance, if in this example the **Precast Bottom 1** pinned span pattern were to be amended so that B1 and B3 regions were set as 25% of the span; because the bottom and link regions would then coincide, region boundaries would only exist at 25% and 75% of the span.

Consequently, when the span is designed in Tekla Tedds, only 3 design sections would be required:



- s1 - positioned half way along the first region (12.5% of the span)

- s2 - positioned half way between the first and second region boundaries (50% of the span)
- s3 - positioned half way along the last region (75% of the span)

NOTE In the above elevation diagrams:

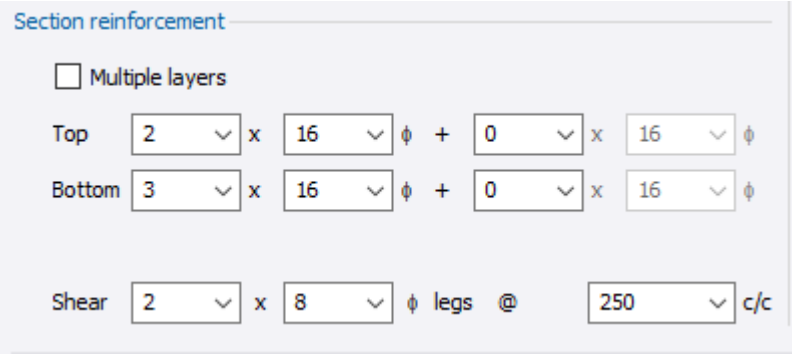
- A black section mark indicates the selected design section is passing the design criteria.
- Grey section marks indicate unselected design sections which are passing.
- A red section mark would indicate a design section which is failing.

Default reinforcement in the Tekla Tedds calculation

Regardless of beam shape and size imported from Tekla Structural Designer, the reinforcement in each beam member is always defaulted to the same values Tekla Tedds calculation.

This default reinforcement is:

- Top: 2x 16 dia bars
- Bottom: 3x 16 dia bars
- Shear: 2 legs of 8 dia bars at 250 cross centres



NOTE The Tekla Tedds calculation contains some limitations to the error trapping when it comes to maximum reinforcement values. Although the main bars in both the top and bottom of the beam have a check allocated to prevent unrealistic distances being entered, the shear links have no such limitation.

The engineer is expected to use their own knowledge to place in shear link values and distances which are both realistic and constructible.

NOTE The Tekla Tedds calculation contains some limitations to the section diagram. It may be noted that the diagram is never altered if the number of shear legs is increased from the default value of 2 legs. It should be noted that shear link legs are indicative only within Tekla Tedds and in reality there are a variety of ways that reinforcement can be detailed. The Tekla Tedds calculation therefore chooses to omit additional shear reinforcement in the section diagram and to flag it instead in the notes and the calculations.

Lifting checks

Lifting checks are disabled by default. This has been done to ensure a rapid design of grouped beams can take place. If lifting checks are required then simply enable the switch within the **Design Options** dialog within the Tekla Tedds calculation.

NOTE The lifting check for beam members makes allowance for reinforcement for both bending and shear checks.

Analysis forces transferred from Tekla Structural Designer

The following values in the Tekla Tedds calculation are populated from the Tekla Structural Designer model.

- Positive moment
  - Positive design moment
  - Positive quasi-permanent moment
  - Redistribution ratio
- Negative moment
  - Positive design moment
  - Negative quasi-permanent moment
  - Redistribution ratio
- Shear
  - Maximum Design shear force
  - Design shear force
- Torsion
  - Design torsional moment

NOTE Minor axis forces (as defined in Design > Settings > Design Forces) are not used in the design of the member. Where minor axis forces are present within a beam, a warning is displayed within the process dialog of Tekla Structural Designer.

Other precast beam properties

The following properties which are common to both cast-in-place and precast beams, are not transferred to the Tekla Tedds calculation, but still require consideration as they will impact the analysis in Tekla Structural Designer.

-
- Design parameters (page 155)

4.5 Precast column design

Specific aspects of the precast column design workflow in Tekla Structural Designer are described below:

NOTE The current Tekla Tedds calculation makes no allowance for any shear checks.

Section shapes

Whist a variety of different section shapes can be defined in Tekla Structural Designer, the Tekla Tedds calculation only supports the design of rectangular and circular column sections.

Concrete type

While you can apply both normal and lightweight concrete in the column properties, column design using lightweight concrete is currently beyond scope.

Nominal cover

The default nominal cover value set in Design Settings> Precast> Column> Reinforcement Settings is automatically passed through to the Tekla Tedds calculation. The nominal concrete cover is the distance between the surface of

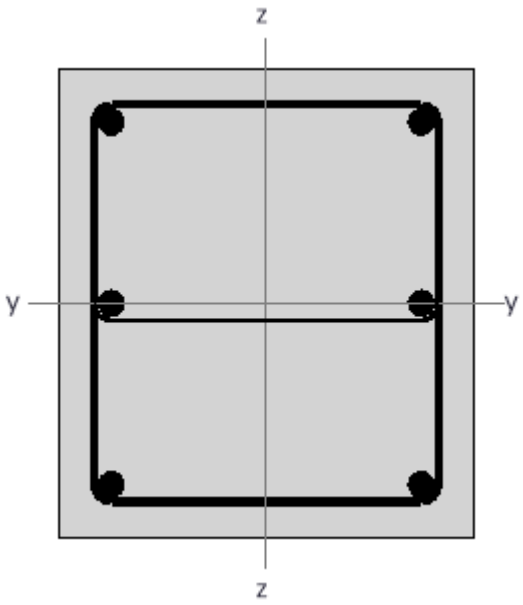
the reinforcement closest to the concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.

Reinforcement

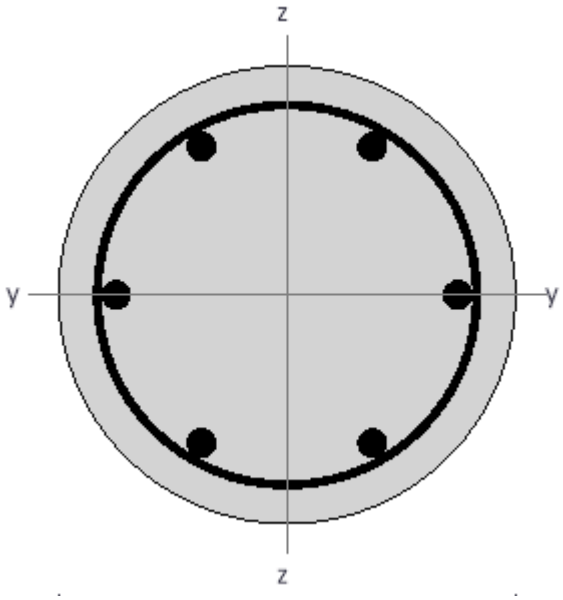
Default reinforcement in the Tekla Tedds calculation

Regardless of the column size imported from Tekla Structural Designer, the initial default reinforcement for all stacks in the Tekla Tedds calculation is as follows:

- Rectangular columns: a 2x3 formation of 25mm dia rebars with 8mm shear links.



- Circular columns: 6x 25mm dia rebars with 8mm shear links.



NOTE Tekla Tedds has no concept of multiple stack columns and Tekla Structural Designer has to associate separate Tekla Tedds calculations with each stack. It is the responsibility of the engineer to ensure that rebar on adjacent stacks are of appropriate quantities and sizes to ensure correct detailing can take place at the construction stage. It should be noted that Tekla Structural Designer does not hold any reinforcement information relating to precast members. Reinforcement is completely managed by the Tekla Tedds calculations.

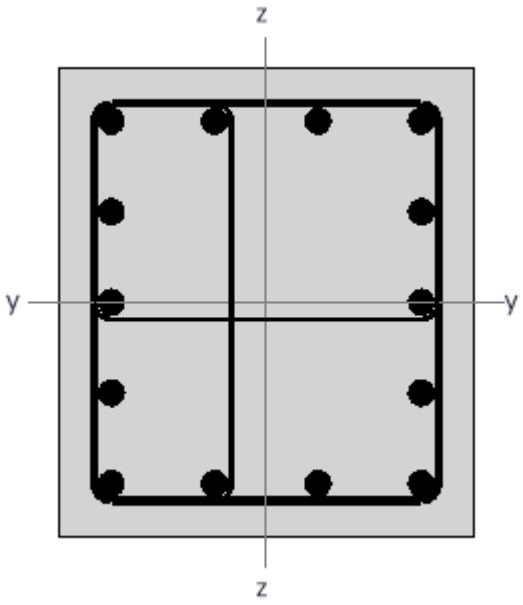
The maximum number of bars which can be entered in either direction for rectangular columns is 10. For circular columns, the maximum total number of bars is 20. Please note that these restrictions will have an effect on the maximum allowable column size for the design workflow.

There is a basic minimum distance calculation within the Tekla Tedds document relating to bar amounts to avoid unrealistic values of rebar from being selected. This amount is to be the minimum of:

- K1 x bar dia
- 20mm
- k2 x max aggregate size

NOTE For values of coefficients k1 and k2, refer to clause 8.2 of the Eurocode.

The number of link legs shown in the Tekla Tedds calculation will increase or decrease depending on the number of reinforcing bars assigned within the document.



NOTE The link legs are displayed to indicate required confinement reinforcement for vertical bars in compression, but have no effect on the overall design process.

Lifting Checks and Splice Design

Lifting checks and splice checks are disabled by default. This has been done to ensure a rapid design of grouped columns can take place. If these checks are required then simply enable the switches within the **Design Options** dialog within the Tekla Tedds calculation.

NOTE The lifting check is performed against the unreinforced area of the column, hence there is no input for the shear link centres within the calculation. (Bending checks (for splice design) do include reinforcement allowances within the design process.)

Analysis forces transferred from Tekla Structural Designer

The following values in the Tekla Tedds calculation are populated from the Tekla Structural Designer model.

- Axial load, 1st case (Max value)
- Axial load, 2nd case (Min value)

    - Where tension exists in the column, the lowest compression value is populated into this parameter and the tension value ignored. A warning will be displayed in the process log within Tekla Structural Designer.
- Moment about y axis at top
- Moment about z axis at top
- Moment about y axis at bottom
- Moment about z axis at bottom

NOTE Tekla Structural Designer populates values from all design combinations. For columns subjected to low axial loading, the moment capacity will not be as high as a column subjected to larger axial loading. Low axial load and high moment can easily be a critical combination for concrete column design.

Other precast column properties

The following properties which are common to both cast-in-place and precast columns, are not transferred to the Tekla Tedds calculation, but require consideration as they impact the overall performance of the model.

-
- Design parameters (page 163)
- Stiffness (page 165)
- Sway/Drift Checks (page 166)

4.6 Precast column connection eccentricity moments

Overview

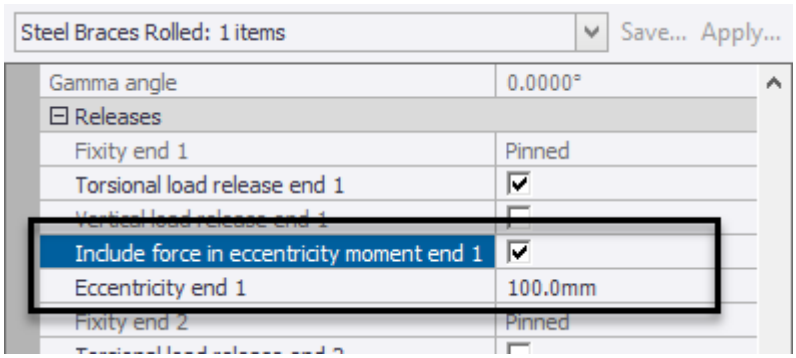
Nominal eccentricity moments arise in precast columns due to the reactions from pin ended beams being applied at eccentricities to the column centerline. The eccentricity moments are calculated for all regional codes, and can be interrogated in the Load Analysis View.

If working to Eurocodes, these eccentricity moments are also taken into account when precast columns are transferred to Tedds for design.

These moments do not come directly from the global analysis but instead are calculated at the ‘load analysis’ post-processing stage as follows:

- At each level the eccentricity of each connection is specified as a user defined offset from the column face. If rigid zones are not being used this is increased by half the depth of the supporting column.

- At each level the pinned beam end reactions connecting to the column at each face are determined.
- If braces also connect to the same face, the force in the brace will also be taken into consideration if the "Include force in eccentricity moment" brace release property is checked for the appropriate end of the brace.



- Taking the beam end reactions (and brace forces if included) on opposite faces multiplied by their connection eccentricities, resultant eccentricity moments are determined.
- These moments are then distributed above and below the level based on the column stiffnesses.

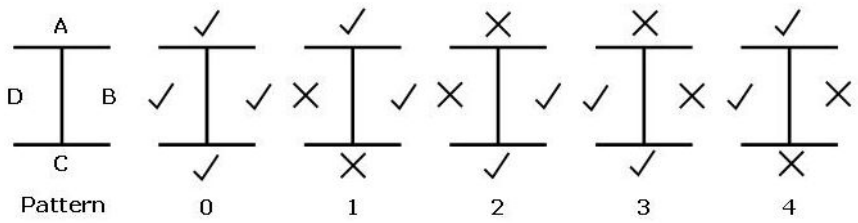
NOTE The eccentricity moments are typically assumed not to be transferred beyond the level at which they are applied.

Patterning of eccentricity moments

The eccentricity moments resulting from live loads can be patterned if required to account for the likelihood that the load is not present on all spans simultaneously.

When eccentricity moment patterning is enabled you must then indicate which of the live cases are to be patterned, (you may for example decide not to pattern storage loads.)

For those live cases with patterning enabled, five patterns are considered. These are:



Pattern 0 is for the full live load at all positions i.e. no patterning - this gives the maximum axial force in any one stack with (usually) lower eccentricity moment.

Patterns 1 to 4 are ‘true’ patterns switching live load ‘on’ and ‘off at each pair of positions around the column in order to generate the maximum live eccentricity moments about the major and minor axes of the column.

NOTE The same pattern is applied at the top and bottom of the stack, so for example it is not possible to have P1 at the top and P4 at the bottom.

Design to Eurocodes

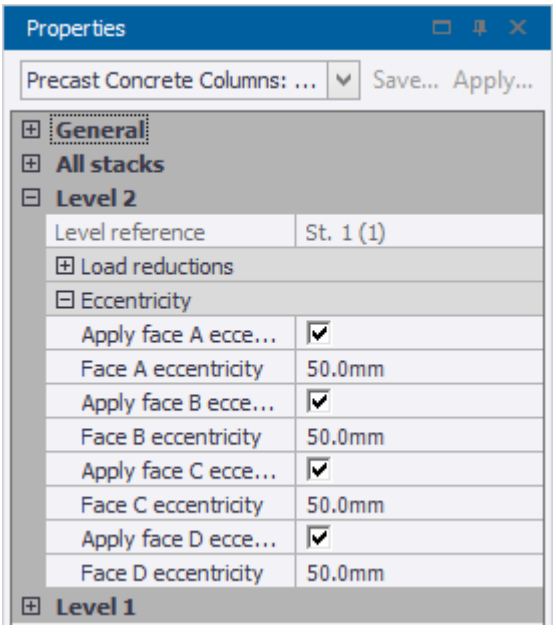
In general, eccentricity moments are only added to the 'real' moments at the ends of each stack and are only added if they make the design worse.

If you have elected to pattern live eccentricity moments these are considered in conjunction with the eccentricity moments from other types of load, and with the 'real' moments.

- As the eccentricity moments are considered localised to each floor the full axial force from other floors is maintained. The axial force at the level under consideration will be slightly reduced with patterning enabled as the live floor loading will not be present on all sides simultaneously.

Define connection eccentricity values

The eccentricities at each level are defined in the column properties and a different eccentricity can be applied to each face.



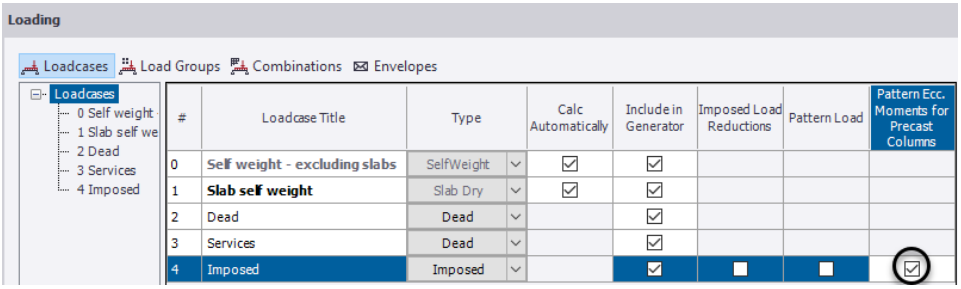
If you uncheck the option to apply eccentricity at a face the end reaction on that face is applied axially.

NOTE Face A can be identified graphically, from which the other faces follow, see: Steel member orientation (page 26)

Pattern eccentricity moments for live loadcases

Patterning can be switched on for specific live loadcases in a two-step process as follows:

1. From the **Home** ribbon:
   a. Click **Model Settings > Loading > General**
   b. Select **Use patterning of eccentricity moments for precast columns**
   c. Click **OK**
2. From the **Loadcases** page of the **Loading dialog**:
   a. Select a live loadcase that you want to be patterned
   b. Select **Pattern Ecc. Moments for Precast Columns**
   c. When patterning has been selected for each of the required loadcases, click **OK**

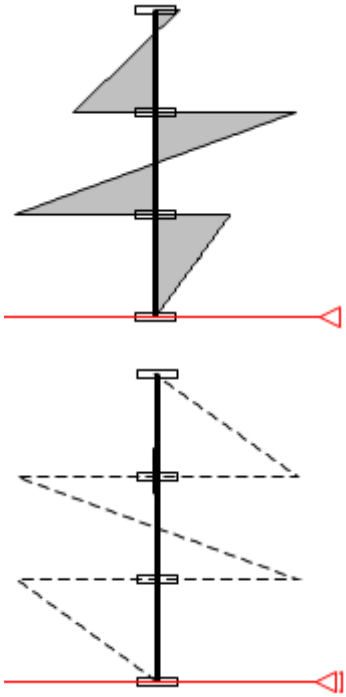


Review connection eccentricity moments

Because eccentricity moments do not come directly from the global analysis they cannot be displayed graphically in a **Results View**, they can only be displayed on a column by column basis by opening a Load Analysis View.

With a **Load Analysis View** open and the required loadcase or combination selected in the **Loading** list, you then select the **Major**, or **Minor** direction in the **Loading Analysis** ribbon.

The 'real' moments are displayed as a shaded diagram using solid lines, the eccentricity moments as an unshaded using dashed lines:

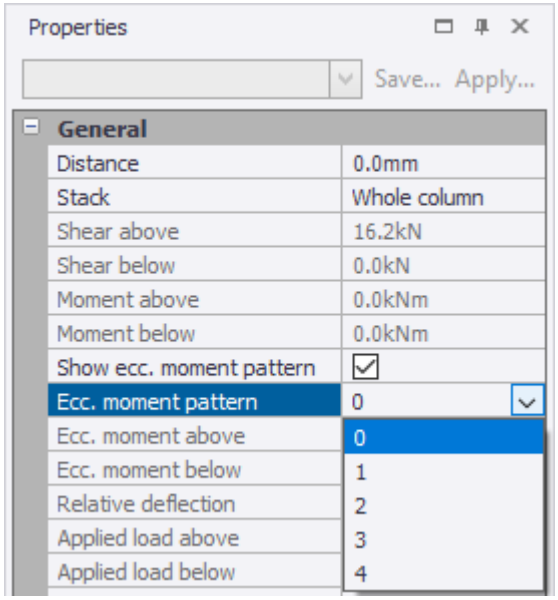


The red marker line can be set to a specified distance in the **Properties** window to allow the real and ecc. moment values above and below the line to be displayed.

Displaying patterned eccentricity moments

When you select a patterned live loadcase a **Show ecc. moment pattern** box will become available in the **Properties** window.

After selecting **Show ecc. moment pattern** you can then click **Ecc. moment pattern** in order to select the pattern to display from the droplist.


Properties
Save...
Apply...
General
Distance
0.0mm
Stack
Whole column
Shear above
16.2kN
Shear below
0.0kN
Moment above
0.0kNm
Moment below
0.0kNm
Show ecc. moment pattern
Ecc. moment pattern
0
Ecc. moment above
Ecc. moment below
Relative deflection
Applied load above
Applied load below
0
1
2
3
4

5 Timber member design handbook

Timber structures can be modeled and analyzed in Tekla Structural Designer. If a license of Tekla Tedds is available, timber beams, columns and braces can then be designed to NDS LRFD, NDS ASD, base Eurocode, and the following Eurocode National Annexes: UK; Ireland; Norway; Finland; Sweden.

The following topics are covered in this handbook:

- Timber member design limitations and assumptions (page 381)
- Timber member design workflow (page 383)
- Timber member design groups (page 397)
- Timber combination sets and strength design cases (page 401)
- Timber beam deflection checks - US (page 403)
- Timber beam deflection checks - EC (page 408)
- Timber column stacks and lifts (page 413)

Learn how to design wood members in Tekla Structural Designer using Tekla Tedds (USA)

Learn how to design timber members in Tekla Structural Designer using Tekla Tedds (Eurocode)

5.1 Timber member design limitations and assumptions

The following limitations and assumptions apply to the design of timber beams, columns and braces in Tekla Structural Designer:

- A compatible version of Tekla Tedds must be installed and licensed.
- The locale must be set correctly in Tekla Tedds before attempting to run the design from Tekla Structural Designer.

- Design is supported for NDS LRFD, NDS ASD, base Eurocode, and the following Eurocode National Annexes: UK; Ireland; Norway; Finland; Sweden.
    - If the Singapore or Malaysia Eurocode National Annex is selected, the calculation will adopt base Eurocode recommended values.
- For models with seismic loading using the RSA method, RSA analysis results are not considered when determining the design forces populated to the Tedds timber design calculation.
    - This limitation can be worked around as follows: the engineer should review the RSA Analysis results for the Seismic Combinations for timber members being designed using Tedds, and note the critical values for design from these where they are greater than those of the static analysis/ combinations. The noted values can then be manually entered into the Tedds timber design calculation when design using Tedds is run.
- If working to the NDS code, size classifications taken from the NDS supplement 'Design Values for Wood Construction' are used in the Tedds calculation to distinguish which grades are valid for particular timber species. The same classification rules are not implemented in Tekla Structural Designer itself. Therefore, after a member has been designed in Tedds, if the member size or grade is subsequently changed in Tekla Structural Designer the associated Tedds calculation is deleted, so that the member cannot be checked and a redesign is required.
- Column design: If designing a multi-stack column you must indicate a splice in the stack properties **before** running the design if a change in cross section is required between stacks. If a splice has not been specified the change in section size will not be correctly returned to the Tekla Structural Designer model.
- When working to the NDS code, the following assumptions have been made:
    - The moisture content of structural composite lumber or glulam sections is less than 16% and the temperature remains below 100°F - analysis modification factors may need to be applied if this is not the case.
    - Glulam sections will be loaded perpendicular to the wide faces of the laminations (bending about the x-x axis).
    - Glulam grades comply with the characteristic properties outlined in AWC NDS supplement 2018.
    - The engineer has verified that the $E_{analysis}$ value of the member is suitable for the particular application.
    - The section and strength class are compatible.
- When working to Eurocodes, the following assumptions have been made:
    - Loading will be perpendicular to the wide faces of the laminations (bending about the major axis).

- Moisture content of the sections will be less than 15% - analysis modification factors may need to be applied if this is not the case.
- Glulam grades comply with the characteristic properties outlined in BS EN 14080:2013.
- The engineer has verified that the $E_{analysis}$ value of the member is suitable for the particular application.
- The section and strength class are compatible.

NOTE For further information relating to the scope, assumptions and limitations, general notes and references specific to the Tekla Tedds calculation itself, please refer to the respective calculation documentation in the Tedds calculation library.

5.2 Timber member design workflow

Key aspects of the Tedds integrated timber design process and workflow in Tekla Structural Designer are described in the sections below.

Learn how to design wood members in Tekla Structural Designer using Tekla Tedds (USA)

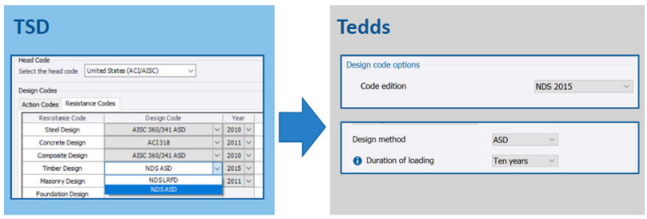
Learn how to design timber members in Tekla Structural Designer using Tekla Tedds (Eurocode)

Set the timber design code

The timber design code is set from the **Home** ribbon by clicking **Model Settings > Design Codes > Resistance Codes**.

- If working to **US codes**, there are two different methods, NDS ASD or NDS LRFD to choose from and also a choice of code year to apply. The design forces consider only the selected design method.
- If working to **Eurocodes**, the timber design code will be applied with a national annex that is appropriate to the regional code that has been selected.

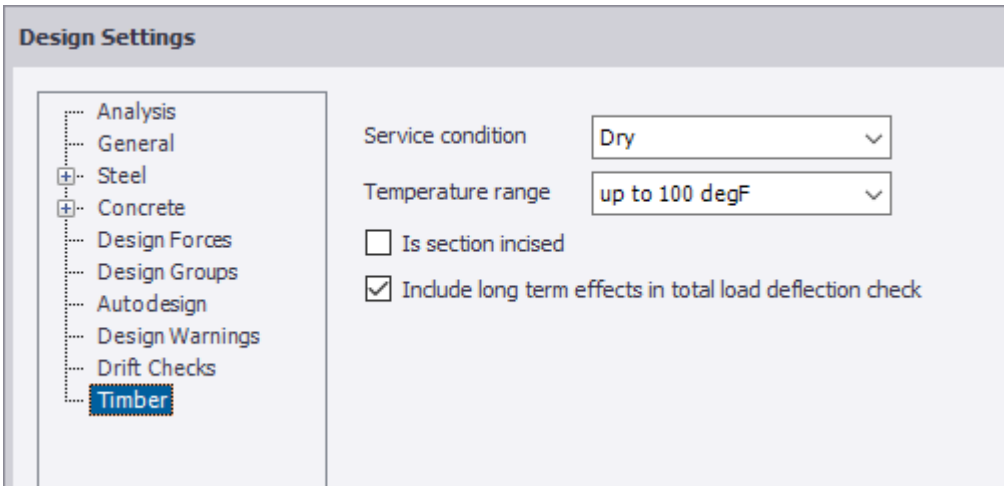
The selected design code is automatically applied in all Tedds timber member designs.



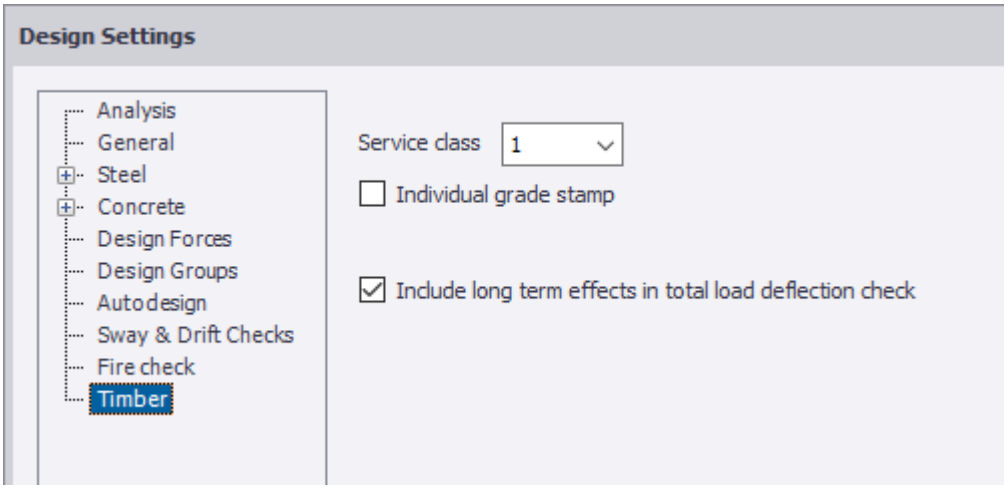
Configure timber design settings

Certain timber design settings which apply to the entire building can be specified from the **Design** ribbon by clicking **Settings > Timber**.

- If working to **US codes**:



- If working to **Eurocodes**:



By ensuring the defaults are set correctly you can avoid having to manually set the values in each Tedds timber calculation as it is run.

You still have control within individual calculations to adjust these settings on a member by member or group basis.

NOTE Tedds data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established.

If you change timber design settings after a member has already been designed you would need to manually clear the Tedds data from the design in order to have it use the revised design settings. To do this highlight the member, right-click and select Clear Tekla Tedds Data from the context menu.

Define and place timber members

To place timber members, select the appropriate member type from the **Timber** group on the **Model** ribbon, ensure the values in the **Properties** window are as required and then pick the points to locate the members in position.

Note the following with regard to the properties:

- **Section shapes** - When defining a timber model in Tekla Structural Designer it may appear that the range of timber sections is somewhat limited. This is because Tekla Structural Designer doesn't currently have a facility to specify timber sections by simply inputting a breadth and depth. Instead you have to select the required size from Tekla Structural Designer's database of timber sections.

  While this might seem to be a significant limitation, it is actually not too great an issue, because the Tedds timber calculation allows you to override the original size to input the breadth and depth you require. Then when the member data is transferred back from Tedds to Tekla Structural Designer the new size is automatically added to the timber section database.

  Once the new section is in the section database it becomes available for future Tekla Structural Designer models also.

- **Timber fabrication and grade** - You can specify timber, glulam, or structural composite lumber fabrication types.

  Initially the grade must be selected from the existing grades in the Tekla Structural Designer materials database, although if the grade is changed in the Tedds timber calculation, when the member data is transferred back from Tedds to Tekla Structural Designer the new grade is automatically added to the materials database.

  NOTE If a situation arises such that the material properties in the programs don’t align, Tedds ignores unmatched data and selects calculation defaults.

- **Deflection limits** - for beams you can specify limits for individual load type deflections and also for the total load deflection. For more information,

see: Timber beam deflection checks - US (page 403) or Timber beam deflection checks - EC (page 408) as appropriate.

- **Column splices** - for columns, splices should be specified in the stack properties if a change in cross section is required between stacks. If a splice has not been specified before the column is designed in Tedds the change in section size will not be correctly returned to the Tekla Structural Designer model. For further information, see: Timber column stacks and lifts (page 413).

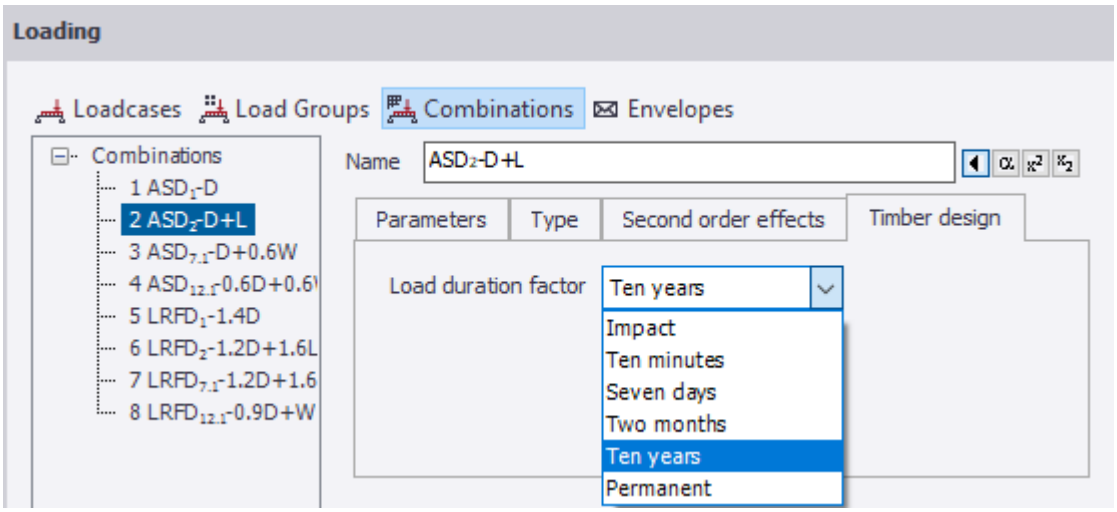
Create load combinations and review load duration/time effect factors

Loads are applied and load combinations defined in the standard manner.

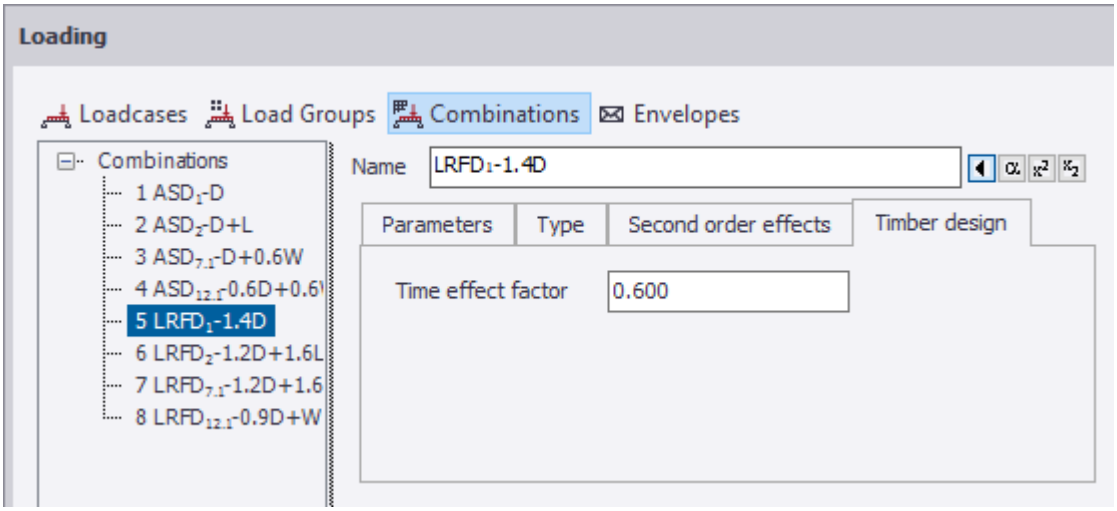
Once your load combinations have been created you should review the default factors associated with load duration that have been applied. (Load duration factor - NDS ASD and EC, Time effect factor - NDS LRFD).

These factors are set on the **Timber design** tab on the **Combinations** page of the **Loading** dialog once you have selected an individual combination from the list in the dialog. The value is set automatically according to the constituent loadcases.

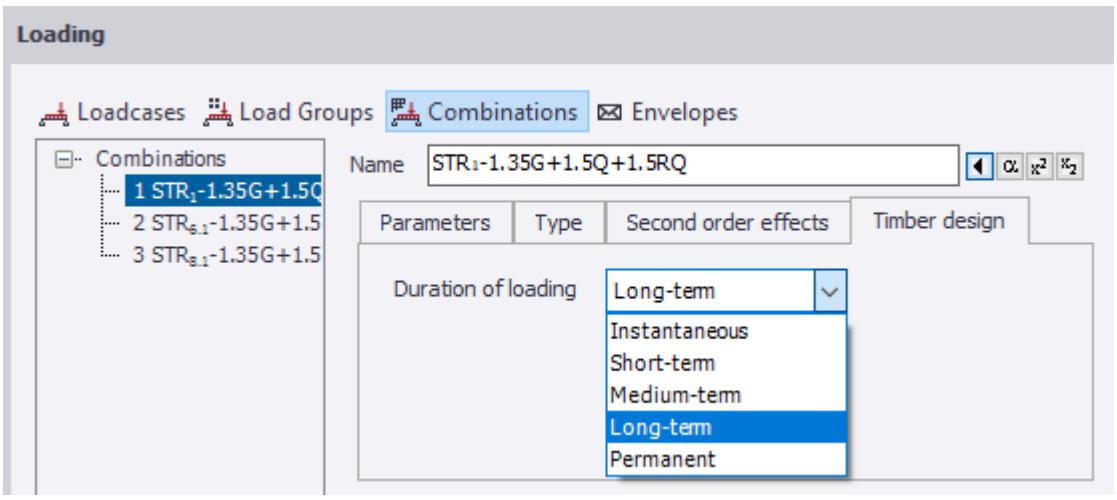
- **ASD combination:**



- **LRFD combination:**



- **Eurocode combination:**



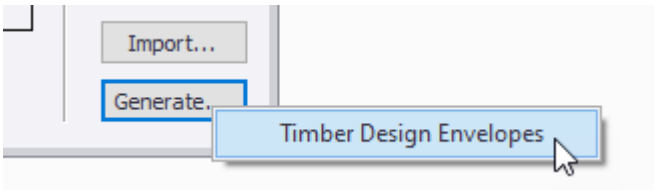
You have control to adjust the default value on a combination by combination basis if required.

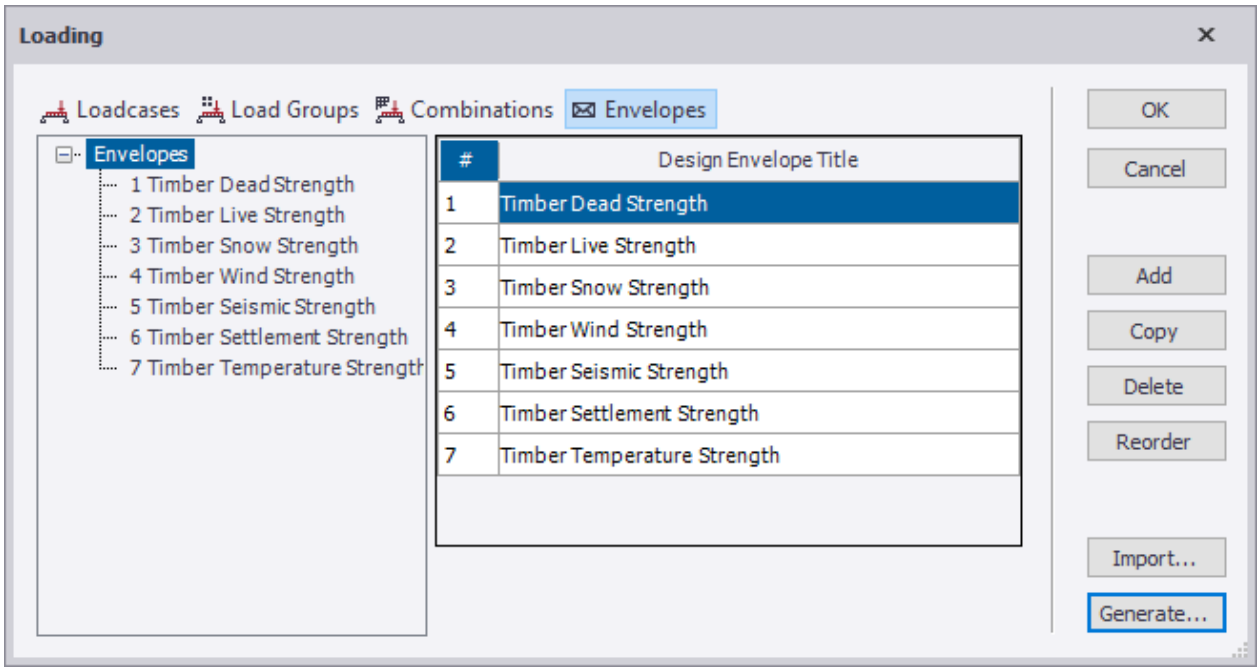
Load combinations are automatically grouped into timber combination sets to enable appropriate design cases to be created when the design is performed. For further details see: Timber combination sets and strength design cases (page 401).

Generate timber design envelopes

Design envelopes can be generated for each of the timber combination sets (page 401) so that the design forces can be easily reviewed.

You can create these from the **Envelopes** page of the **Loading** dialog by clicking **Generate... > Timber Design Envelopes**.





Configure timber groups

When timber members are placed in the model, to make editing easier they are automatically grouped according to a set of rules (page 400).

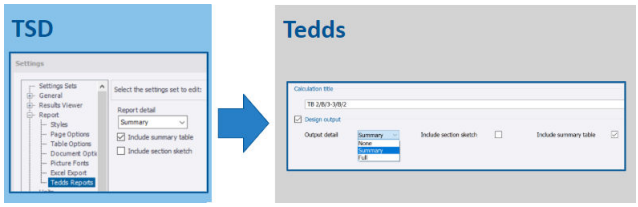
These initial groups can be reviewed from the **Groups** tab of the **Project Workspace** - you can add new groups and move members between groups as required.

Groups are not only useful for editing, they can (optionally) be activated for design purposes also. There are a number of reasons why you would choose to do this.

- Why use timber design groups? (page 397)
- Activating timber member design groups (page 398)

Set the Tedds results output level

You can choose the output level for the Tedds timber calculations in advance from the **Home** ribbon by clicking **Settings > Report > Tedds Reports**.



By ensuring this is set correctly beforehand you can avoid having to manually set the level in each Tedds timber calculation as it is run.

The setting applies to all Tedds linked calculations (precast and timber).

Run analysis

Timber members can only be designed provided a set of analysis results exist. These can be generated by running from the **Analyze** ribbon.

Once the analysis has been performed the **Design using Tekla Tedds** options become available.

Design using Tekla Tedds

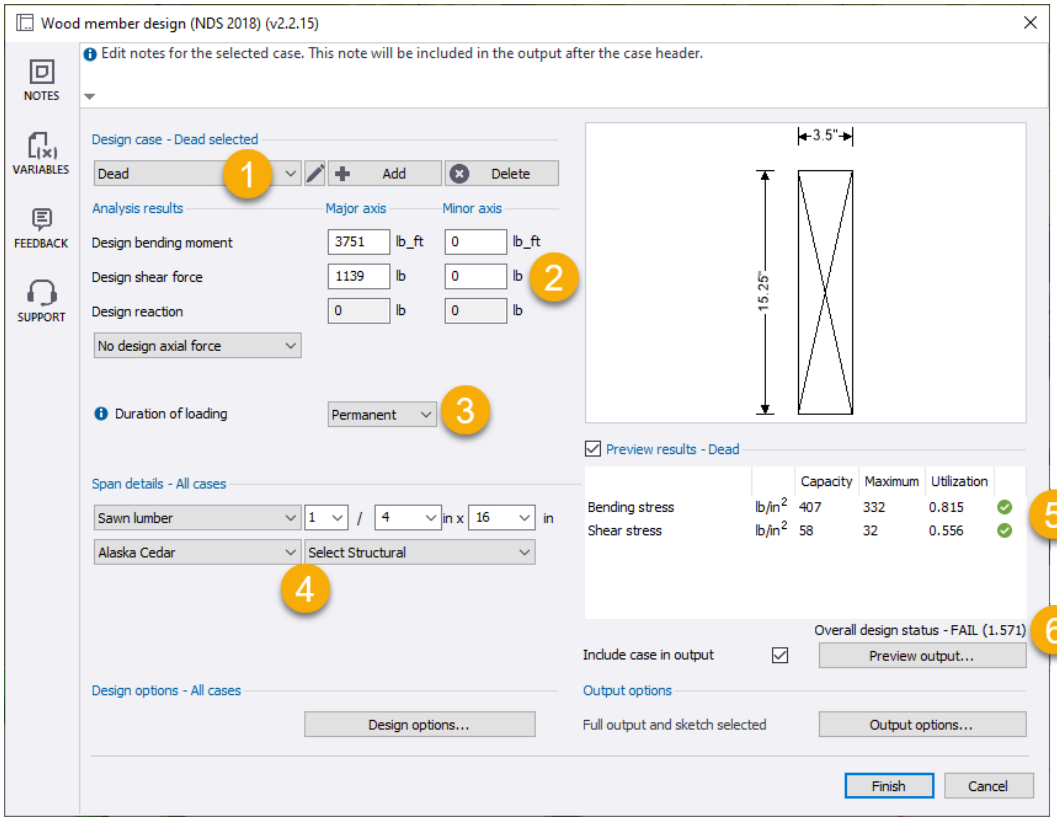
Once the analysis has been performed, a design using Tekla Tedds can be initiated from the Project Workspace or any graphical view of the model.

You can choose to design a member, group, selection, or all timber members in the model.

- Design a selection (page 393)
- Design a group (page 393)
- Design a grouped member (page 395)
- Design an ungrouped member (page 396)
- Design model (page 396)

NOTE Design of a selection provides a useful means to design all members of a truss - which can be selected with one click - in one operation.

When the design is initiated a Tedds calculation interface appropriate to the timber design code being worked to is displayed (the NDS ASD version being shown below).



The main areas within the interface are:

1. Drop list for selecting the design case (page 402).

2. Design forces for selected design case.

3. Duration of loading (for NDS ASD and EC), or time effect factor (NDS LRFD) for the selected design case.

4. Timber section details for the span under consideration.

5. Design pass/fail status and individual check results and utilization ratios for the selected design case.

6. Overall design status and utilization ratio.

You can make changes in the interface - such as to section size and/ or grade - and the design results are immediately refreshed.

When you click **Finish**, all the data you input and changes you make are updated to and stored in the model.

Check the design after changes

If changes are made to the model you can re-analyze and then run a 'check' design to determine if the existing sections are still sufficient. You can conrtinue to perform as many design and analysis loops as necessary so that all members are passing. A check is quicker to perform than a design because the Tedds calculation runs in the background without having to display the Tedds calculation dialog.

You can check the whole model from the **Design** ribbon, or you can make a graphical selection, then right-click to open the context menu listing the available check options. You can also run checks from the Project Workspace.

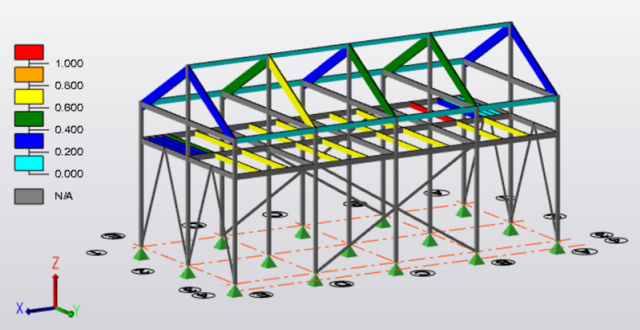
- Check model (page 396)
- Check a selection (page 396)
- Check a member (page 397)
- Check a group (page 397)

The updated utilizations can then be reviewed graphically in a Review View, or as a table in a Design Summary Tabular Data View as described in the next section.

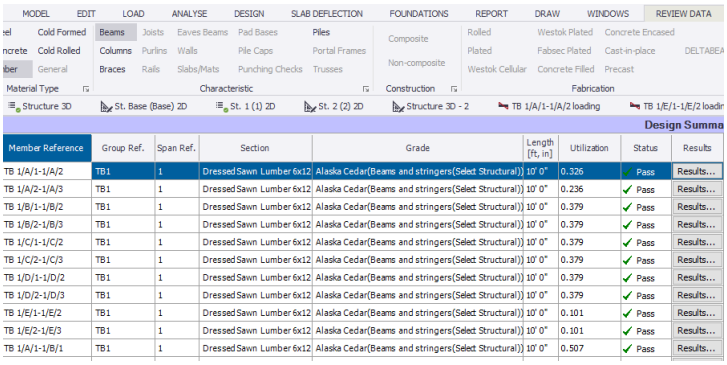
Output the calculations

The Tekla Tedds design results are returned to the Tekla Structural Designer model.

At this stage you can graphically display member design status and utilization ratios from a Review View:

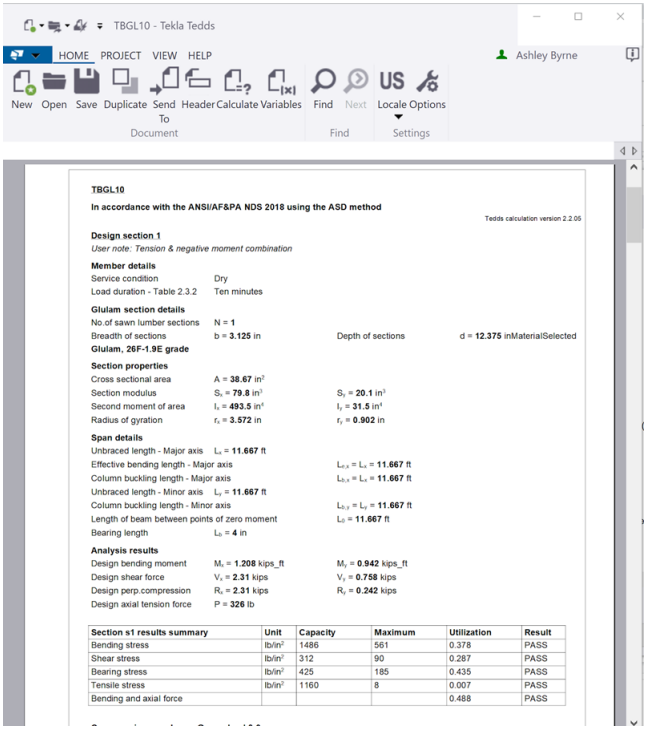


From the **Review** ribbon, by clicking **Tabular Data** and making appropriate selections you can also display the design status table in a Design Summary Tabular Data View:



Member Reference	Group Ref.	Span Ref.	Section	Grade	Length [ft, in]	Utilization	Status	Results
TB 1/A/1-1/A/2	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.326	Pass	Results...
TB 1/A/2-1/A/3	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.236	Pass	Results...
TB 1/B/1-1/B/2	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.379	Pass	Results...
TB 1/B/2-1/B/3	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.379	Pass	Results...
TB 1/C/1-1/C/2	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.379	Pass	Results...
TB 1/C/2-1/C/3	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.379	Pass	Results...
TB 1/D/1-1/D/2	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.379	Pass	Results...
TB 1/D/2-1/D/3	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.379	Pass	Results...
TB 1/E/1-1/E/2	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.101	Pass	Results...
TB 1/E/2-1/E/3	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.101	Pass	Results...
TB 1/A/1-1/B/1	TB1	1	Dressed Sawn Lumber 6x12	Alaska Cedar(Beams and stringers(Select Structural))	10' 0"	0.507	Pass	Results...

Detailed Tekla Tedds calculations for timber members do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds.



Design timber members using Tekla Tedds

Design a selection

To design several timber members or groups in one go:

1. Select all the timber members you want to design.
2. Right click and select Design using Tekla Tedds> Selection

   Tekla Structural Designer highlights the first member or group in the selection and a Tedds dialog opens displaying the design forces at the member's first design section.

   NOTE If grouped design is active, a single grouped design is performed for each group included in the selection using critical design forces established from all members in the group (irrespective of whether or not they were included in the selection). At the end of the process all members in each designed group are checked, (irrespective of whether or not they were included in the selection).

3. Using the results preview, design the member's first design section for its critical design forces, then if there is a second section design that also.
4. When all sections are satisfactory, click **Finish**.

   Tekla Structural Designer highlights the next member or group in the selection and the Tedds dialog re-opens to allow it to be designed.

   Continue in the same way until all the selected members/groups have been designed.

   NOTE If a section is changed during the design to a size that does not exist in Tekla Structural Designer's Timber Section database, it will be automatically added to the local section database that exists for the current model only.

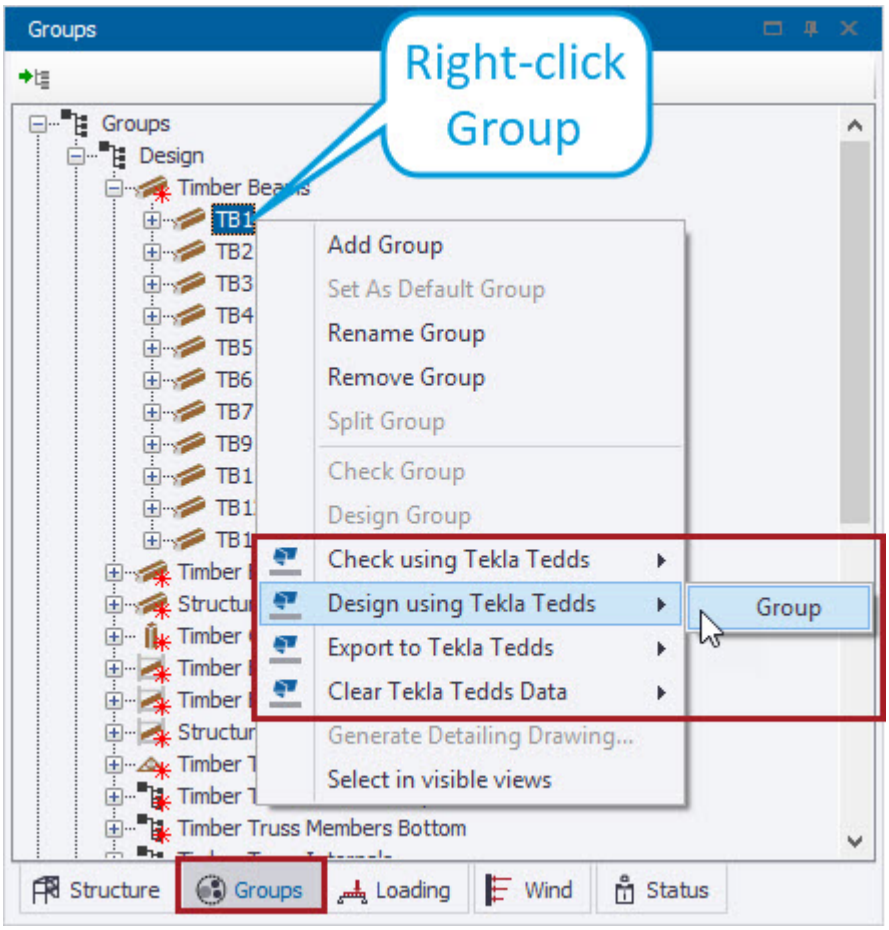
Design a group

If you have activated timber member design groups, each group can be designed as follows:

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, expand **Timber Beams**, **Timber Columns**, or **Timber Braces** as required.

NOTE When designing timber brace pairs, the groups for these are always located under **Timber Braces** and not **Timber X Braces, A Braces, V Braces, K Braces.** These latter groups are used for editing purposes only, but not design.

3. In the **Design** tree, right-click the timber member group you want to design.
   a. It can be useful at this point to click **Select in visible views** to confirm all the group members.
4. In the context menu, select Design using Tekla Tedds> Group.



This design gathers the analysis results for all members in the group and collates them into one set of critical design forces. (In effect assuming that all the worst case loads are happening on one member simultaneously.)

5. Using the results preview, design the first design section in the group for its critical design forces, then if there is a second section design that also.
6. When all sections are satisfactory, click **Finish**.

If the section size was changed during the design, all members of the group are updated to the new section.

Irrespective of whether the section has been changed or not, all members in the group are then automatically checked against only the loads that they see individually. A pass fail status and utilization ratio is calculated accordingly for each one.

7. In the Review View, review the utilization ratios for all members in the group - if these indicate an efficient design, the design can stop at this point.

NOTE For columns in particular, the envelope of critical design forces applied to members of the group can be overly conservative - e.g. if some columns are loaded about one axis, and some loaded about the other, they are all designed as if loaded about *both* axes.

If the utilizations indicate the existing grouping is not very efficient, you can investigate alternative grouping arrangements by adding extra groups as necessary and re-allocating members between the groups. You might also consider individually designing grouped members (as described below) when optimizing the groups.

Design a grouped member

If after running a grouped design some group members have a lower than desired utilization, you can re-design them individually to investigate the effect optimizing them would have on the rest of the group.

1. Highlight a group member which has a lower than desired utilization, right click and select Design using Tekla Tedds> Member

   The analysis results of the selected member are used to establish the set of critical design forces.

2. Optimize the design of the member for its forces.
3. Click **Finish**.
4. If the section size was changed during the design, all members of the group are updated to the new section and checked against only the loads that they see individually.
5. In the Review View, review the new utilizations for the group members - note that some of the group might now be failing.
6. Using the new utilizations to better inform you choice, add extra groups as necessary and re-allocating members between the groups.
7. Run Design using Tekla Tedds> Group for each of the new groups.
8. Iterate the process as necessary until the groups are fully optimized.

Design an ungrouped member

If you have elected not to make use of design groups, each member can be designed individually as follows:

1. Highlight the member, right click and select Design using Tekla Tedds> Member
2. Tekla Structural Designer highlights the member and a Tedds dialog opens displaying the design forces at the member's first design section.
3. Using the results preview, design the member's first design section for its critical design forces, then if there is a second section design that also.
4. When all sections are satisfactory, click **Finish**.
5. If the section size was changed during the design, it is updated to the new section in the Tekla Structural Designer model.
6. The member status and utilization are displayed in the member tooltip.

Design model

If you want to design every timber (and precast) member in the model in one go:

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**

   At the end of the process the status and utilization of each member is displayed in a Review View.

Check timber members using Tekla Tedds

Check model

To check all the existing Tedds member designs,

1. Click Check In Tedds from the **Design** tab.

   This reruns all the Tedds calculations in the background using the latest analysis results.

Check a selection

To check several members or groups in one go,

1. Select the members or groups you want to check.

2. Right click and select **Check using Tekla Tedds> Selection**

   The Tedds calculations for the selected members or groups run in the background using the latest analysis results.

Check a member

To check a single member,

1. Highlight the member you want to check.
2. Right click and select **Check using Tekla Tedds> Member**

   The Tedds calculations for the selected member runs in the background using the latest analysis results.

   The updated status and utilization are displayed in the member tooltip.

Check a group

To check a member group,

1. Highlight a member in the group you want to check.
2. Right click and select **Check using Tekla Tedds> Group**
   - The Tedds calculations for the selected group run in the background using the group critical design forces.
   - All members in the group are then checked against their individual design forces.

5.3 Timber member design groups

Design Groups are automatically created for Timber members and listed in the Project Workspace Groups Tree. The resulting groups can be reviewed and customized as necessary.

Group design is activated by default for timber beams, columns, and braces but can be deactivated if required.

Why use timber design groups?

Timber beams, columns and braces are put into groups for two reasons:

1. For editing purposes - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.
2. For design -
   a. to standardize designs,

b. to reduce the volume of output created,

c. to speed up the design process, particularly so in the case of medium to large size models.

When groups are used for design:

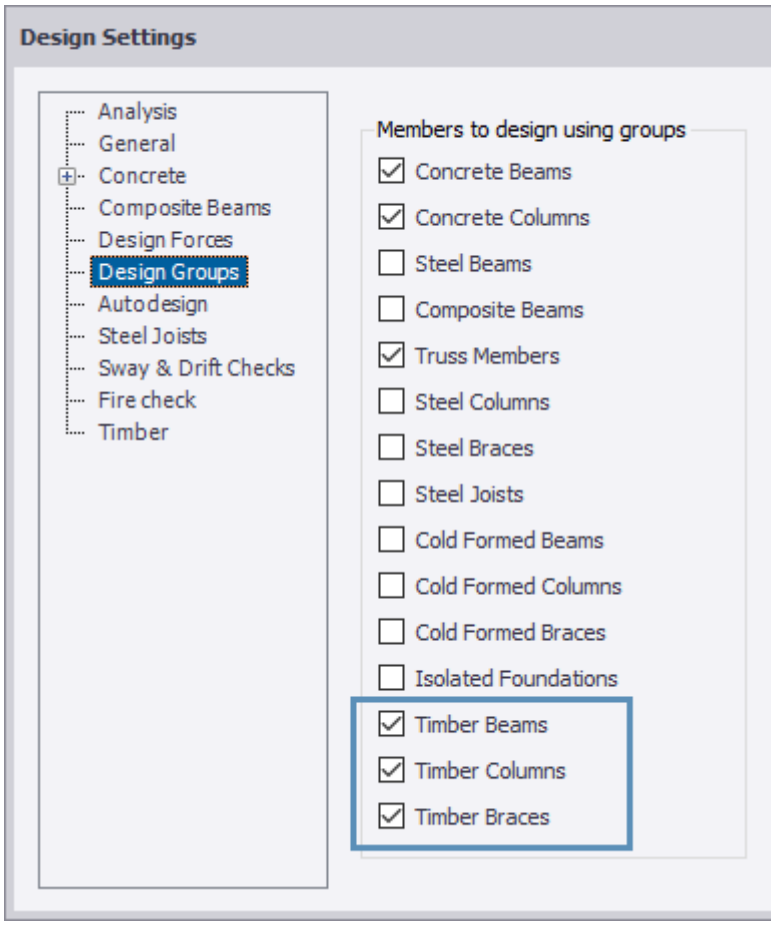
- An envelope of group design forces is passed to a single Tedds calculation.
- User can manipulate design options, member size, material etc.
- Any changes are applied to each member in group.
- Each individual member of group is checked against individual loads without any further user interaction.

NOTE Although timber members are automatically grouped for ease of editing, the use of groups for design is optional and can be deactivated if required: From the Design tab, click Settings> Design Groups, then select or unselect the member types to be designed in groups.

Activating timber member design groups

Timber beam, column and brace design groups are activated as follows:

1. From the **Design** tab, click Settings> Design Groups
2. Select **Timber Beams**, **Timber Columns**, **Timber Braces** as required.



Group management

Automatic Grouping

Timber beams, columns and braces are grouped automatically according to a fixed set of requirements, including section size, grade, and member length.

In Model Settings > Grouping the user defined maximum length variation is used to control whether elements are of sufficiently similar length to be considered equivalent for grouping purposes.

Manual/Interactive Grouping

After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

See:

NOTE When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is

then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.

Regroup Members

If you have made changes in the model that impact on grouping of a specific member type, you can update the affected groups from the Groups tab of the Project Workspace by right-clicking **Timber Beams**, **Timber Columns**, or **Timber Braces** as required and selecting **Regroup Members**.

NOTE Any manually applied grouping will be lost if you elect to re-group!

Regroup ALL Model Members

If you have made changes in the model that impact on grouping, you can update all affected groups accordingly from the Groups tab of the Project Workspace, by clicking Re-group ALL Model Members. (Located at the top of the Groups tab.)

NOTE Any manually applied grouping will be lost if you elect to re-group!

Timber design group requirements

Timber member design groups are formed according to the following rules:

Member type	Design group rules
Timber beam	• A beam may be in only **one** design group. • All beams in the group must have an identical cross section. • All beams in the group must have an identical grade. • All beams in the group must have an identical number of spans. • Individual span lengths must be the same across the group.
Timber column	• A column may be in only **one** design group. • All columns in the group must have an identical cross section. • All columns in the group must have an identical grade. • All columns in the group must have an identical number of stacks. • Any splices must be located in the same stacks for all columns of the group.

Member type	Design group rules
	• Individual stack lengths must be the same across the group.
Timber brace	• A brace may be in only **one** design group. • All braces in the group must have an identical cross section. • All braces in the group must have an identical grade. • All braces in the group must have an identical span length.

5.4 Timber combination sets and strength design cases

Load combinations are automatically divided into groups (called **timber combination sets**) in order to create **strength design cases** with appropriate load duration factors (or time effect factors if working to NDS LRFD) for use in the Tedds calculation.

NOTE For timber beams, **deflection design cases** can also be created. These are described elsewhere, see: Timber beam deflection checks - US (page 403) and Timber beam deflection checks - EC (page 408).

Timber combination sets

Each timber combination set is created based on the load type occurring in the combinations according to the hierarchy defined in the table below.

Timber combination set	Load type included (EC)	Load type included (US)
Seismic set	Seismic	Seismic
Wind set	Wind	Wind
Temperature set	Temperature	Temperature
Snow set	Snow	Snow
Imposed set (EC only) / Live set (US only)	Imposed, Imposed (Other), Roof Imposed	Live, Live (Other), Roof Live
Settlement set	Settlement	Settlement
Dead set	Dead, Slab Wet	Dead, Slab Wet

NOTE • Dead load type includes ‘self weight’ and ‘slab dry’ load cases

- Snow load type includes all types of snow and ice loads; volumes, drifts, ice
- Seismic load type includes ELF and RSA

As load combinations are included in a set, only the remaining load combinations are considered in creating additional sets.

As an example, the following four load combinations:

- **LC1: Dead**
- **LC2: Dead + Temperature**
- **LC3: Dead + Live**
- **LC4: Live + Wind**

would result in the following four timber combination sets:

- **Wind set: LC4**
- **Temperature set: LC2**
- **Live set: LC3**
- **Dead set: LC1**

The factor associated with load duration (Load duration factor - NDS ASD and EC, Time effect factor - NDS LRFD) is typically the same for all combinations in a timber combination set. If this is not the case the critical value will be used. i.e. the shortest time duration for the load duration factor (NDS ASD, and EC), or largest value for the time effect factor (NDS LRFD).

You can generate envelopes that correspond to the active timber combination sets so that the design forces can be easily reviewed. For details, see: Generate timber design envelopes (page 387).

Strength design cases

Timber combination sets relate directly to strength design cases in Tedds. The maximum member forces from all of the combinations in a timber combination set will create at least one design case in Tedds. When a member sees both tension and compression within a timber combination set, two design cases are created for those sets.

As an example, if a member had timber combination sets for **Seismic**, **Wind**, **Live** and **Dead**, and the member saw both tension and compression under seismic and wind sets, six design cases would be created as follows:

- **Seismic: Tens**
- **Seismic: Comp**
- **Wind: Tens**
- **Wind: Comp**

- **Live**
- **Dead**

NOTE For glulam members designed to NDS, more design cases will be created when either tension and compression or positive and negative moment forces are seen across a member or group in a timber combination set.

5.5 Timber beam deflection checks - US

When designing to US codes, timber beam deflection checks can be requested for:

- Dead, Live, Snow, Wind, and Total loads for instantaneous deflections
- Total load deflections can optionally consider the effects of creep deformation in design

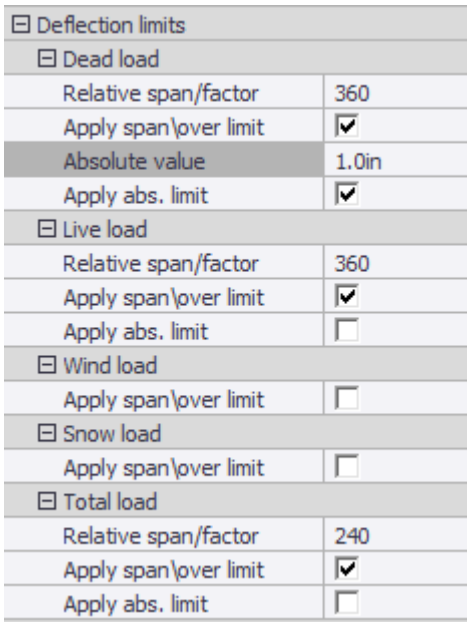
Specify deflection checks

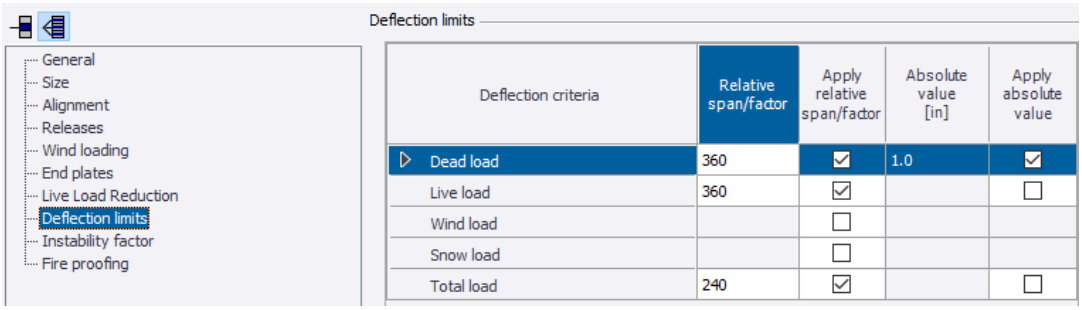
By default the checks are applied as follows:

- Dead - relative span/factor limit of 360 applied
- Live - relative span/factor limit of 360 applied
- Wind - not applied
- Snow - not applied
- Total loads - relative span/factor limit of 240 applied

You can modify the deflection checks you require in either the beam properties window or dialog. Limits can be defined as a span over factor limit, an absolute value limit or both.

The images below show the beam properties window and the dialog respectively after an additional Dead load absolute limit of 1in has been applied.

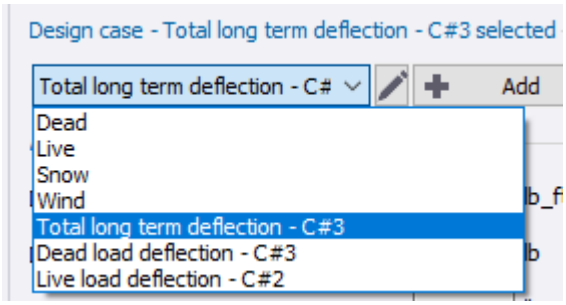




Deflection design cases

When a beam with deflection limits applied is designed using Tedds, a separate deflection design case is created for each deflection check that has been specified.

The deflection load type and the load combination number (-C#XX) with the controlling deflection is identified in each deflection design case name.

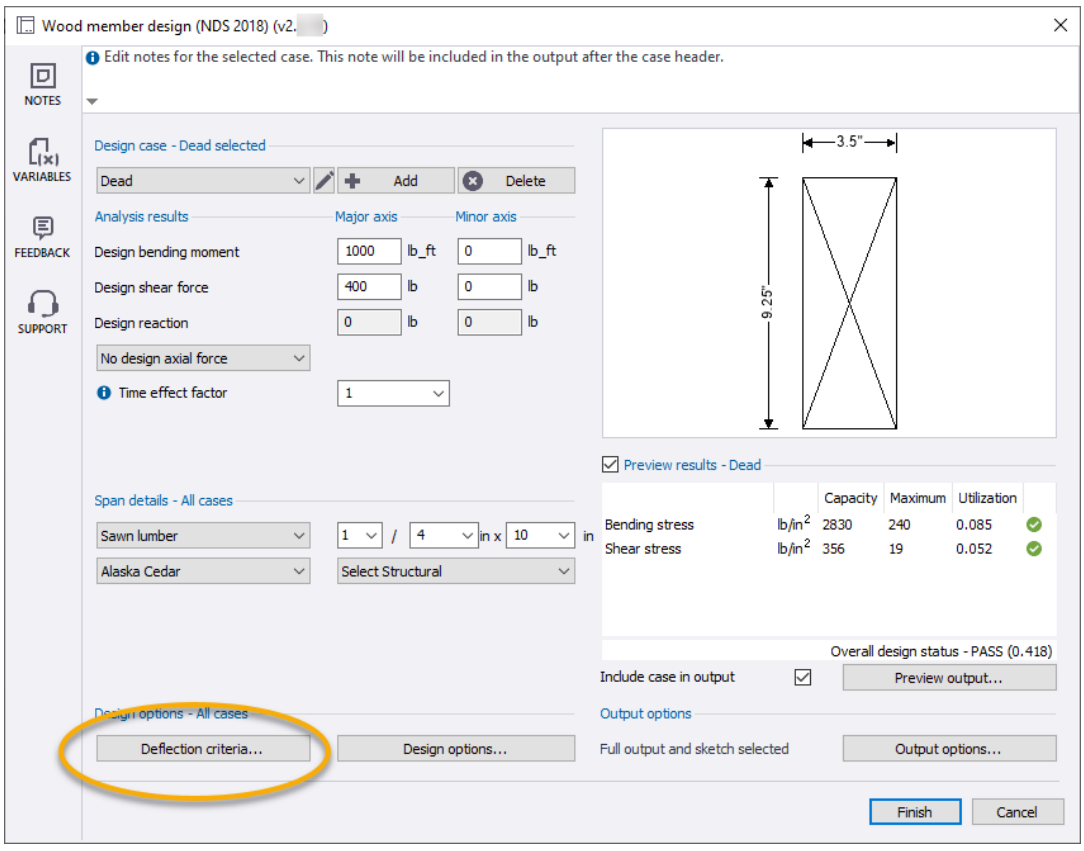


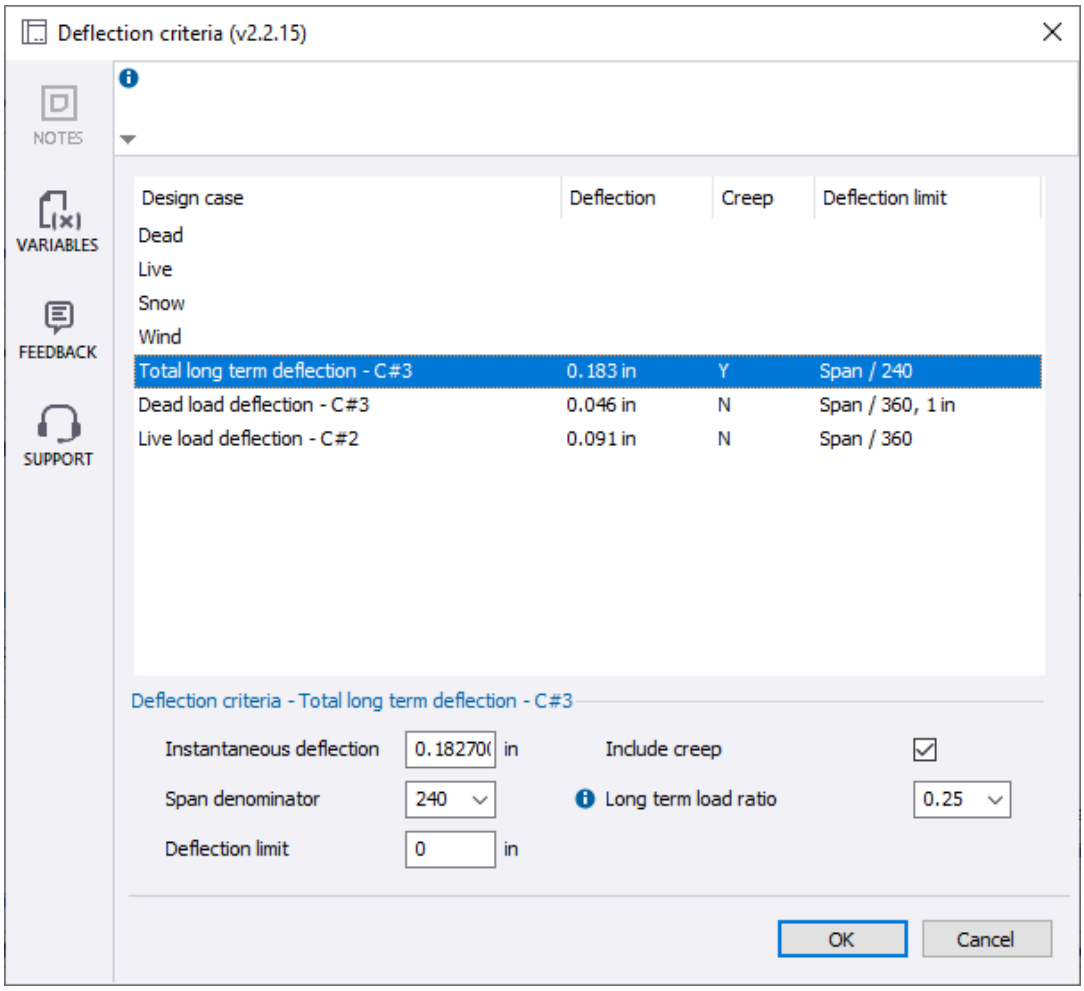
NOTE When the calculated deflections are very small, you might decide that checks are not required. In this situation you can choose to specify a lower limit beneath which checks are not created. This is achieved by specifying an **Ignore deflections for reporting** value on the Design Forces page in Design Settings.

For each deflection design case the controlling deflection is determined from the current analysis results for active combinations only, using service load factors.

- Individual loadcase deflections are taken at the location of maximum deflection for the combination.
- The deflection values are taken as the absolute value of the maximum positive and negative major axis relative deflections.
- The deflection for each load type for a single combination is the sum of all related loadcase deflections factored by the associated load combination factor. The maximum deflection across all load combinations is taken as the controlling value.

When deflection design cases exist, a **Deflection criteria** button is added to the Tedds interface to allow their properties to be viewed and managed.

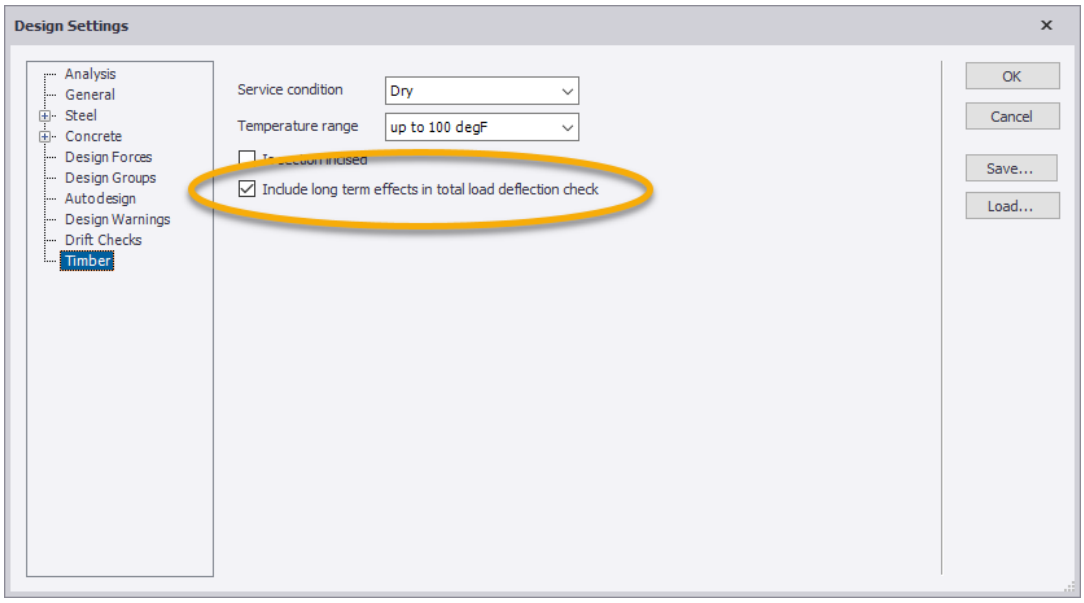




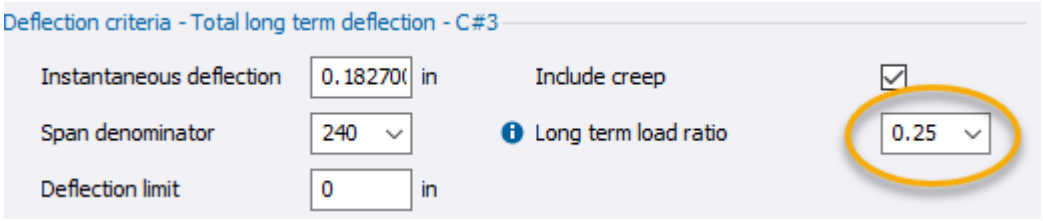
Long term deflections due to creep effects

Tekla Structural Designer automatically determines the effects of creep and considers the long term deflection in the total load deflection check.

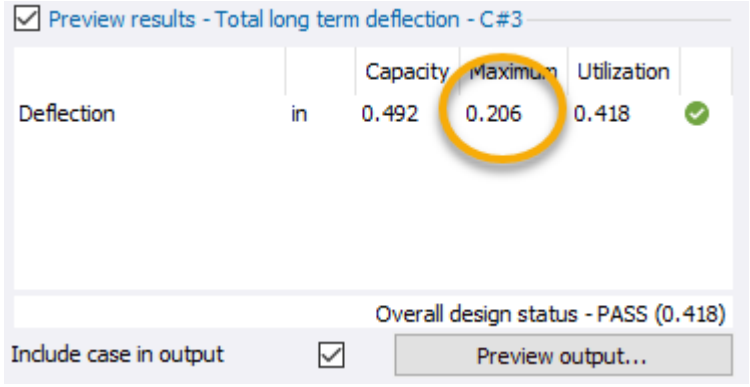
You can choose to exclude these long term effects by clearing the 'Include' option on the Timber page in Design Settings.



Tekla Structural Designer automatically determines the long term load ratio.



Final deflections are determined in Tedds and displayed in the preview results as pictured below.



When long term effects are included, only gravity combinations are considered when determining the critical combination, whereas for the instantaneous total load deflection check both gravity and lateral combinations would be considered.

Automatic adjustment of design deflection values

You can choose to apply Modification Factors to the modulus of elasticity and moment of inertia values in order to create the correct distribution of forces in a model.

The relative deflection values reported in the **Load Analysis View**, (which are determined using $\mathbf{E_{analysis}}$), take these modification factors into account.

When deflection checks are created in the Tedds calculation Tekla Structural Designer automatically adjusts the values so that they are based on unfactored modulus of elasticity and moment of inertia values. They are also further adjusted to use $\mathbf{E_{mean}}$ rather than $\mathbf{E_{analysis}}$ as the $\mathbf{E_{mean}}$ value is typically recommended for serviceability checks under normal conditions.

5.6 Timber beam deflection checks - EC

When designing to Eurocodes, timber beam deflection checks can be requested for:

- Dead, Imposed, Wind, Snow and Total loads for instantaneous deflections
- Total load deflections can optionally consider the effects of creep deformation in design

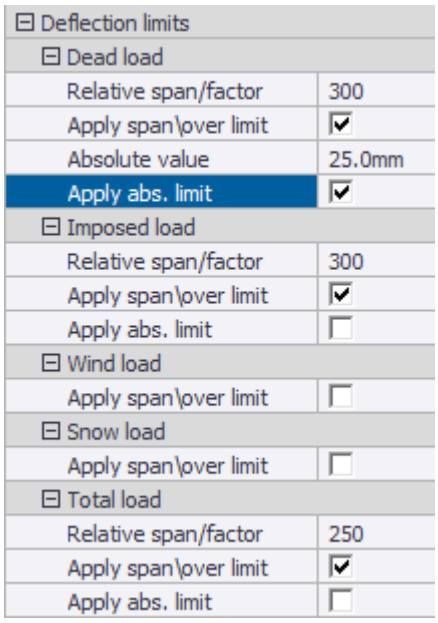
Specify deflection checks

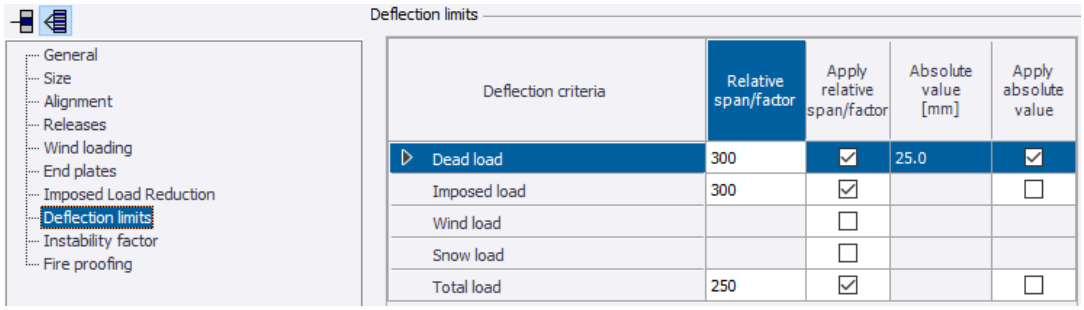
By default the checks are applied as follows:

- Dead - relative span/factor limit of 300 applied
- Imposed - relative span/factor limit of 300 applied
- Wind - not applied
- Snow - not applied
- Total loads - relative span/factor limit of 250 applied

You can modify the deflection checks you require in either the beam properties window or dialog. Limits can be defined as a span over factor limit, an absolute value limit or both.

The images below show the beam properties window and the dialog respectively after an additional Dead load absolute limit of 25mm has been applied.

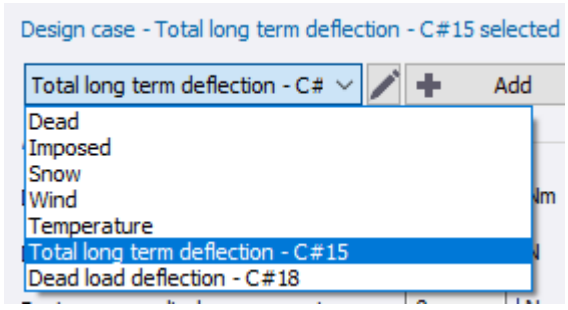




Deflection design cases

When a beam with deflection limits applied is designed using Tedds, a separate deflection design case is created for each deflection check that has been specified.

The deflection load type and the load combination number (-C#XX) with the controlling deflection is identified in each deflection design case name.

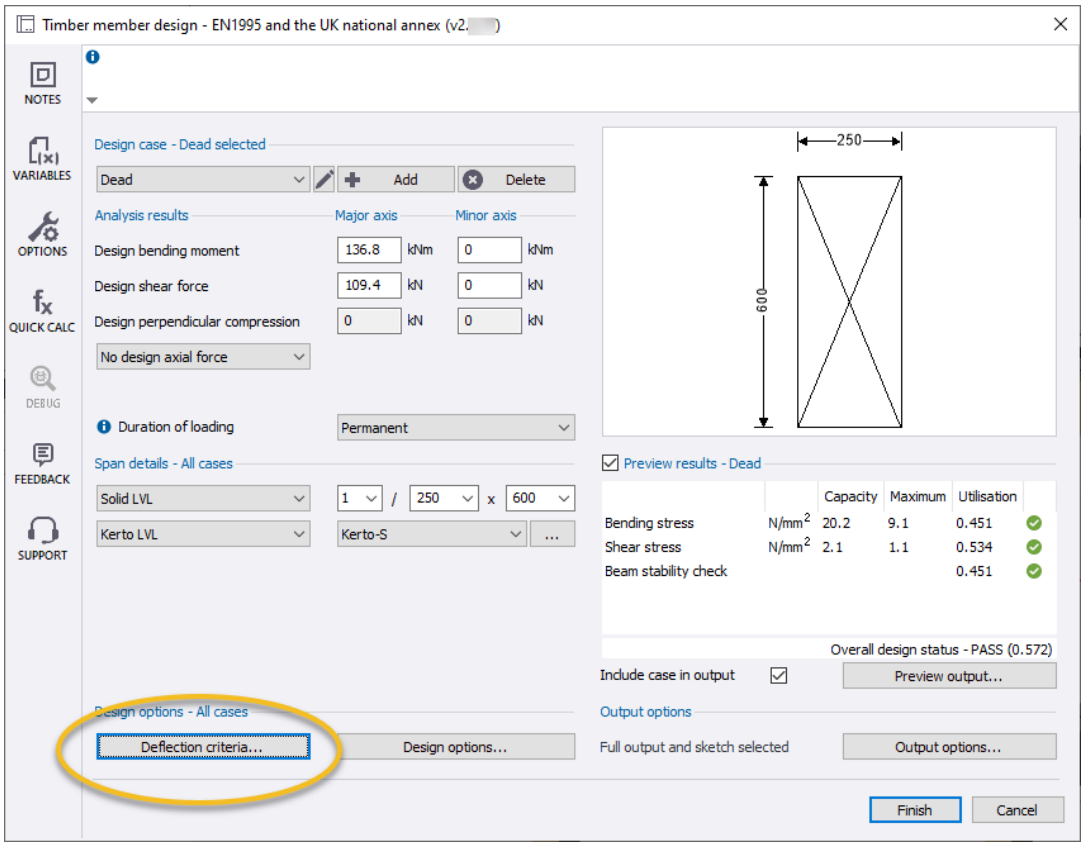


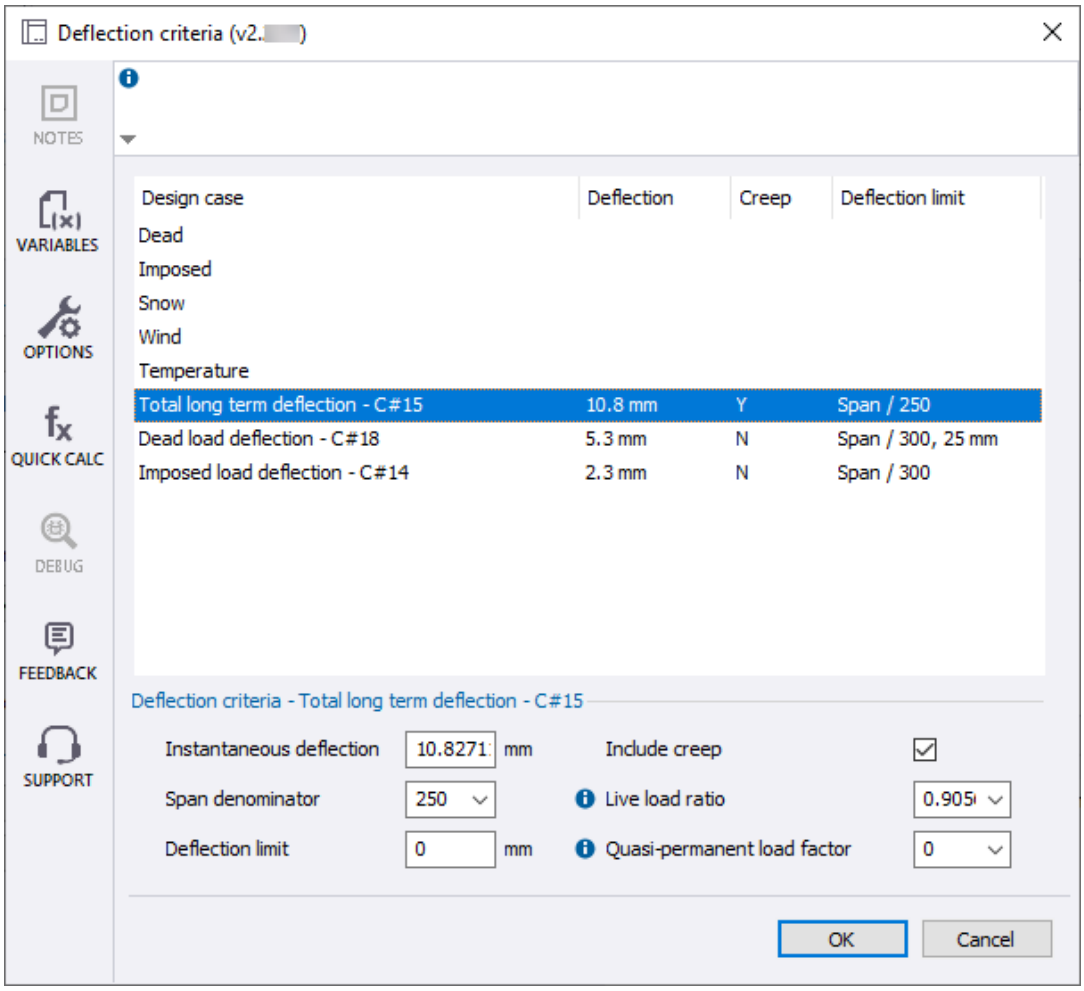
NOTE When the calculated deflections are very small, you might decide that checks are not required. In this situation you can choose to specify a lower limit beneath which checks are not created. This is achieved by specifying an **Ignore deflections for reporting** value on the Design Forces page in Design Settings.

For each deflection design case the controlling deflection is determined from the current analysis results for active combinations only, using service load factors.

- Individual loadcase deflections are taken at the location of maximum deflection for the combination.
- The deflection values are taken as the absolute value of the maximum positive and negative major axis relative deflections.
- The deflection for each load type for a single combination is the sum of all related loadcase deflections factored by the associated load combination factor. The maximum deflection across all load combinations is taken as the controlling value.

When deflection design cases exist, a **Deflection criteria** button is added to the Tedds interface to allow their properties to be viewed and managed.

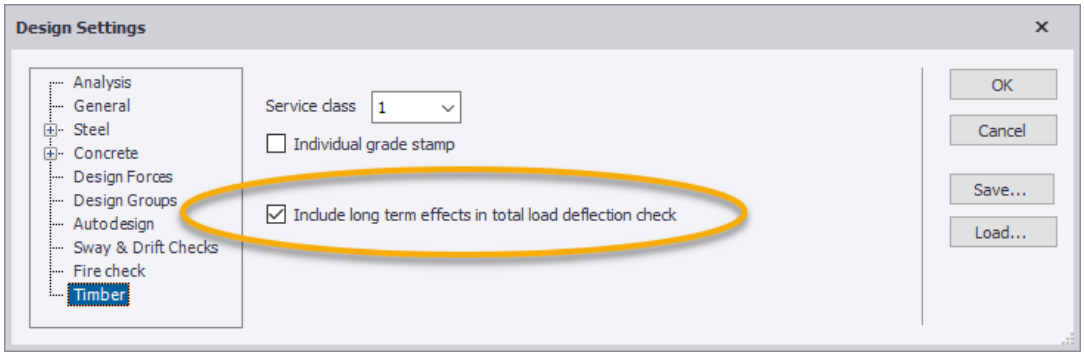




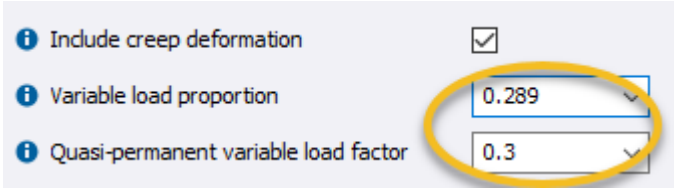
Long term deflections due to creep effects

Tekla Structural Designer automatically determines the effects of creep and considers the long term deflection in the total load deflection check.

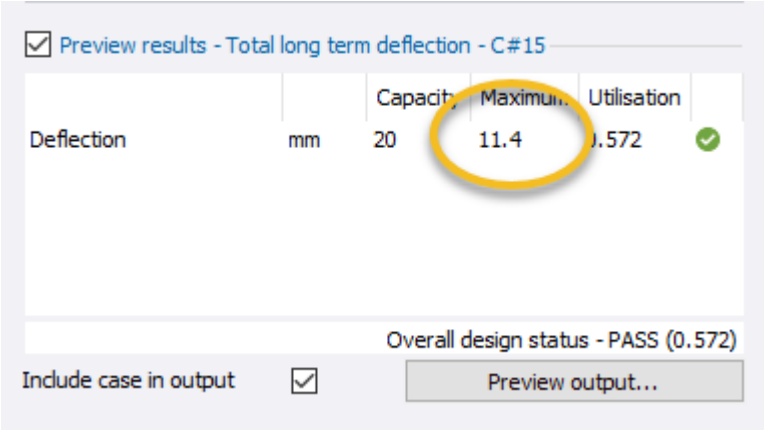
You can choose to exclude these long term effects by unselecting the 'Include' option on the Timber page in Design Settings.



Tekla Structural Designer automatically determines the variable load proportion and the quasi-permanent variable load factor. If more than one type of variable load is included in a load combination the largest associated value is used for the quasi-permanent variable load factor.



Final deflections are determined in Tedds and displayed in the preview results as pictured below.



When long term effects are included, only gravity combinations are considered when determining the critical combination, whereas for the instantaneous total load deflection check both gravity and lateral combinations would be considered.

Automatic adjustment of design deflection values

You can choose to apply Modification Factors to the modulus of elasticity and moment of inertia values in order to create the correct distribution of forces in a model.

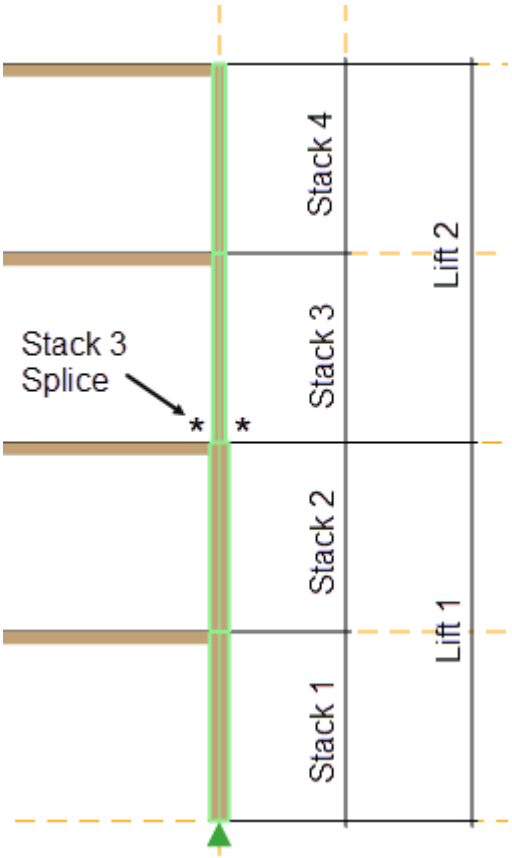
The relative deflection values reported in the **Load Analysis View**, (which are determined using $\mathbf{E_{analysis}}$), take these modification factors into account.

When deflection checks are created in the Tedds calculation Tekla Structural Designer automatically adjusts the values so that they are based on unfactored modulus of elasticity and moment of inertia values. They are also further adjusted to use $\mathbf{E_{mean}}$ rather than $\mathbf{E_{analysis}}$ as the $\mathbf{E_{mean}}$ value is typically recommended for serviceability checks under normal conditions.

5.7 Timber column stacks and lifts

Timber columns are automatically split into stacks at construction levels provided at least one member or slab connects to the column at that level.

Where a change in cross section is required between stacks you should indicate a splice in the stack properties before initiating the column design.



Each splice insertion introduces a new column lift above the splice position.

A lift can either comprise of a single stack, or multiple stacks, depending on the defined splice locations.

Design of a single stack lift

The Tedds calculation for a single stack lift is based on the stack length. The forces used in the design will depend on the command selected as follows:

- **Design using Tekla Tedds > Group**

  uses the GROUP forces in each direction within the stack length.

- **Design using Tekla Tedds > Member**

  uses the MEMBER forces in each direction within the stack length.

Design of a multi-stack lift

Design of a multi-stack lift is performed using a single Tedds calculation for the whole lift. The major and minor axis unbraced lengths are taken from the longest stack in the lift.

The following sets of forces are established for use in the design:

- Stack MEMBER forces - maximum forces in each direction in the stack.
- LIFT forces - maximum forces in each direction in the lift.
- GROUP forces - maximum forces in each direction in the lift from all members in the group.

The design process adopted will depend on whether column grouping is enabled and also on the command selected for the design as follows:

Ungrouped design

- **Design using Tekla Tedds > Member** (with an individual stack selected)
  1. The lift containing the selected stack is designed for its LIFT forces.
  2. All stacks in the lift are then checked for their individual stack MEMBER forces.
- **Design using Tekla Tedds > Member** (with an entire column selected)
  1. All lifts in the column are designed for their LIFT forces.
  2. All stacks in all lifts are then checked for their individual stack MEMBER forces.
- **Design performed from Review Data > Design Summary** (by clicking the Results.. button adjacent to a stack)
  1. The lift containing the selected stack is designed for its LIFT forces.
  2. All stacks in the lift are then checked for their individual stack MEMBER forces

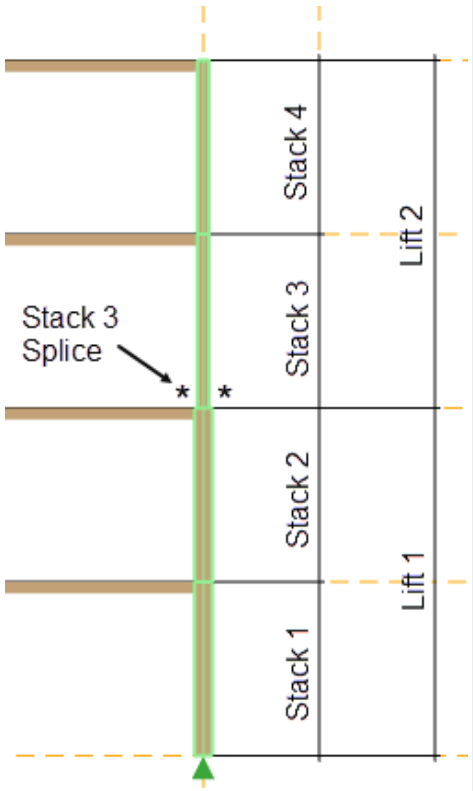
Grouped design

- **Design using Tekla Tedds > Group**
  1. All lifts are designed for the respective lift GROUP forces.
  2. All stacks of all lifts in the group are then checked for their individual stack MEMBER forces.
- **Design using Tekla Tedds > Member** (with an individual stack selected)
  1. The lift containing the selected stack is designed for its LIFT forces.

2. All stacks of this lift in the group are then checked for their individual stack MEMBER forces.
3. The designed lift is also checked for the lift GROUP forces so as to be available for export of group to Tedds.

- **Design using Tekla Tedds > Member** (with an entire column selected)
  1. All lifts in the column are designed for their LIFT forces.
  2. All stacks in all lifts in the group are then checked for their individual stack MEMBER forces.
  3. The designed lift is also checked for the lift GROUP forces so as to be available for export of group to Tedds.
- **Design performed from Review Data > Design Summary** (by clicking the Results.. button adjacent to a stack)
  1. The lift containing the selected stack is designed for its LIFT forces.
  2. All stacks of this lift in the group are then checked for their individual stack MEMBER forces.
  3. The designed lift is also checked for the lift GROUP forces so as to be available for export of group to Tedds.

Example: Column with 2 lifts and 4 stacks



MEMBER unbraced lengths

$unbraced_length_{stack_1}$

$unbraced_length_{stack_2}$

$unbraced_length_{stack_3}$

$unbraced_length_{stack_4}$

Example: Column with 2 lifts and 4 stacks	
MEMBER forces: Stack 1	**MEMBER forces: Stack 2**
$bending_major_{stack_1}$	$bending_major_{stack_2}$
$bending_minor_{stack_1}$	$bending_minor_{stack_2}$
$shear_major_{stack_1}$	$shear_major_{stack_2}$
$shear_minor_{stack_1}$	$shear_minor_{stack_2}$
$axial_{stack_1}$	$axial_{stack_2}$
MEMBER forces: Stack 3	**MEMBER forces: Stack 4**
$bending_major_{stack_3}$	$bending_major_{stack_4}$
$bending_minor_{stack_3}$	$bending_minor_{stack_4}$
$shear_major_{stack_3}$	$shear_major_{stack_4}$
$shear_minor_{stack_3}$	$shear_minor_{stack_4}$
$axial_{stack_3}$	$axial_{stack_4}$
LIFT forces: Lift 1	
$bending_major_{lift_1} = max\ (bending_major_{stack_1}, bending_major_{stack_2})$	
$bending_minor_{lift_1} = max\ (bending_minor_{stack_1}, bending_minor_{stack_2})$	
$shear_major_{lift_1} = max\ (shear_major_{stack_1}, shear_major_{stack_2})$	
$shear_minor_{lift_1} = max\ (shear_minor_{stack_1}, shear_minor_{stack_2})$	
$axial_{lift_1} = max\ (axial_{stack_1}, axial_{stack_2})$	
LIFT unbraced lengths: Lift 1	
$unbraced_length_{lift_1} = max\ (unbraced_length_{stack_1}, unbraced_length_{stack_2})$	
LIFT forces: Lift 2	
$bending_major_{lift_2} = max\ (bending_major_{stack_3}, bending_major_{stack_4})$	
$bending_minor_{lift_2} = max\ (bending_minor_{stack_3}, bending_minor_{stack_4})$	
$shear_major_{lift_2} = max\ (shear_major_{stack_3}, shear_major_{stack_4})$	
$shear_minor_{lift_2} = max\ (shear_minor_{stack_3}, shear_minor_{stack_4})$	
$axial_{lift_2} = max\ (axial_{stack_3}, axial_{stack4})$	
LIFT unbraced lengths: Lift 2	
$unbraced_length_{lift_2} = max\ (unbraced_length_{stack_3}, unbraced_length_{stack_4})$	
GROUP forces: Lift 1	
$bending_major_{lift_1_group}$ = max of $bending_major_{lift_1}$ for all columns in the group	

Example: Column with 2 lifts and 4 stacks
$bending_minor_{lift_1_group}$ = max of $bending_minor_{lift_1}$ for all columns in the group $shear_major_{lift_1_group}$ = max of $shear_major_{lift_1}$ for all columns in the group $shear_minor_{lift_1_group}$ = max of $shear_minor_{lift_1}$ for all columns in the group $axial_{lift_1}$ = max of $axial_{lift_1}$ for all columns in the group
GROUP forces: Lift 2 $bending_major_{lift_2_group}$ = max of $bending_major_{lift_2}$ for all columns in the group $bending_minor_{lift_2_group}$ = max of $bending_minor_{lift_2}$ for all columns in the group $shear_major_{lift_2_group}$ = max of $shear_major_{lift_2}$ for all columns in the group $shear_minor_{lift_2_group}$ = max of $shear_minor_{lift_2}$ for all columns in the group $axial_{lift_2}$ = max of $axial_{lift_2}$ for all columns in the group

6 Masonry wall design handbook

This handbook contains information relating to the design of masonry shear walls in Tekla Structural Designer.

- Masonry wall design - introduction (page 418)
- Masonry wall limitations and assumptions (page 419)
- Masonry wall design workflow (page 421)
- Masonry wall design cases (page 451)

6.1 Masonry wall design - introduction

Masonry walls can be modeled and analyzed in Tekla Structural Designer for all regional codes.

For the US regional code, Tekla Tedds can be used to automatically define material and analytical properties, as well as to perform the design of the masonry wall sections. Result lines associated with a masonry wall panel are checked as part of the panel design and can also be investigated independently in Tekla Tedds from within Tekla Structural Designer.

In the first release there is no particular consideration in modeling, analysis or design, for any other type of masonry wall other than single pane, in-plane loaded masonry walls (typically internal).

Wall types

The following masonry shear wall types can be designed:

- clay masonry - solid, hollow, filled, or partially filled
- concrete masonry - solid, hollow, filled, or partially filled
- unreinforced, or reinforced (vertical and horizontal)
- single skin walls only, double skin walls (i.e. using two layers of masonry) are beyond scope.

Design codes

Currently, design is only available for the US region. The following design code versions are available for both LRFD and ASD methods:

- TMS MSJC 2013
- TMS 402/602 2016
- TMS 402/602 2022

6.2 Masonry wall limitations and assumptions

The following limitations and assumptions apply to the design of masonry shear walls in Tekla Structural Designer:

Limitations

- General
  - Seismic design is not currently considered in masonry wall design.
  - Loads are distributed based on finite element stiffness. As such, specific limitations per TMS 402-22 5.1.1 and 5.2 are not considered.
  - Masonry walls are modeled as homogeneous elements with solid cross sections. As such, shell analysis results may not correspond to actual stresses in a heterogeneous masonry wall construction. Design in Tekla Tedds ensures analysis results are designed appropriately for the defined wall construction.
- Drawings
  - New masonry wall drawing layers are active by default for new models. They are inactive for existing models (created prior to the Tekla Structural Designer 2024 release) which means that masonry walls are not visible by default in drawings if they are added to an existing model. The user needs to remember to activate the respective layers or load the new drawing settings from the active settings set.
  - Reinforcement detailing around openings etc. is the responsibility of the designer.

Assumptions

- General
  - A compatible version of Tekla Tedds must be installed and licensed.
  - The locale (US) must be set correctly in Tekla Tedds before attempting to run the masonry wall design from Tekla Structural Designer.
  - Masonry walls are assumed to primarily resist axial and in-plane shear and bending loads.
- Material definition

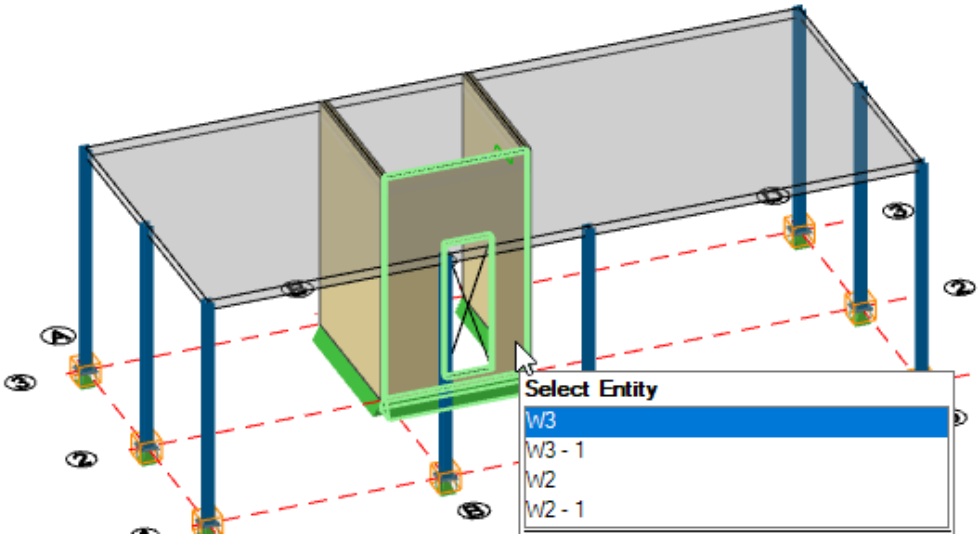
    - Masonry wall material can be defined in the Tekla Structural Designer material database with the assistance of Tekla Tedds. At the material level, all walls are assumed to be fully grouted and isotropic. The calculated material properties consider the specific wall construction defined and property values prescribed by TMS 402/602. Adjustments for grout spacing and/or orthotropic behavior can be made for each panel in the panel properties.
    - Custom masonry materials may be defined without the use of Tekla Tedds. The design feature is not available for these custom materials.
- Analysis
    - At the start of the analytical process, an average density and adjusted orthotropic factors is calculated if required. Both are calculated on a masonry wall panel basis, with the intention of adjusting the wall properties and behavior according to the geometric characteristics of the panel construction.
    - The average density is derived from the geometry and grouting of the masonry wall panel section.
    - The adjusted orthotropic factors will only be calculated automatically if they are active and the option "Apply section adjustment factors" has been selected. Their calculation adjusts analysis properties for average section properties as recommended by *NCMA TEK 14-01B - Section Properties of Concrete Masonry Walls*. The adjustment to Ey (Young's modulus in the Y direction, along the height of the wall) assumes the wall is acting primarily to resist in-plane loads. Calculation of these properties for a specific panel can be viewed in the design documentation accessed via the Export to Tekla Tedds option.
    - Orthotropic analysis properties calculated by Tekla Tedds provide a basic adjustment based on section properties only. Masonry wall stiffness can and should be reviewed and adjusted for specific design applications. Orthotropic analysis adjustments can be made from the panel properties and generally applied modification factors can be applied from the Analysis Settings dialog.
    - Once these properties have been defined they will only be recalculated during the analysis in cases where they become outdated.
- Design
    - The design of masonry wall panels is only available if there are combination results in the model.
    - Result lines associated with masonry wall panels are checked during the design of the wall panel, using the panel design data.
    - The design of a result line can be investigated from the context menu of the result line. This always uses the parent panel design data, and consequently is only available in cases where the associated masonry wall panel has previously been designed.

- There is no result line specific design data saved with the result lines, however the data can be applied automatically to the associated panel at the end of the interactive investigation if necessary.

NOTE For further information relating to the scope, assumptions and limitations, general notes and references specific to the Tekla Tedds calculation itself, please refer to the Masonry wall section design (TMS) calculation in the Tedds calculation library.

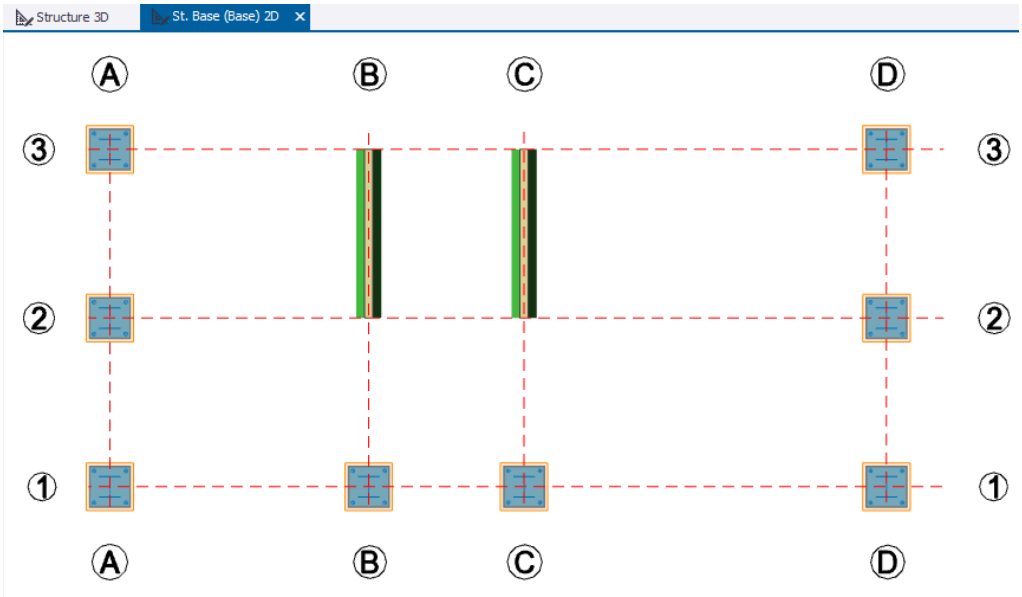
6.3 Masonry wall design workflow

To illustrate the basic workflow for masonry wall design in Tekla Structural Designer, the walls in the below model will be checked/designed using the **TMS 402/602 2022** code and the ASD method.



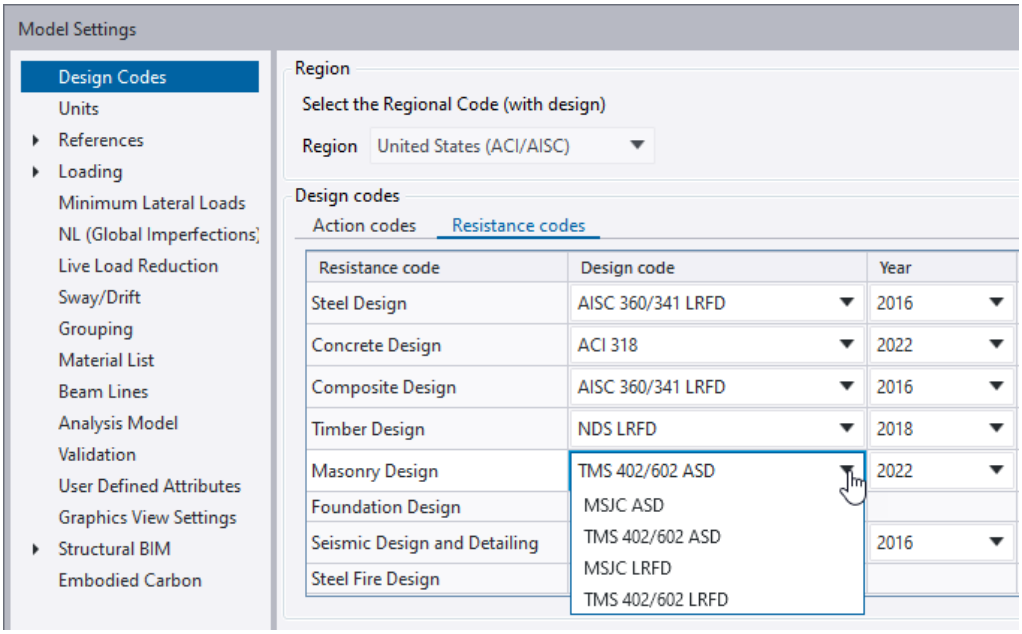
Download and open the tutorial model

1. Download the attachment **TMS.zip** from this page.
2. Extract the tutorial model **TMS 402 602.tsmd** from the zip.
3. Open the model in Tekla Structural Designer.



Choose the masonry design code

1. On the **Home** ribbon, click **Model Settings**.
2. On the **Design Codes** page, click **Resistance Codes**
3. Examine the available **Masonry Design** codes.



As the region is set to **United States (ACI/AISC)** the following design code versions are available for both LRFD and ASD methods:

- MSJC 2013

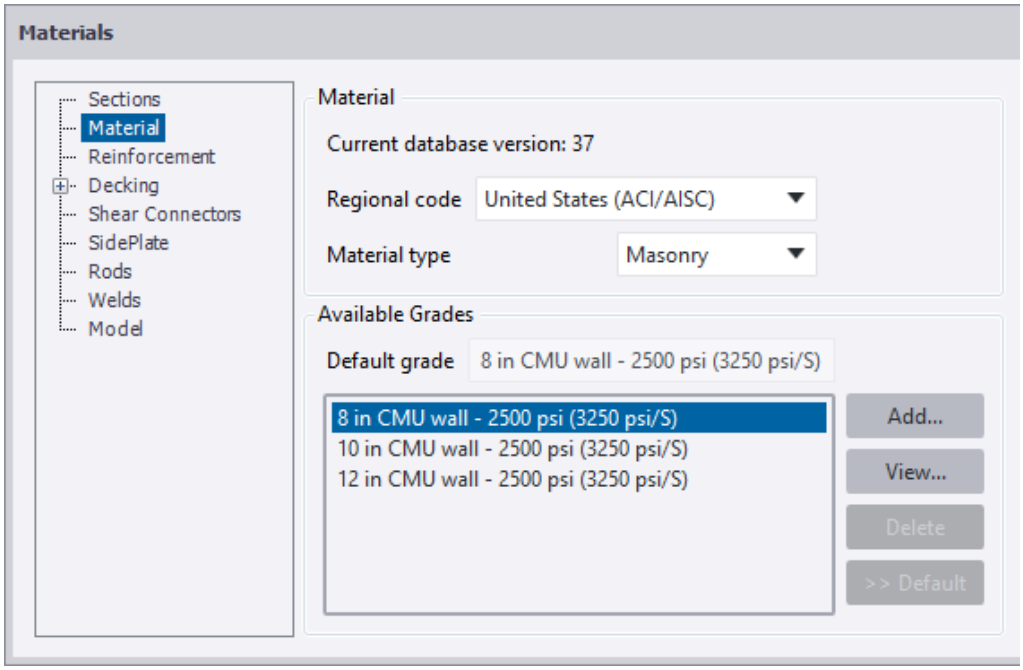
- TMS 402/602 2022
- TMS 402/602 2016

4. Ensure the **TMS 402/602 ASD 2022** code is selected.
5. Click **OK**

View the default masonry wall material grades

The database of available masonry wall grades can be displayed in **Materials** dialog box as follows:

1. On the **Home** ribbon, click **Materials**.
2. On the **Material** page, select the **Masonry** material type.



There are three built-in masonry material grades in Tekla Structural Designer for the US regional code.

The default masonry grade names are obtained as:

- "[Nominal wall thickness] CMU wall-[f'm] ([Unit strength]/[Mortar type])"

DEFINE MASONRY GRADE USING TEDDS	You can easily add new grades to the database using Tedds by following these instructions. When added in this way, the default name is assigned automatically. When editing a grade through Tedds the name is only automatically updated if it has not been previously modified by the user.

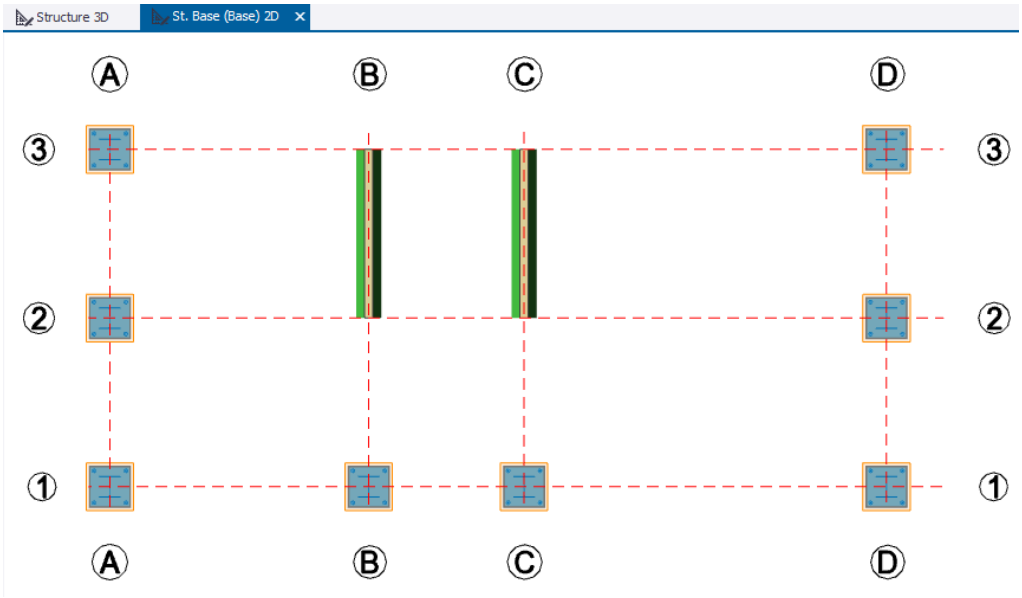
Each new wall you create initially adopts the default grade, although you can change it to any of displayed grades.

3. The walls in this example will use the **8 in CMU wall -2500 psi** grade. No changes are required, so click **Close** to exit.

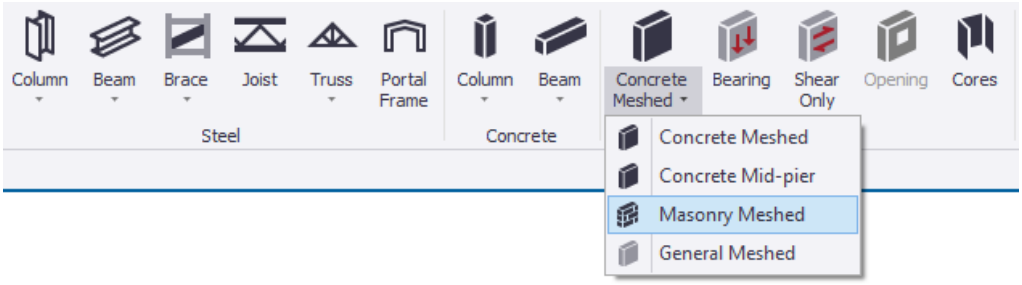
Define masonry wall properties

To define the wall panel data properties before placing the wall, proceed as follows:

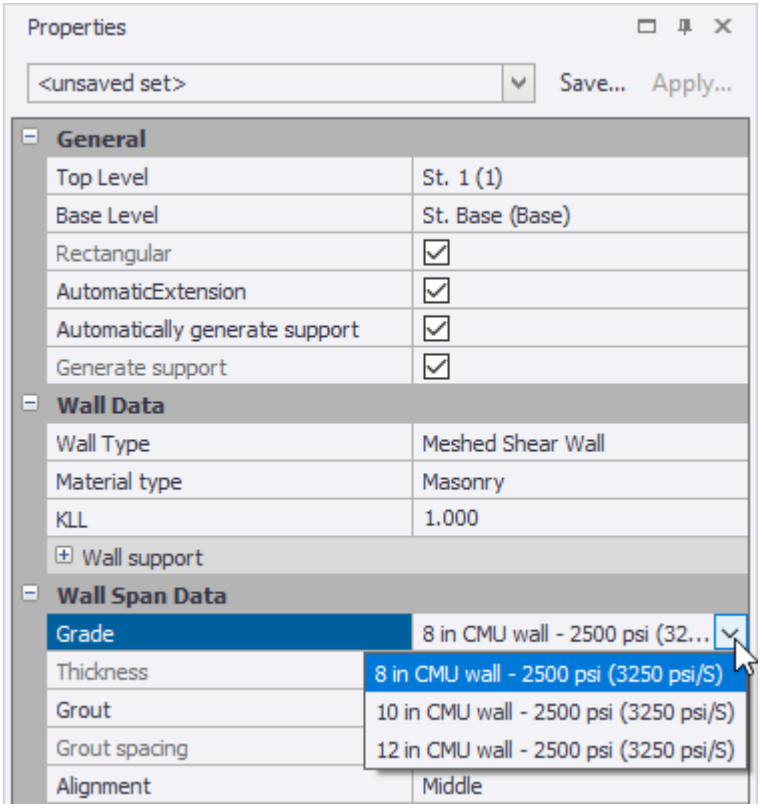
1. Activate the **St. (Base) 2D** view.



2. On the **Model** ribbon, click **Walls > Masonry Meshed**.

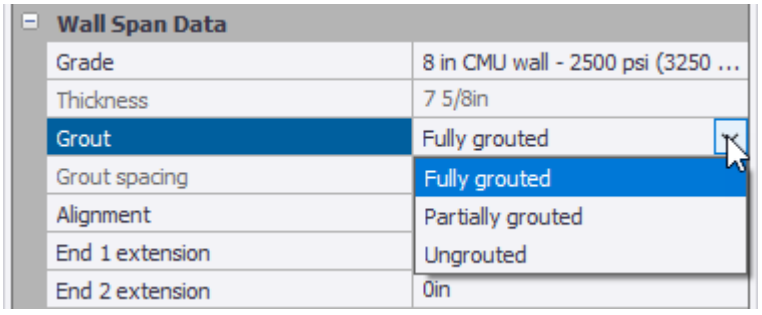


3. In the **Properties** window, select the **Grade** required.



For this example we can accept the default **8 in CMU wall -2500 psi**.

4. Next, select the **Grout** option required.



For this example we can accept the default **Fully grouted**.

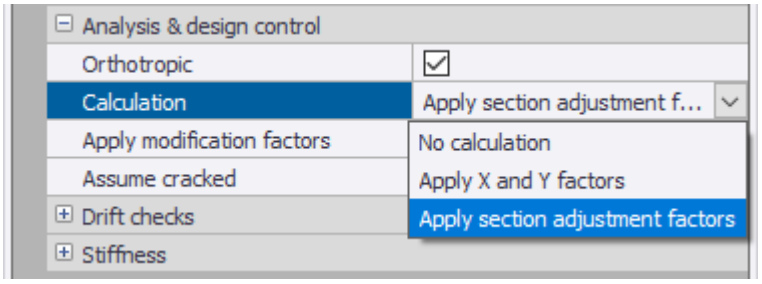
Note that the grout choice affects the subsequent choice of isotropic/ orthotropic properties as follows:

- **Fully grouted** - this is the initial default. With this option the wall defaults to having orthotropic properties disabled. It can also be set to orthotropic if required, by selecting the **Orthotropic** option under **Analysis & Design control**. There shall then be three calculation methods available to the user.
- **Partially grouted** - after selecting this option you should then also specify the grout spacing. With this option the wall is forced to have

orthotropic properties, with the calculation method defaulting to **Apply section adjustment factors** (using Tekla Tedds). Any of the other calculation methods shall also be available for selection.

- **Ungrouted** - with this option the wall is forced to have orthotropic properties, with the calculation method defaulting to **Apply section adjustment factors** (using Tekla Tedds). Any of the other calculation methods shall also be available for selection.

5. Under **Analysis & design control**, select **Orthotropic** and set the orthotropic calculation method as **Apply section adjustment factors**.

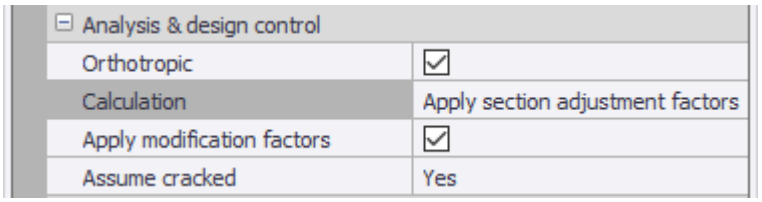


When the **Apply section adjustment factors** option is used, the wall density and orthotropic factors, where necessary, are automatically calculated during the analysis process in Tekla Structural Designer, using Tedds.

NOTE To find out more about the different calculation methods available, see:

---

6. Choose whether to **Apply modification factors** and then if applied, specify if the wall is **Assumed cracked** or not.



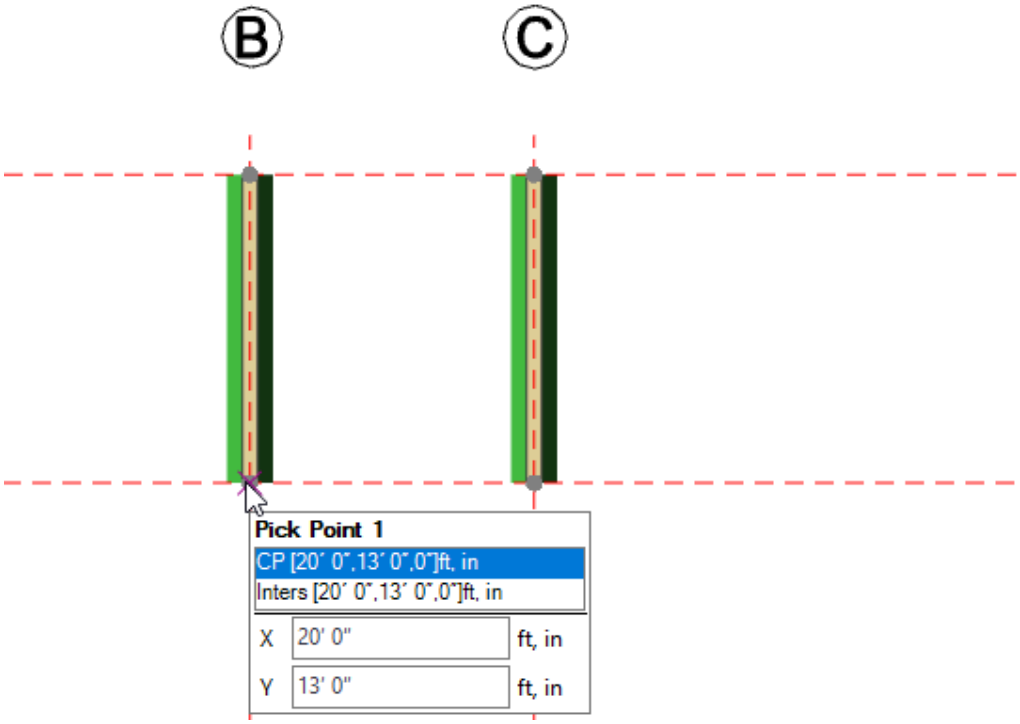
For this example leave **Apply modification factors** selected, and leave the **Assume cracked** setting as "Yes".

7. Ensure the other Masonry wall properties are set as required.

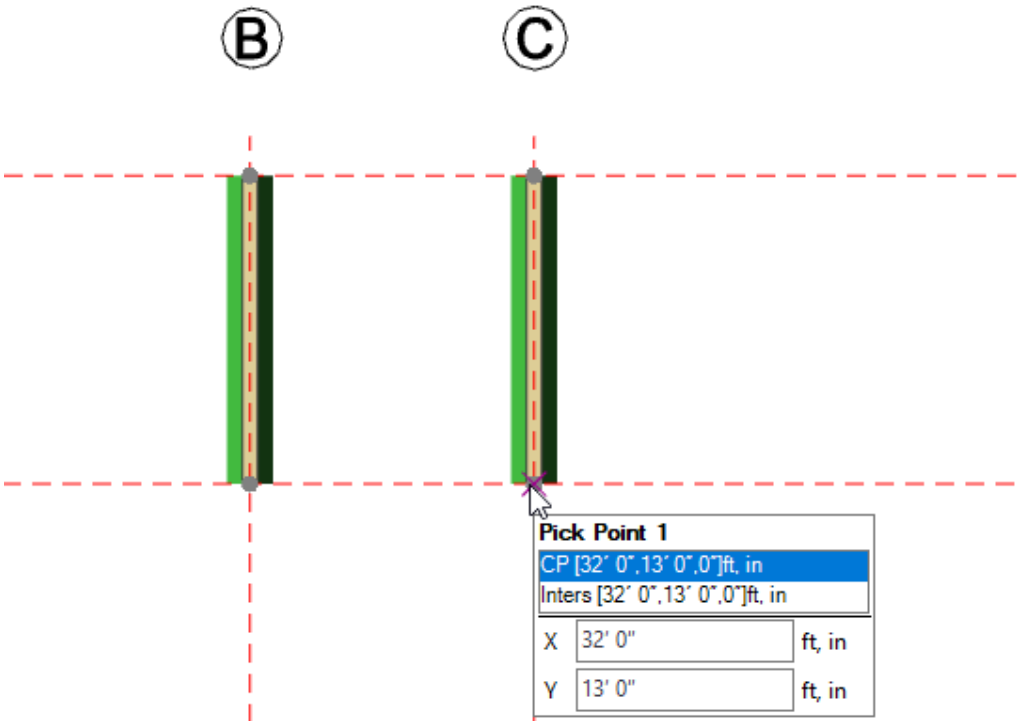
Place a wall

Two masonry walls currently exist in the model, we shall now add a third between grid points B/2 and C/2.

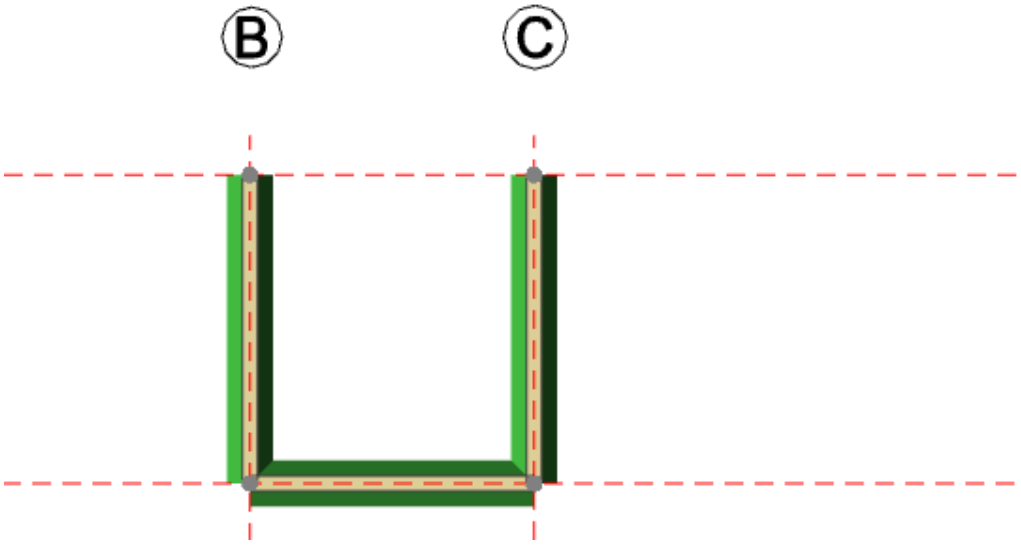
1. To place a new wall, in the **St. (Base) 2D** view, pick point 1 as grid point B/2



2. Then pick point 2 as grid point C/2



The wall is created between the points picked.



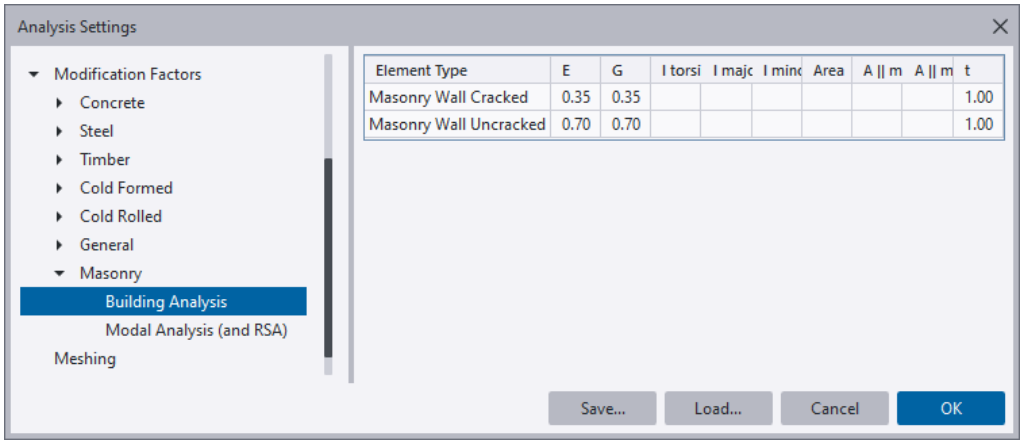
3. Press **Esc** to end the wall creation command.

Review the masonry modification factors

As **Apply modification factors** and **Assumed cracked** were selected in the wall properties, you should check that the factors meet your requirements.

To review these factors:

1. On the **Analyze** ribbon, click **Settings**.
2. On the **Modification Factors** page, select the **Masonry > Building Analysis** sub-page.



Modification factors (page 870) can be applied to the Elastic Modulus, E, Shear Modulus, G and wall thickness, t. Separate values can be specified for building analysis and modal analysis.

3. The walls in this example will use the **Masonry Wall Cracked** factors shown above. No changes are required, so click **Cancel**.

Create a wall opening

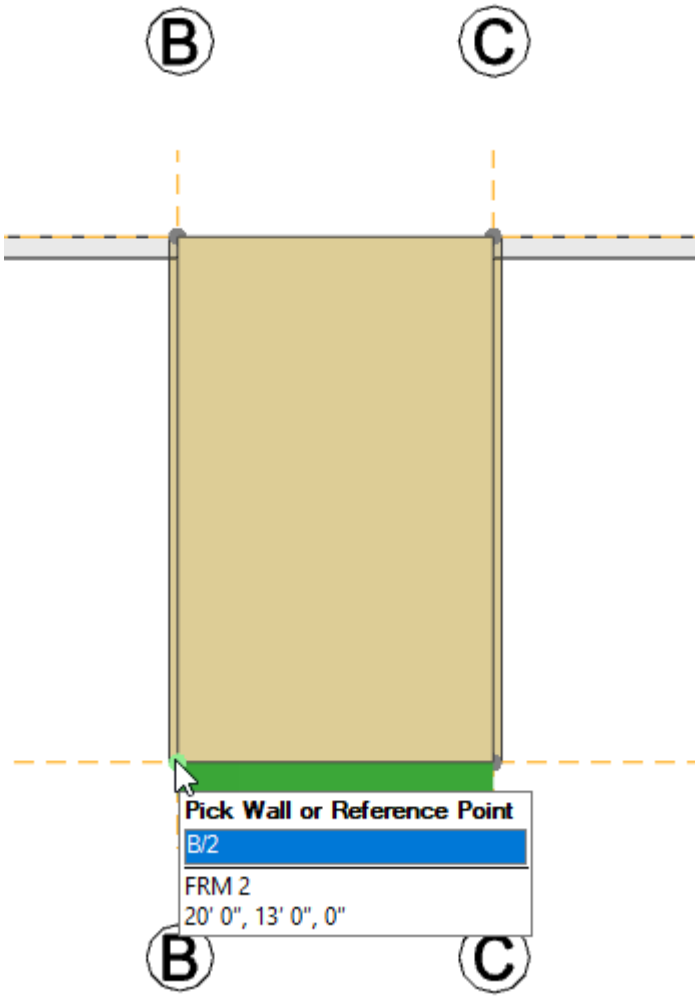
You may need to create openings in masonry walls, so we will now demonstrate how this can be done.

CAUTION Although you can create openings in a masonry wall, currently no extra checks are performed in the Tedds calculation to take the openings into account. (This is similar to how openings are handled in Tekla Structural Designer's concrete wall design.)

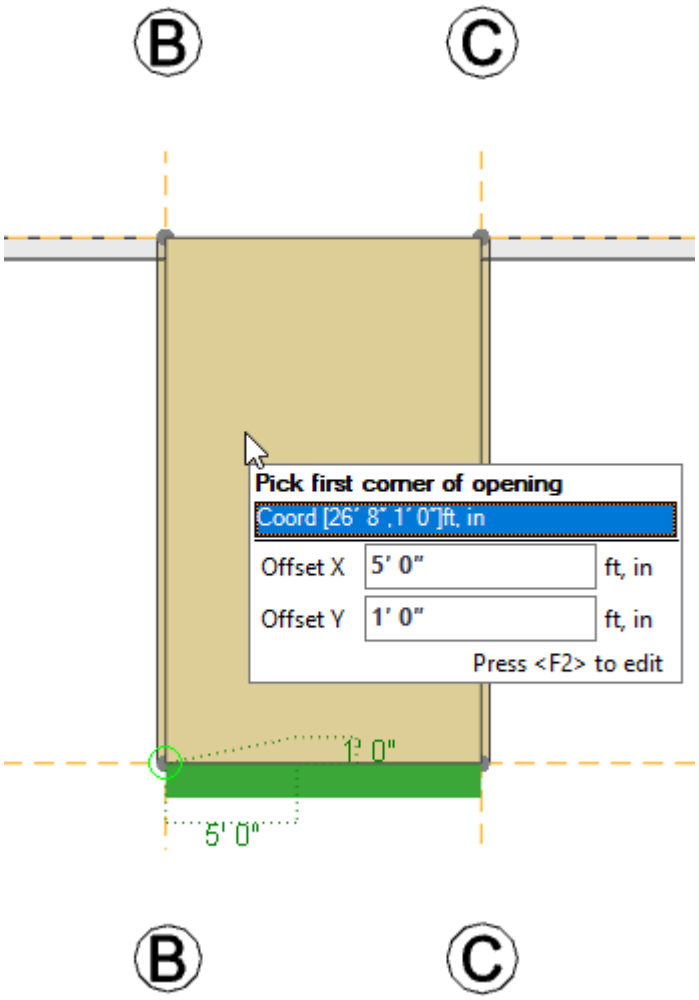
---

1. In the **Project Workspace Structure** window, right click on **Frames > FRM 2** and select **Open view**
2. On the **Model** ribbon, click  **Opening**

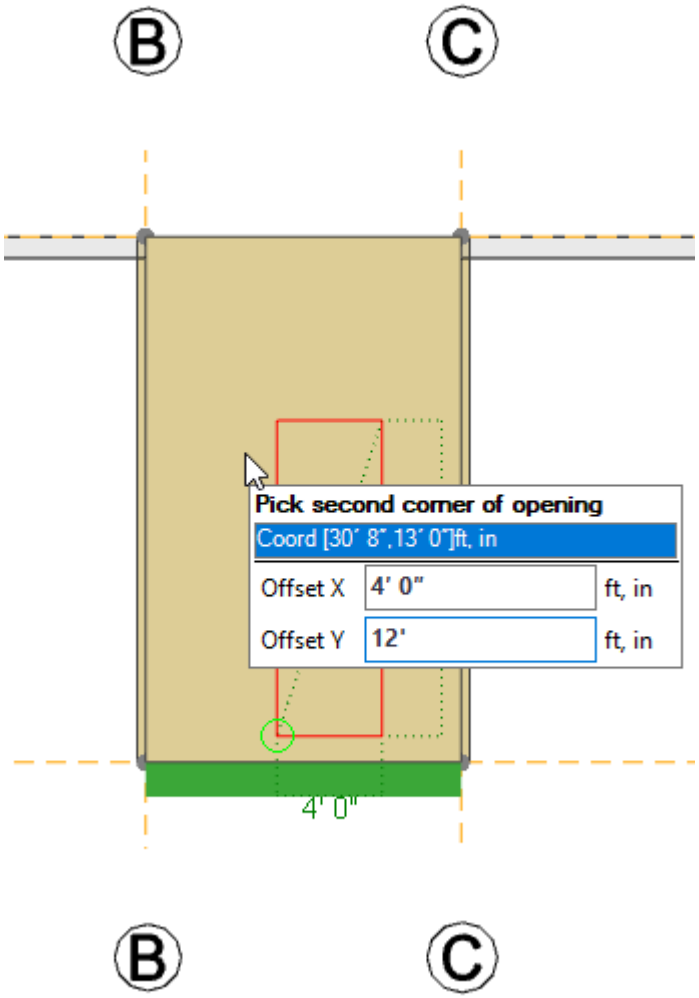
3. To create an opening, pick the reference point where grid B meets the base level grid.



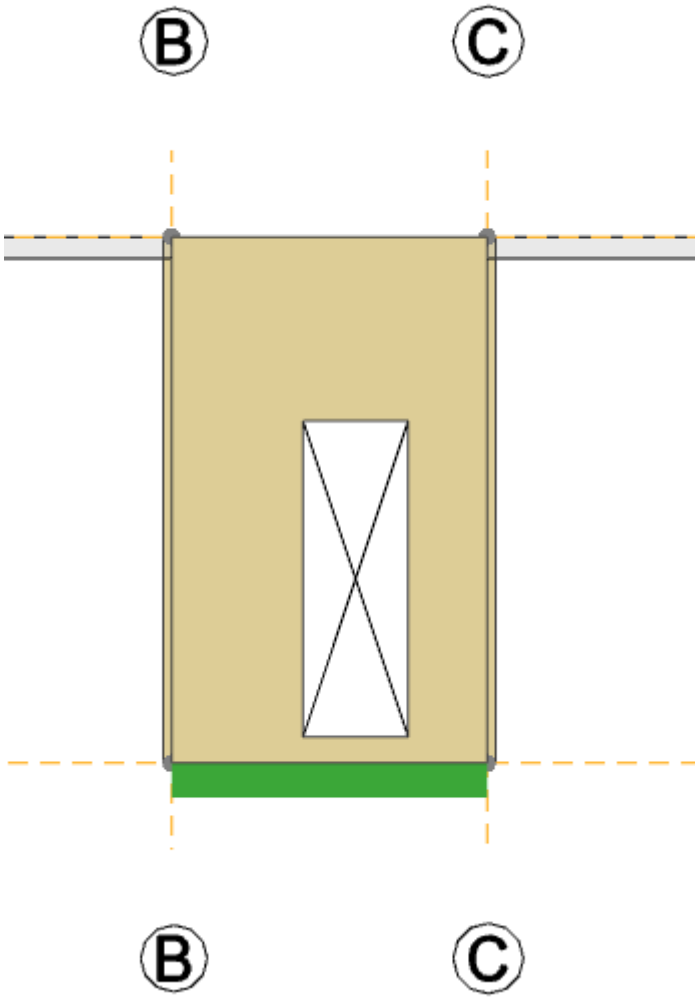
4. To pick the first corner, press **[F2]**, enter **Offset X** as **5'** and **Offset Y** as **1'**, then click on the wall.



5. To pick the second corner, press **[F2]**, enter **Offset X** as **4'** and **Offset Y** as **12'** then click on the wall again.



The wall opening is created between the points picked.



Create result lines around an opening

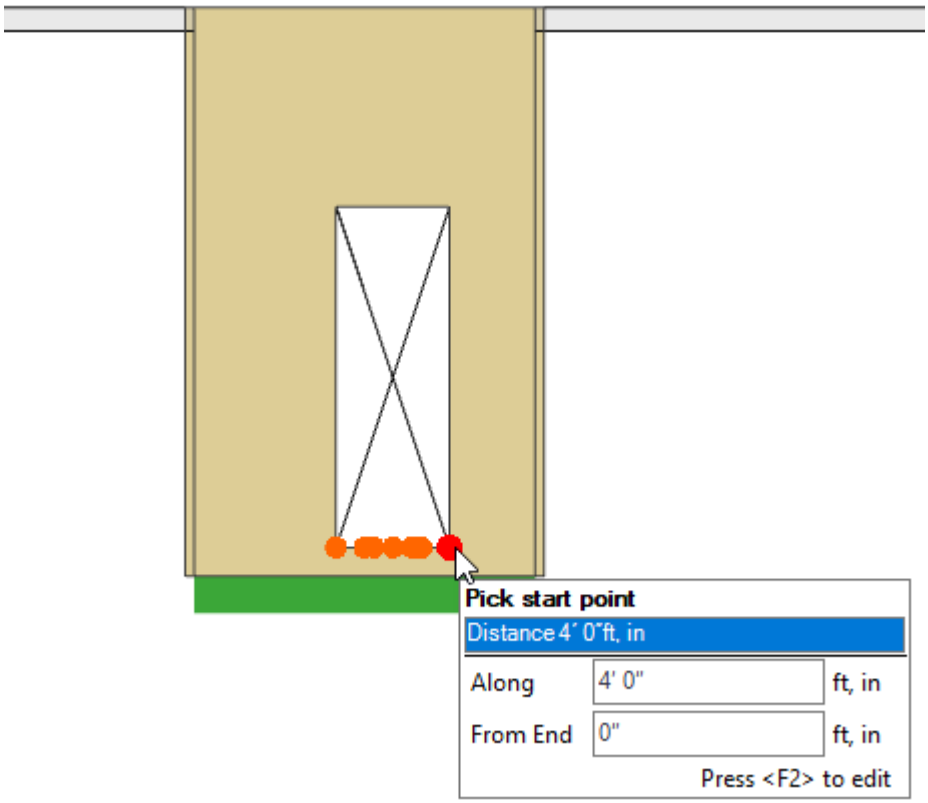
The design cases (page 451) exported from a masonry wall panel to the Tedds calculation are at fixed locations (two at the top of the wall, two in the mid-fifth, and two at the bottom).

Result Lines can be placed to set up additional design cases where required, (for example at a wall opening). When a result line is checked it exports four design cases (two at the left and two at the right of the line) to the Tedds calculation.

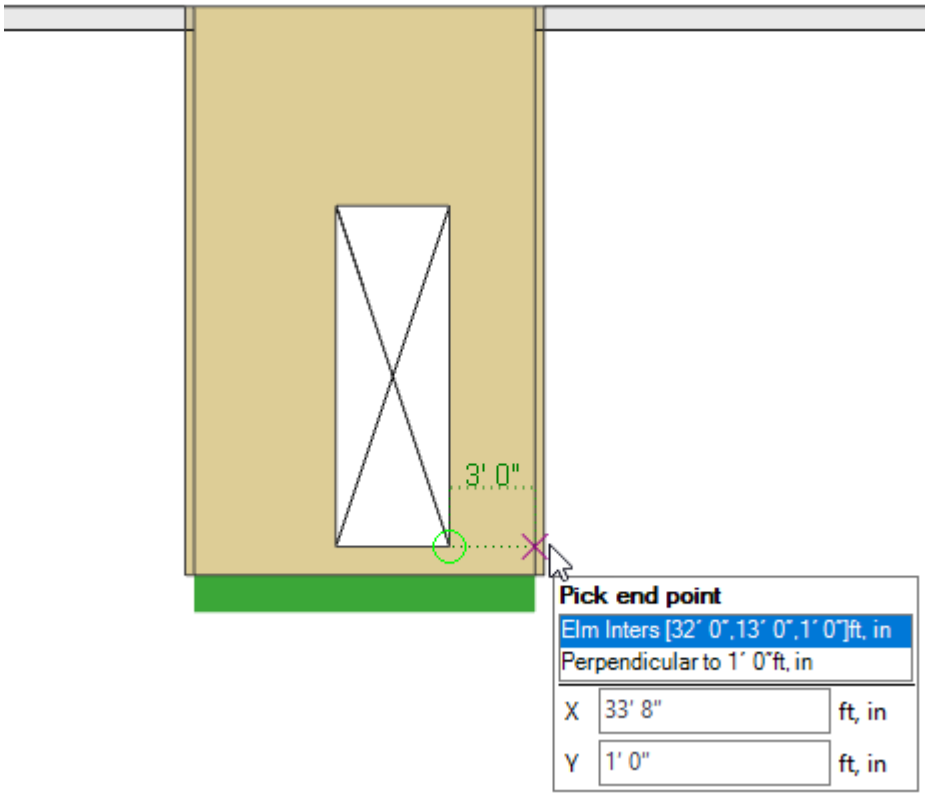
In this example, in order to demonstrate the principle, we shall place a couple of result lines at the base, and a couple more at the top of the opening. In a real model you should use your engineering judgment to decide if, and where, any result lines might need to be placed.

NOTE Checking of result lines in Tedds is limited to horizontal result lines contained in a single panel.

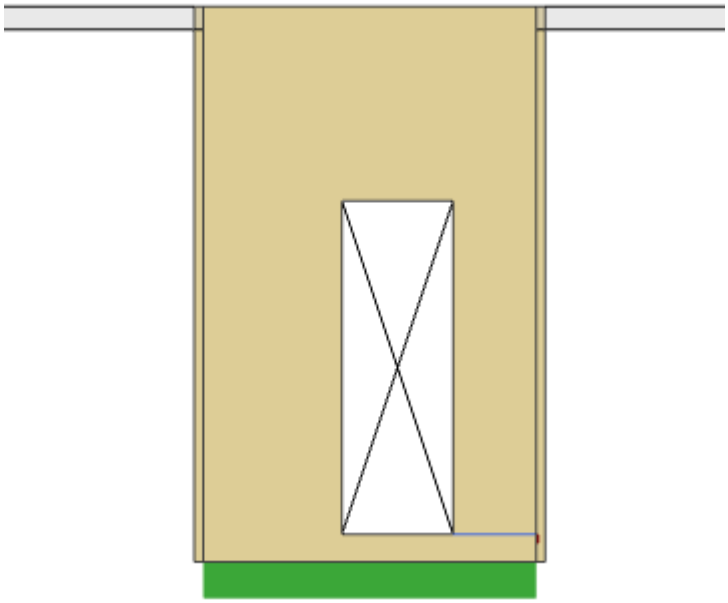
1. In the **FRM 2 2D** view, switch the view regime to **Results View**.
2. On the **Results** ribbon, click **Result Line**
3. To create the first result line, pick the start point by snapping to the bottom right corner of the opening.



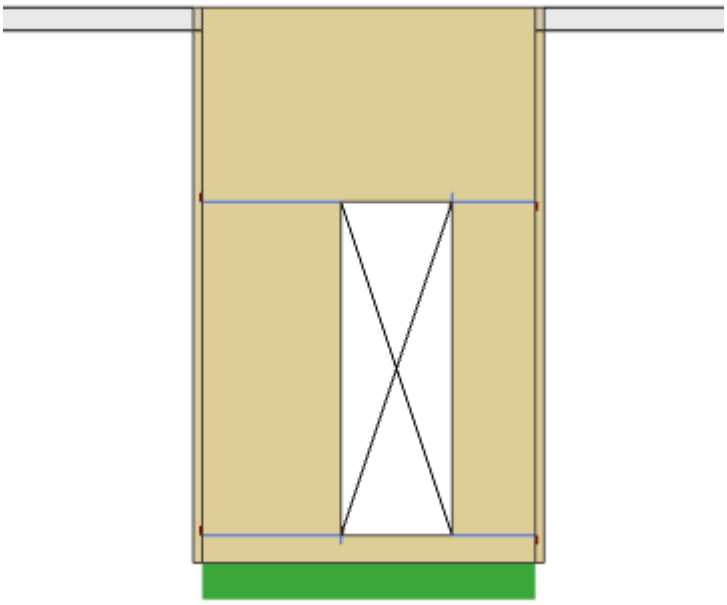
4. Pick the end point by snapping to the horizontally adjacent edge of the wall.



The first result line is created as shown.



5. Repeat the process to create three more result lines from the top right, top left and bottom left of the opening, each extending to the edge of the wall.

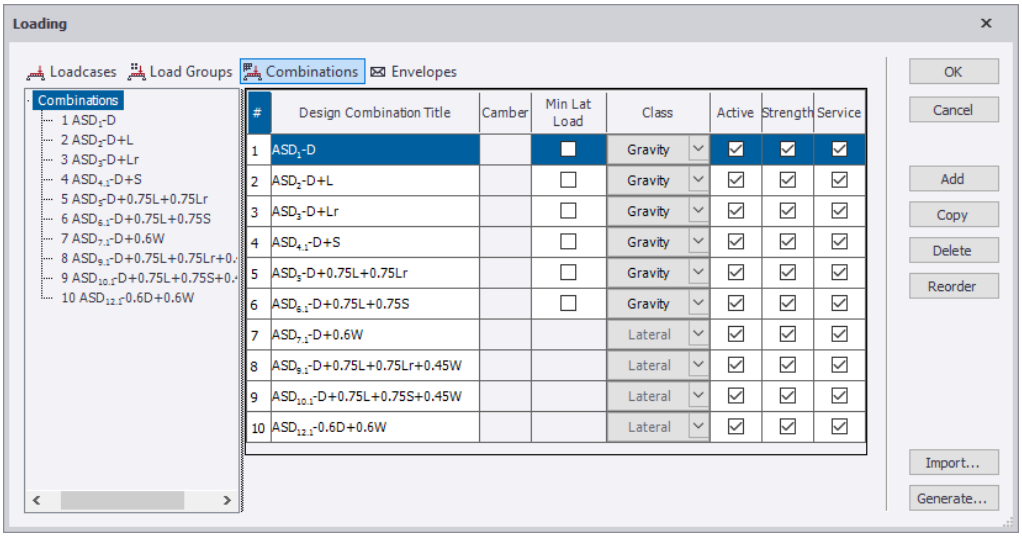


6. Press **[Esc]** to end the command.

Create load combinations

For structures containing masonry walls, loads can be applied and load combinations defined in the standard manner. In this example the load combinations have already been generated.

1. To view the load combinations: on the **Load** ribbon, click **Combination**.



2. No changes are required, so click **Cancel**

Establish design forces by running the analysis

Masonry walls can only be designed provided a suitable set of analysis results exist.

To generate some results:

- On the **Analyze** ribbon, click

In models with masonry walls that are assigned Tedds defined grades, there is an additional step in the early stages of the analysis process. Here, the program checks to see if there is a requirement to update the density or orthotropic factor values of any of the masonry walls in the model, and subsequently updates them if so.

The material density value, obtained in this way, is an average value per length that takes into consideration the geometry of the wall section, and is of particular interest with the existence of non-uniform grouting and voids in the wall section. This newly calculated value then overrides the one originally input in the materials database.

The orthotropic analysis properties, obtained in this way relative to the vertical and horizontal directions in the plane of the wall, are adjusted for average section properties as recommended by *NCMA TEK 14-01B - Section Properties of Concrete Masonry Walls.* The adjustment to Ey (Young's modulus in the Y direction, along the height of the wall) assumes the wall is acting primarily to resist in-plane loads.

The forces to be included in the design envelopes for masonry walls can come from any of the analysis types available for design, and is limited by the selections in Design > Options > Analysis.

Check all the walls in the model

NOTE Unlike timber/precast members, masonry walls can be checked before having been designed. This is possible because a wall that can be designed in Tedds has all the data necessary for check/design from the moment it is created.

You can rapidly check all the masonry wall panels in the model using the initial default reinforcement properties, (which are a single #5 vertical reinforcement bar in each grouted cell and bond beams at 48 inches with 2 #3 bars), as follows:

1. On the **Design** ribbon, click Check In Tedds.

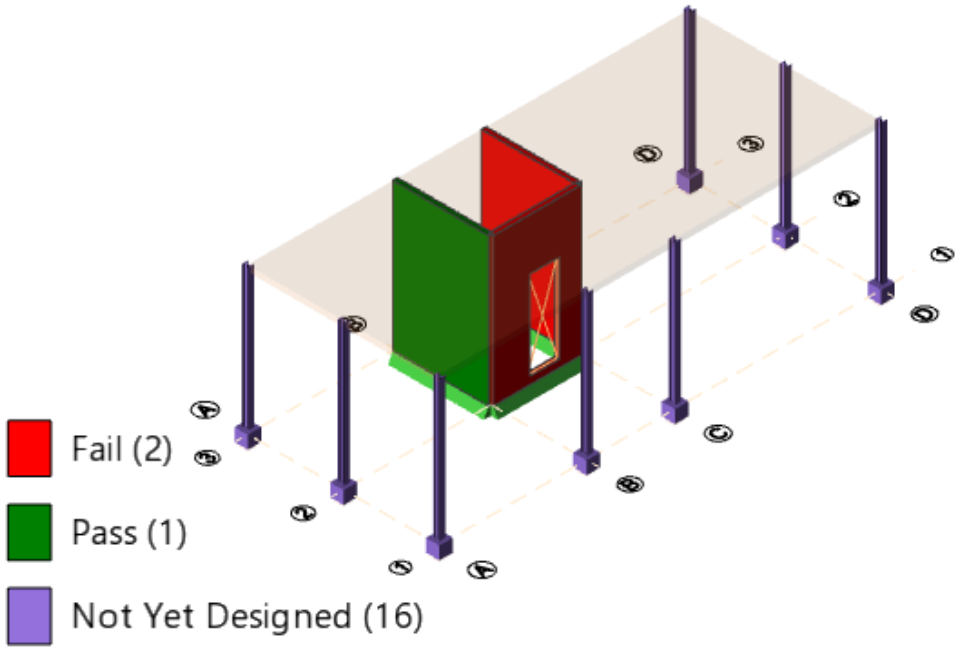
   This runs the necessary Tedds calculations in the background using the latest analysis results.

2. To review the status for all walls:

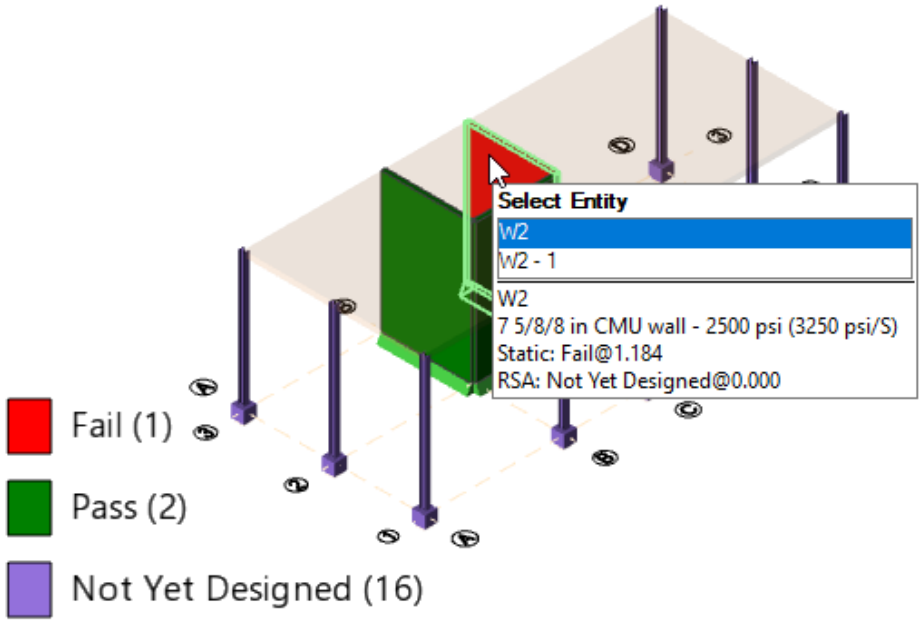
a. Activate the **Structure 3D** view.

b. Switch the view regime to **Review View**.

The initial status of each wall panel using the default reinforcement is shown below.



3. Hover the cursor over wall W2, (the wall running along grid C).



As indicated in the tooltip the wall fails.

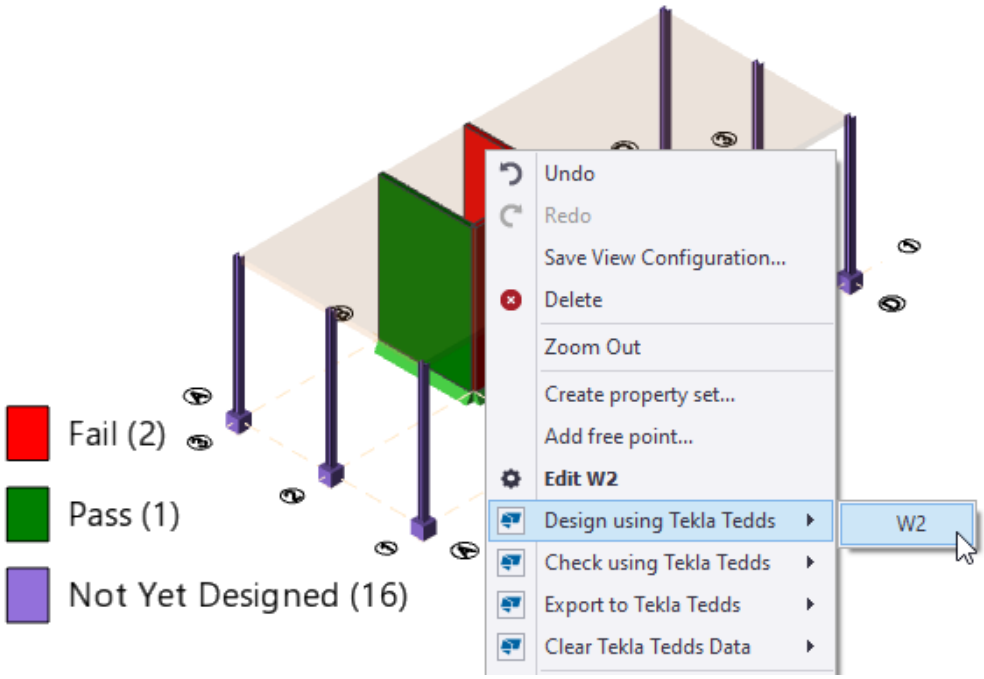
The ratio displayed for a wall is the maximum ratio from the check of all panel(s) and result lines associated with the wall.

In this case, W2 consists of a single panel and has no result lines associated with it.

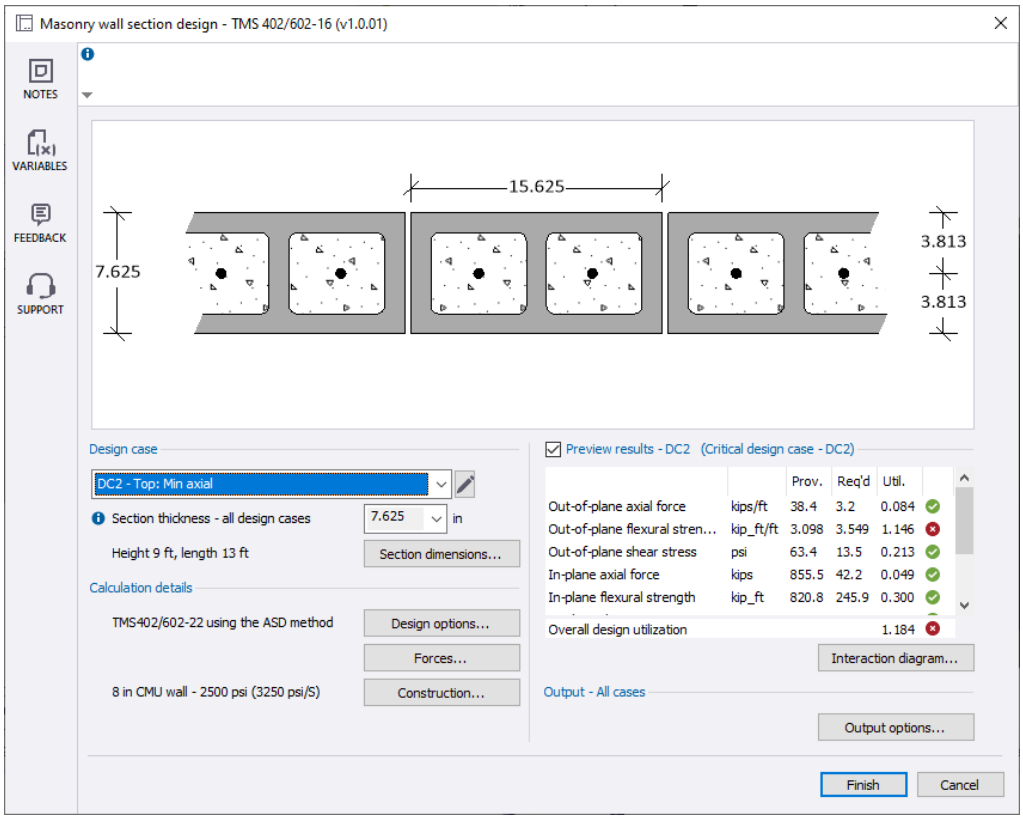
Design failing wall W2

The default reinforcement used in the check of wall W2 is insufficient. It can be updated by designing the wall.

1. Right click on wall W2 (the wall running along grid C) and select Design using Tekla Tedds > W2

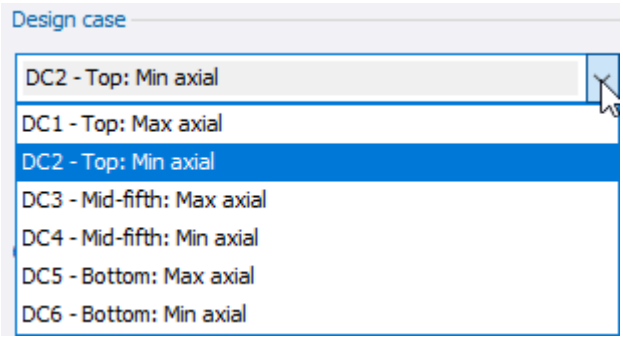


The Tedds masonry wall section design calculation opens.



As shown above, the critical design case is automatically selected from the design case list, and the preview results are displayed for that case.

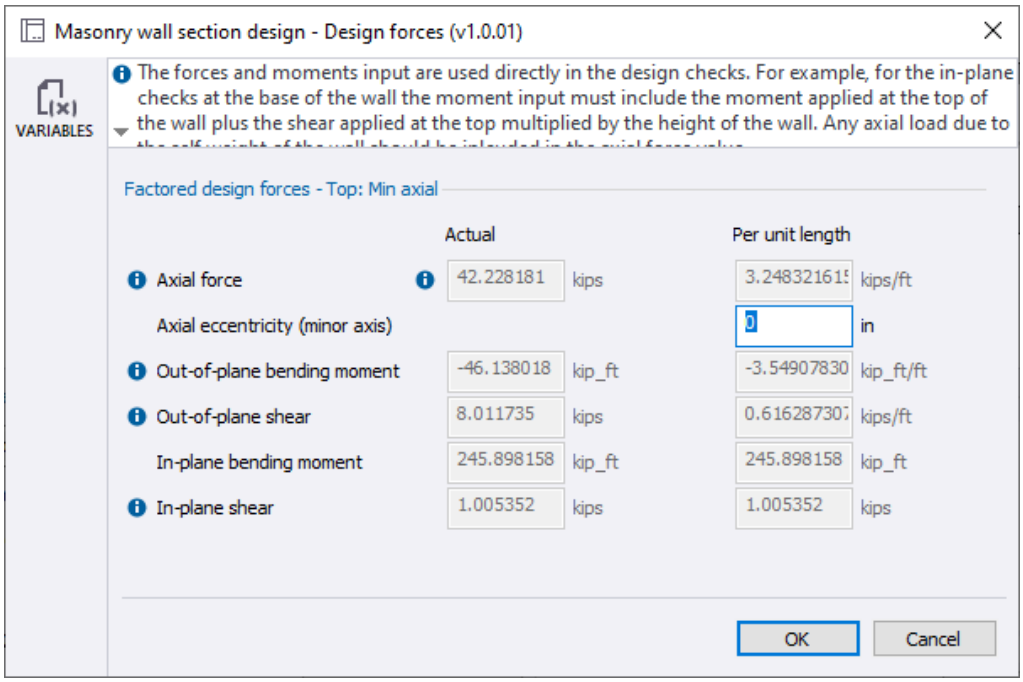
2. Click the **Design case** list.



Each of the six design cases (page 451) is automatically established from the enveloped load combination results.

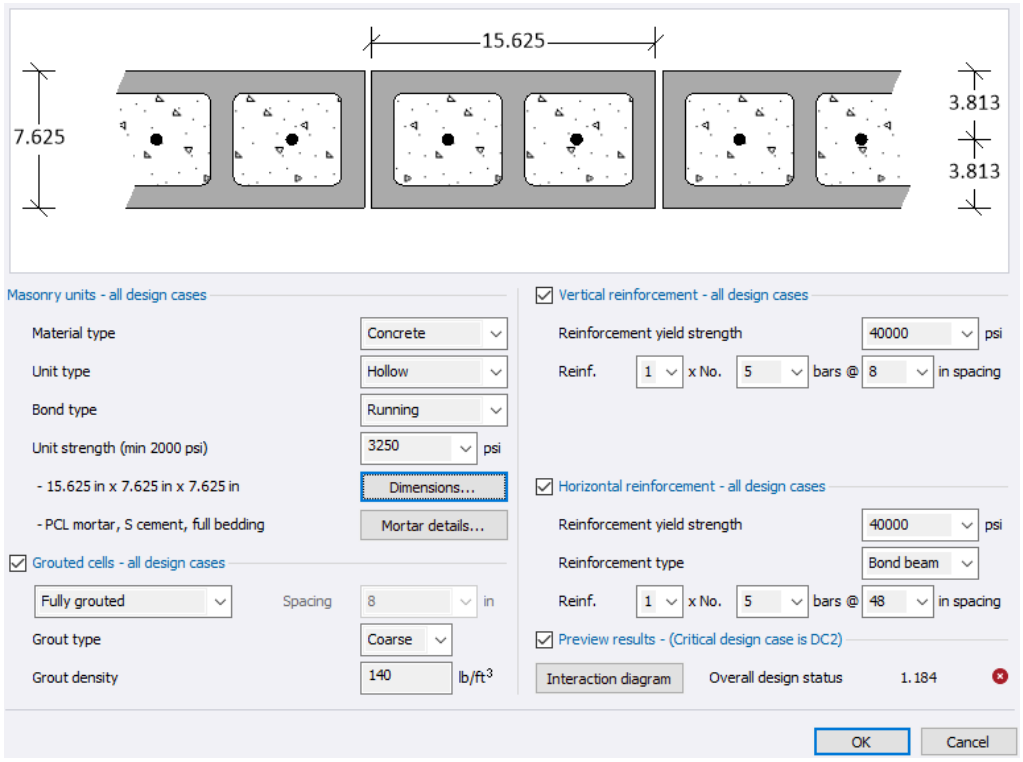
When you select a different design case from the list its results are displayed in the preview pane.

3. Click **Forces...**



Design forces are displayed for the selected design case.

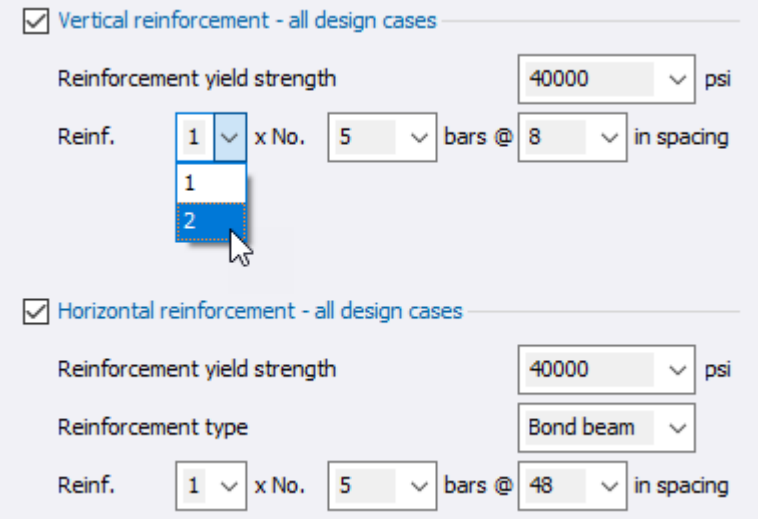
4. Click **Construction...**



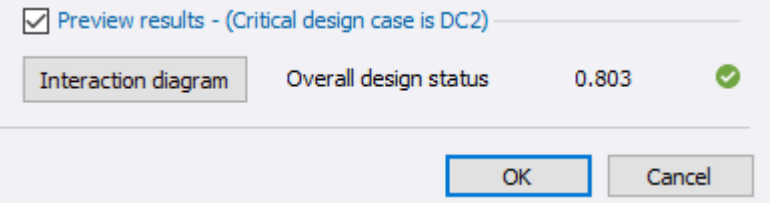
The masonry unit properties on the left are read from the database and are not editable.

The reinforcement properties on the right are editable. For this example we need to update the reinforcement to achieve a satisfactory design.

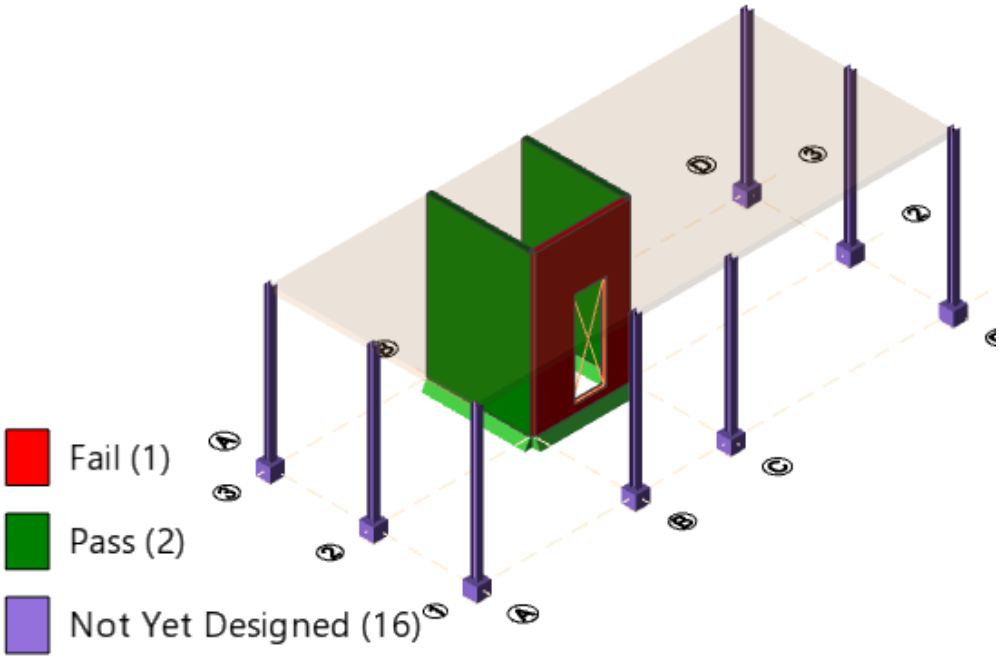
5. Edit the vertical reinforcement to increase the number of bars at each position from 1 to 2.



6. The overall design status should update accordingly.



7. The wall is now passing so click **OK**, and then click **Finish**.



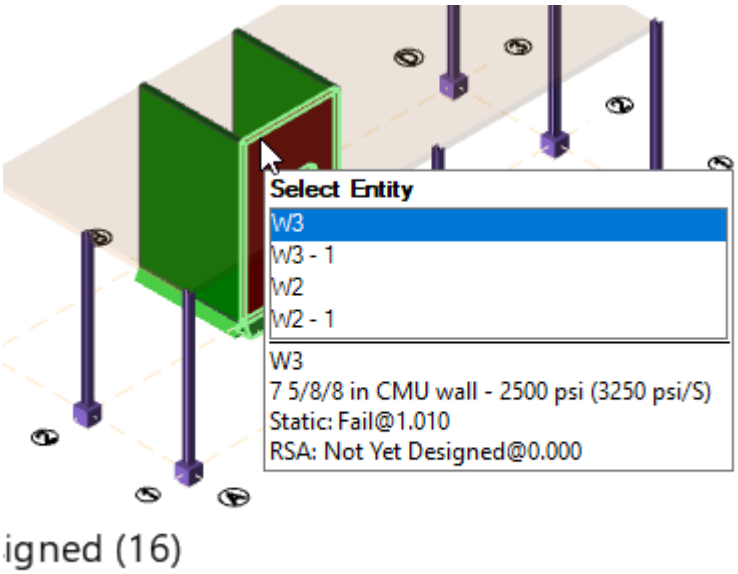
NOTE Tedds calculation data is retained from one run of the design to the next unless changes are made in Tekla Structural Designer that force the Tedds calculation data to be re-established. To manually clear the retained Tedds calculation data, right-click on the wall, then select Clear Tekla Tedds Data from the context menu.

Review the design of W3 and associated result lines

Result line design can be reviewed graphically via the tooltips in the same way as panel design. Another way to review the design is to display a tabular design summary. Both methods are illustrated below.

To review graphically:

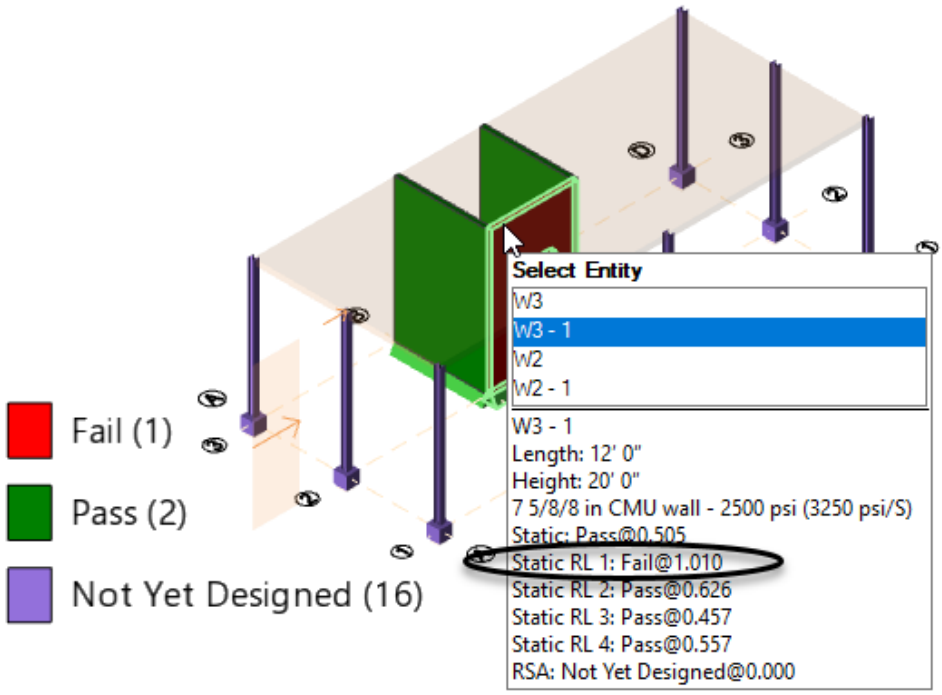
1. Hover the cursor over wall W3, (the wall running along grid 2) and if necessary, press the tab key until W3 is the selected entity.



igned (16)

When W3 is the selected entity, the status is fail, but no details are shown for the associated result lines.

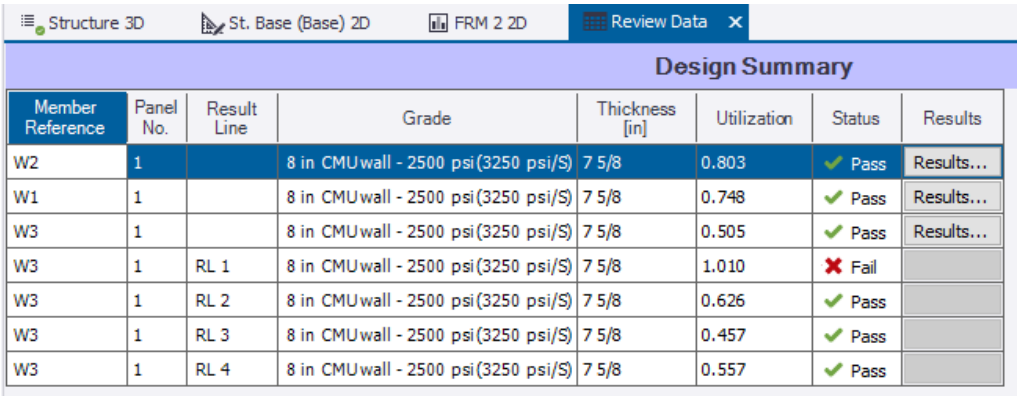
2. In the tooltip press the tab key so that W3-1 is the selected entity.



The tooltip now shows the utilization ratio of the W3-1 panel check, plus the ratios for four result lines associated with it. While the panel check passes, one of the result lines fails. It is the failing result line that dictates the overall status of W3.

To display the design summary:

1. On the **Review** ribbon, click **Tabular Data**
2. On the **Review Data** ribbon, ensure **Design Summary** is selected in list in the View Type group and then click **Masonry** in the Material Type group.



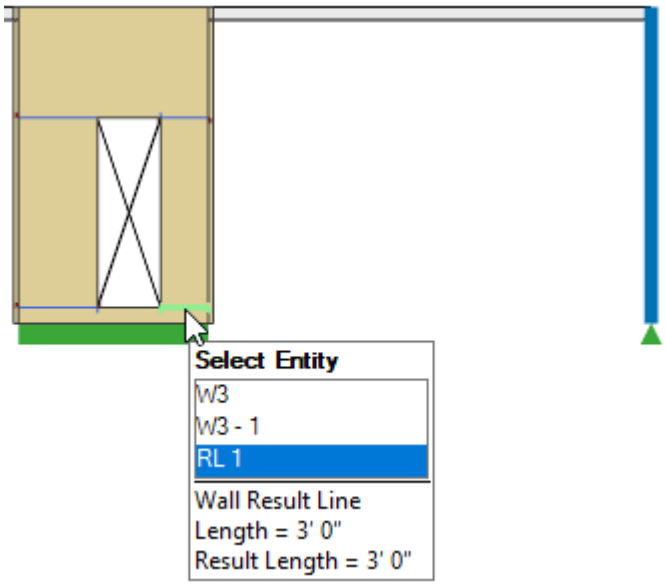
Member Reference	Panel No.	Result Line	Grade	Thickness [in]	Utilization	Status	Results
W2	1		8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.803	Pass	Results...
W1	1		8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.748	Pass	Results...
W3	1		8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.505	Pass	Results...
W3	1	RL 1	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	1.010	Fail	
W3	1	RL 2	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.626	Pass	
W3	1	RL 3	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.457	Pass	
W3	1	RL 4	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.557	Pass	

The design summary confirms what was shown in the tooltips. Panel W3-1 is passing but result line RL1 is failing. The overall status of W3 will only pass when the panel passes AND all the result lines pass. The result line utilizations in the table (and the tooltips) are always calculated using the same design data as the panel.

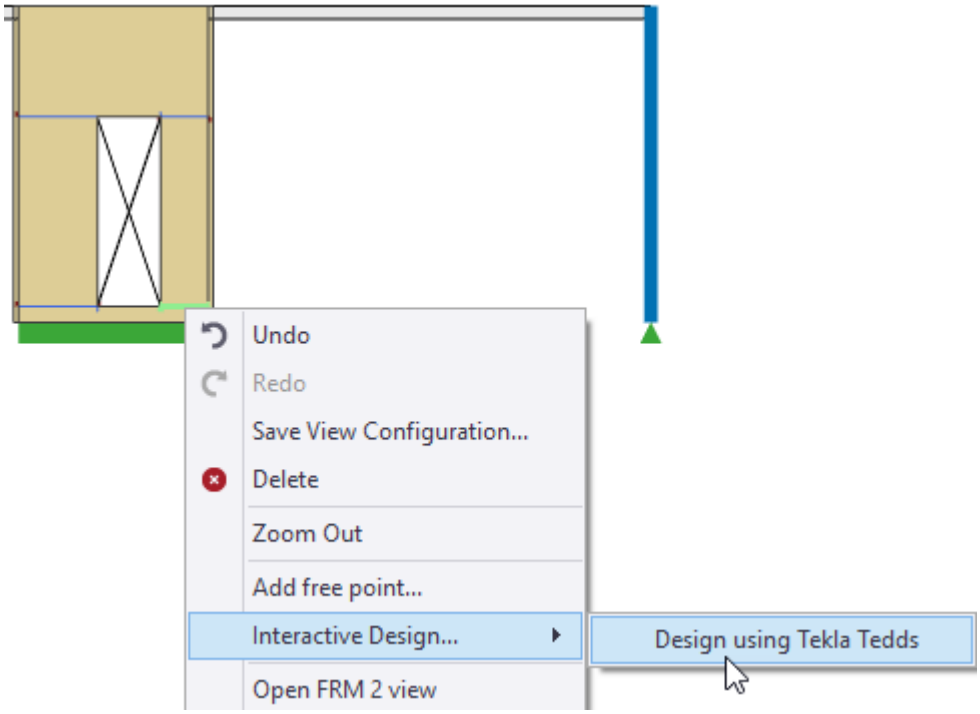
Design failing result line RL 1

Wall W3 is failing because the associated result line, RL 1 is failing. To get W3 to pass, RL1 should be designed to determine suitable reinforcement. When RL1 is passing, the new reinforcement can be applied back to the panel.

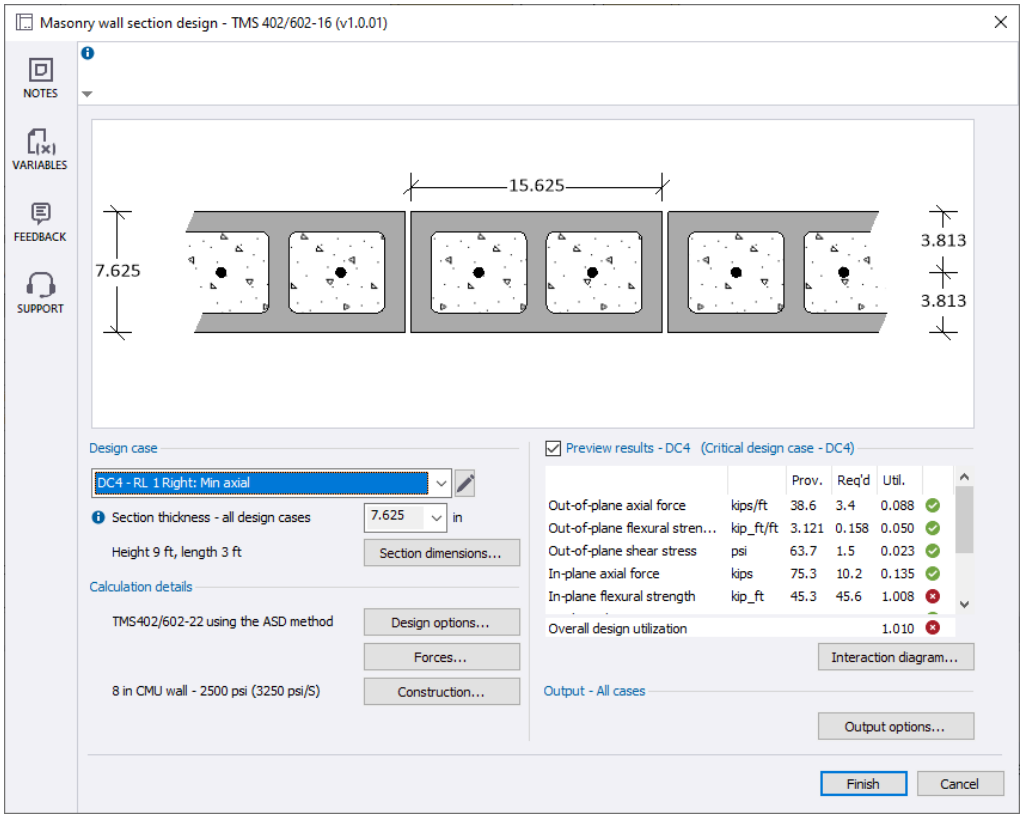
1. Activate the **FRM 2 2D** view and select the failing result line RL 1.



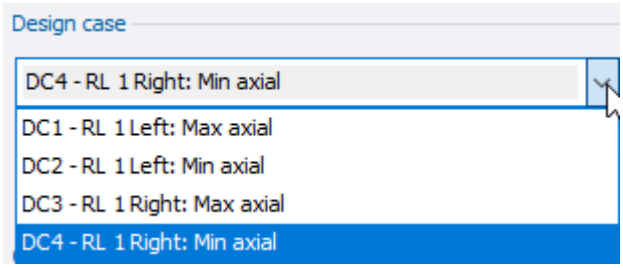
2. Right click and select Interactive Design > Design using Tekla Tedds



The Tedds masonry wall section design calculation opens.

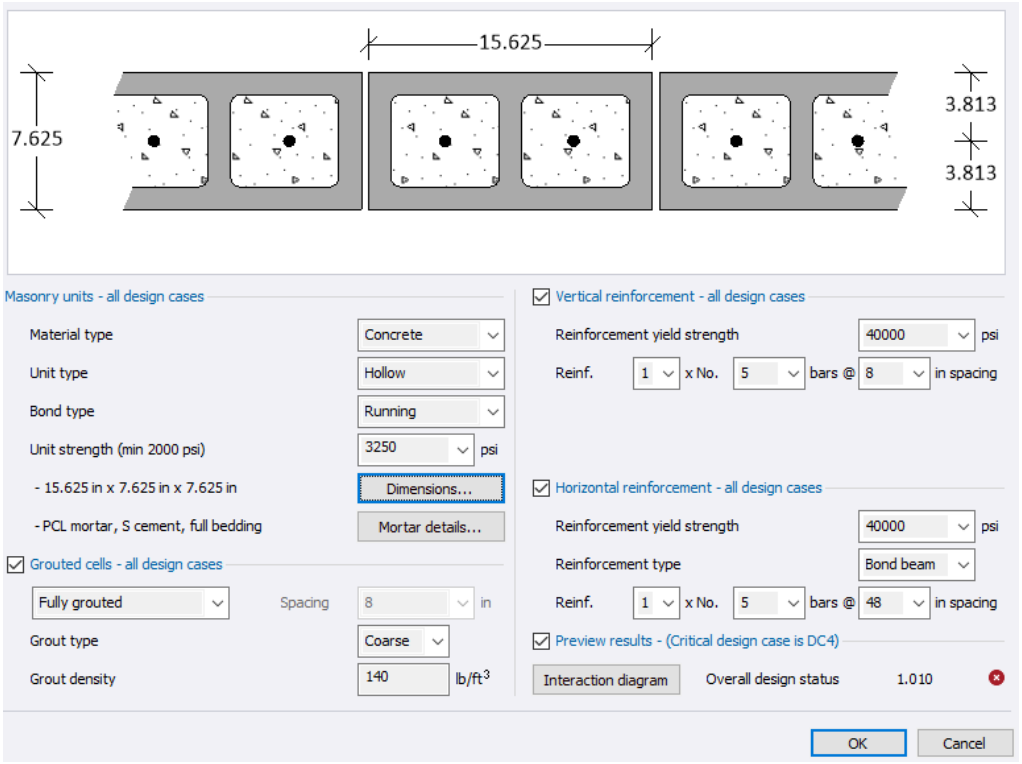


3. Click the **Design case** list.



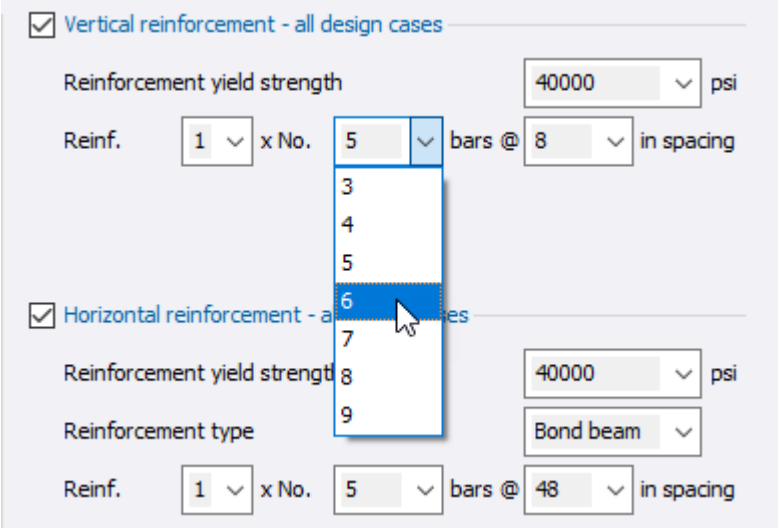
Unlike wall panels, result lines only have four design cases (page 451) established from the enveloped load combination results.

4. Click **Construction...**

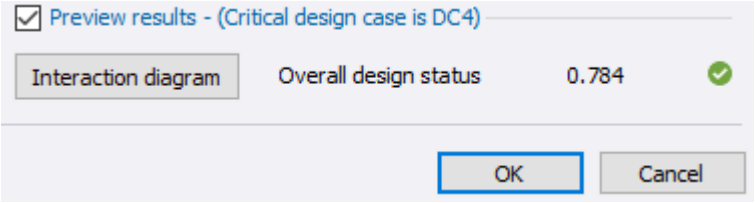


NOTE When a result line is designed Tekla Structural Designer considers this to be an interactive investigation. The construction data is initially populated using data from the associated panel. At the end of the design you have the option to apply the data back to the associated panel; if you choose not to do so, when you next design the result line you will find that it will have reverted back to the original panel construction data.

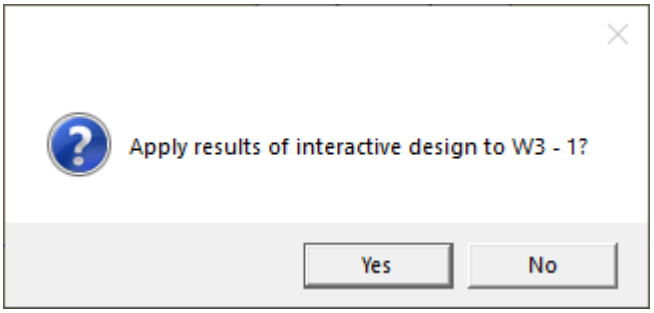
5. Edit the vertical reinforcement, but this time, instead of increasing the number of bars at each location, increase the bar size from 5 to 6.



6. The overall design status updates accordingly.



7. The result line is now passing so click **OK**, and then click **Finish**.
8. At this point you are prompted if you would like to apply the result line design data back to the panel.



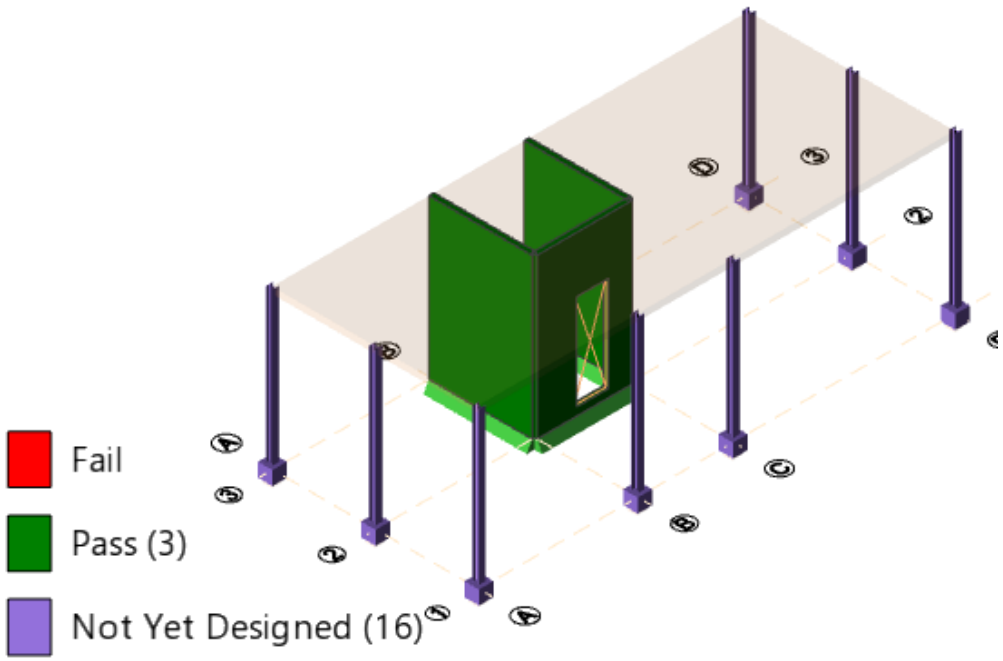
This is particularly useful in two situations:

- the result line design is the critical fail overall, so applying the result line design data to the full panel achieves a general pass using a uniform data set.
- the result line design is the critical pass overall but design is too conservative, so optimizing the design of the result line and the panel.

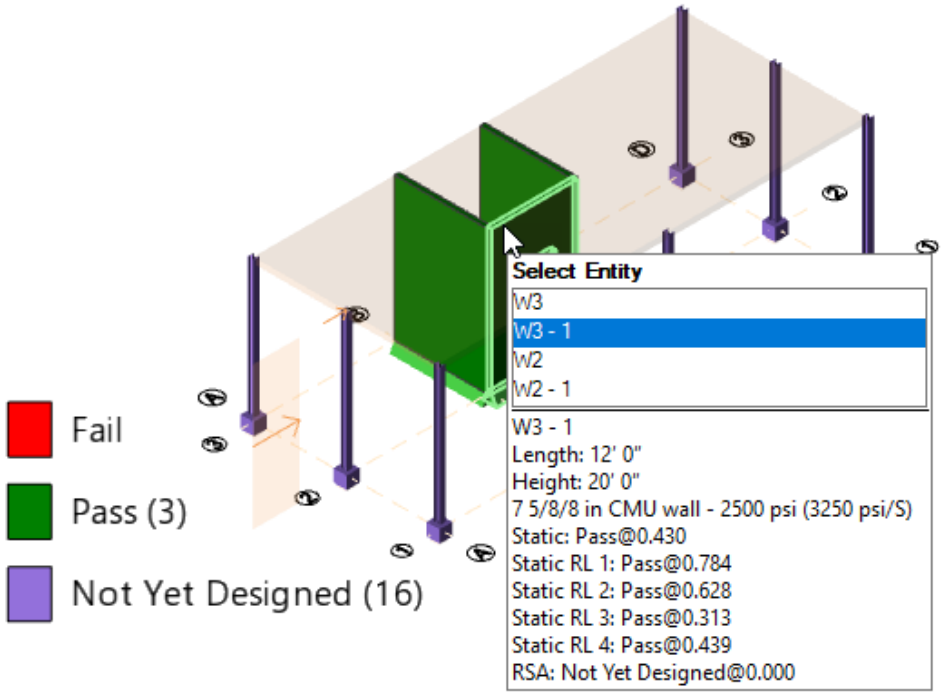
9. Click **Yes**.

The updated result line reinforcement is applied to panel W3-1. The panel and its associated result lines are checked in the background.

10. To review the new status of the checks, activate the **Structure 3D** view.

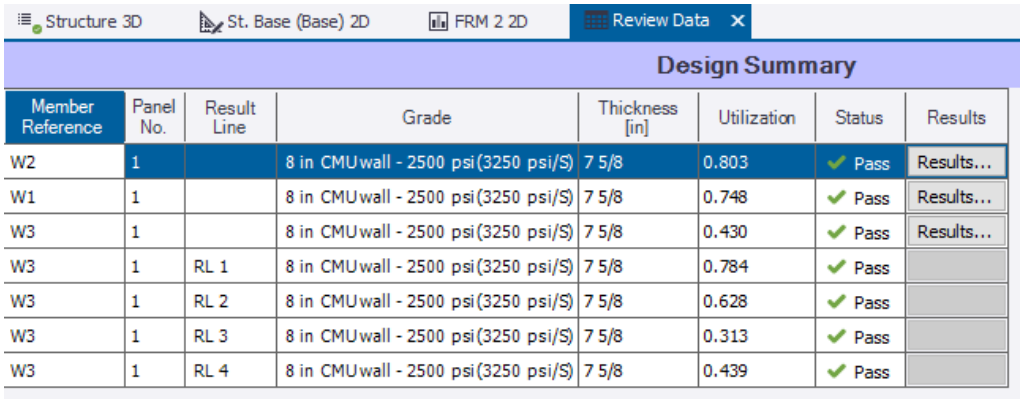


11. Hover the cursor over wall W3, then in the tooltip press the tab key so that W3-1 is selected.



The tooltip shows the utilization ratio of the W3-1 panel check, plus the ratios for four result lines associated with it. The panel check passes and all of the result lines pass, hence the overall status of W3 is now passes.

12. To review the updated Design Summary, activate the **Review Data** view.



Design Summary

Member Reference	Panel No.	Result Line	Grade	Thickness [in]	Utilization	Status	Results
W2	1		8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.803	Pass	Results...
W1	1		8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.748	Pass	Results...
W3	1		8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.430	Pass	Results...
W3	1	RL 1	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.784	Pass	
W3	1	RL 2	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.628	Pass	
W3	1	RL 3	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.313	Pass	
W3	1	RL 4	8 in CMUwall - 2500 psi(3250 psi/S)	7 5/8	0.439	Pass	

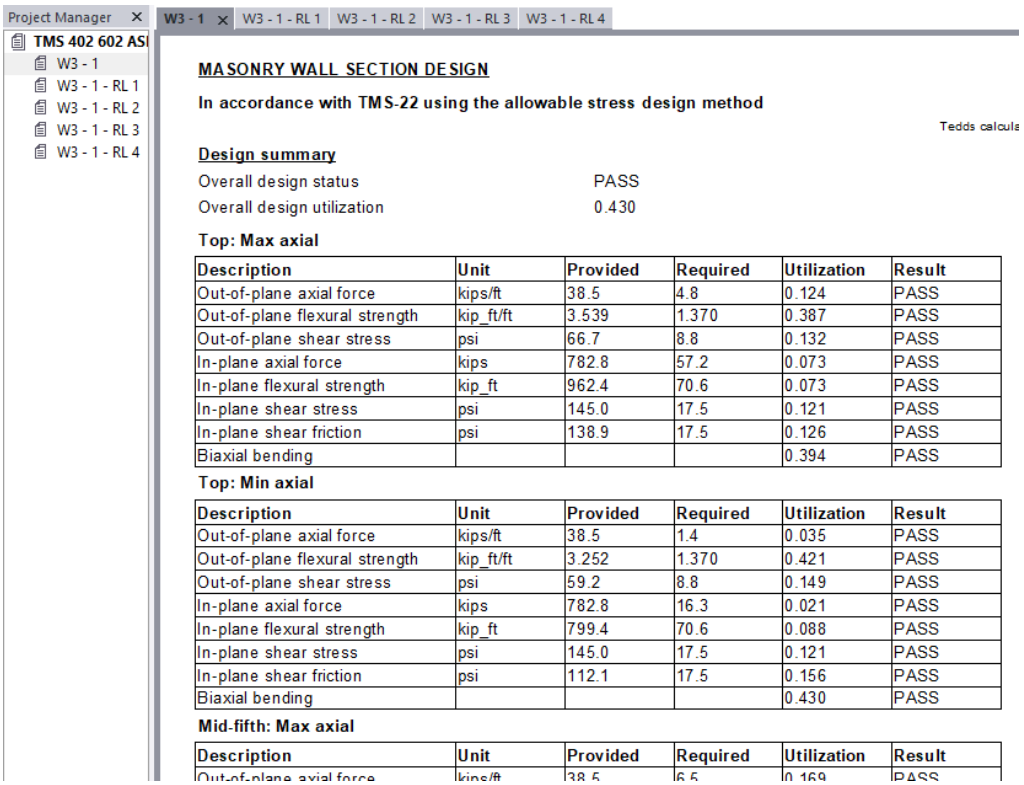
Because the design data from RL1 was applied back to the panel, the utilizations of panel and the associated result lines have all been recalculated accordingly.

Output the calculations

Detailed Tekla Tedds calculations for masonry walls do not exist within Tekla Structural Designer itself, instead they are available by exporting to Tekla Tedds.

1. Right click on the wall W3 and select **Export to Tekla Tedds > W3**

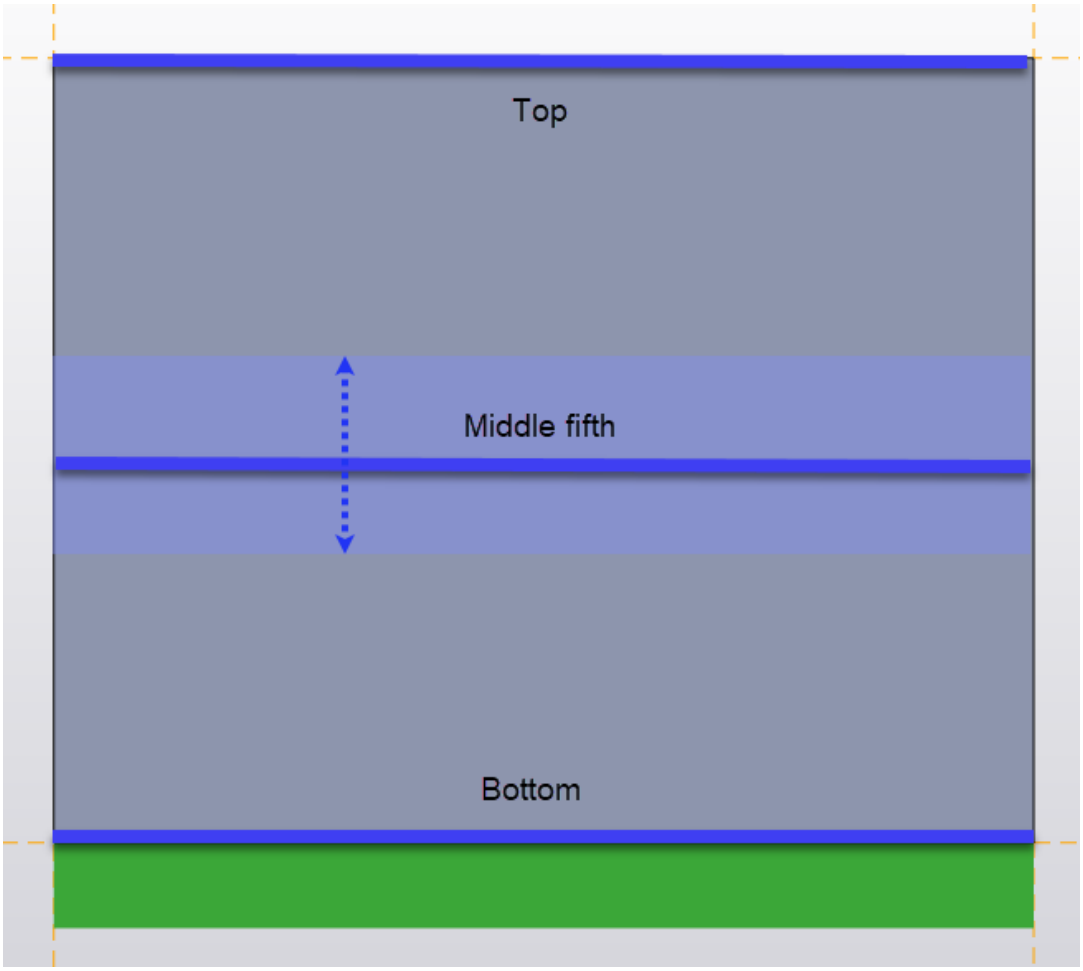
   Tedds opens and the wall calculation is displayed. Separate calculations are created for the panel and each of the associated result lines.



6.4 Masonry wall design cases

The analysis results that are created for meshed walls are grouped and load combination results enveloped to create 6 individual design cases to send to Tedds. For each design case, the envelope considers all selected load combinations and all selected analysis types including RSA and Staged Construction Analysis.

Overall, three design locations are created for each panel: top, bottom, and mid-fifth. Each design location results in two design cases which are the maximum design forces for moment and shear forces combined with either the maximum or minimum axial force. This approach ensures the worst design cases are captured.



The 6 Tedds design cases are therefore:

- DC1 - Top: Max axial
- DC2 - Top: Min axial
- DC3 - Mid-fifth: Max axial
- DC4 - Mid-fifth: Min axial
- DC5 - Bottom: Max axial
- DC6 - Bottom: Min axial

Design forces for each design case are the maximum of the absolute values of all design forces included in the envelope:

- Shear Major
- Shear Minor
- Moment Major (retains sign)
- Moment Minor (retains sign)

Axial Force is either the maximum or minimum value of all axial forces in the envelope as indicated by design case.

For the two "mid-fifth" cases the result locations, positions included in envelope are:

- 3/5 height, Above
- 3/5 height, Below
- 1/2 height, Above
- 1/2 height, Below
- 2/5 height, Above
- 2/5 height, Below

7 Staged construction handbook

This handbook contains the following information relating to staged construction analysis and design in Tekla Structural Designer.

- Introduction (page 454)
- Overview of staged construction analysis (page 456)
- Limitations and assumptions (page 480)
- Design comparison - with and without staged construction (page 481)
- Workflow example - Eurocodes (page 489)
- Workflow example - United States (ACI/AISC) (page 511)

7.1 Introduction to staged construction analysis

What is staged construction analysis?

A staged construction analysis,

- Is a static analysis procedure
- It recognizes the changing state of a structure over time
- It is more relevant for taller (typically concrete) buildings

It typically assumes,

- Buildings are constructed in stages
- Stages are story based
- Not all forces and stiffnesses are in effect from day one (unlike a full 3D analysis).

At each “stage” any or all of the following may change:

- The physical members/supports included in the model

- Material properties (Time dependent effects)
- The applied loading

Benefits of staged construction analysis

By running a staged construction analysis in Tekla Structural Designer you can take advantage of the following benefits:

1. Transfer levels are loaded more realistically,
   - but not to the same extent as a traditional hand calculation - this is more closely emulated by Tekla Structural Designer's existing chasedown procedures - see this example (page 481) for details.
2. Axial deformations are reduced to give a more realistic assessment.
   - This is due to the structure evolving over time, as opposed to being formed instantly.
3. Deflection due to wind and other short term loading events in concrete structures is determined using different properties to those used for sustained loading.
   - This is achieved through the use of load type dependent design cases, each with appropriate modification factors - see Staged construction modification factors (page 470) for more details.
4. A more optimized design is delivered.
   - The engineer can choose between staged construction analysis or 3D building analysis with either or both of FE chasedown and grillage chasedown.
   - The different analyses mean that an envelope of possible design forces have been considered.
   - Swapping out 3D building analysis for staged construction analysis (more realistic) will reduce this envelope of design forces, leading to a more optimized design, potentially reducing the quantities of materials.

     For further details, see: Design comparison - with and without staged construction (page 481)

Note however that staged construction analysis in Tekla Structural Designer doesn't enable you to examine design conditions on partially completed structures (e.g. wind effects on a half built building).

Scope in Tekla Structural Designer

Staged construction analysis is applicable to both concrete and steel structures.

- **All of the concrete design codes** available in Tekla Structural Designer take account of staged construction analysis design forces.
- Only the **Eurocode steel design code** initially considers staged construction analysis design forces.
- The Staged Construction load combination can be analyzed using either 1st or 2nd order linear, 1st or 2nd order nonlinear analysis.
- The static part of a Seismic RSA design combination can also come from a staged construction analysis.

Structures in other materials that can be designed via Tekla Tedds (e.g. timber, precast) also consider staged construction analysis results.

Implementation in Tekla Structural Designer

In Tekla Structural Designer staged construction has been implemented in a way that is both simple and intuitive:

1. Set up, review and edit the construction stages
   - A series of construction stages are defined by the engineer to idealize the evolution of the structure.
2. Define a **Staged Construction** load combination.
   - A single staged construction load combination is defined, typically consisting of self weight plus an additional construction loading allowance.
3. In **Design Settings** choose to include staged construction analysis.
   - The available choices allow you to control which types of analysis results get considered in design - either of 3D building analysis or staged construction analysis with either or both FE chasedown and grillage chasedown.
4. Run **Design** in the normal way.
   - Analysis results can be reviewed on a stage by stage basis.
   - The design is the same, but with new forces.

See also

Overview of staged construction analysis (page 456)

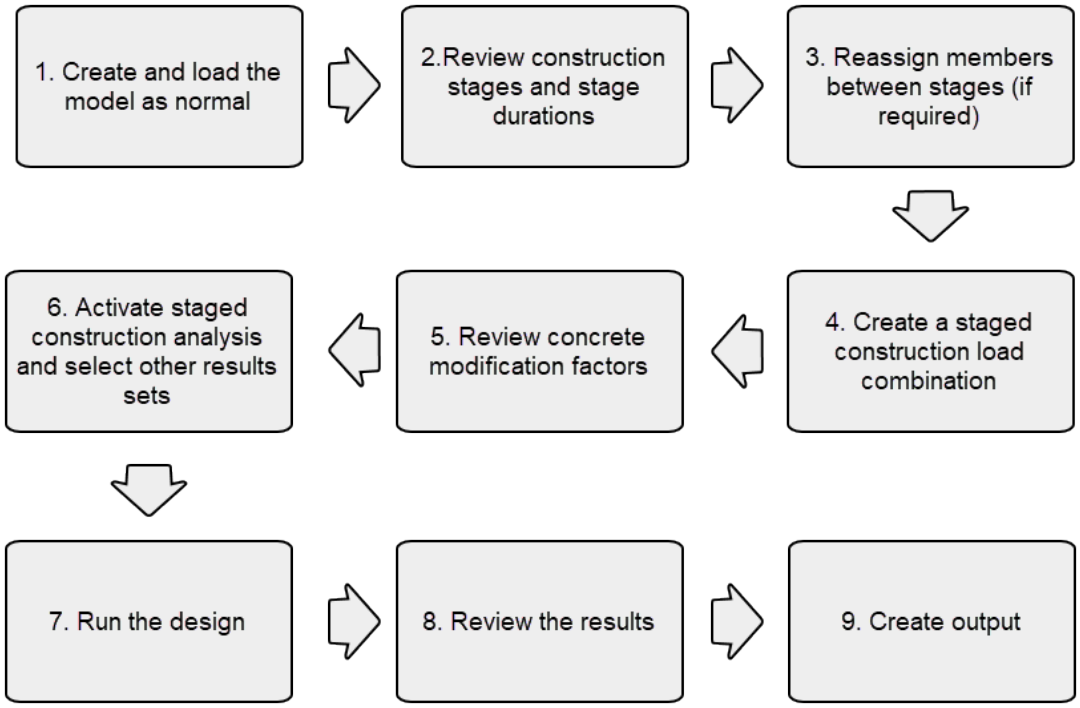
Staged construction limitations and assumptions (page 480)

Staged construction workflow example - Eurocodes (page 489)

Staged construction workflow example - United States (ACI/AISC) (page 511)

7.2 Overview of staged construction analysis

Regardless of the design guide being worked to, the main steps required for a staged construction analysis and design in Tekla Structural Designer are the same.



Tekla Structural Designer's ease-of-use methodology simplifies the above process. Defaults are applied to minimize the amount of user input, but you can then override whichever parameters you deem necessary to satisfy your design requirements.

See below for more information about each of the steps.

Create and load the model as normal

The same general approach for creating the model can be adopted irrespective of whether you intend to run a staged construction analysis or not. The model can be created, loads can be applied and design load combinations generated in the normal fashion.

A staged construction load combination will also need to be created at some point but this can be done after the construction stages have been specified.

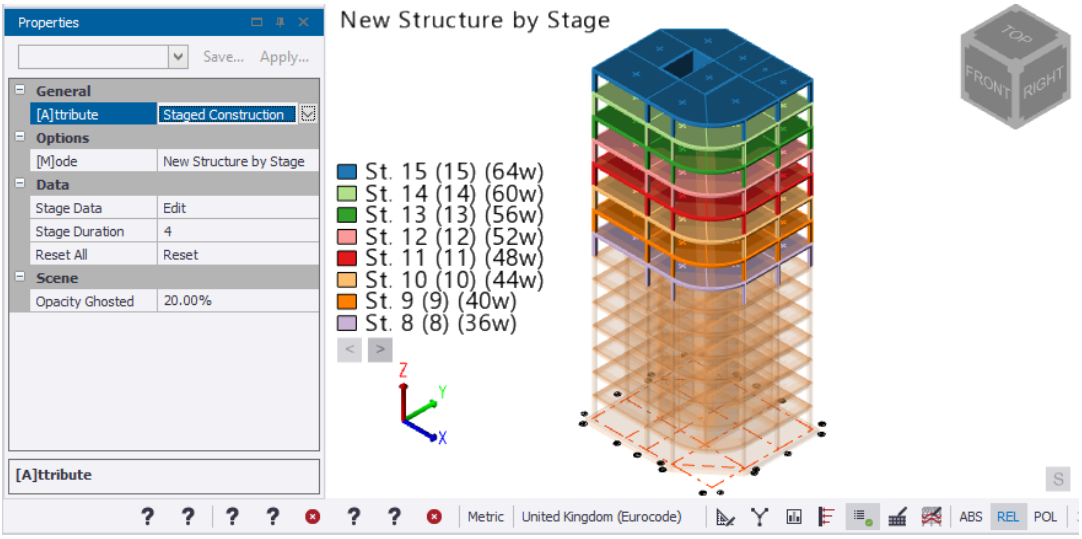
Review construction stages and stage durations

The reason for splitting the model into construction stages is to allow for the fact that not all forces will be applied at the same time, and also to cater for materials with time dependent effects.

In Tekla Structural Designer construction stage definition takes place in the Review View regime.

To see the default stages that are created automatically, select the Show/Alter State command on the Review ribbon and then the Staged Construction attribute in the Properties window.

With the mode set to New Structure by Stage, the default construction stages are shown and their durations indicated in a legend.



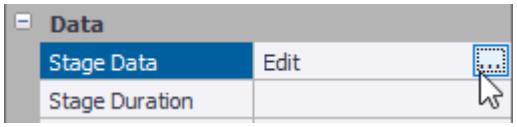
For tall structures click the < > buttons underneath the legend to see more stages.

TIP It can be a lot easier to review stages by using the animate option, pause it, and then step forward/back through the stages.

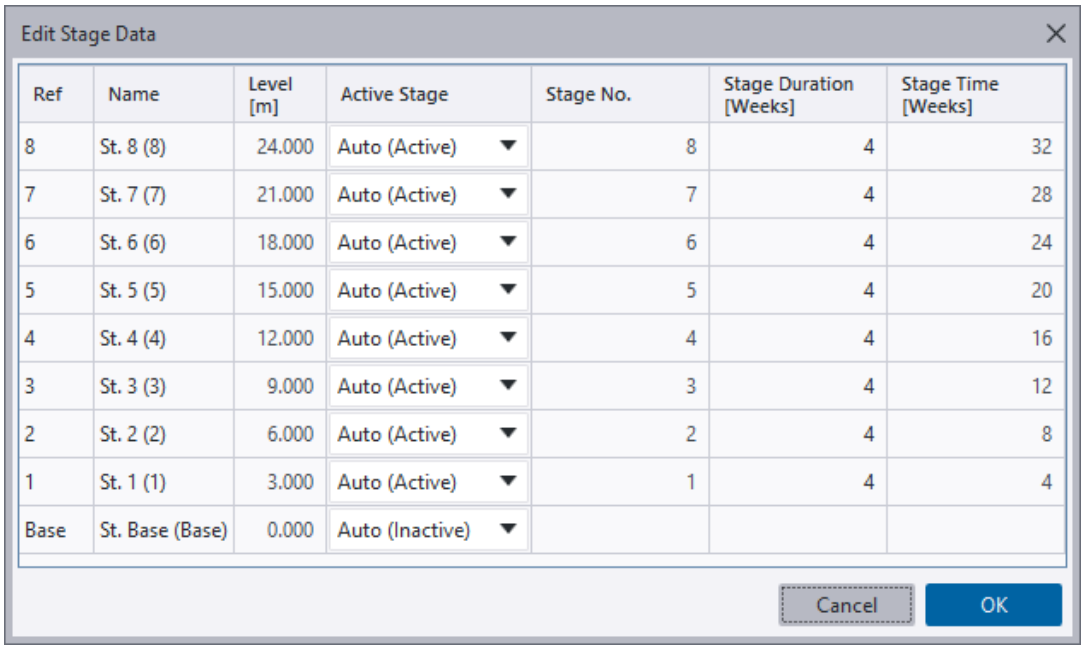
Floor levels (specifically, levels marked as floors) denote potential construction stages. Default stages are automatically created so that members in a floor level or attached to that level from below form an individual stage.

Initially each stage defaults to a stage duration of 4 weeks.

Default stages can be modified as required by clicking the Edit button next to Stage Data.

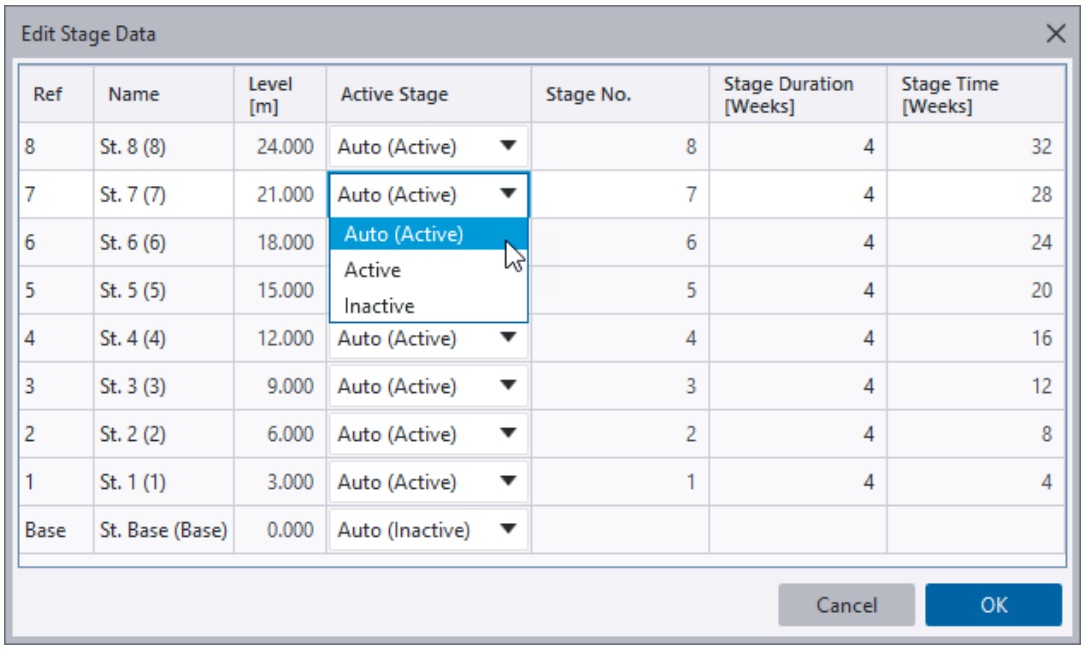


This displays the Edit Stage Data dialog box which allows the number of stages and the stage durations to be edited.



Each row in the 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.



Initially each floor defaults to either Auto (Active), or Auto (Inactive):

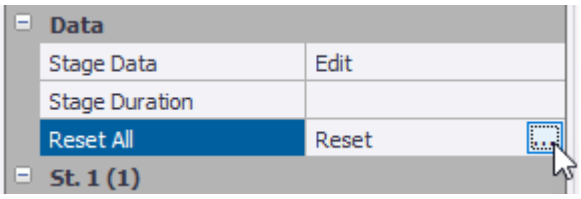
- Auto (Active) - the floor level is an active stage if it contains a slab, (a level with beams and no slabs is not automatically active).

- Auto (Inactive) - the floor level is an inactive stage if it doesn't contain a slab.
- The one exception to the above is the top floor level which is always automatically active.
- 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 (page 476).

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.

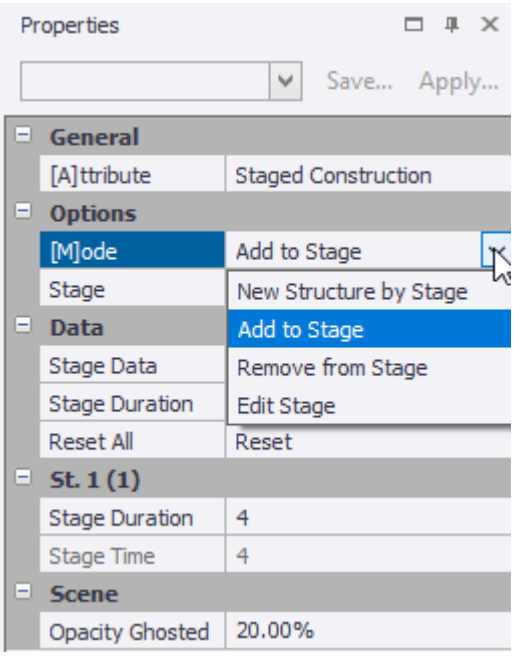


Note that if you do click Reset All:

- 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.

Reassign members between construction stages

Once the active construction stages have been set, members can be reassigned between stages if necessary. Add to Stage and Remove from Stage editing modes are provided in the Properties window for this purpose.



VALIDATION RULES

There are certain validation rules governing the assignment of members to stages.

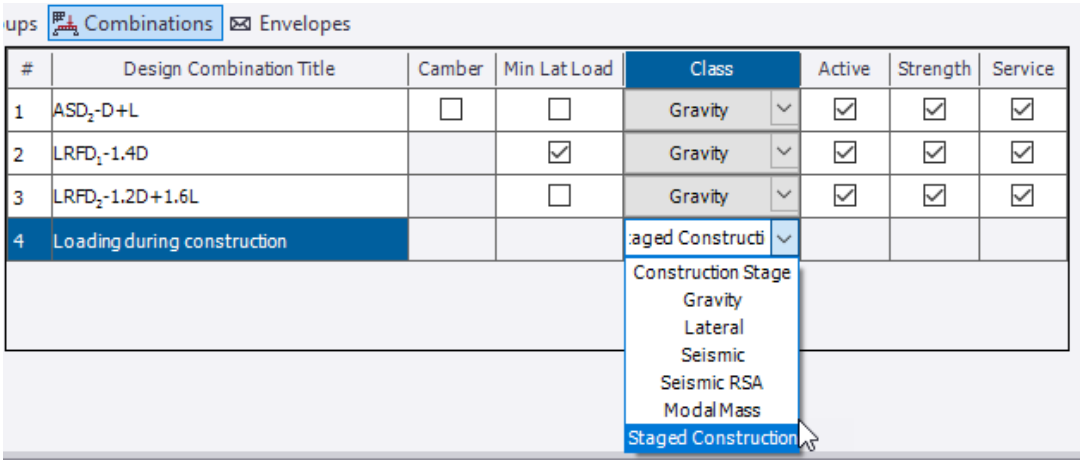
- The final construction stage must include all members in the model (it cannot be edited)
- Only objects that affect the analysis model are available to be included in the earlier stages, this includes:
  - 1D member spans / stacks - all characteristics (beams, columns, joists, etc)
  - Trusses are included / excluded in their entirety
  - Portal frames are included / excluded in their entirety
  - Mid-pier, meshed, and shear only wall spans / stacks
  - Slabs - all slab items in a plane are included / exclude in their entirety
    - Note that a slab can only be included if all the beams that support it are included.
  - Roof panels
    - Similar to slabs, a roof panel can only be included if all the beams that support it are included.
  - Supports are included if any member touching the support is included.

- The following objects don't need to be included in earlier stages:
  - Wall Panels
  - Loading Objects (These are not required as loading on the construction stage models is derived from load cases that have already been defined).

Create a staged construction load combination

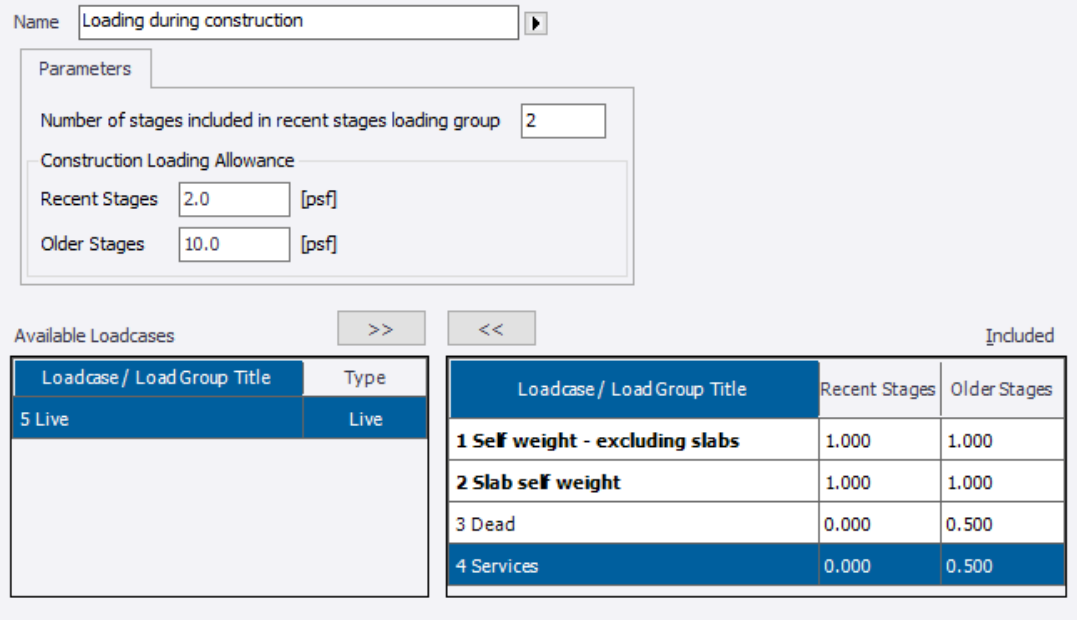
While the standard design combinations should have already been defined, you will not be able to run a staged construction analysis until a dedicated staged construction combination has been created.

This combination has to be added manually in the Loading dialog box, taking care to set its class to Staged Construction.



You would typically want to define this combination so that self weight plus an additional construction loading allowance are applied to the most recent construction stages, while at the same time a different construction allowance and some proportion of dead load is applied to older stages.

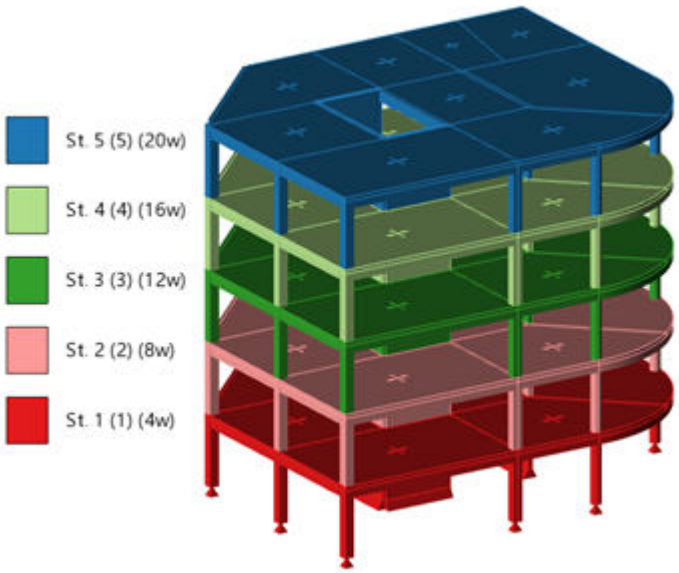
The resulting construction stage combination would then effectively describe the loading for every construction stage in a building of any size.



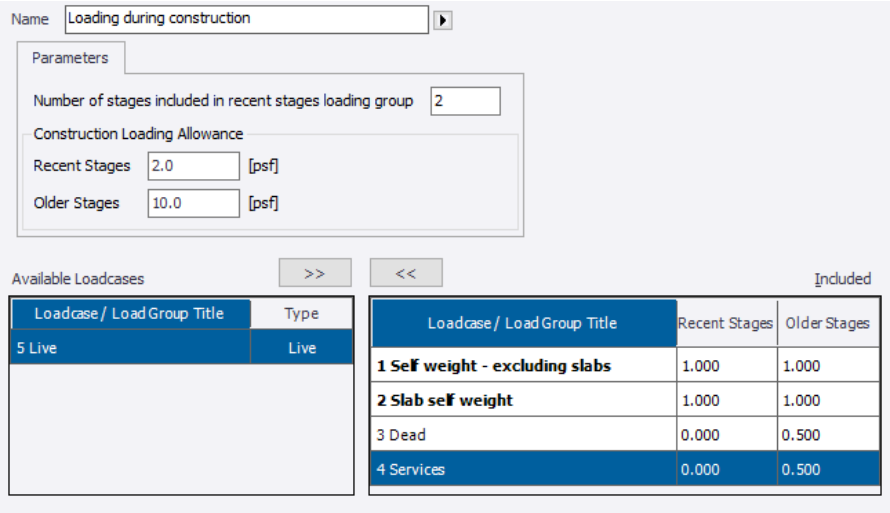
To make this possible, two load combination groups are provided; one for the recent stages and one for older stages. Both groups have their own set of load factors. There is provision for you to say how many stages are included in the recent stages group and also to specify a different construction loading allowance for each group.

RECENT AND OLDER STAGES LOAD FACTORS

To illustrate how the load factors are applied to recent and older stages, consider the following 5 stage (5 floor levels) example.



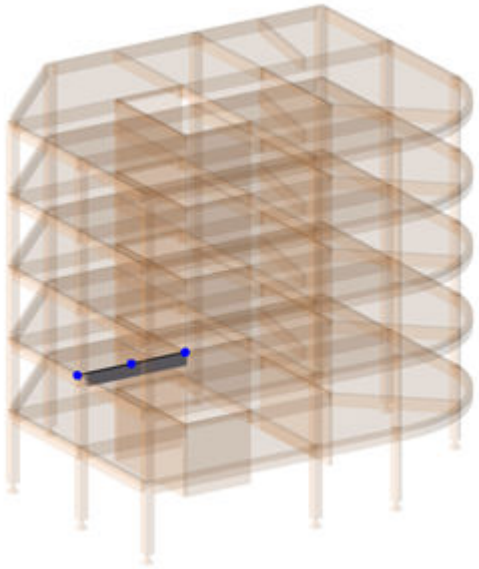
The staged construction combination has been defined as follows:



The number of stages in the recent stages group is set to 2 and the services load factors are set to:

- Recent stages: 0.0
- Older stages: 0.5

Consider the following beam which is introduced in stage 2 and has a services load of 0.7klf (10kN/m). (This might be a directly applied load or a load from 1 or 2 way decomposition.)



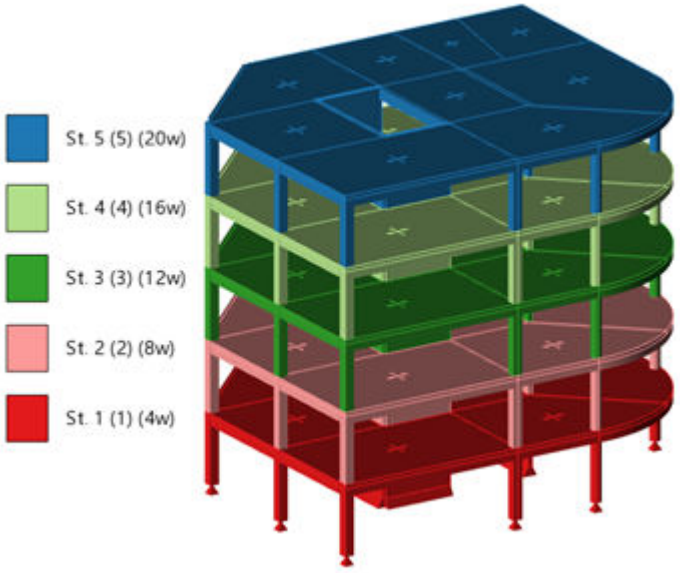
- The beam does not exist in the stage 1 model
- In stages 2 and 3 the beam would be in a recent stage, so the services load contribution = 0.0 * 0.7 = 0klf (0kN/m)

- In stages 4 and 5 the beam would move to an older stage, so the services load contribution = 0.5 * 0.7 = 0.35klf (5kN/m)

The same procedure applies to all the loadcases used in the construction stage loading combination, and is applied at every stage to determine the loading on every member.

CONSTRUCTION LOADING ALLOWANCE

To illustrate how the construction loading allowance is applied to recent and older stages, consider the following 5 stage (5 floor levels) example.



The staged construction combination has been defined as follows:

Name: Loading during construction

Parameters

Number of stages included in recent stages loading group: 2

Construction Loading Allowance

Recent Stages: 2.0 [psf]

Older Stages: 10.0 [psf]

Available Loadcases >> <<

Loadcase / Load Group Title	Type
5 Live	Live

Included

Loadcase / Load Group Title	Recent Stages	Older Stages
1 Self weight - excluding slabs	1.000	1.000
2 Slab self weight	1.000	1.000
3 Dead	0.000	0.500
4 Services	0.000	0.500

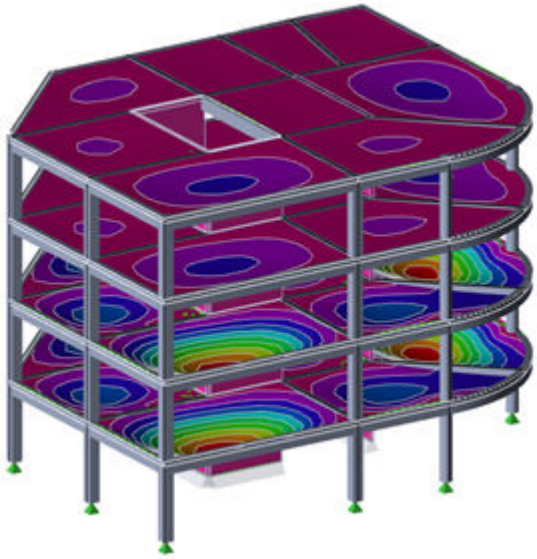
The construction loading allowance is set to:

- Recent stages: 2.0psf (0.1kN/m^2)
- Older stages: 10.0psf (0.5kN/m^2)

There are 2 stages included in the recent stages group, so if we consider one of the construction stages in isolation (for example stage 4), it would be have construction loading allowances applied as follows:

- Construction loading allowance applied to level 4 and 3, (the recent stages for stage 4) = 2.0psf (0.1kN/m^2)
- Construction loading allowance applied to level 2 and 1 (the older stages for stage 4) = 10.0psf (0.5kN/m^2)

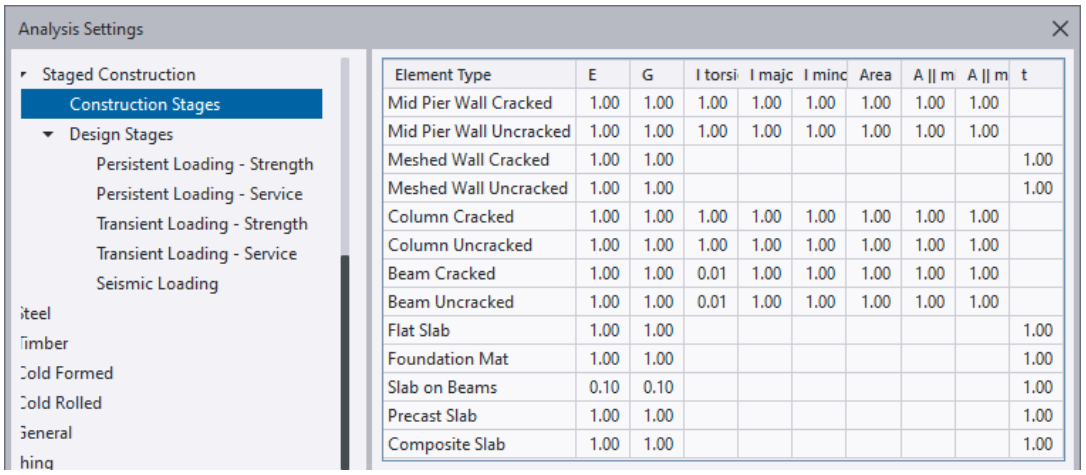
The above loading allowance would therefore deflect construction stage 4 as follows:



Review concrete modification factors

By adopting a staged construction approach the load dependent nature of creep and cracking in concrete is catered for by the application of modification factors that are appropriate to the persistent or transient nature of loading. Furthermore, separate factors take account of the difference between service and ultimate load levels.

They can be reviewed by selecting Settings on the Analyze ribbon and then navigating to the appropriate Modification Factors > Staged Construction page in the Analysis Settings dialog.



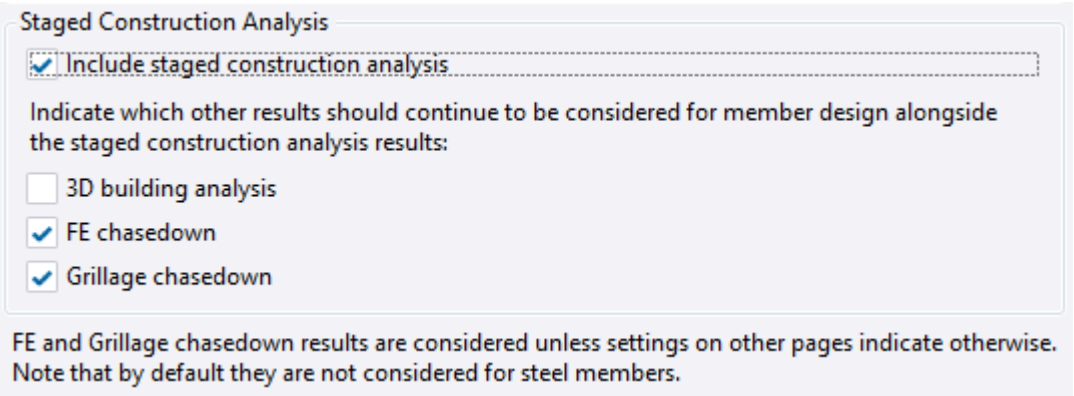
Default values have been specifically calibrated for the construction stages, and for each of the design stages.

For further information, see: Staged construction modification factors (page 470)

Activate staged construction analysis and select other results sets

Before you run a batch design you must specify which analysis type(s) are to be run. This is configured via the Analysis page in Design Settings.

- Firstly, you need to select whether the analysis should be first-order or second-order. The chosen setting gets applied to 3D building analysis and/or staged construction analysis.
- Next you need to include staged construction analysis.
- Finally you can select which other results should also be considered in the design.



When staged construction analysis is included, by default both chasedowns are also included and 3D building analysis is excluded.

In effect this means that the staged construction analysis results are replacing the 3D building analysis results at one end of the design envelope. Refer to design comparison - with and without staged construction (page 481) to find out more about the implications of this.

Run the design

Once the analysis type(s) have been set as required, you can then proceed to run the combined analysis and design. This is initiated by selecting the required design choice (e.g. Design All (Static)) from the Design ribbon.

The choices you made on the Analysis page in Design Settings determine the analysis types that will be performed and also whether a first-order or second-order analysis is used for staged construction analysis and/or 3D building analysis.

If there are any non-linear elements in the model the staged construction analysis and/or 3D building analysis will run as non-linear, otherwise it will be linear.

The Staged Construction combination is analyzed for each construction stage sequentially to determine the evolution of forces during the construction stages. The forces at the final construction stage then become a starting point for the analysis of design stages.

The gravity and lateral combinations are each assigned to a design stage (according to loadcase types they contain), they are then analyzed and designed accordingly.

Review the results

Once the model has been designed you are taken to the Review View from where the design can be examined.

If you switch to the Results View regime and select the staged construction method from the Result Type group in the Results ribbon, you can then investigate the analysis results for the staged construction loading combination and each of the design combinations. When the staged construction loading combination is selected in the Loading toolbar, a Stage list is added to Result Type group to allow you to select each of the construction stages.

If you switch to the Solver Model View regime and select the staged construction method in the Solver ribbon a Stage list is added to the Solver ribbon. This allows you to select and review the solver model applicable for each construction stage and design stage.

Create output

Reports and drawings are created in the normal way from the Report ribbon and Draw ribbon respectively.

See also

Construction stages and design stages (page 469)

Staged construction modification factors (page 470)

Concrete creep allowance during construction stages (page 476)

Staged construction limitations and assumptions (page 480)

Design comparison - with and without staged construction (page 481)

Staged construction workflow example - Eurocodes (page 489)

Staged construction workflow example - United States (ACI/AISC) (page 511)

Construction stages and design stages

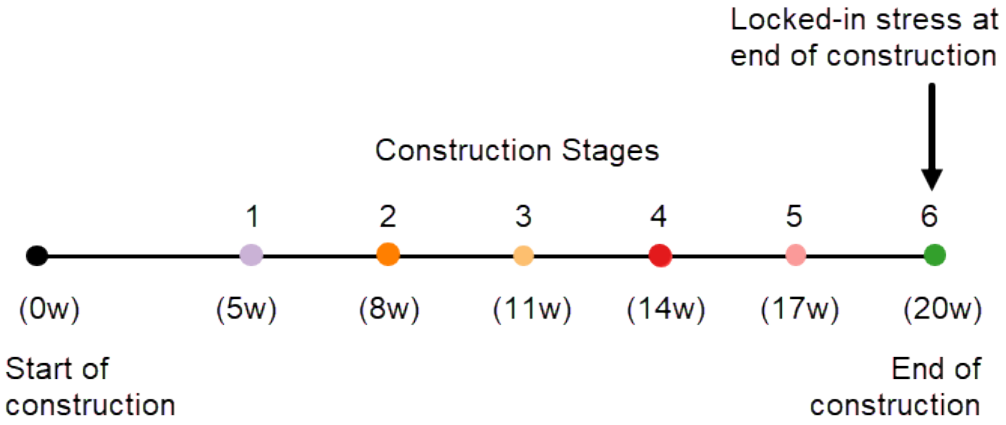
Construction stages

Floor levels (specifically, levels marked as floors) denote potential construction stages. Default construction stages are automatically created so that members in a floor level or attached to that level from below form an individual construction stage. You can reassign members between stages if required.

Each construction stage is assigned a stage duration (in weeks).

When the staged construction analysis is performed, each construction stage is analyzed sequentially for the Staged Construction combination only.

Stresses that develop in one construction stage become “Locked-in stresses” that are input to the following stage. At the end of the construction stages there is a set of “Locked-in stresses” that are carried through to the design stages.



Design stages

Design stages only come in to play after the end of construction, (so they don't have stage durations).

Any model can potentially have up to five design stages, each having its own set of stiffness modification factors (page 470). The design stages are:

- Persistent Loading - Strength
- Persistent Loading - Service
- Transient Loading - Strength
- Transient Loading - Service
- Seismic Loading

When the staged construction analysis and design is performed, the Gravity, Lateral and Seismic combinations are automatically assigned to a design stage according to loadcase types contained within the combination. For example, any combination including wind will be considered transient and will be analyzed using properties appropriate to the "Transient Loading - Strength" and "Transient Loading - Service" design stages.

LOAD COMBINATION ASSIGNMENT

The rules for the assignment are as follows:

1. IF a combination includes any seismic case it is assigned to the **Seismic Loading** design stage.
2. ELSE IF a combination includes any of the following cases it considered as **Transient Loading**:
   - Wind
   - Snow
   - MLL
   - Temperature
3. ELSE the combination is considered as **Persistent Loading**
4. The **Strength** (ULS) and **Service** (SLS) variants of the combination are then separated as appropriate.

The Gravity, Lateral and Seismic combinations are then designed accordingly, (the Staged Construction combination is *not* designed).

See also

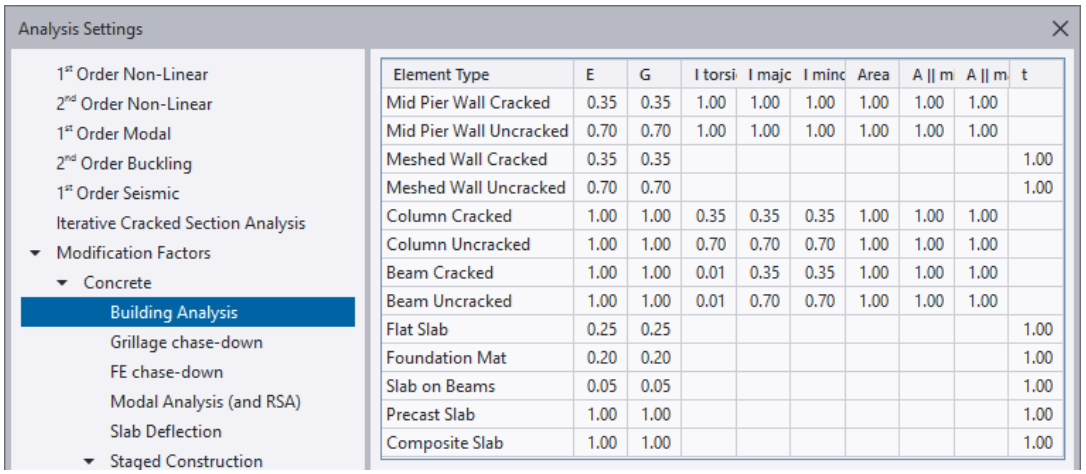
Overview of staged construction analysis (page 456)

Staged construction modification factors

'Unstaged' modification factors

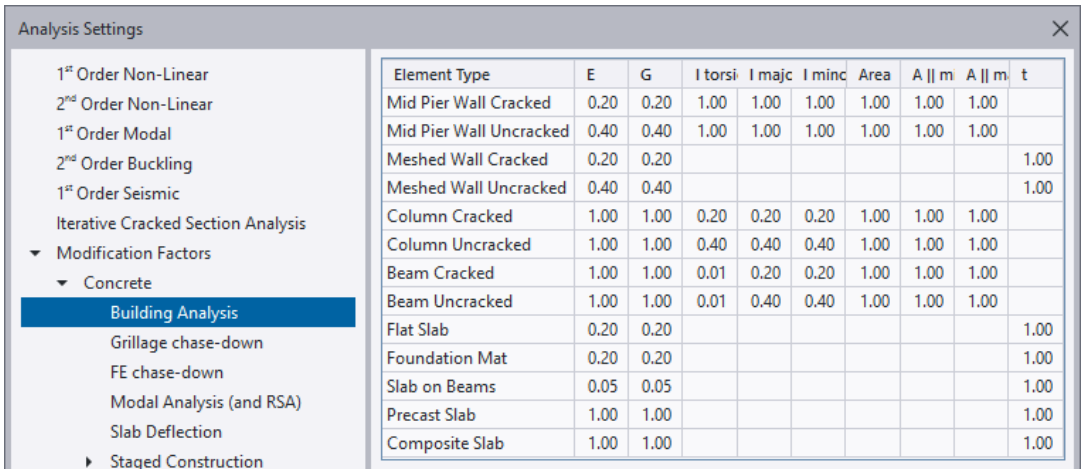
A standard 'unstaged' 3D building analysis in Tekla Structural Designer is limited by the fact that the modification factors defined in the Analysis Settings dialog apply irrespective of the nature of the loading. For the majority of materials the factors default to 1.0, but for concrete different defaults apply, which vary depending on the design code.

If working to ACI/AISC the 'unstaged' defaults are:



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.35	0.35	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	0.70	0.70	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.35	0.35							1.00
Meshed Wall Uncracked	0.70	0.70							1.00
Column Cracked	1.00	1.00	0.35	0.35	0.35	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	0.70	0.70	0.70	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.35	0.35	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	0.70	0.70	1.00	1.00	1.00	
Flat Slab	0.25	0.25							1.00
Foundation Mat	0.20	0.20							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

If working to Eurocodes the 'unstaged' defaults are:



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.20	0.20	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	0.40	0.40	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.20	0.20							1.00
Meshed Wall Uncracked	0.40	0.40							1.00
Column Cracked	1.00	1.00	0.20	0.20	0.20	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	0.40	0.40	0.40	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.20	0.20	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	0.40	0.40	1.00	1.00	1.00	
Flat Slab	0.20	0.20							1.00
Foundation Mat	0.20	0.20							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

In the absence of any more rigorous approach these factors are deemed to cater for creep and cracking effects in concrete to a degree that is sufficient for 2nd order analysis at ULS design load levels. Clearly this is very simplistic because both creep and cracking are load dependent, so the use of a single factor for all load combinations, and also for the service and ultimate load levels in any single combination can never be “correct”.

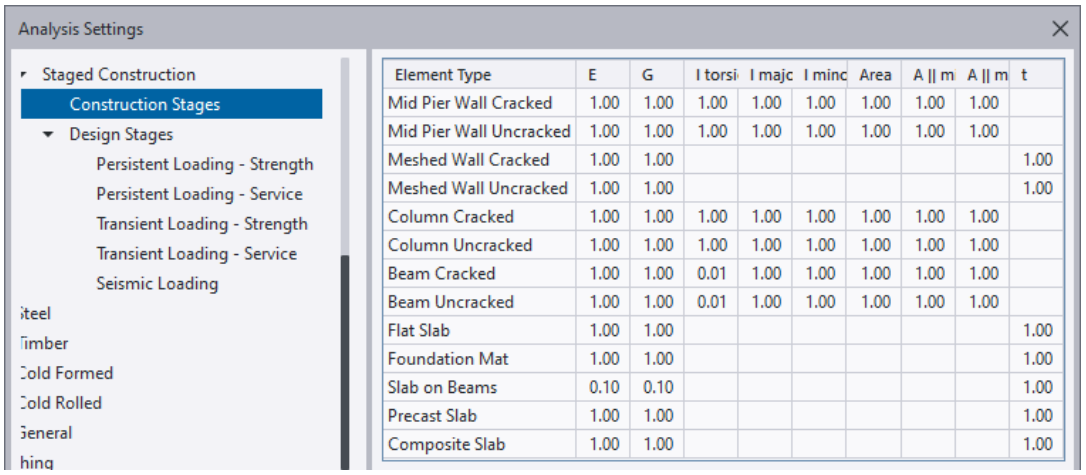
Staged construction modification factors

By adopting a staged construction approach the load dependent nature of creep and cracking in concrete can be catered for.

This is achieved as follows:

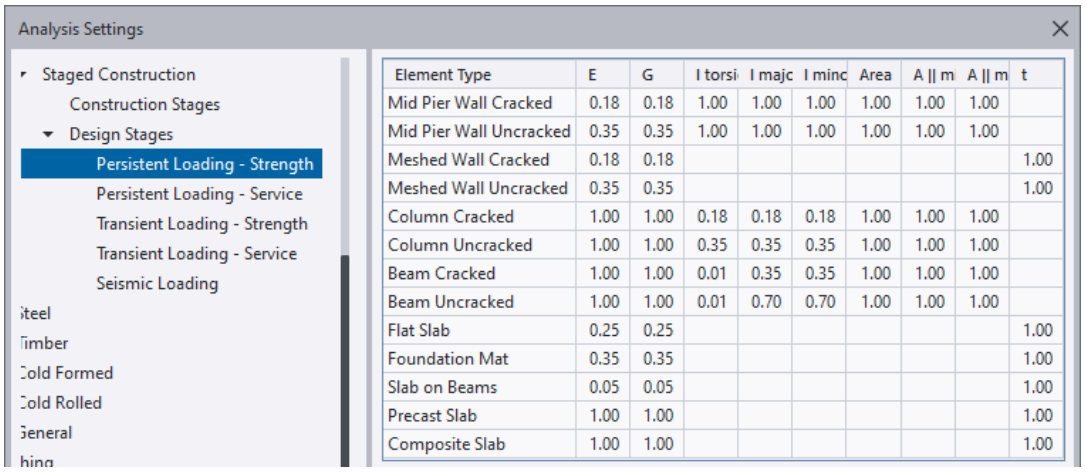
- A single set of 'Construction Stage' modification factors are specified for all of the construction stages.
- Five separate sets of 'Design Stage' modification factors are specified appropriate to the persistent or transient nature of loading, and also to the service or ultimate load level.
- Unlike the solver models used for other analysis types, the staged construction solver model is split into a series of construction stages and design stages, with the section properties of elements in each stage having modification factors applied appropriate to that stage.
    - Each of the construction stages use the same set of 'Construction Stage' modification factors in the solver model. Only those members assigned to the selected construction stage and previous stages are included in the solver model for that stage.
    - Each of the design stages use the appropriate 'Design Stage' modification factors for that particular stage in the solver model. Because the design stages occur after the end of construction, all members are included in the solver model for each of the design stages.
    - To see the effect of the modification factors, select a solver element in the solver model. In the Properties window the section/material properties update accordingly as you move from one stage to the next.
- The 'Staged Construction' load combination is analyzed for each construction stage solver model using the same 'Construction Stage' modification factors.
- The remaining load combinations are analyzed for the appropriate design stage solver model using the appropriate 'Design Stage' modification factors, (load combinations having been automatically assigned to Design Stages according to the loadcases they contain.)

The Construction Stage defaults are:



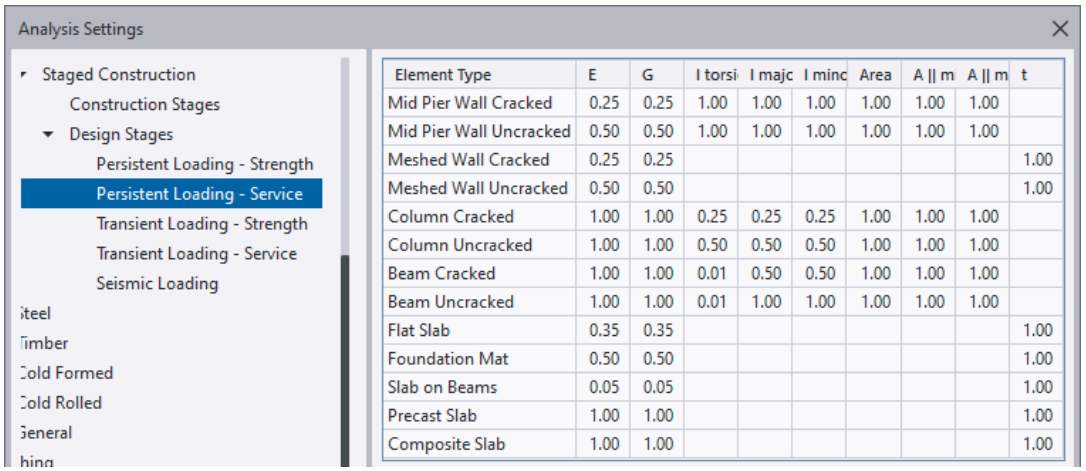
NOTE Creep during construction is catered for separately by an adjustment to the E and G values assigned to each member at each construction stage, (based on the age of the member at that stage). For details of how this is calculated, see: Concrete creep allowance during construction stages (page 476)

The 'Persistent Loading - Strength' Design Stage defaults are:



NOTE The default factors shown above have an allowance for creep built in.

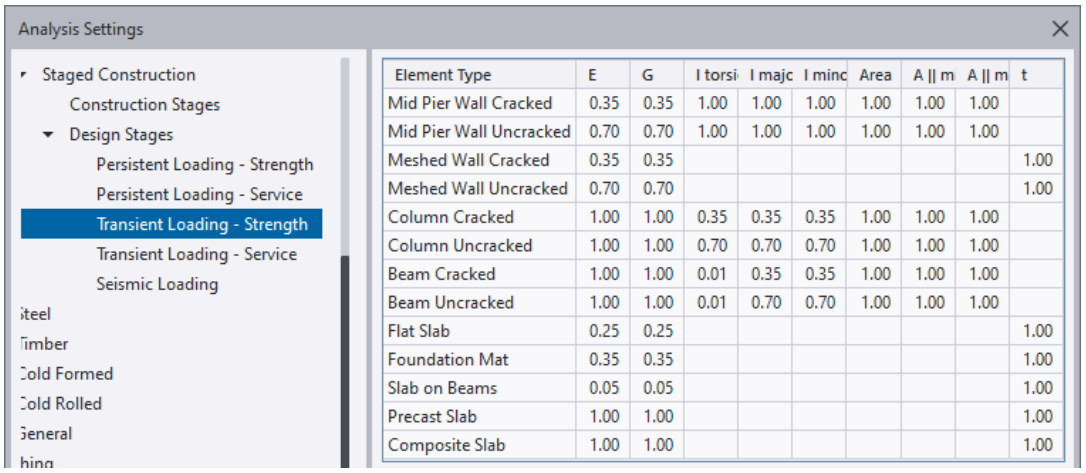
The 'Persistent Loading - Service' Design Stage defaults are:



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.25	0.25	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	0.50	0.50	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.25	0.25							1.00
Meshed Wall Uncracked	0.50	0.50							1.00
Column Cracked	1.00	1.00	0.25	0.25	0.25	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	0.50	0.50	0.50	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.50	0.50	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Flat Slab	0.35	0.35							1.00
Foundation Mat	0.50	0.50							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

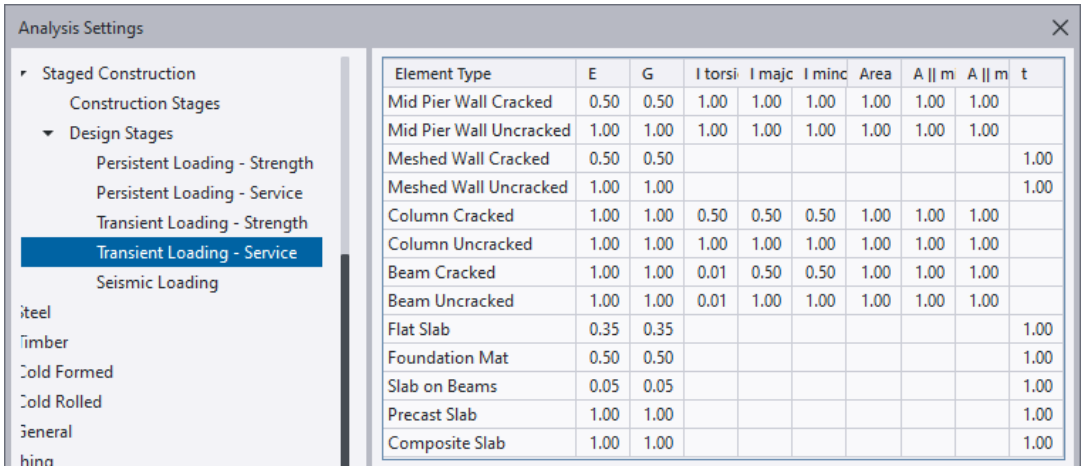
NOTE The default factors shown above have an allowance for creep built in.

The 'Transient Loading - Strength' Design Stage defaults are:



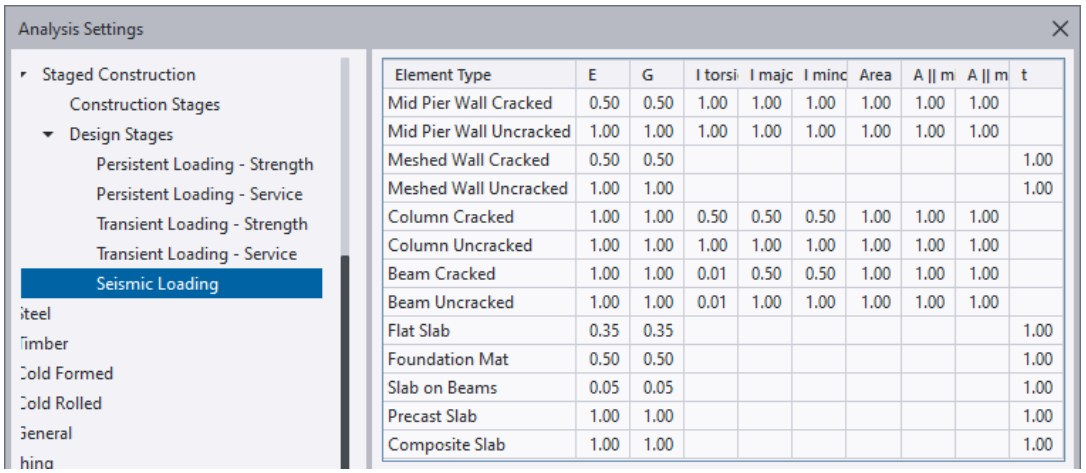
Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.35	0.35	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	0.70	0.70	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.35	0.35							1.00
Meshed Wall Uncracked	0.70	0.70							1.00
Column Cracked	1.00	1.00	0.35	0.35	0.35	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	0.70	0.70	0.70	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.35	0.35	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	0.70	0.70	1.00	1.00	1.00	
Flat Slab	0.25	0.25							1.00
Foundation Mat	0.35	0.35							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

The 'Transient Loading - Service' Design Stage defaults are:



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.50	0.50	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.50	0.50							1.00
Meshed Wall Uncracked	1.00	1.00							1.00
Column Cracked	1.00	1.00	0.50	0.50	0.50	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.50	0.50	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Flat Slab	0.35	0.35							1.00
Foundation Mat	0.50	0.50							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

The 'Seismic Loading' Design Stage defaults are:



USE OF MODIFICATION FACTORS IN SWAY AND DRIFT CHECKS

Staged construction analysis enables assessment of drift using short term properties and member design using long term properties in a single design process as follows:

1. Enable staged construction analysis for design
   - This means that the analysis for gravity design uses the relevant “persistent loading - strength” modification factors and hence analysis results at the relevant level of 2nd order effects.
   - An added benefit is that analysis for ULS design of (transient classed) wind combinations uses the transient modification factors and hence potentially reduced 2nd order effects.
2. Edit the building analysis modification factors to values relevant for wind (or seismic) assessments* then run a single design process.
   - *The recommendation is that these are reset to be the same as those for staged construction analysis “transient loading-service” or “seismic loading”
     - Note that by default these factors are the same (based on ACI guidance).
   - Both Wind and Seismic (ELF) drift checks are based on the analysis of wind loads in isolation. Staged construction analysis does not (cannot logically) solve individual loadcases, so the loadcase results used for wind drift continue to derive from the building analysis. With the adjusted factors these checks will be less onerous (as desired).

See also

Overview of staged construction analysis (page 456)

Concrete creep allowance during construction stages

Creep during construction stages can be allowed for on a simplified basis by using the US code approach described in ACI-318-11 clause 9.5.2.5.

Adopting this approach, Tekla Structural Designer uses the following table to make an automatic age based adjustment to the E & G values applied to concrete element properties.

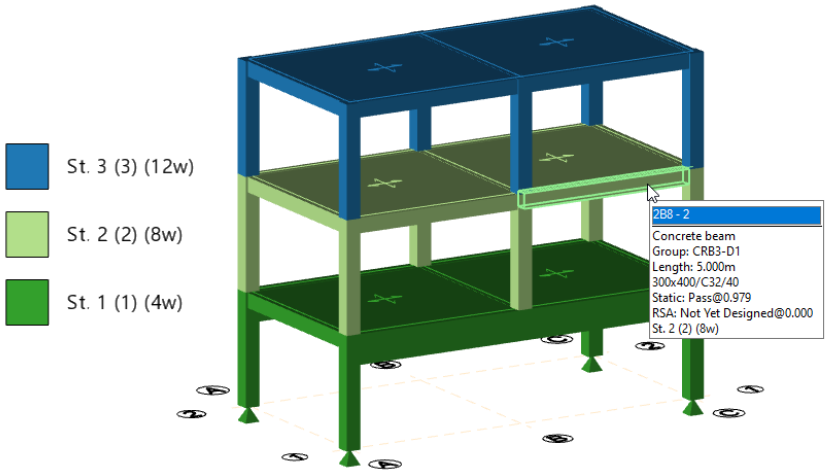
AGE	Beams and Slabs	Columns and Walls
0	1.00	1.00
4.33 w	0.62	0.79
13 w	0.51	0.65
26 w	0.46	0.60
52 w	0.41	0.56
156 w	0.36	0.55
260 w (or greater)	0.34	0.55

The adjustment calculated for each member in each stage is as follows:

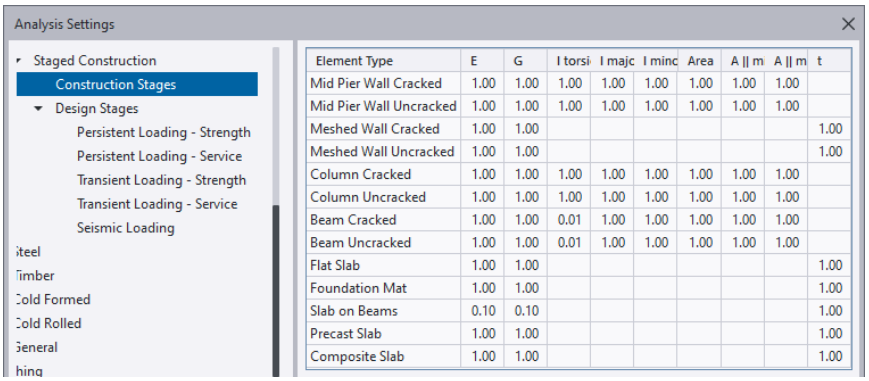
1. Determine the AGE of the member at the end of the stage currently under consideration:
   - ST_end = Stage Time for analysis model stage
   - ST_start = Stage time for the stage **before** the element was introduced
   - AGE = ST_end - ST_start
2. Calculate the additional creep adjustment factor by interpolation between the relevant values in the above table.
3. Adjust the E and G values.

CREEP ALLOWANCE EXAMPLE

For a construction stage 3 analysis model with total construction stage time = 12w, determine the adjustment for the cracked concrete beam introduced in stage 2 shown below.



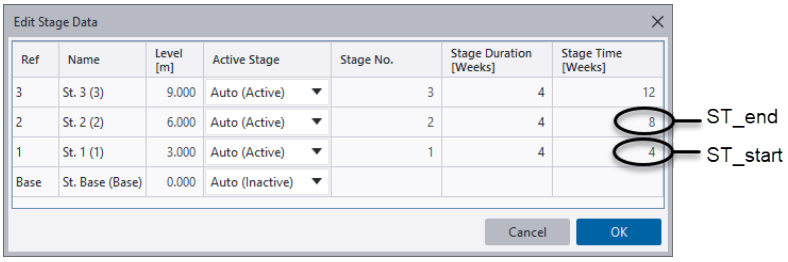
As shown in the Construction Stages modification factors table, the beam's unadjusted E and G modification factors = 1.00



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	1.00	1.00							1.00
Meshed Wall Uncracked	1.00	1.00							1.00
Column Cracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Flat Slab	1.00	1.00							1.00
Foundation Mat	1.00	1.00							1.00
Slab on Beams	0.10	0.10							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

To calculate the age based adjustment for this beam:

1. Determine the AGE of the beam at the end of stage 2:

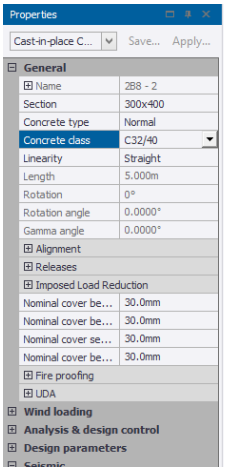


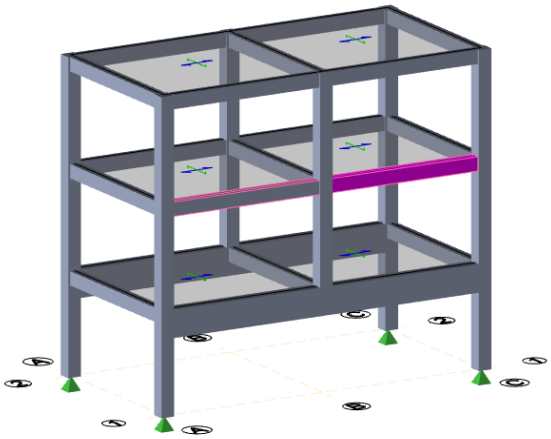
- ST_end = Stage Time for analysis model stage = 8w
- ST_start = Stage time for the stage **before** the element was introduced = 4w
- AGE = ST_end - ST_start = 4w

2. Calculate the additional creep adjustment factor at this age by interpolation between the relevant values in the table.
    - Adjustment factor for a beam at 4w = 0.649
3. Adjust the E value:
    - The beam's original E modification factor = 1.0
    - The total adjustment applied to E = 0.649 * 1.0
4. Adjust the G value:
    - The beam's original G modification factor = 1.0
    - The total adjustment applied to G = 0.649 * 1.0

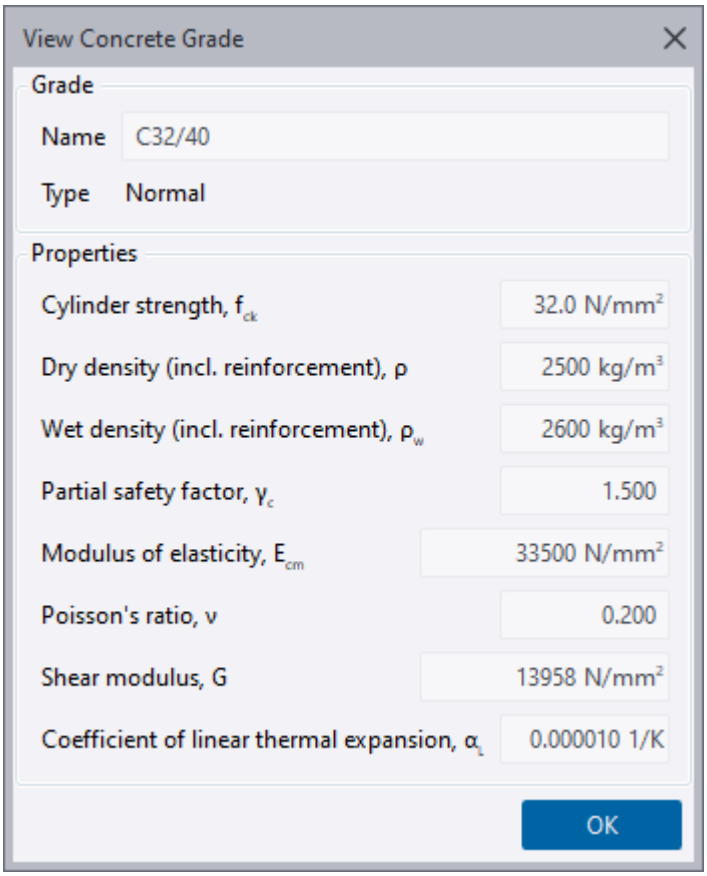
To verify the total adjustment applied:

1. Open a **Structural View** containing the element
2. Select the element and confirm the concrete class in the **Properties** window.

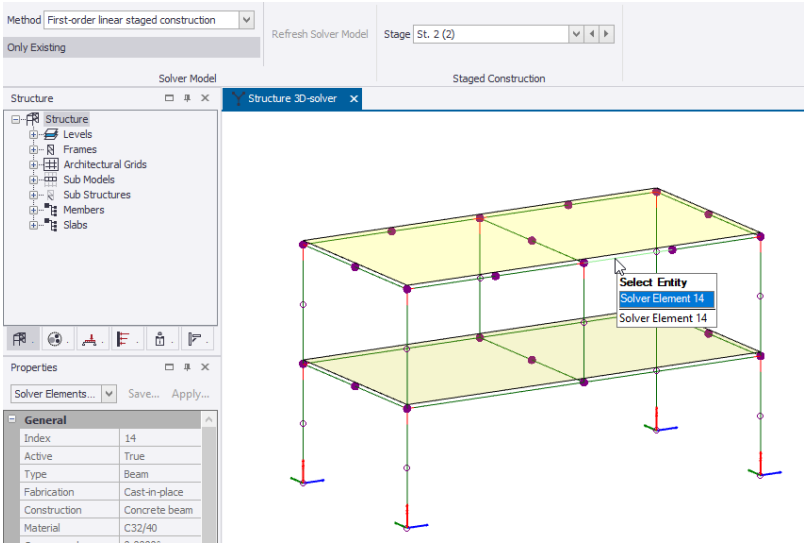




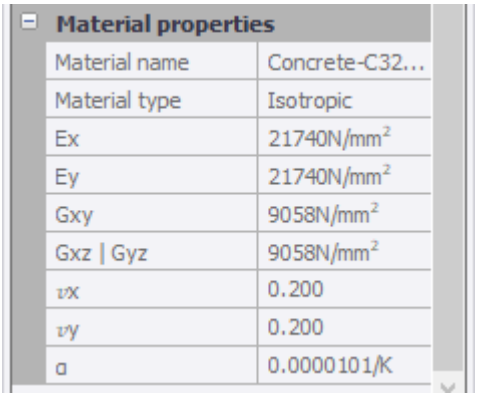
3. The unadjusted E and G values for this class are 33500 and 13958 respectively, as can be confirmed in the Materials database.



4. Switch the view regime to a **Solver View**
5. On the **Solver** ribbon, set the **Method** to First-order linear staged construction
6. On the **Solver** ribbon, select the required Construction Stage
7. Select the element In the **Solver View**



8. In the Material properties section of the **Properties** window, view the adjusted E and G values (21740 and 9058 respectively).

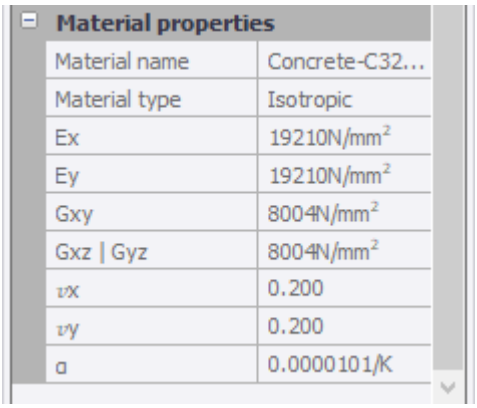


Material properties	
Material name	Concrete-C32...
Material type	Isotropic
Ex	21740N/mm²
Ey	21740N/mm²
Gxy	9058N/mm²
Gxz | Gyz	9058N/mm²
νx	0.200
νy	0.200
α	0.0000101/K

9. The ratio of the adjusted to the unadjusted values should match the calculated creep adjustment factor.

   21740/ 33500 = **0.649** , 9058/13958 = **0.649**

10. Note that as shown below, if you were to select the next stage (St. 3) in the **Solver** ribbon the beam's properties would change to reflect it's age (8 weeks) at the end of stage 3. The adjusted E and G values now being 19210 and 8004 respectively.



Material properties	
Material name	Concrete-C32...
Material type	Isotropic
Ex	19210N/mm²
Ey	19210N/mm²
Gxy	8004N/mm²
Gxz | Gyz	8004N/mm²
νx	0.200
νy	0.200
α	0.0000101/K

   The ratio of the adjusted to the unadjusted values equates to the new creep adjustment factor.

   19210/ 33500 = **0.573** , 8004/13958 = **0.573**

   0.573 being the adjustment obtained by interpolating for an age of 8 weeks in the table.

See also

Overview of staged construction analysis (page 456)

7.3 Staged construction limitations and assumptions

Limitations

The following limitations apply to the first release:

1. Steel design for staged construction results has only been implemented for Eurocodes.
2. No allowance for objects being removed. (e.g. propping during construction).
3. Multiple stage construction scenarios are not supported.
4. Creep during construction - in Tekla Structural Designer a simple approach has been adopted (based on simplified guidance from ACI codes).
5. Construction stage forces are not considered in design. (These are not expected to be controlling design conditions.)
6. There is no option to elevate loading levels to ultimate at intermediate construction stages.

   As a result it is not possible to examine design conditions on partially completed structures (e.g. wind effects on a half built building). To investigate this scenario the workaround is to make a copy of the structure and cut it down to required size.

See also

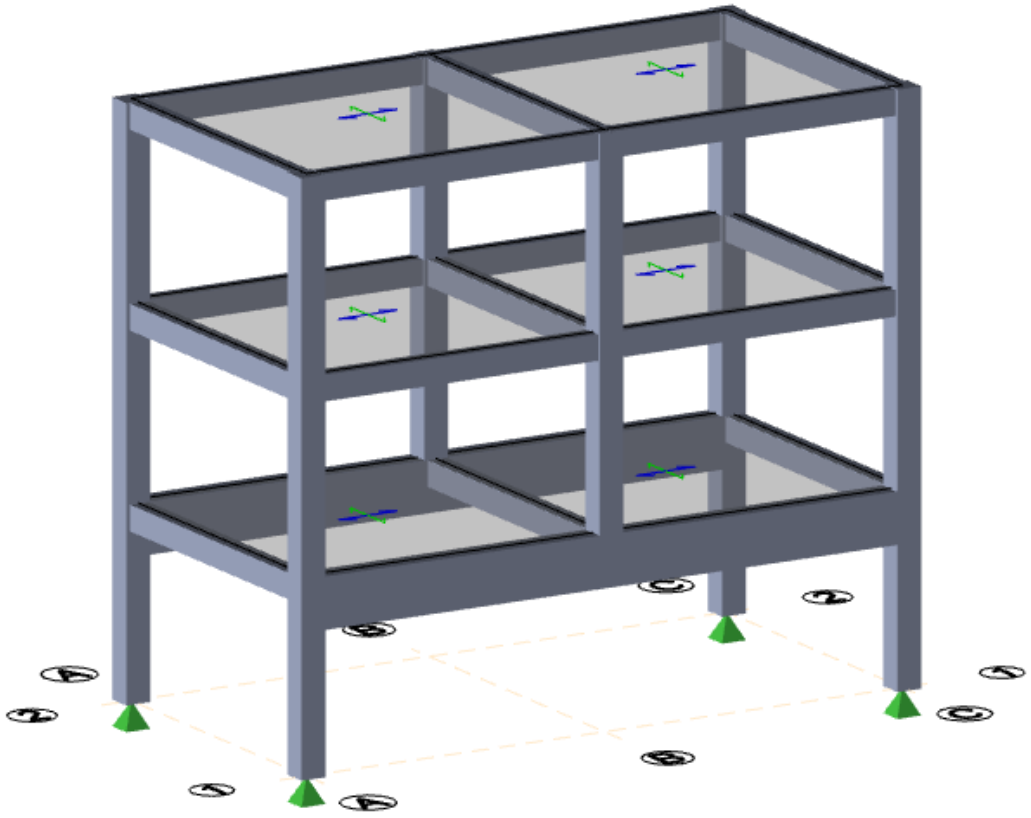
Introduction to staged construction analysis (page 454)

Overview of staged construction analysis (page 456)

7.4 Design comparison - with and without staged construction

The following transfer beam model is used to demonstrate how Tekla Structural Designer effectively considers an envelope of design cases to ensure a realistic and safe design, (irrespective of whether staged construction is considered or not).

The envelope formed when staged construction analysis has not been considered is compared with the similar envelope formed when staged construction is taken into account, to see what effect it has on the design.



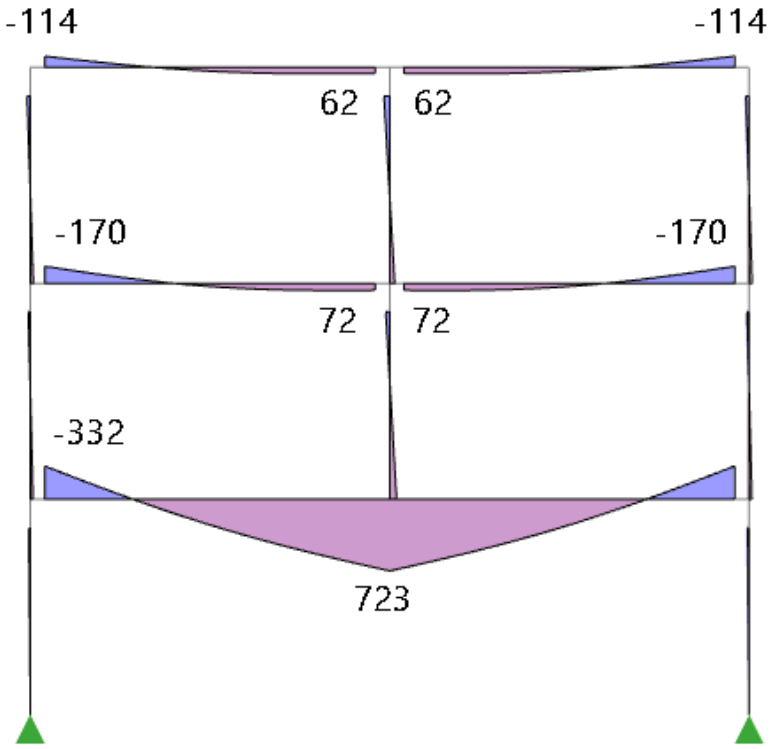
Design force envelope without staged construction

When the model is designed without taking account of staged construction, a design force envelope is established that considers the results of a 3D building analysis, and where appropriate grillage and FE chasedown analyses.

The design moments that result from the three analysis types are as follows:

3D building analysis results

First-order linear analysis moments:

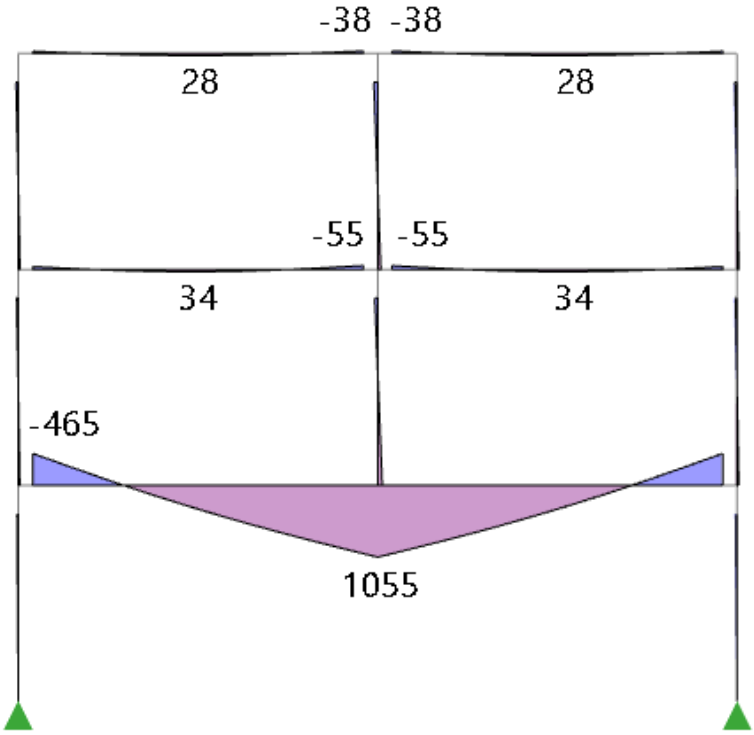


Frame effects in the building analysis model result in some of the load being shared between the transfer beam and the beams in the upper two levels.

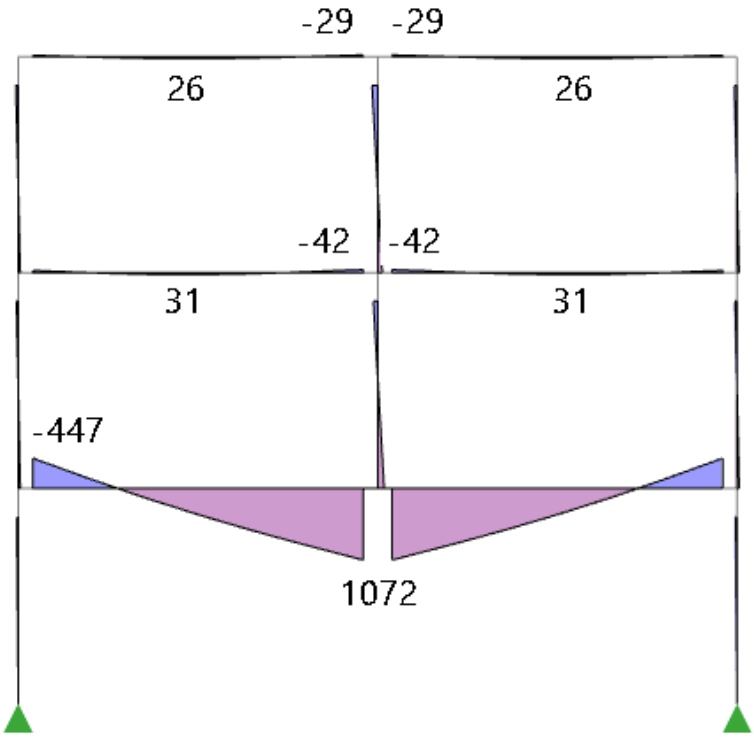
The key question for engineers to consider here is - would you be happy if the second and third floor beams do not get designed for a hogging moment at the central (supporting?) column?

Grillage chasedown and FE chasedown results

Grillage chasedown moments:



FE chasedown moments:



In both of the chasedown analysis models, (which aim to emulate how an engineer might design the structure by hand):

- more load is carried by the transfer beam,

- the second and third floor beams get designed for a hogging moment at the central column.

See the eLearning topic Background to the Analysis/Design procedures to gain a wider understanding of the benefits provided by the chasedown analysis models.

Design force envelope

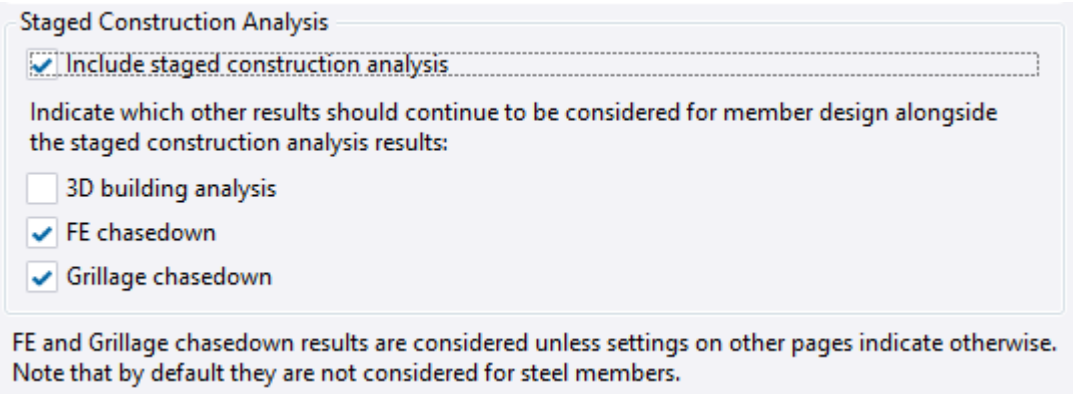
Member	Critical sagging moment (kNm)	Critical hogging moment (kNm)
3rd floor beam	62 (building analysis)	-114 (building analysis)
2nd floor beam	72 (building analysis)	-170 (building analysis)
1st floor (transfer) beam	1072 (FE chasedown)	-465 (Grillage chasedown)

Because the members are designed for an envelope of design forces from each of the analyses this is a safe approach.

The concern of some engineers is that this envelope might be over-conservative and design efficiency can be achieved by another approach.

Design force envelope with staged construction

Staged construction can be considered by selecting **Include staged construction analysis** on the Analysis page of the Design Settings dialog box.

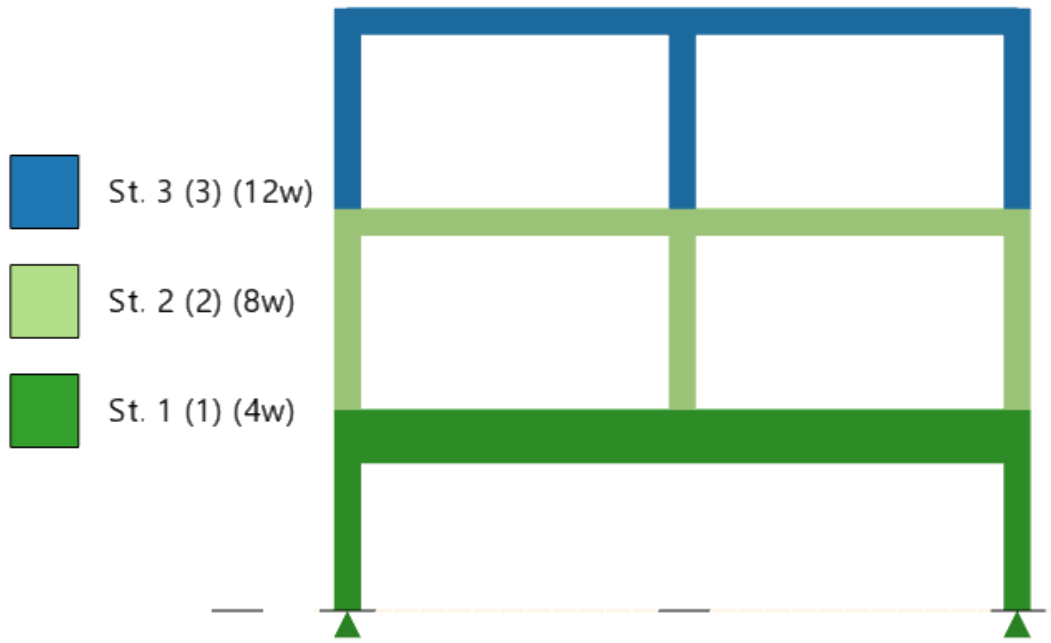


The other sets of design forces to be considered in the envelope are at the user's discretion. The default as shown above is for the 3D building analysis not to be considered, while both the FE and Grillage chasedowns are considered.

The staged construction analysis results that are obtained will be dependent on a number of factors including the pace of construction and the amount of load applied in the construction stages.

Pace of construction

In this example the duration and content of each construction stage has been specified as shown - each floor is placed in a separate stage, and each stage has a duration of four weeks.



If a faster pace of construction were to be adopted this would result in more creep occurring after the construction stages are complete. **Faster construction therefore moves the results closer to those obtained by 3D building analysis.**

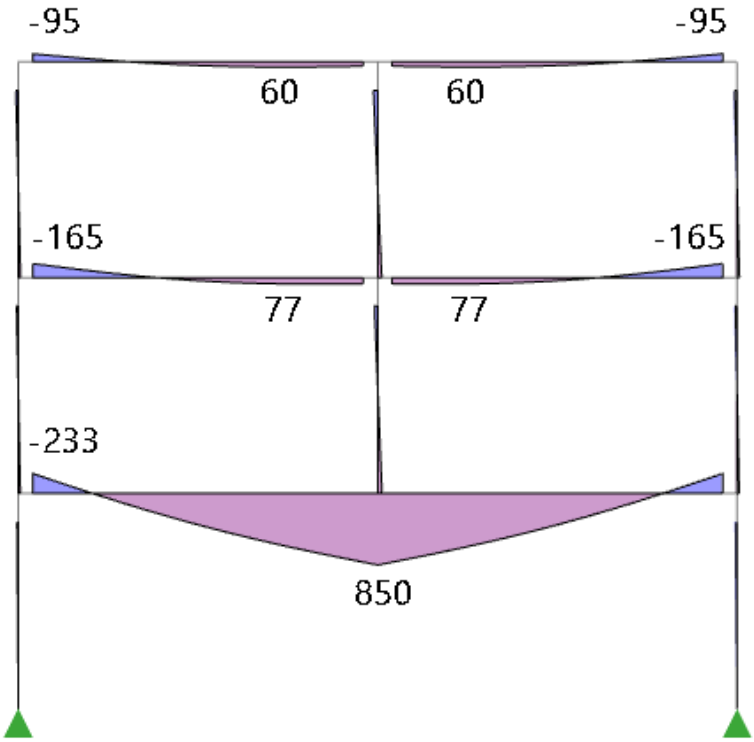
Construction stage loading

By making a realistic estimate of the construction load you will obviously obtain more accurate results.

If minimal load were to be applied during construction, the balance applied in the design stages would be greater. **Minimizing the construction stage loading moves the results closer to those obtained by 3D building analysis.**

Staged construction analysis results

Staged construction analysis moments:



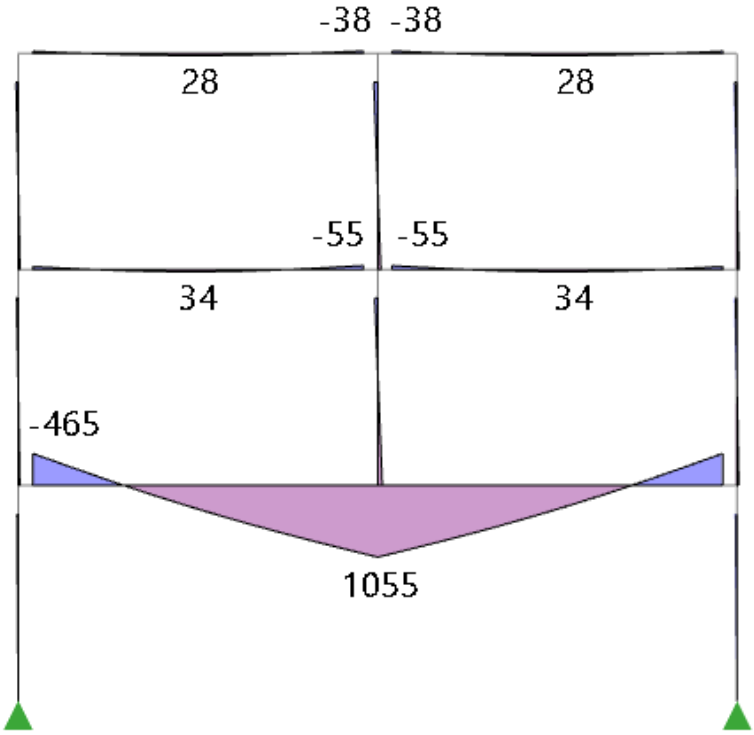
Staged construction analysis recognizes the changing state of a structure over time, (something that a simple 3D analysis completely ignores) and provided that the input parameters have been specified reasonably accurately can be considered more realistic.

Note that the second and third floor beams do not get designed for a hogging moment at the central column, chasedowns are the only way to mimic this traditional design expectation.

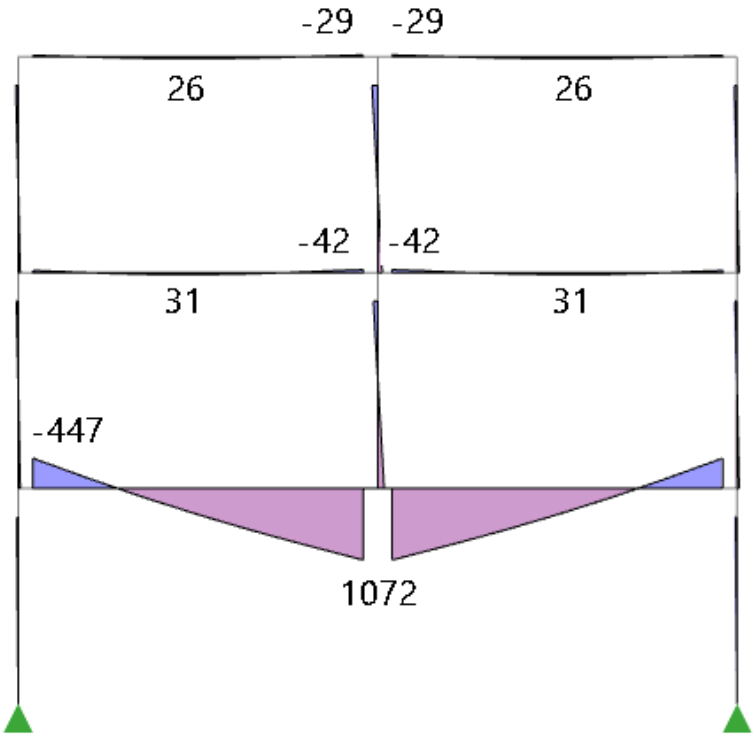
Grillage chasedown and FE chasedown results

The grillage and FE chasedown analysis results are unchanged, (as chasedown analysis models are separate from and not affected by staged construction analysis).

Grillage chasedown moments:



FE chasedown moments:



Design force envelope

The members are now designed for an envelope of design forces in which the 3D building analysis results have been replaced by the 3D staged analysis results.

Member	Critical sagging moment (kNm)	Critical hogging moment (kNm)
3rd floor beam	60 (staged construction analysis)	-95 (staged construction analysis)
2nd floor beam	77 (staged construction analysis)	-165 (staged construction analysis)
1st floor (transfer) beam	1072 (FE chasedown)	-465 (Grillage chasedown)

The chasedown results still govern when determining the maximum hogging and sagging moments in the transfer beam at the first floor, so its design is unchanged.

The 3D building analysis results previously governed the design of the second floor beam, but in the envelope formed for staged construction it is now the smaller 3D staged analysis results that govern.

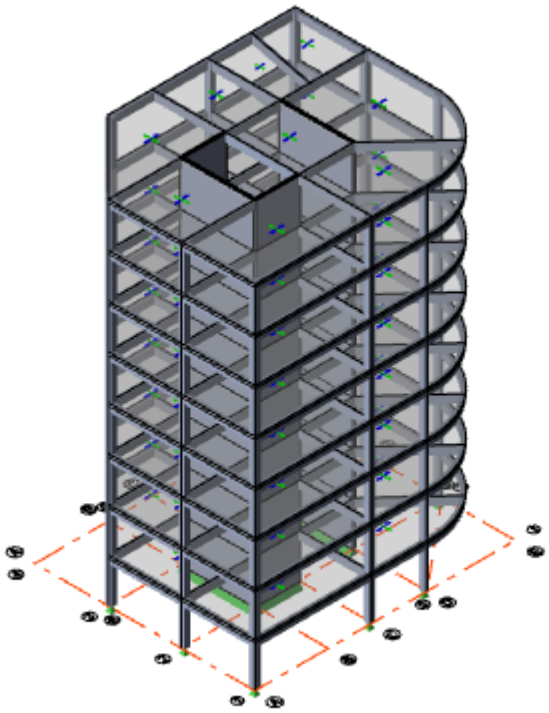
Conclusions

The chasedown results will always govern the design of the transfer beam, so provided they are not deactivated it will always be designed safely.

For other beams in this model the 3D building analysis results initially govern. By replacing them with the staged construction analysis results it is possible to tighten up on the design. By using better estimates for the staged construction input parameters more accurate results can be achieved.

7.5 Staged construction workflow example - Eurocodes

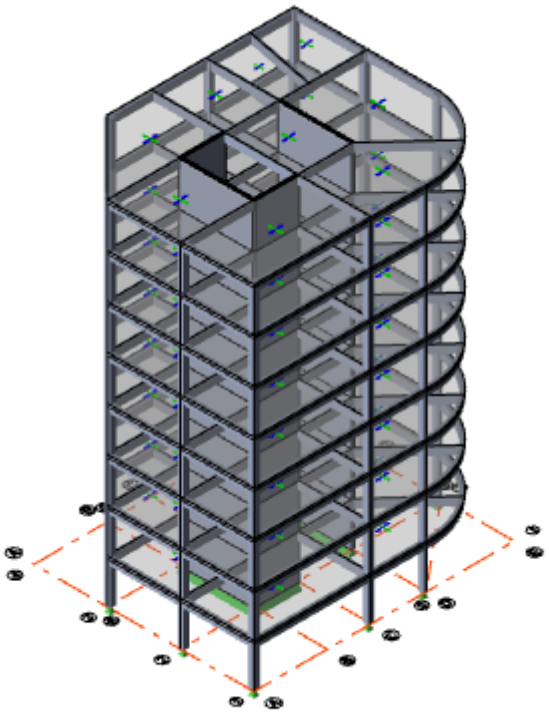
The following model will be used to illustrate the basic workflow for staged construction analysis and design in Tekla Structural Designer when working to Eurocodes.



Video to be supplied.

Download and open the tutorial model

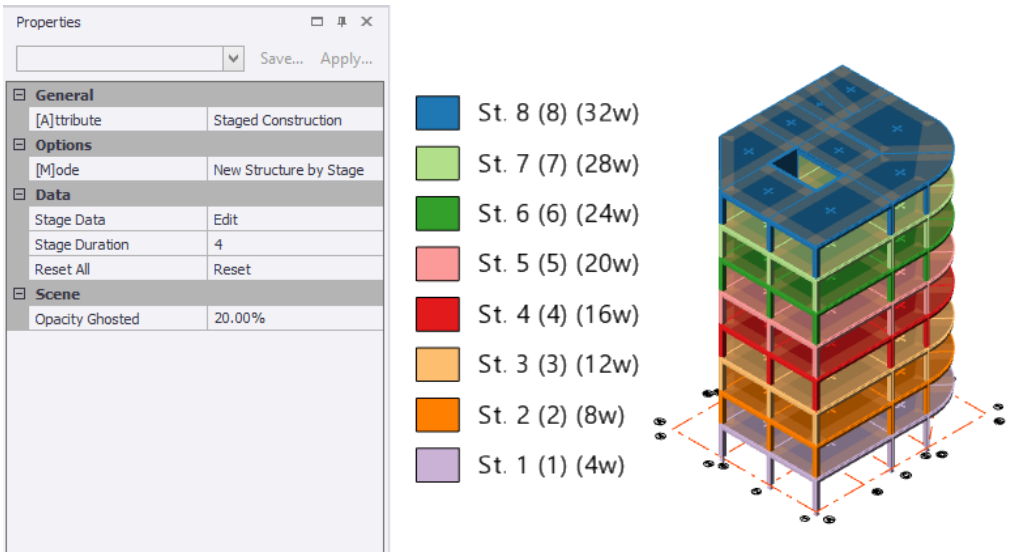
1. Download the attachment **SCA - EC UK.zip** from this page.
2. Extract the tutorial model **SCA - EC UK.tsmd** from the zip.
3. Open the model in Tekla Structural Designer.



Define construction stages and their stage durations

In this section we will examine the default construction stages (page 469) and adjust the stage durations.

1. Switch the view regime for the 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**.

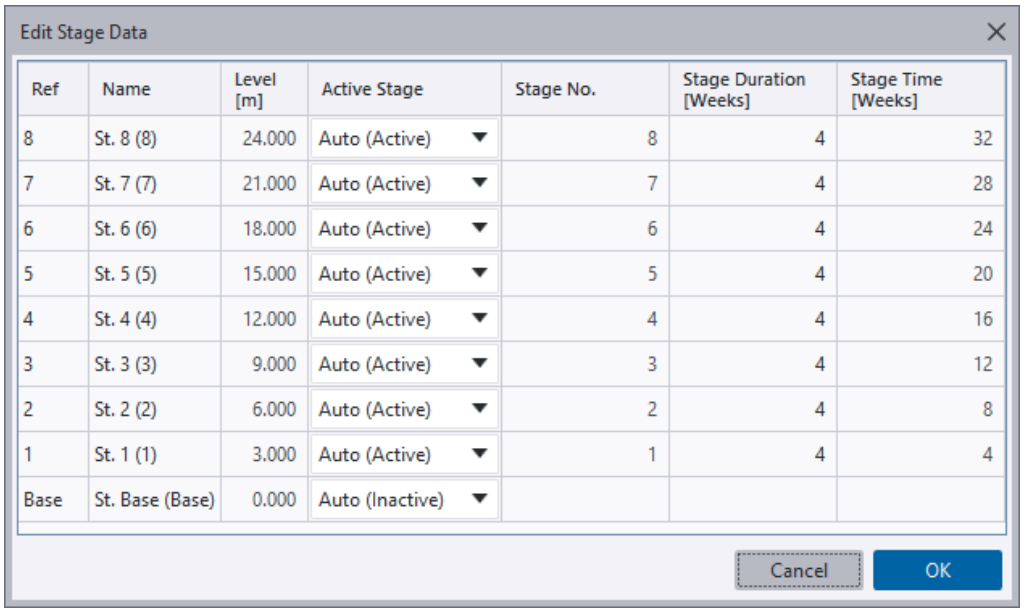


Construction stages are activated by default at every floor level in which slabs have been defined. Because this happens automatically there may be little, or no further editing required.

5. In the **Properties** window, click **Stage Data**, then click the **Edit** button

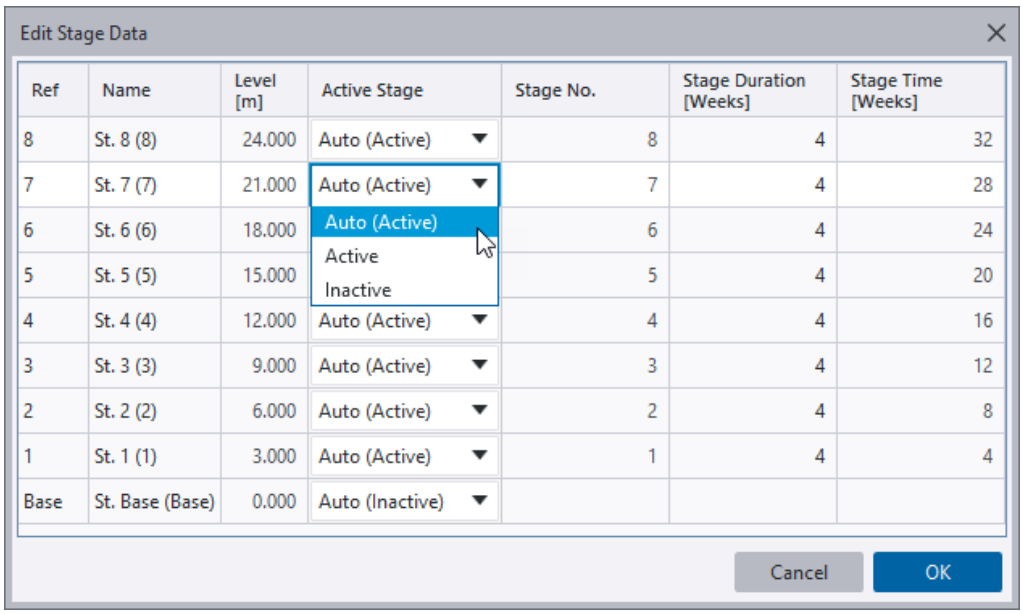


6. This displays the **Edit Stage Data** dialog box.



Each row in the above table corresponds to a level that has been marked as a floor in the **Construction Levels** dialog box.

If you want to reduce the number of stages, you can do so by setting **Active Stage** at a particular level to **Inactive**. The members at that level are then moved to the next active stage.



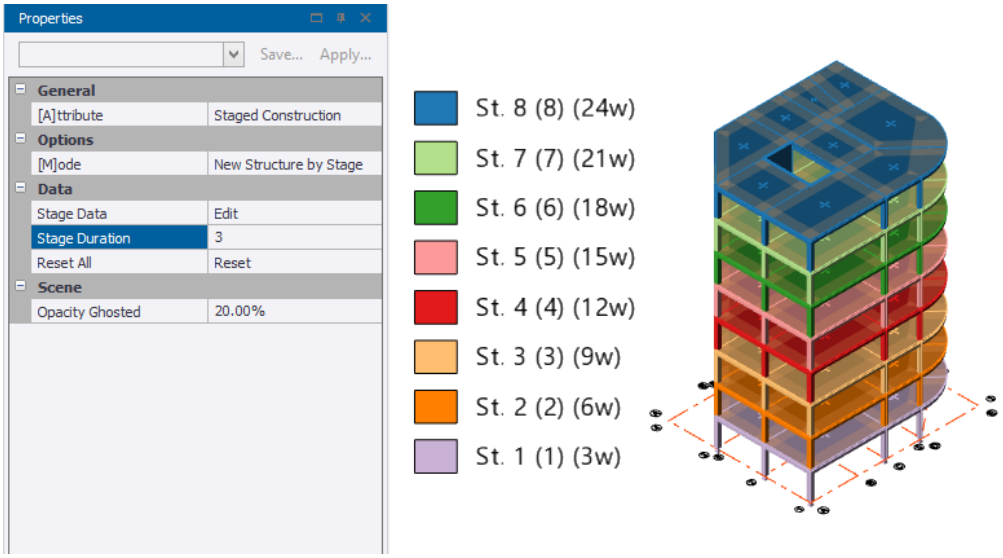
7. The easiest way to adjust individual stage durations is via the above table, however for this example we will use another method, so click **Cancel** to close the dialog.

   The stage durations will now be adjusted directly in the **Properties** window.

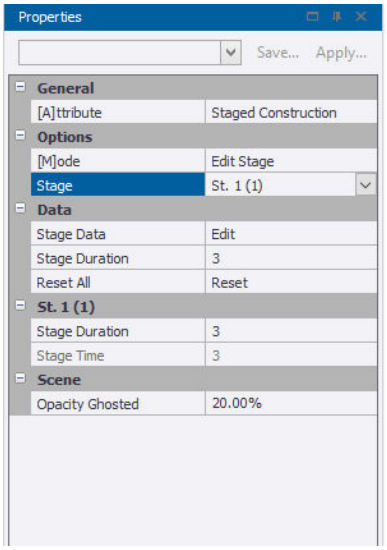
8. In the **Properties** window, first ensure the **Mode** is set to **New Structure by Stage**.

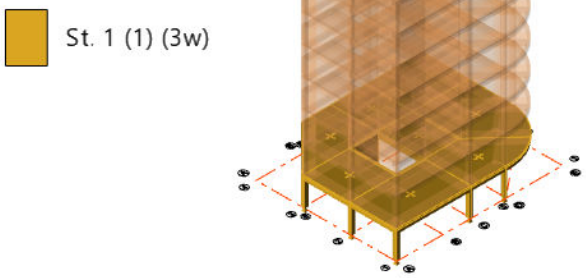
9. In the **Data** section, change the **Stage Duration** to **3**.

   The new duration is applied to all stages.

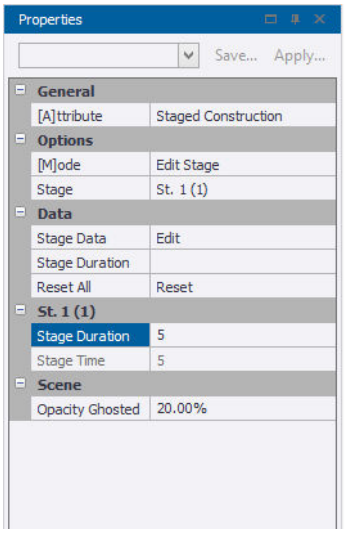


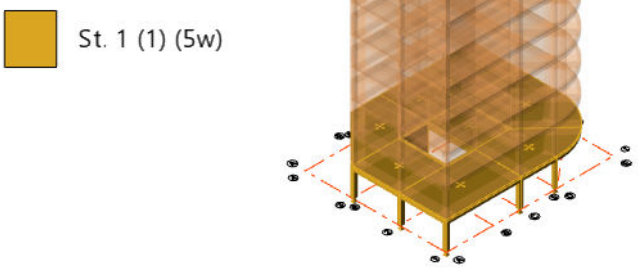
10. In the **Properties** window, change the **Mode** to **Edit Stage**, then in **Options** use the **Stage** list to select stage **St. 1 (1)**.



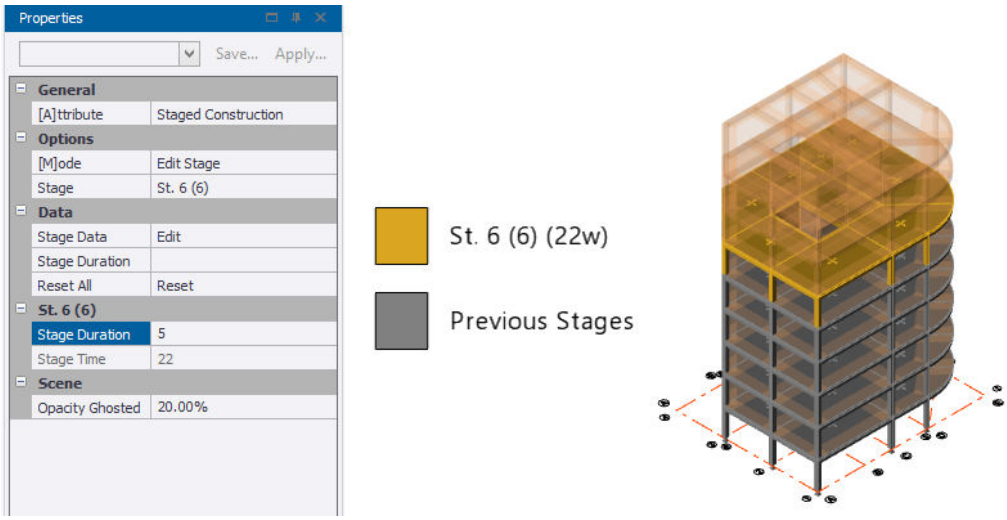


11. In the **Properties** window, in the **St.1 (1)** group, change the **Stage Duration** value to **5** weeks. (Be careful not to change it in the **Data** group as this would change the duration of *all* stages).

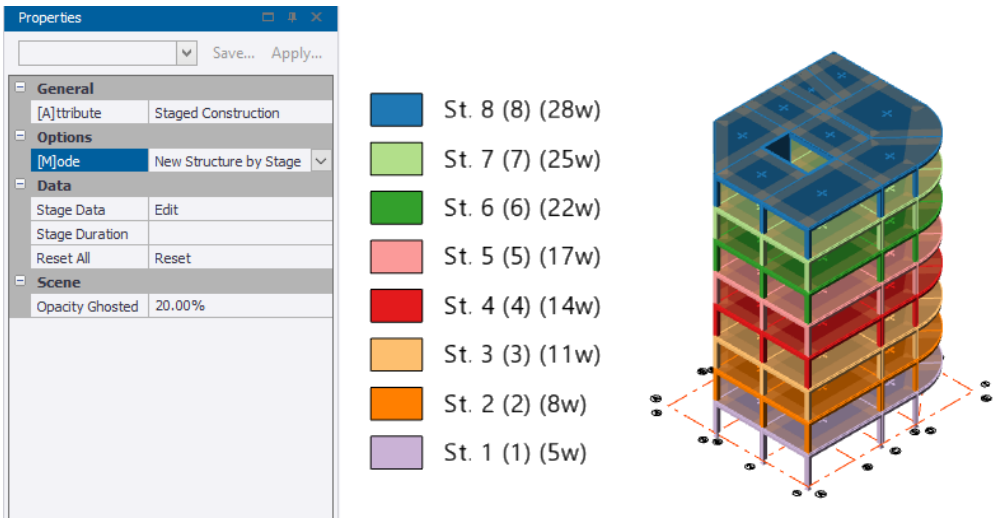




12. In a similar manner, set the **Stage Duration** to **5** weeks for stage **St. 6 (6)**.



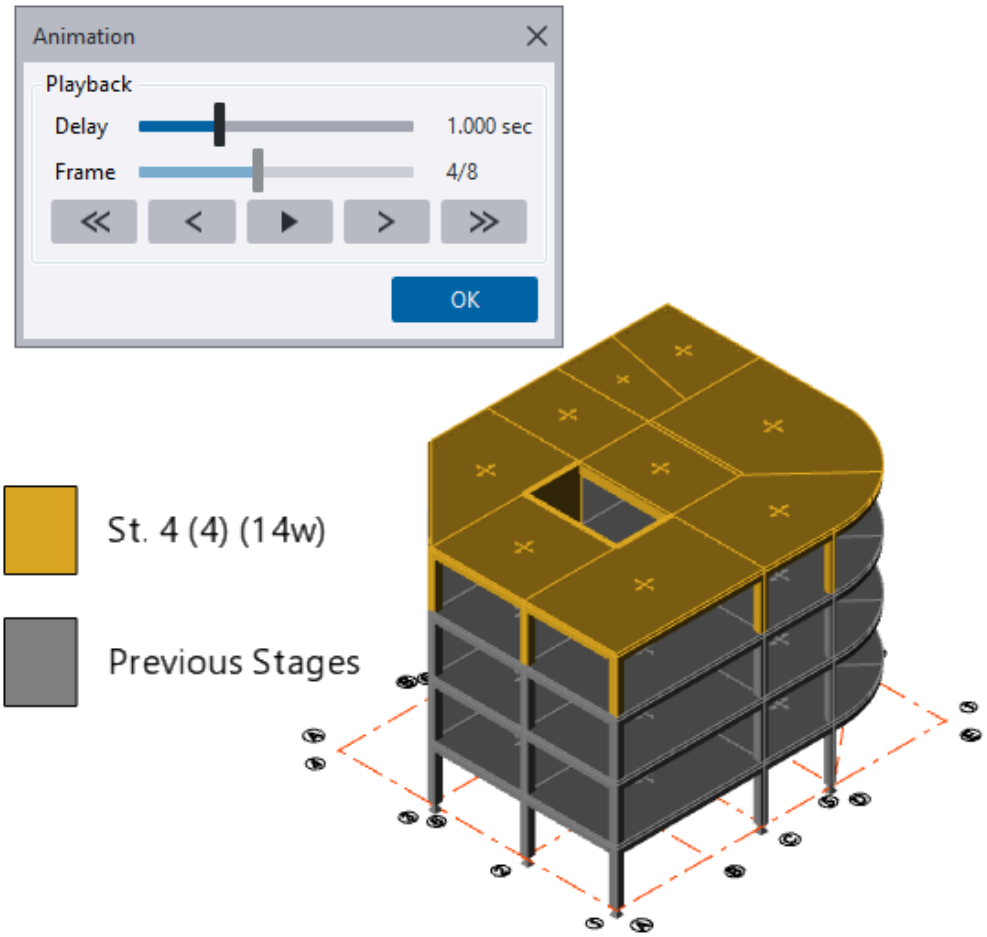
13. To review all the stage durations, change the **Mode** back to **New Structure by Stage**. The stage times shown in the legend should now be as shown below.



14. Right click and select **Animate** from the context menu.

    The **Animate** feature provides a practical way to review stages in more detail.

15. Familiarize yourself with the animation buttons.

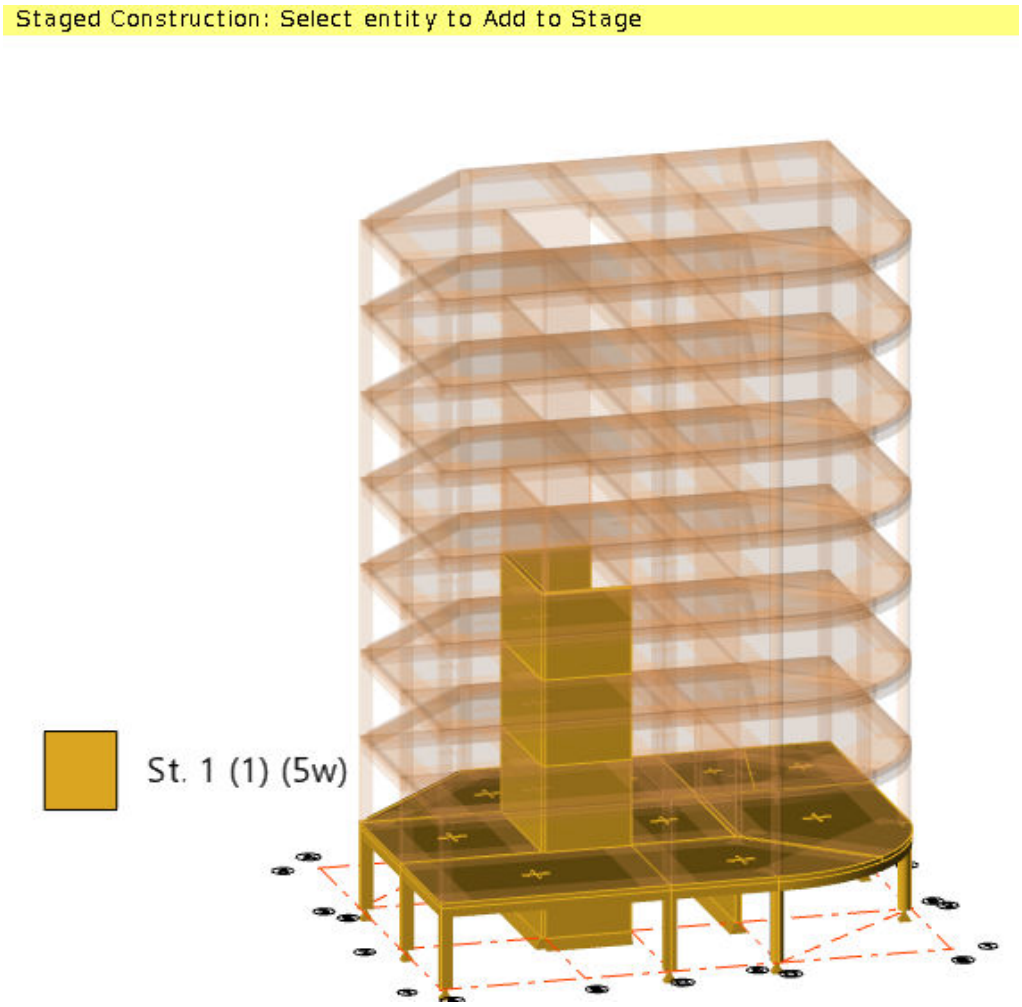


- Press the play button to move through the stages in sequence.
- Press the pause button to stop at a particular stage
- Press the back and forward buttons to display the previous/next stage.

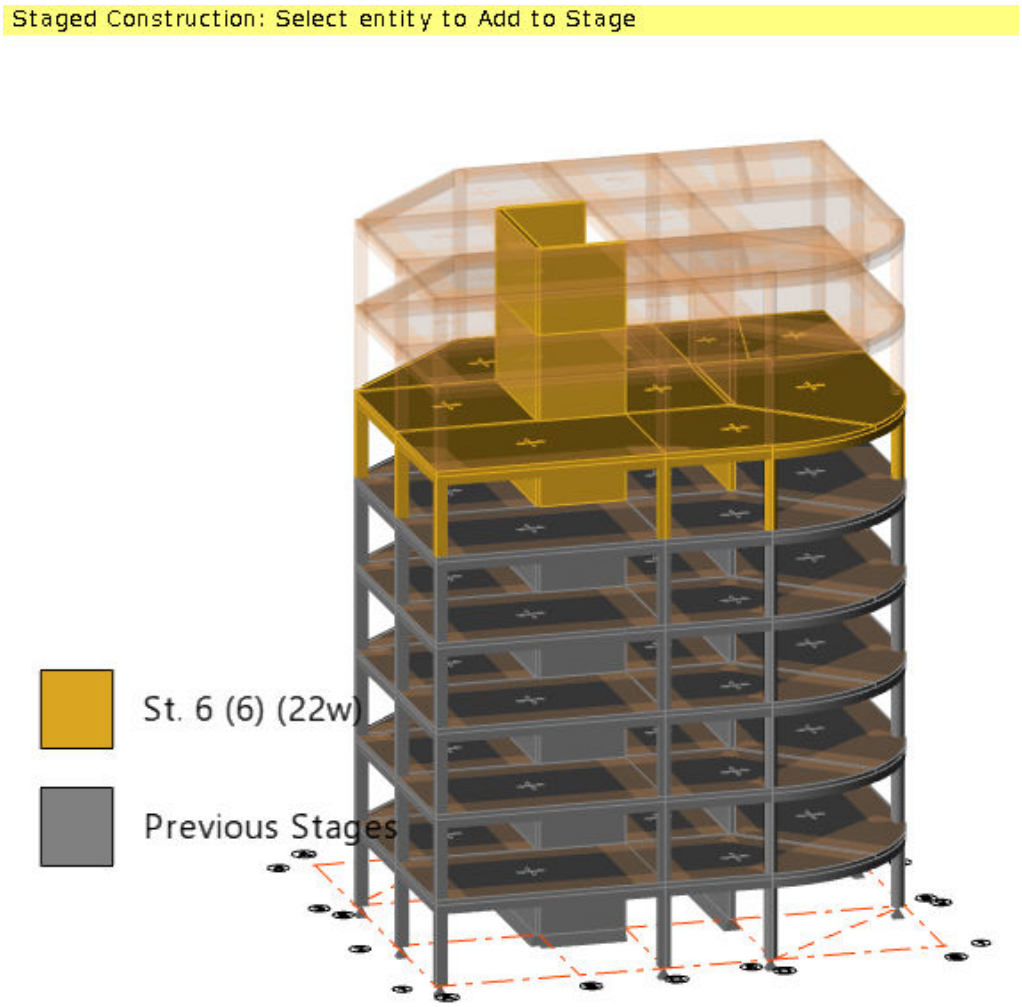
Assign members to construction stages

In this section we will reassign some of the wall panels to a different constructions stage.

1. In the **Properties** window, change the **Mode** to **Add to Stage** and ensure stage **St. 1 (1)** is selected.
2. Select the wall panels in the core up to level 4 to add them to this stage.



3. In the **Properties** window select stage **St. 6 (6)**.
4. Select the wall panels in the core up to the roof to add them to this stage.

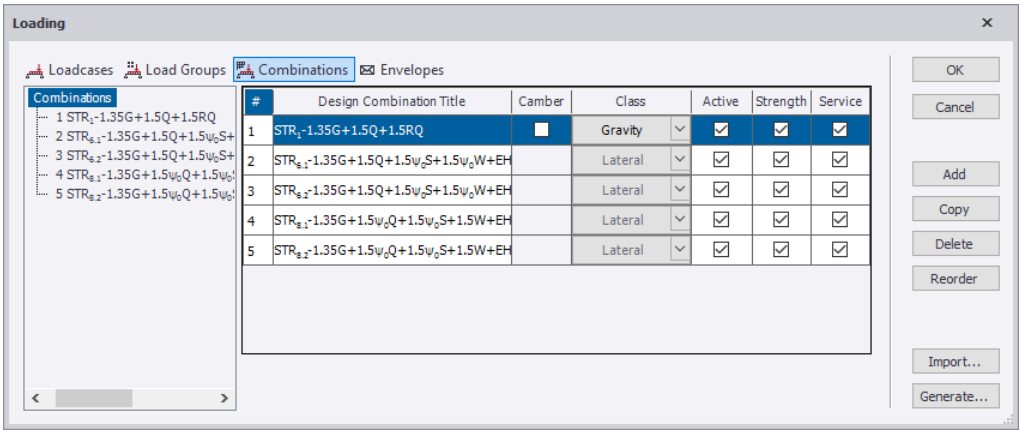


5. When stage **St. 6 (6)** is as shown above, press **Esc** to end the command.

Create a Staged Construction load combination

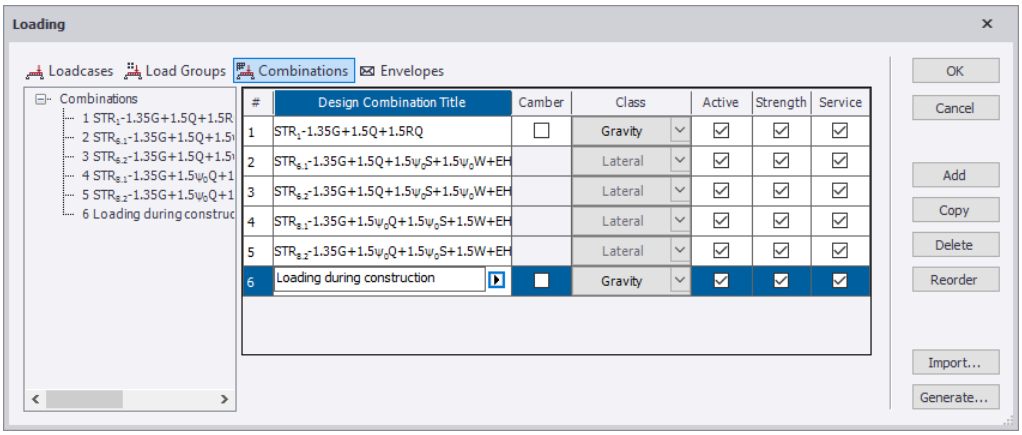
In this section we will create a staged construction load combination (page 462) and then define the loading within it.

1. On the **Load** ribbon, click **Combination**.

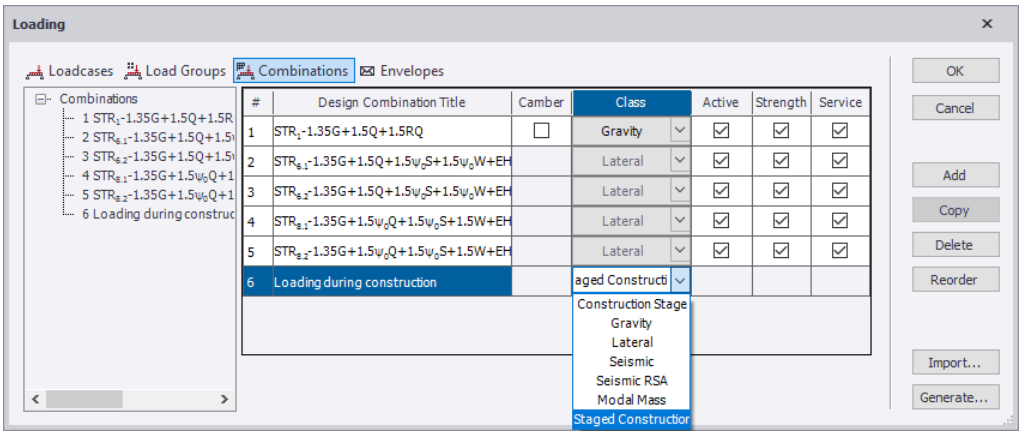


Some standard design combinations have already been created.

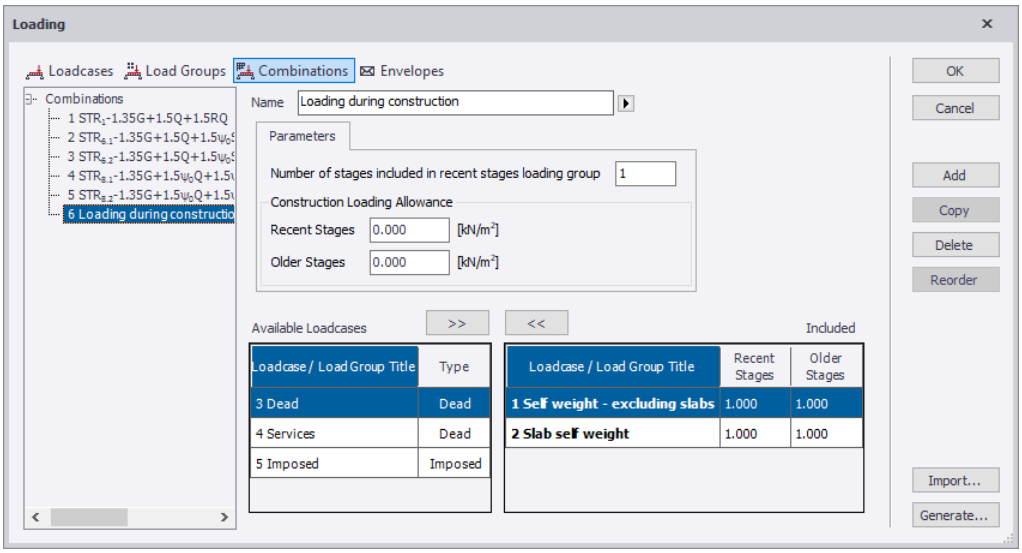
2. Click **Add**
3. Enter the new combination name as **Loading during construction**



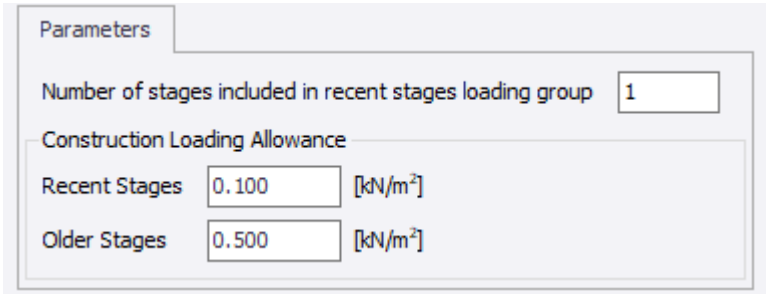
4. Change the combination **Class** to **Staged Construction**.



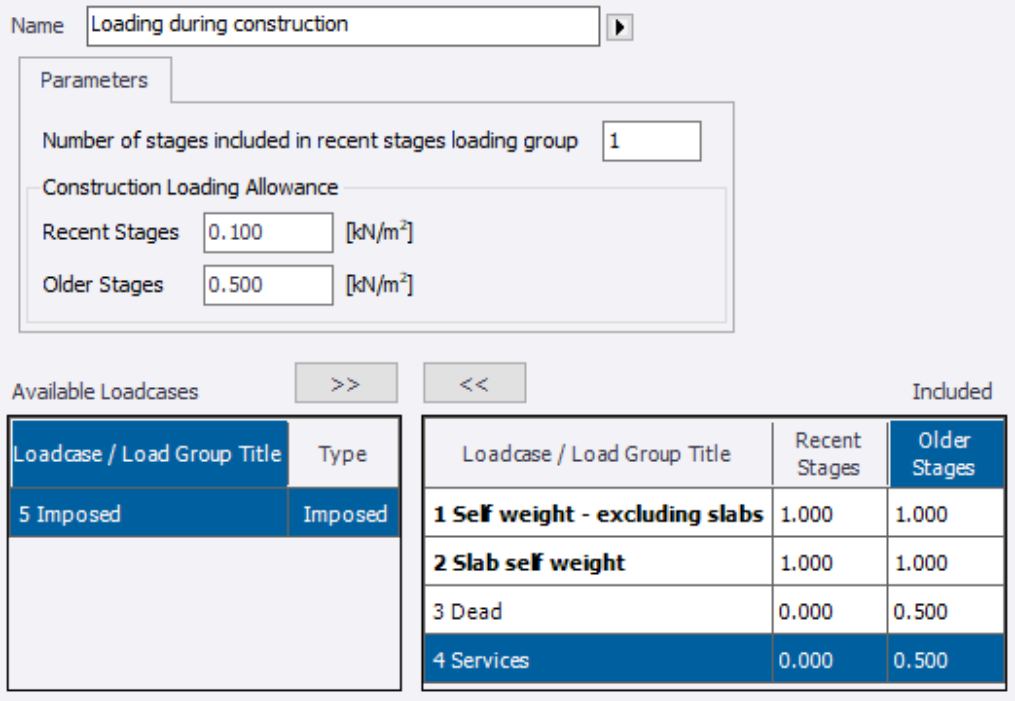
5. Select the new combination in the left hand pane of the dialog.



6. Specify the construction loading allowance parameters as shown below.



7. Include the Dead and Services loadcases in the combination and then set the load factors for recent and older stages as shown below.



In real models engineering judgment will be required when defining the staged construction combination.

8. Click **OK** to save the changes.

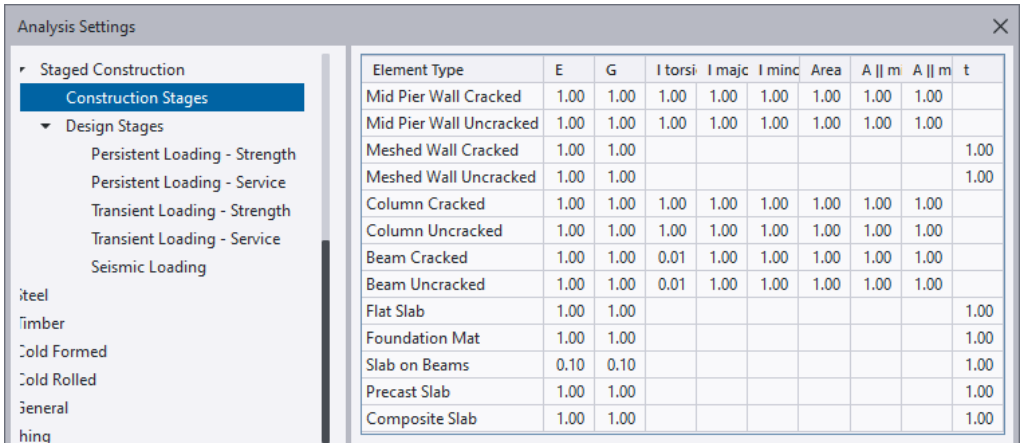
Review the modification factors

The staged construction modification factors (page 470) applied to elements in the staged construction solver model are different to those applied to other solver model types.

To review these factors:

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

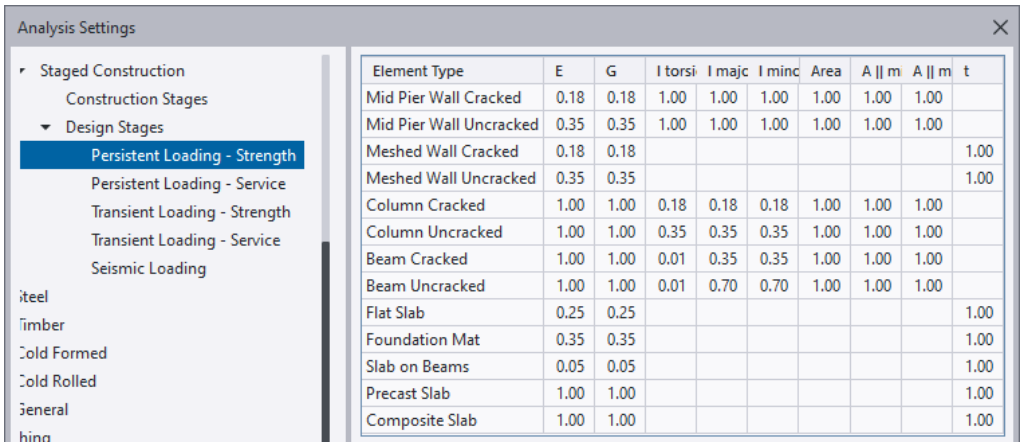
2. On the **Modification Factors** page, select the **Concrete > Staged Construction > Construction Stages** sub-page.



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	1.00	1.00							1.00
Meshed Wall Uncracked	1.00	1.00							1.00
Column Cracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Flat Slab	1.00	1.00							1.00
Foundation Mat	1.00	1.00							1.00
Slab on Beams	0.10	0.10							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

The above modification factors are applied to concrete elements during the construction stages.

3. Select the **Concrete > Staged Construction > Design Stages > Persistent Loading - Strength** sub-page.



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.18	0.18	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	0.35	0.35	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.18	0.18							1.00
Meshed Wall Uncracked	0.35	0.35							1.00
Column Cracked	1.00	1.00	0.18	0.18	0.18	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	0.35	0.35	0.35	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.35	0.35	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	0.70	0.70	1.00	1.00	1.00	
Flat Slab	0.25	0.25							1.00
Foundation Mat	0.35	0.35							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

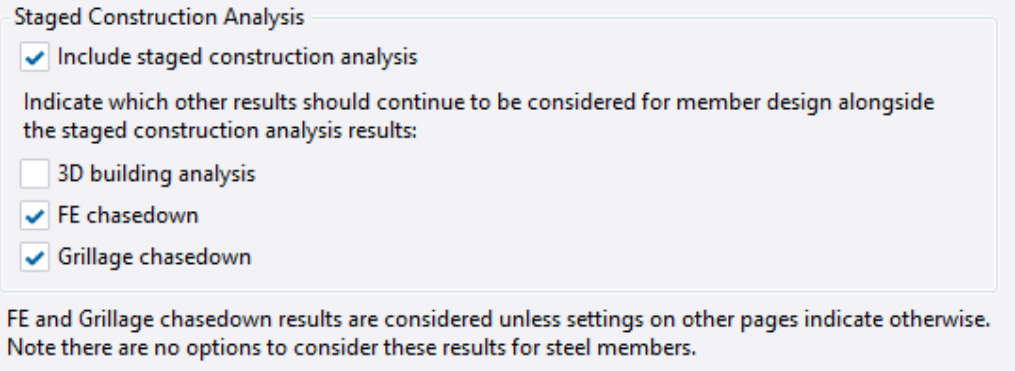
This sub-page and the four pages below it are used to define the modification factors applied to concrete elements for each of the design stages.

4. No changes are required, so click **Cancel**.

Configure the analyses

1. On the **Design** ribbon, click **Settings**.
2. On the page, select whether the analysis should be first-order or second-order. For this particular example we shall accept **First-order analysis**.
3. Next, select **Include staged construction analysis**.

4. We will accept the default for the other results, by leaving 3D building analysis unselected, and the two chasedown analyses selected.



   In effect this means that the staged construction analysis results are replacing the 3D building analysis results at one end of the design envelope. Refer to design comparison - with and without staged construction (page 481) to find out more about the implications of this.

5. Click **OK** to save the changes.

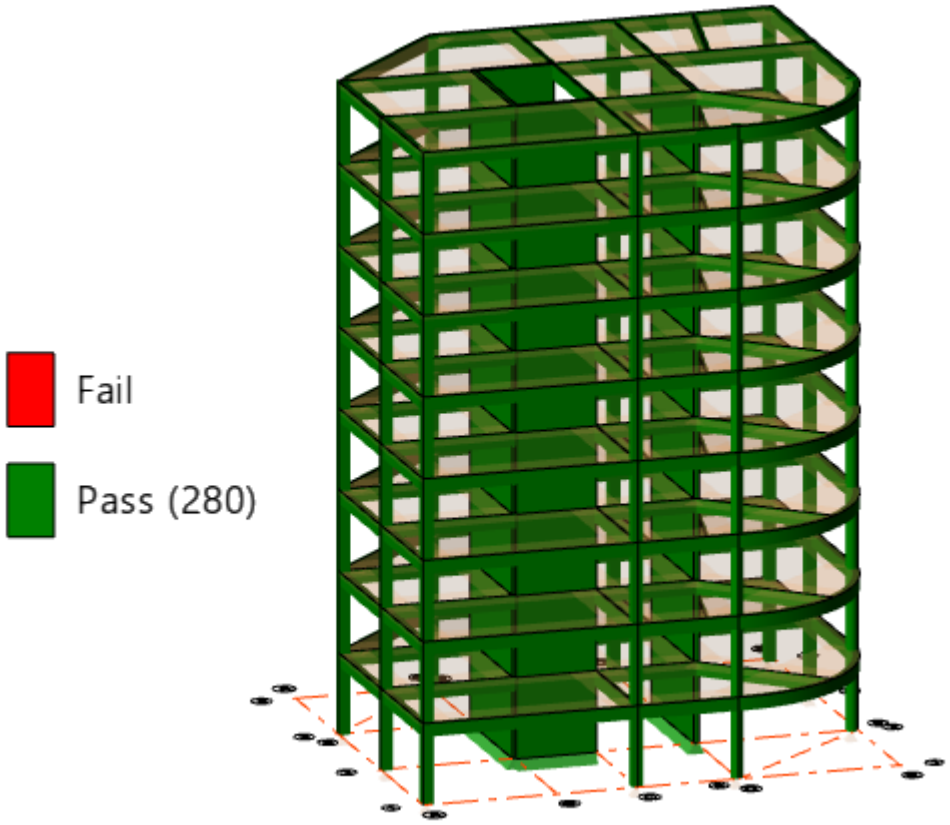
Run the design and review the design results

Now that the analysis type(s) have been set as required, you can proceed to run the combined analysis and design. This is initiated by selecting the required design option from the Design ribbon.

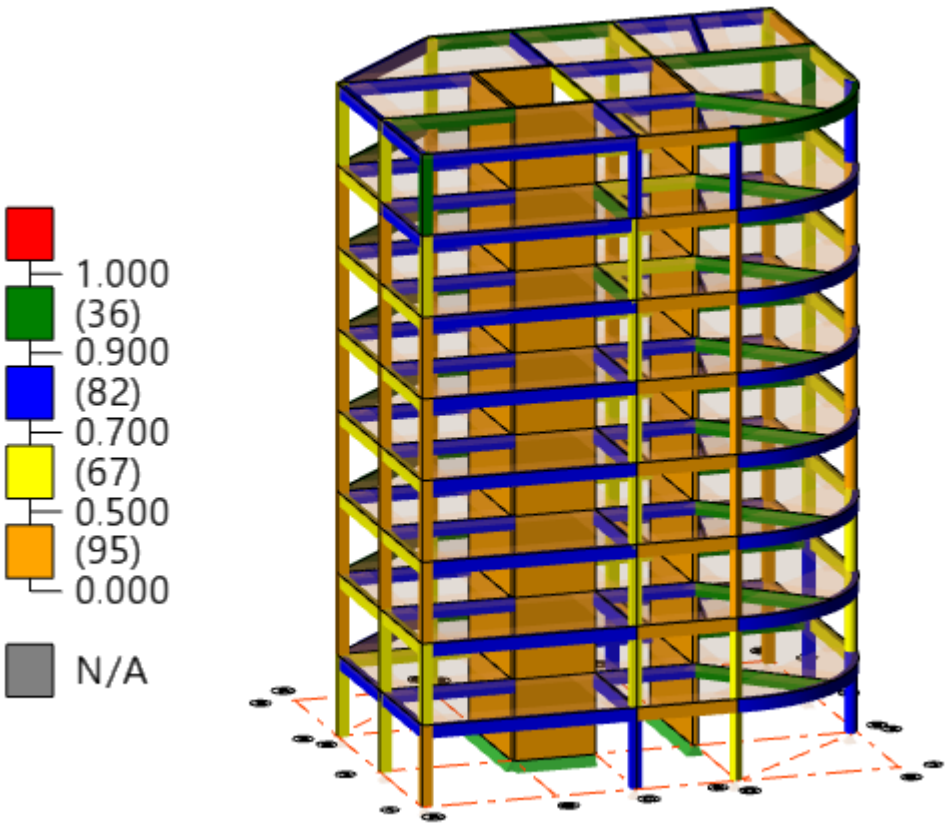
1. On the **Design** ribbon, click **Design All (Static)**.

   As this model doesn't contain any non-linear elements, a *linear* staged construction analysis is performed as part of the analysis and design process.

   Once the analysis and design process is complete the active view switches to the Review View regime and shows the design status.



2. To investigate the efficiency of the design, on the **Review** ribbon, click **Ratio**



3. To view tabular results, on the **Review** ribbon, click **Tabular Data**

4. Ensure the **View Type** is set to **Design Summary**

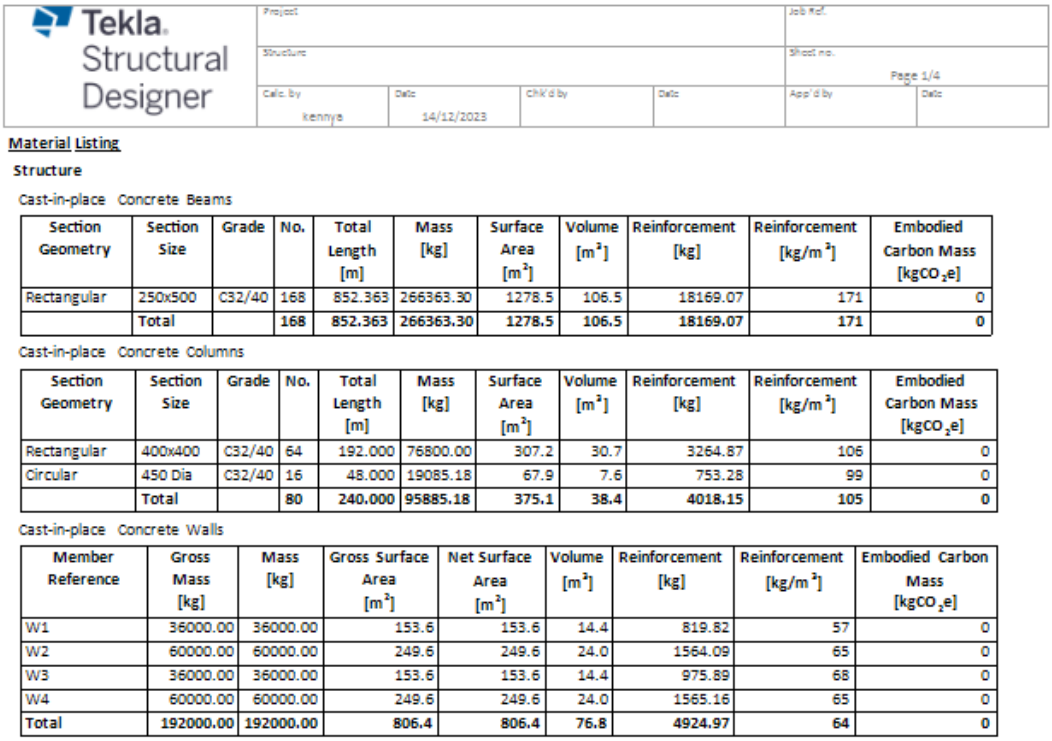
Design Summary

Member Reference	Group Ref.	Stack Ref.	Section	Grade	Length [m]	Utilization	Status	Results
CC A/1	CRC5-D12	1	400x400	C32/40	3.000	0.496	Pass	Results...
CC A/1	CRC5-D12	2	400x400	C32/40	3.000	0.542	Pass	Results...
CC A/1	CRC5-D12	3	400x400	C32/40	3.000	0.455	Pass	Results...
CC A/1	CRC5-D12	4	400x400	C32/40	3.000	0.452	Pass	Results...
CC A/1	CRC5-D12	5	400x400	C32/40	3.000	0.462	Pass	Results...
CC A/1	CRC5-D12	6	400x400	C32/40	3.000	0.486	Pass	Results...
CC A/1	CRC5-D12	7	400x400	C32/40	3.000	0.536	Pass	Results...
CC A/1	CRC5-D12	8	400x400	C32/40	3.000	0.992	Pass	Results...
CC A/2	CRC5-D8	1	400x400	C32/40	3.000	0.611	Pass	Results...
CC A/2	CRC5-D8	2	400x400	C32/40	3.000	0.534	Pass	Results...
CC A/2	CRC5-D8	3	400x400	C32/40	3.000	0.458	Pass	Results...
CC A/2	CRC5-D8	4	400x400	C32/40	3.000	0.381	Pass	Results...
CC A/2	CRC5-D8	5	400x400	C32/40	3.000	0.326	Pass	Results...
CC A/2	CRC5-D8	6	400x400	C32/40	3.000	0.321	Pass	Results...
CC A/2	CRC5-D8	7	400x400	C32/40	3.000	0.338	Pass	Results...

Any failing members should be investigated in the normal way.

Output the calculations

Once the design is satisfactory, reports and drawings can be output in the normal way.



Tekla Structural Designer

Project | Job Ref.
Structure | Sheet no. Page 1/4
Calc. by: kennya | Date: 14/12/2023 | Chk'd by | Date | App'd by | Date

Material Listing

Structure

Cast-in-place Concrete Beams

Section Geometry	Section Size	Grade	No.	Total Length [m]	Mass [kg]	Surface Area [m^2]	Volume [m^3]	Reinforcement [kg]	Reinforcement [kg/m^3]	Embodied Carbon Mass [$kgCO_2e$]
Rectangular	250x500	C32/40	168	852.363	266363.30	1278.5	106.5	18169.07	171	0
	Total		**168**	**852.363**	**266363.30**	**1278.5**	**106.5**	**18169.07**	**171**	**0**

Cast-in-place Concrete Columns

Section Geometry	Section Size	Grade	No.	Total Length [m]	Mass [kg]	Surface Area [m^2]	Volume [m^3]	Reinforcement [kg]	Reinforcement [kg/m^3]	Embodied Carbon Mass [$kgCO_2e$]
Rectangular	400x400	C32/40	64	192.000	76800.00	307.2	30.7	3264.87	106	0
Circular	450 Dia	C32/40	16	48.000	19085.18	67.9	7.6	753.28	99	0
	Total		**80**	**240.000**	**95885.18**	**375.1**	**38.4**	**4018.15**	**105**	**0**

Cast-in-place Concrete Walls

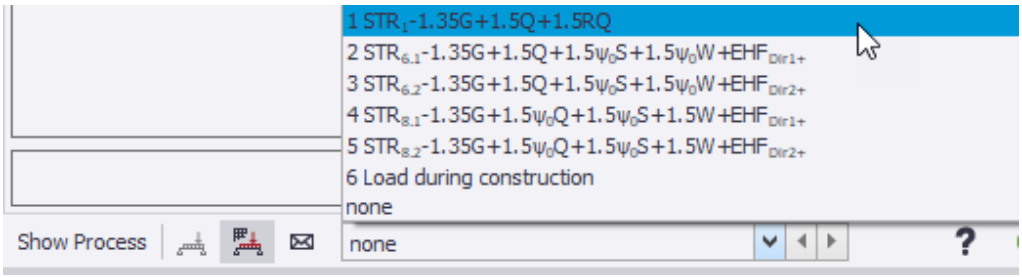
Member Reference	Gross Mass [kg]	Mass [kg]	Gross Surface Area [m^2]	Net Surface Area [m^2]	Volume [m^3]	Reinforcement [kg]	Reinforcement [kg/m^3]	Embodied Carbon Mass [$kgCO_2e$]
W1	36000.00	36000.00	153.6	153.6	14.4	819.82	57	0
W2	60000.00	60000.00	249.6	249.6	24.0	1564.09	65	0
W3	36000.00	36000.00	153.6	153.6	14.4	975.89	68	0
W4	60000.00	60000.00	249.6	249.6	24.0	1565.16	65	0
Total	**192000.00**	**192000.00**	**806.4**	**806.4**	**76.8**	**4924.97**	**64**	**0**

Display staged construction analysis results

You can graphically display staged construction analysis results from a 3D Results View. The steps below demonstrate how to display the results, firstly for a design stage combination, and then for each of the construction stages.

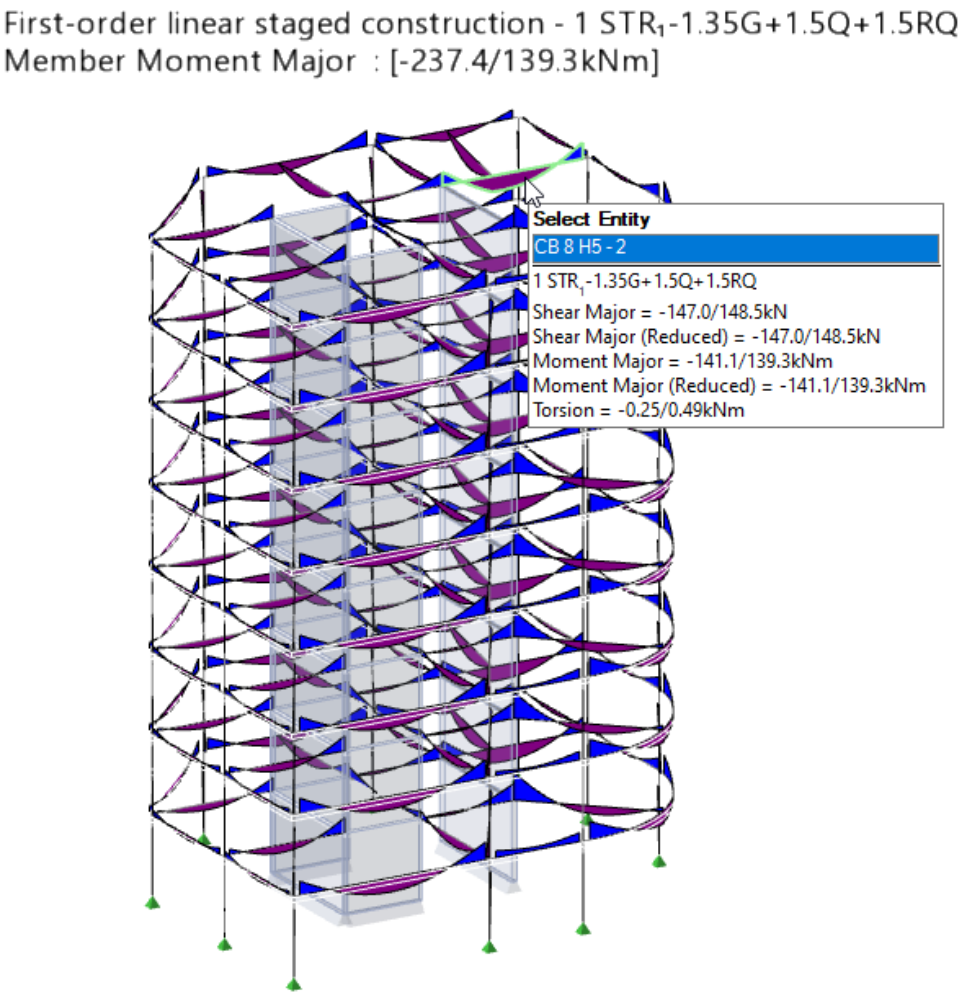
1. Switch the view regime for the Structure 3D view to  **Results View**.
2. On the **Results** ribbon, in the **Method** list, select **First-order linear staged construction**.

   The design stage combinations and the staged construction loading combination are available in the **Loading list**.



3. Select the **STR_{1}-1.35G+1.5Q+1.5RQ** combination.
4. From the **1D Results** group on the **Results** ribbon, select **Moment Major**.

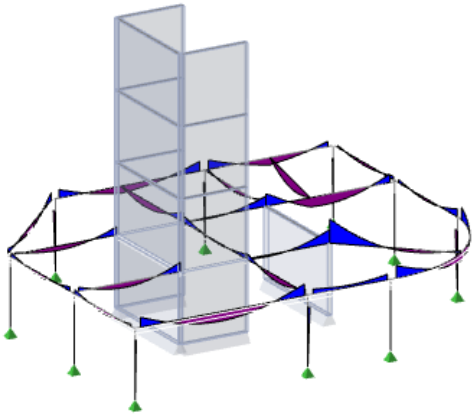
   The results are displayed for the selected combination; when the cursor is positioned over a member the tooltip displays more detailed information.



5. To view results for each of the construction stages, select the **Load during construction** combination from the **Loading list**.

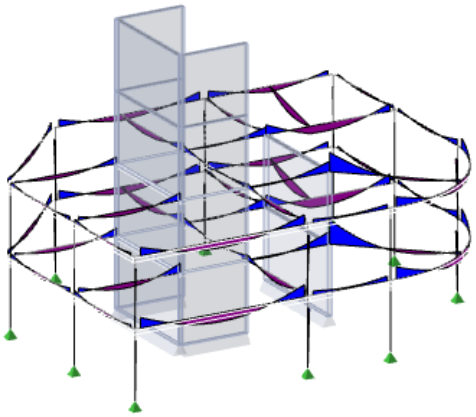
   Initially the first construction stage is displayed as shown below.

First-order linear staged construction - 6 Load during construction - St. 1 (1)
Member Moment Major : [-109.5/44.2kNm]



6. On the **Results** ribbon, in the **Stage** list, you can select each of the stages in turn to display results for each stage.

First-order linear staged construction - 6 Load during construction - St. 2 (2)
Member Moment Major : [-136.0/55.2kNm]

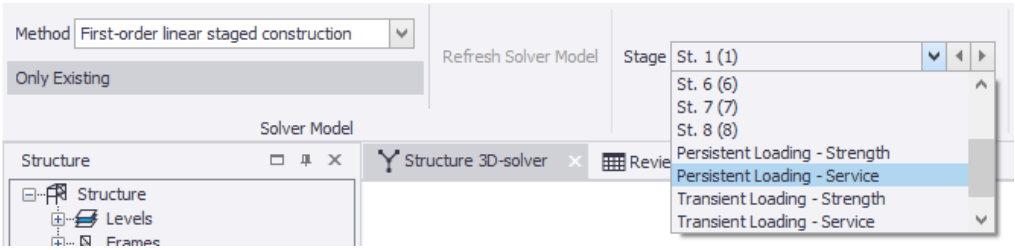


Display staged construction solver models

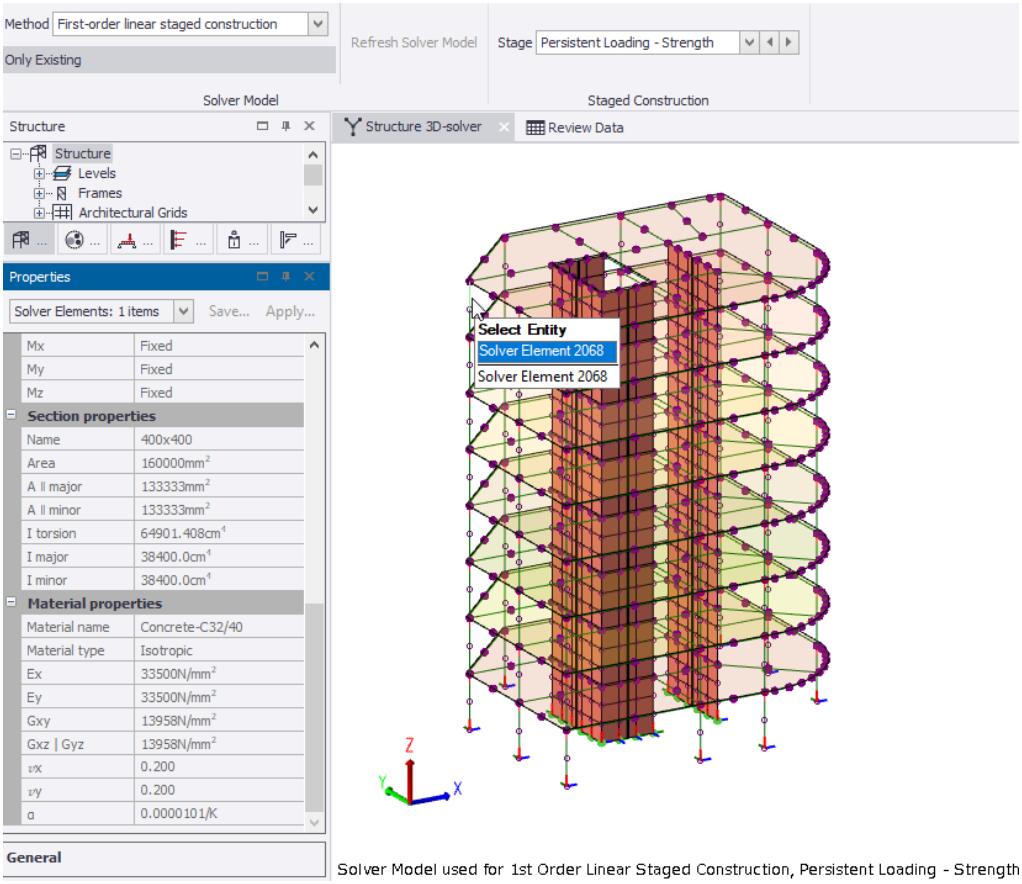
You can display the individual solver models for the different design stages and each construction stage from a Solver View.

1. Switch the view regime to  **Solver View**.
2. On the **Solver** ribbon, in the **Method** list, select **First-order linear staged construction**.

   A **Stage** list is added to the ribbon, to enable each construction stage and the different design stages to be selected.



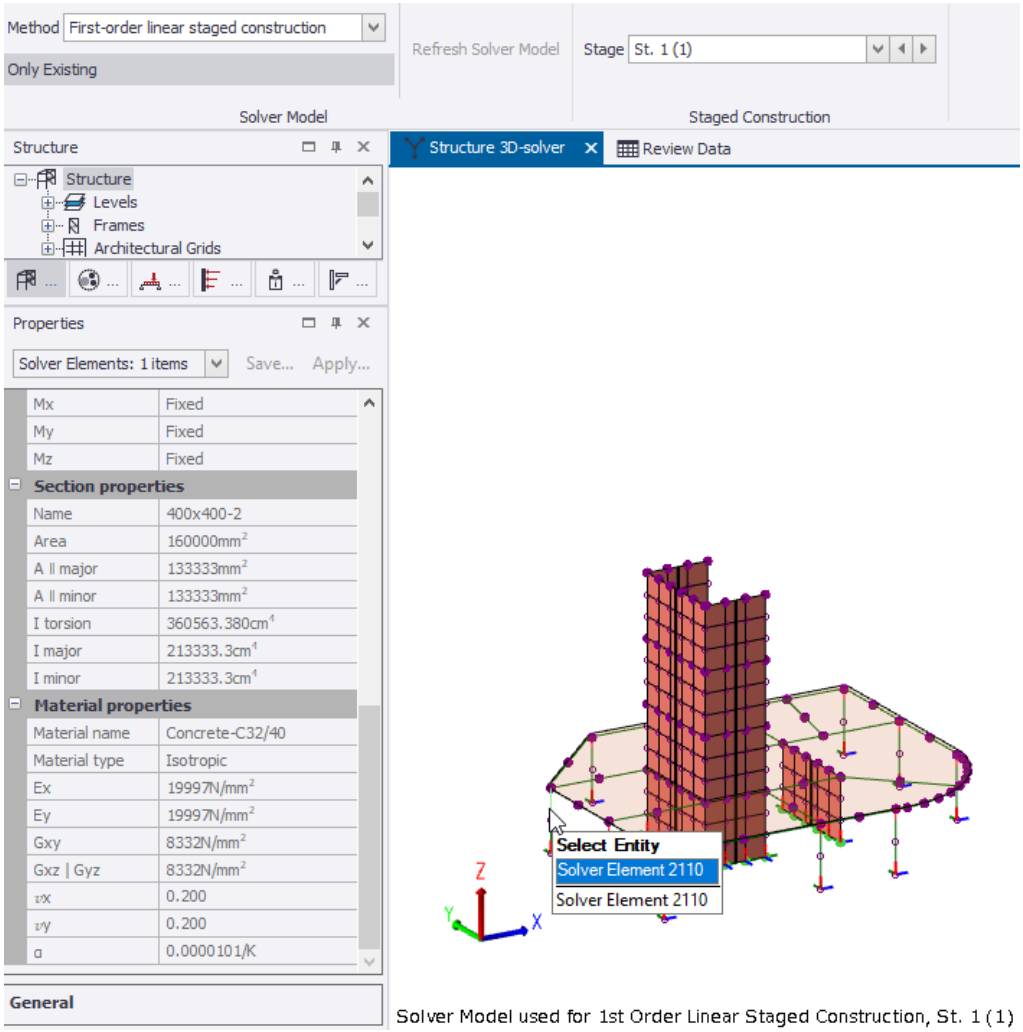
3. In the **Stage** list, select **Persistent Loading - Strength** (one of the design stages) .



Solver Model used for 1st Order Linear Staged Construction, Persistent Loading - Strength

The solver model is displayed accordingly. As shown above, when an element is selected its properties are displayed in the **Properties** window. The properties displayed reflect the modification factors that are applicable for the selected stage.

4. In the **Stage** list, select **St. 1 (1)** (one of the construction stages) .



Solver Model used for 1st Order Linear Staged Construction, St. 1 (1)

The solver model for the first construction stage is displayed. As shown above, when an element is selected its properties are displayed in the Properties window. The properties displayed reflect the construction stage modification factors and also any adjustments made for concrete creep allowance (page 476).

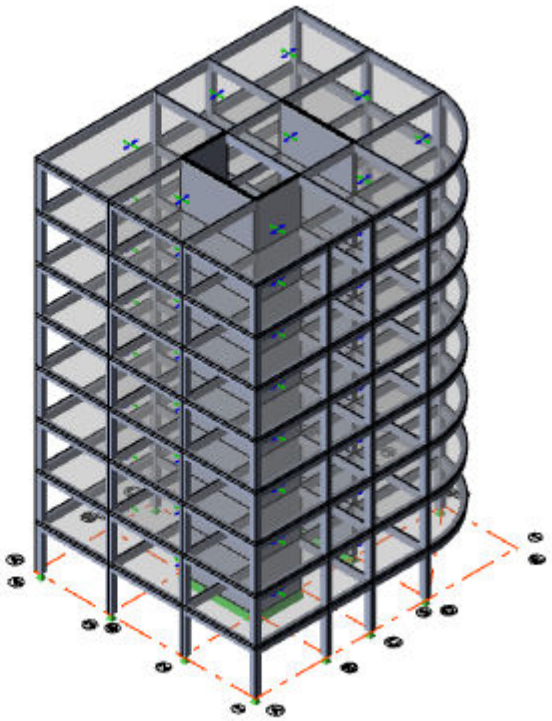
5. You can select each of the construction stages in turn in order to display its corresponding solver model.

See also

Overview of staged construction analysis (page 456)

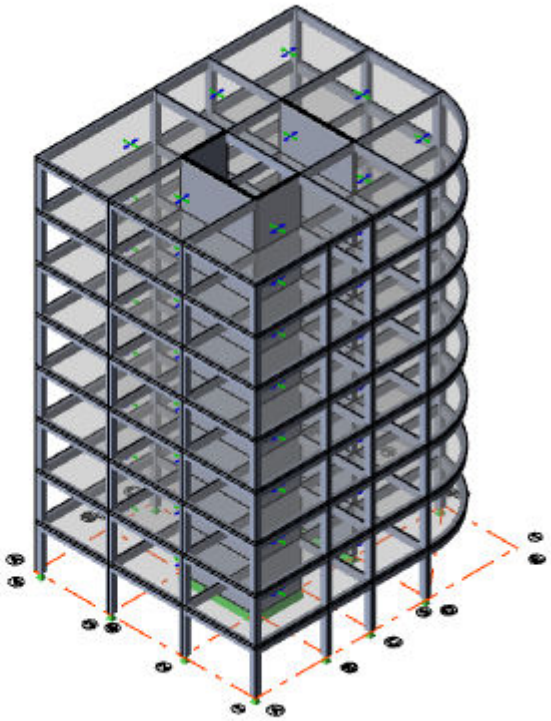
7.6 Staged construction workflow example - United States (ACI/AISC)

The following model will be used to illustrate the basic workflow for staged construction analysis and design in Tekla Structural Designer with the region set as United States (ACI/AISC) .



Download and open the tutorial model

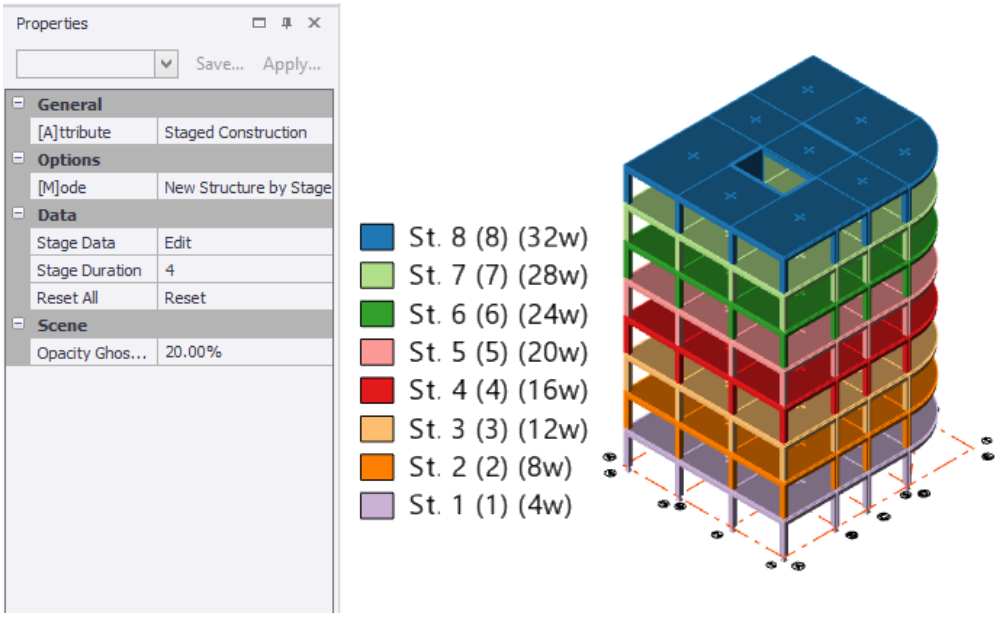
1. Download the attachment **SCA - ACI 318.zip** from this page.
2. Extract the tutorial model **SCA - ACI 318.tsmd** from the zip.
3. Open the model in Tekla Structural Designer.



Define construction stages and their stage durations

In this section we will examine the default construction stages (page 469) and adjust the stage durations.

1. Switch the view regime for the 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**.

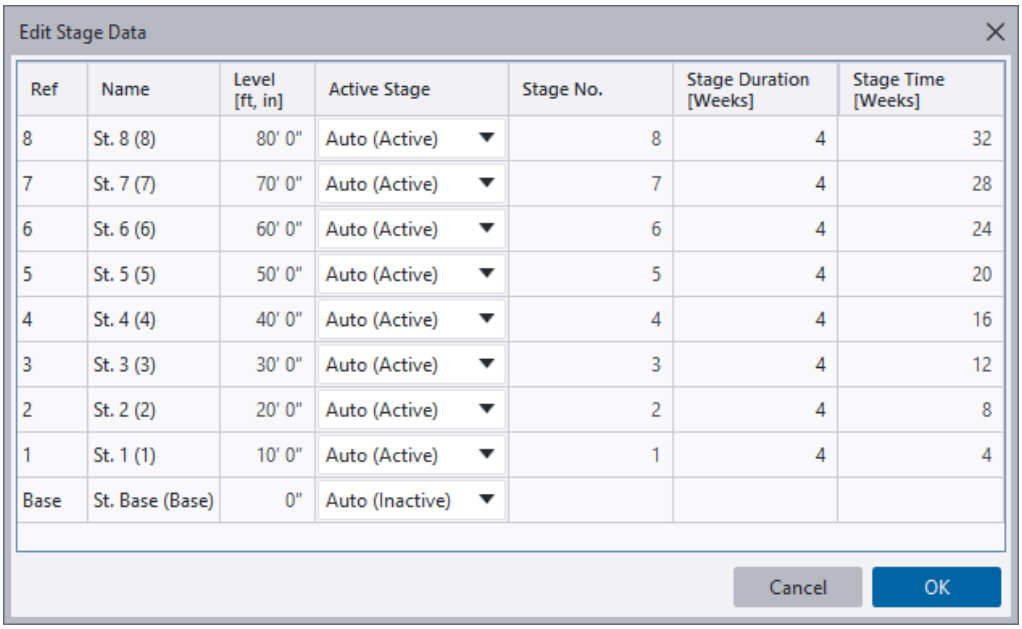


Construction stages are activated by default at every floor level in which slabs have been defined. Because this happens automatically there may be little, or no further editing required.

5. In the **Properties** window, click **Stage Data**, then click the **Edit** button

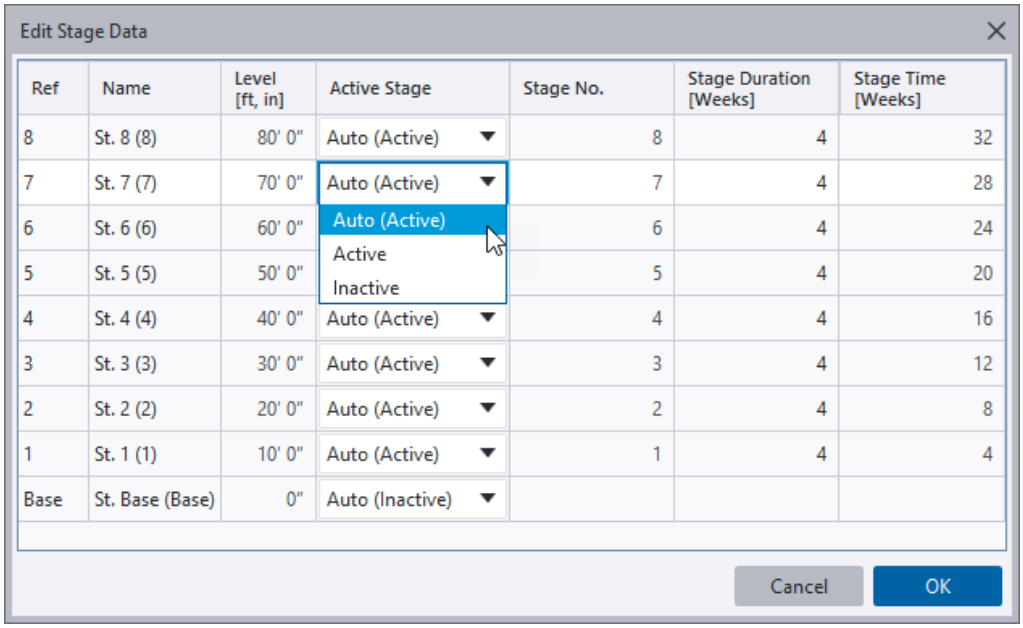


6. This displays the **Edit Stage Data** dialog box.



Each row in the above table corresponds to a level that has been marked as a floor in the **Construction Levels** dialog box.

If you want to reduce the number of stages, you can do so by setting **Active Stage** at a particular level to **Inactive**. The members at that level are then moved to the next active stage.



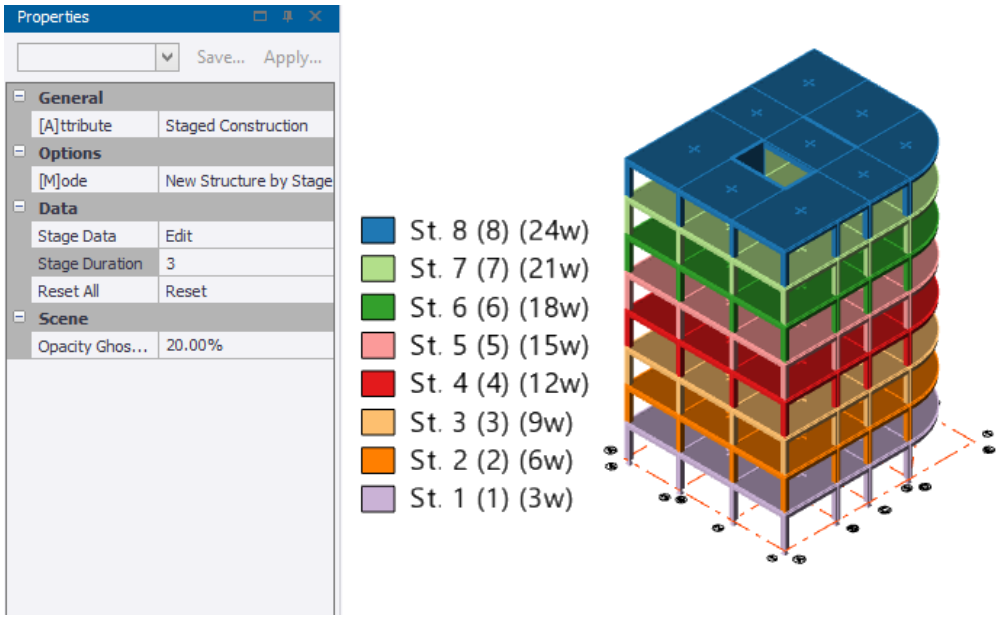
7. The easiest way to adjust individual stage durations is via the above table, however for this example we will use another method, so click **Cancel** to close the dialog.

   The stage durations will now be adjusted directly in the **Properties** window.

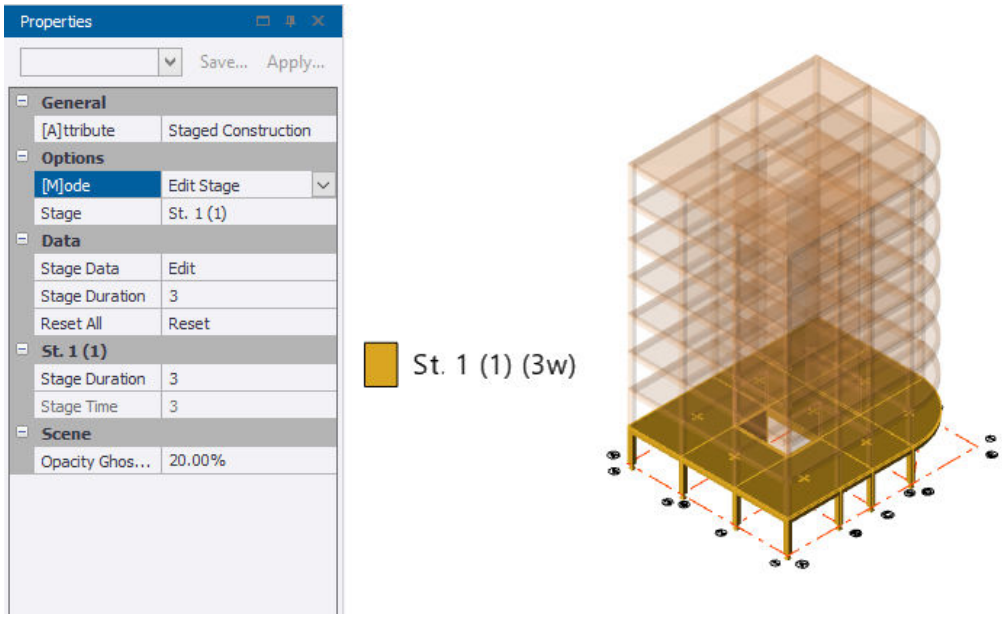
8. In the **Properties** window, first ensure the **Mode** is set to **New Structure by Stage**.

9. In the **Data** section, change the **Stage Duration** to **3**.

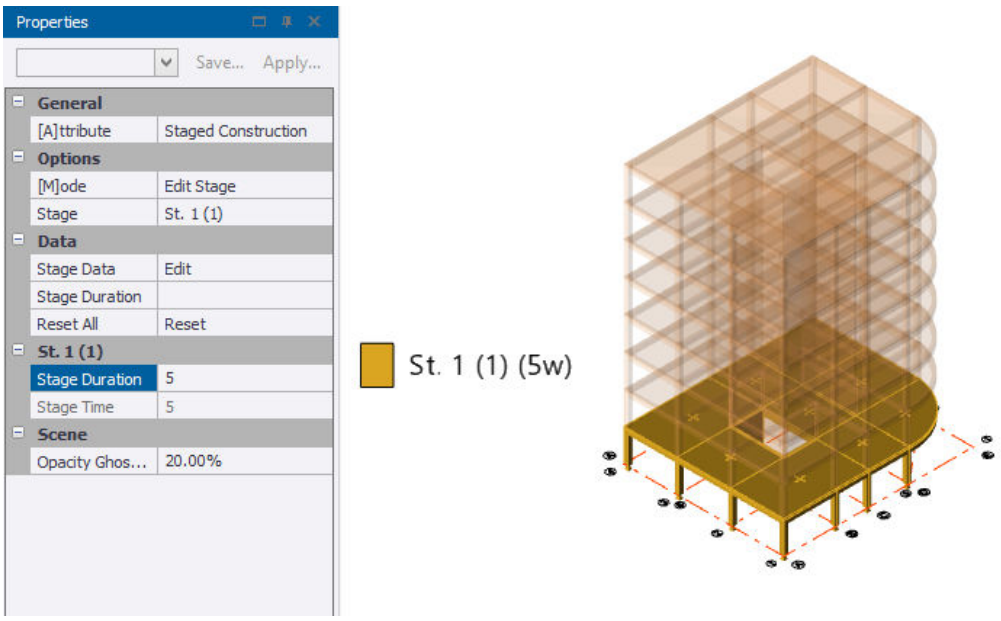
   The new duration is applied to all stages.



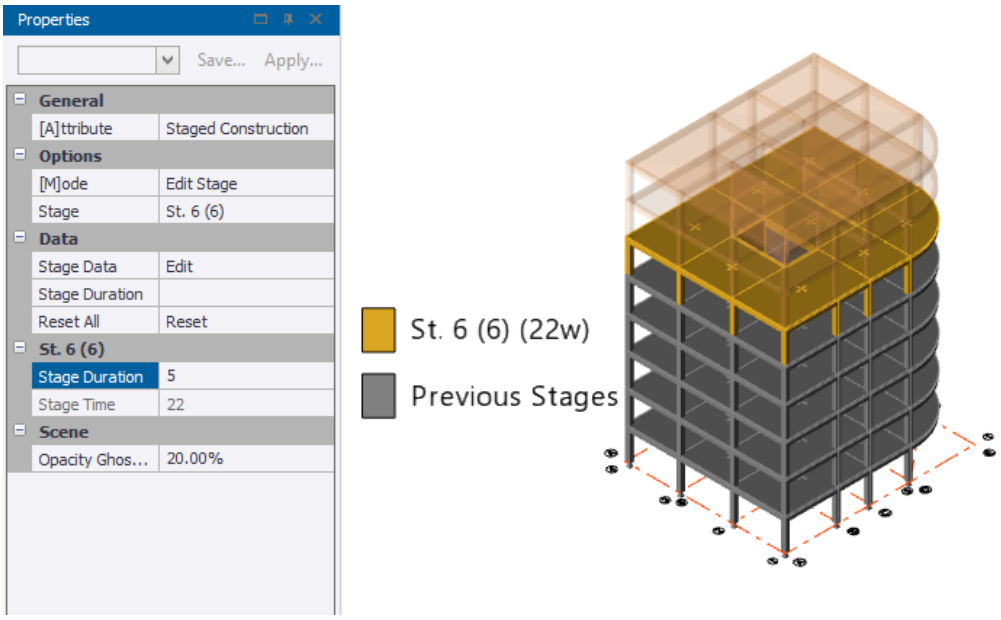
10. In the **Properties** window, change the **Mode** to **Edit Stage**, then in **Options** use the **Stage** list to select stage **St. 1 (1)**.



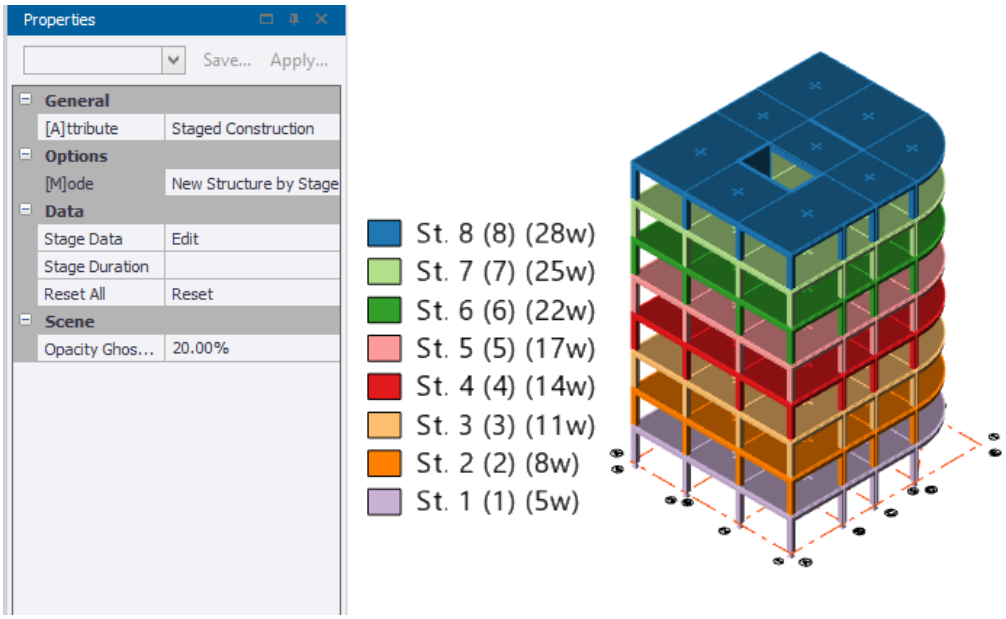
11. In the **Properties** window, in the **St.1 (1)** group, change the **Stage Duration** value to **5** weeks. (Be careful not to change it in the **Data** group as this would change the duration of *all* stages).



12. In a similar manner, set the **Stage Duration** to **5** weeks for stage **St. 6 (6)**.



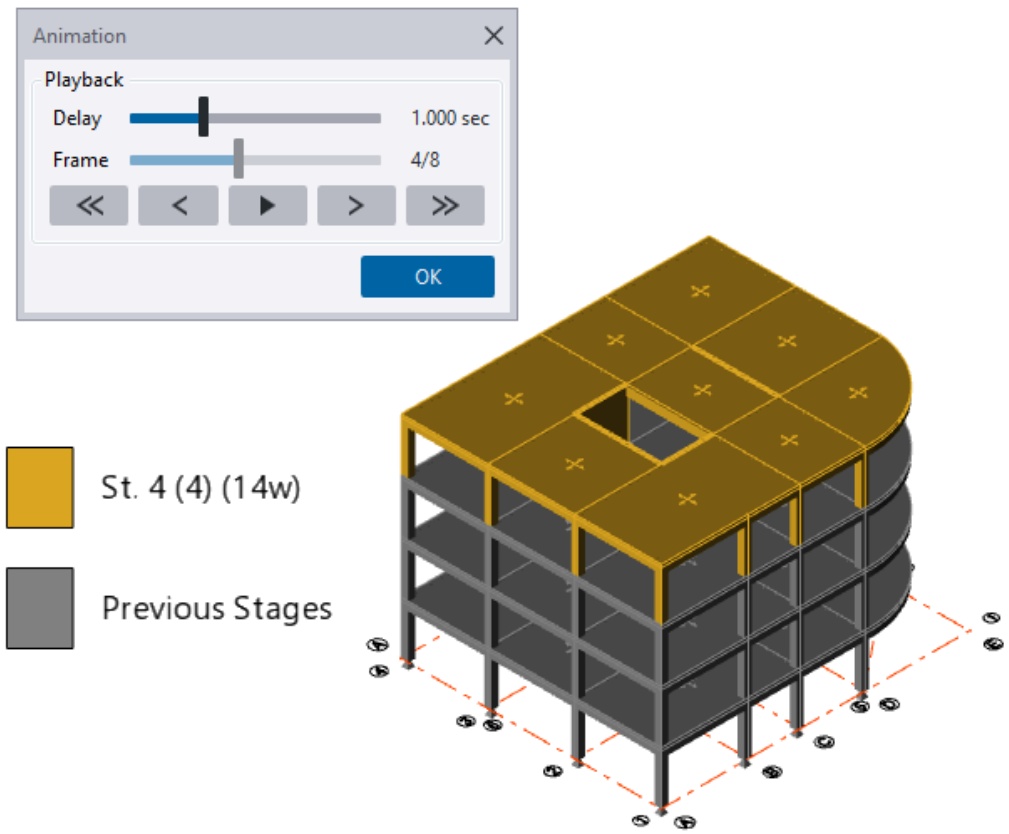
13. To review all the stage durations, change the **Mode** back to **New Structure by Stage**. The stage times shown in the legend should now be as shown below.



14. Right click and select **Animate** from the context menu.

    The **Animate** feature provides a practical way to review stages in more detail.

15. Familiarize yourself with the animation buttons.

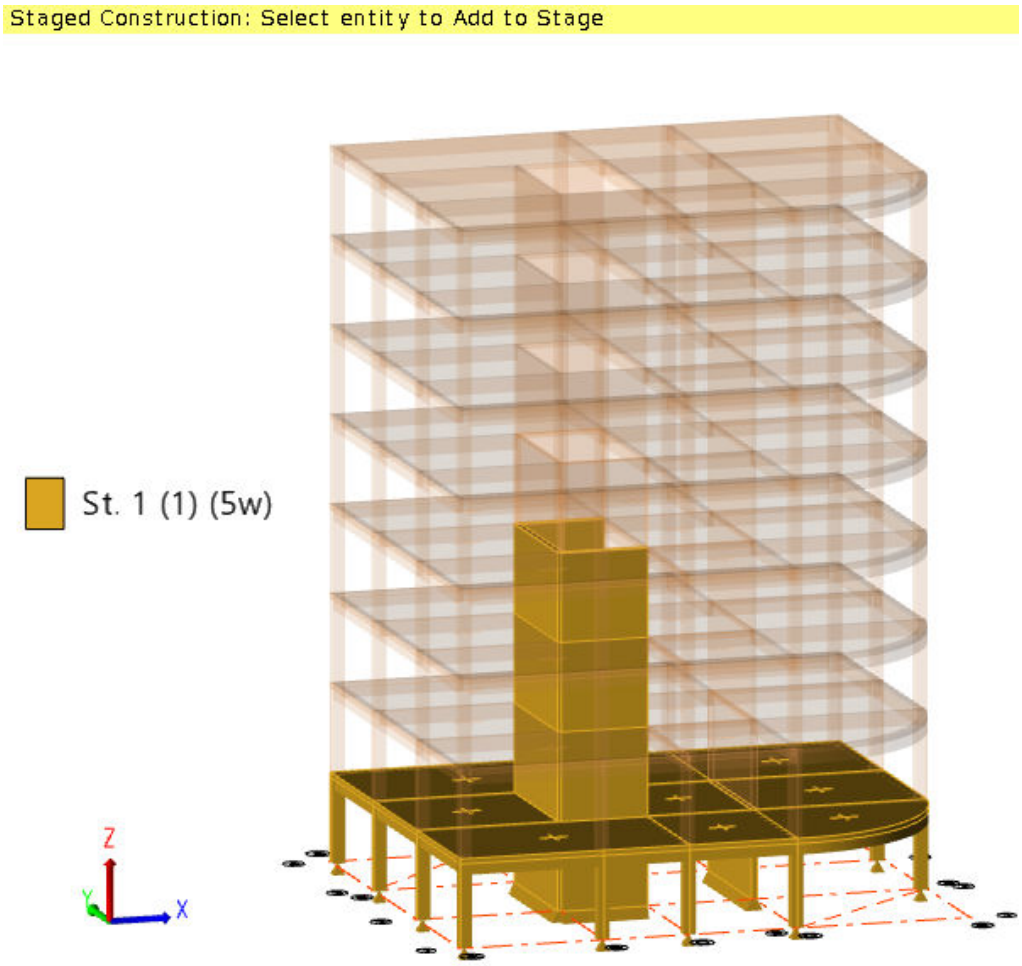


- Press the play button to move through the stages in sequence.
- Press the pause button to stop at a particular stage
- Press the back and forward buttons to display the previous/next stage.

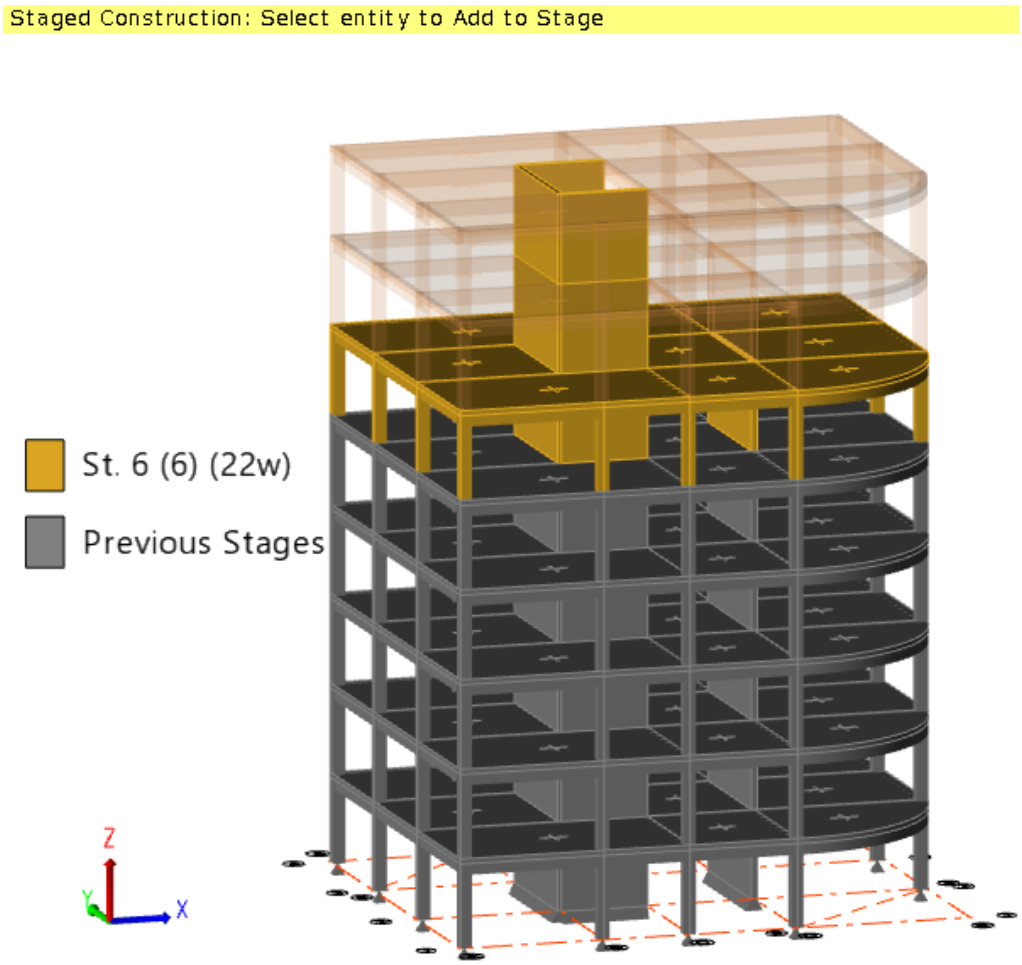
Assign members to construction stages

In this section we will reassign some of the wall panels to a different constructions stage.

1. In the **Properties** window, change the **Mode** to **Add to Stage** and ensure stage **St. 1 (1)** is selected.
2. Select the wall panels in the core up to level 4 to add them to this stage.



3. In the **Properties** window select stage **St. 6 (6)**.
4. Select the wall panels in the core up to the roof to add them to this stage.

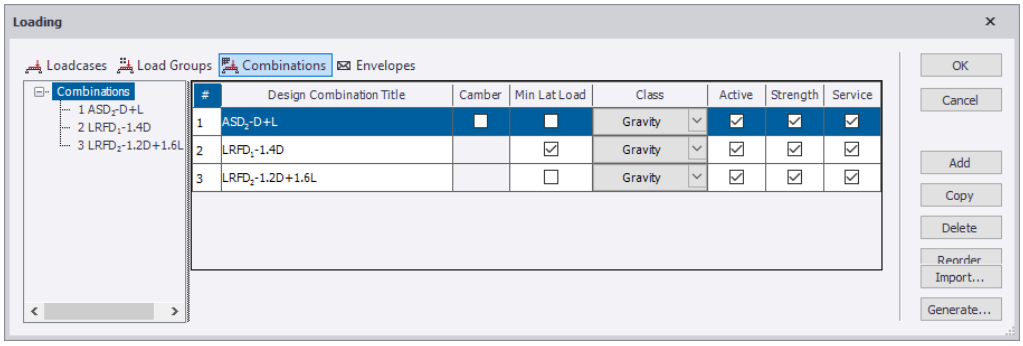


5. When stage **St. 6 (6)** is as shown above, press **Esc** to end the command.

Create a Staged Construction load combination

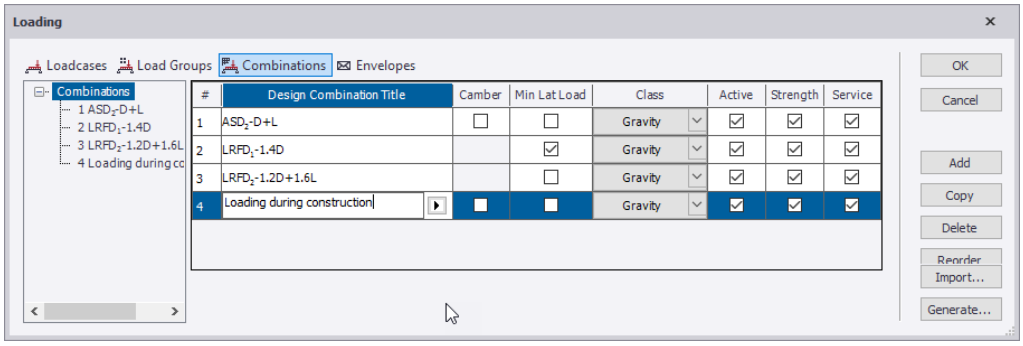
In this section we will create a staged construction load combination (page 462) and then define the loading within it.

1. On the **Load** ribbon, click **Combination**.

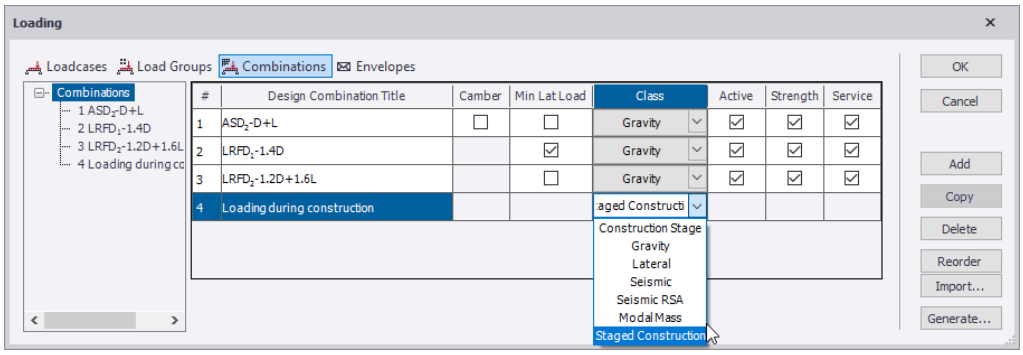


Some standard design combinations have already been created.

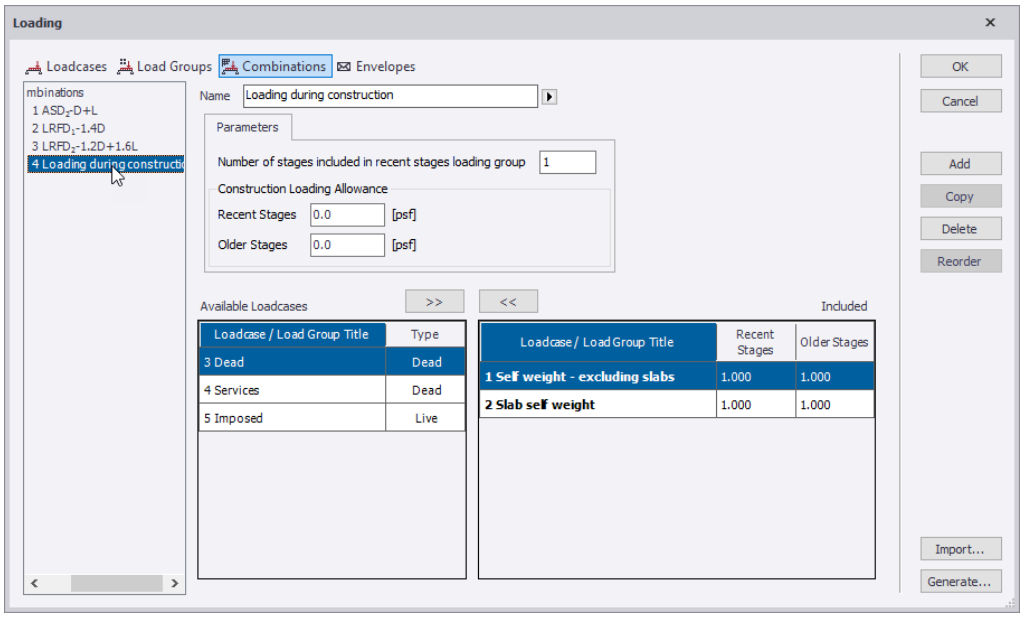
2. Click **Add**
3. Enter the new combination name as **Loading during construction**



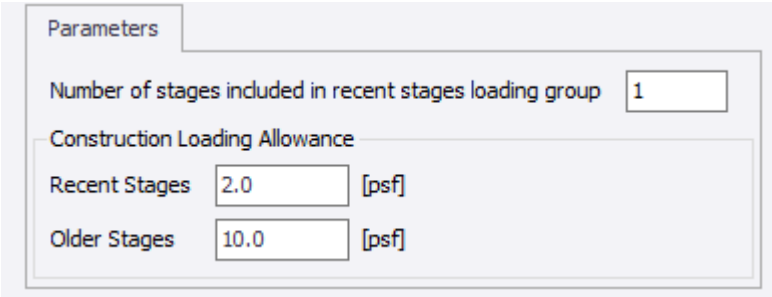
4. Change the combination **Class** to **Staged Construction**.



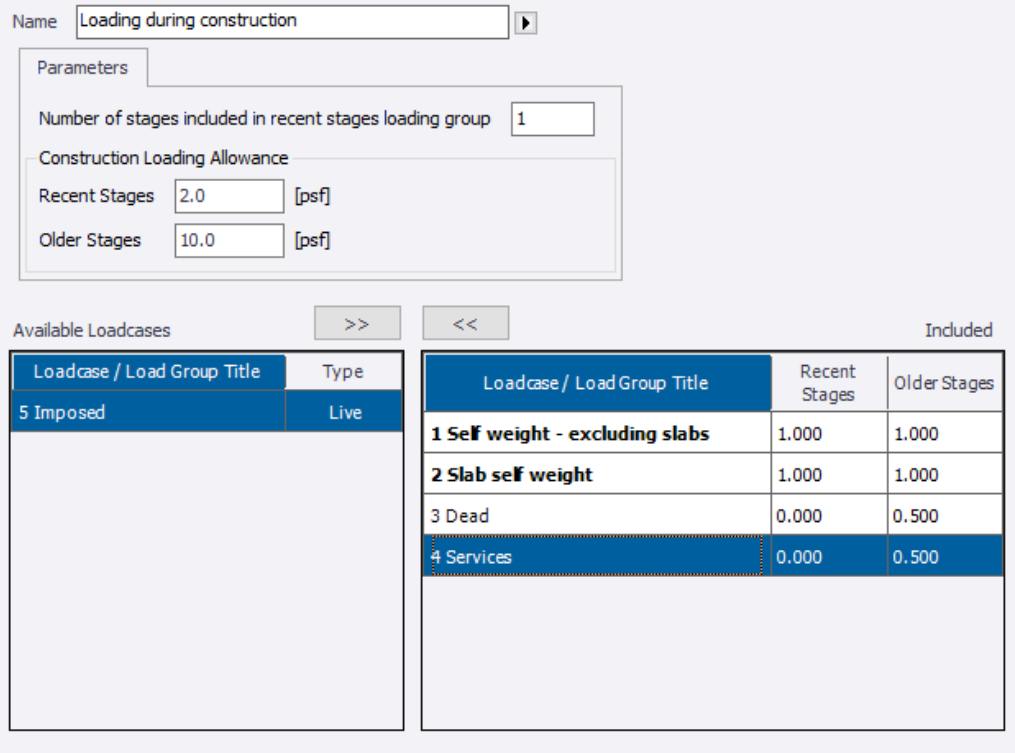
5. Select the new combination in the left hand pane of the dialog.



6. Specify the construction loading allowance parameters as shown below.



7. Include the Dead and Services loadcases in the combination and then set the load factors for recent and older stages as shown below..



In real models engineering judgment will be required when defining the staged construction combination.

8. Click **OK** to save the changes.

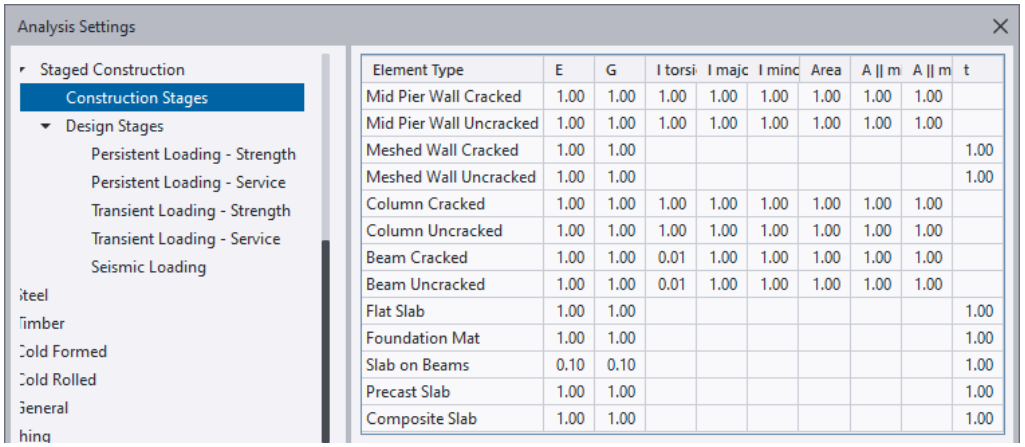
Review the modification factors

The staged construction modification factors (page 470) applied to elements in the staged construction solver model are different to those applied to other solver model types.

To review these factors:

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

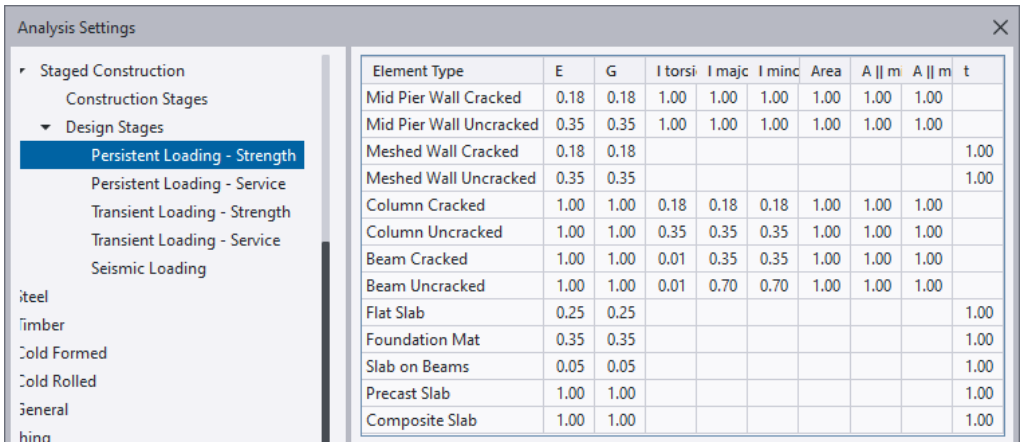
2. On the **Modification Factors** page, select the **Concrete > Staged Construction > Construction Stages** sub-page.



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	1.00	1.00							1.00
Meshed Wall Uncracked	1.00	1.00							1.00
Column Cracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	1.00	1.00	1.00	1.00	1.00	
Flat Slab	1.00	1.00							1.00
Foundation Mat	1.00	1.00							1.00
Slab on Beams	0.10	0.10							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

The above modification factors are applied to concrete elements during the construction stages.

3. Select the **Concrete > Staged Construction > Design Stages > Persistent Loading - Strength** sub-page.



Element Type	E	G	I torsi	I majc	I minc	Area	A || m	A || m	t
Mid Pier Wall Cracked	0.18	0.18	1.00	1.00	1.00	1.00	1.00	1.00	
Mid Pier Wall Uncracked	0.35	0.35	1.00	1.00	1.00	1.00	1.00	1.00	
Meshed Wall Cracked	0.18	0.18							1.00
Meshed Wall Uncracked	0.35	0.35							1.00
Column Cracked	1.00	1.00	0.18	0.18	0.18	1.00	1.00	1.00	
Column Uncracked	1.00	1.00	0.35	0.35	0.35	1.00	1.00	1.00	
Beam Cracked	1.00	1.00	0.01	0.35	0.35	1.00	1.00	1.00	
Beam Uncracked	1.00	1.00	0.01	0.70	0.70	1.00	1.00	1.00	
Flat Slab	0.25	0.25							1.00
Foundation Mat	0.35	0.35							1.00
Slab on Beams	0.05	0.05							1.00
Precast Slab	1.00	1.00							1.00
Composite Slab	1.00	1.00							1.00

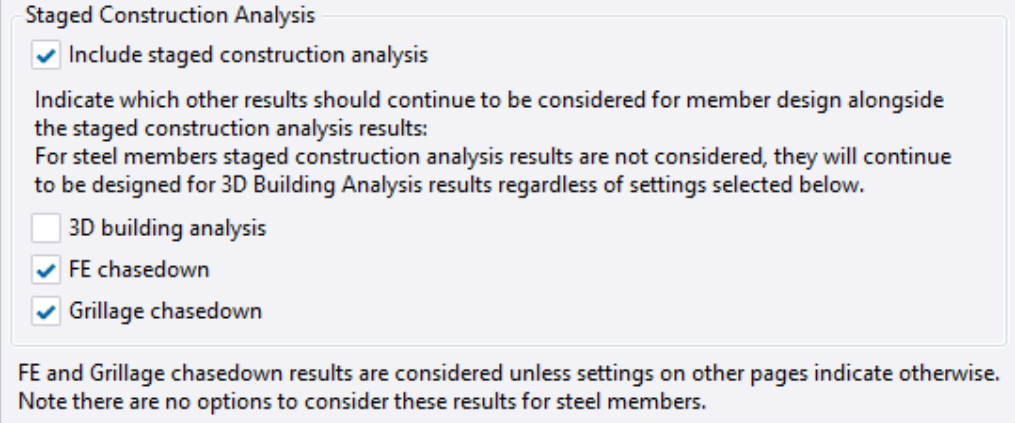
This sub-page and the four pages below it are used to define the modification factors applied to concrete elements for each of the design stages.

4. No changes are required, so click **Cancel**.

Configure the analyses

1. On the **Design** ribbon, click **Settings**.
2. On the page, select whether the analysis should be first-order or second-order. For this particular example we shall accept **First-order analysis**.
3. Next, select **Include staged construction analysis**.

4. We will accept the default for the other results, by leaving 3D building analysis unselected, and the two chasedown analyses selected.



   In effect this means that the staged construction analysis results are replacing the 3D building analysis results at one end of the design envelope. Refer to design comparison - with and without staged construction (page 481) to find out more about the implications of this.

5. Click **OK** to save the changes.

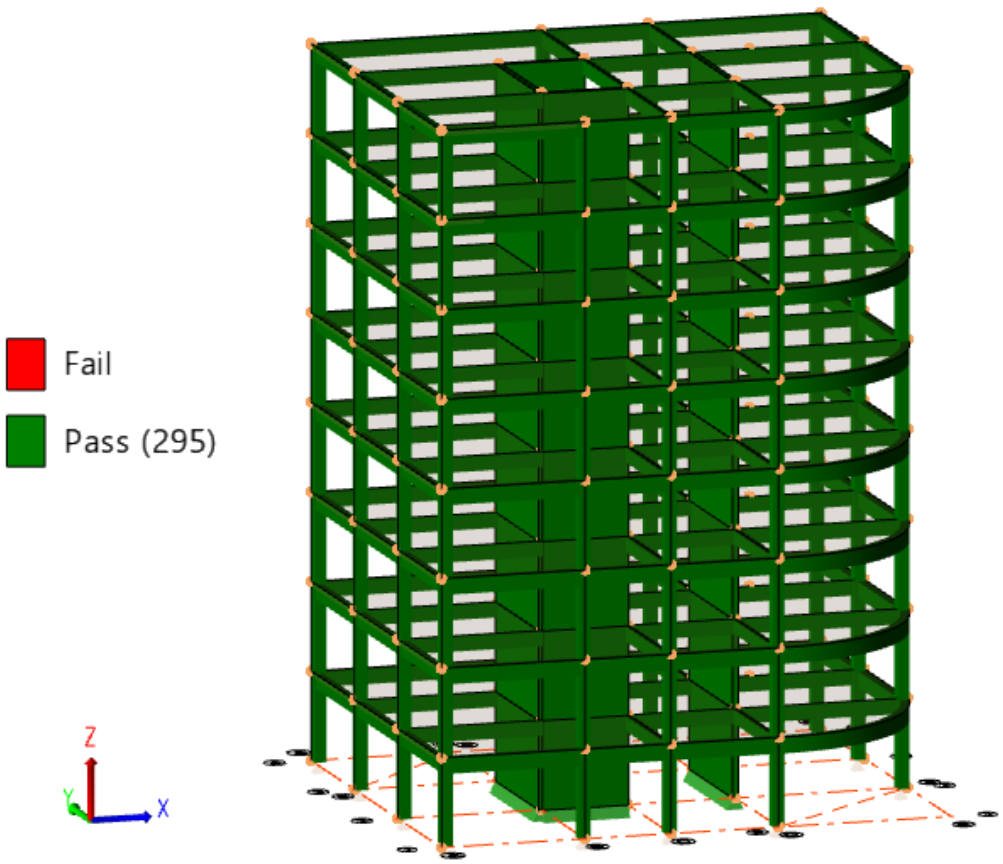
Run the design and review the design results

Now that the analysis type(s) have been set as required, you can proceed to run the combined analysis and design. This is initiated by selecting the required design option from the Design ribbon.

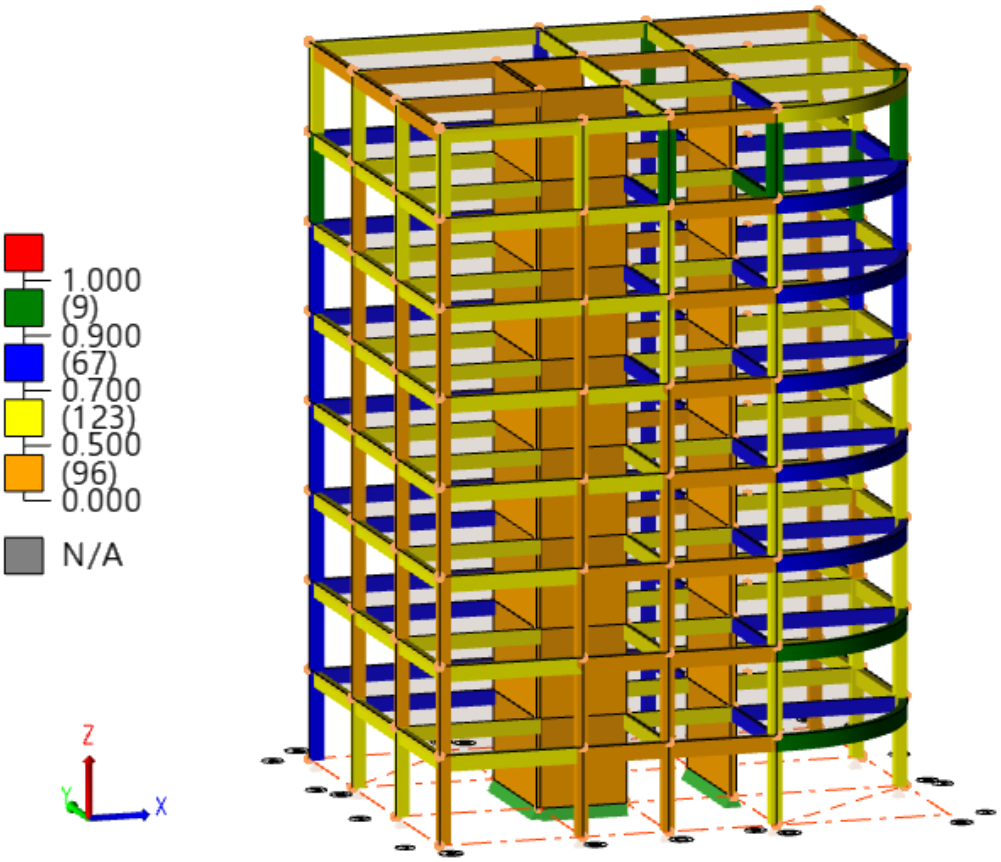
1. On the **Design** ribbon, click **Design All (Static)**.

   As this model doesn't contain any non-linear elements, a *linear* staged construction analysis is performed as part of the analysis and design process.

   Once the analysis and design process is complete the active view switches to the Review View regime and shows the design status.



2. To investigate the efficiency of the design, on the **Review** ribbon, click **Ratio**



3. To view tabular results, on the **Review** ribbon, click **Tabular Data**

4. Ensure the **View Type** is set to **Design Summary**

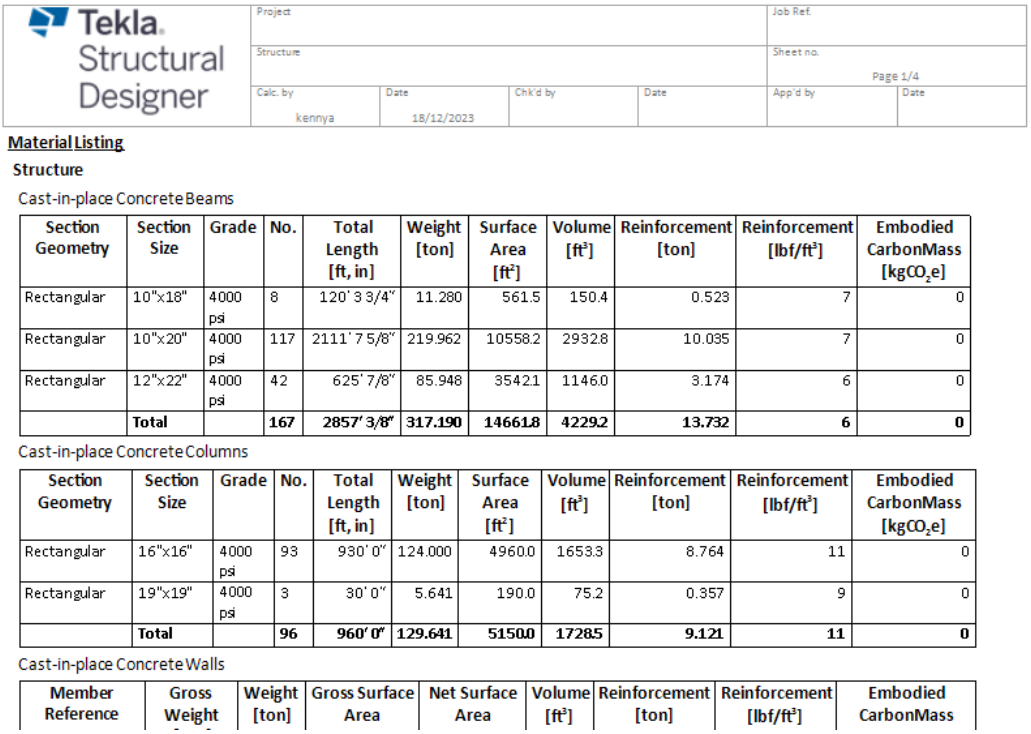
Design Summary

Member Reference	Group Ref.	Stack Ref.	Section	Grade	Length [ft, in]	Utilization	Status	Results
CC A/1	CRC5-D12	1	16x16	4000 psi	10' 0"	0.305	Pass	Results...
CC A/1	CRC5-D12	2	16x16	4000 psi	10' 0"	0.266	Pass	Results...
CC A/1	CRC5-D12	3	16x16	4000 psi	10' 0"	0.226	Pass	Results...
CC A/1	CRC5-D12	4	16x16	4000 psi	10' 0"	0.245	Pass	Results...
CC A/1	CRC5-D12	5	16x16	4000 psi	10' 0"	0.269	Pass	Results...
CC A/1	CRC5-D12	6	16x16	4000 psi	10' 0"	0.299	Pass	Results...
CC A/1	CRC5-D12	7	16x16	4000 psi	10' 0"	0.338	Pass	Results...
CC A/1	CRC5-D12	8	16x16	4000 psi	10' 0"	0.617	Pass	Results...
CC A/2	CRC5-D8	1	16x16	4000 psi	10' 0"	0.535	Pass	Results...
CC A/2	CRC5-D8	2	16x16	4000 psi	10' 0"	0.465	Pass	Results...
CC A/2	CRC5-D8	3	16x16	4000 psi	10' 0"	0.396	Pass	Results...
CC A/2	CRC5-D8	4	16x16	4000 psi	10' 0"	0.356	Pass	Results...
CC A/2	CRC5-D8	5	16x16	4000 psi	10' 0"	0.400	Pass	Results...
CC A/2	CRC5-D8	6	16x16	4000 psi	10' 0"	0.466	Pass	Results...
CC A/2	CRC5-D8	7	16x16	4000 psi	10' 0"	0.557	Pass	Results...
CC A/2	CRC5-D8	8	16x16	4000 psi	10' 0"	0.671	Pass	Results...
CC A/3	CRC1-D1	1	16x16	4000 psi	10' 0"	0.533	Pass	Results...
CC A/3	CRC1-D1	2	16x16	4000 psi	10' 0"	0.462	Pass	Results...
CC A/3	CRC1-D1	3	16x16	4000 psi	10' 0"	0.392	Pass	Results...
CC A/3	CRC1-D1	4	16x16	4000 psi	10' 0"	0.350	Pass	Results...
CC A/3	CRC1-D1	5	16x16	4000 psi	10' 0"	0.350	Pass	Results...
CC A/3	CRC1-D1	6	16x16	4000 psi	10' 0"	0.350	Pass	Results...
CC A/3	CRC1-D1	7	16x16	4000 psi	10' 0"	0.481	Pass	Results...
CC A/3	CRC1-D1	8	16x16	4000 psi	10' 0"	0.652	Pass	Results...
CC D/4	CRC5-D7	1	16x16	4000 psi	10' 0"	0.762	Pass	Results...

Any failing members should be investigated in the normal way.

Output the calculations

Once the design is satisfactory, reports and drawings can be output in the normal way.



Tekla Structural Designer

Project				Job Ref.	
Structure				Sheet no.	Page 1/4
Calc. by	Date	Chk'd by	Date	App'd by	Date
kennya	18/12/2023				

Material Listing

Structure

Cast-in-place Concrete Beams

Section Geometry	Section Size	Grade	No.	Total Length [ft, in]	Weight [ton]	Surface Area [ft²]	Volume [ft³]	Reinforcement [ton]	Reinforcement [lbf/ft³]	Embodied CarbonMass [kgCO₂e]
Rectangular	10"x18"	4000 psi	8	120' 3 3/4"	11.280	561.5	150.4	0.523	7	0
Rectangular	10"x20"	4000 psi	117	2111' 7 5/8"	219.962	10558.2	2932.8	10.035	7	0
Rectangular	12"x22"	4000 psi	42	625' 7/8"	85.948	3542.1	1146.0	3.174	6	0
	Total		**167**	**2857' 3/8"**	**317.190**	**14661.8**	**4229.2**	**13.732**	**6**	**0**

Cast-in-place Concrete Columns

Section Geometry	Section Size	Grade	No.	Total Length [ft, in]	Weight [ton]	Surface Area [ft²]	Volume [ft³]	Reinforcement [ton]	Reinforcement [lbf/ft³]	Embodied CarbonMass [kgCO₂e]
Rectangular	16"x16"	4000 psi	93	930' 0"	124.000	4960.0	1653.3	8.764	11	0
Rectangular	19"x19"	4000 psi	3	30' 0"	5.641	190.0	75.2	0.357	9	0
	Total		**96**	**960' 0"**	**129.641**	**5150.0**	**1728.5**	**9.121**	**11**	**0**

Cast-in-place Concrete Walls

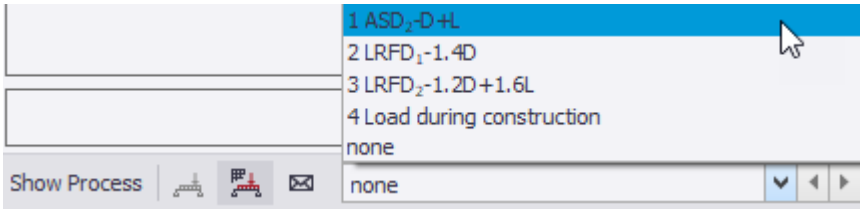
Member Reference	Gross Weight	Weight [ton]	Gross Surface Area	Net Surface Area	Volume [ft³]	Reinforcement [ton]	Reinforcement [lbf/ft³]	Embodied CarbonMass

Display staged construction analysis results

You can graphically display staged construction analysis results from a 3D Results View. The steps below demonstrate how to display the results, firstly for a design stage combination, and then for each of the construction stages.

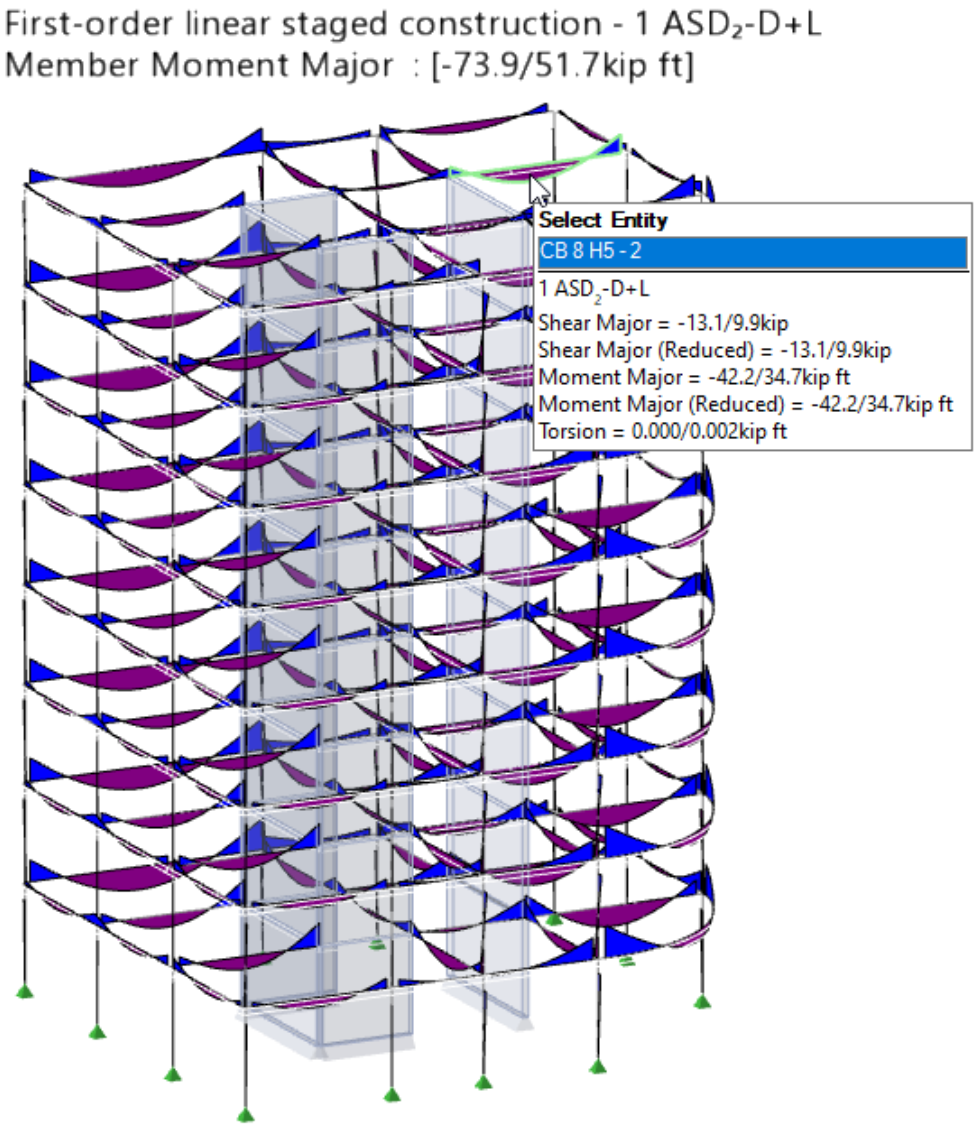
1. Switch the view regime for the Structure 3D view to  **Results View**.
2. On the **Results** ribbon, in the **Method** list, select **First-order linear staged construction**.

   The design stage combinations and the staged construction loading combination are available in the **Loading list**.



3. Select the **ASD_2 - D + L** combination.
4. From the **1D Results** group on the **Results** ribbon, select **Moment Major**.

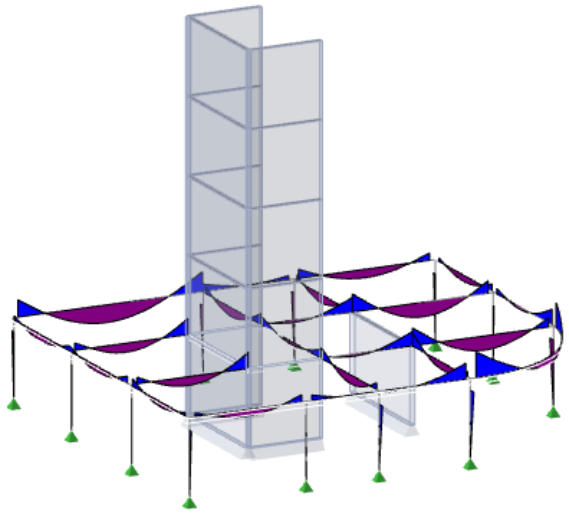
   The results are displayed for the selected combination; when the cursor is positioned over a member the tooltip displays more detailed information.



5. To view results for each of the construction stages, select the **Load during construction** combination from the **Loading list**.

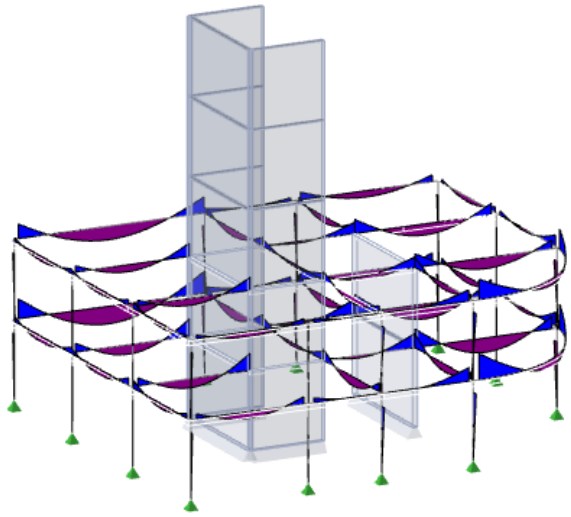
   Initially the first construction stage is displayed as shown below.

First-order linear staged construction - 4 Load during construction - St. 1 (1)
Member Moment Major : [-49.2/33.3kip ft]



6. On the **Results** ribbon, in the **Stage** list, you can select each of the stages in turn to display results for each stage.

First-order linear staged construction - 4 Load during construction - St. 2 (2)
Member Moment Major : [-62.2/41.7kip ft]



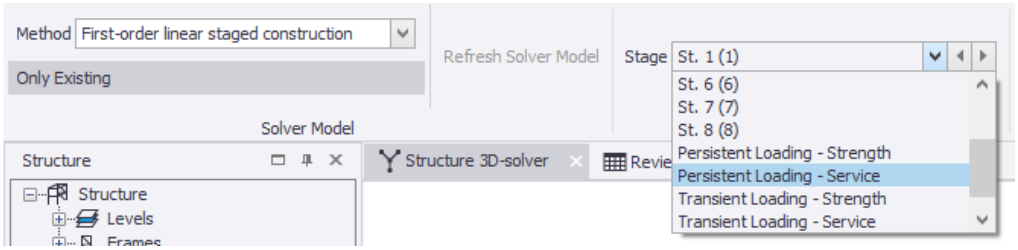
Display staged construction solver models

You can display the individual solver models for the different design stages and each construction stage from a Solver View.

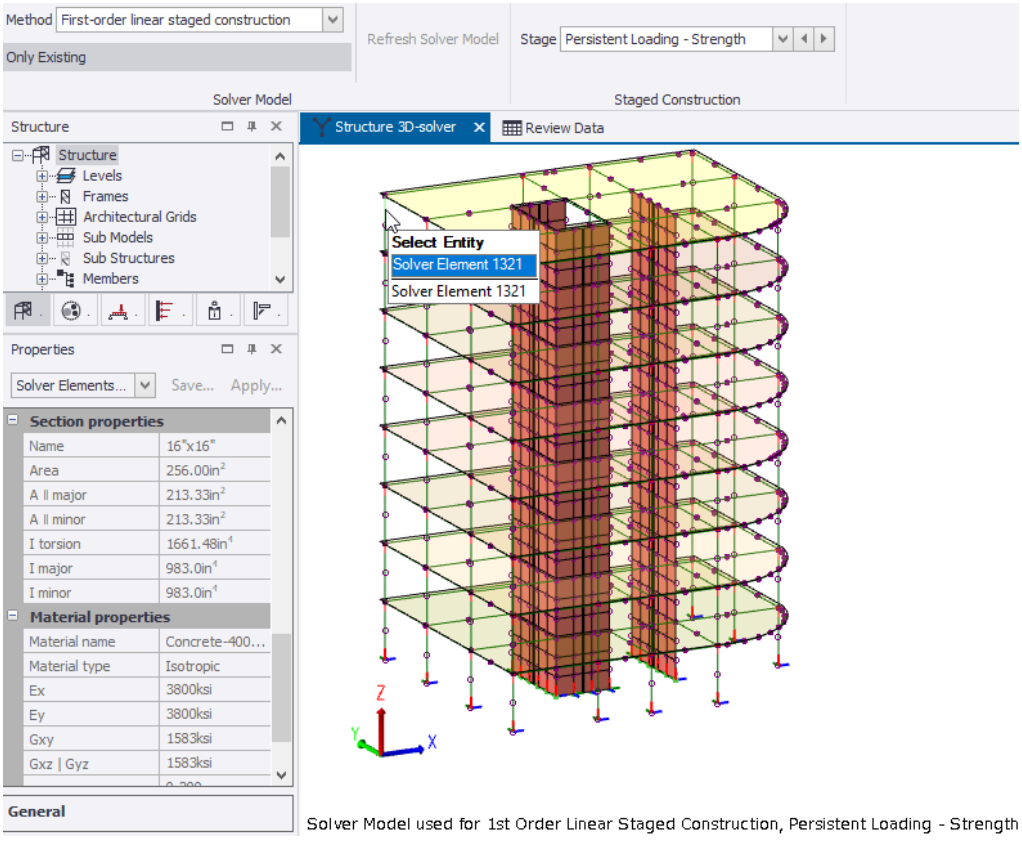
1. Switch the view regime to **Solver View**.

2. On the **Solver** ribbon, in the **Method** list, select **First-order linear staged construction**.

   A **Stage** list is added to the ribbon, to enable each construction stage and the different design stages to be selected.



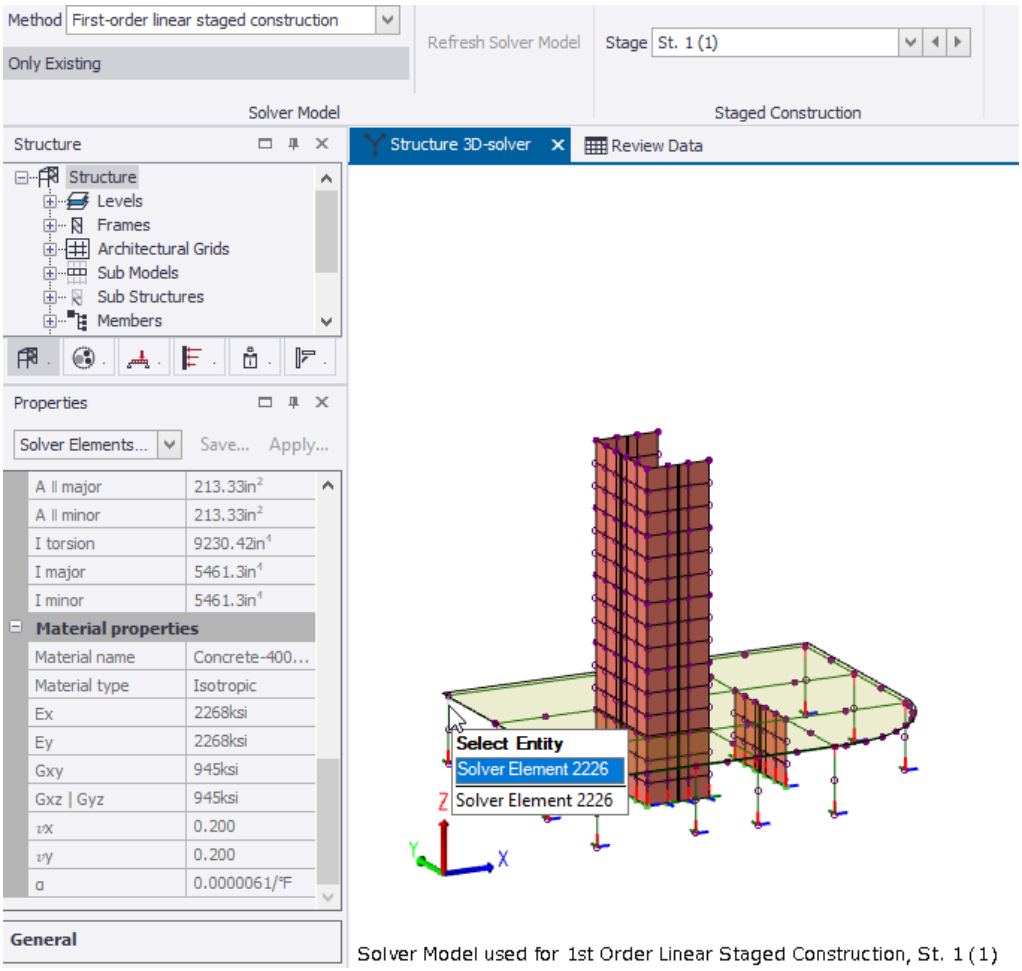
3. In the **Stage** list, select **Persistent Loading - Strength** (one of the design stages) .



Solver Model used for 1st Order Linear Staged Construction, Persistent Loading - Strength

   The solver model is displayed accordingly. As shown above, when an element is selected its properties are displayed in the **Properties** window. The properties displayed reflect the modification factors that are applicable for the selected stage.

4. In the **Stage** list, select **St. 1 (1)** (one of the construction stages) .



Solver Model used for 1st Order Linear Staged Construction, St. 1 (1)

The solver model for the first construction stage is displayed. As shown above, when an element is selected its properties are displayed in the Properties window. The properties displayed reflect the construction stage modification factors and also any adjustments made for concrete creep allowance (page 476).

5. You can select each of the construction stages in turn in order to display its corresponding solver model.

See also

Overview of staged construction analysis (page 456)

8 Seismic analysis and design handbook

You can find the following information in this handbook:

- Introduction to seismic analysis and design (page 535)
- Limitations of Seismic Design (page 550)
- Seismic force resisting systems (page 551)
- Seismic design methods (page 554)

8.1 Introduction to seismic analysis and design

Definitions

The actual method applied to a specific beam will depend upon whether it is required to support sensitive finishes.

Various terms used in Tekla Structural Designer's seismic processes are described below:

Code Spectra

The spectra specified in a country's loading and design codes for use in ELF and RSA analysis.

Site Specific Spectra

User defined spectra for ELF and RSA which are required for locations which use another country's loading and design codes where the code spectra are not relevant.

Base Shear Combination

Also referred to as the Effective Seismic Weight Combination (ASCE 7/UBC) or the Seismic Inertia Combination (Eurocode). This combination is used for modal analysis, and in the calculation of base shears, during the Seismic Analysis Process. This combination is created and modified by the Seismic Wizard only.

RSA Seismic Combinations

These combinations are created by the Combination Generator at the end of the Seismic Wizard, but can also be modified in the standard Combination dialog. They consist of 3 kinds of loadcases: Static, RSA Seismic and RSA Torsion. The Base Shear Combination is not included in this category of combination.

Static Loadcase

Standard loadcases, e.g. "Self weight - excluding slabs", "Dead", etc., and derived cases for NHF/EHF, but no patterns.

RSA Seismic Loadcase

Two loadcases, i.e. "Seismic Dir1" and "Seismic Dir2", which cannot be edited. These are created at the end of the Seismic Wizard. being derived from information supplied in the Seismic Wizard and the results of the modal analysis. No actual loads are available for graphical display.

RSA Torsion Loadcase

These cases can be generated by the Seismic Wizard and are regenerated whenever RSA Seismic Combinations are modified.

Fundamental Period (T)

Separately for Dir 1 & Dir 2, this is either defined in the Seismic Wizard, (user value or calculated), or determined in the modal analysis for the Base Shear Combination.

Level Seismic Weight

For each relevant level, this is the sum of the vertical forces in nodes on that level, for the Base Shear Combination.

Effective Seismic Weight

This is the sum of the level seismic weights for all relevant levels for the Base Shear Combination.

Seismic Base Shear

The base shear is calculated separately for Dir 1 & Dir 2, for the Base Shear Combination.

NOTE In Tekla Structural Designer the base shear is displayed when you display story shear tabular results.

Square root of Summation of Square (SRSS)

The SRSS formula for combining modes in RSA is as follows:

$$\lambda = \sqrt{(\sum_{k=1}^{n} (\lambda_k^2))}$$

λ = Absolute value of combined "response"

λ_k = "response" value for Relevant Mode k

n = Number of Relevant Modes

Complete Quadratic Combination (CQC)

The CQC formula for combining modes in RSA is as follows:

$$\lambda = \sqrt{(\sum_{i=1}^{n} \sum_{j=1}^{n} (\lambda_i \ \rho_{ij} \ \lambda_j))}$$

λ = Absolute value of combined "response"

λ_i = "response" value for Relevant Mode i

λ_j = "response" value for Relevant Mode j

n = Number of Relevant Modes

ρ_{ij} = Cross modal coefficient for i & j

Cross Modal Coefficient

This co-efficient is used in the CQC method for combining modes in RSA.

$$\rho_{ij} = \frac{8\zeta^2 \ (1+\beta)\beta^{1.5}}{(1-\beta^2)^2 + 4\zeta^2(1+\beta)^2}$$

ζ = modal damping ratio

IBC/ASCE assumed = 5% (ASCE Figs 22-1 to 6)

EC8 assumed = 5% where q accounts for the damping in various materials being different to 5% (EC8 Cl 3.2.2.5)

IS codes the user can define the level of damping and this is accounted for in the above equation.

β = Frequency ratio = ω_i / ω_j

ω_i = Frequency for Relevant Mode i

ω_j = Frequency for Relevant Mode j

Overview

All seismic codes work in a similar manner from the loading view point with relatively minor differences in terminology and methodology.

It is worth noting at the start that seismic analysis determines a set of forces for which it is expected (statistically) that if those forces are designed for and other design precautions taken (additional seismic design) then in the event of an earthquake the structure may well suffer extensive damage but will not collapse and for some categories of building should actually remain serviceable.

In Tekla Structural Designer a seismic wizard gathers all the information together to setup the requirements for a seismic analysis.

From this information a number of things are determined:

- If working to ASCE 7 - the seismic design category (SDC) for the building - giving amongst other things the permissible type of analysis
- The Base Shear Combination to determine the seismic base shear in the building
- The natural frequencies of the building in two horizontal directions
- The combination of the gravity and other lateral forces with the seismic loadcases

Earthquakes load a building by a random cyclic acceleration and deceleration of the foundations. These are in both horizontal directions (Dir1 and Dir2) but can also be in a vertical direction too. This ground acceleration excites the building in its natural and higher frequencies.

As a result if the building is:

- In an area of low seismic acceleration, low in height and poses limited risk to life then a gross approximation can be used in analysis - assuming a % of gravity loading is applied horizontally to the building to represent the earthquake (US codes 1%, Australian codes 10%).
- In an area of moderate to low seismic acceleration, medium to low in height and does not house a significant number of people - the predominant mode excited is the 1st mode. An equivalent lateral force (ELF) approximation can be used that applies static horizontal loading distributed up the building to mimic the shape of the 1st mode of vibration in a static analysis.

- Anything else, in an area of high acceleration, tall in height and could be holding many people or be critical post-earthquake then a "more representative" analysis method of Response Spectrum Analysis (RSA) should be used. This analysis is based upon a modal analysis considering all mode shapes in the two horizontal directions in which typically 90% of the structure's mass is mobilized.

The results from the chosen method of seismic analysis are used in combination with other gravity and lateral loadcases to design both normal members and those members in seismic force resisting systems (SFRS). These latter members need additional design and detailing rules to ensure they resist the seismic forces that they subjected to during an earthquake. If working to ASCE 7, the extent of these rules are determined by the SDC noted above (the higher the demand, the 'better' the SFRS that will be required).

NOTE The additional design and detailing requirements of "seismic" design are only supported in Tekla Structural Designer for the ACI/AISC and the Indian regional codes.

Seismic Wizard

In Tekla Structural Designer the Seismic Wizard sets up the information required for either ELF or RSA seismic analysis - the main input parameters being:

- Ground acceleration - strength of the earthquake
- The Importance Level (or Importance Class) of the building - the use to which the building is being put - typically
  - I= very minor, farm and temporary buildings,
  - II= general buildings occupied by people,
  - III = buildings occupied by a large number of people
  - IV = critical buildings with a post-disaster function eg hospitals, police stations, fire stations and buildings along access route to them)
- The ground conditions upon which the building is founded (typically Hard Rock, Rock, Shallow soil, Deep Soil, Very Soft Soil)
- Building height - for low buildings the first mode is totally dominant in taller buildings other modes become significant
- Plan and vertical irregularities in the building

From this input the Seismic Wizard determines the seismic design category for ASCE 7, and also the elastic design response spectrum to be used for the building.

Additionally the Wizard sets up the Base Shear Combination - the combination of loads likely to be acting on the building when the earthquake strikes.

Vertical and Horizontal Irregularities

There are typically 5 types of horizontal irregularity and 5 types of vertical irregularity - all are defined to pick up structures that have lateral framing systems and shapes in plan that will preclude the structure naturally developing a simple first mode of vibration. Since this is a basic assumption of ELF - the presence of these irregularities may preclude the use of ELF.

Vertical Loads

NOTE There is no explicit analysis or consideration of a vertical spectrum in either the ELF or RSA analysis methods in Tekla Structural Designer.

The treatment of seismic vertical load effects in Tekla Structural Designer is code dependent:

ASCE 7

Vertical seismic load effect

The vertical seismic load effect, E_v is defined in ASCE 7-22 clause 12.4.2 as follows:

$E = E_h + E_v$

or

$E = E_h - E_v$

where

E = seismic load effect

E_h *= horizontal seismic load effect*

E_v *= vertical seismic load effect*

E_v is determined from clause 12.4.2.2 as:

$E_v = 0.2 \times S_{DS} \times D$

where

S_{DS} *= design spectral response acceleration parameter at short periods*

D = effect of dead load

In Tekla Structural Designer, when it is to be applied, E_v is calculated when the seismic combinations are generated. The calculation is the same irrespective of the ASCE 7 code year being worked to. Whether it is applied or not depends on on the Seismic Design Category:

- For structures assigned to Seismic Design Category A, E_v is not applied.

- For structures assigned to Seismic Design Category B an 'Include vertical seismic load effect' checkbox is provided on the Basic Information page of the seismic wizard. By unselecting the checkbox E_v will taken as zero.
- For structures assigned to Seismic Design Categories C, D, E or F; E_v is automatically applied.

When E_v is applied, it is accounted for by an adjustment to the dead load factors in the seismic combinations. This adjustment is illustrated in the following examples.

Example 1 - adjustment of dead load factors in the ASD seismic combination $1.0D + 0.7_v + 0.7_h$ when S_{DS} = 26.67 % g	
Original combination including vertical seismic load effect	$1.0D + 0.7E_v + 0.7E_h$
Substituting for E_v	$1.0D + 0.7(0.2 \times S_{DS} \times D) + 0.7E_h$
Which can be rearranged to	$(1.0 + 0.7(0.2 \times S_{DS}))D + 0.7E_h$
Substituting for S_{DS} = 0.2667	$(1.0 + 0.7(0.2 \times 0.2667))D + 0.7E_h$
Which simplifies to	$1.037D + 0.7E_h$
Hence the adjusted factors displayed for this combination in Tekla Structural Designer:	

Loadcase / Load Group Title	Strength	Service
1 Self weight - excluding slabs	1.037	0.000
2 Slab self weight	1.037	0.000
3 Dead	1.037	0.000
7 Seismic Dir1	0.700	0.000
9 Seismic Torsion	0.700	0.000

Example 2 - adjustment of dead load factors in the ASD seismic combination $0.6D - 0.7_v + 0.7_h$ when S_{DS} = 26.67 % g	
Original combination including vertical seismic load effect	$0.6D - 0.7_v + 0.7E_h$
Substituting for E_v	$0.6D - 0.7(0.2 \times S_{DS} \times D) + 0.7E_h$
Which can be rearranged to	$(0.6 - 0.7(0.2 \times S_{DS}))D + 0.7E_h$
Substituting for S_{DS} = 0.2667	$(0.6 - 0.7(0.2 \times 0.2667))D + 0.7E_h$
Which simplifies to	$0.563D + 0.7E_h$

Example 2 - adjustment of dead load factors in the ASD seismic combination 0.6D - 0.7_v + 0.7_h when S_{DS} = 26.67 % g

Hence the adjusted factors displayed for this combination in Tekla Structural Designer:

Loadcase / Load Group Title	Strength	Service
1 Self weight - excluding slabs	0.563	0.000
2 Slab self weight	0.563	0.000
3 Dead	0.563	0.000
7 Seismic Dir1	0.700	0.000
9 Seismic Torsion	0.700	0.000

Vertical spectra

The commentary to ASCE 7-22 states:

> *For most structures, the effect of vertical ground motions is not analyzed explicitly; it is implicitly included by adjusting the load factors (up and down) for permanent dead loads, as specified in Section 12.4. Certain conditions requiring more detailed analysis of vertical response are defined in Chapters 13 and 15 for nonstructural components and nonbuilding structures, respectively.*

Consequently, there is no explicit analysis or consideration of a vertical spectrum in either the ELF or RSA analysis methods in Tekla Structural Designer.

Horizontal cantilevers

Horizontal cantilevers have broadly similar requirements in each of the ASCE 7 versions. For ASCE 7-22 the requirement is stated in clause 12.4.4 as follows:

> *12.4.4 Minimum Upward Force for Horizontal Cantilevers for Seismic Design Categories D through F*
>
> *For a structure assigned to Seismic Design Category D, E, or F, horizontal cantilever structural members shall be designed for a supplemental basic load combination consisting of a minimum net upward force of 0.2 times the dead load for strength design, or 0.14 times the dead load for allowable stress design.*

Tekla Structural Designer **does not** do this automatically and so it is up to the user to deal with it with separate combinations.

UBC

Vertical seismic load effect

The method of application for UBC is the same as that for ASCE 7 - the calculated vertical seismic load effect being automatically applied to either

amplify, or in combinations where the dead load is being considered as beneficial, to reduce the dead load strength factors in the seismic combinations. The only difference being that for UBC the vertical seismic load effect factor is calculated as follows:

- 0.5 x C_a x I x Dead loads for strength design
- 0.0 for allowable stress design

EC8

Vertical component of seismic action

The vertical component of the seismic action is considered in clause 4.3.3.5.2 which states:

> *(l) If a_{vg} is greater than 0.25 g (2.5 m/s^2) the vertical component of the seismic action, as defined in 3.2.2.3, should be taken into account in the cases listed below:*
>
> - *for horizontal or nearly horizontal structural members spanning 20m or more;*
> - *for horizontal or nearly horizontal cantilever components longer than 5m;*
> - *for horizontal or nearly horizontal pre-stressed components;*
> - *for beams supporting columns;*
> - *in base-isolated structures.*

The above cases are **not** dealt with explicitly in Tekla Structural Designer.

IS1893 (Part 1)

Design vertical earthquake effects

Design vertical earthquake effects are considered in IS1893 (Part 1) : 2016 section 6.3.3.

In amendment no.2 November 2020, clause 6.3.3.1 b) states:

> *Effects due to vertical earthquake shaking shall be considered for the safety of buildings, embankments, bridges and dams or their components, including in the following cases:*
>
> *a) structure located in Seismic Zone IV or V,*
>
> *b) structure has vertical or plan irregularities,*
>
> *c) structure is rested on soft soil,*
>
> *d) structures with long spans, which cause amplification of oscillations due to their flexibility,*
>
> *e) structure has large overhangs of structural members or sub-systems,*

f) prestressed structures, and

g) structures with prestressed members.

When both horizontal and vertical seismic forces are to be taken into account, section 6.3.4 then goes on to describe how earthquake load components may be combined to account for three directional earthquake ground shaking.

Clause 6.3.4.1 states:

> *When responses from the three earthquake components are to be considered, the responses due to each component may be combined using the assumption that when the maximum response from one component occurs, the responses from the other two components are 30 percent each of their maximum. All possible combinations of three components (ELX, ELY and ELZ) including variations in sign (plus or minus) shall be considered. Thus, the structure should be designed for the following sets of combinations of earthquake load effects:*
>
> *1. +/- ELX +/- 0.3 ELY +/- 0.3 ELZ*
>
> *2. +/- ELY +/- 0.3 ELZ +/- 0.3 ELX and*
>
> *3. +/- ELZ +/- 0.3 ELX +/- 0.3 ELY*
>
> *where X and Y are orthogonal plan directions and Z vertical direction.*

Currently neither section 6.3.3 or 6.3.4 are automatically catered for in Tekla Structural Designer, the user would therefore have to consider these manually if required.

Vertical spectra

Clause 6.4.6 states:

> *The design seismic acceleration spectral value A_v shall be taken as: (2/3 x Z/2) x 2.5 / (R / I) for buildings*

The above clause is **not** currently considered in Tekla Structural Designer.

Torsion

When a structure's center of mass at a level does not align with the position of the center of rigidity then torsion is introduced in the structure at that level when an earthquake excites the structure. To account for this, there are three types of torsion potentially applied to levels with rigid or semi-rigid diaphragms during a seismic analysis:

- Inherent torsion - in a 3D analysis when the center of mass and center of rigidity at a level do not align, this is taken account of automatically
- Accidental torsion - to allow for the uncertainty in position of loads in a structure, an additional eccentricity of usually 5% of the structure's width in

all relevant directions - this is accounted for with a torsion loadcase in the analysis

- Amplified accidental torsion - structures with certain SDCs and certain horizontal irregularities require an amplified accidental torsion to allow for extra effects. For this to be calculated the following conditions must *all* apply:
  - Seismic Code is ASCE 7
  - SDC is C, D, E or F
  - Either Plan irregularities 1a or 1b apply

  Assuming the above conditions all exist, calculations are performed to determine if amplified accidental torsion is required. As part of these calculations the following are determined at each relevant level from the deflections of the seismic lateral and torsion loadcases (in each permutation of positive and negative directions):
  - δ_{max} = maximum horizontal displacement for all column/wall nodes on that level
  - δ_{min} = minimum horizontal displacement for all column/wall nodes on that level*
  - $\delta_{avg} = (\delta_{max} + \delta_{min})/2$
  - $A_x = \min(3.0, \max(A_x, (\delta_{max} / (1.2 \times \delta_{avg}))^2))$ where A_x is the value for this level for the torsional loadcase

  NOTE * The δ_{min} value currently output in the Seismic Design Report is NOT the minimum value used in the above calculation for δ_{avg}. (It is instead the numerical minimum for all permutations, i.e. the largest negative value).

  At the end of these calculations, for each relevant level with $A_x > 1.0$ the accidental torsion loads on each level are multiplied by A_x only when the ratio $d_{max} / d_{avg} > 1.2$ or 1.4.

  The analysis results are automatically updated if the accidental torsion loads are amplified.

  When (and only when) the accidental torsion loads have been amplified, this is confirmed by the output of additional tables in the Seismic Design Report which give the displacement and calculated A_x values for each direction and level.

TIP Torsional irregularity and amplification must always be checked for. A suggested method for this is presented here: Workflow to check for torsional irregularity and amplification (page 549)

Modal Analysis

Using the Base Shear Combination, a modal analysis is run for two purposes:

- the natural frequencies of the building in two directions are determined to assist with the calculation of the seismic base shear that in turn is used to determine the distribution of applied loads up the building for an ELF analysis
- the frequencies and mode shapes of the building are determined that need to be included in an RSA analysis so that typically 90% of the mass in the building is mobilized during the RSA analysis

% of Gravity Load Method

The % of gravity load method is used as a means of a gross approximation of the earthquake and is only used in situations where seismic effects are considered to be low. This loadcase is combined with the relevant load factor with other gravity and lateral loadcases to determine the design forces and moments to be considered in conventional design.

NOTE This method is not applicable when working to Eurocodes.

Equivalent Lateral Force Method

The ELF method assumes that the first mode shape is the predominant response of the structure to the earthquake.

Based on the natural frequency and the Base Shear Combination, a total base shear on the structure is determined and this is then set up as a series of forces up the structure at each level (in the shape of an inverted triangle) and these deflect the structure in an approximation to the shape of the first mode.

The resulting seismic loadcases are combined with the correct combination factors with the other gravity and lateral loadcases in the seismic combinations to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

Response Spectrum Analysis Method

The RSA method uses a set of vibration modes that together ensure that the mass participation is typically 90% in the structure in a particular direction.

The response of the structure is the combination of many modes that correspond to the "harmonics". For each mode, a response is read from the design spectrum, based on the modal frequency and the modal mass, and

they are then combined to provide an estimate of the total response of the structure.

Combination methods include the following:

- **Square root of Summation of Square (SRSS)**
- **Complete Quadratic Combination (CQC)** - a method that is an improvement on SRSS for closely spaced modes

As a result of the combination methods (SRSS and CQC), the resulting seismic loadcases are without sign and so they are applied with the correct combination factors both + and - around the "static" results of the other gravity and lateral loadcases in the seismic combinations to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

Response Spectrum Analysis Processes

RSA Seismic Analysis (1st or 2nd order) is run as a stand-alone analysis from the **Analyze** ribbon tab, or as part of one of the **Design (RSA)** processes available from the **Design** ribbon tab. In the latter case the use of 1st order or 2nd order is set for the static analysis is set via **Design Settings > Analysis**.

The RSA seismic analysis process consists of the following steps:

Step	Process	Description
1	Model Validation	Run to detect any design issues which might exist. This is similar to standard model validation but also checks: • Base Shear Combination must exist • At least one RSA Seismic Combination must exist including at least one RSA Seismic Loadcase.
2	Modal Analysis	A 1st order modal analysis for the Seismic Inertia Combination only, which returns the standard results for that analysis type, but also the fundamental periods for directions 1 & 2.
3	Pre-Analysis for Seismic	Performs calculations for RSA Torsion Loadcases. The seismic weight and seismic torque are both calculated at this stage.
4	Static Analysis	1st Order Linear or 2nd Order Linear analysis is performed for all RSA Seismic Combinations and all their relevant loadcases, i.e. this includes Static Loadcases, but does not include RSA Seismic and RSA Torsion Loadcases.

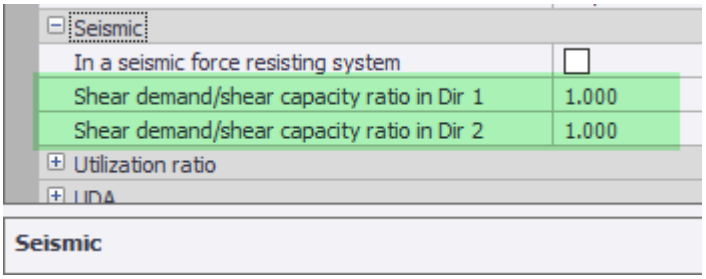
Step	Process	Description
5	RSA Analysis	A set of results is generated for a sub-set of vibration modes for each RSA Seismic Loadcase.
6	Accidental Torsion Analysis	Analysis of any RSA Torsion Loadcases that exist.

NOTE If running one of the **Design (RSA)** processes, an extra member design step is added to the end of the above list.

Seismic Drift

Seismic drift is assessed on a floor to floor horizontal deflection basis and there are limits for acceptability of a structure.

When working to the ASCE 7 code, the engineer can directly define the shear demand / capacity ratio (beta) for columns and walls. The default value of 1.0 could be over-conservative. This option is located in the “Seismic” group of member properties, and can be set separately for 'Direction 1' and 'Direction 2' and for each stack/panel, as shown in the picture below.



- 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.

The building's overall seismic drift status is displayed in the Design branch of the Status Tree in the Project Workspace.

Full details for all columns are presented in a table accessed from the Review ribbon.

These details can be included in printed output by adding the Analysis>Seismic Drift chapter to your model report.

Design Coefficients and Factors (ASCE 7/UBC)

Typically three factors are determined based on the lateral force resisting systems in the structure and which account approximately for the inelastic

response that occurs during the earthquake which is not accounted for directly in the analysis.

- The response modification coefficient which affects the seismic base shear.
- The overstrength factor accounts for the reinforcement steel yielding overstrength and is utilized in the concrete beam and column capacity calculations.
- The deflection amplification factor which is used in the calculation of seismic drift.

Workflow to check for torsional irregularity and amplification

When using the seismic code ASCE 7, torsional irregularity and amplification must always be checked for. Initially it is unknown if torsional irregularity 1a or 1b should be applied, but the only way to determine this is to actually apply them and look at the results. A suggested workflow for proceeding is outlined below:

1. Initially the SFRS is not sized/ designed so you actually have little idea how laterally stiff the structure is hence if 1a or 1b exist, and there is little point checking for this and amplifying torsion at this stage.
2. Hence we recommend you conduct an initial full lateral and seismic design using ELF with 1a/1b OFF (assuming other irregularities don't exist that mandate RSA).
3. SFRS members will be auto designed for the seismic forces to regular code requirements and checked to special seismic.
4. You may then need to manually increase sections sizes to pass special seismic provisions AND seismic drift.
5. This process may require some iteration since the lateral stiffness may increase the seismic loads, since these are a function of this.
6. ONLY when the SFRS/drift checks are all passing at this stage, you can run the seismic wizard again and activate 1a (not 1b) which will also mandate RSA.
7. You can then run a check-design (or just run Analyze All (RSA)) and see if the torsion amplification occurs - the additional amplified torsion result tables will be included in the Seismic design report if it does.
8. If there is no torsion amplification - i.e. no extra tables for this in the seismic design report - the design is complete at this stage. You have checked for torsional irregularity and there is none. You could therefore revert to using ELF at this stage for final output.
9. If there is, size any members of the SFRS that now fail with amplified torsion if this is the case. This will increase the stiffness which may affect seismic loads and drift/torsion checks.

10. Run Analyze All (RSA) again when this has been done.
11. From the Seismic Design Report torsion tables calculate the ratio d_{max}/ d_{avg} for each level to check if the irregularity is 1b (extreme).
12. If the irregularity is 1b (extreme) then run the seismic wizard again and enable 1b. The Wizard will then check if RSA analysis is still allowed for the SDC. Complete the Wizard and check design if so.
13. Iterate as necessary.

Alternatively to the above process, more experienced users might want to go straight to using the RSA method for the pre-design stage (though we would still recommend doing a full gravity and lateral design for regular drift/ wind before attempting seismic design). Note that RSA is generally acknowledged to be less conservative than ELF. So doing an ELF pre-design might actually result in section sizes that are over-conservative (in relation to what RSA might achieve).

8.2 Limitations of Seismic Design

The following limitations apply:

- Where seismic design and detailing is required this is only supported in Tekla Structural Designer for the ACI/AISC and Indian regional codes.
- It is up to the user to assess whether framing is split horizontally or vertically, whether system specific requirements need to be assessed - like mixed system moment frames, whether diaphragms are rigid or flexible - in all instances, the user will need to make the necessary adjustments for the situation in hand. The software does not handle these situations automatically.
- Linear modal analysis with non-linear element properties - currently the modal analysis is limited to a linear model so all non-linear elements are set to be linear.
- ELF can be run as 1st or 2nd order analyses, however if the Fundamental Period is determined using modal analysis the modal analysis is always run as 1st order.
- The RSA analysis itself is a 1st order linear process. For the 2nd Order RSA Seismic analysis, the peak responses are enveloped around the static results for 2nd Order Linear Analysis. Thus when the analysis is set to 2nd order in Design Options, in real terms the results are actually RSA Seismic + 2nd Order.
- Non-linear models (i.e. those containing any non-linear elements) are beyond the scope of the RSA method, so only the ELF method can be used for these.

- We do not consider any of the standard methods for structurally accommodating seismic actions - e.g. base isolators, damping systems
- We do not consider more accurate methods of analysis like time history analysis. As a result there are some situations with very tall buildings and very irregular buildings that Tekla Structural Designer does not cater for.
- Diaphragms - rigid and semi-rigid diaphragms (meshed floors) are available and it is the user's responsibility to ensure they are modelled suitably. Rigid diaphragms are only allowed in limited circumstances and, so called, 'flexible diaphragms' can be modelled as semi-rigid diaphragms with extremely low stiffness. Force transfer into and out of the diaphragm is not checked.
- Collector elements - no checks included.
- Non-structural elements - no checks included.

Specific limitations of steel seismic design

- Coincident V & A braces giving X type are beyond scope
- Various other requirements not checked
  - e.g. V & A braces are restrained at their intersection
  - e.g. tension braces resist between 30% and 70% of total horizontal force
  - e.g. forces in restraining members not checked
- Connections are not designed
- Potentially unconservative results can be obtained for non-linear models - which requires the use of non-linear analysis and the ELF Analysis Procedure. For further details see product bulletin PBTSD-2101-1

A specific limitation of RC seismic design

- While checks can be done in both directions they are direction specific where applicable - there are no biaxial checks.

8.3 Seismic force resisting systems

Available SFRS types

The seismic design requirements for a particular member are based upon which Seismic Force Resisting System (SFRS) the member forms part. Hence, in Tekla Structural Designer you can set appropriate members as part of one of the following systems:

SFRS types included for steel members

Moment Frame Systems

- Special Moment Frames (SMF)
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Braced Frame and Shear Wall Systems

- Ordinary Concentrically Braced Frames (OCBF)
- Special Concentrically Braced Frames (SCBF)
- Special Concentrically Braced Frames (SCBF)

Other Seismic Frame Type

- Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types available for concrete members

Moment Frame Systems

- Special Moment Frames (SMF) - limited design.
- Intermediate Moment Frames (IMF)
- Ordinary Moment Frames (OMF)

Walls

- Special reinforced concrete structural wall - limited design
- Intermediate precast structural wall - no seismic design performed
- Ordinary reinforced concrete structural wall

Other Seismic Frame Type

- Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types excluded

Everything else

- e.g. Eccentrically Braced Frames

Members allowed in the SFRS

The following member types are allowed to be part of a SFRS in Tekla Structural Designer

- Steel columns
- Steel beams
- Steel braces
- Concrete columns
- Concrete beams

- Concrete walls

The following member types are not allowed to be part of a SFRS in Tekla Structural Designer

- Any timber, cold-formed, general
- All other “Characteristics”, e.g. steel joists, truss members, purlins
- Composite members
- Plated, Westok, Fabsec, concrete filled, concrete encased – selectable but no design (i.e. only rolled)

Assigning members to the SFRS

The choice of members to be part of a SFRS is entirely the engineer’s responsibility.

- It is expected that all members in a frame are assigned to the SFRS.
- The assigned members should be specified to act in building Direction 1 or Direction 2

Special Moment Frames - assigning connection types at steel beam ends

For SFRS comprising of steel SMF it is necessary to ensure that the beams fail before the columns. To this end, an assessment of plastic moment capacity is made at each floor. The moment capacity is dependent upon the position of the plastic hinge, typically $(d_{col} + d_{beam})/2$. These locations can be selected appropriate to each beam end either in the beam properties.

Options are provided as follows:

- Plastic hinge position at start
  - Plastic hinge position at end

Either, $(d_{col} + d_{beam})/2$ (default)

or, $d_{col}/2 + L$

L = 0 (default)

Validation of the SFRS

There is only a small amount of validity checking for an SFRS that can be performed automatically; it remains in large part the user’s responsibility to ensure that each SFRS is defined appropriately.

The following validation conditions are however detected:

1. Any A or V brace in a Seismic Force Resisting System must have the A or V as vertically released. A warning is provided in validation if this is not the case.
2. X type bracing is defined as more than one V or inverted V (A) type brace pair on the beam. When more than one A or V brace pair is detected, the additional checks required by AISC 341-05 and AISC 341-10 given in Section 8.3 are out of scope. This situation is not detected during validation, but it is identified in the seismic design, so that the beam is given a “beyond scope” status.
3. The use of K braces is not allowed in AISC 341. An error is provided in validation.
4. Tension only braces were permitted to the 05 version but had no additional requirements. In the 10 version they are only are allowed for OCBF. Thus, an error is provided in validation when a tension only brace is set as part of a SCBF and the code is the 10 version. (The same validation is also applied to compression only braces.)
5. If seismic loadcases are included in combinations and there is not at least one member assigned to each of Direction 1 and Direction 2 then a warning is issued.

Auto design of SFRS members

All SFRS members can be auto-designed to the conventional design requirements both for seismic and non-seismic combinations;

During the automatic design procedure, besides the conventional auto-design, and for seismic combination results only:

- Steel members in the SFRS are checked for seismic provisions;
- Normal weight reinforced concrete members in the SFRS are auto-designed for seismic provisions

8.4 Seismic design methods

To design members and walls for the results of a seismic analysis:

- For geographic regions categorised as “low seismicity” it is acceptable to assume “ductility class low” applies. Under these conditions the seismic analysis results can be fed into “conventional” design, see Seismic analysis and conventional design (page 555)
- Certain conditions (e.g. “high seismicity”) necessitate that a “seismic” design is performed - additional design and detailing requirements have to be applied in this situation, see Seismic analysis and seismic design (page 556)

NOTE The additional design and detailing requirements of “seismic” design are only supported in Tekla Structural Designer for the ACI/AISC and the Indian regional codes.

Seismic analysis and conventional design

ELF seismic analysis and conventional design

The overall modeling, analysis and conventional design process using the ELF method is summarized as follows:

1. Modeling
   - No additional seismic modeling requirements
   - There is no need to assign members to a SFRS
2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required
   b. Set the site class (ASCE 7), or ground type (Eurocode)
   c. Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method
   d. Set up the relevant seismic combinations
3. Static Design

   Run the Design (Static):

   - the results of the ELF seismic combinations are fed into the design and considered in the same way as other combinations.
4. Calculation Output
   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and conventional design

The overall modeling, analysis and conventional design process using the RSA method is summarized as follows:

1. Modeling
   - No additional seismic modeling requirements
   - There is no need to assign members to a SFRS

2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.

   b. Set the site class (ASCE 7), or ground type (Eurocode)

   c. Select RSA method of analysis

   d. Set up the relevant seismic combinations

3. Static Design

   Run the Design (Static):

   - Results of the static combinations are fed into conventional design.

4. RSA Seismic Design

   Run the Design (RSA):

   - Results of the RSA seismic combinations are fed into conventional design and considered in the same way as the static combinations.
   - No additional seismic design is required

5. Calculation Output

   - A Seismic Design Report is available
   - Drift limitations are checked

Seismic analysis and seismic design

NOTE The additional design and detailing requirements of “seismic” design are only supported in Tekla Structural Designer for the ACI/AISC and the Indian regional codes.

These requirements vary depending upon the 'sophistication' of the SFRS. For example OMF have less stringent requirements than SMF.

ELF seismic analysis and seismic design

The overall modeling, analysis and seismic design process using the ELF method is summarized as follows:

1. Modeling

   Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type.

2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.

   b. Set the site class (ASCE 7), or ground type (Eurocode)

   c. Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method.

   d. Set up a modal mass combination

   e. Set up the relevant seismic combinations

3. Static Design

   Run Design (Static) to:

   a. Conventionally design all members for all non-seismic (gravity and lateral) combinations

   b. Conventionally design all members for all seismic combinations in the same way as the other combinations.

   c. Perform additional seismic design for the seismic combinations for those members assigned to a SFRS

4. Calculation Output

   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and seismic design

The overall modeling, analysis and seismic design process using the RSA method is summarized as follows:

1. Modeling

   Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type. These members will be designed and detailed according to the seismic provisions.

2. Loading and Analysis

   Run the Seismic Wizard to:

   a. Determine building height to the highest level and adjust it if required.

   b. Set the site class (ASCE 7), or ground type (Eurocode)

   c. Select RSA method of analysis

   d. Set up a modal mass combination

   e. Set up the relevant seismic RSA combinations

3. Static Design

   Run the Design (Static) to:

   - Conventionally design all members for all non-seismic (gravity and lateral) combinations .

4. Modal Analysis

   At this point it is recommended that you run a 1st order modal analysis in order to confirm the model converges on a solution, (until it is able to do so, it is pointless proceeding with a full RSA Seismic Design).

5. RSA Seismic Design

   Run the Design (RSA) to:

   a. Conventionally design all members for all RSA seismic combinations in the same way as the other combinations.

   b. Perform additional seismic design for the RSA seismic combinations for those members assigned to a SFRS

6. Calculation Output

   - A Seismic Design Report is available
   - Drift limitations are checked

9 Footfall assessment handbook

This handbook contains information relating to the footfall assessment capabilities available within Tekla Structural Designer.

- Introduction (page 559)
- General procedure (page 560)
- Limitations and assumptions (page 570)
- Basic workflow - SCI P354 (page 572)
- Basic workflow - CCIP-016 (page 596)
- Basic workflow - AISC DG11 (page 631)
- Footfall assessment settings (page 683)

Footfall assessment in Tekla Structural Designer

9.1 Footfall assessment - introduction

The current architectural trends tend to produce more sleek and slender designs, contrasting with heavy structures of the past. This is possible thanks to increased design efficiency, allowing the utilization of stronger materials in design considerations. Consequently, this results in lighter structures being built, which is often accompanied by the reduction in structural stiffness and damping. Such structures can struggle to satisfy the vibration serviceability limit state criteria for human occupancy. Therefore, more accurate methods of assessing human induced vibrations on floors are required.

While earlier versions of Tekla Structural Designer included options to run simplified floor vibration checks to either the AISC DG11 or the SCI P354 design guide, the above mentioned need for more accurate methods has resulted in the introduction of a rigorous footfall assessment. This new approach involves the creation and modal analysis of floor analysis models for dedicated footfall mass combinations. The resulting workflow consists of two parts:

1. Prediction of the structural response in terms of acceleration and/or velocity.
2. Footfall checks to evaluate the limiting criteria.

The new approach can be performed to either the SCI P354, CCIP-016 or AISC DG11 design guide. While it offers a comprehensive set of options for advanced users, the workflow it adopts is easy to follow.

NOTE The simplified floor vibration checks to DG11 and SCI P354 have both been retained, should you still want to make use of them.

Relevant design guides and codes

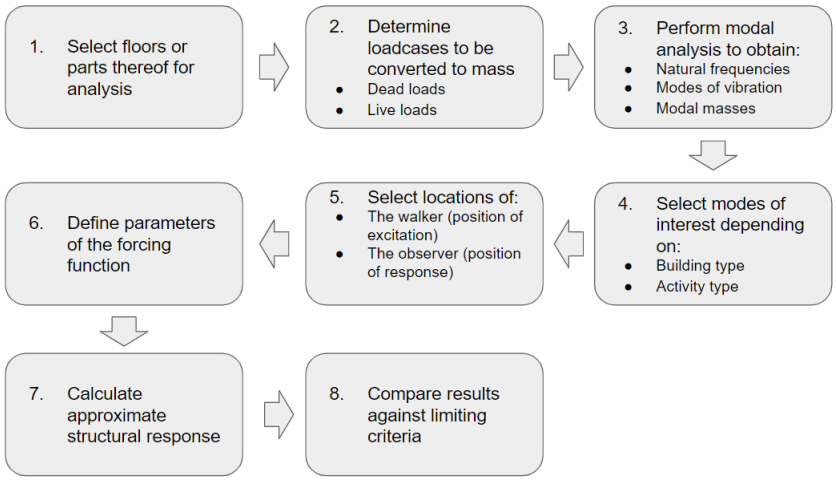
The following design guides and codes provide resources relevant to footfall analysis:

- **SCI P354:** Design of Floors for Vibration: A New Approach by the *Steel Construction Institute* (Ref. 1 (page 698))
- **AISC DG11:** Vibrations of Steel-Framed Structural Systems Due to Human Activity by the *American Institute of Steel Construction* (Ref. 2 (page 698))
- **CCIP-016:** A Design Guide for Footfall Induced Vibration of Structures by the *Concrete Centre* (Ref. 3 (page 698))
- **BS 6472-1:2008:** Guide to evaluation of human exposure to vibration in buildings. Vibration sources other than blasting (Ref. 4 (page 698))
- **ISO 10137:2007:** Bases for design of structures - Serviceability of buildings and walkways against vibrations (Ref. 5 (page 698))

The design guides specify analysis methods, whereas the two codes, BS 6472-1:2008 and ISO 10137:2007, focus on defining the limiting criteria.

9.2 Footfall analysis general procedure

Regardless of the design guide being worked to, the main steps of a footfall analysis are generally the same.



Tekla Structural Designer's ease-of-use methodology simplifies and guides you through the above process and selects most of the analysis parameters automatically, while requiring a minimal amount of user input. However, a comprehensive set of options are available for advanced users, allowing you to override whichever parameters are deemed necessary, to satisfy your design requirements.

See below for design guide recommendations, and for more information about key aspects of the process in Tekla Structural Designer.

Design guide recommendations

The design guides provide a number of recommendations regarding FE modeling, some of which are summarized below:

SCI P354

The SCI P354 design guide (Ref. 1 (page 698)) suggests that the following aspects of the floor analysis model should be considered.

The dynamic Young's modulus of concrete may be taken as 38 GPa and 22 GPa for normal and lightweight concrete, respectively.

All connections should be assumed to be rigid.

Column sections should be provided, and they may either be assumed to be pinned at their theoretical inflexion points, usually mid-height between floors, or fixed at their remote ends.

In some cases non-loadbearing cladding (or partition walls) might be judged to provide vertical restraint (translationally and rotationally free).

The interfaces at core walls should be modeled as fully fixed.

CCIP-016

According to CCIP-016, the presence of full height partitions of (i) masonry, (ii) studwork, and (iii) glass can significantly modify the dynamic properties of floors. As a consequence, if the presence of those elements is required for the floor to serve its purpose, they should be included in the analysis. This is not the case for part-height and movable partitions.

Due to small strains associated with footfall analysis, pinned beam connections should be converted to fixed.

To account for the additional stiffness and restraint provided by facades and concrete core walls, vertical edge restraints should be provided in their locations.

Since columns increase the frequencies of floors, due to their rotational stiffness, the columns both above and below the considered level should be included.

AISC DG11

DG11 recommends that the FAM be limited in its size and indicates that considering a large floor area might be unconservative. For a slab item of interest the FAM should include only those slabs immediately adjacent. So for a regular building a 'middle' slab would create a model of 9 similar slabs including the one of interest. This is entirely at the user's control in Tekla Structural Designer.

Balconies can be included as a whole, but it's recommended to include only the seating sections between several rackers if the relevant vibration modes span over very large areas.

In case of staircases and footbridges, flexible supporting elements should be included, as their flexibility is most likely to provide additional damping and decrease the natural frequencies.

When modeling a floor with composite action, the greater stiffness of concrete on metal deck under dynamic loading needs to be accounted for by increasing the concrete's Young's modulus by a factor of 1.35.

To predict natural frequencies of slabs more accurately, the orthotropic material properties should be defined, with the flexural stiffness parallel to deck ribs being between two to five times higher than the stiffness perpendicular to the ribs.

Rolled beam and girder end connections should be considered as continuous, even if they are designed and detailed as shear connections.

Floor analysis model preparation

Floor analysis models (FAMs) can be created for any floor or part thereof that are occupied, (thus roofs or floors completely devoted to storage can be

ignored). In Tekla Structural Designer multiple floor analysis models can be created at each floor level if required.

Each FAM is created from the main structural model. It is typically composed of the beams and slabs of a floor and the single column/ wall stacks directly above and below it.

Analysis Model Options (page 687) and Concrete Modification Factors (page 688) are automatically applied to the FAM which can be changed at any time by selecting 'Settings' on the Footfall Analysis ribbon.

Slabs

For 2-way spanning slabs the slab mesh is generated using 2D elements in accordance with the meshing controls specified in the FAM Properties.

To have their stiffness included in the analysis, each of the different 1-way spanning slab types can optionally be meshed using orthotropic 2D elements.

If 1-way composite slabs are meshed, the orthotropic properties can either be auto calculated based on their thickness, (using an X factor of 1.000 and a Y factor of 0.010), or user defined properties can be specified. To use the latter option you would also need to select 'Analyze slab' in the composite slab item properties in order to define the analysis parameters required.

For those slabs that are unmeshed you can specify the proportion of their self weight mass to be included.

Each of the above choices are set via Analysis Model Options (page 687)

Column stacks and wall panels

The column stacks are modeled as either full height with fixed supports, or half height with pinned supports.

Walls are modeled either with fixed supports, or modeled supports.

The releases of members attached to the columns and walls can either be fixed or the actual releases defined in the model can be used.

Each of the above choices are set via Analysis Model Options (page 687)

Beams

The supporting beams are subdivided into 1D elements connected to the slab 2D element mesh to produce a well discretized analysis model of both beams and slabs.

A ‘Max node spacing’ parameter in Analysis Model Options (page 687) ensures that beam lengths with no meshed slab attached are still subdivided to some degree.

While any non-composite steel beams included in the FAM are by default treated as non-composite, the general expectation is that any steel beam with a minimum nominal composite behavior should be considered as composite for footfall analysis, even if for design purposes it's still a non-composite beam. To cater for this a 'Treat as composite' option can be set in the 'Floor

vibration' section of the beam properties. Enabling this option allows the concrete flange width above the beam to be calculated taking into account the adjacent composite slab(s) in the same way as it would be for a beam that is defined as composite.

Facade restraints

The restraining effect provided by building cladding can be taken into account by selecting 'Generate Facade restraints' in the FAM Properties. When selected, the solver nodes along the perimeter of the FAM are fixed in their Fz degree of freedom.

Once facade restraints have been generated you can then add or remove them from individual beams and slab edges as required.

Spandrel beams (AISC DG11 only)

Spandrel beams (beams that support the facade) are usually significantly restrained by cladding.

In accordance with DG11 any beams identified as spandrel beams have a multiplier applied to their major axis stiffness for the footfall modal analysis.

They are taken into account by selecting 'Generate Spandrel Beams' in the FAM Properties. When selected, the multiplier is applied to any beams along the perimeter of the FAM.

Once spandrel beams have been generated you can then add or remove them from individual beams as required.

The default multiplier is 2.5, as this is the value recommended by DG11. You can edit the multiplier if required via Analysis Model Options (page 687)

Load to mass conversion

To accurately represent the operational mass of the floors that is subsequently used in the modal analysis, it is necessary to convert certain load cases to mass. Tekla Structural Designer automatically does this for all loadcases included in 'Footfall Mass Combinations'.

SCI P354 and CCIP-016

These loadcases normally include unfactored self-weight of the structure with superimposed dead loads such as the weight of the ceilings and services, unless the analysis is for a bare-state structure, then those can be ignored.

If certain load cases are guaranteed to exist in the finished structure, semi-permanent loads can be included as well. This does not apply for dance or aerobic floors.

Although UK National Annex to EN 1990 (Ref. 6 (page 698)) recommends the inclusion of 30% of the imposed loads in SLS verification, SCI P354 and CCIP-016 recommend to either ignore them or apply no more than 10% of the

nominal imposed loads, following the research conducted by Hicks et al.(Ref. 7 (page 698))

Overestimating the mass used in the modal analysis can result in non-conservative structural response.

AISC DG11

Recommendations are given in DG11 Section 3.3.

Expected day-to-day dead and live loads should be estimated and used. Normal mechanical and ceiling installations can be modeled by adding 4 psf to the structural system weight, although those values can be adjusted if needed.

DG11 Table 3.1 specifies live loads that can be used for vibration analysis based on the occupancy type.

Supported partition wall masses can be applied to slab shell nodes along the wall, as line masses assigned to zero-stiffness frame elements in the plane of the slab, or as uniform mass applied to the slab shells.

Modal analysis

It is necessary to perform a modal analysis in order to establish the dynamic properties of the floor analysis model, including natural frequencies, mode shape and modal masses.

The modal analysis is free vibration. Damping is then taken into account in the post processing.

Excitation and response locations

The guides suggest that considering coincident excitation and response at every point is a reasonable and conservative first step - floors that pass on this basis require no further investigation. While the guides suggest that where excitation and response locations are known/predictable it would be possible to refine the results at critical locations, this procedure is beyond the current scope of Tekla Structural Designer.

The SCI P354 design guide suggests that when using the finite element analysis, all points should be considered. This is a conservative, as it does not allow for there being sections of the floor that will never be occupied, and hence not having a walking force applied.

When considering the excitation and response position as coincident, Tekla Structural Designer checks every node in the floor analysis model.

Analysis post-processing - SCI P354 and CCIP-016

The results of the modal analysis - the mode shapes and their modal properties - are then subject to a fully automated post-analysis process to calculate the response (SRSS) accelerations likely to result from excitation of the floor by walking occupants. The walking excitation is defined in terms of a Footfall (pacing) frequency and a number of other parameters such as the walker's weight, length of stride and walking path. Acceleration values are calculated at every node of the slab 2D element mesh.

The response to excitation can be either *Resonant,* for lower frequencies, or *Transient* for higher frequencies, different methods of calculating the response acceleration being applicable to each. Floors are broadly classified as either Low or High frequency by their first (lowest) frequency; Low being 0 to 10 Hz and High > 8Hz.

- The industry recommendation is to calculate both response types when the floor is nominally classed as Low frequency. Hence both Resonant and Transient response accelerations can be automatically calculated from the modal analysis results for all exciting (pacing) frequencies.

From the response accelerations a *Response Factor* (Rf) - which is a measure of human perception of acceleration - is then calculated and compared to a Target value. The design guides define acceptable Target Rf values for different building uses and circumstances, e.g a workshop or residential building during day time etc.

A number of events (per period) is also calculated at each node for a Target Vibration Dose (VDV) assessment.

Damping ratio - SCI P354

Recommended values of the damping ratio are provided if more accurate values cannot be estimated.

The damping ratio is specified in SCI P354 Table 4.1 based on the type of floor finishes:

Critical damping ratios for various floor types according to SCI P354 Table 4.1.

Floor finishes	Damping ratio, ζ [%]
Fully welded steel structures, e.g. staircase	0.5
Completely bare floors or floors where only a small amount of furnishings are present	1.1
Fully fitted out and furnished floors in normal use	3.0
A floor where the designer is confident that partitions will be approximately located to interrupt the relevant mode(s) of vibration, i.e.	4.5

Floor finishes	Damping ratio, ζ [%]
partition lines are perpendicular to the main vibrating elements of the critical mode shape	

Damping ratio - CCIP-016

Most floors and bridges have between 1% and 5% of critical damping. Appendix A of CCIP-016 provides suggested ranges of critical damping based on the type of the structure, see below:

Suggested damping values for different structures taken from Appendix A of CCIP-016

Component	Critical damping ratio, ζn [%]
Welded steel bridges with little or no services, fixtures or fittings	0.3 – 0.5
Bolted steel, composite and prestressed concrete bridges with little or no services, fixtures or fittings	0.5 – 1.0
Reinforced concrete bridges	0.7 – 1.2
Bare steel composite or post tensioned concrete floors with little or no fit out	0.8 – 1.5
Bare reinforced concrete floors	1.0 – 2.0
Completed steel composite or post tensioned concrete floors with low fit out	1.5 – 2.5
Completed steel composite or post tensioned concrete floors with typical fit out	2.0 – 3.0
Completed reinforced concrete floors with typical fit out	2.2 – 3.5
Completed steel composite, post tensioned or reinforced concrete floors with extensive fit out and full height partitions	3.0 – 4.5

Analysis post-processing - AISC DG11

The results of the modal analysis - the mode shapes and their modal properties - are subject to a fully automated post-analysis process to calculate

the response sinusoidal accelerations likely to result from excitation of the floor by walking occupants.

The post-processing uses the following input parameters entered in the FAM Properties:

- walker's weight (default 168 lbs)
- damping ratio (default 3%)
- Fig.2-1 AISC DG11 curve (default Curve A)

Damping ratio - AISC DG11

When working to DG 11, to establish the damping ratio the designer sums contributions from each of the floor components separately.

Recommended component damping values according to AISC DG11

Component	Ratio of actual damping-to-critical damping, ζ [%]
Structural system	1.0
Ceiling and ductwork	1.0
Electronic office fit-out	0.5
Paper office fit-out	1.0
Lightly furnished quiet spaces	0.5
Churches, schools and malls	0.0
Full-height drywall partitions in bay	2.0 – 5.0

Acceptable accelerations (Figure 2-1 of DG11)

This figure depicts three curves for different situations e.g. office floors and pedestrian bridges. In Tekla Structural Designer these have been named Curves A to C where,

- Curve A - offices and residences, quiet areas
- Curve B - indoor pedestrian bridges, shopping malls
- Curve C - outdoor pedestrian bridges

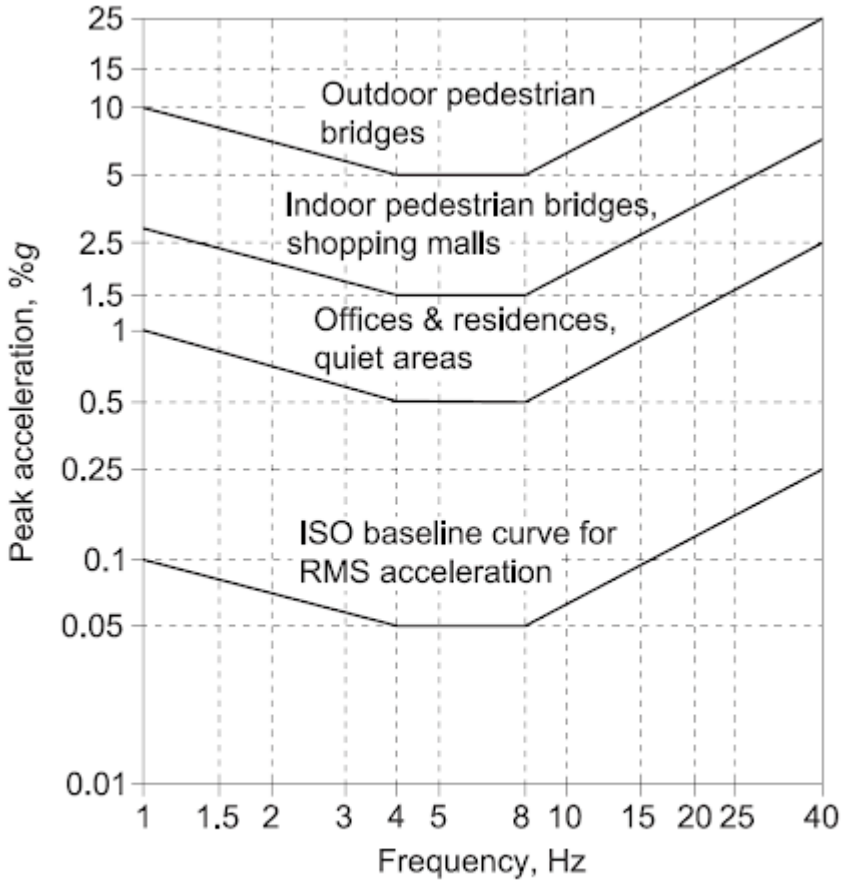
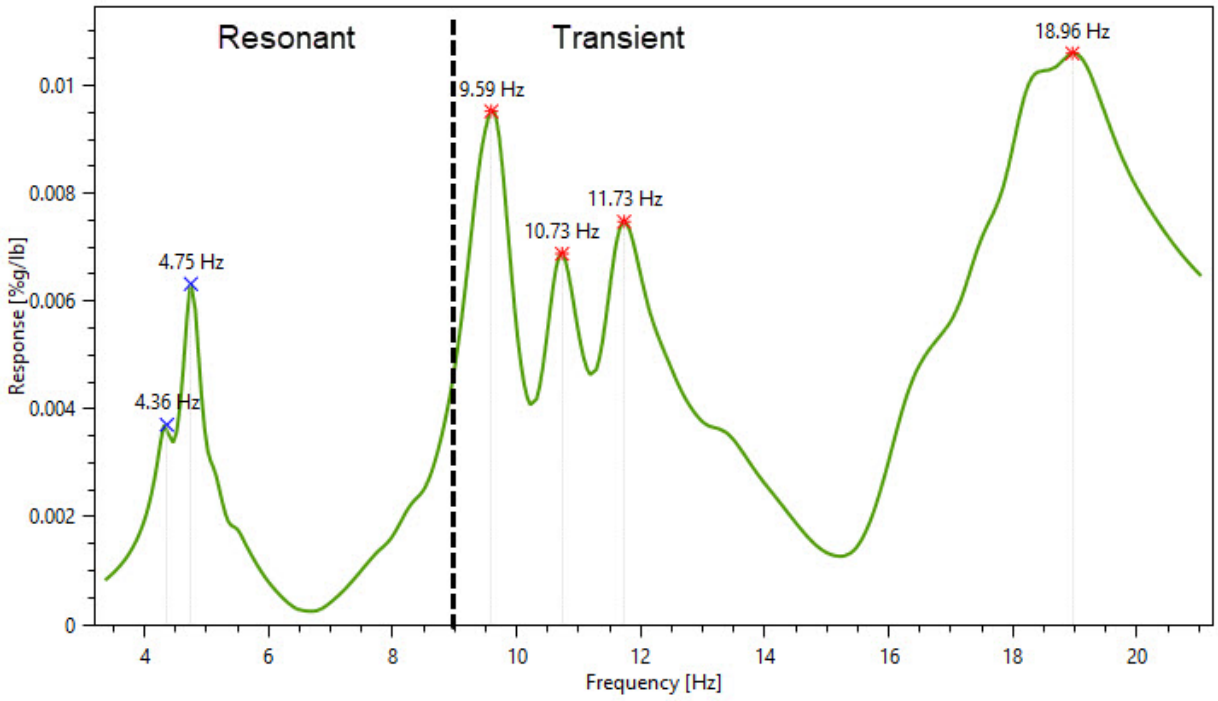


Fig. 2-1. Recommended tolerance limits for human comfort.

Frequency Response Function (FRF)

DG11 uses the Frequency Response Function (FRF) method to identify the worst frequency of the floor. This frequency is called the 'dominant frequency' and has an associated 'magnitude' as a percent of acceleration due to gravity, g.



FRF Frequency

- Resonant Response: FRF for frequencies below 9 Hz
- Transient Response: FRF for frequencies below 20 Hz

The peak accelerations at the dominant frequencies are then compared directly with limiting accelerations obtained from the selected curve in DG11 Figure 2-1 to determine the pass/fail status.

NOTE If the 'dominant frequency' is below 3 Hz, DG 11 suggests that the floor might be susceptible to 'vandal jumping' - deliberate attempt to excite the floor excessively. It recommends an alternative procedure for the FRF using a group rhythmic forcing function. Tekla Structural Designer does not perform this procedure - instead the standard calculations are performed and a warning is issued, "Warning: The dominant frequency is below 3 Hz. The susceptibility to 'vandal jumping' has not been assessed."

9.3 Footfall analysis limitations and assumptions

Tekla Structural Designer can perform footfall assessments to:

- UK practice - using the SCI P354 design guide, or the CCIP-016 design guide.
- US practice - using the AISC DG11 design guide

NOTE If when following UK practice, the same model were to be assessed in Tekla Structural Designer to both the SCI P354 design guide and the CCIP-016 design guide; while the underlying modal analysis results will be identical, there can be significant differences in the footfall check results. This is due to differences in the approaches adopted by the two guides to post process the modal analysis results. For a small number of comparison models it seems that the response calculated to CCIP-016 tends to be lower than when using SCI P354, and that the difference tends to be greater for transient than for resonant response calculations. Two additional points may be worth considering in this regard:

1. The SCI P354 guide includes suggestions that the limits applied can be relaxed in normal situations. Perhaps this more relaxed view should only apply when the response factors are calculated using the SCI guide.
2. The SCI P354 guide permits a VDV assessment for both resonant and transient responses while the CCIP-016 design guide restricts VDV to resonant responses only. Therefore, the higher transient responses determined using the SCI Guide may not be limiting.

Assumptions

- The SCI P354 guidance includes forcing functions due to walking that are either continuous or intermittent. Floor behavior is categorized as low frequency, 0 to 10 Hz, or high frequency, > 8Hz. These are classed as steady state or transient responses respectively and fit neatly with the continuous and intermittent forcing functions. The industry recommendation is to calculate both forcing functions when the floor is nominally classed as low frequency. So both are implemented.
- Excitation and response positions are assumed to be coincident.
- Summing accelerations most commonly uses square root of sum of squares (SRSS) and these are offered in the first release.
- Non-continuous response using techniques such as Vibration Dose Value (VDV) are adopted but for greater ease of use the calculation procedure is ‘inverted’ to provide the number of times the walking path would need to be executed in order to generate an adverse response.
- If precast slabs with reinforced structural topping and with studs to supporting beams are defined, these contribute fully to the modal mass used in the vibration analysis. The mass is determined from the given ‘Weight per units area’ of the pc plank plus a calculated allowance for the topping. The role of primary beams in this situation is unclear as they tend not to have studs but are usually joined with infill concrete. The assumption taken is that their self-weight is included. This is reasonable if you ask for the precast slabs to be meshed in Settings. But if they are not at all composite, as noted in SCI P354 Section 4.1.6, caution is advised.

Limitations

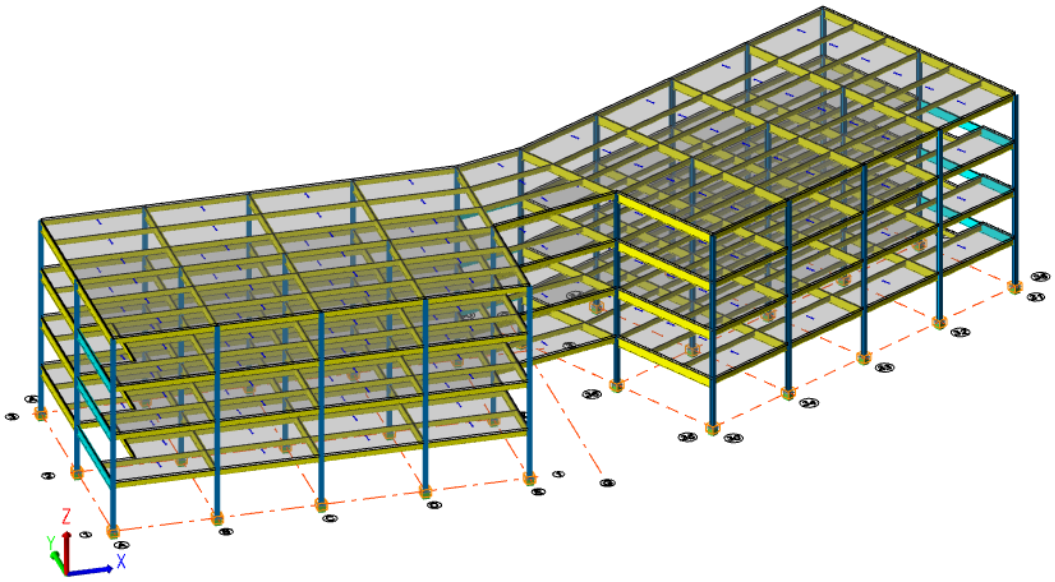
- Steel Plate, timber, and other general material floor areas can be included in the FAM primarily so that the response of the supporting structure can be considered. However, if working to SCI P354, since the guidance is almost entirely based on steel buildings with composite or precast floors, if the option is taken to mesh such slab areas the results should be viewed with caution.
- For Westok beams the self weight is adjusted to allow for the cells but for other beams with web openings the full section is used. Thus for Westok beams, the correct mass is determined for the modal analysis. On the other hand for beams with web openings whilst the full self weight of the unperforated section is used for beam design and this is conservative, for modal analysis this is not conservative. However, given the small proportion of the floor mass that this represents, the degree of unconservatism introduced by using the full self weight is unlikely to be significant.
- If working in the US (ACI/AISC) Region, Steel Joists can be included in the FAM, but will have the analysis properties used in general analysis and fixed ends (or not) as specified in model settings.
- DG11 requires that where transfer or 'hanger' columns are attached to the floor under consideration then the floor above should also be included in the FAM. This is excluded in the current release.
- If tension/compression only braces are included they are treated as linear.
- Trusses are effectively dealt with as fixed lines of support - this is not suggested to be correct or ideal, it simply allows flexibility. For example slab areas one panel remote from the truss can be considered quite reasonably.
- Shear only walls are excluded from the FAM (but this does not make the FAM invalid).
    - Basically these walls are intended only to provide lateral stability, FAMs tend to be inherently laterally stable due to fixed member ends etc.

9.4 Footfall analysis basic workflow - SCI P354

The guidance in SCI P354 (Ref. 1 (page 698)) is almost entirely based on steel buildings with composite or precast floors. However, the footfall assessment process in Tekla Structural Designer can be performed on models of any material with floors of any construction type. Where the structure is not predominantly a steel building with composite or precast floors, it is the engineer's responsibility to determine the applicability of the SCI P354 guide

method, and thus the footfall assessment in Tekla Structural Designer which follows this, to their structure.

To illustrate the basic workflow when working to the SCI P354 design guide, the steel structure with composite floors shown below will be used.



Footfall assessment in Tekla Structural Designer

Download and open the tutorial model

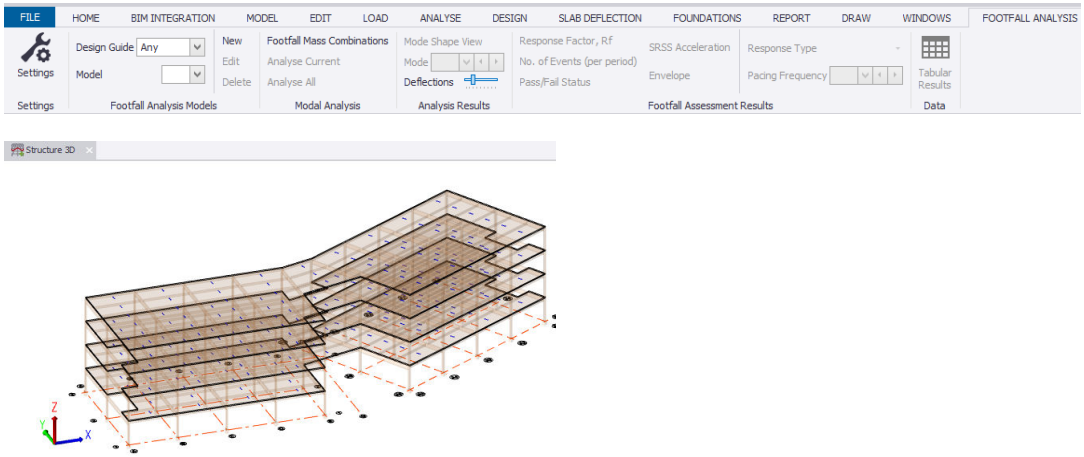
1. Download the tutorial model zip file Footfall - EC UK SCI P354.zip from here.
2. Extract the following tutorial model from the zip:
   - Footfall - EC UK SCI P354.tsmd
3. Open the model in Tekla Structural Designer.

Open an appropriate view

1. Open a **Structure 3D** Scene view.
2. In the **Status bar** at the bottom of the main window, click the **Footfall Analysis View** regime button.



The **Footfall Analysis** ribbon becomes available, and the active view switches to a Footfall analysis view.



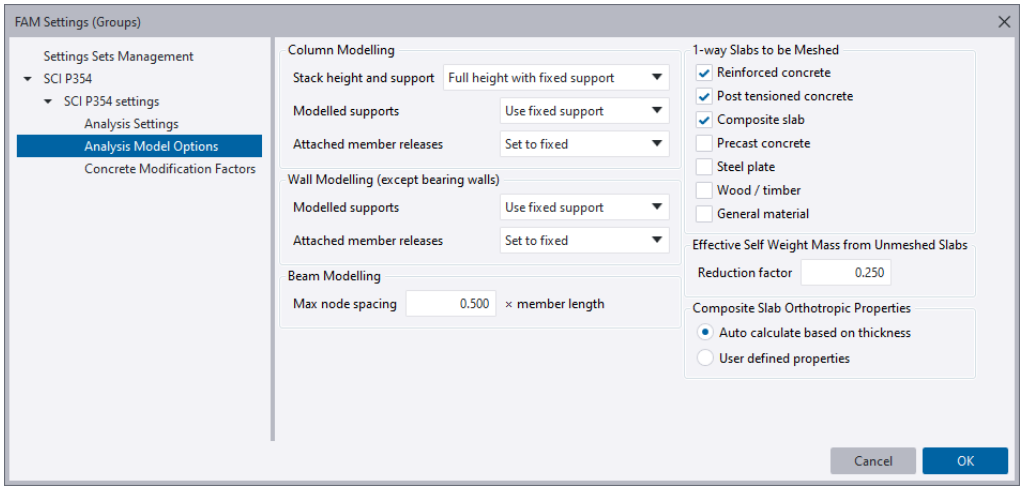
TIP Provided you have slab items displayed in an active Structural view, simply right-click a slab item and select **Create Floor Analysis Model** from the context menu - the Footfall analysis view is opened automatically with the selected slab highlighted and the rest of the floor ghosted.

Confirm the floor analysis model settings

1. On the **Footfall Analysis** ribbon, select **Settings**

   The default settings are displayed. These are used to control the footfall analysis settings, analysis model settings, and modification factors for each available design guide. For more information, see: Footfall assessment settings (page 683)

   Initially there is a single settings set for the SCI P354 design guide, but you can add further sets to allow you to rapidly investigate different setting configurations.

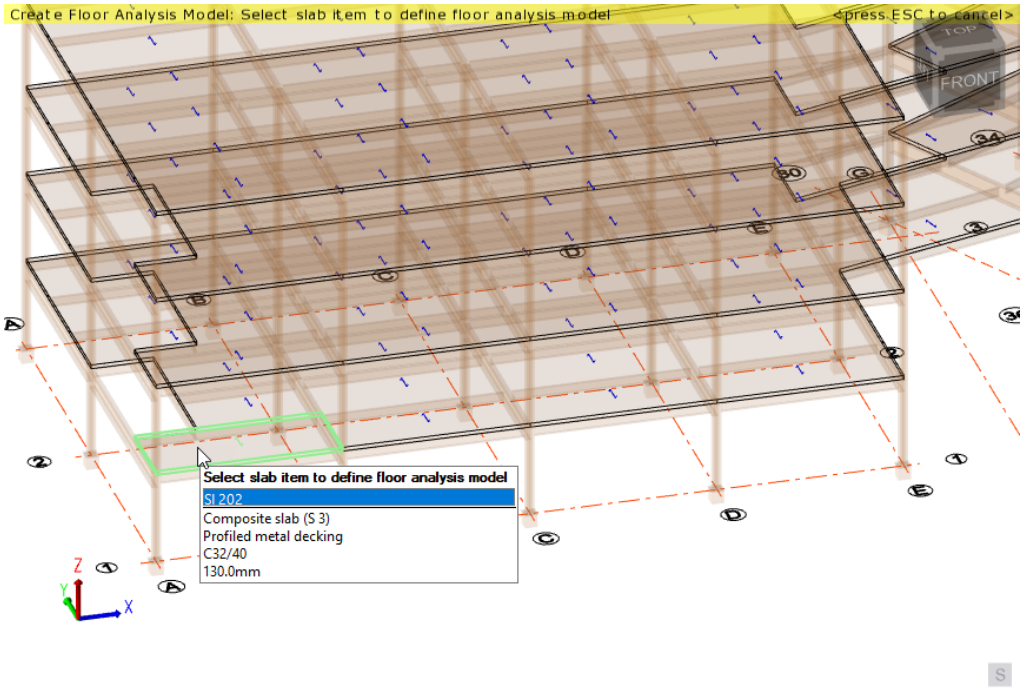


   It is anticipated that most users will not need to adjust these settings from the default set up.

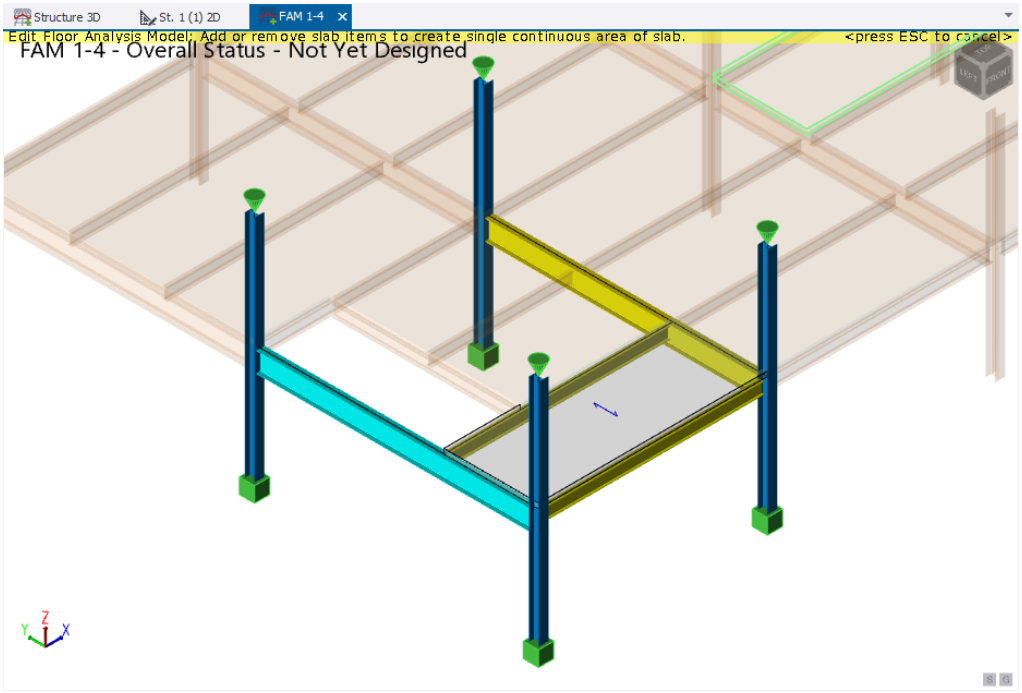
2. Click **Cancel** to exit.

Create a floor analysis model

1. On the **Footfall Analysis** ribbon, select **New**
2. Select the slab item shown below as the first slab to include.



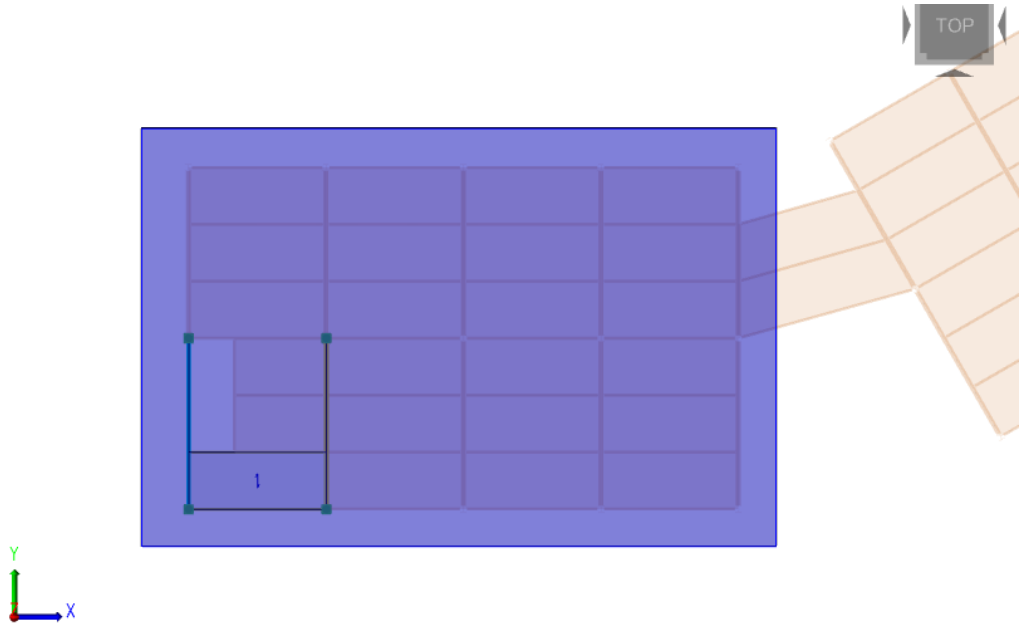
Once the first slab item has been selected a new Floor Analysis Model (FAM) view opens and the are displayed in the **Properties** window.



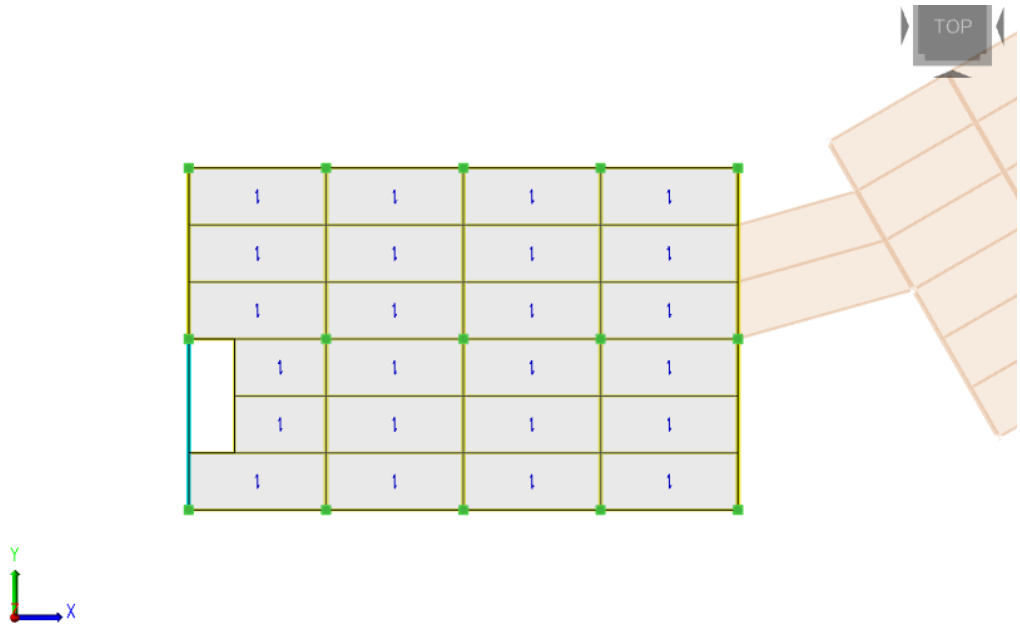
In this view further slab items can then be added to the model, either by clicking them individually, or by box selecting. Note that the **Selection Mode** in the **Properties** window, which defaults to 'Add or Remove' can also be set to 'Add Only' or 'Remove Only' to assist this process.

For this example we want the analysis model to comprise the entire left hand wing of the floor. The quickest way to achieve this is as follows:

3. In the **Properties** window, set the **Selection Mode** to 'Add Only'
4. Change to a **Top** view and then drag a box around the slabs.



The slab items that comprise the analysis model should now be displayed as shown below:

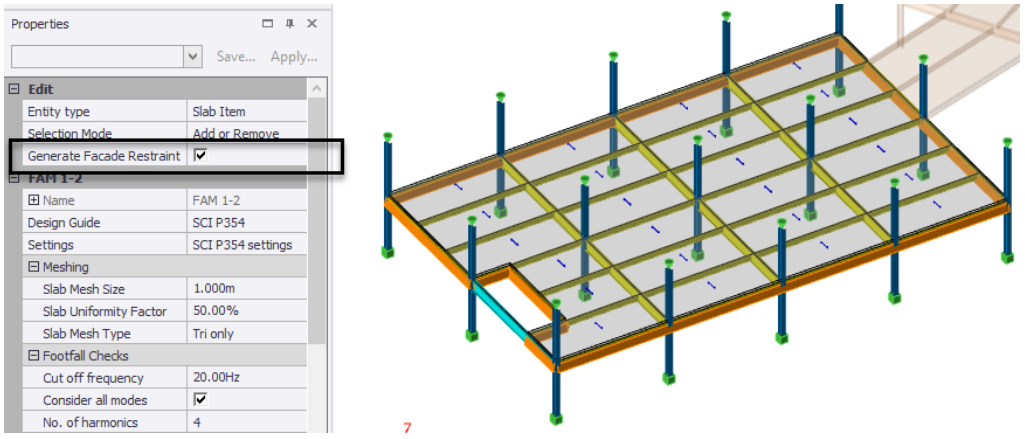


Add and edit facade restraints

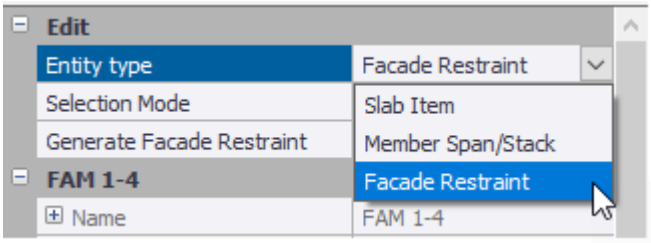
If required, facade restraints can be added very easily.

1. Select **Generate Facade Restraint** in the **Properties** window

   Facade restraints are added around the entire perimeter of the selected slabs.



NOTE By changing the Entity type in the **Properties** window to Facade Restraint, you can then edit the restraints as required - simply clicking on a slab edge will add or remove its facade restraint.

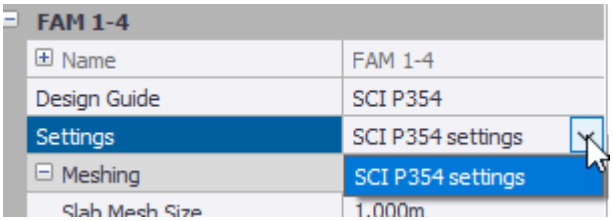


2. Press ESC on the keyboard to exit Edit mode.

Confirm the design guide, settings and meshing

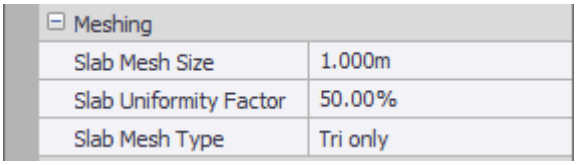
Review the following properties located at the top of the **Properties** window.

1. Ensure the **Design Guide** 'SCI P354' is selected.
2. Ensure the **Settings** set 'SCI P354 settings' is selected.



In a real model you could have created multiple FAM Settings sets, in which case you could then choose which set to apply.

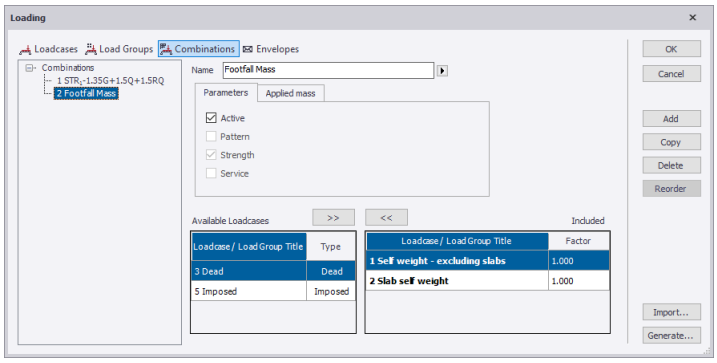
3. Review the meshing properties.



The mesh defaults are reasonable and will be left unchanged.

Create a Footfall mass combination

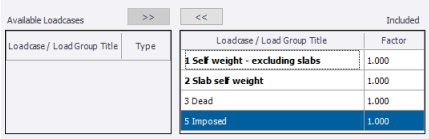
1. On the **Footfall Analysis** ribbon, select **Footfall Mass Combinations**.



   Any existing combinations for static design will be listed in this dialog followed by a **Footfall Mass** combination which is selected as shown above. Two loadcases have automatically been included in this combination.

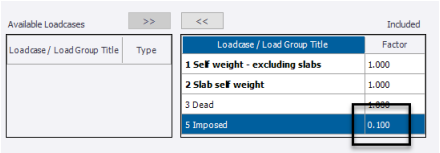
   At this point you complete the Footfall Mass combination by making appropriate selections from the list of available loadcases.

2. Include the Dead and the Imposed loadcases in the Footfall Mass combination.



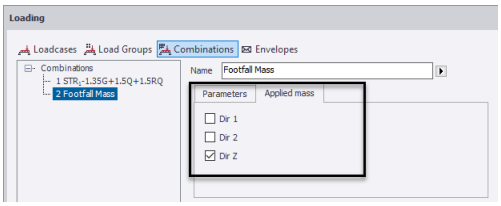
   In real models engineering judgment will be required as the list of available loadcases can potentially be very long.

3. Set the Imposed loadcase factor to 0.100. (SCI P354 suggests 10% for imposed.)



4. Click the **Applied mass** tab in the dialog.

   No changes are required here, but note that for Footfall mass combinations, the applied mass defaults to being applied in the Z direction only.



5. Click **OK** to save the footfall mass combination.

Analyse the model and view the mode shapes

On the **Footfall Analysis** ribbon, there are two analysis choices:

- **Analyse Current** - this runs a first-order modal analysis of **the currently selected FAM** for all active footfall mass combinations.
- **Analyse All** - this runs a first-order modal analysis of **all FAMs** for all active footfall mass combinations.

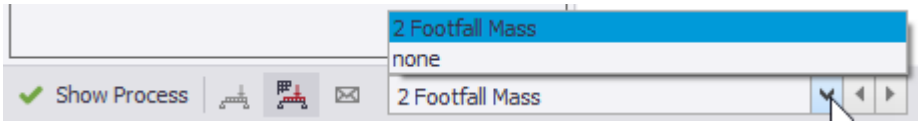
As only one FAM has been created at this stage, it doesn't actually matter which of the above two options is selected.

1. Click **Analyse Current**

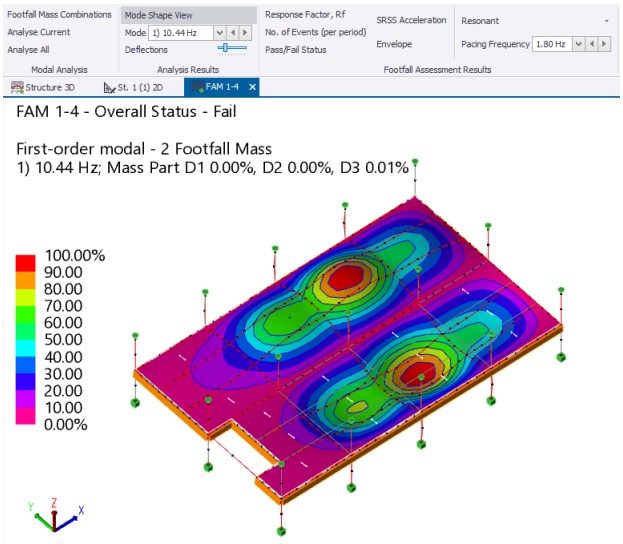
View the mode shapes

To view mode shapes graphically for the Footfall mass combination:

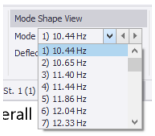
1. Select the **Footfall Mass** combination in the Loading list.



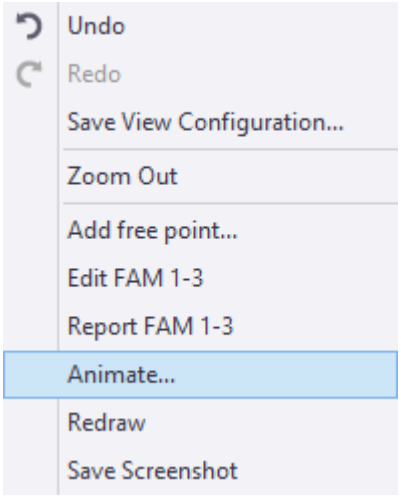
2. On the **Footfall Analysis** ribbon, select **Mode Shape View**.



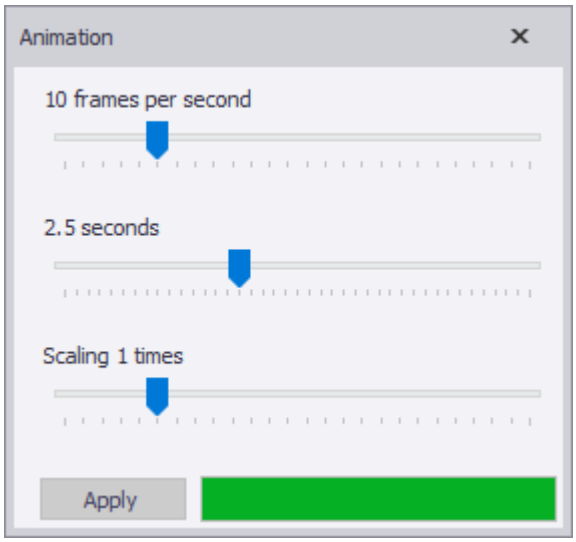
3. To examine subsequent modes, select the required mode from the **Mode** list.



4. To adjust the amplitude of the mode shape, drag the **Deflections** slider in the ribbon as required.
5. To animate the mode shape, right-click in the view, and select **Animate** from the context menu.

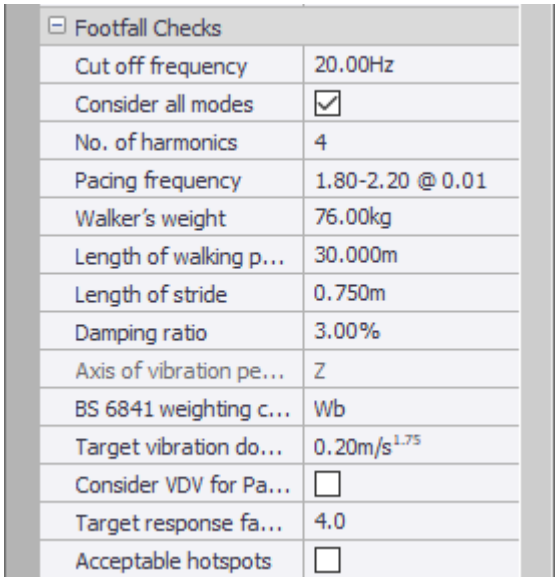


6. To end the animation, close the Animation dialog.



Configure the footfall checks

Default SCI P354 footfall check parameters for all models (predefined on the Footfall Assessment > SCI P354 > Footfall Checks (page 689) page in global settings) are automatically copied into the **Footfall Checks** section of the **Properties** window for each FAM that is created.



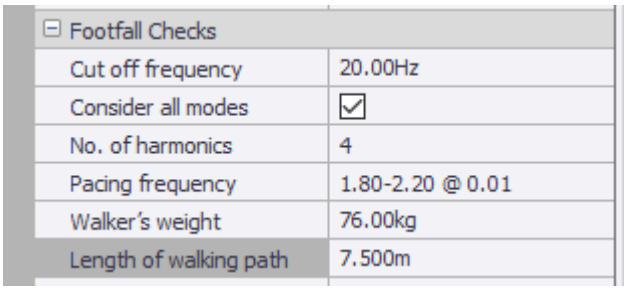
The copied values can then be adjusted as required to suit the individual FAM.

For a detailed explanation of the above parameters, refer the SCI P354 Design Guide.(Ref. 1 (page 698))

1. Take a moment to review the footfall check values in the **Properties** window.

   In this example we shall adjust the Length of walking path as the initial default (**30m**) is too conservative.

2. Set the Length of walking path to **7.5m**.



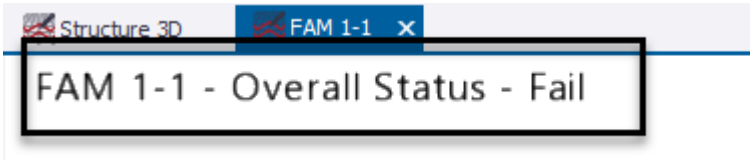
We shall accept all the other default values. Note that the default **Target response factor** is **4.0** so any nodes in the FAM with a greater response will be deemed to have failed.

If you were to change any of the above values, the Footfall Assessment results would update automatically - there is no need to re-perform the analysis.

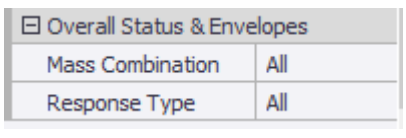
View the footfall check results

Overall status

The overall status is displayed in the top left corner of the FAM view.



Note that the status takes into account any **Mass Combination** or **Response Type** filters applied in the **Overall Status & Envelopes** section of the **Properties** window.

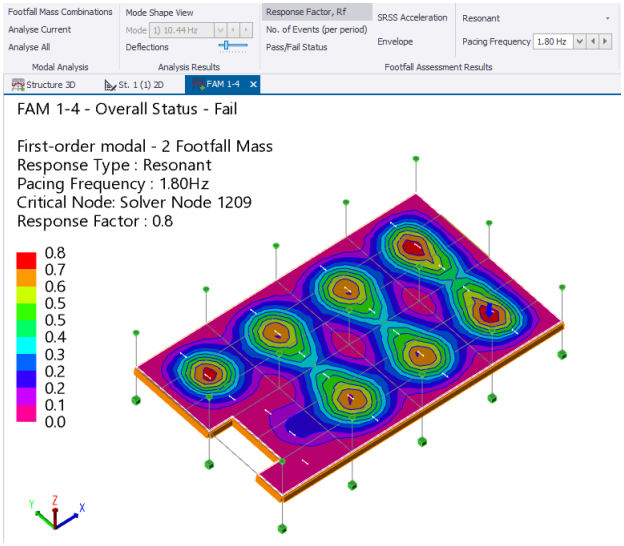


By default, all active footfall combinations are considered, and all (i.e. both resonant and transient) responses are considered.

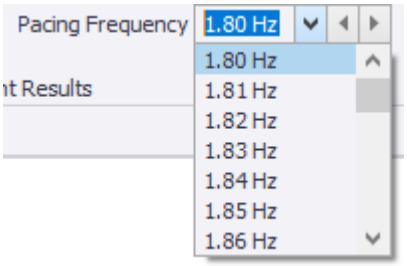
As the overall status is 'Fail', this indicates that the target response factor has been exceeded for the resonant or the transient response at one or more pacing frequency.

View resonant and transient response factors at specific pacing frequencies

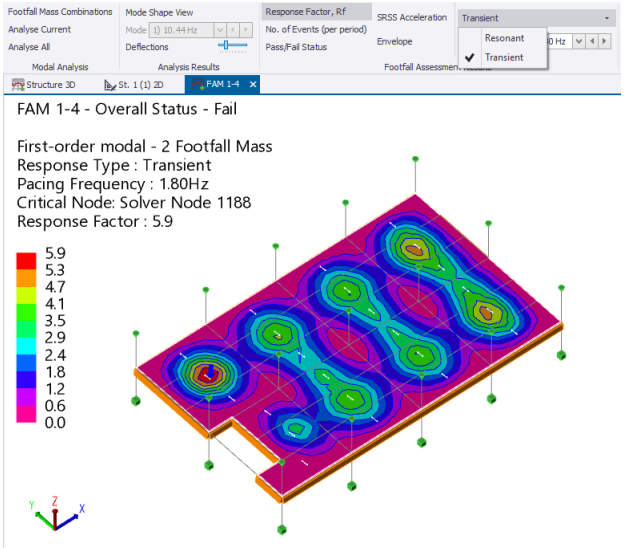
1. On the **Footfall Analysis** ribbon, select **Response Factor, Rf**



Initially the **Resonant** response is displayed for the pacing frequency shown on the ribbon. You can select a different pacing frequency using the drop down option as shown below.

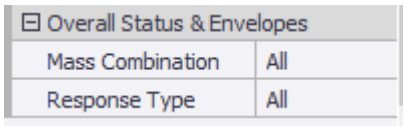


2. To examine the transient response for the selected pacing frequency, select **Transient** on the ribbon.



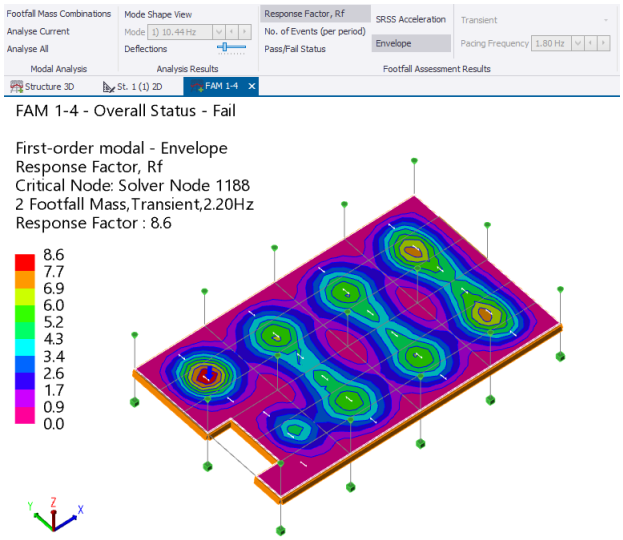
View response factor envelope

Note that envelopes take into account any **Mass Combination** or **Response Type** filters applied in the **Overall Status & Envelopes** section of the **Properties** window.



So by accepting the default filters as shown above, the response factor envelope allows us to locate the most critical response factor of *either type* (resonant or transient), for *all* pacing frequencies and *all* mass combinations (if more than one has been defined).

1. Select **Envelope** on the ribbon.



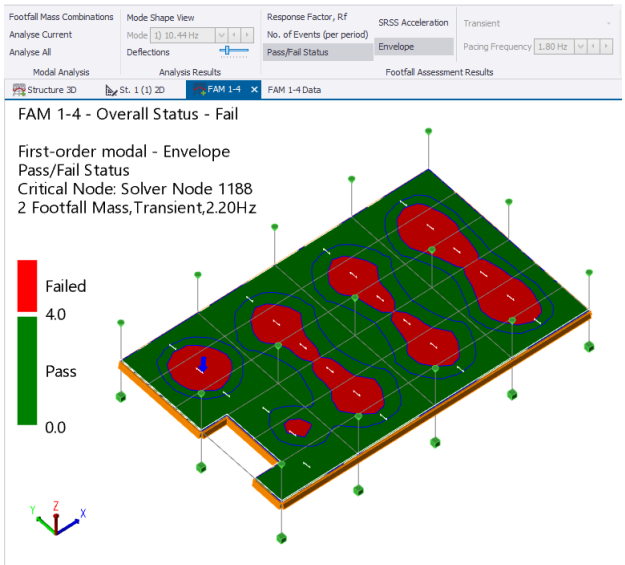
The above envelope reveals that the transient response is critical at a pacing frequency of 2.20Hz with a response factor of 8.6

View Pass/Fail Status

To gain a better understanding of how wide an area is failing, the Pass/Fail status can be examined for the enveloped results.

On the ribbon:

1. Select **Pass/Fail Status**.
2. Select **Pass/Fail Status**



Every node has to be less than the **Target response factor, Rf** to achieve a pass status. Areas where nodal values are greater than the target are shown as failed.

View acceptable hotspots

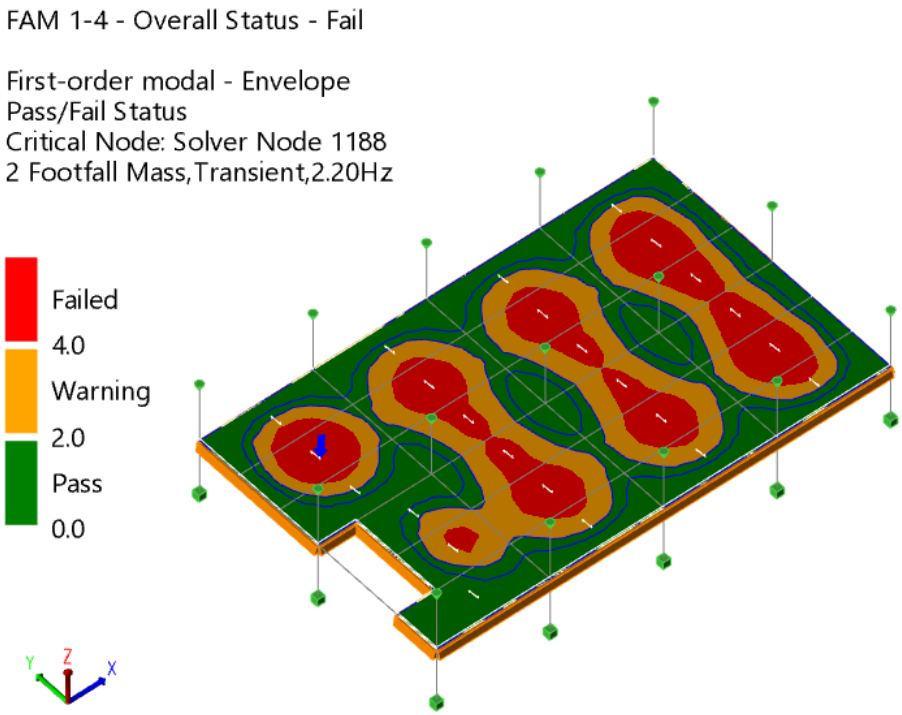
Acceptable hotspots can be used to investigate areas that fall between a general limit and the Target response factor.

1. In the **Properties** window, select **Acceptable hotspots**.

Consider VDV for Pass/Fail	☐
Target response factor, Rf	4.0
Acceptable hotspots	☑
General limit, Rf	2.0

   When acceptable hotspots are activated a **General limit, Rf** setting is added to the properties.

2. Leave the **General limit** as **2.0**



Regions with Rf values between the **General limit** and the **Target response factor** are then given a warning status.

Perform a VDV assessment

If the response factor fails you may be able to make a variable dose value (VDV) assessment instead. VDVs are always calculated, but by default they are not considered when determining the overall status.

For ease of use the VDV calculation procedure is 'inverted' so that the **number of events (per period)** equates to the number of times the walking path would need to be executed in order to generate an adverse response.

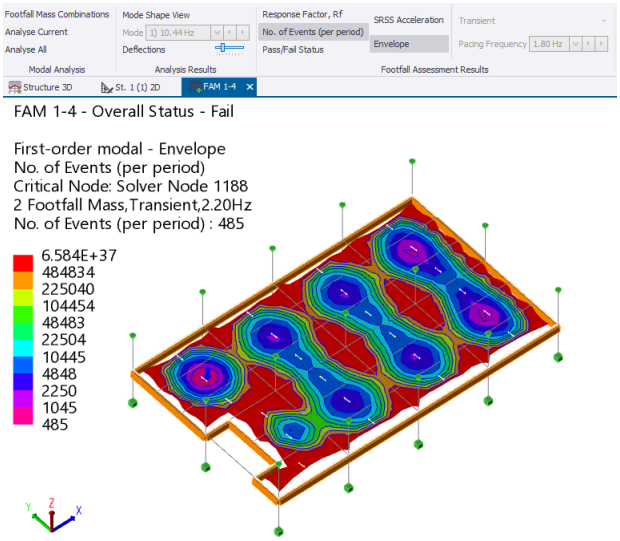
View the number of events at each node

To examine the number of events at the critical node:

1. On the **Footfall Analysis** ribbon, select **No. of Events (per period)**.

   Ensure **Envelope** is selected on the ribbon

   The number of events at each node is displayed on a logarithmic scale. An arrow is used to identify where the critical node occurs.



   In the above display it can be seen that the critical number of events is 485. The arrow identifies the critical node where this occurs.

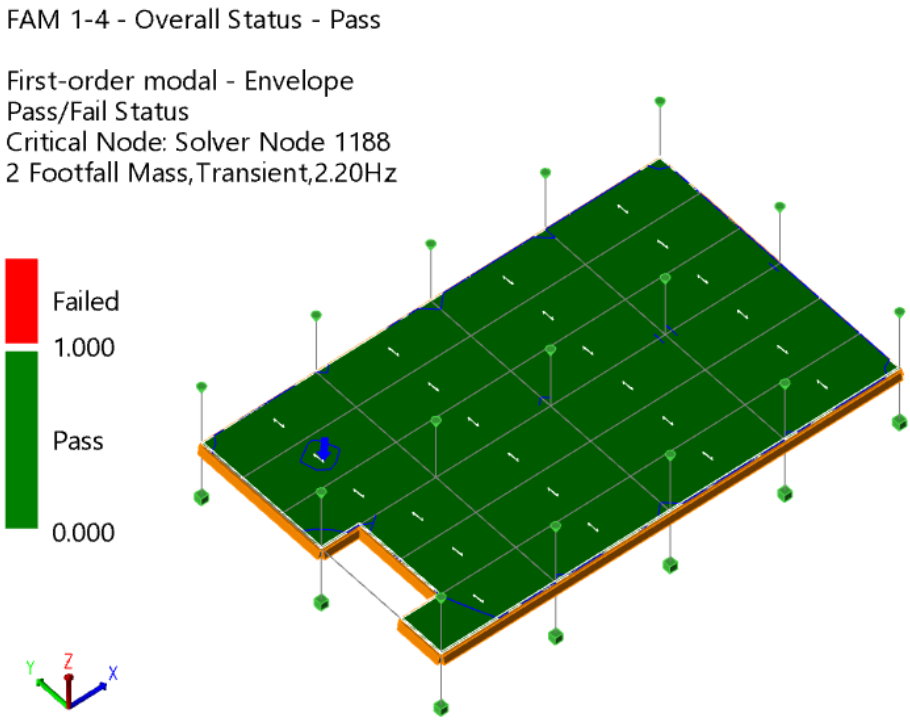
Activate the VDV assessment

When VDV is considered, the **target number of events in a period** that the client will accept has to be less than the **Number of Events (per period)** reported at the critical node.

1. In the **Properties** window, select **Consider VDV for Pass/Fail**.
2. Reduce the target number of events to **400**.

Target vibration dose value (VDV)	0.20m/s$^{1.75}$
Consider VDV for Pass/Fail	☑
Target number of events in a period	400
Target response factor, Rf	4.0

With these settings an overall pass status has been achieved.

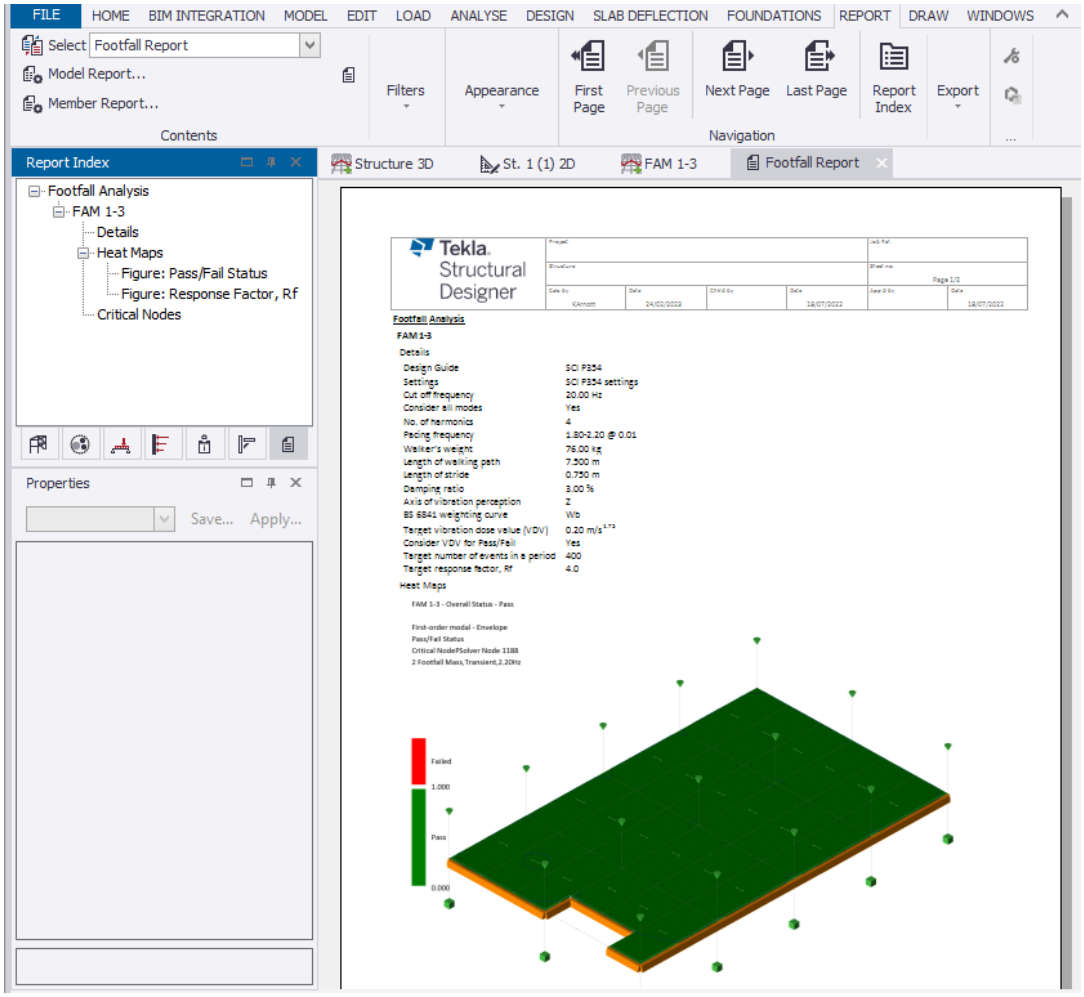


Output the calculations

A **Footfall Analysis** report can be created as follows:

1. On the ribbon, go to the **Report** tab.
2. Click **Model Report**.
3. In the **Report Contents** dialog, click **Add** and then under **Active Style**, enter a **Name** for the report
4. In the list of chapters and options, select **Footfall Analysis** and drag it across to the **Report Structure** area.
5. Click **OK**
6. On the ribbon, in the list on the left side, select the report name you just created.

7. Click **Show Report**.



Additional display and review options

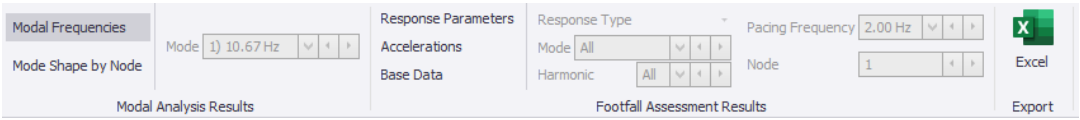
Several tabular data views are available, each of which can be exported to Excel.

To display the **Floor Analysis Model Data** ribbon from where the tabular data views can be accessed:

1. Make the Floor Analysis Model (FAM) view active once more.

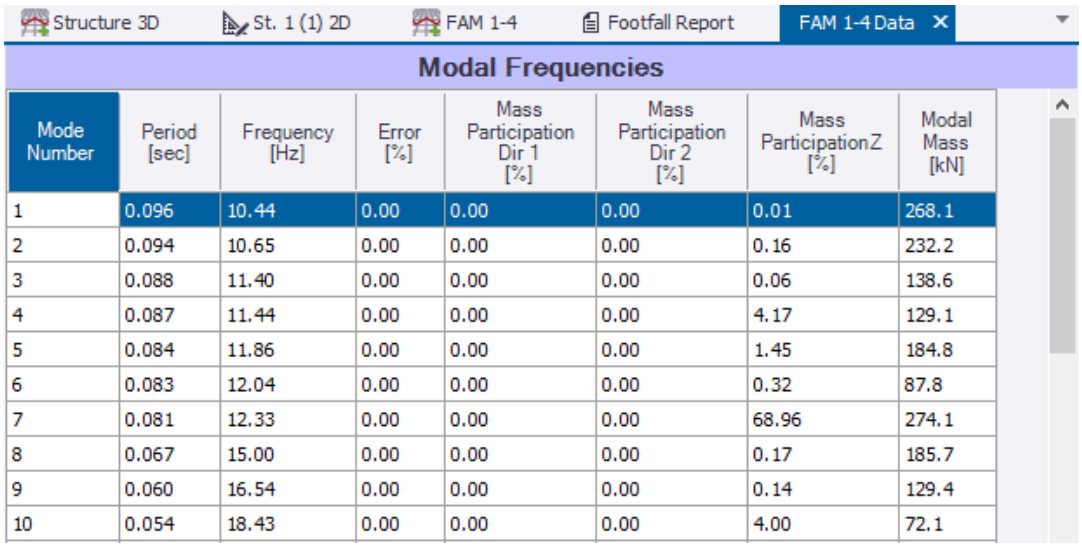
2. On the **Footfall Analysis** ribbon, select **Tabular Results** 

The **Floor Analysis Model Data** ribbon is displayed and an FAM tabular data view opens.



Modal Frequencies

To view modal frequencies, on the **Floor Analysis Model Data** select **Modal Frequencies**



Modal Frequencies

Mode Number	Period [sec]	Frequency [Hz]	Error [%]	Mass Participation Dir 1 [%]	Mass Participation Dir 2 [%]	Mass Participation Z [%]	Modal Mass [kN]
1	0.096	10.44	0.00	0.00	0.00	0.01	268.1
2	0.094	10.65	0.00	0.00	0.00	0.16	232.2
3	0.088	11.40	0.00	0.00	0.00	0.06	138.6
4	0.087	11.44	0.00	0.00	0.00	4.17	129.1
5	0.084	11.86	0.00	0.00	0.00	1.45	184.8
6	0.083	12.04	0.00	0.00	0.00	0.32	87.8
7	0.081	12.33	0.00	0.00	0.00	68.96	274.1
8	0.067	15.00	0.00	0.00	0.00	0.17	185.7
9	0.060	16.54	0.00	0.00	0.00	0.14	129.4
10	0.054	18.43	0.00	0.00	0.00	4.00	72.1

This table lists the period, frequency, mass participation, and modal mass for each mode.

Mode Shape by Node

To view mode shapes by node:

1. On the **Floor Analysis Model Data** ribbon, select **Mode Shape by Node**
2. Select the required **Mode** from the list.

Mode Shape by Node (Unity normalised)

Node No.	X [m]	Y [m]	Z [m]	Diaphragm	Translation X	Translation Y	Translation Z	Rotation X	Rotation Y	Rotation Z
303	16.722	6.251	3.000	-	0.0000	0.0000	-0.3991	0.2926	-0.0933	0.0000
307	2.000	4.346	3.000	-	0.0000	0.0000	0.0000	0.0000	0.0377	0.0000
311	11.504	5.884	3.000	-	0.0000	0.0000	-0.3961	0.2268	-0.0296	0.0000
312	0.000	4.967	3.000	-	0.0000	0.0000	-0.0000	0.0000	0.0002	0.0000
341	2.000	3.725	3.000	-	0.0000	0.0000	0.0000	0.0000	0.0327	0.0000
346	2.000	3.104	3.000	-	0.0000	0.0000	0.0000	0.0000	0.0246	0.0000
347	1.333	7.450	3.000	-	0.0000	0.0000	0.0000	0.0181	-0.0004	0.0000
352	0.667	7.450	3.000	-	0.0000	0.0000	0.0000	0.0095	-0.0001	0.0000

The mode shapes are listed for the selected mode.

Response Parameters

To view response parameters:

1. On the **Floor Analysis Model Data** ribbon, select **Response Parameters**
2. Select the **Resonant** or **Transient** response type
3. Select the **Pacing Frequency** from the list
4. In the **Properties** window, ensure the Footfall Checks parameters are specified as required.

Response Parameters

Node No.	SRSS Acceleration [mm/s²]	Response Factor, Rf	Number of Events in Period, na	Status
1209	0.0	0.8	6.727E+06	Pass
1284	0.0	0.8	6.793E+06	Pass
126	0.0	0.7	7.843E+06	Pass
1282	0.0	0.7	7.891E+06	Pass
1188	0.0	0.7	8.064E+06	Pass
1333	0.0	0.7	9.603E+06	Pass
719	0.0	0.7	9.703E+06	Pass
1119	0.0	0.7	9.859E+06	Pass

Note in particular:

- If **Consider VDV for Pass/Fail** is *unchecked* in the **Properties** window, the response factor is compared to the target response factor to determine the status.
- If **Consider VDV for Pass/Fail** is *checked*, the number of events are compared to the target number of events to determine the status.

Accelerations

To view accelerations:

1. On the **Floor Analysis Model Data** ribbon, select **Accelerations**
2. Select the **Resonant** response type
3. Select the required **Mode** from the list
4. Select the required **Pacing Frequency** from the list
5. Select the required **Harmonic** from the list
6. Select the required **Node** from the list

Accelerations per Mode

Node No.	Harmonic	Amplitude	Acceleration [mm/s²]
1403	1	0.4033	0.000
1403	2	0.4033	0.000
1403	3	0.4033	0.000
1403	4	0.4033	0.000

Base Data

To view base data:

1. On the **Floor Analysis Model Data** ribbon, select **Base Data**

Base Data

Mode Number	Harmonic	Dynamic Load Factor	Weighting Factor	Dynamic Magnification Factor
1	1	0.371	1.000	0.031
1	2	0.095	1.000	0.135
1	3	0.074	1.000	0.365
1	4	0.064	1.000	0.904

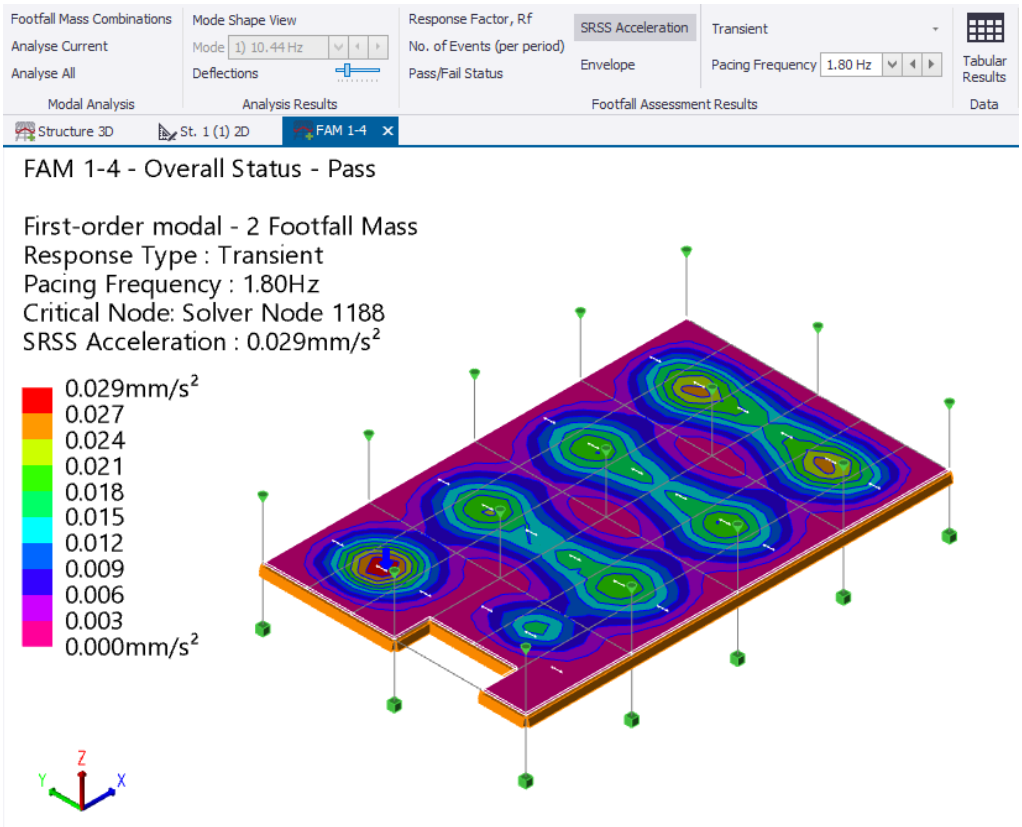
The choice of Resonant, or Transient, plus the Mode, Harmonic and Pacing Frequency to be displayed can be selected from the ribbon.

SRSS Acceleration

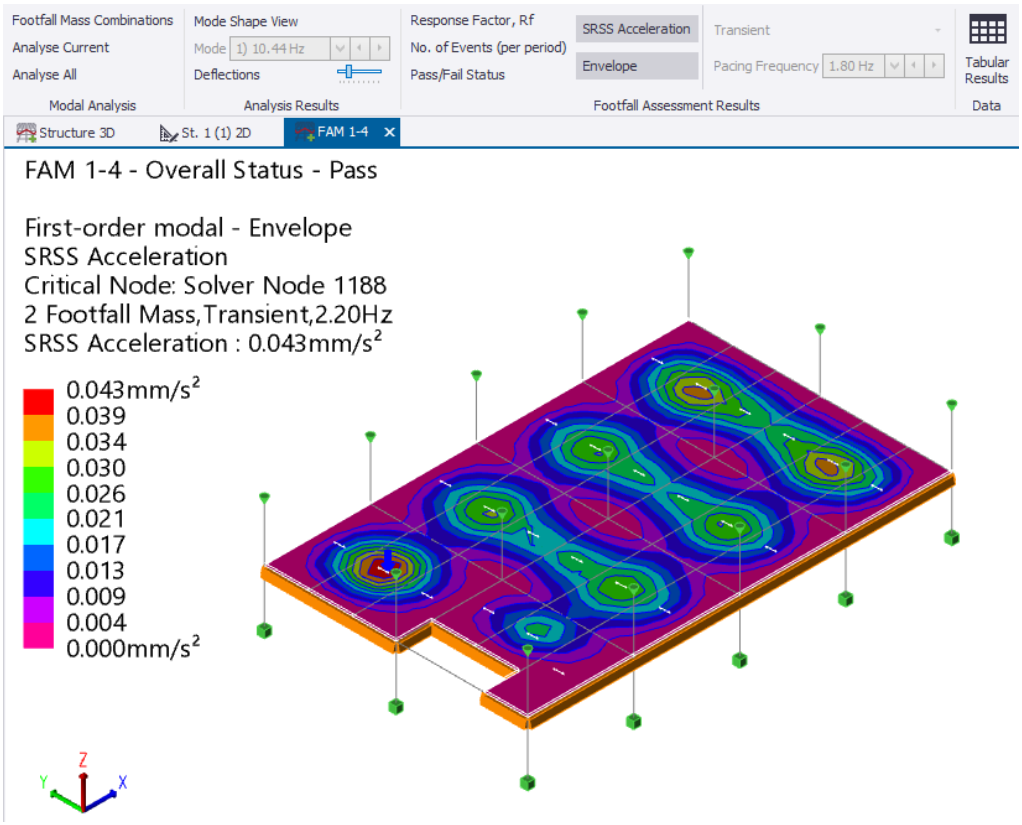
To graphically display SRSS Accelerations, return to the Floor Analysis Model (FAM) view and:

1. On the **Footfall Analysis** ribbon, select **SRSS Acceleration**.
2. Unselect **Envelope** and select **Transient**

The transient acceleration is shown and the critical node identified for the Footfall Mass combination selected in the Loading list.



3. To consider all response types (resonant and transient), as well as all pacing frequencies and mass combinations (if more than one is defined), select **Envelope**.



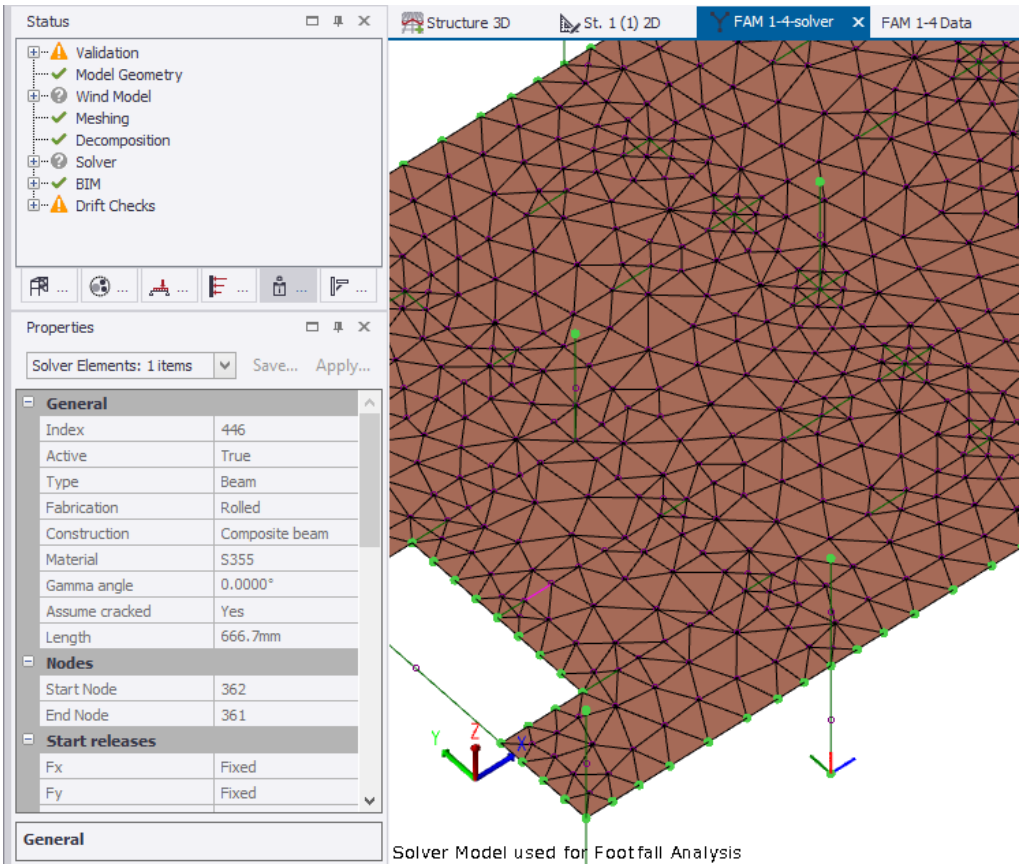
FAM solver model

After the analysis has been performed, if you require you can examine the solver model used for the Footfall Analysis model as follows:

1. Ensure that the required Floor Analysis Model (FAM) view is active.
2. To change the view type of the FAM view, do one of the following:
   - Right-click the FAM **view tab** and in the context menu, select **Solver view**
   - Or, in the **Status bar** at the bottom of the window, click **Solver View**.

   The FAM view changes to an FAM solver view.
3. Right-click anywhere in the solver view.
4. In the context menu, select **Solver models**.
5. In the submenu, select **Solver Model used for Footfall Analysis**.

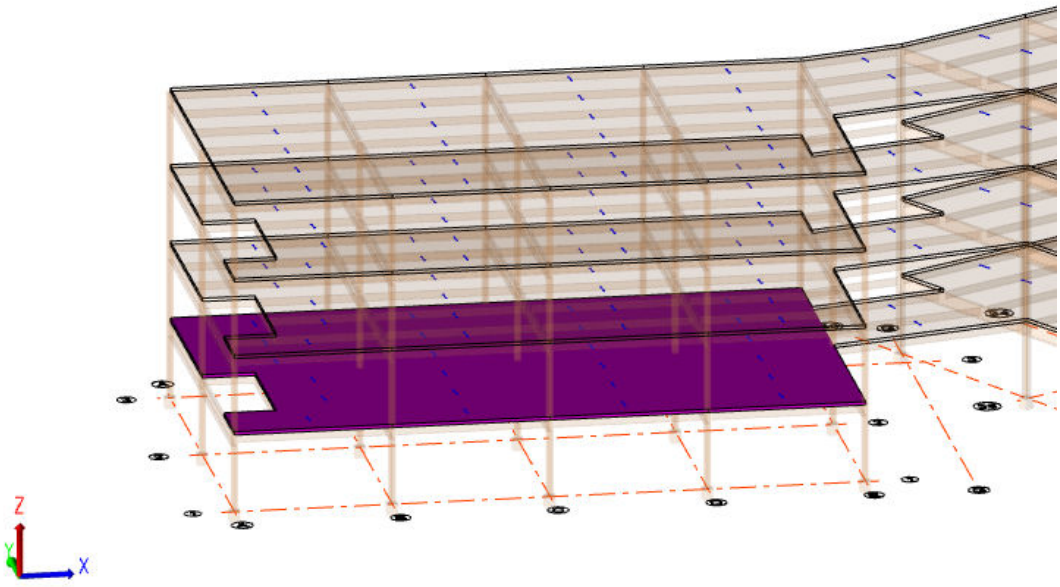
   Tekla Structural Designer opens the selected solver model. You can then select individual elements in the solver model to display their section and material properties in the **Properties** window.



Create additional floor analysis models, edit or delete models

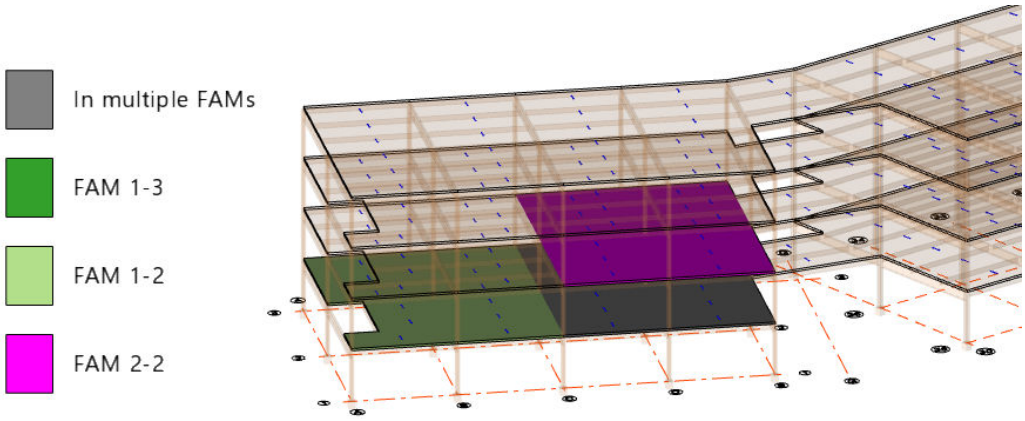
When the **Structure 3D** view is active and set to the **Footfall Analysis View** regime it will display all the floor analysis models that have been created so far.

You can continue to define as many FAM's as you wish in a similar manner, on the same or other levels. Slab items can be in more than one FAM. For example you might create a single FAM for a very large floor area which will expose areas of interest which might then be investigated with other more localized FAMs.



The following options are available:

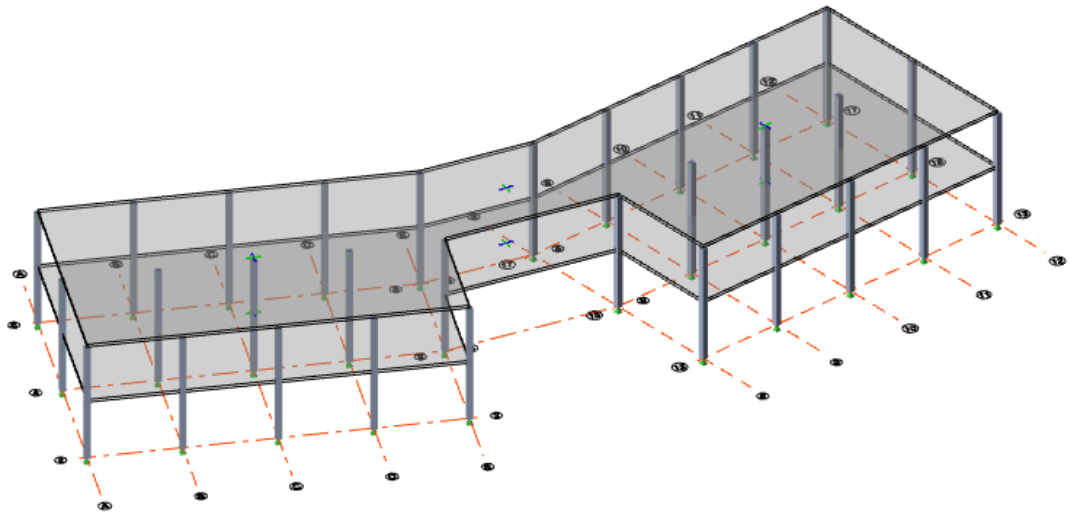
- Click on an existing model in order to edit it in a FAM view.
- Select **New** in the ribbon to start a new model
- Select **Delete** in the ribbon, then click on an existing model in order to delete it.



9.5 Footfall analysis basic workflow - CCIP 016

The CCIP-016 design guide (Ref. 3 (page 698)) is applicable to structures of any construction material, e.g. steel, composite, concrete or timber, making it a highly flexible option applicable to all kinds of buildings.

The concrete flat slab structure shown below can be used to illustrate the basic workflow when working to the CCIP-016 design guide.



Footfall assessment in Tekla Structural Designer

Download and open the tutorial model

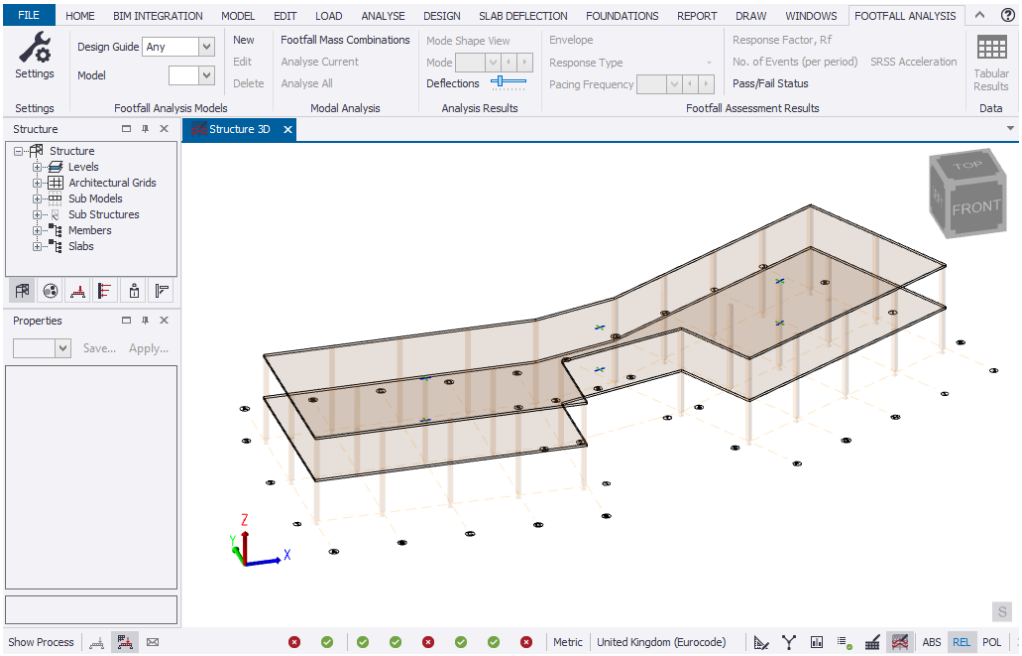
1. Download the tutorial model zip file Footfall - EC UK CCIP-016.zip from here.
2. Extract the following tutorial model from the zip:
   - Footfall - EC UK CCIP-016.tsmd
3. Open the model in Tekla Structural Designer.

Open a footfall analysis view

1. Open a **Structure 3D** Scene view.
2. In the **Status bar** at the bottom of the main window, click the **Footfall Analysis View** regime button.



The **Footfall Analysis** ribbon becomes available, and the active view switches to a Footfall analysis view.



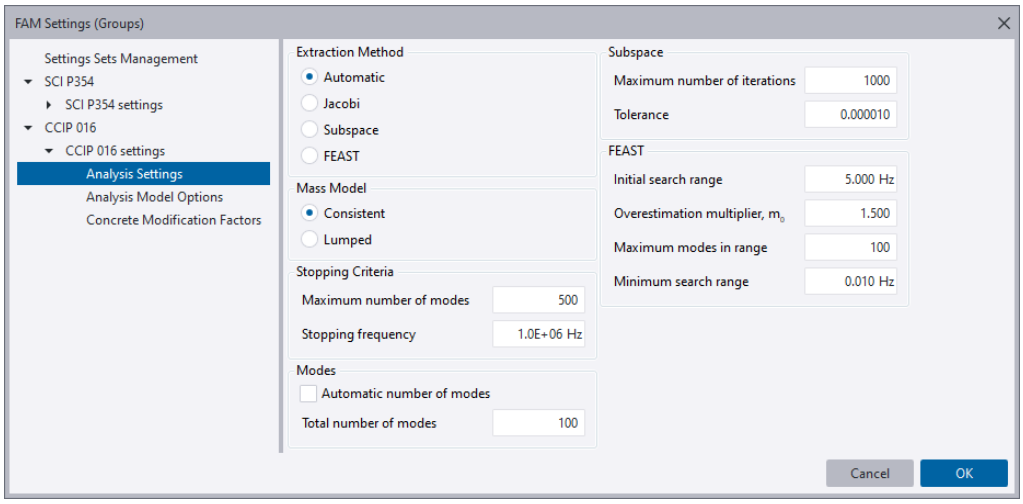
TIP In any view you can simply right-click a slab item and select **Create Floor Analysis Model** from the context menu in order to begin creating a new FAM. It is not essential for you to switch to the **Footfall Analysis View** regime first.

Confirm the floor analysis model settings

1. On the **Footfall Analysis** ribbon, select **Settings**

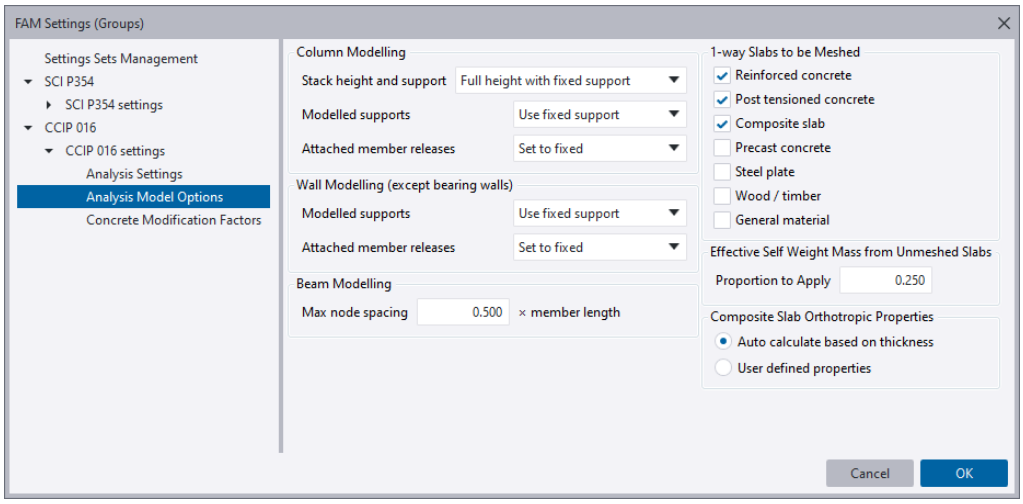
   The available settings sets for each design guide are displayed.

2. In the left hand pane, select **CCIP 016 > CCIP 016 settings > Analysis Settings**



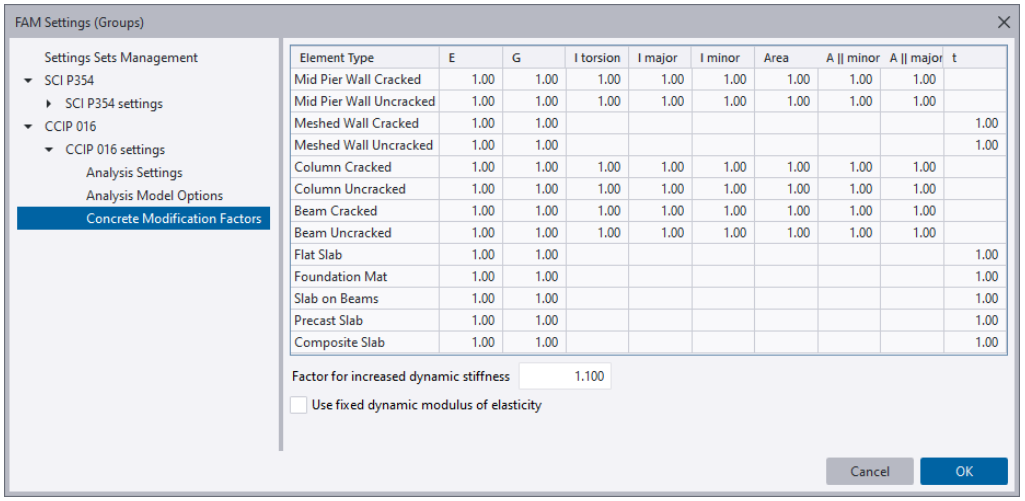
It is anticipated that most users will not need to adjust the default analysis settings which are as shown above.

3. In the left hand pane, select **CCIP 016 > CCIP 016 settings > Analysis Model Options**



These options allow you to configure the FAM to your requirements.

4. In the left hand pane, select **CCIP 016 > CCIP 016 settings > Concrete Modification Factors**

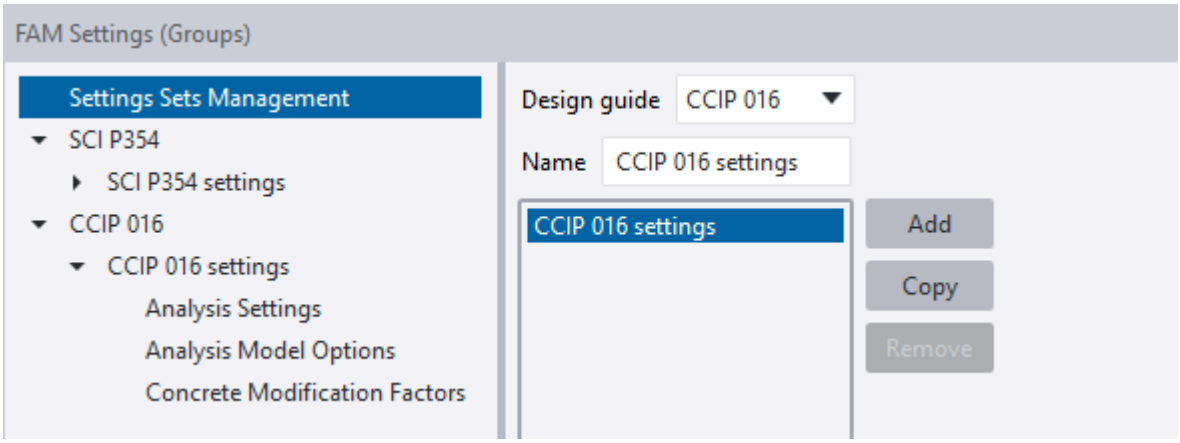


Note that by accepting the defaults shown above the static concrete modulus of elasticity will be factored by 1.1 to account for the greater stiffness under dynamic loading.

For more information on the different settings, options and factors, see: Footfall assessment settings (page 683)

5. In the left hand pane, select **Settings Sets Management**

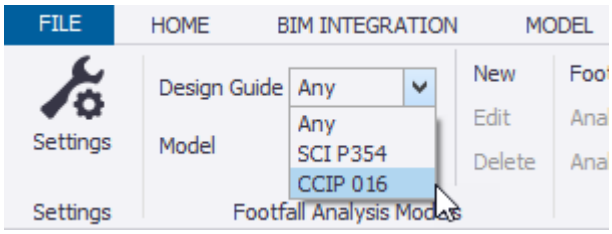
Although It is anticipated that the previously displayed settings will be appropriate in most cases, you can add further sets to investigate different configurations if you require. This would be achieved after selecting the CCIP 016 design guide using the **Add** or **Copy** buttons shown below.



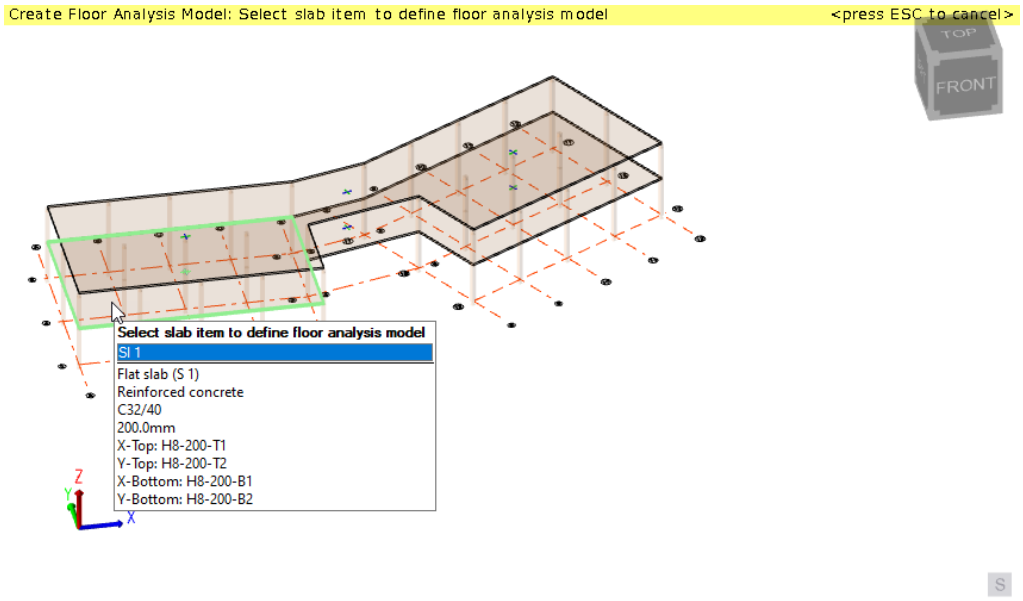
6. We don't need to add a further set so click **OK** to exit.

Create a floor analysis model

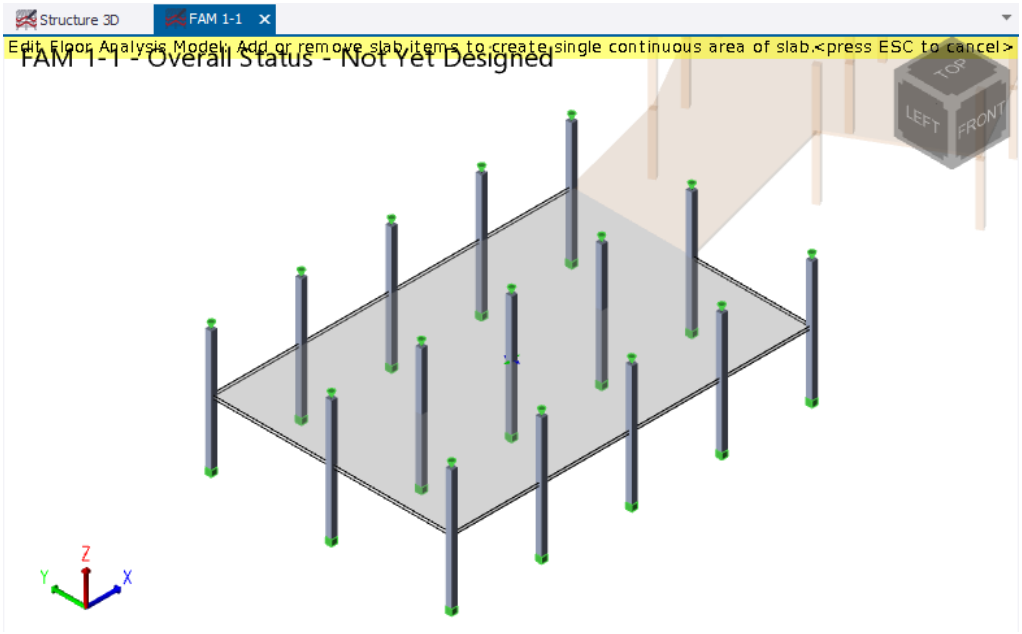
1. On the **Footfall Analysis** ribbon, in the **Design Guide** list, ensure CCIP-016 is selected.



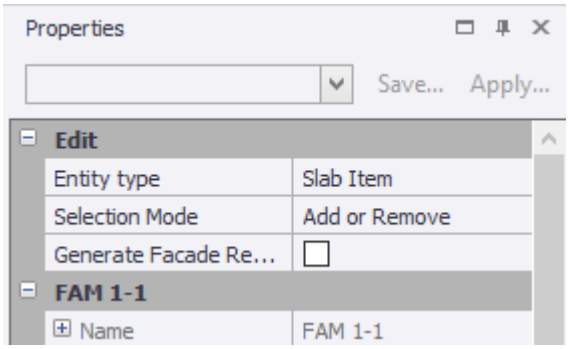
2. On the **Footfall Analysis** ribbon, select **New**
3. Select the slab item shown below.



A new Floor Analysis Model (FAM) view opens in Edit mode and the are displayed in the **Properties** window.



When in Edit mode, editing commands are available at the top of the **Properties** window.



Additional slab items (and attached members) could be added either by clicking slab items individually, or by box selecting. Note that the **Selection Mode** in the **Properties** window, which defaults to 'Add or Remove' can also be set to 'Add Only' or 'Remove Only' to assist this process.

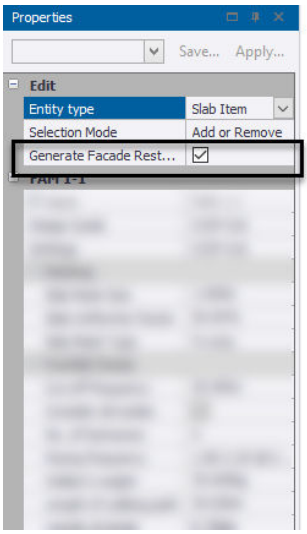
For this example no more slab items are required.

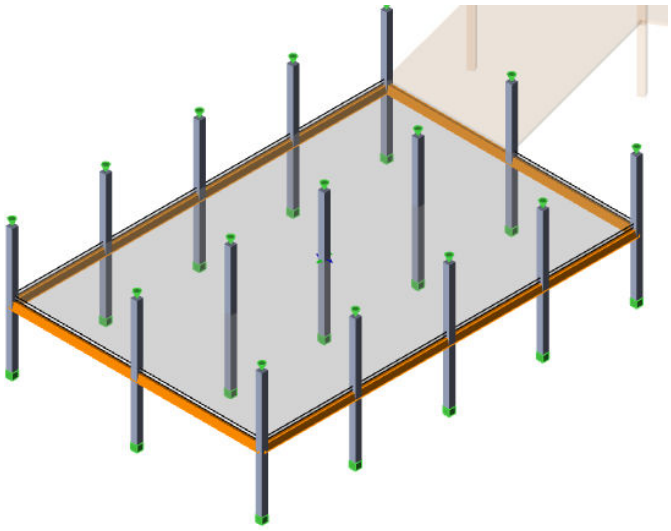
Add and edit facade restraints

Facade restraints can be used to vertically restrain specific slab edges.

1. Select **Generate Facade Restraint** in the **Properties** window

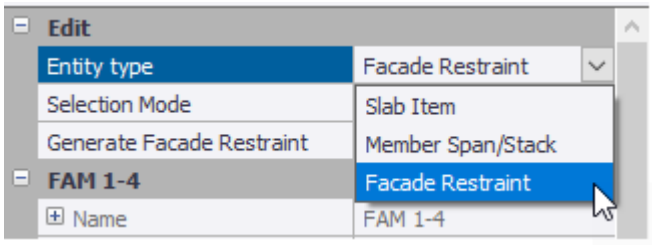
   Facade restraints are added around the entire perimeter of the selected slabs.





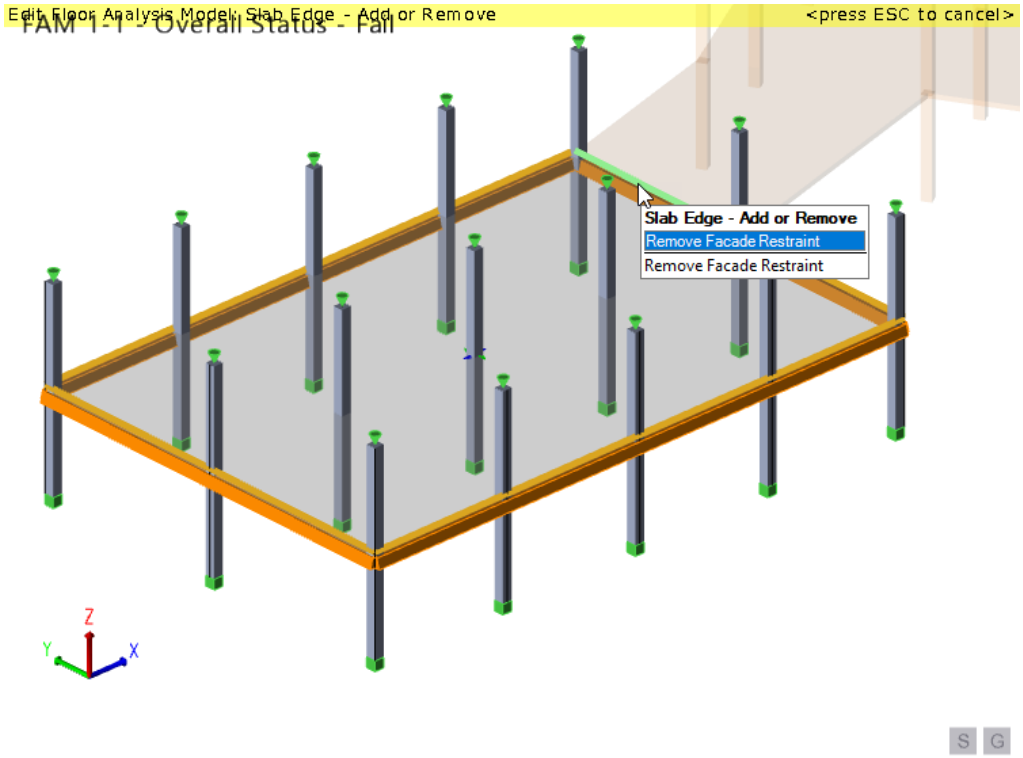
Restraints can also be added or removed from individual slab edges. We will now remove one of the restraints applied above.

2. In the **Properties** window, change the **Entity type** to 'Facade Restraint'.

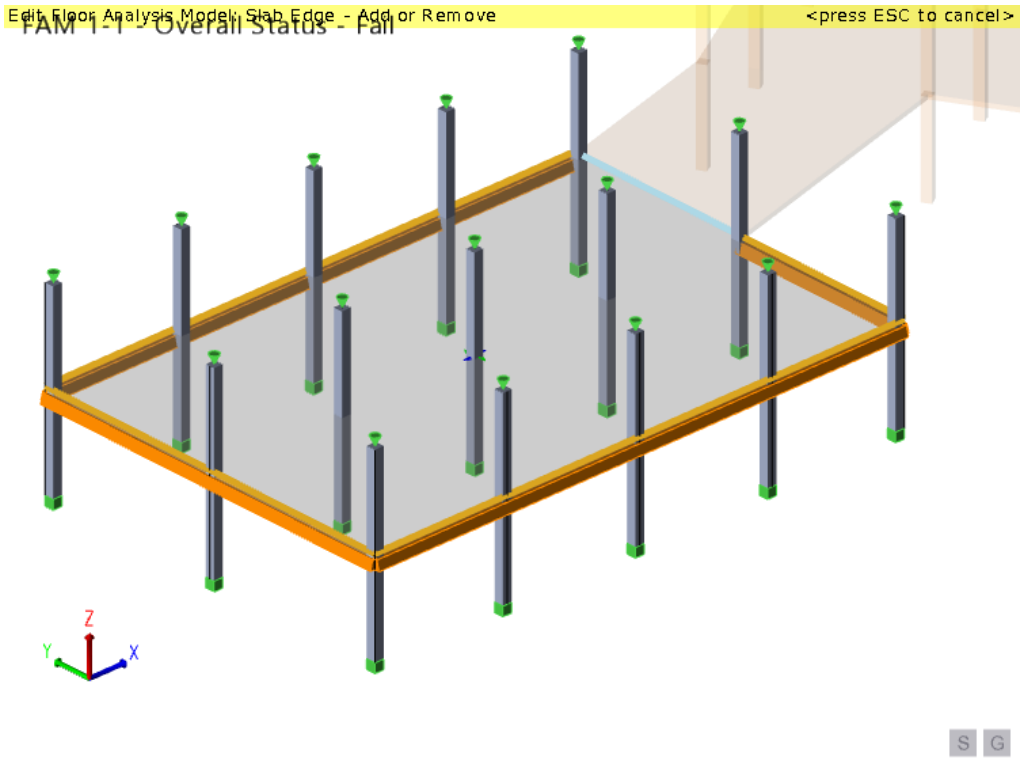


3. Set the **Selection Mode** to 'Remove Only'.

4. Click on the right hand slab edge at the position shown.



The FAM should now look as follows:

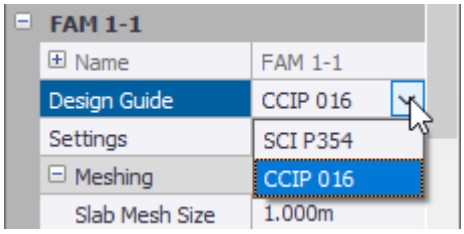


5. The FAM model is now complete, so press ESC to exit Edit mode.

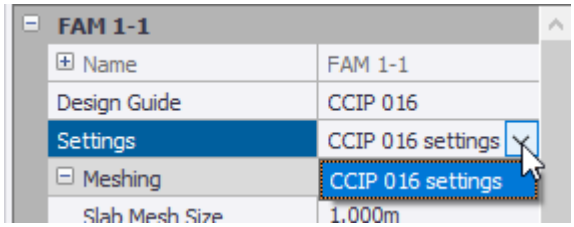
Confirm the design guide, settings and meshing

Review the following properties located at the top of the **Properties** window.

1. Ensure the **Design Guide** 'CCIP 016' is selected.

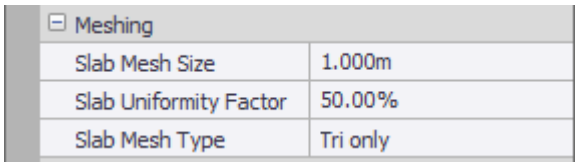


2. Ensure the **Settings** set 'CCIP 016 settings' is selected.



   In a real model you could have created multiple FAM Settings sets, in which case you could then choose which set to apply.

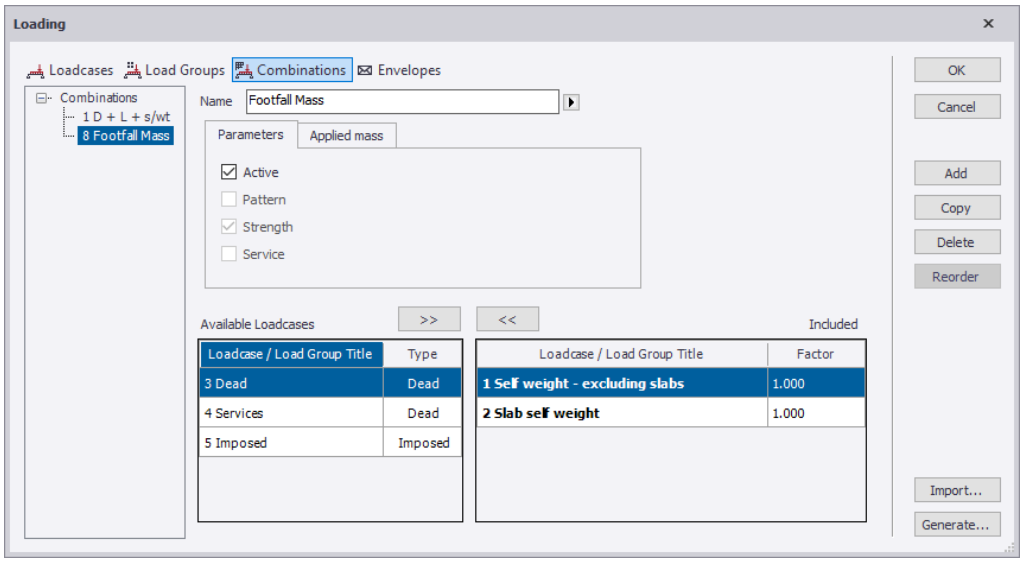
3. Review the meshing properties.



   The mesh defaults are reasonable and will be left unchanged.

Create a footfall mass combination

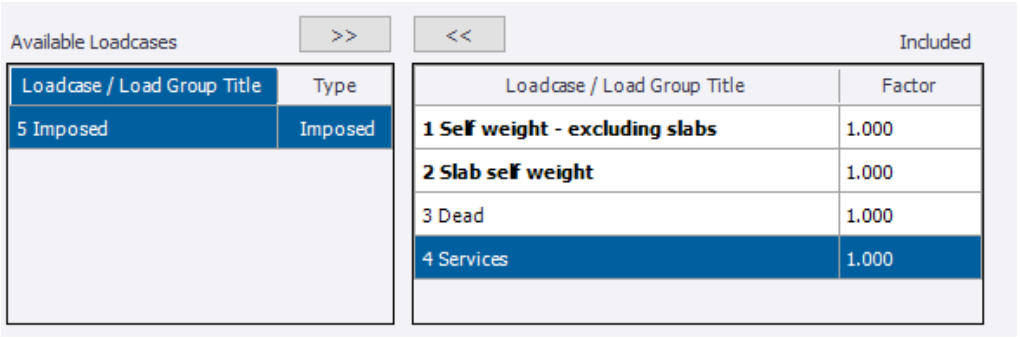
1. On the **Footfall Analysis** ribbon, select **Footfall Mass Combinations**.



Any existing combinations for static design will be listed in this dialog followed by a **Footfall Mass** combination which is selected as shown above. Two loadcases have automatically been included in this combination.

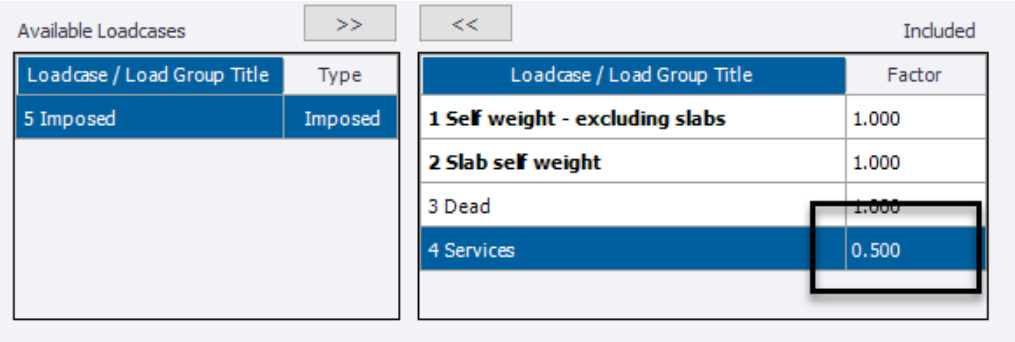
At this point you complete the Footfall Mass combination by making appropriate selections from the list of available loadcases.

2. Include the **Dead** and the **Services** loadcases.

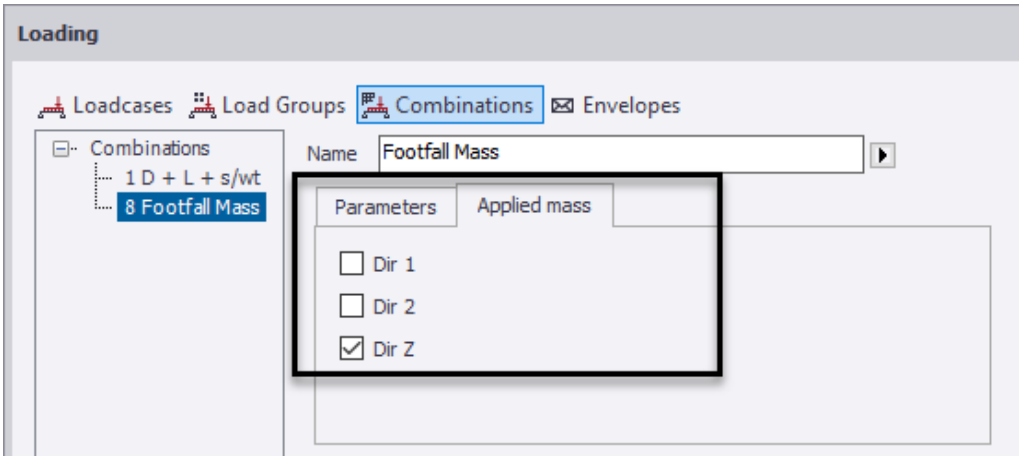


In real models engineering judgment will be required as the list of available loadcases can potentially be very long.

3. Set the Services loadcase factor to **0.500**.



4. Click the **Applied mass** tab in the dialog.



No changes are required here, but note that for Footfall mass combinations, the applied mass defaults to being applied in the Z direction only.

5. Click **OK** to save the footfall mass combination.

Analyse the model and view the results

On the **Footfall Analysis** ribbon, there are two analysis choices:

- **Analyse Current** - this runs a first-order modal analysis of **the currently selected FAM** for all active footfall mass combinations.
- **Analyse All** - this runs a first-order modal analysis of **all FAMs** for all active footfall mass combinations.

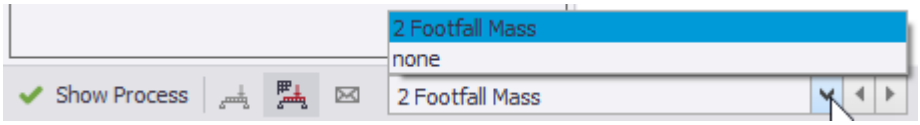
As only one FAM has been created at this stage, it doesn't actually matter which of the above two options is selected.

1. Click **Analyse Current**

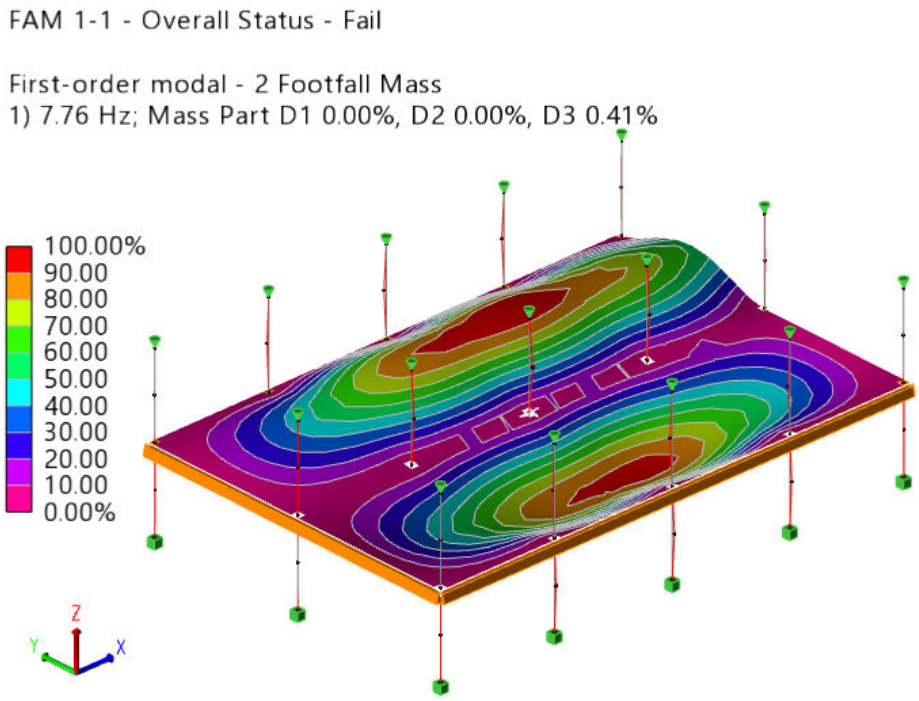
View the mode shapes

To view mode shapes graphically for the footfall mass combination:

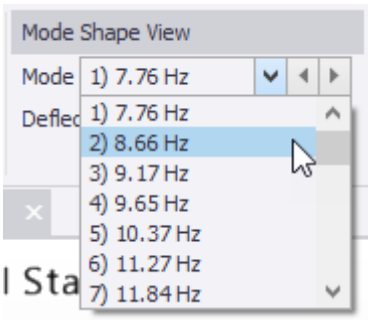
1. Select the **Footfall Mass** combination in the Loading list.

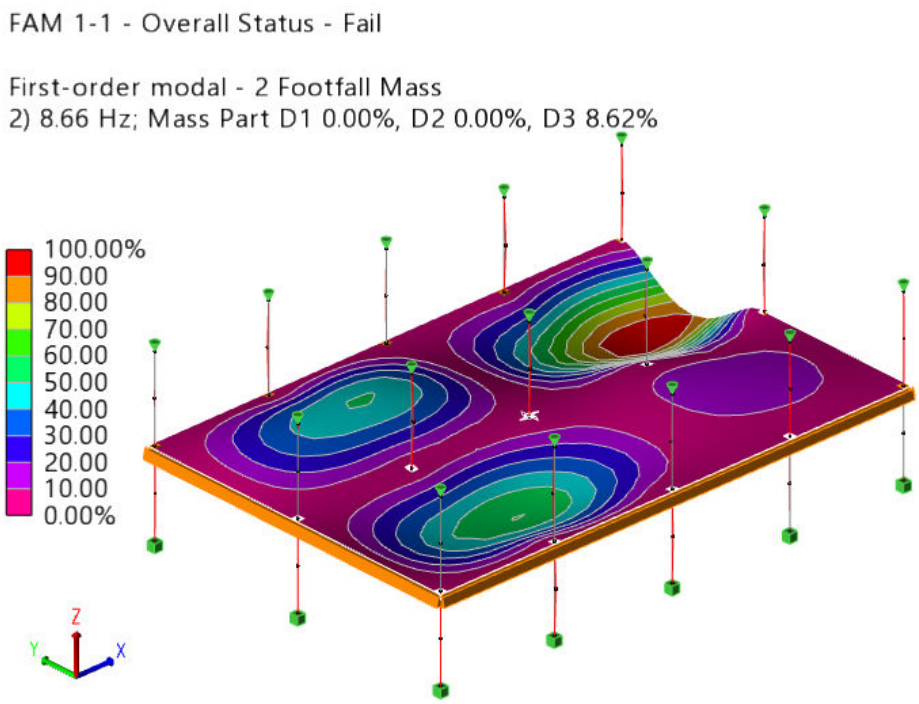


2. On the **Footfall Analysis** ribbon, select **Mode Shape View**.

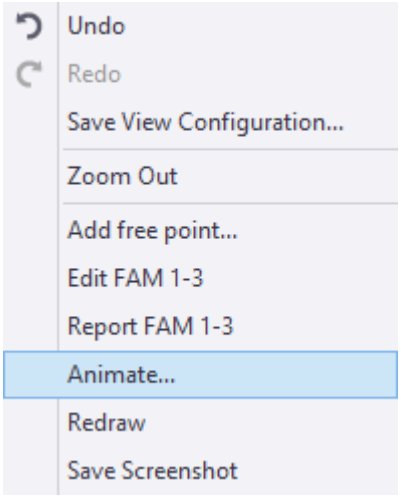


3. Select the next mode from the **Mode** list.

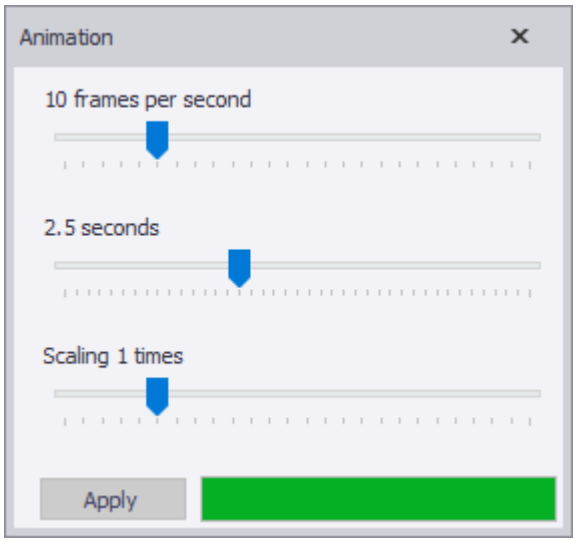




4. To adjust the amplitude of the mode shape, drag the **Deflections** slider in the ribbon as required.
5. To animate the mode shape, right-click in the view, and select **Animate** from the context menu.



6. To end the animation, close the Animation dialog.



Configure the footfall checks

Default CCIP-016 footfall check parameters for all models (predefined on the Footfall Assessment > CCIP-016 > Footfall Checks (page 693) page in global settings) are automatically copied into the **Footfall Checks** section of the **Properties** window for each FAM that is created.

Footfall Checks	
Cut off frequency	20.00Hz
Consider all modes	☑
No. of harmonics	4
Pacing frequency	1.80-2.20 @ 0.01
Walker's weight	76.00kg
Length of walking path	30.000m
Length of stride	0.750m
Damping ratio	3.00%
Axis of vibration perception	Z
Target vibration dose value (VDV)	$0.20 m/s^{1.75}$
Consider VDV for Pass/Fail	☐
Target response factor, Rf	4.0
Acceptable hotspots	☐

The copied values can then be adjusted as required to suit the individual FAM.

For a detailed explanation of the above parameters, refer to CCIP-016: A Design Guide for Footfall Induced Vibration of Structures. (Ref. 3 (page 698))

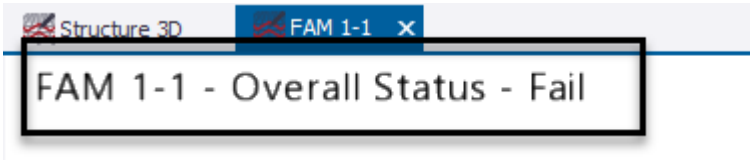
In this example we shall initially accept all the defaults. Note that the default **Target response factor** is **4.0** so any nodes in the FAM with a greater response will be deemed to have failed.

If you were to change any of the above values, the Footfall Assessment results would update automatically - there is no need to re-perform the analysis.

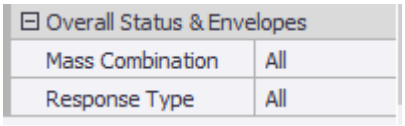
View the footfall check results

Overall status

The overall status is displayed in the top left corner of the FAM view.



Note that the status takes into account any **Mass Combination** or **Response Type** filters applied in the **Overall Status & Envelopes** section of the **Properties** window.



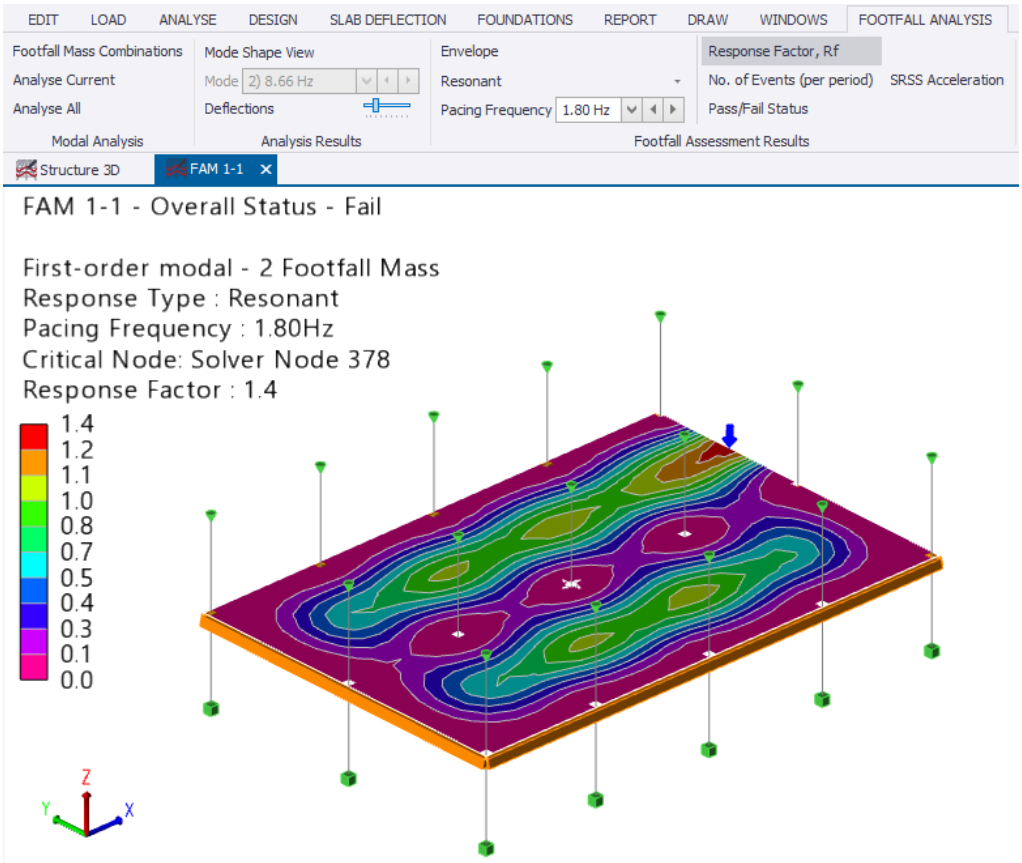
By default, all active footfall combinations are considered, and all (i.e. both resonant and transient) responses are considered.

As the overall status is 'Fail', this indicates that the target response factor has been exceeded for the resonant or the transient response at one or more pacing frequency.

View resonant and transient response factors at specific pacing frequencies

1. On the **Footfall Analysis** ribbon:
    a. Select **Response Factor, Rf**
    b. Ensure the response type is set to **Resonant**

c. Ensure **Envelope** is unselected

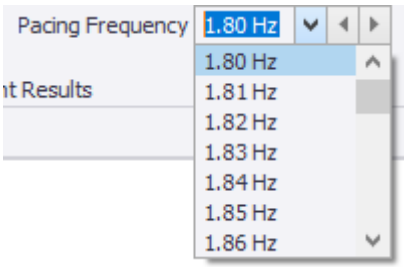


In the above view, the text below the overall status confirms the current response type and pacing frequency selected in the ribbon. The critical node number for this selection is also shown along with its response factor.

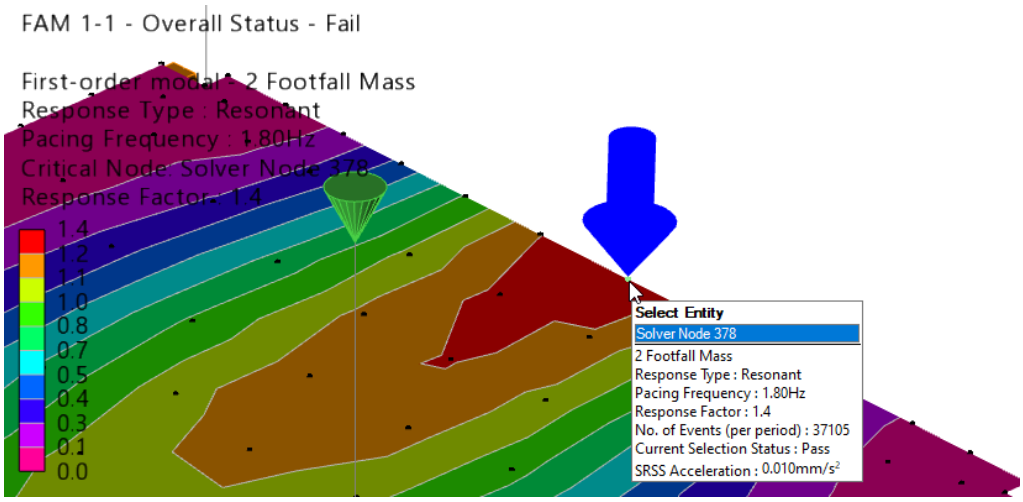
An arrow indicates the location of the critical node.

Initially the response is displayed for the lowest pacing frequency.

Because a range of pacing frequencies has been requested, the list in the ribbon can be used to move between them.



2. Zoom in to the critical node, then hover the cursor over the node to display the tooltip.

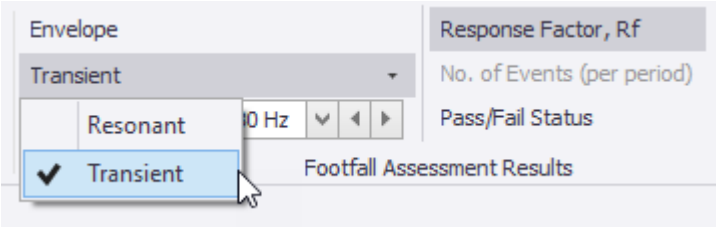


The tooltip shows that the resonant response factor at this node for the selected pacing frequency. As this is less than the target response the 'Current Selection Status' is Pass.

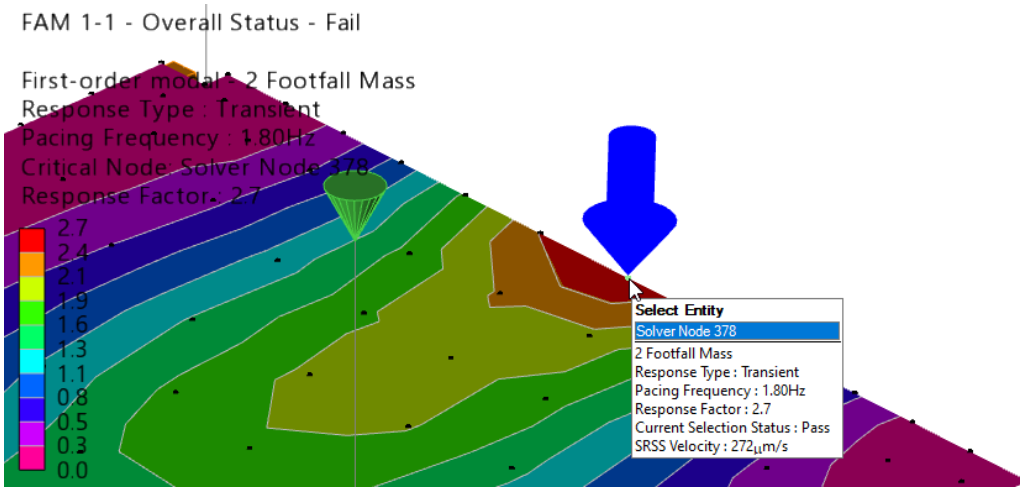
NOTE Node numbers are regenerated when a solver model is remeshed, so it is ok if the critical node's number that you see is not the same as the node number shown above.

---

3. To examine the transient response for the selected pacing frequency, select **Transient** on the ribbon.



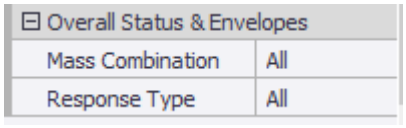
4. Hover the cursor over the critical node to display the tooltip.



The resonant response factor for the selected pacing frequency is less than the target response, so once again the 'Current Selection Status' is Pass.

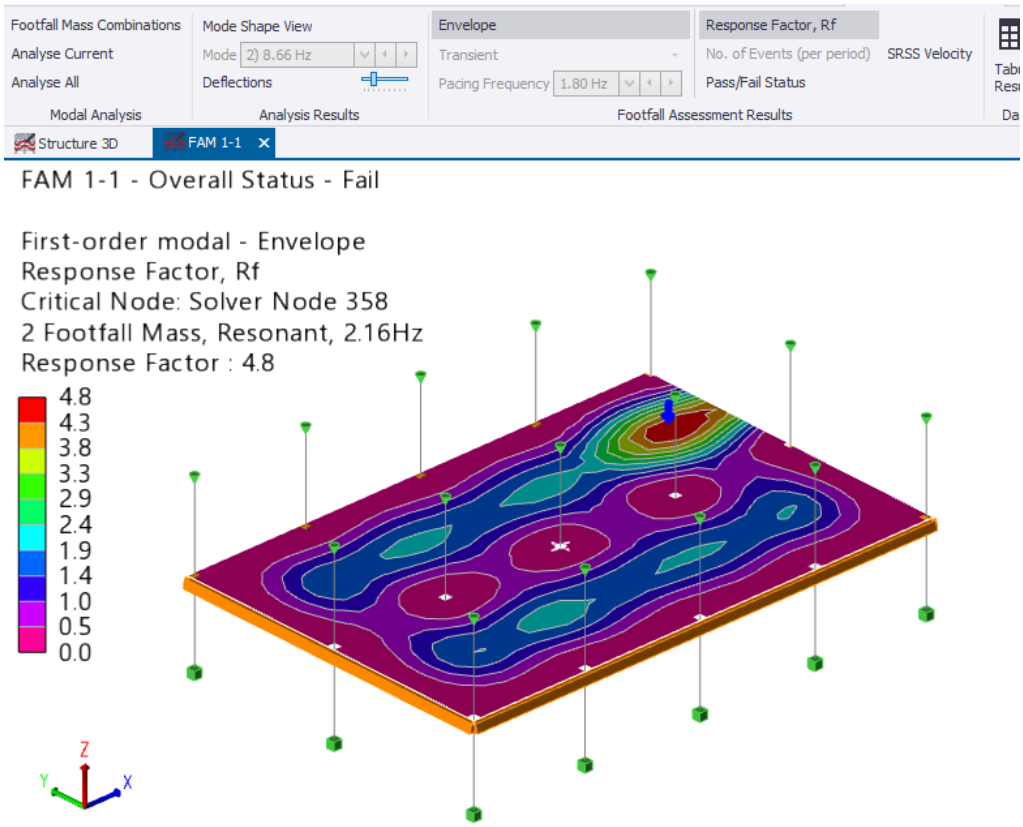
View response factor envelope

Note that envelopes take into account any **Mass Combination** or **Response Type** filters applied in the **Overall Status & Envelopes** section of the **Properties** window.



So by accepting the default filters as shown above, the response factor envelope allows us to locate the most critical response factor of *either type* (resonant or transient), for *all* pacing frequencies and *all* mass combinations (if more than one has been defined).

1. Select **Envelope** on the ribbon.



The above envelope reveals that the resonant response is critical at a pacing frequency of 2.16Hz with a response factor of 4.8.

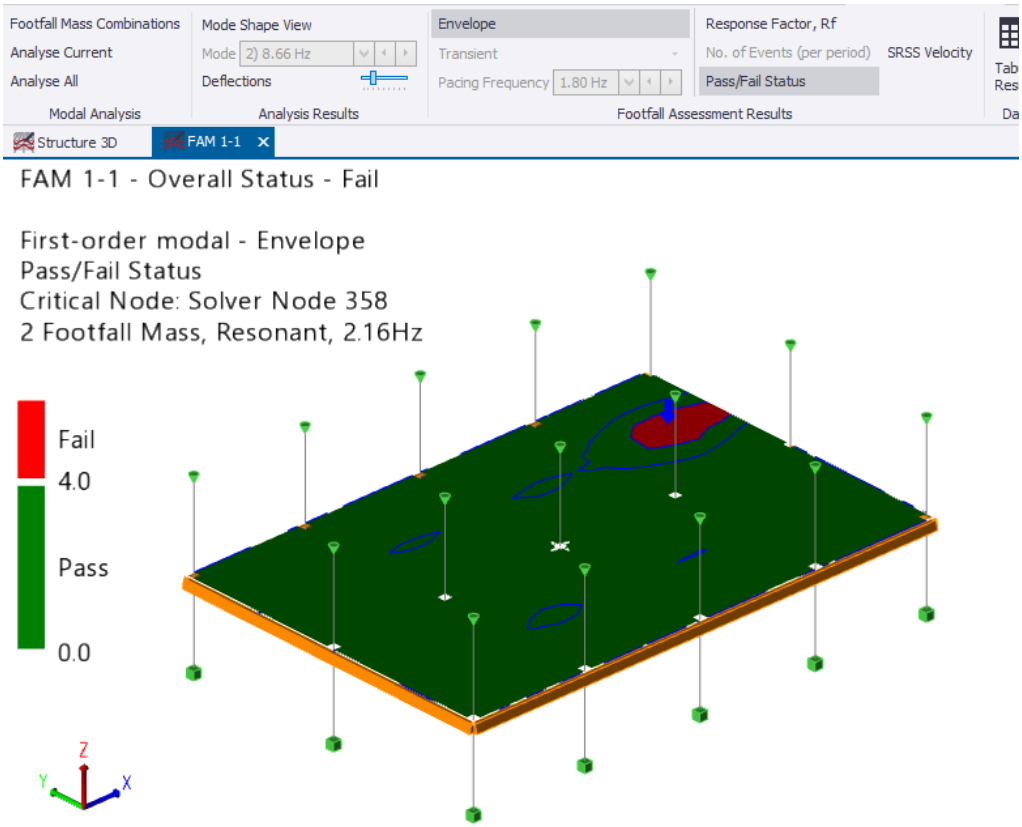
View Pass/Fail Status

To gain a better understanding of how wide an area is failing, the Pass/Fail status can be examined for the enveloped results.

On the ribbon:

1. Ensure **Envelope** is still selected.

2. Select **Pass/Fail Status**



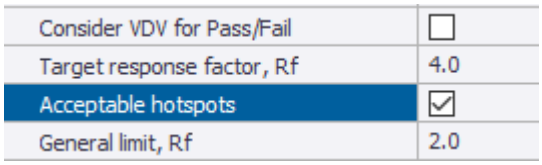
Nodes that exceed the **Target response factor** at any pacing frequency are shown as failed.

View acceptable hotspots

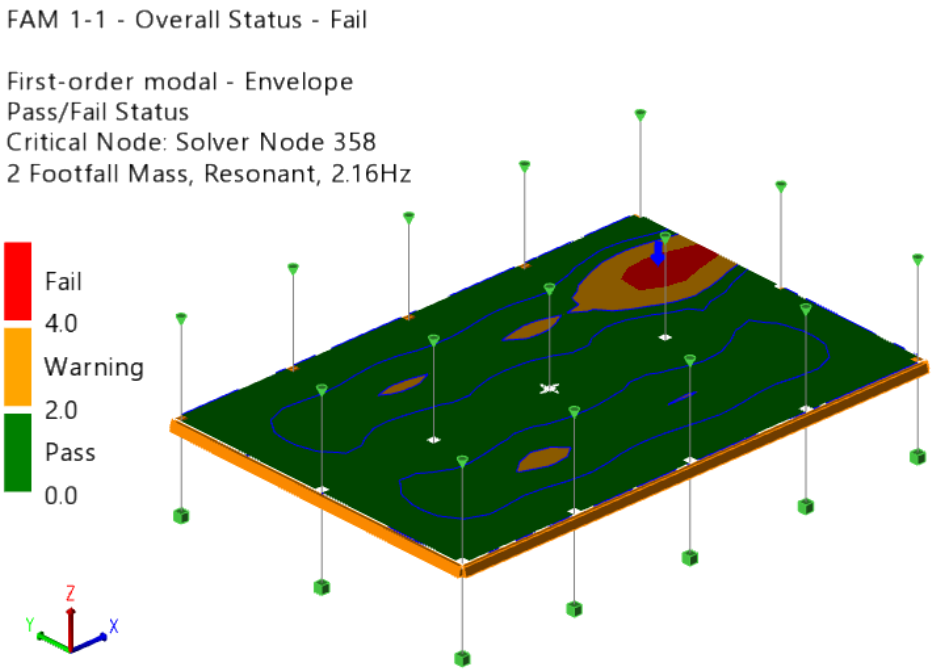
Acceptable hotspots can be used to investigate areas that fall between a general limit and the Target response factor.

In the **Properties** window:

1. Select **Acceptable hotspots**.



2. Leave the **General limit** as **2.0**



Regions with Rf values between the **General limit** and the **Target response factor** are shown with a warning status.

Perform a VDV assessment

If the response factor fails you may be able to make a variable dose value (VDV) assessment instead. VDVs are always calculated, but by default they are not considered when determining the overall status.

For ease of use the VDV calculation procedure is 'inverted' so that the **number of events (per period)** equates to the number of times the walking path would need to be executed in order to generate an adverse response.

Is a VDV assessment applicable for this example?

It is important to note that when working to CCIP-016 a VDV assessment is only allowed for the resonant response - it cannot be used to assess the transient response. So before proceeding, it makes sense to determine the overall status for the individual response types.

This can be done as follows:

1. In the **Properties** window, under the Overall Status & Envelopes heading, change the **Response Type** to 'Resonant'
2. Review the overall status.

   The overall status should still be failing.

3. In the **Properties** window, under the Overall Status & Envelopes heading, change the **Response Type** to 'Transient'
4. Review the overall status.

   The overall status should be passing.

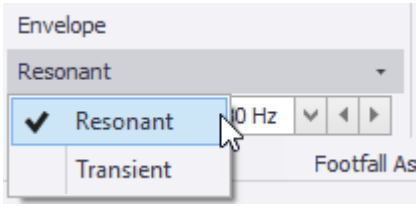
Therefore in this example a VDV assessment is applicable as it could potentially improve the resonant response status. It won't affect the transient response status, but as this is passing already, that is not going to be a problem.

View the number of events at each node

We want to display the number of events for the 'Resonant' response type enveloped for all pacing frequencies.

This is achieved as follows:

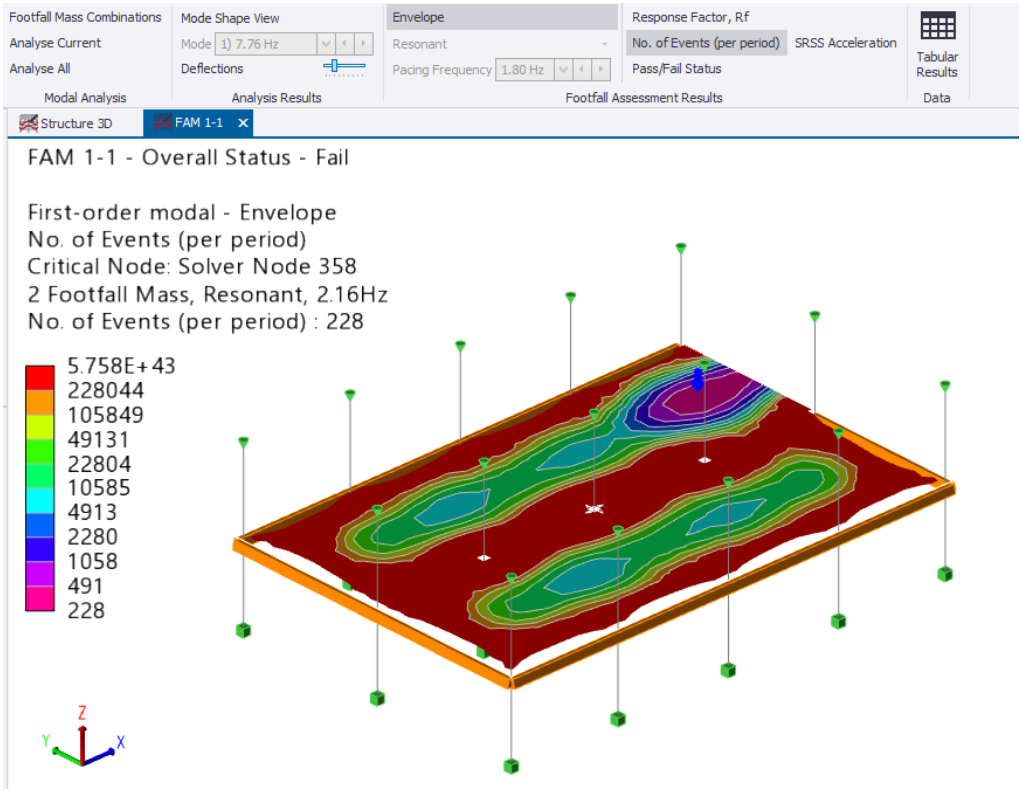
1. In the **Properties** window, under the Overall Status & Envelopes heading, change the **Response Type** to 'All'
2. On the **Footfall Analysis** ribbon:
   a. Ensure **Envelope** is unselected.
   b. If the response type is set to 'Transient' change it to 'Resonant'



3. On the ribbon, select **No. of Events (per period)**.

   The number of events at each node is displayed on a logarithmic scale.
4. Select **Envelope** on the ribbon.

   The enveloped number of events will now be displayed for *all* pacing frequencies and *all* footfall mass combinations, (only one mass combination has been defined in this example).



In the above display it can be seen that the critical number of events is 228. The arrow identifies the critical node where this occurs.

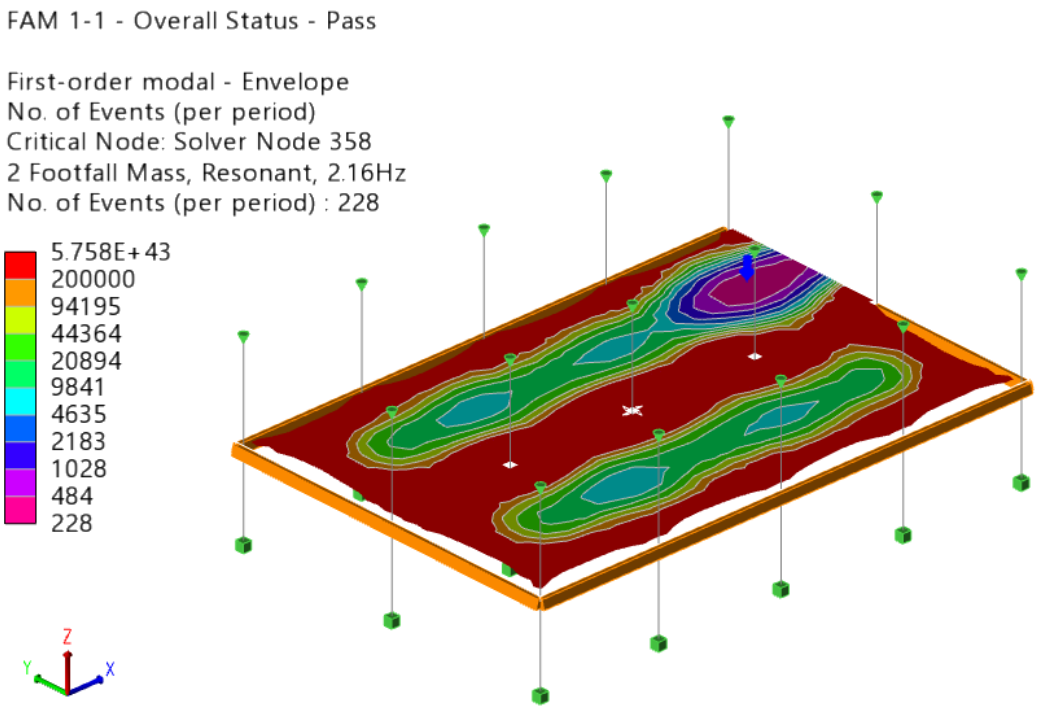
Activate the VDV assessment

For the purpose of the example we will assume the client has specified the target number of events as 200.

To perform a VDV assessment on this basis:

1. In the **Properties** window, select **Consider VDV for Pass/Fail**.
2. Set the **Target number of events in a period** as **200**.

Target vibration dose value (VDV)	0.20...
Consider VDV for Pass/Fail	☑
Target number of events in a period	200
Target response factor, Rf	4.0



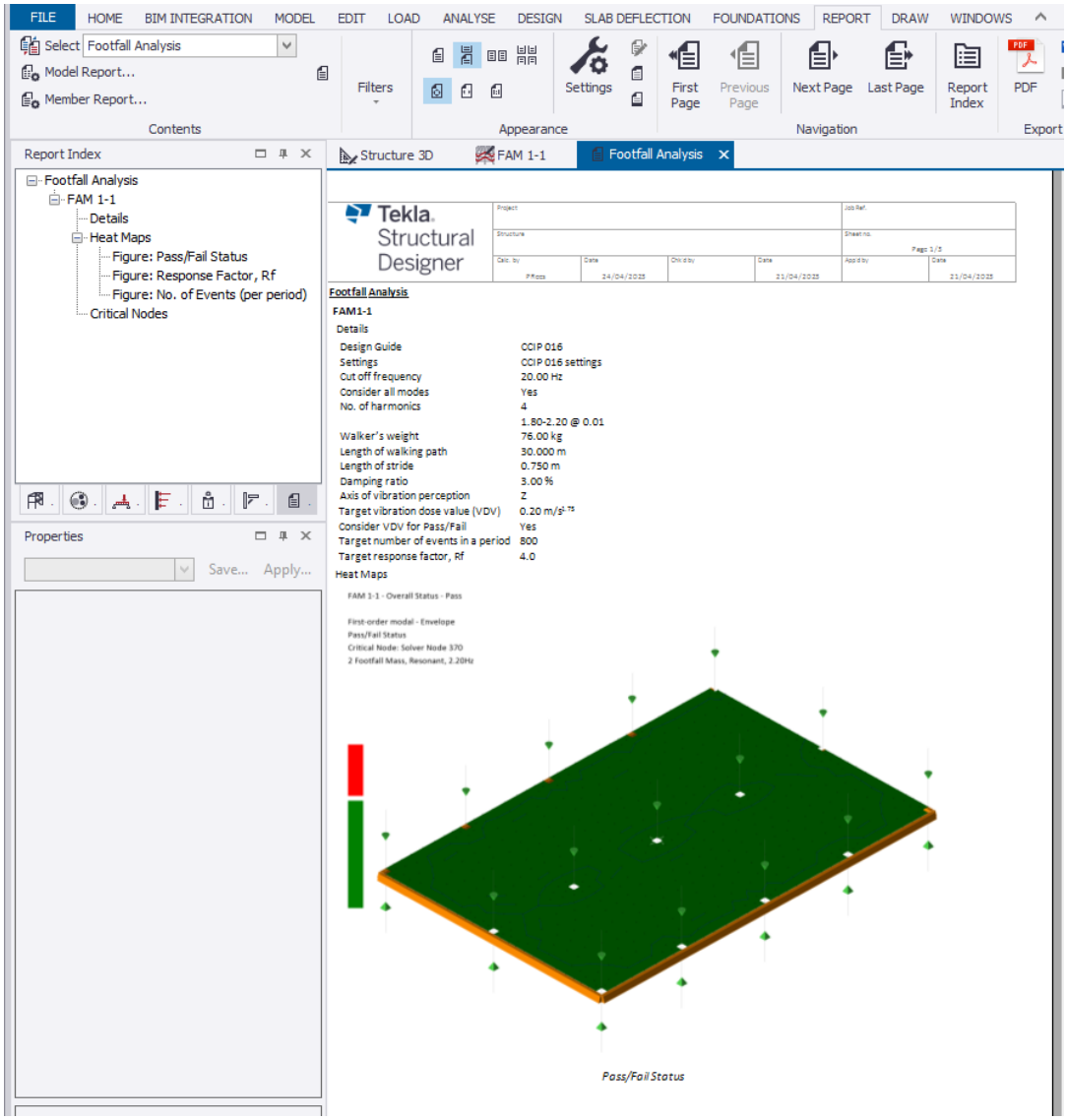
With the above settings, because the critical number of events is higher than the target, an overall pass status is achieved.

Output the calculations

To create a footfall analysis report:

1. On the ribbon, go to the **Report** tab.
2. Click **Model Report**.
3. In the **Report Contents** dialog, click **Add** and then under **Active Style**, enter a **Name** for the report
4. In the list of chapters and options, select **Footfall Analysis** and drag it across to the **Report Structure** area.
5. Click **OK**
6. On the ribbon, in the list on the left side, select the report name you just created.

7. Click **Show Report**.



Additional display and review options

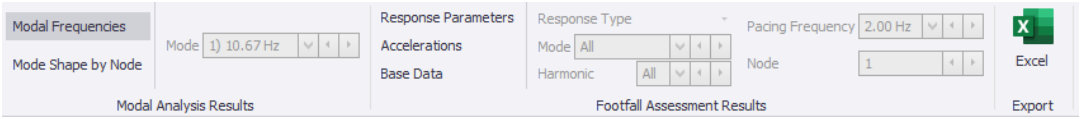
Several tabular data views are available, each of which can be exported to Excel.

To display the **Floor Analysis Model Data** ribbon from where the tabular data views can be accessed:

1. Make the Floor Analysis Model (FAM) view active once more.

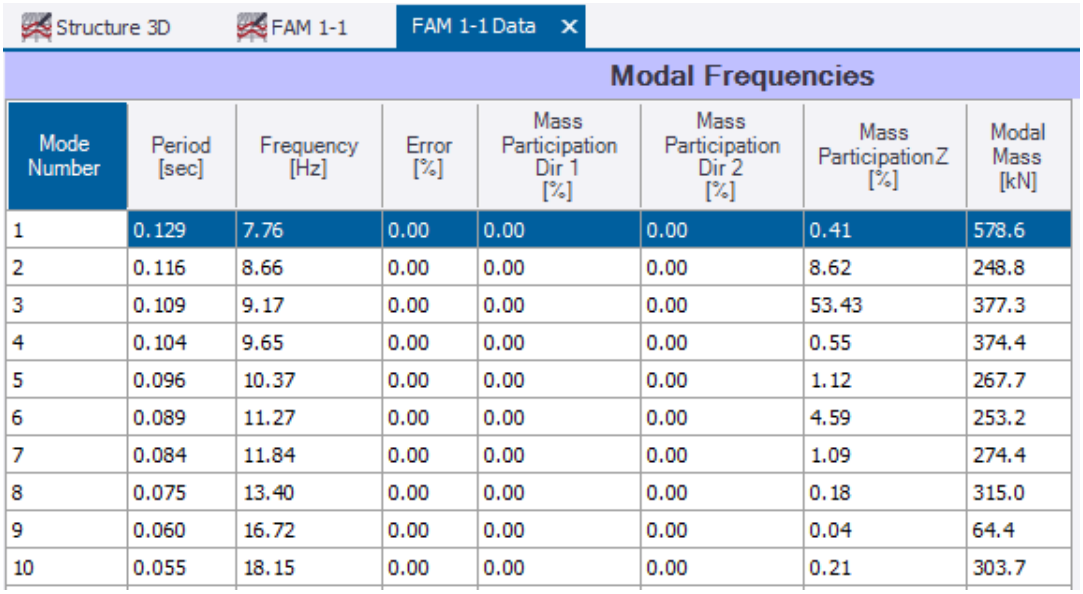
2. On the **Footfall Analysis** ribbon, select **Tabular Results** 

The **Floor Analysis Model Data** ribbon is displayed and an FAM tabular data view opens.



Modal Frequencies

To view modal frequencies, on the **Floor Analysis Model Data** select **Modal Frequencies**



Modal Frequencies

Mode Number	Period [sec]	Frequency [Hz]	Error [%]	Mass Participation Dir 1 [%]	Mass Participation Dir 2 [%]	Mass ParticipationZ [%]	Modal Mass [kN]
1	0.129	7.76	0.00	0.00	0.00	0.41	578.6
2	0.116	8.66	0.00	0.00	0.00	8.62	248.8
3	0.109	9.17	0.00	0.00	0.00	53.43	377.3
4	0.104	9.65	0.00	0.00	0.00	0.55	374.4
5	0.096	10.37	0.00	0.00	0.00	1.12	267.7
6	0.089	11.27	0.00	0.00	0.00	4.59	253.2
7	0.084	11.84	0.00	0.00	0.00	1.09	274.4
8	0.075	13.40	0.00	0.00	0.00	0.18	315.0
9	0.060	16.72	0.00	0.00	0.00	0.04	64.4
10	0.055	18.15	0.00	0.00	0.00	0.21	303.7

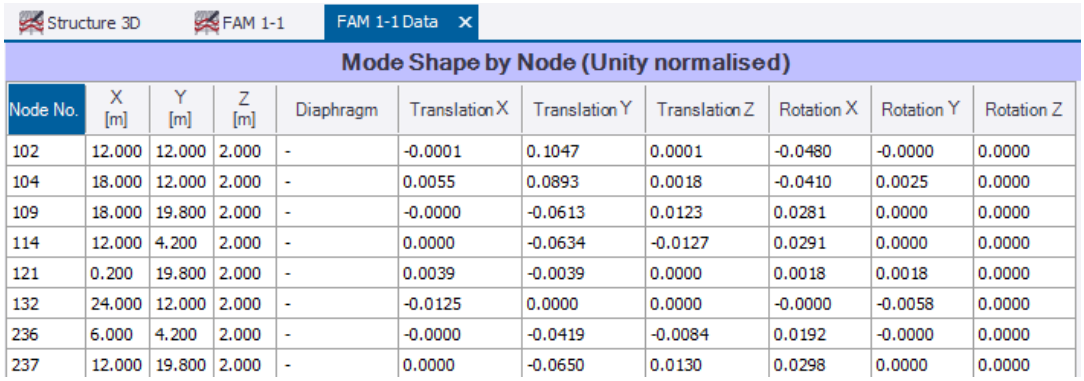
This table lists the period, frequency, mass participation, and modal mass for each mode.

Mode Shape by Node

To view mode shapes by node:

1. On the **Floor Analysis Model Data** ribbon, select **Mode Shape by Node**

2. Select the required **Mode** from the list.



Structure 3D | FAM 1-1 | FAM 1-1 Data

Mode Shape by Node (Unity normalised)

Node No.	X [m]	Y [m]	Z [m]	Diaphragm	Translation X	Translation Y	Translation Z	Rotation X	Rotation Y	Rotation Z
102	12.000	12.000	2.000	-	-0.0001	0.1047	0.0001	-0.0480	-0.0000	0.0000
104	18.000	12.000	2.000	-	0.0055	0.0893	0.0018	-0.0410	0.0025	0.0000
109	18.000	19.800	2.000	-	-0.0000	-0.0613	0.0123	0.0281	0.0000	0.0000
114	12.000	4.200	2.000	-	0.0000	-0.0634	-0.0127	0.0291	0.0000	0.0000
121	0.200	19.800	2.000	-	0.0039	-0.0039	0.0000	0.0018	0.0018	0.0000
132	24.000	12.000	2.000	-	-0.0125	0.0000	0.0000	-0.0000	-0.0058	0.0000
236	6.000	4.200	2.000	-	-0.0000	-0.0419	-0.0084	0.0192	-0.0000	0.0000
237	12.000	19.800	2.000	-	0.0000	-0.0650	0.0130	0.0298	0.0000	0.0000

The mode shapes are listed for the selected mode.

Response Parameters

To view response parameters:

1. On the **Floor Analysis Model Data** ribbon, select **Response Parameters**
2. Select the **Resonant** or **Transient** response type
3. Select the **Pacing Frequency** from the list
4. In the **Properties** window, ensure the Footfall Checks parameters are specified as required.

Note in particular:

- If **Consider VDV for Pass/Fail** is *unchecked* in the **Properties** window, the response factor is compared to the target response factor to determine the status.

Structure 3D | FAM 1-1 | FAM 1-1 Data

Response Parameters

Node No.	SRSS Acceleration [mm/s²]	Response Factor, Rf	Number of Events in Period, na	Status
358	0.036	4.8	228	Fail
370	0.036	4.7	237	Fail
357	0.036	4.7	246	Fail
369	0.035	4.6	258	Fail
377	0.033	4.4	326	Fail
533	0.033	4.3	349	Fail
359	0.033	4.3	358	Fail
378	0.032	4.3	364	Fail
376	0.032	4.2	388	Fail
493	0.031	4.0	463	Fail
677	0.030	4.0	471	Pass

- If **Consider VDV for Pass/Fail** is *checked* and the response type is *Resonant*, the number of events are compared to the target number of events to determine the status.

Structure 3D | FAM 1-1 | FAM 1-1 Data

Response Parameters

Node No.	SRSS Acceleration [mm/s²]	Response Factor, Rf	Number of Events in Period, na	Status
358	0.036	4.8	228	Pass
370	0.036	4.7	237	Pass
357	0.036	4.7	246	Pass
369	0.035	4.6	258	Pass
377	0.033	4.4	326	Pass
533	0.033	4.3	349	Pass
359	0.033	4.3	358	Pass
378	0.032	4.3	364	Pass
376	0.032	4.2	388	Pass
493	0.031	4.0	463	Pass
677	0.030	4.0	471	Pass

- If **Consider VDV for Pass/Fail** is *checked* and the response type is *Transient*, the transient response factor is compared to the target response factor to determine the status.

Structure 3D | FAM 1-1 | FAM 1-1 Data

Response Parameters

Node No.	RMS Velocity per Step [µm/s]	Response Factor, Rf	Status
378	385	3.7	Pass
379	347	3.4	Pass
377	312	3.0	Pass
391	304	3.0	Pass
376	297	2.9	Pass
370	291	2.8	Pass
358	279	2.7	Pass
369	272	2.6	Pass
357	266	2.6	Pass
533	265	2.6	Pass
493	248	2.4	Pass
557	245	2.4	Pass

Accelerations and Velocities

To view accelerations:

1. On the **Floor Analysis Model Data** ribbon, select **Accelerations**
2. Select the **Resonant** response type
3. Select the required **Mode** from the list
4. Select the required **Pacing Frequency** from the list
5. Select the required **Harmonic** from the list
6. Select the required **Node** from the list

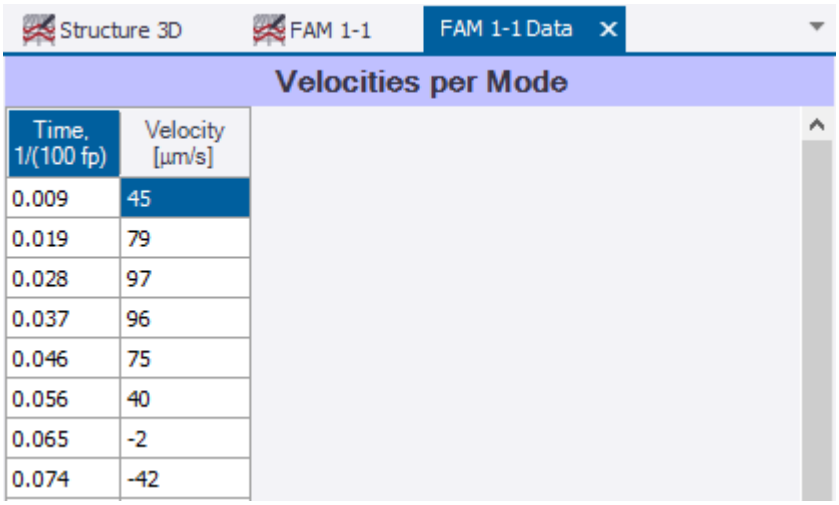
Structure 3D | FAM 1-1 | FAM 1-1 Data

Accelerations per Mode

Node No.	Harmonic	Amplitude	Acceleration Real [mm/s²]	Acceleration Imaginary [mm/s²]
358	1	0.7349	0.000	0.000
358	2	0.7349	0.000	0.000
358	3	0.7349	0.001	0.000
358	4	0.7349	-0.002	0.001

To view velocities:

1. Select **Accelerations**
2. Select the **Transient** response type
3. Select the required **Mode** from the list
4. Select the required **Pacing Frequency** from the list
5. Select the required **Node** from the list

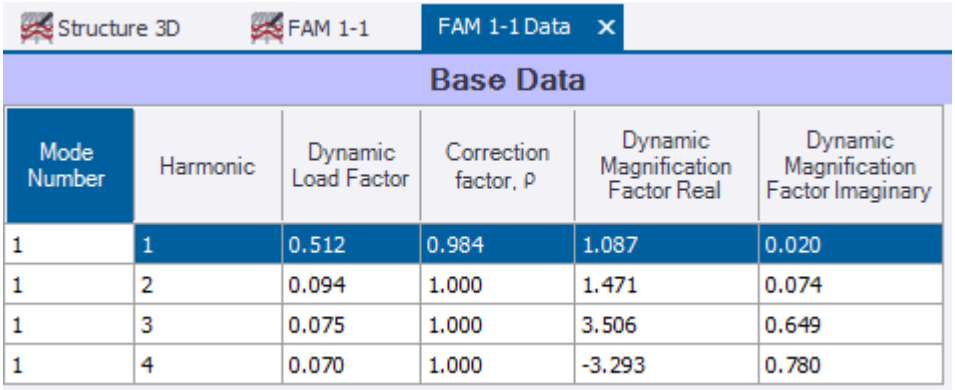


Time, 1/(100 fp)	Velocity [µm/s]
0.009	45
0.019	79
0.028	97
0.037	96
0.046	75
0.056	40
0.065	-2
0.074	-42

Base Data

To view Resonant base data:

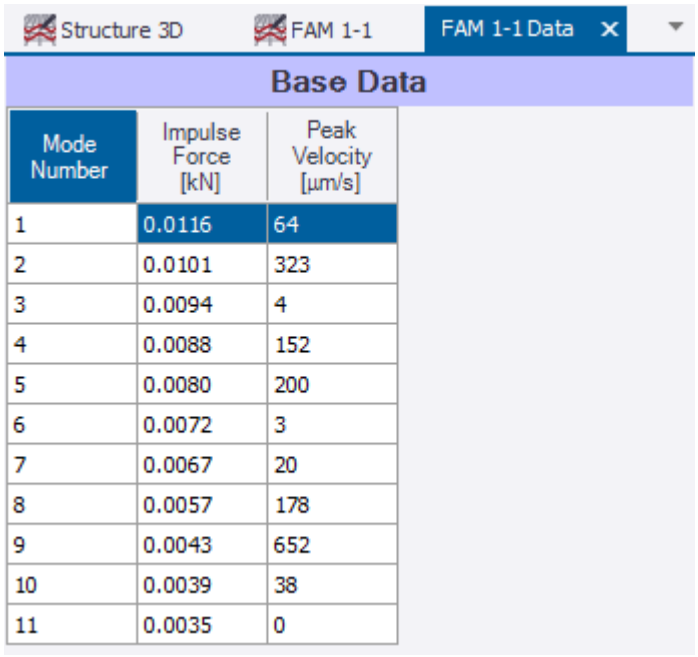
1. On the **Floor Analysis Model Data** ribbon, select **Base Data**
2. Select the **Resonant** response type
3. Select the required **Mode** from the list
4. Select the required **Harmonic** from the list
5. Select the required **Pacing Frequency** from the list



Mode Number	Harmonic	Dynamic Load Factor	Correction factor, ρ	Dynamic Magnification Factor Real	Dynamic Magnification Factor Imaginary
1	1	0.512	0.984	1.087	0.020
1	2	0.094	1.000	1.471	0.074
1	3	0.075	1.000	3.506	0.649
1	4	0.070	1.000	-3.293	0.780

To view Transient base data:

1. Select **Base Data**
2. Select the **Transient** response type
3. Select the required **Mode** from the list
4. Select the required **Pacing Frequency** from the list
5. Select the required **Node** from the list

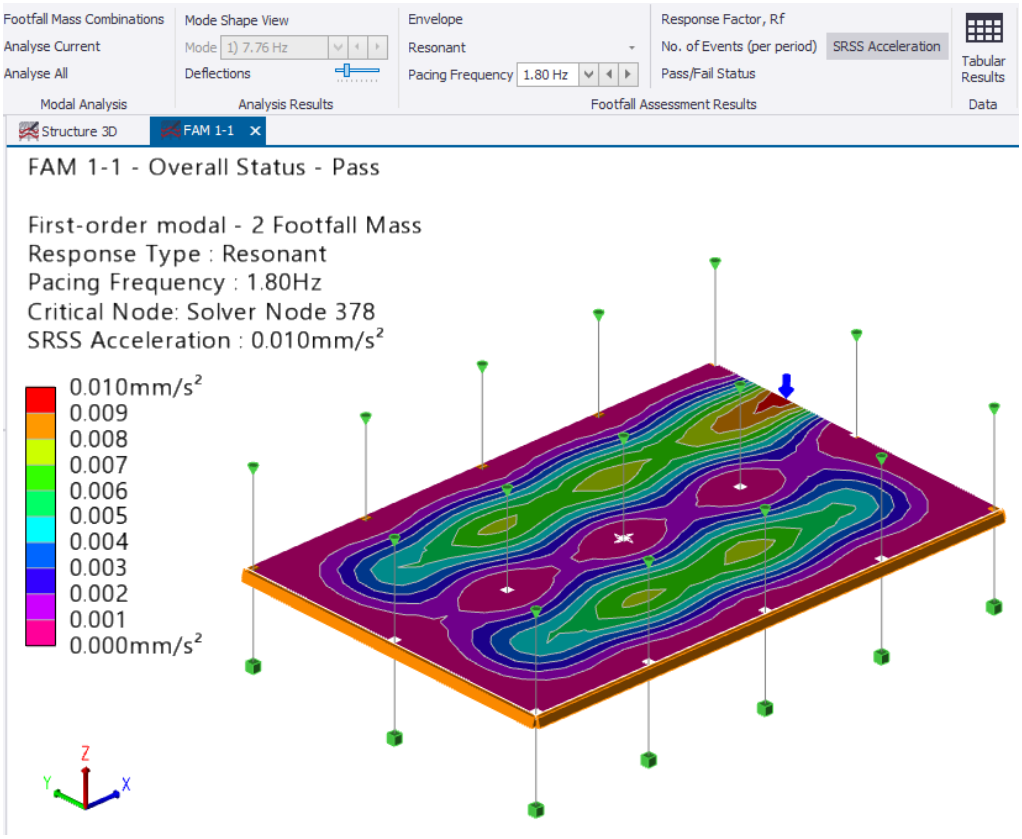


Mode Number	Impulse Force [kN]	Peak Velocity [µm/s]
1	0.0116	64
2	0.0101	323
3	0.0094	4
4	0.0088	152
5	0.0080	200
6	0.0072	3
7	0.0067	20
8	0.0057	178
9	0.0043	652
10	0.0039	38
11	0.0035	0

SRSS Acceleration and SRSS Velocity

To graphically display SRSS Accelerations, return to the Floor Analysis Model (FAM) view and:

1. On the **Footfall Analysis** ribbon, unselect **Envelope** (if selected)
2. Select **Resonant**
3. Select **SRSS Acceleration**

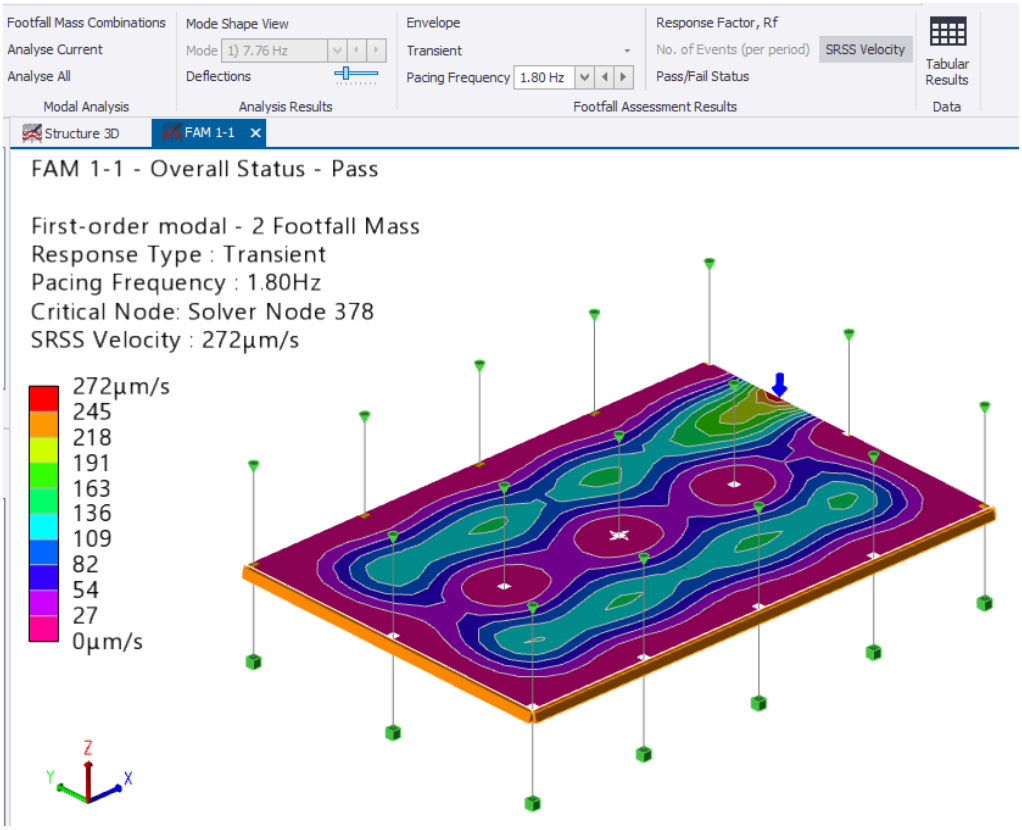


The resonant acceleration is shown and the critical node identified for the Footfall Mass combination selected in the Loading list.

4. To consider all pacing frequencies and mass combinations (if more than one is defined), select **Envelope**

To graphically display SRSS Velocities:

1. On the **Footfall Analysis** ribbon, unselect **Envelope** (if selected)
2. Select **Transient**
3. Select **SRSS Velocity**



The velocities are shown and the critical node identified for the Footfall Mass combination selected in the Loading list.

4. To consider all pacing frequencies and mass combinations (if more than one is defined), select **Envelope**

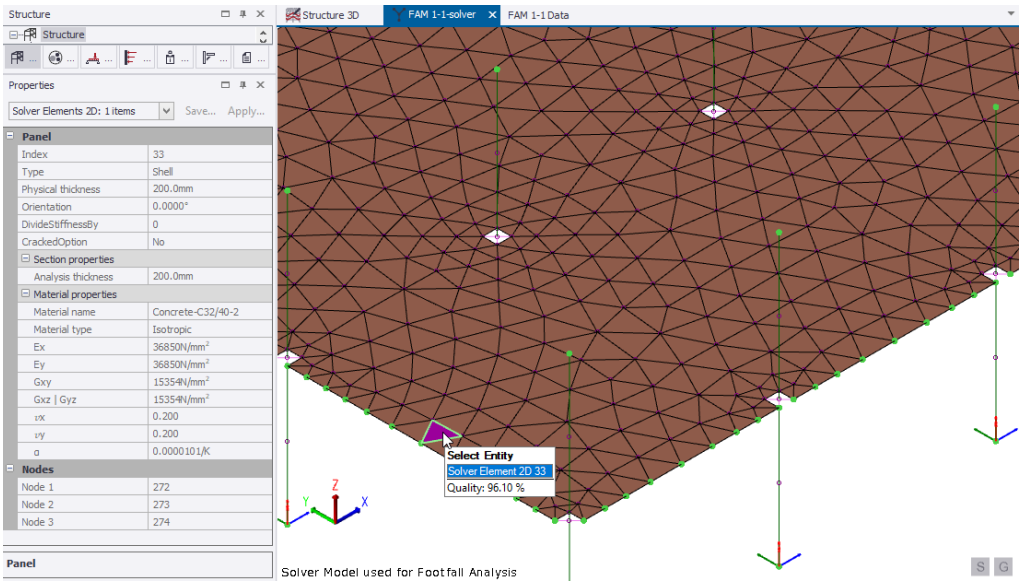
FAM solver model

After the analysis has been performed, if you require you can examine the solver model used for the Footfall Analysis model as follows:

1. Ensure that the required Floor Analysis Model (FAM) view is active.
2. To change the view type of the FAM view, do one of the following:
   - Right-click the FAM **view tab** and in the context menu, select **Solver view**
   - Or, in the **Status bar** at the bottom of the window, click **Solver View**.

   The FAM view changes to an FAM solver view.
3. Right-click anywhere in the solver view.
4. In the context menu, select **Solver models**.
5. In the submenu, select **Solver Model used for Footfall Analysis**.

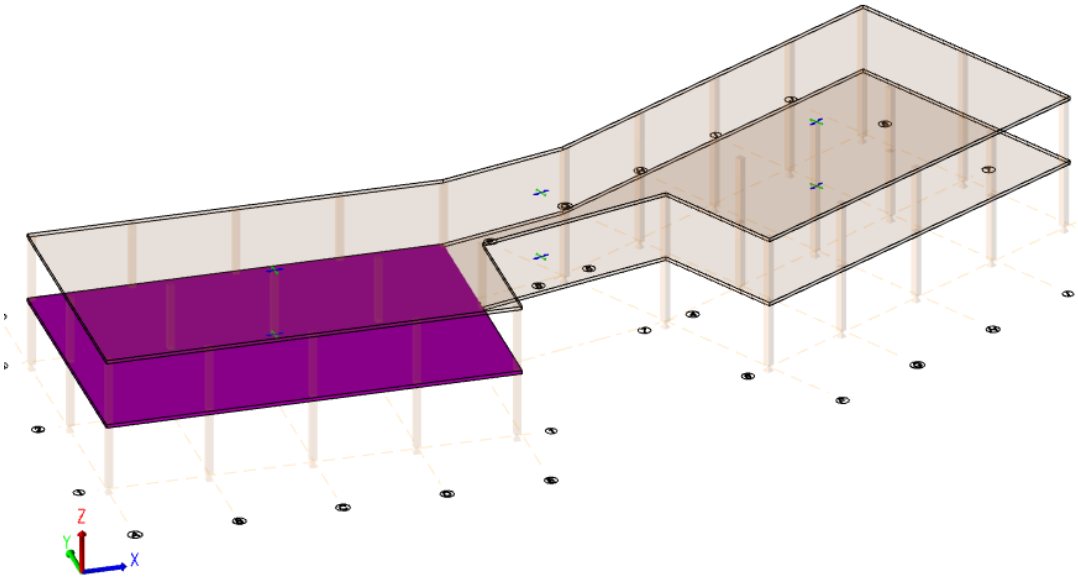
   Tekla Structural Designer opens the selected solver model. You can then select individual elements in the solver model to display their section and material properties in the **Properties** window.



Create additional floor analysis models, edit or delete models

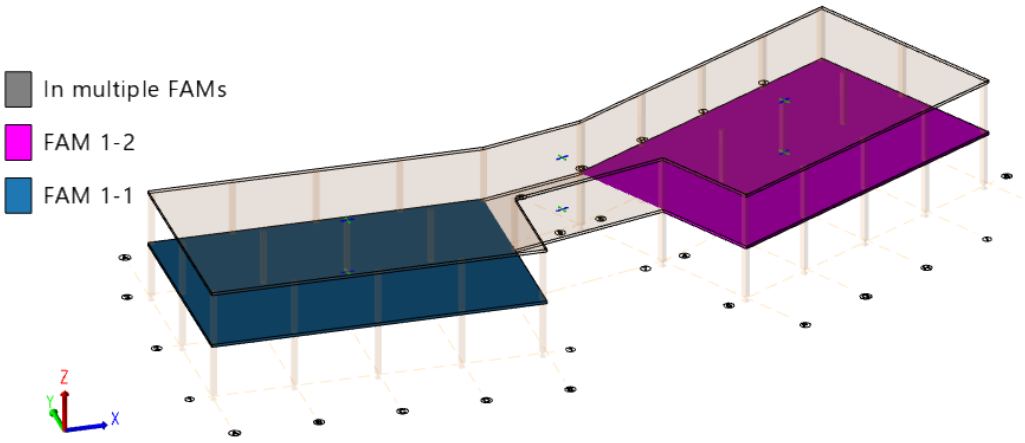
When the **Structure 3D** view is active and set to the **Footfall Analysis View** regime it will display all the floor analysis models that have been created so far.

You can continue to define as many FAM's as you wish in a similar manner, on the same or other levels. Slab items can be in more than one FAM. For example you might create a single FAM for a very large floor area which will expose areas of interest which might then be investigated with other more localized FAMs.



The following options are available:

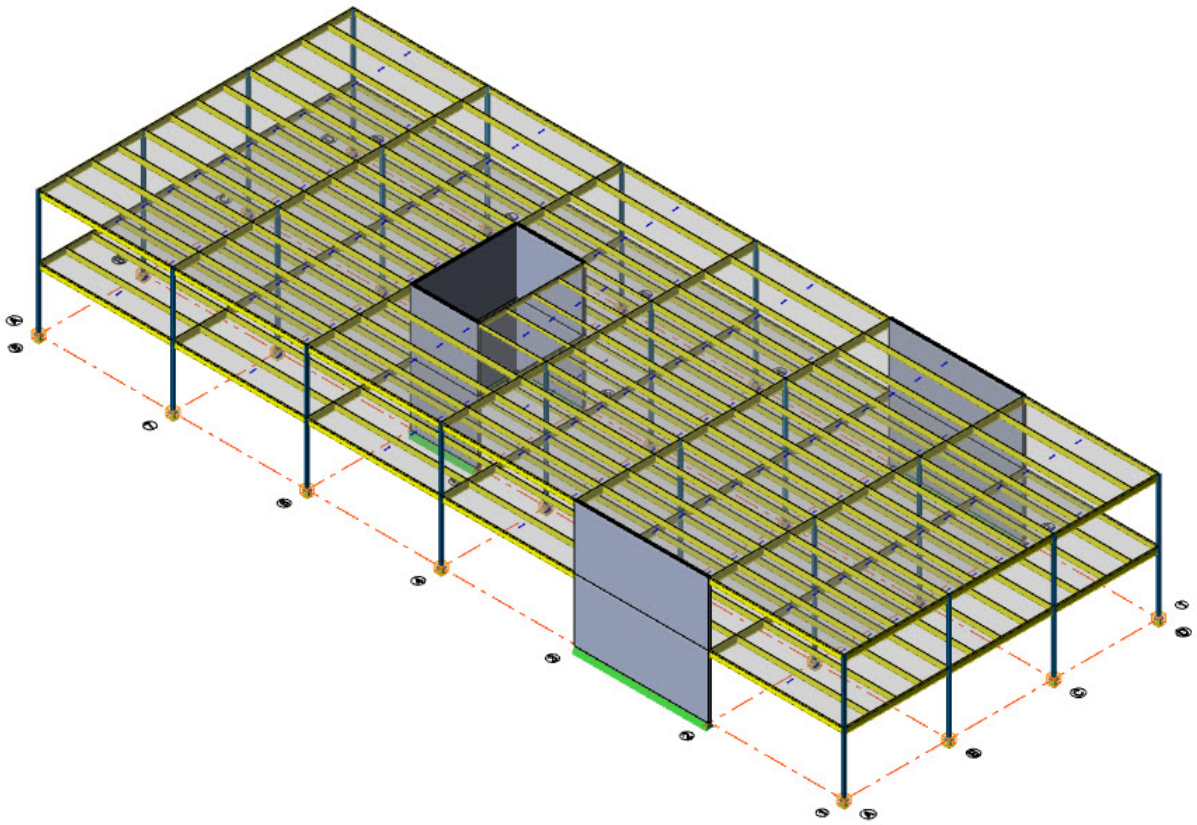
- Click on an existing model in order to edit it in a FAM view.
- Select **New** in the ribbon to start a new model
- Select **Delete** in the ribbon, then click on an existing model in order to delete it.



9.6 Footfall analysis basic workflow - AISC DG11

The guidance in AISC DG11[Ref. 2 (page 698)] is generally based on steel buildings with composite or precast floors. However, the footfall assessment process in Tekla Structural Designer can be performed on models of any material with floors of any construction type. Where the structure is not predominantly a steel building with composite or precast floors, it is the engineer's responsibility to determine the applicability of AISC DG11, and thus the footfall assessment in Tekla Structural Designer which follows this, to their structure.

To illustrate the basic workflow when working to the AISC DG11 design guide, the composite steel beam structure shown below will be used.



Footfall assessment in Tekla Structural Designer

Download and open the tutorial model

1. Download the tutorial model zip file Footfall - AISC DG11.zip from here.
2. Extract the following tutorial model from the zip:

- Footfall - AISC DG11.tsmd

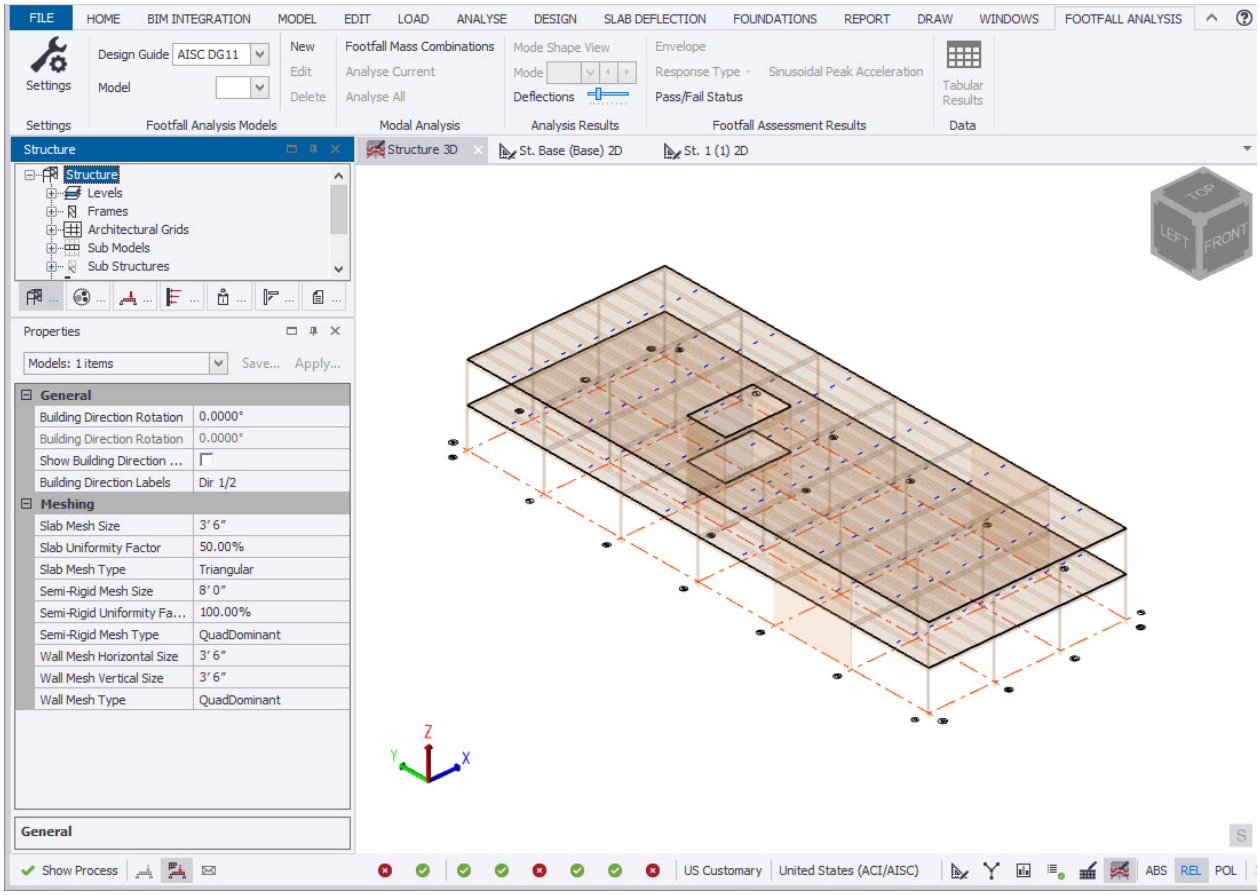
3. Open the model in Tekla Structural Designer.

Open a footfall analysis view

1. Open a **Structure 3D** Scene view.
2. In the **Status bar** at the bottom of the main window, click the **Footfall Analysis View** regime button.



The **Footfall Analysis** ribbon becomes available, and the active view switches to a Footfall analysis view.



TIP In any view you can simply right-click a slab item and select **Create Floor Analysis Model** from the context menu in order to begin

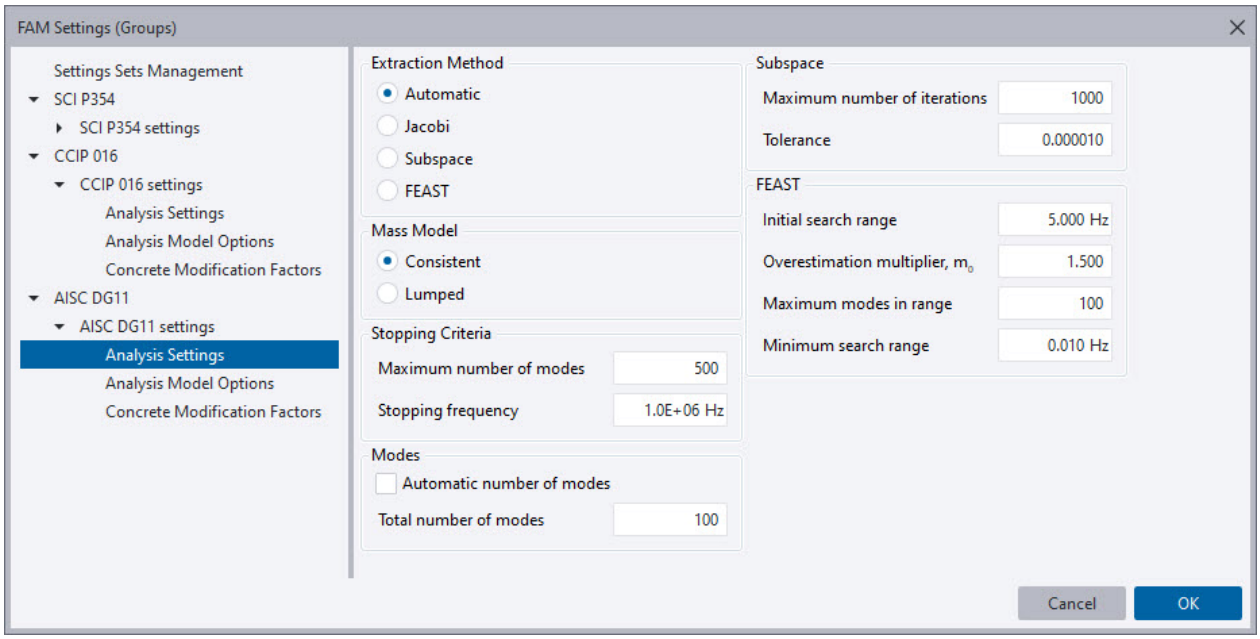
creating a new FAM. It is not essential for you to switch to the **Footfall Analysis View** regime first.

Confirm the floor analysis model settings

1. On the **Footfall Analysis** ribbon, select **Settings**

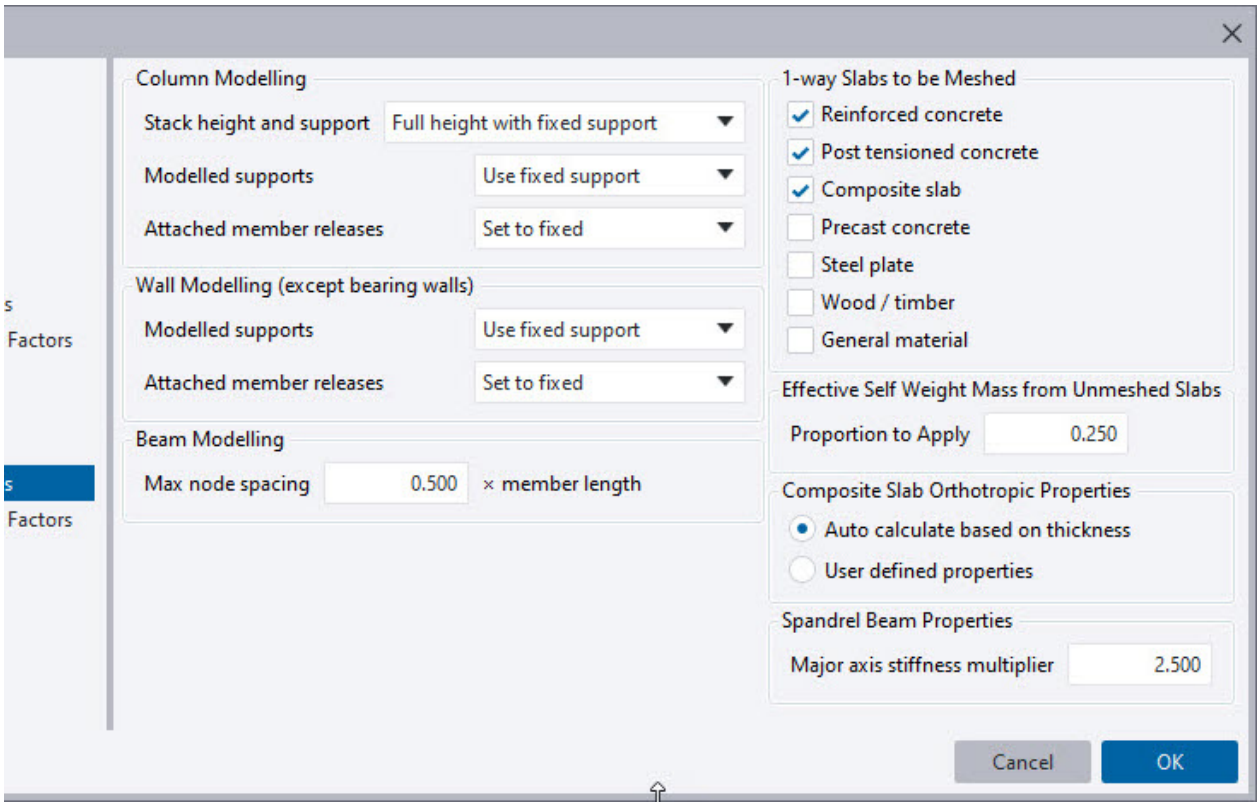
   The available settings sets for each design guide are displayed.

2. In the left hand pane, select **AISC DG11 > AISC DG11 settings > Analysis Settings**



It is anticipated that most users will not need to adjust the default analysis settings which are as shown above.

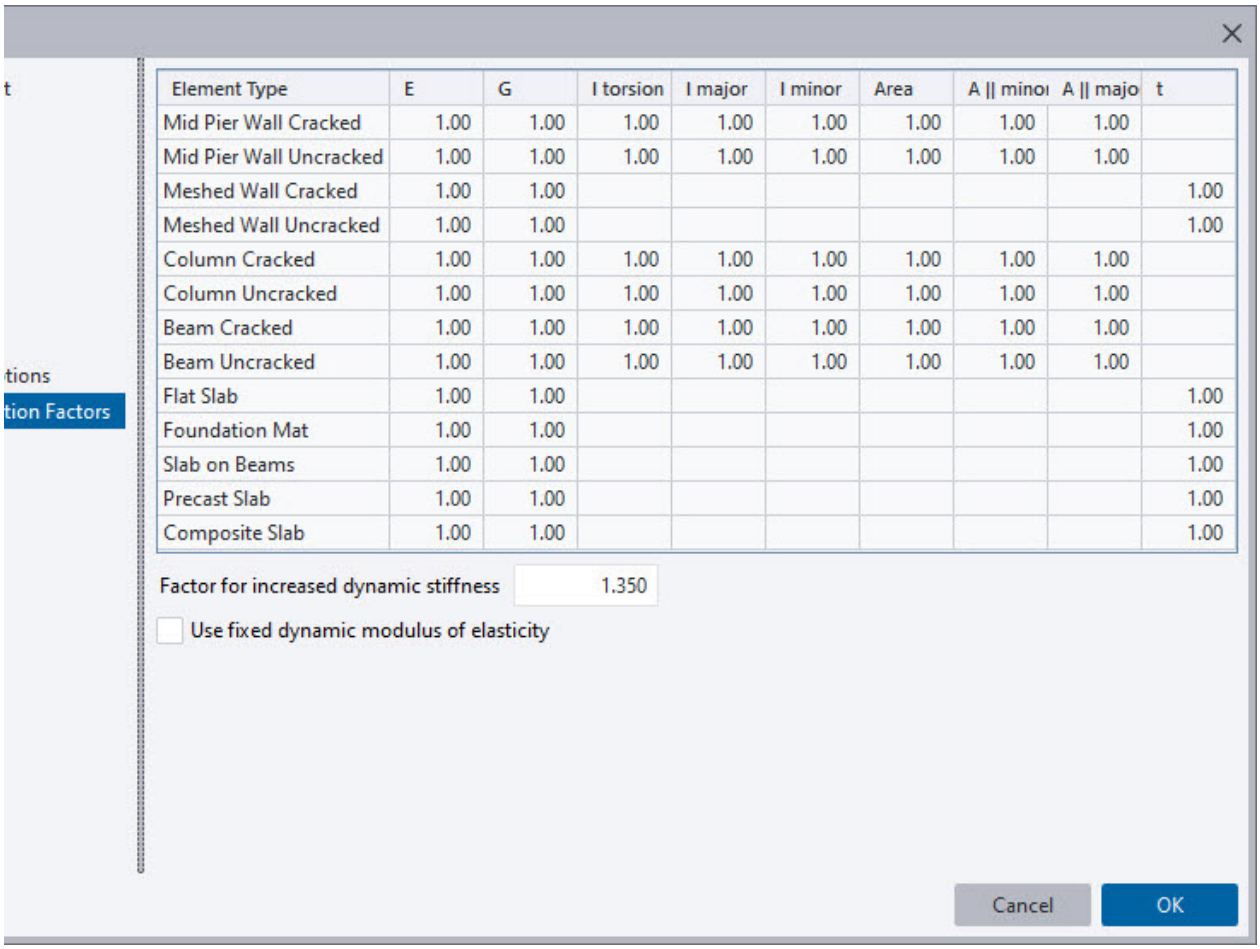
3. In the left hand pane, select **AISC DG11 > AISC DG11 settings > Analysis Model Options**



These options allow you to configure the FAM to your requirements.

Note that as our example model contains 1-way spanning composite slabs, by accepting the default options shown above these will be modeled as 2-way meshed slabs with automatically calculated orthotropic properties.

4. In the left hand pane, select **AISC DG11 > AISC DG11 settings > Concrete Modification Factors**

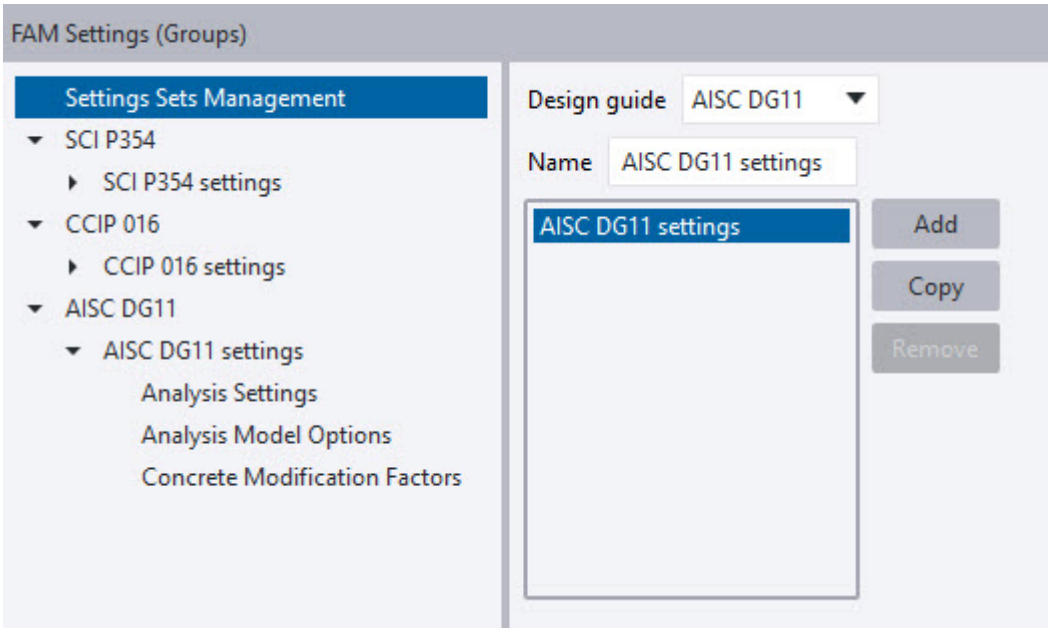


Note that by accepting the defaults shown above the static concrete modulus of elasticity will be factored by 1.35 to account for the greater stiffness of concrete in a composite floor under dynamic loading, as recommended in AISC DG11.

For more information on the different settings, options and factors, see: Footfall assessment settings (page 683)

5. In the left hand pane, select **Settings Sets Management**

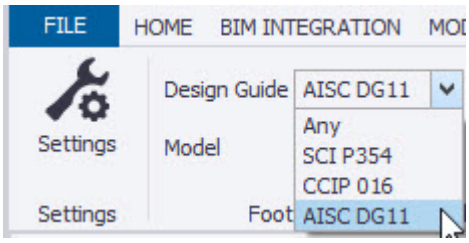
Although It is anticipated that the previously displayed settings will be appropriate in most cases, you can add further sets to investigate different configurations if you require. This would be achieved after selecting the AISC DG11 design guide via the **Add** or **Copy** buttons shown below.



6. We don't need to add a further set so click **OK** to exit.

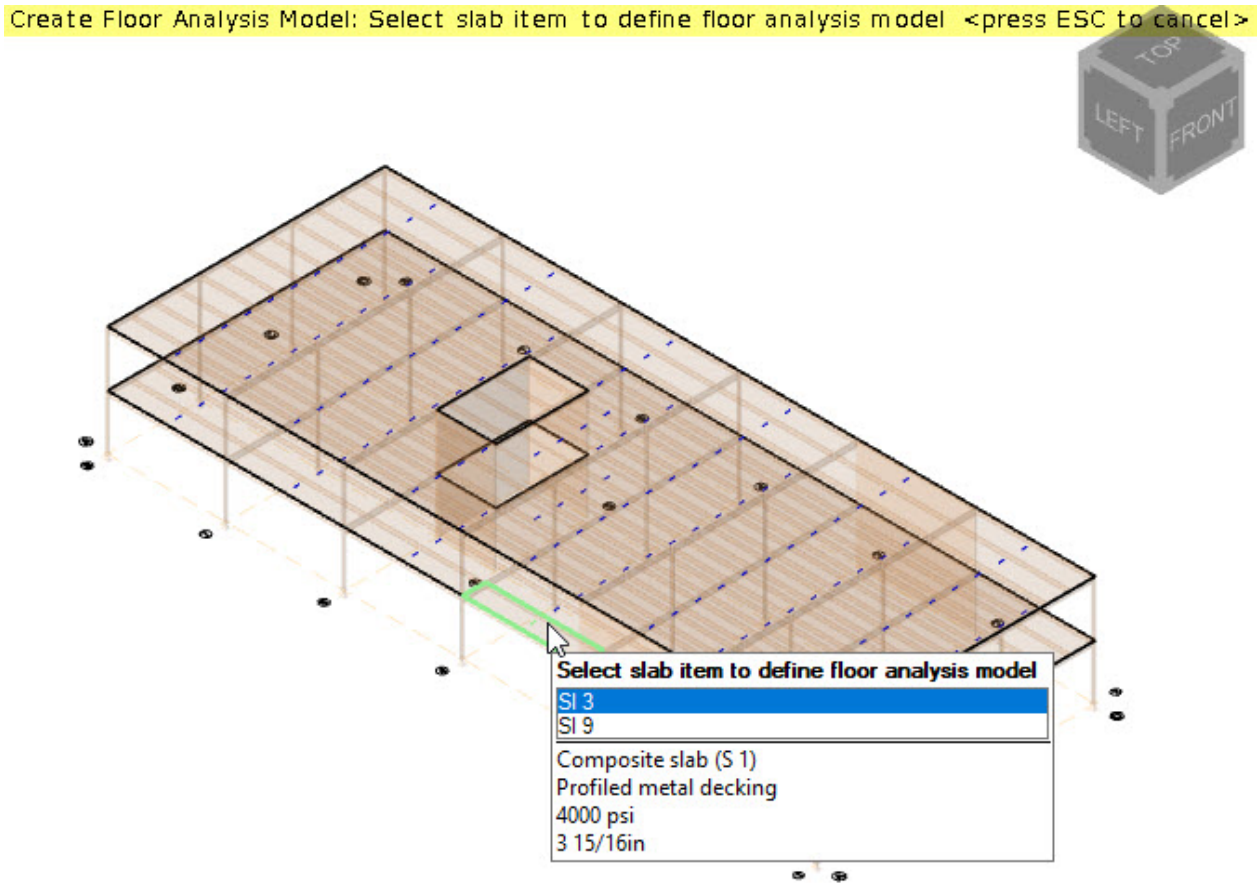
Create a floor analysis model

1. On the **Footfall Analysis** ribbon, in the **Design Guide** list, ensure AISC DG11 is selected.

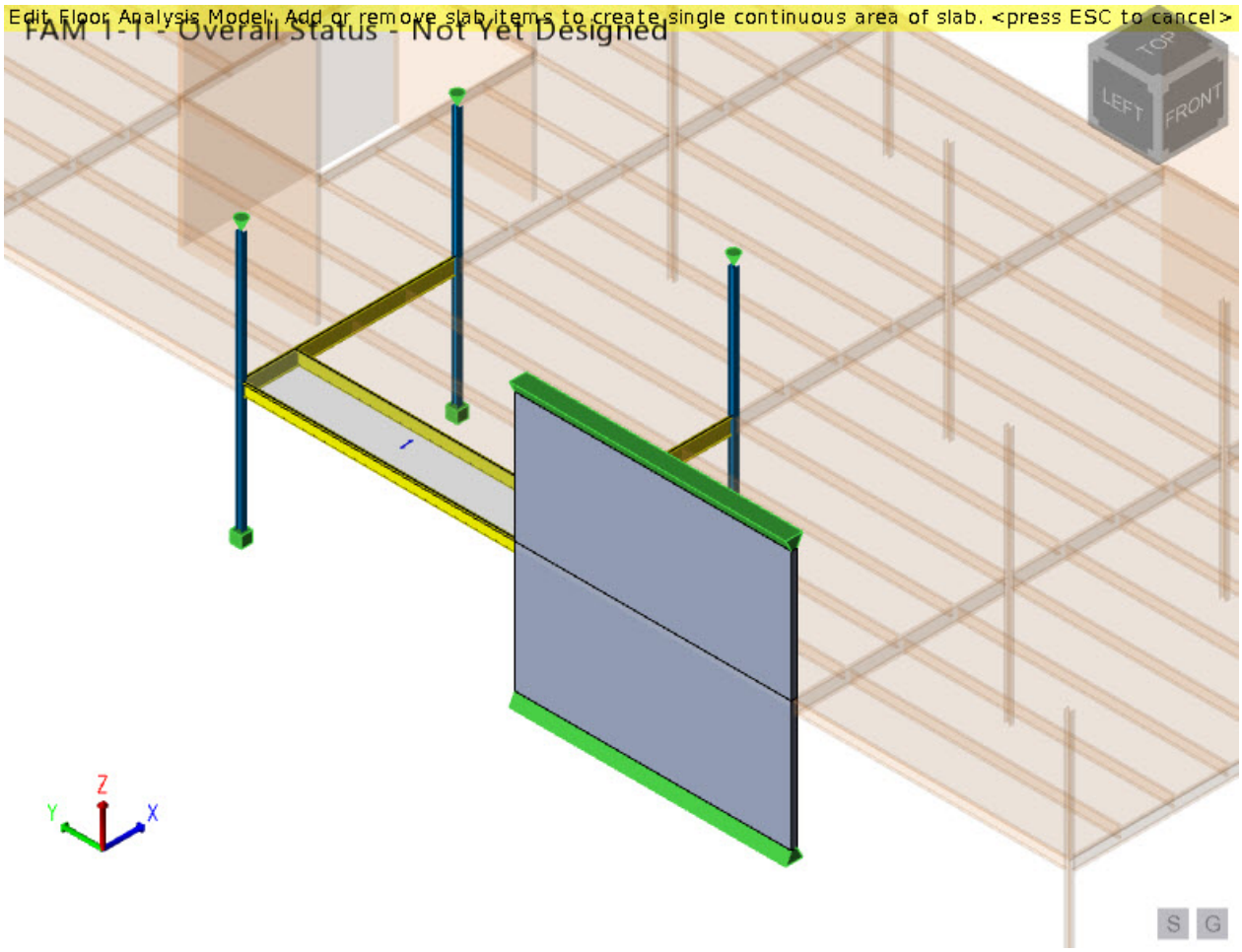


2. On the **Footfall Analysis** ribbon, select **New**

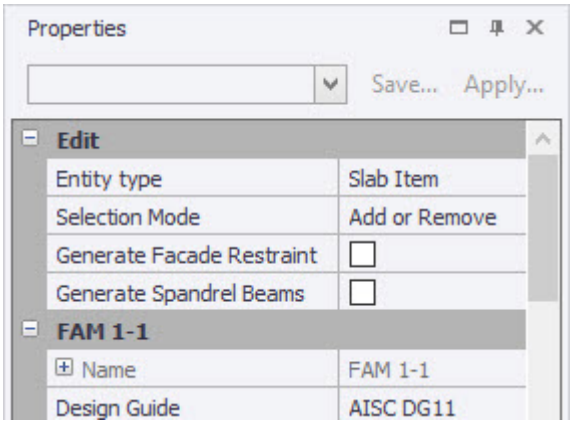
3. Select the slab item shown below.



A new Floor Analysis Model (FAM) view opens in Edit mode and the are displayed in the **Properties** window.



When in Edit mode, editing commands are available at the top of the **Properties** window which enable slab items and other members to be added or removed from the FAM.



Add slabs (and attached members)

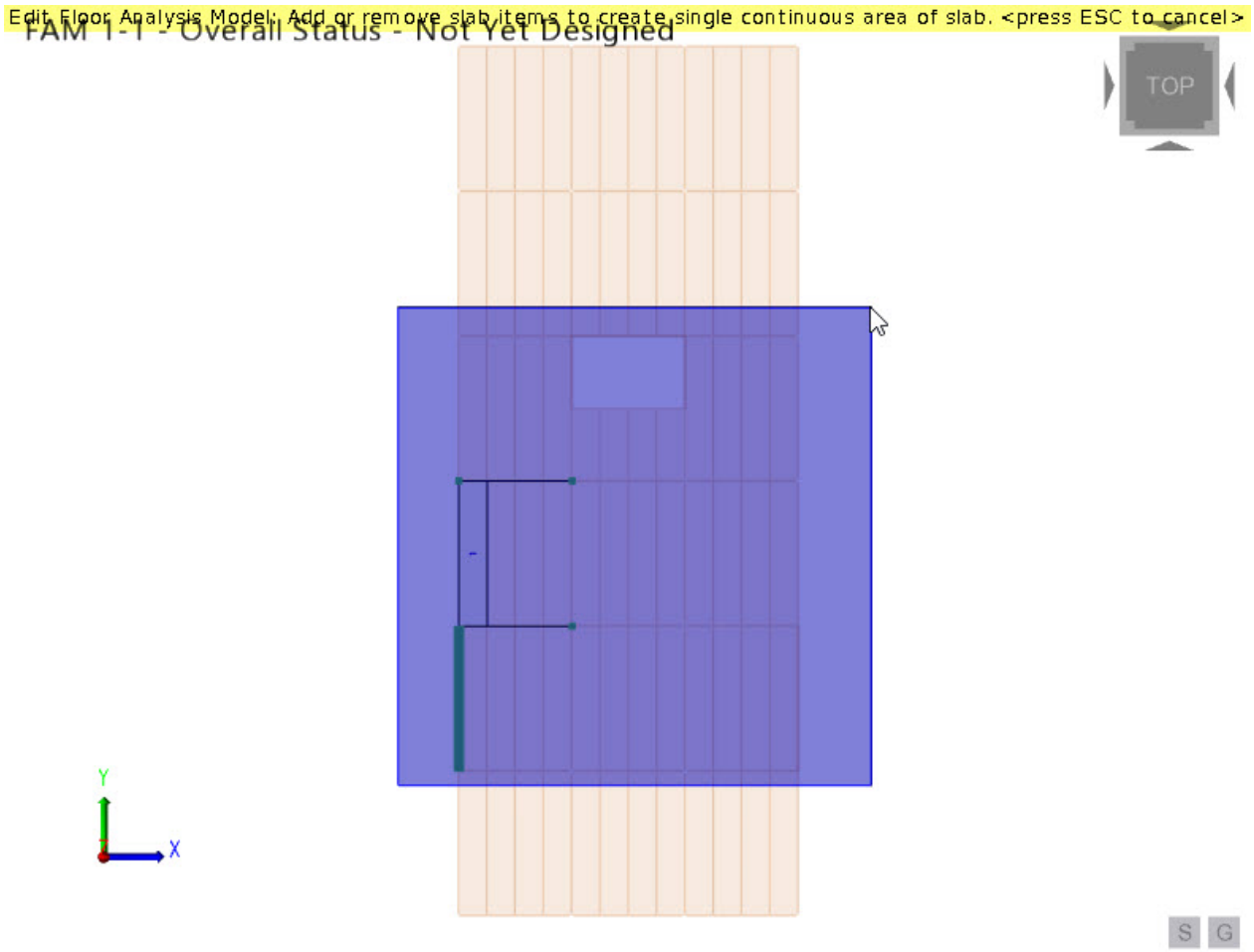
NOTE DG11 recommends that the FAM be limited in its size and indicates that considering a large floor area might be unconservative. For a slab item of interest the FAM should include only those slabs immediately adjacent. So for a regular building a 'middle' slab would create a model of 9 similar slabs including the one of interest. This is entirely at the user's control in Tekla Structural Designer.

Additional slab items can be added either by clicking them individually, or by box selecting. Note that the **Selection Mode** in the **Properties** window, which defaults to 'Add or Remove' can also be set to 'Add Only' or 'Remove Only' to assist this process.

For this example we want the analysis model to only comprise the central area of floor. The quickest way to achieve this is as follows:

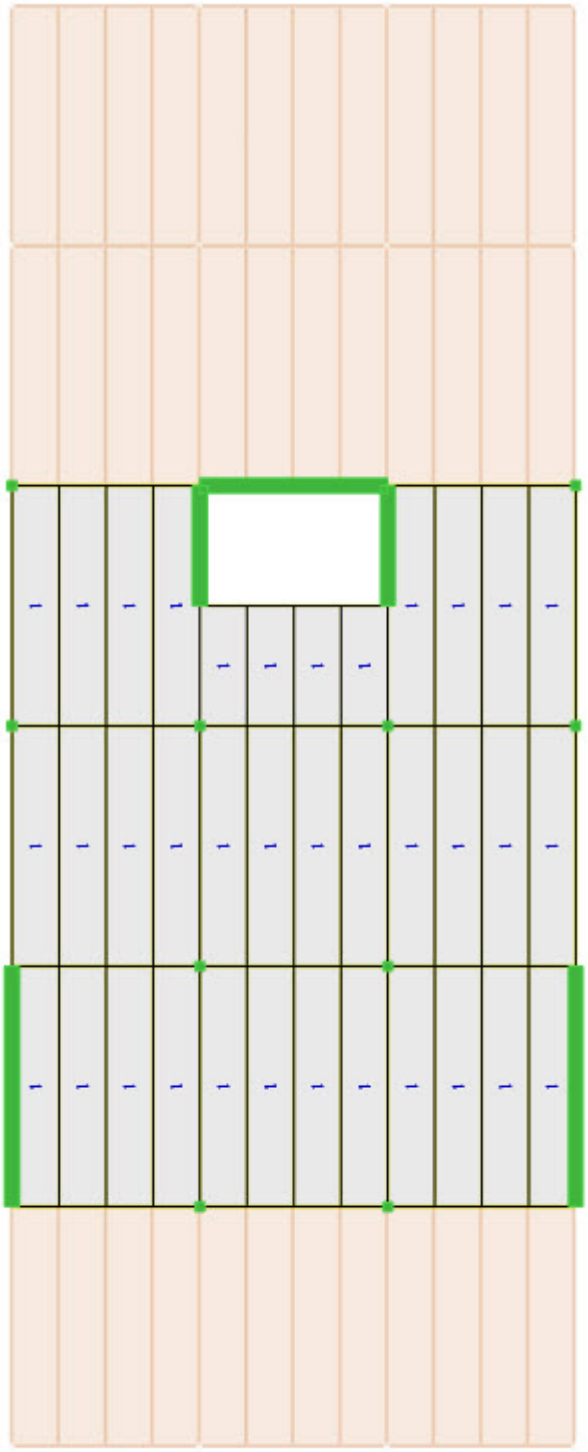
1. In the **Properties** window, set the Selection Mode to 'Add Only'
2. Use the **ViewCube** to switch to a top view.

3. Drag a box from left to right around the slab items as shown:



Any slab items totally enclosed by the box are added to the model along with any attached members/walls.

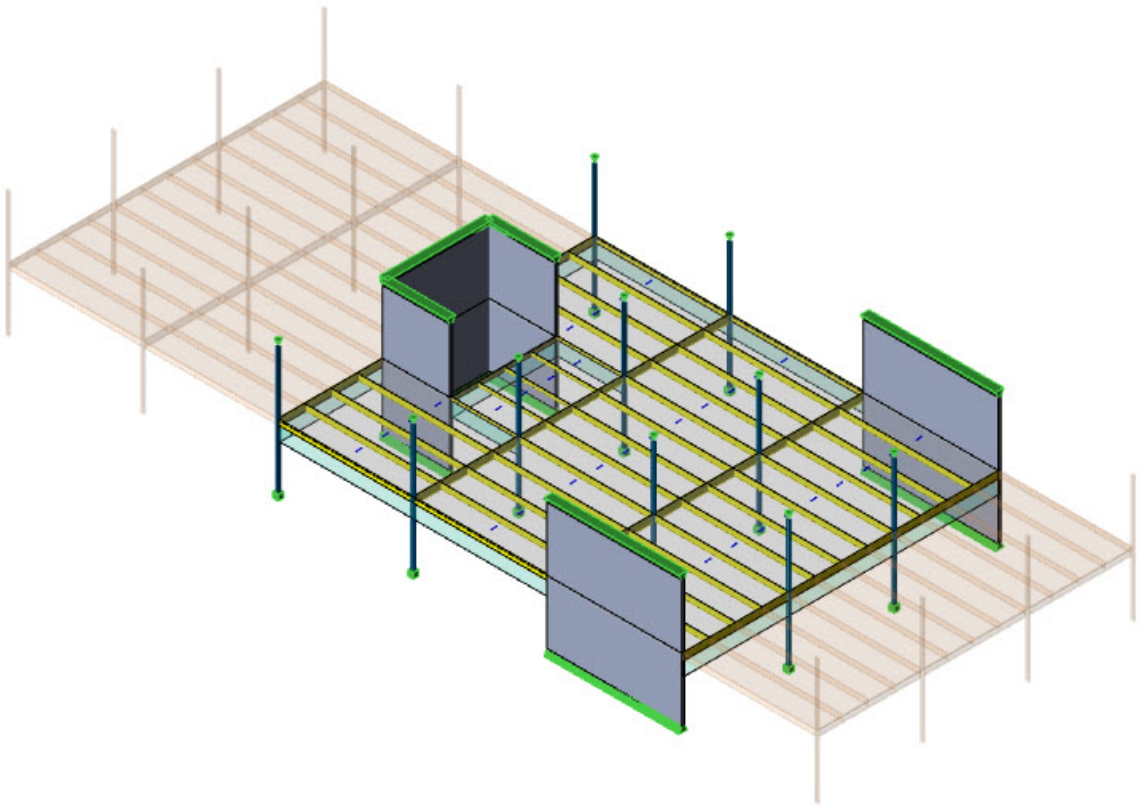
The model should now look as follows:



Add and edit spandrel beams

Spandrel beams can be used to model the restraining effect of cladding on the building perimeter.

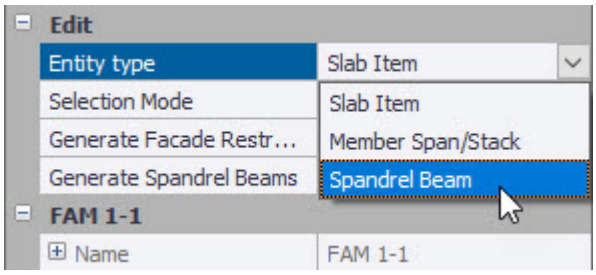
1. Use the **ViewCube** to switch back to an isometric view.
2. In the **Properties** window, select **Generate Spandrel Beams**



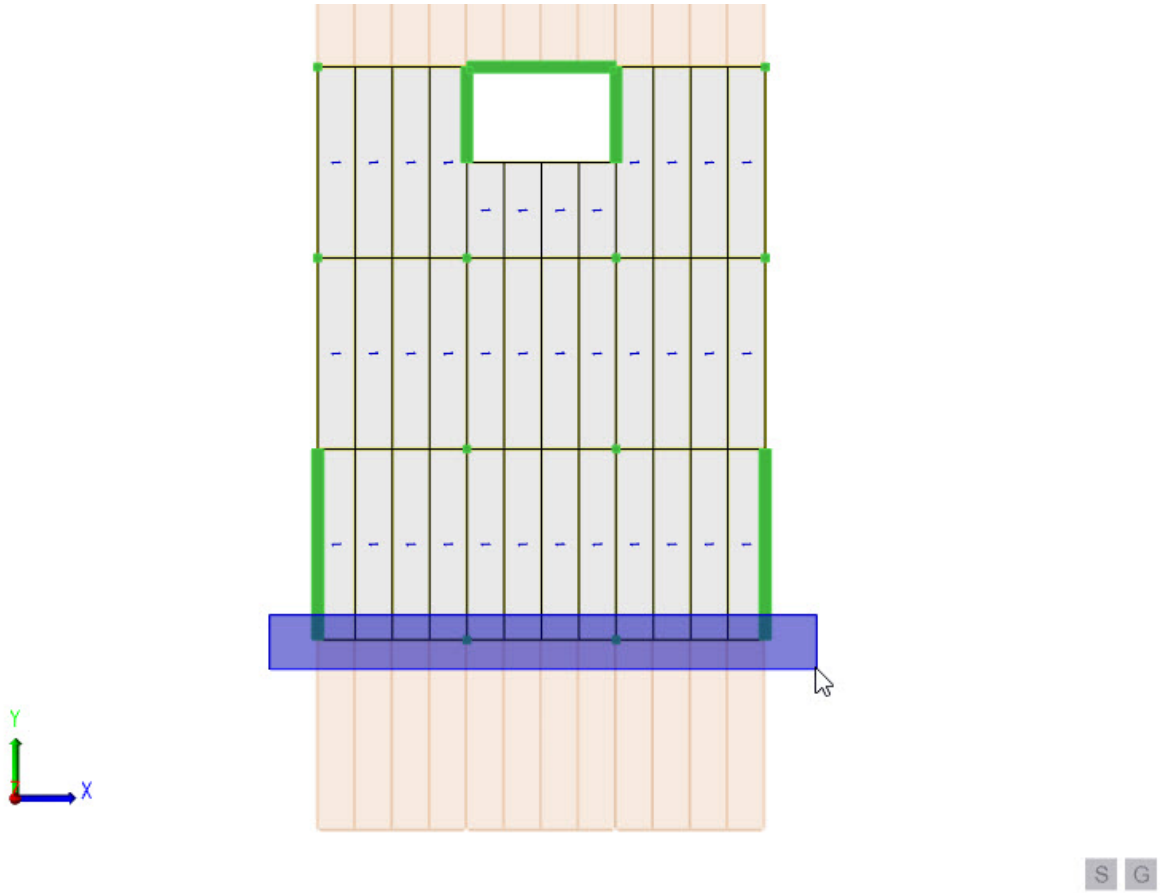
As shown above, all beams around the perimeter of the FAM are initially marked as spandrel beams. Some editing may be required if the default marking is inappropriate.

In this case we will remove the spandrel beam marking from those beams that aren't actually on the building edge as follows.

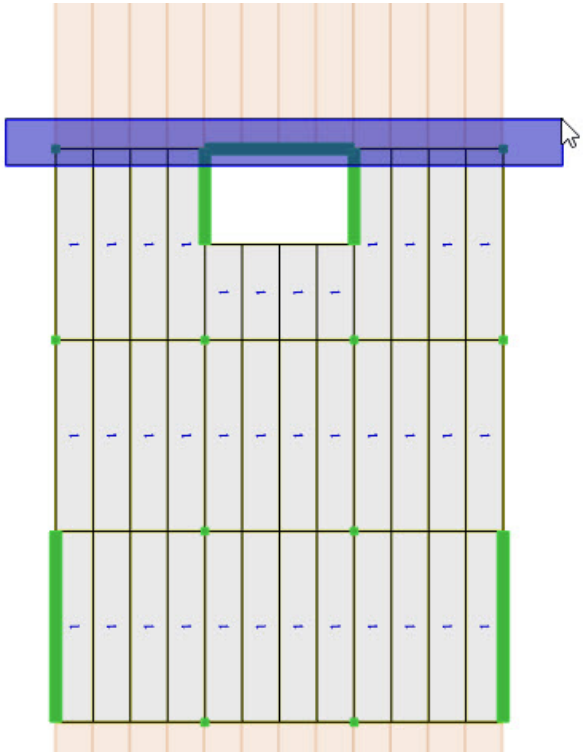
1. In the **Properties** window, change the **Entity type** to 'Spandrel Beam'.



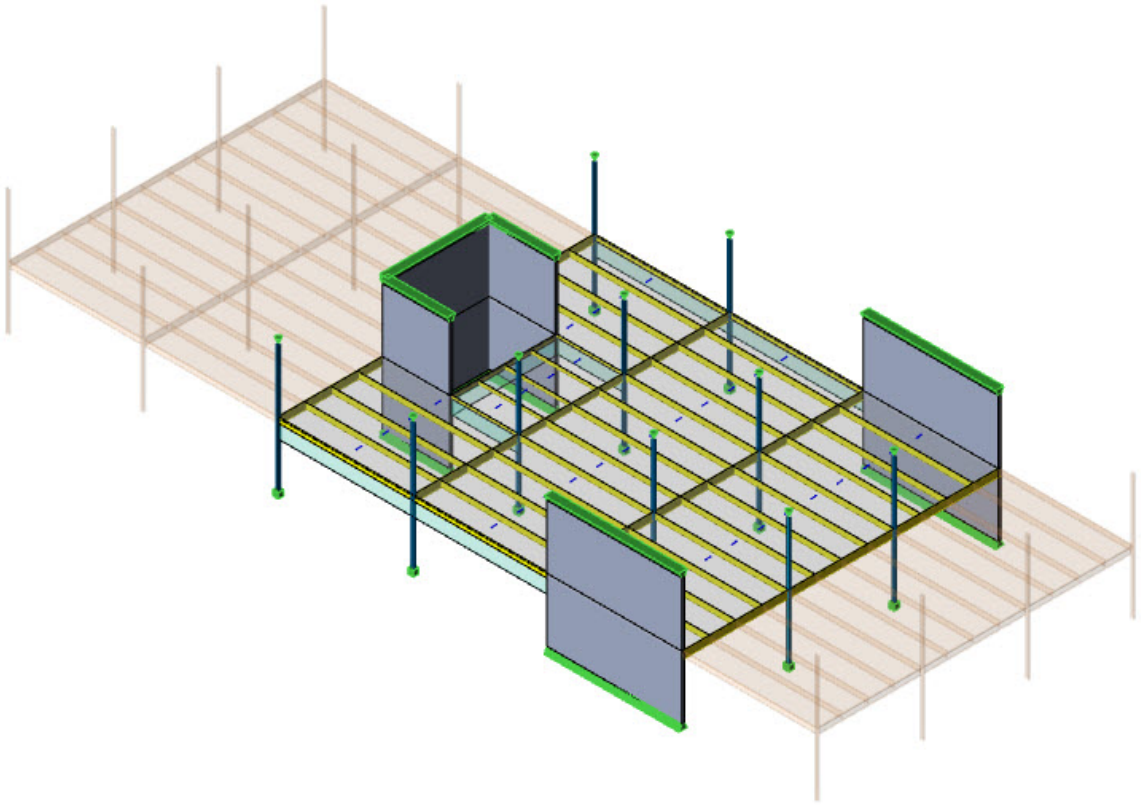
2. Set the **Selection Mode** to 'Remove Only'.
3. Use the **ViewCube** to switch back to a top view.
4. Drag a box from left to right around the beams along the bottom edge of the FAM as shown:



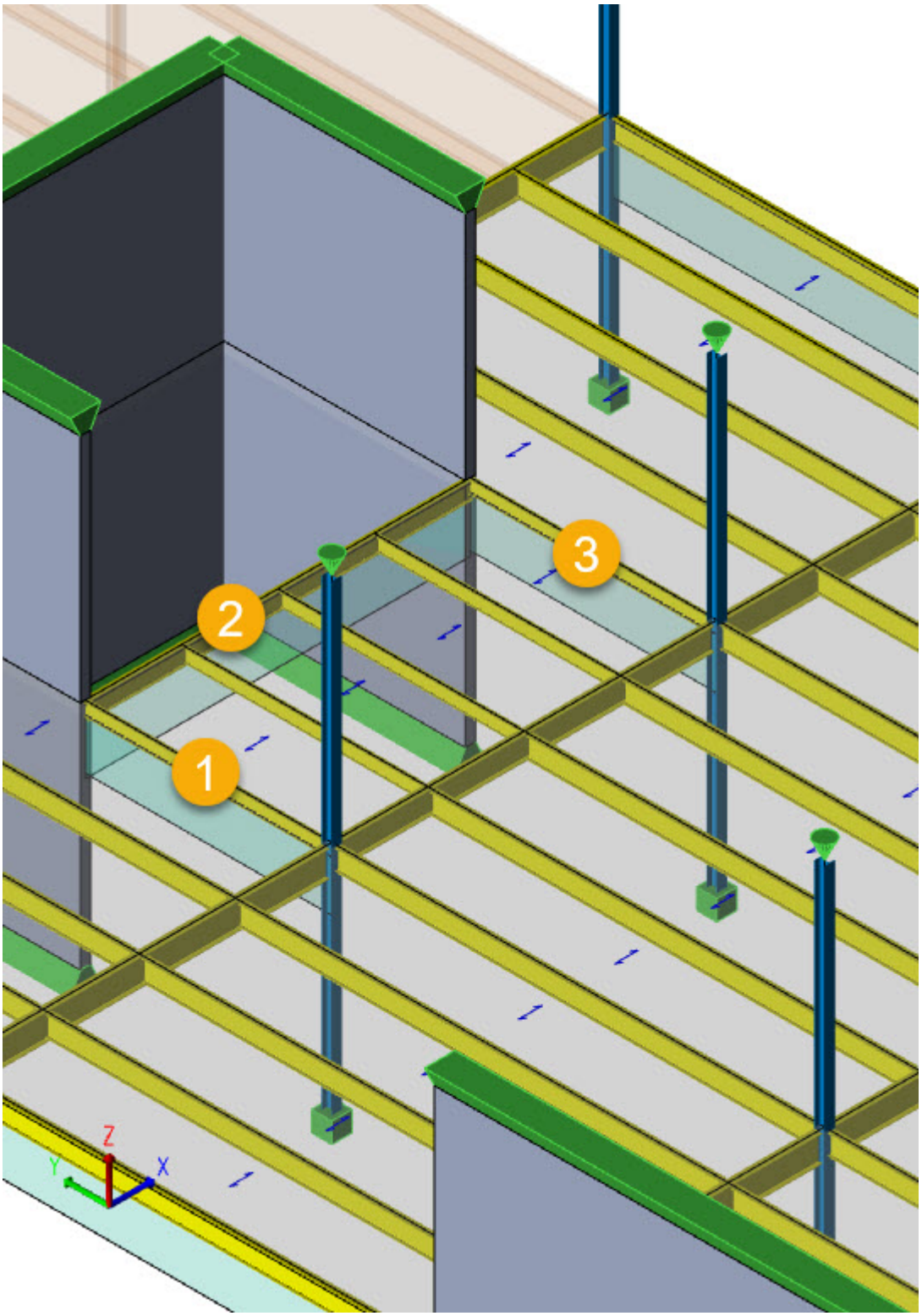
5. Drag a similar box around the spandrel beams along the top edge of the FAM.



6. Use the **ViewCube** to switch back to an isometric view.



7. There are still three remaining spandrel beam markings adjacent to the lift core opening. Click on these individually to remove them.

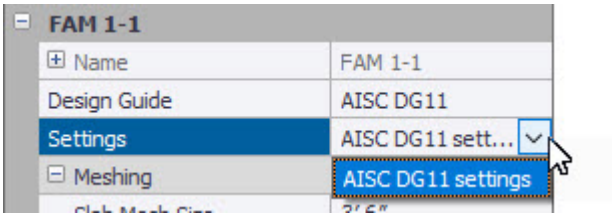


8. The FAM model is now complete, so press ESC on the keyboard to exit Edit mode.

Confirm the design guide, settings and meshing

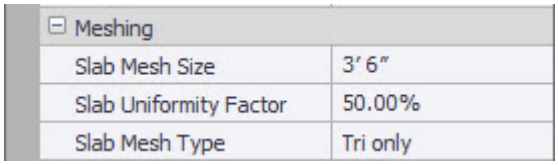
Review the following properties located at the top of the **Properties** window.

1. Ensure the **Design Guide** 'AISC DG11' and the **Settings** set 'AISC DG11 settings' are both selected.



   In a real model you could have created multiple FAM Settings sets, in which case you could then choose which set to apply.

2. Review the meshing properties.



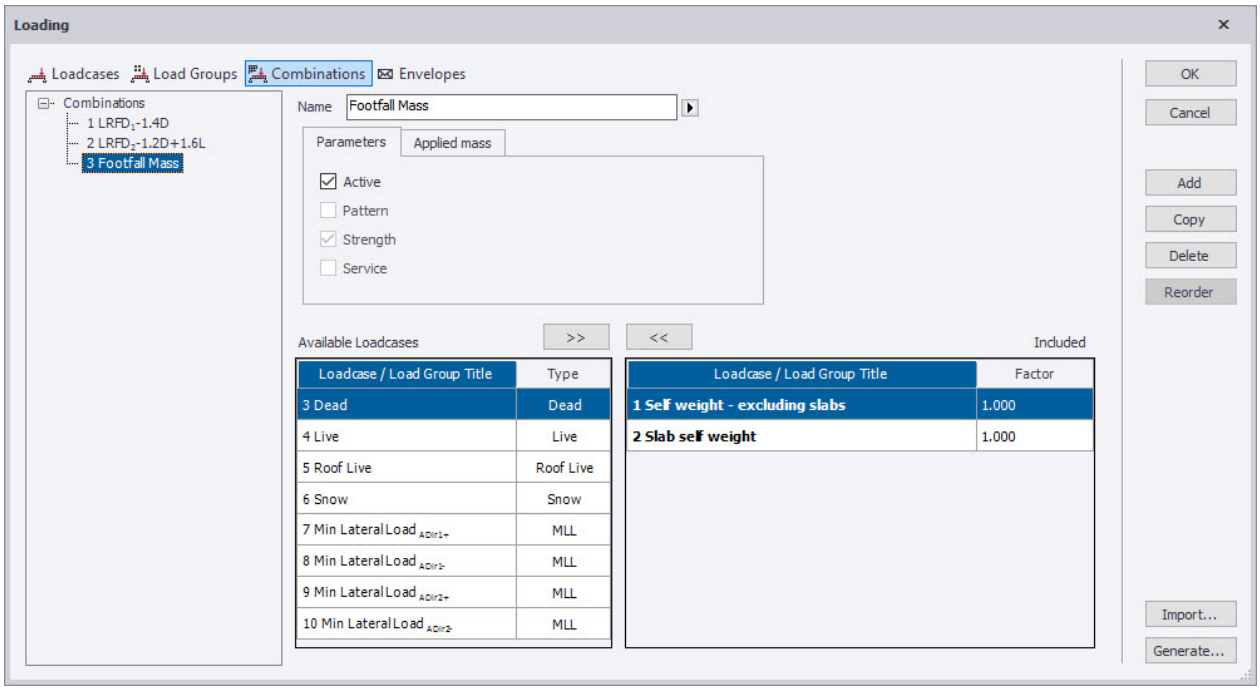
   The mesh defaults are reasonable and will be left unchanged.

NOTE DG11 provides some guidance on meshing. It recommends 1/10 th of the bay size and suggests not to over-refine the mesh as the natural frequency calculation is only accurate to within 5 to 10% anyway. If refinement is felt necessary then that process should stop when change in the natural frequencies is no larger than 0.05 to 1.0 Hz

---

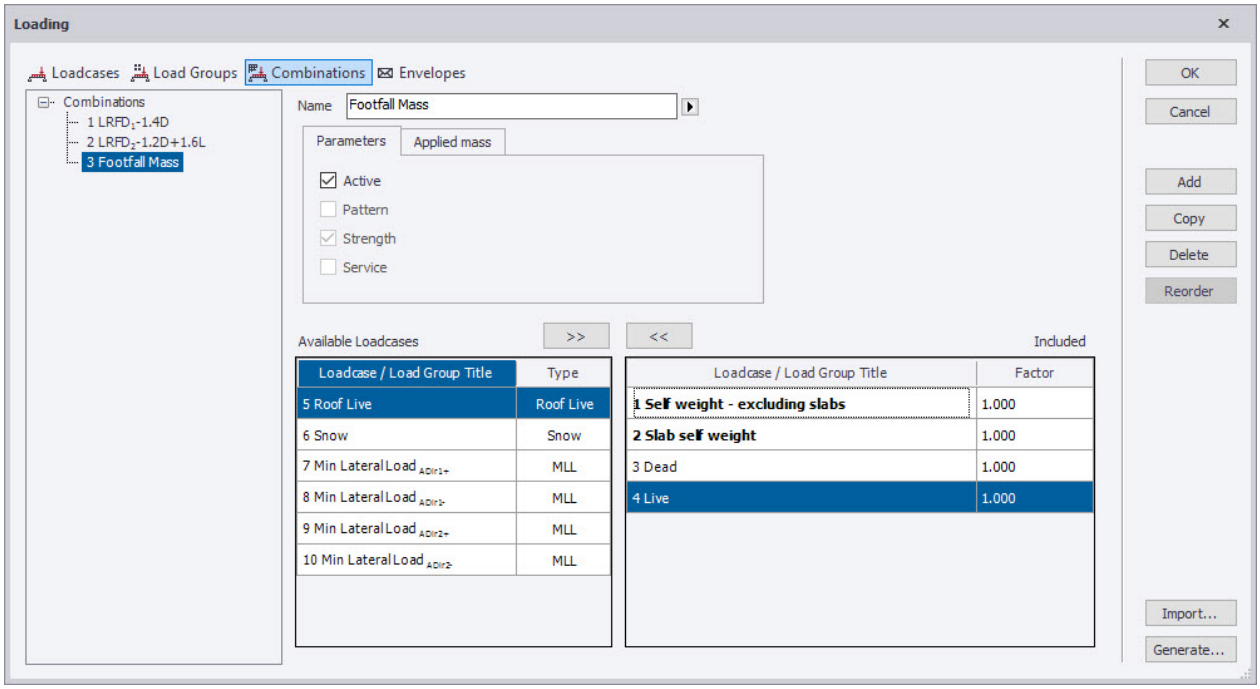
Create a footfall mass combination

1. On the **Footfall Analysis** ribbon, select **Footfall Mass Combinations**.



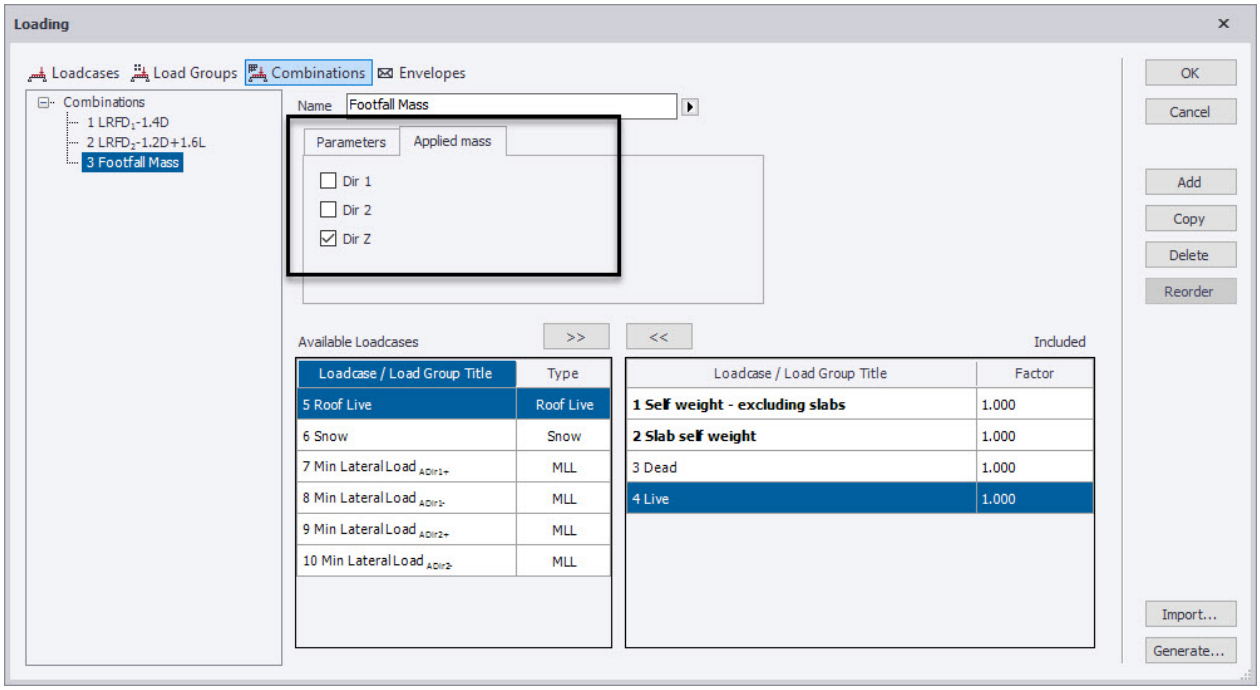
Any existing combinations for static design will be listed in this dialog followed by a **Footfall Mass** combination which is selected as shown above. Two loadcases have automatically been included in this combination.

2. Include the **Dead** and the **Live** loadcases.



3. Click the **Applied mass** tab in the dialog.

   No changes are required here, but note that for Footfall mass combinations, the applied mass defaults to being applied in the Z direction only.



4. Click **OK** to save the footfall mass combination.

NOTE In real models engineering judgment will be required when defining the footfall mass combination. The list of available loadcases can potentially be very long and you will also need to consider if the load factors are appropriate. Certain recommendations are given in DG11 section 3.3.

Analyse the model and view the results

On the **Footfall Analysis** ribbon, there are two analysis choices:

- **Analyse Current** - this runs a first-order modal analysis of **the currently selected FAM** for all active footfall mass combinations.
- **Analyse All** - this runs a first-order modal analysis of **all FAMs** for all active footfall mass combinations.

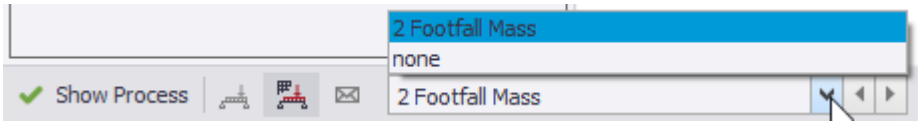
As only one FAM has been created at this stage, it doesn't actually matter which of the above two options is selected.

1. Click **Analyse Current**

View the mode shapes

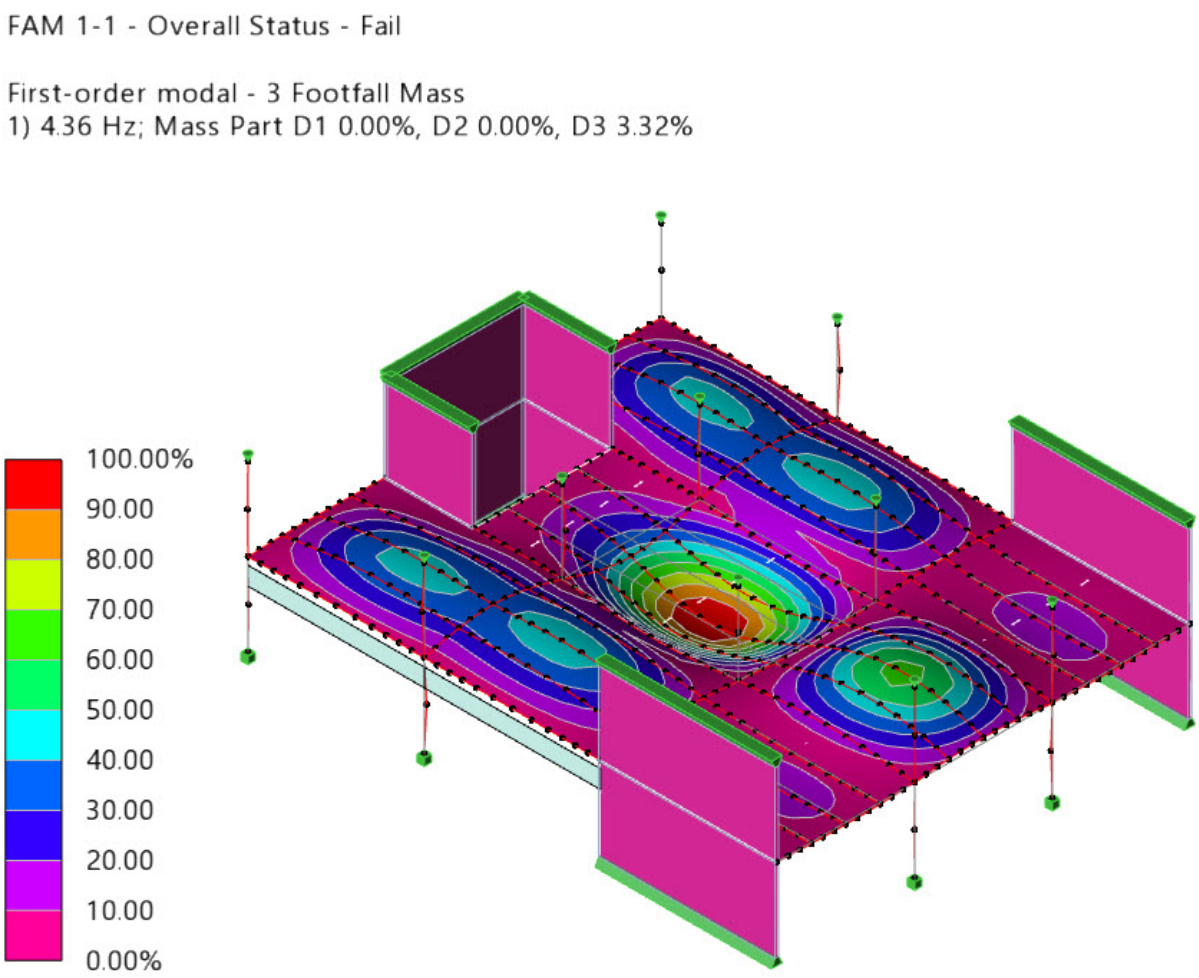
To view mode shapes graphically for the footfall mass combination:

1. Select the **Footfall Mass** combination in the Loading list.

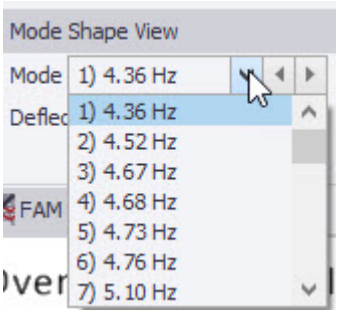


The mode shapes can then be displayed graphically.

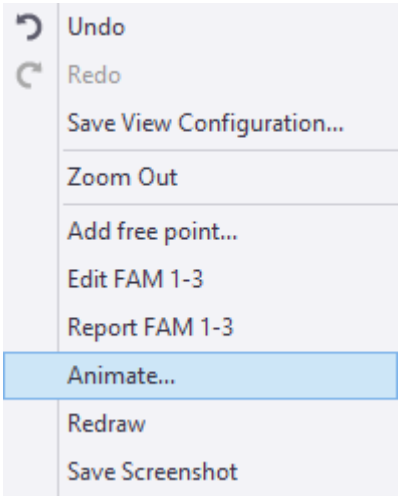
2. On the **Footfall Analysis** ribbon, select **Mode Shape View**.



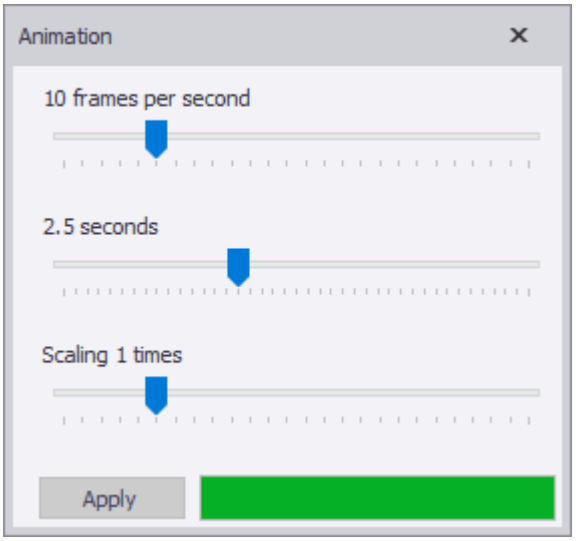
3. Cycle through the modes by using the Next and Previous buttons, or by selecting from the **Mode** list.



4. To adjust the amplitude of the selected mode shape, drag the **Deflections** slider in the ribbon as required.
5. To animate the selected mode shape, right-click in the view, and select **Animate** from the context menu.



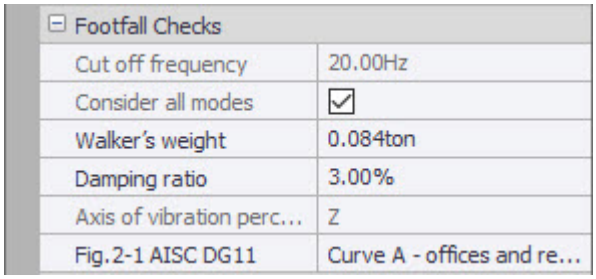
6. To end the animation, close the Animation dialog.



Configure the footfall checks

Default footfall check parameters for all models can be configured on the Footfall Assessment > AISC DG11 > Footfall Checks (page 696) page in global settings.

These defaults are automatically copied into the **Footfall Checks** section of the **Properties** window for each FAM that is created.



The following parameters are shown in gray because they are fixed and cannot be edited:

- Cut off frequency (20Hz)
- Consider all modes (selected)
- Axis of vibration perception (Z)

The following parameters are editable:

- walker's weight (default 168 lbs for US customary units and 76 kg for metric units)

- damping ratio (default 3%)
- Fig.2-1 AISC DG11 curve (default Curve A)

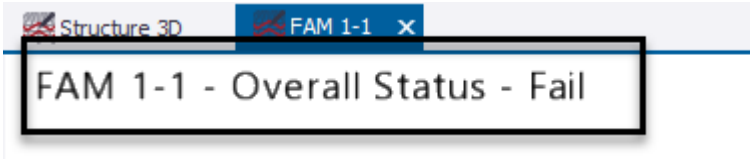
For more information about the above parameters, see: Analysis post-processing - AISC DG11 (page 567)

The default values should be adjusted as required to suit the individual FAM, however in this example we shall simply accept all the defaults.

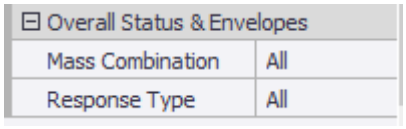
If you were to change any of the above values, the Footfall Assessment results would update automatically - there is no need to re-perform the analysis.

View the footfall check overall results

The footfall checks overall status is displayed in the top left corner of the FAM view.



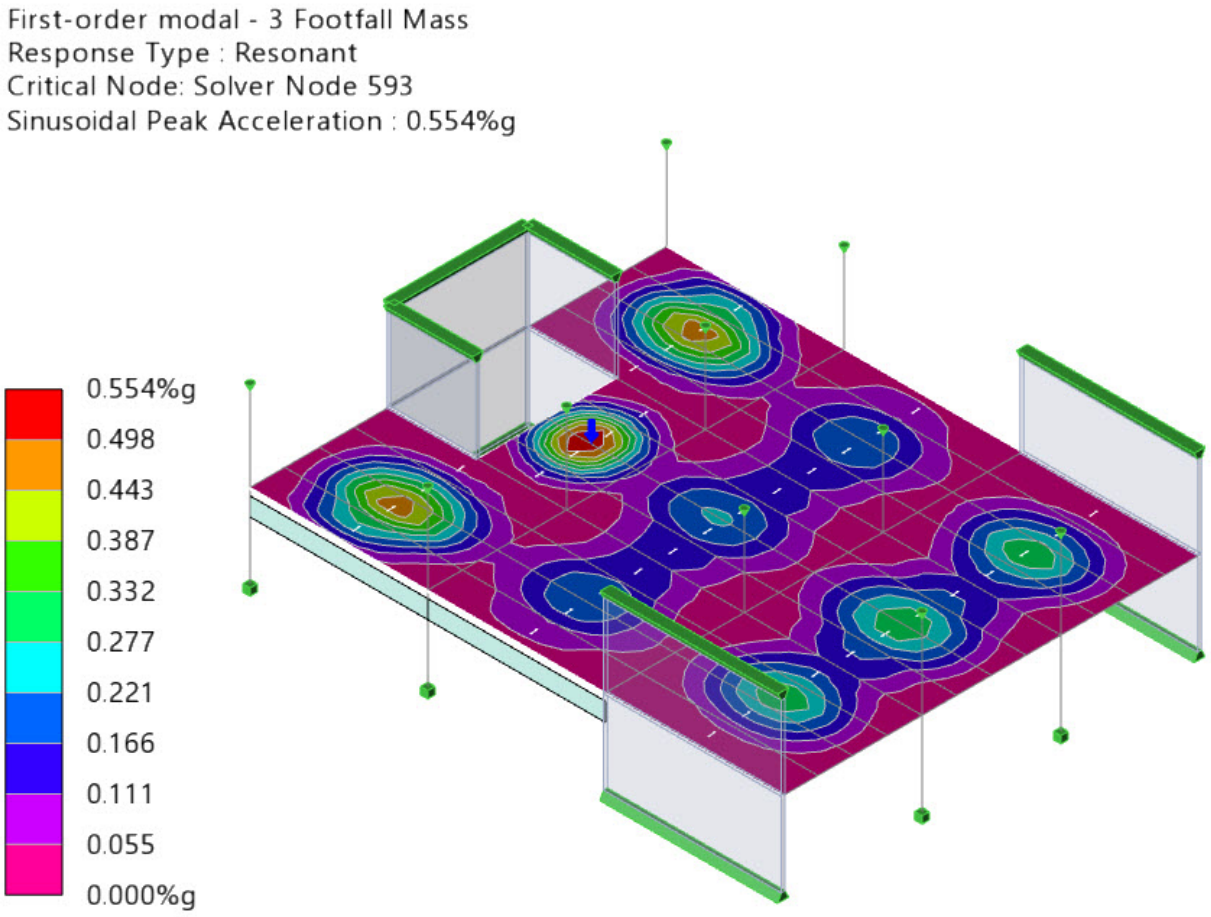
Note that the status takes into account any potential 'Mass Combination' or 'Response Type' filters applied in the **Overall Status & Envelopes** section of the **Properties** window.



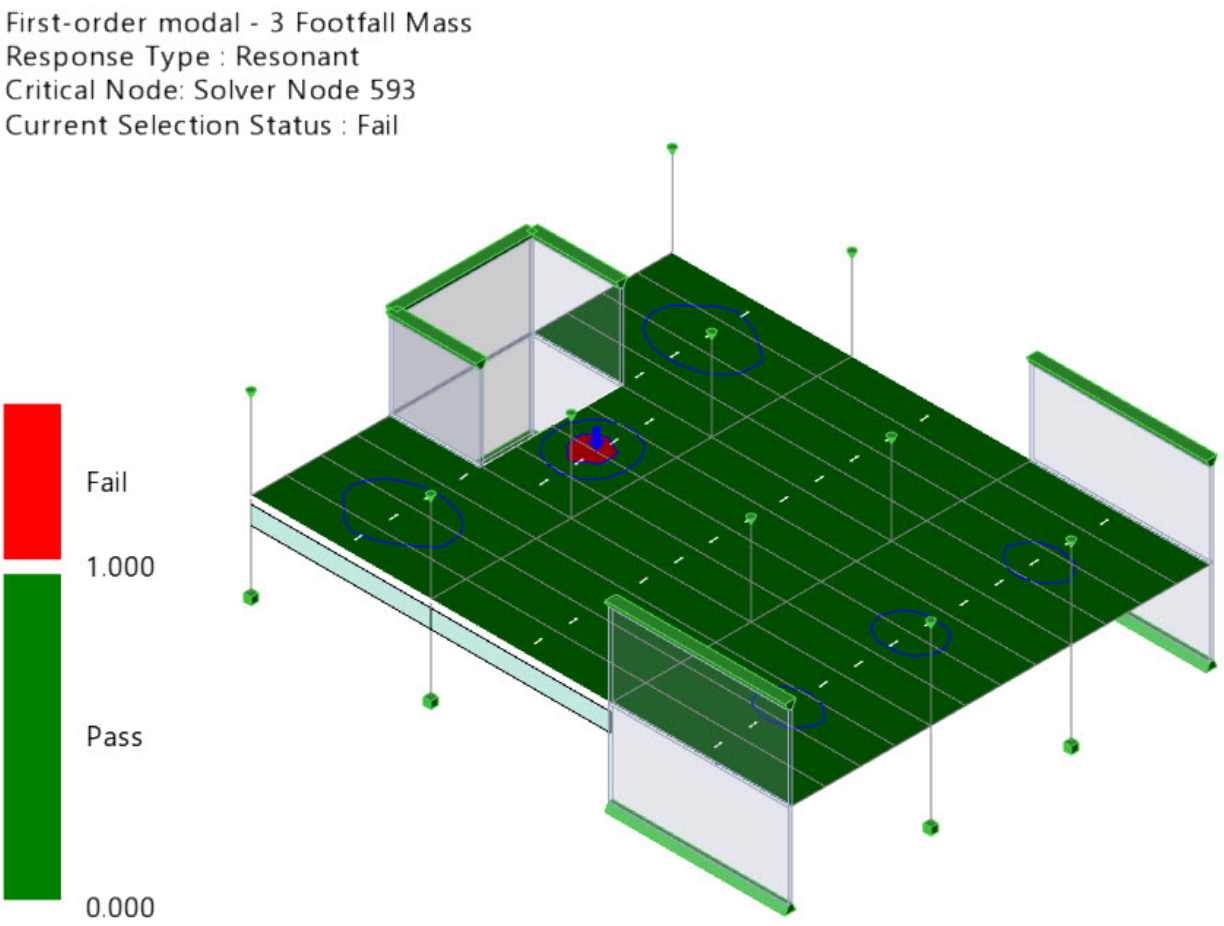
By default, all active footfall combinations are considered, and all (i.e. both resonant and transient) responses are considered.

View resonant response results

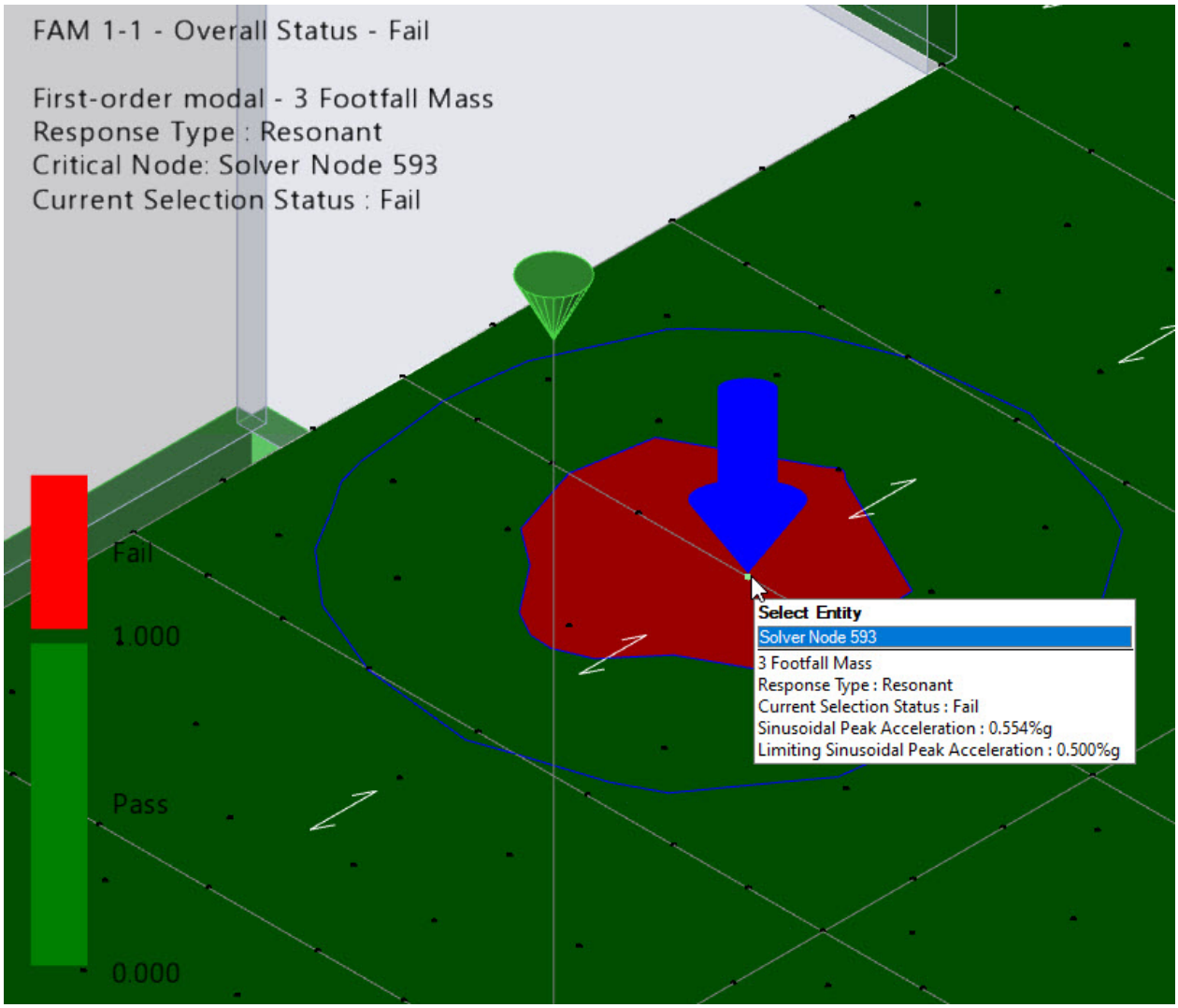
1. On the **Footfall Analysis** ribbon:
    a. Select **Sinusoidal Peak Acceleration**
    b. Ensure the response type is set to **Resonant**
    c. Ensure **Envelope** is unselected.


First-order modal - 3 Footfall Mass
Response Type : Resonant
Critical Node: Solver Node 593
Sinusoidal Peak Acceleration : 0.554%g
0.554%g
0.498
0.443
0.387
0.332
0.277
0.221
0.166
0.111
0.055
0.000%g

2. Select **Pass/Fail Status**



3. Hover the cursor over the critical node (indicated by the blue arrow) for more details.



This node is failing the resonant response because its peak acceleration (0.554%g) exceeds the limiting acceleration (0.500%g).

NOTE As previously mentioned, the 'Overall Status' in the top corner of the view is controlled from the Properties window, and is therefore considering all response types (resonant and transient, whereas the 'Current Selection Status' is controlled from the ribbon and is therefore currently considering the resonant response only.

Resonant response results in detail

Firstly, we can view the data from which the Pass/Fail Status above contours are plotted:

1. On the **Footfall Analysis** ribbon, select **Tabular Results.**

   The Floor Analysis Model Data ribbon is displayed.

2. From the Footfall Assessment Results group on the ribbon, select **Response Parameters**

3. Ensure the response type on the ribbon is set to **Resonant**

Response Parameters

Node No.	Dominant Frequency [Hz]	Sinusoidal Peak Acceleration [%g]	Limiting Sinusoidal Peak Acceleration [%g]	Acceleration Ratio	Status
593	7.06	0.554	0.500	1.107	✖ Fail
350	7.06	0.533	0.500	1.066	✖ Fail
1165	7.05	0.506	0.500	1.013	✖ Fail
1147	7.05	0.501	0.500	1.001	✖ Fail
1013	7.05	0.466	0.500	0.932	✔ Pass
882	4.76	0.464	0.500	0.928	✔ Pass
739	4.76	0.464	0.500	0.928	✔ Pass
833	4.76	0.463	0.500	0.926	✔ Pass
1124	4.76	0.463	0.500	0.926	✔ Pass
551	4.76	0.453	0.500	0.907	✔ Pass
584	4.76	0.453	0.500	0.907	✔ Pass
222	7.06	0.444	0.500	0.887	✔ Pass

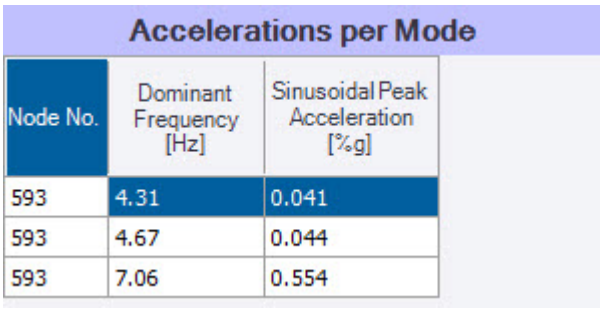
For each node the following data is displayed:

- **Dominant Frequency** - the dominant frequency calculated by the FRF method. (Because we are looking at resonant results, the dominant frequencies are restricted to a maximum of 9Hz.)
- **Sinusoidal Peak Acceleration, (ap)** - determined from FRF_{max}
- **Limiting Sinusoidal Peak Acceleration** - determined at the dominant frequency from DG11 Fig.2-1.
- **Acceleration Ratio** - ratio of peak, to limiting peak acceleration
- **Status** - Pass/Fail status by comparing peak acceleration results with limiting acceleration.

At each node multiple peak accelerations may have been identified, with the worst case being used to create the graphical **Pass/Fail Status** contour plot. A filtered view tabulating these for an individual node can be displayed as follows:

4. From the **Footfall Assessment Results** group on the ribbon, select **Accelerations**

5. Enter the critical node number (for example) in the **Node** field.

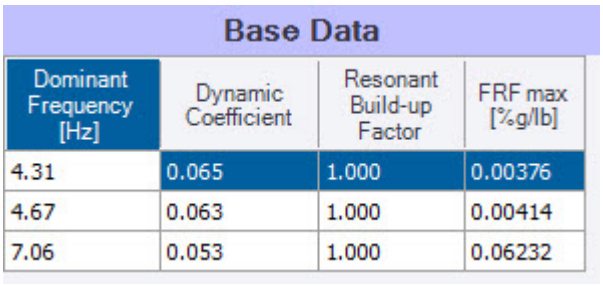


Accelerations per Mode

Node No.	Dominant Frequency [Hz]	Sinusoidal Peak Acceleration [%g]
593	4.31	0.041
593	4.67	0.044
593	7.06	0.554

For this particular node, three separate sinusoidal peak accelerations have been calculated as per Equ. 7.1 of DG11.

6. To see the data that was used in the above Accelerations calculations for this node, from the **Footfall Assessment Results** group on the ribbon, select **Base Data**



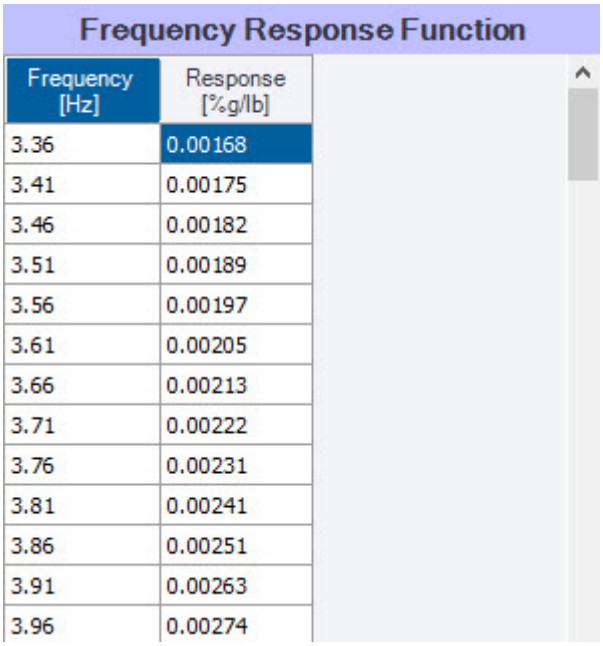
Base Data

Dominant Frequency [Hz]	Dynamic Coefficient	Resonant Build-up Factor	FRF max [%g/lb]
4.31	0.065	1.000	0.00376
4.67	0.063	1.000	0.00414
7.06	0.053	1.000	0.06232

For the selected node the following data is displayed:

- **Dominant Frequency** - the dominant frequency calculated by the FRF method.
- **Dynamic Coefficient** - determined from DG11 Equ. 7.2
- **Resonant Build-up Factor** - determined from DG11 Equ. 7.3.
- **FRF_{max}** - the maximum magnitude of acceleration from the FRF procedure

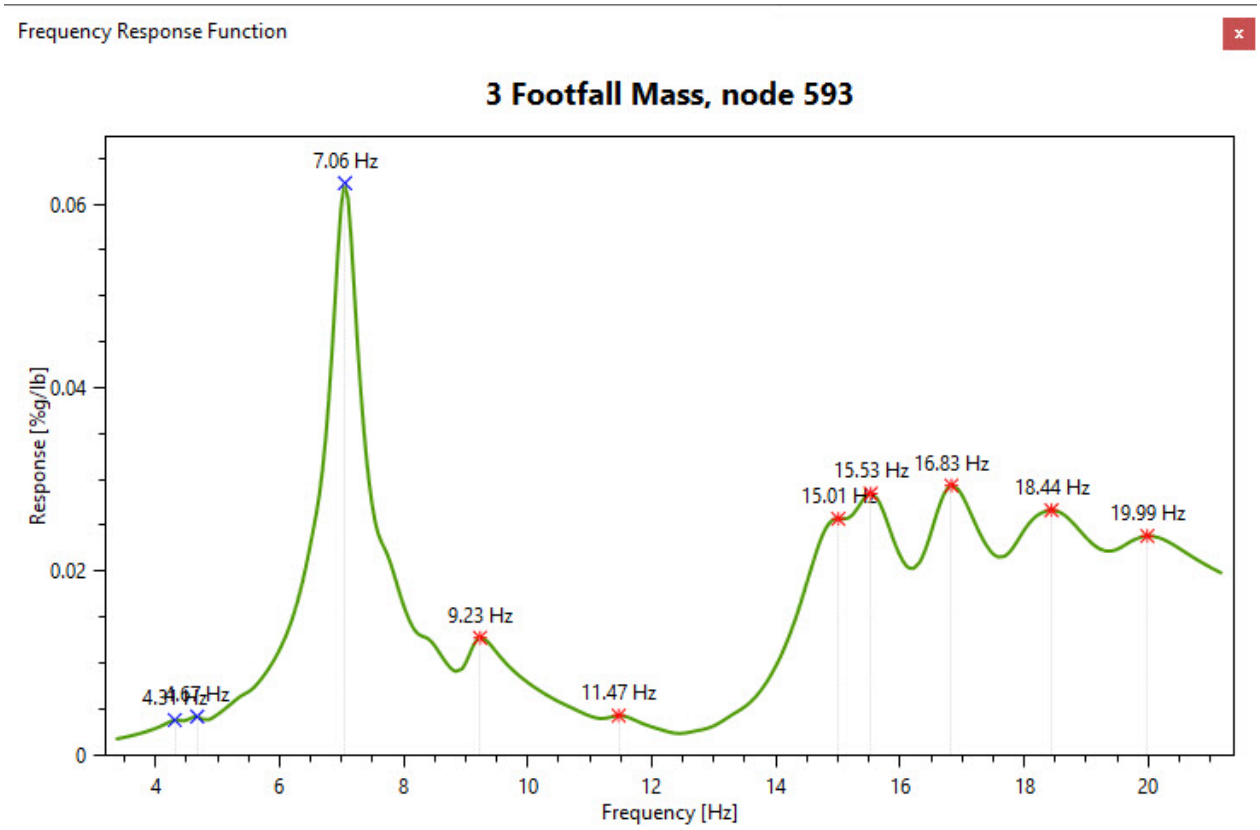
7. To see the tabulated response for this node at all frequencies, from the **Footfall Assessment Results** group, select **FRF**.



Frequency Response Function

Frequency [Hz]	Response [%g/lb]
3.36	0.00168
3.41	0.00175
3.46	0.00182
3.51	0.00189
3.56	0.00197
3.61	0.00205
3.66	0.00213
3.71	0.00222
3.76	0.00231
3.81	0.00241
3.86	0.00251
3.91	0.00263
3.96	0.00274

8. To see the above values plotted, select **FRF** from the Plots group on the ribbon.

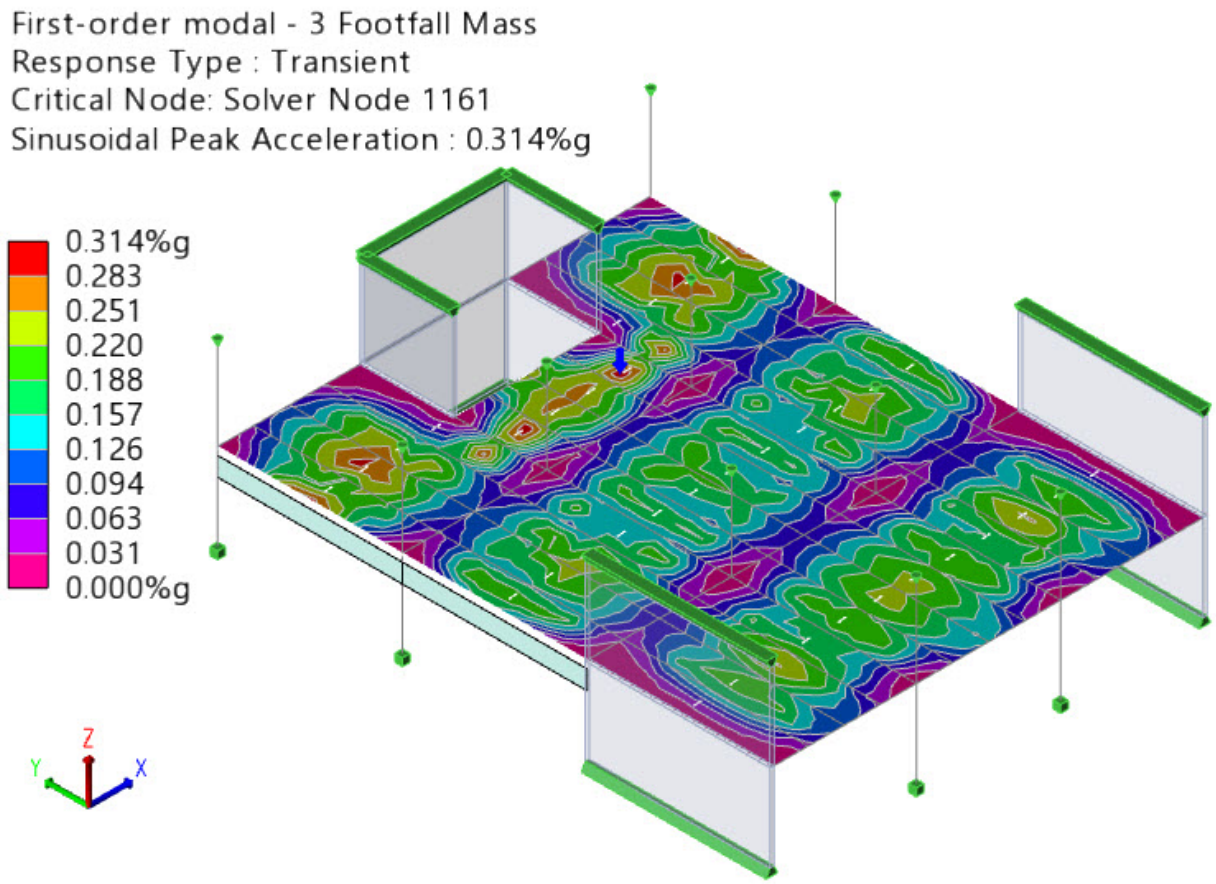


As shown above, for this node three peaks are identified occurring before 9Hz. These are the FRF_{max} values and the corresponding dominant frequencies that were displayed earlier in the Base Data table for the resonant response.

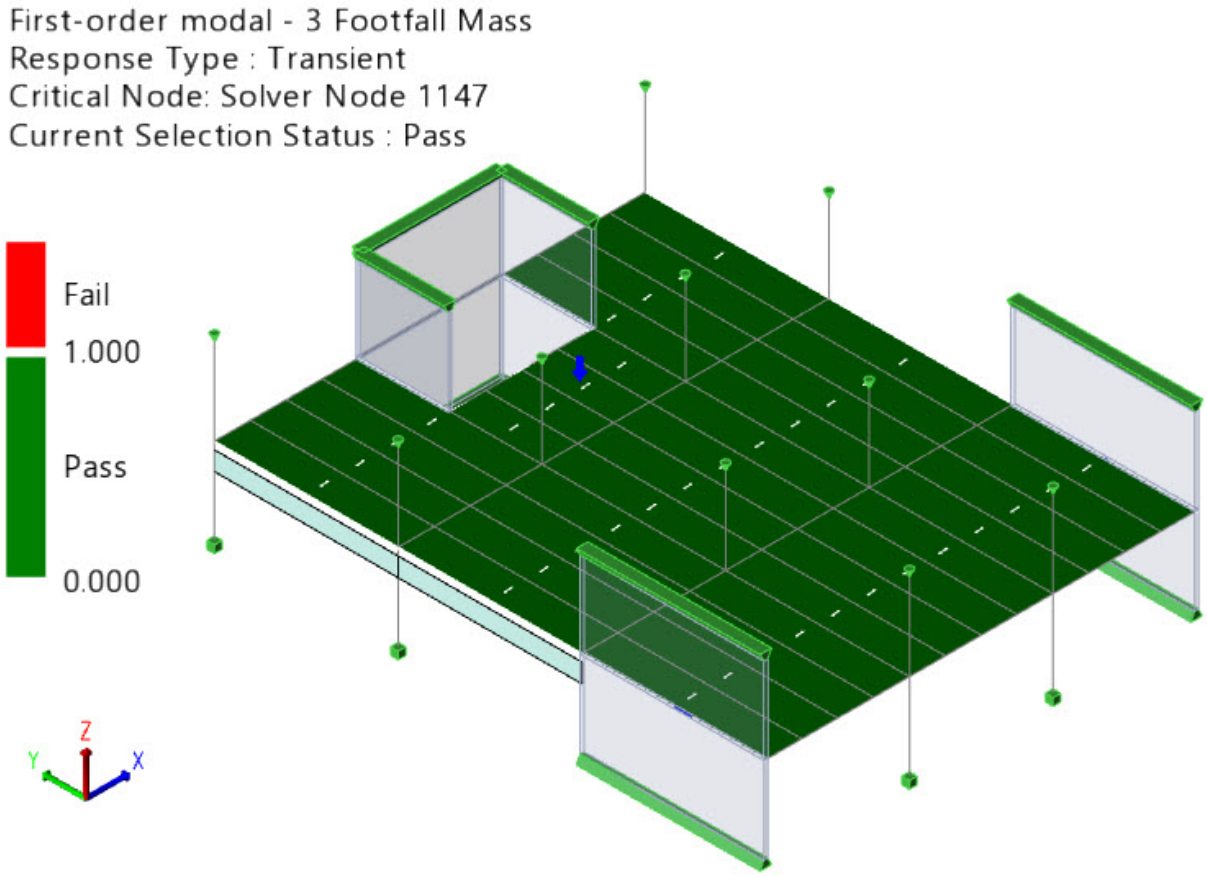
View transient response results

1. Return to the graphical view of the FAM once more.
2. On the **Footfall Analysis** ribbon, select **Sinusoidal Peak Acceleration**

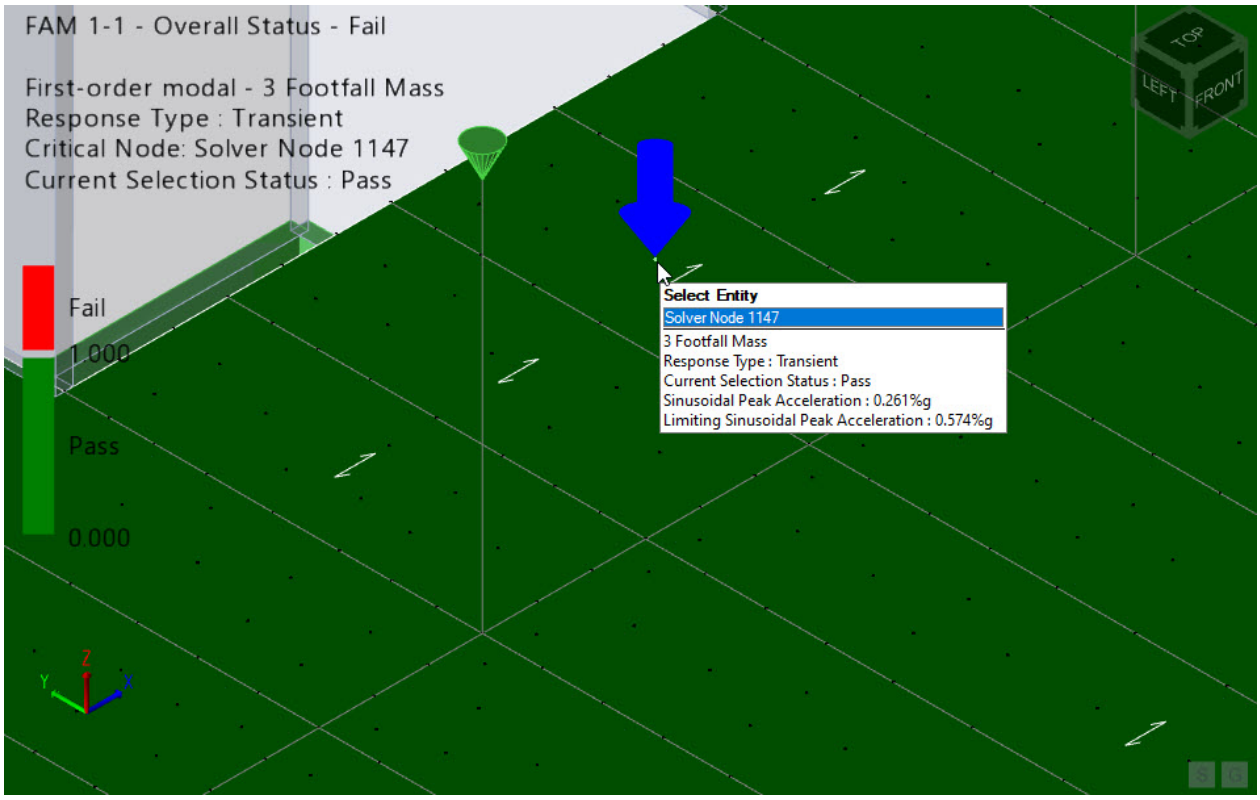
3. Change the response type from **Resonant** to **Transient** (keeping **Envelope** unselected).



4. Select **Pass/Fail Status**



Hover the cursor over the critical node (indicated by the blue arrow) for more details.



This node is passing the transient response because its peak acceleration (0.261%g) is less than the limiting acceleration (0.574%g) .

Transient response results in detail

To view the data from which the Pass/Fail Status above contours are plotted:

1. On the **Footfall Analysis** ribbon, select **Tabular Results.**

   The Floor Analysis Model Data ribbon is displayed.

2. From the Footfall Assessment Results group on the ribbon, select **Response Parameters**

3. Change the response type from **Resonant** to **Transient**

Response Parameters

Node No.	Dominant Frequency [Hz]	Sinusoidal Peak Acceleration [%g]	Limiting Sinusoidal Peak Acceleration [%g]	Acceleration Ratio	Status
1147	9.18	0.261	0.574	0.456	Pass
1165	9.18	0.261	0.574	0.455	Pass
593	9.23	0.257	0.577	0.446	Pass
139	9.04	0.242	0.565	0.428	Pass
350	9.23	0.247	0.577	0.428	Pass
963	10.96	0.293	0.685	0.427	Pass
1124	10.96	0.292	0.685	0.427	Pass
833	10.96	0.292	0.685	0.427	Pass
594	9.09	0.242	0.568	0.427	Pass
592	9.09	0.242	0.568	0.427	Pass
1161	11.88	0.314	0.743	0.423	Pass
1179	11.88	0.314	0.743	0.422	Pass
1086	10.91	0.287	0.682	0.421	Pass

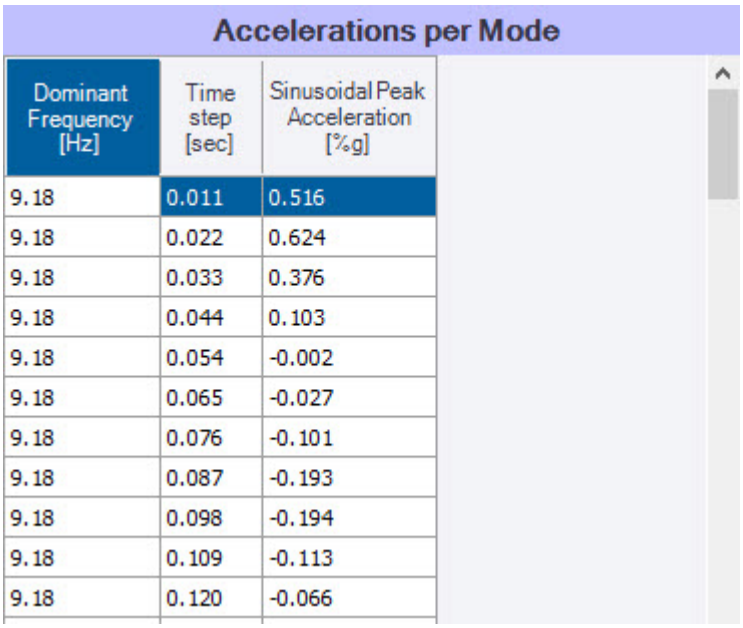
For each node the following data is displayed:

- **Dominant Frequency** - the dominant frequency calculated by the FRF method. (Because we are looking at transient results, the dominant frequencies are calculated for the range of 9-20Hz.)
- **Sinusoidal Peak Acceleration** - the root mean square sinusoidal peak acceleration, (aESPA), determined from Equ. 7-6 of DG11
- **Limiting Sinusoidal Peak Acceleration** - determined at the dominant frequency using DG11 Fig.2-1.
- **Acceleration Ratio** - ratio of peak, to limiting peak acceleration
- **Status** - Pass/Fail status by comparing peak acceleration results with limiting acceleration.

While only the worst case **Status** is used to create the graphical **Pass/Fail Status** contour plot, multiple peak accelerations are identified at each node.

4. From the **Footfall Assessment Results** group on the ribbon, select **Accelerations**

5. Enter the critical node number (for example) in the **Node** field.



Accelerations per Mode

Dominant Frequency [Hz]	Time step [sec]	Sinusoidal Peak Acceleration [%g]
9.18	0.011	0.516
9.18	0.022	0.624
9.18	0.033	0.376
9.18	0.044	0.103
9.18	0.054	-0.002
9.18	0.065	-0.027
9.18	0.076	-0.101
9.18	0.087	-0.193
9.18	0.098	-0.194
9.18	0.109	-0.113
9.18	0.120	-0.066

6. For the selected node the following data is displayed:
   - **Dominant Frequency** - the dominant frequency calculated by the FRF method.
   - **Time Step** - determined from the step frequency
   - **Sinusoidal Peak Acceleration** - determined from DG11 Equ. 7.4
7. Scroll down the list and you will see that multiple dominant frequencies have been identified for this node. Each dominant frequency has been assessed with its own step frequency, (determined using Table 7-1 from DG11). The acceleration is calculated for an individual mode or all modes.

8. From the **Footfall Assessment Results** group on the ribbon, select **Base Data**

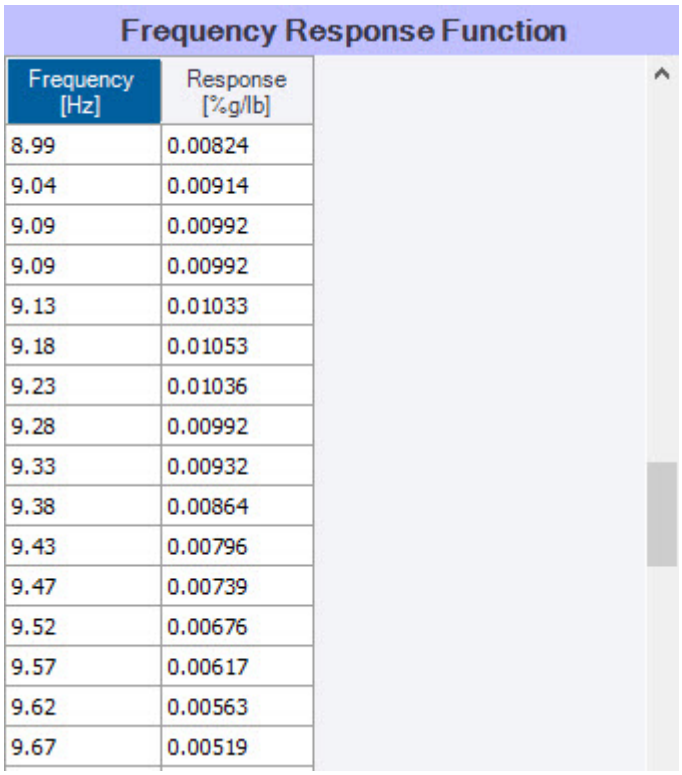
Base Data

Mode Number	Dominant Frequency [Hz]	Step Frequency [Hz]	Effective Impulse Force [kip]	Sinusoidal Peak Acceleration [%g]
1	9.18	1.84	0.0	0.003
1	11.93	1.99	0.0	0.003
1	16.42	2.05	0.0	0.003
1	17.73	1.97	0.0	0.003
1	19.73	2.19	0.0	0.003
2	9.18	1.84	0.0	0.001
2	11.93	1.99	0.0	0.001
2	16.42	2.05	0.0	0.001
2	17.73	1.97	0.0	0.001
2	19.73	2.19	0.0	0.001
3	9.18	1.84	0.0	0.000
3	11.93	1.99	0.0	0.000
3	16.42	2.05	0.0	0.000
3	17.73	1.97	0.0	0.000
3	19.73	2.19	0.0	0.000
4	9.18	1.84	0.0	0.001

For the selected node the following data is displayed:

- **Dominant Frequency** - the dominant frequency calculated by the FRF method.
- **Step Frequency** - determined from DG11 Table 7.1
- **Effective Impulse Force** - determined from DG11 Equ. 1.6.
- **Sinusoidal Peak Acceleration** - determined from DG11 Equ. 7.4 and 7.6

9. To see the tabulated response for this node at all frequencies, from the **Footfall Assessment Results** group, select **FRF**.



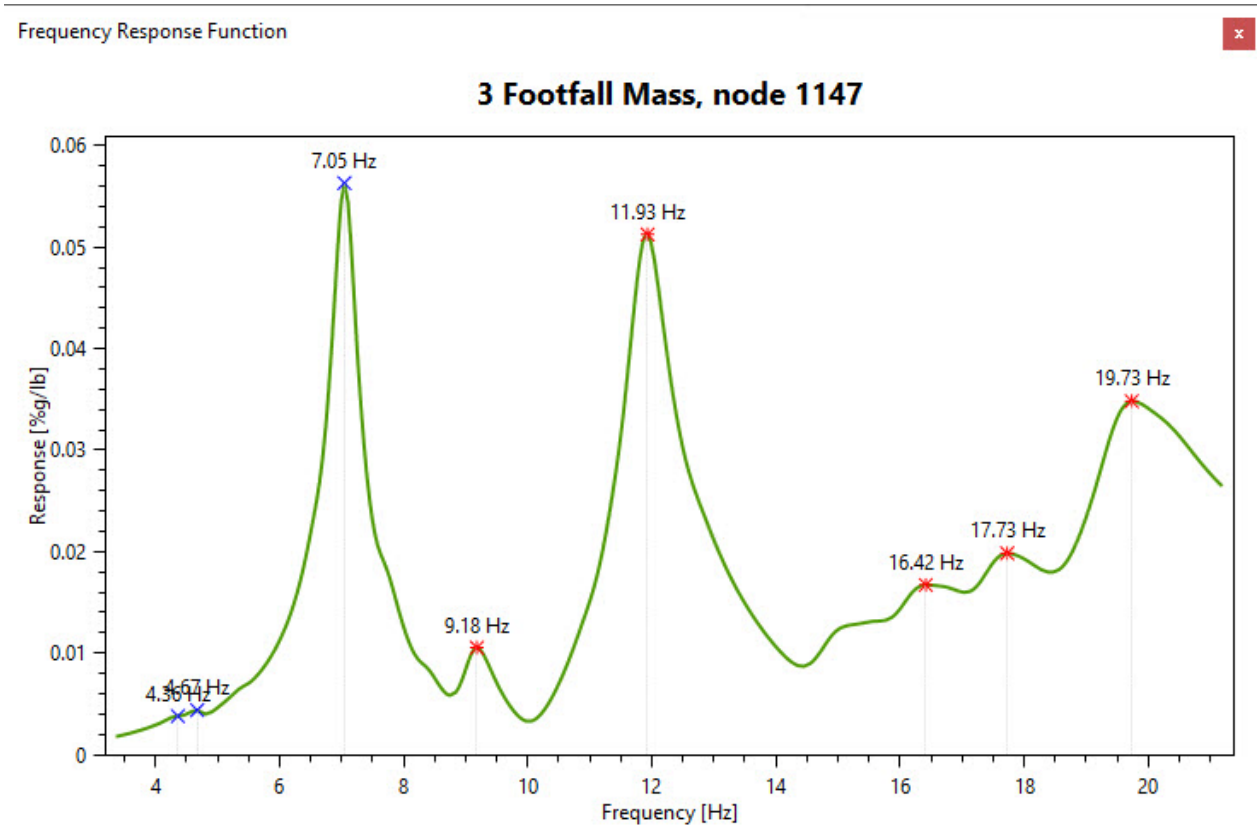
Frequency Response Function

Frequency [Hz]	Response [%g/lb]
8.99	0.00824
9.04	0.00914
9.09	0.00992
9.09	0.00992
9.13	0.01033
9.18	0.01053
9.23	0.01036
9.28	0.00992
9.33	0.00932
9.38	0.00864
9.43	0.00796
9.47	0.00739
9.52	0.00676
9.57	0.00617
9.62	0.00563
9.67	0.00519

For the selected node the following data is displayed:

- **Frequency**
- **Response** (in %g/lb) - see section 7.1 of DG11.

10. To see the above values plotted, select **FRF** from the Plots group on the ribbon.

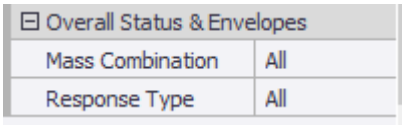


As shown above, for this node multiple peaks are identified after 9Hz. These are occurring at the dominant frequencies displayed in the Base Data table for the transient response.

View enveloped response results

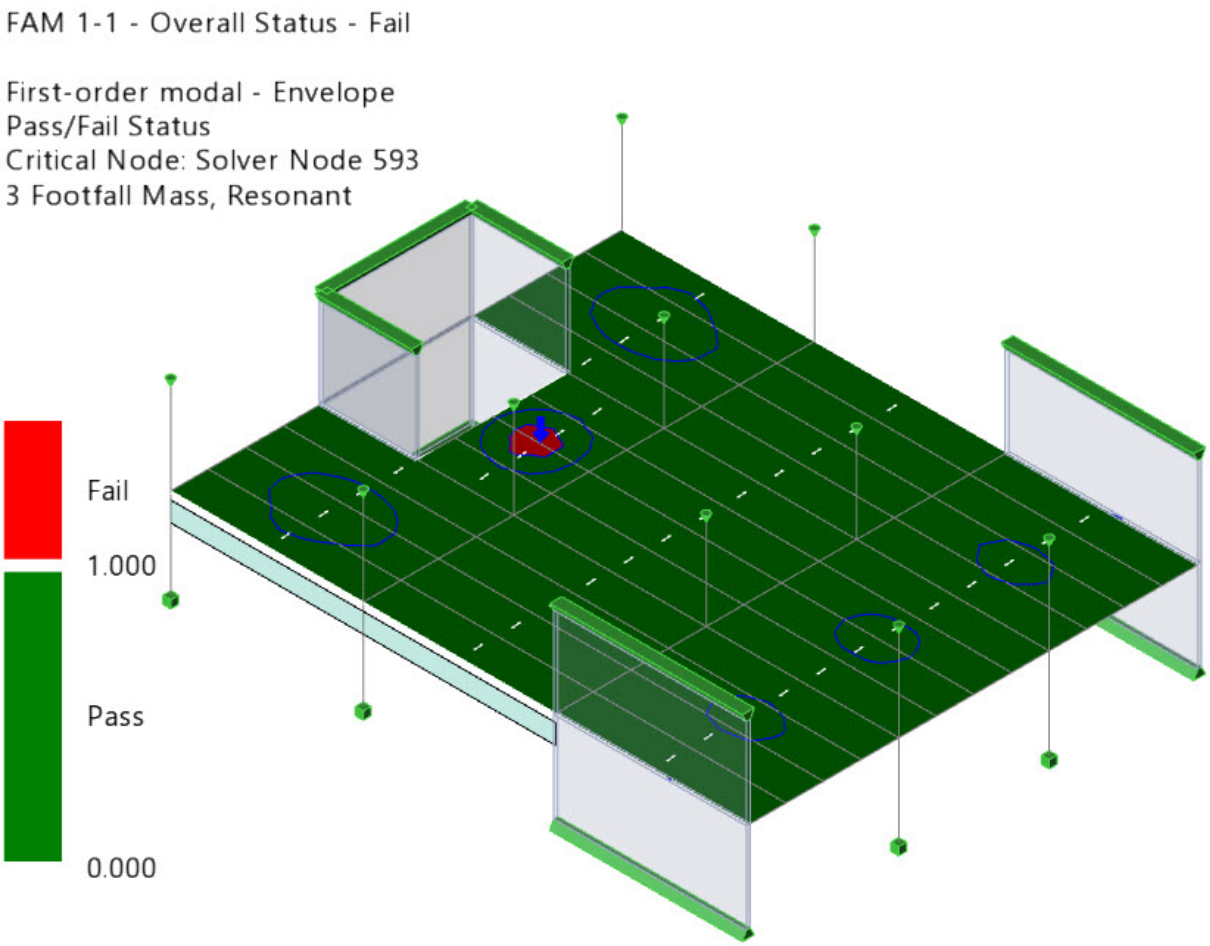
Depending on how it is configured, the envelope option on the ribbon can be used to display results for a selected, or all Mass Combinations; and for respondent, transient, or both response types.

It is configured in the same way as the 'Overall Status' via the **Overall Status & Envelopes** section of the **Properties** window.



Accepting the default settings shown above, the enveloped results will now be investigated.

1. Return to the graphical view of the FAM once more.
2. On the **Footfall Analysis** ribbon, select **Pass/Fail Status**
3. Select **Envelope**



In the above enveloped view, both resonant and transient responses have been considered (and if there were multiple active footfall combinations they would also have been considered).

The most critical node is failing under a resonant response for the Footfall Mass combination, (the only active combination in this example).

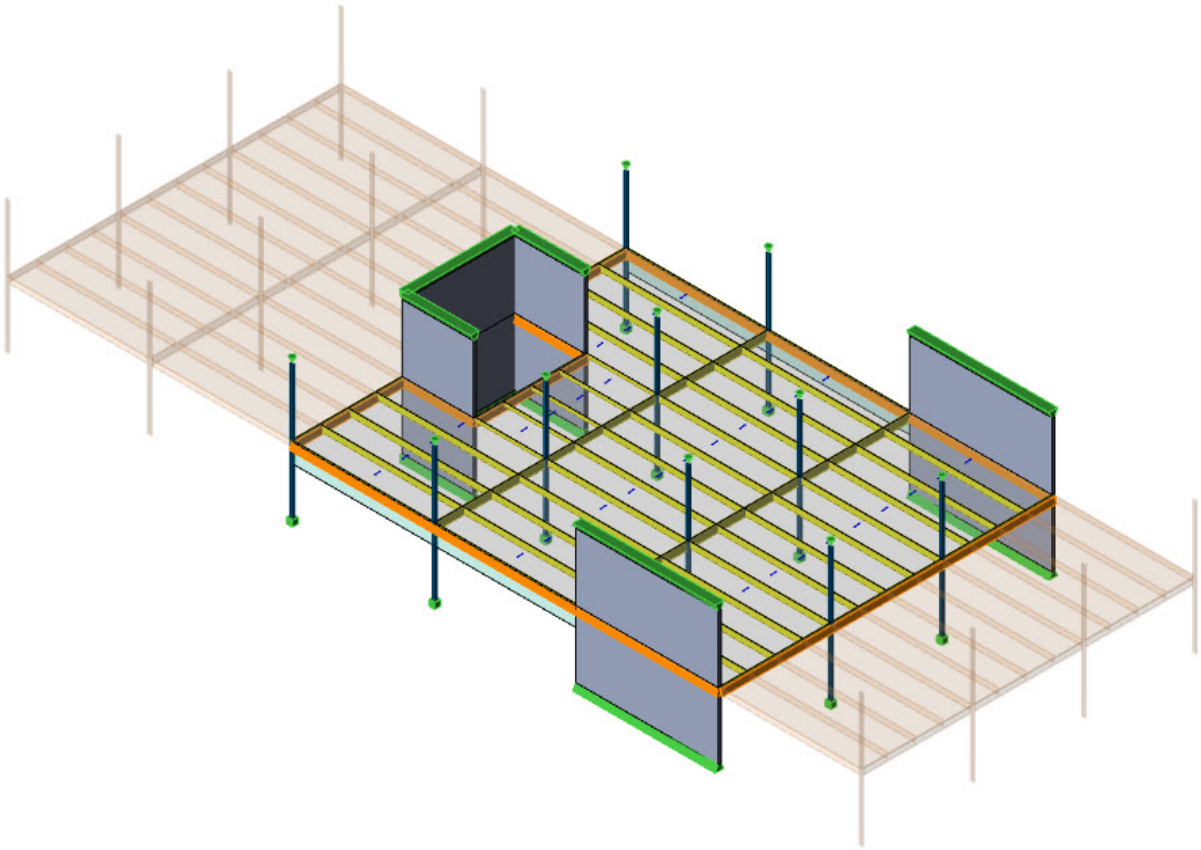
The FAM is failing.

Edit the FAM and add a facade restraint

Facade restraints can be used to vertically restrain specific slab edges.

The existing slab edges are currently unrestrained, we will now apply a facade restraint to one edge to investigate its effect on the results.

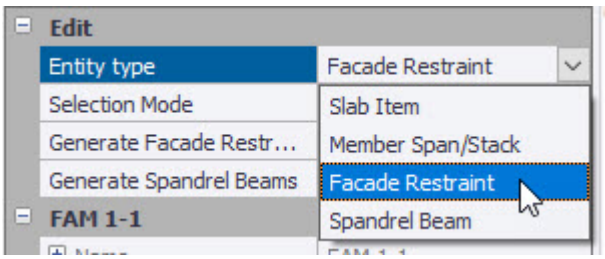
1. From the **Footfall Analysis Models** group on the ribbon, select **Edit**
2. In the **Properties** window, select **Generate Facade Restraint**



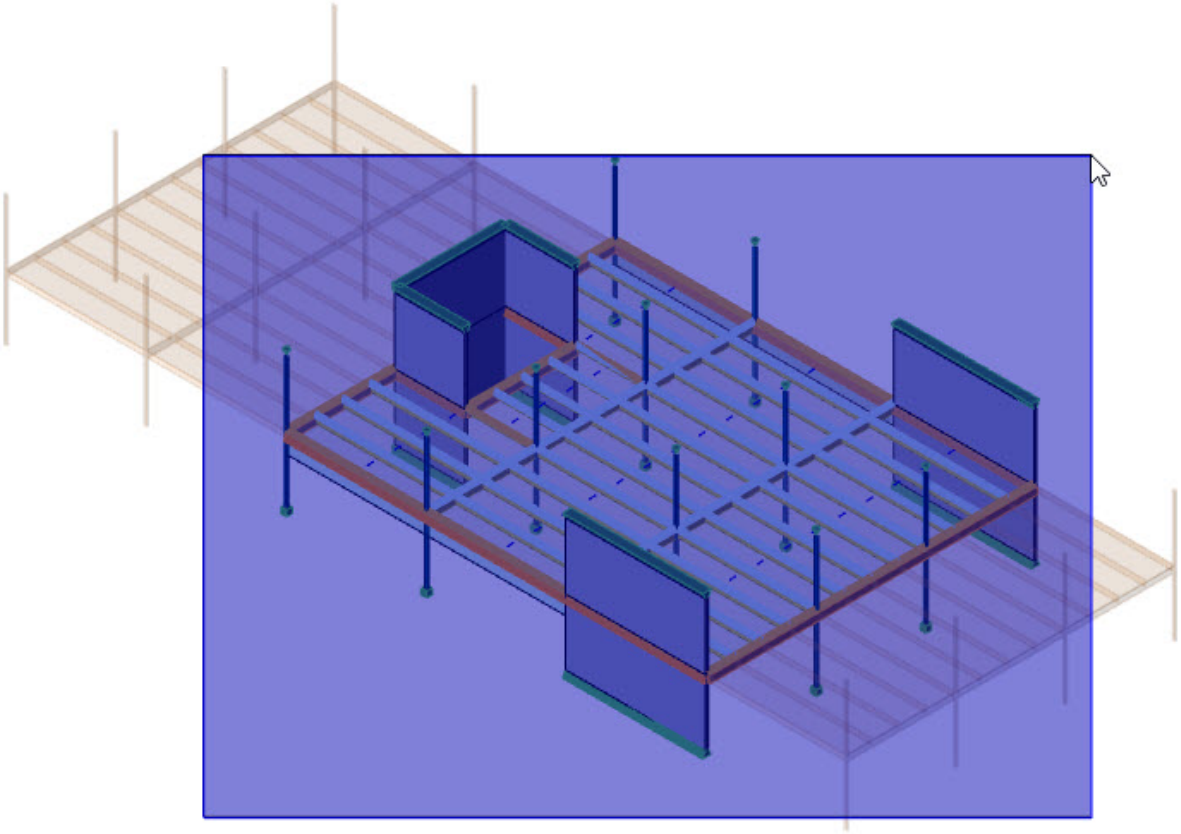
As shown above this action has generated facade restraints around the entire FAM perimeter.

We only want to restrain a single edge, so some further editing is required.

3. In the **Properties** window, change the **Entity type** to 'Facade Restraint'.

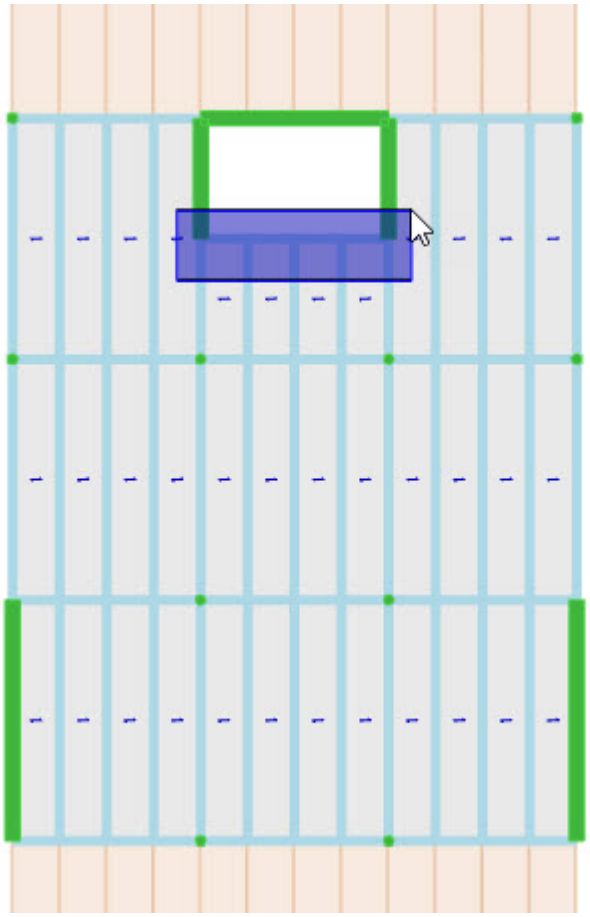


4. Set the **Selection Mode** to 'Remove Only'.
5. Drag a box from left to right around the entire FAM.

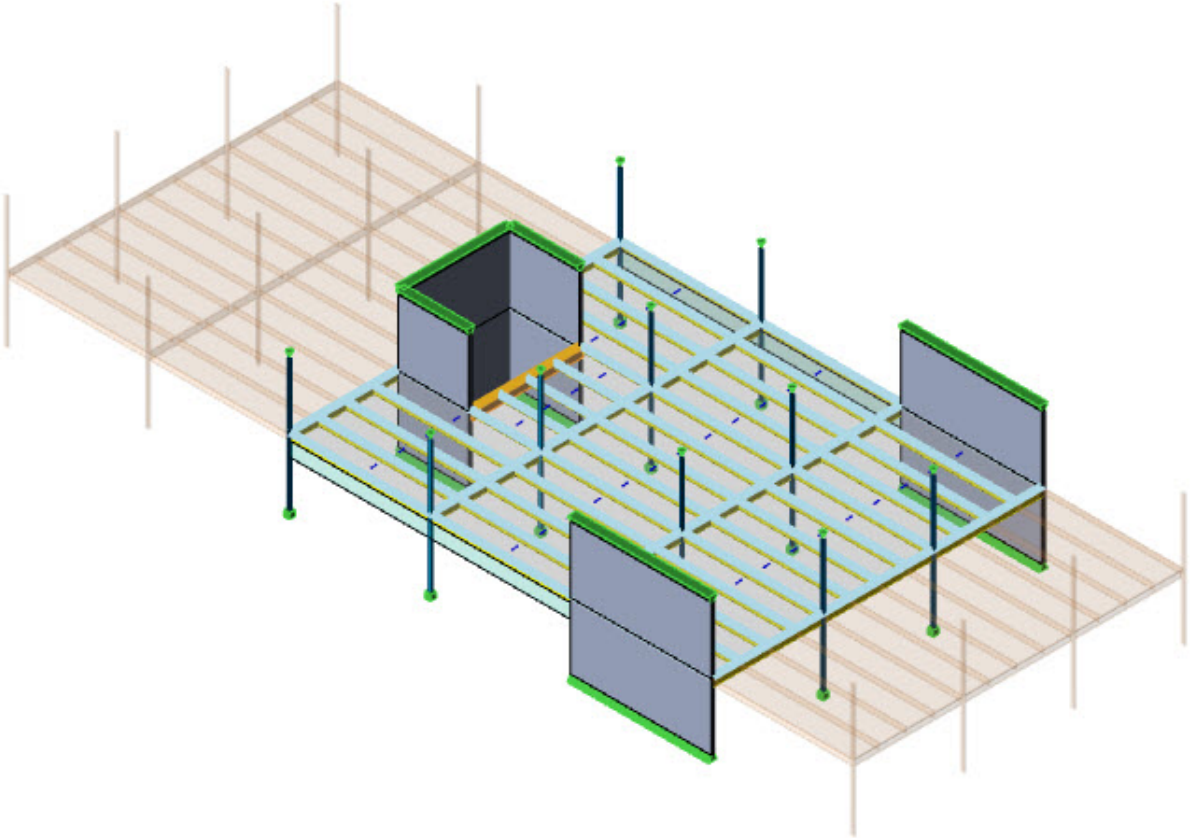


6. Set the **Selection Mode** to 'Add Only'.
7. Use the **ViewCube** to switch to a top view.

8. Drag a box around the beam spanning the lift core.



9. Use the **ViewCube** to switch back to an isometric view.

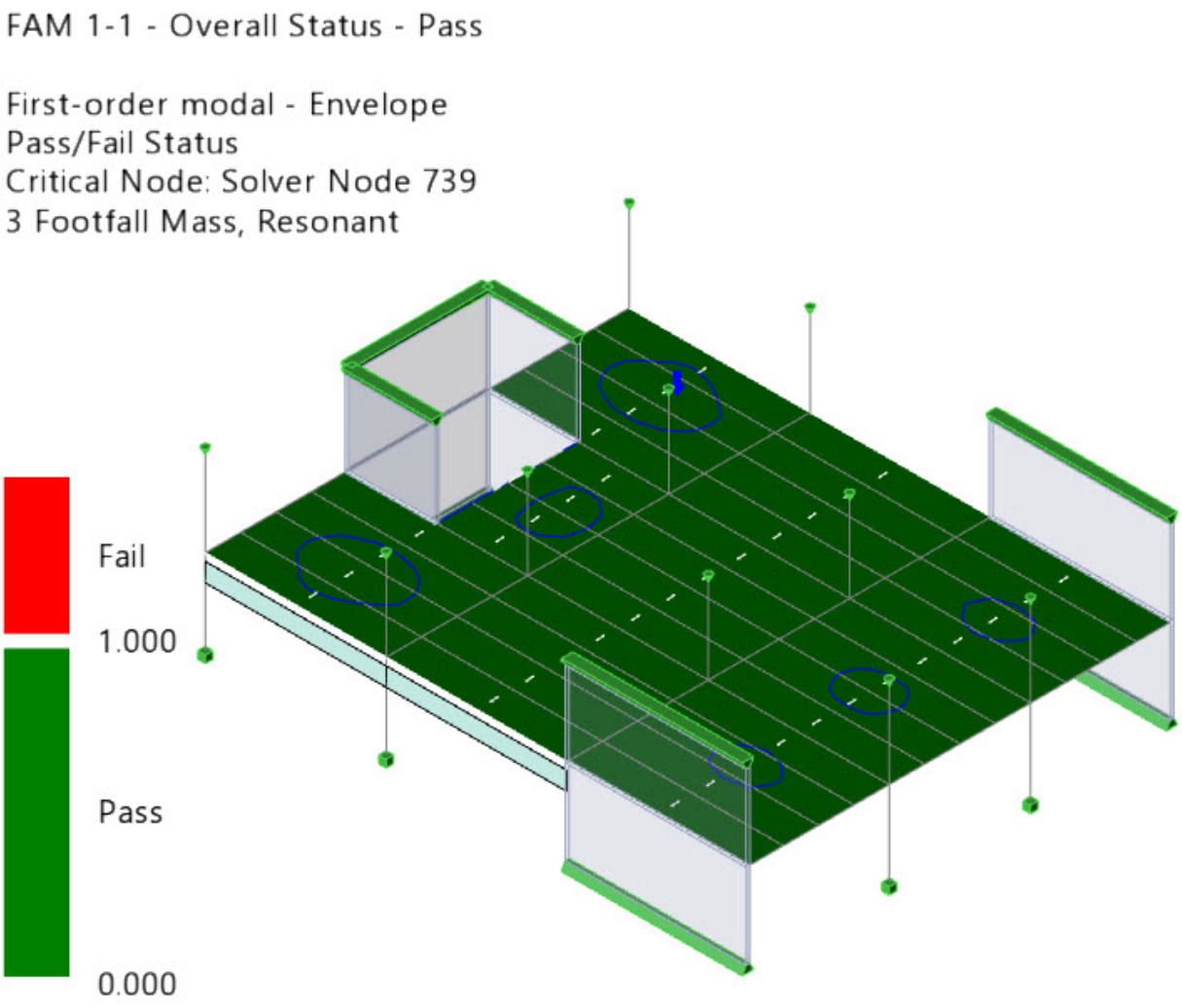


10. The editing is now complete, so press ESC on the keyboard to exit Edit mode.

Reanalyse and view the updated results

1. Click **Analyze Current**
2. On the **Footfall Analysis** ribbon, select **Pass/Fail Status**

3. Select **Envelope**



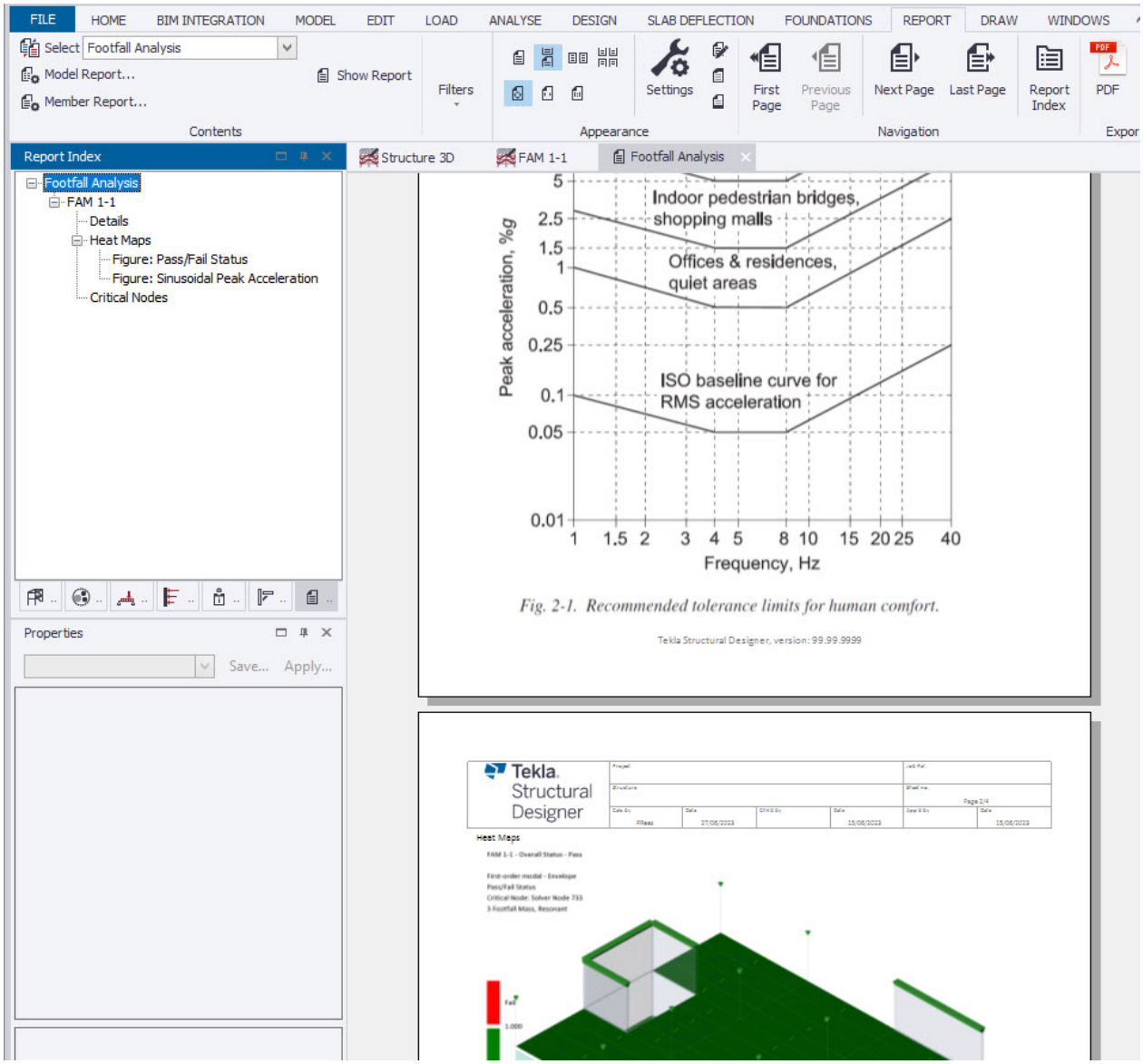
The FAM is now passing.

Output the calculations

To create a footfall analysis report:

1. On the ribbon, go to the **Report** tab.
2. Click **Model Report**.
3. In the **Report Contents** dialog, click **Add** and then under **Active Style**, enter a **Name** for the report
4. In the list of chapters and options, select **Footfall Analysis** and drag it across to the **Report Structure** area.

5. Click **OK**
6. On the ribbon, in the list on the left side, select the report name you just created.
7. Click **Show Report**.



Additional display and review options

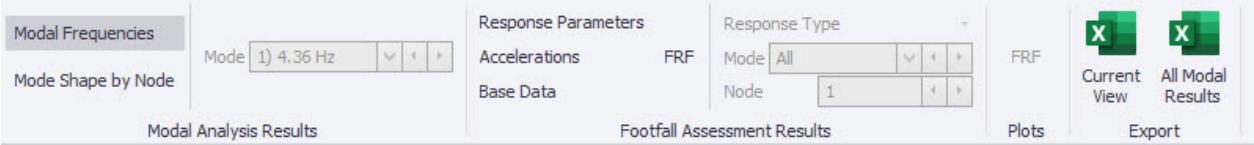
Several tabular data views are available, each of which can be exported to Excel.

To display the **Floor Analysis Model Data** ribbon from where the tabular data views can be accessed:

1. Make the Floor Analysis Model (FAM) view active once more.

2. On the **Footfall Analysis** ribbon, select **Tabular Results**

The **Floor Analysis Model Data** ribbon is displayed and an FAM tabular data view opens.



Modal Frequencies

To view modal frequencies, display the **Floor Analysis Model Data** ribbon and select **Modal Frequencies**

Modal Frequencies

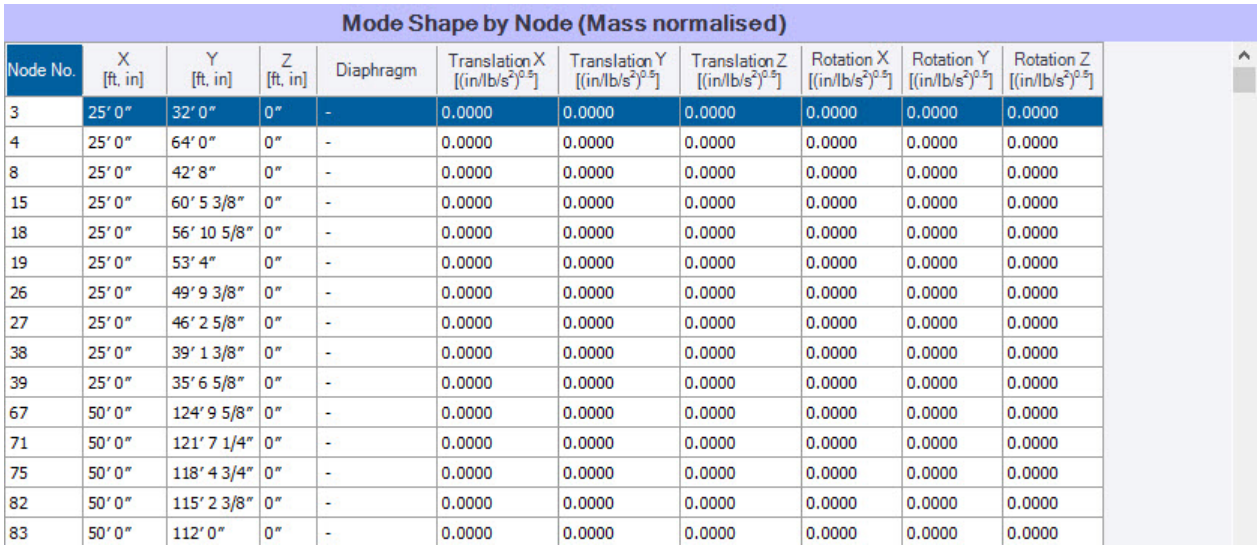
Mode Number	Period [sec]	Frequency [Hz]	Error [%]	Mass Participation Dir 1 [%]	Mass Participation Dir 2 [%]	Mass ParticipationZ [%]	Modal Mass [kip]
1	0.229	4.36	0.00	0.00	0.00	3.26	85.1
2	0.221	4.52	0.00	0.00	0.00	0.40	65.4
3	0.214	4.67	0.00	0.00	0.00	0.00	140.4
4	0.214	4.68	0.00	0.00	0.00	0.57	103.5
5	0.211	4.75	0.00	0.00	0.00	4.70	81.9
6	0.210	4.77	0.00	0.00	0.00	0.00	67.6
7	0.196	5.10	0.00	0.00	0.00	0.00	156.7
8	0.183	5.45	0.00	0.00	0.00	44.70	130.0
9	0.176	5.68	0.00	0.00	0.00	0.00	112.6
10	0.151	6.62	0.00	0.00	0.00	0.17	128.6

This table lists the period, frequency, mass participation, and modal mass for each mode.

Mode Shape by Node

To view mode shapes by node:

1. On the **Floor Analysis Model Data** ribbon, select **Mode Shape by Node**
2. Select the required **Mode** from the list.



Mode Shape by Node (Mass normalised)

Node No.	X [ft, in]	Y [ft, in]	Z [ft, in]	Diaphragm	Translation X [(in/lb/s^2)$^{0.5}$]	Translation Y [(in/lb/s^2)$^{0.5}$]	Translation Z [(in/lb/s^2)$^{0.5}$]	Rotation X [(in/lb/s^2)$^{0.5}$]	Rotation Y [(in/lb/s^2)$^{0.5}$]	Rotation Z [(in/lb/s^2)$^{0.5}$]
3	25′ 0″	32′ 0″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
4	25′ 0″	64′ 0″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
8	25′ 0″	42′ 8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
15	25′ 0″	60′ 5 3/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
18	25′ 0″	56′ 10 5/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
19	25′ 0″	53′ 4″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
26	25′ 0″	49′ 9 3/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
27	25′ 0″	46′ 2 5/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
38	25′ 0″	39′ 1 3/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
39	25′ 0″	35′ 6 5/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
67	50′ 0″	124′ 9 5/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
71	50′ 0″	121′ 7 1/4″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
75	50′ 0″	118′ 4 3/4″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
82	50′ 0″	115′ 2 3/8″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
83	50′ 0″	112′ 0″	0″	-	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000

The mode shapes are listed for the selected mode.

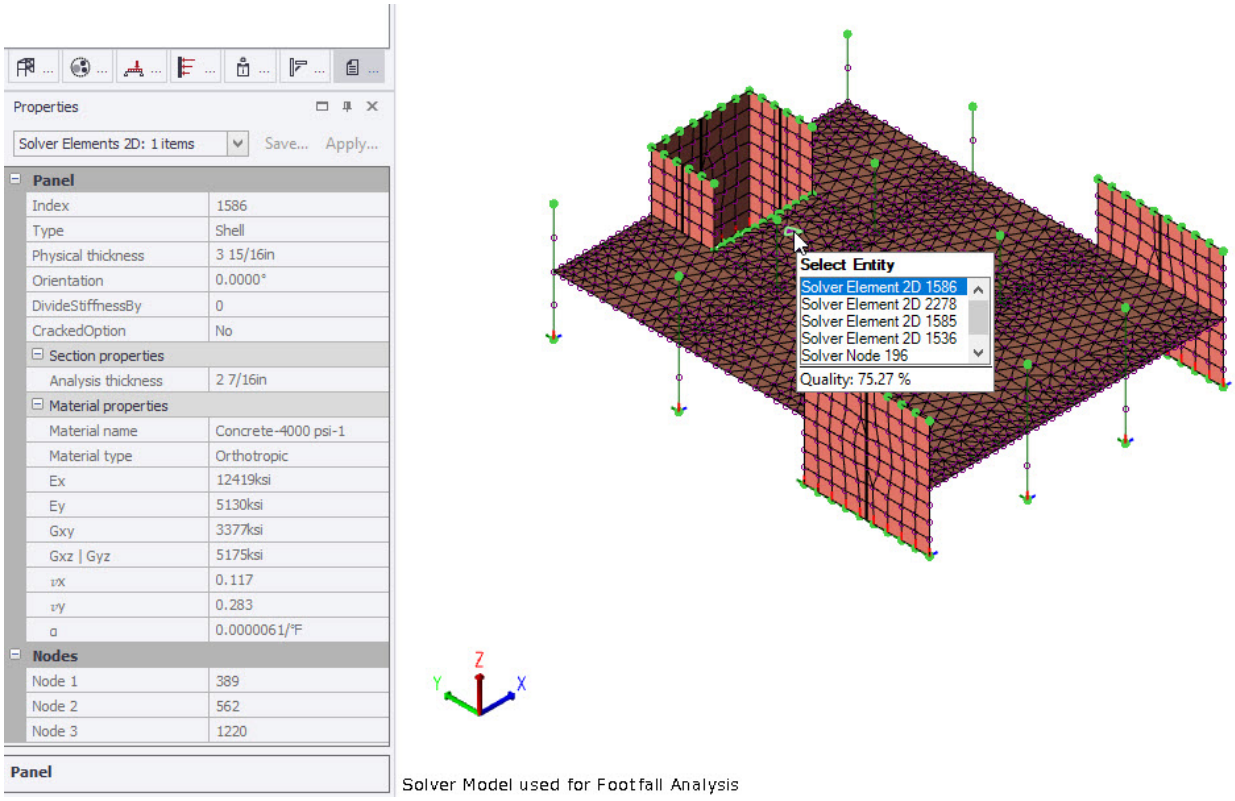
FAM solver model

After the analysis has been performed, if you require you can examine the solver model used for the Footfall Analysis model as follows:

1. Ensure that the required Floor Analysis Model (FAM) view is active.
2. To change the view type of the FAM view, do one of the following:
   - Right-click the FAM **view tab** and in the context menu, select **Solver view**
   - Or, in the **Status bar** at the bottom of the window, click **Solver View**.

   The FAM view changes to an FAM solver view.
3. Right-click anywhere in the solver view.
4. In the context menu, select **Solver models**.
5. In the sub-menu, select **Solver Model used for Footfall Analysis**.

   Tekla Structural Designer opens the selected solver model. You can then select individual elements in the solver model to display their section and material properties in the **Properties** window.

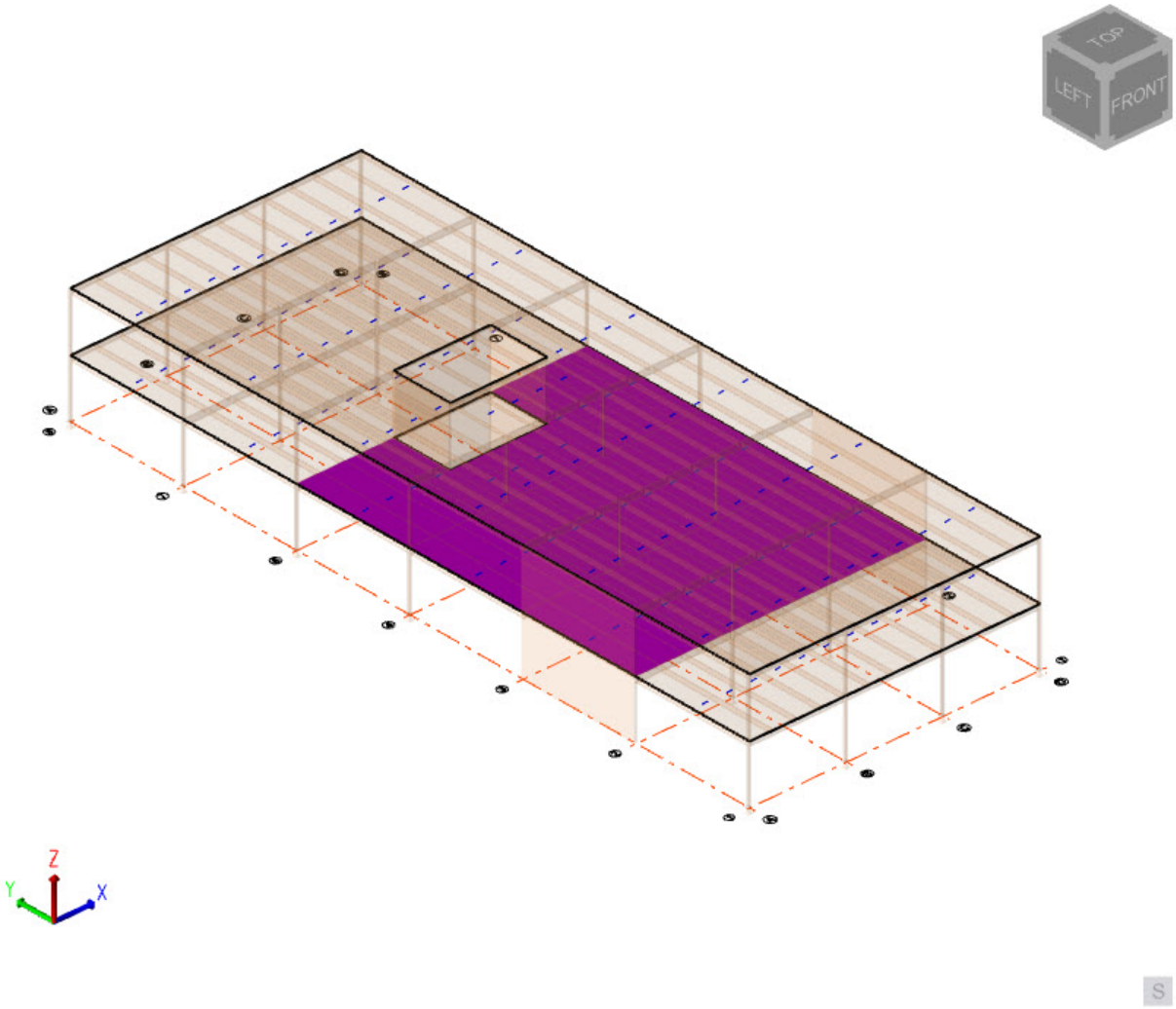


Solver Model used for Footfall Analysis

Manage FAMs in a Structure 3D view

When a **Structure 3D** view is active and set to the **Footfall Analysis View** regime it will display all the floor analysis models that have been created so far.

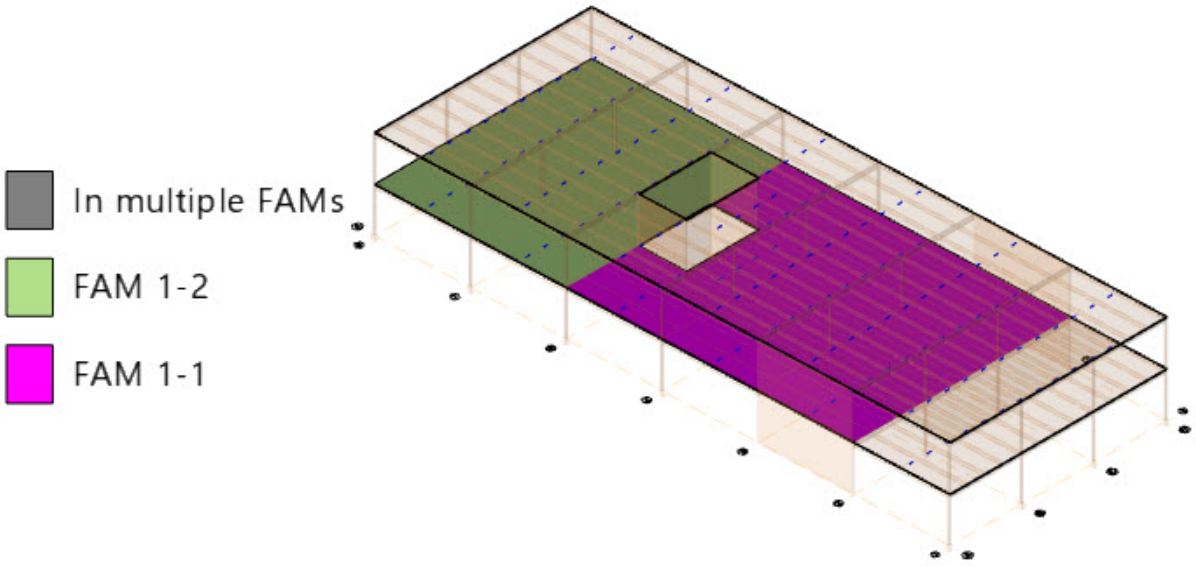
In this example, only one FAM has been created, so it has been automatically selected.



The following options are available in the ribbon:

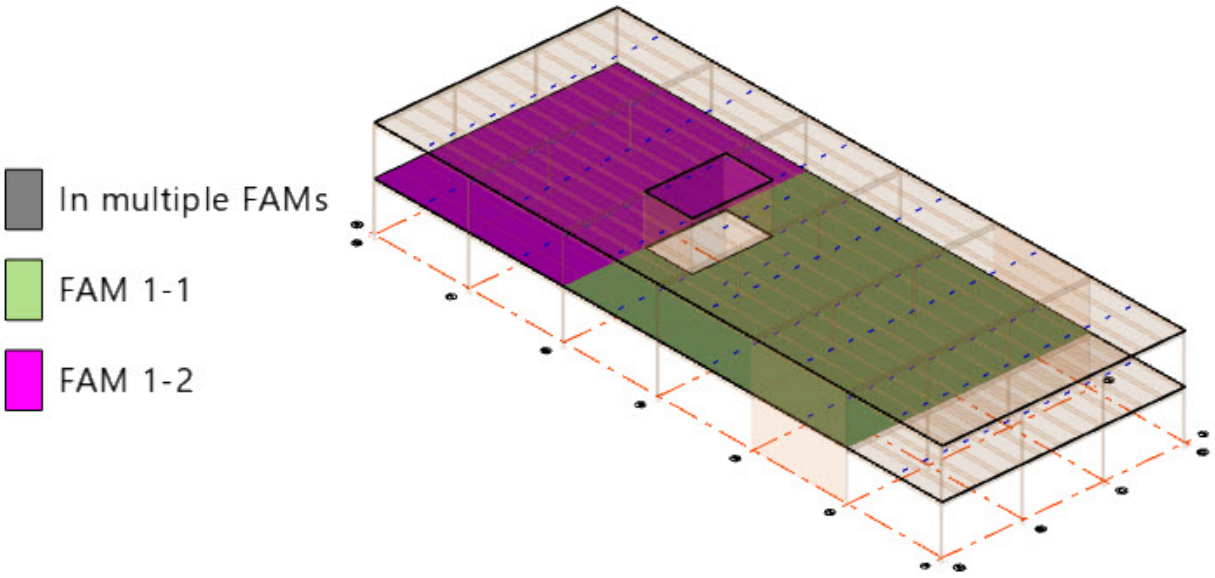
- **New** - create a new FAM by selecting a slab item.
- **Edit** - open the currently selected FAM.
- **Delete** - delete an existing FAM by selecting it.
- **Pass/Fail Status** - show the pass fail/status of all FAMs

When more than one FAM exists, a color coded legend is displayed and the currently selected FAM is highlighted in purple.



You can change the selected FAM from the Model list in the ribbon.





9.7 Footfall assessment settings

When you create a new model, global settings for footfall assessment are copied to the model file and become model specific settings applicable to the new model only.

- To edit the global footfall settings: click **Home** > **Settings** > **Footfall Assessment**.
  - Under each available design guide, the following settings pages are then provided:

    Analysis Settings, **Analysis Model Options**, **Concrete Modification Factors** and **Footfall Checks**.
- To edit the model specific footfall settings: open a Footfall Analysis View and then click **Footfall Analysis** > **Settings** on the ribbon.
  - The first model specific page is **Settings Sets Management**. Subsequent pages are then provided for the available design guides. Each design guide contains one (or more) settings set. Each settings set contains the following sub-pages: **Analysis Settings**, **Analysis Model Options**, **Concrete Modification Factors**.

Settings Set Management

NOTE This page is only available in the model specific settings, not the global settings.

From the **Design guide** list, select the design guide containing the footfall settings sets you want to manage.

A table then lists the available settings sets for the selected design guide.

- When a settings set is highlighted in the table, you can change its description by editing the **Name** field.
- Click **Add** to create a new settings set. The following sub-pages are added beneath the new set in the left hand pane of the dialog:
  - **Analysis Settings**
  - **Analysis Model Options**
  - **Concrete Modification Factors**
- When a settings set is highlighted in the table, click **Copy** to duplicate it to a new settings set.
- When a settings set is highlighted in the table, click **Remove** to remove it. If there is only one settings set it cannot be removed.

Locate the new set in the left hand pane of the dialog to edit its **Analysis Settings**, **Analysis Model Options** and **Concrete Modification Factors** as required.

Analysis Settings

NOTE It is anticipated that most users will not need to adjust the default analysis 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 higher modes until the stopping criteria is fulfilled.

Item	Description
	• **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**
Stopping Criteria	Stopping criteria prevent the analysis from 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

Item	Description
	number of modes required. Note that sufficient modes are required to attain the **Cut off frequency** set in the FAM Footfall Checks properties (typically 20 Hz). With the default number of 100 it will be extremely unlikely modes above this target are not found. However for larger area models it may start to take several minutes to find 100 modes. Hence if analysis speed is felt to be an issue you can consider reducing the target number of modes. For small/normal sized FAMs speed will not be an issue and there is no need to reduce the default number. See also: **Cut off frequency** in the **Footfall checks** section below).
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 against the previous iteration results. If the difference is less than the tolerance specified here,

Item	Description
	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.

Analysis Model Options

The options given below generally relate to analysis model guidance in the guides, e.g. connections assumed to be pinned for static design may be assumed to be fixed for footfall assessment.

Column Modeling	
Stack height and support	• Full height with fixed support (default) • Half height with pinned support
Modeled supports	• Use fixed support (default) • Use modeled support
Attached beam releases	• Set to fixed (default) • Use modeled releases

Wall Modeling (except bearing walls)	
Modeled supports	• Use fixed support (default) • Use modeled support
Attached beam releases	• Set to fixed (default) • Use modeled releases

Beam Modeling	
Max node spacing	Entered as a ratio of the member length (default 0.5) The purpose of this is to ensure that beam lengths with no meshed slab attached are subdivided to some degree.

Slabs	
1-way slabs to be meshed	Select the slab types that are to be meshed in the FAM.
Effective Self Weight Mass from unmeshed slabs	Specified as a proportion (default 0.25)
Composite slab orthotropic properties	By default these are auto calculated based on the thickness, but user defined properties can be adopted if required.

Spandrel Beam Properties (AISC DG11 only)	
Major axis stiffness multiplier	Any beams identified as spandrel beams have a multiplier applied to their major axis stiffness for the footfall modal analysis. The default multiplier is 2.5, as this is the value recommended in AISC DG11.

Concrete Modification Factors

Item	Description
modification factors table	These are modification factors specific to the Footfall assessment modal analysis for concrete entities (default for all = 1.0) Different factors can be applied in the model in order to adjust the following

Item	Description
	properties used in the footfall analysis: • Young's modulus, E • G: Shear modulus, G • Section inertia about local X, $I_{torsion}$ • Section inertia about local Y, I_{major} • Section inertia about local Z, I_{minor} - • Section area in compression or tension, Area • Section shear area in local Y, $A_{parallel\ to\ minor}$ • Section shear area in local Z, $A_{parallel\ to\ major}$ • Shell thickness, t If you make changes to any of the factors, footfall analysis must be repeated.
Factor for increased dynamic stiffness	This factor (default 1.1) is applied to the Young's modulus value for all concrete element types in addition to the E factors specified in the above table, (unless the below option to **Use fixed dynamic modulus of elasticity** is selected).
Use fixed dynamic modulus of elasticity	When this option is enabled you can specify the value to be used for all concrete entities regardless of their concrete grade (which will otherwise determine their E value). When this option is enabled the **Factor for increased dynamic stiffness** is not applied.

Footfall Checks (SCI P354)

NOTE This page is only available in the global settings, not the model specific settings.

Each Floor Analysis Model (FAM) inherits these values from the global settings as it is created, the values can then be edited in the properties window for any selected FAM.

Refer to SCI P354 for further guidance.

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 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.

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
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 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

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
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.
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**.

Footfall Checks (SCI P354)			
Item	**Default**	**Range**	**Description**
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 be given a warning status.

Footfall Checks (CCIP-016)

NOTE This page is only available in the global settings, not the model specific settings.

Each Floor Analysis Model (FAM) inherits these values from the global settings as it is created, the values can then be edited in the properties window for any selected FAM.

Refer to CCIP-016 for further guidance.

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
Cut off frequency	20 Hz	1-50	Frequency above which post processing calculations are not performed.

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
			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 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.

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
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 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

Footfall Checks (CCIP-016)			
Item	**Default**	**Range**	**Description**
			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 be given a warning status.

Footfall Checks (AISC DG11)

NOTE This page is only available in the global settings, not the model specific settings.

Each Floor Analysis Model (FAM) inherits these values from the global settings as it is created, the values can then be edited in the properties window for any selected FAM.

Refer to AISC DG11 for further guidance.

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

Footfall Checks (AISC DG11)			
Item	**Default**	**Range**	**Description**
			more accurate values are not available.
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.

See also

Footfall assessment handbook (page 559)

9.8 Footfall assessment references

1. **A. L. Smith, S. J. Hicks, and P. J. Devine, "Design of Floors for Vibration: A New Approach," Ascot, Berkshire, 2009.**
2. **T. M. Murray, D. E. Allen, E. E. Ungar, and D. B. Davis, "Steel Design Guide 11: Vibrations of Steel-Framed Structural Systems Due to Human Activity," Chicago, USA, 2016.**
3. **M. R. Willford and P. Young, "A Design Guide for Footfall Induced Vibration of Structures," Camberley, Surrey, 2006.**
4. **British Standards Institution, "Guide to evaluation of human exposure to vibration in buildings. Vibration sources other than blasting," London, UK, 2008.**
5. **International Organization for Standardization, "Bases for design of structures - Serviceability of buildings and walkways against vibration," 2007.**

6. **British Standards Institution, "UK National Annex for Eurocode - Basis of structural design," London, UK, 2005.**
7. **S. J. Hicks, J. Brozzetti, B. Rémy, and R. M. Lawson, "Dimensionnement Des Planchers Mixtes Acier Béton Vis-a-Vis Des Vibrations," *Constr. Métallique*, vol. 40, no. 1, pp. 3–31, 2003.**

10 Slab deflections handbook

In Tekla Structural Designer you can choose to adopt a rigorous approach to slab deflection calculation using iterative cracked section analysis.

NOTE Tekla Structural Designer's iterative cracked section analysis for slab deflections is only available for the Eurocode and ACI/AISC Head Code.

- To begin, see Get started with slab deflection analysis (page 700)
- To define event sequences in Tekla Structural Designer, see Work with event sequences (page 702).
- To set up the deflection checks and place check lines, see Work with check lines (page 704).
- To perform the analysis, see Run a slab deflection analysis (page 706)
- To investigate the results, see Slab deflection results and reports (page 707).
- For further information, see Slab deflection concepts, workflows and examples (page 713)

Related videos

Comprehensive slab deflection calculations - part 1

Comprehensive slab deflection calculations - part 2

Rigorous Slab Deflection Design (ACI)

Rigorous Slab Deflection (Eurocode)

10.1 Get started with slab deflection analysis

Two different ways of checking slab deflections in Tekla Structural Designer are available. Either deemed-to-satisfy checks, or a rigorous theoretical deflection

estimation. These are introduced in the topic Slab deflection methods (page 714).

If you decide to adopt the latter approach the video links at the bottom of this page are a good start point, and the Rigorous slab deflection workflow (page 716) topic also provides an overview of the steps required.

Many topics are then discussed in more detail in Slab deflection concepts, workflows and examples (page 713), which also provides Tutorial worked examples for you to download and work through.

Related videos

Comprehensive slab deflection calculations - part 1

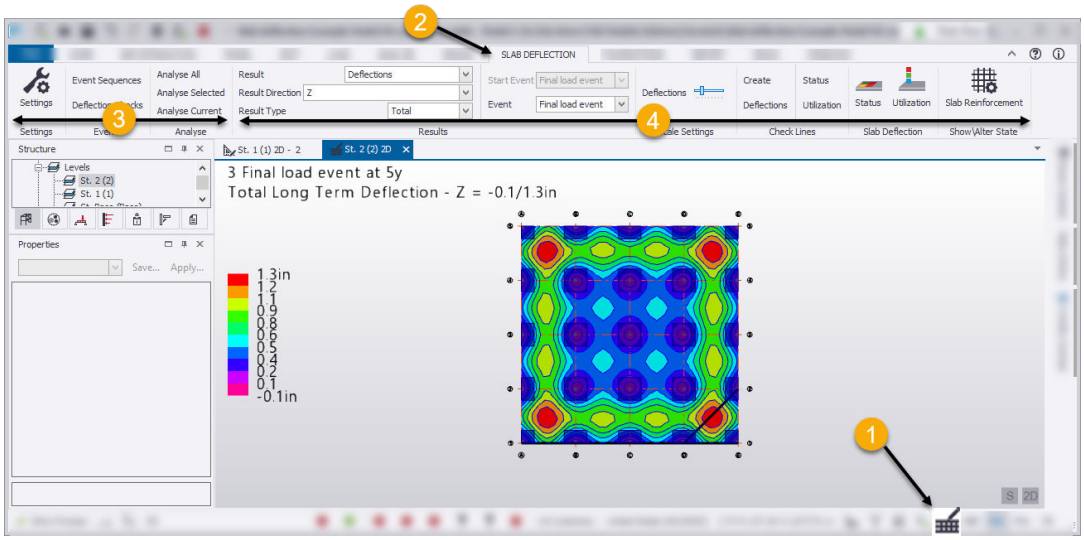
Comprehensive slab deflection calculations - part 2

Rigorous Slab Deflection Design (ACI)

Rigorous Slab Deflection (Eurocode)

10.2 Open a Slab Deflections View

To display slab deflection analysis results switch the view to the  **Slab Deflections View (1)** regime, (this also activates the **Slab Deflections** ribbon **(2)**).



Prior to analysis only the **Settings**, **Events** and **Analyse** groups **(3)** are available on the ribbon.

After a slab deflection analysis has been performed, additional groups **(4)** are added to the ribbon to allow the results to be interrogated.

See also:

Rigorous slab deflection workflow (page 716)

10.3 Work with event sequences

Event sequences are edited in the .

Add an event to the end of the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Click **Add** to Add the Event as the last event in the sequence.
4. Name the new event and specify the event parameters according to your needs. If you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
5. Click **OK**.

Insert an event within the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Select the event above which you want to insert the new event.
4. Click **Insert** to insert the event.
5. Name the new event and specify the event parameters according to your needs. If you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
6. Click **OK**.

Re-order events in the event sequence

You can Add, Insert, Remove and Re-Order Events from the Model Event Sequence page.

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.

2. Go to the **Model Event Sequence** sub page.
3. In the **Drawing Variant** list, select the Event you want to move.
4. Click**Move Up** or **Move Down** until the Event is in the required position.
5. Click **OK**.

Remove an event from the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Select the event which you want to remove.
4. Click **Remove** to remove the event.
5. Click **OK**.

Edit event parameters

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page, or specific custom event sub page as required.
3. Edit the event parameters according to your needs. If editing the Model Event Sequence and you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
4. Click **OK**.

Edit event loadcases

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page, or specific custom event sub page as required.
3. In the left hand pane, expand the Event Sequence if required so that you can select the Event.

   The available and included loadcases are displayed.
4. Move loadcases between the available and included lists as required. If editing the Model Event Sequence and you want the same loadcases to

apply to custom events, ensure the **Update custom event sequences** box is selected.

5. Click **OK**.

Create a custom event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.

2. Go to the **Event Sequences** sub page.
3. Click **Add** to add a new Custom Event.
4. In the right hand pane click on the event name to rename it.
5. In the left hand pane click on the event name in order to define the event parameters in the sequence.
6. Click **OK**.

Apply a custom event sequence to a submodel

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.

2. Go to the **Submodels** sub page.

   The table displays the event sequence assigned to each submodel.

3. Use the droplist to change the Event Sequence assigned to a specific submodel as required.
4. Click **OK**.

10.4 Work with check lines

User defined Check Lines can be placed across 2D element meshes, They are similar to 2D results strips, (but with zero width and different properties).

From these check lines, deflection results are determined from the shell/plate/membrane nodal analysis results - these can then be checked against specified limits for slab design purposes.

Engineering judgment is required when positioning the check lines to ensure worst case deflections are obtained.

Create the deflection checks to be applied to check lines

1. On the **Slab Deflection** tab, click **Deflection Checks**

   The is displayed, containing any checks that have been defined.

2. Click **Add**

   A new check is created in the table.

3. Select the name in order to give it a more descriptive title.
4. Choose the check **Type**: (Total, Instantaneous, or Differential).
   a. If the Type is Differential, select the **Start Event** from the droplist.
5. Select the **Event** to check from the droplist.
6. Enter the **Deflection Limit** to be checked.
7. Select **Use in new Check Lines** if you want this check to be performed in each check line that is created.

Create a check line

Check lines can only be created while in the **Slab Deflections View** regime.

1. Open a 2D view of the FE mesh where the check line is to be placed.
2. On the **Slab Deflection** tab, click **Create**

   The Check Line properties are displayed in the **Properties** window.

3. Adjust the properties to specify:
   a. The number of stations
   b. The upper and lower Flat Zone Reductions
   c. Up to 6 deflection checks to be performed (these can either be selected from the predefined Deflection Checks Catalogue, or a new check can be defined)
4. Click a point where the strip is to start.
5. Click a 2nd point where the strip is to end. (Neither start or end points have to match nodes in the mesh.)

   A check line is created between the points that you identified.

6. Either continue to place further check lines, or if done, press **Esc** to exit.

Delete a check line

To be able to delete a check line you must first ensure that Slab Deflection Check Lines are switched on in Scene Content.

1. Open a view displaying the check lines to be deleted.

2. Click or press **Delete**
3. Click on the check lines to be deleted.

10.5 Run a slab deflection analysis

The same basic analysis process is followed irrespective of whether the current level (sub-model), a selected level, or all slabs in the model are analyzed.

In simple terms, events are considered in sequence.

For each event:

- An iterative cracked section analysis including long term effects determines the deflection at the end of the event.
- An additional analysis using the determined state of cracking along with short term cracked properties is undertaken to calculate the total instantaneous deflection associated with the event.
- The state of cracking is carried forward to the next event as the starting point.

To run the analysis for specific or all sub models see the following instructions.

Run a slab deflection analysis for the current sub model

To analyze a specific sub model:

1. Open a view of the required sub model from the Structure Tree in the Project Workspace.
2. On the **Slab Deflection** tab, click **Analyze Current** At the end of the analysis the active sub model view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

Run a slab deflection analysis for all sub models

To analyze all sub models:

1. On the **Slab Deflection** tab, click **Analyze All** At the end of the analysis the active view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

Run a slab deflection analysis for selected sub models

To analyze selected sub models:

1. On the **Slab Deflection** tab, click **Analyze Selected** At the end of the analysis the active view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

10.6 Slab deflection results and reports

Click the links below to find out more:

- Display slab deflection analysis results (page 707)
- Display check line results (page 709)
- Display slab deflection status and utilization (page 710)
- Slab deflection optimization (page 712)
- Slab Deflection Reports
- Slab Reinforcement

Display slab deflection analysis results

Once a slab deflection analysis has been performed analysis results can be reviewed for the chosen level (sub-model), or the entire structure - dependent upon your chosen analysis.

The following results are available for display in the **Slab Deflections View** regime by making appropriate selections from the **Results** group of the **Slab Deflection** ribbon.

- **Deflections**.
- **Extent of Cracking** at any load event.
- **Relative Stiffness** in a particular result direction for any specified event.
- **Effective Reinforcement** for a chosen result direction for each FE element.

Display deflection contours

Three deflection types are available for review:

- **Total** deflection at the end of any event.

- **Differential** deflection between any two events (Start of Event and End of Event).

  For further details, see: Interrogating slab deflection calculations (page 739)
- **Instantaneous** deflection (not actually needed for TR 58).
  - This is the deflection when the entire event loading is applied to a version of the model using the established extent of cracking along with short term analysis properties.
  - US codes require an assessment of the instantaneous deflection associated with the imposed load only. This is achieved by adding extra events at the same time as the final event where only the required imposed load is applied.

To display deflection contours:

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Deflections**
2. Then from the same group:
   a. Make your selection from the **Result Direction** list
   b. Make your selection (Instantaneous, Total, or Differential) from the **Result Type** list
3. From the **Results** group select the **Event**, and if viewing differential deflections also select the **Start Event**.

NOTE Hover the cursor over a specific solver node in order to see the nodal displacement at that location displayed in a tooltip.

Display extent of cracking

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Extent of Cracking**
2. Make your selection from the **Result Direction** list
3. From the **Results** group select the **Event**.

NOTE Hover the cursor over a specific solver element in order to see the detailed cracking calculation parameters for that element displayed in a tooltip.

Display relative stiffness

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Relative Stiffness**

2. Make your selection from the **Result Direction** list
3. From the **Results** group select the **Event**.

NOTE Hover the cursor over a specific solver element in order to see the detailed relative stiffness calculation parameters for that element displayed in a tooltip.

Display effective reinforcement

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Effective Reinforcement**
2. Make your selection from the **Result Direction** list

NOTE Hover the cursor over a specific solver element in order to see the detailed effective reinforcement calculation parameters for that element displayed in a tooltip.

Display check line results

Once a slab deflection analysis has been performed, the Check Line results are available for display in the **Slab Deflections View** regime.

Display deflections along all check lines

1. From the **Slab Deflection** ribbon **Check Lines** group click **Deflections**

   The deflected shape of each check line and an accompanying legend is displayed in the current view.

   For clarity you may want to switch off other results to obtain a clearer view.

Display detailed deflections and average slopes along an individual check Line

1. Right click on the check line you want to view and select Open deflections check view from the context menu that is displayed.
2. On the ribbon, select the Result Type and the Event from the Loading Analysis tab.

   The deflections and average slopes are displayed in a deflections view.

Display check line status and utilization

Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio

Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization.

To view check line status or utilization:

1. From the **Slab Deflection** ribbon **Check Lines** group
   - Click **Status**
   - Click **Utilization**

   Color codes are used to graphically display the status or utilization of each Check Line.

Display slab deflection status and utilization

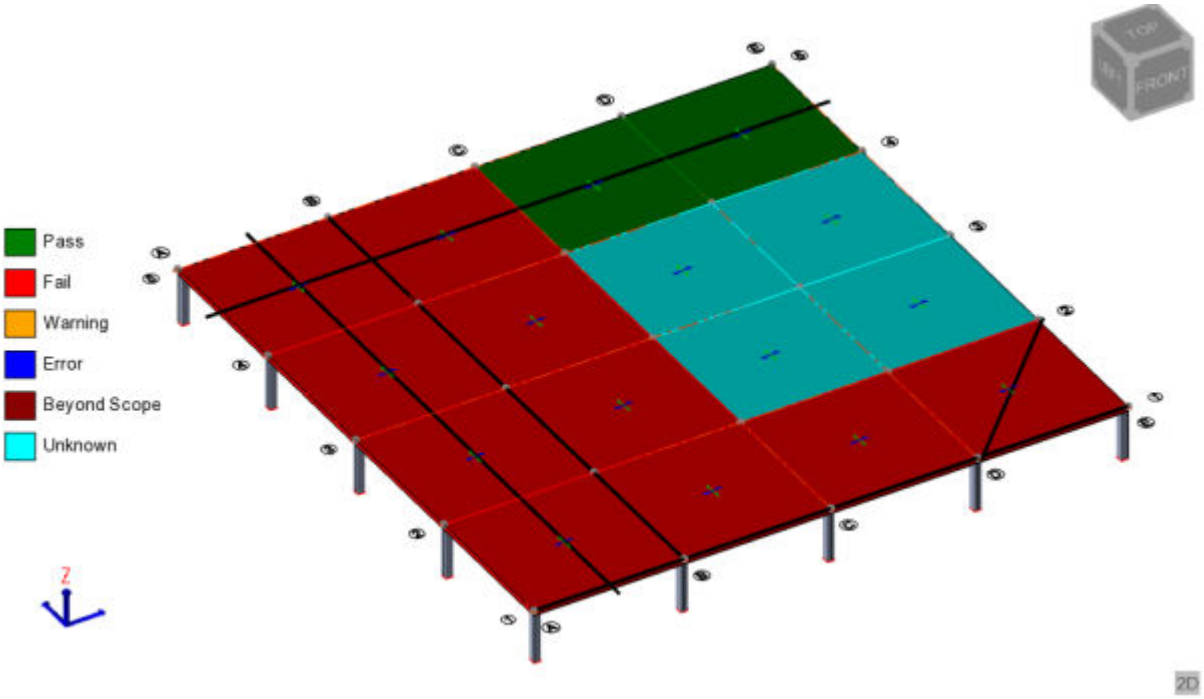
Once a slab deflection analysis has been performed, the Slab deflection status and utilization are available for display in the **Slab Deflections View** regime.

Display slab deflection status

1. From the **Slab Deflection** ribbon **Slab Deflection** group click **Status**

   Color codes are used to graphically display the status of each slab item.

In the screenshot below;



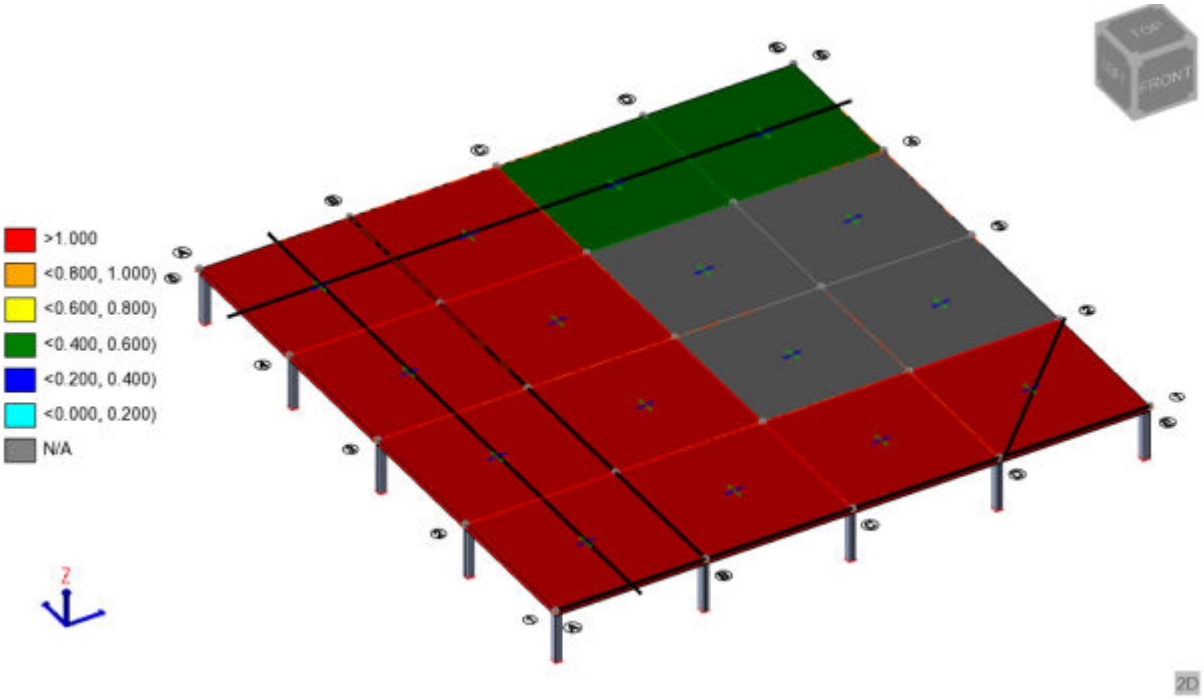
- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

Display slab deflection utilization

1. From the **Slab Deflection** ribbon **Slab Deflection** group click **Utilization**

   Color codes are used to graphically display the utilization of each slab item.

The worst utilization from all associated check lines is displayed.



Slab deflection optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

View slab deflection reports

A Slab Deflection Check Lines report can be generated for multiple check lines from the Reports toolbar. The same report can also be generated for individual check lines from the right click menu.

View an individual check line report

1. Right click on the check line you want the report for and select **Report for Member** from the context menu that is displayed.

   A report is displayed consisting of a defection check summary table.

NOTE Optionally the report can be configured to include a picture showing the Check Line location within the slab, (by clicking Member Report on the Reports ribbon and choosing the Slab Deflection Check Line as the Member Type.)

View all/multiple check line reports

1. From the **Report** tab click **Model Report...**

   A report is displayed consisting of a defection check summary table.

2. Create a report that includes the Slab Deflection Check Lines chapter (located under Concrete>Slab/Mat Design per Plane).
3. Apply Model filters to the report as required.

View an effective modulus report

A slab Effective Modulus Report can be exported to Excel for an individual slab panel, or all panels.

1. Right click on a slab panel and select **Export Eff. Modulus Report to Excel** from the context menu that is displayed.
2. Choose whether to export for the current slab item, or all slab items from the sub menu that is displayed.

   The report opens in Excel, (assuming Excel is installed).

10.7 Slab deflection concepts, workflows and examples

These topics take a detailed look at the concepts and workflows involved when applying iterative cracked section analysis within Tekla Structural Designer.

NOTE Tekla Structural Designer's iterative cracked section analysis for slab deflections is only available for the Eurocode and ACI/AISC Head Code.

- Slab deflection methods (page 714)
- Rigorous slab deflection workflow (page 716)
- Factors that affect rigorous slab deflection estimates (page 716)
- Event sequences (page 724)

- Slab deflection analysis sequence (page 737)
- Total, differential, and instantaneous deflection types (page 738)
- Slab deflection calculations in depth (page 739)
- Check lines (page 751)
- Slab deflection status and utilization (page 753)
- Slab deflection example (Eurocode) (page 755)
- Slab deflection example (ACI) (page 788)

Related video

Rigorous Slab Deflection (Eurocode)

Rigorous Slab Deflection Design (ACI)

Slab deflection methods

Concrete is considered a durable and economic material for floors systems. However, reinforced concrete slabs deflect. The magnitude of the deflection is more complicated for concrete as deflection increases with time. It's long term behavior is characterized by cracking caused by flexure, shrinkage and creep. If this is not taken into consideration by allowing adequate tolerances to glass facades and internal partitions for example then problems can arise.

Tekla Structural Designer provides two alternative methods for checking deflections. Either:

- Deemed-to-satisfy checks, or
- A rigorous theoretical deflection estimation using iterative cracked section analysis

Deemed-to-Satisfy Checks

A couple of deemed-to-satisfy checks are presented here.

1. The use of a limiting span-to-depth ratio (L/d) method. This method is assumed to 'be adequate for avoiding deflection problems in normal circumstances'. It can only be considered as a rough deflection estimate and is not intended to predict how much a member will actually deflect. Total deflection is expected to be < span / 250 (Eurocode) or < span / 250 (US) and deflection affecting sensitive finishes is expected to be < span / 500 (Eurocode) or < span / 480 (US)
2. A linear analysis with adjusted analysis properties.

It is important to appreciate that these deemed-to-satisfy methods do not predict actual deflections even though the linear analysis method provides a

total deflection that can be checked against the span / 250 (Eurocode) or < span / 250 (US) limit mentioned above.

When normal deflection limits do not apply, for example, due to stricter usage limits, glazed cladding systems or where a faster pace of construction is applied then the 'deemed-to-satisfy' checks are no longer applicable - the alternative is a rigorous deflection estimation which is the primary topic for this guide.

See also

- Deemed to satisfy slab deflection checks example (Eurocode) (page 756)
- Deemed to satisfy slab deflection checks example (ACI) (page 789)

Rigorous theoretical deflection estimation

The rigorous theoretical deflection assessment takes into account cracking, creep and shrinkage over time.

In the UK, rigorous deflection estimation is taken to mean deflection estimation in accordance with the Concrete Society Technical Report 58.

The principle of assessing deflections rigorously involves assessing the curvatures induced by both load and shrinkage, adding them together and then the total curvature is translated into a deflection.

The Technical Report discusses the importance of construction events. Total deflection at the end of every event comprises:

- An instantaneous deflection which is influenced by the extent of cracking
- An additional accumulated creep deflection
- An additional accumulated shrinkage deflection.

Once these totals are known, differential deflections between any two events can be calculated.

The Technical Report gives detailed guidance on some very complex looking calculations - it all seems very "rigorous". However, we must not lose sight that the material - Concrete is a very variable material. Furthermore, how accurately can we really predict input parameters such as event loads and timings? The report advises that deflection accuracy can only be considered an estimate in the range +15 to -30%.

In the US, the basic approach described in ACI 318 has a similar approach to cracking, interpolating between the fully cracked and the uncracked states, (although it doesn't recognize the reduction in tension stiffening). For creep and shrinkage, in ACI 318 there is a single multiplier for the deflection calculated from the cracked flexural rigidity. There are additionally two ACI Committees 435 and 209 which go into more detail about creep and shrinkage calculations. The US user can therefore either adopt the basic ACI 318 approach, or, take on board the ACI Committee 435 & 209 guidance. In special situations the TR58 approach could even be considered.

Expectations - It is incorrect to think that rigorous methods will provide greater economy. i.e. by allowing the engineer to reduce slab thickness or the quantity of reinforcement. The end result is greatly influenced by the various input parameters which each can impact on the deflection.

See also

- Rigorous slab deflection workflow (page 716)
- Rigorous slab deflection analysis example (Eurocode) (page 759)
- Rigorous slab deflection analysis examples (ACI) (page 791)

Rigorous slab deflection workflow

You perform rigorous slab deflection analysis in Tekla Structural Designer from the **Slab Deflection** toolbar.

The suggested workflow is as follows:

1. Define the slab loading event sequences (page 724).
2. Run a slab deflection analysis (page 706) and review deflections for the final load event.

   You can analyze the current or selected level (sub-model) or choose to analyze every slab in the model. Obviously your choice has an effect on the time necessary to undertake the iterative slab deflection analysis.
3. Define the slab deflection checks via the **Slab Deflection Check Catalogue** and then place Check lines (page 751) as required.
4. Graphically review the Slab deflection status and utilization (page 753).
5. Make adjustments as necessary until the status passes. For more information, see Slab deflection optimization (page 712).

NOTE A reasonable level of slab reinforcement should be established prior to running the Slab Deflection Analysis as this will have a significant impact on the deflection calculations.

Factors that affect rigorous slab deflection estimates

Many input parameters have an impact on the rigorous deflection estimates. These include:

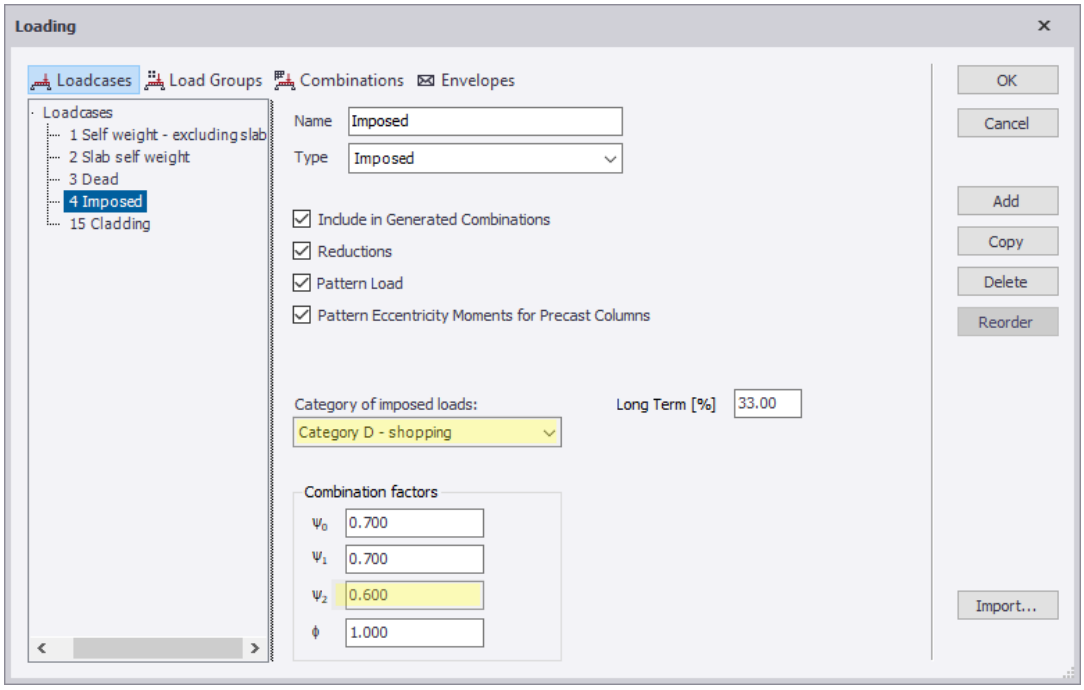
- Level of Restraint
- Concrete properties
- Stiffness adjustments
- Allowance for shrinkage effects

- Event sequence parameters
- and even the assumed analysis properties of connected columns and walls

Some of the key factors are described below, these vary depend on the regional code being worked to.

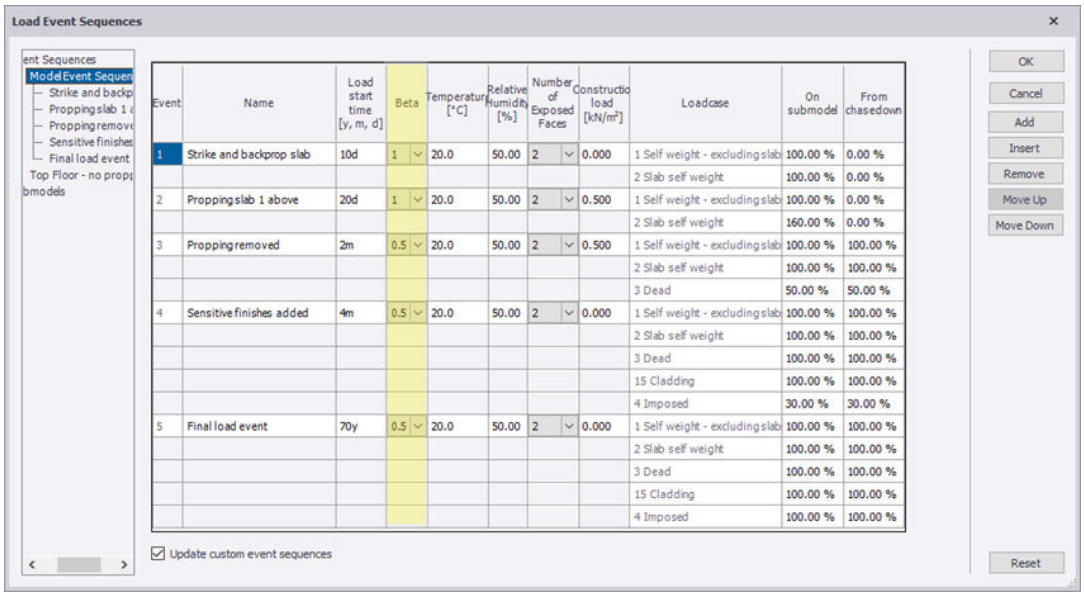
Quasi-permanent load factors (EC2)

An accurate prediction of deflection requires a realistic assessment of the loading. To the Eurocodes long term loads are termed quasi-permanent and a ψ_2 (Psi_2) factor is applied to the imposed load which varies based on the use of the structure. These are dealt with when defining the imposed loadcase.



Beta coefficient (EC2)

This property is set in the Event Sequence dialog.



Beta relates to the duration of the load and tension stiffening effects.

NOTE Tension stiffening occurs when the concrete is not fully cracked - because there is still concrete in the tension zone that transfers some tensile forces, the stiffness is greater than that of the fully cracked stiffness (and less than the uncracked stiffness).

Due to phenomena such as increased cracking or local bond failure, tension stiffening effects reduce over time - Beta is used to account for these, (see EC2 Clause 7.4.3 and TR58).

- For loads with a short duration, and for cyclic loads, Beta should be set to 1.0.
- For loads with a long duration, Beta should be set to 0.5.

NOTE 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.

Since these phenomena are irreversible, it is not recommended that Events have a value of Beta greater than that set for any previous Event and a warning will be issued if you do this.

However, there are circumstances where you may wish to have a value of Beta=0.5 in the earlier events, and Beta=1.0 in a later Event. For example, TR58 suggests that Beta=1.0 be used for the variable part of imposed load (if you wish to consider that at all).

In this case the analysis will permit you to enter these values. However, because the reduction in tension stiffening is carried forward and is irreversible in the analysis, caution is advised.

To explain this issue more fully, the impact of choosing Beta=1.0 at a later event, for different extents of cracking, is explored below.

Consider 3 possible configurations of a model, where:

1. There are many elements that are uncracked at the end of earlier events. For a later event where the duration is short (which implies you should set Beta=1), setting Beta=0.5 could lead to an overestimation of incremental cracking, and subsequently overestimation of deflection.
2. There are many elements that are cracked in an earlier event, for which you have chosen Beta=0.5. For a later event where the duration is short (which implies you should set Beta=1), if cracking increases in that Event, setting Beta=1.0 could lead to an underestimate of incremental cracking, and subsequently an underestimate of deflection for this later event.
3. The majority of cracking occurs in an earlier event, for which Beta=0.5, if cracking does not increase in a later Event, the value of Beta in that later Event will have minimal impact on deflection. To ascertain whether the cracking increases, you may run separate analyses, with values of Beta=0.5 & 1.0, and compare the results.

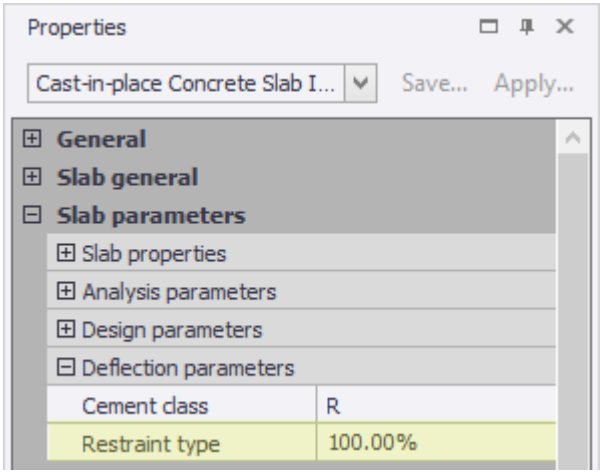
In summary, it is prudent to consider whether cracking is likely to increase in later events for which you wish to set Beta=1.0.

However, it is likely that you will observe minimal difference between the total deflection estimates, and that other assumed values will be of much greater significance.

It is straightforward to run the analysis with different values of Beta to determine whether this is the case.

Restraint type (EC2)

Restraint type, which is located under the Deflection parameters heading in the slab item Properties Window is used for specifying the degree of restraint resisting shrinkage.

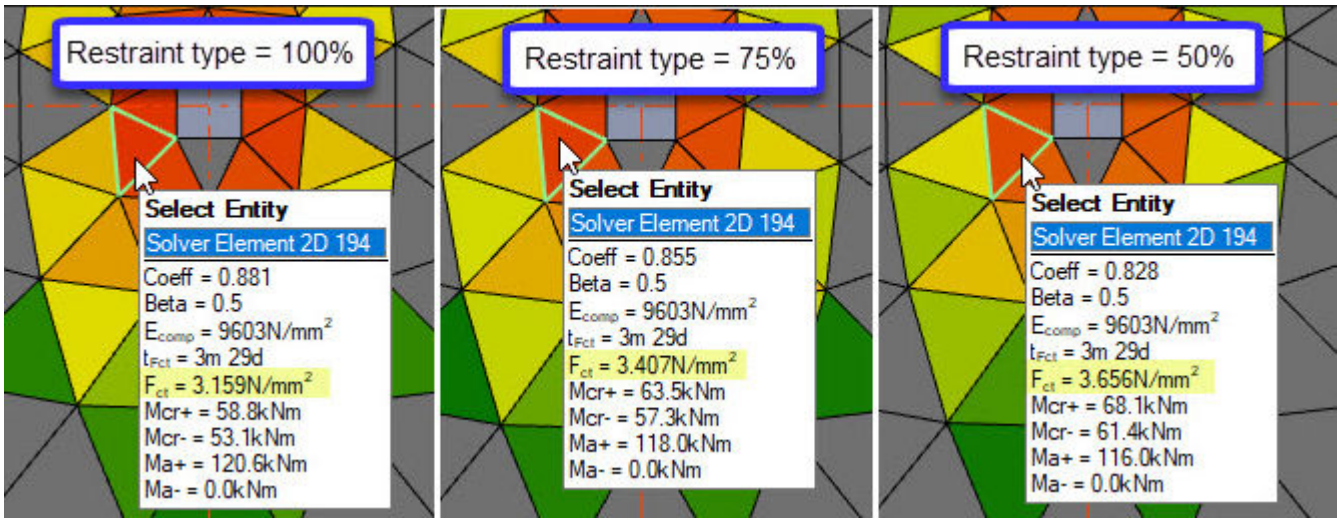


When setting its value the following should points should be considered:

- The default of 100% corresponds to high restraint and is conservative.
- The range of input is 50%-100%
- 50% corresponds to low restraint and is not conservative.

Changing the value will affect tensile strength (f_{ct}) as follows:

- **For Restraint Type = 100% (High) -** the tensile strength ***f_ct = f_ctm(t)*** which is the strength adjusted for the concrete age at time of loading (for each load event)***t*** per EC2 cl 3.1.2 (9). This is the minimum value of **f_ct** (at any given age).
- **For Restraint Type = 50% (Low) -** following Cl 7.4.3(4), we refer to cl 3.1.8(1) for flexural tensile strength and set ***f_ct = f_ctm,fl*** - again adjusted for the concrete age per EC2 cl 3.1.2 (9). This is the maximum value of ***f_ct*** (at any given age)
- **For 50% < Restraint Type < 100%** - we linearly interpolate between these max/min values.
    - Note you can verify the value of ***f_ct*** calculated and used by the program for the current input for each load event - the value is reported in cursor tooltip for any selected element when viewing the "Extent of cracking" Results as shown in the picture below.

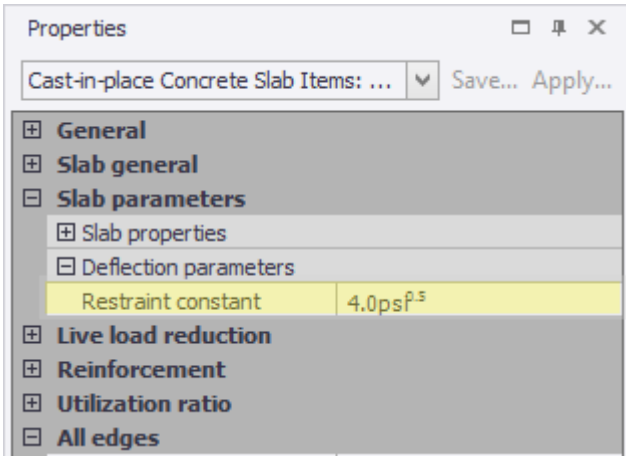


The value to adopt is a matter of judgment - for guidance on this refer to EC2 cl 7.4.3(4) and TR58.

NOTE Restraint type is actually a property of the parent slab, so if you change it for one slab item it will also be updated for all other slab items in the same slab.

Restraint constant (ACI)

This property is located under the Deflection parameters heading in the slab item Properties Window.



NOTE The Restraint constant is actually a property of the parent slab, so if you change it for one slab item it will also be updated for all other slab items in the same slab.

For US customary units, restraint constant values from ACI 435 are:

- For situations with significant restraint - 4.0
- For insignificant restraint - 7.5

The default is conservatively set to 4.0.

The allowable input range is between 1 and 10 and is a user specified value by the engineer.

The restraint constant is used to determine the Modulus of Rupture, f_r = "restraint constant" x λ x √(f_c')

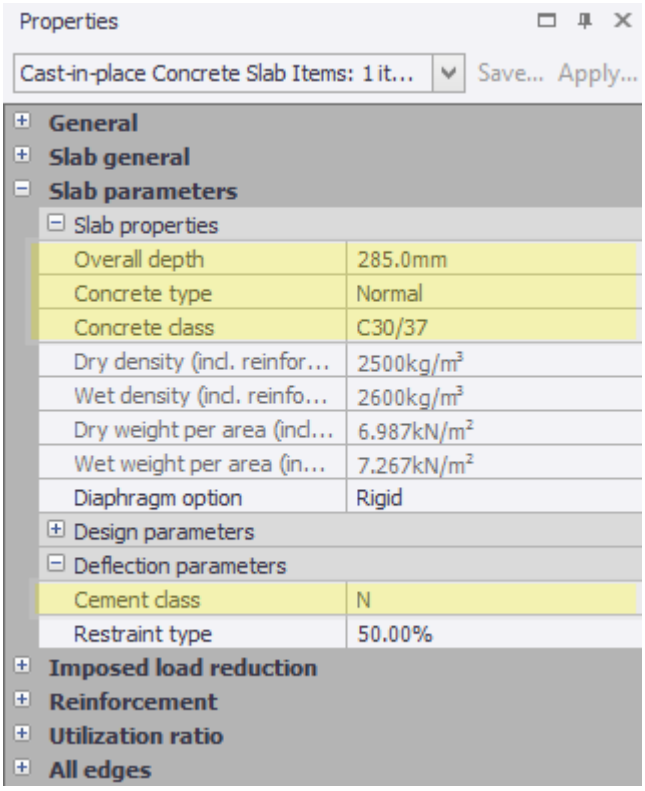
Where:

f_c' = specified compressive strength (psi)

λ = 1.0 (for normal weight concrete)

Concrete Properties (Eurocode)

The concrete properties that exist as a property of the individual slabs defined in the model are very important. Stiffness is variable and is aggregate dependent. The Cement class can also affect deflection.



The elastic deformation of concrete largely depends on its composition (especially the aggregates). For C35/45 concrete, Eurocode 1992-1-1:2004

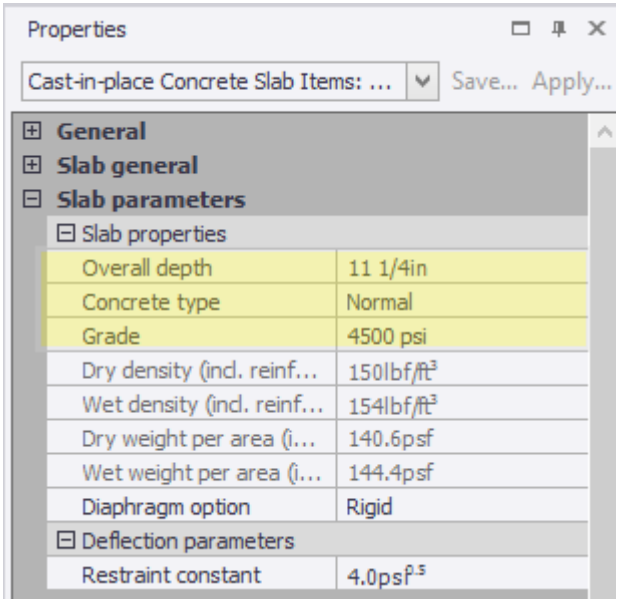
Table 3.1 provides a modulus of elasticity, E of 34GPa (34000 N/mm^2) as a mid range value using quartzite aggregate. For different aggregates this can range between -30% and +20%.

What is the correct E value for your concrete?

If you know the value, you should set up a new material grade and assign it to the elements in the model. It may, however, be easier when assessing the impact, to use the stiffness adjustment option, for details of which see the **Stiffness Adjustments** section below.

Concrete Properties (ACI)

The concrete properties that exist as a property of the individual slabs defined in the model are very important. Stiffness is variable and is aggregate dependent.



What is the correct E value for your concrete?

If you know the value, you should set up a new material grade and assign it to the elements in the model. It may, however, be easier when assessing the impact, to use the stiffness adjustment option, for details of which see the **Stiffness Adjustments** section below.

Stiffness Adjustments

The relative stiffness of the interconnecting elements will play a role in the force distribution and hence the deflection results.

For existing models the values can be reviewed and amended by clicking **Settings > Modification Factors** on the Slab Deflection ribbon. These are user defined values with assumed defaults.

As mentioned in the above **Concrete Properties (Eurocode)** and **Concrete Properties (ACI)** sections, an alternative to assessing the impact of a different grade could be to alter the Modification Factors specified for flat slab E and G values. i.e. Concrete material property of slab = 34000 N/mm2. What impact would using a value of 32000 N/mm^2 make? Flat slab stiffness adjustment = 32000/34000 = 0.9412

Shrinkage

Shrinkage is the strain in hardened concrete that can occur due to moisture loss.

Shrinkage is taken into consideration by making an overall adjustment to the total deflection. This approach is in line with many other software products.

The adjustment is specified in the **Slab Deflection Settings** dialog.

We recommend an allowance with an upper limit of 30%. The default is set as 0.25 (25%).

Event sequences

An Event sequence defines all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

By default a model event sequence is initially applied to all sub models in the structure; if required custom event sequences can also be defined to override the model event sequence for specific sub models.

NOTE Event sequences are NOT structure event sequences, i.e. they do not describe all the events starting from the first day of the overall construction.

Click the links below to find out more:

- Construction stage events (page 725)
- A typical model event sequence (page 725)
- Custom event sequences (page 733)
- Understanding event sequence deflections (page 736)

See also

Work with event sequences (page 702)

Construction stage events

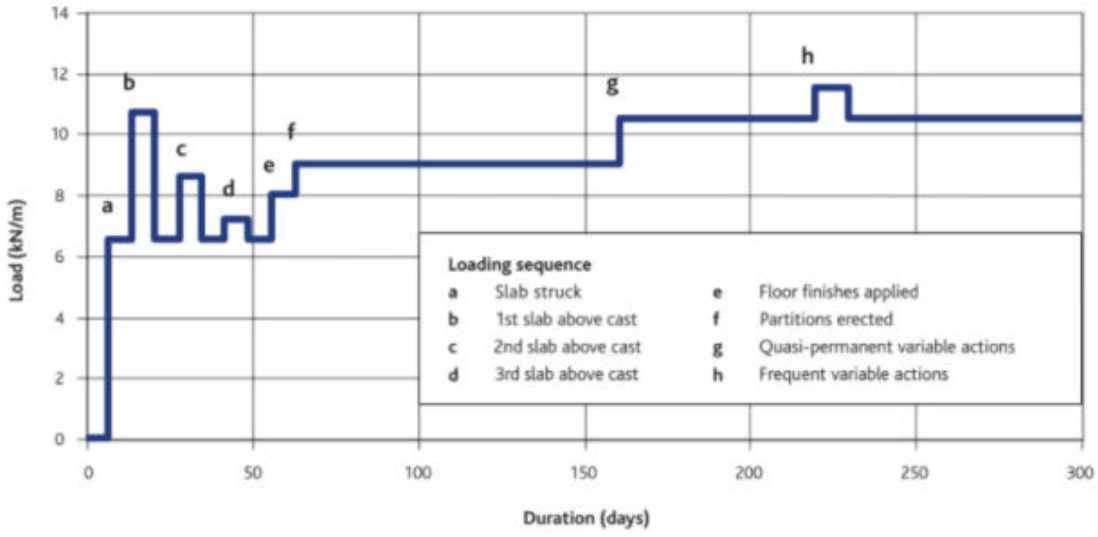
An Event Sequence defines all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

Deflection is very dependant on the extent of cracking. Once the slab has cracked then it is assumed to remain cracked. The tensile stress of concrete varies with time so careful consideration of load events are required so predicted stresses can be compared with allowable at the appropriate age. Propping loads from slabs being cast above are acknowledged to be significant early loading events which will cause cracking.

In addition to the Final load event, construction stages that could be considered are:

1. Striking
2. Casting the floor above

NOTE Propping loads from casting slabs above have an impact on already struck slabs - dependant upon the extent of back-propping adopted.

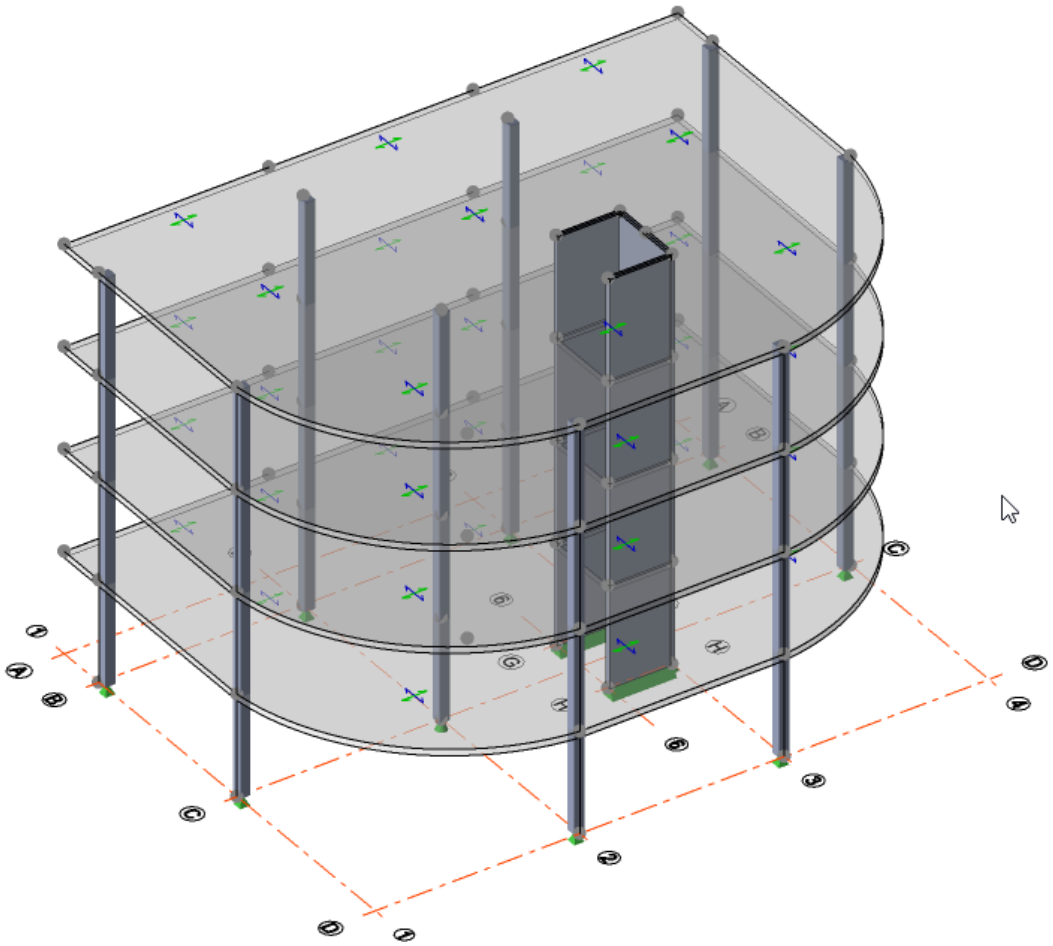


Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

3. Adding Partitions
4. Finishes

A typical model event sequence

Let us consider a multistory building where the slab layout is the same on each level.

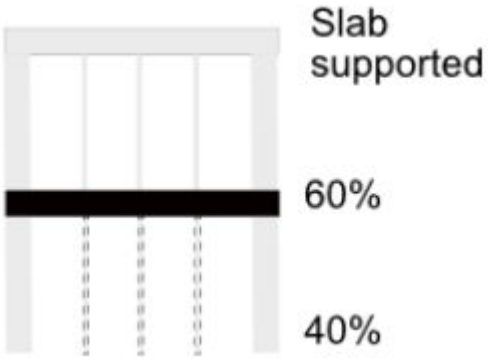


If we think about the slab model event sequence that occurs for the slab at level 1 it could be something like this:

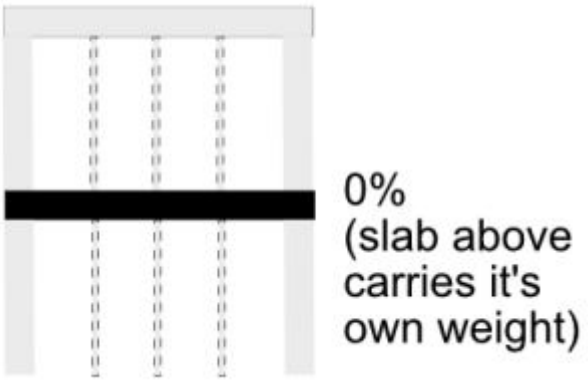
1. Strike and backprop slab (slab carries it's own weight)



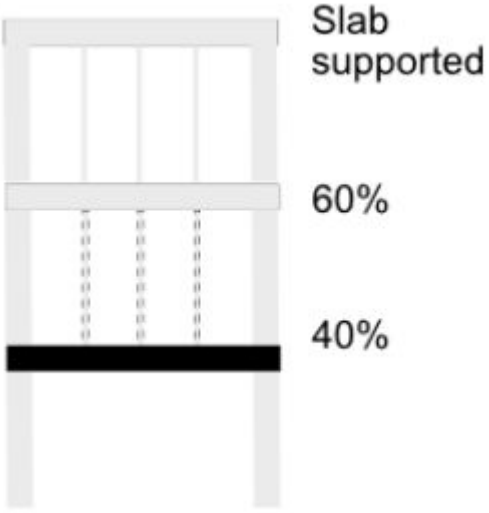
2. Cast Slab above (slab carries a proportion of the weight of slab above - the proportion is dependent on the number of levels of backpropping and there is also some debate about the efficiency of load sharing between the supporting levels. In this example we will assume 2 levels of propping and that the level directly below supports 60% and level 2 below supports 40%.)



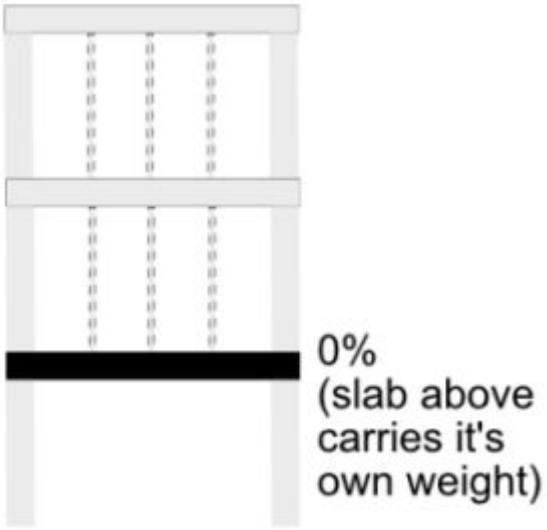
3. Strike and backprop slab above (slab above now carries its own weight)



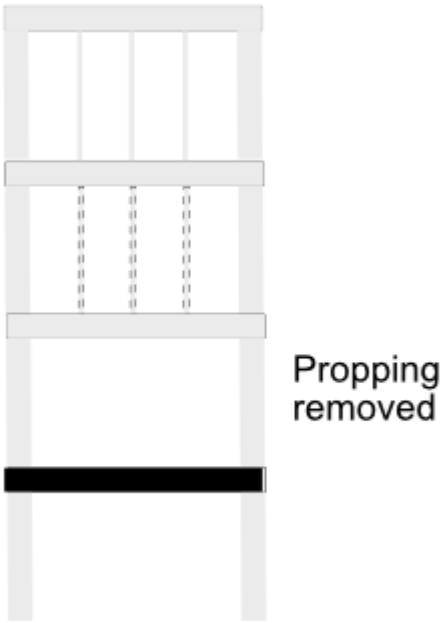
4. Cast slab 2 above (slab carries a proportion of the weight of the slab 2 above)



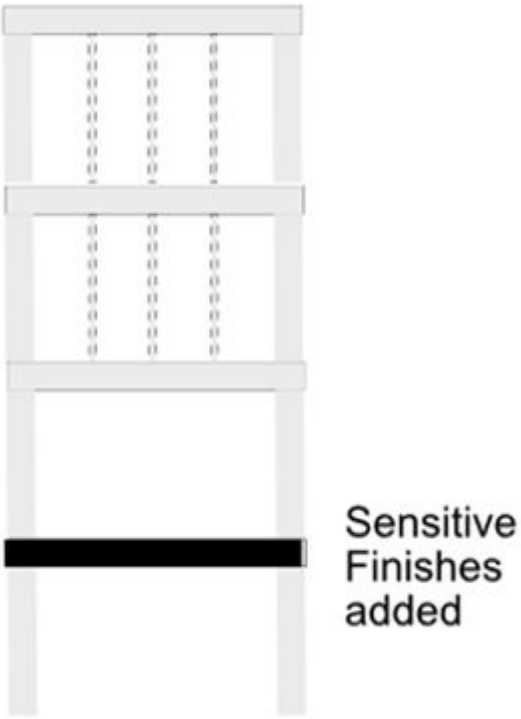
5. Strike and backprop slab 2 above (slab 2 above now carries its own weight)



6. If propping extends through 3 levels then there can be events for casting and striking the slab 3 above. In this example, the propping is removed and used two levels above.



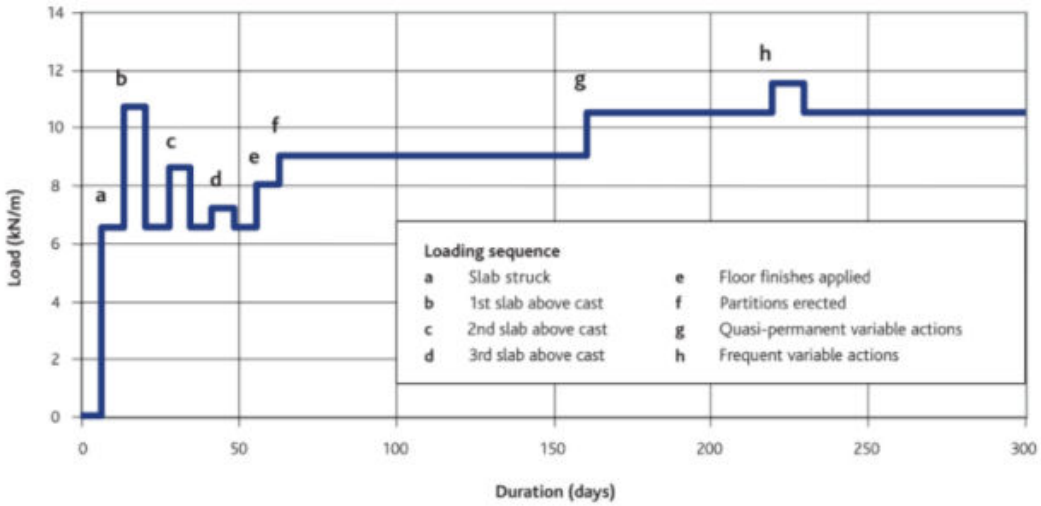
7. Additional load from finishes (in particular from sensitive finishes)



8. Start of occupation.
9. Final Load Event

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

The view shown below is a graphical representation of the above but in this case is recognizing 3 levels of backpropping



Extract from How to design reinforced concrete flat slabs using Finite Element Analysis, The Concrete Centre - Figure 2

This sort of event sequence can be described within the Event Sequences dialog as shown below.

NOTE Some of the event parameters vary between regional codes, so both the ACI and Eurocode versions of the dialog are shown.

Model Event Sequence ACI (US Imperial units)

:vent	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Constructio load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
							1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	160.00 %	100.00 %
3	Strike slab 1 above	17d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	140.00 %	100.00 %
5	Strike slab 2 above	27d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	50.00 %	50.00 %
							3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	2.350	0.800	2	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 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	33.00 %	100.00 %
9	Final load event	70y	2.350	0.800	2	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 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	100.00 %	100.00 %

Model Event Sequence Eurocode (metric)

:vent	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Constructio load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	160.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	140.00 %	100.00 %
5	Strike slab 2 above	27d	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	50.00 %	50.00 %
								3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	33.00 %	100.00 %
9	Final load event	70y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

Note the increased slab self weight when casting slab 1 above and slab 2 above where an additional 60% and 40% of the slab weight is being supported respectively. This is defined in the On submodel % column.

A slab model event loading sequence for any other internal slab i.e. level 2 would be identical to that described above.

If we now consider the uppermost slab - the roof.

- Is the slab event sequence for the roof any different?
- Can we use the slab model event sequence above for the roof?

The event sequences for casting any slabs above are not required, since there is no need to make an allowance for additional propping loads. This means that a different event sequence is necessary to deal with the roof. In Tekla

Structural Designer differences in event sequences are dealt with using a Custom Event Sequence.

Technically, we could deal with the propping events in one of two ways.

1. Delete the event
2. Keep the event, but adjust the included slab self weight to allow only for the roof load as the previous event. i.e the event has no change in load to that of the previous load event.

In Tekla Structural Designer, we must deal with the change using method 2 above. This is due to the way Tekla Structural Designer deals with Custom event sequences (page 733).

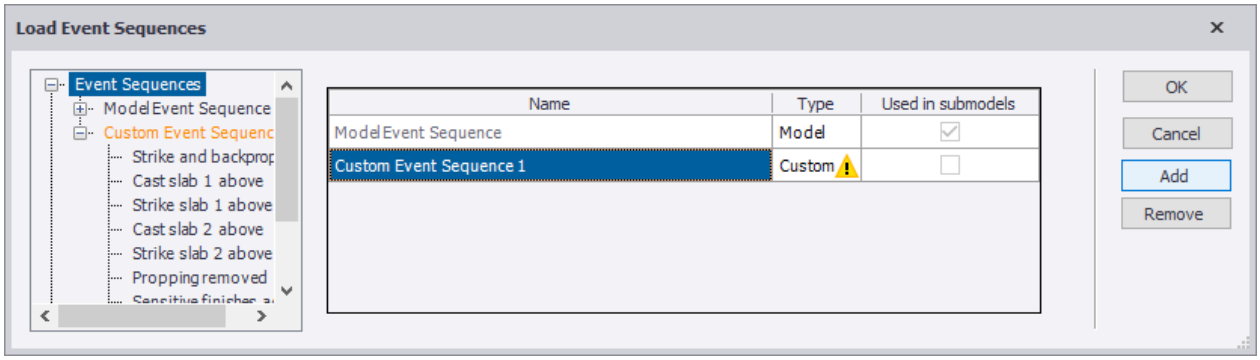
See also

Custom event sequences (page 733)

Custom event sequences

Custom event sequences are required to deal with different slab loading sequences, such as for the roof slab and special cases like transfer slabs.

A custom event sequence can be created via the Add button on the Event Sequences page (highlighted below).



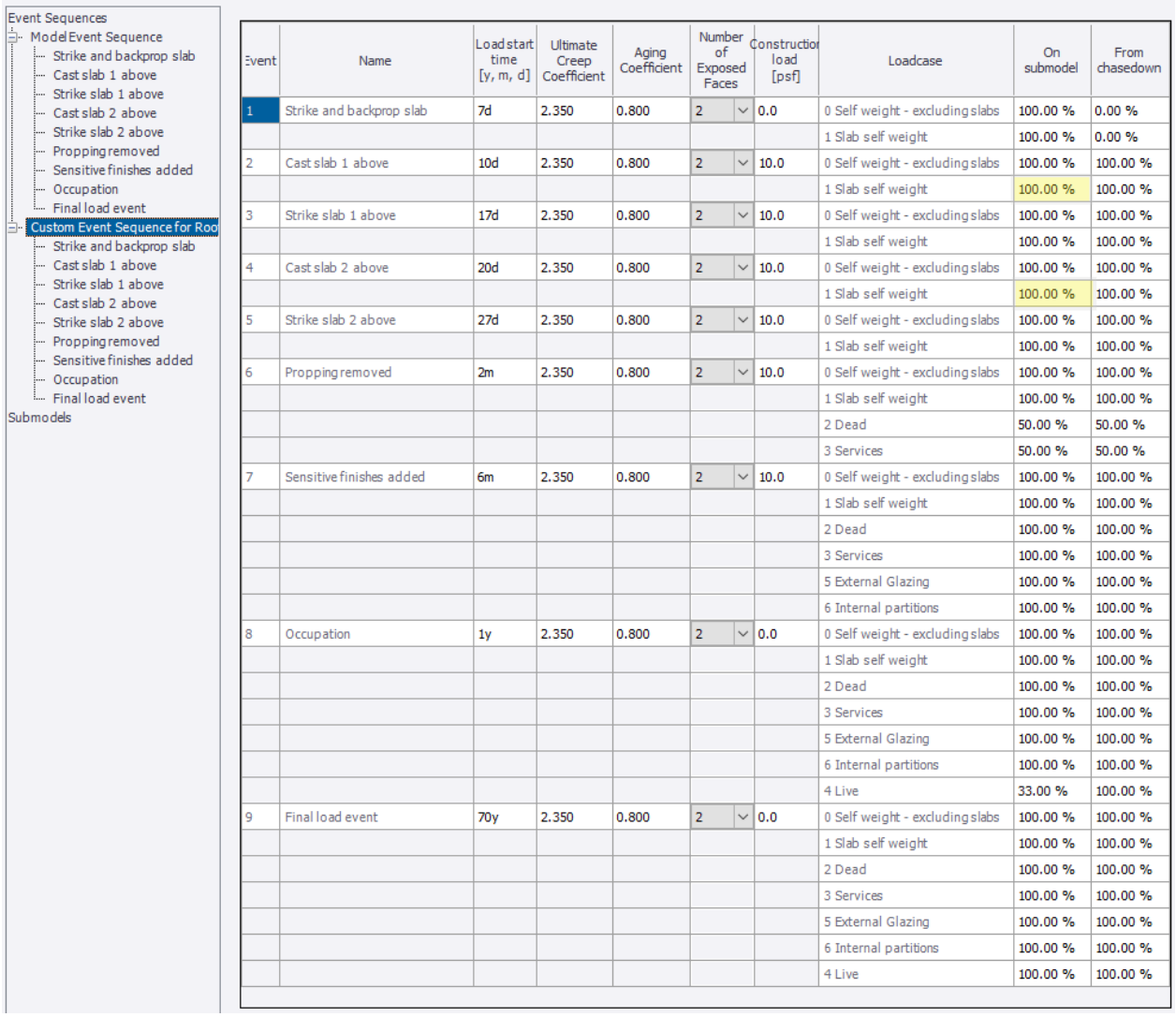
Once it has been added and given a name you can then edit the Custom event sequence by selecting it in the list.

Editing a custom event sequence

When first added, a Custom Event Sequence is identical to the Model Event Sequence. It therefore needs to be edited to achieve the required slab loading sequence.

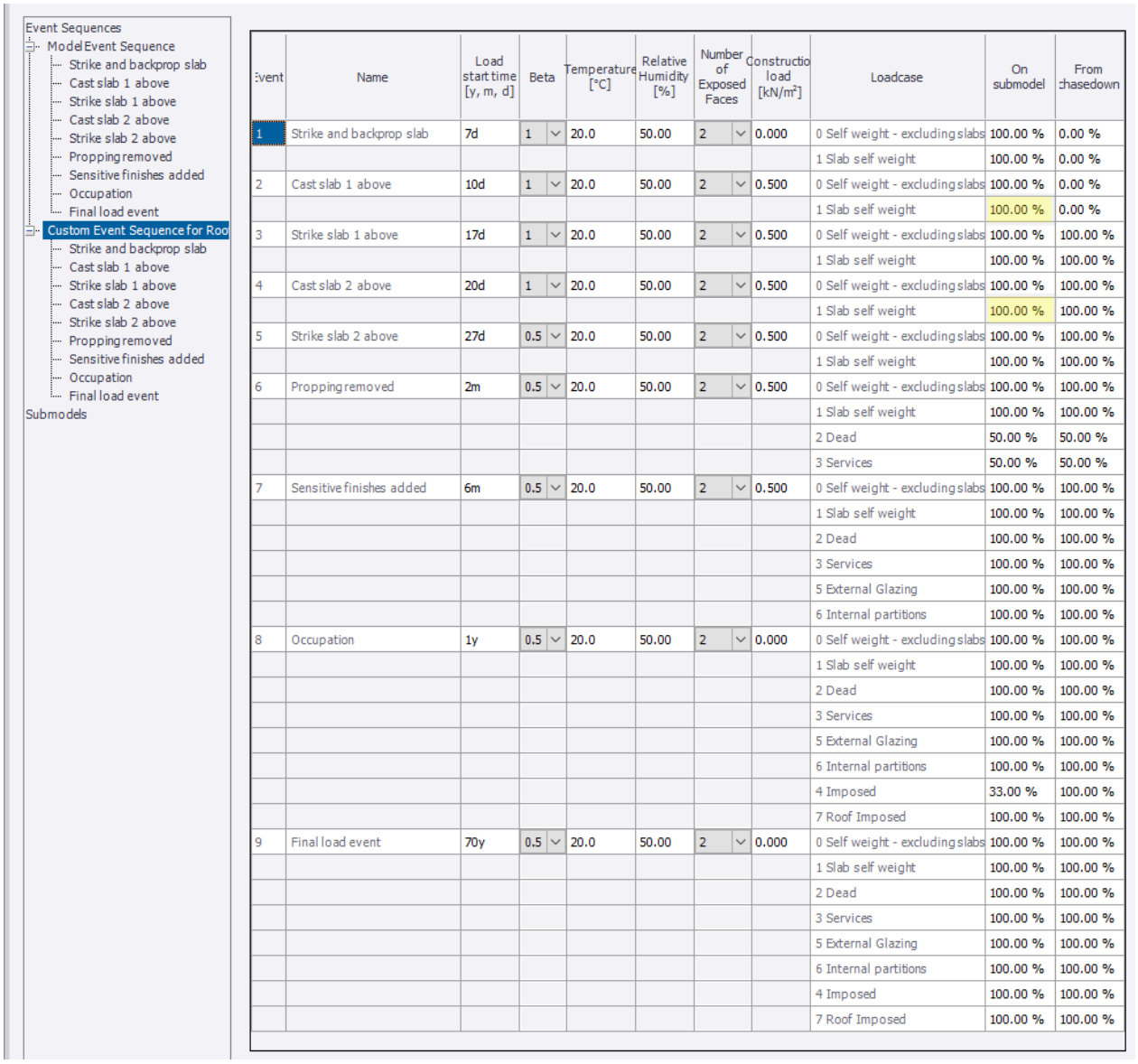
In the screenshot below, to create a custom event sequence for the roof we have edited the slab self weight load in the Cast slab 1 above and Cast slab 2 above events to 100% i.e. it is only supporting the roof slab self weight and not any extra propping load from any slabs above.

Custom Event Sequence ACI (US Imperial units)



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	Strike and backprop slab	7d	2.350	0.800	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
							1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
3	Strike slab 1 above	17d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	50.00 %	50.00 %
							3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	2.350	0.800	2	10.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	2.350	0.800	2	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 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	33.00 %	100.00 %
9	Final load event	70y	2.350	0.800	2	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 %
							3 Services	100.00 %	100.00 %
							5 External Glazing	100.00 %	100.00 %
							6 Internal partitions	100.00 %	100.00 %
							4 Live	100.00 %	100.00 %

Custom Event Sequence Eurocode (metric)



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	7d	1	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
2	Cast slab 1 above	10d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	0.00 %
								1 Slab self weight	100.00 %	0.00 %
3	Strike slab 1 above	17d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
4	Cast slab 2 above	20d	1	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
5	Strike slab 2 above	27d	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
6	Propping removed	2m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	50.00 %	50.00 %
								3 Services	50.00 %	50.00 %
7	Sensitive finishes added	6m	0.5	20.0	50.00	2	0.500	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
8	Occupation	1y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	33.00 %	100.00 %
								7 Roof Imposed	100.00 %	100.00 %
9	Final load event	70y	0.5	20.0	50.00	2	0.000	0 Self weight - excluding slabs	100.00 %	100.00 %
								1 Slab self weight	100.00 %	100.00 %
								2 Dead	100.00 %	100.00 %
								3 Services	100.00 %	100.00 %
								5 External Glazing	100.00 %	100.00 %
								6 Internal partitions	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %
								7 Roof Imposed	100.00 %	100.00 %

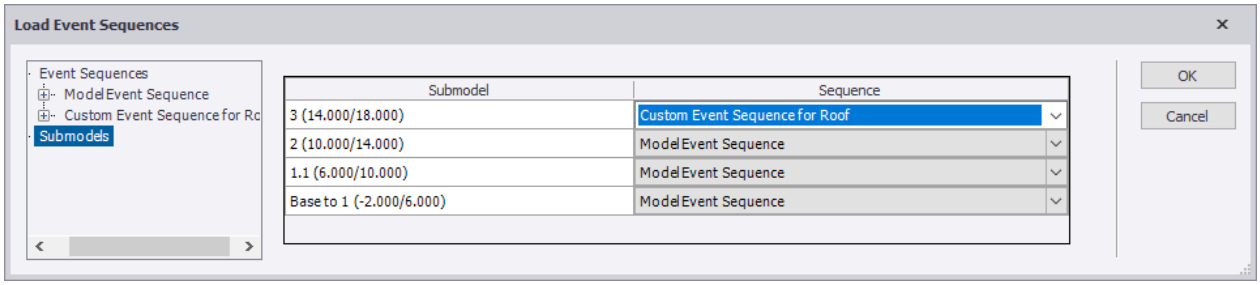
It should be noted that it is not possible to Add, Insert, Remove or change the order of Custom event sequences. It is also not possible to alter any item that is greyed out within the table. This includes events and loadcases. You can, however, edit the Load start time, Number of Exposed faces, Construction loads and the % of load to apply in the combinations (and if working to ACI: Ultimate Creep and Aging Coefficients; or if working to Eurocodes: Beta, Temperature, Relative Humidity). This could mean that careful consideration of the slab model event sequence is necessary to ensure any necessary events are included.

Assigning a custom event sequence to a submodel

When first added, a Custom Event Sequence is identical to the Model Event Sequence. It therefore needs to be edited to achieve the required slab loading sequence.

You can assign different event sequences to different submodels (slabs) using the Submodels page of the Load Event Sequence.

In the screenshot below, the slab model event sequence is assigned to all slabs, except the roof, which has its own custom event sequence defined.



Hence, when you run a rigorous slab deflection estimate on a selected slab, it runs the Load Event Sequence specified on the Submodel page above.

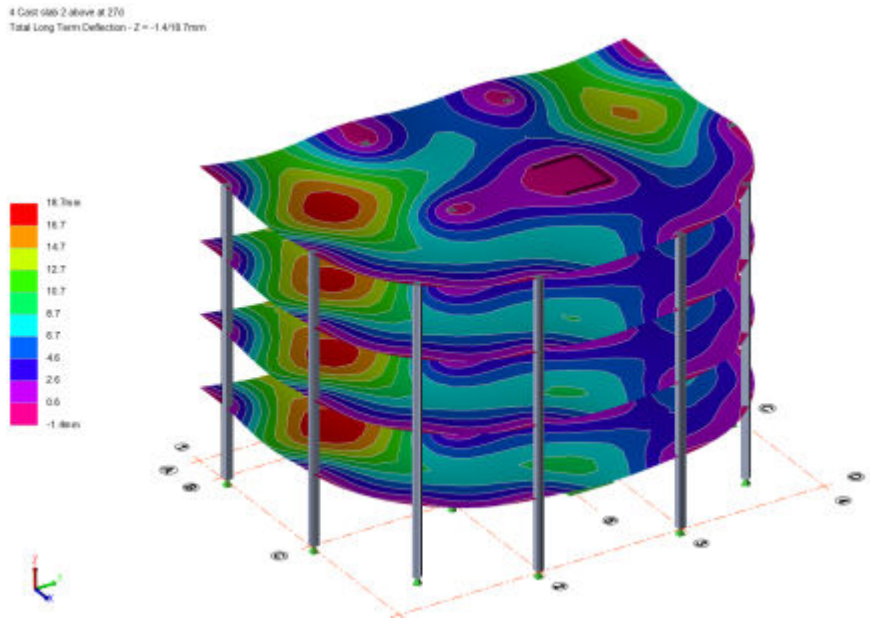
Understanding event sequence deflections

The model described in A typical model event sequence (page 725) consists 3 identical levels and then a roof level with different loading applied. To cater for this a Custom Event Sequence has been applied to the roof submodel.

When you run **Slab Deflection > Analyse All** for this model, 4 sub-models are dealt with one after the other:

- Each one considers 9 events
- Each one does an iterative cracked section analysis at every one of these events
  - assume say, average of 20 iterations in each cracked section analysis
  - That's 180 analyses for each sub-model - over 700 analyses for the whole structure.
- There is an additional instantaneous analysis for each event.

After analysis the deflections can be reviewed for individual events. The view below shows the deflections for Event 4 “Cast slab 2 above”.



Event 4 ends when the slab is 27 days old. The view above is the deflection estimate when each of the slabs are this age. These things don't happen at the same time but it's a very convenient way to display things.

Initially, this has the potential to be confusing, however it should be easy to understand provided you remember that:

- Event Sequences are “Slab Event Sequences”. They describe the events a slab goes through where Day 0 = the day the slab is cast.
- Event Sequences are NOT structure event sequences. They do not describe the all the events starting on day 1 of the overall construction.

Slab deflection analysis sequence

The same basic process is followed irrespective of whether the current level (sub-model), a selected level, or all slabs in the model are analyzed.

In simple terms, events are considered in sequence.

For each event:

- An iterative cracked section analysis including long term effects determines the deflection at the end of the event.
- An additional analysis using the determined state of cracking along with short term cracked properties is undertaken to calculate the total instantaneous deflection associated with the event.
- The state of cracking is carried forward to the next event as the starting point.

Having run a Slab Deflection Analysis the following analysis results are then available to review for either the chosen level (sub-model) or the entire structure dependent upon your chosen analysis

- Deflections (page 738) - Three deflection types are available: Total, Differential, and Instantaneous.
- Extent of Cracking (page 742) - You can also review contours to display the extent of cracking at any load event.
- Relative Stiffness (page 743) - You can also review the relative stiffness in a particular result direction for any specified event.
- Effective Reinforcement (page 745) - You can also review the area of effective reinforcement for a chosen result direction for each FE element.

Total, differential, and instantaneous deflection types

Once the slab deflection analysis has been run, three deflection types are available for review:

- **Total** deflection at the end of any event.
- **Differential** deflection between any two events (Start of Event and End of Event).
- **Instantaneous** deflection (not actually needed for TR 58).
    - This is the deflection when the entire event loading is applied to a version of the model using the established extent of cracking along with short term analysis properties.
    - US codes require an assessment of the instantaneous deflection associated with the imposed load only. This is achieved by adding extra events at the same time as the final event where only the required imposed load is applied.

Understanding differential deflections

When looking at differential deflections it is important to appreciate that an event is not a single point in time, it is a time period which has different deflections at the start and the end - so it is logical that there can be a differential deflection for a single event.

To clarify the selections requested in the ribbon:

- **Start Event** defines the **start** of the start event
- **Event** defines the **end** of the event

Therefore, if you specify the same event as the start and the end event, you will still see a differential deflection.

To clarify further, consider the scenario below:

- Event 1 - Construction loading

- Event 2 - Application of Sensitive Finishes
- Event 3 - Application of Additional Finishes
- Event 4 - Occupation (all loading with live load set at long term factor)
- Event 5 - Final loading event (as above with live load set at 100%)

To determine the maximum differential deflection relevant to the application of sensitive finishes you would define this as the deflection between Events 2 and 5 (Start of Event 2 to End of Event 5). If you opted to pick Events 1 and 5 you would get an un-conservative deflection reported since the effect of construction loading is also considered.

Slab deflection calculations in depth

Interrogating slab deflection calculations

Tekla Structural Designer's Slab Deflection Analysis is not a "black box" - the calculated results are all exposed for interrogation as required.

Several different Results views are available: Deflection, Extent of Cracking, Relative Stiffness and Effective Reinforcement. There is also a Composite modulus report available for each slab.

To help pull all the results together, the items noted above are tied together in the following way:

1. Every element is part of a slab item. Each slab item can have different effective concrete properties.

   The best way to understand this is to look at the summary of information from the Composite Modulus report for each event and consider things like:

   • Adjusted event times due to temperature and cement class

   • Adjusted creep properties due to the number of exposed faces

   • Incremental loading factors

   • The Composite Modulus Calculation
2. Every shell element can have different Effective Reinforcement:

   • This is determined automatically

   • The Effective Reinforcement results view allows you to confirm the values used.
3. Cracking and Stiffness Calculations for each event:

• This is dependent upon the effective concrete properties, effective reinforcement, and the forces that develop

• So this calculation is unique for each direction of each shell for each event.

• The stiffness of cracked sections is dependent on the degree of cracking - so the procedure is iterative (force -> stiffness -> new force, etc)

• At the converged conclusion of this you can see (and check):

- the extent of cracking via the Extent of Cracking results view

- the stiffnesses determined via the Relative Stiffness results view

Composite creep

Design codes typically provide a way of calculating an effective creep modulus at time, t for a constant load/stress applied at time t_0. Typically this is presented as $\phi(t, t_0)$.

An effective Young's Modulus is then calculated as $E_{c,\,eff}\,(t, t_0) = E_{c,28} / [\,1 + \phi(t, t_0)\,]$

However, codes typically do not give guidance on how to deal with a loading history where loads vary over the time period being considered.

Technical Report 58 introduces guidance on this topic and proposes a method by which the loading history can be taken into account. (Reference Section 8.4.1, equation 8.37 and also the example on page 36):

$$\left(\frac{\Sigma w}{E_{comp}}\right)n = \frac{w_1}{E_{eff,1}} + \frac{w_2}{E_{eff,2}} + \ldots\ldots + \frac{w_i}{E_{eff,i}} + \ldots\ldots + \frac{w_n}{E_{eff,n}}$$

Where:

n = event under consideration

w_i = incremental load in event i (= load in event i - load in event (i-1)) (note that this will be a negative value when load is removed)

$E_{eff,i} = E_{c,\,eff}\,(t_{end,n}, t_i)$ (i.e. covers period from start of event i to end of event n)

The above is logical when you consider a single member subjected to a constant loading arrangement that is increased or decreased at each event. However, when you consider an entire slab with many panels receiving different loading increments in different events it does not seem reasonable to consider all the panels together. Two examples of this are:

1. Why should the addition of cladding loads affect internal panels to the same extent as edge panels?
2. In a transfer slab why should panels that don't support columns be affected to the same extent as those which do?

The aim of TR58 is clear, in that loading on a span/panel is taken as an indication of stress. What this fails to consider is that loading on another span/panel can also induce stress (although in most situations this will be a secondary effect). It is also clear that you do not actually need "loads", you just need "relative loads" or some other measure of the relative work done in each event. With this in mind a more general approach has been developed where the relative "work done" by each panel is determined by considering the strain energy in each event:

1. Calculate total strain energy 'Q_0' for each Slab Item for a unit loadcase

   • 'Q_0' = sum of 2D Element strain energies

2. For each Load Event 'i'

   • Calculate incremental strain energy 'Q_i' 'Q_i' = sum of incremental 2D Element strain energies

   • Calculate equivalent incremental load factor 'λ_i

   ' $\lambda_i = Q_i / Q_0$

3. E_{comp} can be established from the equation below where "incremental work done" replaces "incremental load" in the TR58 equation.

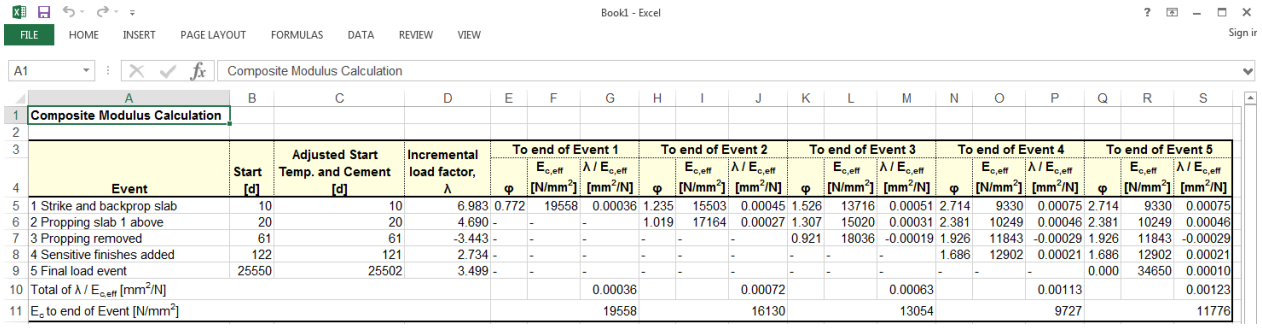
$$\left(\frac{\Sigma\lambda}{E_{comp}}\right)n = \frac{\lambda_1}{E_{eff,1}} + \frac{\lambda_2}{E_{eff,2}} + \ldots\ldots + \frac{\lambda_i}{E_{eff,i}} + \ldots\ldots + \frac{\lambda_n}{E_{eff,n}}$$

There is an array of intermediate values which lie behind the calculation of the composite modulus, E_{comp} for each slab item, for each event. The composite modulus calculation is provided as an excel spreadsheet report.

You can either generate a report for a chosen slab, selected slabs or all slabs, dependant upon your selection method.

- To obtain slab modulus reports for all slab items, right click anywhere in a scene view and choose Export Eff. Modulus report to Excel > For all slab items
- To obtain slab modulus reports for selected slab items, select the slabs in the structure view regime, right click anywhere in the scene view and choose Export Eff. Modulus report to Excel > For selected slab items
- To obtain slab modulus reports for a chosen slab items, right click a slab panel in the structure view regime and choose Export Eff. Modulus report to Excel > For current slab items

A typical composite modulus report for a slab panel is shown below.



Composite Modulus Calculation

Event	Start [d]	Adjusted Start Temp. and Cement [d]	Incremental load factor, λ	To end of Event 1 φ	To end of Event 1 $E_{c,eff}$ [N/mm²]	To end of Event 1 λ / $E_{c,eff}$ [mm²/N]	To end of Event 2 φ	To end of Event 2 $E_{c,eff}$ [N/mm²]	To end of Event 2 λ / $E_{c,eff}$ [mm²/N]	To end of Event 3 φ	To end of Event 3 $E_{c,eff}$ [N/mm²]	To end of Event 3 λ / $E_{c,eff}$ [mm²/N]	To end of Event 4 φ	To end of Event 4 $E_{c,eff}$ [N/mm²]	To end of Event 4 λ / $E_{c,eff}$ [mm²/N]	To end of Event 5 φ	To end of Event 5 $E_{c,eff}$ [N/mm²]	To end of Event 5 λ / $E_{c,eff}$ [mm²/N]
1 Strike and backprop slab	10	10	6.983	0.772	19558	0.00036	1.235	15503	0.00045	1.526	13716	0.00051	2.714	9330	0.00075	2.714	9330	0.00075
2 Propping slab 1 above	20	20	4.690	-	-	-	1.019	17164	0.00027	1.307	15020	0.00031	2.381	10249	0.00046	2.381	10249	0.00046
3 Propping removed	61	61	-3.443	-	-	-	-	-	-	0.921	18036	-0.00019	1.926	11843	-0.00029	1.926	11843	-0.00029
4 Sensitive finishes added	122	121	2.734	-	-	-	-	-	-	-	-	-	1.686	12902	0.00021	1.686	12902	0.00021
5 Final load event	25550	25502	3.499	-	-	-	-	-	-	-	-	-	-	-	-	0.000	34650	0.00010
Total of λ / $E_{c,eff}$ [mm²/N]						0.00036			0.00072			0.00063			0.00113			0.00123
E_c to end of Event [N/mm²]						19558			16130			13054			9727			11776

It should be noted that Tekla Structural Designer takes into account the cement class when determining the temperature adjusted age of loading so minor variations will occur.

The effective modulus is used in determining the properties for each load event.

Extent of cracking

The effective modulus, and the effective reinforcement are used to determine the cracked or uncracked state of each shell for each event, for each direction.

Eurocode 2 provides an expression that predicts the behavior between the cracked and uncracked states. This expression uses a Distribution factor, ζ that apportions the behavior between a fully-cracked state (±1.0) and an uncracked state (0) for interpolating the stiffness when a state of partial cracking exists.

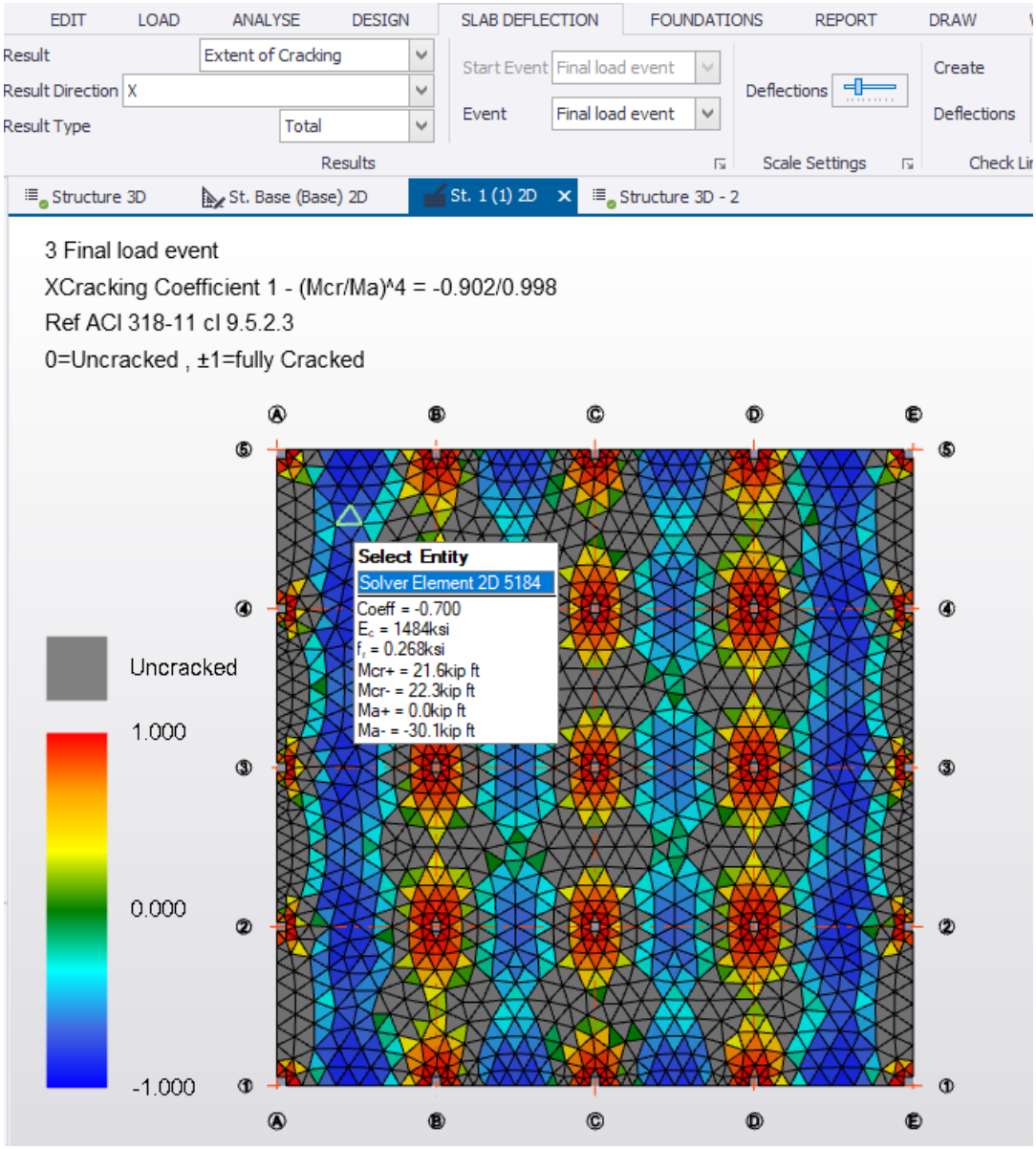
$\zeta = 1 - ß\,(M_{cr} / M)^2$

Where:

- ß is a user defined value specified in the Event Sequences and is either: 1.0 for single short-term loading 0.5 for sustained loads or many cycles of repeated loading
- M_{cr} is the hogging (positive) or sagging (negative) cracking moment.
- $M_{a(+|-)}$ is the relevant Wood-Armer moment in the direction for which the display is shown (X or Y). This is calculated from M_x, M_y & M_{xy} in the usual way, when determining the extent of cracking for a shell element for each iteration for each Event.

If you view Extent of Cracking results for a chosen result direction and cycle through the events you will see each of the FE elements shaded to indicate the extent of cracking.

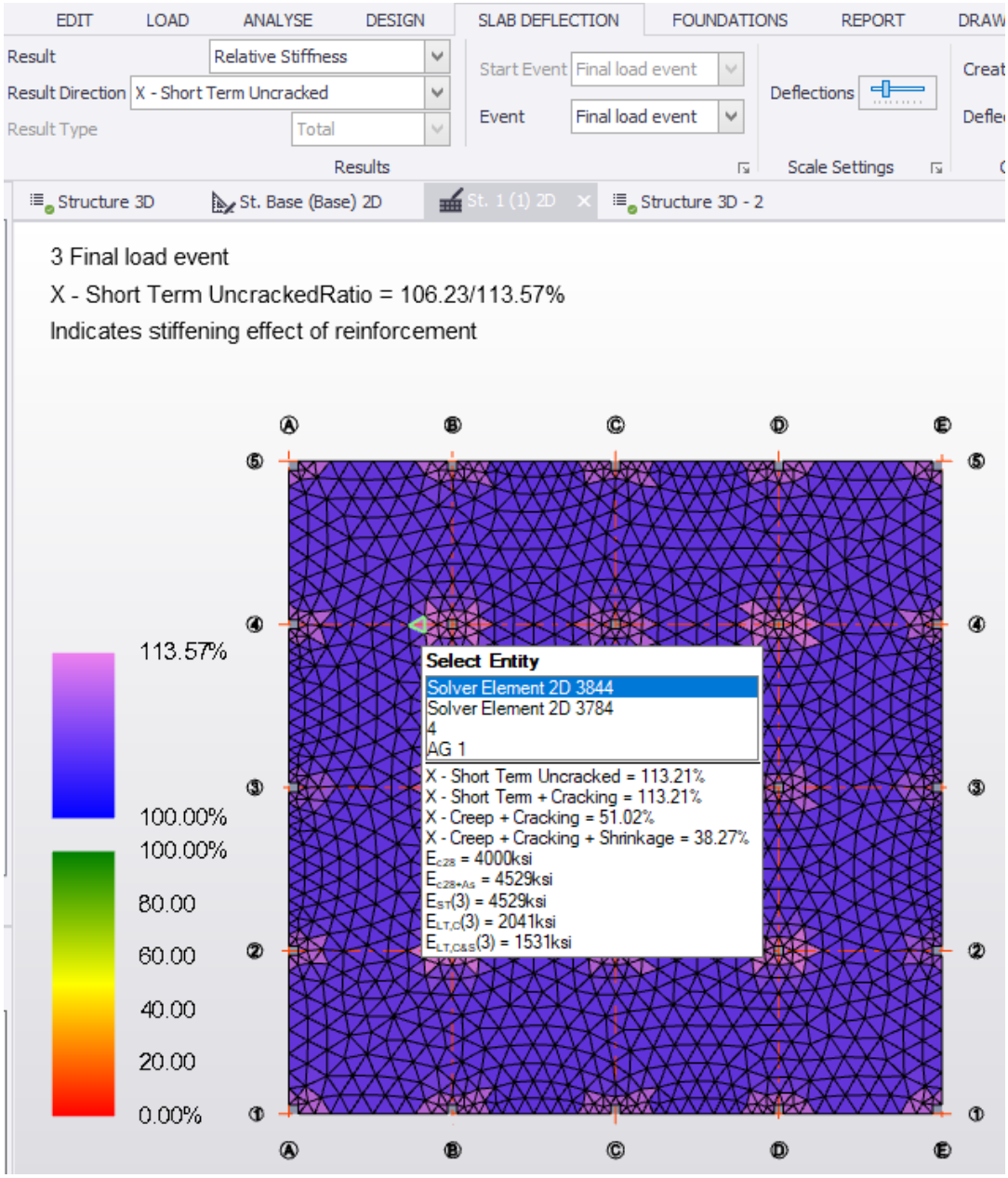
At the first sign of cracking, $M_{cr} > M_a$. If you hover over an FE element the tooltip provides some intermediate calculation results to verify this distribution factor.



The tooltip shows the applied moment for the Event in question, rather than the worst moment of all Events up to and including the one being looked at - this allows engineers to deduce that a greater level of cracking was caused by an earlier Event.

Relative stiffness

The tooltip provides detailed information on the relative stiffness calculations.



The tooltip lists:

- E_{c28} : E_{db}*1.05*stiffness adjustment factor. (Where E_{db} = the short term modulus from the concrete materials database). The stiffness adjustment factor used is determined from the Slab Deflection ribbon > Settings >, Modification Factors page. Provided for information - not directly used in any analysis.
- Ec_{28+As} : the short term modulus including for reinforcement. Provided for information - not directly used in any analysis.

- $E_{ST}(i)$: the short term modulus used in the instantaneous analysis (i.e. includes area of reinforcement and cracking if cracking has occurred) for the selected event.
- $E_{LT,C}(i)$: the modulus used in the final iteration of long term deflection estimation (i.e. includes area of reinforcement and cracking if cracking has occurred, and effective creep) for the selected event.
- $E_{LT,C\&S}(i)$: $E_{LT,C}(i)$ with further adjustment to allow for effect of shrinkage (= $E_{LT,C}$ / multiplier) for the selected event. The shrinkage multiplier to determine the shrinkage contribution is determined for the chosen event based upon the ratio of the maximum panel Z deflection (including shrinkage) / total Z deflection (excluding shrinkage). This provides an indication of the overall effective stiffness adjustment. Provided for information - not directly used in any analysis.

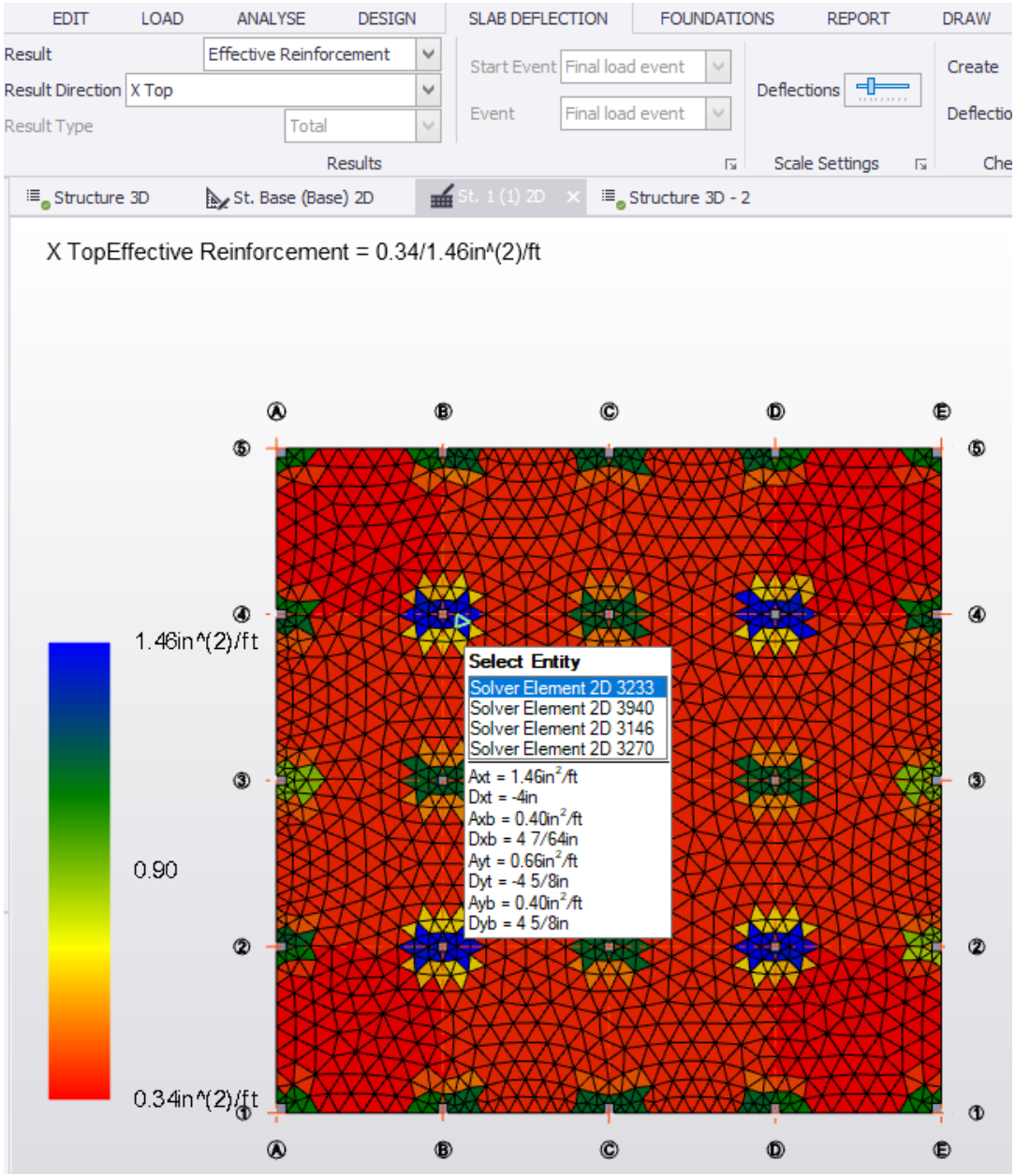
Based on the modulus, E defined above, a number of ratios are provided in the tooltip for the chosen result direction.

- Short Term Uncracked = E_{c28+As} / E_{c28}
- Short Term + Cracking = E_{ST} / E_{c28}
- Creep + Cracking = $E_{LT,C}$ / E_{c28}
- Creep + Cracking + Shrinkage = $E_{LT,C\&S}$ / E_{c28}

Effective reinforcement

Effective reinforcement for each shell element is also required for the determination of the shell's effective properties at the end of each load event.

The effective reinforcement used in the property calculations are reported to you in the 4 layers for every shell element. The information is provided as a color coded shell display from minimum to maximum reinforcement area.



If you hover over a shell the tooltip provides the following information:

- A_{xt} : X Top Effective Reinforcement.
- D_{xt} : Distance from the section centroid to the center of X top reinforcement.
- A_{xb} : X Bottom Effective Reinforcement.
- D_{xb} : Distance from the section centroid to the center of X bottom reinforcement.
- A_{yt} : Y Top Effective Reinforcement.

- D_{yt} : Distance from the section centroid to the center of Y top reinforcement.
- A_{yb} : Y Bottom Effective Reinforcement.
- D_{yb} : Distance from the section centroid to the center of Y bottom reinforcement.

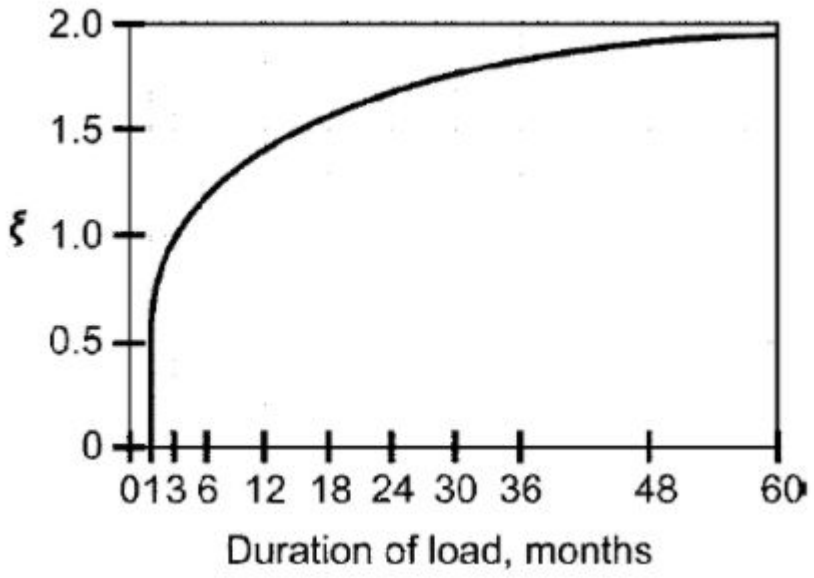
Shrinkage allowance

Shrinkage is the strain in hardened concrete that can occur due to moisture loss.

- Eurocode 2 provides a method to estimate shrinkage strains and curvatures based on exposed surface area, member size, relative humidity and reinforcement quantity and position.
- Asymmetry of reinforcement leads to curvature which leads to deflection. It is estimated this effect can contribute up to 30% to the long-term deflection.
- Technical Report 58 provides a theoretical method of estimating the additional shrinkage deflection effect in the analysis

At this time the TR58 method has not been implemented within Tekla Structural Designer. Shrinkage is taken into consideration using a multiplier, by making an overall adjustment to the total deflection (excluding shrinkage) in line with simpler adjustment proposals of the ACI code. This approach is in line with many other software products.

The simplest ACI approach makes an allowance for all long term effects (creep and shrinkage) by using an adjustment factor. This is based on the graph below and also provides some specific values.



Time	Multiplier for long-term deflections
5 years or more	2.0
12 months	1.4
6 months	1.2
3 months	1.0

Note that we said shrinkage effects and not creep and shrinkage. Creep is dealt with rigorously in Tekla Structural Designer so we need to ascertain the proportional effect of shrinkage only. ACI 435 provides some indication of the separate contribution of creep and shrinkage.

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 $\sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 $\sqrt{f_c'}$ 4 $\sqrt{f_c'}$	1.0 1.0	2.0 1.5	2.0 1.0	5.0 3.5
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

Comparing the different sources the ratio of shrinkage is as follows:

- Sbarounis 1.2 / 5 = 24%
- Branson 1 / 4 = 25%
- Graham and Scanlon 1 / 3.5 = 28%

  (ignore higher modulus of rupture because reduced values are considered automatically in the cracked section analysis).

The above provides a shrinkage ratio of between 24% and 28%. Hence we recommend a value of between 20%-30% is used. A 25% default is provided via the Slab deflection ribbon > Options dialog and the Creep and Shrinkage page.

The total deflection due to shrinkage effect is determined based on an identified “Total Shrinkage Event” towards the end of the event sequence. The event sequence with the latest load start time is used for calculating the shrinkage adjustment. If multiple events exist with the latest load start time then the first one is considered.

A special note about the Final load event - It is perfectly ok to have multiple events at the same final time, you should not separate these events by a small number of days.

Using this “total shrinkage effect” we can then assign a proportion of the total shrinkage to each event.

With reference to earlier versions of ACI 435 (1966) on which the values in the graph above are based, additional values for 1 month and 3 years can be obtained. This follows the case where As’ = 0 (because compression steel is allowed for differently in the code). By comparison with the graph above, we can see closely matched values.

Duration of loading	Factor F		
	As' = 0	As' = 0.5As	As' = As
1 month	0.58	0.42	0.27
3 months	0.95	0.77	0.55
6 months	1.17	0.95	0.69
1 year	1.42	1.08	0.78
3 years	1.78	1.18	0.81
5 years	1.95	1.21	0.82

The values we have adopted for considering shrinkage effects are as tabulated below. The final column provides the proportion of the total shrinkage at a given time.

Time	Long Term Effects Factor	Proportion of Total Shrinkage
0	0	0.00
1 month	0.6	0.30
3 months	1	0.50
6 months	1.2	0.60
1 year	1.4	0.70
3 years	1.8	0.90
5 years and above	2	1.00

From the above, for any event, the end of event time is used to calculate a “Proportion of total shrinkage” using linear interpolation between the values discussed in the table above.

Deflection calculations on the Z deformation are then adjusted to account for shrinkage effects.

As an example, let’s assume the following event sequence.

- Event 1 = 7 days, Event 2 = 10 days, Event 3 = 17 days, Event 4 = 20 days, Event 5 = 27 days, Event 6 = 2 months, Event 7 = 6 months, Event 8 = 1 year and Event 9 = 70 years

Assuming a shrinkage factor of 25% (user defined input value), a basic multiplier can be determined = 1/(1-25%) = 1.333

Event 9 Final event at 70 years analysis deflection (excluding shrinkage) = 32.4mm

Therefore, the Total deflection (including shrinkage) = 32.4 x 1.333 = 43.2 mm

Total Deflection from shrinkage alone is 43.2 - 32.4 = 10.8 mm

We can now apportion this deflection due to shrinkage, to each event based upon the event time and a proportion value.

i.e.

At 0 days proportion of total shrinkage is 0, At 1 month proportion is 0.3. Therefore using linear interpolation between these values;

- Event 1 (7 days) Shrinkage multiplier = 0.3 * 7/30 = 0.07
- Event 2 (10 days) Shrinkage multiplier = 0.3 * 10/30 = 0.1
- Event 3 (17 days) Shrinkage multiplier = 0.3 * 17/30 = 0.17
- Event 4 (20 days) Shrinkage multiplier = 0.3 * 20/30 = 0.2
- Event 5 (27 days) Shrinkage multiplier = 0.3 * 27/30 = 0.27

At 1 month proportion of total shrinkage is 0.3, At 3 month proportion is 0.5. Therefore using linear interpolation

- Event 6 (2 months) Shrinkage multiplier = 0.4

At 6 month proportion of total shrinkage is 0.6

- Event 7 (6 months) Shrinkage Multiplier = 0.6

At 1 year proportion of total shrinkage is 0.7

- Event 8 (1 year) Shrinkage Multiplier = 0.7

At 70 years proportion of total shrinkage is 1.0

- Event 9 (70 years) Shrinkage Multiplier = 1.0

The Shrinkage deflection that occurs at each event is then the total shrinkage 10.8 mm x the shrinkage multiplier calculated above.

- Event 1 (7 days) Shrinkage = 0.07 x 10.8 = 0.76 mm
- Event 2 (10 days) Shrinkage = 0.1 x 10.8 = 1.08 mm
- Event 3 (17 days) Shrinkage = 0.17 x 10.8 = 1.84 mm
- Event 4 (20 days) Shrinkage = 0.2 x 10.8 = 2.16 mm
- Event 5 (27 days) Shrinkage = 0.27 x 10.8 = 2.92 mm
- Event 6 (2 months) Shrinkage = 0.4 x 10.8 = 4.32 mm
- Event 7 (6 months) Shrinkage = 0.6 x 10.8 = 6.48 mm
- Event 8 (1 year) Shrinkage = 0.7 x 10.8 = 7.56 mm
- Event 9 (70 years) Shrinkage = 1.0 x 10.8 = 10.8 mm

For each event, the total deflection (including shrinkage) reported in the Slab deflection view regime and the tooltips is the event analysis deflection

(excluding shrinkage) + the proportion calculated above using the shrinkage multiplier.

Check lines

Check lines - a unique feature inTekla Structural Designer - provide a practical way to automate deflection checking and reduce the possibility of errors.

Each check line defines a line along which deflection checks are required. Multiple check lines can be created and each line can have several different deflection checks performed (against either a total, instantaneous, or differential deflection limit).

Whilst the check lines have to initially be positioned using engineering judgment, once they are in place they provide an instantaneous means to evaluate revised deflections following changes to the model parameters and re-analysis.

Assessing slab deflections without check lines would be quite an arduous task, since deflection limits are of the form (span / fixed value) and the spans can vary on an irregular slab, hence the permissible limit would also vary. Furthermore, you may wish to check total or differential deflections at or between different load events. Would the critical location be obvious?

Setting up the checks in advance (via the slab deflection check catalogue)

The deflection checks to be performed for the check lines are set up in the Slab Deflection Check Catalogue before the check lines are positioned.

Each check in the catalogue is specified as a Total, Differential or Instantaneous deflection limit that is checked for a specified event, (or in the case of a Differential check between the start of an event and the end of an event).

Related task Create the deflection checks to be applied to check lines (page 704)

Application of check lines

The application of check lines is an iterative activity. They should be created at all locations where you deem deflection checks to be required.

Check lines can only be created in a 2D view. If the Create command is greyed out ensure you switch to a 2D view.

When you click Create, the Properties Window automatically includes those slab deflection checks from the Slab Deflection Check Catalogue where "Use in new Check Lines" was checked. Note that a maximum of six checks can be assigned to a single check line; if you require more than six checks, then multiple check lines can be applied at the same location.

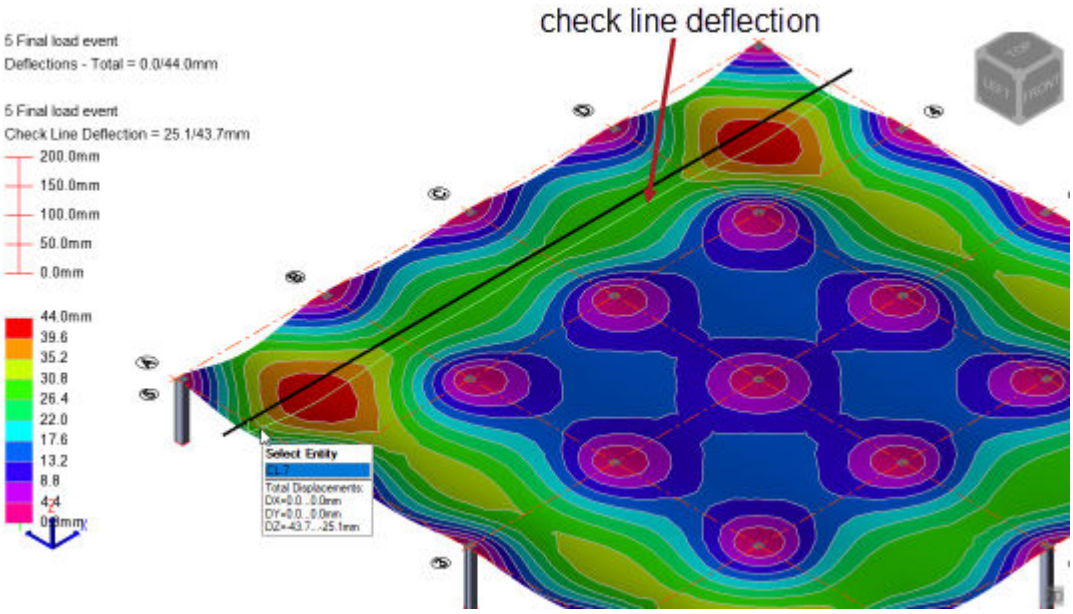
You can add further checks from the catalogue or add new checks directly from the Properties Window.

You can also edit the "Use in New Check Lines" option in the catalogue prior to running the Create command to change the default checks automatically assigned to each new check line.

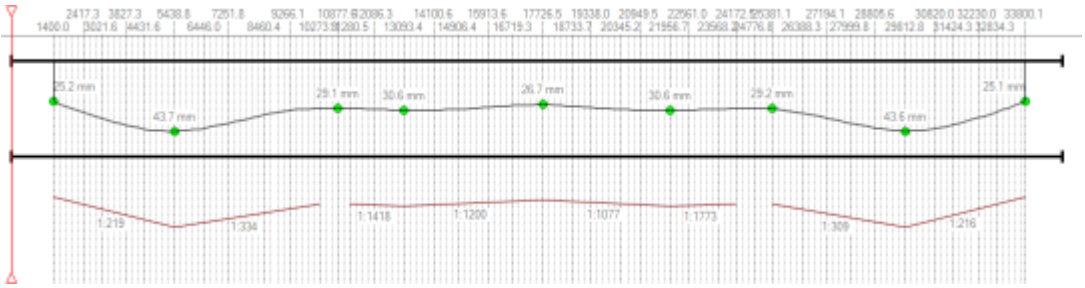
Related task Create a check line (page 705)

Displaying check line results

The Check Line deflection can be overlayed on the slab by using the Deflections command in the ribbon, it is perhaps easier to visualize if you switch the view to a 3D view of the slab using the 2D/3D toggle button in the bottom right of the window.



You can also right click on a check line and open the deflections check view. This displays a cut line through the slab, showing the deflected shape with the maximum and minimum deflection values. Beneath this it also draws the average slopes between maximum and minimum points and reports the average slope ratios.



The above deflection check view is controlled from the Loading Analysis ribbon which has droplists to enable you choose the Result Type (Total, Instantaneous, or Differential) and the Event(s).

The average slope ratios (for the appropriate result type/event) are checked against the deflection limits that have requested for the check line.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

Related task Display check line results (page 709)

Check line reports

A tabulated report is available for each check line which itemizes each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

You can also generate a Slab deflection check line report for the entire structure, selected level, planes or sub structures via the Model Report command on the Report ribbon.

Related task View slab deflection reports (page 712)

Slab deflection status and utilization

The status and the utilization can be graphically displayed for both Check lines and slabs.

Check lines Status and Utilization

Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical utilization ratio

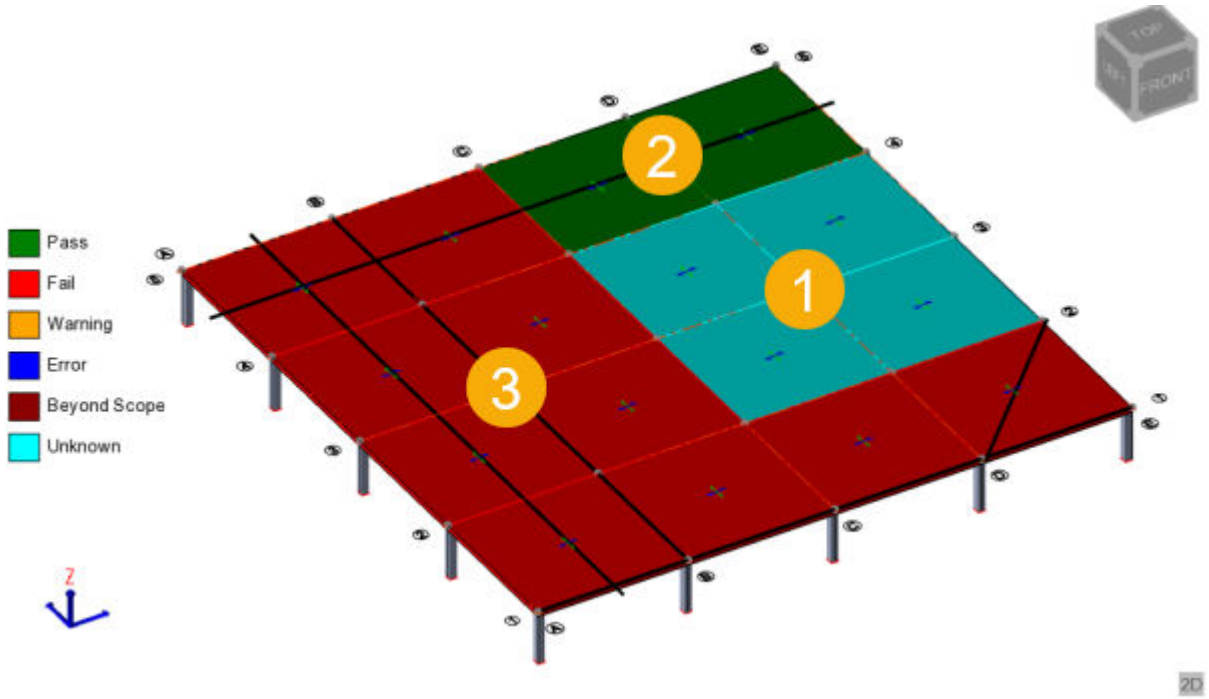
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines

are automatically re-checked so you will obtain instant feedback on status and utilization.

Related task: Display check line status and utilization (page 709)

Slab Deflection Status and Utilization

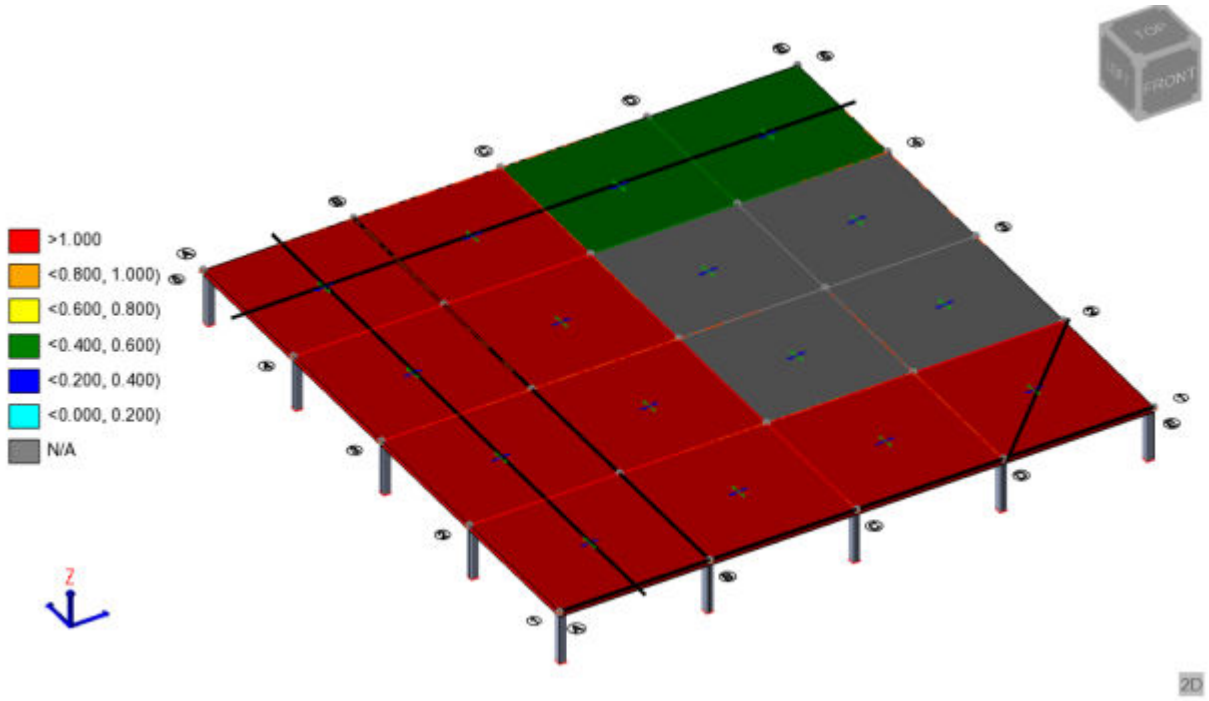
Slab deflection status is determined for each slab item as the worst status from all associated check lines detected within the slab item or touching the slab item boundary.



In the above status view:

1. No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed
2. One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
3. A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

Slab deflection utilization is similarly determined for each slab item as the worst utilization from all associated check lines.

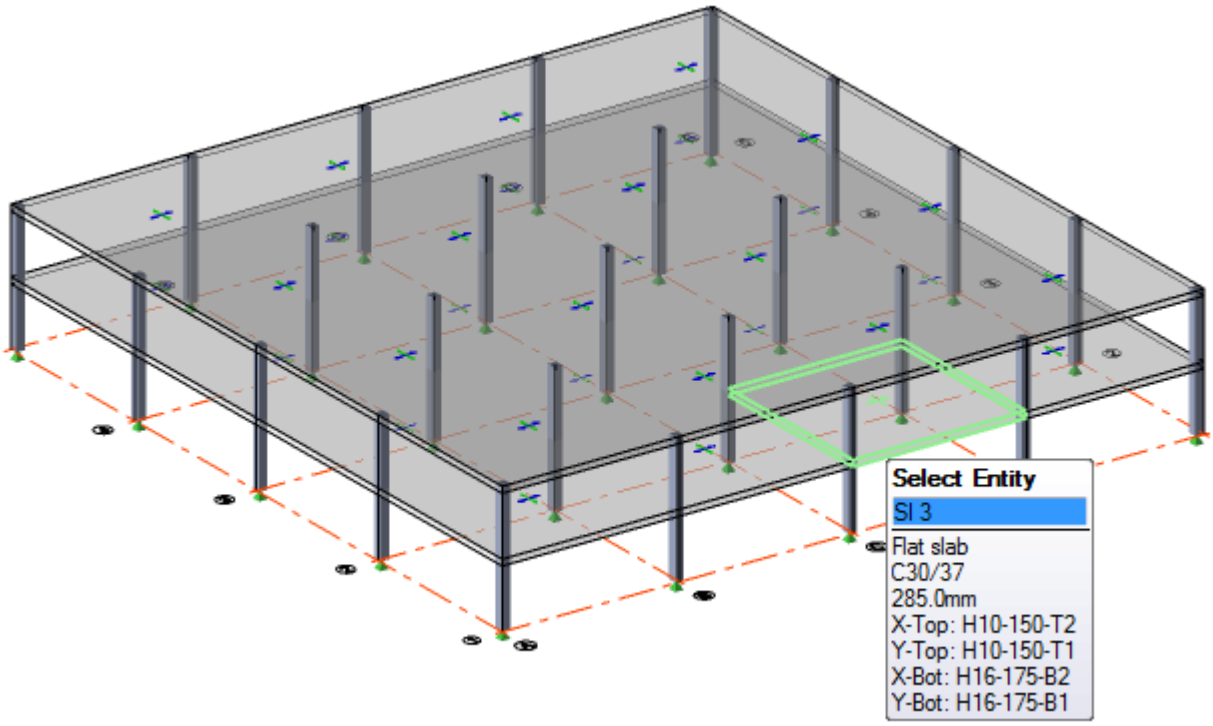


Related task: Display slab deflection status and utilization (page 710)

Slab deflection example (Eurocode)

In the following exercise slab deflections will be checked for the tutorial model shown below.

It is a simple multi-bay flat slab structure on an 8m square grid of columns. The slabs have been sized based on deemed-to-satisfy rules taken from “Economic Concrete Frame Elements to Eurocode 2 - The Concrete Society”.



Geometry:

- 8m grid
- 285 thick C30/37 slab
- 400 square columns

Loading:

- Finishes - 1.5kN/m^2
- Imposed - 5.0kN/m^2
- Perimeter Cladding - 10.0kN/m

Deemed to satisfy checks (page 756) provide one method of checking, however the main focus of the exercise will be to investigate rigorous slab deflection checks (page 759).

Deemed to satisfy slab deflection checks example (Eurocode)

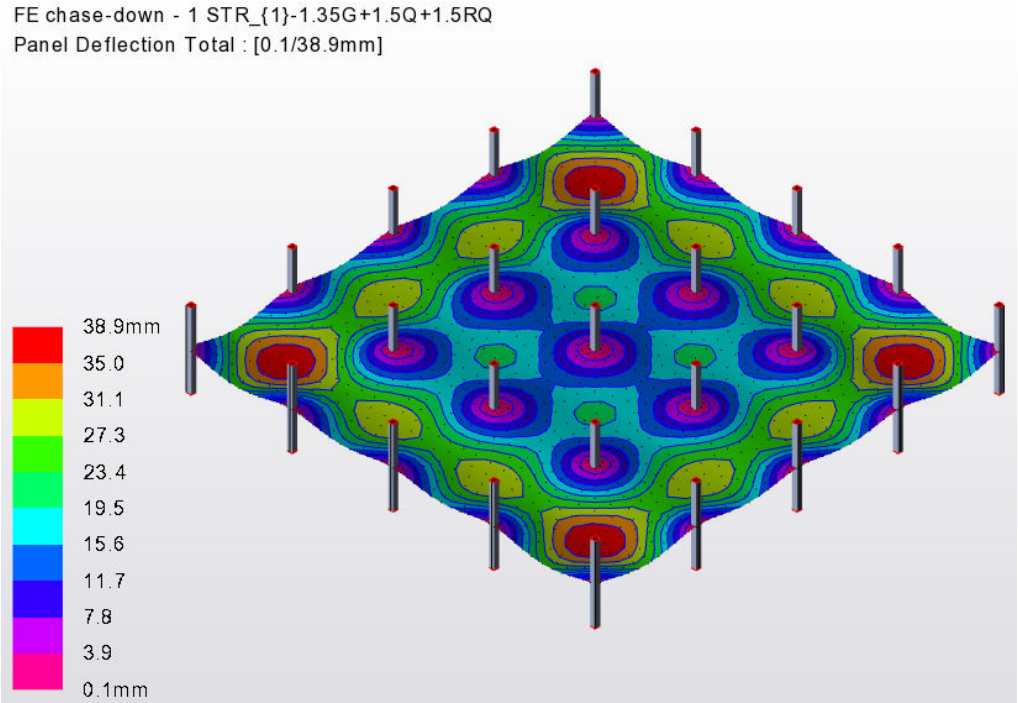
A simple way to assess slab deflection in Tekla Structural Designer is to run a linear analysis using adjusted analysis properties, and then check the resulting deflections by manually determining critical spans.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection EC.tsmd

Perform Linear Analysis

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. Open a view of the **Typical floor** level
3. Switch to the **Results View**
4. From the Results toolbar, review **2D deflections** for the **FE chase-down analysis** for the load combination 1, service load results



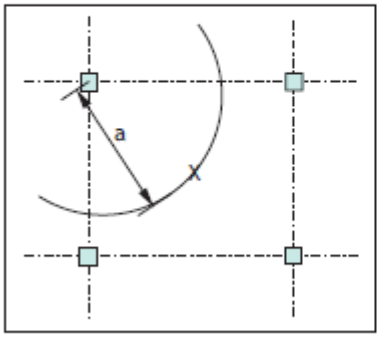
Identify critical check locations

We can see that the maximum reported deflection is 38.9mm, occurring in the middle of a corner bay. This should be assessed by taking the slab span diagonally across the bay.

NOTE In 'real world' flat slabs some engineering judgment might be required when assessing which deflections and span lengths require checking.

Guidance exists in How to Design Concrete Structures using Eurocode 2, The Concrete Centre, Figure 9

Figure 9
Recommended acceptance criteria for flat slabs



Notes

If maximum permitted $\delta = L/n$ and X is the position of maximum δ

where

L – Span

n – Limiting span-to-depth ratio, e.g. 250

then the deflection at X should not be greater than $2a/n$.

(Maximum deflection on gridlines may be more critical.)

In our example, taking the diagonal dimension across the columns, the deemed-to-satisfy span / 250 rule provides a deflection of [$\sqrt{(8000^2+8000^2)}$] / 250 = 45.3mm.

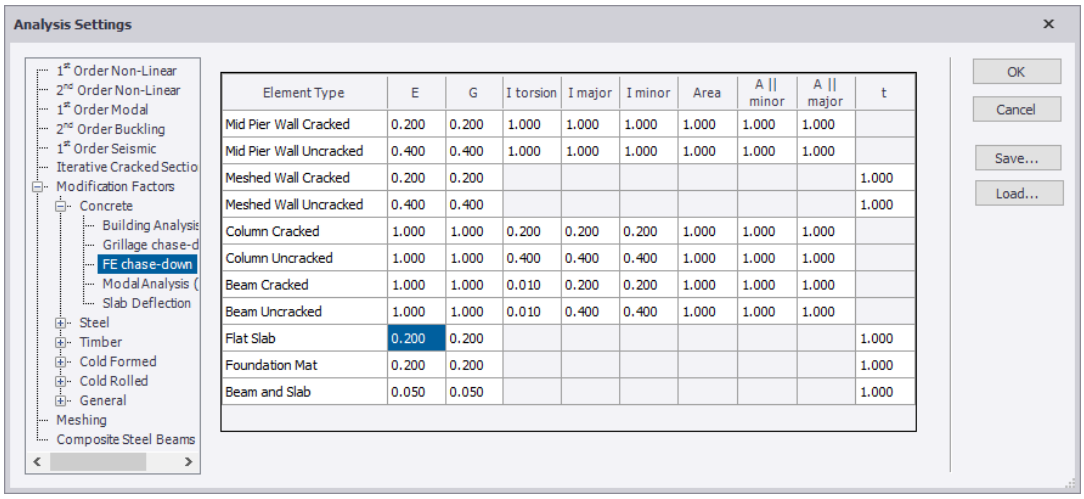
This compares favorably.

NOTE Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 250.

Concrete properties used in the analysis

The Tekla Structural Designer deflection result is completely dependent upon the concrete elastic modulus used in the analysis which is adjusted by a modification factor to consider such things as creep, cracking and shrinkage.

The modification factor is set from the Settings dialog on the **Analyze** ribbon. As shown below, for the FE chase-down analysis of flat slabs this defaults to 0.2.



Rigorous slab deflection analysis example (Eurocode)

There are many input parameters that will have an impact on the rigorous deflection estimates and would therefore need to be considered. For details see: Factors that affect rigorous slab deflection estimates (page 716)

For this exercise, initially it will be assumed that the default settings have already been reviewed and set as required.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection EC.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

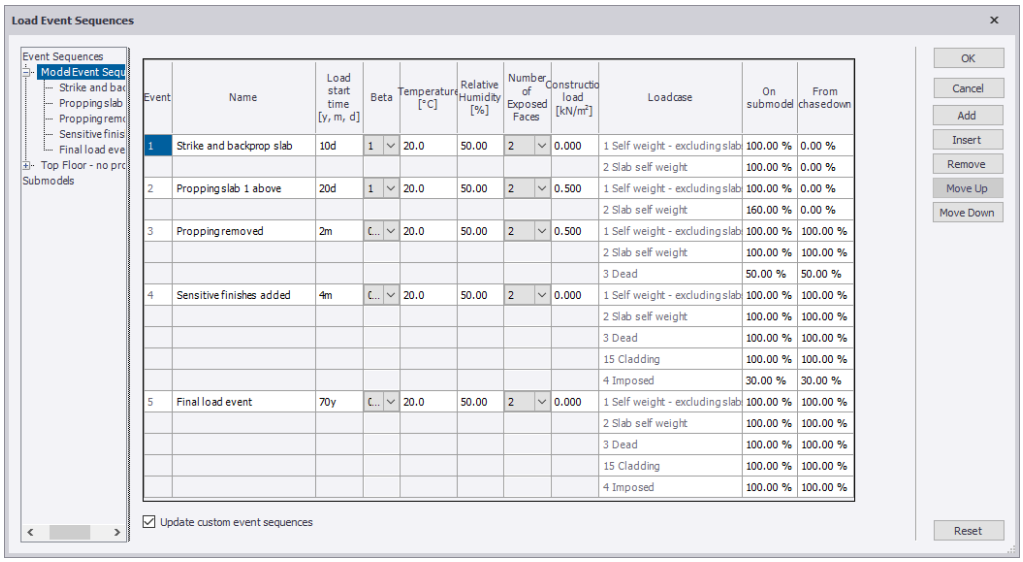
1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Review the Model Event Sequence

To review the model event sequence:

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**

This has already been setup for this example as shown below:



Deflections to the end of each of the above event periods will be calculated by the analysis.

3. Click **OK** to close the dialog.

Perform Iterative Slab Deflection Analysis

To establish some initial results (with all parameters left as default values) you could just click **Analyse All** from the Slab Deflection toolbar, however, in a real model rather than analysing all levels at once, it can be more efficient to work on just the current level, or a selected level. Obviously considering just a single level reduces the time necessary to undertake the iterative slab deflection analysis.

1. Open a 2D plan view of the level **Typical floor**
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.

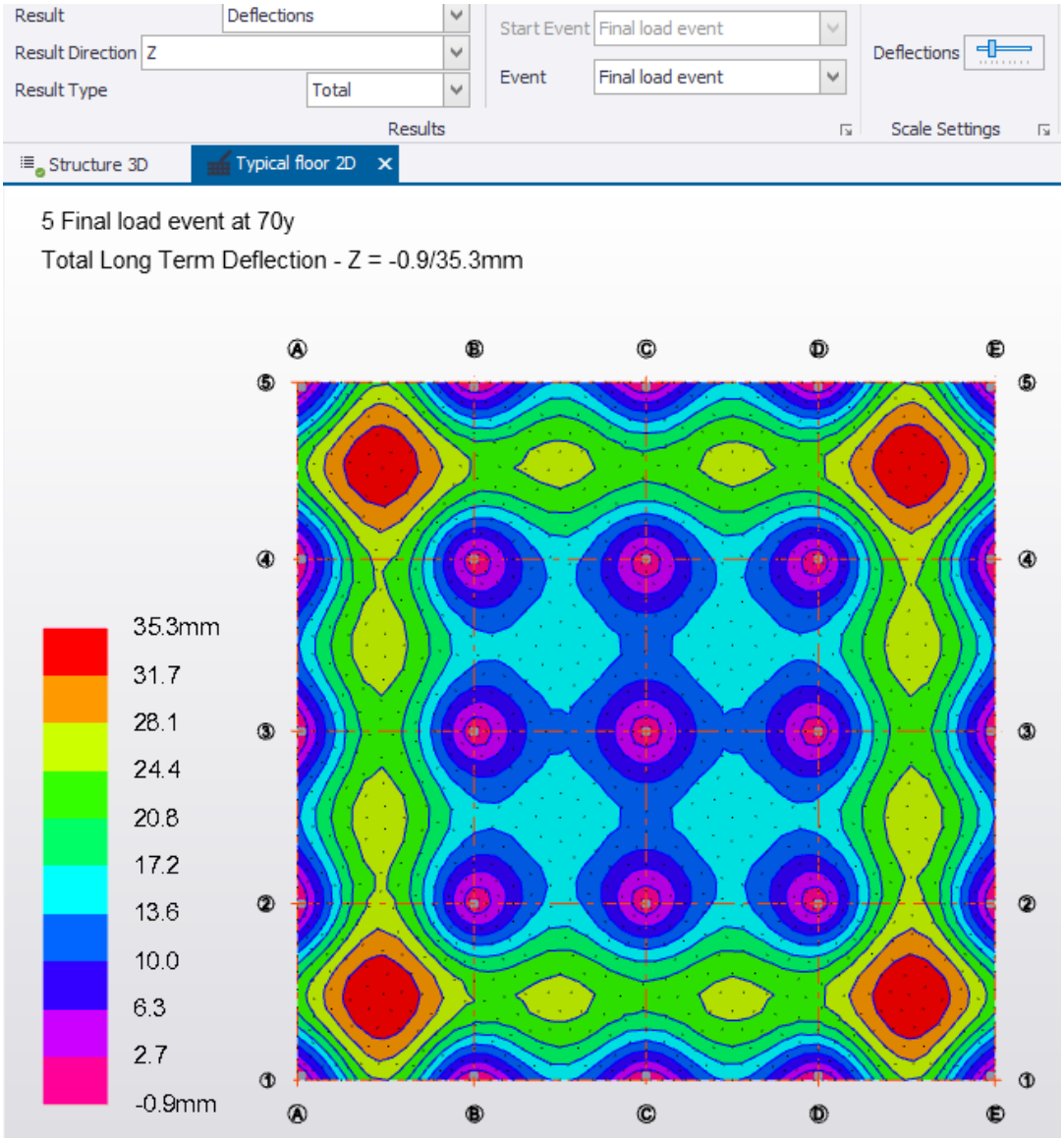
Review Deflections for Events

Deflections can be reviewed for each event by making selections from the Event droplist in the ribbon.

You are able to review:

- Total deflection at the end of any event.
- Differential deflection between any two events (Start of Event and End of Event).
- Instantaneous deflection (not actually needed for TR 58).

The image below shows total deflection contours for the final load event.



NOTE To see the image as shown, you may need to adjust some of the selections in Scene Content, for example **Slab Patches** should be de-selected.

As a comparison with the simple approach earlier (39.7mm), the Total deflection at the final load event for the chosen location is 35.2mm.

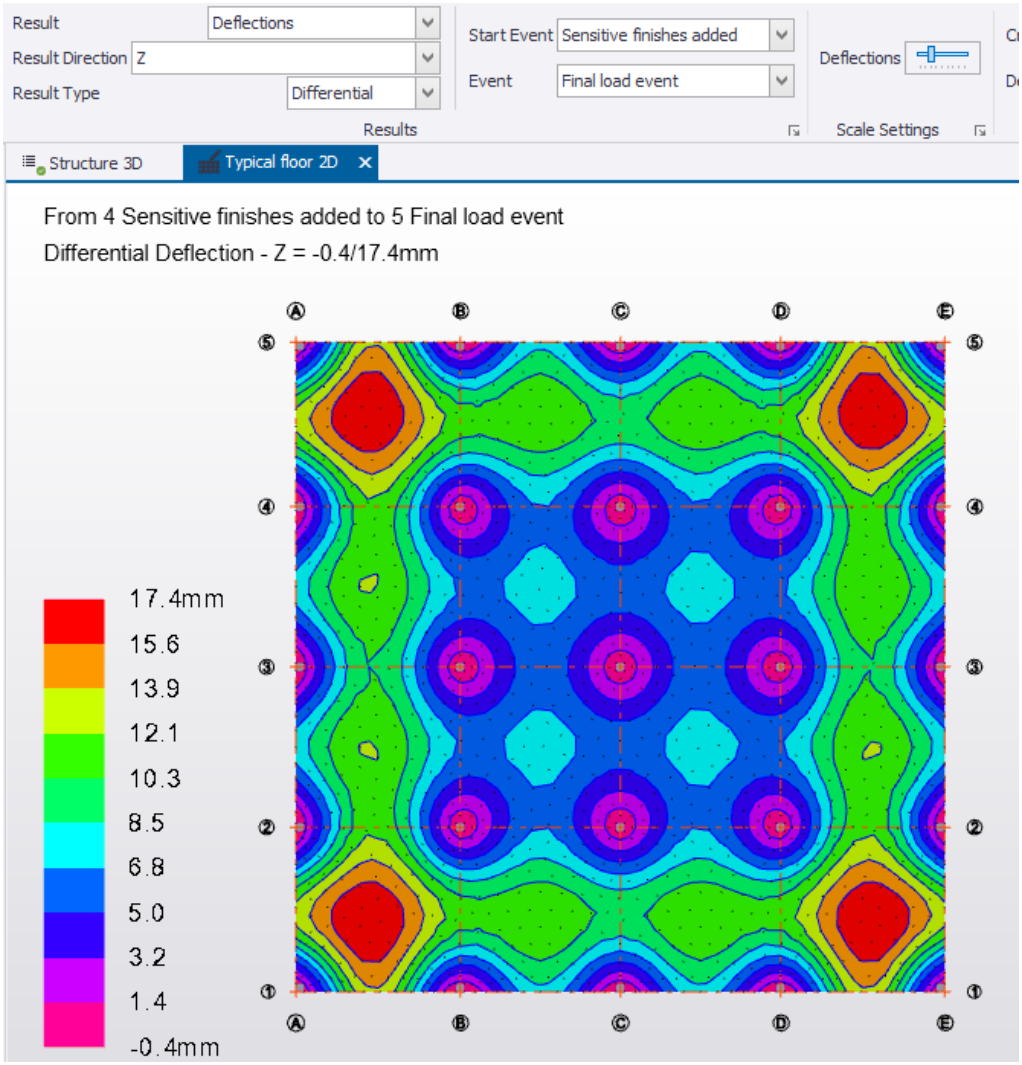
NOTE In the above contour plot the deflections are not exactly symmetrical - this is because reinforcement is in the outer layer in the Y direction making the slab stiffer in that direction.

Total deflections to the end of each of the event periods in the Event Sequence are available and could also be displayed as required.

In the Event Sequence there is an event for “Sensitive Finishes added” - we shall now show differential deflection between this and the final event.

1. From the **Slab Deflection** toolbar, change the Result Type to **Differential**
2. Select the Start Event as **Sensitive Finishes added**
3. Select the Event as **Final load event**

   The 2D view now displays the differential deflections between “Sensitive Finishes added” and the final event.



Review Other Results

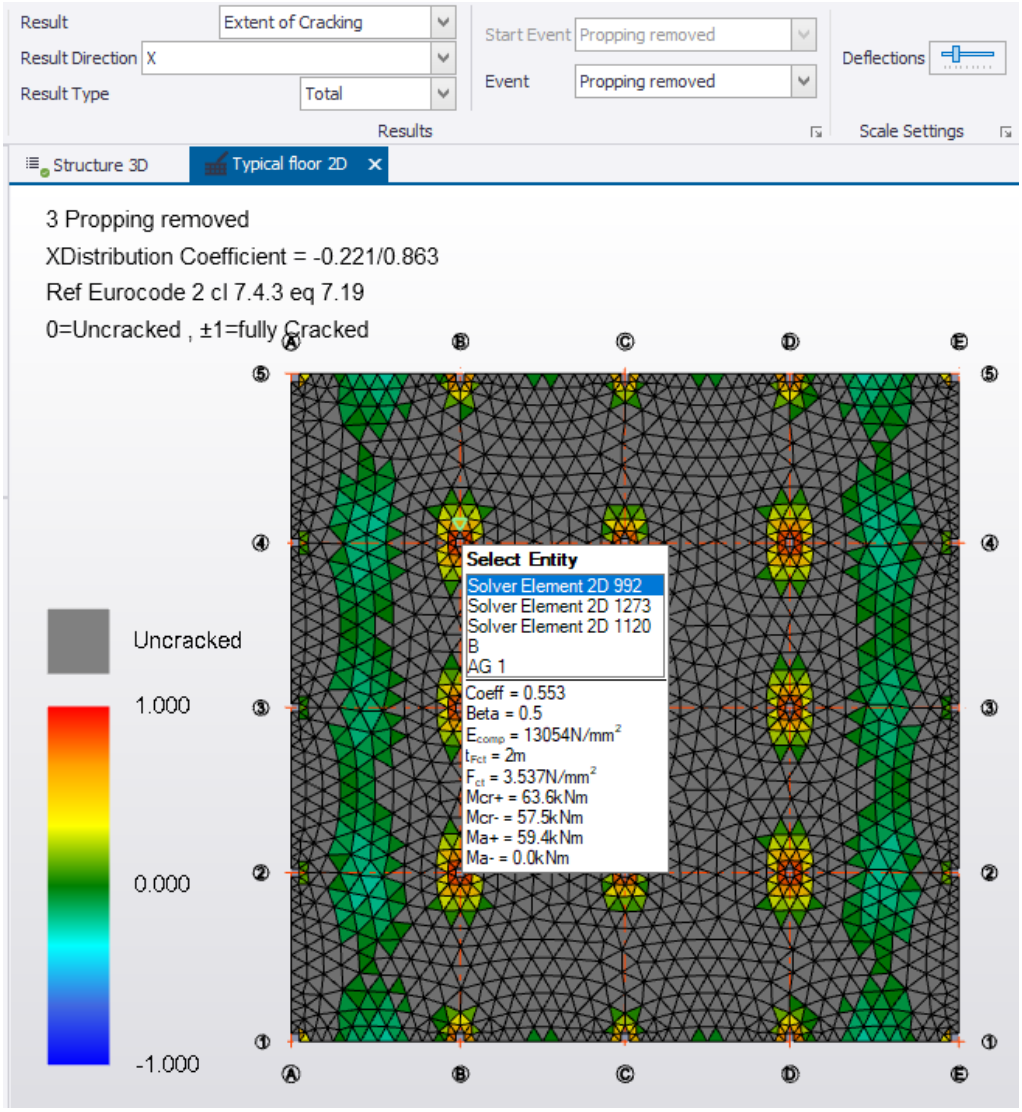
In addition to the deflections you can display the extent of cracking at any load event.

You can also review the relative stiffness in a particular result direction for any specified event.

You can also review the area of effective reinforcement for a chosen result direction for each FE element.

1. Select the Result as **Extent of Cracking**, and then:
   a. Select the Result Direction as **X**,
   b. Select the Result Type as **Total**,
   c. Select the Event as **Propping removed**,

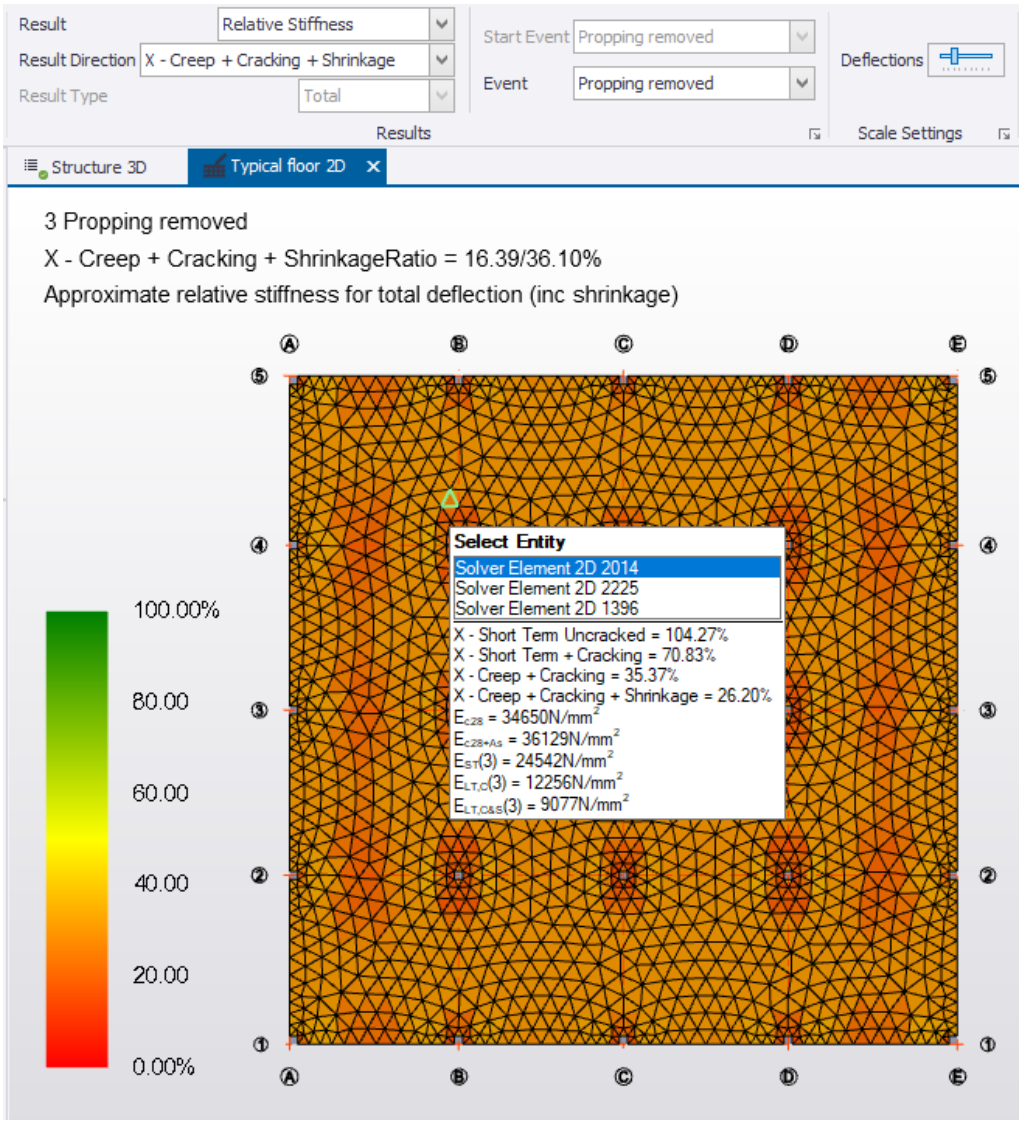
   The extent of cracking for the **Propping removed** load event is displayed.



2. Select the Result as **Relative Stiffness**, and then:

   a. Select the Result Direction as **X - Creep + Cracking + Shrinkage**,

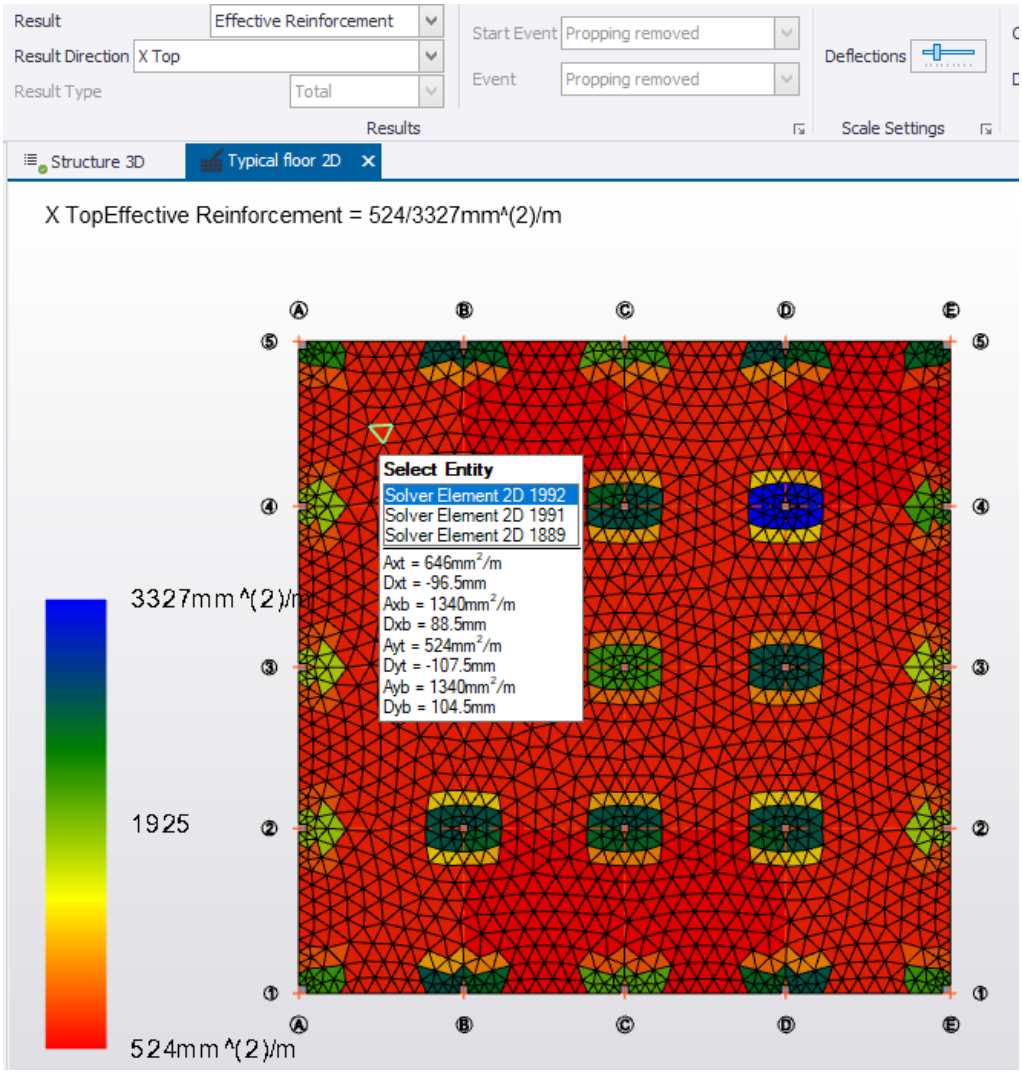
   The relative stiffness for the **Propping removed** load event is displayed.



3. Select the Result as **Effective Reinforcement**, and then:

   a. Select the Result Direction as **X Top**,

The effective reinforcement for the **X Top** direction is displayed.



NOTE When you hover over any FE element in the slab deflection view regime - values are provided within the tooltip.

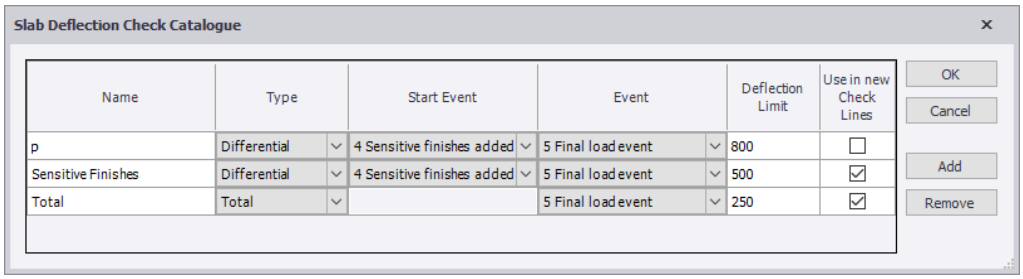
Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgment.

The deflection checks associated with each check line are selected from a predefined Deflection Check Catalogue. This is viewed by clicking Deflection Checks in the ribbon.

You can add new checks to the catalogue as required.

1. From the **Slab Deflection** toolbar, click **Deflection Checks**



Whilst three checks have been defined above, only two of these have been set to be used in new Check Lines:

- **Sensitive finishes** will check the differential deflections from when the sensitive finishes are applied to the final load event against a deflection limit of 1/500
- **Total** will check the total deflections to the final load event against a deflection limit of 1/250

2. Click **OK** to close the dialog.

Place Check Lines

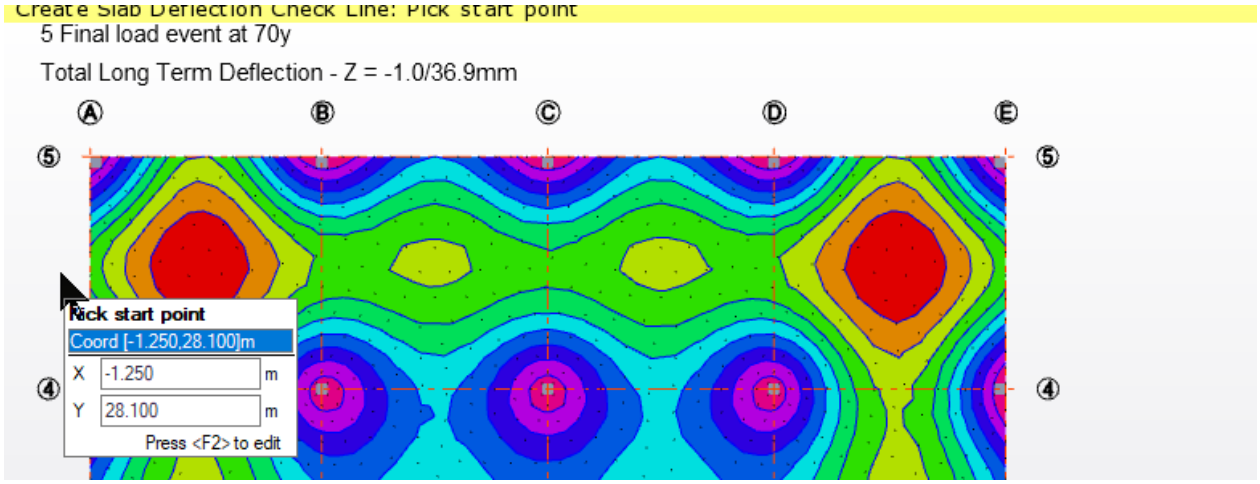
NOTE Check lines can only be created in a 2D view.

We will define several check lines in this example:

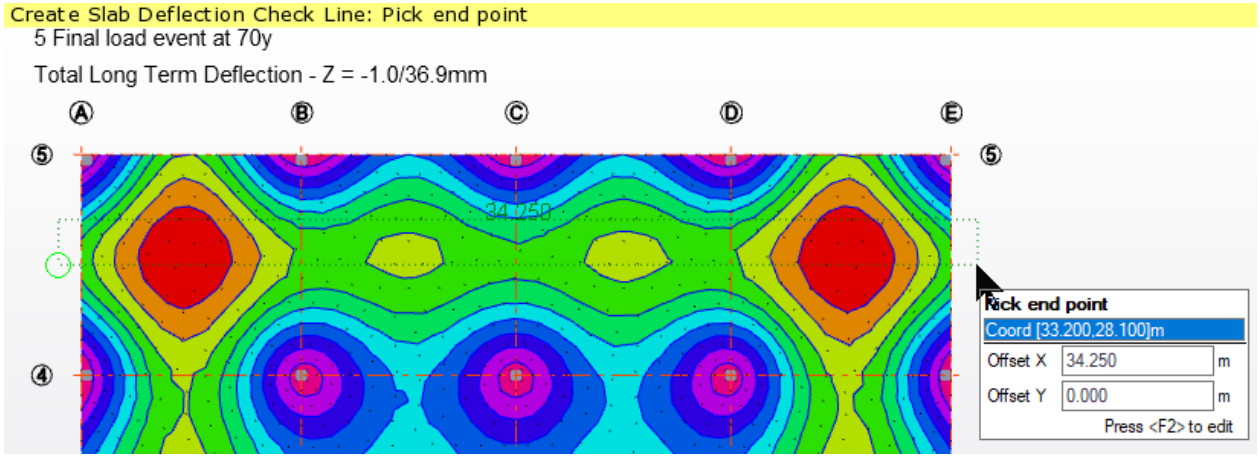
1. Ensure you are in the **Typical floor** 2D plan view and if necessary change:
   a. the Result back to **Deflections**,
   b. the Result Type to **Total**,
   c. the Event to **Final load event**
2. Click **Create**

NOTE When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue for which “Use in new Check Lines” was checked.

3. To place the check line at approximately mid-span between grid lines 4-5 from grid line A to E:
   a. Pick the start point to the left of grid line A as shown:

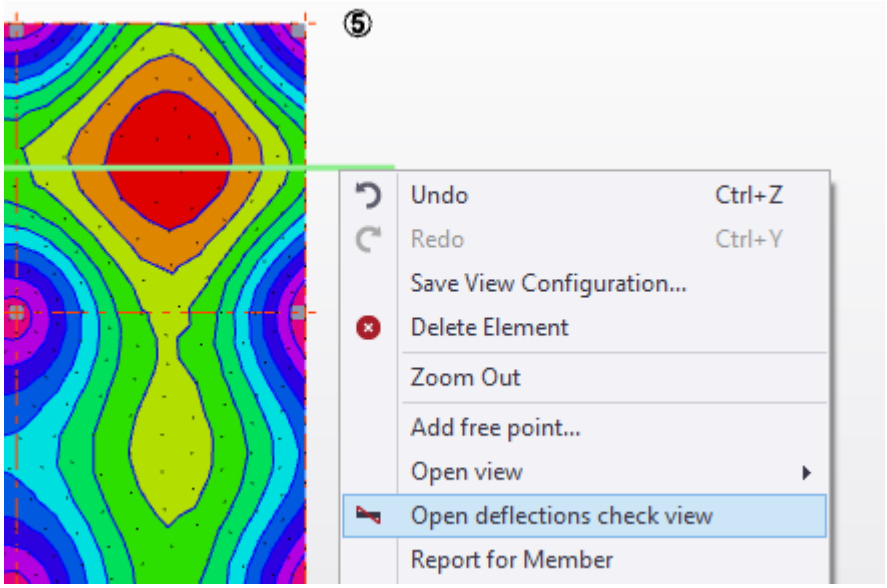


b. Pick the end point to the right of grid line E as shown:

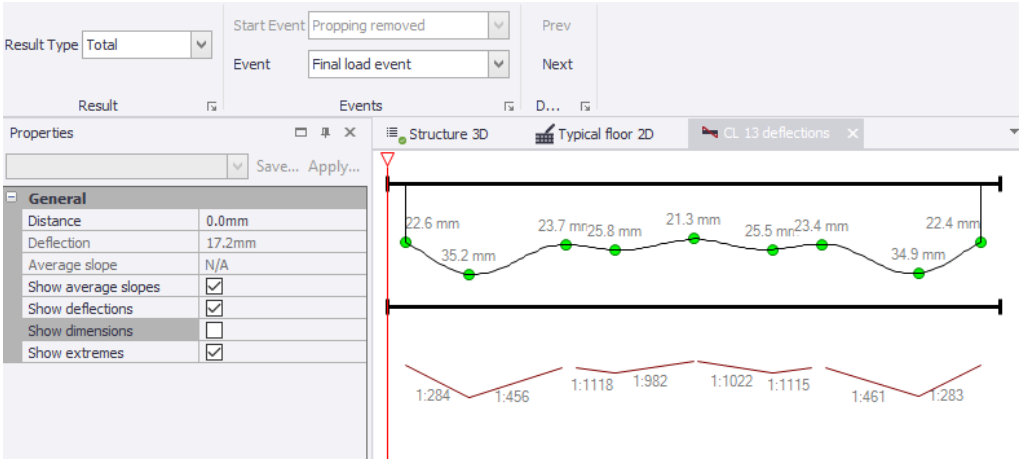


c. Press **Esc** to end the command.

4. Right click on the check line and choose **Open deflections check view** from the context menu.



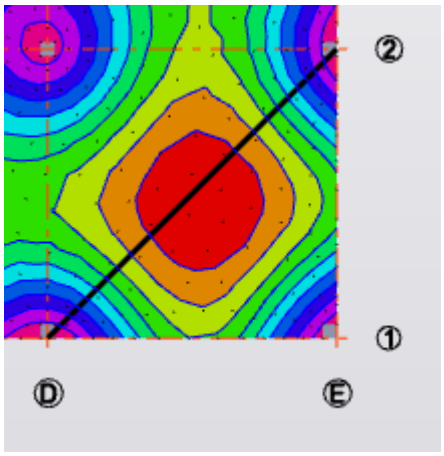
The deflection results along the length of the check line are displayed.



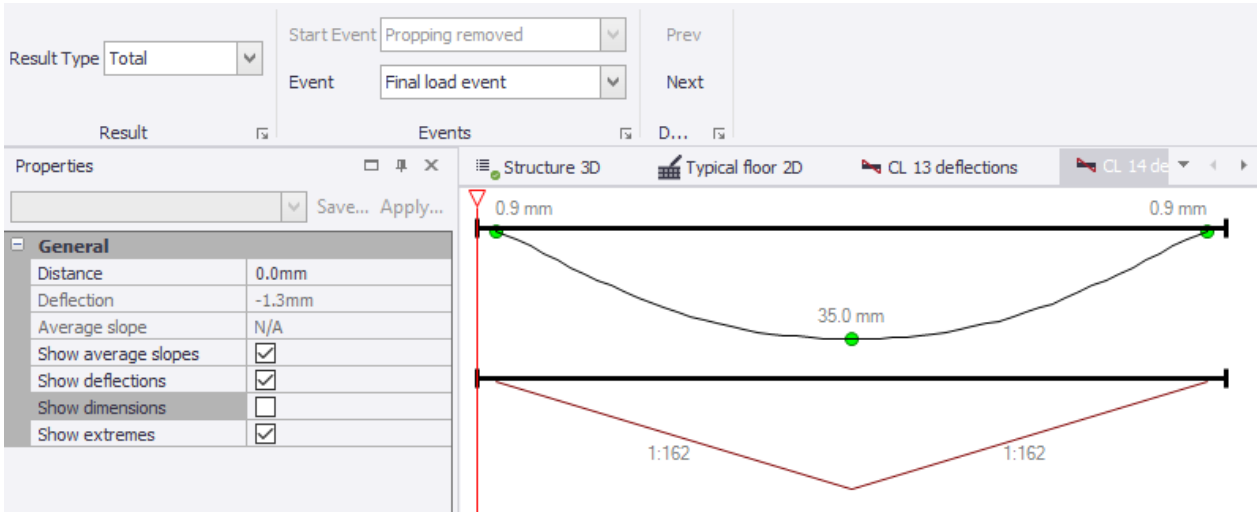
The **Slab Deflection** toolbar allows you to specify the total (as shown above), or differential or instantaneous results for the selected events. Tekla Structural Designer then draws average slopes between maximum and minimum points.

If we return to the original deemed-to-satisfy check - this was performed diagonally between columns in bottom right corner panel of the slab - we will now revisit this using a check line.

5. Create a check line running diagonally between columns in bottom right corner panel (from D/1 to E/2) where the peak deflection occurs.



6. Right click to open the deflections check view for the new check line.



A total deflection limit of Span / 250 is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 125. In the view above the average slope between these points is 1 / 155, i.e. less than the limit.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

Generate Check Line Reports

A tabulated report is available for each check line which itemises each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

1. Return to the **Typical floor 2D** view, right click on the diagonal check line and select **Report for Member**

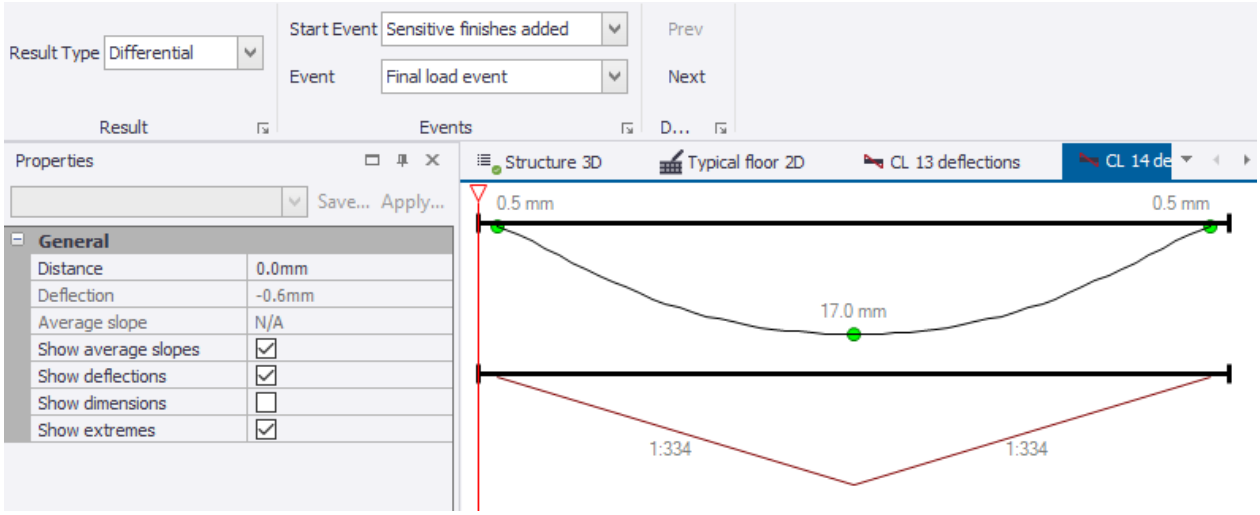
Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	16.5	5515.4	1 : 334	✓ Pass	0.749
Total	250	1 : 125	34.1	5515.4	1 : 162	✓ Pass	0.772

As previously noted, the slope above is reported as 1:155 which is not less than the allowable slope limit of 1:125 and hence a Pass.

If you click within the Load Analysis Deflection view, the ribbon changes to allow you to display deflection results and slopes for the Result Type - Total or Instantaneous for a chosen event, or Differential between chosen events.

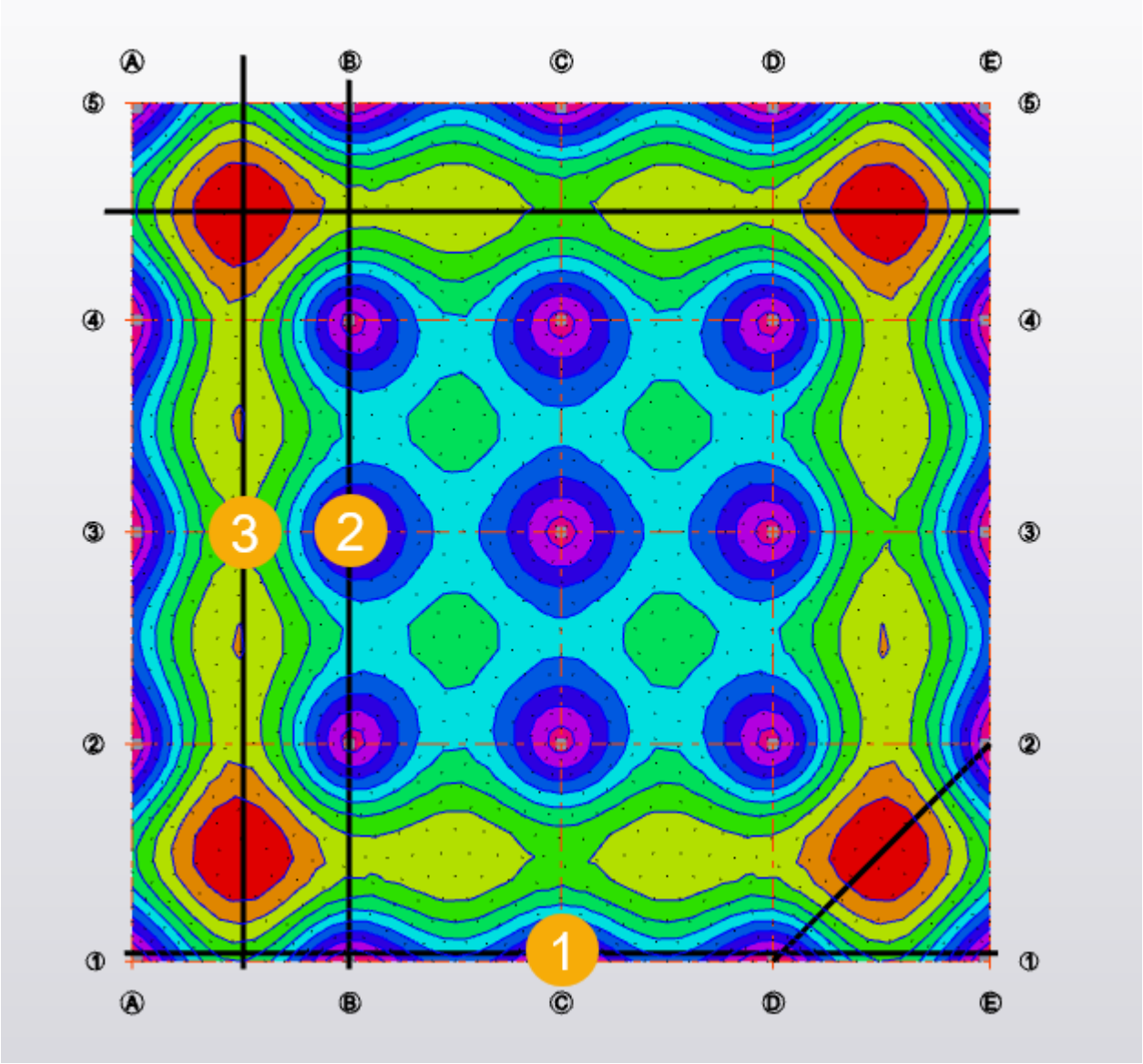
2. Switch to the Load Analysis Deflection view for the diagonal check line.
3. From the **Slab Deflection** toolbar, change the Result type to **Differential** and check deflection and slopes between the**Sensitive finishes added** and **Final load event**.



We can add as many check lines to the model as we consider appropriate.

4. Return to the **Typical floor 2D** view.
5. Add three further check lines using the default deflection checks in the catalogue as follows:
   a. along grid line 1, from A/1 to E/1
   b. along grid line B, from B/1 to B/5

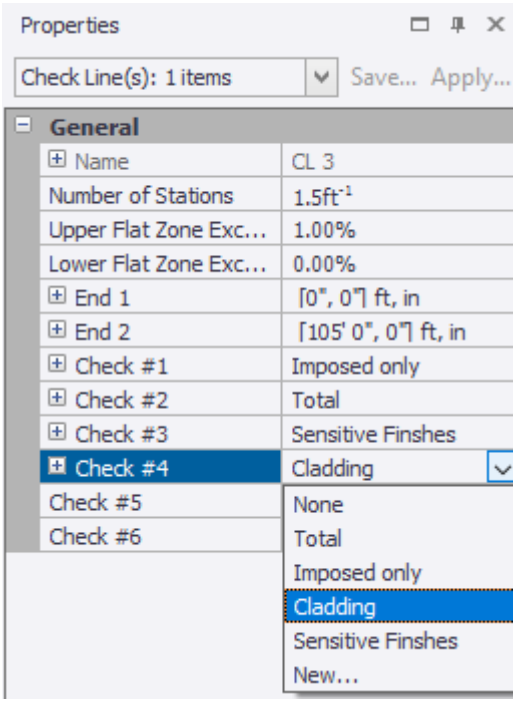
c. half way between grid line A and B, starting at grid line 1 and finishing at grid line 5



6. Press **Esc** to end the command.

   If you select each Check Line in turn you are able to edit the deflection checks associated with it in the Properties Window.

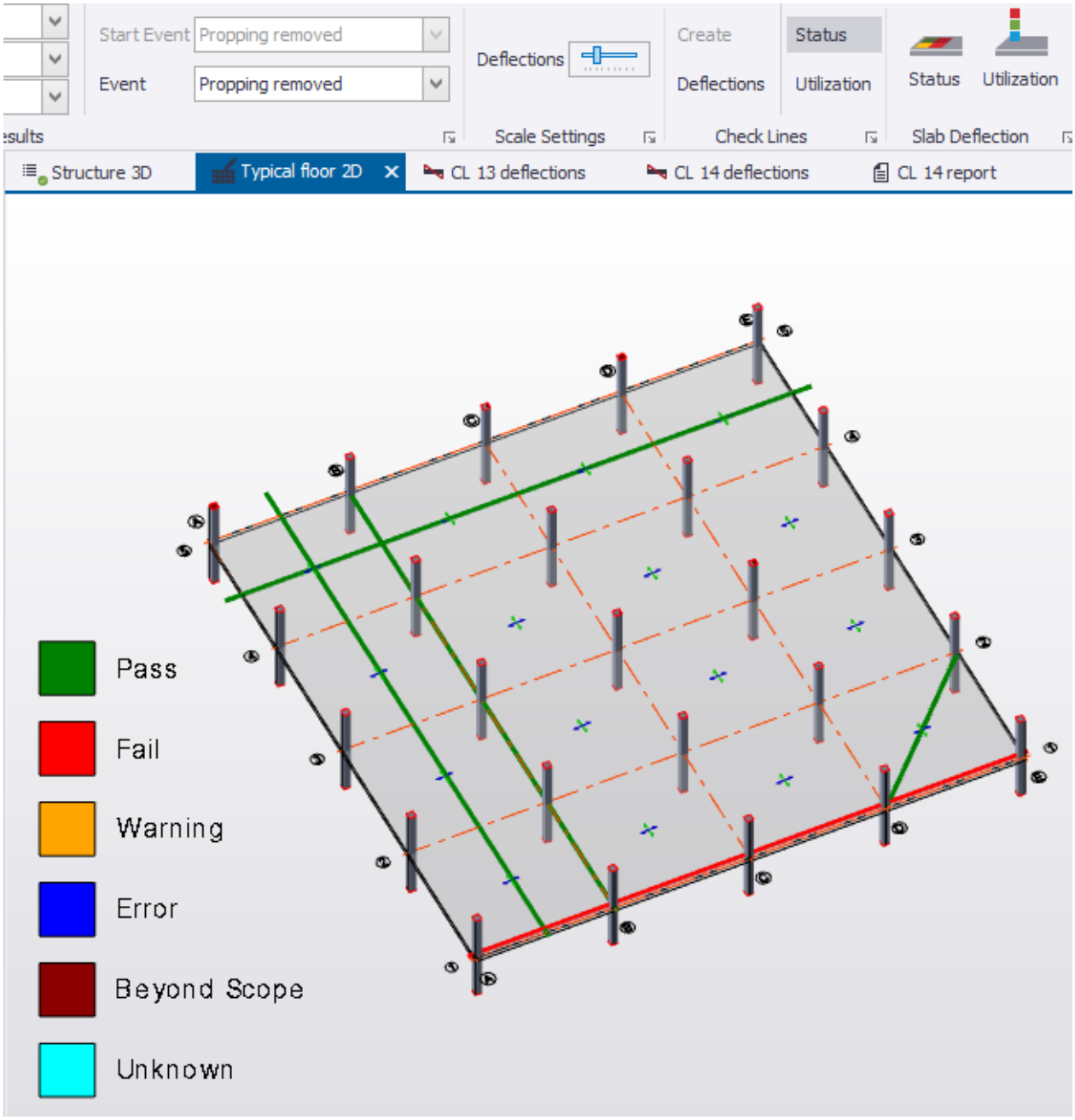
7. Select the check along grid line that runs along grid line 1 and ensure that it also has a **Check #3** defined as **Cladding**.



Review Check Line Status and Utilization

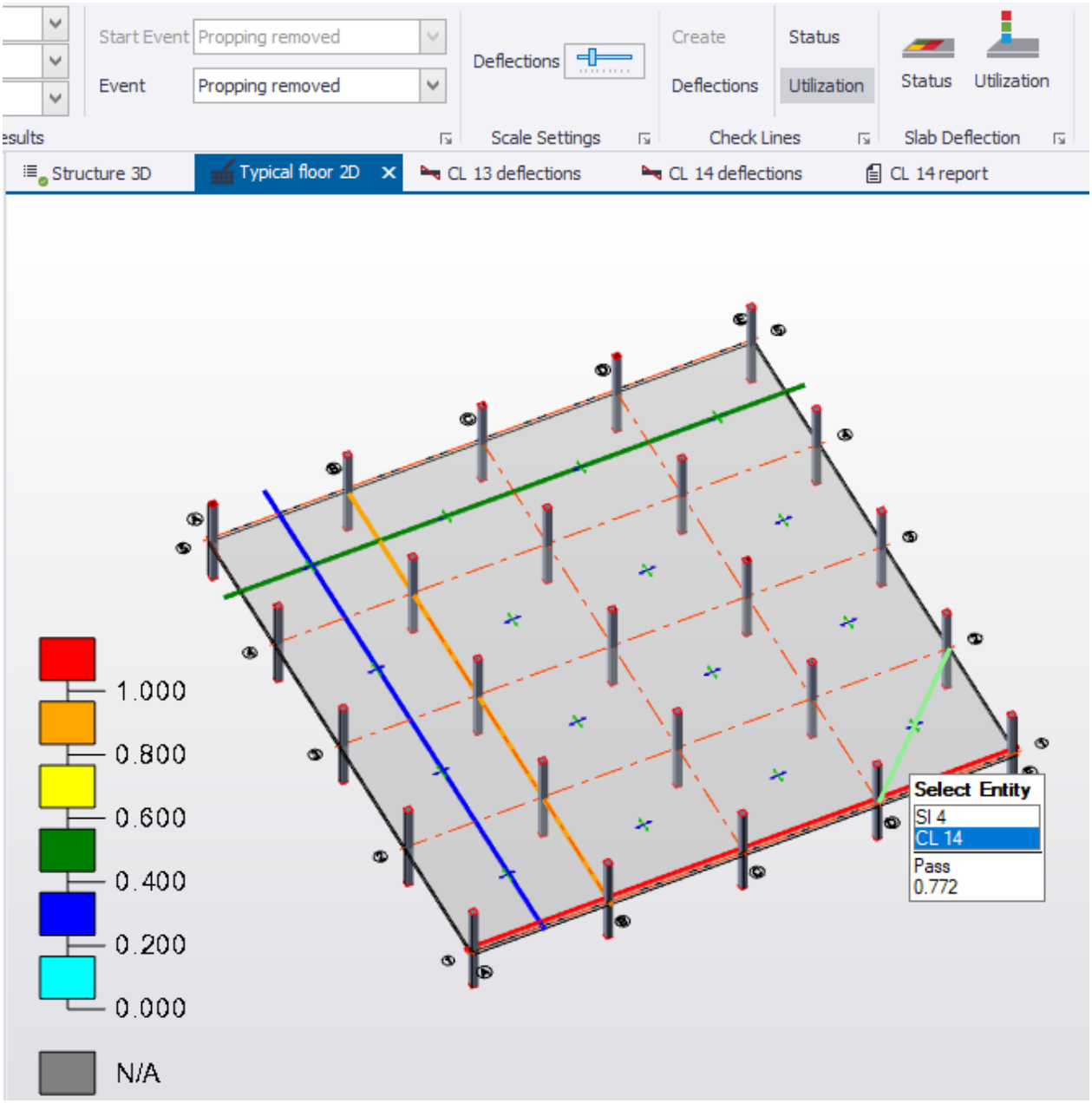
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio.

1. Click on the Typical floor 2D view to make it active.
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.
3. Click **Status** in the Check Lines group of the ribbon to see the pass/fail status graphically displayed for each check line.



TIP You can also hover over a check line and the tooltip displays the utilization and pass/fall status.

4. Click **Utilization** in the Check Lines group to show the critical utilization for each check line and investigate the tooltip results.



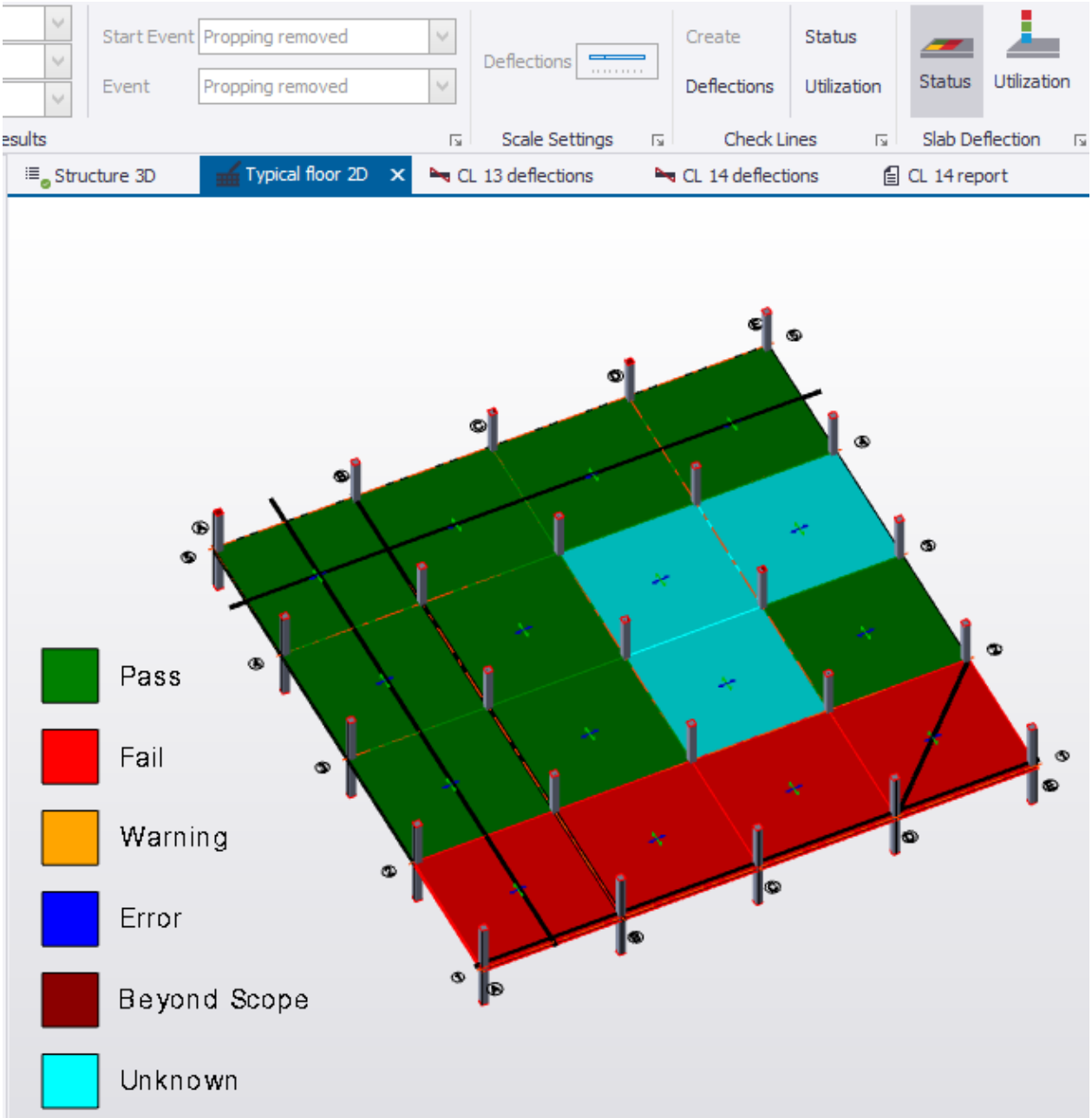
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization

Review Slab Status and Utilization

Every check line is associated with at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

Both the Status and the Utilization can be reviewed.

1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/fail status of each slab.

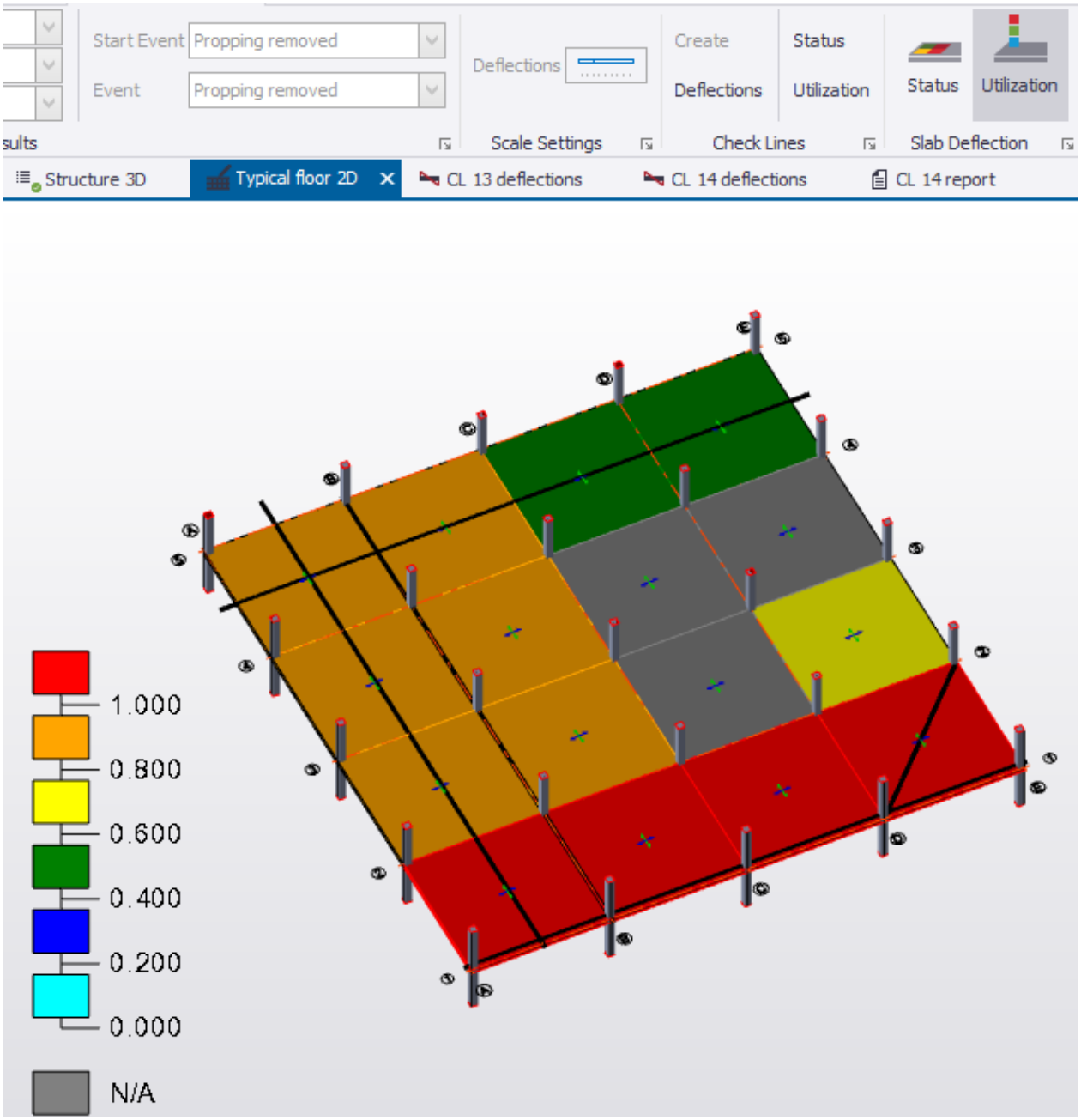


- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed.
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.

- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the slab items Fail where the check lines runs along A/1-E/1.

2. Click **Utilization** in the Slab Deflections group to show the Utilization of each slab item.

   This is the worst utilization from all associated check lines.



Optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

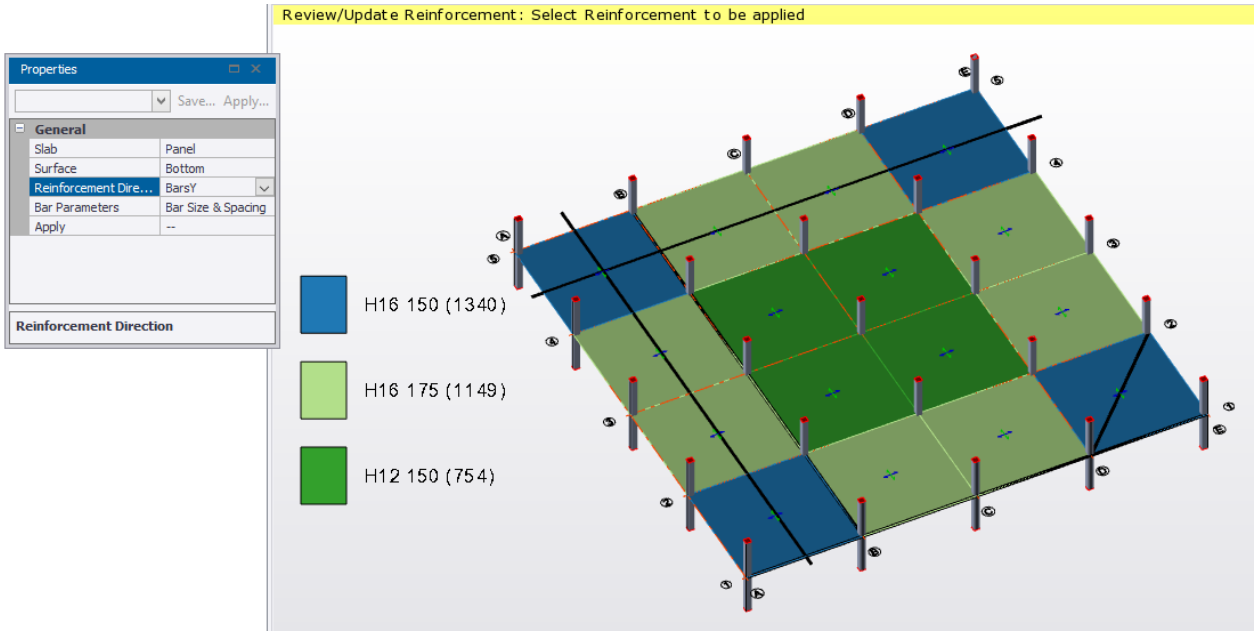
The analysis is extremely quick and since everything is contained within one model file, it allows “What If” scenarios to be considered to find the optimum solution.

In this exercise we will start by looking at the impact of adjusting the reinforcement.

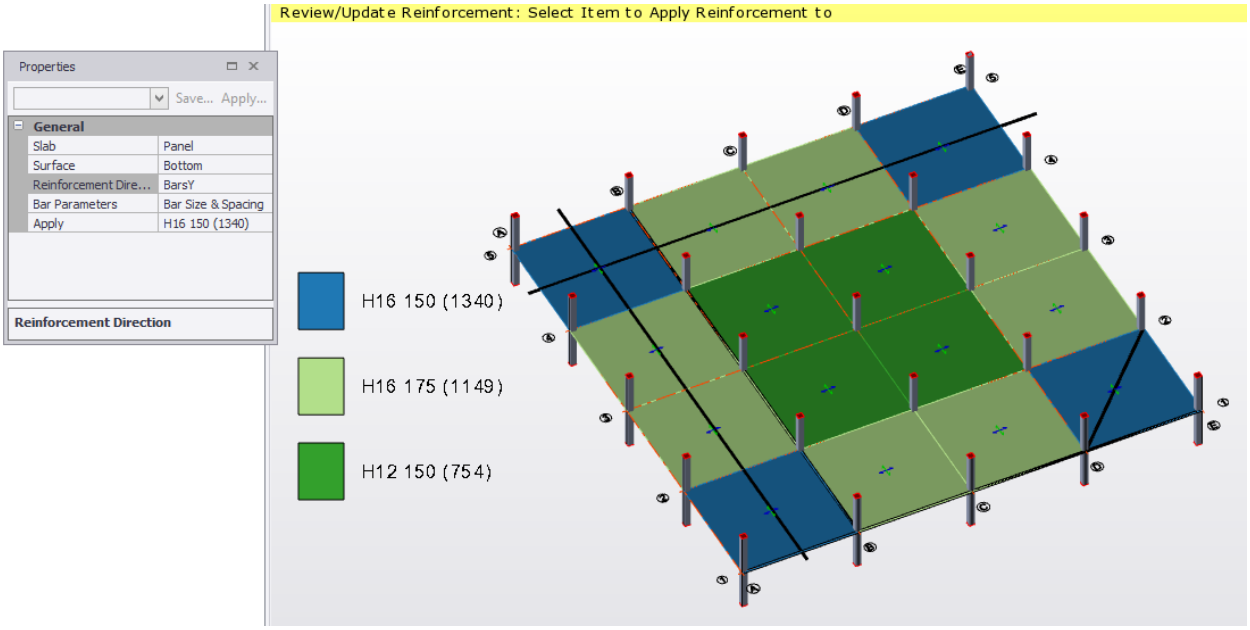
1. Highlight the check line running along grid line B and make a note of its utilization.

   TIP Press the tab key if necessary to highlight the check line when it is directly under the mouse cursor.

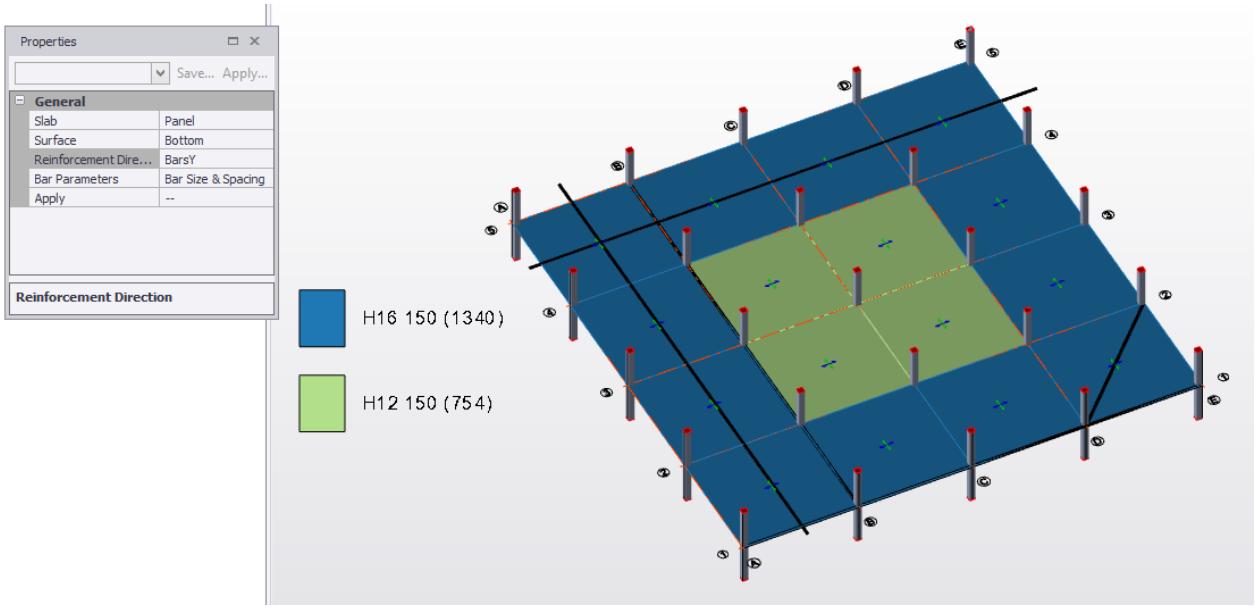
2. Click **Slab Reinforcement** in the Show/Alter State group to show the existing reinforcement.
3. In the Properties Window:
   a. Leave the Slab Reinforcement to modify as **Panel**
   b. Leave the Slab Layer to modify as **Bottom**
   c. Change the Reinforcement Direction to **BarsY** to see the bars in that direction



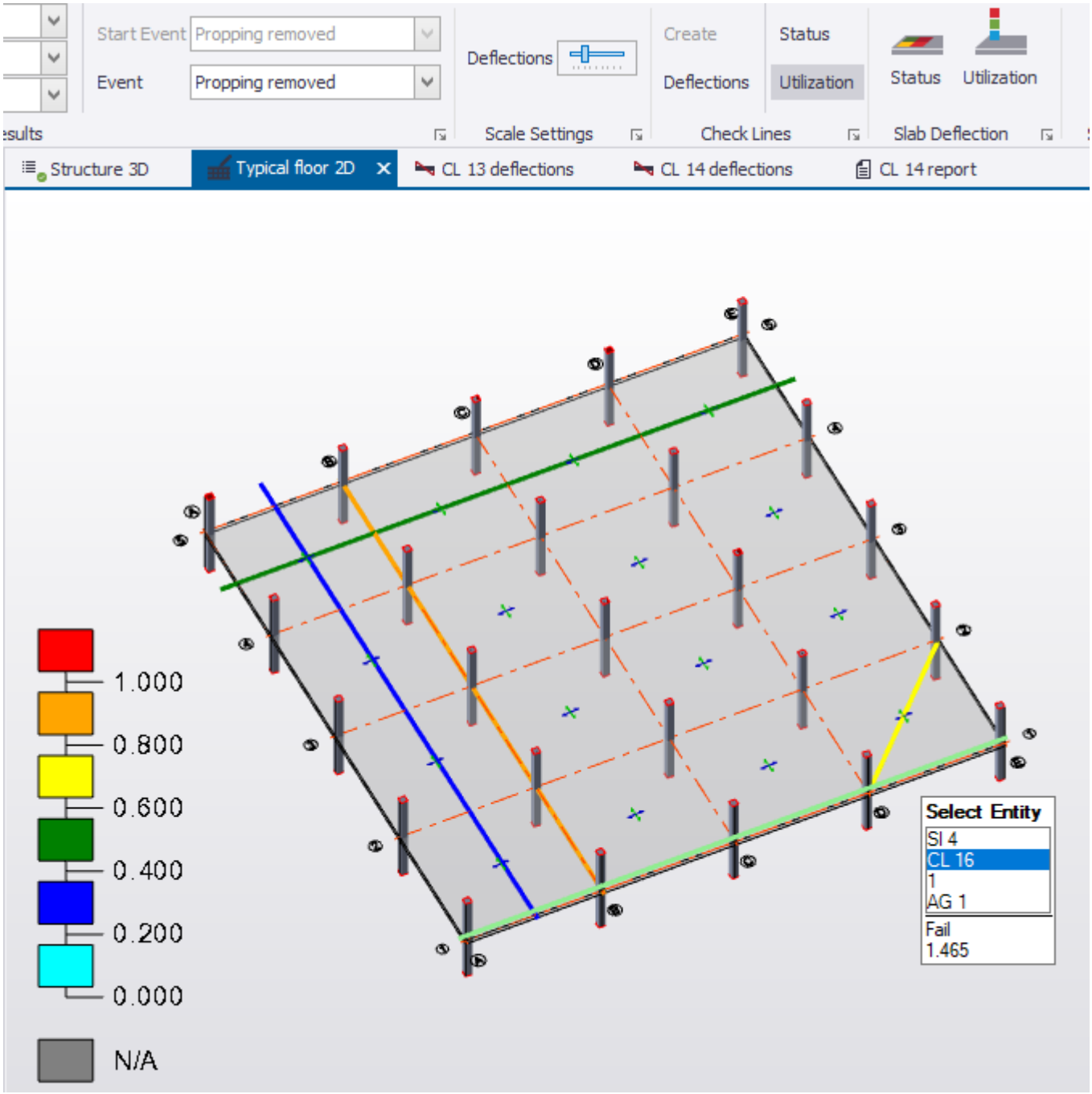
4. Click on one of the corner slab panels to select **H16 150** as the reinforcement to be applied.



5. Click on the eight slab panels currently showing H16 bars at 175 spacings to change them to the 150 spacings.



6. Change the Reinforcement Direction to **BarsX** to see the bars in that direction.
7. Edit the same 8 panels in exactly the same way, so that the bars in X match the bars in Y.
8. Click **Analyse Current** to update the results
9. Click **Utilization** in the Check Lines group to show the critical utilization for each check line once again.
10. Investigate the tooltip results

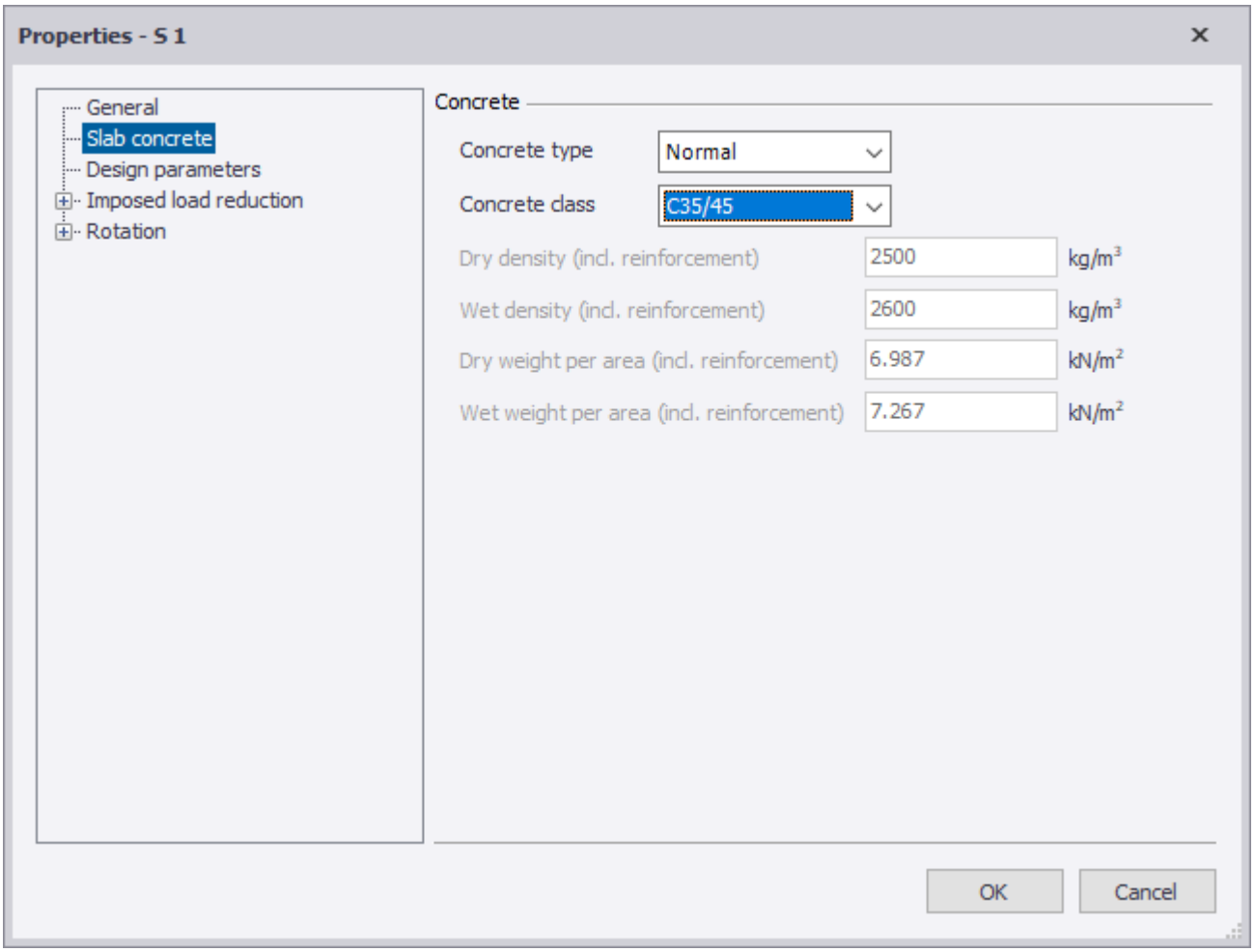


You should find that although some utilizations have reduced, the line along grid line B is still failing. At this point you could begin to look at the impact of the various other input parameters. For now, we will adjust the concrete grade of the typical floor slab group from C30/37 to C35/45.

11. Right click slab **SI 4** between D-E/1-2 on the typical floor level and choose **Edit....** from the context menu.

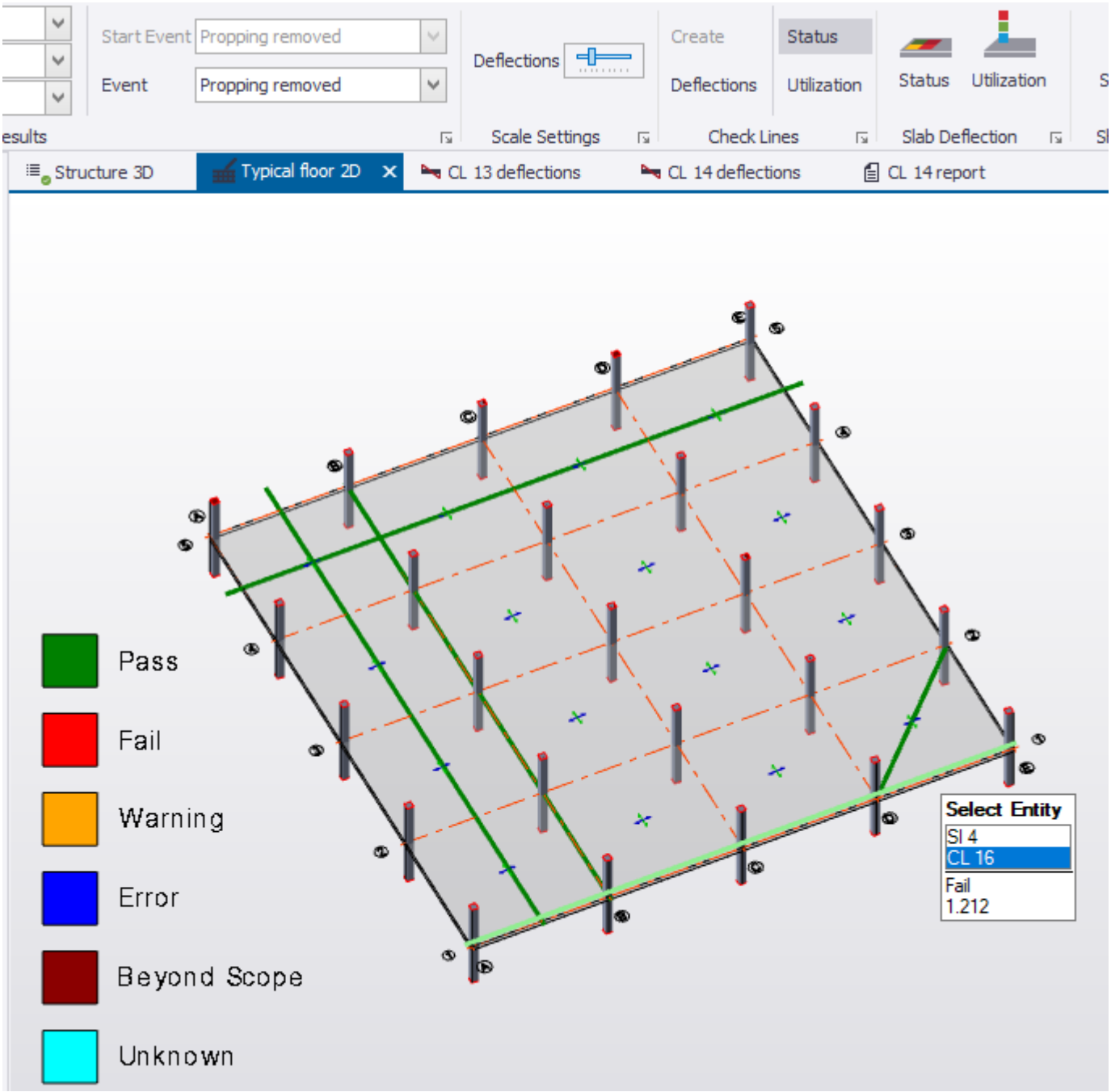
NOTE Editing a slab item via the right click context menu updates the parent slab properties, not just the individual slab item. Hence any changes will be applied to all the slab items in S 1.

12. On the **Slab concrete** page of the dialog, change the Concrete Strength to **C35/45** and click OK



To update the check line results we need to re-run the analysis. A chasedown analysis is automatically performed as part of the slab deflection analysis, however, it should be borne in mind that some edits could affect the element design i.e. reducing the concrete slab thickness would result in an increase to the required reinforcement and hence a Design Concrete (Static), Slab or patch design may be required again.

13. Click **Analyse Current** again to update the results.
14. Review the Check Lines **Status**.



The check line with the more onerous Cladding deflection limit is still failing, but we can clearly see improvements in the results. It has reduced from a Utilization ratio of 1.538 to 1.262.

In fact, to obtain a Pass we would have needed to increase the concrete grade to C45/55.

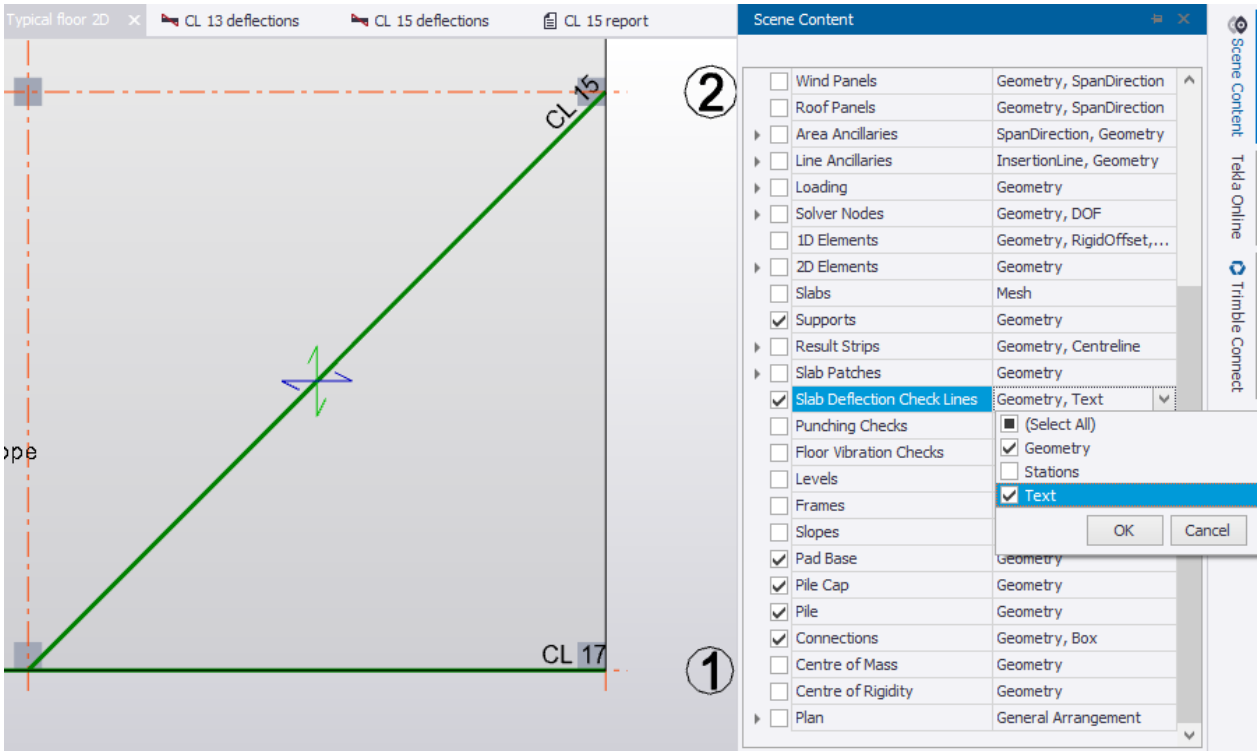
NOTE For the purpose of this exercise, as the cladding check is not a code requirement we will simply disable it before proceeding to generate the model report.

15. Select the check line along grid 1 and then in the **Properties window**, reset Check #3 to **None**.

Generate Model report

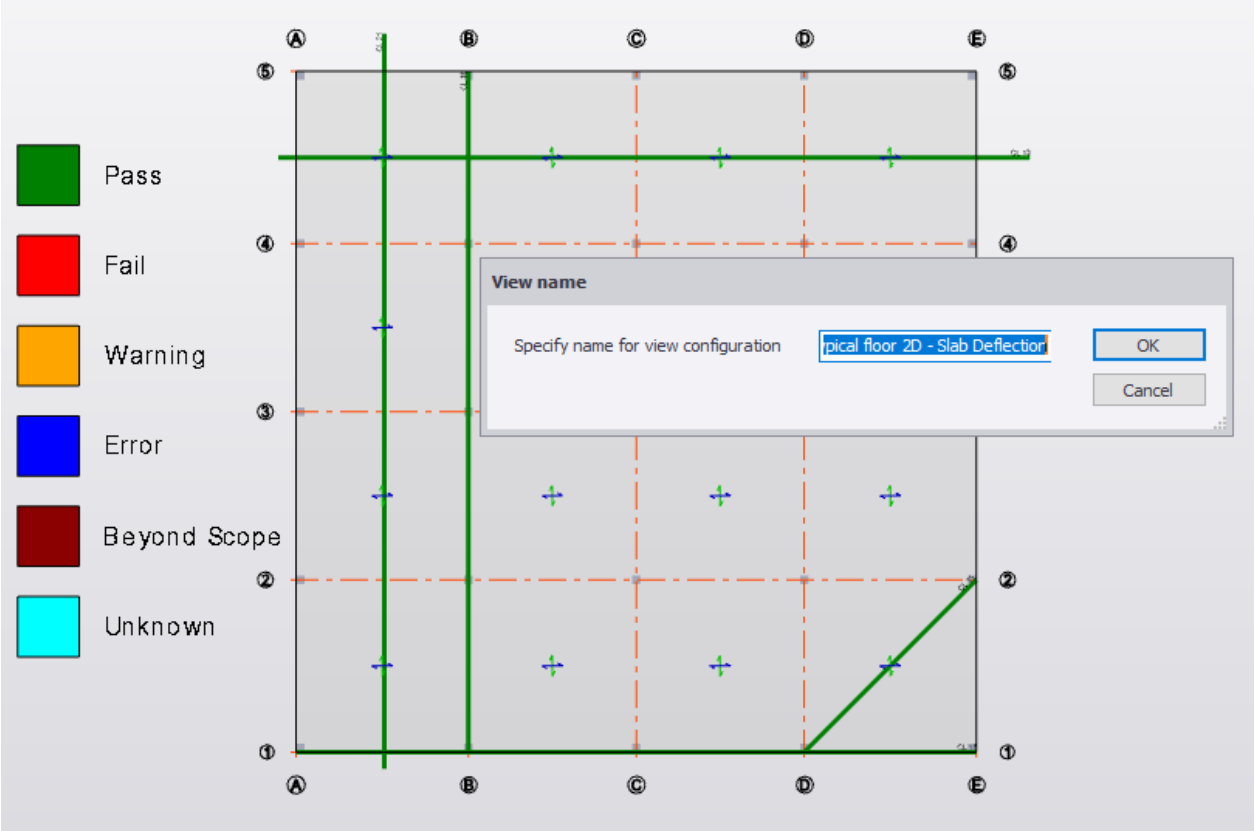
A Slab Deflection Check Lines model report can be created for the selected Model Filter (entire structure, level, plane or sub structure). This lists all the check lines for the chosen model filter. To help identify the check lines in the report it is sensible to include a saved picture of the scene view displaying check lines and their associated reference within the report.

1. In Scene Content, switch on the Text display for the Slab Deflection Checks.



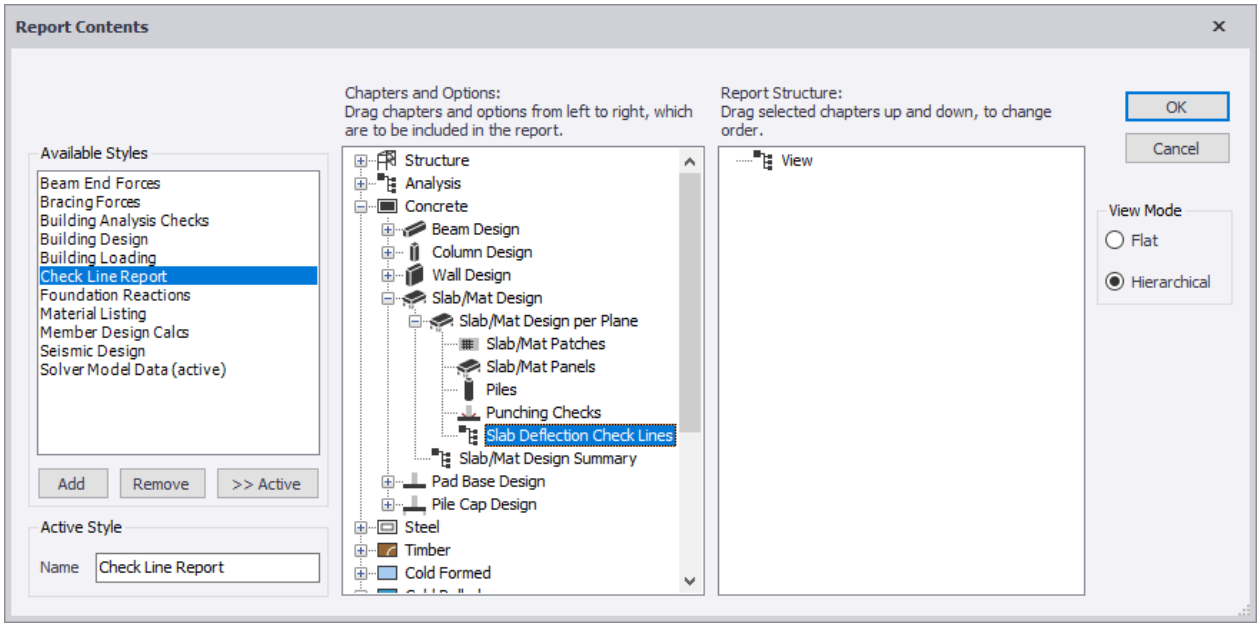
TIP The check line references can be customised using the Name property for each individual check line.

2. Right click in the Typical floor 2D view and choose **Save View Configuration...** from the context menu, then specify a name.

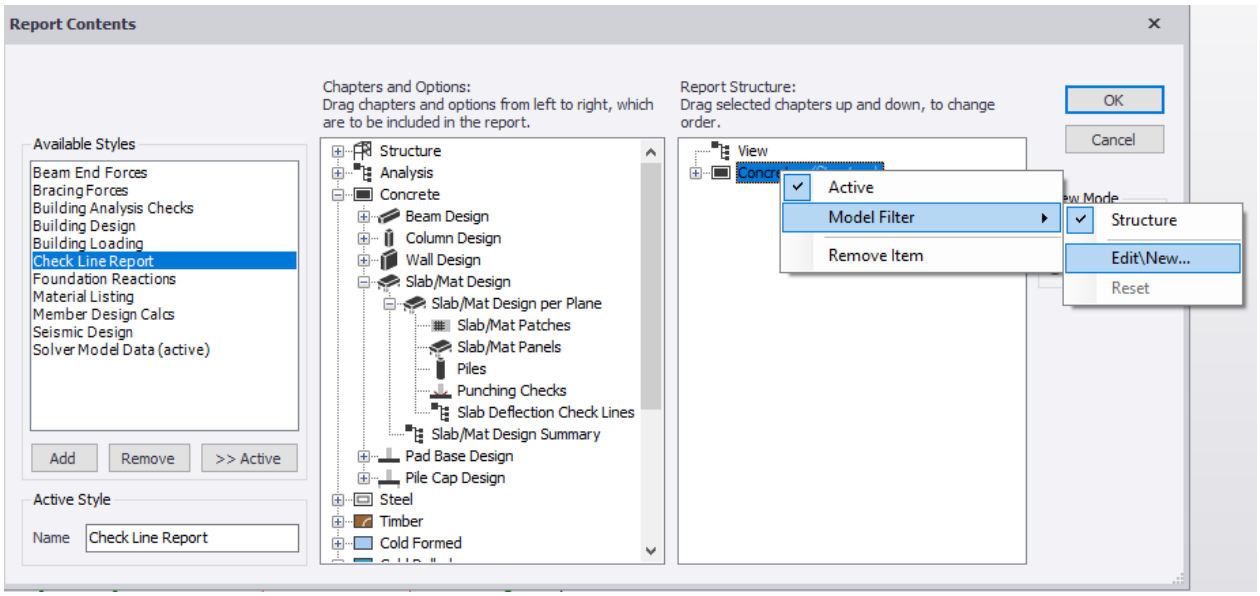


This saved view can be included in the Check line report.

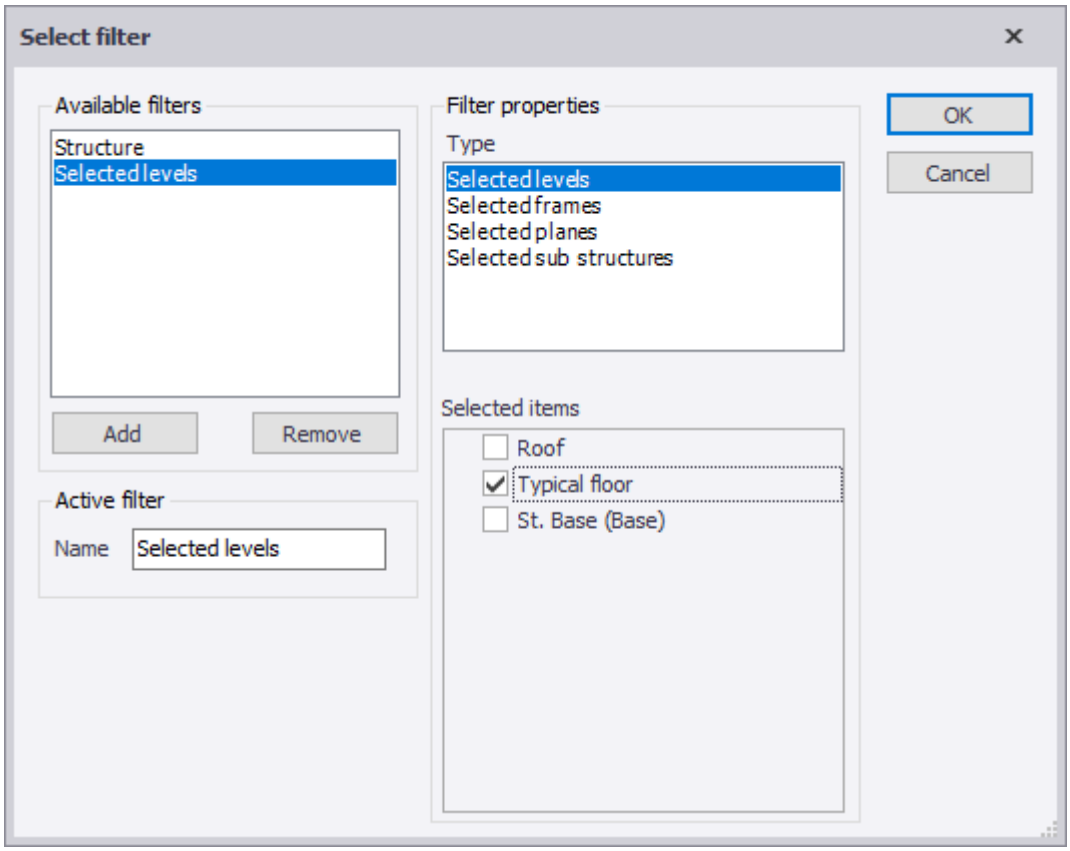
3. On the Report ribbon, click **Model Report...**
4. Click **Add** and provide a Name “Check Line Report” for the report.
5. In Chapters and Options, drag **View** to the Report Structure area
6. In Chapters and Options, drag **Concrete>Slab/Mat Design per Plane>Slab Deflection Check Lines** to the Report Structure area



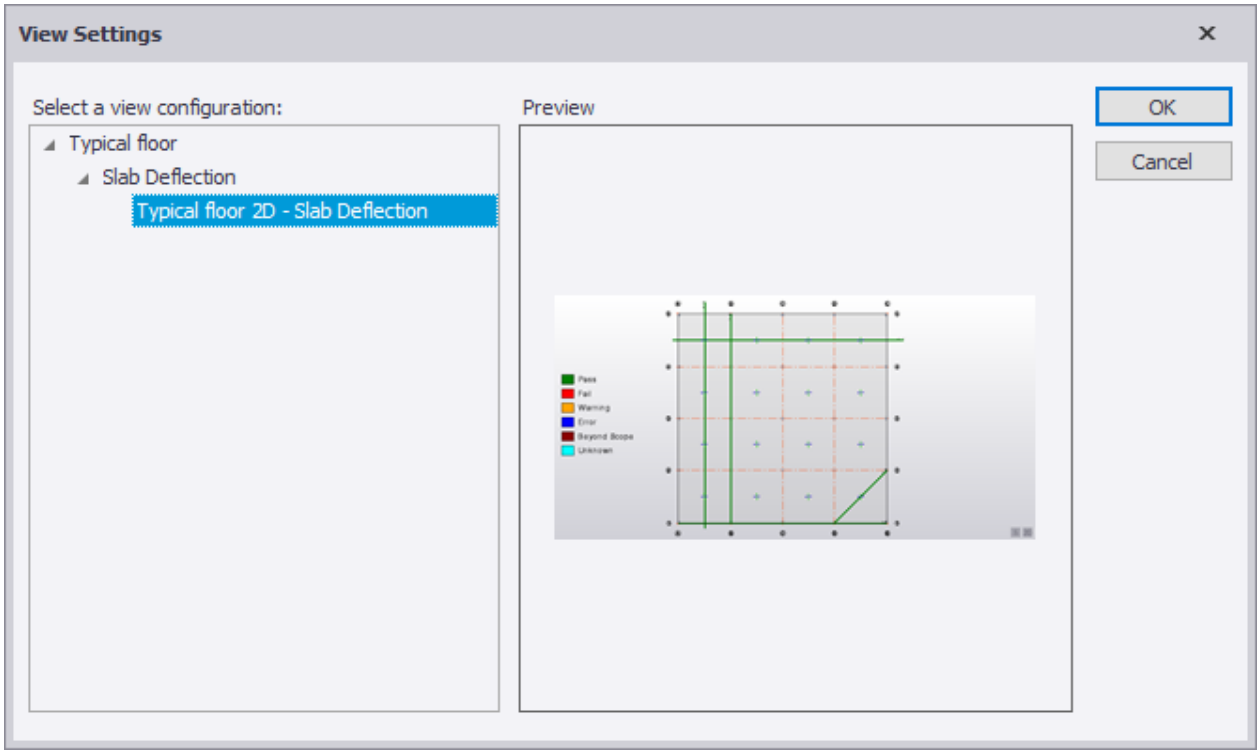
7. In the Report Structure, expand **Concrete> Slab/Mat Design per Plane> Slab Deflection Check Lines** and right click, **Model Filter> Edit/New**



8. In the Filter dialog, click **Add** and select Selected levels and ensure a check against Typical floor

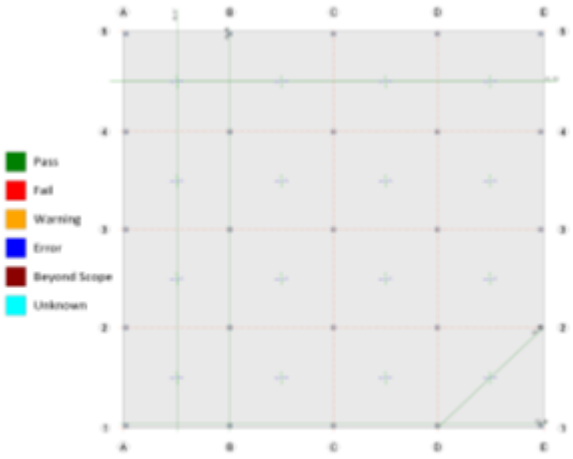


9. Click **OK** to return to the Report Contents dialog.
10. In the Report Structure, right click **View**, then choose **Settings...**
11. Select the Slab Deflection view you created earlier



12. Click **OK** to return to the Report Contents dialog
13. Click **OK** to exit and save the report.

    A report structure called Check Line Report has now been saved that contains a view and the check lines.
14. To display the report.
    a. Use the Select drop list in the ribbon to select “Check Line Report”
    b. Click the Show Report command to open the report.



Typical floor 2D - Slab Deflection

Concrete

Slab/Mat Design

Slab/Mat Design per Plane

Typical floor

Slab Deflection Check Lines

CL 13

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	-3.9	3520.1	1 : 900	✓ Pass	0.278
Total	250	1 : 125	-10.7	3520.1	1 : 330	✓ Pass	0.379

CL 14

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	14.0	5515.4	1 : 394	✓ Pass	0.635
Total	250	1 : 125	-29.3	5515.4	1 : 188	✓ Pass	0.665

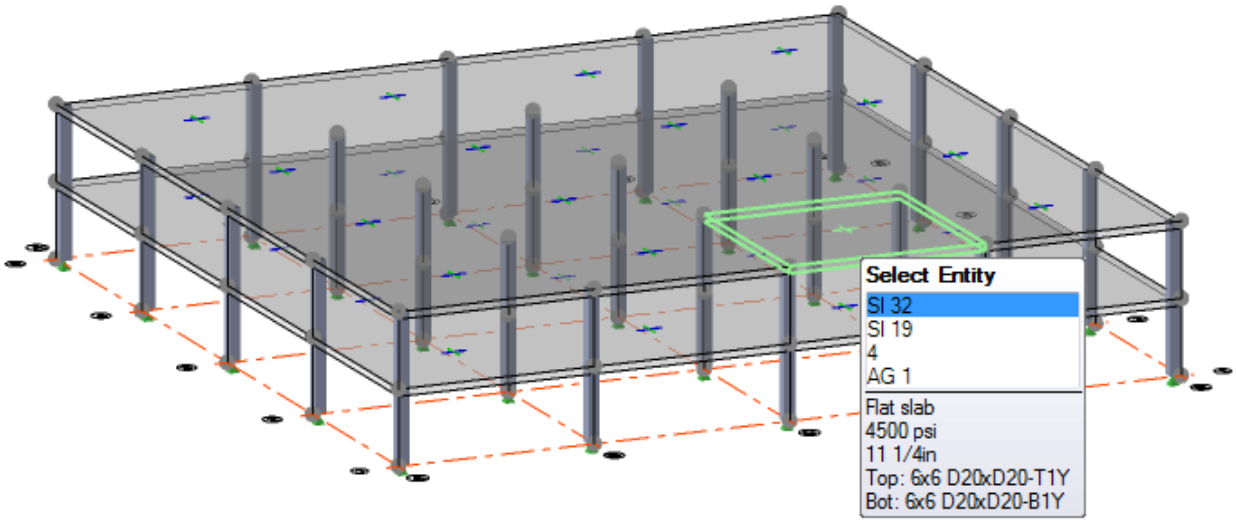
CL 15

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [mm]	Length [mm]	Slope	Status	Utilization
Sensitive Finishes	500	1 : 250	8.7	3324.4	1 : 381	✓ Pass	0.656
Total	250	1 : 125	19.2	3324.4	1 : 173	✓ Pass	0.723

Slab deflection example (ACI)

In the following exercises deflections will be checked for the two story multi-bay flat slab tutorial model shown below.



Geometry:

- Bay centers 26’ 3”
- Floor to Floor spacing 9’ 10”
- Slab thickness 11 ¼ “, concrete grade 4500 psi
- 18” x 18” columns, concrete grade 4500 psi
- Analysis stiffness adjustment factors in accordance with ACI code guidance

Loading:

- Dead area load 30 psf
- Live area load 105 psf
- Cladding perimeter load 0.7klf

Deemed to satisfy checks (page 789) provide one method of checking, however the main focus of these exercises will be to investigate rigorous slab deflection analysis (page 791) and check lines (page 822) as the method of checking.

Deemed to satisfy slab deflection checks example (ACI)

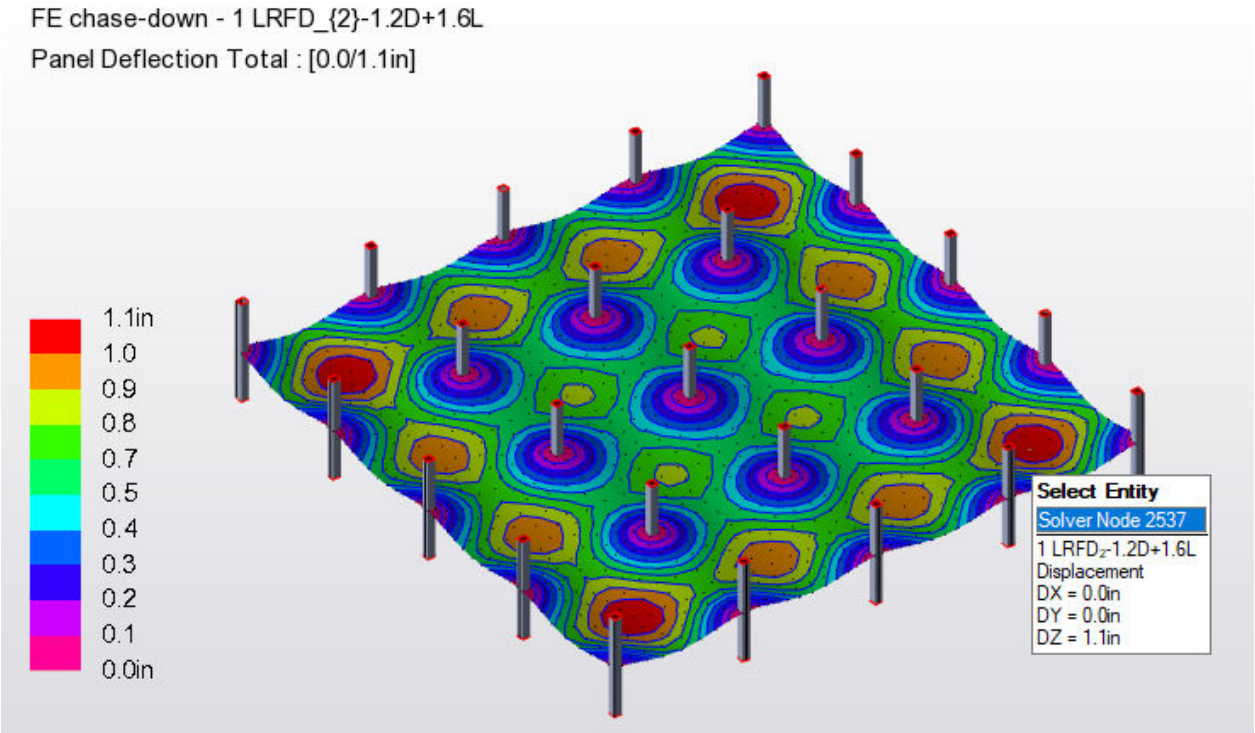
A simple way to assess slab deflection in Tekla Structural Designer is to run a linear analysis using adjusted analysis properties, and then check the resulting deflections by manually determining critical spans.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI.tsmd

Perform Linear Analysis

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. Open a view of the **Typical floor** level
3. Switch to the **Results View**
4. From the Results toolbar, review **2D deflections** for the **FE chase-down analysis** for the load combination 1, service load results



Identify critical check locations

We can see that the maximum reported deflection is 1.1in, occurring in the middle of a corner bay. This should be assessed by taking the slab span diagonally across the bay.

NOTE In 'real world' flat slabs engineering judgment will be required to assess which deflections and span lengths require checking.

The deflection at the identified location now needs to be checked against a limiting span-to-depth ratio which we will assume for this example can be taken as span /240.

Taking the diagonal dimension across the columns, the deemed-to-satisfy span / 240 rule provides a deflection of [√(26' 3"^{2}+26' 3"^{2})] / 240 = 1.8in.

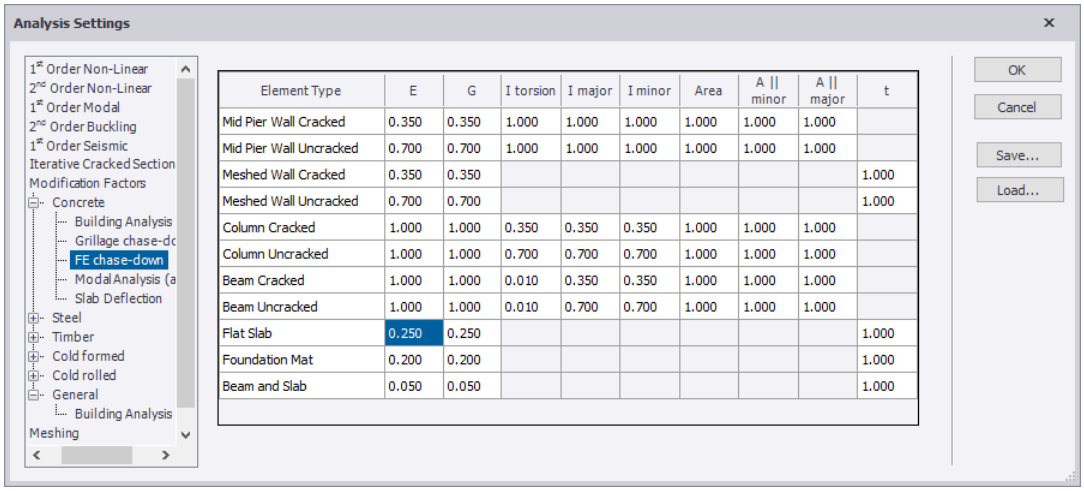
This compares unfavorably.

NOTE Remember, the method does not predict actual deflections. The total deflection is simply expected to be less than span / 240.

Concrete properties used in the analysis

The Tekla Structural Designer deflection result is completely dependent upon the concrete elastic modulus used in the analysis which is adjusted by a modification factor to consider such things as creep, cracking and shrinkage.

The modification factor is set from the Settings dialog on the **Analyze** ribbon. As shown below, for the FE chase-down analysis of flat slabs this defaults to 0.25.



Element Type	E	G	I torsion	I major	I minor	Area	A || minor	A || major	t
Mid Pier Wall Cracked	0.350	0.350	1.000	1.000	1.000	1.000	1.000	1.000	
Mid Pier Wall Uncracked	0.700	0.700	1.000	1.000	1.000	1.000	1.000	1.000	
Meshed Wall Cracked	0.350	0.350							1.000
Meshed Wall Uncracked	0.700	0.700							1.000
Column Cracked	1.000	1.000	0.350	0.350	0.350	1.000	1.000	1.000	
Column Uncracked	1.000	1.000	0.700	0.700	0.700	1.000	1.000	1.000	
Beam Cracked	1.000	1.000	0.010	0.350	0.350	1.000	1.000	1.000	
Beam Uncracked	1.000	1.000	0.010	0.700	0.700	1.000	1.000	1.000	
Flat Slab	0.250	0.250							1.000
Foundation Mat	0.200	0.200							1.000
Beam and Slab	0.050	0.050							1.000

Rigorous slab deflection analysis examples (ACI)

Three alternative methods of allowing for creep and shrinkage in the analysis are investigated.

- Slab Deflection Property Choices (page 792)
- Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance (page 793)
- Method 2: Simplified Event Sequence + Rigorous Creep and Shrinkage Allowance (page 804)
- Method 3: Detailed Event Sequence + Rigorous Creep and Shrinkage Allowance (page 810)

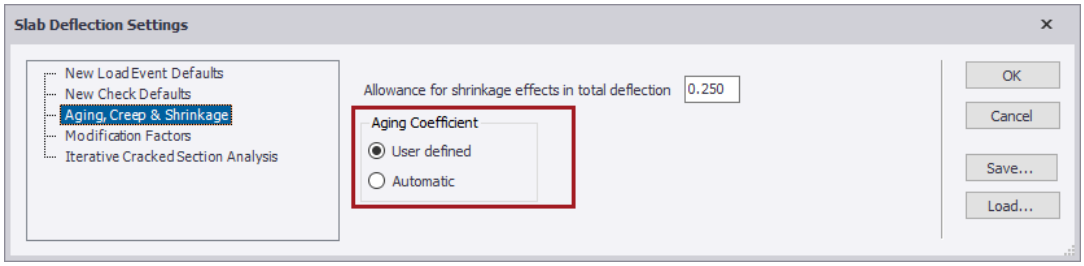
- Observations on the Different Methods (page 817)
- Use of check lines to check deflections (page 822)

Slab Deflection Property Choices

Aging Coefficient - User Defined or Automatic?

Tekla Structural Designer calculates a modulus of elasticity for each load event which accommodates both creep and aging.

The **Aging, Creep and Shrinkage** page on the Slab Deflection Settings dialog allows you to choose the method for this.



- **Automatic** - the modulus of elasticity used in the analysis (termed the composite modulus) is calculated rigorously according to the procedure defined in the Concrete Society Technical Report 58. This requires an early age event history to be defined by way of a detailed event sequence.
- **User defined** - the modulus of elasticity used in the analysis is calculated in accordance with ACI 435 Equation 3.33c:

  $\bar{E}_c(t, t_0) = E_c(t, t_0) / (1+ \chi C_t)$

  Where:

  - The aging coefficient χ must be specified in the event sequence. The recommended value is 0.8. This is a user specified value.
  - The creep coefficient C_t required in the above equation is automatically determined by *Tekla Structural Designer* using equation A-18 from ACI 209 taking account of shape and size effects.

  The equation for C_t is $[(t_i-t_0) / (26\, e^{(0.36 \times V/S)}) + (t_i-t_0)] \times C_{ui}$

  Where:

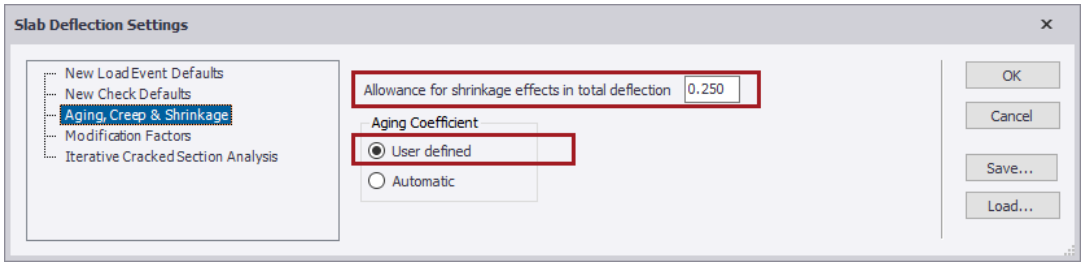
  - For a slab the volume to exposed surface ratio V/S is taken as h/n where h is the slab thickness and n is the number of exposed surfaces. If n is set to zero then the creep coefficient is taken as zero thus eliminating creep from the analysis properties.
  - Start of first event, t_0
  - End time of event under consideration, t_i
  - The Ultimate Creep coefficient, C_u must be specified in the event sequence. In normal conditions the recommended value from ACI 435R

page 55 is C_u = 2.35. A different value can be specified according to guidance in ACI 209 if desired.

Shrinkage Allowance, or Combined Creep and Shrinkage Allowance?

When creep effects are included in the analysis, the Allowance for Shrinkage Effects multiplier can be set at a value which caters for shrinkage only. Alternatively you can increase the multiplier to allow for both creep and shrinkage, provided that you ensure that creep is not also accounted for in the analysis.

When using the latter approach it is important to ensure that the Aging coefficient is set to User defined.



Restraint Constant - Significant or Insignificant Restraint?

For each of the three methods considered in this example the Restraint constant (ACI) (page 721) will be varied in order to model different restraint assumptions.

Restraint constant values from ACI 435 are:

- For situations with significant restraint - **4.0**
- For insignificant restraint - **7.5**

This will allow us to compare deflections estimations based on the assumption of significant restraint against those assuming insignificant restraint.

Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance

In this approach the simplest method suggested in ACI 318 is emulated. All creep and shrinkage effects are introduced as a single amplification factor and a cracked section analysis is run on a simplified event sequence without addressing the possibility of early age loading.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
    - Slab deflection ACI - Simplified Event Sequence.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Set up the Simplified Combined Creep and Shrinkage Allowance (ignoring ACI 435)

You can increase the **Allowance for Shrinkage Effects** multiplier to allow for both creep and shrinkage based on the multipliers from ACI 435 Table 4.1.

Using the **ACI code** recommendation (highlighted in the below table), multipliers are immediate **1.0**, creep and shrinkage **2.0**. Total = **3.0**.

Thus a creep and shrinkage contribution = 2.0/3.0 = 0.66 is to be applied which equates to 66% of the total deflection.

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 $\sqrt{f_c'}$	1.0	2.8	1.2	5.0
Branson (1977)	7.5 $\sqrt{f_c'}$	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 $\sqrt{f_c'}$ 4 $\sqrt{f_c'}$	1.0 1.0	2.0 1.5	2.0 1.0	5.0 3.5
ACI Code	**7.5 $\sqrt{f_c'}$**	**1.0**	**2.0**		**3.0**

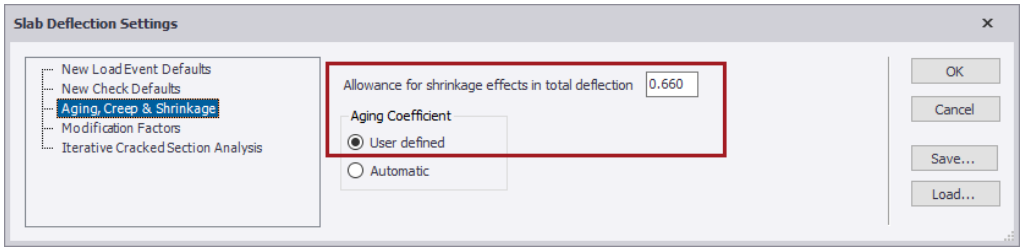
When using this approach:

- The automatic procedure using Technical Report 58 is not considered - this is ensured by setting the **Aging coefficient** to **User defined**.
- Creep is not also accounted for in the analysis - it can be excluded by setting the **Number of Exposed Faces** to **Zero** in the event sequence.

1. **To adopt the above ACI 318 Code multipliers for simplified creep and shrinkage:**

1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**

3. Ensure the **Aging Coefficient** is set to **User defined** and the **Allowance for shrinkage effects in total deflection** is set to **0.66**.



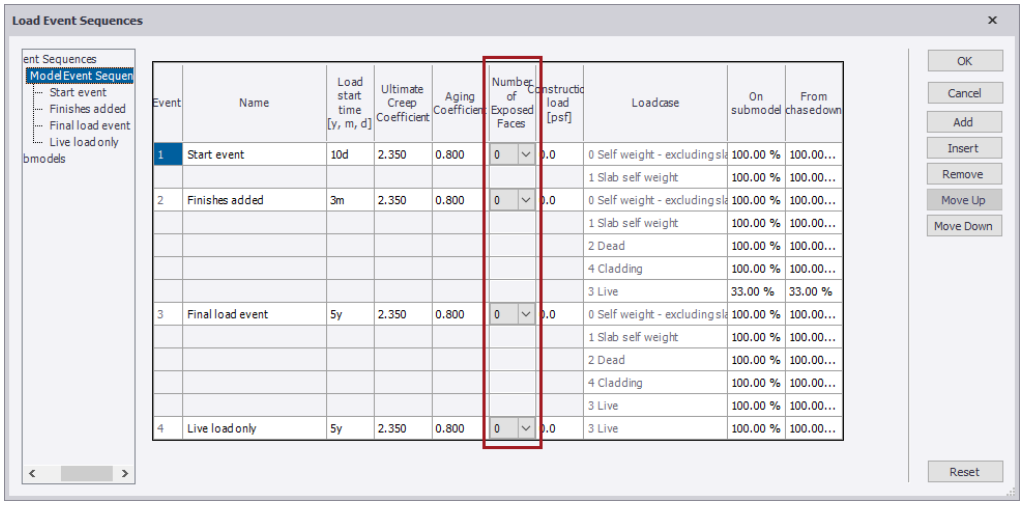
The above factor allows for both creep and shrinkage, (i.e. creep and shrinkage effects will account for 66% of the total deflection).

4. Click **OK** to close the dialog.

2. **To exclude additional creep effects from the analysis**

As creep is already being catered for by the amplification factor, we have to exclude additional creep effects from the analysis. This can be done by setting the number of exposed faces in each event to zero as follows:

5. From the **Slab Deflection** toolbar, click **Event Sequences**
6. Click **Model Event Sequence**
7. For each listed event in the model sequence, ensure the **Number of Exposed Faces** is set to **0**.



If the number of faces = 0 then the creep coefficient C_t = 0 . See: Aging Coefficient - User Defined or Automatic? (page 792)

Review the Individual Events in the Model Event Sequence

A simplified event sequence has been defined that does not include any construction stage propping events.

Various guidance documents discuss slab deflection analysis without addressing the possibility of significant early age loading events such as propping loads.

An early start event should always be created since time is measured from this start event. In this example we have assumed a start event of 10 days and allowed self weight only.

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 %

You will also note that:

- Each event has a load start time. The Final load event is set to the normal ACI requirement of 5 years.
- ACI requires that instantaneous deflection due to live loads only should be considered based upon a span/360 limit. To account for instantaneous deflection due to live load only, an end event has been included with the same load start time as the preceding event but only including live load.
- Ultimate Creep Coefficient, C_u is set with the default value of 2.350. This value can be set separately for each event.
- Aging Coefficient is set at the default value of 0.8. This value can be set separately for each event. It may be more logical to set higher values for the earlier event times, however, if your primary concern is differential deflection between later events then it will be conservative to use the same value everywhere.
- Number of exposed Faces is set to 0 - this has been done to exclude additional creep effects from the analysis, (as explained in the previous section).

Discussion:

- You need to think about the accumulation of deflection through time and hence the checks that you ultimately wish to consider.
- As an example, if you are interested in differential deflection between the final load event and the start of finishes (deflection at end of event 1), if you underestimate deflection to the end of event 1 then this check becomes more onerous.
- Is it reasonable to assume no construction load and no self weight from finishes during this period?
- How much load is reasonable to assume at this starting event is ultimately the responsibility of the engineer.

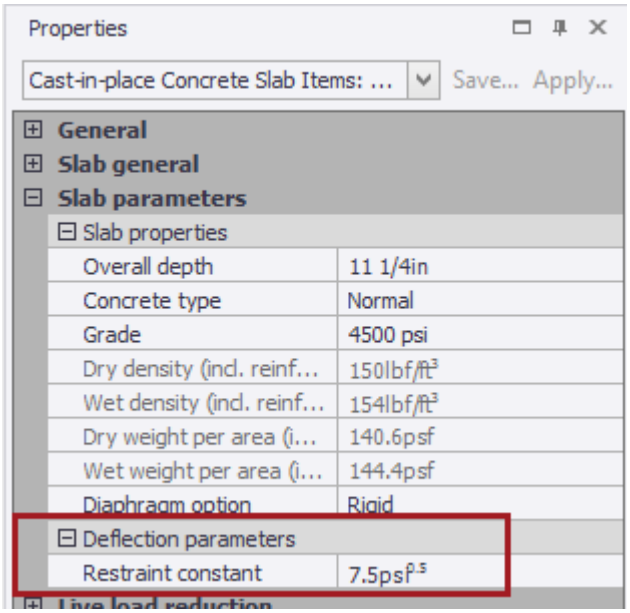
1. After reviewing the Event Sequence, click **OK** to close the dialog.

Set up the Restraint Constant

The modulus of rupture is set using the **Restraint Constant** slab deflection parameter. Since we are using the ACI code multipliers from Table 4.1 above this should be set to 7.5.

1. **To specify an appropriate restraint constant**

1. Open the **Structure** 3D view
2. Select all the slab items in the model and via the Properties Window, ensure the **Restraint Constant** is set to **7.5**



Perform Iterative Slab Deflection Analysis

To establish some initial results:

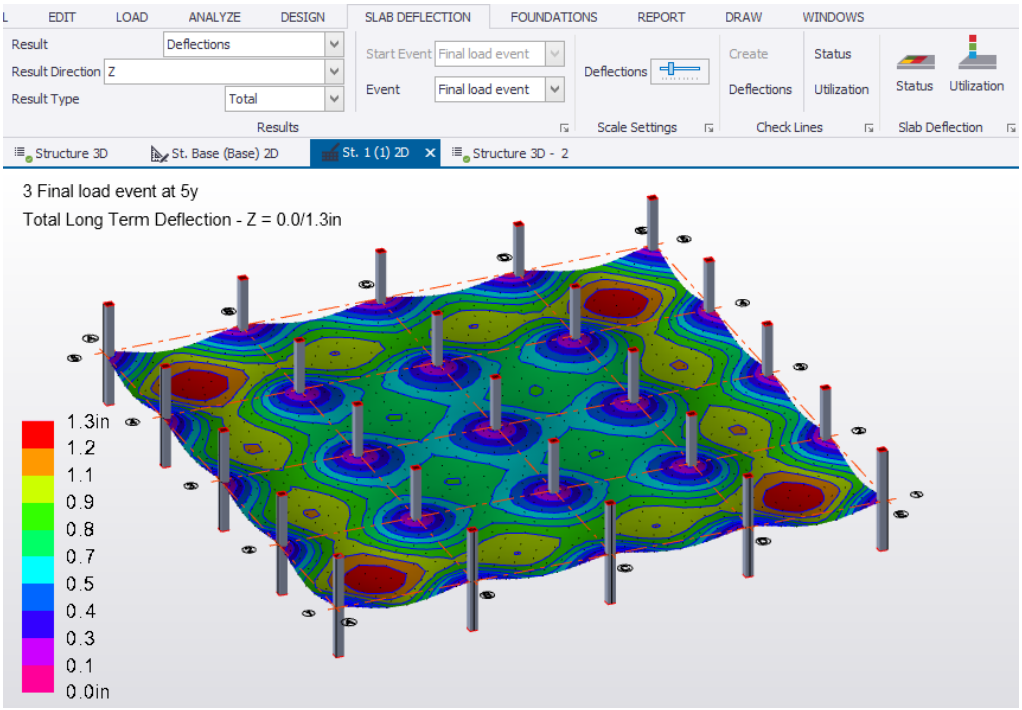
1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections for the events

   Deflections can be reviewed for each event by making selections from the Event droplist in the ribbon. You are able to review:

   - Total deflection at the end of any event.
   - Differential deflection between any two events (Start of Event and End of Event).
   - Instantaneous deflection.

   The Total deflection at the final load event for the chosen location is 1.3”



TIP If slab patches are obscuring the above deflection contours, these can be switched in Scene Content.

Consider the Sustained Load Multiplier Effect from ACI 435

ACI 435 Clause 4.3.4.2 needs to be reviewed carefully at this stage, as this suggests that the above deflection estimation is unconservative.

- It states that if the restraint stresses are expected to be **insignificant**, (so that the restraint constant is set at 7.5) then:
  - *"the multiplier for sustained-load deflection should be increased from 2 to 4, as recommended by Sbarounis (1984) and Graham and Scanlon [1986(b)]"*

Table 4.1 - Multipliers recommended by different authors

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	**7.5 √f_c'**	**1.0**	**2.8**	**1.2**	**5.0**
Branson (1977)	7.5 √f_c'	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	**7.5 √f_c'** 4 √f_c'	**1.0** 1.0	**2.0** 1.5	**2.0** 1.0	**5.0** 3.5
ACI Code	7.5 √f_c'	1.0	2.0		3.0

Hence the combined creep and shrinkage contribution = 4.0/5.0 = **0.8** (i.e. 80% of the total deflection).

- Alternatively, if the restraint stresses are likely to have a **significant** effect then Clause 4.3.4.2 states that:
  - a reduced restraint constant of 4 can be used, along with a long-term sustained-load multiplier of 2.5.

Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
Sbarounis (1984)	7.5 √f_c'	1.0	2.8	1.2	5.0
Branson (1977)	7.5 √f_c'	1.0	2.0	1.0 1.0	4.0
Graham and Scanlon (1986b)	7.5 √f_c' **4 √f_c'**	1.0 1.0	2.0 **1.5**	2.0 **1.0**	5.0 **3.5**

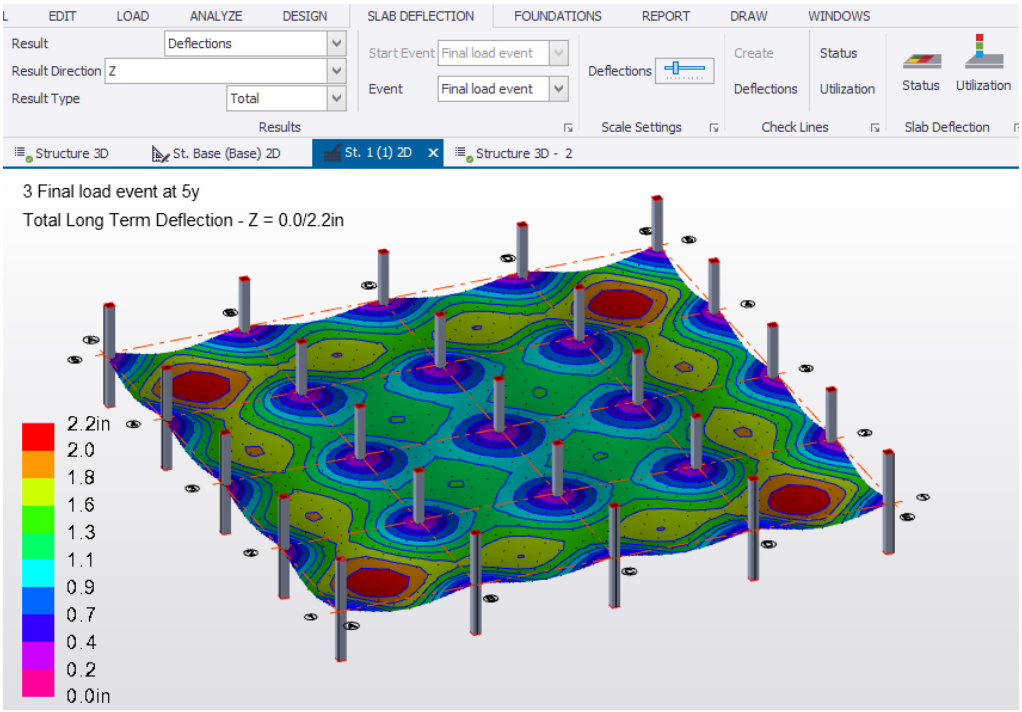
Source	Modulus of rupture, psi	Immediate	Creep λ_c	Shrinkage λ_{sh}	Total λ_t
ACI Code	7.5 $\sqrt{f_c'}$	1.0	2.0		3.0

In this case the combined creep and shrinkage contribution should be reduced to 2.5/3.5 = **0.714** (i.e. 71.4% of the total deflection).

1. **To adopt the ACI 435 recommendation for insignificant restraint stresses:**

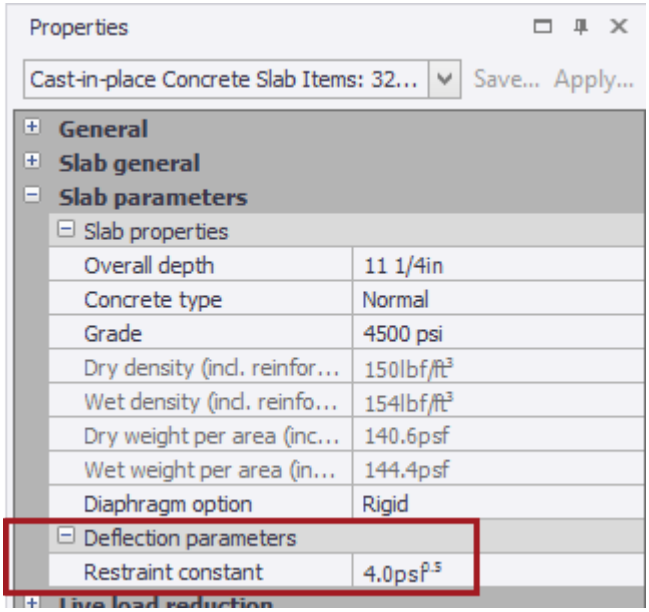
1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Increase the **Allowance for shrinkage effects in total deflection** to **0.8** then click **OK**
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   Using this value, the revised deflection prediction increases to 2.2”



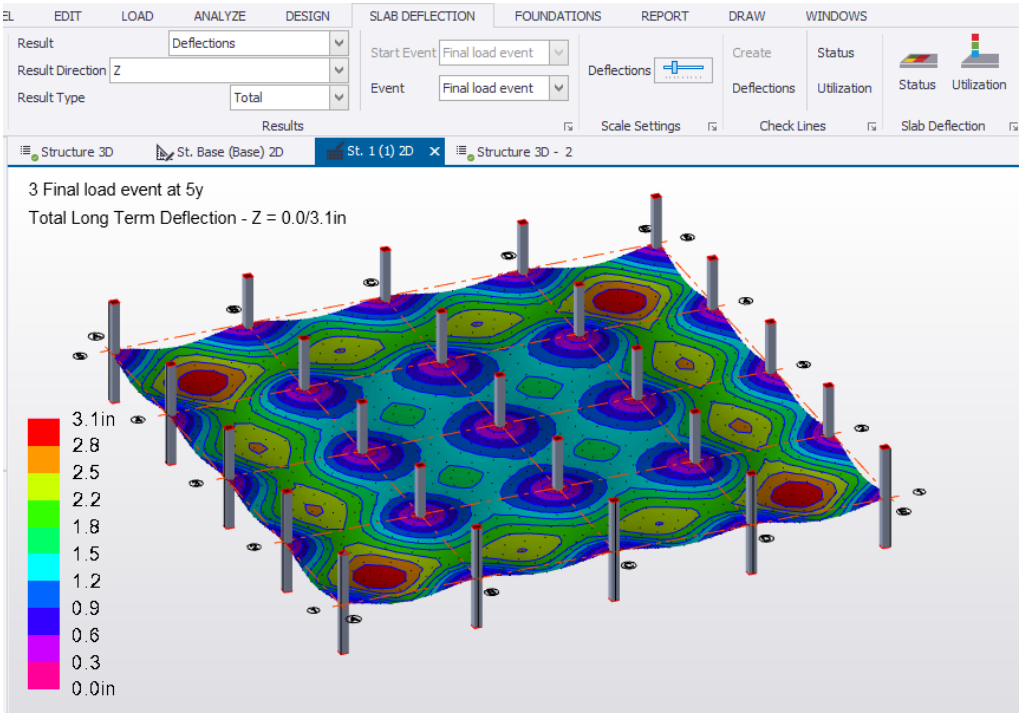
2. **To adopt the ACI 435 recommendation when restraint stresses are expected to be significant:**

5. Open the **Structure** 3D view
6. Select all the slabs in the model and via the Properties Window, adjust the **Restraint Constant** to **4.0**



7. From the **Slab Deflection** toolbar, click **Settings**
8. In the dialog, click **Aging, Creep & Shrinkage**
9. Change the **Allowance for shrinkage effects in total deflection** to **0.714** then click **OK**
10. Return to the **St.1 (1)** 2D view.
11. From the **Slab Deflection** toolbar, click **Analyze Current**

Using this value, the revised deflection prediction increases further to 3.1”

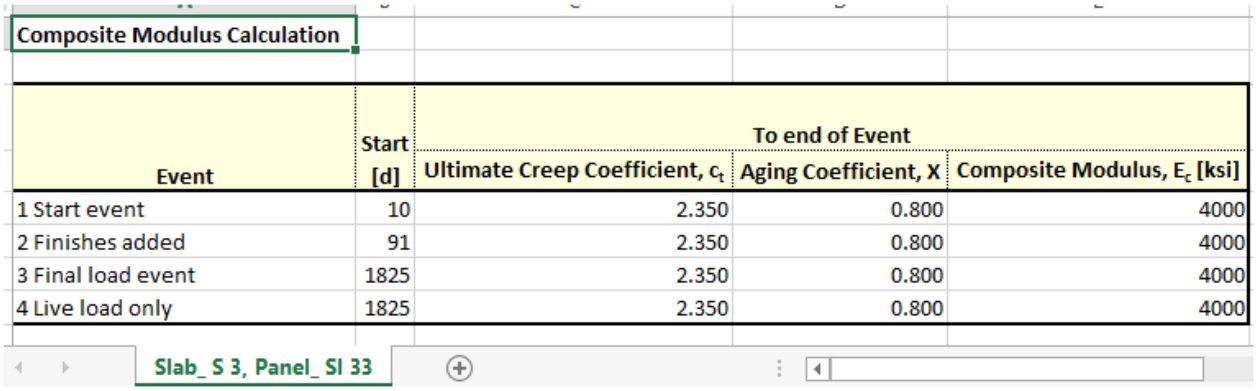


Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel > For the current slab item**



Composite Modulus Calculation

Event	Start [d]	To end of Event		
		Ultimate Creep Coefficient, c_t	Aging Coefficient, X	Composite Modulus, E_c [ksi]
1 Start event	10	2.350	0.800	4000
2 Finishes added	91	2.350	0.800	4000
3 Final load event	1825	2.350	0.800	4000
4 Live load only	1825	2.350	0.800	4000

Slab_ S 3, Panel_ SI 33

We can see from the report that the short term composite modulus, E_c is used.

Summary of Results

Using the Simplified event sequence + simplified combined creep and shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Assumed Creep and Shrinkage combined allowance %	Total deflection (Final load event)
ACI 318 (ignoring ACI 435)	7.5	66%	1.3”
ACI 435 (simple approach - insignificant restraint)	7.5	80%	2.2”
ACI 435 (simple approach - significant restraint)	4.0	71.4%	3.1”

Next steps

- In Method 2 (page 804) we will re-use the same model, but edit the simplified event sequence so that creep is considered in the cracked section analysis. The multiplier will also be edited so that it only considers shrinkage effects.
- In Method 3 (page 810) a modified version of the model using a detailed event sequence is investigated.
- Having obtained results for all three methods, observations on the different methods (page 817) are discussed.

- Finally, using results from one of the methods, deflection checks are performed using check lines (page 822) and an output report is generated.

Method 2: Simplified Event Sequence + Rigorous Creep and Shrinkage Allowance

In Method 1 (page 793) we used a simplified event sequence (without addressing the possibility of early age loading), to see how a single multiplier can be applied to allow for the combined effects of creep and shrinkage.

In **Method 2** we will re-use the same model, but edit the previous simplified event sequence so that creep is considered in the cracked section analysis. The multiplier will also be edited so that it only considers shrinkage effects.

Download and open the tutorial model (if required)

NOTE If you are reusing the model from the Method 1 (page 793) exercise you can skip this step.

---

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI - Simplified Event Sequence.tsmd

Establish some slab reinforcement

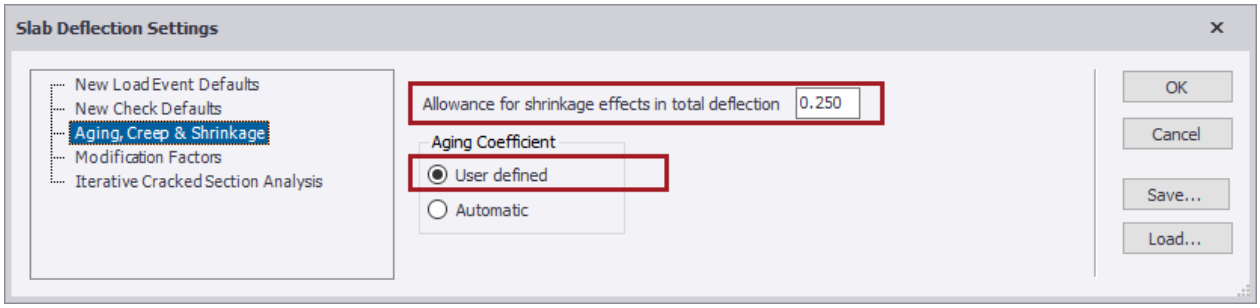
Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Set up the rigorous creep and shrinkage allowance

In this method the **Allowance for Shrinkage Effects** multiplier is set to allow for shrinkage only.

1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Aging Coefficient** is set to **User defined** and the **Allowance for shrinkage effects in total deflection** is set to **0.25**.



The above factor allows for shrinkage only.

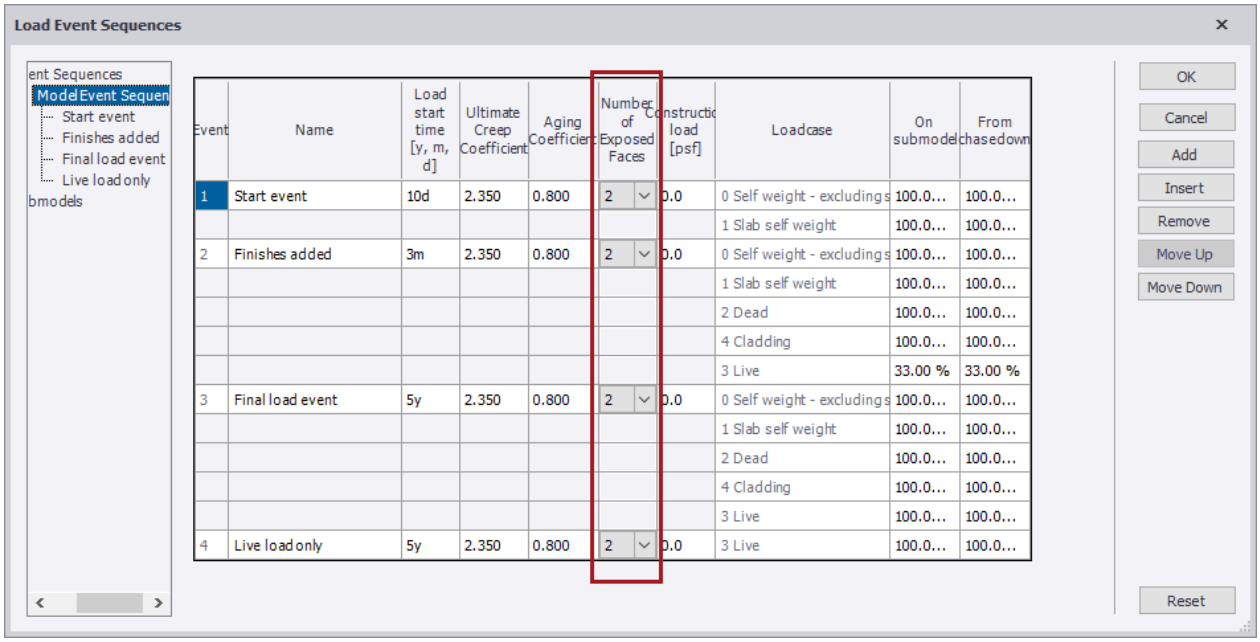
Refer to Shrinkage allowance (page 747) for an explanation of where this value comes from.

4. Click **OK** to close the dialog.

Edit the Event Sequence to ensure that creep is accounted for in the analysis

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**
3. Ensure that the **Number of Exposed Faces** is set to **2** for each event.

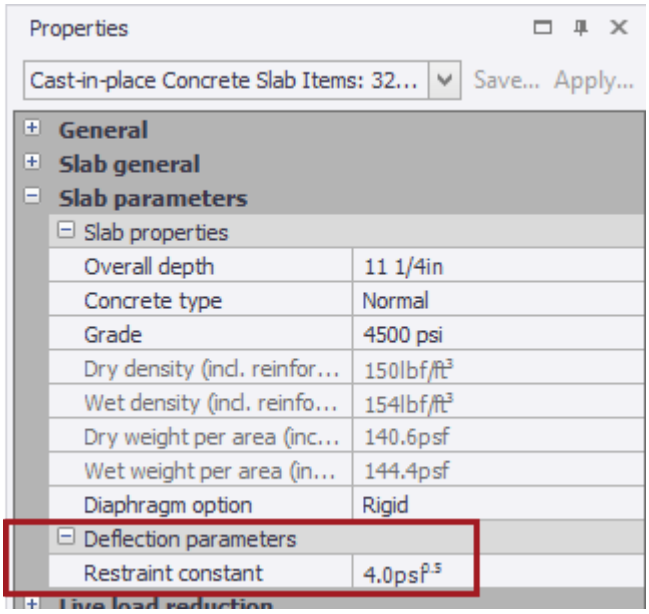
   The creep coefficient C_t will be calculated based on this, thus affecting the modulus of elasticity used in the analysis. For further details of this calculation, see: Aging Coefficient - User Defined or Automatic? (page 792)



Review the Restraint Constant

1.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, ensure the **Restraint Constant** is set to **4.0**

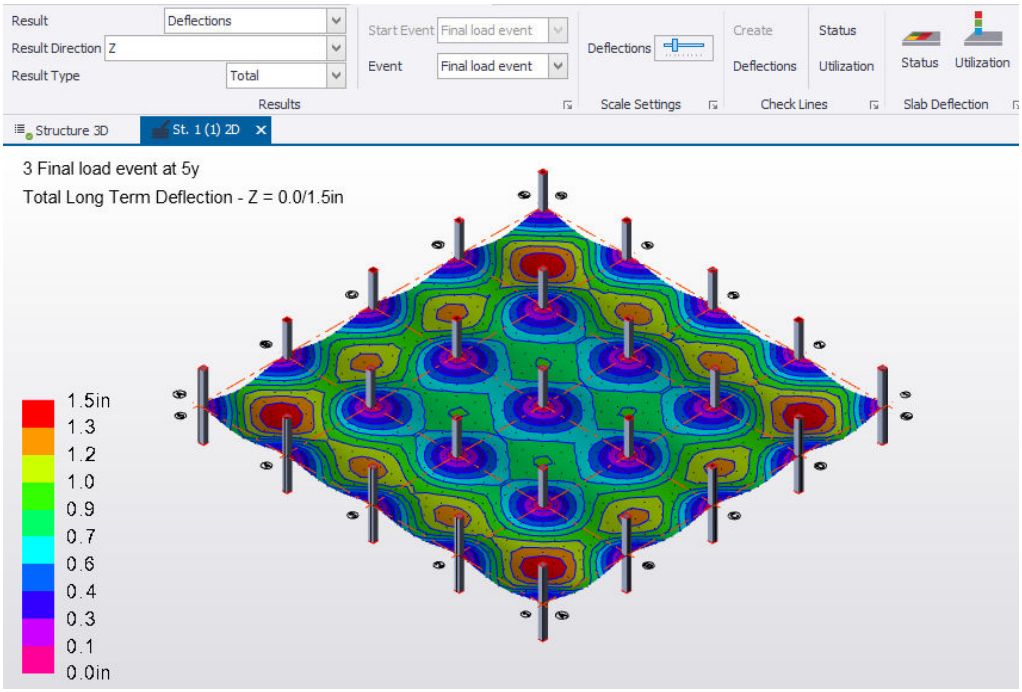


Perform Iterative Slab Deflection Analysis

1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections.

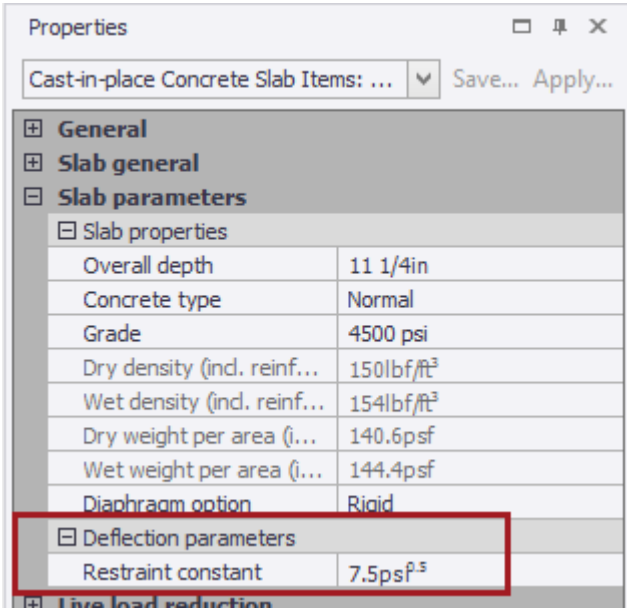
The predicted deflection estimate is 1.5” (using a Restraint Constant of 4.0).



Adjust the Restraint Constant and Re-analyze

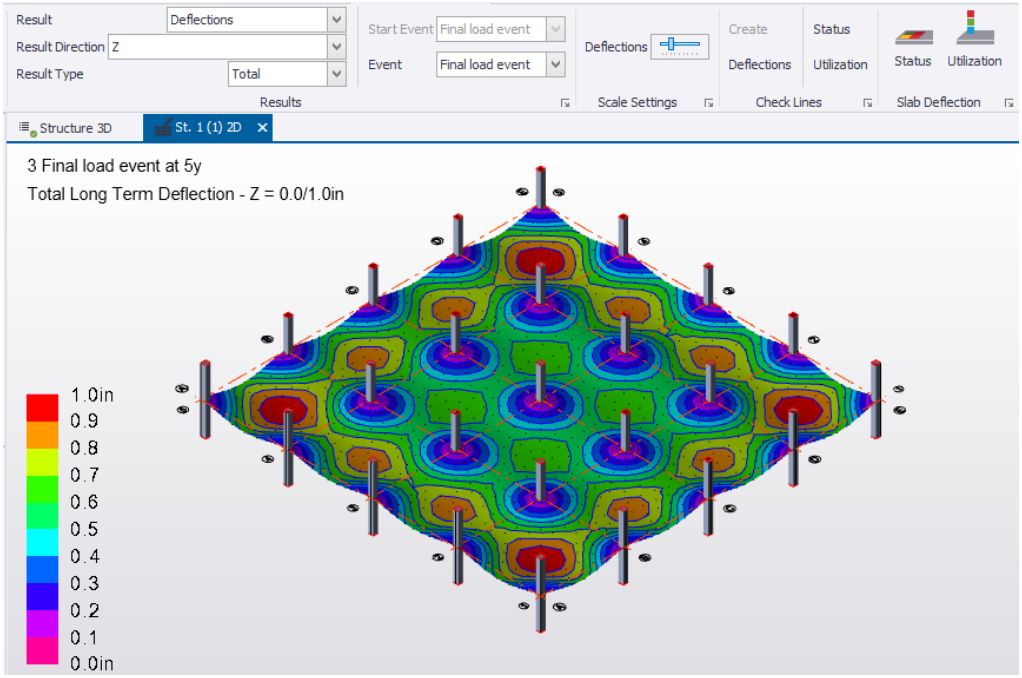
Initially the Restraint Constant was set assuming significant restraint; we will now investigate the effect of assuming insignificant restraint.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, change the **Restraint Constant** to **7.5**



3. Return to the **St.1 (1)** 2D view.
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   With these settings the total deflection predicted at 5 years reduces to 1”



Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel>For the current slab item**

Composite Modulus Calculation

Event	Start [d]	To end of Event			
		Ultimate Creep Coefficient, c_u	c_t	Aging Coefficient, X	Composite Modulus, E_c [ksi]
1 Start event	10	2.350	0.684	0.800	2585
2 Finishes added	91	2.350	2.119	0.800	1484
3 Final load event	1825	2.350	2.119	0.800	1484
4 Live load only	1825	2.350	2.119	0.800	1484

Slab_ S 3, Panel_ SI 33

Comparing this report to the report obtained when using the simplified creep and shrinkage allowance (page 802), we can see a difference in the value of the Composite Modulus E_c used. In this method, the age of concrete is taken into account.

To verify the result, consider Event 2, 3 and 4 where the event ends at 1825 days:

- Start event t_0 = 10 days, Event being considered, t_i = 1825 days
- Time between events (t_i-t_0) = 1825-10 = 1815 days
- Modulus of Elasticity, E for 4500 psi concrete grade = 4000 ksi (from Material database)
- Assumed Ultimate Creep Coefficient, C_u = 2.35
- Assumed Aging coefficient χ = 0.8. This is typically in the range 0.7 to 0.9
- Slab depth, d = 11 ¼ in
- Number of exposed faces, h = 2.

  $C_t = [(t_i-t_0) / (26\ e^{(0.36 \times d/h)}) + (t_i-t_0)] \times C_{ui} = [1815 / (26\ e^{(0.36 \times 11\ ¼ / 2)} + 1815] \times 2.35 = 2.1199$

  Thus

  $\bar{E}_c = (t, t_0) = E_c\ (t, t_0) / (1 + \chi\ Ct)$

  For event 2, 3 and 4, Composite Modulus, $E_c = 4000 / (1 + 0.8 \times 2.1199) = 1484$ ksi

Summary of Results

Using the Simplified event sequence plus rigorous creep plus shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = 0.8	25%	1.5”
ACI 435 (creep included in analysis)	7.5	C_u = 2.35, h = 2 and χ = 0.8	25%	1.0”

Next steps

- In Method 3 (page 810) a modified version of the model using a detailed event sequence is investigated.
- Having obtained results for all three methods, observations on the different methods (page 817) are discussed.
- Finally, using results from one of the methods, deflection checks are performed using check lines (page 822) and an output report is generated.

Method 3: Detailed Event Sequence + Rigorous Creep and Shrinkage Allowance

In this approach, a detailed event sequence has been defined which includes propping loads at early events to create an overall load history. Creep is considered rigorously alongside this load history by setting the Aging coefficient to Automatic. This means that an effective composite modulus value to the end of each event is calculated in accordance with the Concrete Society Technical Report 58.

NOTE The aging coefficient method should only be set to Automatic when defining a detailed event sequence that includes a realistic assessment of early propping load events.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:

- Slab deflection ACI - Detailed Event Sequence.tsmd

Establish some slab reinforcement

Prior to running a Slab Deflection Analysis, a reasonable level of slab reinforcement should already be provided. This can be achieved by designing all slabs and patches as follows:

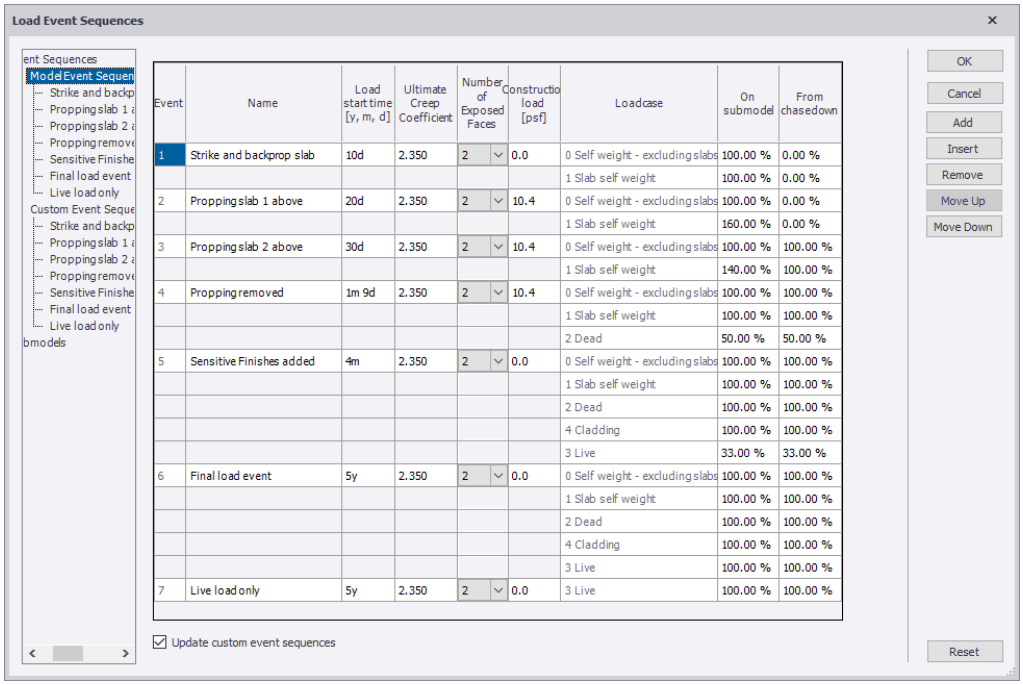
1. From the **Analyze** toolbar, click **Analyze All (Static)**
2. From the **Design** toolbar, click **Design Slabs**
3. From the **Design** toolbar, click **Design Patches**

Review the Detailed Event Sequence

1. **To display the Event Sequence**

1. From the **Slab Deflection** toolbar, click **Event Sequences**
2. Click **Model Event Sequence**

   A detailed event sequence has already been created:



Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	2.350	2	0.0	0 Self weight - excluding slabs	100.00 %	0.00 %
						1 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	0.00 %
						1 Slab self weight	160.00 %	0.00 %
3	Propping slab 2 above	30d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	140.00 %	100.00 %
4	Propping removed	1m 9d	2.350	2	10.4	0 Self weight - excluding slabs	100.00 %	100.00 %
						1 Slab self weight	100.00 %	100.00 %
						2 Dead	50.00 %	50.00 %
5	Sensitive Finishes added	4m	2.350	2	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 %
6	Final load event	5y	2.350	2	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 %
7	Live load only	5y	2.350	2	0.0	3 Live	100.00 %	100.00 %

You will note that:

- Ultimate Creep Coefficient, C_u set at **2.35** for this example
- Number of exposed faces, h set at **2** for this example
- Construction load assumed set at **10.4 psf** in this example
- Propping load % assumed during early age propping events **160%** and **140%** assumed.

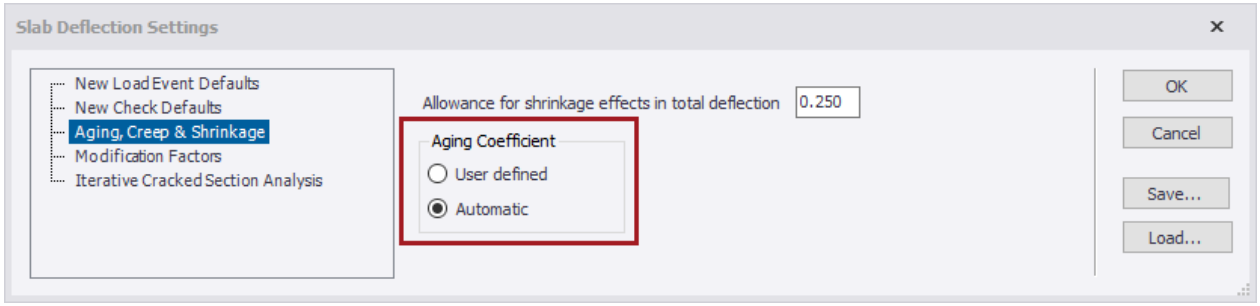
Set up Rigorous Creep

This method requires that the **Aging coefficient** is set to Automatic. Creep is then considered automatically as part of the analysis for each event sequence.

A user defined allowance for Shrinkage effects in the total deflection is also specified.

1. **To specify the multiplier for shrinkage effects only but also consider creep in the analysis**

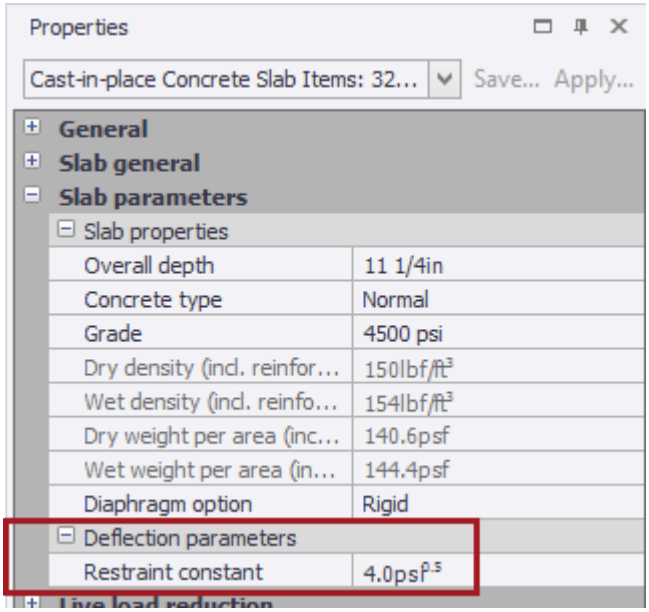
1. From the **Slab Deflection** toolbar, click **Settings**
2. In the dialog, click **Aging, Creep & Shrinkage**
3. Ensure the **Allowance for shrinkage effects in total deflection** is set to **0.25**, and the **Aging Coefficient** is set to **Automatic**



Review the Restraint Constant

1.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, ensure the **Restraint Constant** is set to **4.0**

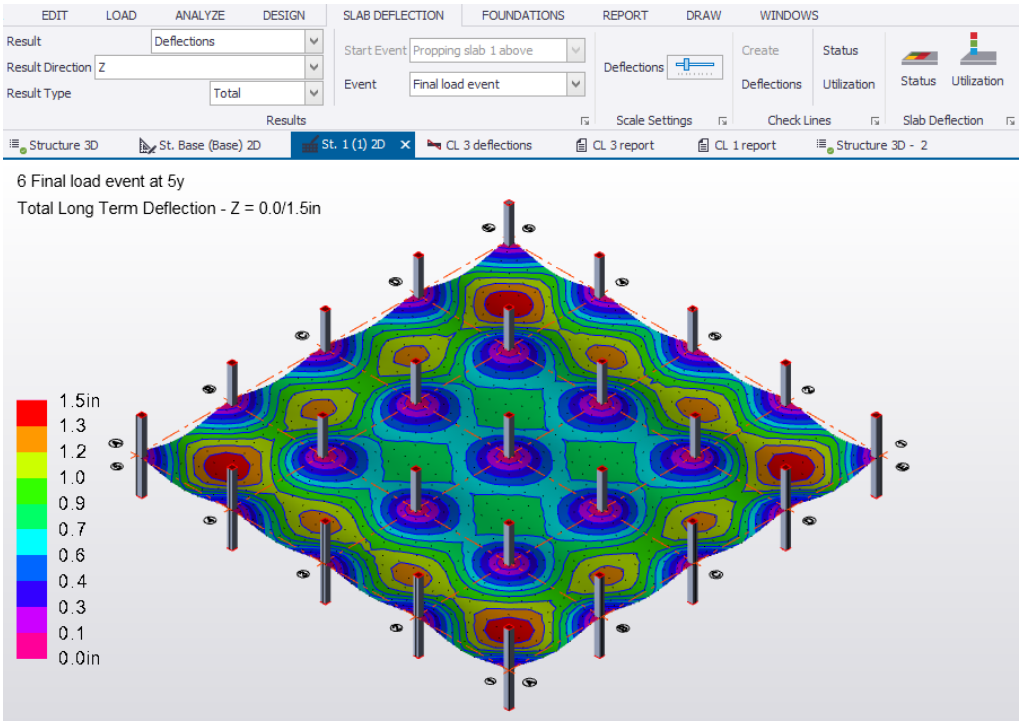


Perform Iterative Slab Deflection Analysis

1. Open a **St.1 (1)** 2D view.
2. From the **Slab Deflection** toolbar, click **Analyze Current**

   After analysis the current view automatically switches into the Slab Deflections View regime.
3. Review the deflections.

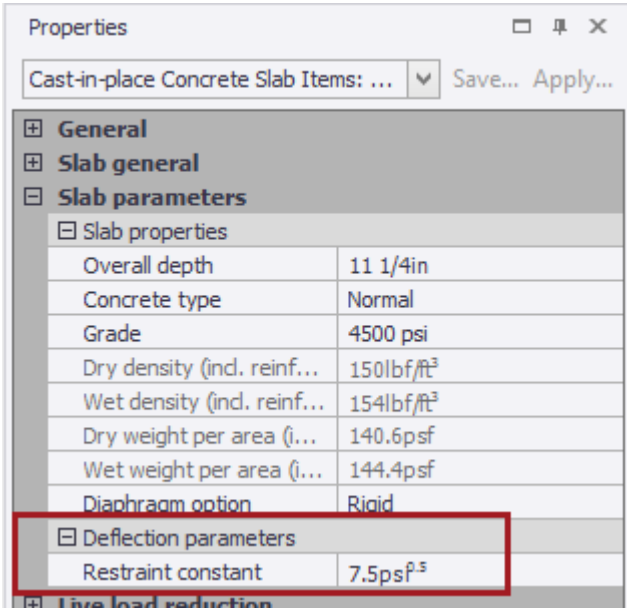
The predicted deflection estimate is 1.5” (using a Restraint Constant of 4.0).



Adjust the Restraint Constant and Re-analyze

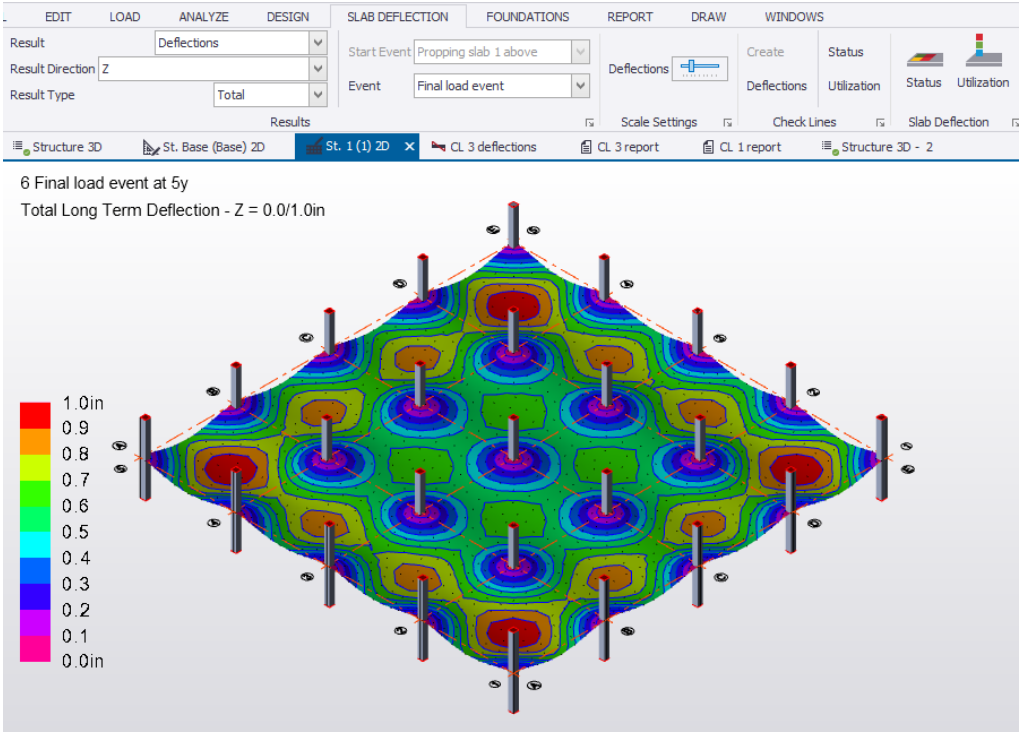
Initially the Restraint Constant was set assuming significant restraint; we will now investigate the effect of assuming insignificant restraint.

1. Open the **Structure** 3D view.
2. Select all the slabs in the model and via the Properties Window, change the **Restraint Constant** to **7.5**



3. Return to the **St.1 (1)** 2D view.
4. From the **Slab Deflection** toolbar, click **Analyze Current**

   With these settings the total deflection predicted at 5 years reduces to 1”

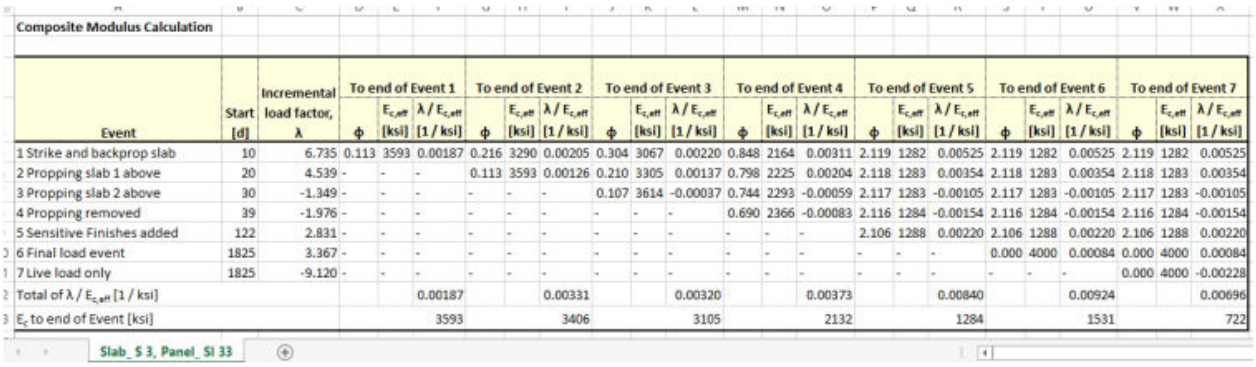


Generate Composite Modulus Report

An effective composite modulus report can be obtained by right clicking a slab and choosing **Export Eff. modulus report to Excel**. The report details the composite modulus E_c determined for each event.

To display the Composite Modulus Report:

1. Set the **Result** to **None** to display the slabs.
2. Right click the slab between gridline D-E/1-2 and **Export Eff Modulus report to Excel>For the current slab item**



Composite Modulus Calculation

Event	Start [d]	Incremental load factor, λ	To end of Event 1			To end of Event 2			To end of Event 3			To end of Event 4			To end of Event 5			To end of Event 6			To end of Event 7		
			ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]	ϕ	$E_{c,eff}$ [ksi]	λ / $E_{c,eff}$ [1 / ksi]
1 Strike and backprop slab	10	6.735	0.113	3593	0.00187	0.216	3290	0.00205	0.304	3067	0.00220	0.848	2164	0.00311	2.119	1282	0.00525	2.119	1282	0.00525	2.119	1282	0.00525
2 Propping slab 1 above	20	4.539	-	-	-	0.113	3593	0.00126	0.210	3305	0.00137	0.798	2225	0.00204	2.118	1283	0.00354	2.118	1283	0.00354	2.118	1283	0.00354
3 Propping slab 2 above	30	-1.349	-	-	-	-	-	-	0.107	3614	-0.00037	0.744	2293	-0.00059	2.117	1283	-0.00105	2.117	1283	-0.00105	2.117	1283	-0.00105
4 Propping removed	39	-1.976	-	-	-	-	-	-	-	-	-	0.690	2366	-0.00083	2.116	1284	-0.00154	2.116	1284	-0.00154	2.116	1284	-0.00154
5 Sensitive Finishes added	122	2.831	-	-	-	-	-	-	-	-	-	-	-	-	2.106	1288	0.00220	2.106	1288	0.00220	2.106	1288	0.00220
6 Final load event	1825	3.367	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	0.000	4000	0.00084	0.000	4000	0.00084
7 Live load only	1825	-9.120	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	0.000	4000	-0.00228
Total of λ / $E_{c,eff}$ [1 / ksi]					0.00187			0.00331			0.00320			0.00373			0.00840			0.00924			0.00696
E_c to end of Event [ksi]					3593			3406			3105			2132			1284			1531			722

Slab_S 3, Panel_Sl 33

Comparing this report to the report for the more simplified ACI "rigorous" approach to creep (page 808) undertaken earlier.

- ACI composite modulus calculation for the Final Load Event = 1484 ksi (also verified by hand calculation earlier)
- TR58 composite modulus calculation for the Final Load Event = 1531 ksi
- These values compare favorably.

Note, however, that if we compared event 5 above where the composite modulus, E_c = 1284 ksi using the ACI method, we can establish the aging coefficient as:

$E_c = E / (1 + X * Ct)$

$1284 = 4000 / (1 + \chi \times 2.1199)$

$\chi = [(4000 / 1284) - 1] / 2.1199 = 0.9978$

This increased aging coefficient occurs due to the significant propping loads at early events, which creates an overall load history that is almost equivalent to constant loading.

Summary of Results

Using the Detailed event sequence plus rigorous creep and shrinkage allowance method, the results can be tabulated below:

Approach	Restraint Constant for Modulus of Rupture	Creep approach	Assumed Shrinkage %	Total deflection (Final load event)
ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = automatic based on TR58	25%	1.5″
ACI 435 (creep included in analysis)	7.5	C_u = 2.35, h = 2 and χ = automatic based on TR58	25%	1.0″

Next steps

- Having obtained results for all three methods, observations on the different methods (page 817) are discussed.
- Finally, using results from one of the methods, deflection checks are performed using check lines (page 822) and an output report is generated.

Observations on the Different Methods

Combined Table of Results

Approach		Restraint Constant for Modulus of Rupture	Creep approach	Assumed Creep and Shrinkage combined allowance %	Assumed Shrinkage %	Total deflection (Final load event)
Method 1: Simplified Event Sequence +	ACI 318 (ignoring ACI 435)	7.5		66%		1.3″
	ACI 435 (simple approach	7.5		80%		2.2″

Approach		Restraint Constant for Modulus of Rupture	Creep approach	Assumed Creep and Shrinkage combined allowance %	Assumed Shrinkage %	Total deflection (Final load event)
Simplified Combined Creep and Shrinkage Allowance	- insignificant restraint)					
	ACI 435 (simple approach - significant restraint)	4.0		71.4%		3.1”
Method 2: Simplified Event Sequence + Rigorous Creep + Shrinkage Allowance	**ACI 435 (creep included in analysis)**	**4.0**	**C_u = 2.35, h = 2 and χ = 0.8**		**25%**	**1.5”**
		7.5	C_u = 2.35, h = 2 and χ = 0.8		25%	1.0”
Method 3: Detailed Event Sequence + Rigorous Creep + Shrinkage Allowance	ACI 435 (creep included in analysis)	4.0	C_u = 2.35, h = 2 and χ = automatic based on TR58		25%	1.5”
		7.5	C_u = 2.35, h = 2 and χ = automatic based on TR58		25%	1.0”

Discussion

When the guidance in ACI 435 about the use of a combined allowance for creep and shrinkage is taken into account, the simplified approach using a combined creep and shrinkage multiplier seems to determine very conservative deflection estimations.

Looking at table 4.1 in ACI 435, it is clear that the creep contribution is the most significant part of the overall deflection estimate. It is also the area where there seems to be greatest variation in opinion on the contribution level.

The creep contribution can be dealt with more rigorously by including it in the analysis. So rather than using a short term E value and then amplifying the result to account for creep, the analysis at the end of each event uses an effective E value which includes for creep up to that point. This impacts on the analysis properties of every shell in the FE analysis and even on the extent of cracking.

It is suggested that when the creep is included in the analysis the use of a modulus of rupture which assumes low restraint (restraint constant = 7.5) be used cautiously.

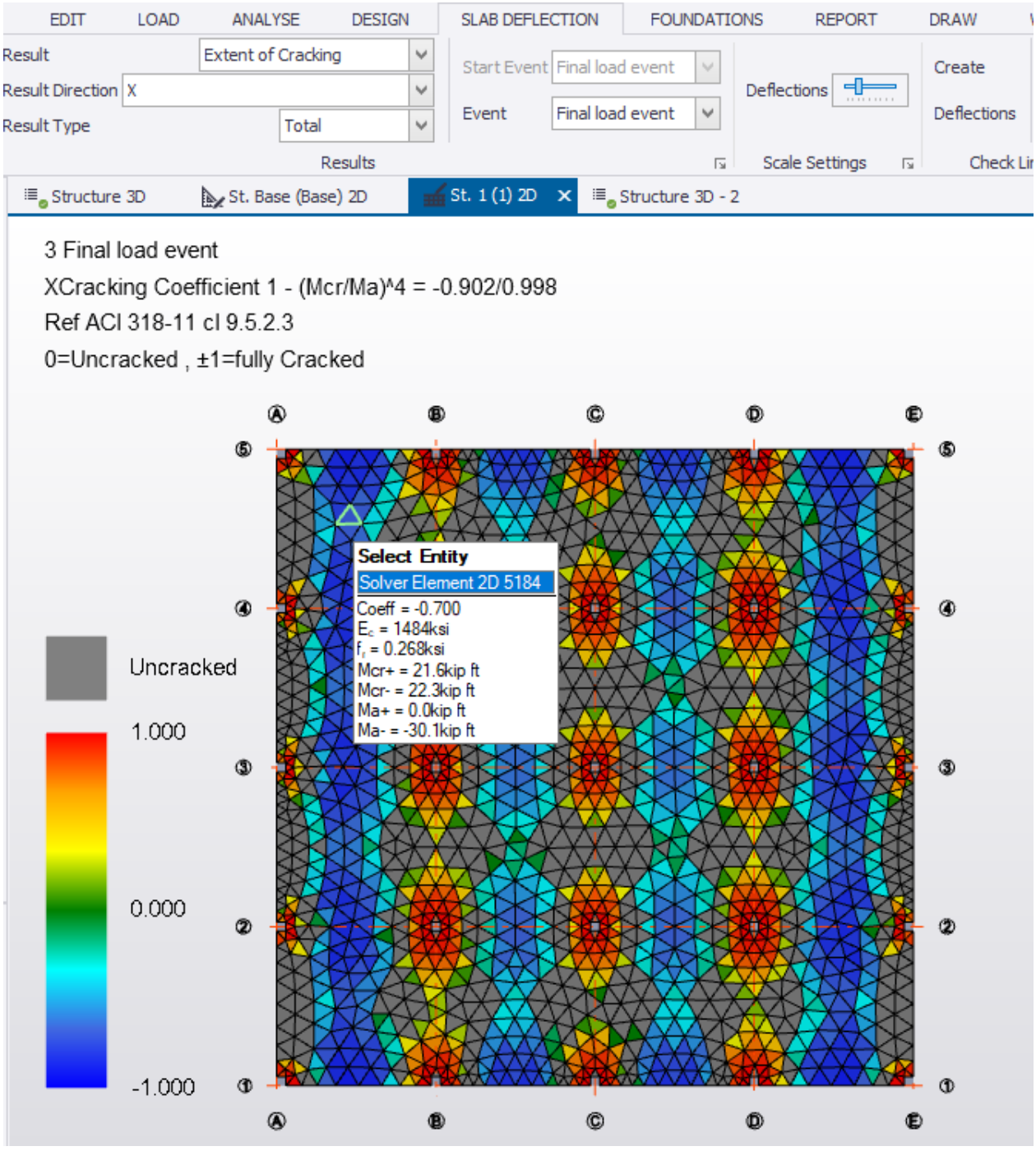
The option to calculate the effective E value based on some UK guidance in TR58 makes little difference in this example. It should be borne in mind that the aging coefficient χ is user defined and 0.8 is just a default for “normal” situations. In situations such as a transfer slab where load will accrue more slowly over time 0.8 is likely to be conservative and it may be of interest to consider the TR58 alternative.

Overall, it is expected that most engineers would use the settings highlighted in bold in the table above as a default starting point and then make refinements if necessary.

Extent of Cracking

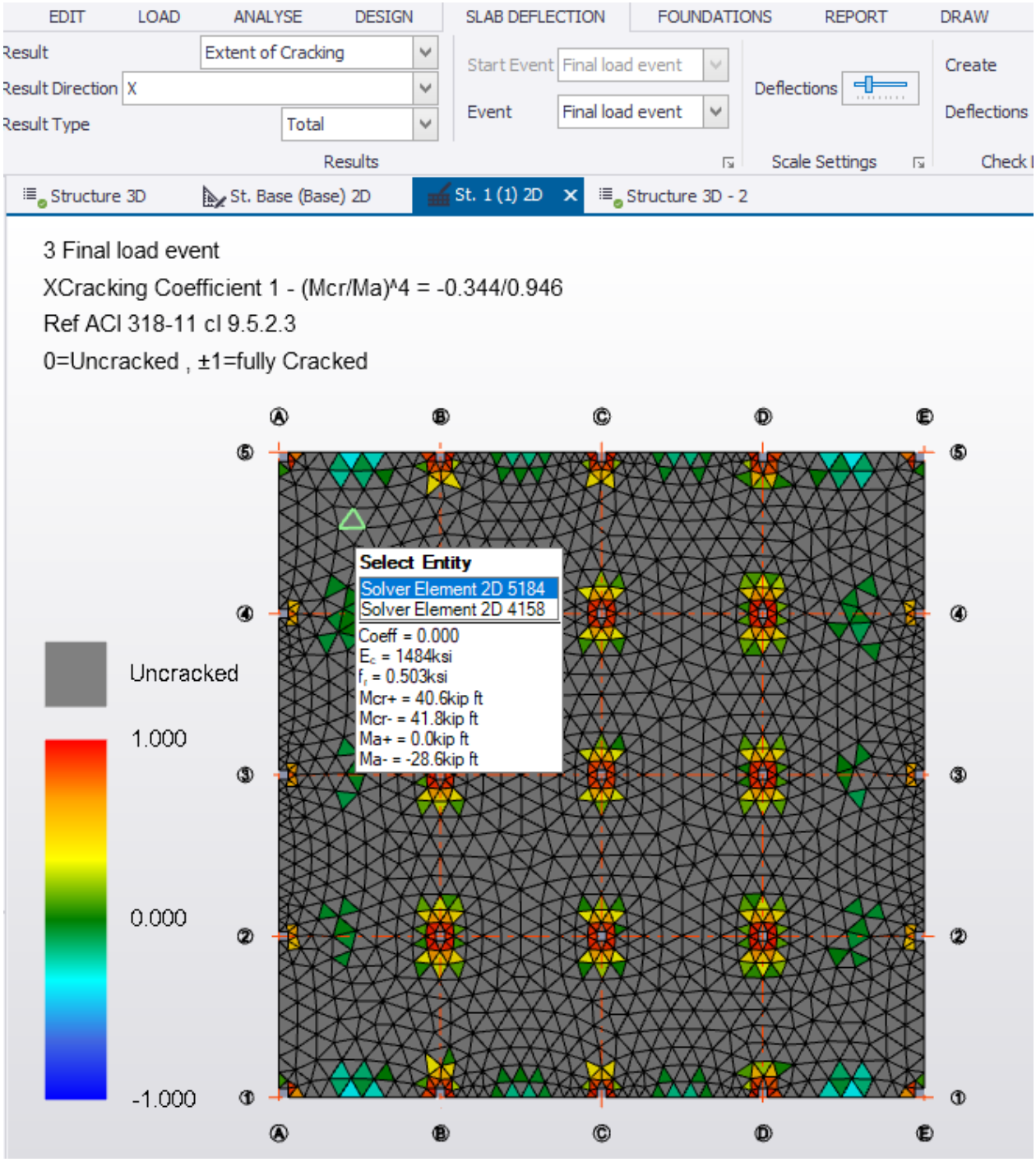
It is clear from the **Combined Table of Results**, that the Restraint Constant and hence the Modulus of Rupture has a huge impact on the predicted deflections. This is due to the impact the modulus of rupture has on the extent of cracking that develops. This is illustrated below where the extent of cracking for the final load event is displayed for the Method 2 model.

Extent of cracking resulting from a restraint constant of 4.0



You can clearly see the majority of the slabs have cracked where the restraint constant is 4.0.

Extent of cracking resulting from a restraint constant of 7.5



The majority of the slabs remain uncracked where the restraint constant is 7.5.

Next steps

- Using results from one of the methods, deflection checks are performed using check lines (page 822) and an output report is generated.

Use of check lines to check deflections (ACI)

Once the input parameters have been set as required to allow for creep and shrinkage, and the analysis has been performed, check lines can then be placed in order to check the deflections.

Download and open the tutorial model

1. Download the tutorial models from here.
2. Open the following tutorial model:
   - Slab deflection ACI - Check Lines.tsmd

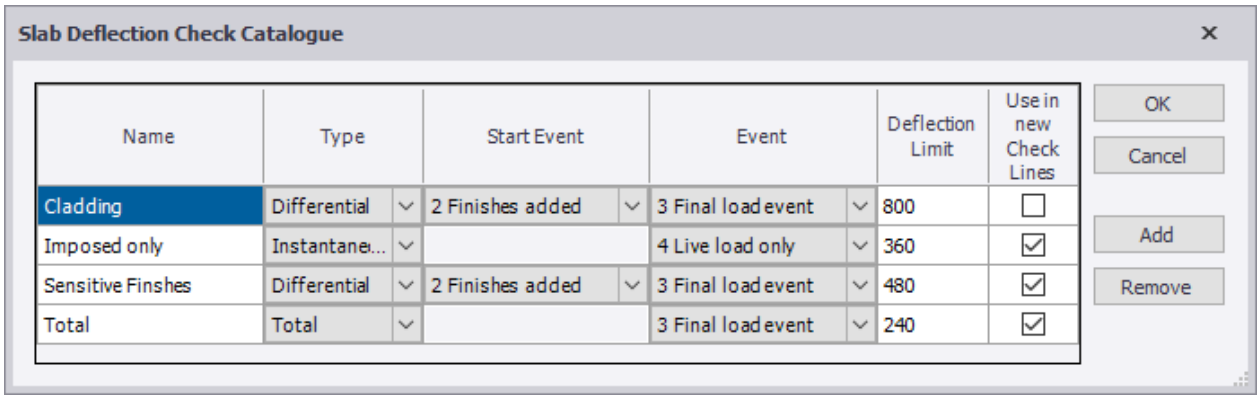
Define Check Line Deflection Checks

Check lines have to initially be positioned using engineering judgment.

The deflection checks associated with each check line are selected from a predefined Deflection Check Catalogue. This is viewed by clicking Deflection Checks in the ribbon.

You can add new checks to the catalogue as required.

1. From the Slab Deflection ribbon, click **Deflection Checks**



Whilst four checks have been defined above, only three of these have been set to be used in new Check Lines. The more onerous cladding check which does not come from code requirements is not applied by default:

- **Sensitive finishes** will check the differential deflections from when the sensitive finishes are applied to the final load event against a deflection limit of 1/480
- **Total** will check the total deflections to the final load event against a deflection limit of 1/240

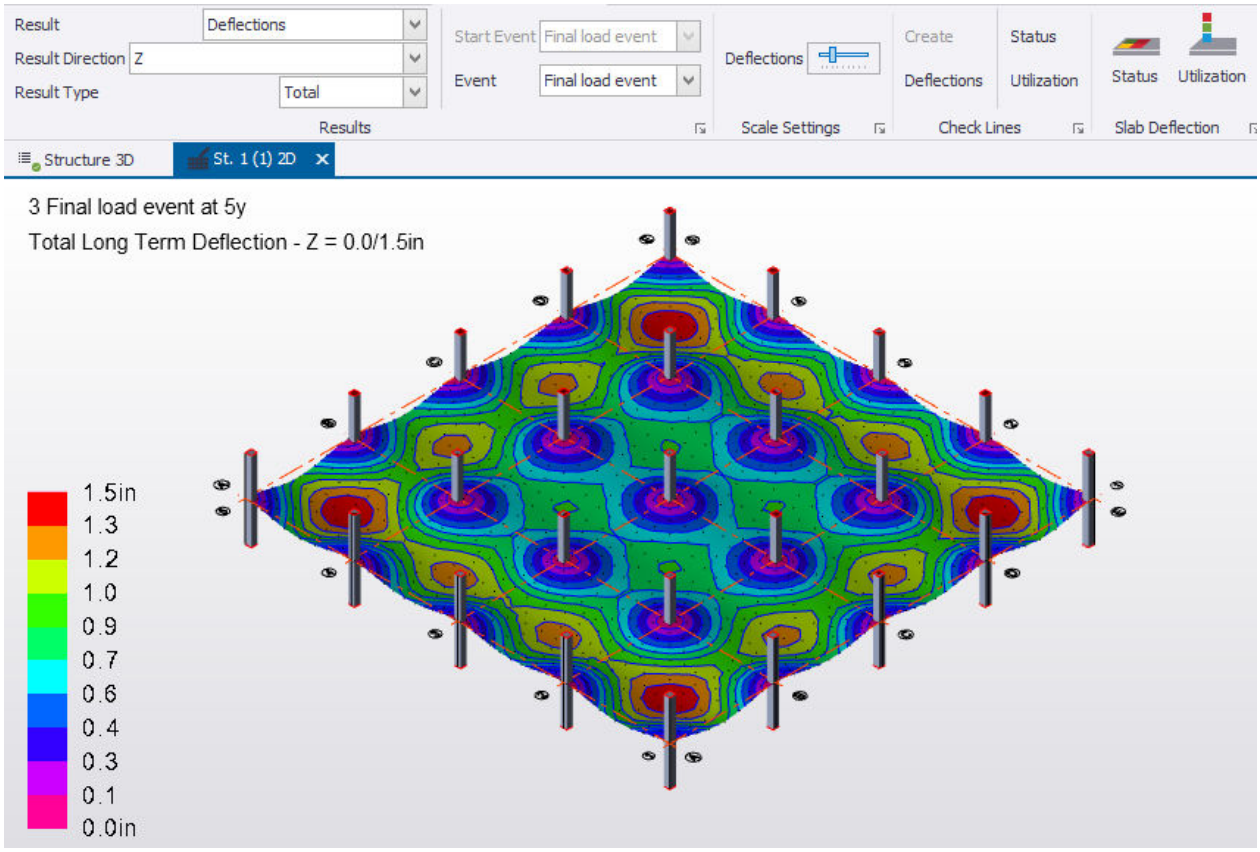
- **Live load only** will check the instantaneous deflections for the live load only event against a deflection limit of 1/360

2. Click **OK** to close the dialog.

Place Check Lines

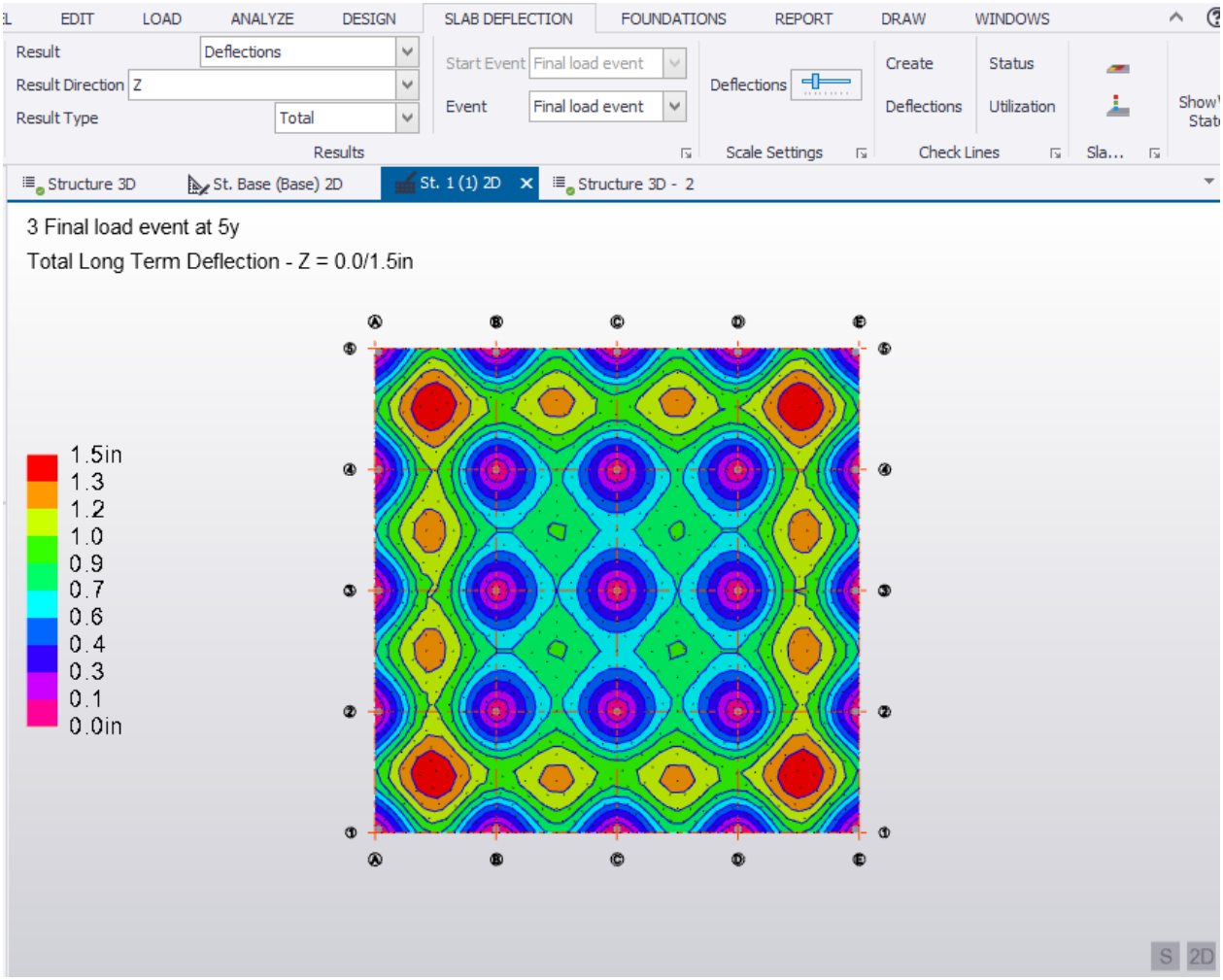
We will define two check lines in this example:

1. Open the **St.1 (1)** 2D plan view



NOTE When the 2D plan view is being viewed in 3D as above, the **Create** button on the **Slab Deflection** toolbar is grayed out.

2. Click 3D in the bottom right corner of the view to toggle the view until it is displayed in 2D.

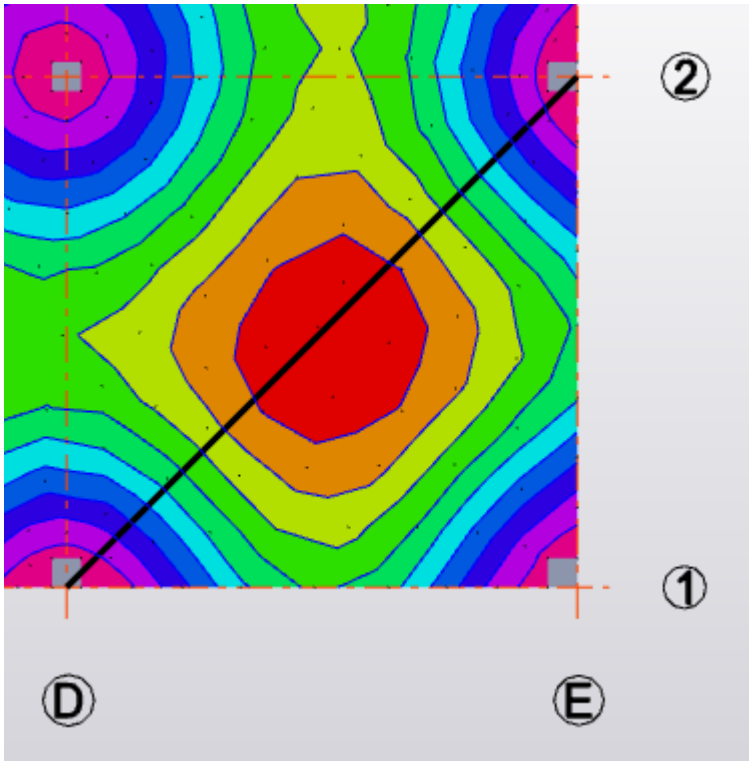


3. From the **Slab Deflection** toolbar, ensure the Results are set as above:
   a. Result - **Deflections**,
   b. Result Direction - **Z**,
   c. Result Type - **Total**,
   d. Event - **Final load event**
4. Click **Create**

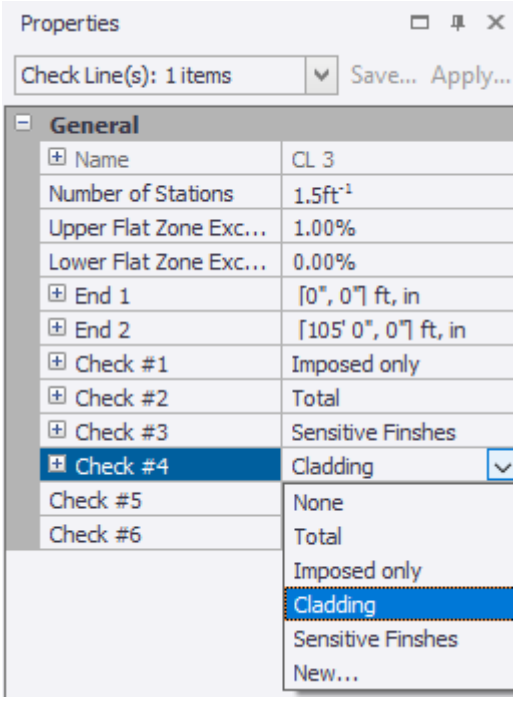
NOTE Check lines can only be created in a 2D view.

NOTE When you click Create, the Properties Window automatically includes the slab deflection checks from the catalogue for which "Use in new Check Lines" was checked.

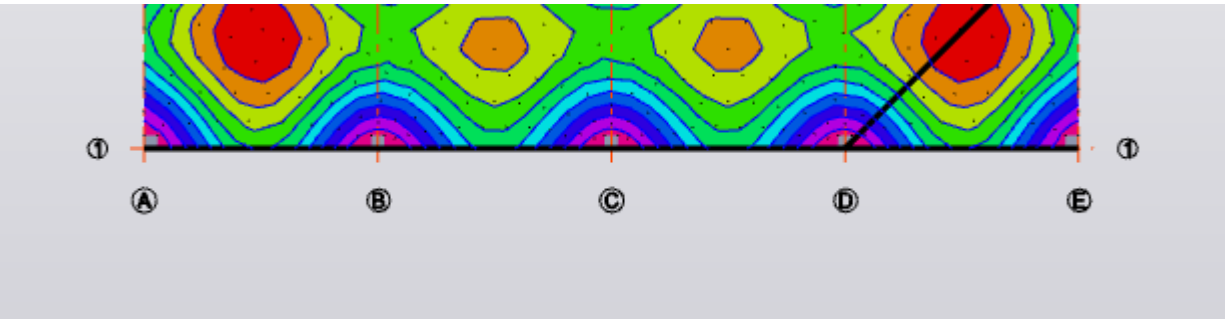
5. To place a check line running diagonally between columns in bottom right corner panel where the peak deflection occurs:
    a. Pick the start point as grid line intersection D/1.
    b. Pick the end point as grid line intersection E/2.



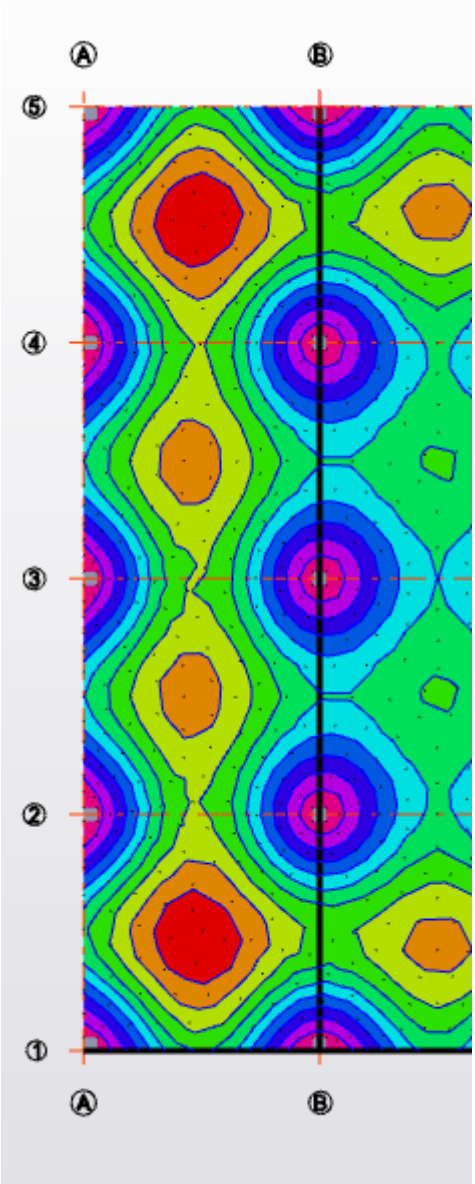
Before creating a second check line, in the Properties Window add a check #3, by selecting Cladding from the droplist.



6. Create the second check line along grid line 1.
   a. Pick the start point as grid line intersection A/1.
   b. Pick the end point as grid line intersection E/1.

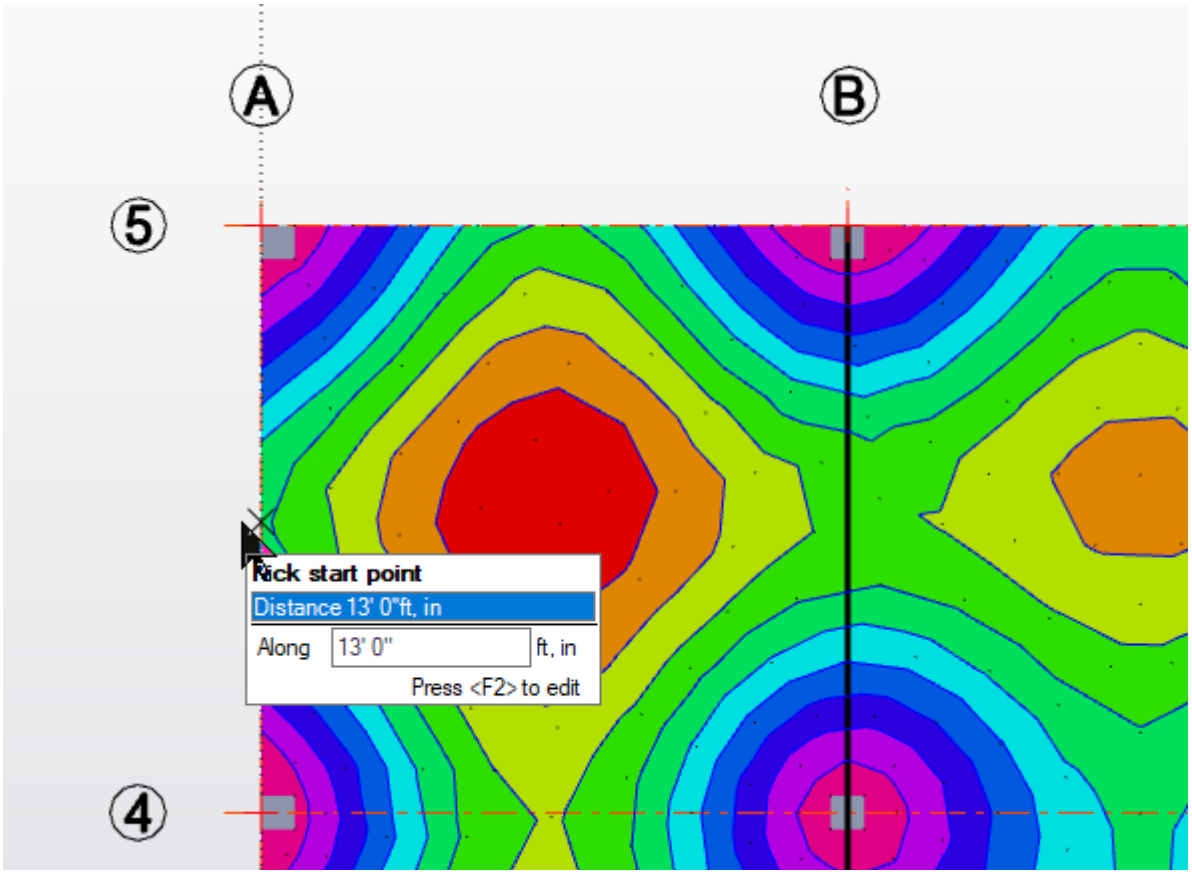


7. Create the third check line along grid line B.
   a. Pick the start point as grid line intersection B/1.
   b. Pick the end point as grid line intersection B/5.

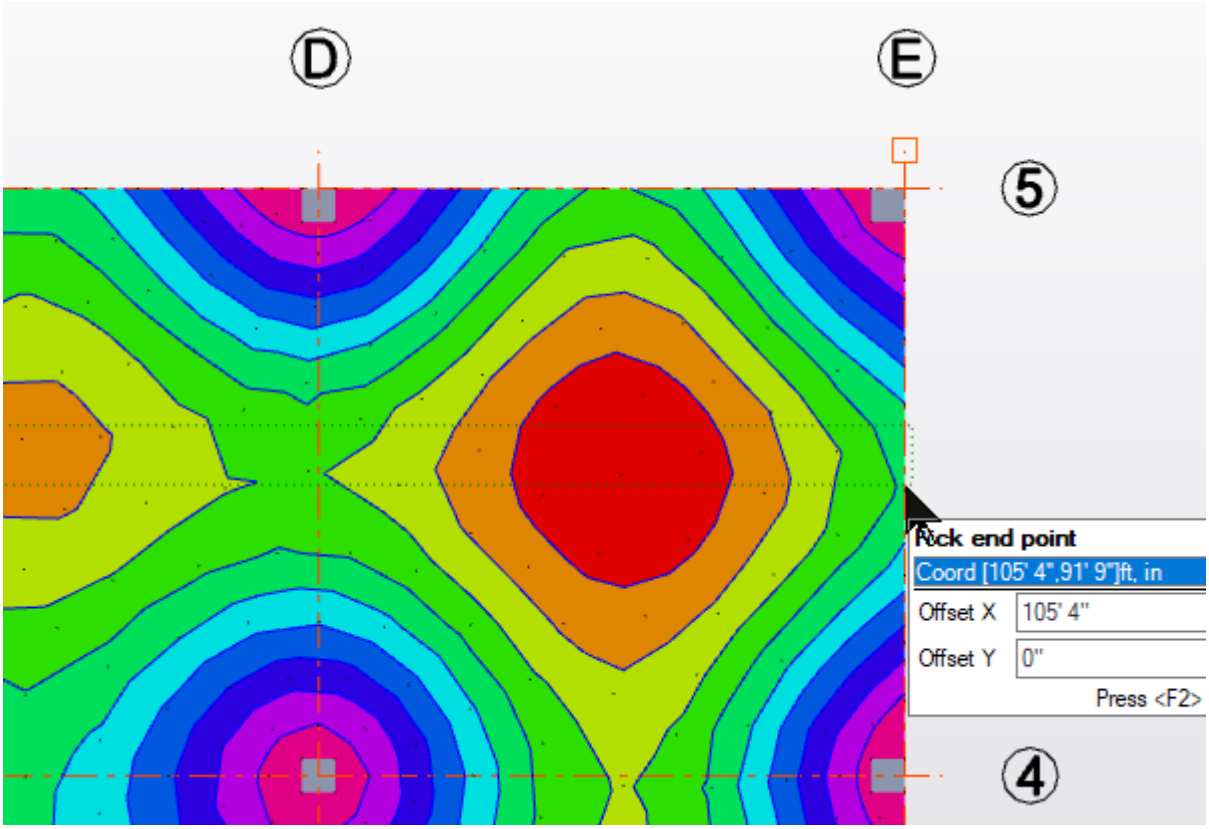


8. Create the final check line mid-way between grid lines 4-5 from grid line A to E.

    a. Pick the start point approximately as shown:



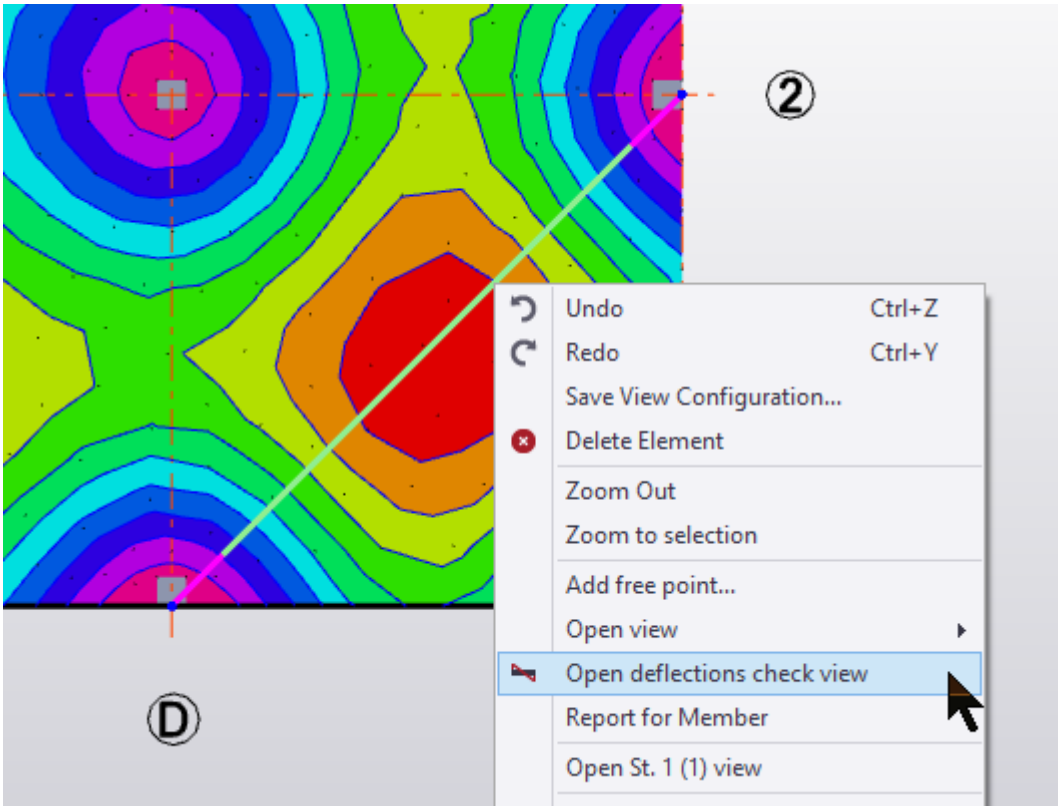
b. Pick the end point approximately as shown:

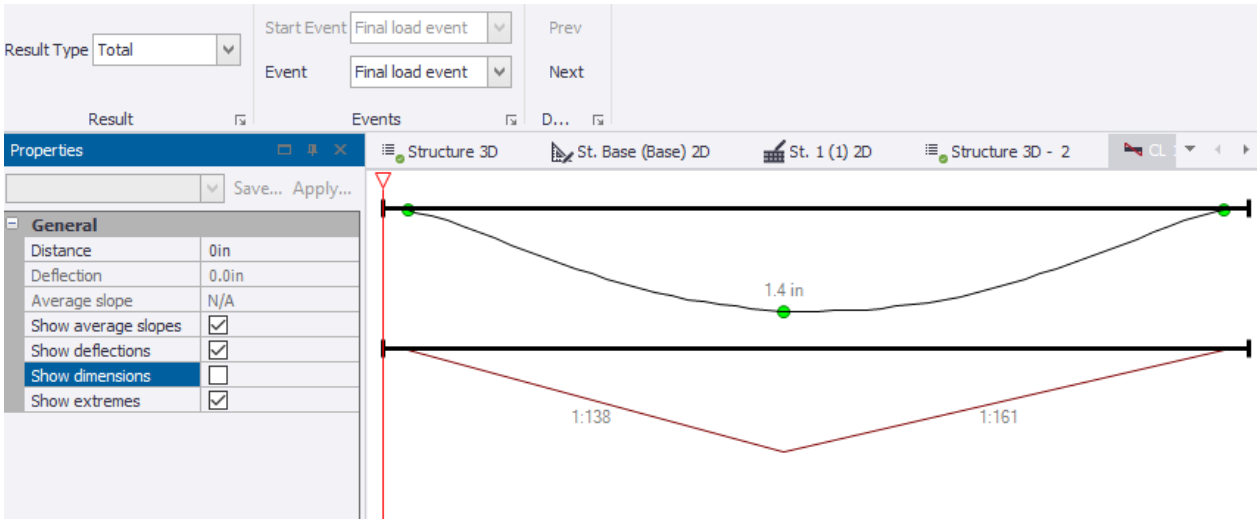


9. Press **Esc** to end the command.

NOTE In a real model you can add as many check lines to the model as you consider appropriate.

10. Right click on the diagonal check line and from the context menu select **Open deflections check view**.





The ribbon allows you to specify the total (as shown above), or differential or instantaneous results for the selected events. Tekla Structural Designer then draws average slopes between maximum and minimum points.

A total deflection limit of Span / 240 is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 120. In

the view above the average slope between these points is 1 / 138, so the check passes.

Doing the checks this way inherently deals with offset peak deflections (non-uniform load) and also cantilevers.

11. Click within the Deflection load analysis view and change the Result type to **Differential** and check deflection and slopes between the **Finishes added** and **Final load event**. The resulting differential deflection is as shown:



For the Sensitive Finishes check of differential deflection between finishes added and the final load limit had a limit of Span / 480, which is the same as saying that the average slope between points of maximum and minimum deflection = 1 / 240. In the view above the average slope between these points is 1 / 194, so this check fails.

Generate Check Line Reports

A tabulated report is available for each check line which itemizes each requested deflection check along the check line.

You can generate a check line report for an individual check line by right clicking the check line and selecting Member Report.

1. Return to the **St.1 (1) 2D** view, right click on the diagonal check line and select **Report for Member**

Deflection Checks Summary

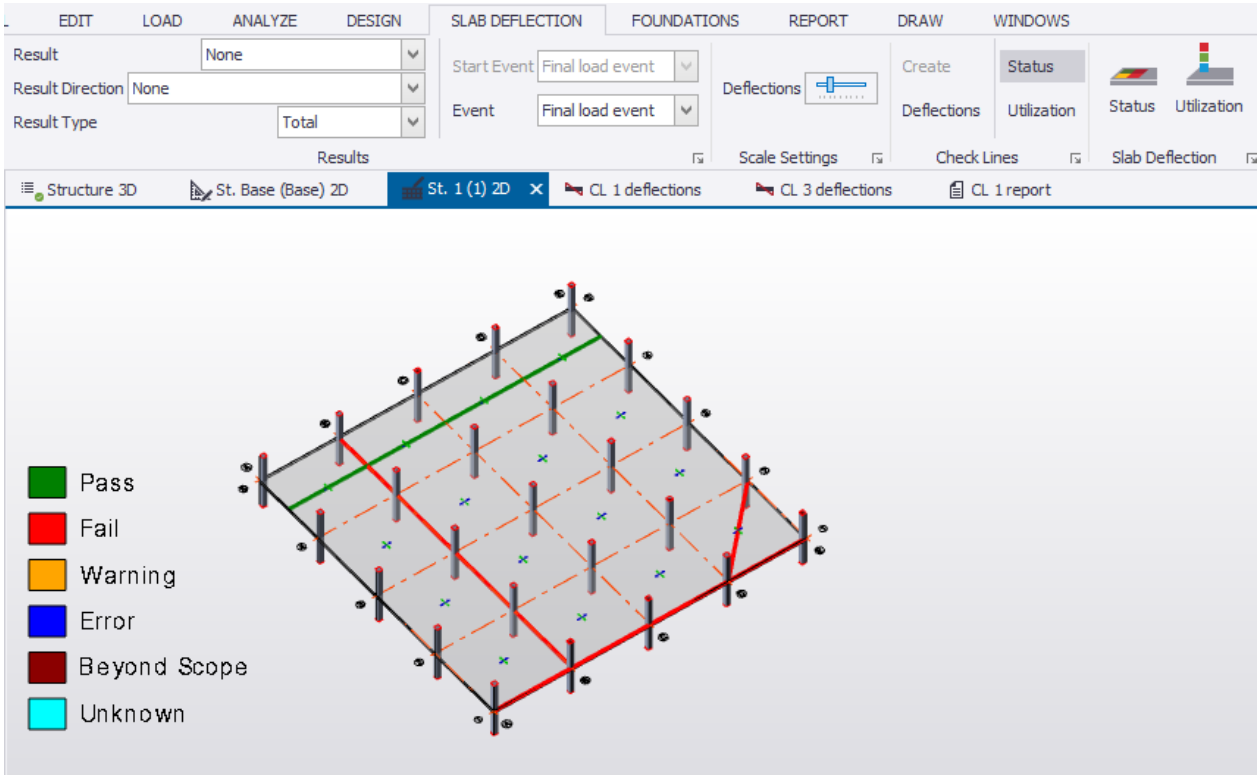
Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.2	193 33/64	1 : 811	✓ Pass	0.222
Sensitive Finshes	480	1 : 240	1.0	193 33/64	1 : 194	✗ Fail	1.237
Total	240	1 : 120	1.4	193 33/64	1 : 138	✓ Pass	0.870

As previously stated, the Sensitive Finishes check fails since the slope for differential deflection is reported as 1:206 which is greater than the allowable slope limit of 1:240.

Review Check Line Status and Utilization

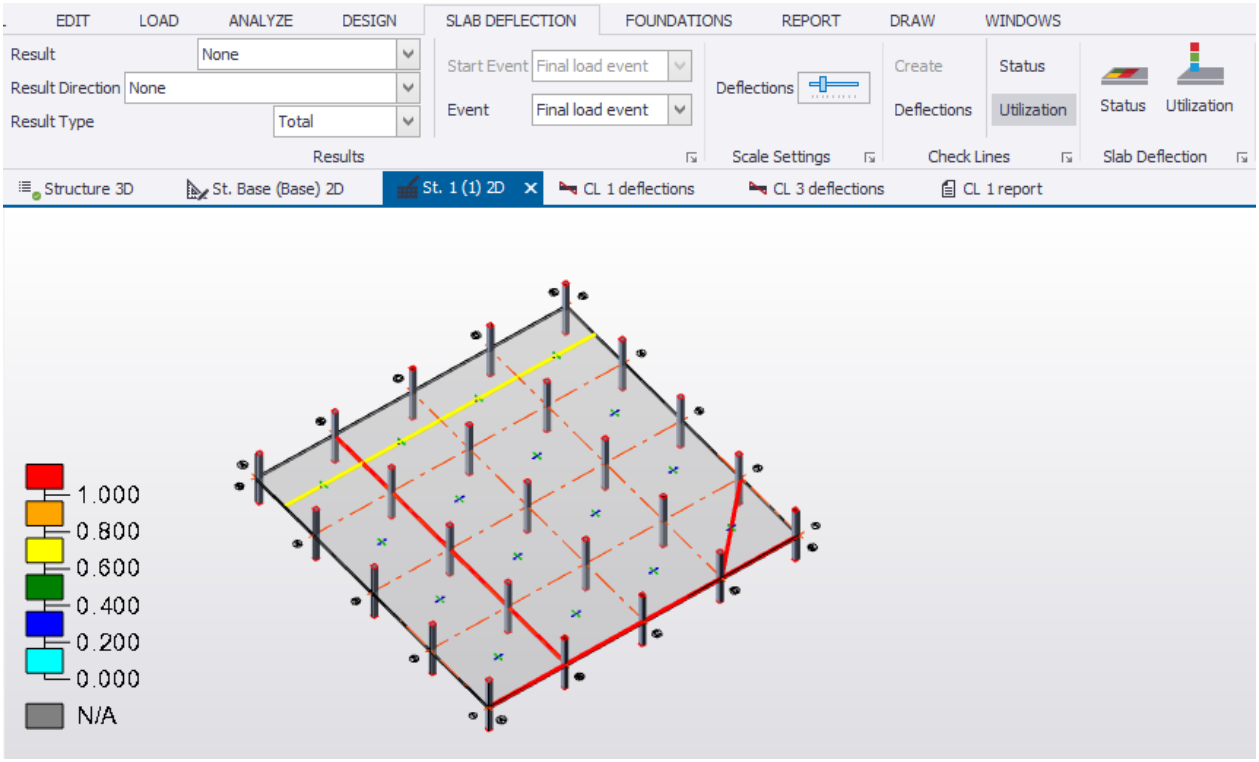
Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio.

1. Click on the **St.1 (1) 2D** 2D view to make it active.
2. To make it easier to see the check lines, change the Result droplist from Deflections to **None**.
3. Click **Status** in the Check Lines group of the ribbon to see the pass/fail status graphically displayed for each check line.



TIP You can also hover over a check line and the tooltip displays the utilization and pass/fall status.

4. Click **Utilization** in the Check Lines group to show the critical utilization for each check line and investigate the tooltip results.



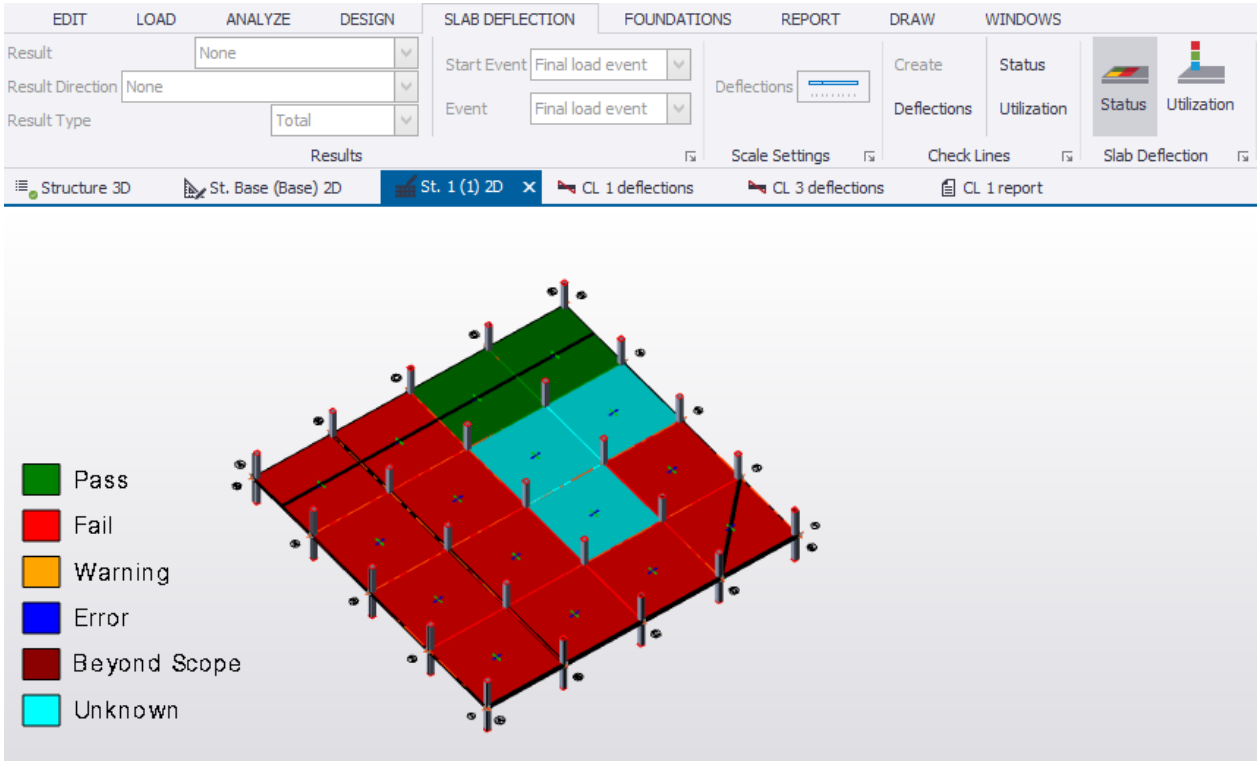
Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization

Review Slab Status and Utilization

Every check line is associated with at least one slab item. If all checks lines associated with the slab item pass then the slab item passes.

Both the Status and the Utilization can be reviewed.

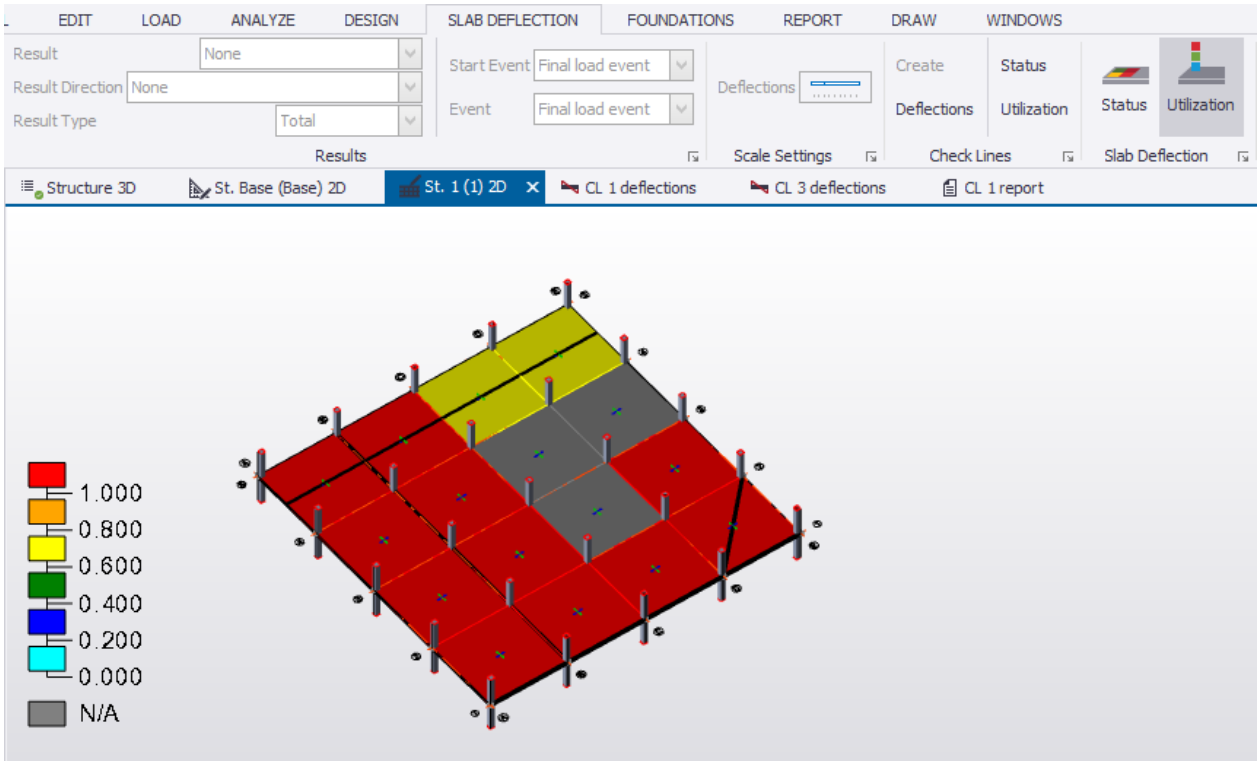
1. Click **Status** in the Slab Deflections group of the ribbon to see the pass/fail status of each slab.



- No check lines cross the majority of the slab items so they currently have a status of Unknown.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, the 4 slab items crossed by the horizontal check line Fail.
- One slab item is being crossed (only just, in one corner) by the passing diagonal check line, this is the only slab to have a pass status.

2. Click **Utilization** in the Slab Deflections group to show the Utilization of each slab item.

This is the worst utilization from all associated check lines.



Optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

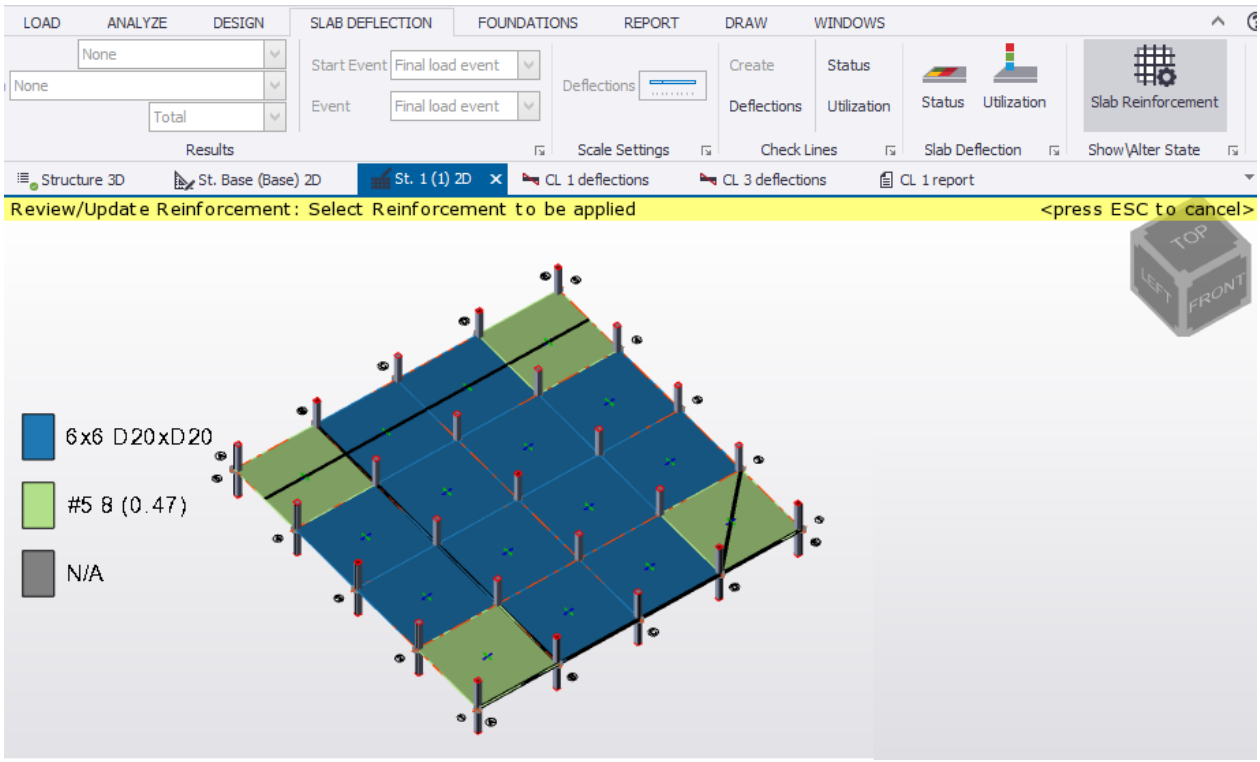
Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

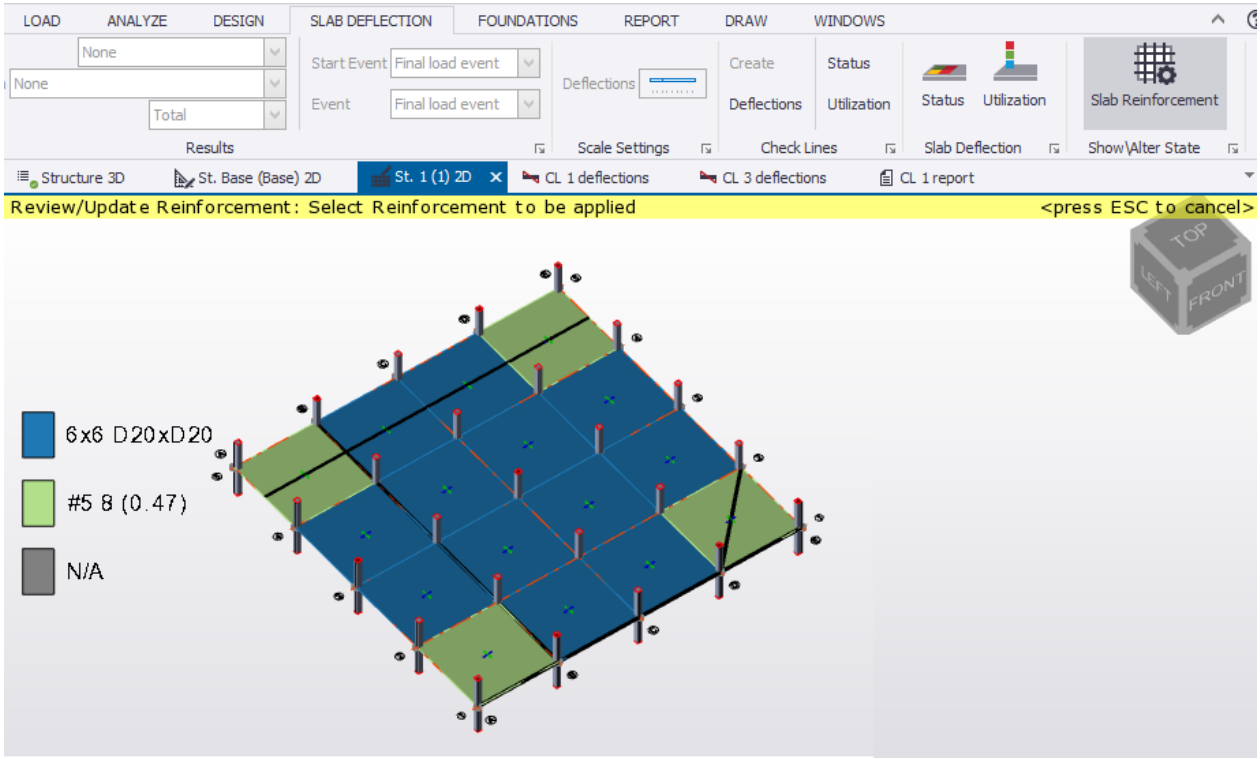
The analysis is extremely quick and since everything is contained within one model file, it allows “What If” scenarios to be considered to find the optimum solution.

In this exercise we will start by looking at the impact of adjusting the reinforcement.

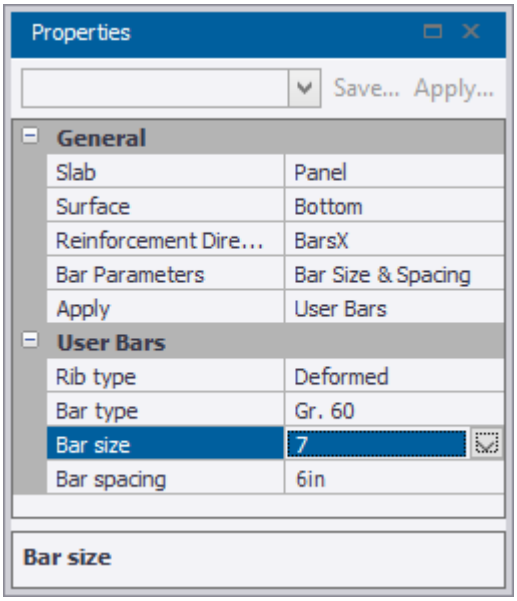
1. Click **Slab Reinforcement** in the Show/Alter State group to show the existing reinforcement.



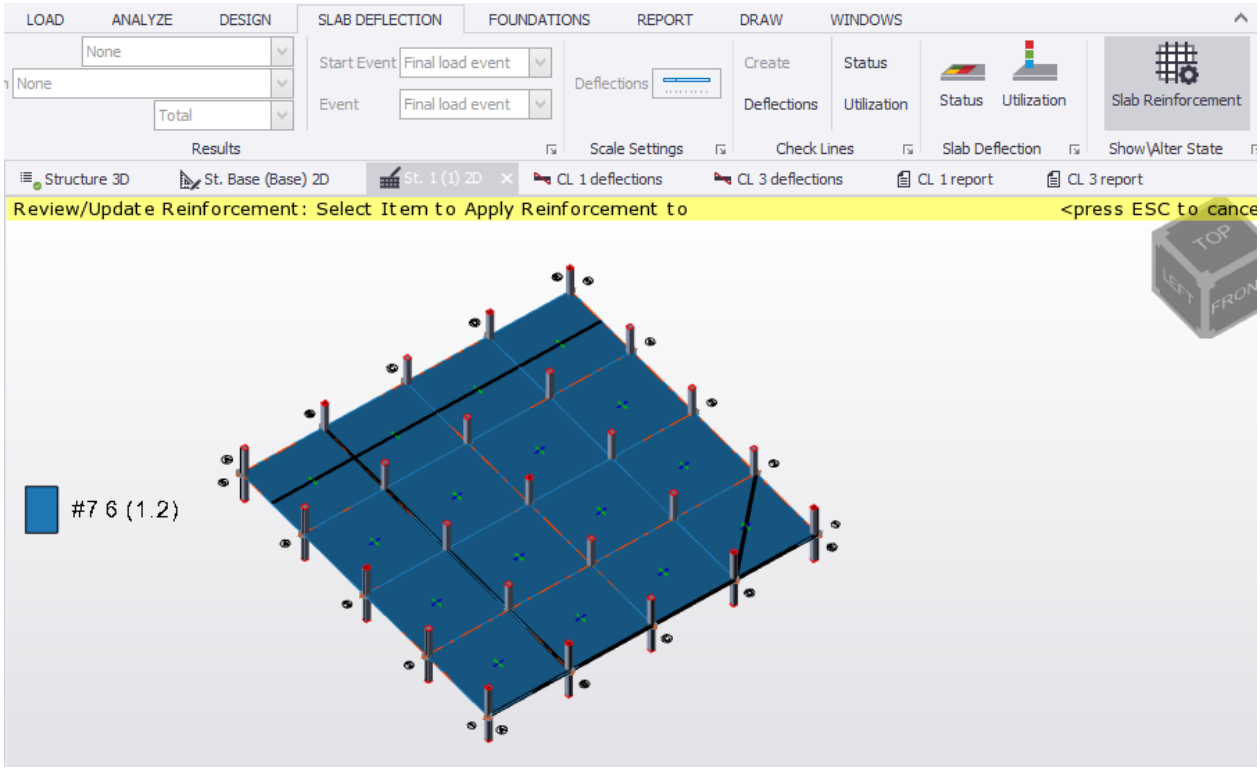
2. In the Properties Window:
   a. Leave the Slab Reinforcement to modify as **Panel**
   b. Leave the Slab Layer to modify as **Bottom**
   c. Change the Reinforcement Direction to **BarsY** to see the bars in that direction



3. In the Properties Window:
    a. Leave the Slab Reinforcement to modify as **Panel**
    b. Leave the Slab Layer to modify as **Bottom**
    c. Change Apply to **User Bars**
    d. Change Bar size to **7**
    e. Change Bar spacing to **6in**

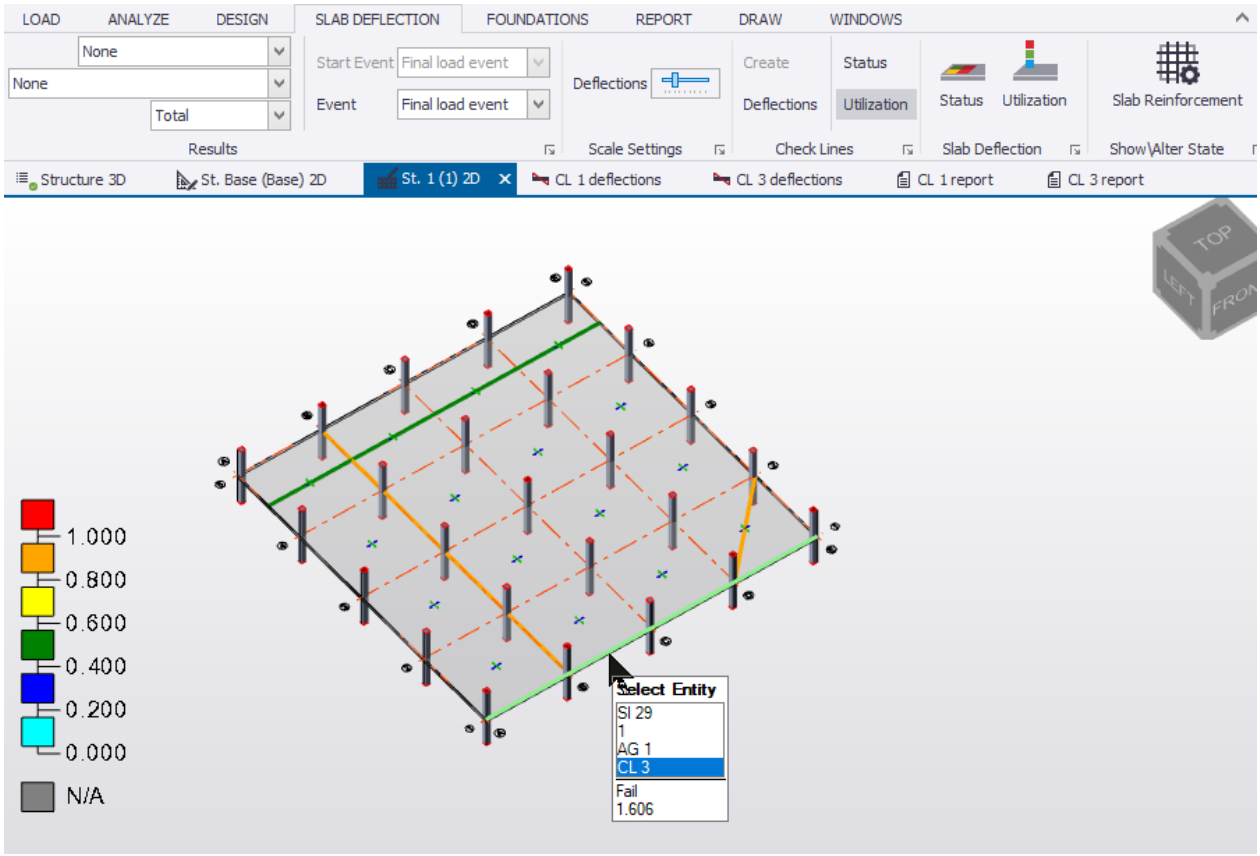


4. Click on each panel to apply the new reinforcement.



5. Repeat the above process to apply the same bars in the X reinforcement direction also.
6. Click **Analyze Current** to update the results

7. Click **Utilization** in the Check Lines group to show the critical utilization for each check line once again.
8. Investigate the tooltip results

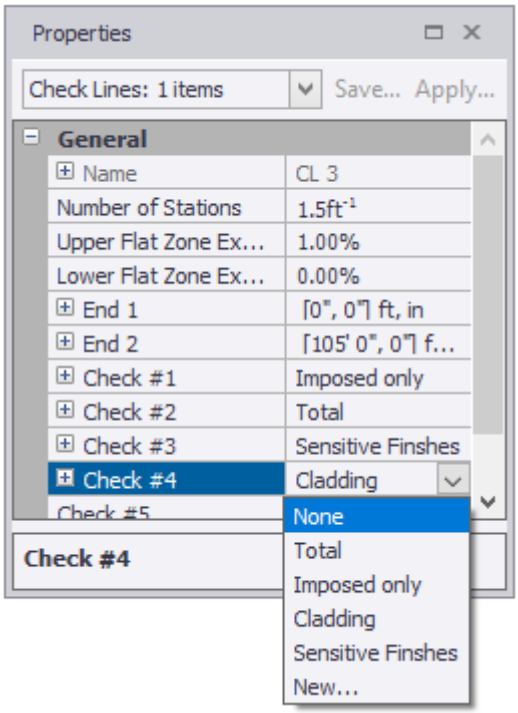


We can clearly see improvements in the results, however the check line along grid 1 with the more onerous Cladding deflection limit is still failing.

At this point in a real model you could begin to look at the impact of the various other input parameters, such as changing the concrete grade.

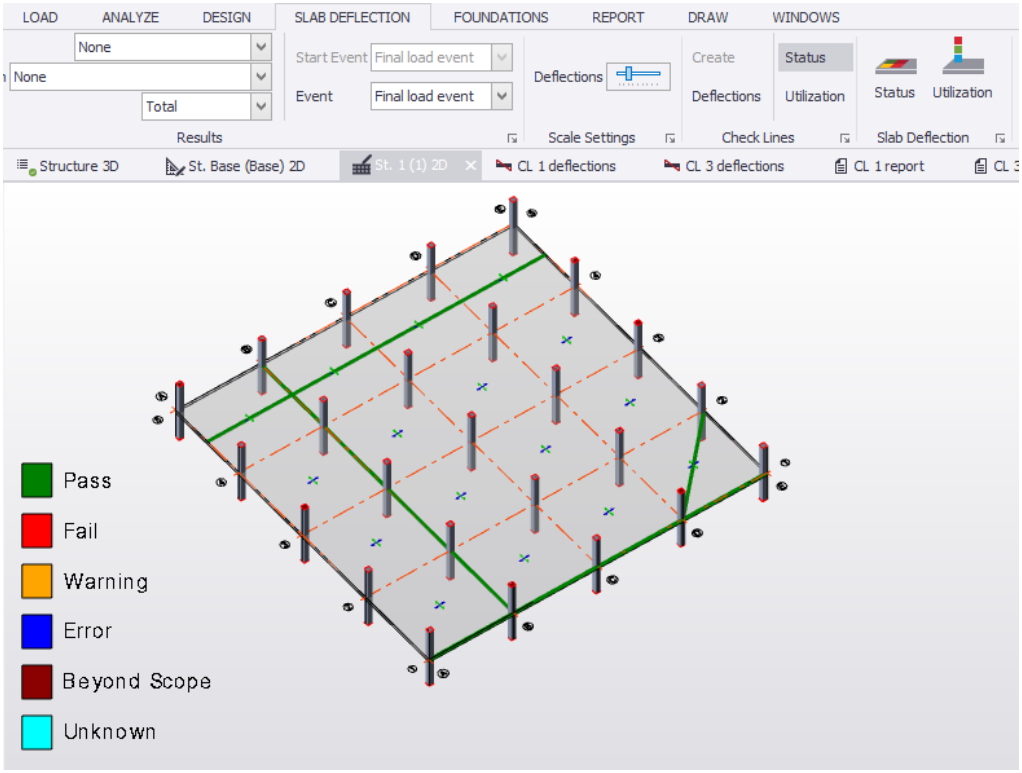
For the purpose of this exercise, and as the cladding check is not a code requirement, we will simply disable it, as follows:

9. Select the check line along grid 1
10. In the Properties window, reset Check #4 to **None**.



11. Click **Analyze Current** to update the results
12. Click **Status** in the Check Lines group.

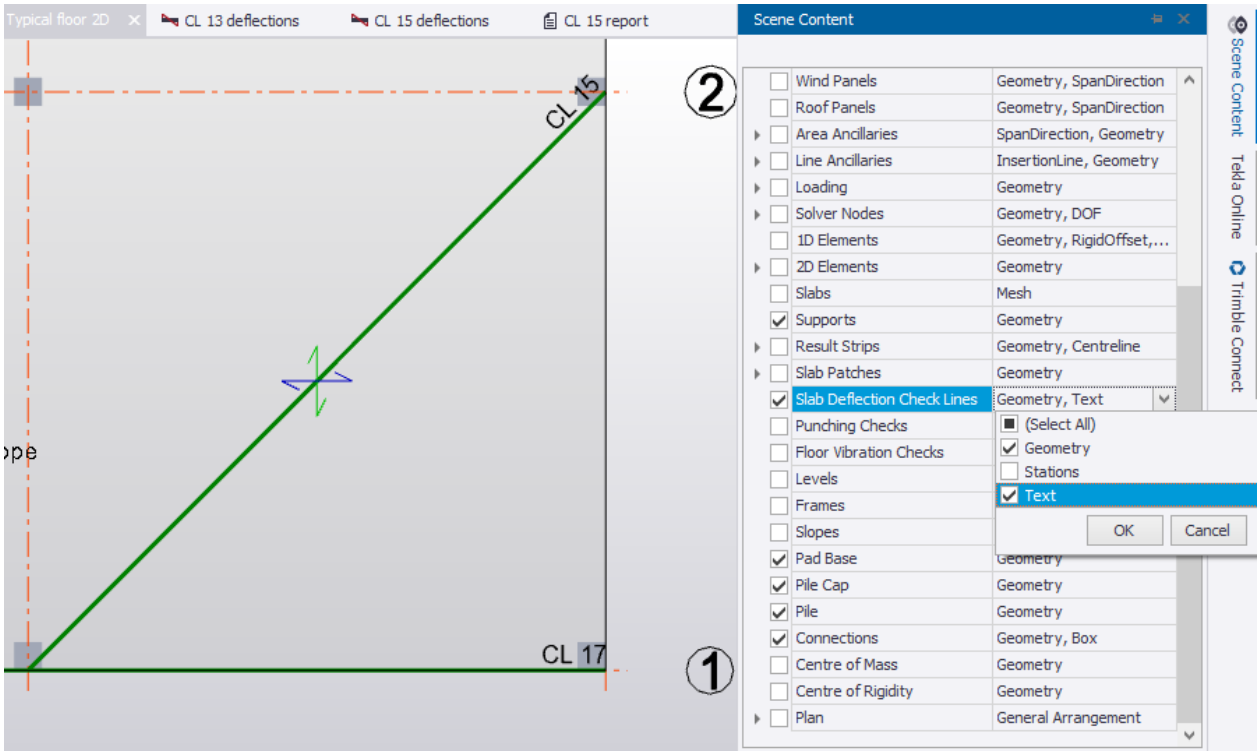
All of the check lines should now pass.



Generate Model report

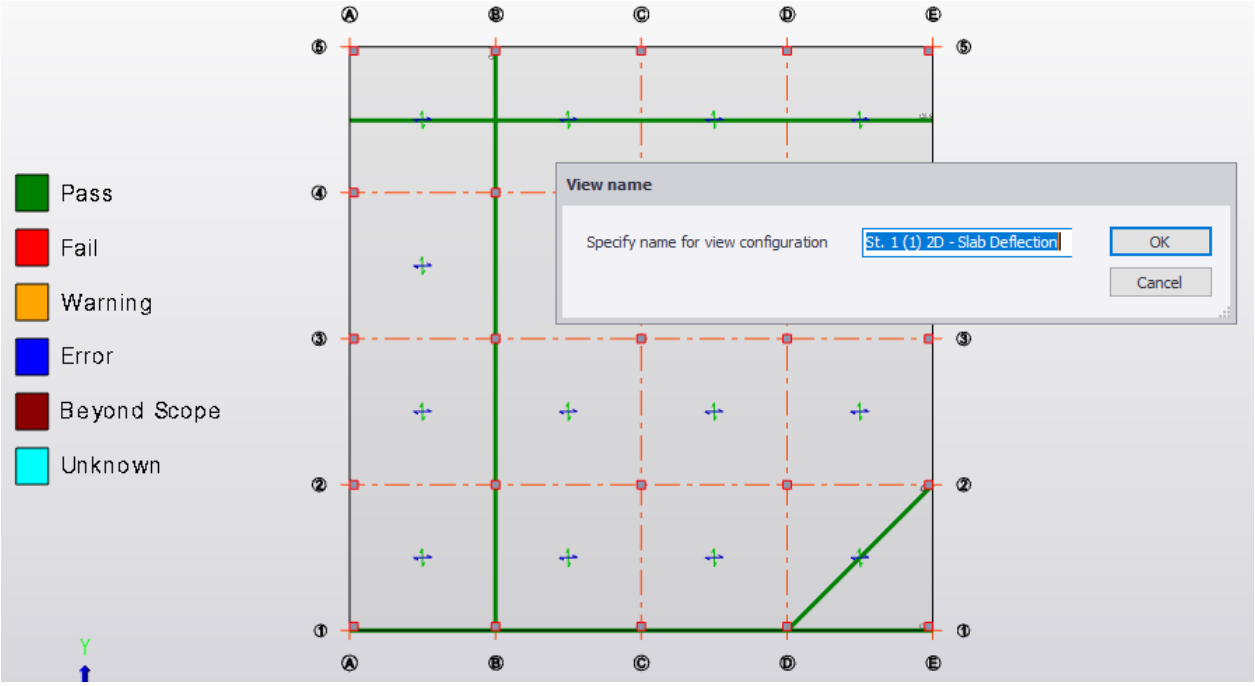
A Slab Deflection Check Lines model report can be created for the selected Model Filter (entire structure, level, plane or sub structure). This lists all the check lines for the chosen model filter. To help identify the check lines in the report it is sensible to include a saved picture of the scene view displaying check lines and their associated reference within the report.

1. In Scene Content, switch on the Text display for the Slab Deflection Checks.



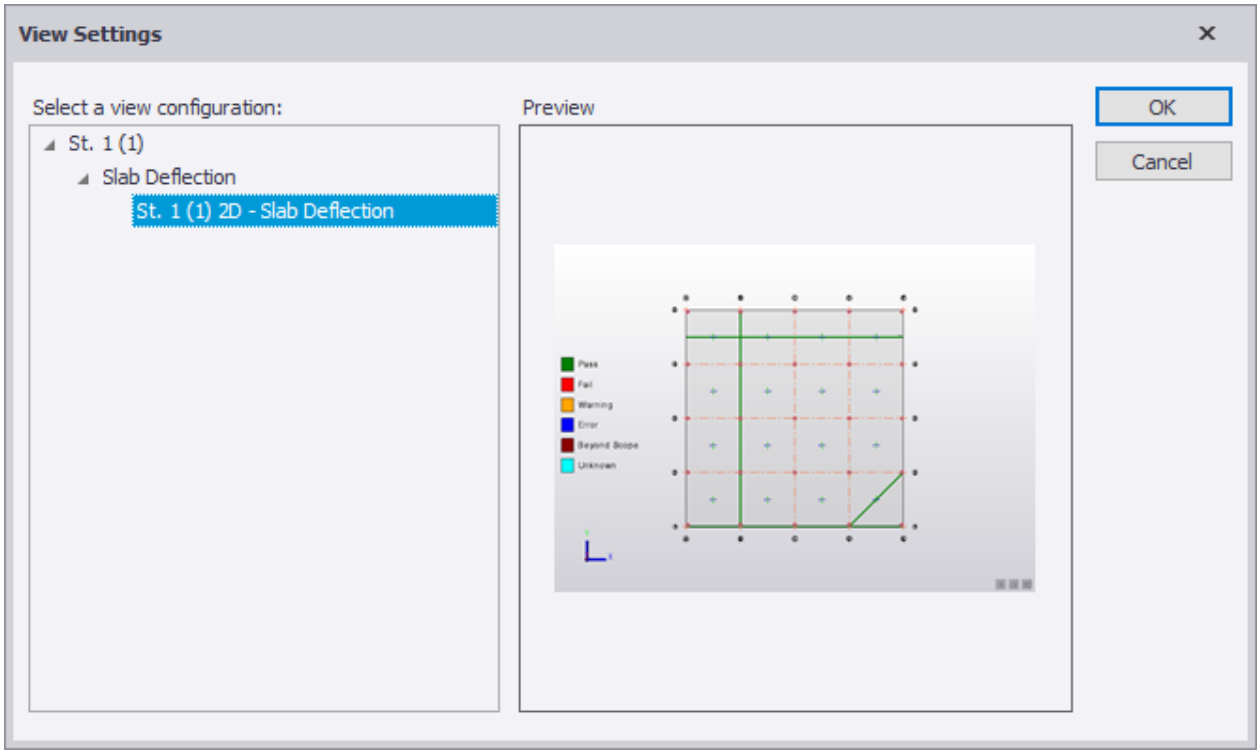
TIP The check line references can be customized using the Name property for each individual check line.

2. Right click in the Typical floor 2D view and choose **Save View Configuration...** from the context menu, then specify a name.

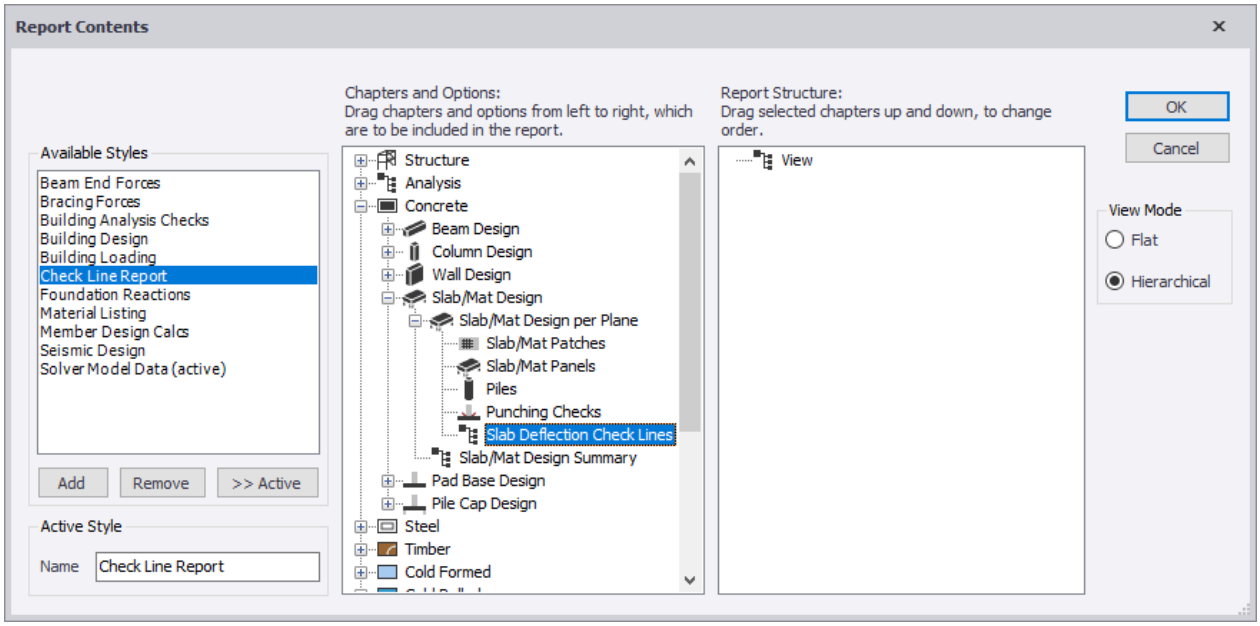


This saved view can be included in the Check line report.

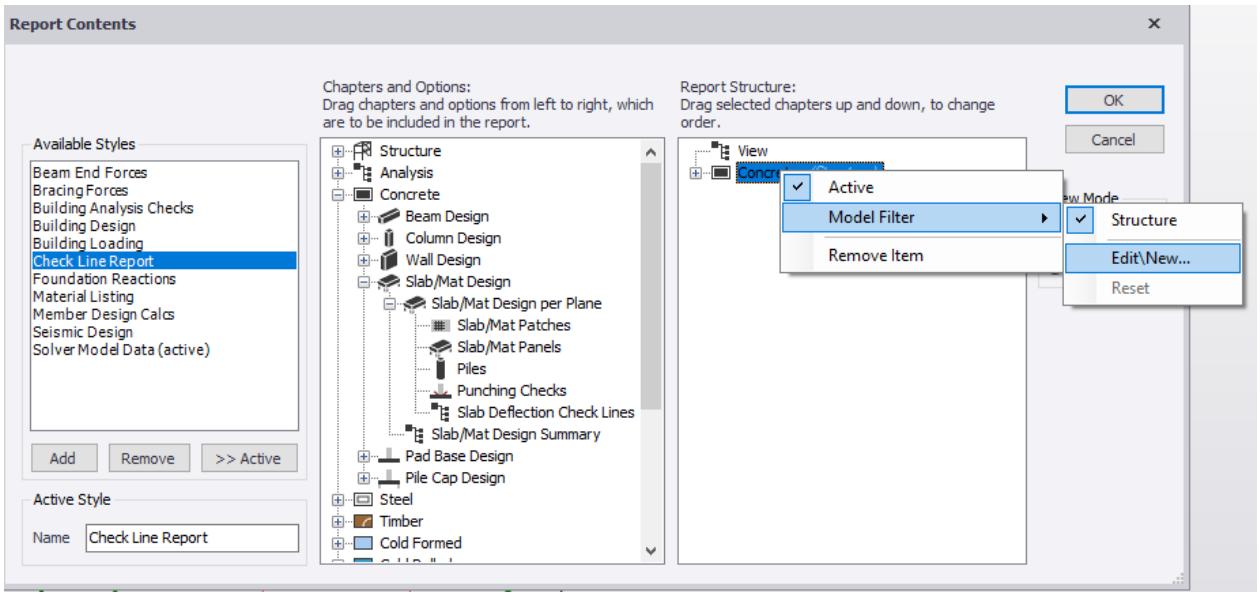
3. On the Report ribbon, click **Model Report...**
4. Click **Add** and provide a Name “Check Line Report” for the report.
5. In Chapters and Options, drag **View** to the Report Structure area
6. In the Report Structure, right click **View**, then choose **Settings...**
7. Select the Slab Deflection view you created earlier



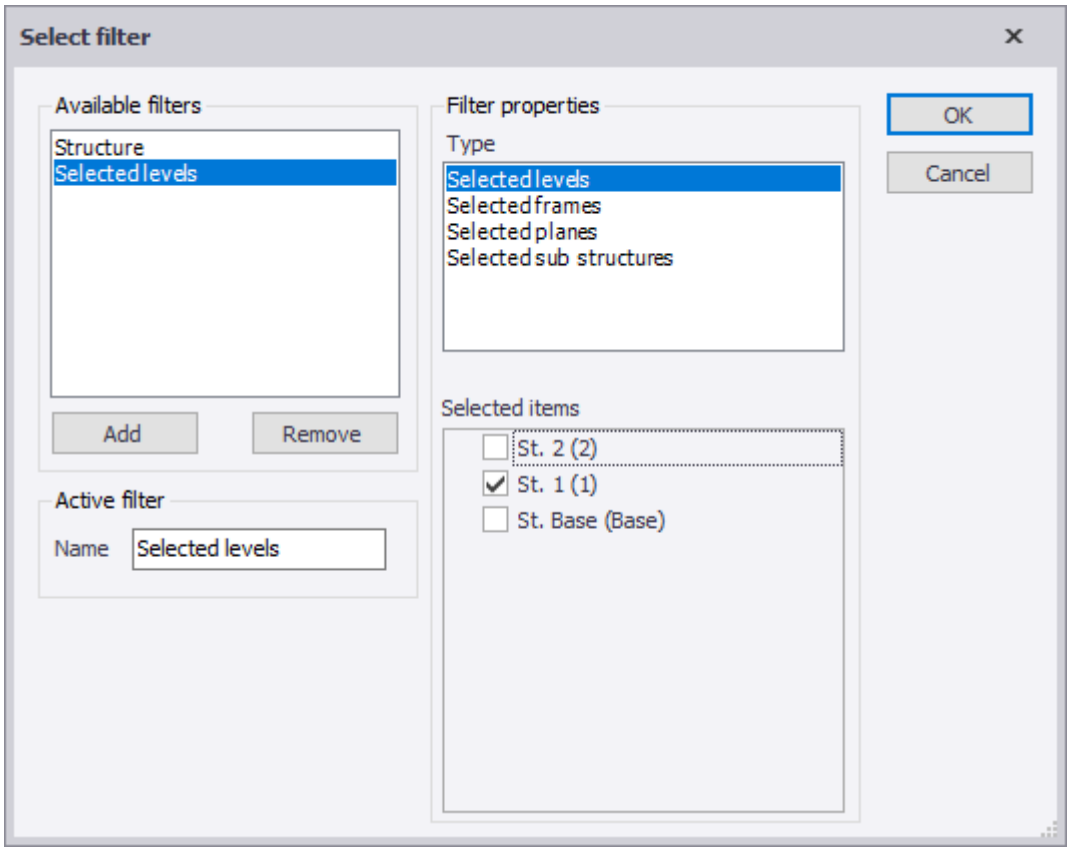
8. Click **OK** to return to the Report Contents dialog
9. In Chapters and Options, drag **Concrete>Slab/Mat Design per Plane>Slab Deflection Check Lines** to the Report Structure area



10. In the Report Structure, expand **Concrete> Slab/Mat Design per Plane> Slab Deflection Check Lines** and right click, **Model Filter> Edit/New**

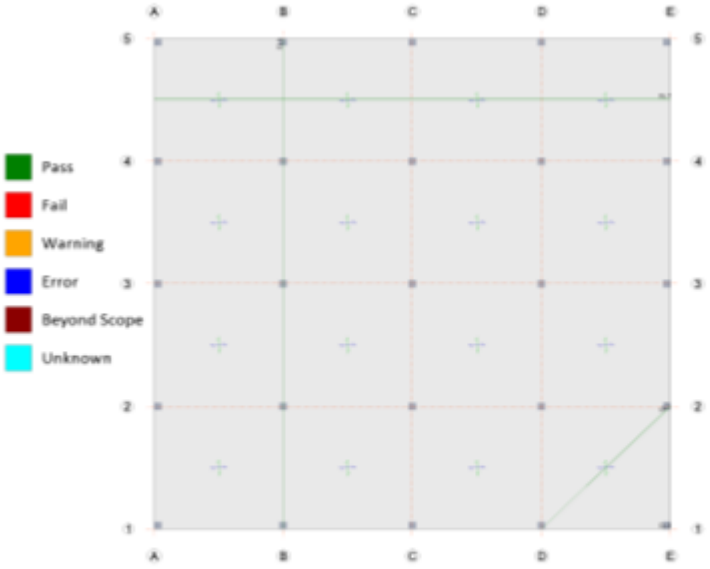


11. In the Filter dialog, click **Add** and select Selected levels and ensure a check against **St. 1 (1)**



12. Click **OK** to return to the Report Contents dialog.
13. Click **OK** to exit and save the report.

    A report structure called **Check Line Report** has now been saved that contains a view and the check lines.
14. To display the report.
    a. Use the Select drop list in the ribbon to select **Check Line Report**
    b. Click **Show Report** to open the report.



St. 1 (1) 2D - Slab Deflection

Concrete

Slab/Mat Design per Plane

St. 1 (1)

Slab Deflection Check Lines

CL 1

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.2	201 49/64	1 : 1087	✓ Pass	0.166
Sensitive Finshes	480	1 : 240	0.7	201 49/64	1 : 273	✓ Pass	0.879
Total	240	1 : 120	1.1	201 49/64	1 : 185	✓ Pass	0.649

CL 3

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	-0.1	144 15/32	1 : 1232	✓ Pass	0.146
Total	240	1 : 120	-0.8	144 15/32	1 : 183	✓ Pass	0.656
Sensitive Finshes	480	1 : 240	-0.6	144 15/32	1 : 254	✓ Pass	0.945

CL 5

Deflection Checks Summary

Reference	L/? Limit	Slope Limit	Deflection [in]	Length [in]	Slope	Status	Utilization
Imposed only	360	1 : 180	0.1	135 15/32	1 : 1059	✓ Pass	0.170
Sensitive Finshes	480	1 : 240	0.5	135 15/32	1 : 284	✓ Pass	0.845

10.8 Slab deflection settings

Click **Slab Deflection** --> **Settings** to edit the following slab deflection settings specific to the current model. After they have been edited, click Save, Load, Cancel, or OK according to your requirements.

Click **Home** --> **Settings** --> **Slab Deflection** to edit the .

New Load Event Defaults

Setting	Description
Start time offset	Allows you to define a default start time offset value for new event load start times. The value of a new event load start time is always the previous event start time + the default start time offset.
Construction load	Allows you to define a default value for new construction loads. **NOTE** The default construction load value depends on the regional code that you are using.

New Check Defaults

Setting	Description
Deflection limit, L /	When a new check is added to the Slab Deflection Check Catalogue it initially defaults to the deflection limit set here.

Aging, Creep & Shrinkage

Setting	Description
Allowance for shrinkage effects in total deflection	Allows you to define an amplification base factor for shrinkage. The value must be within the range from 0.1 to 0.9.
Aging Coefficient	Allows you to select between 2 creep/ aging methods: **User defined** and **Automatic**. **NOTE** The **Aging Coefficient** option is only available for the US regional code.

Modification factors

NOTE This sub-page is only displayed when the dialog box is accessed from the **Slab Deflection** ribbon.

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.

NOTE 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. If you make changes to any of the modification factors, analysis must be repeated.

Iterative Cracked Section Analysis

NOTE This sub-page is only displayed when the dialog box is accessed from the **Slab Deflection** ribbon.

Setting	Description
Global Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 200.

Setting	Description
	• **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**. • 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**.

Setting	Description
	• If you are using the Eurocode regional code, increase the value in **Smoothing Parameter**.

Load Event Sequence

This sub-page is only displayed when accessed from **Settings** on the **Home** tab. It allows you to modify the global event sequence according to your needs.

TIP The global event sequence provides the basis for the model event sequence that you can modify, whereas the model event sequence provides the basis for a submodel event sequence that you can modify.

11 Stability and imperfections handbook

This handbook introduces you to the following topics:

- The four sources of stability requirements that may need to be considered, see Overview of stability requirements (page 853).
- Choosing the correct analysis type for your model and regional code, see Allowing for global second-order effects (page 857).
- Catering for global imperfections, see Allowing for global imperfections (page 870).
- Catering for member imperfections, see Allowing for member imperfections (page 872).

11.1 Overview of stability requirements

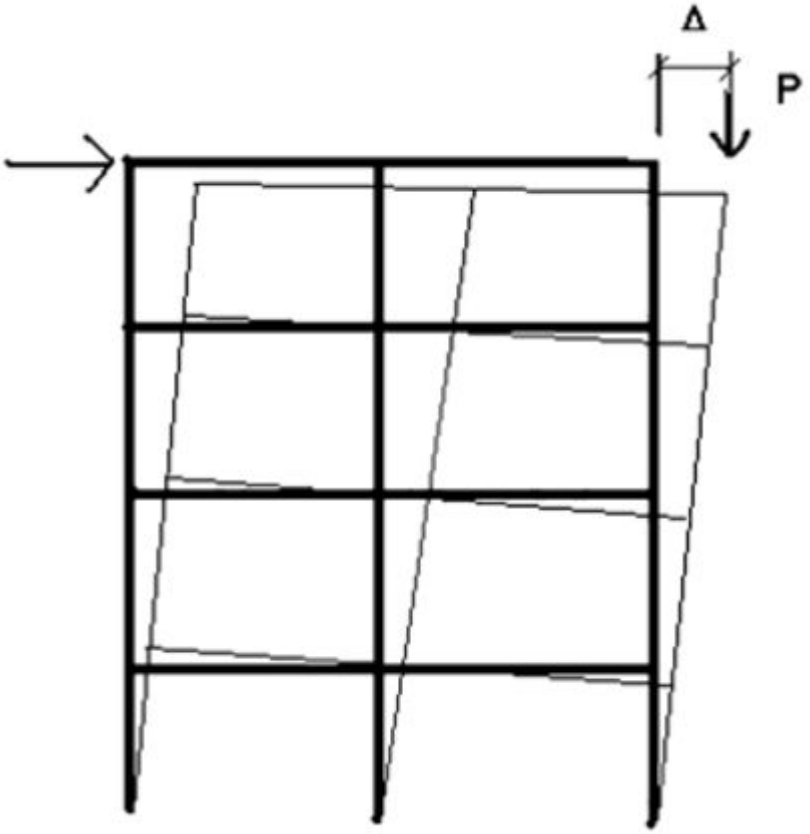
The analysis and design process has to allow for the differences between a theoretical mathematical model of a building and a more realistic representation. For example, buildings are not truly vertical when first built nor do they remain so when subject to load. These are called stability requirements and are from four sources:

1. **Global second-order (P-Δ) effects** to allow for deformation of the structure under load,
2. **Member second-order (P-δ) effects** to allow for deformation of the members under load,
3. **Global imperfections** due to the structure not being built plumb and square,
4. **Member imperfections** due to initial lack of straightness of the member.

There are various methods of allowing for each of these and they can be different for steel and concrete. There is also some variation based on country code.

Global second-order (P-Δ) effects

If gravity loads are applied to the deflected shape of a structure the load P applied at eccentricity Δ generates additional forces.



Provided the deflection is small:

- Structure is 'Non-sway'
- Second order effects can be ignored.

At some level these effects are no longer ignorable:

- Structure is 'Sway sensitive'
- And you have to do something to account for the second order effects.

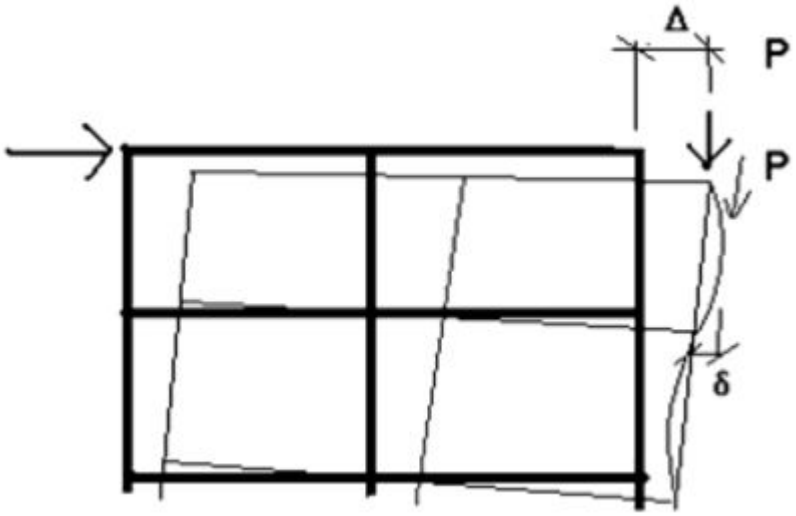
NOTE The terminology: 'Non-sway' and 'Sway sensitive' can vary between codes.

Related concept

Allowing for global second-order effects (page 857)

Member second-order (P-δ) effects

Under load members will deform between their ends:



- Member curvature introduces a displacement δ between the member ends.
- The member axial loads applied at eccentricity δ generates additional forces.

In concrete structures:

- Where deflections are small :
  - Member is 'Short' or 'Stocky' or 'Non-Slender'
  - member second order effects are considered ignorable
- At some level they are no longer ignorable:
  - Member is 'Slender'
  - effects must be catered for in the design calculations

In steel structures: these effects are intrinsically allowed for in the design equations.

NOTE The terminology: 'Short' , 'Stocky' and 'Non-slender' can vary between codes.

When must global and member second order effects be considered?

Depending on the building's overall sway classification and each member's slenderness, global and member effects must be considered as follows:

Member Effects	Global	Effects
	Non-sway	Sway
Short Member	A	C
Slender Member	B	D

A - All second order effects can be ignored

B - Global effects can be ignored - member effects must be considered

C - Global effects must be considered - member effects can be ignored

D - Global effects must be considered - member effects must be considered

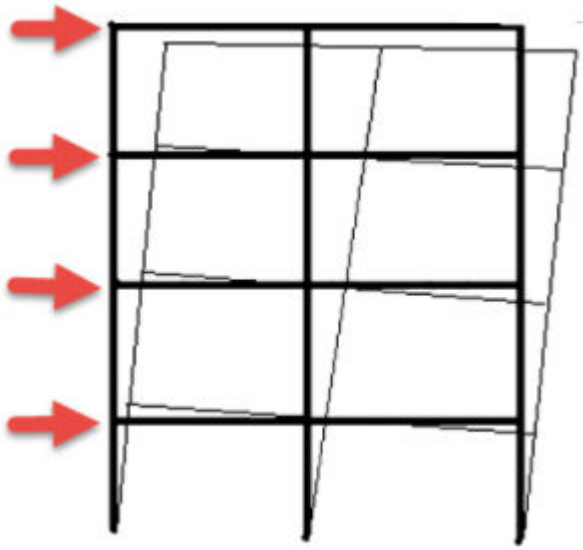
NOTE REMEMBER - every structure and every member has 2 directions!

Global imperfections

When the design code requires it you need to account for some degree of inclination (slope); typically in the range 0.2 to 0.5%

The codes allow you to cater for this in different ways:

- You could build multiple analysis models that are inclined
- You could have a single analysis model where you apply Equivalent Horizontal Forces [Eurocodes] / Notional Loads [AISC/ACI] / Notional Horizontal Forces [AUS/BS/IS] that will induce the same effect. Basically this means applying horizontal forces = 0.2 to 0.5% of the vertical forces in any combination.



In Tekla Structural Designer we use the second option.

NOTE Global Imperfections apply regardless of whether the structure is 'Non-sway' or 'Sway sensitive'.

Related concept

Allowing for global imperfections (page 870)

Member imperfections

Member Imperfections apply regardless of whether members are slender or not. They are normally dealt with as part of the member design.

Related concept

Allowing for member imperfections (page 872)

11.2 Allowing for global second-order effects

The type of analysis required to meet stability requirements and the checks performed will vary depending on the regional code and material.

Choice of analysis type (ACI/AISC)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of two analysis types. These are,

- First-order analysis (not suitable for the final design of steel structures),
- Second-order analysis.

For steel structures: - Unless a number of specific criteria can be met, it is essential that your final design utilizes second-order analysis. However, second order analysis can be more sensitive to parts of your model that lack stiffness. For this reason it is recommended that you initially use first-order analysis to obtain sections and an overall building performance with which you are satisfied before switching to second-order analysis.

For concrete structures: - By choosing second-order analysis the global second-order effects are automatically catered for. However, if you determine that the structure is non-sway (see When should a concrete building be classed as non-sway? (page 858)) you may instead opt for the design to be based on the first order analysis.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 870).

When should a concrete building be classed as non-sway?

ACI 318-11 clause 10.10.5.2 states that a story can be considered as non-sway when the stability index Q is less than or equal to 0.05. (Basically, second order effects can be ignored below this value.)

Tekla Structural Designer does not report Q directly, but the Drift report does calculate an indicative stability coefficient (Δ_2/Δ_1) which is effectively the same as a moment magnifier.

From ACI Eq. (10-20):

Moment magnifier, $\delta_s = 1/(1-Q)$

Equating this to the stability coefficient gives

$1/(1-Q) = (\Delta_2/\Delta_1)$

$1-Q = 1/(\Delta_2/\Delta_1)$

$Q = 1 - 1/(\Delta_2/\Delta_1)$

For $Q < 0.05$

$1 - 1/(\Delta_2/\Delta_1) < 0.05$

$(\Delta_2/\Delta_1) < 1/0.95$

$(\Delta_2/\Delta_1) < 1.0526$

Therefore, if the Drift report determines an indicative stability coefficient (Δ_2/Δ_1) less than 1.05 it can be assumed non-sway; whereas if greater than this value a second-order analysis is required.

NOTE The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 870).

Choice of analysis type (BS)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of three analysis types. These are,

- First-order (Elastic) analysis,
- Amplified forces (k_{amp}) method (uses first-order analysis),
- Second-order analysis.

First-order (Elastic) analysis

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - - see: **A practical approach to setting the analysis type** below.

Amplified forces (k_{amp}) method

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (regular frameworks with $\lambda_{cr} > 4$).

If the amplified forces method is selected you must also indicate which formula to use for determining the amplification factor,

If the structure is clad and if the stiffening effect of cladding is not taken into account explicitly:

$k_{amp} = \lambda_{cr}/(1.15\lambda_{cr} - 1.5)$

If the structure is unclad or clad with a direct allowance made for the stiffening effect:

$k_{amp} = \lambda_{cr}/(\lambda_{cr} - 1)$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the

'appropriate' value of k_{amp}. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of λ_{cr} for all stacks in the building.

The k_{amp} results are summarized for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review tab.
2. Select 'k_{amp}' from the View Type drop list on the Review tab.
3. The k_{amp} results in both directions are tabulated for each column in the building.

Second-order analysis

Full second-order analysis is not restricted to regular frameworks, but requires $\lambda_{cr} > 2$.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 870).

A practical approach to setting the analysis type

Unless λ_{cr} is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections and an overall building performance with which you are satisfied, before switching to P-Δ analysis.

Note: If the rigorous second-order (P-Δ) analysis approach is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.
2. Perform Design All (Gravity) using first-order analysis in order to size members for the gravity loads.
3. Once the members are adequately sized for the gravity combinations obtain a figure for the building's elastic critical load factor, λ_{cr}. See: **How do I assess the worst elastic critical load factor for the building?** section below.

4. If the λ_{cr} that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.
5. If λ_{cr} is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
   - Either, refine the design until λ_{cr} is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces - see below),
   - Or, in order to use the amplified forces (k_{amp}) approach, refine the design further until λ_{cr} is greater than 4.0.
6. When a suitable λ_{cr} has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate. (If the k_{amp} approach is selected you must also indicate which formula to use for determining the amplification factor, This will depend on whether the structure is clad or not and if the cladding is taken into account explicitly or not.)
7. With the analysis type still set to the full second-order, or the k_{amp} approach perform Design All (Static).

If you use the k_{amp} approach be aware that BS5950-1:2000 classes certain structures outside the scope of this method. e.g. tied portals, or structures containing tension only braces. For such structures, you would need to ensure that λ_{cr} is greater than 2.0, and use the full second-order analysis approach for the static design.

How do I assess the worst elastic critical load factor for the building?

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review tab.
2. Select 'Sway' from the View Type drop list on the Review tab.
3. The elastic critical load factor in both directions ((λ_{Dir1} & (λ_{Dir2}) is tabulated for each column in the building.
4. Make a note of the smallest λ value from all of the columns in either direction.

NOTE If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the λDir1

header until the columns are arranged in increasing order of λDir1, then repeat for λDir2.

In BS 5950-1:2000 a building can be considered as 'non-sway' when $\lambda_{cr} \geq 10$ else it is 'sway sensitive' and (global) second-order effects must be taken into account.

Note however that you are not restricted in your choice of analysis type irrespective of the value of λ_{cr} (it is your call, although we will warn you about it).

How is the elastic critical load factor calculated?

In order to determine whether a building is 'non-sway' or 'sway sensitive', Tekla Structural Designer calculates the elastic critical buckling load factor, λ_{cr}. It is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Notional Horizontal Forces (NHF) are determined. It uses 0.5% of the vertical load at the column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the NHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of λ_{cr} for that combination as follows,

$\lambda_{cr} = h/(200 * \delta_s)$

Where

h= the storey height

δ_s =the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads.

NOTE Within each column's properties, a facility is provided to exclude particular column stacks from the sway check calculations to avoid spurious results associated with very small stack lengths.

What are the twist results?

A ‘measure’ of twist is also tabulated for each column - this indicates the degree to which if you push the column one way, how much it moves orthogonally as well. If you have a building where the 'lateral load resisting system' is not well dispersed then pushing one way can cause significant movement in the other direction.

The twist is reported as a ratio of: distance moved in the direction of loading/ absolute distance moved.

Example:

When a column node moves in X and Y then the 'total' deflection is $\sqrt{(\delta_x^2 + \delta_y^2)}$ in other words the diagonal of the triangle and not either of the sides.

So if a node moves say 10mm in X and 2mm in Y, its diagonal i.e. absolute deflection in this plane is $\sqrt{(100 + 4)} = 10.198$.

Hence its twist is what it should have been with just X loading i.e. 10mm divided into what it actually moved i.e. 10.198.

So Twist = 1.0198.

Choice of analysis type (Eurocode)

In Tekla Structural Designer on the Analysis page of the Design Options dialog you have the choice of three analysis types. These are,

- First-order (Elastic) analysis,
- Amplified forces (k_{amp}) method (uses first-order analysis),
- Second-order analysis.

First-order (Elastic) analysis

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - see: **A practical approach to setting the analysis type** below.

Amplified forces (k_{amp}) method

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (basically regular frameworks with $\alpha_{cr} > 3$ - see **Validity of the amplified forces method** section below). Full second-order analysis is preferred for steelwork and since it is not precluded by EC2 it can be used for concrete.

The amplified forces method is described differently in EC3 compared to EC2, whilst the presentations are different, they are both based on the amplifier, k_{amp} given as,

$k_{amp} = 1/(1 - 1/\alpha_{cr})$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of k_{amp}. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of α_{cr} for all stacks in the building.

The k_{amp} results are summarized for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review tab.
2. Select 'k_{amp}' from the View Type drop list on the Review tab.
3. The k_{amp} results in both directions are tabulated for each column in the building.

NOTE The amplified forces method is not recommended for non-linear structures - a full second-order analysis should be performed instead.

Validity of the amplified forces method

EC3 Clause 5.2.2 (6)B lists limitations on the applicability of the Amp. Forces method. It is therefore your responsibility when selecting this method to ensure all of the following:

- all storeys have a similar distribution of vertical load
- all storeys have a similar distribution of horizontal load
- all storeys have a similar distribution of frame stiffness with respect to the applied storey shear forces

Also according to clause 5.2.1 (4)B limitation:

- roof slope shallow - not steeper than 1:2 (26 degs)
- axial compression in beams or rafters - N_{cr} / N_{ed} <= 11.1

Second-order analysis

Full second-order analysis is more widely applicable for steelwork structures and since it is not precluded by EC2 it can be used for concrete.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see Use of Modification Factors (page 870)

A practical approach to setting the analysis type

Unless α_{cr} is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections

and an overall building performance with which you are satisfied, before switching to one of the P-Δ analysis methods.

The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.
2. Perform Design All (Gravity) using first-order analysis in order to size members for the gravity loads.
3. Once the members are adequately sized for the gravity combinations obtain a figure for the building's elastic critical load factor, α_{cr}. See **How do I assess the worst elastic critical load factor for the building?** section below.
4. If the α_{cr} that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.
5. If α_{cr} is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
   - Either, refine the design until α_{cr} is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces),
   - Or, in order to use the amplified forces approach, refine the design further until α_{cr} is greater than 3.0.
6. When a suitable α_{cr} has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate.
7. With the analysis type still set to the full second-order, or the amplified forces method, perform Design All (Static).

NOTE If full second-order analysis is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

NOTE If you use the 'Second-order analysis - Amp. forces method' be aware that EC3 classes certain structures outside of its scope (see above section: **Validity of the amplified forces method**). Such structures would need to be refined during gravity sizing until the elastic critical load factor is at least greater than 2.0, so that the full second-order approach can be used for the full design.

When should a building be classed as sway sensitive?

Susceptibility to second order effects is a general characteristic and is not material specific, it has just been presented differently in EC3 and EC2:

- In EC3 a building can be considered as 'non-sway' when the elastic critical load factor $\alpha_{cr} \geq 10$, else the building is 'sway sensitive' and (global) second-order effects must be taken into account.
- In EC2 the definition is slightly different - it does not use the terms 'non-sway' and 'sway sensitive'. Rather it simply defines when second-order effects are small enough to be ignored. The principle is given in Clause 5.8.2 (6) which states that they can be ignored if they are less than 10% of the corresponding first order effects. Because of the way in which the amplification factor, k_{amp} is calculated the change point is at an α_{cr} of 11 not 10. See: **Derivation of the k_{amp} formula for concrete structures** section below.

However, the intent is the same in both cases and so in Tekla Structural Designer $\alpha_{cr} \geq 10$ is taken as the change point. In any event, you are not restricted in your choice of analysis type irrespective of the value of α_{cr} (it is your call, although we will warn you about it)

How do I assess the worst elastic critical load factor for the building?

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review tab.
2. Select 'Sway' from the View Type drop list on the Review tab.
3. The elastic critical load factor in both directions (α_{Dir1} & α_{Dir2}) is tabulated for each column in the building.
4. Make a note of the smallest elastic critical load factor from all of the columns in either direction - this is the α_{cr} value for your building.

NOTE If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click the αDir1 header until the columns are arranged in increasing order of αDir1, then repeat for αDir2.

Having determined an α_{cr} for your building, you then use it when deciding which is the most appropriate analysis type for design.

How is the elastic critical load factor calculated?

The elastic critical load factor, α_{cr} is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing

gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Equivalent Horizontal Forces (EHF) are determined. These consist of 0.5% of the vertical load at each column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the EHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of α_{cr} for that combination as follows,

$\alpha_{cr} = h/(200 * \delta_{EHF})$

Where

h = the storey height

δ_{EHF} = the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads

---

NOTE Within each column's properties, a facility is provided to exclude particular column stacks from the sway check calculations to avoid spurious results associated with very small stack lengths.

What are the twist results?

A ‘measure’ of twist is also tabulated for each column - this indicates the degree to which if you push the column one way, how much it moves orthogonally as well. If you have a building where the 'lateral load resisting system' is not well dispersed then pushing one way can cause significant movement in the other direction.

The twist is reported as a ratio of: distance moved in the direction of loading/ absolute distance moved.

Example:

When a column node moves in X and Y then the 'total' deflection is $\sqrt{(\delta_x^2 + \delta_y^2)}$ in other words the diagonal of the triangle and not either of the sides.

So if a node moves say 10mm in X and 2mm in Y, its diagonal i.e. absolute deflection in this plane is $\sqrt{(100 + 4)} = 10.198$.

Hence its twist is what it should have been with just X loading i.e. 10mm divided into what it actually moved i.e. 10.198.

So Twist = 1.0198.

Derivation of the k_{amp} formula for concrete structures

EC2 provides two specific approaches to determine the change point below which second-order effects are small enough to be ignored:

The first specific approach is contained in Clause 5.8.3.3 which provides a pass/fail criterion to check whether the global second-order effects may be ignored. It is given as,

$F_{VEd} = k_1 * n_s/(n_s + 1.6) * S(E_{cd} * I_c)/L^2$

where

F_{VEd} = the total vertical load (on 'braced' and 'bracing' members)

k_1 = a factor that allows for cracking in the concrete of the LLRS and is a Nationally Determined Parameter (NDP)

* n_s = number of storeys

E_{cd} = the design value of the modulus of elasticity of the concrete

I_c = the second moment of area of the uncracked bracing members

L = the total height of the building

However, the above approach has a number of restrictions in its application and as a result it is not applied in Tekla Structural Designer.

The second specific approach is given in Annex H.

The method given in Annex H.1.2 is the background for the more limited method given in Clause 5.8.3.3 as described above, but it does not apply where there is significant shear deformation in the LLRS e.g. for shear walls with significant openings, hence again it is not considered in Tekla Structural Designer.

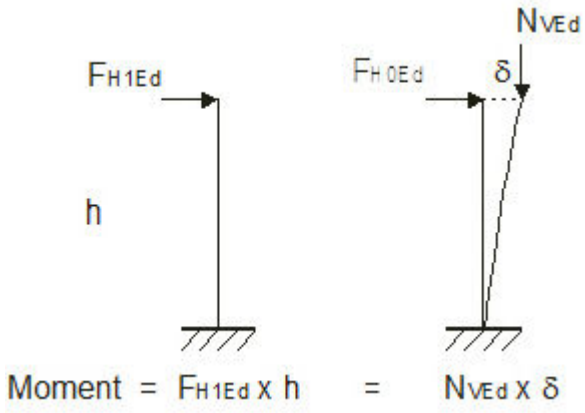
Instead, recourse is made to determining the level of second-order effect using Annex H.2. Using this approach, by rearranging Equation H.8 it is possible to provide a 'stability coefficient' $1/\alpha_{cr}$ which can be applied as the change point between non-sway and sway sensitive structures.

$F_{HEd} = F_{H0Ed} /(1-F_{H1Ed}/F_{H0Ed})$ Equation H.8

Where:

F_{H1Ed} = fictitious horizontal force, giving the same bending moments as vertical load N_{VEd} acting on the deformed structure, with deformation caused by F_{H0Ed} (first order deformation), and calculated with nominal stiffness values according to 5.8.7.2

Considering how this definition of F_{H1Ed} might apply to an imaginary cantilever of height, h, we arrive at:



1. The moment due to F_{H1Ed} is the same as that due to the vertical load N_{VEd}, so:

$F_{H1Ed} * h = N_{VEd} * \delta$

which can be rearranged to:

$F_{H1Ed} = (N_{VEd} * \delta)/h$

2. Substituting for F_{H1Ed} in Equation H.8, we have:

$F_{HEd} = F_{H0Ed} /(1 - (N_{VEd} * \delta)/(F_{H0Ed} * h)$

3. By defining $k_{amp} = F_{HEd}/F_{H0Ed}$ the above can be rearranged to:

$k_{amp} = 1/(1 - (N_{VEd} * \delta)/(F_{H0Ed} * h)$

4. Now, the EC3 Equation 5.2 for the elastic critical buckling load is:

$\alpha_{cr} = H_{Ed}/V_{Ed} * h/\delta_{HEd}$

which, when re-expressed in the terminology used in H.2 becomes:

$\alpha_{cr} = F_{H0Ed} /N_{VEd} * h/\delta_{HEd}$

and when further rearranged becomes:

$1/\alpha_{cr} = (N_{VEd} *\delta_{HEd})/(F_{H0Ed} * h)$

5. Hence $1/\alpha_{cr}$ can be substituted into the above equation for k_{amp} so that we arrive at the more well-known formula for amplification:

$k_{amp} = 1/(1 - 1/\alpha_{cr})$

NOTE Strictly, the watershed for concrete structures should be at a k_{amp} factor of 1.1 (amplification of no more than 10% due to second-order effects). Setting k_{amp} to be 1.1 and rearranging gives $\alpha_{cr} \geq 11$ i.e. a stability coefficient ≤ 0.0909 not 0.1

NOTE It is important to note that the resulting values of α_{cr} and k_{amp} are very dependent upon the analysis properties that are used and the you therefore need to carefully consider the modification factors you choose to apply via the Analysis Options.

Use of modification factors

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. Consequently design codes can require that analysis stiffness adjustment factors are applied to concrete members, (as the appropriate properties to use in analysis are load and time dependent).

For non-concrete members it is also possible that you will want to apply an adjustment to properties for various other investigations. One suggested example is the assessment of steel structures subject to corrosion. Another classic requirement in this regard relates to torsion, it is common engineering practice to assume that if it will work without assuming any torsional strength, then torsion can be ignored.

In order to make these adjustments in Tekla Structural Designer you can apply modification factors from the **Analysis Settings** dialog.

Note that while modification factors affect analysis (and hence the forces considered in design), they have no effect on Tekla Structural Designer member design checks, including deflection calculations.

USE OF MODIFICATION FACTORS IN SWAY AND DRIFT CHECKS	If the design considers building analysis, (but not staged construction analysis), the building analysis results are used both for sway and drift checks AND for member design for all combinations at both strength and service levels. The default building analysis modification factors are set for determination of second-order effects and so reflect long term properties. This means that, by default, the building analysis displacements used for the drift checks for transient loading - Wind and Seismic (ELF) - would be overestimated. To obtain more realistic deflections and hence drift checks, it is necessary to run a second analysis process after adjusting the building analysis modification factor values to reflect short term properties (similar to the default values for Modal Analysis (and RSA)). When the design considers staged construction analysis, this enables assessment of drift using short term properties and member design using long term properties in a single design process. For further details see: Staged construction modification factors.

11.3 Allowing for global imperfections

These are typically represented by the application of Notional Loads \ Equivalent Horizontal Forces.

Allowing for global imperfections (ACI/AISC)

Steel code

Steelwork design to AISC 360-10 requires an allowance for global imperfections. Columns are assumed to be out of plumb by some amount and this is replicated by applying Notional Loads, (NL), in the analysis. The requirements are given in Clause C2.2b and the value of NL is given as 0.2%. However, to accommodate the requirements of Clause C2.3 an additional 0.1% has been included - so that the actual NL used is 0.3%.

Concrete code

For concrete design to ACI 318-11 there are no requirements. However, given that a building can use mixed materials, (and even for an entirely concrete building), you have the choice to include the NLs via the combinations. By default these are to the AISC requirements - i.e. 0.3%

Allowing for global imperfections (Eurocode)

The formula to calculate the global imperfections (using EHFs) is the same for both steel and concrete, see : EC2 Cl 5.3.2 (3) a) and EC3 Cl 5.2 (5)

$\varphi = \varphi_0 \alpha_h \alpha_m$

Where:

φ_0 is the basic value of inclination.

α_h is the reduction factor for length or height : $\alpha_h = 2/\sqrt{h}$; $2/3 \leq \alpha_h \leq 1$

h is the length or height of the structure.

When height = 9m the maximum reduction of 2/3 will apply.

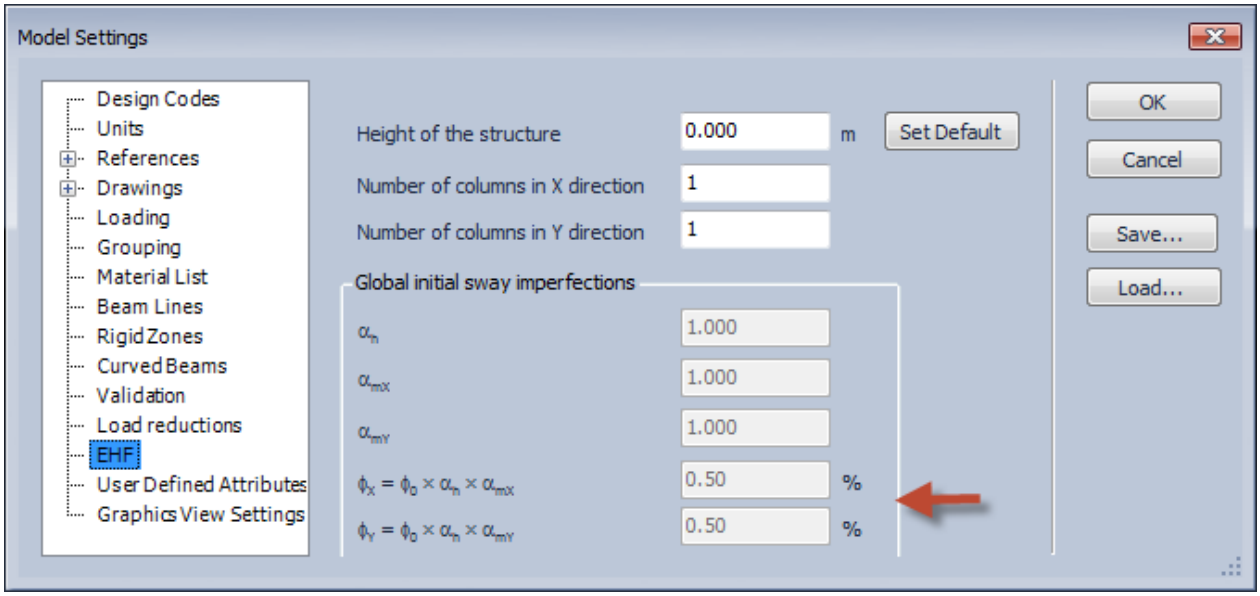
α_m is the reduction factor for number of members (EC2) or columns in a row (EC3), : $\alpha_m = \sqrt{(0.5(1+1/m))}$

m is the number of vertical members contributing to the total effect (EC2), or is the number of columns in a row including only those columns which carry a vertical load not less than 50% of the average value of the column in the plane considered (EC3).

m =	1	5	10	1000
α_m =	1	0.775	0.742	0.707

Guidance on how to count 'm' is vague and varied - however as demonstrated above, once you get above 5 or 10 it starts to make very little difference.

In Tekla Structural Designer the allowance is applied in the same manner for steel and concrete - being controlled in Model Settings as shown below:



It does require some user intervention to provide structure height and number of columns to consider. These user inputs cause adjustment of the default value of imperfection of 0.5% of the vertical load and this can be a different adjustment in the two orthogonal directions (Direction 1 and Direction 2). For example, the adjustment factor might give an EHF of 0.4% in the X-direction and 0.3% in the Y-direction.

Allowing for global imperfections (BS)

Steel code

BS5950 requires up to 0.5% of vertical loading.

Concrete code

BS8110 does not have a requirement (minimum lateral load is not the same thing).

11.4 Allowing for member imperfections

Allowing for member imperfections (ACI/AISC)

Steel code

AISC indicates that these are inherently catered for in the strut design.

Concrete code

For concrete design to ACI 318-11 there are no requirements.

Allowing for member imperfections (Eurocode)

Steel code

For steel structures designed to EC3, member imperfections are intrinsically included in the design routines for all members (beams, columns, braces). Apart from one explicit requirement, carrying out the design is all that is necessary.

The explicit requirement is from Clause 5.3.2 (6) in which member imperfection should be included as part of the analysis when the frame is sway sensitive and the axial force in members with moment connections is above a certain limit. If this situation arises Tekla Structural Designer issues a warning.

Concrete code

For concrete structures designed to EC2, explicit calculations which consider an imperfection effect are carried out as part of the design.

In Tekla Structural Designer this is achieved by adding an additional moment to the analysis results before starting design.

Allowing for member imperfections (BS)

Steel code

BS5950 inherently caters for member imperfections in the strut design curves.

Concrete code

BS8110 does not have a requirement (minimum eccentricity is not the same thing).

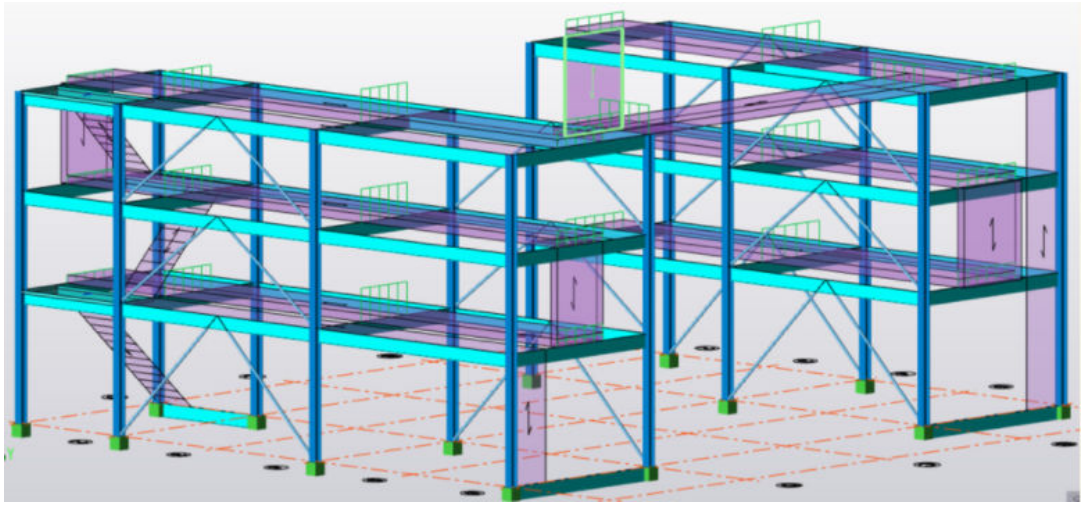
12 Industrial structures handbook

The following Tekla Structural Designer features specifically developed for industrial structures are covered here:

- Ancillaries (page 874) - these enable the quick and efficient application of loading from ancillary items such as ladders, stairs and pipework etc - that are not part of the main structural frame.
- Equipment (page 886) - these enable the quick and efficient application of loading from plant items, tanks & cylinders etc. - that are not part of the main structural frame.
- Inactive members (page 900) - these are particularly useful for industrial structures, as frequently not all the secondary beams are included in the analysis and design model, but they are still required to distribute load in the structure and act as restraints to supporting members.
- Fireproofing (page 907) - can be applied to selected members as thickness and a density.

12.1 Ancillaries

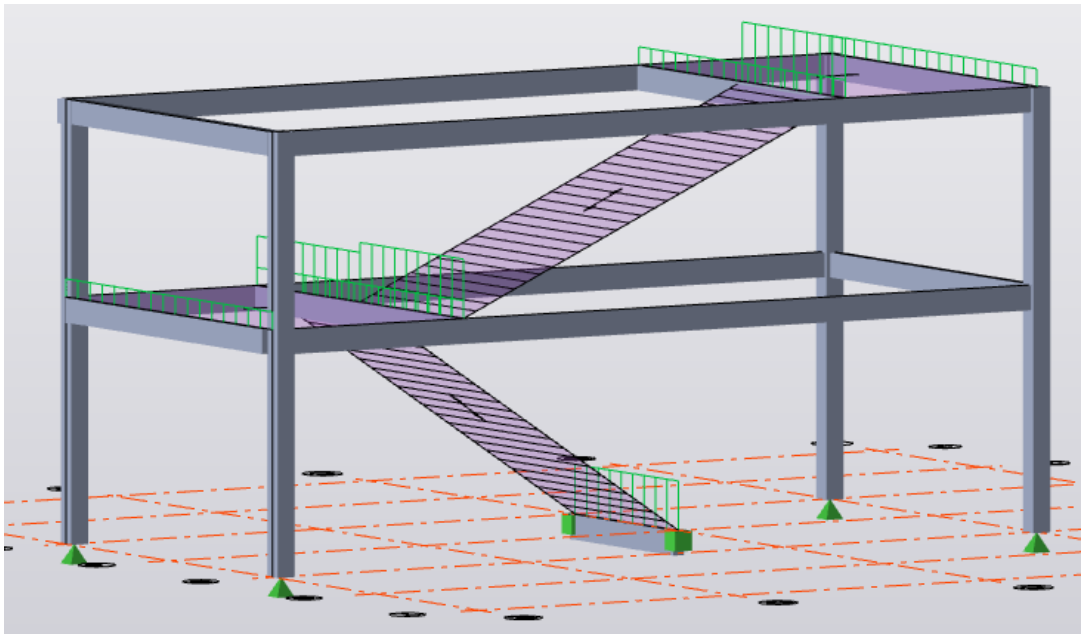
Ancillaries enable the quick and efficient application of loading from ancillary items such as ladders, stairs and pipework etc - that are not part of the main structural frame.



What are ancillaries used for?

While primarily intended for use in industrial structures, some features associated with ancillary loads are also beneficial to the AEC market - eg:

- Modeling of stairs in steel, concrete, and timber structures,
- The use of stairs in combination with inactive members.



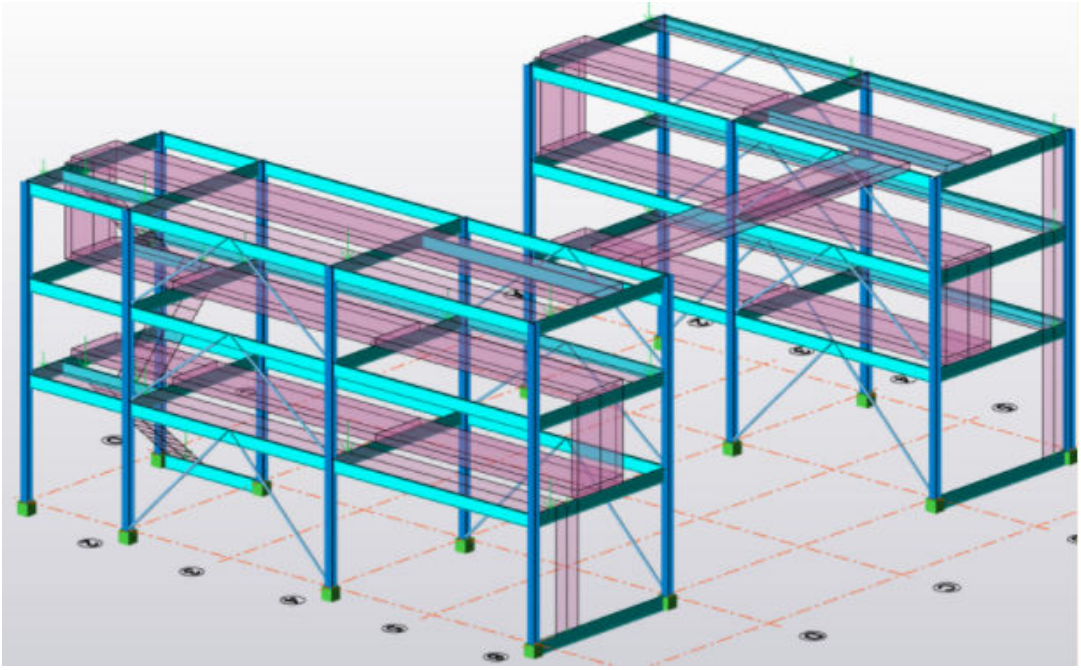
Ancillaries can also be used to model loads from:

- Walkways/Catwalks
- Ladders (with/without cages)
- Access Platforms
- Operating Platforms (Storage/Standard)
- Lines of Pipework
- Lines of Cable Tray

In Tekla Structural Designer each of the above types can be created in the form of either a *line*, or *area* ancillary.

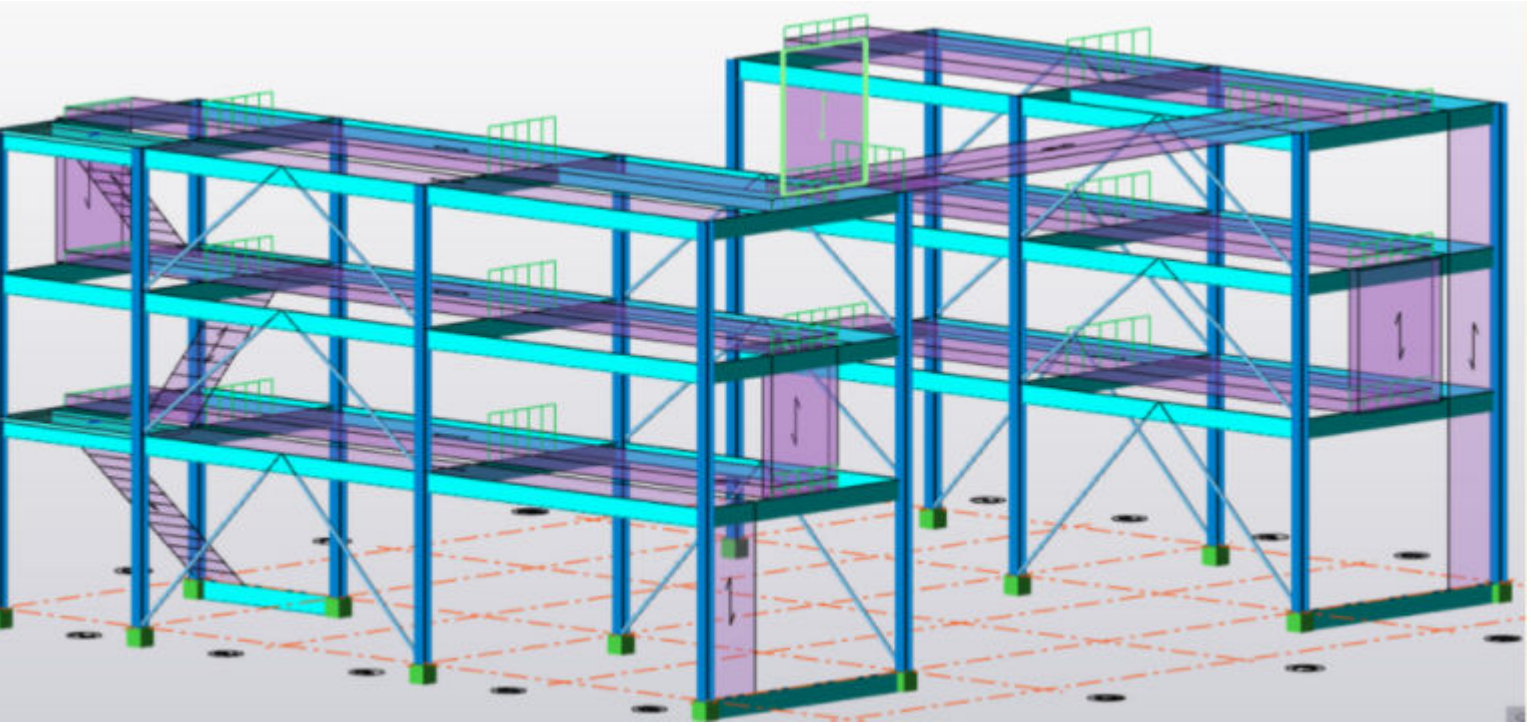
Line ancillaries

Line ancillaries can be defined horizontally, vertically or sloped. They are created by selecting a series of support points in order to create a "run". Each line has a defined width in a user defined plane (perpendicular to length).



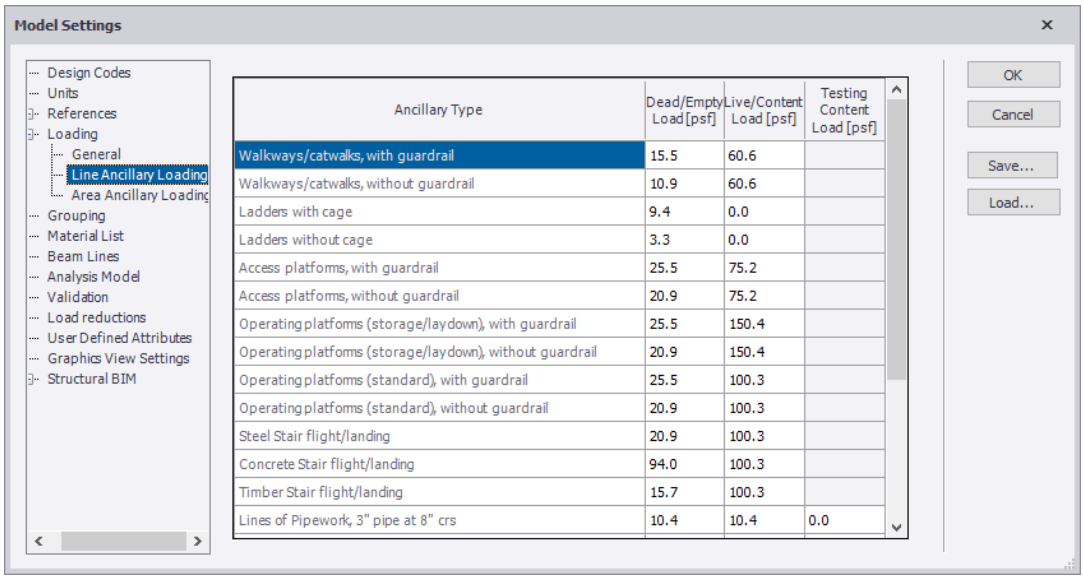
Area ancillaries

Area ancillaries can be defined horizontally, vertically or sloped. They are created in the same way as a panel, by selecting the vertices of the area.



Ancillary load default values

Each Line or Area Ancillary has default Dead Loads and Imposed/Live Loads which can be preset to be project specific from the subpages under **Loading** in **Model Settings**.

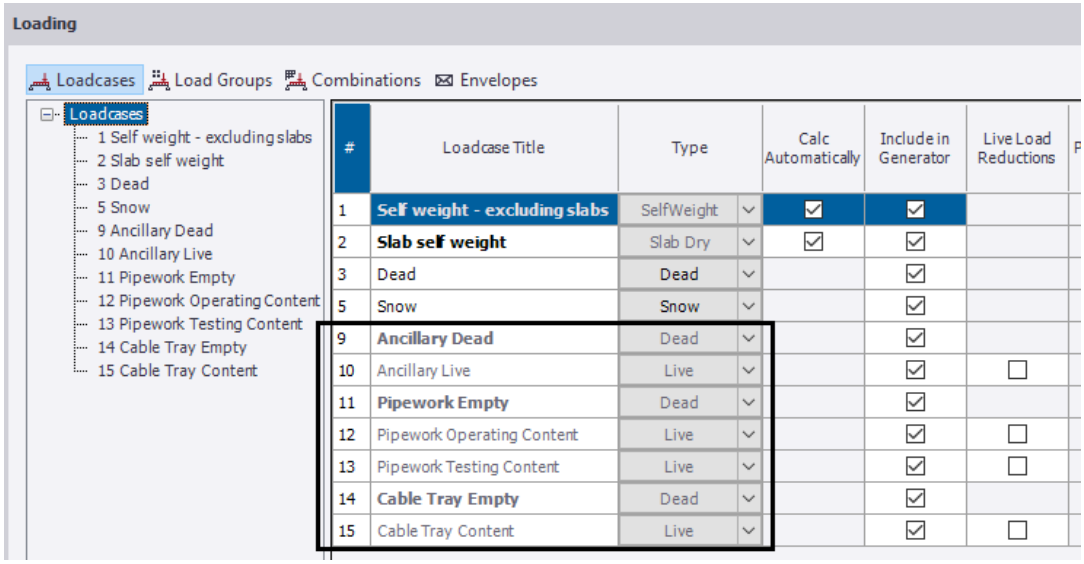


The default values can be overridden when individual ancillaries are created.

NOTE For Pipework Operating & Testing Content Loadcase Types, you can select whether these are considered as Dead or Imposed (Live) loads from **Model Settings > Loading > General**.

Ancillary loadcases

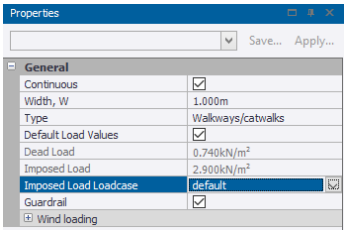
Ancillary loads are created in dedicated loadcases which are automatically added and removed as the ancillary loads are added/deleted. These dedicated loadcases specifically aid combination building for Industrial design.



Most types of ancillary load are created in the **Ancillary Dead** & **Ancillary Live (Imposed)** loadcases. The two exceptions are lines of pipework which are created in the **Pipework Empty**, **Pipework Operating Content** & **Pipework Testing Content** loadcases, and lines of cable tray which are created in the **Cable Tray Empty** & **Cable Tray Content** loadcases.

NOTE

If working to Eurocodes, the ancillary imposed case Ψ and ϕ factors default to $\Psi_1 = 1.0$, $\Psi_2 = 0.9$, $\Psi_3 = 0.8$, $\phi = 1.0$.



Some ancillaries may require different values in which case a new imposed loadcase should be manually added with the desired Ψ and ϕ factors. Then

when the ancillary load is being created the loads can be put into the new loadcase instead of the dedicated loadcase by changing the **Imposed Load Loadcase** in the **Properties** window from 'default' to that required.

Ancillary load decomposition

All decomposed loads from ancillaries are present in the analysis and design.

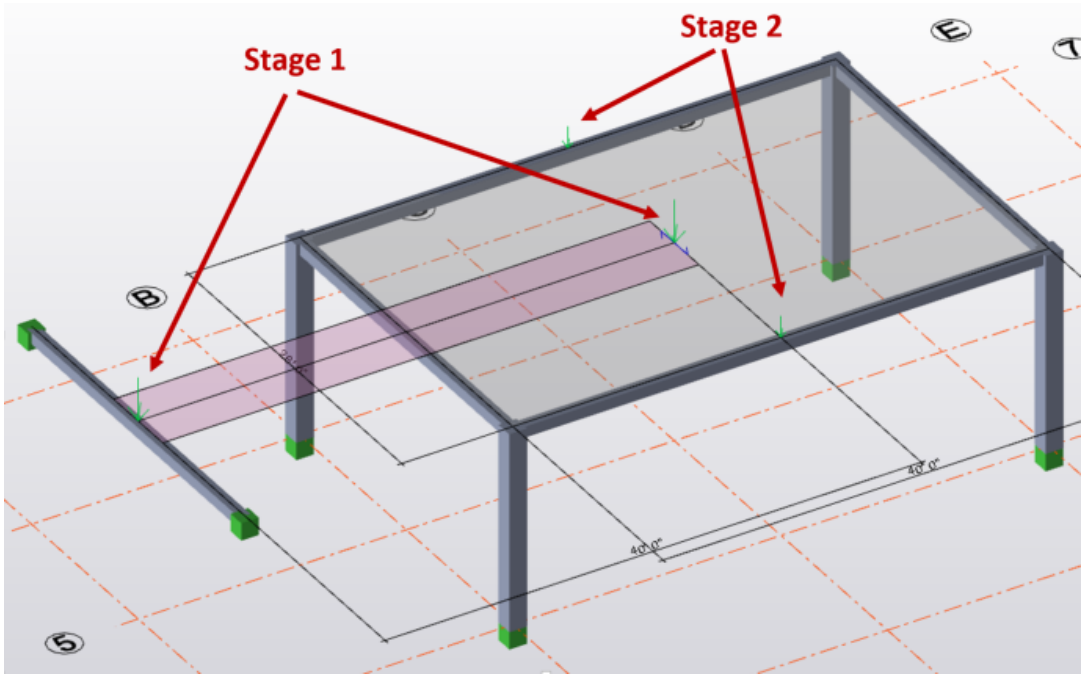
Once loads are decomposed the ancillaries themselves play no further part in analysis and design.

Line ancillary decomposition

Line ancillaries are treated as simply supported beams, spanning onto supporting members/slabs.

Decomposition is in two stages from the line ancillary to its supports then onwards.

In the below example the line ancillary is supported by a beam at one end and a slab spanning perpendicular to the ancillary at the other. At the slab end, stage 2 decomposition occurs to distribute the load on to beams supporting the slab.



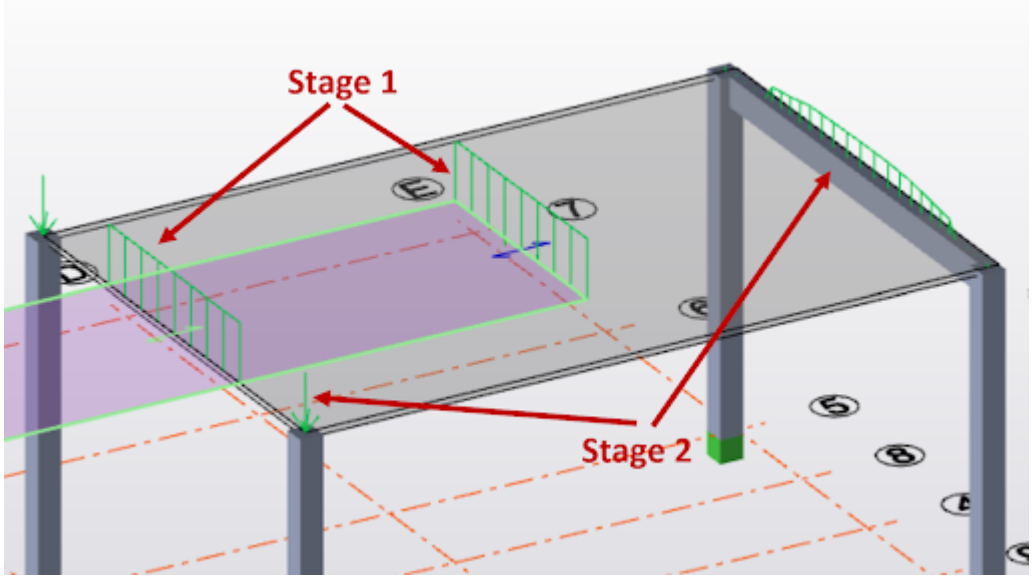
Area ancillary decomposition

In the same way as a roof panel, an area ancillary has a span direction which determines the load decomposition onto surrounding supporting members/ slabs.

Decomposition is in two stages from the area ancillary to its supports then onwards.

In the below example the area ancillary is supported by a slab which spans in the same direction as the ancillary.

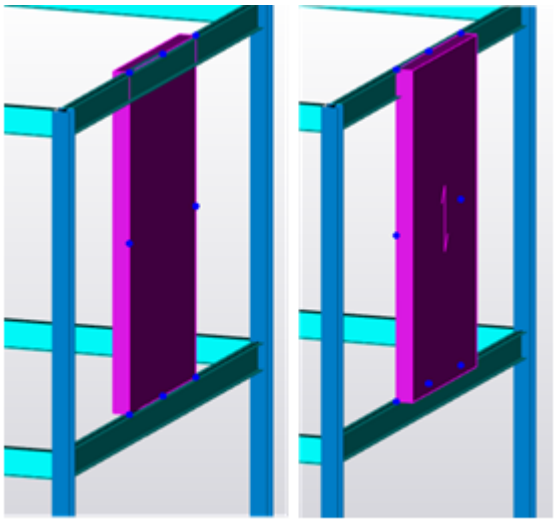
The UDLs generated in the stage 1 decomposition are then decomposed on to the members supporting the slab in stage 2.



Create line ancillary loads

1. On the **Model** tab, click **Line Ancillary**.
2. In the **Properties** window,
   a. If the load is to only span between two points, uncheck the **Continuous** setting, otherwise leave it checked.
   b. Enter the load width
   c. Select the required **Type**.
   d. Accept the default load values, or define your own.
   e. Specify the remaining properties as appropriate to selected Type.

NOTE
1. For walkways and platforms the presence of a guardrail and channel height alters dead loading.
2. For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking the Reversed property.



Reversed line of pipework or cable tray

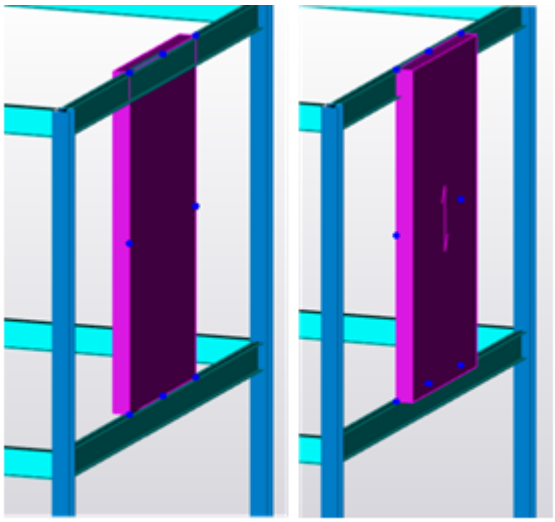
3. Click the start point of the line.
4. Click the remaining points of the line.
5. Double-click to define the end point of the line.
6. Pick a reference point to define the ancillary plane.

   Tekla Structural Designer creates a line ancillary between the selected points.

Create area ancillary loads

1. On the **Model** tab, click **Area Ancillary**.
2. In the **Properties** window,
   a. The rotation angle defines the span direction measured relative to the first two points clicked to identify the area.
   b. Select the required **Type**.
   c. Accept the default load values, or define your own.
   d. Specify the remaining properties as appropriate to selected Type.

NOTE
1. For walkways and platforms the presence of a guardrail and channel height alters dead loading.
2. For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking the Reversed property.



Reversed line of pipework or cable tray

3. Click the start point of the area.
4. Click the remaining points of the area.
5. To define the end point of the area, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates an area ancillary load between the selected points.

Generate ancillary combinations

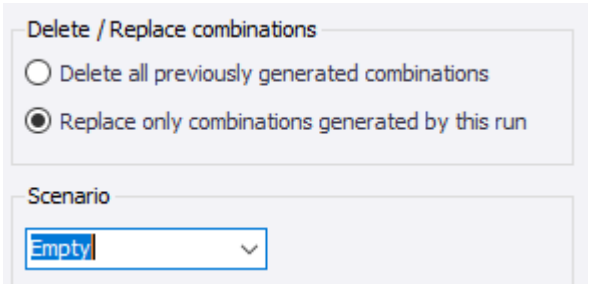
You can generate combinations automatically in the normal way, however if you have created pipework or cable tray loadcases you may need to make use of combination 'scenarios' to prevent the empty and content loadcases from being included in the same combination. The steps for doing this for pipework loadcases are illustrated below (cable tray loadcases being similar):

1. On the **Load** tab, click **Loadcases**.

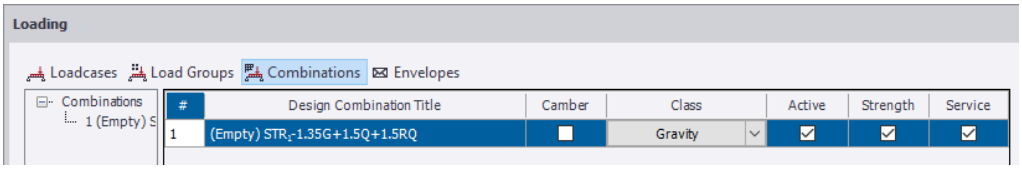
   The opens on the **Loadcases** page. On this page, you can see the loadcases that have been created and select which ones are to include when combinations are generated.
2. To generate combination(s) for the pipework empty loadcase, check 'Include in Generator' for **Pipework Empty** but uncheck the other pipework loadcases.

Loadcase Title	Type	Calc Automatically	Include in Generator
Pipework Empty	Dead		☑
Pipework Operating Content	Dead		☐
Pipework Testing Content	Dead		☐

3. At the top of the dialog, click  **Combinations**.
4. Click **Generate**.
5. In the Combination Generator, enter an appropriate name for the 'empty' scenario and select the **Replace only combinations generated by this run** option.



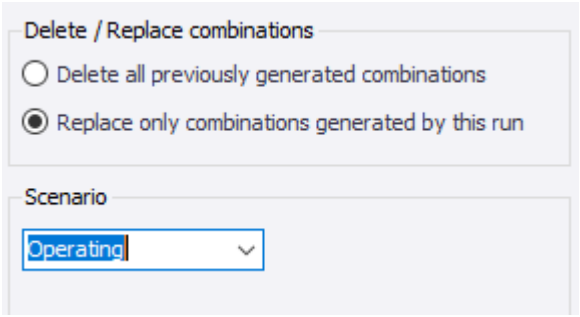
6. Make appropriate selections to generate the combination(s) required for this scenario, then click **Finish** to see the result.



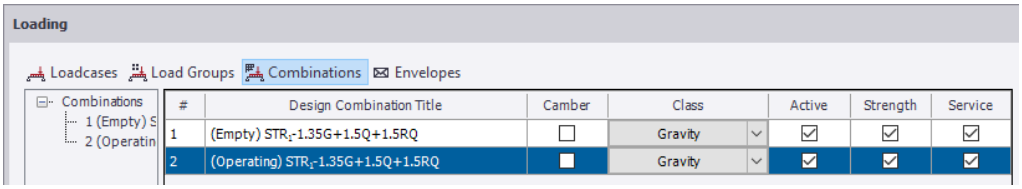
7. Click  **Loadcases** to return to the **Loadcases** page.
8. Check 'Include in Generator' for **Pipework Operating Content** but uncheck the other equipment loadcases.

Loadcase Title	Type	Calc Automatically	Include in Generator
Pipework Empty	Dead		☐
Pipework Operating Content	Dead		☑
Pipework Testing Content	Dead		☐

9. On the **Combinations** page, click **Generate**.
10. In the Combination Generator, enter an appropriate name for the 'operating' scenario and select the **Replace only combinations generated by this run** option.



11. Make appropriate selections to generate the combination(s) required for this scenario, then click **Finish** to see the result.

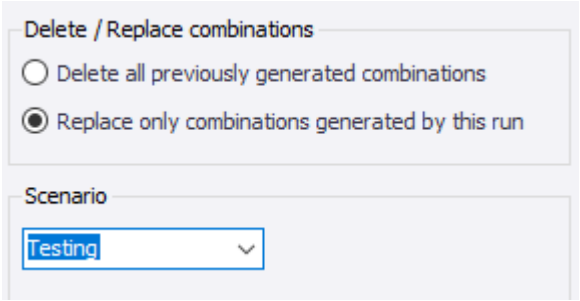


12. On the **Loadcases** page, check 'Include in Generator' for **Pipework Testing Content** but uncheck the other pipework loadcases.

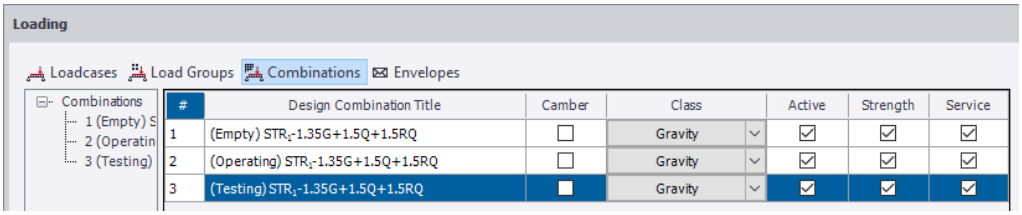
Loadcase Title	Type	Calc Automatically	Include in Generator
Pipework Empty	Dead		☐
Pipework Operating Content	Dead		☐
Pipework Testing Content	Dead		☑

13. On the **Combinations** page, click **Generate**.

14. In the Combination Generator, enter an appropriate name for the 'testing' scenario and select the **Replace only combinations generated by this run** option.



15. Make appropriate selections to generate the combination(s) required for this scenario, then click **Finish** to see the result.



16. Click **OK**

Create an ancillary loads report

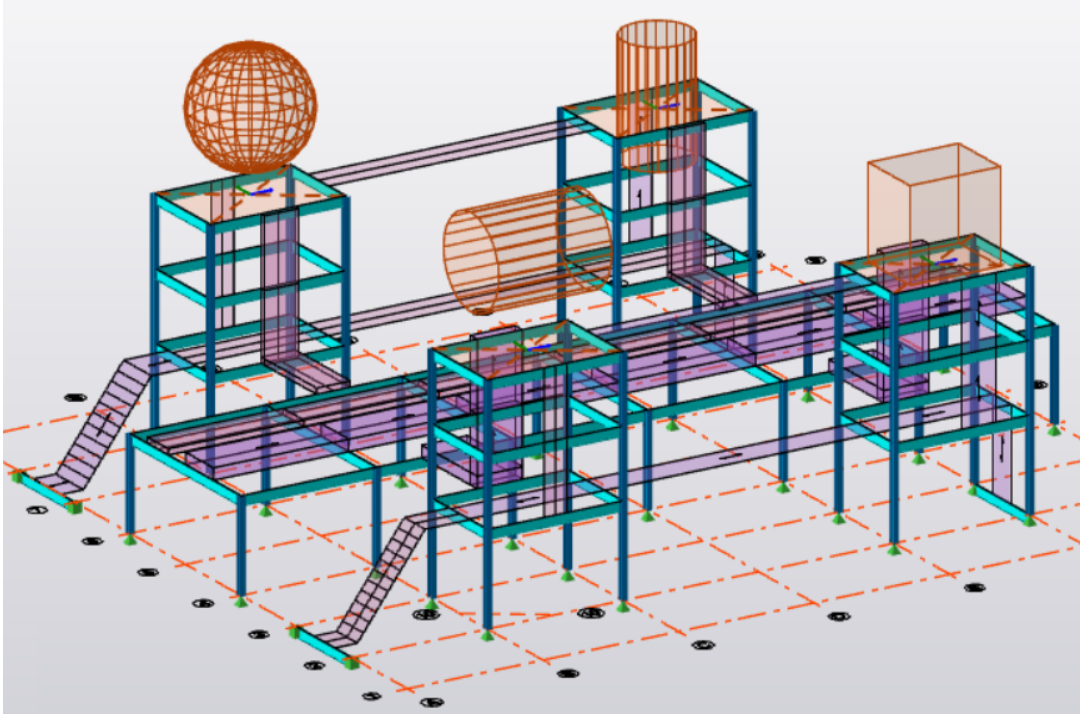
A table of the ancillary loads applied to the model is available in the **Industrial Structure Loading** report, which is created as follows:

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.

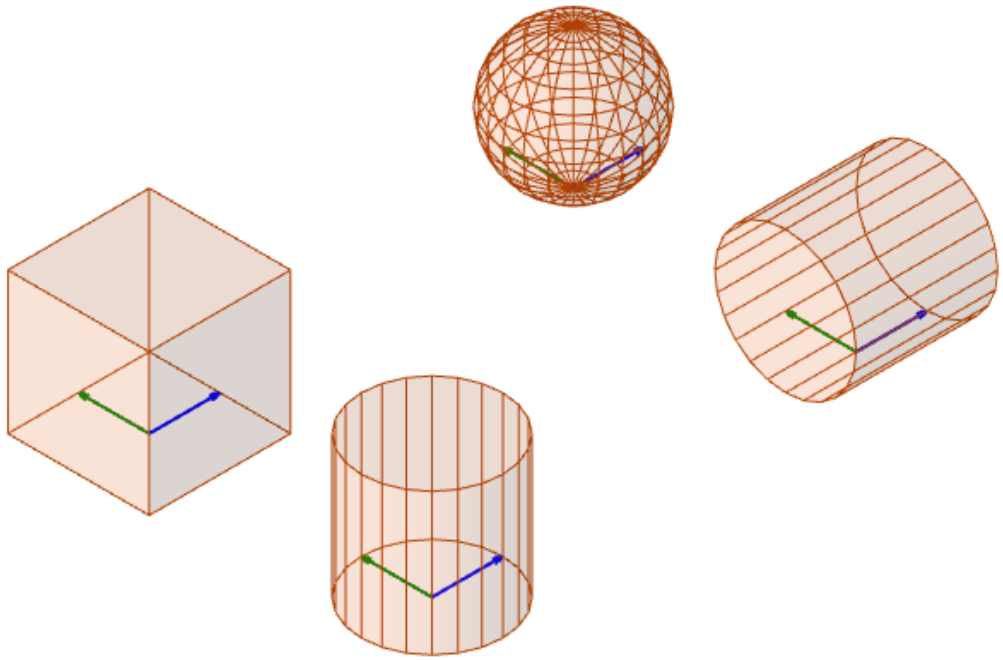
12.2 Equipment

Equipment is primarily intended for use in industrial structures, enabling the quick and efficient application of loading from plant items, tanks & cylinders etc. - that are not part of the main structural frame.



Overview

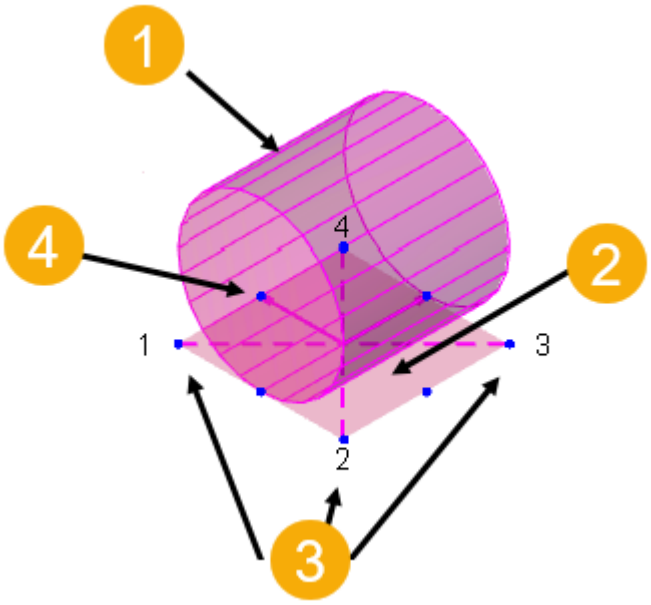
Equipment can be displayed in Tekla Structural Designer in the form of a sphere, vertical cylinder, horizontal cylinder or cuboid.



Equipment has an associated Center of Gravity (CoG) which is indicated by the user co-ordinate system (UCS) as shown above.

Equipment also has a number of support points through which load is transferred to the structure. At least three support points are required and there is no limit on the maximum. The area bounded by these support points is referred to as the 'Loading Area'.

The support points are numbered and become active when the equipment is selected. Unnumbered nodes at the mid-points of loading area edges also become active.

Equipment support points and loading area	
	
1	Equipment
2	Loading area
3	Support points
4	Unnumbered node **NOTE** **Unnumbered** nodes are provided to facilitate editing of the Loading Area - they are **not** support points.

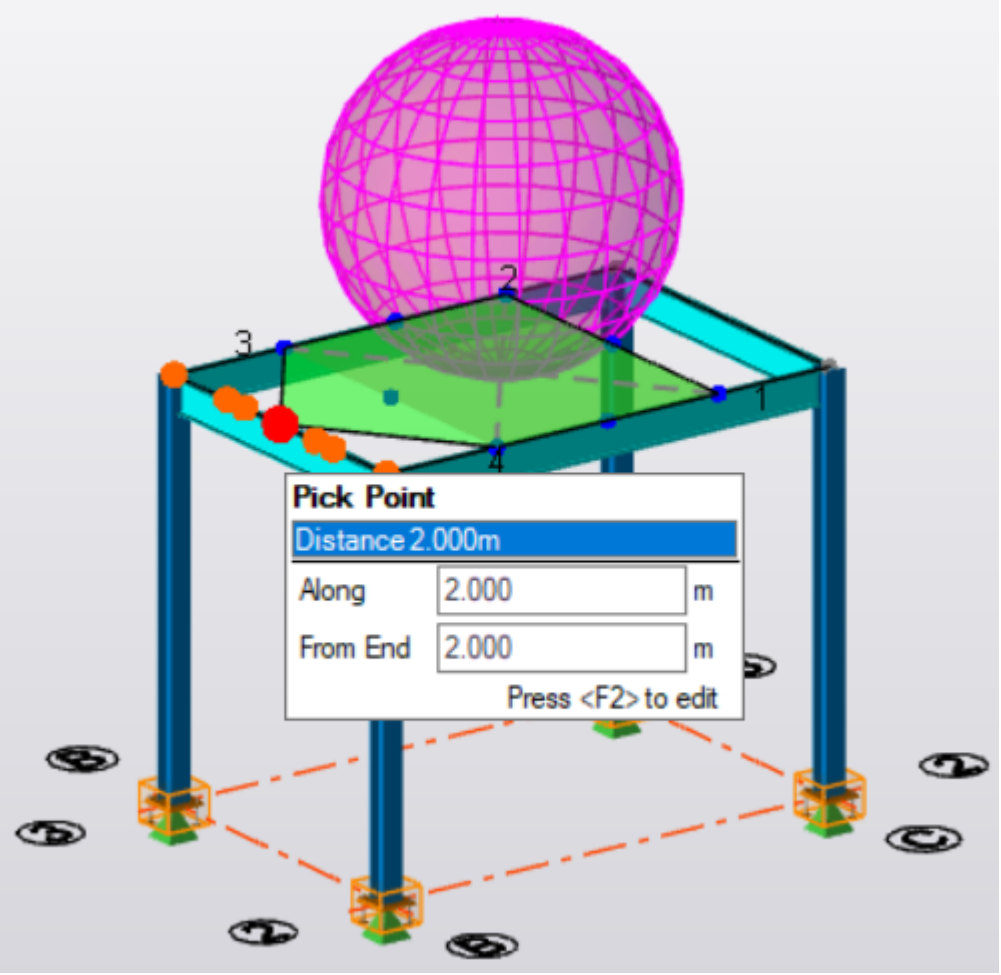
The 3D geometry, UCS and loading area can be switched on or off for each equipment shape within Scene Content.

Once placed, equipment is listed in the Project Workspace Structure Tree under the Loading Elements group.

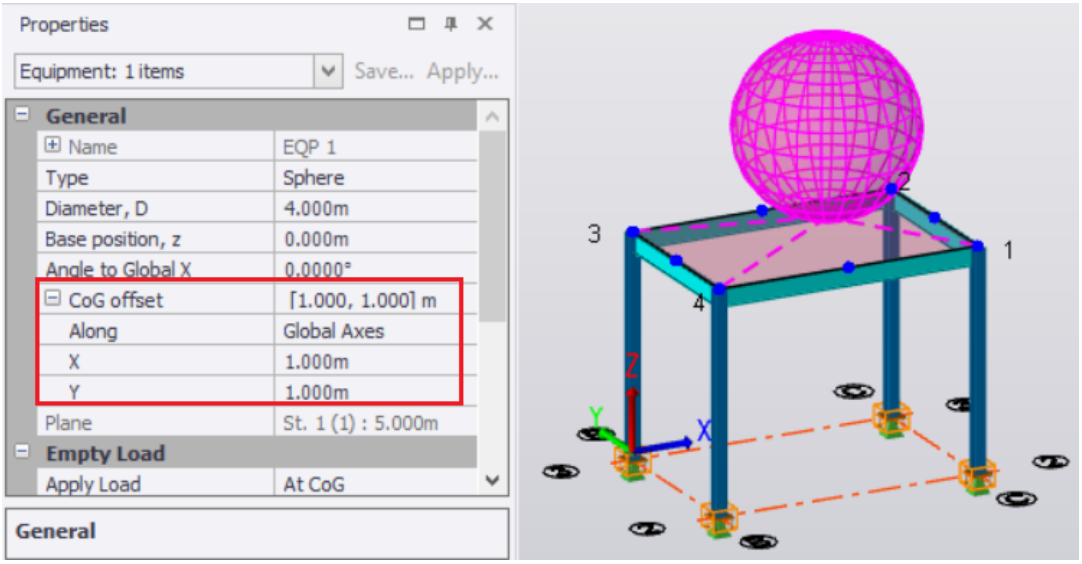
Modeling

Equipment must be defined in the horizontal plane.

Once it has been defined, you can alter the shape of the loading area by dragging any of its nodes. Dragging a numbered support point moves it, dragging an unnumbered node (as shown below) introduces a new support point.



By default the CoG of the equipment and the CoG of the support points coincide. You can specify an offset to the equipment CoG to facilitate the adjustment of equipment loads on the structure.



Equipment must be placed in such a way that each support point is itself supported by a structural item or slab item.

No member, ancillary, slab, wall or other equipment may be attached to an equipment entity.

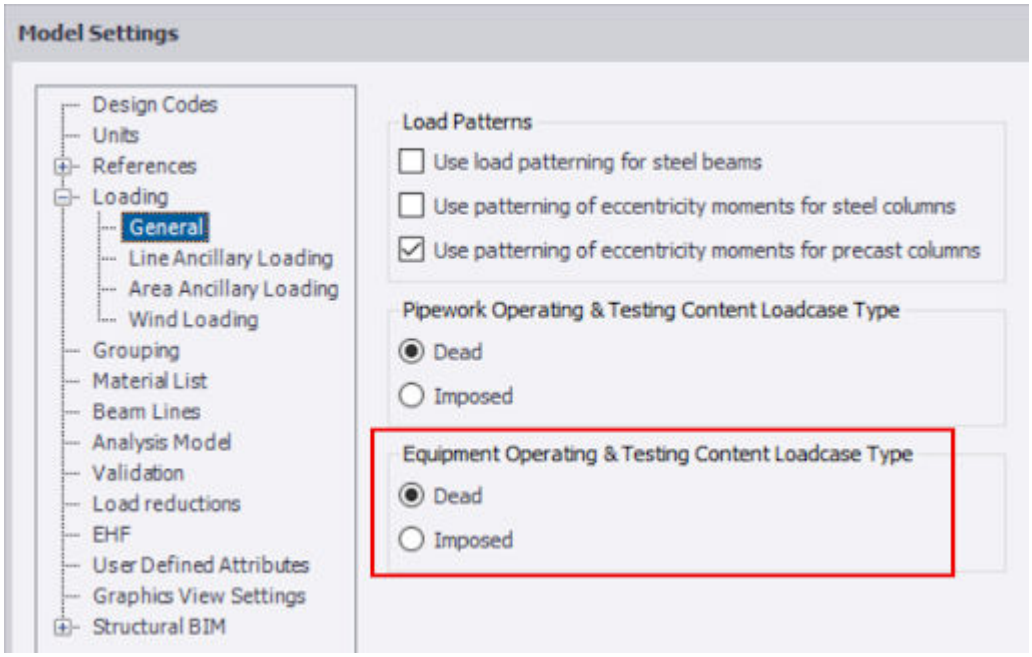
Equipment loadcases

Equipment loads have three dedicated gravity loadcases which are automatically added and removed as the equipment loads are added/deleted.

- Equipment Empty
- Equipment Operating Content
- Equipment Testing Content

These dedicated loadcases specifically aid combination building for Industrial design.

NOTE For Equipment Operating & Testing Content Loadcase Types, you can select whether these are considered as Dead or Imposed (Live) loads from **Model Settings > Loading > General**.



When equipment is added, initially the loads in the dedicated loadcases default to zero. When the load is specified it can either be applied at the CoG of the equipment, or at the equipment support points. This is important when the CoG is different between the loaded and unloaded conditions.

Once equipment has been defined, additional horizontal or vertical point loads can be applied to it in other dead and imposed/live loadcases.

The **Equipment**  load command for doing this is located on the **Load** ribbon in the **Structure Loads** group.

Equipment load decomposition

All decomposed loads from equipment are present in the analysis and design.

Once loads are decomposed the equipment itself plays no further part in analysis and design.

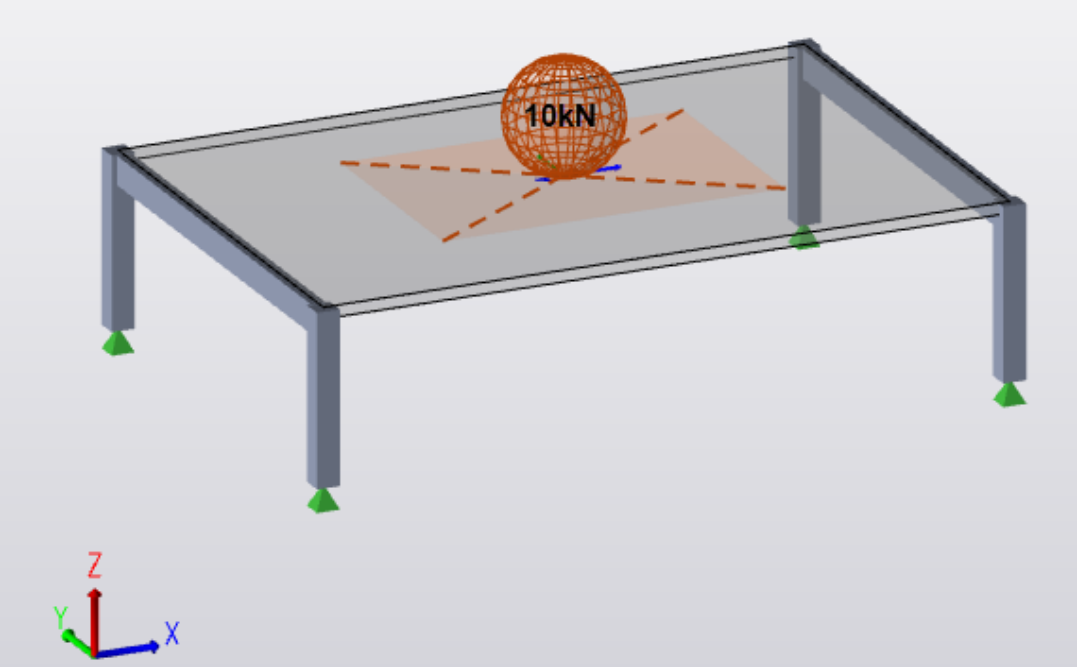
Equipment decomposition for loads applied at CoG

Decomposition is in two stages: from the equipment to the equipment support points, then into the structure.

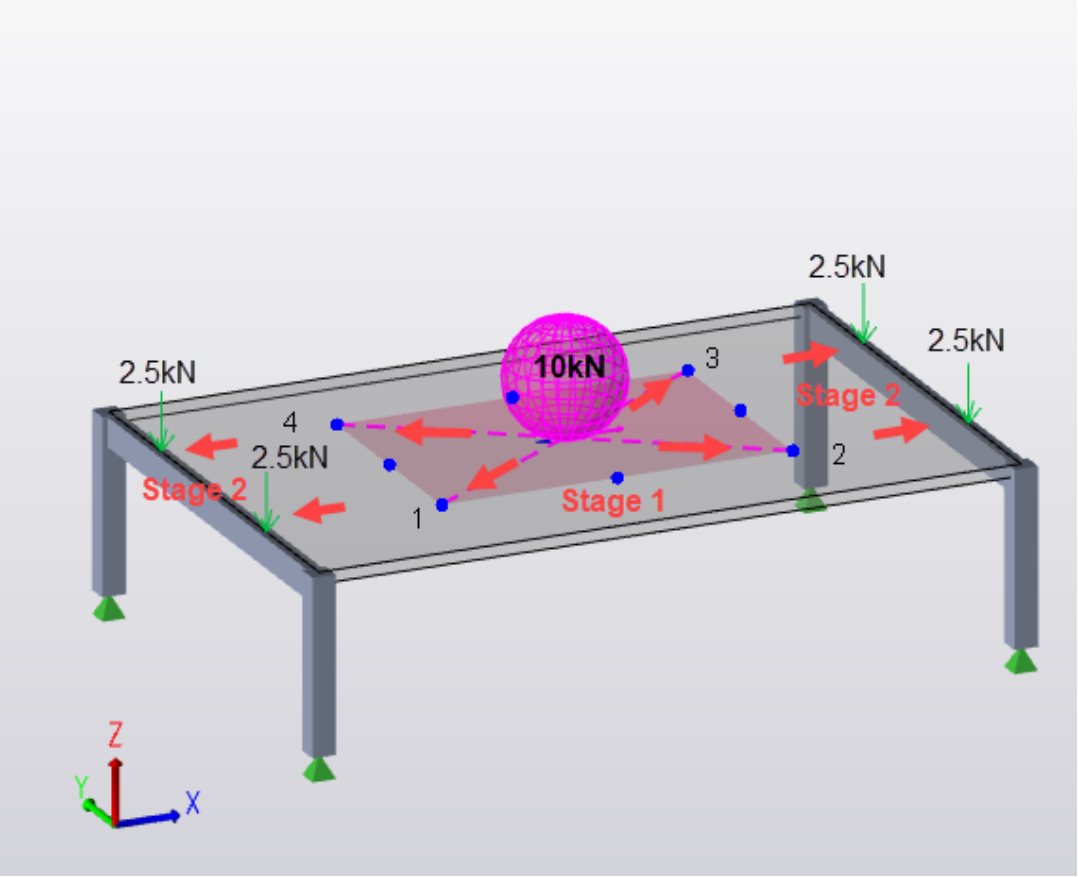
For each equipment, the load distribution distributes the total load proportionately to the equipment support points based on an iterative decomposition approach in which the loading area acts as rigid.

NOTE The distribution of a point load applied to a rigid plate is a very complex problem to solve, if done by FE then results are determined by stiffness of the plate. The iterative approach is approximate but believed to be sufficiently accurate for the purpose. If you do not want to use this approach, the loads should be applied at supports instead.

In the following example equipment is placed on a one-way slab which spans in the global X direction. The equipment has 4 support points positioned symmetrically about the slab center-line. A load of 10kN is applied at the equipment CoG.

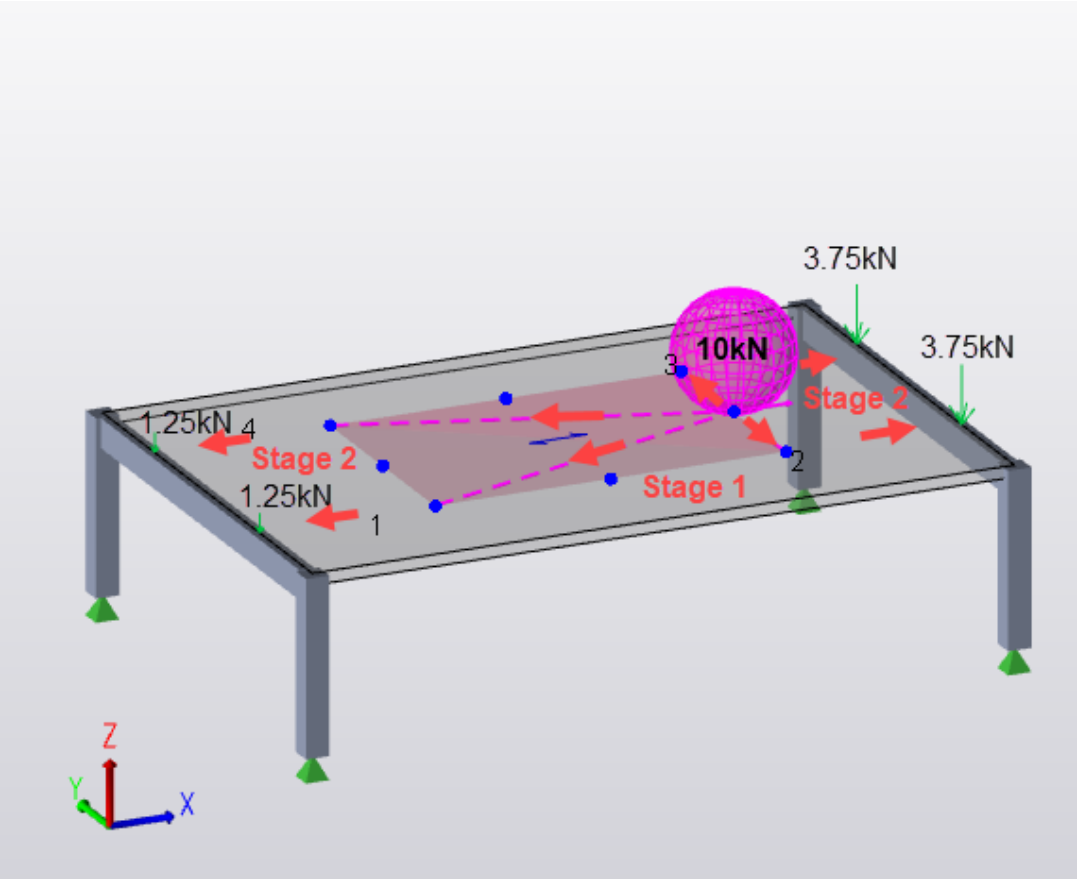


When the model is analyzed the equipment load is decomposed to the equipment support points, then on to the supporting members as shown below.



- **Stage 1** - the 10kN load is decomposed to the 4 numbered support points, due to symmetry these each receive 2.5kN.
- **Stage 2** - support point loads are decomposed on to the members supporting the slab. Again due to symmetry, the resulting member point loads are each 2.5kN

If the CoG of the equipment is offset along the global X axis so that it now lies at the edge of the loaded area, when the model is re-analyzed the decomposed loads are updated as shown below:



- **Stage 1** - because the CoG has been offset, support points 2 and 3 each receive 5kN whereas support points 1 and 4 remain unloaded.
- **Stage 2** - support point loads are decomposed on to the members supporting the slab, resulting in 2 point loads of 1.25kN on the left, and 2 point loads of 3.75kN on the right.

Equipment decomposition for loads applied at supports

In this case, only the second stage decomposition is required on to the supporting members.

Create equipment and equipment loads

1. Open a 3D View, or 2D Level View in which to create the equipment.

   NOTE Equipment must be defined in a horizontal plane, and so cannot be created in a 2D Inclined Plane View.

2. On the **Model** tab, click  **Equipment**.
3. In the **Properties** window, define the equipment shape and size:
   a. Select the required **Type**.
   b. Enter the geometric properties (Length, Breadth, Height, or Diameter) required to define the size.
   c. Leave **Base position, z** as 0 to place the equipment directly on it's loaded plane, or enter a non-zero value to raise/lower it.
   d. Leave **Angle to Global X** as 0 , or enter an angle to rotate it.
4. In the **Properties** window, select whether the empty, operating content, and testing loads are to be applied at the center of gravity of the equipment, or through the supports.
5. Define the equipment load values:
   - If applied **At CoG**, enter each value.
   - If applied **At Supports**, click the ellipsis button  then enter the required value at each support.
6. In the 3D or 2D View, click the start point of the loaded area.
7. Click the remaining points of the loaded area.
8. To define the end point of the loaded area, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   NOTE At least three support points are required, no limit on maximum.

   Tekla Structural Designer creates the equipment positioned at the center of gravity of the loaded area.

Create additional equipment loads in other loadcases

Once equipment has been defined you can apply additional loads to it in loadcases other than the dedicated 'equipment' ones. These loads can be applied in any global or local direction.

1. In the **Loading** list, select an appropriate loadcase.

NOTE This cannot be one of the existing dedicated 'equipment' loadcases.

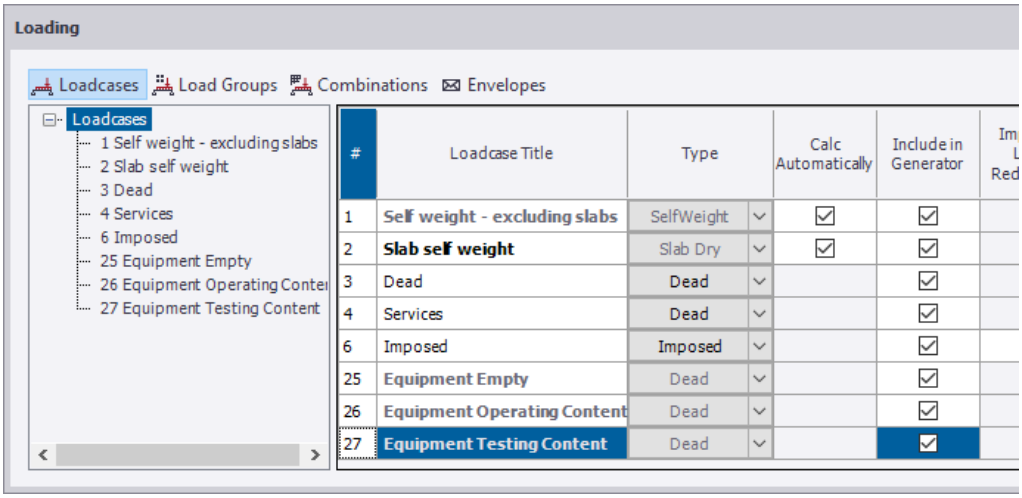
2. On the ribbon, go to the **Load** tab.
3. Click  **Equipment**.
4. In the **Properties** window, select the load direction.
5. In the **Properties** window, select whether the load is to be applied at the center of gravity of the equipment, or through the supports.
6. Define the load value:
   - If applied **At CoG**, enter the load value and the **Offset from mid height** above or below the loaded area at which the load is to be applied.
   - If applied **At Supports**, click the ellipsis button  then enter the required value at each support.
7. In a 2D or 3D View, pick the equipment to apply the load to.

Generate equipment combinations

If you are going to generate combinations automatically you will need to make use of combination 'scenarios' to prevent equipment empty, operating content and testing content loadcases from being included in the same combination. The steps for doing this are as follows:

1. On the **Load** tab, click  **Loadcases**.

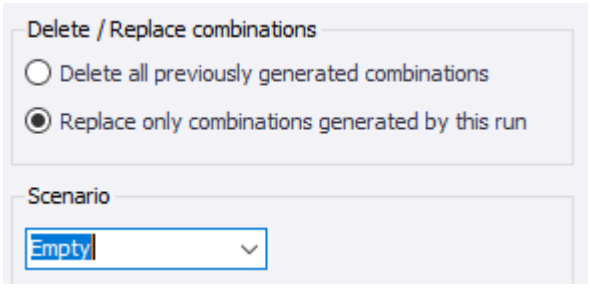
The opens on the **Loadcases** page. On this page, you can see the loadcases that have been created and select which ones are to include when combinations are generated.



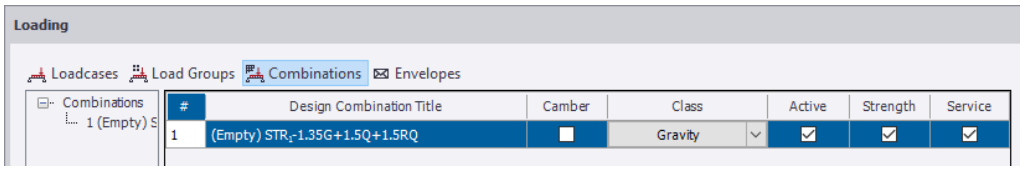
2. To generate combination(s) for the equipment empty loadcase, check 'Include in Generator' for **Equipment Empty** but uncheck the other equipment loadcases.



3. At the top of the dialog, click **Combinations**.
4. Click **Generate**.
5. In the Combination Generator, enter an appropriate name for the 'empty' scenario and select the **Replace only combinations generated by this run** option.



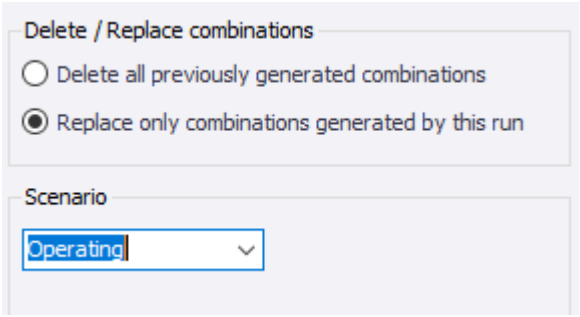
6. Make appropriate selections to generate the combination(s) required for this scenario, then click **Finish** to see the result.



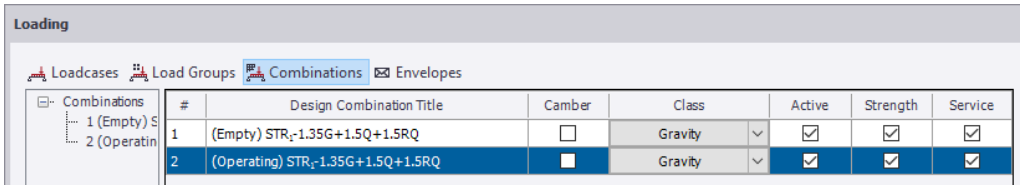
7. Click **Loadcases** to return to the **Loadcases** page.
8. Check 'Include in Generator' for **Equipment Operating Content** but uncheck the other equipment loadcases.



9. On the **Combinations** page, click **Generate**.
10. In the Combination Generator, enter an appropriate name for the 'operating' scenario and select the **Replace only combinations generated by this run** option.



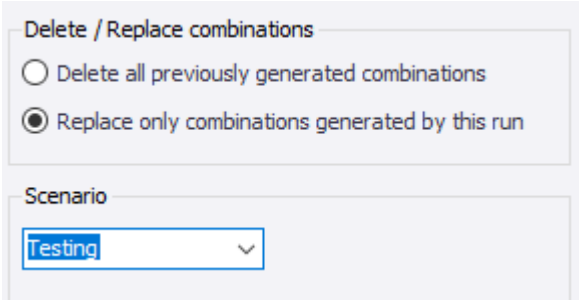
11. Make appropriate selections to generate the combination(s) required for this scenario, then click **Finish** to see the result.



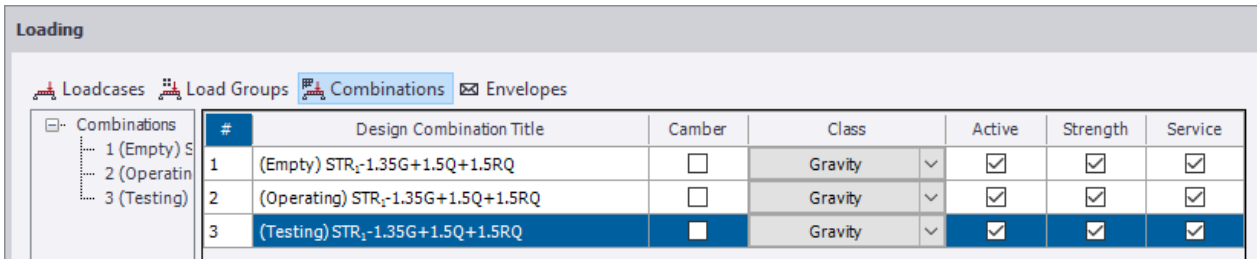
12. On the **Loadcases** page, check 'Include in Generator' for **Equipment Testing Content** but uncheck the other equipment loadcases.



13. On the **Combinations** page, click **Generate**.
14. In the Combination Generator, enter an appropriate name for the 'testing' scenario and select the **Replace only combinations generated by this run** option.



15. Make appropriate selections to generate the combination(s) required for this scenario, then click **Finish** to see the result.



16. Click **OK**

Create an equipment loads report

A table of the equipment loads applied to the model is available in the **Industrial Structure Loading** report, which is created as follows:

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.

12.3 Inactive members

Single span members can be made inactive for analysis and design while still being kept in the model for load distribution and for determination of effective lengths.

This feature is particularly useful for industrial structures, as frequently not all the secondary beams are included in the analysis and design model, but they are still required to distribute load in the structure and act as restraints to supporting members.

An inactive member is totally ignored in the solver model as is any load carried by it. Therefore, to allow the load to be distributed it is instead decomposed to nodal forces in a pre-analysis decomposition stage - this happens for all applied loads on inactive members on the basis of the members being simply supported.

Which members can be made inactive?

Only single span beams, braces, analysis elements, purlins and rails have the potential to be inactive. They each have an **Active** setting in the properties which defaults to 'on' but can be unchecked.

NOTE Members cannot be made inactive if:

- they are multi-span, or,
- they support another member (active or inactive)

To make a member inactive

By selecting the Show/Alter State Active attribute in a **Review View**, inactive members are color coded allowing the active/inactive setting to be toggled graphically.

Alternatively, you can change the setting manually in the **Properties** window as follows:

1. Select the member in a 2D or 3D view.
2. In the **Properties** window deselect **Active**.

Inactive member load decomposition

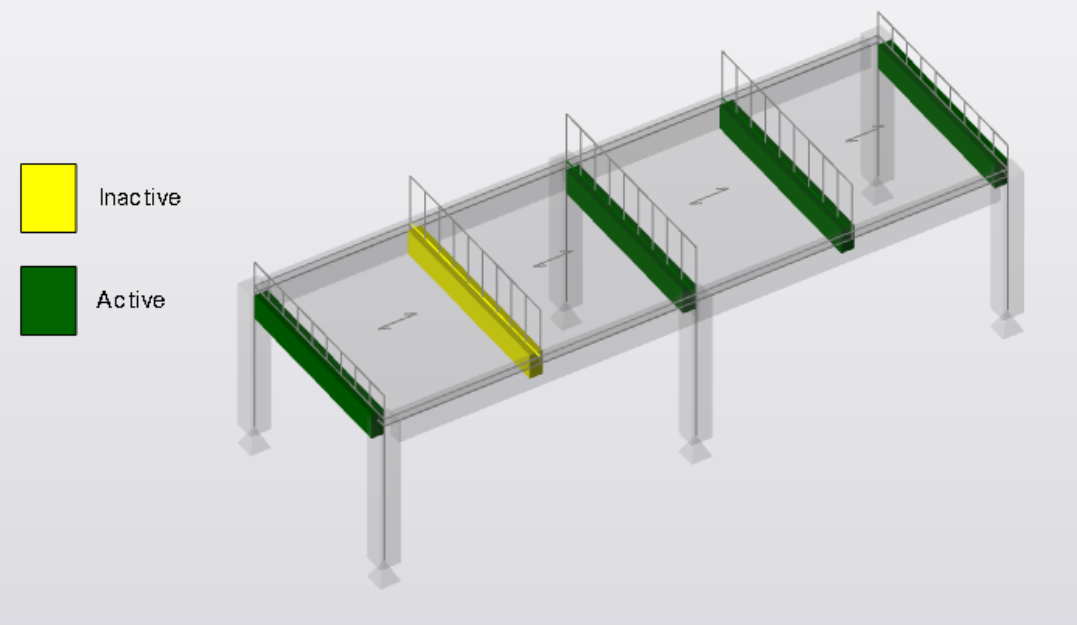
The first stage of load decomposition treats active and inactive members in exactly the same way.

This is then followed by a second stage of load decomposition for the inactive members only, as follows:

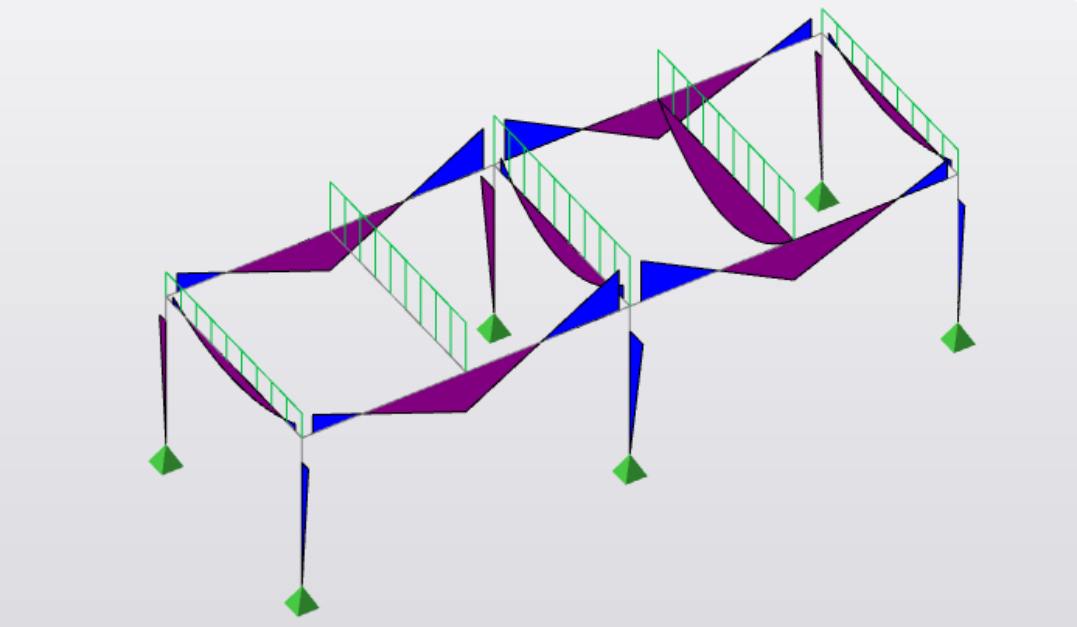
- End reactions are calculated for the inactive members.
- These are then applied as nodal forces to the supporting members.

After decomposition has taken place analysis can proceed with the inactive members removed from the solver model.

In the below example, by using the Show/Alter State Active attribute one beam has been made inactive. The view shows decomposed loads - note that the inactive beam still has slab load decomposed to it in the same way as the active beams (as a result of the first decomposition stage).



The effects of the second decomposition stage can be seen by switching to a Results View showing major moments.



In the second stage the end reactions of the inactive beam have been applied as nodal forces to the supporting edge beams in the first bay causing moments to be generated in them, (without this second stage the two supporting beams would have otherwise been completely unloaded.)

No moments are generated in the inactive beam itself because after the decomposition has taken place it does not exist in the solver model.

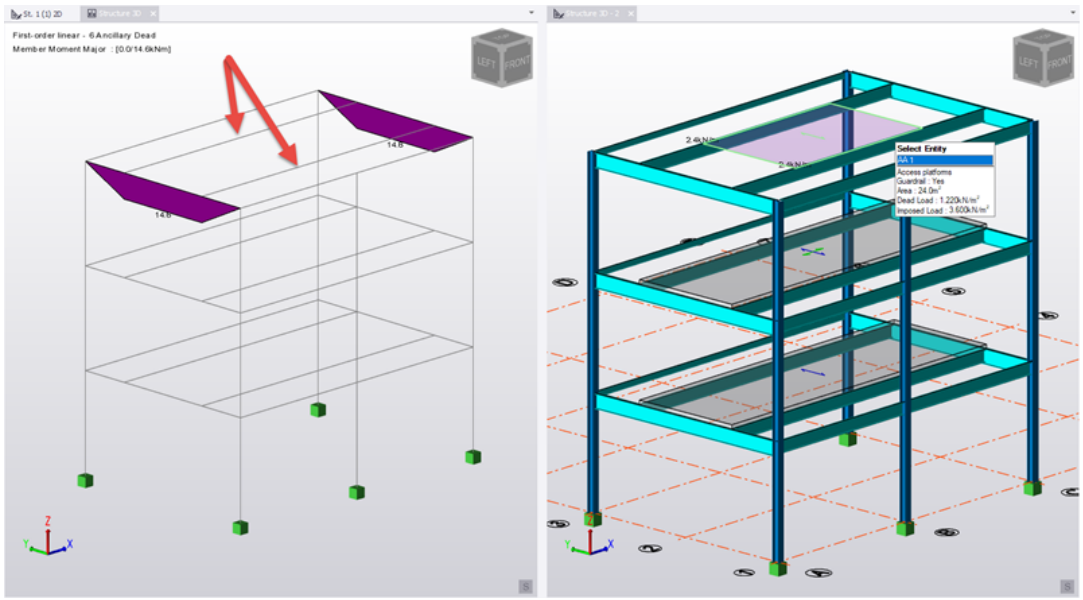
Typical usage cases for inactive members

There are potentially many uses for inactive members, the following being some examples.

Industrial Structures

Inactive members can be used in conjunction with ancillaries in order to apply loads to the structure while not themselves participating in the analysis.

In the below example an area ancillary decomposes load on to two inactive beams, which is then applied as nodal forces on to the supporting members.

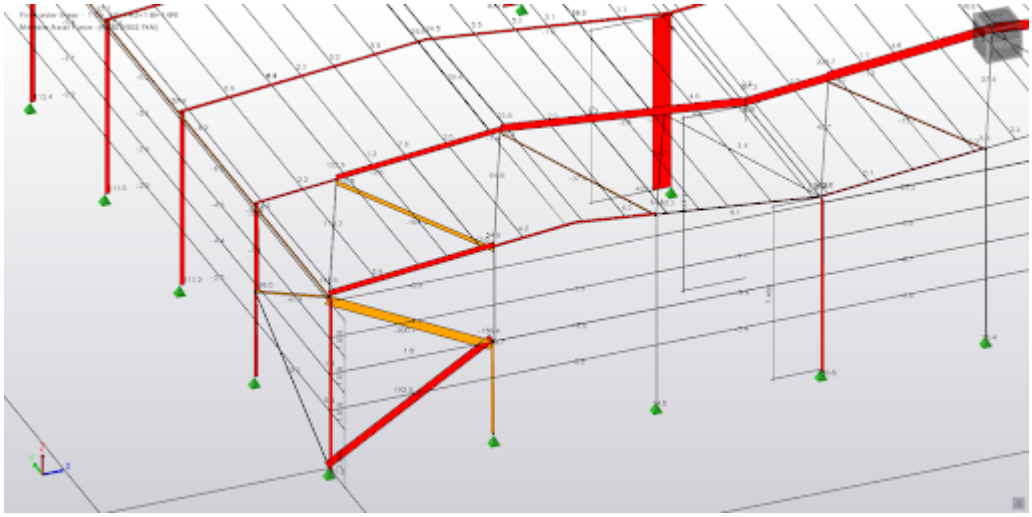


Purlins and Rails

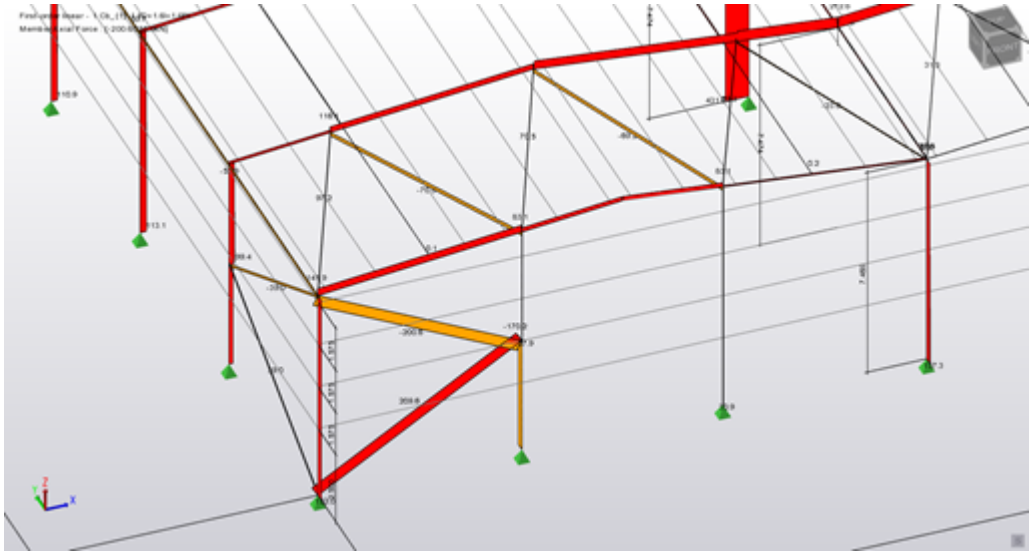
In large portal sheds, purlins and rails may be included in the model to act as a means of load distribution; the engineer would not want to see the forces in them when reviewing forces in the main structural members, or consider the small axial loads that develop in purlins and rails when gravity loadcases are analysed.

By setting the purlins and rails to be inactive these goals are achieved.

This is demonstrated in the below example - in the top view the purlins are rails are active, consequently small forces develop in them in gravity combinations and the results views are quite cluttered with values that are of no interest to the engineer.



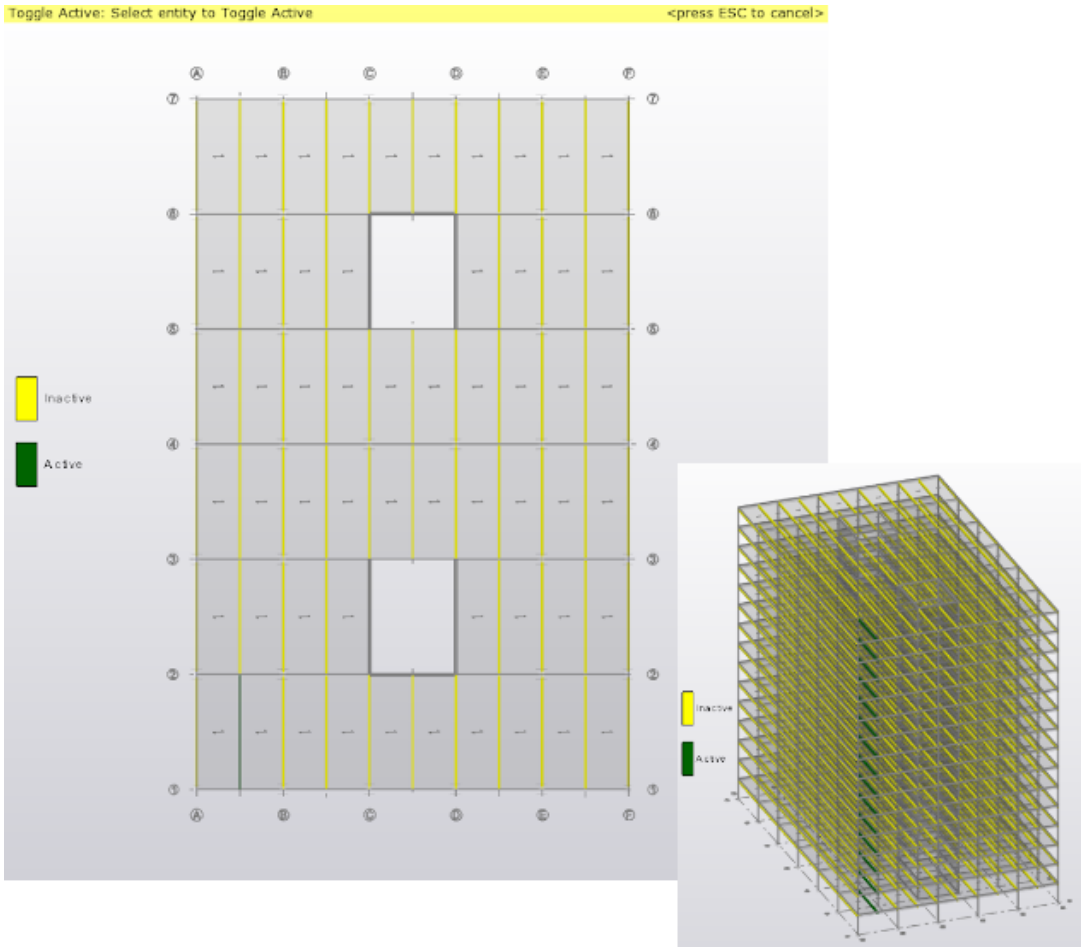
In the second view the purlins and rails have been set inactive, providing a clearer display and showing the values the engineer would expect/assume from such a combination.



Grouped Design

Design can be sped up by enabling group design and limiting the number of active members in a group.

This is demonstrated in the below example of a regular floor, all beams in the group apart from one have been made inactive which significantly reduces the design time.

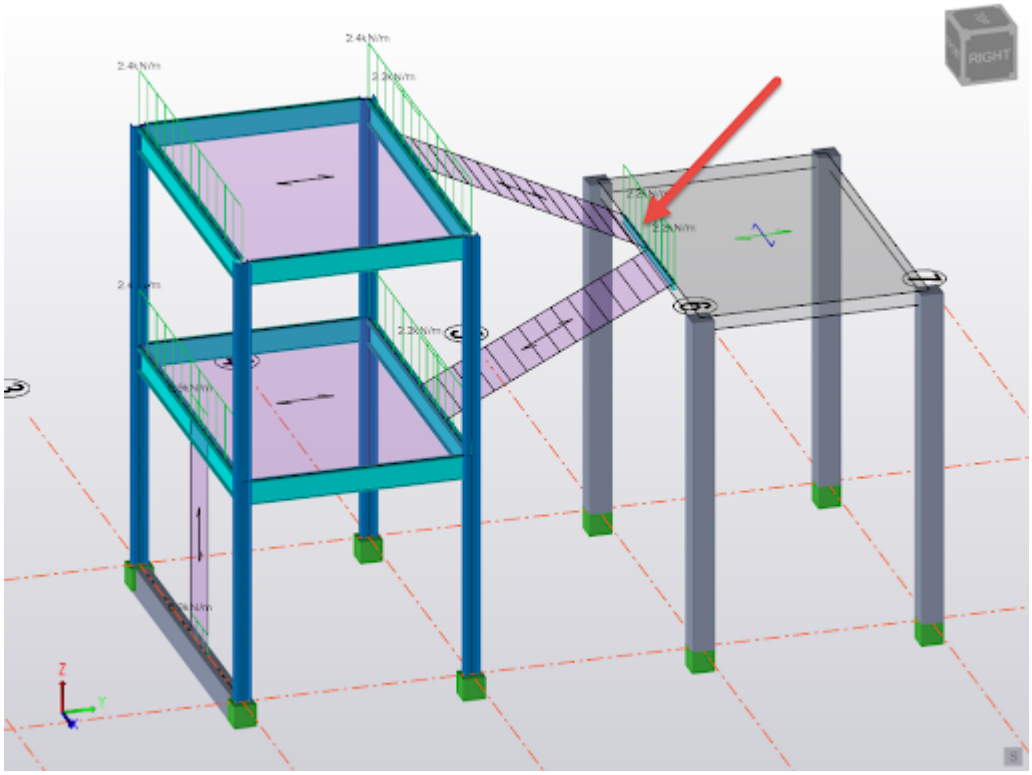


NOTE When making the non-critical beams inactive to speed up design:

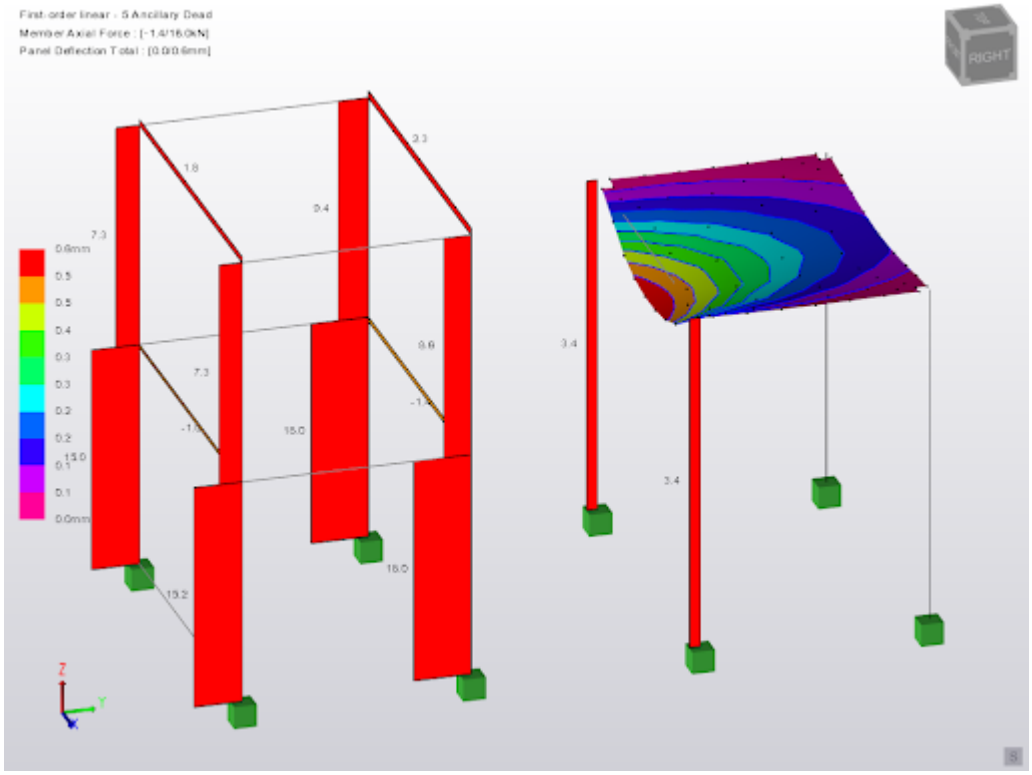
- You are responsible for choosing critical beam(s)
- Primary beams cannot be set to inactive

To support stairs at a slab edge

If an area ancillary stair spans onto a slab edge, in order to decompose the stair load on to the slab you can add an inactive member along just that part of the slab edge to which the stair attaches.



Provided that the slab is meshed for 3D analysis, the inactive member provides a load path - the load becomes point loads on slab at each end inactive member (i.e. at the edges of the stair).



12.4 Fire proofing

Fire proofing is the application of fire resistance material on the surface of structural members in the form of:

- protective paints
- thin/thick film coating
- sprays
- fire boards

In Tekla Structural Designer fire proofing can be applied as a surface coat to members, defined by a thickness and density.

NOTE Fire proofing can be applied to all 1D member types apart from analysis elements, DELTABEAM, and FABSEC beams.

The additional self weight of fire protection material is then added to the "Self-weight - excluding slabs" loadcase.

NOTE If **Calc Automatically** is unselected for the "Self-weight - excluding slabs" in the Loading dialog, neither the member self weight, or the fire proofing self weight are calculated.

The extra thickness also increases the area exposed to wind if open structure wind loads are applied to the member.

How to apply fire proofing

Fireproofing is applied to a member by selecting ‘Protected’ under the 'Fire proofing' heading in the member properties. It can also be applied graphically using the Fire proofing attribute in Show/Alter state.

Fire protection material details

Property	Description
Exposure	• Exposed on three sides (Applicable for beams only) • Exposed on all sides
Shape	Contour encasement

Property	Description
	 Hollow encasement 
Thickness	Thickness of fire proofing material
Density	Density of fire proofing material
Selfweight	Calculated self weight of fire proofing material per unit length.

13 Embodied carbon handbook

Click the link below to find out about the carbon impact of structures and how Tekla Structural Designer helps engineers to assess this for their projects:

- Measuring the carbon impact of a structure (page 909)
- Add and manage embodied carbon factors (page 911)
- Embodied carbon workflow (page 915)

Embodied carbon assessment (video)

Embodied carbon assessment in Tekla Structural Designer (eLearning)

13.1 Measuring the carbon impact of a structure

Global impact of construction industry

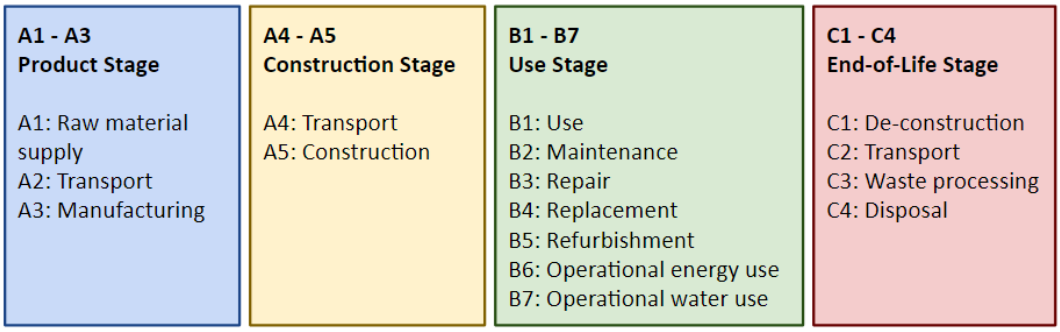
According to The Institution of Structural Engineers (IStructE):

- Buildings and construction account for about **40%** of energy related CO2 emissions
- Manufacture and disposal of materials used in structures accounts for about **11%** of all greenhouse gas emissions
- Analysis suggests that inefficiency in material use of up to **50%** is common

Typical emissions at each stage of the structure's life

In order to reduce carbon emissions it is first necessary to measure them. This can be done in different ways - in the UK for example by using BS EN 15978[1]

the life cycle of the structure can be split into four stages as shown below, and the emissions assessed at each stage.



The distribution of emissions at each stage breaks down in approximately in these proportions:

Stage	Typical distribution of emissions
Product Stage	50%
Construction Stage	5%
Use Stage	43%
End-of-Life Stage	2%

It can be seen that the largest portion of emissions occurs at the Product Stage, and this is the stage that can be targeted by Tekla Structural Designer to make savings.

1. BS EN 15978:2011: Sustainability of construction works. Assessment of environmental performance of buildings. Calculation method. London: BSI, 2011

Measuring Product Stage Carbon

In the UK, the Product Stage is split into modules A1-A3, which are:

- Raw material extraction
- Transport of raw materials to manufacturing facilities
- Manufacture

The carbon impact of materials used in manufacture can then be measured by using an Embodied Carbon Factor (ECF).

This factor is specific to each material and is multiplied by the quantity of that material in order to give a quantity of CO_2 emitted in the Product Stage.

Many construction materials have standard ECF values that the engineer can use, and in addition manufacturers may also provide third-party verified Environmental Product Declarations (EPDs) which can be used when the engineer knows exactly which manufacturer the material will be sourced from.

Reporting and export of embodied carbon data

Tekla Structural Designer includes an automated and comprehensive built-in calculation of *Embodied Carbon* quantities, together with powerful options for graphical review/optimization and reporting of this aspect of design. For more information, see Embodied carbon workflow (page 915).

In addition Tekla Structural Designer also enables model material data to be exported to One Click LCA, which:

- is a widely used cloud-based software for life-cycle assessment and the calculation of embodied carbon.
- complies with a large number of standards including BREEAM and LEED.
- includes a huge database of generic and manufacturer-specific Environmental Product Declarations (EPDs)

The data can either be exported to a spreadsheet, or directly to the One Click LCA service, from where the environmental impact of the design can be reviewed and reports can be created.

NOTE The One Click LCA service can only be used if you have a One Click LCA account.

Both of the above features enable the Engineer to quickly and easily determine the embodied carbon for the part of a project they are responsible for. This also helps them to develop and compare different scheme options. For a selected scheme they can then track progress as they make refinements to drive down the carbon impact.

See also

Embodied carbon workflow (page 915)

13.2 Add and manage embodied carbon factors

To enable embodied carbon to be calculated in Tekla Structural Designer default embodied carbon factor (ECF) values have to be set up and saved. You can set up a local set of factors which apply to the current model only and a global set of factors which serve as the defaults for new models.

See also

Embodied carbon workflow (page 915)

Set up a global set of factors

When setting up standard embodied carbon factors you should add them directly to a global settings set so that they are available for all new models.

This can be done as follows:

1. On the **Home** tab, click **Settings**
2. On the **Embodied Carbon** page of the dialog, click **Edit...**

   The is displayed from where you can manage the global set of factors for the active setting set.
3. When you have finished, click **OK** to close the dialog and **OK** to close **Settings**

Set up and edit the local set of factors

The local set of factors are initially inherited from the global set. Often you might want to customize these by making specific factors inactive. You might also want to add project specific factors, for example cladding for walls and roofs.

These operations should be performed using the , which is accessed as follows:

1. On the **Home** tab, click **Embodied Carbon Factors**

   The is displayed from where you can manage the local set of factors.
2. When you have finished, click **OK** to close the dialog.

NOTE Changes to the local set do not get applied to new models unless they are saved to a global settings set.

Add factor

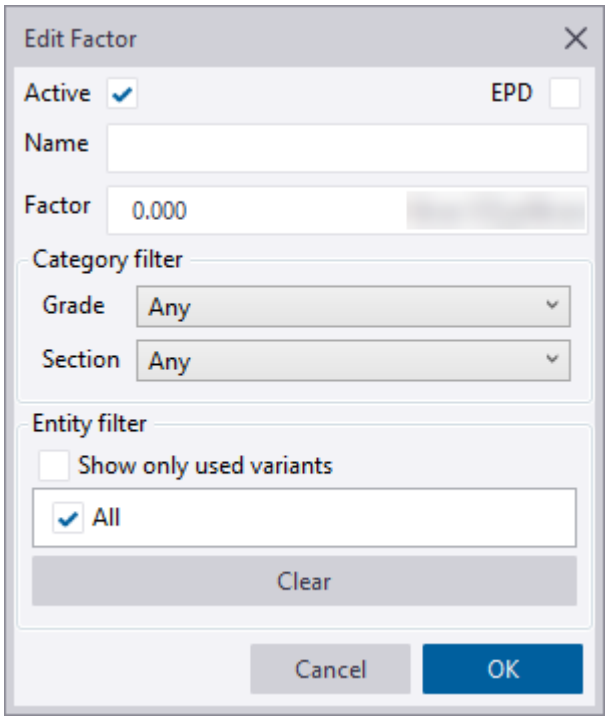
1. Open the
2. In the **Category filter** select the category appropriate to the factor you are about to add.

   TIP If you intend to enter several factors with similar characteristics you can specify additional Category/Entity filters in advance, before clicking **Add factor**. This way you don't have to respecify the filters for each factor.

   ---

3. Click **Add factor**

The **Add factor** dialog is displayed with any Category/Entity filters that you specified preselected.

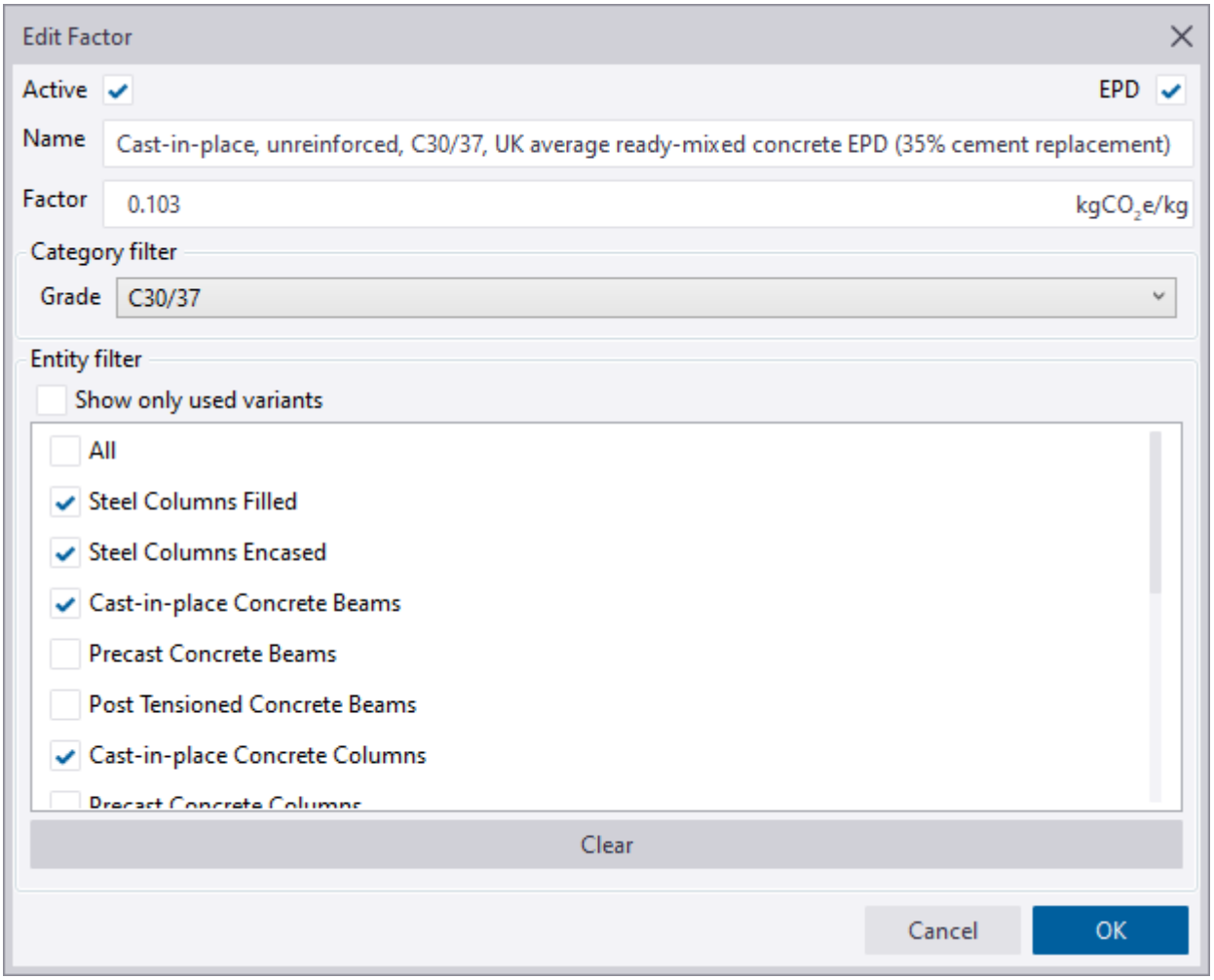


4. If the factor being added is from manufacturers data and has an Environmental Product Declaration (EPD), check the EPD box.
5. In the Name box enter a description for the factor that can be used to identify it uniquely.
6. Enter the factor.
7. Confirm that the Category/Entity filters are set as you require.
8. Click **OK** The factor is added to the list.
9. You can now continue to add further factors, or if finished, click **OK** to close the dialog.

Edit factor

1. Open the
2. In the **Category filter** select the category appropriate to the factor you want to edit.
3. Select the factor to be edited from the list.
4. Click **Edit factor**

   The **Edit factor** dialog is displayed.



5. Edit the values and filters as required.
6. Click **OK**

Remove factor

1. Open the
2. In the **Category filter** select the category appropriate to the factor you want to remove.
3. Select the factor to be removed from the list.
4. Click **Remove factor**

   The factor is removed from the list.
5. Click **OK**

Reorder the list of factors

1. Open the
2. In the **Category filter** select the category appropriate to the factors you want to reorder.
3. Select a factor to be move and drag it to move it up or down the list.
4. Continue moving factors as required, or click **OK** when done.

Set a factor as active or inactive

1. Open the
2. In the **Category filter** select the category appropriate to the factors you want to activate/deactivate.
3. Check the box to the left of a factor to make it active, or uncheck to make inactive.



4. Click **OK** when done.

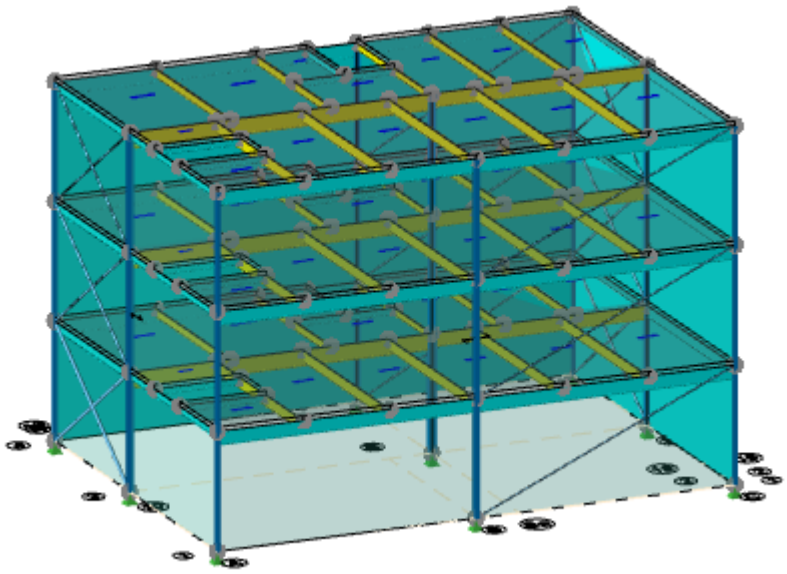
Export factors to a spreadsheet

1. Open the
2. Click **Export** and choose to export either all factors, or active factors.

   A spreadsheet opens listing the factors.

13.3 Embodied carbon workflow

The small steel model shown below will be used to demonstrate the basic steps required to calculate, optimize and report embodied carbon content. The model features 3 identical levels of composite slabs and rolled I-section beams; columns are rolled I-sections and bracing is from circular hollow sections. Wind wall panels have also been applied to each face.



Embodied carbon assessment (video)

Embodied carbon assessment in Tekla Structural Designer (eLearning)

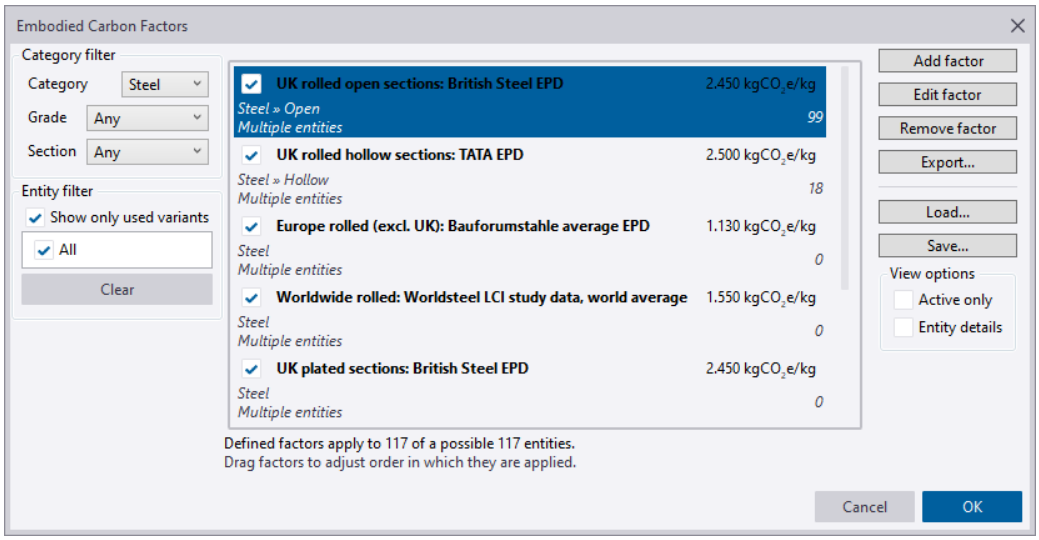
Set up embodied carbon factors

Setting up and managing embodied carbon factor (ECF) values is the engineer's responsibility. They can be set up from the **Home** ribbon by clicking:

- 

  Embodied Carbon Factors

  This displays the . Depending on the regional code being worked to you will find sample ECF definitions will have been added for some of the categories. For example, if working to the United Kingdom (Eurocode) regional code, the following ECF definitions will have been added in the steel category.



NOTE If working to the United States (ACI/AISC) regional code, it is understood that metric units (as shown above) are more commonly required for carbon mass quantities than US Customary units. Therefore these have been set as the default. You can however swap carbon mass quantities back to US Customary units if you require. For details, see:

NOTE There is potentially considerable variability/ uncertainty in ECFs - for example for concrete, many mix variations are possible for every strength grade and all could be unique. Hence the sample ECFs are included principally for guidance and are not a complete set - the engineer is advised to review, edit and add to them as they need.

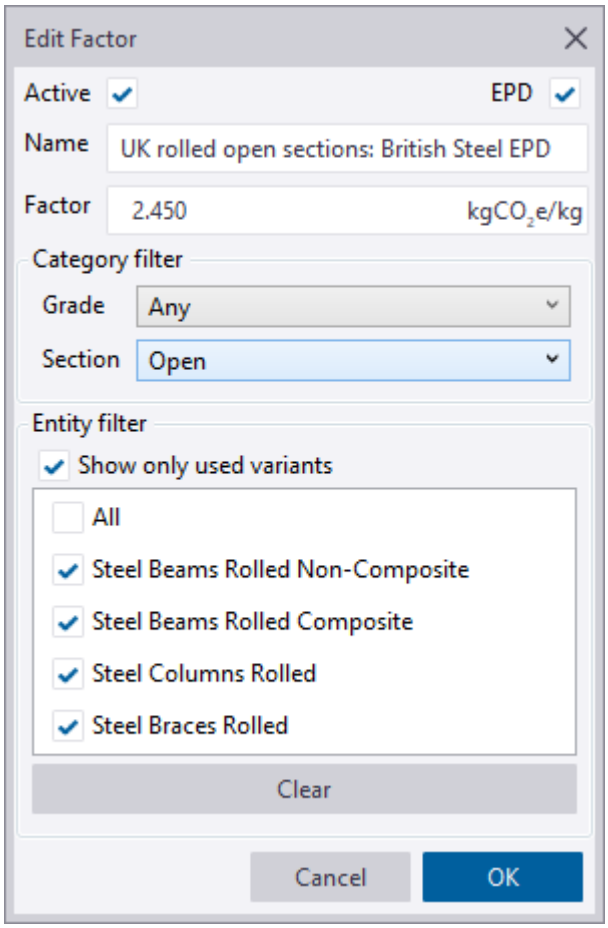
Many construction materials have ECF values that the engineer can use, and in addition manufacturers may also provide third-party verified Environmental Product Declarations (EPDs) which can be used when the engineer knows exactly which manufacturer the material will be sourced from.

To set up ECFs using the above dialog:

- You can add (page 912) additional ECFs using the **Add factor** button.

  NOTE When adding factors, the values displayed in the **Category filter** and **Entity filter** on the main dialog are used as presets for the new factor. Apart from the category, each of these can then be adjusted if required within the **Edit Factor** dialog.

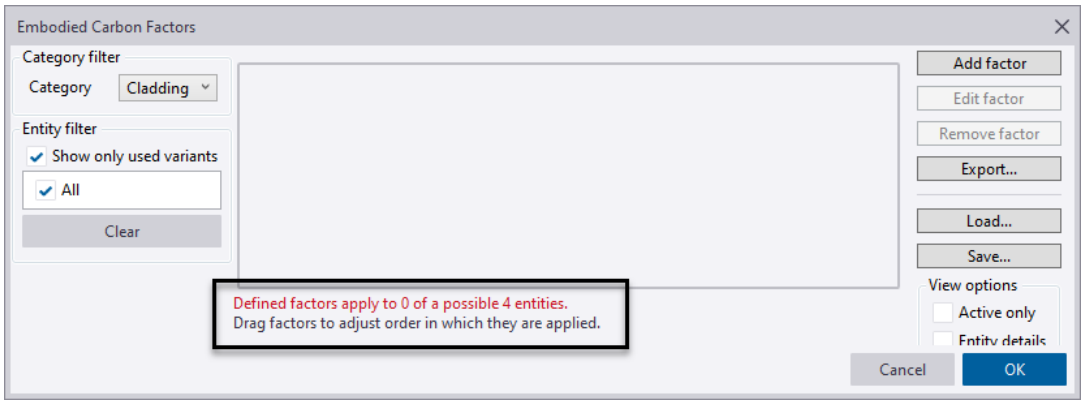
- You can remove (page 914) ECFs using **Remove factor**.
- You can edit (page 913) the description and/or value of ECFs using **Edit factor**.



- Once defined, standard ECF values should be saved in global settings (page 911) using **Save...** so that they can be applied to future models also. However there are some project specific ones (like cladding for walls and roofs) that you would not want to make available for all models.
- If you open an old model (i.e. one created before you established a global set of ECF values), initially the won't contain any values, but you can click **Load...** in order to inherit any factors that exist as global settings.

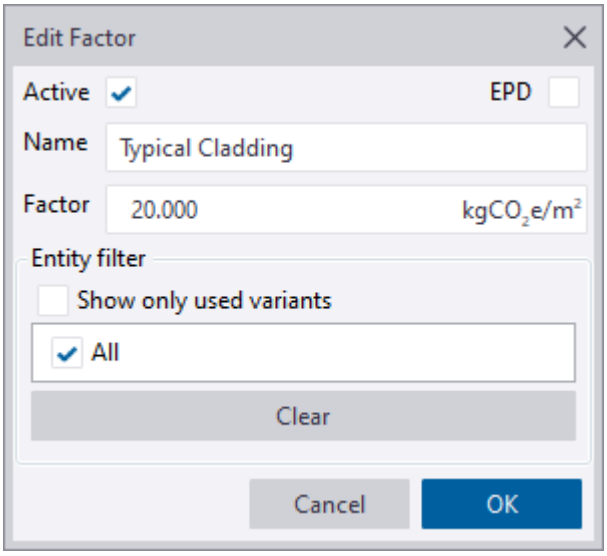
By cycling through each category, you can quickly confirm that ECFs have been defined for all the entities in the model.

In this model, when the 'Cladding' category is reached a message is displayed in red text, as shown below.

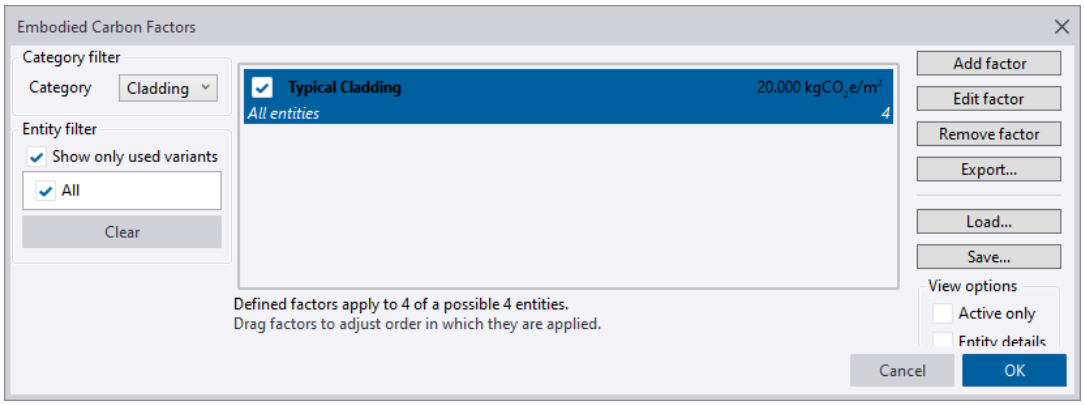


This indicates that there are entities in the model (wind walls in this case) for which there are no defined factors in the current category. The message serves to highlight that the total embodied carbon mass of the structure may be incomplete until the factor has been added.

For this model this can be resolved by clicking **Add factor** in order to define a suitable factor for the cladding.



Clicking on OK, the factor is assigned to wind wall entities and the warning message is removed.



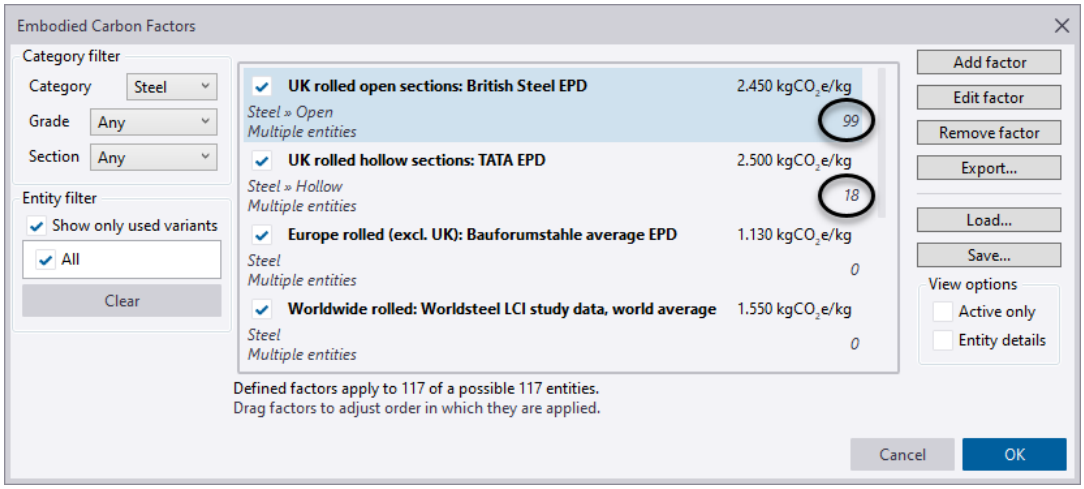
Related tasks

- Add and manage embodied carbon factors (page 911)

Manage how factors are applied to the model

Once the factors have been set up, the same dialog is used to manage how the ECFs in each category are applied to entities in the model.

You first select the main category you wish to view/edit at the top left e.g. 'Steel'. You can then review for this category the ECF list order, the description of each factor (which can be any variation of text and numbers etc) value, and the current *number of assigned entities* for each factor. This last figure is key to the management process and is circled below.



The current assignment of ECFs is a constantly active process, starting at the top of the list and working down, so the top factor has first priority and so on.

If certain factors are inappropriate for the current model they can be deactivated via the checkbox at the left of the factor to prevent them from being applied.

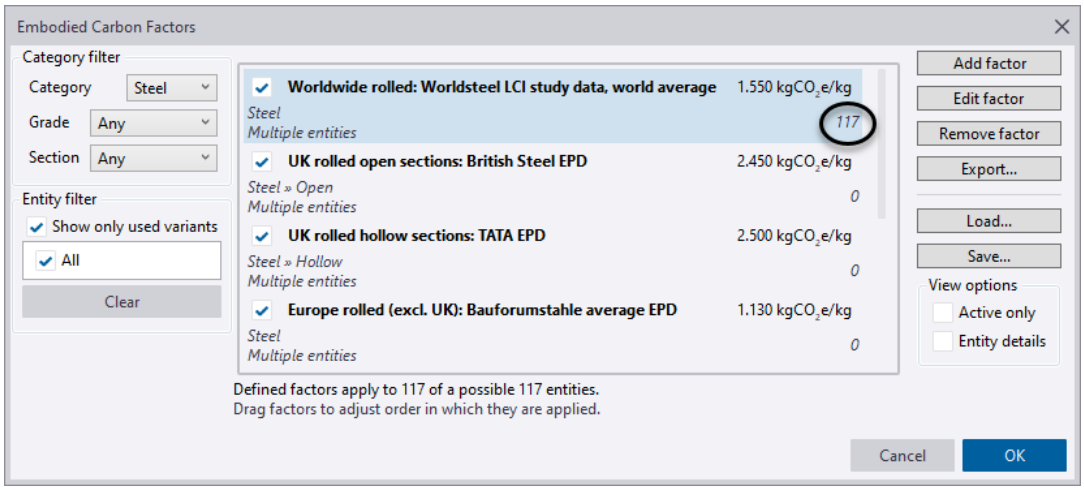
Once an entity in the model has a factor from the list assigned the process stops for that entity and category, hence there is no duplication of factor

assignment within a category where lower (in the list) ECF definitions are also applicable.

Initially in this example 99 steel entities can be assigned to the factor at the top of the list, and the remaining 18 entities (the bracing members) can be assigned to the second factor on the list.

From here you might choose to reorder the list of factors to suit the current model by dragging factors upwards or downwards with the mouse. The assignment of factors updates automatically as edits are made - applicable factors at the top of the list always being assigned wherever possible in preference to factors occurring lower down the list.

To demonstrate, if the fourth factor in this example were dragged to the top of the list, the assignment of factors would automatically update as shown, with all 117 entities being able to be assigned to the factor newly positioned at the top of the list.



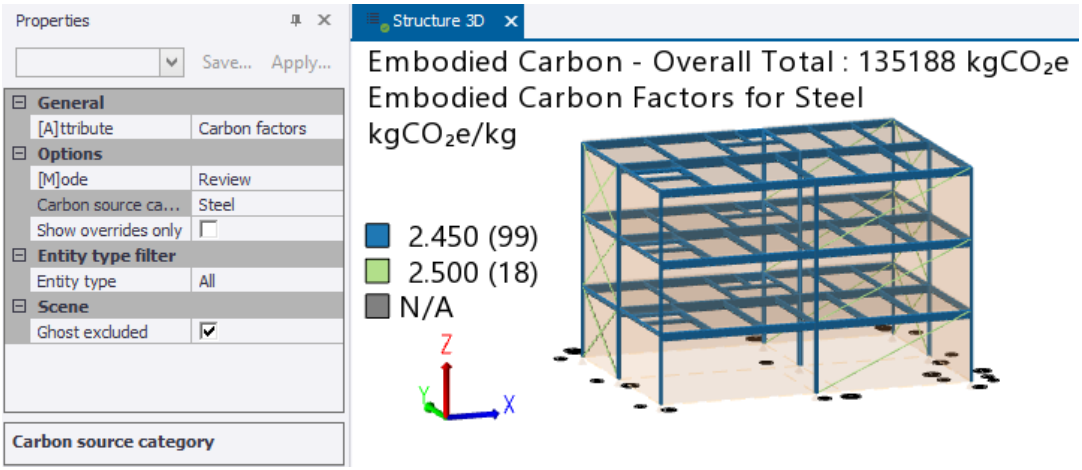
NOTE There is not a one-to-one relationship between model entities and ECFs - for example a composite slab will have different ECFs assigned for its decking, concrete material, and its reinforcement.

NOTE Any changes to the list order, or amendments to the factors themselves will only apply to the current model unless you choose to save them back to the global settings set.

Related tasks

- Add and manage embodied carbon factors (page 911)

Review how factors have been applied



Probably the easiest way to review how the ECFs have been applied to the model is graphically in a Review View.

By stepping through the applicable carbon source categories you can quickly review how each of the factors have been applied.

An option is also provided here to apply overrides if required, (although preferably you should consider adding new factors instead using the). Setting overrides is primarily recommended for entities that require different factors but are indistinguishable in terms of their data (and so cannot be assigned automatically) such as Roof/ Wind panels (in the “Cladding” category).

NOTE There is not a one-to-one relationship between model entities and ECFs - for example a composite slab will have different ECFs assigned for its decking, concrete material, and its reinforcement.

Related task

•

Examine embodied carbon overview

At an early stage you might choose to display the embodied carbon overview in a tabular data view. This is very useful because it rapidly provides a grand total of the embodied carbon mass for the entire structure.

The overview can grouped in a variety of ways as shown below.

- **Group by Construction Type**

  The total embodied carbon mass for each construction type is listed.

Review Data

Embodied Carbon Overview

Construction Type	Embodied Carbon Mass [$kgCO_2e$]
Composite Slabs	54013
Steel Beams Rolled Composite	27747
Steel Beams Rolled Non-Composite	24668
Wind Walls	14880
Steel Columns Rolled	11632
Steel Braces Rolled	2248
Total	**135188**

- **By Level - Summary**

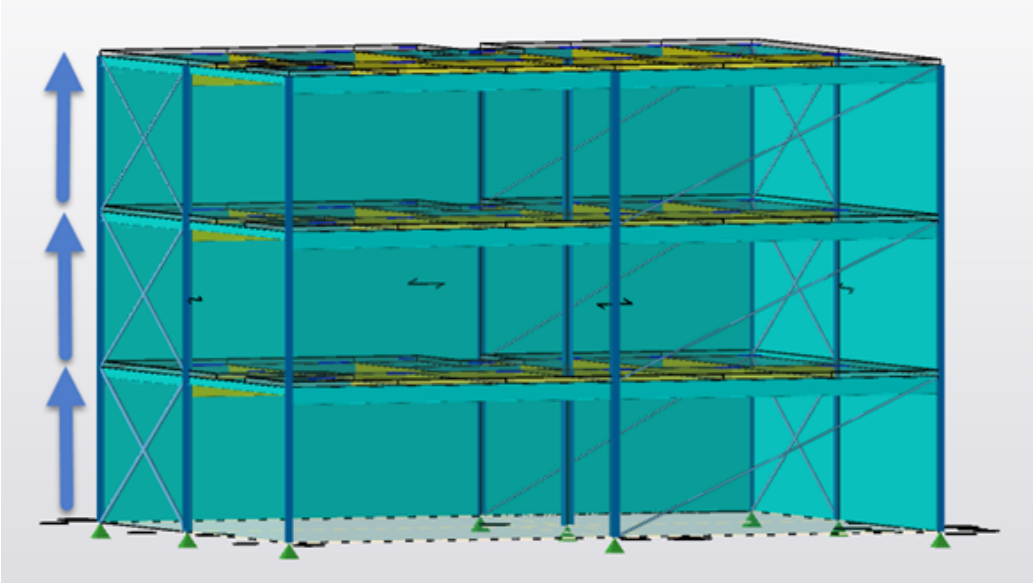
  The total embodied carbon mass between one level and the next is listed.

Review Data

Embodied Carbon Overview

Reference	Level [m]	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
St. 3 (Roof)	12.000	44923	223	201.7
St. 2 (2nd Floor)	8.000	45000	223	202.0
St. 1 (1st Floor)	4.000	45265	223	203.2
Total	-	**135188**	**668**	**202.3**

i.e. - level 1 includes everything above foundation level up to and including items in level 1, level 2 includes everything above level 1 up to and including items in level 2, etc.



A total floored area is determined for each level and hence the embodied carbon per unit area can be reported for each story height of the building.

- **Group by Material**

  The embodied carbon mass is reported as a single total for each material. In the below example, Steel appears twice because there are two types of steel with different EPDs (i.e. they are considered to be different materials) - the same would apply to different concrete or timber grades.

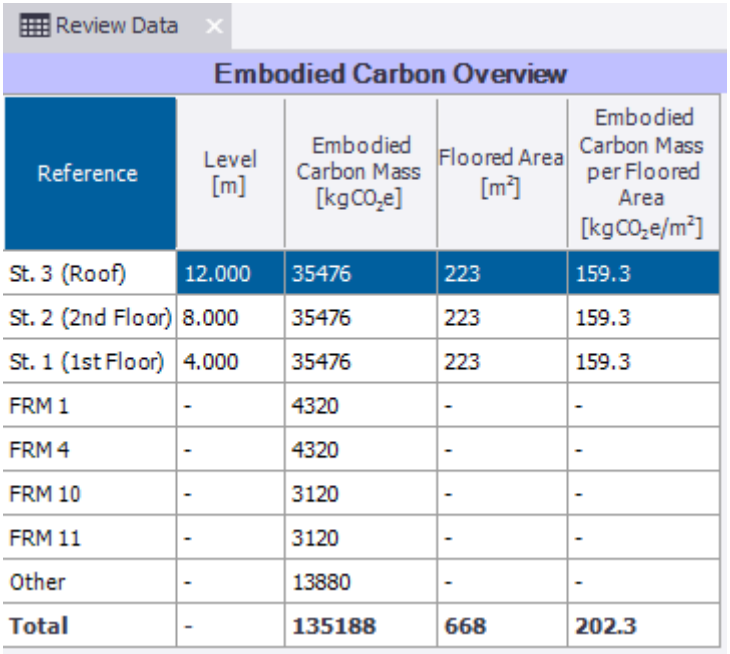
Review Data

Embodied Carbon Overview

Material	Quantity	Unit	Carbon Factor	Embodied Carbon Mass [kgCO_2e]
Steel	25338.32	kg	UK rolled open sections: British Steel EPD (2.450)	62079
Concrete	211612.50	kg	Cast-in-place, unreinforced, generic (0.150)	31742
Metal Deck	6348.38	kg	Steel Deck (2.740)	17395
Cladding	744.0	m^2	Typical Cladding (20.0)	14880
Reinforcement Mesh class 500	1950.77	kg	Reinforcement mesh (2.500)	4877
Steel	899.12	kg	UK rolled hollow sections: TATA EPD (2.500)	2248
Shear Connectors	909.00		Typical (2.000)	1818
Reinforcement class 500	197.18	kg	Reinforcement UK CARES Sector Average EPD (0.760)	150
Coating	734.2	m^2	<not set>	0
Floor Finishes	668.3	m^2	<not set>	0
Total				**135188**

- **Group by Plane**

  The embodied carbon mass is reported for each plane, items that not referenced to a specific plane being referenced as 'Other'. The total floored area and embodied carbon per unit area are reported for each plane.



Reference	Level [m]	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
St. 3 (Roof)	12.000	35476	223	159.3
St. 2 (2nd Floor)	8.000	35476	223	159.3
St. 1 (1st Floor)	4.000	35476	223	159.3
FRM 1	-	4320	-	-
FRM 4	-	4320	-	-
FRM 10	-	3120	-	-
FRM 11	-	3120	-	-
Other	-	13880	-	-
Total	-	**135188**	**668**	**202.3**

- **Group by Level & Construction Type**

  This works in a similar way to **By Level - Summary**, but is broken down into the individual construction types that exist between one level and the next.

  A total floored area is determined for each level, allowing the embodied carbon per unit area to be reported for each construction type for each story height of the building.

Review Data

Embodied Carbon Overview

Reference	Level [m]	Construction Type	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
St. 3 (Roof)	12.000	Composite Slab Items	18004	223	80.8
St. 3 (Roof)	12.000	Steel Beams Rolled Composite	9249	223	41.5
St. 3 (Roof)	12.000	Steel Beams Rolled Non-Composite	8223	223	36.9
St. 3 (Roof)	12.000	Wind Walls	4960	223	22.3
St. 3 (Roof)	12.000	Steel Columns Rolled	3835	223	17.2
St. 3 (Roof)	12.000	Steel Braces Rolled	653	223	2.9
St. 2 (2nd Floor)	8.000	Composite Slab Items	18004	223	80.8
St. 2 (2nd Floor)	8.000	Steel Beams Rolled Composite	9249	223	41.5
St. 2 (2nd Floor)	8.000	Steel Beams Rolled Non-Composite	8223	223	36.9
St. 2 (2nd Floor)	8.000	Wind Walls	4960	223	22.3
St. 2 (2nd Floor)	8.000	Steel Columns Rolled	3835	223	17.2
St. 2 (2nd Floor)	8.000	Steel Braces Rolled	729	223	3.3
St. 1 (1st Floor)	4.000	Composite Slab Items	18004	223	80.8
St. 1 (1st Floor)	4.000	Steel Beams Rolled Composite	9249	223	41.5
St. 1 (1st Floor)	4.000	Steel Beams Rolled Non-Composite	8223	223	36.9
St. 1 (1st Floor)	4.000	Wind Walls	4960	223	22.3
St. 1 (1st Floor)	4.000	Steel Columns Rolled	3963	223	17.8
St. 1 (1st Floor)	4.000	Steel Braces Rolled	866	223	3.9
Total	-	**Total**	**135188**	**668**	**202.3**

- Because the 3 floor levels in this model are identical, as expected items that exist in the identical level planes have identical carbon masses (e.g. the composite slabs).
- Items which exist between the planes are also assigned to a level:
  - e.g. steel column stacks, braces and wind walls.
  - These can be different at each level.
- Items that exist between reporting levels are assigned to the level above:
  - e.g. Braces and top stacks of columns assigned to level 3.
- Spans/stacks are not split - they are assigned to one level based on their highest point. (If a beam started below level 2 and finished above then it would be assigned to level 3.)
- Wind walls do get split and are shared between levels as appropriate.

- **Group by Construction Type & Material**

  This works in a similar way to **Group by Construction Type**, but with the totals further broken down into individual materials, (so for example the steel total is separated for beams, composite beams, columns, braces, etc.)

Review Data

Embodied Carbon Overview

Construction Type ▲	Material	Quantity	Unit	Carbon Factor	Embodied Carbon Mass [kgCO_2e]
Composite Slabs	Concrete	211612.50	kg	Cast-in-place, unreinforced, generic (0.150)	31742
Composite Slabs	Metal Deck	6348.38	kg	Steel Deck (2.740)	17395
Composite Slabs	Reinforcement Mesh class 500	1950.77	kg	Reinforcement mesh (2.500)	4877
Composite Slabs	Floor Finishes	668.3	m^2	<not set>	0
Steel Beams Rolled Composite	Steel	10522.00	kg	UK rolled open sections: British Steel EPD (2.450)	25779
Steel Beams Rolled Composite	Shear Connectors	909.00		Typical (2.000)	1818
Steel Beams Rolled Composite	Reinforcement class 500	197.18	kg	Reinforcement UK CARES Sector Average EPD (0.760)	150
Steel Beams Rolled Composite	Coating	307.5	m^2	<not set>	0
Steel Beams Rolled Non-Composite	Steel	10068.41	kg	UK rolled open sections: British Steel EPD (2.450)	24668
Steel Beams Rolled Non-Composite	Coating	275.7	m^2	<not set>	0
Steel Braces Rolled	Steel	899.12	kg	UK rolled hollow sections: TATA EPD (2.500)	2248
Steel Braces Rolled	Coating	34.6	m^2	<not set>	0
Steel Columns Rolled	Steel	4747.91	kg	UK rolled open sections: British Steel EPD (2.450)	11632
Steel Columns Rolled	Coating	116.4	m^2	<not set>	0
Wind Walls	Cladding	744.0	m^2	Typical Cladding (20.0)	14880
Total					**135188**

Use of Level Properties 'Reporting Level' when grouping by level

In there is a **Reporting Level** checkbox. Unselecting this affects how the overview is presented when using the **Group by Level & Construction Type** or **By Level - Summary**' options.

This is demonstrated in the below example showing the overview by level summary before and after switching off the **Reporting Level** checkbox at levels 1 and 3:

Review Data

Embodied Carbon Overview

Reference	Level [m]	Embodied Carbon Mass [kgCO_2e]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [kgCO_2e/m^2]
St. 3 (Roof)	12.000	44923	223	201.7
St. 2 (2nd Floor)	8.000	45000	223	202.0
St. 1 (1st Floor)	4.000	45265	223	203.2
Total	-	**135188**	**668**	**202.3**

Review Data

Embodied Carbon Overview

Reference	Level [m]	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
Top	-	44923	223	201.7
St. 2 (2nd Floor)	8.000	90265	446	202.6
Total	-	**135188**	**668**	**202.3**

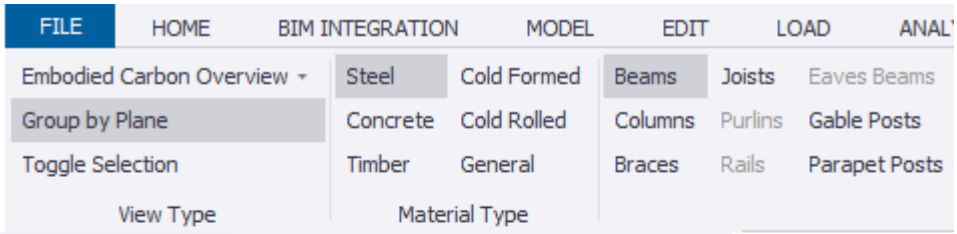
- Everything that was in St.1 and St.2 is now combined in the St.2 totals
- St.2 becomes the highest designated reporting level, everything above that level must be assigned to something so a “Top” level is used for that.
- Note that the overall totals are unaffected.

Reporting for specific construction types only

Clicking **Toggle Selection** in the ribbon switches between all construction types being selected and being unselected.

After unselecting, you can then choose specific construction types to be considered by choosing the required Material Types, Characteristics, Constructions and Fabrications.

This is demonstrated below - the overview is being grouped by plane, but is reporting the masses for steel beams only.



Review Data

Embodied Carbon Overview

Reference	Level [m]	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
St. 3 (Roof)	12.000	17471	223	78.4
St. 2 (2nd Floor)	8.000	17471	223	78.4
St. 1 (1st Floor)	4.000	17471	223	78.4
Total	-	**52414**	**668**	**78.4**

Related task

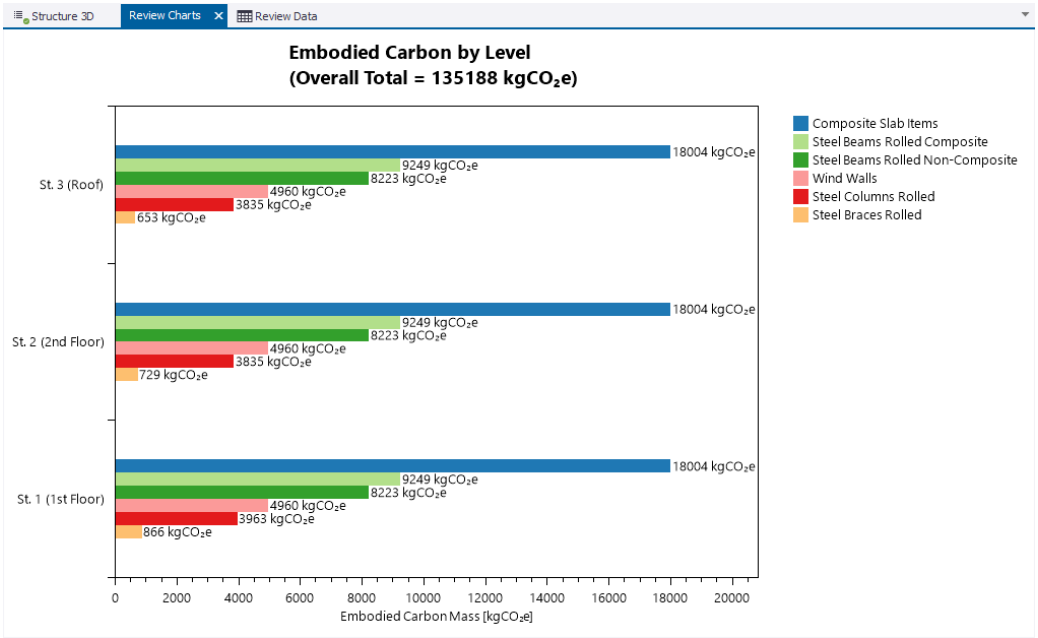
- Create embodied carbon overview tabular results

Display the overview data as a chart

The embodied carbon overview data can also be displayed in charts, with three types being available. Various settings are available to configure the appearance of each chart.

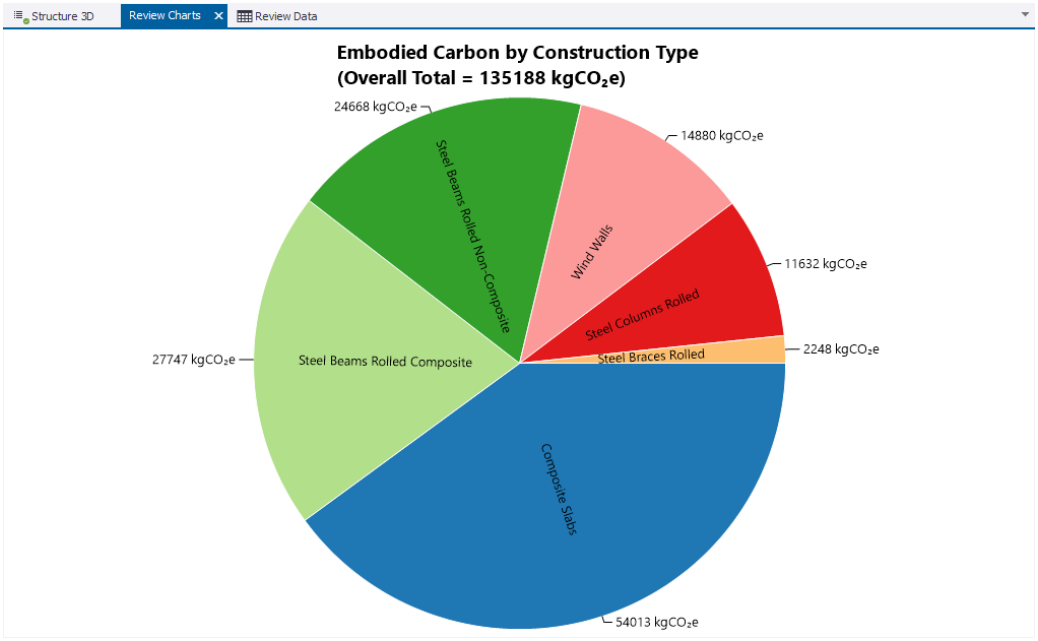
- **Embodied Carbon by Level**

  The total embodied carbon mass between one level and the next is displayed in a bar chart. One of the settings provided is to group by construction type if required (as shown below).



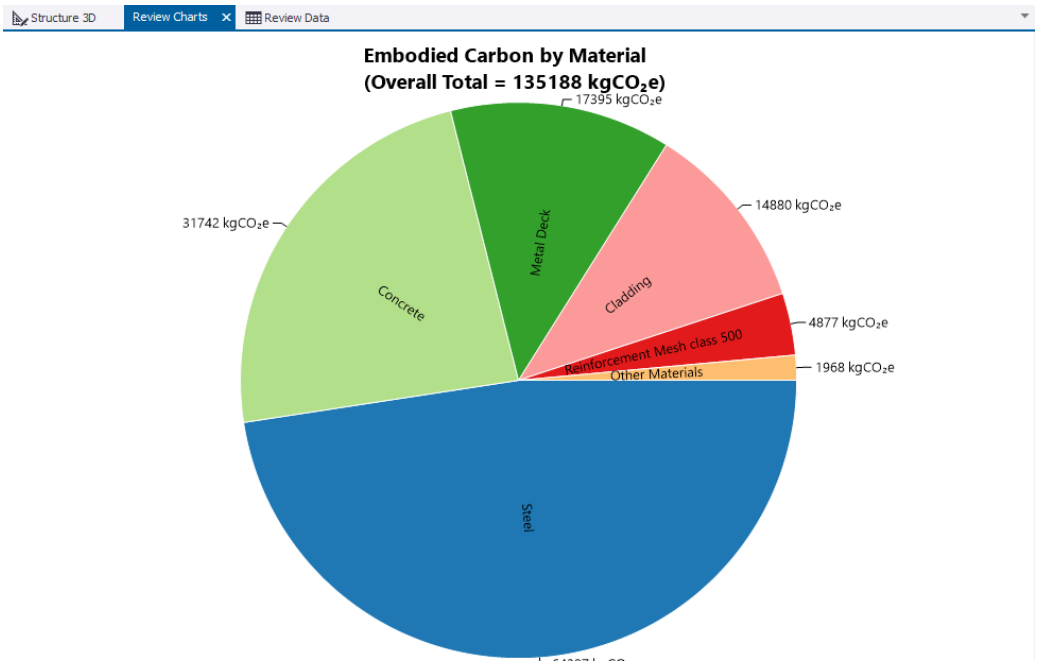
- **Embodied Carbon by Construction Type**

  The total embodied carbon mass for each construction type is displayed in a pie chart.



- **Embodied Carbon by Material**

  The total embodied carbon mass is displayed by material in a pie chart. In this chart materials with the same name but different carbon factors are combined.



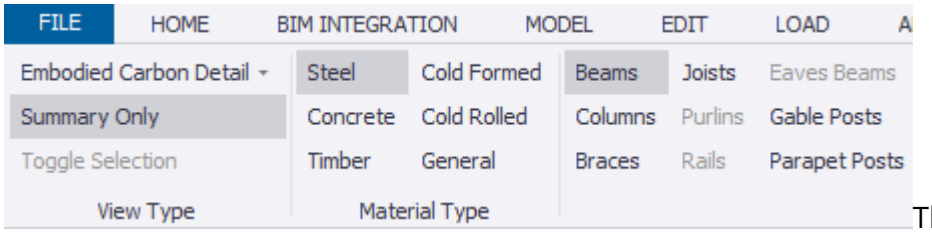
NOTE For this chart the "Merge segments contributing less than" setting defaults to 1.5%. Segments smaller than this value are displayed as "Other Materials".

Related tasks

-
-
-

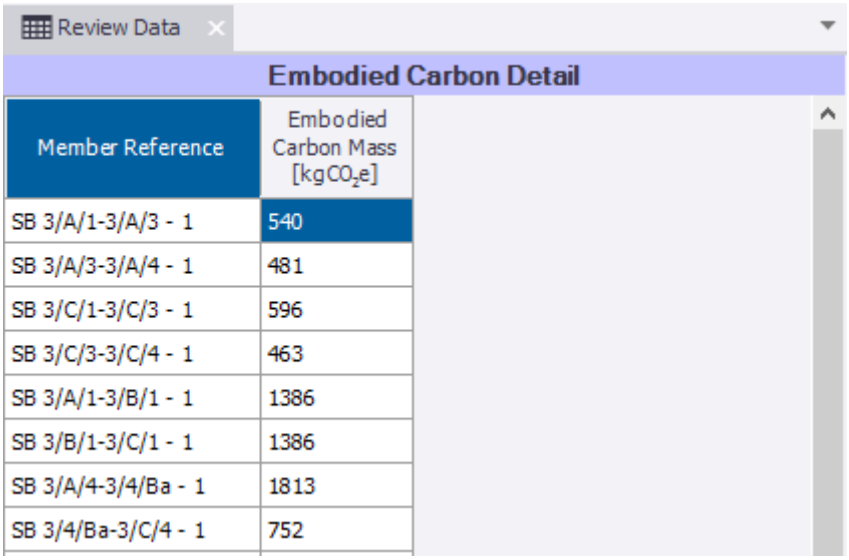
Examine embodied carbon details

If you want to investigate where the values in the overview data table and charts have come from, you also have the option to display the embodied carbon detail for individual construction types in a tabular data view.



This can be displayed

- as a summary...



- or, broken down to shown the separate components that contribute to the total for each entity...

Review Data

Embodied Carbon Detail

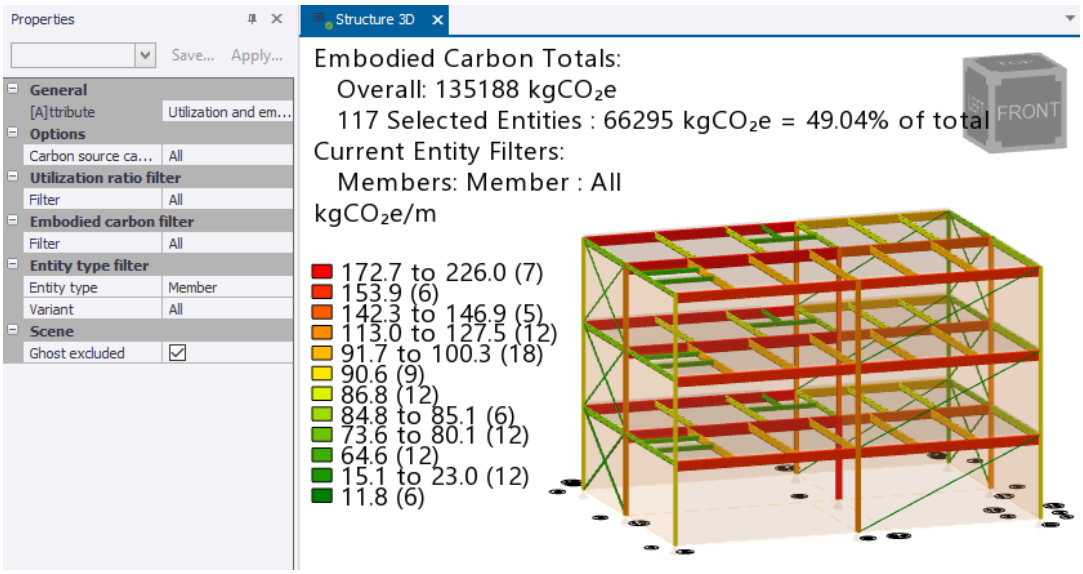
Member Reference	Material	Quantity	Unit	Carbon Factor	Embodied Carbon Mass [kgCO_2e]
SB 3/A/1-3/A/3 - 1	Steel	209.97	kg	UK rolled open sections: British Steel EPD (2.450)	514
SB 3/A/1-3/A/3 - 1	Connection End 1	5.25	kg	UK rolled open sections: British Steel EPD (2.450)	13
SB 3/A/1-3/A/3 - 1	Connection End 2	5.25	kg	UK rolled open sections: British Steel EPD (2.450)	13
SB 3/A/1-3/A/3 - 1	Coating	6.5	m^2	<not set>	0
SB 3/A/3-3/A/4 - 1	Steel	186.88	kg	UK rolled open sections: British Steel EPD (2.450)	458
SB 3/A/3-3/A/4 - 1	Connection End 1	4.67	kg	UK rolled open sections: British Steel EPD (2.450)	11
SB 3/A/3-3/A/4 - 1	Connection End 2	4.67	kg	UK rolled open sections: British Steel EPD (2.450)	11
SB 3/A/3-3/A/4 - 1	Coating	6.4	m^2	<not set>	0
SB 3/C/1-3/C/3 - 1	Steel	231.51	kg	UK rolled open sections: British Steel EPD (2.450)	567
SB 3/C/1-3/C/3 - 1	Connection End 1	5.79	kg	UK rolled open sections: British Steel EPD (2.450)	14
SB 3/C/1-3/C/3 - 1	Connection End 2	5.79	kg	UK rolled open sections: British Steel EPD (2.450)	14

NOTE For steel members with pinned or fixed connections, the carbon mass of the connections at each end is determined as a percentage of that of the member. This percentage can be adjusted via

Related tasks

- Create embodied carbon detail tabular results

Review utilization and embodied carbon



Once the model has been designed you can investigate the potential for optimizing utilization and embodied carbon in a Review View.

This view shows the embodied carbon usage for all, or selected entity types. It also has filters that can be configured to locate any parts of the model where high carbon usage coincides with low utilization.

NOTE There is not a one-to-one relationship between model entities and ECFs - for example a composite slab will have different ECFs assigned for its decking, concrete material, and its reinforcement.

Having rapidly identified inefficiency in this way, you can then consider the potential for optimizing your design in some way.

Related tasks

•

Create reports

Embodied carbon mass is automatically included in the .

Additionally, a separate can be created which by default summarizes the embodied carbon in the model by level and by construction type.



Tekla Structural Designer	Project: Training Course				Job Ref.: 123	
	Structure: Model 1				Sheet no.: Page 1/1	
	Calc. by	Date: 10/05/2021	Chk'd by	Date: 10/05/2021	App'd by	Date: 10/05/2021

Embodied Carbon Overview by Level

Reference	Level [m]	Embodied Carbon Mass [$kgCO_2e$]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [$kgCO_2e/m^2$]
St. 3 (Roof)	12.000	44923	223	201.7
St. 2 (2nd Floor)	8.000	45000	223	202.0
St. 1 (1st Floor)	4.000	45265	223	203.2
Total		135188	668	202.3

Embodied Carbon Overview by Construction Type

Construction Type	Embodied Carbon Mass [$kgCO_2e$]
Composite Slabs	54013
Steel Beams Rolled Composite	27747
Steel Beams Rolled Non-Composite	24668
Wind Walls	14880
Steel Columns Rolled	11632
Steel Braces Rolled	2248
Total	135188

NOTE You can also include a table of Embodied Carbon Factors in the same report if required, or group the summary by plane, or by level & construction type.

Related tasks

- Configure and display a Material Listing report
- Configure and display an Embodied Carbon report
-

Index

.....698

A practical approach to setting the analysis type.....860,864
Amplified forces method 859,863
Application of Check Lines 751
Check Line Reports..... 753
Connector layout.....53
Construction stage events 725
Custom event sequences..... 735,752
Custom event sequences 733
Description of a typical event sequence . 725
Design connections 115
Editing a Custom Event Sequence..... 733
First-order (Elastic) analysis.....859,863
How do I assess the worst elastic critical load factor for the building?..... 861,865,866
How is the elastic critical load factor calculated?.....862,866
How to use the Project Workspace to manage connections 123
Limitations of Seismic Design.....550
Method 1: Simplified Event Sequence + Simplified Combined Creep and Shrinkage Allowance.....793
Method 3: Detailed Event Sequence + Rigorous Creep and Shrinkage Allowance.... 804,810
Observations on the Different Methods..817
Precast concrete planks 63
Second-order analysis.....860,864
Setting up Slab Deflection checks in advance 751
Stud strength..... 52
Understanding event sequence deflections 736
Use of check lines to check deflections (ACI) 822
Validity of the amplified forces method...864

A

Allowing for global imperfections.... 870,871,872
Allowing for global second-order effects. 857
Allowing for member imperfections.....872
Allowing for member imperfections (ACI/AISC).....872
Allowing for member imperfections (BS). 873
Allowing for member imperfections (Eurocode).....873
Ancillaries.....874
Autodesign optimization..... 23
Autodesign (concrete beam)..... 143
Autodesign (concrete column)..... 144
Autodesign (concrete wall)..... 145

C

Camber 46
Check Line Results..... 709
Check lines..... 751
Check timber members using Tekla Tedds.... 396
Choice of analysis type (ACI/AISC).....857
Choice of analysis type (BS)..... 859
Choice of analysis type (Eurocode).....863
Column base plate design workflow..... 80
Column base plate design..... 80
Column interaction diagrams (metric units) 186
Column interaction diagrams (US customary units).....182
Composite beam design..... 41
Concrete beam and column groups.....146
Concrete column design fundamentals...162
Concrete creep allowance during construction stages..... 476
Concrete design handbook..... 126
Concrete member autodesign..... 143
Concrete member design fundamentals. 150

Concrete member design workflow..... 127
Concrete slab design fundamentals.....255
Concrete wall design fundamentals..... 169
Construction stages and design stages....469
Create and check column base plates.....114

D

Derivation of the k_{amp}
formula for concrete structures.....867
Design comparison - with and without
staged construction.....481
Design timber members using Tekla Tedds
.....393
Differential deflection.....738

E

Embodied carbon workflow..... 915
Equipment..... 886
Event sequences 724
Export connections to another application
for design 122

F

Fire proofing 907
Flanged concrete beams.....159
Flat slab design workflow.....215
Footfall analysis handbook.....559
Footfall analysis workflow - CCIP-016....
596,631
Footfall analysis workflow SCI P354..... 572
Footfall Assessment Settings.....683

G

Global imperfections..... 856
Global second-order (P-Δ) effects..... 854

I

Ignoring small shear forces in pad base
design checks..... 287
Inactive members..... 900
Industrial structures..... 874
Instantaneous deflection..... 738
Interactive concrete beam design.....174
Interactive concrete column design..... 178
Interactive concrete member design 173
Interactive concrete wall design.....198
Introduction to seismic analysis and design
.....535

L

Limitations when using Tekla Connection
Designer with Tekla Structural Designer..121

M

Masonry wall design workflow.....421
Masonry wall design handbook..... 418
Masonry wall design - introduction 418
Masonry wall design cases..... 451
Masonry wall limitations and assumptions
.....419
Mat foundation design workflow (metric
units).....297
Mat foundation design workflow (US
customary units)..... 311
Measuring the carbon impact of a structure
.....909
Member imperfections..... 857
Member second-order (P-δ) effects.....855

O

Overview of stability requirements.....853

P

Pad base design workflow..... 281
Pad base, strip base and pile cap design
forces 296
Pile cap design workflow.....291
Piled mat foundation design workflow
(metric units)..... 336
Piled mat foundation design workflow (US
customary units)..... 326
Plated steel beams.....35

Portal frame design..... 95
Precast beam design..... 357
Precast column connection eccentricity moments.....375
Precast column design..... 371
Precast concrete design handbook 346
Precast member design groups..... 355
Precast member design workflow..... 347
Precast member design limitations and assumptions..... 346
Punching shear check fundamentals..... 264

R

Recommended workflows for specific connection types.....119
Reporting and export of embodied carbon data.....911
Rigorous slab deflection analysis examples (ACI).....791
Rotation angle for panels 263

S

Seismic analysis and conventional design.... 555
Seismic analysis and design handbook ...535
Seismic analysis and seismic design.....556
Seismic Design Methods..... 554
Seismic force resisting systems.....551
Select bars starting from.....145
SidePlate connections
Theory.....105
SidePlate connections..... 105
Beam Properties..... 113
Create..... 112
Slab deflections
Methods 714
Concepts, workflows and examples... 714
Calculations in depth..... 739
Factors that affect rigorous slab deflection estimates.....716
Edit Event Sequences..... 702
Reports..... 712
Results..... 710
Rigorous slab deflection analysis example (Eurocode)..... 759
Run Slab Deflection Analysis..... 706
Work with check lines..... 704
Slab deflections
Results and reports.....707
Slab deflection example (ACI) 788
Slab deflection example (Eurocode) 755
Slab Deflection Property Choices).....792
Slab deflection settings 848
Slab deflections 700
Slab deflections
Status and utilization 753
Concepts, workflows and examples... 713
Analysis sequence..... 737
Rigorous slab deflection workflow 716
Get started with slab deflection analysis 700
Open a Slab Deflections View..... 701
Optimization..... 712
Results..... 707
Slab on beam idealized panels 261
Slab on beams design workflow..... 239
Span effective depth checks for irregular shaped panels 261
Stability and imperfections handbook 853
Staged construction handbook.....454
Staged construction - introduction.....454
Staged construction limitations and assumptions..... 480
Staged construction modification factors470
Staged construction workflow example - Eurocodes..... 489
Staged construction workflow example - United States (ACI/AISC).....511
Steel beam design.....28
Steel brace design..... 84
Steel column connection eccentricity moments..... 74
Steel column design.....66
Steel connection formation rules..... 118
Steel joist design..... 86
Steel member autodesign.....21
Steel member design groups..... 24
Steel truss design..... 91
Sustainability and Tekla Structural Designer 909

T

Timber beam deflection checks - EC..... 408
Timber beam deflection checks - US..... 403
Timber column multi-stack lifts.....413
Timber combination sets and strength design cases..... 401
Timber design envelopes.....387
Timber member design groups..... 397
Timber member design workflow.....383
Timber member design handbook 381
Timber member design limitations and assumptions..... 381

U

Use of modification factors..... 870

W

Wall interaction diagrams (metric units).. 202
Wall interaction diagrams (US customary units).....198
When must global and member second order effects be considered?.....855
When should a concrete building be classed as non-sway?..... 858