Tekla Structural Designer 2018

User Guides

July 2018

© 2018 Trimble Solutions Corporation.

Table of Contents

Basics of Tekla Structural Designer 1
About Tekla Structural Designer 1
Tekla Structural Designer Philosophy 1
How does the Tekla Structural Designer way of working differ from traditional methods? 1
Interface Overview 2
Interface components 2
File menu 2
Quick Access Toolbar 3
Viewcube 4
Ribbon 4
Project Workspace 4
Structure Tree 5
Groups Tree 5
Loading Tree 6
Wind Model Tree 6
Status Tree 6
Cutting Planes 7
Report Index 7
Scene Views 7
Basic Tooltip 8
Select Entity Tooltip 8
Properties Window 9
Property Dialog 9
Scene Content 10
Loading drop list 11
Process Window 11
Status Bar 12
Hiding, re-displaying and moving windows 13
To auto hide a window: 13
To close a window: 13
To re-display a window that has been closed: 14
To move a window to a new location 14
To dock a window as a tabbed page in another window 14
To open a tabbed page in another window 14
To dock a window using the docking control 14
Commands on the ribbon toolbars 15
Right click menu commands 15

Apply property set... 15
Check Member 15
Check Panel 16
Check Patches 16
Check plane 16
Check plane patches 17
Check plane slabs 17
Check Slabs 17
Check Truss 18
Check Wall 18
Copy 18
Create property set... 18
Delete Element 18
Design Member 18
Design Patches 19
Design plane 20
Design plane patches 20
Design plane slabs 21
Design Slabs 21
Design Truss 22
Design Wall 22
Edit 23
Generate Detailing Drawing 23
Interactive Design... 23
Open load analysis view 23
Open member view 23
Open view 23
Redraw 23
Report for member 23
Save Screenshot 23
Save View Configuration... 24
Show Member Loading 24
Undo/Redo 24
Zoom Out 24
Zoom to selection 24
Keyboard functions 24
Customising the appearance of the user interface 25
Working with the Project Workspace 25
Working with the Structure Tree 25

Working with the Groups Tree 25
How do I re-apply automatic grouping in order to reset manually edited groups? 26
How do I split an existing member group into smaller groups? 26
How do I manually move an existing member between groups? 26
How do I remove an existing member group? 27
How do I rename groups? 27
Working with the Loading Tree 27
Working with the Wind Model Tree 28
Pressure Zones 28
Wind Directions 28
Wind Loadcases 28
How do I use the Wind Model Tree to display a Wind Direction View? 28
Working with the Status Tree 28
The Report Index 28
Working with Scene Views, View Regimes and Scene Content 29
Opening and Closing and Saving Scene Views 29
How do I open a 3D view of my entire structure? 29
How do I open a 3D view of an existing sub model? 29
How do I open a 3D view of a single member? 29
How do I open a 2D view of an existing construction level? 30
How do I open a 2D view of an existing frame? 30
How do I open a 2D view of an existing sloped plane? 30
View Configurations 31
How do I close a view? 31
Zooming/Panning/Rotating and Walking through Scene Views 31
How do I zoom in/zoom out/zoom extents? 31
How do I pan the view? 32
How do I manually rotate the view 32
How do I use the ViewCube to display one of the preset views? 32
How do I walk through the model in a 3D view? 34
How do I display a 2D view in 3D? 34
Controlling Scene Content 34
How do I display the Scene Content window? 35
How do I make selections in the Scene Content Window? 36
How do I reinstate the default Scene Content selections? 37
Scene Content Entity Categories 38
Scene Content Plan Category 54
View Regime 61
Structural View 61

Solver View....62
Results View....62
Wind View....62
Review View....62
Changing the View Regime....62
Scene View Tab Groups....63
To create a new Tab Group from an existing view tab....63
To create a new Tab Group using the docking control....64
To move a view between Tab Groups....64
Working with Command Prompts....64
Selecting Entities....65
How do I select an individual entity?....65
How do I add further entities to the current selection?....66
How do I select multiple entities by dragging a box?....66
How do I deselect a single entity from the current selection?....67
How do I deselect all entities?....67
Entity selection using Find....67
Member selection using the Structure Tree....68
Member selection using the Groups Tree....68
Entity Nodes....68
Starting a New Project....71
Home toolbar....71
Creating a new project from scratch....73
How do I create a new project from scratch?....74
Working with templates....74
Working with templates....75
How do I create a new template?....75
How do I create a new project from a template?....75
Model Settings....76
How to apply and manage Model Settings....76
Model Settings dialog....76
Editing project details using Project Wiki....76
How do I edit the project details and view the revision history?....76
How do I record revisions?....77
Modeling and Editing Guide....79
Model toolbar....79
Levels group....79
Grid and Construction Lines group....80
Steel group....82

Concrete group 84
Slabs group....84
Timber group 85
Cold Formed group....86
Panels group....87
Miscellaneous group 87
Validate 88
Construction Levels....88
Construction Levels dialog....88
How do I open the Construction Levels dialog? 90
How do I insert a single Construction Level?....91
How do I insert multiple Construction Levels?....91
How do I make one level an identical copy of another? 92
How do I make one level an independent copy of another? 92
How do I delete a construction level?....92
How do I modify the properties associated with a level? 93
Frames and Slopes....93
How do I create a Frame?....93
How do I create a Slope?....94
Grid Lines and Construction Lines....94
Architectural Grids and Grid Lines 95
Why are the grid lines not being shown at a particular level? 95
How to I apply an existing architectural grid to a specific level?....95
How do I create a rectangular grid line system?....96
How do I create a radial grid line system? 97
How do I create a single grid line between two points? 99
How do I create a parallel grid line?....99
How do I create one or more parallel (quick) grid lines? 100
How do I create a perpendicular grid line? 101
How do I create a grid arc?....102
How do I import grids from a .dxf, or import a .dxf as a shadow? 102
How do I set the initial number or letter used for grids? 103
How do I change the name or color of an existing architectural grid? 103
How do I change the name of an individual grid-line, -arc?....103
How do I renumber all grids? 103
How do I extend, move or rotate grid lines and arcs? 103
How do I stretch, shorten, or rotate a grid line?....104
How do I move a grid line in a perpendicular direction?....104
How do I stretch or shorten a grid arc?....104

How do I adjust the radius of a grid arc? 104
How do I move a grid arc? 105
Construction Lines 105
How do I create a rectangular construction line system? 105
How do I create a radial construction line system? 106
How do I create a single construction line between two points? 107
How do I create a parallel construction line? 108
How do I create one or more parallel (quick) construction lines? 109
How do I create a perpendicular construction line? 109
How do I create a construction line arc? 110
How do I extend, move or rotate construction lines and arcs? 110
How do I stretch, shorten, or rotate a construction line? 111
How do I move a construction line in a perpendicular direction? 111
How do I stretch or shorten a construction arc? 111
How do I adjust the radius of a construction arc? 111
How do I move a construction arc? 112
Dimensions 112
How do I create a single dimension? 112
Steel, Cold Rolled, and Cold Formed Member modeling 112
Modeling Steel Columns and Cold Formed Columns 112
Steel column overview 112
Parapet post overview 113
How do I specify the column type and section size? 114
How do I create a single column in a 2D View? 114
How do I create a series of columns in a 2D View? 114
How do I create a single column in a Frame, or Structure View? 115
How do I align a column to a specific angle, or an angled gridline? 115
How do I create an inclined column? 115
How do I create a cranked column? 116
How do I create a plated or compound section steel column? 116
How do I create a concrete filled, or encased concrete steel column? 117
How do I create a gable post, or parapet post? 117
How do I specify a column splice? 117
How do I modify the position of a single column stack? 118
How do I modify the position of an entire column? 118
Modeling Steel Beams and Cold Formed Beams 118
Steel beam overview 118
How do I specify the beam type and section size? 120
How do I create a single span beam? 120

How do I create a plated or compound section steel beam? 121
How do I create a Westok Cellular, Westok Plated, or Fabsec beam? 121
How do I create a series of single span beams? 122
How do I create a continuous beam? 123
How do I add a haunch to an existing beam? 123
How do I create a curved beam? 124
How do I modify the position of a beam? 124
Modeling Steel Braces and Cold Formed Braces 125
Steel brace overview 125
How do I specify the brace type and section size? 126
How do I create a single brace? 126
How do I create an X, K, V or A brace? 127
How do I modify the position of a brace? 127
Modeling Steel Joists 128
How do I specify the joist type and size? 128
How do I create a steel joist? 128
How do I modify the position of a steel joist? 128
Modeling Steel Trusses 129
How do I use the Steel Truss Wizard? 129
How do I use the Space Truss Wizard? 129
How do I define a Free Form Truss? 130
How do I edit the geometry of an existing steel truss? 130
How do I edit the section sizes, material grades and section orientations in an existing steel truss? 130
Modeling Portal Frames 130
How do I create a single span portal frame? 130
How do I create a multi-span portal frame? 132
How do I edit the properties of an existing portal frame? 132
Modeling Cold Rolled Sections 133
How do I create a single cold rolled section? 133
How do I modify the position of a cold rolled section? 133
Modeling Web Openings 133
How do I add web openings using Quick Layout? 134
How do I add web openings manually? 136
Compound Sections 138
Concrete Member modeling 139
Modeling Concrete Walls 139
Concrete wall overview 139
How do I specify whether the wall is to be meshed or mid-pier? 140

How do I create a concrete wall in a 2D View? 140
How do I create a concrete wall in a Frame, or Structure View? 141
How do I specify extensions? 141
How do I specify releases? 141
Automatic support generation 142
How do I edit a wall support? 142
How do I create a door or window opening in an existing concrete wall? 143
Modeling Concrete Columns 143
Concrete column overview 144
How do I specify the column shape and size? 144
How do I create a single concrete column in a 2D View? 144
How do I create a series of concrete columns in a 2D View? 144
How do I create a single concrete column in a Frame, or Structure View? 145
How do I align a column to a specific angle, or an angled gridline? 145
How do I specify the column alignment relative to the grid? 145
How do I create a hollow column? 147
How do I create an inclined column? 147
How do I create a cranked column? 148
How do I modify the position of a single column stack? 148
How do I modify the position of an entire column? 149
How do I edit the column alignment or specify an offset? 149
Modeling Concrete Beams 150
Concrete beam overview 150
How do I specify the beam size? 150
How do I create a single span concrete beam? 150
How do I create a continuous concrete beam? 151
How do I create a series of concrete beams? 152
How do I create a curved concrete beam? 152
How do I specify the beam alignment relative to the grid? 153
How do I specify and use beam flanges for an existing beam? 155
How do I specify and use beam flanges for multiple beams simultaneously? 156
How do I edit the beam alignment or specify an offset? 156
How do I modify the position of a beam? 158
Slab modeling 158
Overview of slab modeling 159
The Concept of Slabs and Slab Panels 159
Panel sub-division 160
Creating slab panels 161
How do I choose the slab type and specify its properties? 161

How do I specify the parent slab to which the panel belongs? 162
How do I create slab panels by bay? 162
How do I create slab panels by points? 163
Modeling slab and mat openings 163
How do I create a rectangular slab opening? 163
How do I create a circular slab opening? 164
How do I delete a slab opening? 164
Adding overhangs to existing slab or mat edges 164
How do I add an overhang to a slab edge? 165
How do I add a curved overhang to a slab edge? 165
Modeling column drops 166
How do I create a column drop? 166
Splitting and joining slabs and mats 166
How do I split a slab? 167
How do I join slabs? 167
Slab steps 167
How do I modify the shape or size of a panel? 168
How do I modify a panel by moving a node? 168
How do I modify a panel by moving an edge? 169
How do I apply curved edges to existing slab items? 170
How do I delete an entire slab from my model? 170
Timber Member modeling 170
Modeling Timber Columns 170
How do I create a single timber column in a 2D View? 170
How do I create a series of timber columns in a 2D View? 171
How do I create a single timber column in a Frame, or Structure View? 171
How do I align a column to a specific angle, or an angled gridline? 172
How do I create an inclined column? 172
How do I create a cranked column? 172
How do I modify the position of a single column stack? 173
How do I modify the position of an entire column? 173
Modeling Timber Beams 173
How do I create a single span timber beam? 173
How do I create a series of single span timber beams? 174
How do I create a continuous timber beam? 174
How do I create a curved timber beam? 175
How do I modify the position of a beam? 176
Modeling Timber Braces 176
Modeling Timber Braces 176

How do I create a single timber brace? 176
How do I create an X, K, V or A brace? 177
How do I modify the position of a brace? 177
Modeling Timber Trusses 178
How do I use the Timber Truss Wizard? 178
How do I use the Timber Space Truss Wizard? 178
How do I define a Free Form Timber Truss? 179
How do I edit the geometry of an existing timber truss? 179
How do I edit the section sizes, material grades and section orientations in an existing timber truss? 179
Panel modeling 179
Modeling Roof Panels 179
How do I create a roof panel? 180
How do I edit the properties of a roof panel? 180
Modeling Wall Panels 180
How do I create a wall panel? 181
How do I edit the properties of a wall panel? 181
How do I create a wall panel with a parapet? 181
How do I reverse a wall panel? 182
Support, Analysis Element and Bearing Wall modeling 182
Modeling Supports 182
How do I create a single support? 182
How do I create a rotated support using 3 Grid Points? 183
How do I create a spring support? 183
How do I edit the properties of supports? 183
Modeling Analysis Elements 183
How do I create an analysis element? 183
How do I modify the position of an analysis element? 184
Modeling Bearing Walls 184
Bearing wall overview 184
How do I create a bearing wall in a 2D View? 184
How do I create a bearing wall in a Frame, or Structure View? 185
Sub Models 185
Sub Model Characteristics 186
Definitions 186
Basic rules of sub-models 186
Sub Models dialog 187
Working with Sub Models 188
How do I open the Sub Models dialog? 188

How do I create Sub Models? 188
How do I delete Sub Models? 188
How do I open a 3D view of an existing Sub Model? 188
Sub Structures 188
Sub Structure Characteristics 189
Working with Sub Structures 189
How do I create a new sub structure? 189
How do I add or remove existing elements in an existing sub structure? 189
How do I review all existing sub structures? 190
How do I open a 3D view of an existing sub structure? 190
Measure commands 190
How do I Measure distances? 190
How do I Measure Angles? 190
What are the points I can click to create a member? 191
Model Validation 193
How do I run model validation? 193
How do I control which conditions are considered during model validation? 193
Edit commands 193
Edit toolbar 193
Copying, moving and mirroring objects 195
How do I copy elements to a new location? 195
How do I copy and rotate elements to a new location? 196
How do I move elements to a new location? 196
How do I move and rotate elements to a new location? 196
How do I mirror elements to a new location? 197
Copy Loads 199
How do I copy all member loads from one span to another? 199
How do I copy just one member load if several are applied to the same span? 200
How do I copy panel area, level and slab loads? 200
How do I copy panel point, line and patch loads? 201
How do I copy structure loads? 201
How do I copy loads to another loadcase? 202
Joining and splitting members 202
How do I Join Members (Make Continuous)? 202
How do I Split Members? 203
Concrete Beam Lines 203
How do I Automatically Join All Concrete Beams (Make Continuous)? 204
Reversing member axes and panel faces 204
How do I reverse the local x axis of a beam? 204

How do I reverse the outward face of a wind panel? 204
Cutting Planes 204
How do I activate or deactivate a cutting plane? 204
How do I move a cutting plane in order to hide part of the model? 205
How do I reshow the hidden part of the model? 205
Moving the model, or the DXF shadow 205
How do I move the model to a new location? 205
How do I move the dxf shadow? 206
Rationalizing the model 206
How do I remove unused sloped planes, frames, grids, and construction lines? 207
How do I update grid and construction lines to extend only a short length beyond where they are required? 207
Creating infill members 207
How do I create infills? 207
Loading Guide 211
Load toolbar 211
Structure group 211
Wind Load group 212
Snow Load group 213
Seismic Load group 213
Decomposition group 214
Panel Loads group 214
Member Loads group 215
Structure Loads group 216
Validate 217
Working with Load Cases 217
Loading Dialog Loadcases Page 217
How do I create load cases? 219
How do I indicate that reductions apply to live (imposed) load cases? 220
How do I renumber all loadcases? 220
How do I add loads into a load case? 220
Working with Combinations 221
Loading Dialog Combinations Page 221
How do I generate load combinations automatically? 223
How do I create load combinations manually? 223
How do I create a Vibration Mass combination? 224
How do I renumber all combinations? 224
Working with Envelopes 225
Loading Dialog Envelopes Page 225

How do I create envelopes? 226
Working with Load Patterns 226
Overview of Load Patterns 227
How do I set an imposed load case to be patterned? 228
How do I set a combination to be patterned? 228
Pattern Load Cases 228
How do I update load patterns? 229
Working with the Wind Wizard 230
Running the Wind Wizard 230
How do I run the Wind Wizard? 230
How do I add a new wind direction? 231
How do I delete a wind direction? 231
How do I delete the entire wind model? 231
Reviewing wind zones and wind zone loads 231
Zone Loads toolbar 231
How do I open a Wind View? 232
How do I view the Wind Zones? 233
How do I edit a Wind Zone? 233
How do I view the Wind Zone Loads? 233
How do I change a Wind Zone Load? 234
What happens if I make changes to my model? 234
Wind loadcase definition 235
How do I Define Wind Loadcases? 235
How do I add new Wind loadcases? 235
How do I delete Wind loadcases? 235
Working with Simple Wind 236
How do I manually create a loadcase for Simple Wind? 236
How do I create a Simple Wind Load? 236
How do I edit Simple Wind load vertical properties? 237
How do I edit Simple Wind load width? 237
Working with the Snow Wizard 237
How do I run the Snow Wizard 238
Manually applying Snow Loads to Snow Loadcases 238
How do I manually apply a Uniform Snow load? 238
How do I manually apply a Valley Snow load? 238
How do I manually apply a Local Drift Snow load? 239
How do I apply a user override to a snow load? 239
A note about the graphical display of Snow Loads 240
Update Snow Loads 240

Delete Snow 240
Working with Seismic Loads 240
Running the Seismic Wizard 241
How do I run the Seismic Wizard? 241
How do I display the Horizontal Design Spectrum? 241
How do I delete Seismic Loads? 241
Load Decomposition 242
How do I manually decompose slab loads for an individual construction level? 242
How do I manually decompose slab loads for all the required levels? 242
How do I view the decomposed loads (either with, or without load values)? 243
Working with Panel Loads 243
How do I create a point load? 243
How do I create a line load? 244
How do I create a patch load? 244
How do I create a polygonal load? 245
How do I create a perimeter load? 246
How do I create a variable patch load? 246
How do I create an area load? 247
How do I create a variable area load? 247
How do I create a slab load? 248
How do I create a level load? 248
How do I edit a load? 248
How do I delete a load? 248
Working with Member Loads 249
How do I create a full UDL? 249
How do I create a partial length UDL or VDL? 249
How do I create a trapezoidal load? 249
How do I create a point or moment load? 250
How do I create a torsion full UDL? 250
How do I create a partial length torsion UDL or torsion VDL? 250
Working with Structure Loads 251
How do I create a nodal load? 251
How do I create a temperature load? 251
How do I create a settlement load? 251
Analysis Guide 253
Analyze toolbar 253
Analysis Options 257
How to apply and manage Analysis Options 257
To modify analysis options in the current project 257

To modify analysis option defaults for future projects 257
1st Order Non-Linear Options 257
2nd Order Non-Linear Options 258
1st Order Vibration Options 258
2nd Order Buckling Options 260
1st Order Seismic Options 260
Modification Factors 262
Meshing 263
Composite Steel Beams 263
Working with FE meshed slabs 264
How do I specify whether a level uses meshed slabs in analysis or FE load decomposition? 264
How do I control the mesh parameters to be used? 265
How do I apply different mesh parameters at different levels? 265
How do I review the slab mesh prior to running the analysis? 265
Slab mesh groups 266
Example of mesh groups at a slab step 266
Discontinuity of force contours at slab steps (and column drops) 267
Mesh group boundary warnings 269
Working with FE meshed walls 269
How do I control the global wall mesh parameters to be used? 269
How do I override the global wall mesh parameters for an individual wall? 269
How do I review the wall mesh prior to running the analysis? 270
Working with rigid and semi-rigid diaphragms 270
How do I activate rigid diaphragm action within a slab? 270
How do I activate semi-rigid diaphragm action within a slab? 270
How do I identify which nodes are constrained by rigid diaphragms? 271
How do I intentionally exclude individual nodes from a rigid diaphragm? 272
How do I intentionally exclude a slab panel from a diaphragm? 273
Running an analysis 273
How do I run 1st order linear or non-linear analysis? 273
How do I run a 1st order linear analysis? 273
How do I run a 1st order non-linear analysis? 273
How do I run 2nd order linear or non-linear analysis? 273
How do I run a 2nd order linear analysis? 273
How do I run a 2nd order non-linear analysis? 274
How do I run 1st order vibration analysis? 274
How do I run 2nd order buckling analysis? 274
How do I run 1st or 2nd order seismic analysis? 274

How do I run a 1st order RSA seismic analysis? 274
How do I run a 2nd order RSA seismic analysis? 275
How do I run FE chasedown and Grillage chasedown analysis? 275
How do I run a Analyze All (Static)? 275
Graphical display of the solver model (Solver View) 275
How do I open a Solver View? 276
How do I display the Solver Model used for a particular analysis type? 276
How do I see Solver Node and Solver Element properties? 276
Tabular data (Solver Model Data) 276
Tabular display of Node and Element Data 277
How do I view tabulated node co-ordinates and degrees of freedom? 277
How do I view tabulated solver element properties? 277
Tabular display of Reactions, Deflections and End Forces 277
How do I view tabular results for support reactions? 277
How do I view tabular results for nodal deflections? 277
How do I view tabular results for solver element end forces? 278
What does the asterisk next to certain nodes signify in the Element End Forces table? 278
How do I view tabular results for wall lines? 278
Tabular display of Vibration and Buckling Analysis results 279
How do I view tabular results for mode shapes? 279
How do I view the summed mass for vibration mass combinations? 279
How do I view the dynamic masses for vibration mass combinations? 279
How do I view the active masses by node? 279
How do I view the total masses by node? 279
How do I view vibration frequencies and modal masses? 280
How do I view buckling factors? 280
Graphical display of the analysis results (Results View) 280
Results toolbar 280
Result Type group 280
Scale Settings group 281
1D Results group 282
Deflections group 282
Notional Horizontal Forces 283
2D Results group 283
AsReq group 284
2D Deflections group 285
2D Strip Results group 285
2D Wall Results group 286

Text group 286
Cross checking the sum of reactions against the load input 287
Loadcase Properties 287
Displaying 1D and 2D results, deflections and reactions 289
How do I choose which analysis to see the results for? 289
How do I choose the loadcase, combination, or envelope to see the results for? 290
How do I display Support Reactions? 290
How do I display 1D Results and 1D Deflections? 290
How do I display 2D Results and 2D Deflections? 291
How do I display AsReq contours? 291
What are 2D Wall Results and how do I display them? 291
How do I increase the amplitude of the diagrams? 292
How do I see a 3D display of the results in a 2D view? 293
How do I customise the display of 2D contours? 293
Displaying Mode Shapes 294
How do I display mode shapes? 294
Displaying Notional Forces and Seismic Equivalent Lateral Forces 294
EHF 294
Notional Loads 294
NHF 295
Seismic 295
Displaying RSA Seismic Results 295
Result Type 295
Mode Shapes 295
1D Element Results 295
Story Shear 296
Support Reactions 298
2D Wall Results 298
Working with 2D Strips and displaying Strip Results 300
Result Strip Overview 300
How do I create a Result Strip? 300
How do I delete a Result Strip? 301
How do I view the results for a Result Strip? 301
How are the strip results calculated? 302
Displaying analysis results for individual members (Loading Analysis View) 302
Loading Analysis toolbar 302
How do I open a Loading Analysis View? 303
Displaying RSA Seismic Results in a Loading Analysis View 303
Loading Analysis View Properties for Columns 305

Loading Analysis View Properties for Beams 308
Analysis results sign conventions 311
Introduction 311
Axis Systems 311
General 311
Object Orientation 312
Diagram Conventions 312
Global Coordinate System 312
Global Axis System and Applied Load Directions 313
Resulting Deflection Directions 313
Building Directions 1 and 2 313
Building Directions and Applied Load Directions 313
Resulting Deflection Directions 314
User Coordinate System 314
UCS Axis System and Applied Load Directions 314
Resulting Deflection Directions 315
1D Member Local Coordinate System 315
General case for 1D members 315
Special case for 1D members 320
2D Member Local Coordinate System 323
Horizontal Panel Local Axis System and Applied Load Directions 323
Vertical and Sloped Panel Local Axis System and Applied Load Directions 323
2D Member Forces Sign Convention 325
Result Line Coordinate System 326
Result Axis System 326
General Case 326
Special Case 326
Result Strip Coordinate System 327
Result Axis System 327
General Case 327
Special Case 328
Foundation Reaction Coordinate System 328
Result Axis System and Directions 328
Reactions 329
Design Guide 331
Design toolbar 331
Design Options 333
How to apply and manage Design Options 333
To modify design options in the current project 333

To modify design option defaults for future projects....334
Design Options-General 334
Design Options-Analysis....335
Design Options-Concrete-Reinforcement Parameters 336
Design Options-Concrete-Beam 336
Concrete - Beam - Reinforcement Settings 336
Concrete - Beam - Detailing Options 337
Concrete - Beam - Top Longitudinal Bar Pattern 339
Concrete - Beam - Bottom Longitudinal Bar Pattern....340
Concrete - Beam - Link Settings 340
Concrete - Beam - General Parameters 342
Design Options-Concrete-Column....343
Concrete - Column - Reinforcement Layout 343
Concrete - Column - Detailing Options 345
Concrete - Column - General Parameters....346
Design Options-Concrete-Wall 347
Concrete - Wall - Reinforcement Layout 347
Concrete - Wall - Detailing Options 348
Concrete - Wall - General Parameters 349
Design Options-Concrete-Slab 350
Concrete - Slab - Reinforcement Layout 350
Concrete - Slab - Patches....351
Concrete - Slab - General Parameters 352
Design Options-Concrete Punching Shear 352
Design Options-Concrete Isolated Foundations....354
Concrete - Isolated Foundations - Reinforcement Layout 354
Concrete - Isolated Foundations - Foundation Size 354
Concrete - Isolated Foundations - General Parameters 355
Concrete - Isolated Foundations - Piles 356
Design Options-Concrete Mat Foundations....356
Design Options-Composite Beams....356
Design Options-Design Forces 356
Design Options-Design Groups 357
Design Options-Autodesign 357
Design Options-Design Warnings 358
Design Options-Steel Joists 360
Design Options-Wind Drift....360
Static Analysis & Design 360
Member design modes....361

Do I run Design Steel, Design Concrete or Design All? 361
Design Steel 361
Design Concrete 362
Design All 362
Do I run Design...(Gravity), or Design...(Static)? 362
Design (Gravity) 362
Design (Static) 362
Design Steel (Gravity) 363
How do I run Design Steel (Gravity)? 363
Design Steel (Static) 364
How do I run Design Steel (Static)? 364
Design Concrete (Gravity) 365
How do I run Design Concrete (Gravity)? 365
Design Concrete (Static) 365
How do I run Design Concrete (Static)? 366
Design All (Gravity) 366
How do I run Design All (Gravity)? 367
Design All (Static) 367
How do I run Design All (Static)? 368
How can I check if Autodesign is on or off? 368
RSA Analysis & Design 369
Design Steel (RSA) 369
How do I run Design Steel (RSA)? 369
Design Concrete (RSA) 370
How do I run Design Concrete (RSA)? 370
Design All (RSA) 371
How do I run Design All (RSA)? 371
Slab design, patches and punching shear 372
Patch Creation 372
What is a patch? 372
How do I create a column patch? 372
How do I create a beam patch? 373
How do I create a wall patch? 375
How do I create a panel patch? 376
How do I edit the properties of an existing patch? 377
How do I resize an existing patch? 378
Design Slabs 379
How do I design or check all slab panels in the model? 379
How do I check an individual slab panel? 379

How do I design an individual slab panel? 379
Design Patches....379
How do I design or check all patches in the model? 379
How do I check an individual patch? 379
How do I design an individual patch? 380
Add Punching Shear Checks 380
How do I create a Punching Check item? 380
Punching Check axis orientation 380
How do I specify stud rail reinforcement? 381
Overview of the Edit Reinforcement Dialog 381
How do I edit the properties of an existing Punching Check item? 384
Floor Vibration 385
How do I create a Floor Vibration Check item? 385
How do I create a check which considers two or three adjoining spans? 386
How do I edit the properties of an existing Floor Vibration Check item? 386
How do I check vibration for all Floor Vibration Check items? 387
How do I check floor vibration for an individual Floor Vibration Check item? 387
Foundations Guide 389
Foundations toolbar 389
Foundation Options 391
Pile Type Catalogue 391
How do I set up the pile type catalogue? 391
Pad and Strip Base modelling 391
How do I create a pad base column? 392
How do I create a strip base wall? 392
Pile Cap modelling 393
How do I create a pile cap? 393
User Defined Pile Arrangement 393
Pile Arrangement dialog 393
How do I open the Pile Arrangement dialog? 395
How do I create a user defined pile arrangement? 395
Isolated Foundation design 395
How do I design or check all pad and strip bases in the model? 395
How do I design or check all pile caps in the model? 396
How do I check an individual isolated foundation? 396
How do I design an individual isolated foundation? 396
Mat Foundation modelling 396
How do I create a minimum area or rectangular mat? 397
How do I create a strip mat? 397

How do I create an area mat? 397
How do I create a mat within bays? 398
How do I create a piled mat? 398
How do I specify if a piled mat is ground bearing? 398
How do I place an individual pile in a mat? 398
How do I place a pile array in a mat? 398
How do specify the pile direction of an inclined pile? 399
Mat design, patches and punching shear 400
Design Mats 400
How do I design or check all mats in the model? 401
How do I check all mats in a single floor? 401
How do I design all mats in a single floor? 401
How do I check an individual mat? 401
How do I design an individual mat? 401
Design Review Guide 403
Review toolbar 403
Review Data toolbar 408
Working with the Review View 408
Setting the Design Type to Review 408
Reviewing Member Design 408
Member Design Status 408
Member Design Ratio 409
Depth Ratio 410
Reviewing Foundation and Pile Design 410
Foundation Design and Pile Design Status 410
Foundation Design and Pile Design Ratios 411
Reviewing Slab Design 411
Slab/Mat Design Status 411
Slab/Mat Design Ratio 411
Slab/Mat Design droplist 411
Show and Alter State 412
Show/Alter State 412
Auto\Check Design 412
BIM Status 413
Braced/Bracing 414
Column Splices 415
Concrete Beam Flanges 415
Copy Properties 416
Deflection Limits 416

Diaphragm On\Off....417
Fixed\Pinned....419
Gravity Only....419
Property Sets....420
Punch Check Position....421
Report Filter....421
Restrained\Unrestrained....422
SFRS Direction....423
SFRS Type....424
Shear Connectors....425
Size Constraints....426
Slab Reinforcement....427
Stud auto layout....428
Sub Structures....429
Transverse reinforcement....429
UDA....429
Web openings....429
Westok openings....430
Working with the Review Data view....430
How do I display a Design Summary in a Review Data table?....430
How do I display Sway Results in a Review Data table?....430
How do I display Storey Shear in a Review Data table?....430
How do I display Drift Results in a Review Data table?....431
How do I display Seismic Drift Results in a Review Data table?....431
How do I display Wind Drift Results in a Review Data table?....431
How do I display Connection Resistance checks in a Review Data table?....431
How do I display a Material List in a Review Data table?....432
Reporting Guide....433
Report toolbar....433
Contents group....433
Filters Group....434
Appearance group....434
Navigation group....435
Export group....435
Reports and Report Terminology Explained....435
Creating Reports....438
How do I configure a Model Report?....438
How do I display a Model Report?....439
How do I configure a Member Report?....439

How do I display an individual Member Report? 440
How do I select the Member Report style to use in the Model Report? 440
How do I select the Member Report style to use in an individual Member Report? 440
How do I modify the Report Structure? 440
How do I print a Report? 441
Filtering Reports 441
What are the different types of filter? 441
How do I create and edit a filter? 442
How do I apply a filter? 442
Formatting Reports 443
Report Settings 443
How do I apply report settings? 443
Report Settings-Styles 444
Report Settings-Page Options 444
Report Settings-Table Options 444
Report Settings-Document Options 444
Report Settings-Picture Fonts 445
Headers and Footers 445
How do I enter company details (logo, address etc.) in the header? 445
How do I enter project specific details (job ref, structure etc.) in the header? 446
How do I create a new header layout? 446
Exporting Reports 447
How do I export a report to PDF? 447
How do I export a report to Word? 447
How do I export a report to Excel? 448
How do I export a report to Tedds? 448
Example Reports 448
Solver Model Data Report 448
How do I show a Solver Model Data Report? 448
Building Loading Report 448
How do I show a Building Loading Report? 449
Building Analysis Checks Report 449
How do I show a Building Analysis Checks Report? 449
Building Design Report 449
How do I show a Building Design Report? 449
Material Listing Report 449
How do I show a Material Listing Report? 450
Beam End Forces Report 450
What data is output in each Beam End Forces Report configuration? 450

How do I change the default Beam End Forces Report configuration? 451
How do I show a Beam End Forces Report? 452
Bracing Forces Report 452
How do I show a Bracing Forces Report? 452
Foundation Reactions Report 452
How do I show a Foundation Reactions Report? 454
Seismic Design Report 454
How do I show a Seismic Design Report? 454
Member Design Report 454
How do I show a Member Design Report? 454
Setting up the Report Page Header and Footer 454
Drawing Guide 457
Draw toolbar 457
Overview of Drawings 458
Drawing Settings 460
How to apply and manage Drawing Settings 460
To modify drawing settings in the current project 460
To modify drawing setting defaults for future projects 460
Export Preferences 461
How do I add a new drawing scale? 461
Layer Configuration 462
How do I add, modify, copy or remove a layer configuration? 462
Layer Style 463
How do I add, modify, copy or remove a drawing style? 464
Drawing Options 464
Options - Planar Drawings 465
General tab 465
Beams tab 466
Columns tab 466
Walls tab 467
Slabs and Mats tab 467
Loads tab 468
Options - Member Details 469
Beam Detail - Content 469
Grouped Beams tab 469
Levels tab 469
Cross sections tab 469
Bar Labels tab 470
Dimensions tab 470

Quantities tab 471
Beam Detail - Style 471
Beam Labels tab 471
Cross sections tab 472
Longitudinal Bars tab 472
Stirrups/Links tab 472
Dimensions tab 473
Column Detail - Content 473
Grouped Columns tab 473
Levels tab 473
Cross sections tab 473
Dimensions tab 473
Column Detail - Style 474
Column Labels tab 474
Cross sections tab 474
Longitudinal Bars tab 474
Ties/Links tab 474
Dimensions tab 474
Wall Detail - Content 475
Levels tab 475
Cross sections tab 475
Dimensions tab 475
Wall Detail - Style 475
Wall Labels tab 476
Cross sections tab 476
Horizontal & Vertical Bars tab 476
Horizontal Bars and Ties/Links tab 476
Dimensions tab 477
Options - Member Schedules 477
Beam Schedule Options 477
General tab 477
Bar Key tab 477
Column Schedule Options 478
Wall Schedule Options 478
Options - Slabs and Mats 479
Layout 479
Slabs and Mats tab 479
Patches tab 481
Punching Shear tab 481

Punching Check Detail - Content 481
Dimensions tab 481
Quantities tab 482
Punching Check Detail - Style 482
General tab 482
Options - Foundations 482
Foundation Layout 482
Slabs and Mats tab 482
Isolated Foundations tab 483
Isolated Foundation Detail-Content 484
Isolated Foundation Detail-Style 485
Planar Drawings 485
How do I create a General Arrangement drawing? 486
How do I create a Beam End Forces drawing? 486
How do I create a Column Splice Loads drawing? 487
How do I create a Foundation Reactions drawing? 487
How do I create a Loading Plan drawing? 488
DXF Export Preferences dialog 488
Member Details 489
How do I create a concrete beam detail? 490
How do I create a concrete column detail? 490
How do I create a concrete wall detail? 491
How do I create a non concrete beam detail? 491
How do I create a non concrete column detail? 491
Slabs and Mats 492
How do I create a Punching Check Detail drawing? 492
How do I create a Slab or Mat Layout drawing? 493
Foundations 493
How do I create a Base or Pile Cap Detail drawing? 494
How do I create a Foundation Layout drawing? 494
Drawing Management 494
How do I add new drawings and specify their content? 495
How do I specify the layout? 496
For load dependant drawing variants, how do I specify the loading? 496
How do I view a drawing? 496
How do I consolidate the bar marks used on concrete detail drawings? 496
How do I apply a revision to a drawing? 497
How do I view the revision history of a drawing? 497
Concrete Member Schedules 497

Concrete Member Schedules 497
How do I create a concrete beam schedule? 498
How do I create a concrete column schedule? 498
How do I create a concrete wall schedule? 498
Schedule Management 499
How do I add new drawings and specify their content? 499
How do I specify the layout? 499
How do I view a drawing? 500
How do I consolidate the bar marks? 500
How do I apply a revision to a drawing? 500
How do I view the revision history of a drawing? 501
Object Properties 503
Structure, Level, Frame and Slope Properties 503
Structure Properties 503
Level Properties 505
Frame Properties 507
Slope Properties 508
Steel, Concrete and Timber Member Properties 509
Steel Member Properties 509
edit steel column properties set 510
edit steel beam properties set 514
edit composite steel beam properties set 517
edit steel brace properties set 521
edit steel joist properties set 524
edit gable post properties set 526
edit parapet post properties set 529
edit steel truss properties set 532
Concrete Member Properties 532
edit concrete wall properties set 533
edit concrete wall span properties set 537
edit concrete column properties set 540
edit concrete beam properties set 546
Timber Member Properties 551
Timber property assumptions 552
edit timber column properties set 552
edit timber beam properties set 555
edit timber brace properties set 557
edit timber truss properties 559
Cold Formed Member Properties 560

create cold formed column properties set 561
create cold formed beam properties set 563
create cold formed brace properties set 565
edit cold formed column properties set 566
edit cold formed beam properties set 570
edit cold formed brace properties set 572
edit cold formed truss properties 574
Cold Rolled Properties 575
create cold rolled section properties set 576
edit cold rolled section properties set 577
Slab and Mat Properties 579
Slabs Properties 579
slab properties 579
edit slab on beams item properties set 580
edit flat slab item properties set 584
edit precast item properties set 588
edit steel deck item properties set 591
edit timber deck item properties set 592
edit composite slab item properties set 594
Mat properties 597
Foundation Mat properties 597
edit foundation mat properties set 598
Openings, Overhangs and Column Drops 602
slab opening and mat opening properties 602
slab overhang and mat overhang properties 603
column drop properties 603
Rotation angle for panels 604
Roof Panel and Wall Panel Properties 605
Roof panel properties 605
Wall panel properties 606
Support, Element and Bearing Wall Properties 607
Support properties 608
Element properties 610
Bearing wall properties 612
Definition of rotation angle and gamma angle 613
Rotation angle 613
Gamma angle 616
Patch, Result Strip and Punching Check Properties 616
result strip properties 616

Isolated Foundation and Pile Properties 617
Pad Base Column Properties 617
Strip Base Wall Properties 622
Pile Cap Column Properties 625
Pile Type Properties 629
Pile Properties 631
create pile properties set 631
edit pile properties set 631
Pile Array Properties 632
Sub Model Properties 633
Solver Model Properties 634
Solver Node properties 634
Solver Element properties 634
Solver Element 2D properties 635
Glossary 637

Basics of Tekla Structural Designer

About Tekla Structural Designer

Tekla Structural Designer is a tool for structural engineers. It is an integrated model-based 3D solution for analysis and design[1] of multi-material structures. *Tekla Structural Designer* features interactive modeling, automated structural analysis and design, drawing and report creation.

Tekla Structural Designer supports multiple design codes including ACI/AISC, Eurocodes, British, Indian and Australian Standards.

Tekla Structural Designer Philosophy

One of the main aims of *Tekla Structural Designer* is to allow you to rapidly build your model, apply loads and then design it for an appropriate set of design forces. On a day to day basis, you don't need to be involved with the underlying analysis models required to achieve this, instead you can focus on the design results…

To make this possible, the program automatically creates and analyses multiple solver models, each one being based on a different but widely accepted approach.

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

How does the Tekla Structural Designer way of working differ from traditional methods?

The traditional modelling, analysis and design process might be summarized under the following headings:

1. Provide a way to input/describe the model.
2. Analyze it.
3. Design it.
4. Produce Calculations.
5. Produce Drawings.

A similar process is followed in *Tekla Structural Designer* with the exception that analysis and design are merged into a single process.

As a result the workflow is as follows:

1. **Input the Geometry and Loads:**
   A key requirement these days is "BIM Integration" - the ability to be able to transfer data from one application to another. *Tekla Structural Designer* has tools to automatically import model data from Neutral Files and from 3D DXF to facilitate this. Of course you can always build the model directly - the Quick Start Guides covers many of the modeling techniques required to achieve this so no further mention is required here.
   Once the physical model has been created, the next step is to load it. A wide range of loads can be applied in a flexible system of loadcases, with a wind load generator available to automatically create the wind load cases. Load combinations can also be generated automatically.
   You should also consider pattern loading - patterned beam loads can be created automatically; patterned slab loads can be manually created for design of slabs.

2. **Combined Analysis and Design:**
   Tekla Structural Designer automatically performs the analyses required to enable member design to proceed: in effect Analysis and Design (with the exception of slab design) are combined into a single automated process.

3. **Produce Reports:**
   A wide range of calculations can be created, which you can tailor extensively to meet your specific requirements

4. **Produce Drawings:**
   Beam and Column detail drawings can be produced and member schedules can also be produced.

1. Whilst multiple materials can be analysed, design is only supported for concrete and steel members.

Interface Overview

Interface components

File menu

The **File menu** contains those commands that can be used to perform file related operations

Button	Description
	New (button) creates a new blank project from scratch.

	Open an existing project.
	Save the currently open project.
	Save As saves the currently open project to a new name, or to a template.
	Save Model Only saves smaller "model only" files without the analysis results, which can be easily shared amongst the project team.
	Close the currently open project.
	Print the currently open view.
	Start Revision records changes to this revision of the project.
	Send As Email creates a new e-mail with the project attached.
	Exit prompts you to save any open project then closes the program.

Quick Access Toolbar

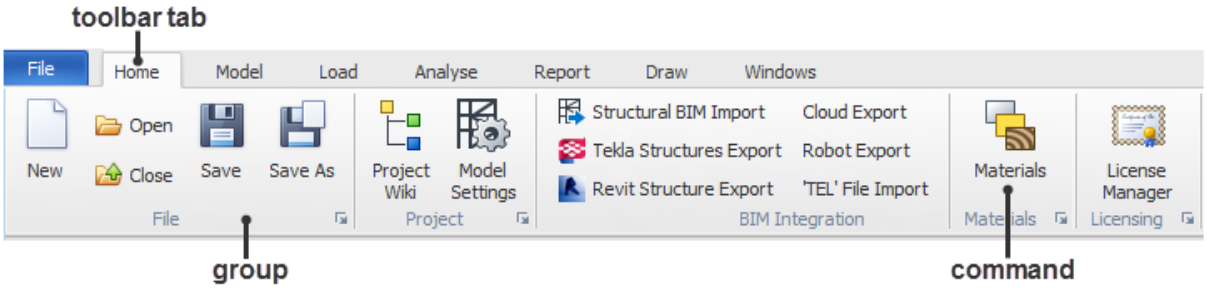
The **Quick Access Toolbar** displays commonly used commands: New, Open, Save, Undo, Redo, Delete and Validate.

Viewcube

In 3D Views - click a vertex, edge, or face of the **Viewcube** to rotate the model to a preset view.

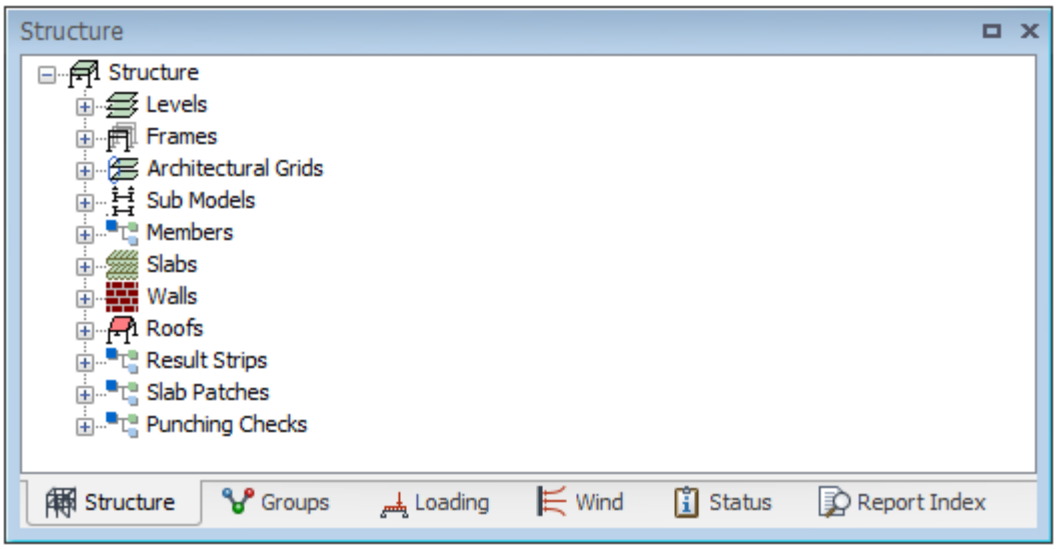
Ribbon

The **Ribbon** consists of a number of toolbars placed on tabs. Each toolbar contains related commands organized into logical groups.



Project Workspace

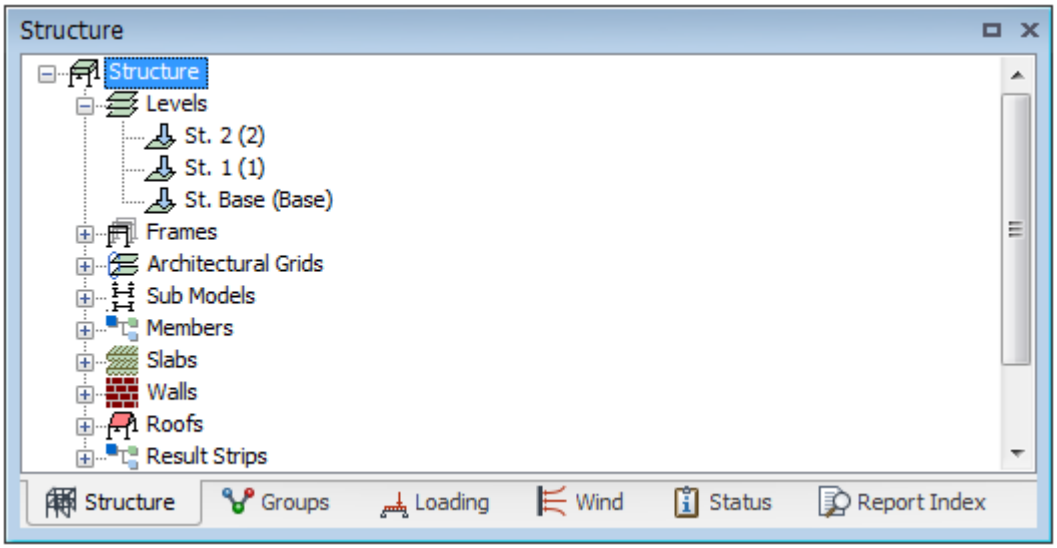
The **Project Workspace** is a central location for organising the entire model into a hierarchical structure.



Because the **Project Workspace** provides access to a whole range functions it is a key tool for creating and controlling your model.

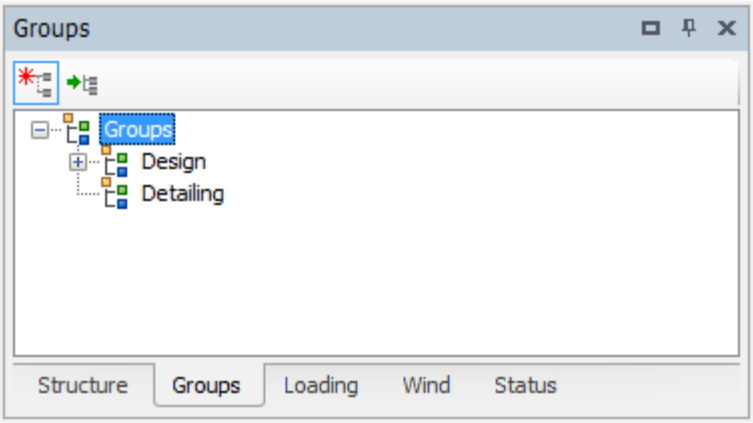
Structure Tree

Levels, grids, elements and certain other model properties are accessible from the **Structure Tree** located on the Structure tab of the **Project Workspace**.



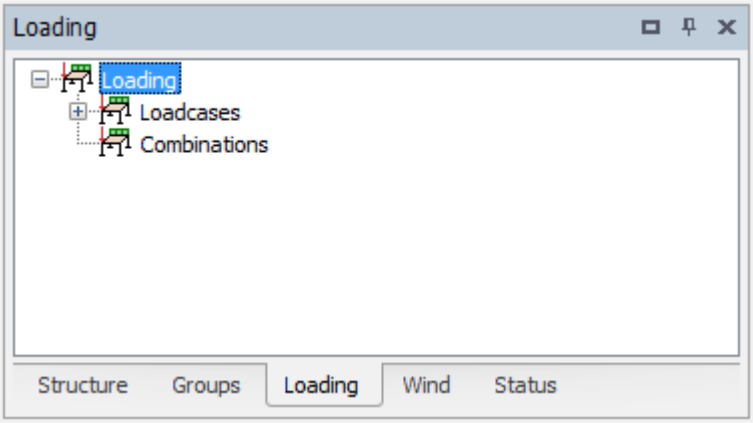
Groups Tree

The **Groups Tree** located on the Groups tab of the **Project Workspace** is used to organize members into Design and Detailing groups.



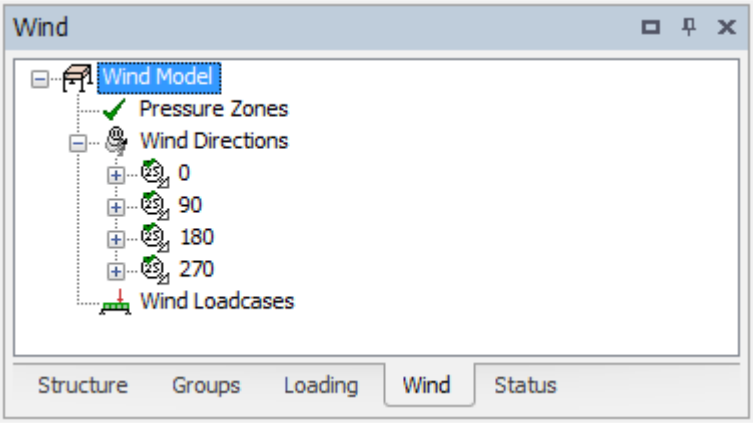
Loading Tree

The **Loading Tree** located on the Loading tab of the **Project Workspace** is used to summate the total loads applied in each of the loadcases and combinations.



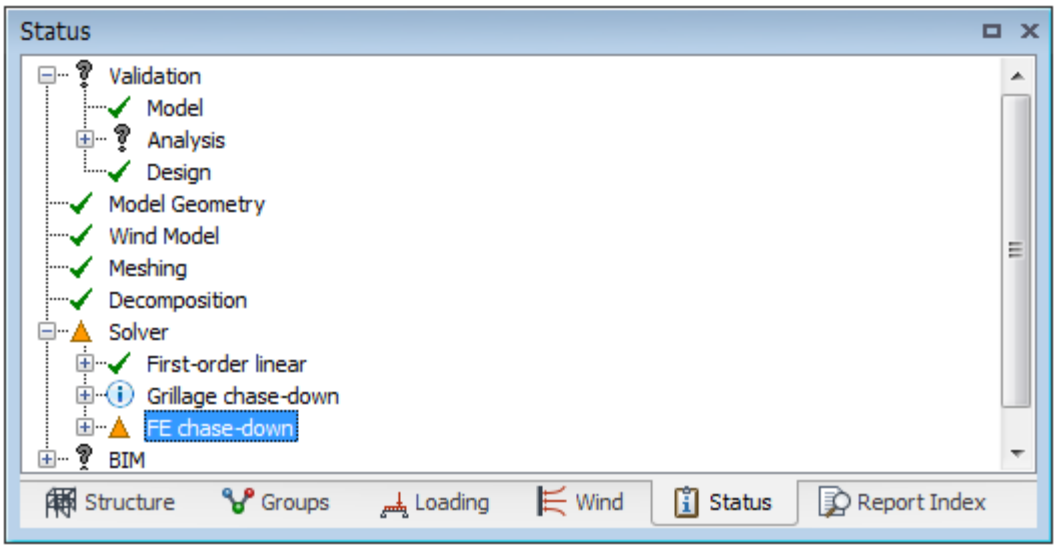
Wind Model Tree

The **Wind Model Tree** located on the Wind tab of the **Project Workspace** is used to access Wind Direction Views and Wind Loadcases.



Status Tree

The **Status Tree** located on the Status tab of the **Project Workspace** is used to review the status of the model and analyses that have been performed. It also displays the import/export status.

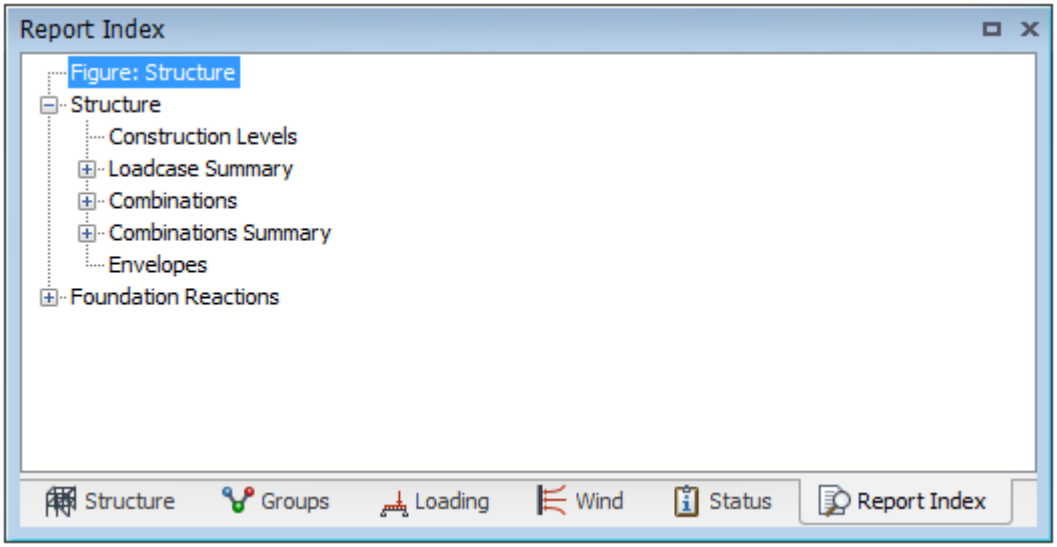


Cutting Planes

The **Cutting Planes** symbol is displayed in the bottom right hand corner of the Scene View when any of the cutting planes are currently active.

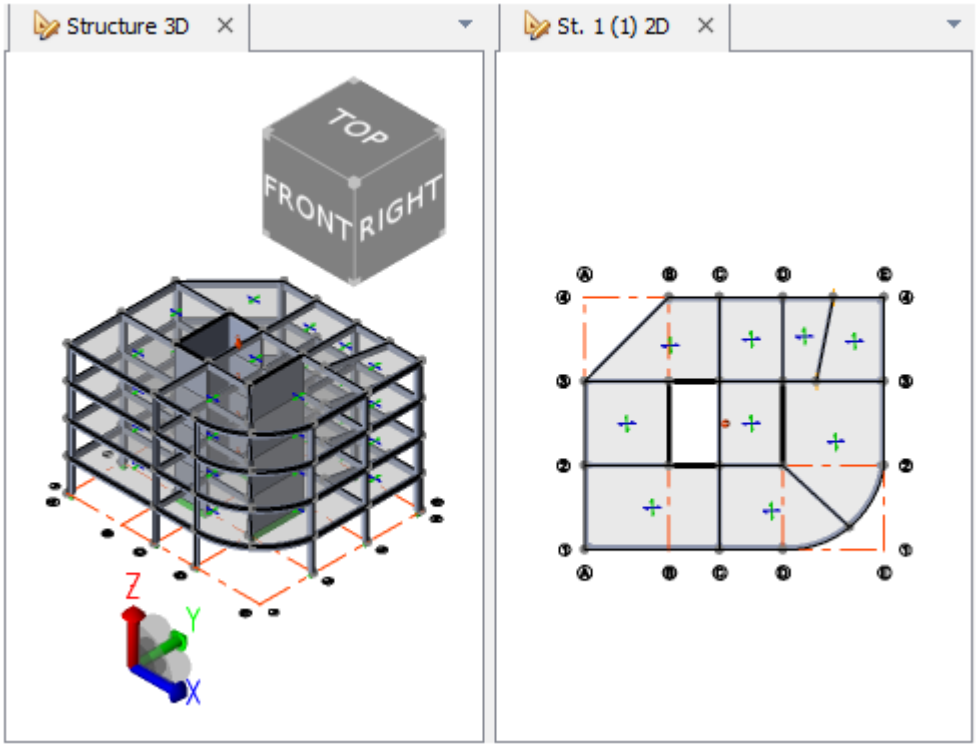
Report Index

The **Report Index is** located in the **Project Workspace**. When a Report is displayed it contains bookmarks that can be used to move around the report.



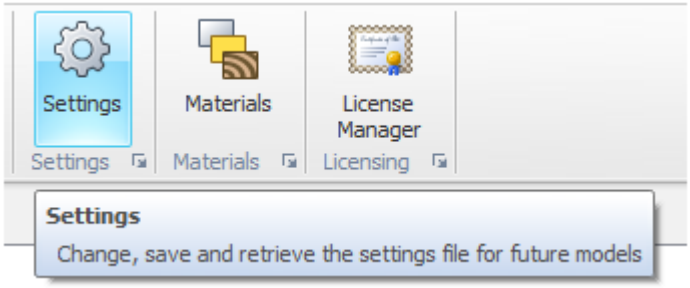
Scene Views

2D and 3D **views** (and **solver views**) of the model, sub models, frames, construction levels and individual members can be displayed in tabbed windows.



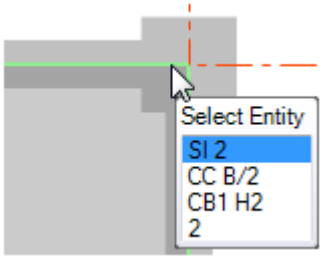
Basic Tooltip

Basic tooltips display the name of the command and may also include a brief description of its function.



Select Entity Tooltip

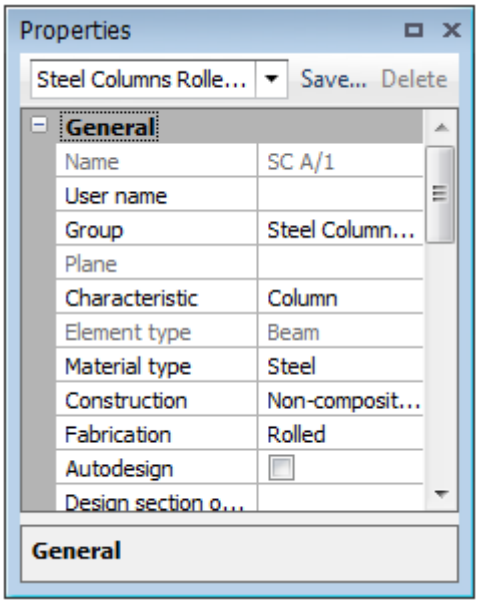
The program is automatically in "select mode" when no other commands are being performed. In this mode you can hover the cursor over an entity and its name will be displayed in the **Select Entity Tooltip**.



When the correct entity is displayed click it to select it. If several entities are displayed you control which one will be selected by using the tab or cursor keys.

Properties Window

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



It is used as follows:

- to input data when a command is run from a toolbar
- to review/edit existing properties when individual or multiple items are selected in the active **Scene View**
- to review/edit properties when a branch of the **Project Workspace** is selected

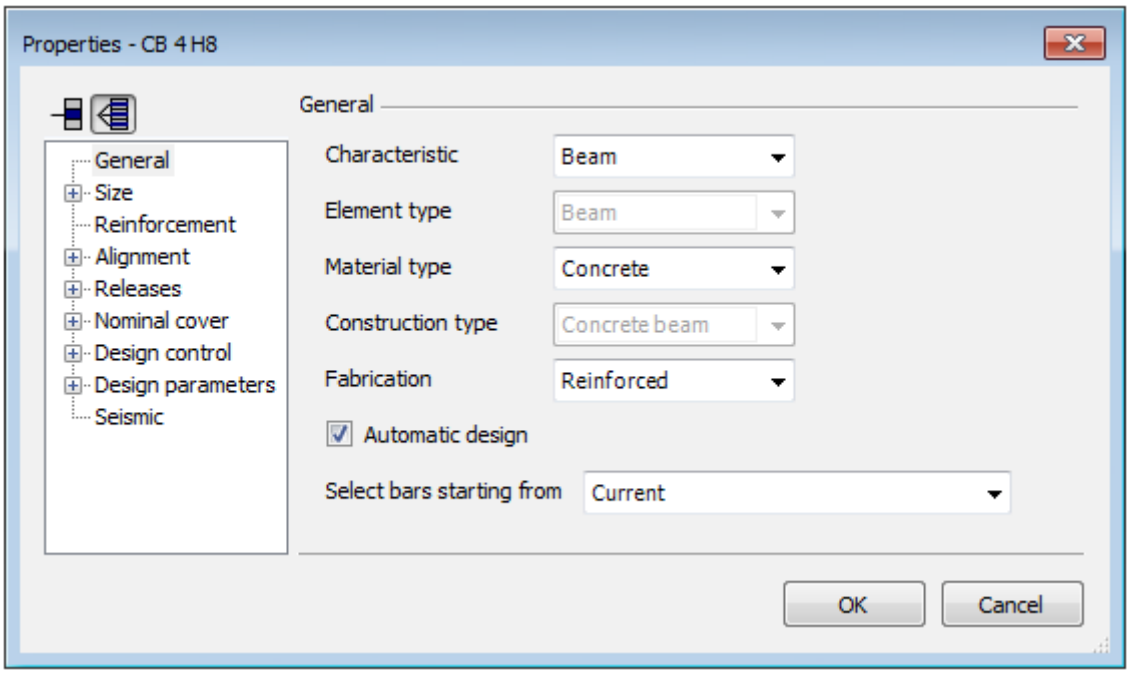
The properties displayed vary according to the selection - they are generally editable unless they are dimmed, in which case they cannot be changed.

A key feature of the Properties Window is that it enables the editing of multiple selected items in one go. Existing properties of selected items are only displayed where they are identical for all selected items - where they differ the property field is left empty.

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

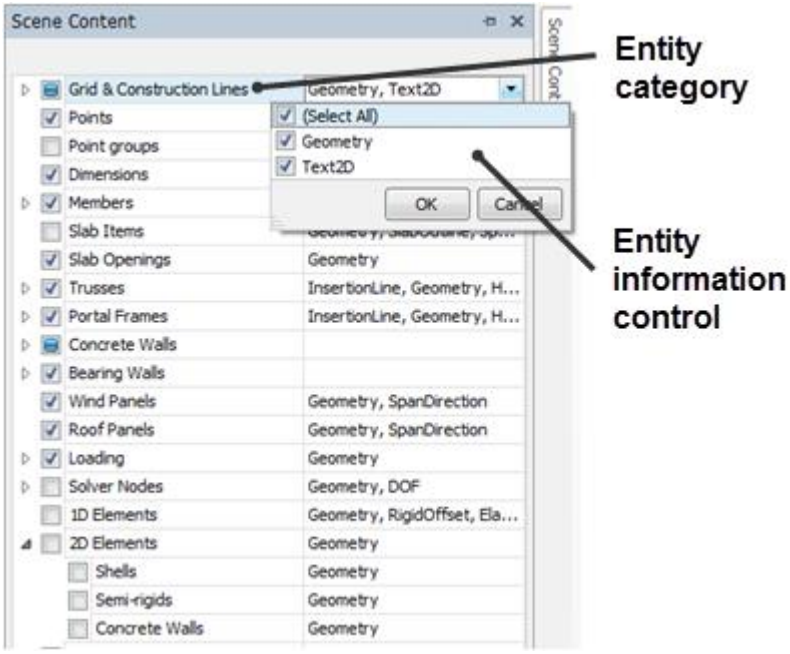
Property Dialog

The **Property Dialog** is used for viewing and editing properties associated with an individual model object. It is displayed by right-clicking on the object in the graphical display and picking the **Edit** option from the context menu that appears.



Scene Content

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



The window contains:

Entity categories (with check boxes)

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

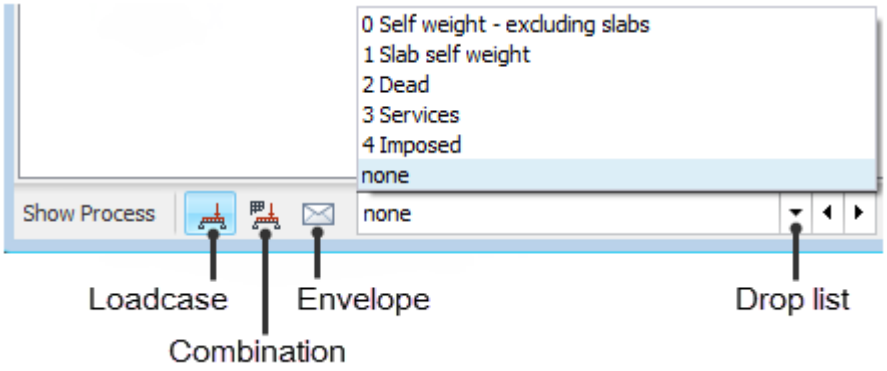
- The check box controls whether the entity category and its associated information is displayed - Simply select the entities you want to see and clear those that you do not.

Entity information controls

- These list the information in each category that will be displayed. When clicked they expand to drop list menus allowing you to select the information required.

Loading drop list

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



It has two main functions:

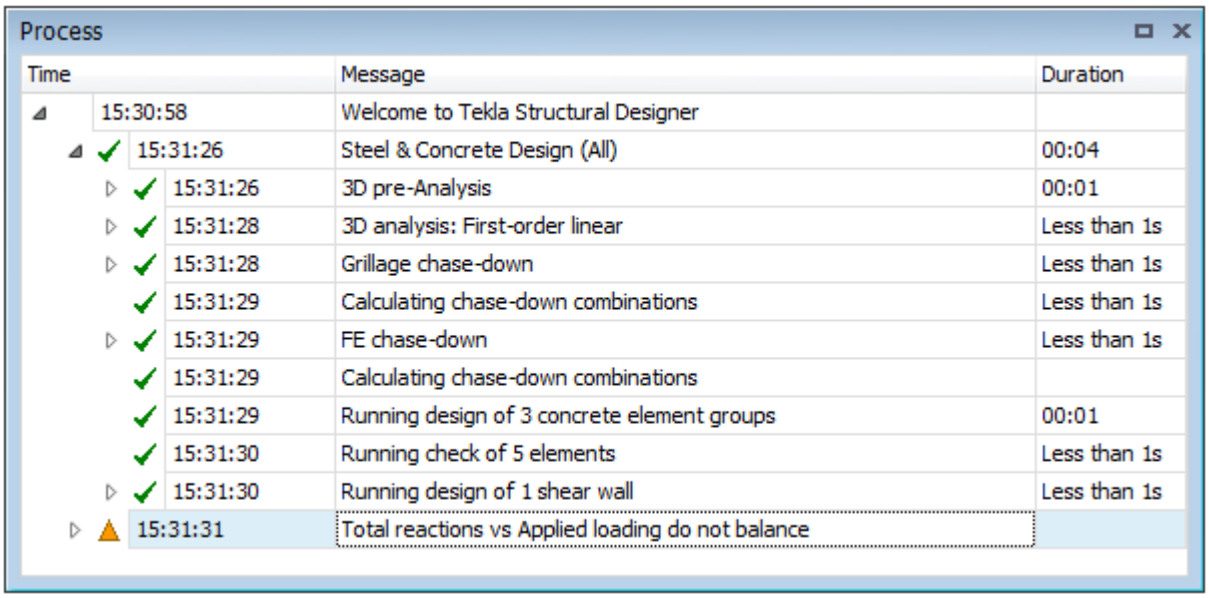
- selecting a specific loadcase to add loads in to,
- selecting a specific loadcase, combination, or envelope to view the results for.

When viewing results, first click the **Loadcase**, **Combination**, or **Envelope** button as required, then choose the specific loadcase, combination, or envelope name from the drop list.

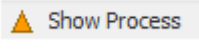
Process Window

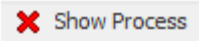
Initially the **Process Window** is minimized; it can be displayed by clicking the **Show Process** button located at the left end of the **Status Bar**.

When you analyze or design the model, each step of the process is logged and displayed in the window.



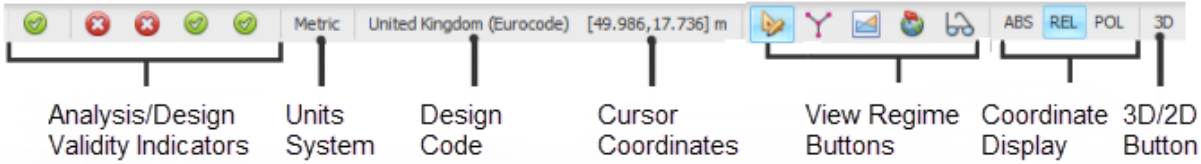
When the window is minimized to the **Status Bar**, if warnings or errors have been detected these are flagged thus:

- 

- 

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

Status Bar



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

It provides feedback by indicating:

- The **Analysis and Design Validity**, (hover over each indictor to display further details).
- The **Units System** (Metric, or US Customary)
- The **Design Code**
- In 2D Views - the **Coordinates** of the cursor relative to the global origin.

The **View Regime** buttons can be used to switch the view regime applied to the active scene view:

- Structural View (shows the geometry and loading)
- Solver View (shows the analysis model)
- Results View (shows the analysis results)

- Wind View (shows the wind model)
- Review View (for graphically interrogating the model properties/status)

It sets the display method for coordinate tooltips:

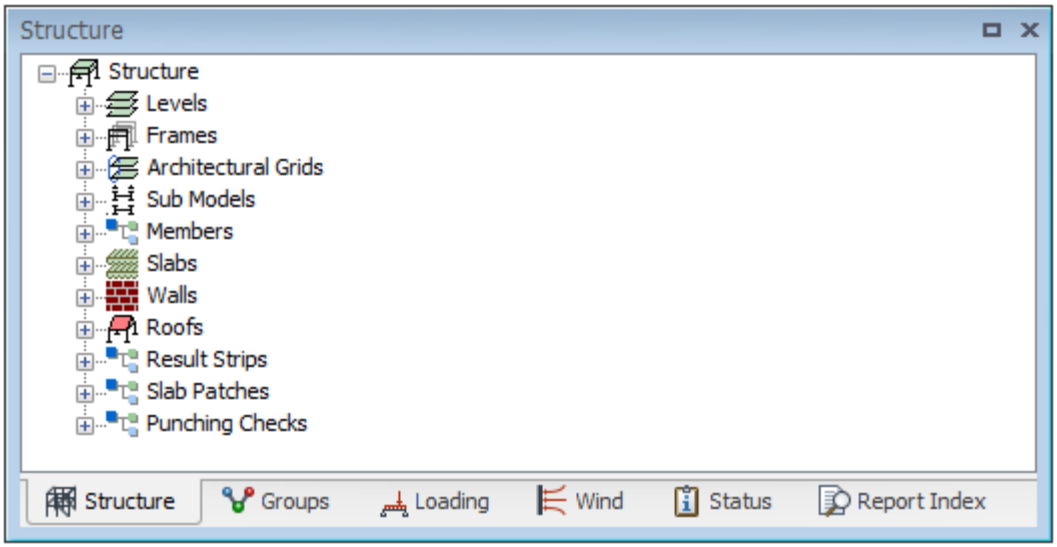
- **ABS** (Absolute)
- **REL** (Relative)
- **POL** (Polar)

In 2D views - the **3D/2D toggle button** is activated:

- 2D - the content of the 2D view is displayed in plan
- 3D - the content of the 2D view is displayed in isometric

Hiding, re-displaying and moving windows

The **Properties Window**, **Process Window**, **Scene Content** and each of the **Project Workspace** tabs (shown below) are displayed in windows that can be resized and repositioned, or docked to an edge of another window.



To auto hide a window:

To increase the area available for graphical display you can choose to Auto Hide any of the windows.

1. Click the window **Auto Hide** option icon ( ).
2. The window contracts immediately to a tab.
3. Click the window tab to expand it.

You can only do this if the window is docked against an edge of the main window.

To close a window:

- Click the icon at the top-right of the window.

To re-display a window that has been closed:

- Click the Windows tab
- In the **View** group click the window name.

If the window is already switched on its control will be emphasized on the View group.

To move a window to a new location

- Grab the window by its handle (the title bar at the top of the window) and drag it to its new location.
- If you place the window over an edge of the main window, or over a divider within it, then it will dock to that edge or divider.

To dock a window as a tabbed page in another window

- Grab the window by its handle (the title bar at the top of the window) and drag it to over the title bar of the other window, or over the tab group of the other window.

To open a tabbed page in another window

- Click the tab and drag it to a new location. It will then open in a new window.

To dock a window using the docking control

When you grab a window, and drag it over another window, a docking control appears in the middle of that window. You can then use this control to dock the window in a number of ways.

1. To dock to a tabbed page of the current window:

- Drag the grabbed window over the central button of the docking control and then release the mouse.

  The grabbed window becomes a tab of the current window.

2. To dock as a new window at an edge of the current window:

- Drag the grabbed window over one of the buttons adjacent to the central button of the docking control and then release the mouse.

  The grabbed window is docked above, below, to the left, or to the right of the current window (depending on the button chosen in the docking control).

3. To dock as a new window at an edge of the main window:

- Drag the grabbed window over one of the outer buttons of the docking control and then release the mouse.

  The grabbed window is docked at the top, bottom, left, or right of the main window (depending on the button chosen in the docking control).

Commands on the ribbon toolbars

The majority of commands are organized on toolbars located on the ribbon.

- Home toolbar
- Model toolbar
- Edit toolbar
- Load toolbar
- Zone Loads toolbar
- Analyze toolbar
- Design toolbar
- Report toolbar
- Draw toolbar
- Results toolbar
- Review toolbar
- Review Data toolbar
- Loading Analysis toolbar

Right click menu commands

Apply property set...

Use the **Apply property set** command to display the **Select property set** dialog, so that a previously saved property set can be applied to the currently highlighted element.

Check Member

Use the **Check Member** command to check an individual member using the existing Static or RSA analysis results. A results dialog is then displayed allowing the detailed calculations to be viewed.

Do any of the following:

To check	Do this

a member from a 2D or 3D view	**1.** Hover the mouse pointer over the member. **2.** If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in t **Select Entity tooltip** **3.** Right click and select **Check Member** > **Static/RSA** (as required)
a member from the Structure Tree	**1.** In the **Members** branch locate the member reference. **2.** Right click and select **Check Member** > **Static/RSA** (as required)

Check Panel

Use the **Check Panel** command to check the highlighted slab panel using the existing analysis results. The results dialog is then displayed allowing the detailed calculations to be viewed.

1. Hover the mouse pointer over the panel.

2. If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in the **Select Entity tooltip**

3. Right click and select **Check Panel**

Check Patches

Use the **Check Patches** command to check all slab patches in the model or just those in the current level/slope using the existing analysis results.

Do any of the following:

To check	Do this
all patches in the model	**1.** Right click in the 3D Structure view. **2.** Select **Check Patches**
all patches in the current level/slope	**1.** Right click in the level or slope view. **2.** Select **Check Patches**

Check plane

Use the **Check plane** command to perform a check of all members in the selected plane using the existing analysis results.

Do any of the following:

To check	Do this
all members in a level	**1.** Right click the required **Level** in the Structure Tree. **2.** Select **Check plane**
all members in a frame	**1.** Right click the required **Frame** in the Structure Tree. **2.** Select **Check plane**
all members in a slope	**1.** Right click the required **Slope** in the Structure Tree. **2.** Select **Check plane**

Check plane patches

Use the **Check plane patches** command to perform a check of all patches in the selected plane using the existing analysis results.

Do any of the following:

To check	Do this
all patches in a level	**1.** Right click the required **Level** in the Structure Tree. **2.** Select **Check plane patches**
all patches in a slope	**1.** Right click the required **Slope** in the Structure Tree. **2.** Select **Check plane patches**

Check plane slabs

Use the **Check plane slabs** command to perform a check of all slabs in the selected plane using the existing analysis results.

Do any of the following:

To check	Do this
all slabs in a level	**1.** Right click the required **Level** in the Structure Tree. **2.** Select **Check plane slabs**
all slabs in a slope	**1.** Right click the required **Slope** in the Structure Tree. **2.** Select **Check plane slabs**

Check Slabs

Use the **Check Slabs** command to check all the slabs in the model or just those in the current level/slope using the existing analysis results.

Do any of the following:

To check	Do this
all slabs in the model	**1.** Right click in the 3D Structure view. **2.** Select **Check Slabs**
all slabs in the current level/slope	**1.** Right click in the level or slope view. **2.** Select **Check Slabs**

Check Truss

Use the **Check Truss** command to perform a check of all truss members in the highlighted truss using the existing analysis results.The results dialog is then displayed allowing the detailed calculations to be viewed.

1. Hover the mouse pointer over the truss.
2. If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in the **Select Entity tooltip**
3. Right click and select **Check Truss**

Check Wall

Use the **Check Wall** command to check the highlighted wall using the existing analysis results. The results dialog is then displayed allowing the detailed calculations to be viewed.

1. Hover the mouse pointer over the wall.
2. If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in the **Select Entity tooltip**
3. Right click and select **Check Wall**

Copy

Copies existing selected entities to new locations in the model.

Create property set...

Creates a new property set based on the properties of the currently highlighted element.

Delete Element

Deletes the currently highlighted element.

Design Member

Use the **Design Member** command to perform a design of the highlighted member or slab panel. The results dialog is then displayed allowing the detailed calculations to be viewed.

Design Member *ignores the auto-design setting of the member. (i.e. it will always perform a design even if the auto-design property is off.*

Design Member *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this
a member from a 2D or 3D view	**1.** Hover the mouse pointer over the member. **2.** If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in t **Select Entity tooltip** **1.** Right click and select **Design Member** > **Static/RSA** (as required)
a member from the Structure Tree	**1.** In the **Members** branch locate the member reference. **2.** Right click and select **Design Member** > **Static/RSA** (as required)

Design Patches

Use the **Design Patches** command to perform a check or design (according to individual autodesign settings) of all slab patches in the model or just those in the current level/slope.

Design Patches *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this
all patches in the model	**1.** Right click in the 3D Structure view. **2.** Select **Design Patches**
all patches in the current level/slope	**1.** Right click in the level or slope view.

	2. Select **Design Patches**

Design plane

Use the **Design plane** command to perform a check or design (according to individual autodesign settings) of all members in the plane according to individual autodesign settings.

This command requires at least one member in the selected plane to be in auto-design mode.

If any grouped members are designed and the grouped member design option is on, all members of the group are considered in the design even if they are not in the selected plane.

Design plane *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this
all members in a level	**1.** Right click the required **Level** in the Structure Tree. **2.** Select **Design plane**
all members in a frame	**1.** Right click the required **Frame** in the Structure Tree. **2.** Select **Design plane**
all members in a slope	**1.** Right click the required **Slope** in the Structure Tree. **2.** Select **Design plane**

Design plane patches

Use the **Design plane patches** command to perform a check or design (according to individual autodesign settings) of all patches in the plane according to individual autodesign settings.

This command requires at least one patch in the selected plane to be in auto-design mode.

Design plane patches *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this
all patches in a level	**1.** Right click the required **Level** in the Structure Tree. **2.** Select **Design plane patches**
all patches in a slope	**1.** Right click the required **Slope** in the Structure Tree. **2.** Select **Design plane patches**

Design plane slabs

Use the **Design plane slabs** command to perform a check or design (according to individual autodesign settings) of all slabs in the plane according to individual autodesign settings.

This command requires at least one slab in the selected plane to be in auto-design mode.

Design plane patches *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this
all slabs in a level	**1.** Right click the required **Level** in the Structure Tree. **2.** Select **Design plane slabs**
all slabs in a slope	**1.** Right click the required **Slope** in the Structure Tree. **2.** Select **Design plane slabs**

Design Slabs

Use the **Design Slabs** command to perform a check or design (according to individual autodesign settings) of all the slabs in the model or just those in the current level/slope.

Design Slabs *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this

all slabs in the model	**1.** Right click in the 3D Structure view. **2.** Select **Design Slabs**
all slabs in the current level/slope	**1.** Right click in the level or slope view. **2.** Select **Design Slabs**

Design Truss

Use the **Design Truss** command to perform a check or design (according to individual autodesign settings) of all truss members in the highlighted truss. The results dialog is then displayed allowing the detailed calculations to be viewed.

Design Truss *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

1. Hover the mouse pointer over the truss.
2. If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in the **Select Entity tooltip**
3. Right click and select **Design Truss**

   Requires at least one truss member in the highlighted truss to be in auto-design mode.

Design Wall

Use the **Design Wall** command to perform a design of the highlighted wall. The results dialog is then displayed allowing the detailed calculations to be viewed.

Design Wall *ignores the auto-design setting of the member. (i.e. it will always perform a design even if the auto-design property is off.*

Design Wall *does* **not** *perform an analysis before the design - it always uses the existing analysis results, (even if the analysis status is 'Out of Date').*

Do any of the following:

To design	Do this

a wall from a 2D or 3D view	**1.** Hover the mouse pointer over the wall. **2.** If a different entity is highlighted, press the **<down arrow>** cursor key until the correct reference is shown in t **Select Entity tooltip** **1.** Right click and select **Design Wall** > **Static/RSA** (as req
a wall from the Structure Tree	**1.** In the **Walls** branch locate the wall reference. **2.** Right click and select **Design Wall** > **Static/RSA** (as req

Edit

Displays the **Edit Element Properties** dialog, opened at the General page for the currently highlighted element.

Generate Detailing Drawing...

Creates a **Drawing** for the currently highlighted member.

•

Interactive Design...

This command is only available for concrete beams and columns.

When clicked, it opens a dialog showing the existing design of the highlighted member for the currently supplied reinforcement. From here you can interactively modify the reinforcement and instantly see the result.

Open load analysis view

Opens a **Loading Analysis View** for the currently highlighted member.

Open member view

Opens a **Member View** containing the currently highlighted member.

The Member toolbar includes commands to edit the member, design it, create a member report and generate a member detail drawing.

Open view

Opens a 2D view of the frame, slope or level that contains the currently highlighted panel.

Redraw

Redraws and updates the current view.

Report for member

Opens a **Report View** for the currently highlighted member.

•

Save Screenshot

Saves the currently displayed view as a png or jpg file.

Save View Configuration...

Saves the currently displayed orientation and zoom state of the active Scene View under a given name.

View Configurations can be used in two ways:

1. They can be included in **Model Reports**.
2. They can be re-opened in a new Scene View at a subsequent time.

Show Member Loading

The **Member Loading** dialog tabulates all the loads applied to the currently highlighted member.

To filter the Member Loading data by Loadcase, Source, Direction, or Type

1. Click the appropriate column header to filter by (Loadcase..., Source..., Direction..., or Type...)
2. From the drop list that appears clear the categories that you don't want to be displayed.
3. Click the Close button under the drop list.

Undo/Redo

Undo and redo can be used to repetitively undo/redo operations in the sequence they were performed.

Zoom Out

Zooms out to the extents of the model.

Zoom to selection

Zooms out to the extents of the current selection.

Keyboard functions

Keys which perform specific functions in *Tekla Structural Designer* are listed below:

Key	Description
F1	Displays Help.
F2	Enables keyboard input of values into "Data Entry" tooltips.
F8	Toggles display of the ViewCube in 3D views.
Up/Down cursor keys	Used to scroll between entities in the **Select Entity** tooltip when multiple entities have been detected.

Ctrl	Holding Ctrl when selecting adds to the current selection.
Esc	Cancels the current command.
ZA	Zoom Extents.
ZS	Zoom to Selection.

Customising the appearance of the user interface

You can use skins to change the appearance of the user interface.

To apply a new skin:

1. Click **Window** > **Skins** (drop list)
2. Select a new skin from the list.

Working with the Project Workspace

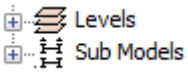
The Project Workspace is the central control area for your model, providing access to a range of functions.

The different Project Workspace pages are accessed by clicking the appropriate tab. Each page contains an expandable tree.

Working with the Structure Tree

The purpose of the Structure Tree is to organize the model geometry in a hierarchical way.

When opened for a new model it contains only two sub branches as shown below.



As the model geometry develops new branches are added as and when required.

When a branch or sub-branch is selected, the common properties of the branch/sub branch are also displayed in the Properties Window from where they can be edited.

Working with the Groups Tree

The Groups Tree is used to organize members into design groups.

If concrete members have been defined these are also organized into further groups for detailing purposes.

The application of grouping is of most benefit in concrete structures.

You can right-click a group name in order to select all of its members simultaneously in the visible views.

If any user defined attributes (UDAs) have been defined these are also displayed on the Groups Tree, so that all members with a particular attribute value can similarly be selected in the visible views.

How do I re-apply automatic grouping in order to reset manually edited groups?

There are occasions where it becomes necessary to re-apply automatic grouping for a specific member type.

To re-group a specific member type

1. In the **Groups Tree**, right-click the member type to be re-grouped.
2. Choose **Regroup Members** from the right-click menu.
   Grouping is reapplied for the chosen member type only.

To regroup ALL member types

1. Click (located at the top of the **Groups Tree**) to automatically regroup all member types and undo any manual grouping.

How do I split an existing member group into smaller groups?

1. Right-click the group name to be split in the **Groups Tree**.
2. Choose **Split Group** from the right-click menu.
   All members of the group are placed into individual groups - in effect making them un-grouped.

How do I manually move an existing member between groups?

1. In the **Groups Tree** open the group containing the member to be moved.
2. Click and drag the member name over the group name to which it is to be moved.

   Provided that the member meets the geometric criteria to belong to the group, a small rectangle will be displayed alongside the cursor. At this point release the mouse button to move the member to the new group.

   If the member does not meet the geometric criteria to belong to the group, the cursor will display a barred circle.

How do I remove an existing member group?

1. Right-click the group name to be removed in the **Groups Tree**.
2. Choose **Remove Group** from the right-click menu.
   All members of the group are placed into individual groups - in effect making them un-grouped.

How do I rename groups?

To manually rename individual groups in the current project:

1. In the **Groups Tree** right-click the group to be renamed.
2. Choose **Rename Group** from the right-click menu.

To modify the group name defaults for future projects:

1. Click **Home** > **Settings** ( )
2. In the **Settings Sets** page of the dialog select the settings set to be updated.

You can update any settings set simply by selecting it from the droplist, it does not need to be active.

3. In the **Grouping** page of the dialog, review and edit the default group names as required.
4. If you change any of the settings, click:

- **OK** - to save the changes to the selected settings set (to act as defaults for future projects when that set is active), or
- **Cancel** - to cancel the changes

Working with the Loading Tree

The Loading Tree is used to organize the loadcases and combinations into a hierarchical structure.

It also provides a summary of each loadcase that can be used to cross check the sum of reactions determined by each of the analyses performed against the sum of the loads applied.

The status of each loadcase or combination is indicated as follows:

- ✓ - **Total Reaction** in equilibrium with the **Total Load on Structure**
- ✗ - **Total Reaction** not in equilibrium with the **Total Load on Structure**
- ? - **Total Reaction** not available

Working with the Wind Model Tree

Once the wind modeling process has been completed, the Wind Model Tree can be used to display the resulting Wind Model views. It comprises the following branches:

Pressure Zones

Solely indicates if the pressure zone calculations have been performed.

Wind Directions

Each Wind Direction View can be displayed from here. Wind direction dependent properties can be accessed through the relevant Wind Direction View and then edited by selecting the relevant roof- or wall-elements.

Wind Loadcases

Wind Loadcases can be created and edited from here

How do I use the Wind Model Tree to display a Wind Direction View?

Wind direction dependent properties can be accessed through the relevant **Wind Direction View** and then edited by selecting the relevant roof- or wall-elements.

1. Right-click the direction required in the **Wind Directions** branch of the tree.
2. Choose **Open View** from the right-click menu.

Working with the Status Tree

The Status Tree is used to review:

- Validation status for the model.

  See: Model Validation.
- Validation status for the analysis.

  The analysis model validation is performed automatically as part of the analysis process.
- Decomposition status.

  This branch indicates if load decomposition has been successfully completed.
- Solver status.

  This branch lists those analyses which have been performed. A tick indicates the analysis was successful so that results are available to view.
- BIM validation

  If the model has been imported/exported (e.g. to Robot, or Revit) then any warnings or errors that relate to the import/export are listed on this branch.

The Report Index

When a report view is active the **Report Index** can be used to locate and display a specific section in a multi-page report.

Working with Scene Views, View Regimes and Scene Content

display views of the model (or a part of it) in tabbed windows. Different scene views can be created to show 3D views, 2D plans, frames, planes or individual members.

For each scene view you can choose a **View Regime** appropriate to the task being performed. Different regimes are available including a regime for creating the structure, and separate regimes for viewing the analysis model, the wind model, the analysis results, and the design.

There is also a separate **Load Analysis View** for displaying the force and moment diagrams for individual members.

Opening and Closing and Saving Scene Views

Multiple scene views can be displayed simultaneously in tabbed windows docked within the main window. If required, specific views can be permanently saved as View Configurations.

How do I open a 3D view of my entire structure?

1. Open the Structure Tree
2. Double click Structure to open a
   (or right-click to open a)

How do I open a 3D view of an existing sub model?

1. Open the Structure Tree
2. Click the plus sign (⊞) to the left of Sub Models to show all the currently defined sub models, then:
3. Double click an individual sub model to open a
   (or right-click to open a)

How do I open a 3D view of a single member?

To open a single member view from within another view:

1. Hover the cursor over the member until it becomes highlighted, then,
2. Right-click and then click **Open (member ref) view** from the context menu.

To open a single member view from the Project Workspace:

1. Open the Structure Tree

2. Click the plus sign (⊞) to the left of Members

3. Expand the appropriate sub-branches to display the member references:

4. Right-click the required member reference and choose **Open view**

How do I open a 2D view of an existing construction level?

1. Open the Structure Tree

2. Click the plus sign (⊞) to the left of Levels to show all the construction levels in your model, then:

3. Double click an individual level to open a
(or right-click to open a)

Before you can view a 2D view of a construction level, you must have created that level in your model.

How do I open a 2D view of an existing frame?

1. Open the Structure Tree

2. Click the plus sign (⊞) to the left of Frames to show all the frames in your model, then:

3. Double click an individual frame to open a
(or right-click to open a)

Before you can view a 2D view of a frame, you must have created that frame in your model.

How do I open a 2D view of an existing sloped plane?

1. Open the Structure Tree

2. Click the plus sign (⊞) to the left of Slopes to show all the slopes in your model, then:

3. Double click an individual slope to open a
(or right-click to open a)

Before you can view a 2D view of a sloped plane, you must have created that plane in your model.

View Configurations

Once a Scene View has been orientated to show a specific area of the model, it can be saved if required to a View Configuration.

View Configurations can then be used in two ways:

1. They can be included as Views in **Model Reports** (in which case they retain the Scene Content settings that were in place when the View Configuration was saved).
2. They can be re-opened in a new Scene View at a subsequent time (in which case they adopt whatever Scene Content settings are currently in place).

To save a View Configuration:

1. right-click in the View and choose **Save View Configuration...** from the context menu.
2. Enter a descriptive name, then click OK.

To re-open a saved a View Configuration:

1. Click **Home** > **Manage View Configurations**
2. Select the view configuration required.
3. Click **Open**

To delete a a View Configuration:

1. Click **Home** > **Manage View Configurations**
2. Select the view configuration required.
3. Click **Delete**

How do I close a view?

1. Click the icon at the top-right of the view tab.

Zooming/Panning/Rotating and Walking through Scene Views

The mouse is used to manually, zoom, pan, or rotate the model to any orientation you require. Additionally, the model can be rotated to a range of preset views using the **ViewCube**.

A walk through mode is available for 3D views.

How do I zoom in/zoom out/zoom extents?

To manually zoom in and out:

1. The mouse wheel is used to zoom in and out.

To zoom extents:

1. Simply right-click anywhere within the view and select **Zoom Out** from the menu.

The keyboard shortcut "ZA" can also be used to zoom extents.

How do I pan the view?

1. Simply hold down the middle mouse button and drag.
2. Once you have achieved the pan you require release the mouse button.

How do I manually rotate the view

If none of the standard **Viewcube** views are appropriate, you can rotate the model to get to just the view you require.

1. Simply right-click and hold over the 3D view, and move the mouse to perform the rotation.
2. Once you have achieved the view you require simply release the mouse button.

How do I use the ViewCube to display one of the preset views?

The **ViewCube** provides quick access to a range of preset views in any 3D view.

1. Move the cursor over the **ViewCube** to make it active.
2. Simply use one of the methods below to choose the view that you want to see.

a) To display one of the eight isometric views click the required vertex on the **ViewCube**. If the required vertex is not visible spin the **ViewCube** by clicking on one of the other vertices until the required one appears.

b) To display one of the twelve edge views click the required edge on the **ViewCube**. If the required edge is not visible spin the **ViewCube** by clicking on a vertex adjacent to the required edge.

c) To display one of the six face views click the required face on the **ViewCube**. If the required face is not visible spin the **ViewCube** by clicking on a vertex adjacent to the required face.

With a face view displayed additional **ViewCube** controls become active.
d) If the required face is not visible you can roll the **ViewCube** on to an adjacent face by clicking on one of triangular controls.

e) If the required face view is displayed, but the orientation of the face is not correct you can rotate it clockwise, or anti-clockwise by clicking on one of the two arrow controls.

Pressing F8 toggles the display of the **ViewCube** *on and off.*

How do I walk through the model in a 3D view?

When you are working in a 3D view it is sometimes useful to be able to walk through the model.

1. Click **Home** > **Walk** ( )

   This puts you into **walk mode**:

   Use the arrow keys to move back/forward/left /right.

   Use Q/Z to move up/down.

   To rotate click and drag the right mouse button.

   Press [Esc] to exit walk mode.

How do I display a 2D view in 3D?

When you are working in a 2D view it is often useful to display it in 3D, as this gives you a 3D view whilst hiding the rest of the structure.

You can switch between the 2D and 3D by clicking the 3D/2D toggle button.

The 3D/2D toggle button is located at the bottom right hand corner of the 2D view and also at the right hand end of the **Status Bar** at the bottom right of the screen*.*

1. If the 2D view is currently displayed in plan, the 3D/2D toggle button will be labeled **3D**.
2. Click the 3D/2D toggle button.
3. The 2D view is now displayed in 3D (and the 3D/2D toggle button changes to **2D**.
4. To change back to a plan view click the 3D/2D toggle button once more.

Controlling Scene Content

Different entities types have different levels of information associated with them. You can choose how much of this information is displayed for each combination of scene view and view regime.

For example, in a solver view it is generally sufficient to represent beams by their insertion lines, however in a physical model view you are likely to also include their geometric

outlines. In either of the views you may also choose to display their direction arrows and possibly their reference texts also.

The information displayed in each scene view is controlled by making appropriate selections in the Scene Content window.

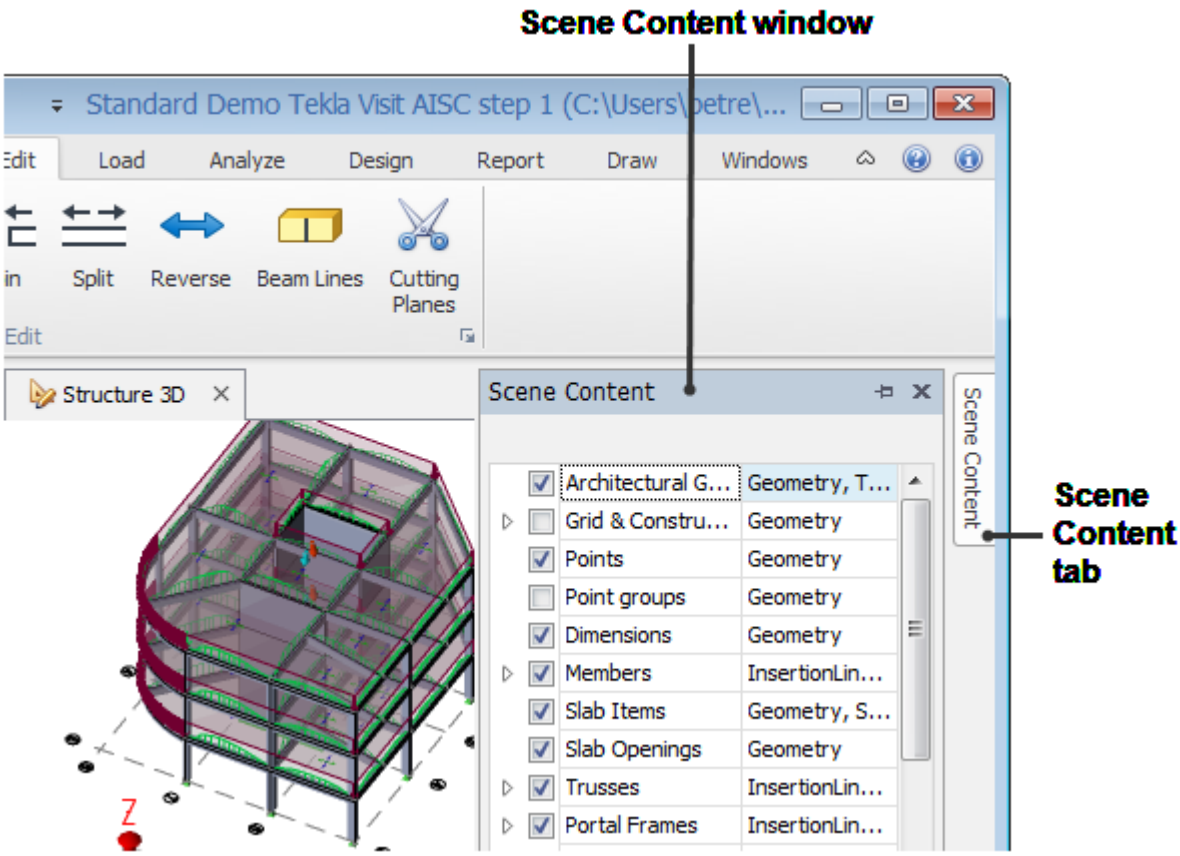
Scene content selections are saved independently with each scene view.

How do I display the Scene Content window?

If the **Scene Content** window is not currently visible, this is either because it is set to auto-hide, or because it has been closed.

Displaying the Scene Content window when it is set to auto-hide:

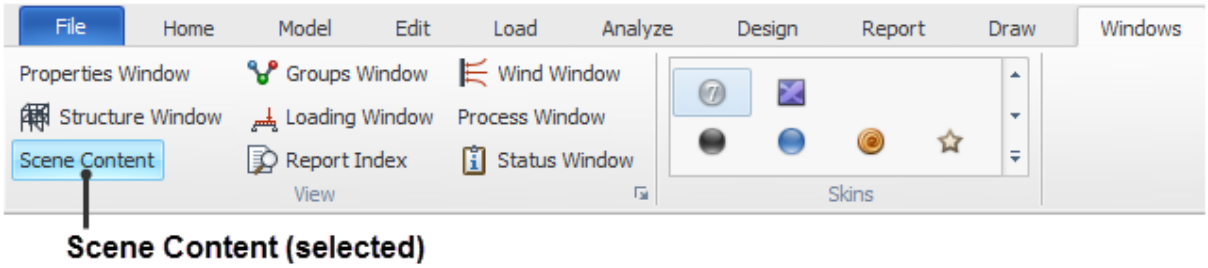
- If Scene Content is set to auto-hide there will be a **Scene Content** tab docked on one edge of the interface - click the tab to expand the window.



- When you have finished making your selections, click anywhere in the scene view to hide it once more.

Re-displaying the Scene Content window after it has been closed:

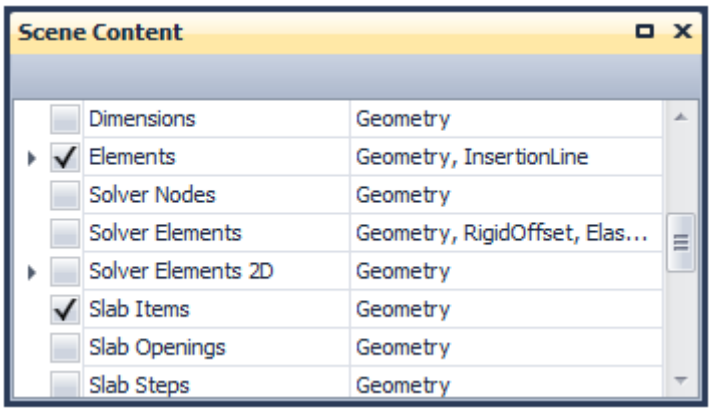
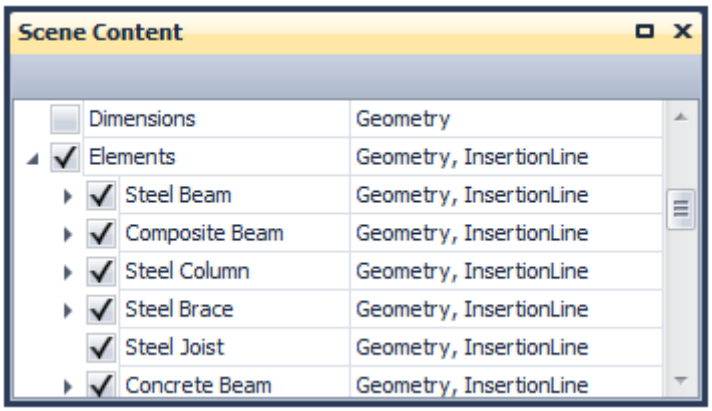
- If Scene Content has been closed, it can be reopened by re-selecting **Scene Content** on the **View** group on the **Windows** toolbar.

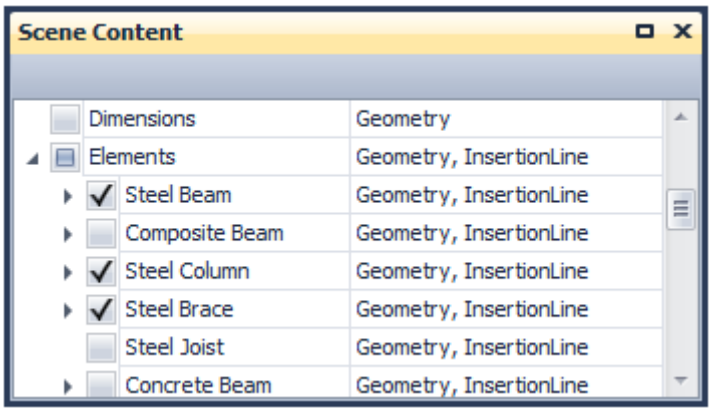
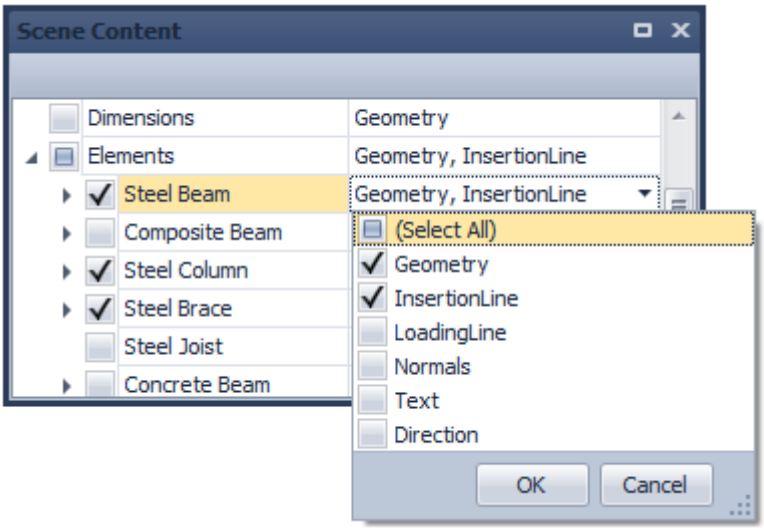


How do I make selections in the Scene Content Window?

The **Scene Content** window displays two columns of information:

- In the left column is a list of the different **entity categories**. Some entities have arrow symbols to their left, indicating sub categories - click the arrow symbol in order to see these.
  - A check box controls whether an entity and its associated information is displayed - Simply select the entities you want to see and clear those that you do not.
- In the right column the **entity information control** lists the information in each category that will be displayed. When clicked this expands to a drop list menu, allowing you to select the information required.

Example	View
Certain check boxes have sub categories (indicated by the arrow sign). The selected check box indicates that all the sub categories are currently displayed.	
Clicking the arrow expands the category....	

To hide a particular sub category clear it... When some, but not all sub categories are cleared, the main category is shown as a shaded box.	
The information displayed for each entity type can be controlled by expanding the entity information control...	

If you switch off the display of a particular entity type, then some commands that affect that entity type can no longer be performed. For instance if you switch off the display of Slab Items, then you cannot define slab or area loads on a floor, since there are no slab panels for you to select.

If planar objects (slabs, roofs, area loads etc.) lie over the start point for performing a selection in the current scene view, you can use Scene Content to switch off their display, (assuming you don't want to have them considered in your selection.

How do I reinstate the default Scene Content selections?

You do not have to worry about changing the initial **Scene Content** configuration as you can reinstate the default selections at any time.

To discard your current selections and reinstate the defaults:

1. Close the View.

2. Re-open the View from the Structure Tree.

Scene Content Entity Categories

When the Scene Content window is accessed it indicates which entities are selected for display in the current scene view.

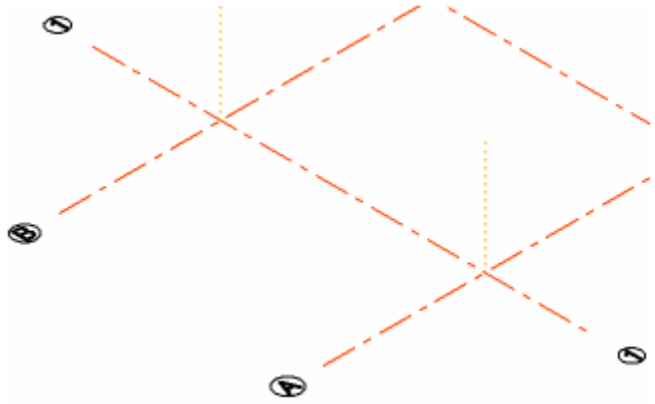
Although the entity categories listed in the content vary depending on the view type (3D/2D) and the view regime (Structural/Solver/Results/Review etc.), most categories are common to multiple views.

The entity categories are generally self-explanatory, but some clarification may be useful.

Architectural Grids

This category is not displayed for 2D views.

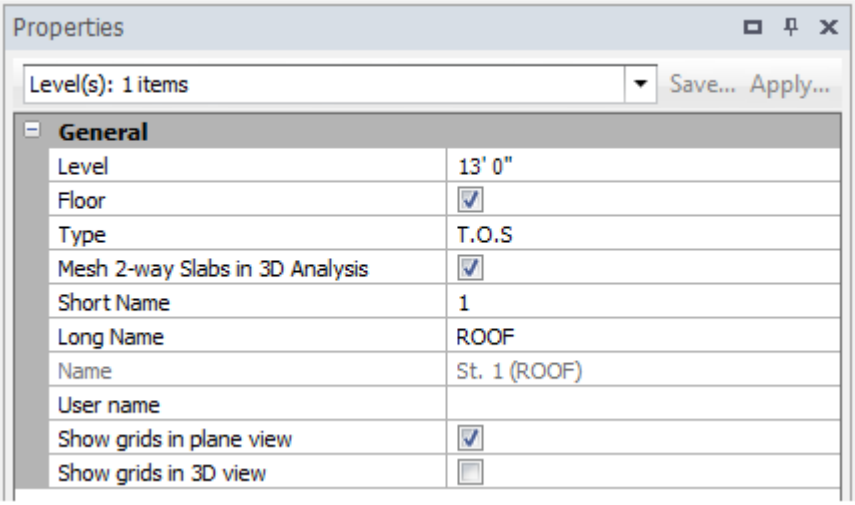
This category is used to show/hide architectural grids in 3D views. When selected they are displayed at the lowest level in the model. (



Grid & Construction Lines can be displayed at any level.)

Grid & Construction Lines

This category is used to show/hide grid and construction lines at individual levels in both 2D and 3D windows.



The levels at which grids are displayed/not displayed are controlled via the Properties Window for each individual level. Grids are only displayed in 2D windows if **Show grids in plane view** *is selected and only displayed in 3D windows if* **Show grids in 3D view** *is selected.*

Points

These are the construction points (CP) that are formed at connections between members.

Point Groups

These are for advanced use only - in normal use they should be switched off.

Dimensions

Shows any dimensions that have been applied.

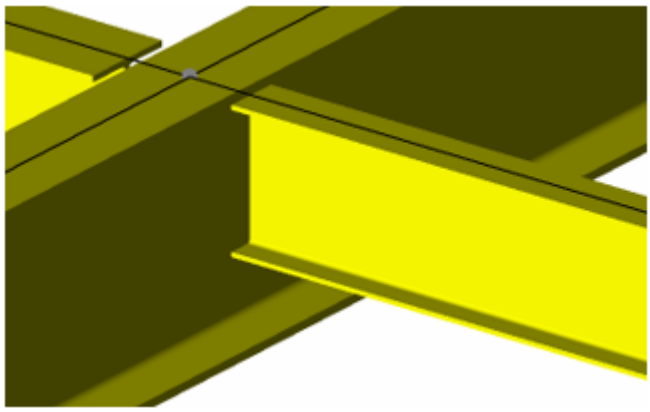
Members, Trusses and Portal Frames

The Members category has a sub category for each of the member types.

Members, Trusses and Portal Frames can each have the following entity information displayed.

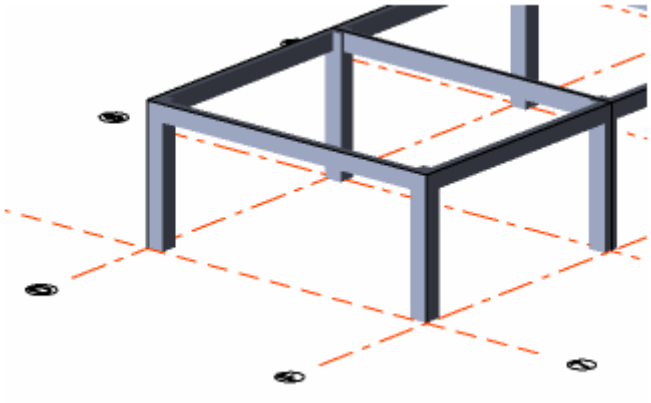
Geometry

This shows the faces of the member by shading them.



Different colors are used for the different member types.

The view below shows a concrete beam and column example with **Geometry** and **Architectural grids** selected.

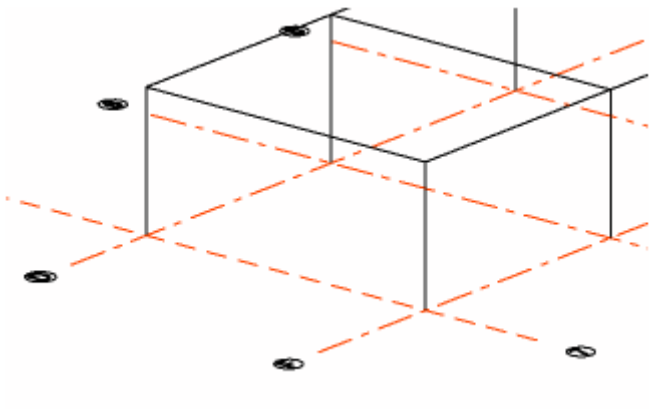


The colors used are customizable from Settings > Scene on the Home toolbar.

InsertionLine

This shows a solid line between the start and end node of the member.

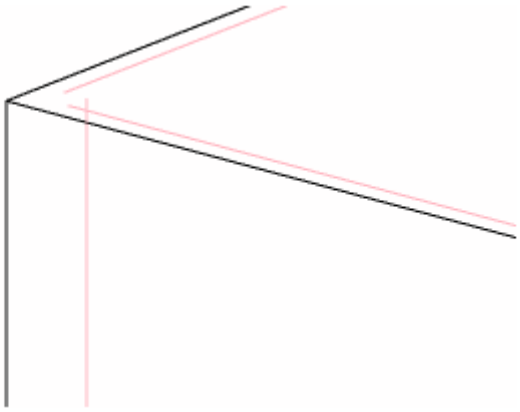
The view below shows the same concrete beam and column example with **InsertionLine** and **Architectural grids** selected.



LoadingLine

This shows a solid line through the center of the member. Any member loads are applied along this line, (which may be at an offset to the InsertionLine).

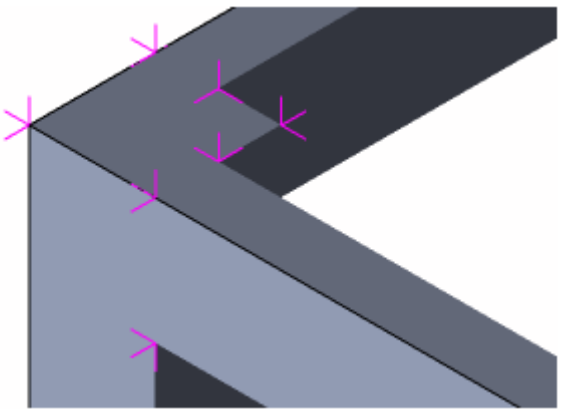
The zoomed in view below shows the same concrete beam and column example with **InsertionLine** and **LoadingLine** selected.



Normals

These are unlikely to be required in general usage, but can be selected in order to see the normal directions at each corner of the member.

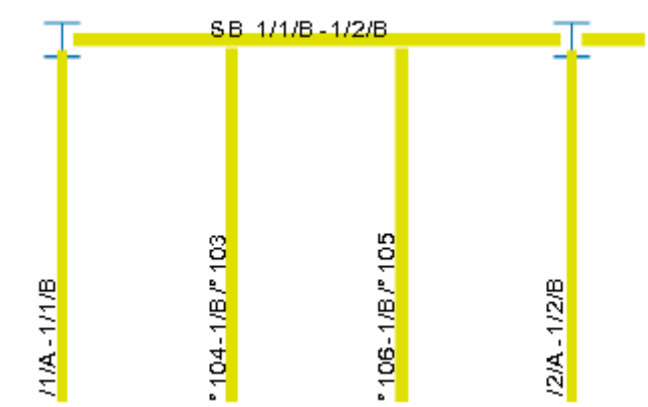
The view below shows the same concrete beam and column example with **Geometry** and **Normals** selected.



Text

This shows the member name.

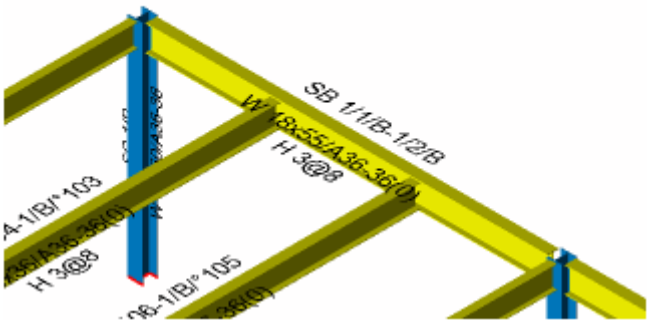
The view below shows a composite steel beam example with **Geometry** and **Text** selected.



Text2D

This shows the member name, section and class in the 2D plane of the member.

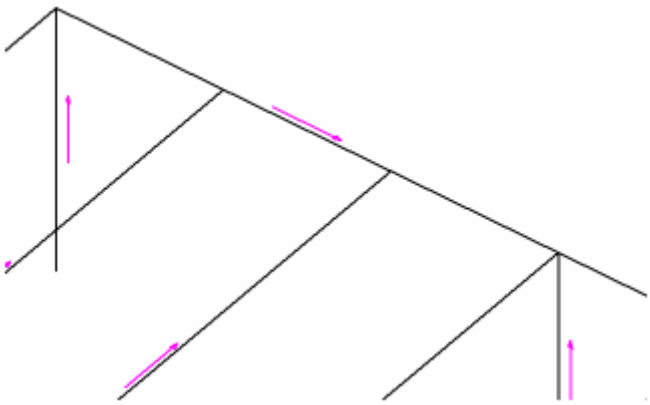
The view below shows the same composite steel beam example with **Geometry** and **Text2D** selected.



Direction

This shows the direction from end 1 to end 2 of the member. (If the direction is incorrect it can be changed using the Reverse command on the Edit menu.)

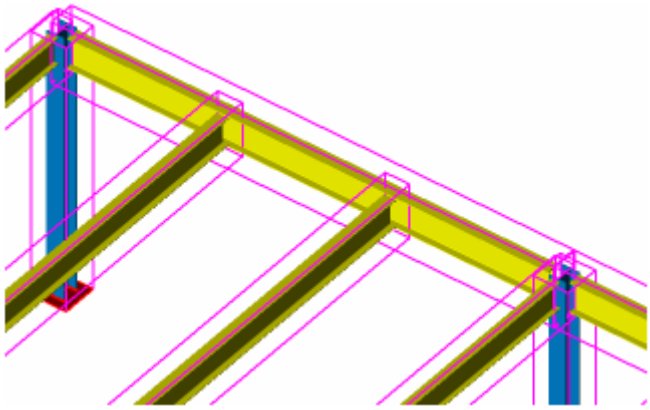
The view below shows the same composite steel beam example with **InsertionLine** and **Direction** selected.



BoundingBox

This is unlikely to be required in general usage, but can potentially be selected in order to make it easier to right-click sections of small cross section.

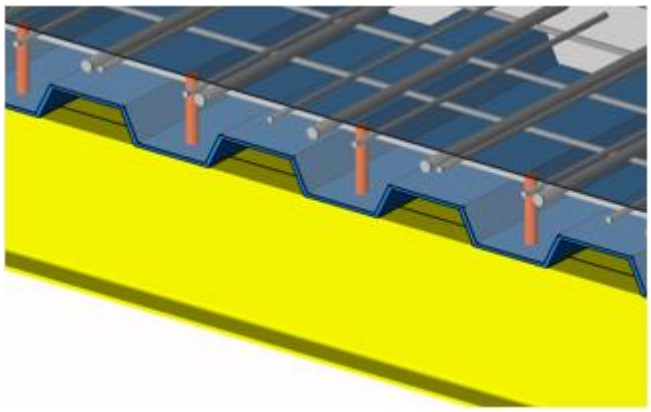
The view below shows the same composite steel beam example with **Geometry** and **BoundingBox** selected.



Decking

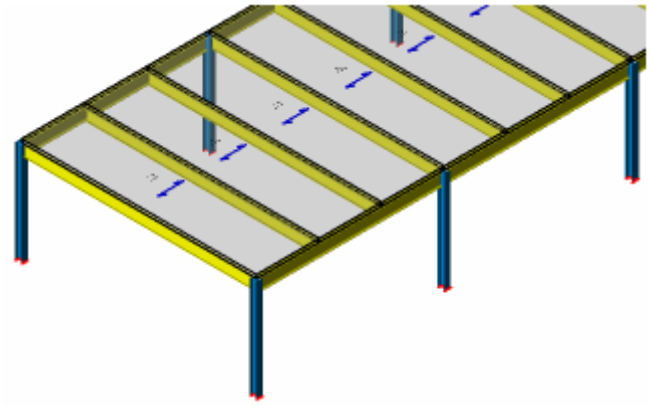
This shows the strip of decking that is connected to the member.

The zoomed in view below shows the same composite steel beam example with **Geometry** and **BoundingBox** selected.



Slab Items

This category is used to show/hide slab panels.

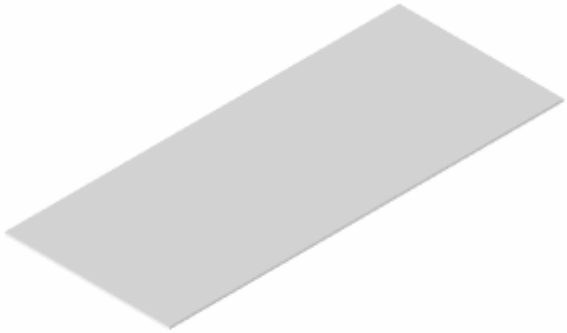


Slab Items can have the following entity information displayed.

Geometry

This shades in the top, bottom and side faces of the slab panel.

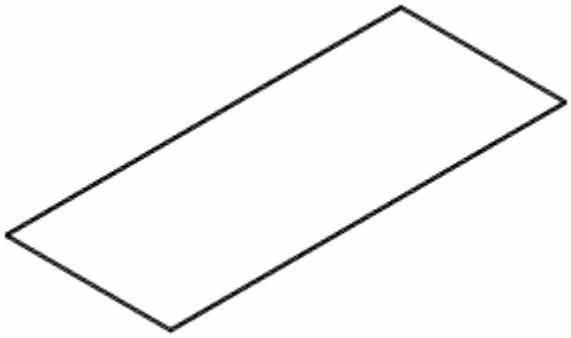
The view below shows the above composite steel beam example with only **Geometry** selected.



SlabOutline

This shows the overall outline of the parent slab.

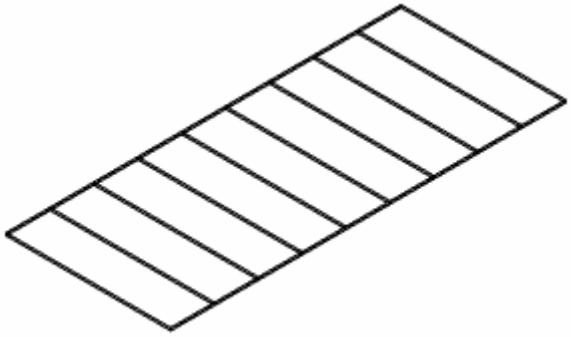
The view below shows the same example with only **SlabOutline** selected.



SlabItemOutline

This shows the overall outline of each individual slab item.

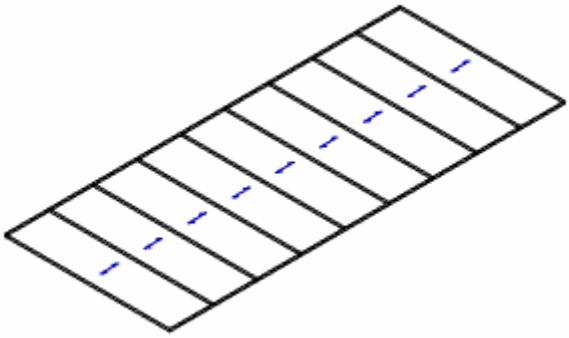
The view below shows the same example with only **SlabItemOutline** selected.



SpanDirection

This shows the span direction symbol.

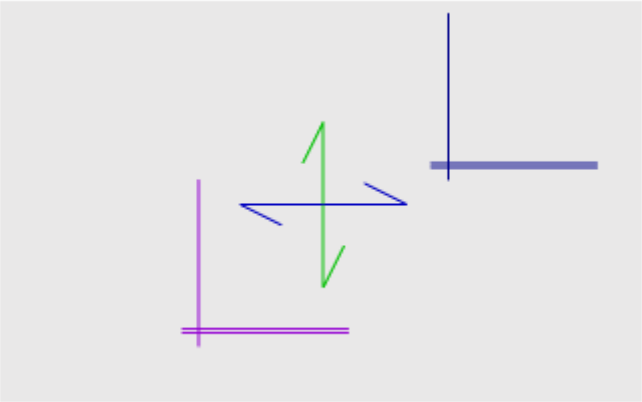
The view below shows the same example with **SlabItemOutline** and **SpanDirection** selected.



TopReinforcement and BtmReinforcement

These show the top and bottom reinforcement symbols.

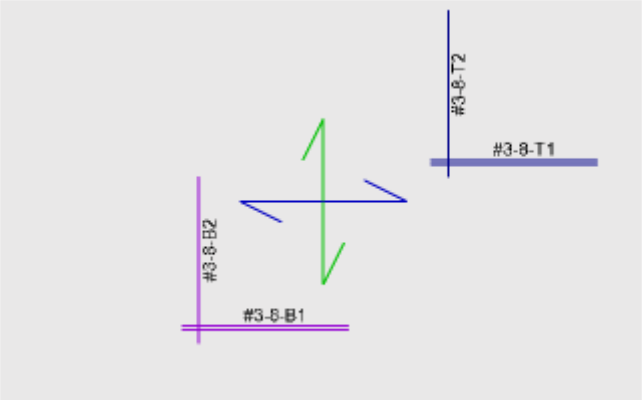
The view below shows a concrete slab example with **TopReinforcement** and **BtmReinforcement** (and **SpanDirection**) selected.



ReinforcementText2D

This labels the reinforcement.

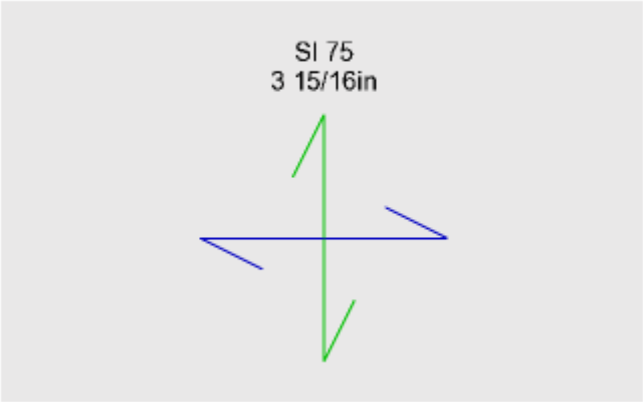
The view below shows a concrete slab example with **TopReinforcement**, **BtmReinforcement**, **ReinforcementText2D** and **SpanDirection** selected.



Text2D

This shows the slab item name and thickness in the 2D plane of the slab.

The view below shows a concrete slab example with **Text2D** and **SpanDirection** selected.



Slab Openings

Geometry

Crossed lines are used to represent the location of each opening.

Concrete Walls

This entity has three sub categories

Concrete Walls

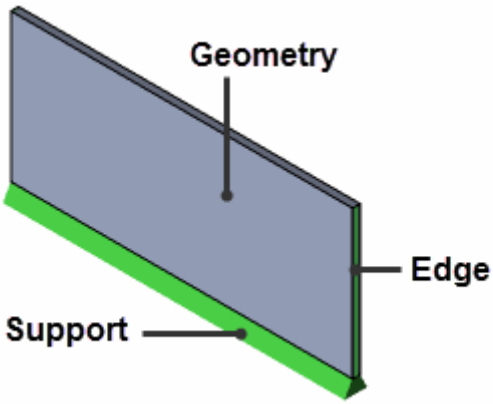
This sub category can have the following entity information displayed.

Geometry - This shades in the top, bottom and side faces of the wall panel.

Mesh - This option is not used! To see the mesh you must first clear **Geometry** and then select **Concrete Walls** inside the 2D Elements entity category instead.

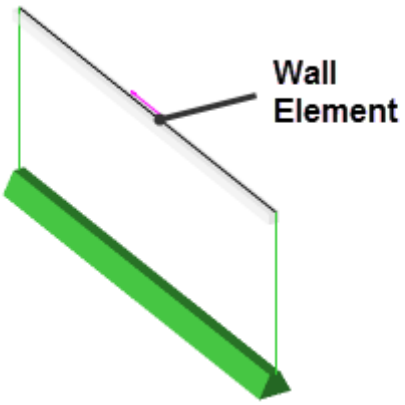
Edges - This shows the left and right edges of the wall.

Support - This shows the wall support.



Wall Elements

This sub category is used to show/hide the 1D elements that exist within concrete walls. When modelled using FE shells a single wall element exists along the top of the wall (as shown below). When modelled as mid-pier additional wall elements are also created along the base of the wall and vertically through the middle of the wall). These wall elements have the same entity information available as Members..

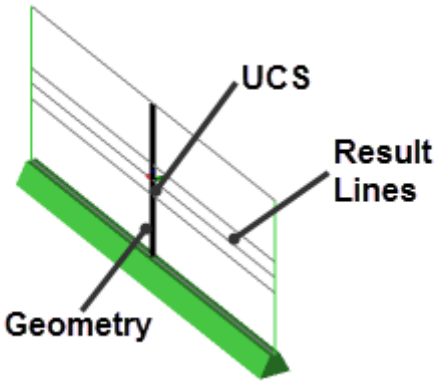


Wall Lines

This sub category only relates to walls modelled using FE shells.

Wall lines are not displayed for the working solver model because they are only formed at the point of analysis. Therefore if you have switched them on in Scene Content but they are still not being shown, trying changing the Solver Model that is being displayed, (from the right-click menu when in the Solver View).

It shows/hides the result lines (used to determine the total internal forces in a wall mesh), and also the wall line itself (used to display the forces from the result lines in an intuitive way).



Wall lines have the following entity information.

Geometry - Shows/hides the wall line.

Result Lines - Shows/hides the result lines.

UCS - Shows/hides the user co-ordinate system that applies to the wall line.

Bearing Walls

This entity has two sub-categories

Bearing Walls

This sub category can have the following entity information displayed.

Geometry - This shades in the top, bottom and side faces of the wall panel.

Support - This shows the wall support.

Wall Elements

This sub category is used to show the analysis elements that form the bearing wall. It has the same entity information available as Members.

Wind Panels, Roof Panels

Wind and Roof Panels can have the following entity information displayed.

Geometry

This shades the inner and outer faces of each panel in different colors.

SpanDirection

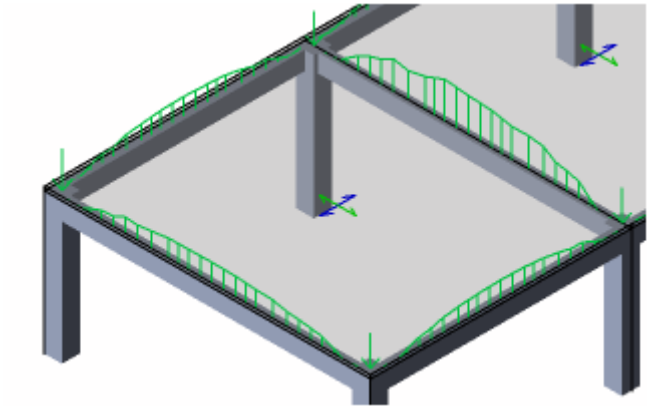
This shows the span direction symbol.

Local Axes

This shows the positive directions of the local axes.

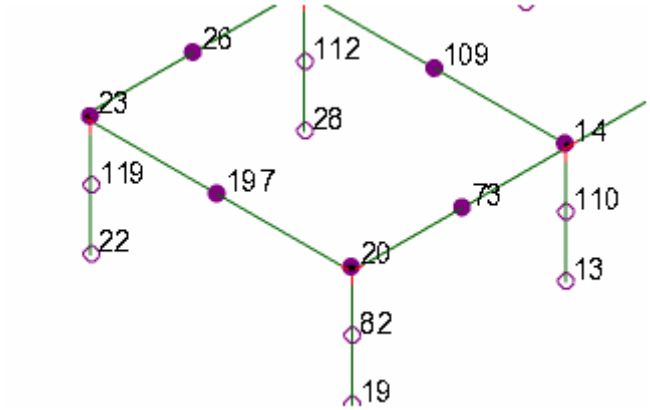
Loading

This has sub categories for different applied loads and also has a Decomposed sub-category which is useful for visually checking how slab loads have been decomposed onto members post analysis.



Solver Nodes

These nodes define the ends and middle of each 1D element and the vertices of each 2D element.



1D Elements

These are 1D elements in the analysis model. They have the following entity information.

Geometry - A single line is used.

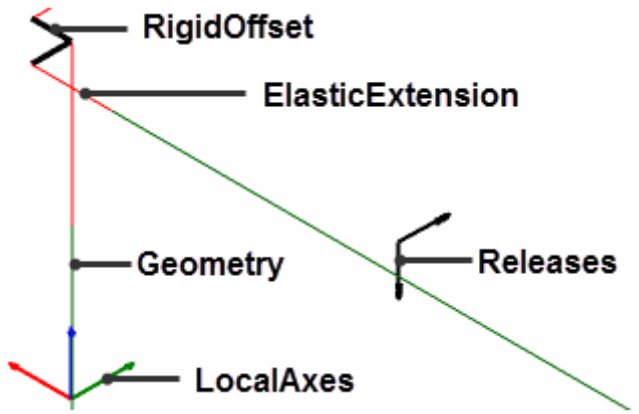
Text - This shows the element number.

Rigid Offset - This is shown as a thicker line.

ElasticExtension - This is the portion of 1D Element that exists inside the boundary of a connected member.

LocalAxes - The local axis system that applies to the 1D Element can optionally be displayed.

Releases - The end releases can also be displayed.



2D Elements

These are 2D finite elements that may have been used in the analysis.

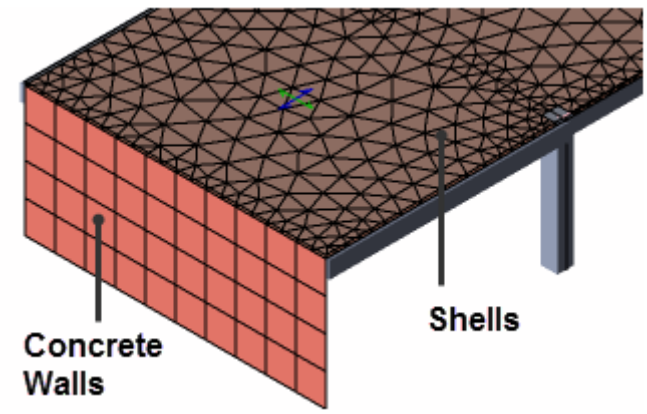
2D elements are not displayed for the working solver model, (even when you have chosen to mesh slabs/walls). This is because 2D elements are only formed at the point of analysis. Therefore if you have switched them on in Scene Content but they are still not being shown, trying changing the Solver Model that is being displayed, (from the right-click menu when in the Solver View).

There are three sub-categories

Shells - This sub-category specifically refers to the 2D elements used in slab meshes, (in the FE chasedown model, and also in the 3D Analysis model when slabs have been meshed).

Semi-rigids - This sub-category specifically refers to the 2D elements used in to model one way spanning slabs when they have been modelled as semi-rigid diaphragms

Concrete Walls - This sub-category specifically refers to the 2D elements used in concrete walls when the option to use mid-pier has not been selected.



Diaphragms

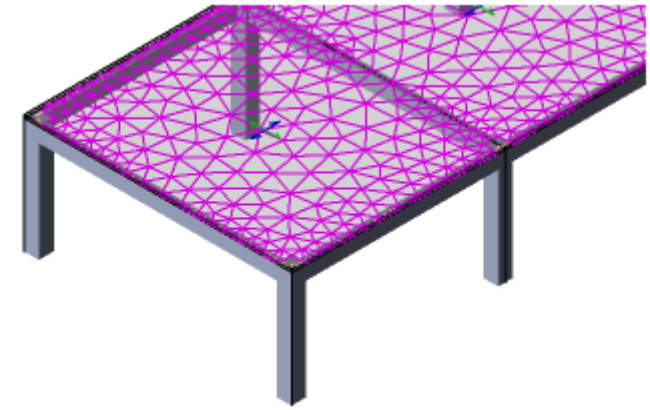
This category is only available when in a Solver View.

This category shows/hides the diaphragm as a shaded plane.

For flexible diaphragms the mesh can also be shown.

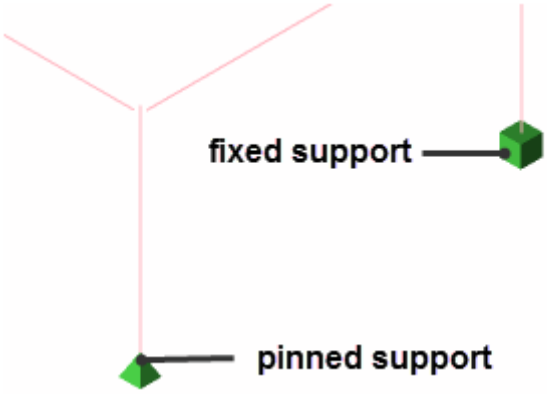
Slabs

This category shows/hides the slab mesh used for load decomposition.



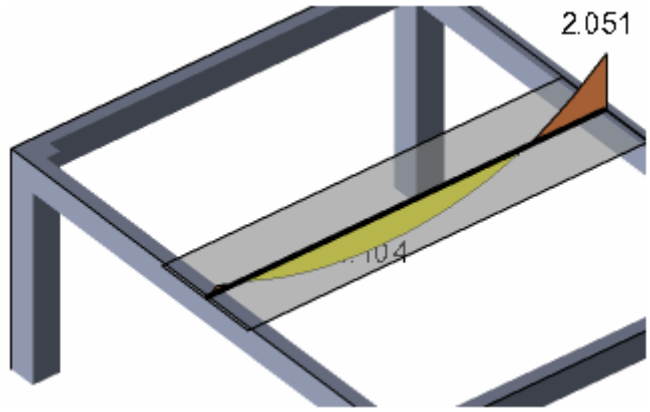
Supports

This category shows/hides the supports under columns and manually defined supports, but not the line supports under concrete walls, (which are controlled by the Concrete Walls entity category).



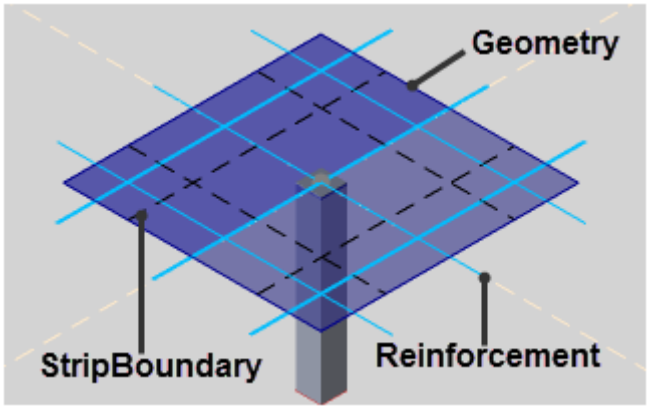
Result Strips

User defined Result Strips can be placed across 2D element meshes. From these strips, force and moment results are determined from the shell/plate/membrane nodal analysis results.



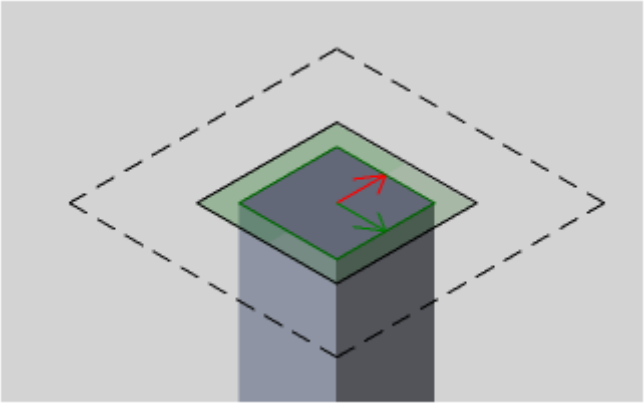
Slab Patches

In concrete models if you have created any slab patches, you use this entity to control their display.



Punching Checks

In concrete models if you have created any punching check items, you use this entity to control their display.



Floor Vibration Checks

Beams
Shades the critical beams in each check.

CriticalSlab
Shades the critical slab item in each check.

Levels, Frames, Slopes

Geometry
Shades the extents of each Level, Frame, or Slope entity.

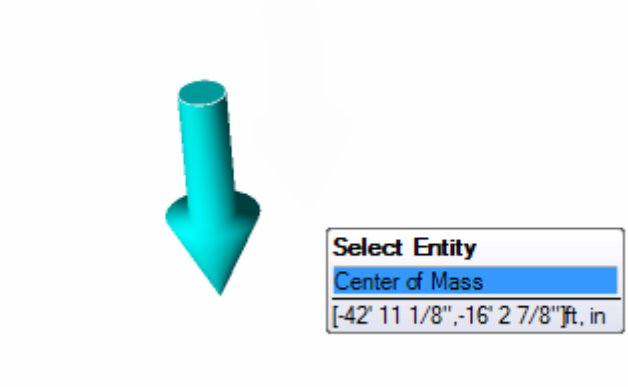
Pad Base, Pile Cap, Pile

Geometry
Shows the faces of each Pad Base, Pile Cap, or Pile by shaping them.

Center of Mass

For any given load case or combination, all gravity loads (self weight, slab dry, live, etc.) applied to a given floor have a center of action (or centre of mass), a point about which the loads would balance if a pinned support were positioned at this location in plan.

You can graphically review this for each floor and each loadcase/combination. By hovering the cursor over the center of mass a tooltip displays its coordinates.



Center of Rigidity

Any given floor has a center of rigidity or bending stiffness based on the stiffness of the structure that supports it (i.e. the columns, walls etc. below).

Its location can be rapidly approximated "on the fly" for each floor - you will see the position constantly changing as the model develops - and by hovering the cursor over it, approximate coordinates can also be displayed.

Since the center of rigidity never actually gets used in Tekla Structural Designer for any analysis or design it is not calculated accurately - the complex nature of the calculation would unnecessarily extend the overall model solution time.

Plan

This is a special category only available in 2D Views - see: Scene Content Plan Category

Scene Content Plan Category

The initial display for 2D Views is configured for modeling purposes and consequently by default does not show all the information that will be output when drawings are created. However, at any point you can select the **Plan** category in **Scene Content** in order to overlay one of the following planar drawing variants on the 2D View:

- General Arrangement
- Beam End Forces
- Foundation Layout
- Loading Plan
- Slab/Mat Layout

In each of these variants you can then selectively choose the individual drawing layers to be overlaid. This can be very useful for displaying information that would otherwise not be available whilst modelling.

Plan Sub Categories

The main sub categories beneath Plan are as follows:

General

This sub-category is used for displaying grids, construction lines and dimensions as they would appear on the general arrangement drawings.

Members

This sub-category is used for displaying the various different member types labelled as they would appear on the general arrangement drawings.

Walls

This sub-category is used for displaying concrete walls labelled as they would appear on the general arrangement drawings.

Slabs/Mats

This sub-category is used for displaying concrete slabs labelled as they would appear on the general arrangement drawings.

Reinforcement

This sub-category is used for displaying slab reinforcement as it would appear on the slab detailing drawings.

Other

This sub-category is used for displaying various other items that can be output to the general arrangement drawings.

In the first release it is not yet possible to overlay Beam End Forces or Foundation Reactions on a scene view. They can however be displayed on general arrangement drawings.

Tables

This sub-category is used for displaying tables of information that can be output to the general arrangement drawings.

Connections

This sub-category is used for displaying connection names, attributes and reactions.

To use the Plan Category

1. Ensure a 2D View is active.
2. In Scene Content, select **Plan**
3. Adjacent to the **Plan** check box use the drop list to choose the drawing variant.

Only those layers that exist in the General Arrangement, Foundation Layout, Loading Plan or Slab/Mat Layout drawing variants can be displayed.

The sub categories beneath **Plan** correspond to the drawing layers in the chosen drawing variant. (By default the layers that are initially active match those in the first **Layer Configuration** for the drawing variant in Drawing Settings).

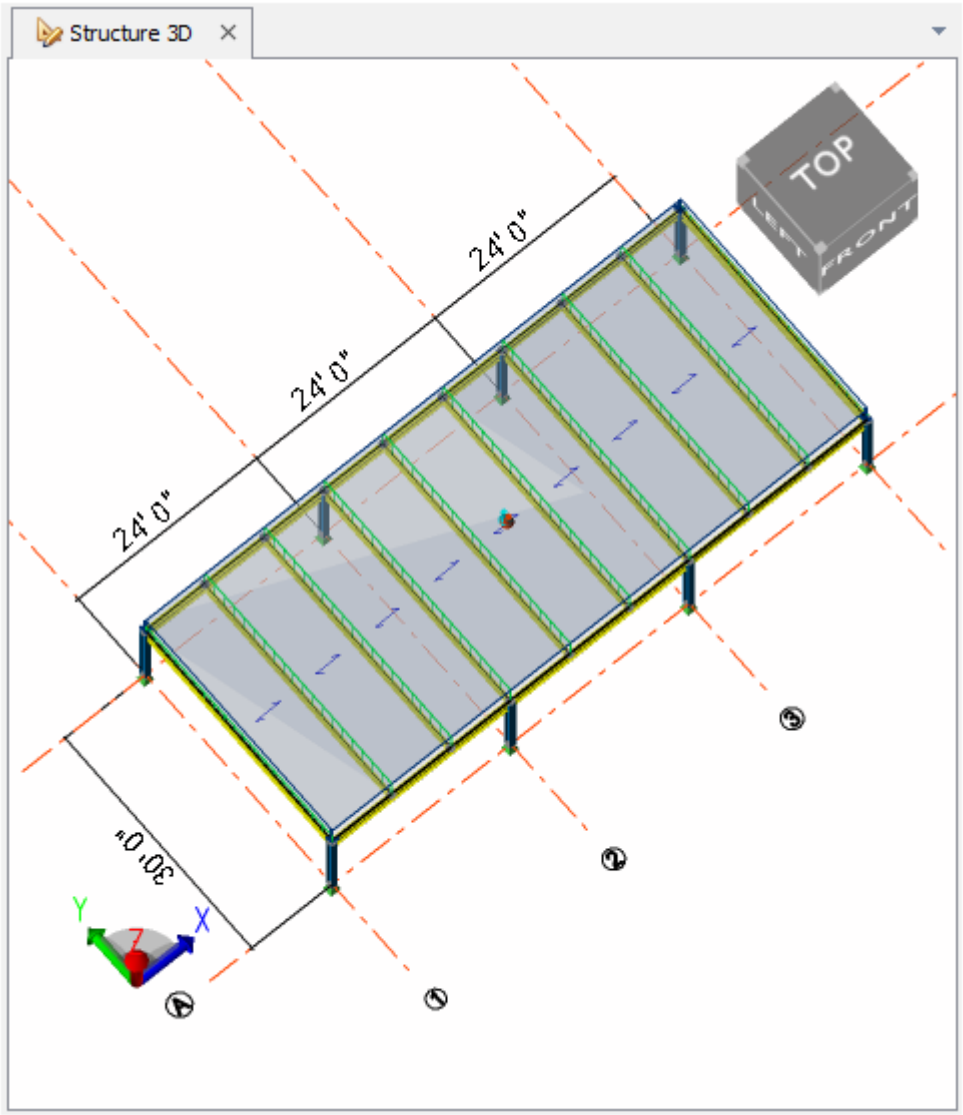
4. Adjust the layers to be displayed by selecting/clearing the **Plan** sub categories as required.

Layer content can be further controlled via Drawing Options in exactly the same way that it is controlled when creating the drawings themselves.

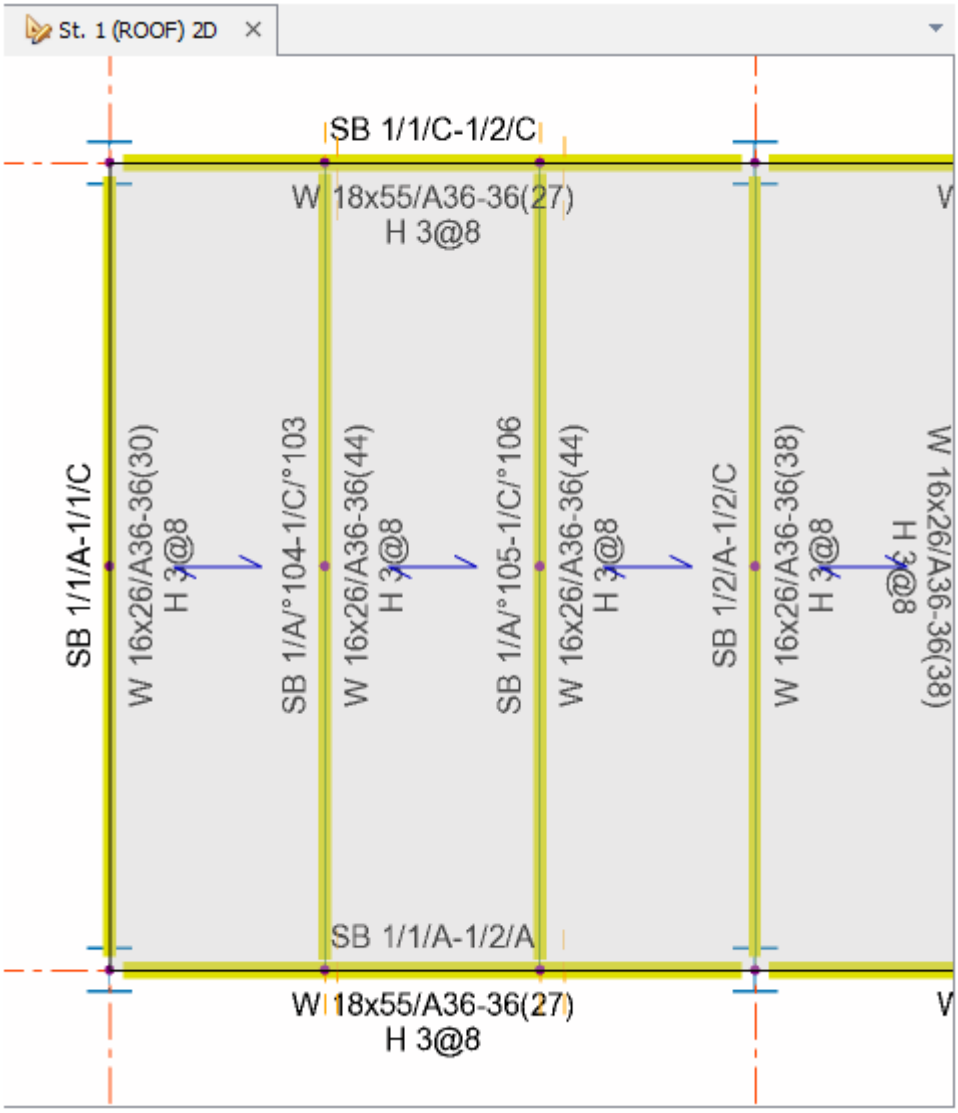
5. Layer colors, line types, fonts and font sizes match those in the first **Layer Style** for the drawing variant in Drawing Settings).

Example

To illustrate how the Plan category would typically be used, consider the following composite beam example designed to the AISC 360 ASD code:



In the below floor view the standard beam labelling for modelling is being applied. This consists of the beam name, section, grade, number of connectors and transverse reinforcement.



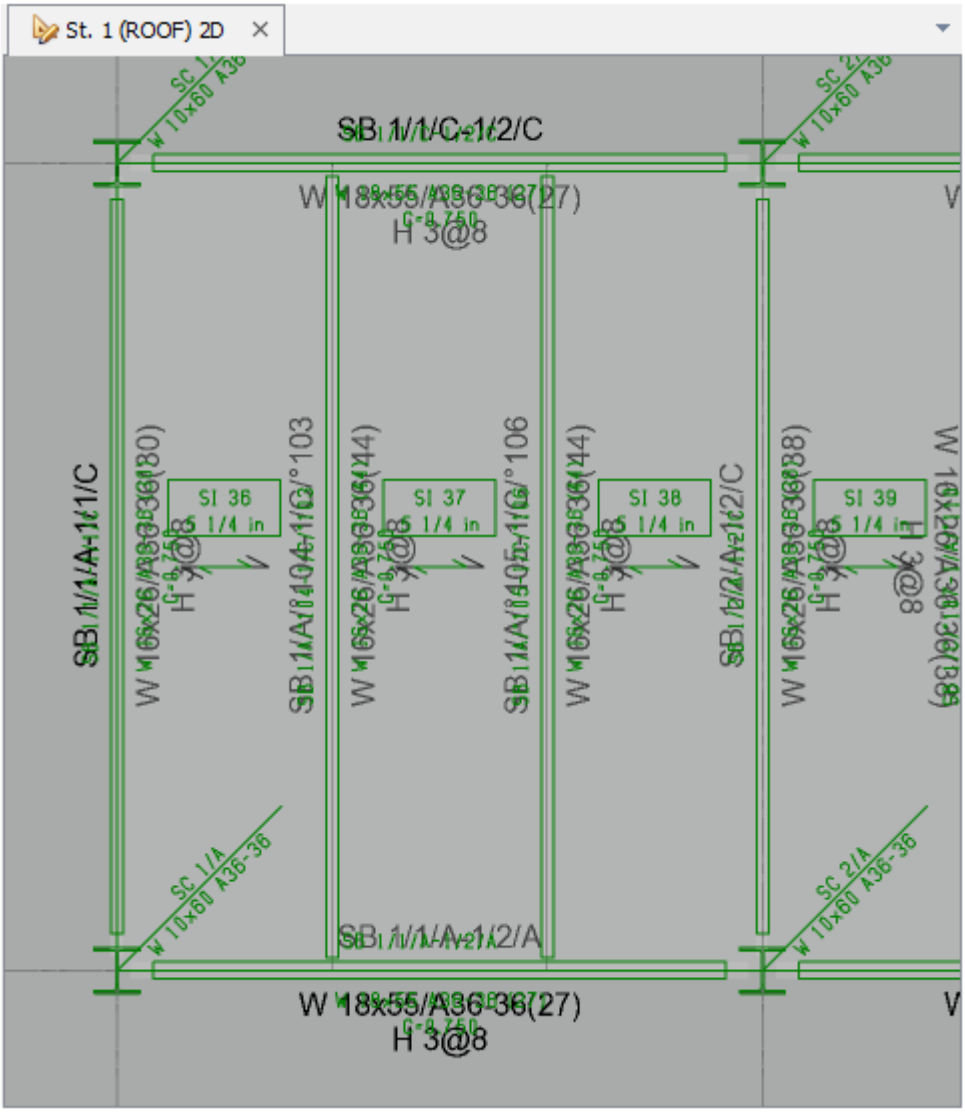
When the Tekla Structural Designer drawings are produced, it is likely that additional design information would also be conveyed, for example the amount beam camber required.

The camber is included in one of the drawing layers, so it should also be possible to include it in the 2D scene view - (provided we know the layer it belongs to).

The procedure to follow is therefore:

1. Make the 2D scene view showing the beams the active view.

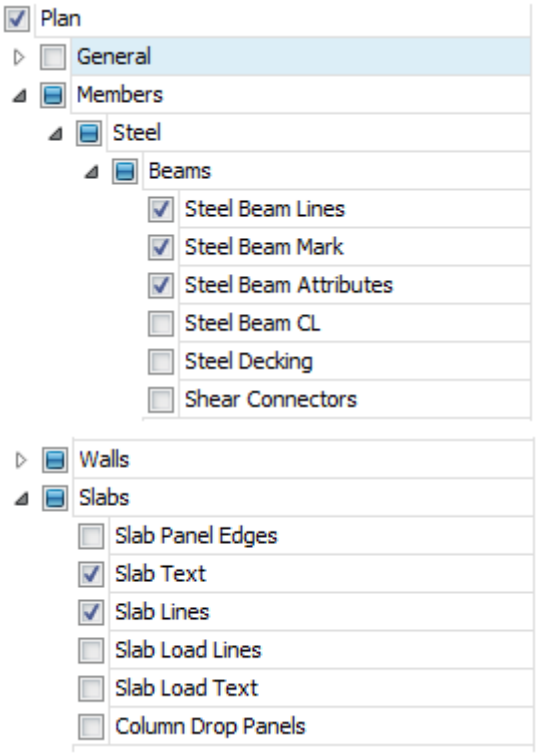
2. Open Scene Content and select the Plan category.



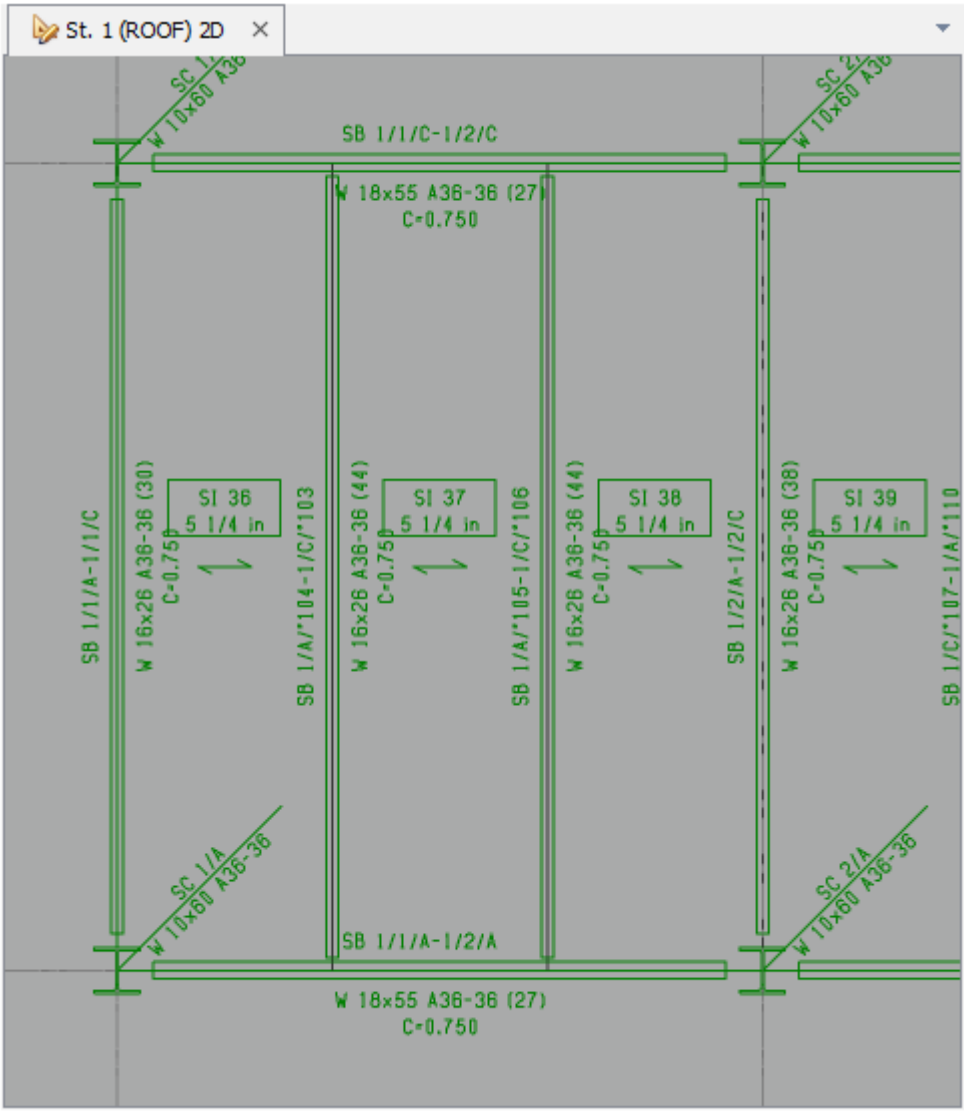
Depending on the current scene content selection there is potentially going to substantial duplication of axes, members and labelling etc.

3. In this example we will display all members in the view as they would appear on the drawing - this is achieved by:

- Clearing the following main Categories from the top of the scene content list: Members, Slab Items
- Selecting the following sub categories beneath Plan as shown:



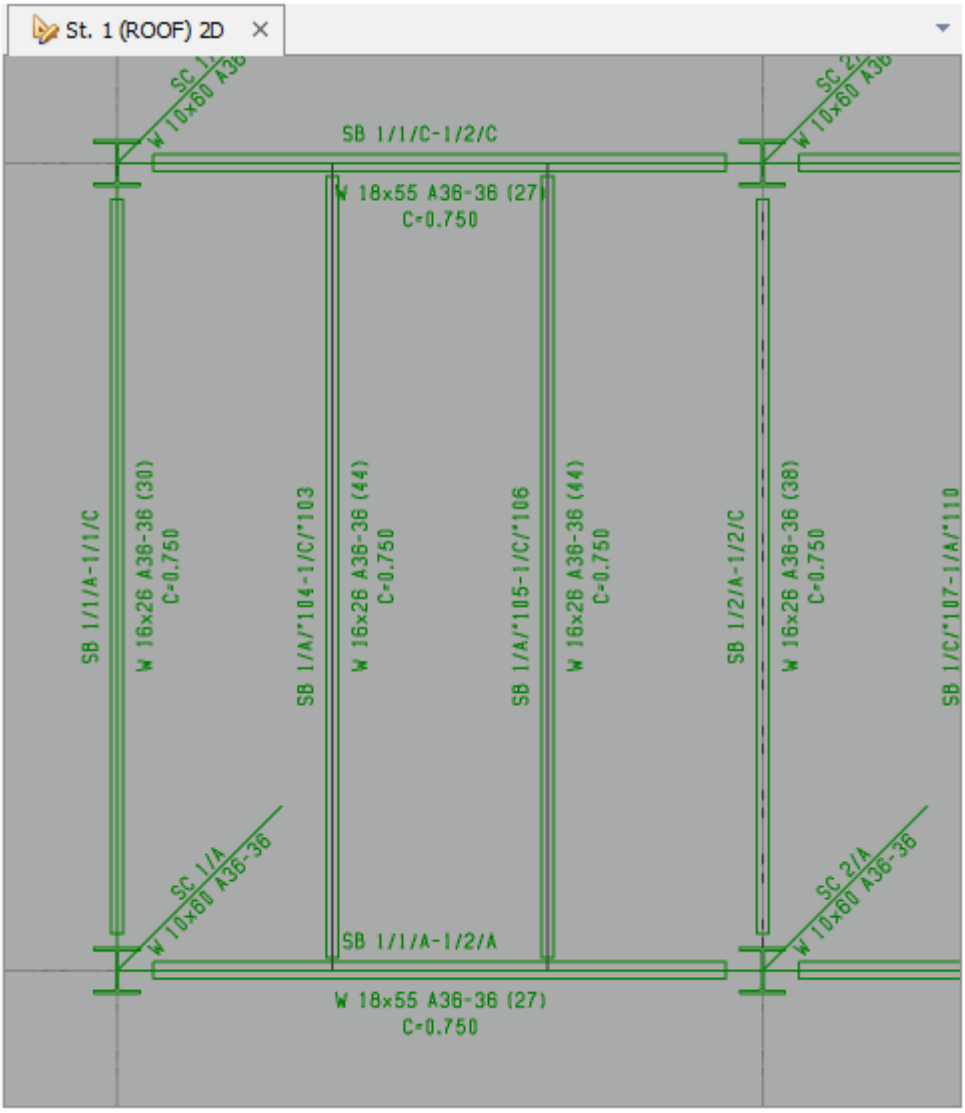
4. The scene view is now less cluttered as shown below:



5. In this example it will be further adjusted using the drawing options as follows:

- From the Draw toolbar, click **Edit...** and then:
- Click **Options > General Arrangement > Beams**
- Select **Append camber to section**
- Click **Options > General Arrangement > Slabs**
- Clear all the panel labelling and the span direction symbol
- Click **OK**

The drawing is re-displayed with the revised options applied.



View Regime

Different view regimes are available for:

- physical modeling of the structure geometry and loading, (**Structural View**)
- displaying the analysis model, (**Solver View**)
- displaying analysis results, (**Results View**)
- displaying the wind models, (**Wind View**)
- graphically interrogating the model properties/status, (**Review View**)

Structural View

Structural Views offer physical modeling of the structure with the aid of grids in either 3D or 2D.

Both geometry and loading are typically defined within this type of view.

A new scene view can be opened in this regime from the Structure Tree in the Project Workspace. You can change an existing scene view to this regime from the Status Bar, or by right-clicking on its view tab.

Solver View

2D and 3D **Solver Views** are mainly used for previewing and interrogating the analytical model, but they can also be used for modeling purposes.

A new scene view can be opened in this regime from the Structure Tree in the Project Workspace. You can change an existing scene view to this regime from the Status Bar, or by right-clicking on its view tab.

Results View

A **Results View** opens automatically at the end of an analysis - it is used for graphically displaying the various analysis results.

You can change an existing scene view to this regime from the Status Bar, or by right-clicking on its view tab.

Wind View

Wind Views becomes available after running the **Wind Wizard** - these are used for graphically displaying the wind zones and zone load details.

Once the wind wizard has been run, you can change an existing scene view to this regime from the Status Bar, or by right-clicking on its view tab.

Review View

A **Review View** opens automatically at the end of the Design All (Static) process - it is used for graphically interrogating the model properties/status.

You can change an existing scene view to this regime from the Status Bar, or by right-clicking on its view tab.

Changing the View Regime

An existing scene view can be switched to another mode, either from the status bar, or by right-clicking on its view tab.

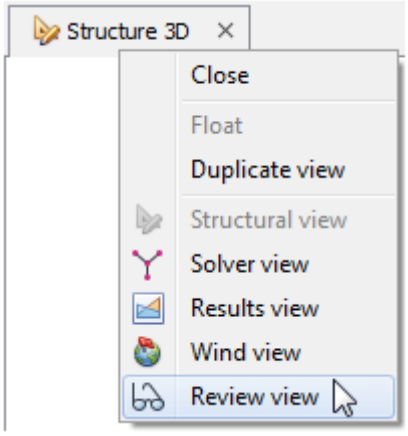
To change the view regime from the status bar

The different view regimes are shown on buttons in the Status Bar at the bottom right of the screen.

Click one of the unselected buttons to make the active scene view display in that regime.

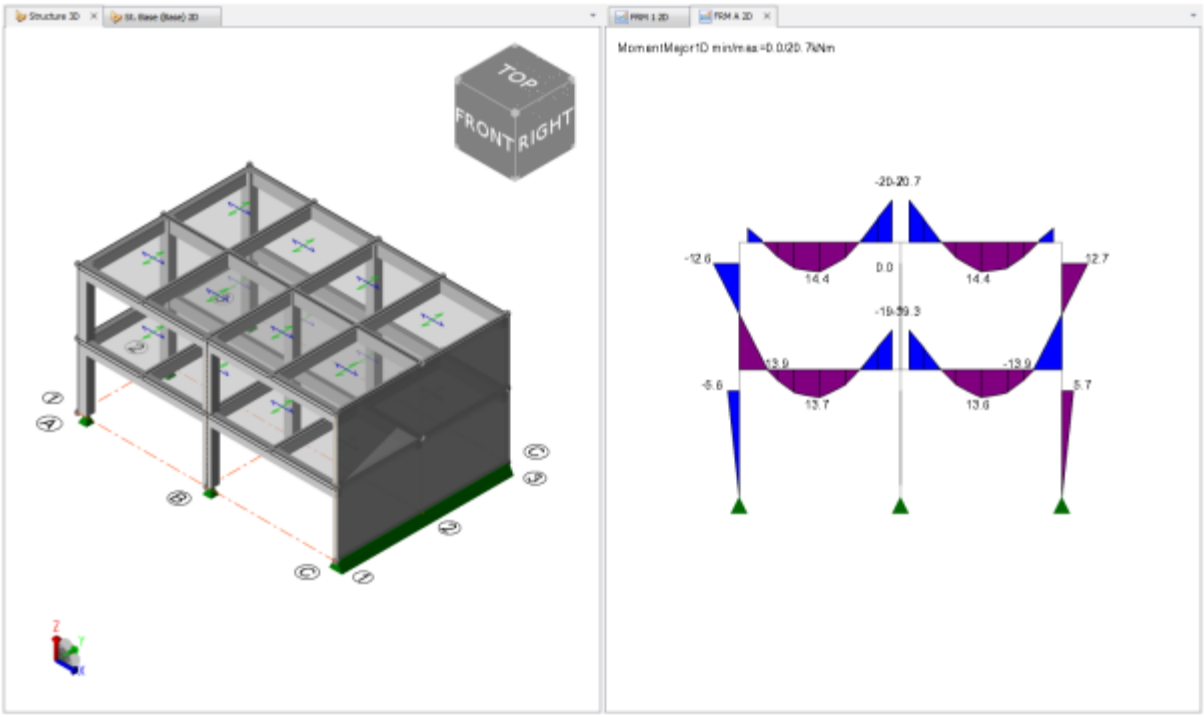
To change the view regime from the active view tab

An open scene view can be made active by left clicking its tab. Once a view has been made active its regime can be changed by right-clicking the tab and then selecting one of the other view regimes from the right-click menu.



Scene View Tab Groups

When multiple scene views have been created, by default only the active view visible. It is often useful to instead display views side by side; this can be achieved by creating new Tab Groups.



To create a new Tab Group from an existing view tab

1. right-click an existing view tab and from the right-click menu select New Horizontal Tab Group, or New Vertical Tab Group as required.

To create a new Tab Group using the docking control

When you grab a view by its tab and drag it, a docking control appears in the middle of the view. You can then use this control to create a new Tab Group.

To create a new Vertical Tab Group:

1. Drag the grabbed view over the left or right button of the docking control and then release the mouse.

To create a new Horizontal Tab Group:

1. Drag the grabbed view over the top or bottom button of the docking control and then release the mouse.

To move a view between Tab Groups

1. right-click the view tab and from the right-click menu select 'Move to Next Tab Group.

Working with Command Prompts

Yellow command prompts are displayed across the top of the active view to guide you through each *Tekla Structural Designer* command.

Press <Esc> in a command prompt to cancel, or go back

- Press <Esc> at the first command prompt to cancel.
- Press <Esc> at subsequent command prompts to go back to the first prompt.
- Selecting another command also cancels the current command.

Example

The parallel grid line command consists of four steps - hence four consecutive command prompts are displayed:

Add Parallel Grid/Construction Line: Select Reference Line **<press ESC to cancel>**

Add Parallel Grid/Construction Line: Specify Distance **<press ESC to cancel>**

Add Parallel Grid/Construction Line: Pick Point 1 **<press ESC to cancel>**

Add Parallel Grid/Construction Line: Pick Point 2 **<press ESC to cancel>**

The grid line is created after the fourth prompt, after which the second prompt is automatically redisplayed:

Add Parallel Grid/Construction Line: Specify Distance **<press ESC to cancel>**

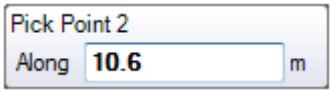
At this point you could either continue to create additional lines parallel to the original, or by pressing <Esc> go back to the first prompt:

Add Parallel Grid/Construction Line: Select Reference Line **<press ESC to cancel>**

You could then select a new reference line and continue again, or by pressing <Esc> again cancel the command.

Using the tooltip for input in a command

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



To enter a value either:

- move the cursor to the point required and click to use it,
- alternatively, press <F2> to type an exact number and press <Enter> to use it.

Undoing a command

Once a command has completed, to undo it do the following:

- Click **Undo** in the Quick Access Toolbar

Undo and redo can be used to repetitively undo/redo commands in the sequence they were performed.

Selecting Entities

In order to perform an operation (edit, delete etc.) on an entity, or group of entities they must first be selected.

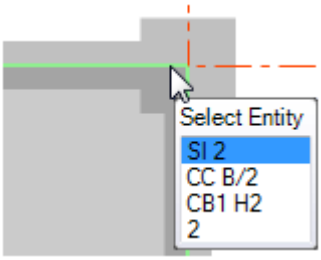
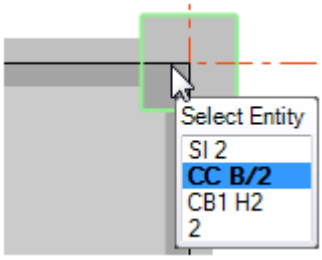
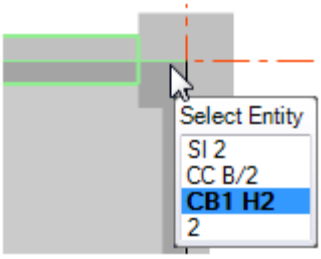
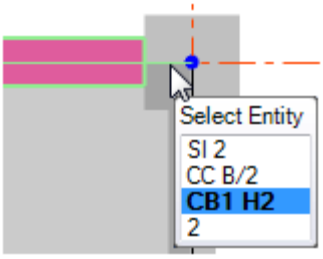
All entity types can be selected/deselected using the mouse provided they are visible in the scene view you are working in. Alternatively they can be selected using the **Find** command. Members can also be selected from the Structure Tree and the Groups Tree.

If planar objects (slabs, roofs, area loads etc.) lie over the start point for performing your selection, you may want to use Scene Content to switch off their display, (assuming you don't want to have them considered in your selection).

How do I select an individual entity?

1. Move the cursor over the required entity in one of the 2D or 3D Views.
   - If the entity is the only one at that location it will become highlighted (it will also be the one listed in the **Select Entity tooltip**).
   - If several entities exist at the same location they will all be listed in the **Select Entity tooltip**, only the first one being highlighted. If this is not the required entity, use the tab key or up/down cursor keys to scroll through the list.
2. When the required entity is highlighted, you can either press the Enter key or left click to select it.

The selected entity's properties are displayed in the Properties Window.

Example	View
Four entities are located under the current cursor position, so each of these are listed in the **Select Entity tooltip**. We require to select the horizontal beam, however the currently highlighted entity shown in the tooltip is a slab panel.	
By pressing the tab key once, the next entity in the list (a column) becomes highlighted.	
This is still not correct, so press the tab key once more - the required beam is now highlighted.	
Either press the Enter key or left click and the beam is selected.	

Individual members can also be selected by clicking their name in the **Structure Tree***.*

How do I add further entities to the current selection?

1. Hold the Ctrl key whilst clicking on each subsequent entity.

   The selected entity's properties are added to those already displayed in the Properties Window.

How do I select multiple entities by dragging a box?

If you only want to select those entities which are totally encompassed by the box, then:

1. Move the cursor to the **left** corner of an imaginary box which will encompass the entities that you want to select.
2. Click and hold the left mouse button.
3. Drag to the diametrically opposite corner of the box (you will see a purple rectangle on the screen which follows your mouse movements and helps you to check the area you are selecting).
4. Release the mouse button.

If you want to select those entities which are totally encompassed by the box, and which it crosses, then:

1. Move the cursor to the **right** corner of an imaginary box which will encompass the entities that you want to select.
2. Click and hold the left mouse button.
3. Drag to the diametrically opposite corner of the box (you will see a green rectangle on the screen which follows your mouse movements and helps you to check the area you are selecting).
4. Release the mouse button.

   The selected entities' properties are displayed in the Properties Window.

How do I deselect a single entity from the current selection?

1. Click the entity you want to de-select.

   The entity's properties are removed from the Properties Window.

How do I deselect all entities?

1. Press the Esc key to deselect all currently selected entities.

   The information displayed in the Properties Window is also cleared.

Entity selection using Find

If you know the name of the entity you are looking for it can be selected using the **Find** command.

1. Click **Home** > **Find** ( )
2. Type a part of the name you are looking for into the box.

   Any matching entity names are immediately displayed in the table below.
3. Click an entity name that you want to select.

Hold the Ctrl key whilst clicking to append further selections.

4. Click **Select**.

The selected entities' properties are displayed in the Properties Window.

Member selection using the Structure Tree

Only individual members can be selected from the Structure Tree , (it in not possible to select multiple members in this way).

1. Expand the Members branch of the **Structure Tree** and then the appropriate sub-branches until the member names are displayed.

EG

If you want to find a particular reinforced rectangular concrete column click the plus sign (⊞) to the left of the Concrete Column entry, then the plus sign (⊞) to the left of the Reinforced Concrete entry, then the plus sign (⊞) to the left of the Rectangular entry.

2. Right-click the required member name and then pick **Select in visible views** from the context menu.

Member selection using the Groups Tree

Member groups and individual members can both be selected from the Groups Tree.

1. Expand the appropriate branch of the **Groups Tree** until you can click the required group or member name.

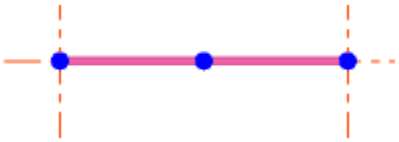
2. Right-click the required group or member name and then pick **Select in visible views** from the context menu.

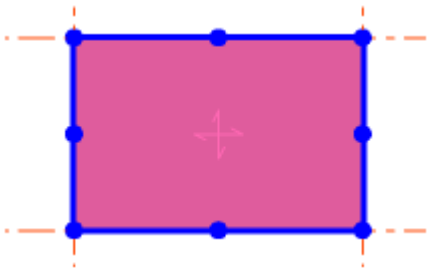
Entity Nodes

When an individual entity is selected the nodes that define it are also displayed; each entity type having a different number of nodes associated.

For example:

- Grid lines and single span 1D elements have nodes at each end and at the mid-point.
- Panels have nodes at each vertex and at the mid-point of each edge.
- Concrete walls have nodes at each corner.

grid line	
1D element	

panel	
wall	

A node can be selected and moved in order to stretch, shorten, reshape or move the entity.

Starting a New Project

New models are initiated from the Home toolbar. They can either be created from scratch, or, if you intend to create several models from a common start point, you might consider setting up a template in advance.

Either way, before commencing you should take a few moments to review the defaults that will be applied to the model. These are held in the "active" settings set in the Settings dialog. Ensuring these match your typical requirements at the outset saves time as it eliminates unnecessary editing at a later stage.

Once a model is open, its settings can be edited from the Model Settings dialog.

Home toolbar

The **Home** toolbar contains the following commands:

Button	Description
New	**New** (icon) creates a new blank project from scratch. See: Creating a new project from scratch
New (drop list) New	**New** (droplist) creates a new project based on an existing template. See: How do I create a new project from a template?
Open	**Open** an existing project.
Close	**Close** the currently open project.
Save	**Save** the currently open project.
Save As	**Save As** saves the currently open project to a new name, or to a template.
Project Wiki	Opens the **Project Wiki** dialog which is used to record miscellaneous properties associated with the project, and to record revisions. See: Editing project details using Project Wiki

Model Settings	Opens the Model Settings dialog which is used to specify the settings **for the current project**.
Manage Property Sets	Import, Export and Delete property sets. See: Manage Property Sets
Manage View Configurations	Opens or deletes saved View Configurations. See: View Configurations
Find	Find and then select objects in the model by typing a part of their name. See: Entity selection using Find
Walk	Walk through 3D views. See: How do I walk through the model in a 3D view?
Structural BIM Import	Import a model from a Neutral File. See: How do I import a project from a Structural BIM Import file?
Tekla Structures Export	Export a model to Tekla Structures. See: How do I export a model to Tekla Structures?
Autodesk Revit Export	Export a model to Revit. See: How do I export a model to Autodesk Revit Structure?
IFC Export	Export a model to IFC. See: How do I export a model to IFC?
Cellbeam Export	Export a beam to Westok Cellbeam. See: How do I export a beam to Westok Cellbeam? Cellbeam import and Export are not yet available in the current release.
Cellbeam Import	Import a beam from Westok Cellbeam. See: How do I import a beam from Westok Cellbeam? Cellbeam import and Export are not yet available in the current release.
STAAD Export	Export a model to the STAAD. See: How do I export a model to STAAD?

Robot Export	Export a model to Robot. See: How do I export a model to Autodesk Robot Structural Analysis?
Cloud Export	Export a model to the Cloud. See: How do I export a model to the Cloud?
'TEL' File Import	Import a model from a TEL File. See: How do I import a project from a TEL file?
3D DXF Import	Import a model from a 3D DXF File. See: How do I import from a 3D DXF file?
TCD Export	Export connections to Tekla Connection Designer.
TPFD Export	Export portal frames to Tekla Portal Frame Designer. See: How do I export portal frames to Tekla Portal Frame Designer?
ADAPT Export	Export a model to ADAPT. See: How do I export a model to ADAPT?
Settings	Opens the Settings dialog which controls the defaults that are applied **prior to starting a new project**.
Materials	Opens the Materials dialog for viewing and adding to the list of materials available to work with.
Check for Updates	Click this button to check that the program is up to date. If critical updates are available, you will be notified, and you can select whether you want to install the updates.
License Manager	Opens the License Manager.

Related topics

- Commands on the ribbon toolbars

Creating a new project from scratch

Before creating a new project you should ensure an appropriate settings set is active.
See: How do I specify the active settings set?

How do I create a new project from scratch?

1. Click **Home** > **New** ( )
(being careful to click the icon button as shown, rather than the drop list button beneath it).

2. Your new project opens and you will see:

   - two Scene Views - ***St. Base (Base) 2D*** (which will show the Base level of your model) in front of ***Structure 3D*** (which will show the whole model in 3D).
   - a tabbed Project Workspace - in which the **Structure tab** displays a Structure Tree containing a **Structure** main entry below which there two major branches — **Levels** and **Sub Models**
   - a Properties Window - which, (provided the **Structure** main entry is selected in the **Structure Tree**), shows the **General** and **Meshing** defaults for the current model.

3. Click **Structure** in the **Structure Tree** and review the **General** defaults in the **Properties Window**:

   - Select **Show Building Direction Arrows**.
   - Since by default the **Building Direction Rotation** is 0 degrees the building's principal axes (Dir 1 and Dir 2) are initially aligned with global X and Y. If you are defining a model that is aligned differently you should adjust the Building Direction Rotation to suit.

4. Click the plus sign ( ) to the left of **Levels** and you will see that your model contains only a base level. Double click **Levels** to display the Construction Levels dialog from where you can add more levels as needed.

If you are creating a steel structure each level you add should be set as T.O.S. (top of steel). If you are creating either a concrete or mixed material structure you should instead set the type as S.S.L.(structural slab level)

Once the levels have been set up you can:

   - left click an existing level in the **Structure Tree** to show the details for that level in the **Properties Window**. If you amend these details they are applied to the level immediately.
   - double-click the name of a **Construction Level** to open a 2D view for that level.

5. You can then proceed to add building objects, either in the Structure 3D view, or any of the 2D level views.

Working with templates

Working with templates

If the projects you will be creating share a common start point, a template can be employed to avoid repetition of input. In order to use a template you must have created it previously. It can contain as much or as little of a model that you consider applicable to serve as the start point for subsequent models. For example, a particular template may comprise a simple grid and the height of the First Floor Construction Level only.

How do I create a new template?

1. Start a new project. (See Creating a new project from scratch)
2. Create only the model data that you want to be included in the template.
3. Click **Home** > **Save As** ( )
4. In the **Save As** dialog, change the **Save as type** to **Tekla Structural Designer template file (*.tsmdt)**.
5. Enter a file name to describe the template and ensure the dialog is pointing to the folder where you want the template to reside.
6. Click **Save.**

How do I create a new project from a template?

Before creating a new project from a template you should ensure an appropriate settings set is active.
See: How do I specify the active settings set?

To create a new project from a template:

1. Click **Home** > **New** ( )
   (being careful to click the lower drop list button, rather than the icon button above it).

   The drop list menu consists of the most recently used templates and an **Open Template** option to navigate to any other templates not shown.
2. Select the required template from the list. The template opens and you will see:
   - One or more Scene Views - their content matching the selected template,
   - a Project Workspace displaying a Structure Tree matching the selected template,
3. Click **Home** > **Save As** ( )
4. In the **Save As** dialog, change the **Save as type** to **Tekla Structural Designer project files (*.tsmd)**.
5. Enter a file name to describe the project and ensure the dialog is pointing to the folder where you want the project to reside.

6. Click **Save.**

7. You can then proceed to add the remaining building objects and loads as necessary to complete the project.

Model Settings

Various defaults and settings that apply to **the current project only** can be controlled via the Model Settings dialog.

How to apply and manage Model Settings

To access the model settings:

- Click **Home** > **Model Settings** ( )

If you change any model settings, click:

- **OK -** to apply the changes directly to the current project, or
- **Save... -** to save the changes back to the active settings set for future use, or
- **Cancel... -** to cancel the changes

You can also click:

- **Load... -** to revert to the model settings specified in the active settings set. (This loads all of the **model settings** in the set, not just those displayed on the current page. It does not load **Analysis Options** or **Design Options**.)

Model Settings dialog

Located on the **Home** toolbar, the **Model Settings** dialog ( ) contains various pages for controlling defaults and settings that apply to **the current project only**.

Editing project details using Project Wiki

Model statistics and revision control information are stored in **Project Wiki**.

How do I edit the project details and view the revision history?

You set the initial project details as you create a new project. If you want to change these later, or view other project parameters this is done as follows:

- Click **Home** > **Project Wiki** ( )

From the **Project Wiki** dialog you can see:

Project Summary

Parameters are entered here for inclusion in the output reports.

Revisions

Revision history can be tracked here.

Sessions

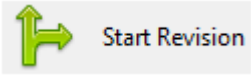
Shows the time at which each revision was started and last saved.

Changes
Displays the changes associated with each revision.

Metrics
Displays statistics related to the model size.

How do I record revisions?

You set the initial project details as you create a new project. If you want to change these later, this is easily done.

1. You must save the project before you can start recording revisions. (eg "ABC123")
2. Click the File menu > 

3. Enter the Revision ID and add a notes related to this revision.
4. In order to keep a record of the changes made in this revision, check **Track Changes**.
5. Click OK.
6. Continue to develop the model then save it to a new name.(eg "ABC123 rev A")
7. Repeat steps 2-5 above as required.

Modeling and Editing Guide

Most of the operations required to build your model can be performed from the Model toolbar; typically Construction Levels and Architectural Grids and Grid Lines are set out first, after which the model can be created from a variety of member types. Further commands for editing the model can be accessed from the Edit toolbar.

Model toolbar

The groups listed below are displayed when a 2D or 3D View is active.
If any other view is active only the "Levels", "Edit" and "Validate" groups are displayed.

Levels group

Button	Description
Construction Levels	Opens the Construction Levels dialog from where you can define the levels required in order to construct your model. See: Construction Levels
Frame	Creates a **Frame** when in a 3D View, (when a 2D View is displayed **Frame** is inactive). The drop list underneath can be used to delete existing Frames. A **Frame** is simply a 2D View of the model created in a vertical plane defined by an existing grid line. Because only those members that lie within the plane of the Frame are displayed, this type of view can be particularly useful for defining bracing. See: Frames and Slopes

Sloped Plane	Creates a **Sloped Plane** when in a 3D View, (when a 2D View is displayed **Sloped Plane** is inactive). The droplist underneath can be used to delete existing Sloped Planes. A **Sloped Plane** is simply a 2D View of the model created in a sloped plane. It is defined by selecting 3 existing grid points. Because only those members that lie within the plane of the Slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps. See: Frames and Slopes

Grid and Construction Lines group

The **Grid and Construction Lines** group contains a **Grid Line** drop list, a **Construction Line** drop list and the **Dimension** command.

The commands on the two drop lists are only available when a 2D view representing the construction level on which you want to create your grid or construction lines is active.

Button	Description
Grid Line (on 1st drop list)	Inserts a grid line between two specified points.
Parallel (on 1st drop list)	Inserts a parallel grid line at a specified distance from the selected line.
Parallel (quick) (on 1st drop list)	Inserts multiple parallel grid lines at specified distances from the selected line. For example: **3*5** - inserts 3 parallel lines, each at 5m spacing **3,4,5** - inserts 3 parallel lines at spacings of 3,4, and 5m respectively.
Perpendicular (on 1st drop list)	Inserts a perpendicular grid line at a specified location on the selected line.
Rectangular Wizard (on 1st drop list)	Used to insert a system of grid lines in two directions at a specified angle.

Sector Wizard (on 1st drop list)	Used to insert a system of radial and arc grid lines.
Arc (on 1st drop list)	Used to insert an arc grid line.
Import DXF (on 1st drop list)	Used to import grid lines from a dxf file.
Construction Line (on 2nd drop list)	Inserts a construction line between two specified points.
Parallel (on 2nd drop list)	Inserts a parallel construction line at a specified distance from the selected line.
Parallel (quick) (on 2nd drop list)	Inserts multiple parallel construction lines at specified distances from the selected line. For example: **3*5** - inserts 3 parallel lines, each at 5m spacing **3,4,5** - inserts 3 parallel lines at spacings of 3,4, and 5m respectively.
Perpendicular (on 2nd drop list)	Inserts a perpendicular construction line at a specified location on the selected line.
Rectangular Wizard (on 2nd drop list)	Used to insert a system of construction lines in two directions at a specified angle.
Sector Wizard (on 2nd drop list)	Used to insert a system of radial and arc construction lines.
Arc (on 2nd drop list)	Used to insert an arc construction line.

Dimension	Dimensions allow you to show distances in your structure between appropriate points. The dimension lines are included on any drawings you create.

Related topics

- Grid Lines and Construction Lines
- Dimensions

Steel group

The **Steel** group contains the following commands:

Button	Description
Steel Column (on 1st drop list)	Creates a steel column which adopts the specified steel column properties set
Plated (on 1st drop list)	Creates a plated steel column which adopts the specified steel column properties set
Concrete Filled (on 1st drop list)	Creates a concrete filled steel column which adopts the specified steel column properties set
Concrete Encased (on 1st drop list)	Creates a concrete encased steel column which adopts the specified steel column properties set
Steel Beam (on 2nd drop list)	Creates a steel beam which adopts the specified steel beam properties set
Plated (on 2nd drop list)	Creates a plated steel beam which adopts the specified steel beam properties set
Westok Cellular (on 1st drop list)	Creates a Westok cellular steel beam which adopts the specified steel beam properties set

Westok Plated (on 1st drop list)	Creates a Westok plated steel beam which adopts the specified steel beam properties set
Fabsec (on 1st drop list)	Creates a Fabsec steel beam which adopts the specified steel beam properties set
Steel Brace (on 3rd drop list)	Creates a steel brace which adopts the specified steel brace properties set
X Brace (on 3rd drop list)	Creates an X steel brace which adopts the specified steel brace properties set
K Brace (on 3rd drop list)	Creates a K steel brace which adopts the specified steel brace properties set
V Brace (on 3rd drop list)	Creates a V steel brace which adopts the specified steel brace properties set
A Brace (on 3rd drop list)	Creates an A steel brace which adopts the specified steel brace properties set
Steel Joist	Creates a steel joist which adopts the specified steel joist properties set
Steel Truss (on 4th drop list)	Runs the **Steel Truss Wizard** to define a truss with the specified edit steel truss properties set
Space (on 4th drop list)	Runs the **Space Truss Wizard** to define a truss with the specified edit steel truss properties set
Portal Frame	Creates a portal frame. To create the frame, you firstly define the two column base positions between which the frame will lie. The base positions are restricted to lie on existing grid points. All other details of the portal frame are subsequently entered in the **Portal Frame** dialog. See: Modeling Portal Frames

Related topics

- Steel, Cold Rolled, and Cold Formed Member modeling

Concrete group

The **Concrete** group contains the following commands:

Button	Description
Mid-pier Concrete Wall	Creates a mid-pier shear wall adopting the properties specified in the concrete wall properties set
Meshed Concrete Wall	Creates a meshed shear wall adopting the properties specified in the concrete wall properties set
Wall Opening	Creates a rectangular opening in an existing meshed shear wall when in a Frame View. (**Wall Opening** is inactive in 3D Views and Level Views). See: How do I create a door or window opening in an existing concrete wall?
Concrete Column	Creates a reinforced concrete column adopting the properties specified in the concrete column properties set
Concrete Beam	Creates a reinforced concrete beam adopting the properties specified in the concrete beam properties set

Related topics

- Modeling Concrete Walls
- Modeling Concrete Columns
- Modeling Concrete Beams

Slabs group

The **Slabs** group contains the following commands:

Button	Description
Slab on Beams (on 1st drop list)	Creates a concrete slab which adopts the specified slab on beams item properties set

Flat Slab (on 1st drop list)	Creates a concrete flat slab which adopts the specified flat slab item properties set
Precast (on 1st drop list)	Creates a precast concrete slab which adopts the specified precast item properties set
Steel Deck (on 1st drop list)	Creates a steel deck which adopts the specified steel deck item properties set
Timber Deck (on 1st drop list)	Creates a timber deck which adopts the specified timber deck item properties set
Composite Slab (on 1st drop list)	Creates a composite slab which adopts the specified composite slab item properties set
Column Drop (on 2nd drop list)	Creates a column drop which adopts the specified column drop properties
Slab Opening (on 2nd drop list)	Creates a slab opening which adopts the specified slab opening and mat opening properties
Slab Overhang (on 2nd drop list)	Creates a slab overhang which adopts the specified slab overhang and mat overhang properties
Slab Split	This command is used to split an existing slab. See: How do I split a slab?
Slab Join	This command is used to join an existing slabs. See: How do I join slabs?

Related topics

- Slab modeling

Timber group

The **Steel** group contains the following commands:

Button	Description
Timber Column	Creates a timber column which adopts the specified timber column properties set
Timber Beam	Creates a timber beam which adopts the specified timber beam properties set
Timber Brace (on 1st drop list)	Creates a timber brace which adopts the specified timber brace properties set
X Brace (on 1st drop list)	Creates an X timber brace which adopts the specified timber brace properties set
K Brace (on 1st drop list)	Creates a K timber brace which adopts the specified timber brace properties set
V Brace (on 1st drop list)	Creates a V timber brace which adopts the specified timber brace properties set
A Brace (on 1st drop list)	Creates an A timber brace which adopts the specified timber brace properties set
Timber Truss (on 2nd drop list)	Creates a steel truss which adopts the specified edit timber truss properties
Space (on 2nd drop list)	Creates a space truss which adopts the specified edit timber truss properties

Related topics

- Timber Member modeling

Cold Formed group

The **Cold Formed** group contains the following commands:

Button	Description

Column	Creates a column which adopts the specified create cold formed column properties set
Beam	Creates a beam which adopts the specified create cold formed beam properties set
Brace (on drop list)	Creates a brace which adopts the specified create cold formed brace properties set
X Brace (on drop list)	Creates an X brace which adopts the specified create cold formed brace properties set
K Brace (on drop list)	Creates a K brace which adopts the specified create cold formed brace properties set properties
V Brace (on drop list)	Creates a V brace which adopts the specified create cold formed brace properties set
A Brace (on drop list)	Creates an A brace which adopts the specified create cold formed brace properties set

Related topics

- Steel, Cold Rolled, and Cold Formed Member modeling

Panels group

The **Panels** group contains the following commands:

Button	Description
Roof Panel	Creates a roof panel which adopts the specified Roof panel properties
Wall Panel	Creates a wall panel which adopts the specified Wall panel properties

Related topics

- Modeling Roof Panels
- Modeling Wall Panels

Miscellaneous group

The **Miscellaneous** group contains the following commands:

Button	Description
Support	Creates additional supports underneath existing members in the model, which adopt the specified Support properties
Element	Creates a new analysis element which adopts the specified Element properties.
Measure	Measures the distance between any existing construction points or intersection points in the model. See: How do I Measure distances?
Measure Angle	Measures angles between existing points in 2D Views. See: How do I Measure Angles?
Bearing Wall	Creates a bearing wall. See: Modeling Bearing Walls

Related topics

- Modeling Supports

Validate

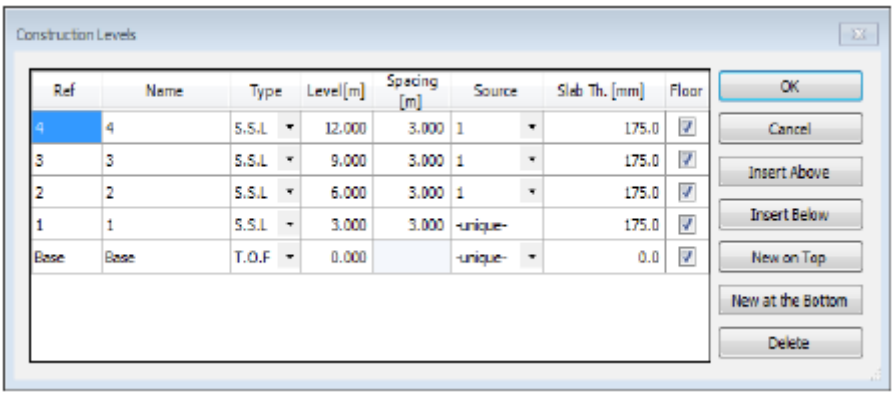
Button	Description
Validate	This command is used to perform the model validity checks. See: Model Validation

Construction Levels

The Construction Levels dialog is used to define those levels that you need to identify in order to construct your model. They could be floors or roofs, or they may simply be intermediate levels required in order to define specific items...

Construction Levels dialog

From this dialog you can define the levels required in order to construct your model.



Fields

Ref

Each construction level should be given a unique reference. Typically this might be a storey number, 1, 2 , 3 etc.

Name

Each construction level can also be given a name to further assist identification. "First Floor", or "Mezzanine" etc.

Type

Select the type from the drop down menu:

- T.O.S = Top of Steel
- S.S.L = Structural Slab Level
- T.O.F = Top of Foundation

Slabs are modeled above the level when it is set to T.O.S but below the level when it is set to S.S.L or T.O.F

Level

The height of the construction level above the base level.

Spacing

The height of the construction level above the level immediately below.

Source

Unique levels can be edited independently, whereas edits to identical levels are applied at both levels simultaneously.

See: How do I make one level an identical copy of another? and How do I make one level an independent copy of another?

Slab Th.

The default thickness of any slabs created at the level.

Floor

By setting a construction level to be a Floor you are indicating that it is a major level in the building. Floor levels determine the number of sub models that are created for the

chasedown analysis. Floor levels are also used to determine items such as your inter story height.

There will certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define:

- half landing levels and stairs,
- K Bracing - you require a construction level for the intermediate bracing connection points,
- steps in the building floor levels.

Where you define a level which is clearly not a floor, then you should clear the **Floor** check box. If a floor exists only at some locations in the level (e.g. in a building with stepped floor levels), then you should select **Floor**.

Buttons

Insert Above

Click this button to insert a single construction level above the current level at the same spacing. See: How do I insert multiple Construction Levels?

Insert Below

Click this button to insert a single construction level below the current level at the same spacing. See: How do I insert multiple Construction Levels?.

New on Top

Click this button to insert multiple construction levels above the topmost level at variable spacings.

EG

*3*5 - inserts 3 levels, each at 5m spacing*
3,4,5 - inserts 3 levels at spacings of 3,4, and 5m respectively

New at the Bottom

Click this button to insert multiple construction levels below the bottommost level at variable spacings.

Delete

Deletes the selected level.

Related topics

- Construction Levels
- How do I open the Construction Levels dialog?
- Sub Models

How do I open the Construction Levels dialog?

To open from the ribbon:

- Click **Model** > **Construction Levels** ( )

To open from the Structure Tree:

- Double-click Levels

Related topics

- Construction Levels dialog

How do I insert a single Construction Level?

1. In the Construction Levels dialog, select an existing level.
2. Click **Insert Above** or **Insert Below** as appropriate.
3. If necessary change the default level name to something more appropriate.
4. For the new level specify the height above the base in the **Level** field, the inter-storey **Spacing** is then calculated automatically; alternatively if you specify the inter-storey **Spacing** the **Level** is calculated automatically.

A default height is calculated for the new level based on the spacings of any existing levels immediately above/below it. Either accept, or adjust as required.

5. Indicate if the level is to be treated as a **Floor** by selecting the appropriate check box.

How do I insert multiple Construction Levels?

1. In the Construction Levels dialog, select an existing level.
2. Click **New on Top** or **New at the Bottom** as appropriate.
3. Enter the level spacings.
4. Click **OK**

EG

*If you have new levels at 12′ 6″, 18′ 6″, 27′ 6″, 36′ 6″ and 27″ above the current level, you can specify this as 12′ 6″, 18′ 6″, 3*9′*

5. If necessary change the default level names to something more appropriate.

If you are designing to BS 5950 and the model contains simple columns, then you must set each level at which beams trim into the simple column so that the Floor setting is Yes.

How do I make one level an identical copy of another?

1. Open the Construction Levels dialog.
2. At the level you want to be a copy, click in the **Source** column.
3. Choose the level you want it to be identical to.
4. Click **OK**

Edits to either the source or the identical level are automatically applied at both levels.

How do I make one level an independent copy of another?

1. Open the Construction Levels dialog.
2. At the level you want to be a copy, click in the **Source** column.
3. Choose the level you want it to be identical to.
4. Click **OK**
5. Reopen the Construction Levels dialog.
6. Click in the **Source** column at the same level and change the setting back to **unique**
7. Click **OK**

Edits to a unique level are applied to that level only.

How do I delete a construction level?

To delete from the Construction Levels dialog:

1. Select the level to be removed.
2. Click **Delete**
3. Click **OK**

To delete from the Structure Tree:

1. Open the Levels branch of the Structure Tree.
2. Right-click the title of the construction level to be removed.
3. Click **Delete** from the context menu that appears.

This option completely deletes the entire level and all its associated details — all beams, members, slabs… You cannot recover the level once you have deleted it. Please take due care when you use this feature.

How do I modify the properties associated with a level?

Certain properties can be modified directly from the Construction Levels dialog, however there are a number of other parameters associated with levels that can only be edited in the Properties Window.

To edit in the Properties Window:

1. Open the Levels branch of the Structure Tree.
2. Select the title of the construction level to be edited.
3. Make your changes to the Level Properties displayed in the Properties Window.

Frames and Slopes

A **Frame** is simply a 2D View of the model created in a vertical plane defined by an existing grid line - because only those members that lie within the plane of the Frame are displayed, this type of view can be particularly useful for defining bracing.

A **Slope** is simply a 2D View of the model created in a sloped plane. It is defined by selecting 3 existing grid points - because only those members that lie within the plane of the Slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps.

A sloped plane must be entirely contained within a single Sub Model because the Sub Model determines the mesh parameters to be applied.

In a sloped plane positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

How do I create a Frame?

1. Obtain a 3D view of your structure in which you can see the base grid line associated with the frame that you want to create.

2. Click **Model** > **Frame** ( )

3. Position the cursor over the grid line for the frame you want to create.

4. Click to create the frame.

5. This creates a frame view for the selected grid line. You can open this view by clicking the plus sign ( ) to the left of Frames in the Structure Tree and then double-clicking the name of the frame whose view you want to open.

How do I create a Slope?

In order to create a slope you need to be able to click three existing grid points (that are not co-linear) which lie in the plane of the slope.

Grid points are formed at grid line intersections on construction levels. Therefore, if the points required to define the slope don't currently exist, you may need to insert new construction levels and/or grid lines to form them.

To create a slope:

1. Obtain a 3D view of your structure in which you can see three grid points which define the sloped plane.

If you cannot see grid line intersections on a particular construction level in the 3D view select the level in the Structure Tree and then select **Show grids in the 3D view** *in the Properties Window. (At the same time you can also clear the same property at other levels in order to simplify the display.)*

2. Click **Model** > **Sloped Plane** ( )

3. Click the three points which define the sloped plane.

4. This creates a sloped plane view. You can open this view by clicking the plus sign ( ) to the left of Slopes in the Structure Tree and then double-clicking the name of the sloped plane whose view you want to open.

Grid Lines and Construction Lines

Grid lines facilitate the placing of objects in your model. Construction lines serve the same purpose, only without displaying a grid bubble.

Architectural Grids and Grid Lines

An **Architectural Grid** is a collection of **Grid Lines** that lie in a horizontal plane. Each architectural grid is only displayed in the 3D View at the lowest level in the structure (they are not displayed in 2D Views). Vertical lines can be projected from each grid intersection of the architectural grid - these extend to the topmost level at which the architectural grid has been applied.

Each architectural grid can be assigned a color and can be set to display grid line names and vertical lines via the architectural grid properties. (Note that when these options are checked they will only be displayed provided the Architectural Grids>Text2D and VerticalLines options are also checked in Scene Content.)

Separate architectural grids could be created as a means to allow two disconnected structures to be created in the same model (where the only 'connection' is a movement joint between the two structures). They can also be used to allow distinct grids to be applied at different levels in the same structure (for example where there might be a concrete car park at one grid spacing at lower levels that supports a steel framed structure above it set out at different grid spacings).

This works provided the two architectural grids are not shared by members at the same level. Errors could result if members that are set out by different architectural grids are connected to each other.

Each **Grid Line** is associated with an architectural grid and is only created at a given level provided the architectural grid has been applied at that level.

Grid lines that have been created at a given level can optionally be shown or not shown in both the 2D and 3D views via the level properties. (Note that when these show options are checked the lines will only be displayed provided the relevant Grid & Construction Lines options are also checked in Scene Content.)

Why are the grid lines not being shown at a particular level?

Having activated the display of grid & construction lines in Scene Content for a particular 2D or 3D view, you may still find that the grid lines are not displayed on particular levels.

In this situation you should first check the level properties to ensure that grids are set to be shown. This is done as follows:

1. Open the **Levels** branch of the Structure Tree.
2. Left-click the level at which you want to have the grids shown.
3. In the Properties Window for the level - select **Show grids in plane view** to make the grids visible in the 2D view, select **Show grids in 3D view** to make the grids visible at that level in 3D views.

   If the grids are still not visible in the 3D view at certain levels, you will need to ensure they have been applied at the levels in question - see: How to I apply an existing architectural grid to a specific level?

How to I apply an existing architectural grid to a specific level?

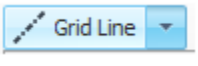
1. Open the **Architectural Grids** branch of the Structure Tree.

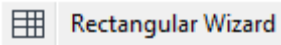
2. Right-click the architectural grid name required and then pick **Edit...** from the context menu.

3. Ensure the levels at which you want the architectural grid to be applied are all checked.

How do I create a rectangular grid line system?

1. Make sure that the 2D view representing the construction level on which you want to create your rectangular grid system is active.

2. Click **Model** > 

, and select 

3. Enter a name for the grid and choose a color for the grid lines you want to create.

4. If you want to create the same grid layout on every construction level in your building ensure that each level is selected in the dialog. If you don't want the grid to be displayed at a particular level clear it.

5. Define the origin of the grid system, either by clicking in the 2D view, or by entering the coordinates in the **Rectangular Grid Wizard** dialog.

6. Choose which grid lines you want to create, in the X direction only, in the Y direction only, or in both directions, and choose the grid line representation - dash dot, dot… that you want to use for these grid lines.

7. Define the layout of grids for the bays in the X direction.

If you are only creating grid lines in the X direction, then the wizard skips this step.

You can define a regular or irregular grid layout:

- for a regular grid you define the number of bays you want to create and the bay centers,
- for an irregular grid you define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

EG

If you have bay centres of 18′, 27′ , 18′ 6″ , 18′ 6″ , 18′ 6″ and 27″ you can specify this as 18′ , 27′ , 3x18′ 6″ , 27″

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

EG

Numerical grid line numbering is self explanatory. For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M…

8. Define the layout of grids for the bays in the Y direction.

If you are only creating grid lines in the Y direction, then the wizard skips this step.

You can define a regular or irregular grid layout:

- for a regular grid you define the number of bays you want to create and the bay centers,
- for an irregular grid you define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

EG

If you have bay centres of 18′, 27′ , 18′ 6″ , 18′ 6″ , 18′ 6″ and 27″ you can specify this as 18′ , 27′ , 3x18′ 6″ , 27″

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

9. Define the rotation of the grid. You can do this graphically by moving the mouse over the 2D view and clicking, or by entering values into the wizard's dialog. If you use the latter approach you can specify the rotation either with respect to the grid system's local x or local y directions.

This is useful if you are going to create a grid system which is not orthogonal.

10. Finally specify the angle between the grid's axes, you can do this with respect to either the X or Y axis system and click **Finish** to create your grid layout. Again you can do this graphically, or you can use the wizards dialog - it's your choice.

How do I create a radial grid line system?

1. Make sure that the 2D view representing the construction level on which you want to create your radial grid system is active.

2. Click **Model** > Grid Line, and select Sector Wizard

3. If you want to create the same grid layout on every construction level in your building ensure that each level is checked in the dialog. If you don't want the grid to be displayed at a particular level clear it.

4. Define the origin of the grid system, either by clicking in the 2D view, or by entering the coordinates in the **Radial Grid Wizard** dialog.

5. Choose which grid lines you want to create, the radial lines only, the arcs only, or both of these, and choose the grid line representation - dash dot, dot… that you want to use for these grid lines.

6. Define the layout of the arcs that you will create.

If you are only creating radial grid lines the wizard skips this step.

You can define a regular or irregular grid layout:

- for a regular grid you define the number of arcs you want to create and the distance between them,
- for an irregular grid you define the distance between successive pairs of arcs lines, separating the numbers by commas. If you have a number of arcs that are the same distance apart, then you can specify these as a single entry.

EG

If you have arcs at distances of 9′, 12′ , 10′ 6″ , 10′ 6″ , 10′ 6″ and 12″ you can specify this as 9′ , 12′ , 3x10′ 6″ , 12″

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

EG

Numerical grid line numbering is self explanatory. For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M…

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

7. Define the layout of radial grid segments that you want to achieve.

If you are only creating grid arcs the wizard skips this step.

You can define a regular or irregular radial line layout:

- for a regular layout you define the number of radial lines you want to create and the angle between them,
- for an irregular radial line layout you define the angle between successive pairs of radial lines, separating the numbers by commas. If you have a number of lines that are at the same centers, then you can specify these as a single entry.

*If you have angles of 30°, 45°, 30°, 30°, 30°, 30° and 45° you can specify this as 30,45,4*30,45.*

Again you can specify the reference of the first grid line, and how you want to increment the reference to give the references of the other radial lines that you create.

8. Finally define the rotation of the grid. You can do this graphically by moving the mouse over the 2D view and clicking, or by entering values into the wizard's dialog. If you use the latter approach you can specify the rotation either with respect to the grid system's local x or local y directions.

How do I create a single grid line between two points?

1. Make sure that the 2D view representing the construction level on which you want to create your grid line is active.

If you want to create a series of grid lines which form a regular or irregular, rectangular or radial grid system, the Tekla Structural Designer Rectagular and Sector Wizards give the speediest solutions.

2. Click **Model** > Grid Line , and select Grid Line

3. Pick the point where you want the grid line to start (Point 1).

The tooltip gives the cursor's coordinates exactly. If it has not snapped to an existing point you can press <F2> to enter the exact coordinates of the point required.

4. Pick the point where you want the grid line to end (Point 2).

The tooltip displays either the absolute, relative, or polar coordinates of Point 2 depending on whether the ABS, REL or POL button is highlighted in the **Status Bar** *at the bottom right of the screen. To switch the display simply click one of the other buttons.*

5. The grid line is created between the points you have picked.

The grid line does not extend to infinity. Please take care to ensure that the grid line is of sufficient length to meet your needs.

How do I create a parallel grid line?

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

1. Click **Model** > Grid Line, and select Parallel
2. Enter a name for the grid line in the Properties Window.
3. Select the grid line to which your new grid line is to be parallel.
4. You will see a dotted line which is parallel to the grid line you selected in step 3, and which follows the cursor.

> *The tooltip gives the distance of the dotted line from the initial grid line you selected in step 3. You can press <F2> to enter the exact distance if required.*
>
> *As you zoom further and further into the model the distance by which the cursor moves alters in smaller and smaller increments, so for more accuracy zoom in.*

5. Once you have achieved the distance you require:
   - click to locate your new grid line at this position
   - pick a point to define the end 1 extent
   - pick a second point to define the end 2 extent
6. The new grid line is created, you can now:
   - Move the cursor and continue placing other grid lines with respect to the line you picked in step 3.
   - Press <Esc> to end grid placement.

How do I create one or more parallel (quick) grid lines?

This option creates one or more grid lines parallel to, and the same length as, an existing one. If you want to use this option, then you must have at least one existing grid line in the current 2D-window.

1. Click **Model** > Grid Line, and select Parallel (quick)
2. Select the grid line to which your new grid lines will be parallel.
3. You will see a line which is parallel to the grid line you selected in step 2, and which follows the cursor.

The tooltip gives the distance of the new line from the initial grid line you selected in step 2.

As you zoom further and further into the model the distance by which the cursor moves alters in smaller and smaller increments, so for more accuracy zoom in.

You can define a single grid line, or a series of grid lines on a regular, or irregular layout:

- for a single grid line click to place it, or press <F2> to define it's offset exactly.
- for a series of grid lines press <F2> then define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

EG

If you have bay centres of 18′, 27′ , 18′ 6″ , 18′ 6″ , 18′ 6″ and 27″ you can specify this as 18′ , 27′ , 3x18′ 6″ , 27″

How do I create a perpendicular grid line?

If you want to use this option, then you must have at least one existing grid line in the current 2D-window.

1. Click **Model** > Grid Line, and select Perpendicular
2. Enter a name for the grid line in the Properties Window.
3. Select the grid line to which your new grid line is to be perpendicular.
4. You will see a dotted line overlying the grid line you selected in step 3, and which follows the cursor.

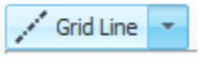
The tooltip gives the perpendicular distance to your new grid line from the middle of the grid line you selected in step 3.

As you zoom further and further into the model the distance by which the cursor moves alters in smaller and smaller increments, so for more accuracy zoom in.

5. Once you have achieved the distance you require:
   - click to locate your new grid line at this position,
   - pick a point to define the end 1 extent
   - pick a second point to define the end 2 extent.
6. The new grid line is created, you can now:
   - Move the cursor and continue placing other grid lines with respect to the line you picked in step 3.

- Press <Esc> to end grid placement.

How do I create a grid arc?

1. Click **Model** > 

, and select

2. Enter a name for the grid line in the Properties Window.

3. Select the point which lies at the center of the grid arc which you want to create.

The tooltip gives the cursor's coordinates exactly.

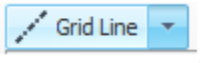
4. As you move the cursor you will see a line which rotates about the center defined in step 3. This end of this line marks the start of the grid arc you are creating. Once you have achieved the required location click to set this.

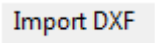
5. Now you will see two dotted lines, a radial line through the center defined in step 3, and an arc which indicates the sweep of the grid arc you are creating. Once you have achieved the required sweep click to create your new arc at this position.

How do I import grids from a .dxf, or import a .dxf as a shadow?

The .dxf file that you want to use must be available before you start the import process. You can either have been sent the file, or have created it yourself in some other application.

1. Open a 2D View of a construction level (to enable the **Grid Line** dropdown).

2. Click **Model** > 

, and select 

   You will see the Open dialog, navigate to the folder which contains the file you want to import, select the appropriate file name and click **Open**

3. You will now see the **Dxf Import Wizard**. This first page of the dialog allows you to control the layers and colors which you want to import (self-explanatory). Of more interest are the output options which allow you choose either Architectural Grids or Shadow.

   - If you import the Architectural Grids from the .dxf file, then all the elements in the selected layers of the .dxf file are mapped to *Tekla Structural Designer* grid lines. You must therefore ensure that you switch off all the layers in the dxf apart from the layers in which the grids have been defined.
   - If you import the .dxf as a Shadow, then *Tekla Structural Designer* imports the .dxf file but does not create any *Tekla Structural Designer* objects. You can then use the intersection points and such like as the source on which to add the *Tekla Structural Designer* objects you require.

4. If required you can adjust the scale and/or offsets for the dxf before proceeding.

5. Click **Next** to move to the second page of the dialog.

6. The **Architectural grids** options allow you to control how the grids are created.
   - Select **By layer** to have separate named grids for each layer imported from the dxf.
   - Select **By color** to have separate named grids for each color imported from the dxf.
   - Select **Merged** to have a single merged grid containing every layer/color imported from the dxf.
7. If you want to import the same grid layout to every construction level in your building ensure that each level is checked in the dialog. If you don't want the grid to be displayed at a particular level clear it.
8. Click **Finish** to close the wizard and complete the import.

How do I set the initial number or letter used for grids?

The initial number and letter to be used for grid numbering is specified in the Model Settings. New grids automatically use the next available number or letter depending on whether they are labeled numerically, or alphanumerically.

1. Click **Home** > **Model Settings** ( )
2. Use the References - General sub page to set the initial values and also to set the naming style.

How do I change the name or color of an existing architectural grid?

1. Expand Architectural Grids in the Structure Tree to display the existing grids.
2. Pick the architectural grid name to be changed from the list.
3. Use the Properties Window to see and / or amend the details of the architectural grid.

How do I change the name of an individual grid-line, -arc?

1. Select the grid line or -arc to be renamed.
2. Edit the **User name** for the grid line as shown in the Properties Window.

How do I renumber all grids?

1. Right-click Architectural Grids in the Structure Tree and choose **Renumber** from the menu.

   Every grid line in the model is renumbered in sequence.

How do I extend, move or rotate grid lines and arcs?

A grid line can be modified as follows:

- It can be stretched, shortened, or be rotated by moving one of its end nodes.
- It can also be moved in a perpendicular direction by moving its center node.

Similarly a grid arc can be modified as follows:

- It can be stretched or shortened by moving one of its end nodes.
- It can have its radius adjusted by moving the middle node on the arc perimeter.
- It can be moved in any direction by moving its center node.

Grid lines and arcs can only be moved in a 2D, not 3D view.

How do I stretch, shorten, or rotate a grid line?

1. In a 2D View, select only the grid line you want to modify.

   The two end nodes and the center node of the grid line should become visible.

2. Click to select one of the end nodes of the grid line.

3. Click where this node is to be moved to.

   The grid line moves to its new position accordingly.

How do I move a grid line in a perpendicular direction?

1. In a 2D View, select only the grid line you want to modify.

   The two end nodes and the center node of the grid line should become visible.

2. Click to select the center node of the grid line.

3. Click where this node is to be moved to.

   The grid line moves to its new position accordingly.

How do I stretch or shorten a grid arc?

1. In a 2D View, select only the arc you want to modify.

   The two end nodes and a middle node on the arc perimeter should become visible, along width a node at the center of the arc.

2. Click to select one of the end nodes.

3. Click where this node is to be moved to.

   The arc stretches or shortens accordingly.

How do I adjust the radius of a grid arc?

1. In a 2D View, select only the arc you want to modify.

   The two end nodes and a middle node on the arc perimeter should become visible, along width a node at the center of the arc.

2. Click to select the middle node on the perimeter.

3. Click where this node is to be moved to.

   The arc radius adjusts accordingly.

How do I move a grid arc?

1. In a 2D View, select only the arc you want to modify.

   The two end nodes and a middle node on the arc perimeter should become visible, along width a node at the center of the arc.

2. Click to select the center node of the arc.

3. Click where this node is to be moved to.

   The arc moves to its new position accordingly.

Construction Lines

Construction lines serve the same purpose as Architectural Grids and Grid Lines, only without displaying a grid bubble.

How do I create a rectangular construction line system?

1. Make sure that the 2D view representing the construction level on which you want to create your rectangular construction line system is active.

2. Click **Model** > Construction Line, and select Rectangular Wizard

3. Define the origin of the construction line system, either by clicking in the 2D view, or by entering the coordinates in the **Rectangular Grid Wizard** dialog.

4. Choose which construction lines you want to create, in the X direction only, in the Y direction only, or in both directions, and choose the line representation - dash dot, dot… that you want to use for these construction lines.

5. Define the layout of construction lines for the bays in the X direction.

> *If you are only creating construction lines in the X direction, then the wizard skips this step.*

You can define a regular or irregular construction line layout:

- for a regular grid you define the number of bays you want to create and the bay centers,
- for an irregular grid you define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

If you have bay centres of 18′, 27′ , 18′ 6″ , 18′ 6″ , 18′ 6″ and 27″ you can specify this as 18′ , 27′ , 3x18′ 6″ , 27″

6. Define the layout of construction lines for the bays in the Y direction.

If you are only creating construction lines in the Y direction, then the wizard skips this step.

7. Define the rotation of the grid. You can do this graphically by moving the mouse over the 2D view and clicking, or by entering values into the wizard's dialog. If you use the latter approach you can specify the rotation either with respect to the construction line system's local x or local y directions.

This is useful if you are going to create a construction line system which is not orthogonal.

8. Finally specify the angle between the construction line axes, you can do this with respect to either the X or Y axis system and click **Finish** to create your construction line layout. Again you can do this graphically, or you can use the wizards dialog - it's your choice.

How do I create a radial construction line system?

1. Make sure that the 2D view representing the construction level on which you want to create your radial construction line system is active.

2. Click **Model** > Construction Line, and select Sector Wizard

3. Define the origin of the grid system, either by clicking in the 2D view, or by entering the coordinates in the **Radial Grid Wizard** dialog.

4. Choose which construction lines you want to create, the radial lines only, the arcs only, or both of these, and choose the line representation - dash dot, dot… that you want to use for these construction lines.

5. Define the layout of the arcs that you will create.

If you are only creating radial construction lines the wizard skips this step.

You can define a regular or irregular layout:

- for a regular layout you define the number of arcs you want to create and the distance between them,
  - for an irregular layout you define the distance between successive pairs of arcs lines, separating the numbers by commas. If you have a number of arcs that are the same distance apart, then you can specify these as a single entry.

If you have arcs at distances of 9′, 12′ , 10′ 6″ , 10′ 6″ , 10′ 6″ and 12″ you can specify this as 9′ , 12′ , 3x10′ 6″ , 12″

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

6. Define the layout of radial construction line segments that you want to achieve.

If you are only creating arcs the wizard skips this step.

You can define a regular or irregular radial line layout:

- for a regular layout you define the number of radial lines you want to create and the angle between them,
  - for an irregular radial line layout you define the angle between successive pairs of radial lines, separating the numbers by commas. If you have a number of lines that are at the same centers, then you can specify these as a single entry.

*If you have angles of 30°, 45°, 30°, 30°, 30°, 30° and 45° you can specify this as 30,45,4*30,45.*

7. Finally define the rotation of the layout. You can do this graphically by moving the mouse over the 2D view and clicking, or by entering values into the wizard's dialog. If you use the latter approach you can specify the rotation either with respect to the construction line system's local x or local y directions.

How do I create a single construction line between two points?

If you want to create a series of construction lines which form a regular or irregular, rectangular or radial system, the Tekla Structural Designer wizards give the speediest solutions.

1. Make sure that the 2D view representing the construction level on which you want to create your construction line is active.

2. Click **Model** > Construction Line

3. Pick the point where you want the construction line to start (Point 1).

> *The tooltip gives the cursor's coordinates exactly. If it has not snapped to an existing point you can press <F2> to enter the exact coordinates of the point required.*

4. Pick the point where you want the construction line to end (Point 2).

> *The tooltip displays either the absolute, relative, or polar coordinates of Point 2 depending on whether the ABS, REL or POL button is highlighted in the* **Status Bar** *at the bottom right of the screen. To switch the display simply click one of the other buttons.*

5. The construction line is created between the points you have picked.

> *The grid line does not extend to infinity. Please take care to ensure that the grid line is of sufficient length to meet your needs.*

How do I create a parallel construction line?

This option creates a construction line parallel to an existing line, but of a different length. If you want to use this option, then you must have at least one existing grid or construction line in the current 2D-window.

1. Click **Model** > Construction Line, and select Parallel

2. Select the line to which your new construction line is to be parallel.

3. You will see a dotted line which is parallel to the line you selected in step 3, and which follows the cursor.

> *The tooltip gives the distance of the dotted line from the initial line you selected in step 3. You can press <F2> to enter the exact distance if required.*
>
> *As you zoom further and further into the model the distance by which the cursor moves alters in smaller and smaller increments, so for more accuracy zoom in.*

4. Once you have achieved the distance you require:

- click to locate your new construction line at this position
- pick a point to define the end 1 extent
- pick a second point to define the end 2 extent

5. The new construction line is created, you can now:
   - Move the cursor and continue placing other construction lines with respect to the line you picked in step 3.
   - Press <Esc> to end construction line placement.

How do I create one or more parallel (quick) construction lines?

This option creates one or more construction lines parallel to, and the same length as, an existing line. If you want to use this option, then you must have at least one existing grid or construction in the current 2D-window.

1. Click **Model** > Construction Line, and select Parallel (quick)
2. Select the line to which your new construction lines will be parallel.
3. You will see a line which is parallel to the line you selected in step 3, and which follows the cursor.

The tooltip gives the distance of the new line from the initial line you selected in step 3.

As you zoom further and further into the model the distance by which the cursor moves alters in smaller and smaller increments, so for more accuracy zoom in.

You can define a single construction line, or a series of lines on a regular, or irregular layout:

- for a single construction line click to place it, or press <F2> to define it's offset exactly.
- for a series of lines press <F2> then define the distance between successive pairs, separating the numbers by commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

EG

If you have bay centres of 18′, 27′, 18′ 6″, 18′ 6″, 18′ 6″ and 27″ you can specify this as 18′, 27′, 3x18′ 6″, 27″

How do I create a perpendicular construction line?

If you want to use this option, then you must have at least one existing grid or construction in the current 2D-window.

1. Click **Model** > Construction Line, and select Perpendicular
2. Select the line to which your new construction line is to be perpendicular.
3. You will see a dotted line overlying the line you selected in step 3, and which follows the cursor.

The tooltip gives the perpendicular distance to your new line from the middle of the line you selected in step 3.

As you zoom further and further into the model the distance by which the cursor moves alters in smaller and smaller increments, so for more accuracy zoom in.

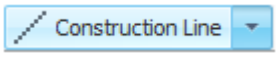
4. Once you have achieved the distance you require:

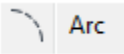
- click to locate your new construction line at this position,
- pick a point to define the end 1 extent
- pick a second point to define the end 2 extent.

5. The new construction line is created, you can now:

- Move the cursor and continue placing other construction lines with respect to the line you picked in step 3.
- Press <Esc> to end construction line placement.

How do I create a construction line arc?

1. Click **Model** > 

, and select 

2. Select the point which lies at the center of the arc which you want to create.

The tooltip gives the cursor's coordinates exactly.

3. As you move the cursor you will see a line which rotates about the center defined in step 3. This end of this line marks the start of the arc you are creating. Once you have achieved the required location click to set this.

4. Now you will see two dotted lines, a radial line through the center defined in step 3, and an arc which indicates the sweep of the arc you are creating. Once you have achieved the required sweep click to create your new arc at this position.

How do I extend, move or rotate construction lines and arcs?

A construction line can be modified as follows:

- It can be stretched, shortened, or be rotated by moving one of its end nodes.
- It can also be moved in a perpendicular direction by moving its center node.

Similarly a construction arc can be modified as follows:

- It can be stretched or shortened by moving one of its end nodes.
- It can have its radius adjusted by moving the middle node on the arc perimeter.
- It can be moved in any direction by moving its center node.

Construction lines and arcs can only be moved in a 2D, not 3D view.

How do I stretch, shorten, or rotate a construction line?

1. In a 2D View, select only the construction line you want to modify.

   The two end nodes and the center node of the construction line should become visible.

2. Click to select one of the end nodes of the construction line.

3. Click where this node is to be moved to.

   The construction line moves to its new position accordingly.

How do I move a construction line in a perpendicular direction?

1. In a 2D View, select only the construction line you want to modify.

   The two end nodes and the center node of the construction line should become visible.

2. Click to select the center node of the construction line.

3. Click where this node is to be moved to.

   The construction line moves to its new position accordingly.

How do I stretch or shorten a construction arc?

1. In a 2D View, select only the arc you want to modify.

   The two end nodes and a middle node on the arc perimeter should become visible, along width a node at the center of the arc.

2. Click to select one of the end nodes.

3. Click where this node is to be moved to.

   The arc stretches or shortens accordingly.

How do I adjust the radius of a construction arc?

1. In a 2D View, select only the arc you want to modify.

   The two end nodes and a middle node on the arc perimeter should become visible, along width a node at the center of the arc.

2. Click to select the middle node on the perimeter.

3. Click where this node is to be moved to.

   The arc radius adjusts accordingly.

How do I move a construction arc?

1. In a 2D View, select only the arc you want to modify.

   The two end nodes and a middle node on the arc perimeter should become visible, along width a node at the center of the arc.

2. Click to select the center node of the arc.

3. Click where this node is to be moved to.

   The arc moves to its new position accordingly.

Dimensions

Dimensions allow you to show distances in your structure between appropriate points. The dimension lines are included on any drawings you create.

How do I create a single dimension?

1. Click **Model** > **Dimension** ( )

2. Click the grid point at the start of the dimension.

3. Click the grid point at the end of the dimension. *Tekla Structural Designer* will show a line between these two points.

4. Move this line to the point where you want the dimension line to lie (somewhere where it will be easily visible, and will not conflict with the rest of your model's details). *Tekla Structural Designer* shows the dimension line itself, and the two locator lines at its ends. Simply move the dimension line to the correct location and click to create it.

Steel, Cold Rolled, and Cold Formed Member modeling

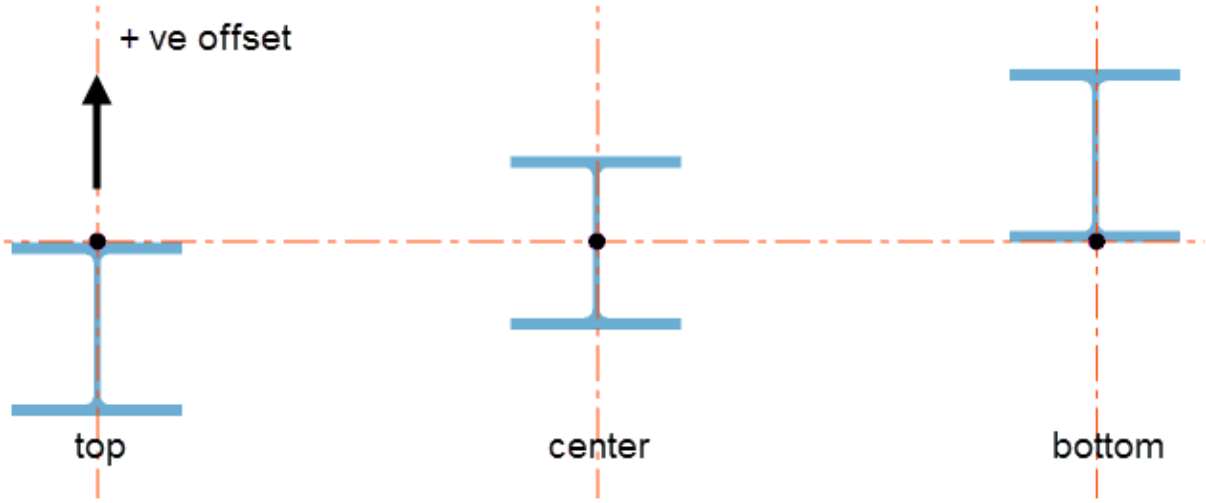
Modeling Steel Columns and Cold Formed Columns

Steel column overview

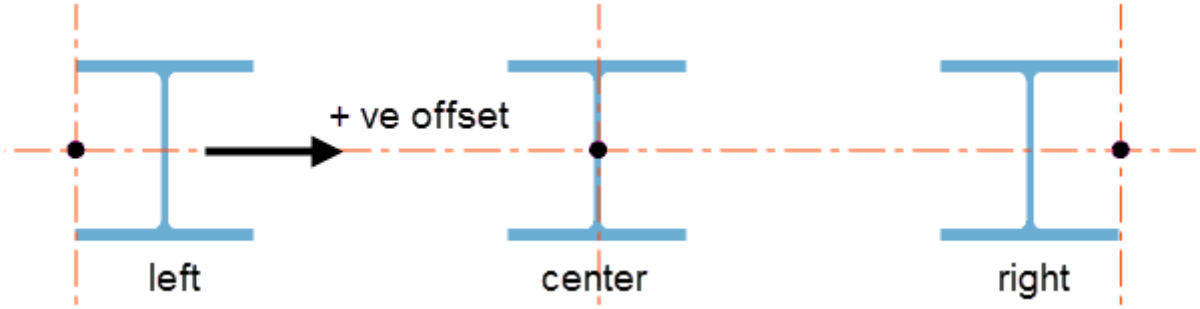
A single steel column can be created over several story heights (stacks) and can start and finish at any level. Different column sections can be defined in each stack, (provided a splice is defined at the change point).

Each steel column stack is placed on an insertion line between points, but its geometry is drawn to reflect the major and minor snap points (and any offsets) specified in the column properties.

Major snap and offset



Minor snap and offset



Steel column alignment snap points and offsets have no effect on the positioning of the 1D solver elements in the solver models - the solver elements are always inserted along the steel column insertion lines. This is different to the approach adopted for concrete column solver elements (in which the alignment snap points and offsets are structurally significant).

Parapet post overview

Steel parapet posts are single span members with fixed end connections.

Their specific purpose is to act as a means to transfer load from wind wall panels into columns - the decomposed load from the panel being applied as a point load and moment at the node connecting the parapet post to the column.

Parapet posts are not designed in Tekla Structural Designer.

How do I specify the column type and section size?

To specify the type of steel column:

1. Click **Model** > Steel Column drop list.
2. Select the type ("steel", "plated", "concrete filled", or "encased concrete") from the drop list.

Alternatively, to specify a cold formed column:

1. Click **Model** > **Column** ( ) (in the **Cold Formed** group).

To specify the column size:

1. In the **Section** box, click **<New\Edit...>**.
2. Pick the new section from the Select Section dialog, then click **Select**.

Before proceeding to create the column, check the remaining properties displayed in the steel column properties set and adjust as required.

How do I create a single column in a 2D View?

1. Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie.
2. Select the column type and size.
   (How do I specify the column type and section size?)
3. Check that the Base Level and the Top Level displayed in the Properties Window are correct, or adjust if necessary.
4. Click the point where the column is to be placed.
   (What are the points I can click to create a member?)

How do I create a series of columns in a 2D View?

1. Ensure that you have defined the construction levels between which the columns will run and the grid points between which they will lie.
2. Select the column type and size.
   (How do I specify the column type and section size?).
3. Check that the Base Level and the Top Level displayed in the Properties Window are correct, or adjust if necessary.

4. Move the cursor to one corner of an imaginary box which will encompass the grid intersection points at which you want to create columns.

5. Click and hold the left mouse button.

6. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating).

7. Once the rubber rectangle encompasses the grid intersection points at which you want to create columns, release the mouse button.

How do I create a single column in a Frame, or Structure View?

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

To create the column:

1. Select the column type and size.
(How do I specify the column type and section size?)

2. Click the point where the column is to start.
(What are the points I can click to create a member?)

3. Click the point where the column is to end. *Tekla Structural Designer* creates the column between these two points.

How do I align a column to a specific angle, or an angled gridline?

Three options are provided for specifying the alignment of a new column - achieved by setting the **Rotation** property as follows:

- **0, 90, 180, 90** - aligns the column to the global axes
- **Angle** - aligns the column to the exact rotation angle you specify
- **Define** - aligns the column to the angle of any grid line you select

How do I create an inclined column?

An inclined column can only be created in a Frame, or Structure View.

In order to define the column you must have defined the construction levels between which the column will run and the grid points between which it will lie.

1. Select the column type and size.
(How do I specify the column type and section size?)

2. Click the point where the column is to start.
(What are the points I can click to create a member?)

3. Click the point where the column is to end. *Tekla Structural Designer* creates the column between these two points.

Related topics

- How do I create a cranked column?

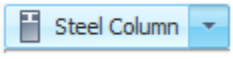
How do I create a cranked column?

A cranked column can only be created in a Frame, or Structure View.

In order to define the column you must have defined the construction levels between which the column will run and the grid points between which it will lie.

1. Select the column type and size.
   (How do I specify the column type and section size?).
2. Click the point where the column is to start.
3. Press and hold the Ctrl key on the keyboard and click the next node of your cranked column.
4. Repeat step 3 until you have defined the penultimate node of your cranked column. Release the Ctrl key and click the point where the column is to end.

How do I create a plated or compound section steel column?

1. Click **Model** > 

, and select **Plated**.
2. In the **Section** box, click **<New/Edit...>**.
3. In the Select Section dialog, choose the required compound section type from the left hand pane.
4. Select the required section from the list of sections that appear for the selected type, or if the section is not listed click Add...
   - Enter sections, plate dimensions and gaps as required to fully define the section
   - Either accept the generated name or rename it
   - Click OK to add the section
   - Once the new section has been added to the list it can then be selected.

The section is automatically added to the Materials database so that it doesn't have to be redefined when required in subsequent models.

5. Review the remaining properties in the properties window and adjust as necessary.

How you then proceed to place the column depends on whether you are in a 2D View, or a Frame/Structure View.

To apply a plated or compound section to an existing steel column

In the column properties:

1. In the **Fabrication** box, click **Plated**.

2. From the Section droplist, either select a compound section that has already been applied in the model, or choose <New/Edit...> to select a new one.

How do I create a concrete filled, or encased concrete steel column?

Concrete filled and concrete encased steel columns can be analysed, but cannot currently be designed in Tekla Structural Designer.

1. Click **Model** > Steel Column, and select **Concrete Filled**, or **Concrete Encased**.
2. The column will adopt its properties from those currently displayed in the steel column properties set. Review the properties and adjust as necessary.

How you then proceed to place the column depends on whether you are in a 2D View, or a Frame/Structure View.

How do I create a gable post, or parapet post?

1. Click **Model** > **Steel Column** ( )

   (or any of the other element types - it does not actually matter which.)

2. In the **Characteristic** box, click **Gable post**, or **Parapet post** as required.

   The properties displayed are updated appropriate to the type of post chosen.

3. Before proceeding to create the post, check the remaining properties in the gable post properties set or parapet post properties set and adjust as necessary.

For gable posts, consider in particular:
- if an axial load release at the top is required (set under the Releases heading)
- what wind load deflection parameters are required (set under the Deflection limits heading)

4. Click where the member is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the member is to end (Point 2).

How do I specify a column splice?

Splices can be added at the base of each column stack (apart from stack 1) as required. The splice offset is then used to locate each splice at a practical distance above the floor level.

To add a splice:

1. Select the columns.
2. The properties are displayed in the edit steel column properties set - expand the properties for the stack within which the splice is required.
3. Select **Splice**.
4. A splice can be added at the base of each stack (apart from stack 1) as required. The splice offset can be set to locate the splice at a practical distance above the floor level.

You should notice that the **Section** property, (which was previously dimmed) is now editable, allowing you to specify a different section size above the space position.

How do I modify the position of a single column stack?

A column stack's position can best be modified in either a Frame, or Structure View.

1. In Scene Content, ensure that **Grid & Construction Lines** is selected.
2. Select the column for which a stack is to be moved.

   The column is highlighted in one color, the column end nodes and middle node in another.

3. Select one end nodes for the column stack to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)
4. Click a grid or construction point to redefine the column stack end node position.

   The entire column is redrawn with the selected node moved to the new position.

5. Select the next end node. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)
6. Click a grid or construction point to redefine the end node position.

   The column is redrawn once more with the selected node moved to the new position.

How do I modify the position of an entire column?

Simply use the **Move** command located on the Edit ribbon.

Modeling Steel Beams and Cold Formed Beams

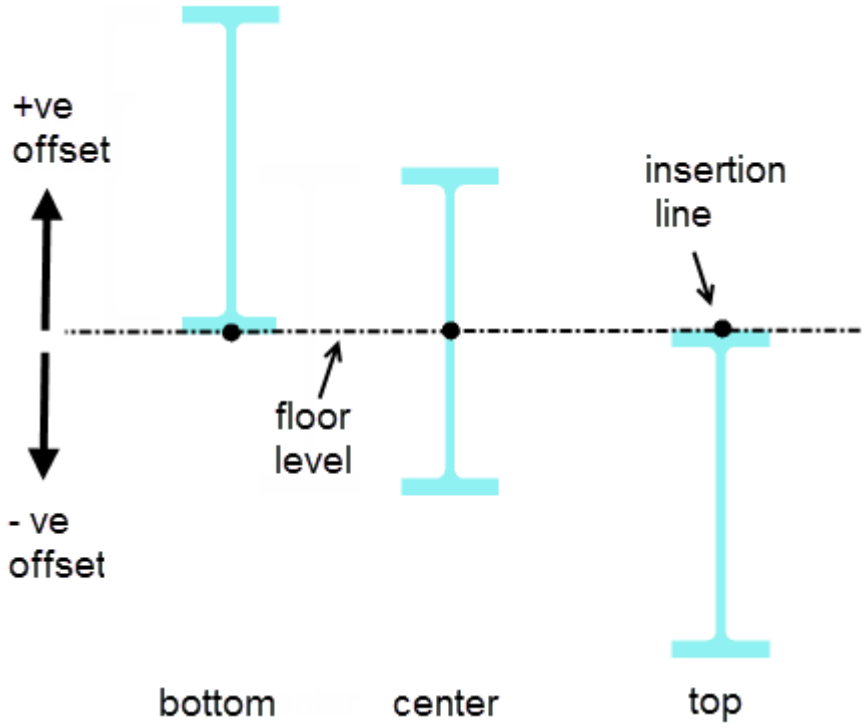
Steel beam overview

Steel beam members can be defined as single span, or continuous over multiple spans. If continuous, although a single beam member is created, this does not prohibit different beam sections from being defined in each span.

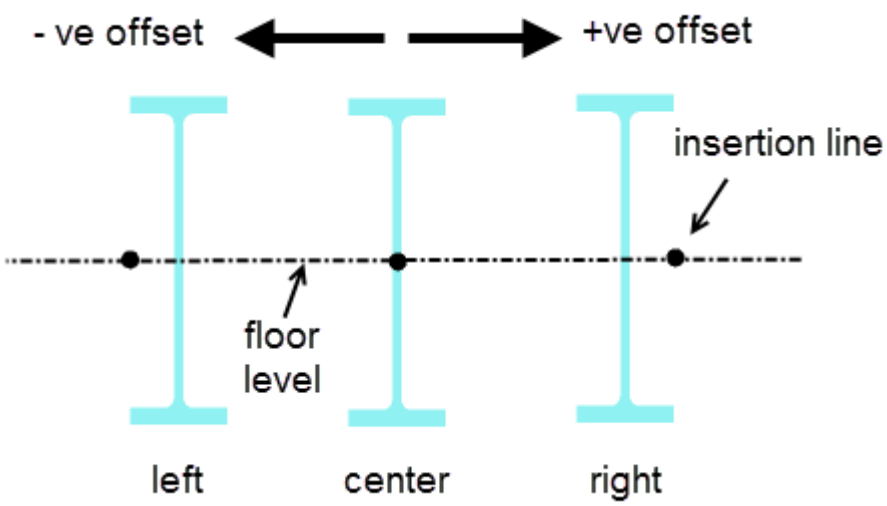
Each steel beam span is placed on an insertion line between points, but its geometry is drawn to reflect:

- the major and minor snap points (and any offsets) specified in the beam properties.
- the level "type" specified in the construction level dialog

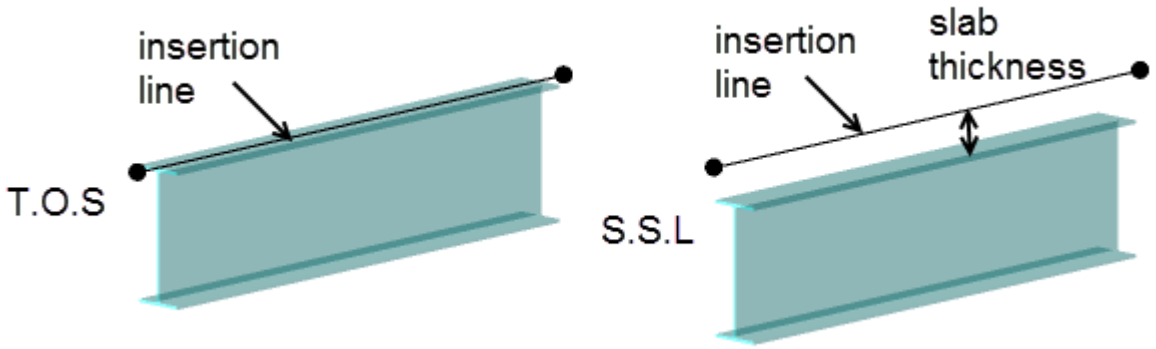
Major snap and offset



Minor snap and offset



Construction level "type"



- When the level type is set to **T.O.S** (top of steel), each beam is displayed according to the alignment snap points and offsets specified.
- When the level type is set to **S.S.L** (structural slab level), each beam is in addition lowered by the slab thickness specified in the construction level dialog.

Steel beam alignment snap points and offsets and the construction level type have no effect on the positioning of the 1D solver elements in the solver models. This is a different approach to that adopted for concrete beam solver elements.

How do I specify the beam type and section size?

To specify the type of steel beam:

1. Click **Model** > Steel Beam drop list.
2. Select the type ("Steel", "Plated", "Westok Cellular", "Westok Plated", or "Fabsec") from the drop list.

Alternatively, to specify a cold formed beam:

1. Click **Model** > **Beam** ( ) (in the **Cold Formed** group).

To specify the beam size:

1. In the **Section** box, click **<New\Edit...>**.
2. Pick the new section from the Select Section dialog, then click **Select.**

Before proceeding to create the beam, check the remaining properties in the steel beam properties set and adjust as required.

How do I create a single span beam?

1. Select the beam type and size.
   (How do I specify the beam type and section size?)
2. Click where the beam is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

3. Click where the beam is to end (Point 2).

If you are prompted to pick another point (Point 3), this will be due to **Continuous** *being selected in the beam property set. In order to create a single span beam simply click once again on Point 2, or press the Enter key.*

How do I create a plated or compound section steel beam?

1. Click the **Model** > Steel Beam, and select **Plated**.

2. In the **Section** box, click **<New/Edit...>**.

3. In the Select Section dialog, choose the required compound section type from the left hand pane.

4. Select the required section from the list of sections that appear for the selected type, or if the section is not listed click Add...

- Enter sections, plate dimensions and gaps as required to fully define the section
- Either accept the generated name or rename it
- Click OK to add the section
- Once the new section has been added to the list it can then be selected.

The section is automatically added to the Materials database so that it doesn't have to be redefined when required in subsequent models.

5. Review the remaining properties in the properties window and adjust as necessary.

How you then proceed to place the beam depends on whether you are in a 2D View, or a Frame/Structure View.

To apply a plated or compound section to an existing steel beam

In the beam properties:

1. In the **Fabrication** box, click **Plated**.

2. In the **Section** box, either click an existing compound section, or click **<New/Edit...>** to select a new one.

How do I create a Westok Cellular, Westok Plated, or Fabsec beam?

1. Click **Model** > **Steel Beam** (drop list).
2. Select the beam type required from the drop list.
3. The beam will adopt its properties from those currently displayed in the steel beam properties set. Review the property set and adjust as necessary.
4. Click where the beam is to start (Point 1).
   (What are the points I can click to create a member?)

> If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the beam is to end (Point 2).

> If you are prompted to pick another point (Point 3), this will be due to **Continuous** being selected in the beam property set. In order to create a single span beam simply click once again on Point 2, or press the Enter key.

Related topics

- How do I create a single span beam?
- How do I create a series of single span beams?
- How do I create a plated or compound section steel beam?
- How do I create a continuous beam?
- How do I create a curved beam?

How do I create a series of single span beams?

In order to create beams using this method the floor or construction level must already contain the columns between which the beams will run. You must also use a 2D view of the floor or construction level to use this option.

1. Select the beam type and size.
   (How do I specify the beam type and section size?).
2. Move the cursor to one corner of an imaginary box which will encompass the columns between which you want to create beams.
3. Click and hold the left mouse button.
4. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating).
5. Once the rubber rectangle encompasses the columns between which you want beams creating, release the mouse button.

6. Beams are created between each adjacent pair of columns within the area you select.

How do I create a continuous beam?

1. Select the beam type and size.
   (How do I specify the beam type and section size?)

2. In the Properties Window**,** ensure that the **Continuous** option is selected and adjust any other details as necessary.

3. Click where the beam is to start.
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

4. Click the next node of your continuous beam.

5. Repeat step 4 until you have defined the penultimate node of your continuous beam.

6. Click where the beam is to end.

7. You are now prompted to pick another point, simply click once again on the same point, or press the Enter key in order to create the continuous beam.

You cannot define continuous beams which are curved either horizontally or vertically.

How do I add a haunch to an existing beam?

1. Hover the cursor over the beam in which you want to add haunches so that it becomes highlighted.

2. Right-click and select Edit... from the dropdown menu.

3. Click the **Haunches** page of the properties dialog

4. Select **Quick layout**.

5. Click the **Create** button.

6. Define the haunch and end plate geometry as required, then click OK.

Although haunches are considered in the analysis, they are not considered in the beam design.

Caution: In the current release, you should avoid connecting any other member into the haunched portion of a beam.

How do I create a curved beam?

1. Select the beam type and size.
   (How do I specify the beam type and section size?).

2. In the Properties Window**,** ensure that the **Linearity** is set to **Curved Major** (if it is to curve vertically), or **Curved Minor** (if it is to curve horizontally), and an appropriate **Chord height** value is specified to define the curve.

You can control the direction in which horizontally curved beams curve. When you place the beam you select its start point and its end point. The beam always curves such that if you were to look along the theoretical line from the start point to the end point, then the curve on the beam will always lie to the right of that line.

For vertically curved beams, a negative chord height value can be used to reverse the curve direction.

3. Adjust any other details in the property set as necessary.

4. Click where the beam is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the beam is to end (Point 2).

When defining beams which curve vertically, if you are using a 2D view of the floor or construction level, then the vertical projection of the beam is out of the plane of the floor or construction level that you are using. In this case you will not see the beam on the graphical display. You will need to change to a 3D view of the floor, construction level or the entire Structure in order to see them.

Horizontally curved beams always take the chord height defined in the property set, they do not curve automatically to fit on any curved grid line that you may have defined.

How do I modify the position of a beam?

You can modify an individual beam's position in both 2D and 3D views.

1. Select the beam to be moved.

   The beam is highlighted in one color, the beam end nodes and middle node in another.

2. Now select the beam end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

3. Click a grid or construction point to redefine the beam end node position.

   The beam is redrawn spanning to the new end position.

Modeling Steel Braces and Cold Formed Braces

Steel brace overview

Tekla Structural Designer allows you to define braces to provide lateral stability to your structure. Alternatively you can specify rigid frames to achieve the same effect. You can also use both methods within a single structure should this be necessary.

Steel braces are single span members with pinned end connections, that are only able to resist axial compression and tension.

Each steel brace is placed on an insertion line between points, with its geometry being drawn to reflect the major and minor snap points (and any offsets) specified in the brace properties.

The alignment snap points and offsets have no effect on the positioning of the solver elements created in the solver models, as these will be created directly between the insertion points.

Element loads cannot be applied directly to the brace itself and moments due to self weight loading are ignored.

As well as single braces, you are also able to define pairs of braces to form X, K, V and A type braces.

These brace pairs can be defined in any vertical, horizontal or sloped plane within bays formed by the intersections of column and beams. Each brace in the pair has independent properties.

A vertical load release can be applied to the end of a V or A type brace pair so that they don't prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).

V and A brace pairs in models imported from Revit are only recognised as a pair if they have already been released vertically in Revit. If they have not, they will be imported as two single braces without a vertical release.

The split command is available to split pairs into individual braces.

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.

Inactive steel braces

Individual steel braces can be made inactive in the solver model by clearing **Active** in the brace properties.

Tension only and compression only braces

Once a steel brace has been placed its properties can be edited if required to specify that it is Tension only or Compression only.

Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements.

How do I specify the brace type and section size?

To specify the type of steel brace:

1. Click **Model** > Steel Brace drop list.
2. Select the type (**Steel Brace**, **X**, **K**, **V**, or **A**) from the drop list.

Alternatively, to specify a cold formed brace:

1. Click **Model** > **Brace** ( ) (in the **Cold Formed** group).

To specify the brace size:

1. In the **Section** box, click **<New\Edit...>**.
2. Pick the new section from the Select Section dialog, then click **Select.**

Before proceeding to create the brace, check the remaining properties in the steel brace properties set and adjust as required.

How do I create a single brace?

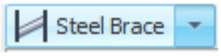
1. Select the brace type and size.
   (How do I specify the brace type and section size?)
2. Click where the brace is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

3. Click where the brace is to end (Point 2).

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 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 EHF calculations with the result that the calculated EHFs are not correct.

How do I create an X, K, V or A brace?

1. Click **Model**, then from the 

drop list select the pattern required (X, K, V, or A).

2. The braces will adopt their properties from the currently displayed steel beam properties set. Review the properties and adjust as necessary.

3. Click to identify the bottom corner of the bay to be braced, (Point 1).

4. Click to identify the opposite bottom corner of the bay to be braced, (Point 2).

5. Click to identify the top corner of the bay to be braced, (Point 3). You will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the orientation is correct.

6. Click to identify the opposite top corner of the bay to be braced, (Point 4).

7. The brace pattern is created within the area you have selected.

How do I modify the position of a brace?

You can modify an individual brace's position in both 2D and 3D views.

1. Select the brace to be moved.

The brace is highlighted in one color, the brace end nodes in another.

2. Now select the end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

3. Click a grid or construction point to redefine the brace end node position.

The brace is redrawn spanning to the new end position.

Modeling Steel Joists

Steel joists are a specific type of member used in the United States. They are constrained to standard types specified by the US Steel Joist Institute. They are standardized in terms of span, depth and load carrying capacity.

Steel joists (or bar joists) are simply supported secondary members, which do not support any other members - they only support loaded areas.

- Steel joists can be defined with ends at differing levels.
- They cannot support any other member.
- Slab and roof loads are supported by steel joists and loads are distributed to them.

How do I specify the joist type and size?

To specify the type of joist:

1. Click **Model** > **Steel Joist**
2. In the **Section** box, click **<New\Edit...>**.
3. Pick the series and then the size required from the Select Section dialog, then click **Select**

Before proceeding to create the joist check the remaining steel joist properties set in the Properties Window and adjust as required.

How do I create a steel joist?

1. Select the joist type and size.
   (How do I specify the joist type and size?)
2. Click where the joist is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

3. Click where the joist is to end (Point 2).

How do I modify the position of a steel joist?

You can modify the joist position in both 2D and 3D views.

1. Select the joist to be moved.

   The joist is highlighted in one color, the joist end nodes in another.

2. Now select the end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

3. Click a grid or construction point to redefine the joist end node position.

   The joist is redrawn spanning to the new end position.

Modeling Steel Trusses

Trusses are particular arrangements of members which *Tekla Structural Designer* calculates automatically for you. Once you have created a truss you can copy this throughout your model as necessary, and you can pick a truss and move it to the location you require.

Once you have defined and loaded your trusses *Tekla Structural Designer* automatically checks the member sizes you have specified to determine their adequacy.

How do I use the Steel Truss Wizard?

1. Click **Model** > **Steel Truss** ( )

2. Choose the truss shape from the list of standard truss patterns.

3. Click the truss Start Point.
   (What are the points I can click to create a member?)

> *If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.*

4. Click the truss End Point, then click **Next**

5. Specify the truss alignment parameters, then click **Next**

6. Specify the truss details, then click **Finish**

How do I use the Space Truss Wizard?

1. Click **Model** > Steel Truss drop list.

2. Select **Space** from the drop list.

3. Choose either a linear or planar truss type and then specify the alignment and number of bays, then click **Next**

4. In the 2D or 3D View, click the truss location points:
   - for a linear truss - click the start and end points
   - for a planar truss - click the four corners,
5. Specify the height, then click **Next**
6. Define either a straight or curved truss geometry then click **Finish**
7. Specify the truss alignment parameters, then click **Next**
8. Specify the truss details, then click **Finish**

How do I define a Free Form Truss?

A free form truss can be created by placing a series of truss members in the shape required.

1. Open a 2D Frame view in which the truss is to be created. (Note that free form trusses cannot be created in a 3D view, or 2D level view).
2. Click **Model** > Steel Truss drop list.
3. Select **Free Form** from the drop list.
4. Click where the first truss member is to start.
5. Click where the first truss member is to end.
6. Continue in the same way to pace each truss member until the truss geometry is complete.
7. Press **<Esc>** to finish.

How do I edit the geometry of an existing steel truss?

1. Hover the cursor over the truss to be edited so that it becomes highlighted.
2. Right-click and select Edit... from the dropdown menu.

   The **Truss Wizard** is displayed.
3. Edit the properties as required and then click **OK**

How do I edit the section sizes, material grades and section orientations in an existing steel truss?

1. Hover the cursor over the truss to be edited so that it becomes highlighted.
2. Left click the truss to display the edit steel truss properties set. Review the properties, adjust as necessary and then click **OK**

Modeling Portal Frames

How do I create a single span portal frame?

1. Click **Model** > **Portal Frame**

2. Click the grid point for the first column base.
   (This also set the local X and Y co-ordinate origin for the frame.)

3. Click the grid point for the second column base.
   (This also defines the positive local X axis direction for the frame.)

The second grid point must lie in the same construction level as the first grid point.

The Portal Frame dialog is displayed allowing you to specify the geometry and section property details for all the elements within the frame.

Initially the local co-ordinates of the Lh and Rh Eaves are invalid (as denoted by the red crosses), this is because the eaves height has yet to be defined.

4. Click **Edit...** to specify the eaves level, choose the frame type required, and complete the frame geometry for the first span.

5. Once the span geometry has been defined, click OK to return to the **Portal Frame** dialog.

6. Click **Rafters** to review the rafter section sizes.

7. Click **Columns** to review the column section sizes.

8. Click **Haunches** to specify any haunches.

9. Click **Bases** to specify the base fixity as either **Pinned** or **Fixed** and if necessary adjust the base levels.

Nominally Pinned and Nominally Fixed bases can also be specified, but only by editing individual support properties in the Properties Window after closing the Portal Frame dialog.
When partially fixed bases have been defined, their fixity type is subsequently displayed as "User" in the Portal Frame dialog. See: Partial fixity of column bases

---

10. If required click the other pages of the **Portal Frame** dialog in order to specify where required any:

- Valleys
- Ties
- Tie Members
- Parapets
- Parapet Members

The horizontal, vertical and rotational stiffnesses that can be defined on the Valleys page for a valley beam support are only ever used when the frame is exported to the Portal Frame Designer program for elastic plastic analysis and design. Because the frame is exported in 2D, the valley beam (which is out of plane) is replaced by a spring support with these stiffnesses.
In Tekla Structural Designer, as there is no spring support at the valley, these stiffnesses have no effect on the analysis or design.

11. Click **Spans** to add extra spans into the frame.
See: How do I create a multi-span portal frame?

How do I create a multi-span portal frame?

1. Start by defining a single span frame.
See: How do I create a single span portal frame?

2. Then, from the **Spans** page of the Portal Frame dialog use the buttons to introduce extra spans as required:

- Click **Add** to introduce a new span at the end of the frame.
- Click **Insert** to insert a new span below the currently highlighted span.
- Click **Copy** to copy the currently highlighted span to a target span. (This can only be used to replace a pre-existing span with the copied span, it does not insert a new span).
- Click **Mirror** to mirror the currently highlighted span to a target span. (This can only be used to replace a pre-existing span with the mirrored span, it does not insert a new span).

How do I edit the properties of an existing portal frame?

To edit the overall frame properties:

1. Hover the cursor over the portal frame to be edited so that it becomes highlighted.

2. Right-click and select Edit... from the context menu.

The Portal Frame dialog is displayed.

3. Edit the properties as required and then click **OK**

To edit the properties of an individual portal frame member:

1. Hover the cursor over the portal frame member to be edited.

2. Use the down arrow key to scroll through the objects listed in the Select Entity tooltip.

3. When the individual portal frame member becomes highlighted (as opposed to the entire portal frame), right-click and select the option to edit the individual member from the context menu.

The **Properties Dialog** for the member is displayed.

4. Edit the properties as required and then click **OK**

Modeling Cold Rolled Sections

The following characteristic types of cold rolled section can be specified in Tekla Structural Designer: Track, Stud, Joist, Purlin, Rail, Eaves Beam.

Cold rolled sections can be modelled and analysed in Tekla Structural Designer but they are not designed.

How do I create a single cold rolled section?

1. Click **Model** > **Steel Beam** (Steel Beam)

   (or any of the other element types - it does not actually matter which.)

2. Change the **Characteristic** in the property set to the type of cold rolled section you require.

   The properties displayed are updated appropriate to the chosen cold rolled section type.

3. Review the Cold Rolled Properties and adjust as necessary.

4. Click where the member is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the member is to end (Point 2).

How do I modify the position of a cold rolled section?

You can modify a cold rolled section in both 2D and 3D views.

1. Click the section you want to modify. You will see two handles at its ends. To move the end of the section, click its handle and move it to its new location with the mouse. When the end is where you want it click again to fix the end to that point.

Modeling Web Openings

Although web openings can be added to steel beams and columns they are only considered in the design of composite and non composite beams designed to either Eurocodes, BS codes, or AISC

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

'Quick-layout' creates web openings at the maximum depth and spaced at the minimum centers recommended for the section size while also meeting certain geometric and proximity recommendations.

Web openings can be defined manually in two ways from the Web Openings dialog page. 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, or alternatively, use of the **Add** button opens the Web Opening Details dialog page which gives access to more help and guidance when defining the opening. Both methods make use of 'Warning' and 'Invalid' text for data entry checks to provide assistance as the opening parameters are defined.

On the Web Opening Details dialog page, the Center button will position the opening on the beam centre whilst the Auto button will position the opening to meet the spacing recommendations. Also on this page tool tips give information on the recommended values for all the opening parameters.

As web openings are defined, they are immediately visible in the diagram on the Web Openings dialog page. This diagram displays the results of the geometric and proximity checks that are carried out on the opening parameters using 'Warning' and 'Invalid' display colours to highlight those areas that are outside the recommended limits.

The areas that are subjected to the checks are end posts, web posts, web opening dimensions and tee dimensions.

This display helps you to decide whether to make any adjustments to the opening parameters before their design is checked.

You should bear in mind that the checks carried out at this stage are geometric checks only and compliance with recommended limits is no guarantee that the opening will pass the subsequent engineering design checks.

Related topics

- Web Openings to AISC
- Web Openings to SCI P355
- Web Openings to SCI P068

How do I add web openings using Quick Layout?

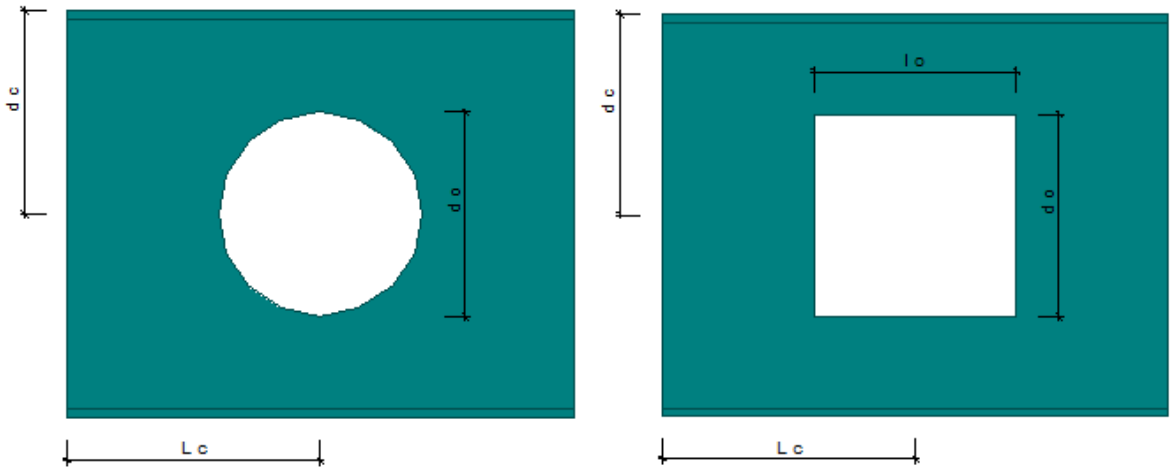
This method enables you to create maximum depth openings spaced at the minimum centers appropriate to the section size.

1. Hover the cursor over the member in which you want to add the web openings so that it becomes highlighted.
2. Right-click and select Edit... from the dropdown menu.
3. Ensure **Automatic design** in the properties is cleared.
   (Web openings cannot be added to members that are in Autodesign mode.)
4. Click the **Web openings** page of the properties dialog
5. Select **Quick layout**.
6. Select where you want to start setting out from via the **Label openings from** droplist.

7. Select the type (square, rectangular or circular) from the **Type** droplist.

Data for the first web opening is automatically created as follows:

- l_O - the length of opening (for rectangular openings only)
- d_O - the depth of opening
- L_{CR} - the distance from the setting out point to the center of the opening
- L_C - the distance from end 1 of the member to the center of the opening.
- d_C - the distance from the top of the member to the center of the opening
- L_{CR} relative to - indicates the setting out point from which L_{CR} is measured (defaults to your selection from the **Label openings from** droplist)
- Nr. rel. to - this field only applies if you have chosen the **Opening ->** option from the **LCR relative to droplist**. It is used to specify an existing opening number that you want to use as the setting out point for the new opening.

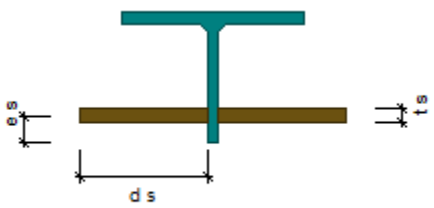


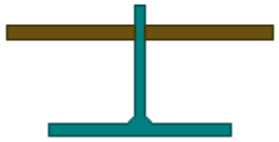
8. If stiffening is required select the location from the **Stiffening** droplist.

Data for the stiffeners is entered manually as follows:

You may need to widen the dialog page to see the relevant column headings.

- d_S - depth of stiffener
- t_S - thickness of stiffener
- L_S - length of stiffener
- e_S - the distance from edge of the opening to the center of the stiffener.





9. Click [Add] then select from the **Type** droplist to create further openings from the chosen setting out point as required.

10. At any point you can use the **Label openings from** droplist to switch to a new setting out point for the next opening.

Related topics

- How do I add web openings manually?
- Web Openings to AISC
- Web Openings to SCI P355
- Web Openings to SCI P068

How do I add web openings manually?

This method enables you to create openings to your own specified depth and spacing.

1. Hover the cursor over the member in which you want to add the web openings so that it becomes highlighted.

2. Right-click and select [Edit]... from the dropdown menu.

3. Ensure **Automatic design** in the properties is cleared.
(Web openings cannot be added to members that are in Autodesign mode.)

4. Click the **Web openings** page of the properties dialog

5. Ensure the **Quick layout** check box is cleared.

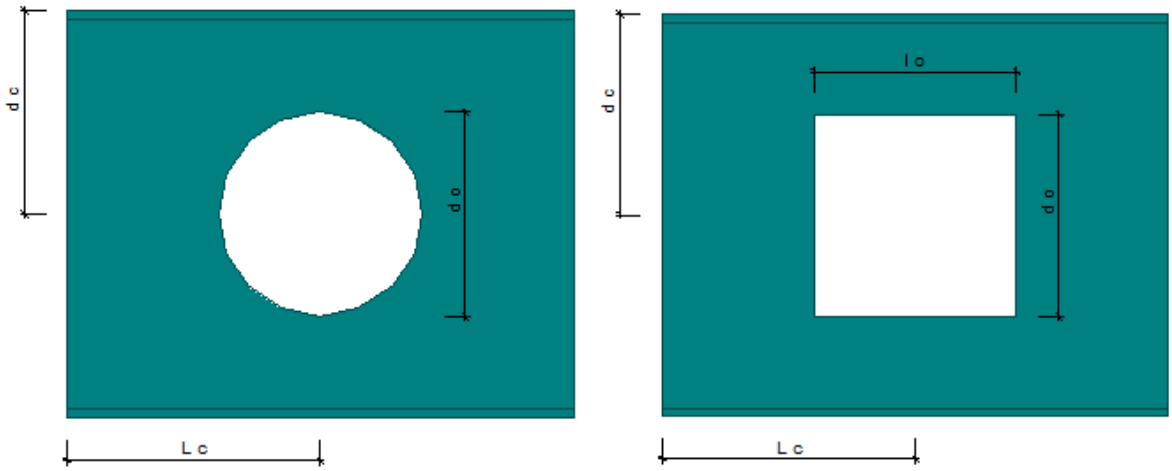
6. Select where you want to start setting out from via the **Label openings from** droplist.

7. Click [Add...] ***(as opposed to*** [Add] ***)***

8. Select the type (square, rectangular or circular) from the **Type** droplist.

9. Manually specify the dimensions of the web opening:

- l_O - the length of opening (for rectangular openings only)
- d_O - the depth of opening



10. Manually specify the distance from the setting out point for the web opening

- L_{CR} relative to - indicates the setting out point from which L_{CR} is measured (defaults to your selection from the **Label openings from** droplist)
- Nr. rel. to - this field only applies if you have chosen the **Opening ->** option from the **LCR relative to droplist**. It is used to specify an existing opening number that you want to use as the setting out point for the new opening.
- L_{CR} - the distance from the setting out point to the center of the opening
- L_C - the distance from end 1 of the member to the center of the opening.

Click **Add** *for L_{CR} and L_C to be automatically calculated as the minimum values appropriate to the size of opening.*

- d_C - the distance from the top of the member to the center of the opening

Click **Center** *for d_C to be automatically calculated to position the opening centrally in the section depth.*

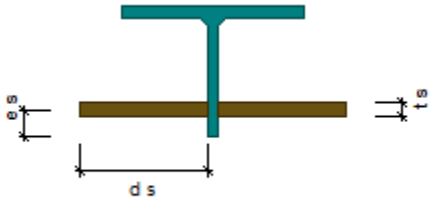
11. If stiffening is required check the **Stiffened** check box - this creates a second page on the dialog.

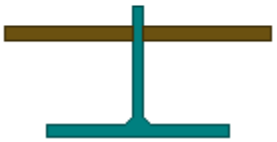
Select the location from the **Stiffening** droplist.

Data for the stiffeners is entered as follows:

- d_S - depth of stiffener
- t_S - thickness of stiffener
- L_S - length of stiffener

- e_S - the distance from edge of the opening to the center of the stiffener.





12. Click OK to create the opening.

13. To create further openings, either:

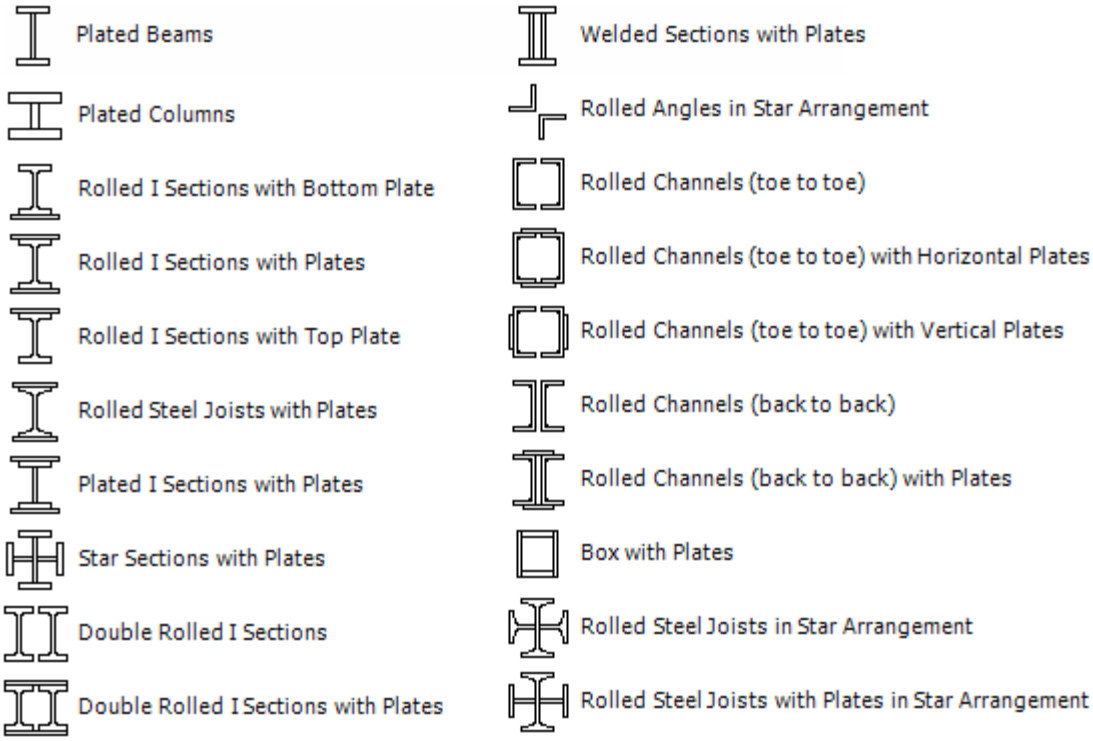
- click Copy... to create multiple copies of a selected opening, or
- click Add... to create a single opening of a different size or spacing,

Related topics

- How do I add web openings using Quick Layout?
- Web Openings to AISC
- Web Openings to SCI P355
- Web Openings to SCI P068

Compound Sections

A comprehensive list of predefined compound section types is available in *Tekla Structural Designer* including many common arrangements such as: Rolled I sections with plates to the bottom, top or both flanges; rolled channels toe to toe and back to back and with or without plates; rolled angles and steel joists in a star arrangement.



You add user defined sections to the database for each compound section type as and when required. Once they have been defined, compound sections are then available for all models.

Compound sections can only be applied to steel beams and steel columns, and while they participate fully in analysis, generally they are not designed (apart from for the limited circumstances detailed below).

Compound sections can only be designed in the following circumstances:
- **Head Code** *must be set to either Eurocode or BS*
- **Characteristic** *must be set to Beam*
- **Compound section type** *must be set to either Plated Beam or Plated Column*

Concrete Member modeling

Modeling Concrete Walls

Concrete wall overview

Concrete walls can be created over several story heights, and can start and finish at any level. Although a single wall is created, this does not prohibit different thicknesses from being defined in each panel, (in which case the panels will be set back on one or both faces, depending on the alignment that has been specified).

The creation of continuous wall in this way, (as opposed to defining a new wall at each story) does not have any significance for analysis or design purposes, but is ultimately important for detailing purposes.

The points used to place a concrete wall define the exact size and position of the wall's analysis model. Its alignment and extension properties have no effect on this model. Either an FE meshed, or mid-pier wall analysis can be adopted on a wall by wall basis.

Provided **AutomaticGenerateSupport** is selected, if no slab or other member exists beneath the wall when it is first created, a support is automatically placed underneath it.

Meshed walls default to the model's mesh parameters, but these can be overridden to allow a user defined mesh to be applied to an individual wall.

Minor axis releases can be applied at the top and bottom of each panel - pinned connections to incoming slabs and members can be modelled in this way.

If you want to create door or window openings in the wall it must be defined as a meshed wall - openings cannot be catered for in mid-pier walls.

Both meshed and mid-pier concrete walls introduce structural strength and stiffness to your model, but they do not perform the same function as Wall Panels, i.e. they do not act as a medium via which loads calculated by the Wind Wizard get applied to your structure. Therefore, in order for these wind loads to be applied you should create additional **Wall Panels** in the same physical locations as the **Concrete Walls**.

How do I specify whether the wall is to be meshed or mid-pier?

You initially choose whether a meshed or mid-pier model is to be adopted by making the appropriate selection from the concrete wall droplist in the Model toolbar.

Subsequently, you can change the model type for an existing wall by changing the **Wall Type** property in the wall properties.

How do I create a concrete wall in a 2D View?

1. Ensure that you have defined the construction levels between which the wall will run and the grid points between which it will lie.

2. Click **Model** > **Concrete Wall drop list** ( )

3. Click the model type required (**Meshed**, or **Mid-pier**).

4. The wall will adopt properties from the concrete wall properties set displayed in the Properties Window.

5. Check that the **Base Level** and the **Top Level** shown in the property set are correct, or adjust if necessary.

6. Check that the **Thickness** and other properties shown in the property set are also correct, and again adjust if necessary.

7. Click where the wall is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

8. Click where the wall is to end (Point 2).

How do I create a concrete wall in a Frame, or Structure View?

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

To create the wall:

1. Click **Model** > **Concrete Wall drop list** ( )
2. Click the model type required (**Meshed**, or **Mid-pier**).
3. The wall will adopt its properties from the concrete wall properties set displayed in the Properties Window.
4. Check that the **Base Level** and the **Top Level** shown in the property set are correct, or adjust if necessary.
5. Check that the **Thickness** and other properties shown in the property set are also correct, and again adjust if necessary.
6. Click the point where the base of the wall is to start.
   (What are the points I can click to create a member?)
7. Click the point where the base of the wall is to end.
8. Click the point where the top of the wall is to start.
9. Click the point where the top of the wall is to end.
10. *Tekla Structural Designer* creates the wall between these four points.

How do I specify extensions?

To automatically trim a new wall back to the face of existing columns or walls:

1. Select the **AutomaticExtension** check box in the Properties Window when creating the wall.

Or, to manually trim or extend existing walls:

1. Select the wall to be trimmed or extended.
2. Specify the required **End 1 extension** or **End 2 extension** in the Properties Window:

- A positive extension extends the wall length beyond its insertion point.
- A negative extension trims the wall back from the insertion point.

How do I specify releases?

1. Select the wall to be released.
2. In the Properties Window, open the **Releases** properties.

3. Select the appropriate release from the **Minor Top** or **Minor Bottom** droplist as required:

- Fixed
- Pinned
- Continuous (incoming members pinned) - only available for FE meshed walls

To specify a pinned connection to a supported slab you should use an FE meshed wall and then select **Continuous (incoming members pinned)** *rather than* **pinned**. *This is because the* **pinned** *option also releases the wall panel above from the wall panel below - which may result in a mechanism during the analysis.*

Automatic support generation

The **AutomaticGenerateSupport** check box in the wall properties is used to control whether a support is automatically created at the wall base level as follows:

- When **AutomaticGenerateSupport** is **on**:
  - **Generate support** cannot be **off**.
  - if there **are** members/slabs underneath the wall capable of providing support - no support is generated.
  - if there **are no** members/slabs underneath the wall capable of providing support - a support is generated.
- When **AutomaticGenerateSupport** is **off**:
  - If **Generate support** is **on** - a support is generated.
  - If **Generate support** is **off** - no support is generated.

When a support is required, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall.

When a support is required, its degrees of freedom are as specified in the **Wall support** *area of the wall properties.*

How do I edit a wall support?

Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

To edit wall support fixity:

1. Expand **Wall Support** in the wall properties

2. Specify the degrees of freedom as required.

Because the discrete supports at each node are angled in the global axis system (always) and not aligned with the wall major/minor axes; it is necessary to set both Mx and My as **Free** *in order to ensure that angled walls are pinned out of plane. (It is not strictly necessary if the wall is aligned in global X or Y, you could set just Mx, or My free as appropriate). Similarly, both Mx and My should be set as* **Fixed** *in order to ensure that angled walls are fixed out of plane.*

To remove a wall support:

1. Uncheck **Generate support** in the wall properties.

The **Generate support** *property will only be available for editing if the* **AutomaticGenerateSupport** *property is inactive. If* **AutomaticGenerateSupport** *is on,* **Generate support** *will be automatically cleared if members are created underneath a wall to support it - similarly it will be automatically rechecked if these members are deleted.*

How do I create a door or window opening in an existing concrete wall?

Openings are only active in meshed (not mid-pier) concrete walls.

1. Open a Frame view of the frame containing the meshed concrete wall within which you want to create an opening.
2. Click **Model** > **Wall Opening**
   - Either, click within the outline of an existing wall panel to define the opening relative to the bottom left corner of the wall,
   - or, click a specific node within an existing wall panel to define the opening relative to the selected node.
3. Pick the first corner of the opening.
   (Press **<F2>** if required to enter its exact position.)
4. Drag to the diametrically opposite corner of the opening (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating the opening).
5. Click to define the opposite corner of the opening.
   (Again, press **<F2>** if required to enter its exact position.)

Modeling Concrete Columns

Concrete column overview

A single concrete column can be created over several story heights (stacks) and can start and finish at any level. Different column sections can be defined in each stack, and the column is not restricted to being co-linear between stacks.

The creation of continuous concrete columns in this way, (as opposed to defining a new column at each story) does not have any significance for analysis or design purposes, but is ultimately important for detailing purposes.

The physical location of the column is a determined from the alignment snap points and offsets specified in the column properties, and the insertion point(s) picked.

How do I specify the column shape and size?

1. Click **Model** > **Concrete Column** ( )
2. In the Properties Window the **Section** box shows the default shape and size.
3. In the **Section** box, click **<New\Edit...>**.
4. Select the shape then enter the size in the dialog.
5. Click **OK** to save. (Do not click **Add** unless you want to create a hollow column - See How do I create a hollow column?).

How do I create a single concrete column in a 2D View?

Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie, then:

1. Click **Model** > **Concrete Column** ( )
2. Check the **Base Level** and **Top Level** shown in the Properties Window are correct - adjust if necessary.
3. Check the column **Section** is correct - adjust if necessary. (How do I specify the column shape and size?)
4. Check the other concrete column properties set values are also correct - adjust if necessary.
5. Click the point where the column is to be placed. (What are the points I can click to create a member?)

How do I create a series of concrete columns in a 2D View?

Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie, then:

1. Click **Model** > **Concrete Column** ( )

2. Check the **Base Level** and **Top Level** shown in the Properties Window are correct - adjust if necessary.

3. Check the column **Section** is correct - adjust if necessary.
(How do I specify the column shape and size?)

4. Check the other concrete column properties set values are also correct - adjust if necessary.

5. Move the cursor to one corner of an imaginary box which will encompass the grid intersection points at which you want to create columns.

6. Click and hold the left mouse button.

7. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating).

8. Once the rubber rectangle encompasses the grid intersection points at which you want to create columns, release the mouse button.

How do I create a single concrete column in a Frame, or Structure View?

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

To create the column:

1. Click **Model** > **Concrete Column** ( )

2. Check the column **Section** is correct - adjust if necessary.
(How do I specify the column shape and size?)

3. Check the other concrete column properties set values are also correct - adjust if necessary.

4. Click the point where the column is to start.
(What are the points I can click to create a member?)

5. Click the point where the column is to end. *Tekla Structural Designer* creates the column between these two points.

How do I align a column to a specific angle, or an angled gridline?

Three options are provided for specifying the alignment of a new column - achieved by setting the **Rotation** property as follows:

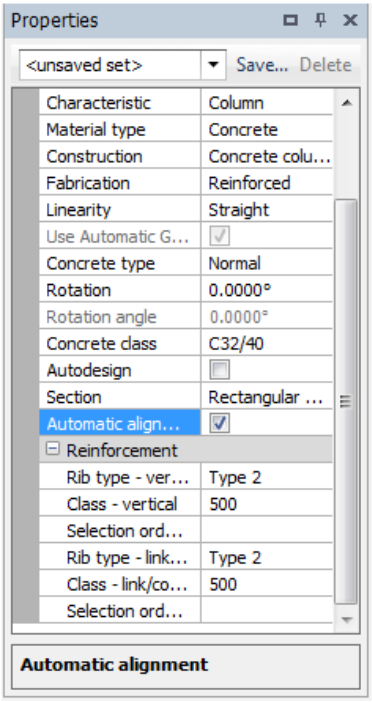
- **0, 90, 180, 90** - aligns the column to the global axes
- **Angle** - aligns the column to the exact rotation angle you specify
- **Define** - aligns the column to the angle of any grid line you select

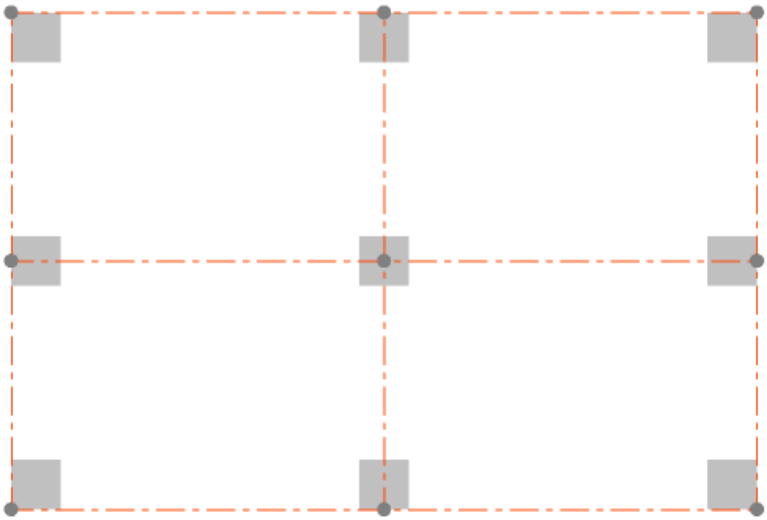
How do I specify the column alignment relative to the grid?

How each column is initially placed relative to the grid depends on the **Automatic alignment** setting in the concrete column properties set.

If Automatic alignment is on:

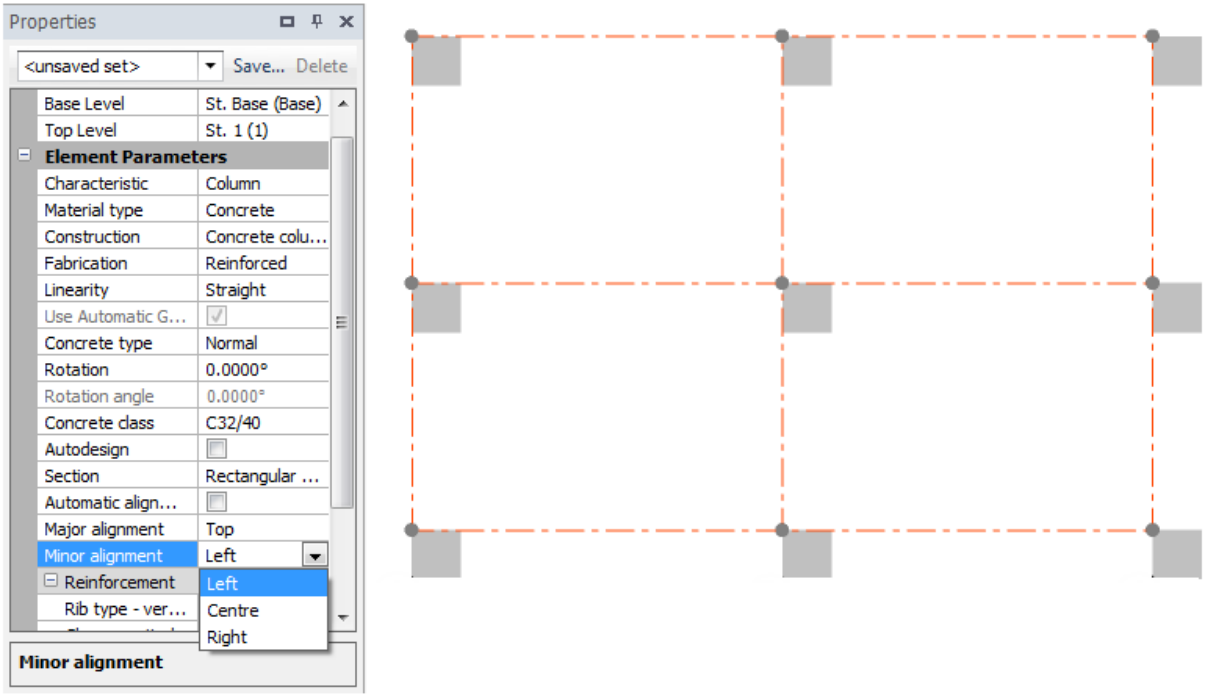
Columns on the perimeter of the grid are aligned with their faces flush to the perimeter and internal columns are aligned centrally on the grid.





If Automatic alignment is off:

Columns are aligned according to the **Major** and **Minor alignment** settings in the concrete column properties set.



How do I create a hollow column?

1. Click **Model** > **Concrete Column** ( )
2. In the Properties Window the **Section** box shows the default shape and size.
3. In the **Section** box, click **<New\Edit...>**.
4. Select the shape then enter the size in the dialog.
5. In the same dialog, click **Add**
6. In the tabular part of the dialog, select the shape and dimensions of the void.
7. Leave the minor and major offsets as 0.0 to position the void centrally in the column, or adjust as necessary to create an offset.

How do I create an inclined column?

An inclined column can only be created in a Frame, or Structure View.

In order to define the column you must have defined the construction levels between which the column will run and the grid points between which it will lie.

1. Click **Model** > **Concrete Column** ( )
2. The column will adopt properties from the currently displayed concrete column properties set.
3. Check that the properties shown in the property set are correct or adjust if necessary.

4. Click the point where the column is to start.
   (What are the points I can click to create a member?)

5. Click the point where the column is to end. *Tekla Structural Designer* creates the column between these two points.

Related topics

- How do I create a cranked column?

How do I create a cranked column?

A cranked column can only be created in a Frame, or Structure View.

In order to define the column you must have defined the construction levels between which the column will run and the grid points between which it will lie.

1. Click **Model** > **Concrete Column** ( )

2. The column will adopt properties from the currently displayed concrete column properties set.

3. Check that the properties shown in the property set are correct or adjust if necessary.

4. Click the point where the column is to start.

5. Press and hold the Ctrl key on the keyboard and click the next node of your cranked column.

6. Repeat step 3 until you have defined the penultimate node of your cranked column. Release the Ctrl key and click the point where the column is to end.

How do I modify the position of a single column stack?

A column stack's position can best be modified in either a Frame, or Structure View.

1. In Scene Content, ensure that the Grid & Construction Lines box is checked.

2. Select the column for which a stack is to be moved.

   The column is highlighted in one color, the column end nodes and middle node in another.

3. Select one end nodes for the column stack to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

4. Click a grid or construction point to redefine the column stack end node position.

   The entire column is redrawn with the selected node moved to the new position.

5. Select the next end node. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

6. Click a grid or construction point to redefine the end node position.

   The column is redrawn once more with the selected node moved to the new position.

How do I modify the position of an entire column?

Simply use the **Move** command located on the Edit ribbon.

How do I edit the column alignment or specify an offset?

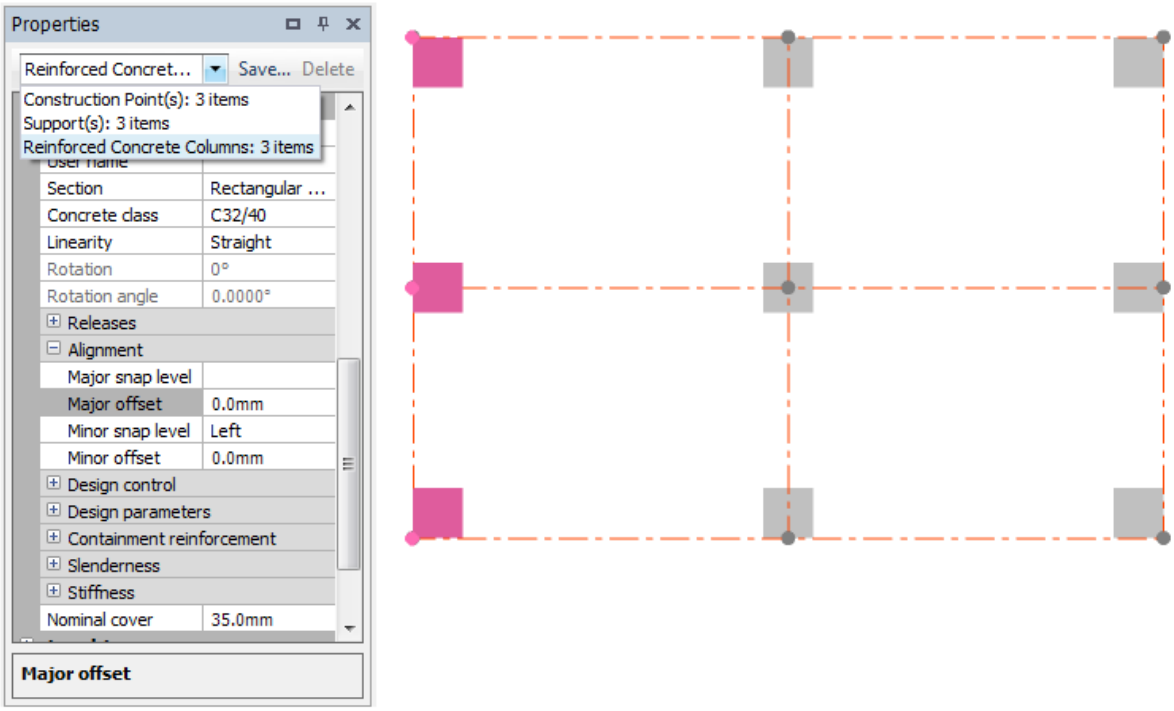
Once columns have been placed their alignments can be adjusted and further offsets specified if required.

- A single column's alignment can be adjusted either in the Properties Dialog or in the Properties Window
- Multiple columns can only be realigned using the Properties Window

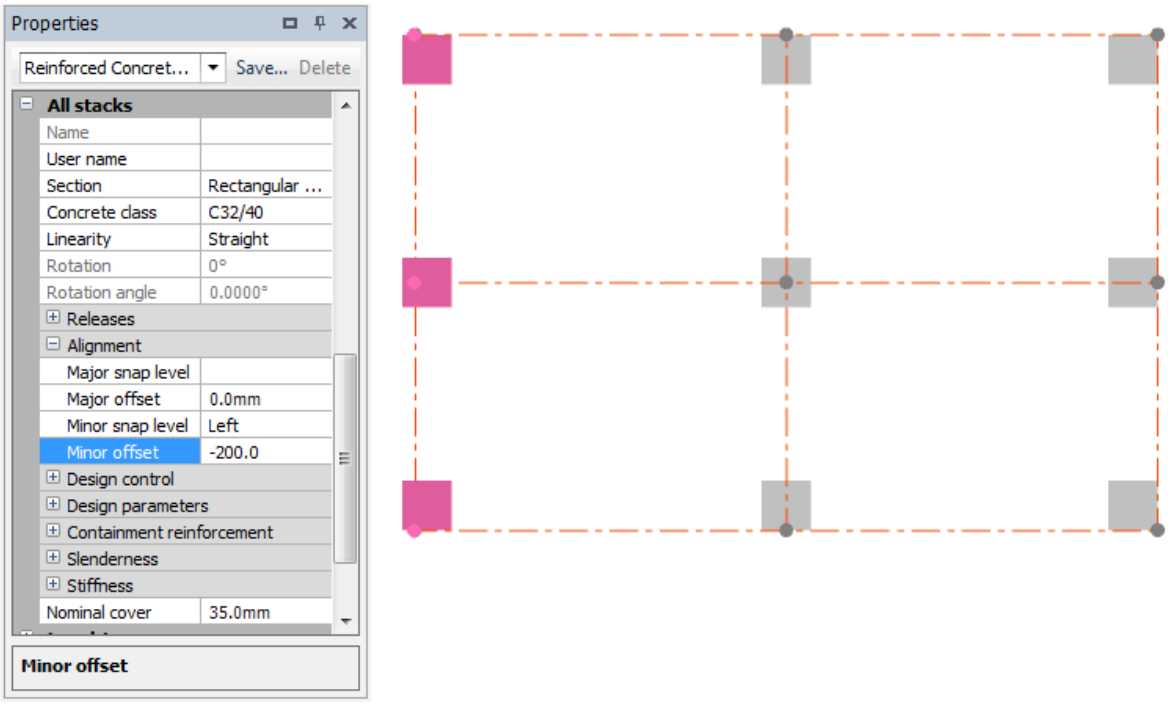
The example below illustrates editing the alignment in the Properties Window.

1. Select the columns to be offset.

2. Ensure the column properties are displayed in the Properties Window, (use the drop list at the top of the window if necessary).



3. Edit **Major offset** and/or **Minor offset** required to move the columns relative to the Major/Minor snap levels by the amount specified.



Modeling Concrete Beams

Concrete beam overview

Concrete beam members consist of one or more beam spans. Although a single beam member is created, this does not prohibit different beam sections from being defined in each span.

The physical location of the beam is a determined from the alignment snap points and offsets specified in the beam properties, and the insertion points picked.

How do I specify the beam size?

1. Click **Model** > **Concrete Beam** ( )
2. In the Properties Window the **Section** box shows the default shape and size.
3. In the **Section** box, click **<New\Edit...>**.
4. Select the shape then enter the size in the dialog.
5. Click **OK** to save.

How do I create a single span concrete beam?

1. Click **Model** > **Concrete Beam** ( )

2. In the Properties Window, check the **Section** box shows the correct size - adjust if necessary.
(How do I specify the beam size?)

3. Check the other concrete beam properties set values are also correct - adjust if necessary.

4. Click where the beam is to start (Point 1).
(What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the beam is to end (Point 2).

If you are prompted to pick another point (Point 3), this will be due to **Continuous** *being selected in the beam property set. In order to create a single span beam simply click once again on Point 2, or press the Enter key.*

How do I create a continuous concrete beam?

1. Click **Model** > **Concrete Beam** ( )

2. In the Properties Window, check the **Section** box shows the correct size - adjust if necessary.
(How do I specify the beam size?)

3. Check the other concrete beam properties set values are also correct - adjust if necessary.

4. Click where the beam is to start.
(What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click the next node of your continuous beam.

6. Repeat step 4 until you have defined the penultimate node of your continuous beam.

7. Click where the beam is to end.

8. You are now prompted to pick another point, simply click once again on the same point, or press the Enter key in order to create the continuous beam.

You cannot define continuous beams which are curved either horizontally or vertically.

How do I create a series of concrete beams?

In order to create beams using this method the floor or construction level must already contain the columns between which the beams will run. You must also use a 2D view of the floor or construction level to use this option.

1. Click **Model** > **Concrete Beam** ( )
2. Check that Section in the Properties Window shows the correct size - adjust if necessary.
   (How do I specify the beam size?)
3. To create continuous beams between the supports - check the Continuous box.
4. Else to create a series of single span beams between the supports - clear the Continuous box.
5. Review the other concrete beam properties set values and adjust if necessary.
6. Move the cursor to one corner of an imaginary box which will encompass the columns between which you want to create beams.
7. Click and hold the left mouse button.
8. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating).
9. Once the rubber rectangle encompasses the columns between which you want beams creating, release the mouse button.
10. Beams are created between each adjacent pair of columns within the area you select.

How do I create a curved concrete beam?

1. Click **Model** > **Concrete Beam** ( )
2. The beam will adopt its properties from the displayed concrete beam properties set; ensure that the **Linearity** is set to **Curved Major** (if it is to curve vertically), or **Curved Minor** (if it is to curve horizontally), and appropriate **Chord height** and Maximum facet error values have been specified to define the curve.

You can control the direction in which horizontally curved beams curve. When you place the beam you select its start point and its end point. The beam always curves such that if you were to look along the theoretical line from the start point to the end point, then the curve on the beam will always lie to the right of that line.

For vertically curved beams, a negative chord height value can be used to reverse the curve direction.

3. Adjust any other details in the property set as necessary.

4. Click where the beam is to start (Point 1).
(What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the beam is to end (Point 2).

When defining beams which curve vertically, if you are using a 2D view of the floor or construction level, then the vertical projection of the beam is out of the plane of the floor or construction level that you are using. In this case you will not see the beam on the graphical display. You will need to change to a 3D view of the floor, construction level or the entire Structure in order to see them.

Horizontally curved beams always take the chord height defined in the property set, they do not curve automatically to fit on any curved grid line that you may have defined.

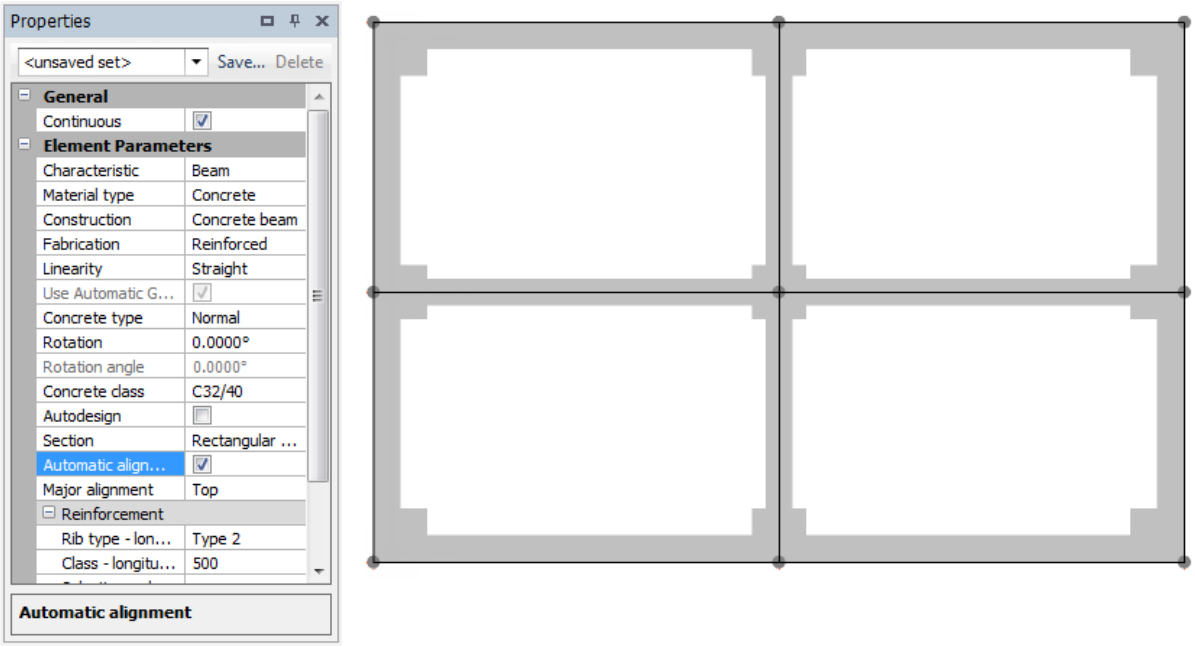
How do I specify the beam alignment relative to the grid?

For initial placement, beams are aligned relative to the grid depending on the **Automatic alignment** setting in the concrete beam properties set.

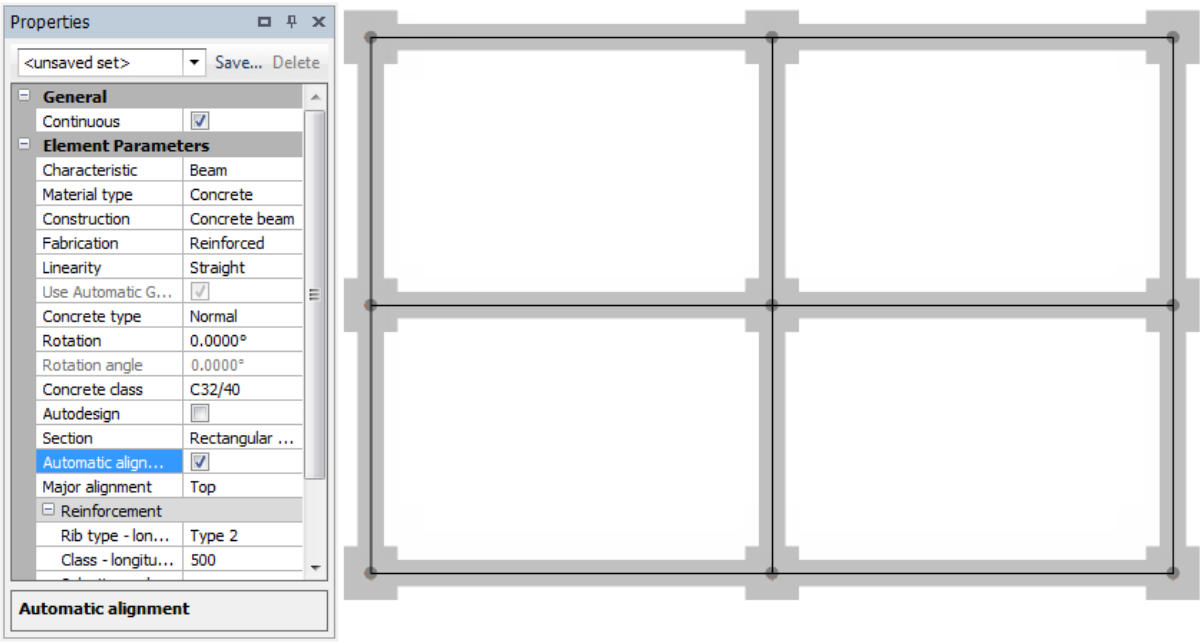
If Automatic alignment is on:

When placed by boxing around columns, each beam's alignment follows that of the columns between which it spans.

In the below example, because the edge columns were aligned flush with the grid, when the beams are placed (by boxing around the columns) with automatic alignment on, they are flush also.

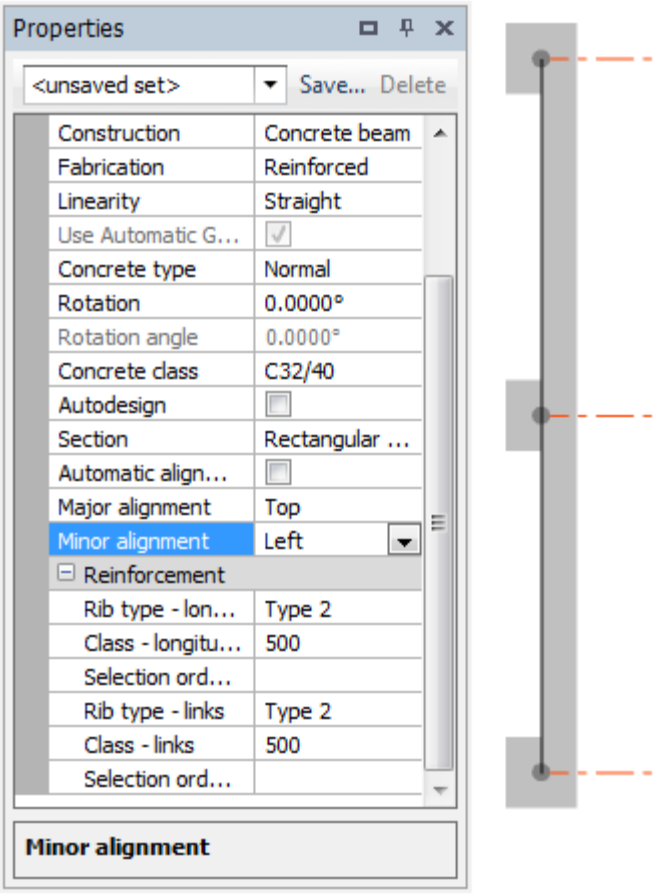


However, if the edge columns had been aligned centrally, when the beams are placed with automatic alignment on, they would be central also.



If Automatic alignment is off:

Beams are aligned according to the **Major** and **Minor alignment** settings in the concrete beam properties set.



How do I specify and use beam flanges for an existing beam?

1. Hover the cursor over the beam to be edited so that it becomes highlighted.

2. Right-click and select Edit... from the dropdown menu.

The **Concrete Beam Property Dialog** is displayed.

3. Open the Design Control page of the dialog for a particular span and select **Consider flanges**.

Further boxes are opened up displaying the flange dimensions (initially all zero).

4. To specify the flange dimensions, click **Calculate flanges**.

- the flange dimensions are automatically calculated.

5. If required you can add an allowance for openings to reduce the calculated widths by a specified amount.

6. Repeat the above for additional spans in the beam as required.

Once flanges have been specified in this way they will be considered in the concrete beam design calculations, however the flanged beam properties are not by default used in the analysis.

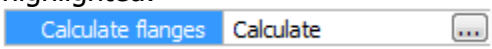
To use flanged beam properties in the analysis (in addition to the design calculations)

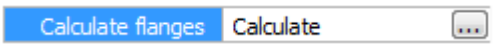
1. Select **Include flanges in analysis**.
2. Repeat for additional spans in the beam as required.

How do I specify and use beam flanges for multiple beams simultaneously?

1. Select the beams to be edited.
2. The edit concrete beam properties set shared by the beams are displayed in the Properties Window.
3. Located under All spans - Design Control, select **Consider flanges**.

   Further boxes are opened up displaying the flange dimensions (initially all zero).
4. To specify the flange dimensions, firstly select **Calculate flanges** so that it becomes highlighted:

5. Then click the  button - the flange dimensions are automatically calculated.

> *Only flange dimensions which are common to all the selected beams can be shown in the fields; where different flange dimensions have been calculated for the selected beams, the fields are left blank.*

6. If required you can add an allowance for openings to reduce the calculated widths by a specified amount.

Once flanges have been specified in this way they will be considered in the concrete beam design calculations, however the flanged beam properties are not by default used in the analysis.

To use flanged beam properties in the analysis (in addition to the design calculations)

1. Located under All spans - Design Control, select **Include flanges in analysis**.

How do I edit the beam alignment or specify an offset?

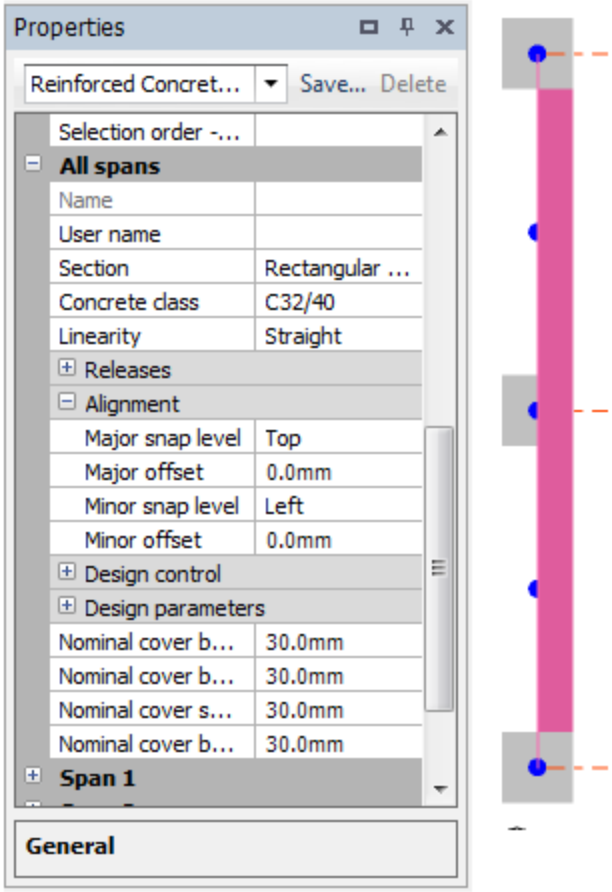
Once beams have been placed their alignments can be adjusted and further offsets specified if required.

- A single beam's alignment can be adjusted either in the Properties Dialog or in the Properties Window
- Multiple beams can only be realigned using the Properties Window

The example below illustrates editing the alignment in the Properties Window.

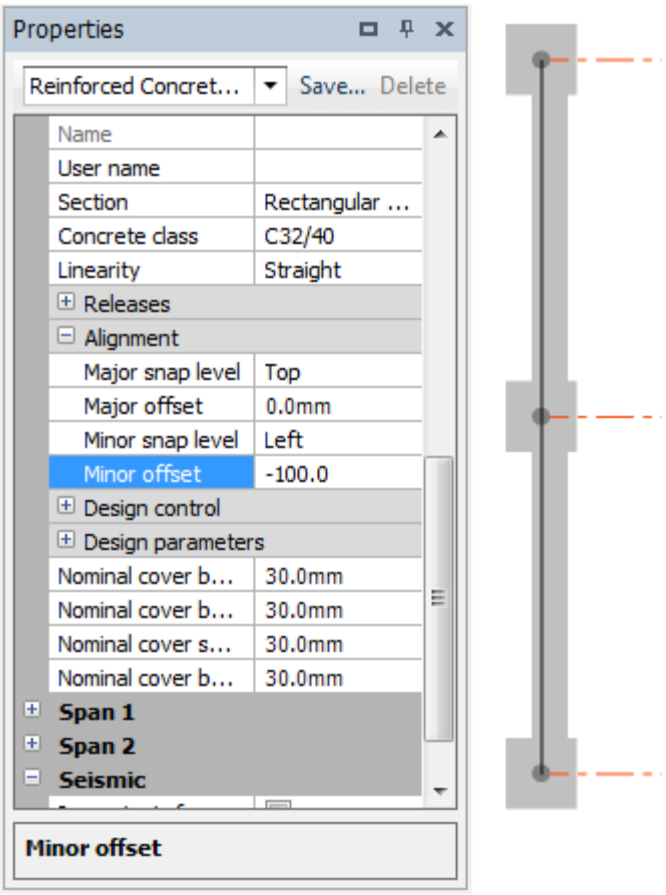
1. Select the beams to be edited.

2. Ensure the beam properties are displayed in the Properties Window, (use the drop list at the top of the window if necessary).



3. Edit the **Major** and/or **Minor snap level** as required to realign the beams in the vertical and horizontal planes respectively.

4. Edit the **Minor offset** to move the beams horizontally relative to the Minor snap level by the amount specified.



5. Edit the **Major offset** to move the beams vertically relative to the Major snap level by the amount specified.

How do I modify the position of a beam?

You can modify an individual beam's position in both 2D and 3D views.

1. Select the beam to be moved.

   The beam is highlighted in one color, the beam end nodes and middle node in another.

2. Now select the beam end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

3. Click a grid or construction point to redefine the beam end node position.

   The beam is redrawn spanning to the new end position.

Slab modeling

Slabs are the medium via which loads placed on a floor are decomposed back to the supporting structure.

Slabs can be created in levels or inclined planes. The vertical alignment of all slabs in a given level is determined by the "type" assigned to the level:

- T.O.S - the base of each slab item aligns with the level

- S.S.L - the top of each slab item aligns with the level

Each "parent" slab consists of a number of individual panels (slab items); these can either be connected or separated from each other, however they must be on the same level.

Every parent slab has a unique name. A slab name used at one level cannot be re-used at a different level.

The individual panels initially inherit their properties from the parent slab, but once they have been created they can be edited to allow certain of properties to be amended: the rotation angle can be changed; the slab depth can be overridden; a panel can be excluded from the diaphragm. When a slab depth override has been applied you can also apply a vertical offset in order to model a slab step.

Vertical offsets **are not** *structurally significant as they have no effect on the analysis mesh position relative to the top of the slab in the solver model. The mesh properties would however reflect any change to the slab thickness.*

Slab boundaries can be straight or curved.

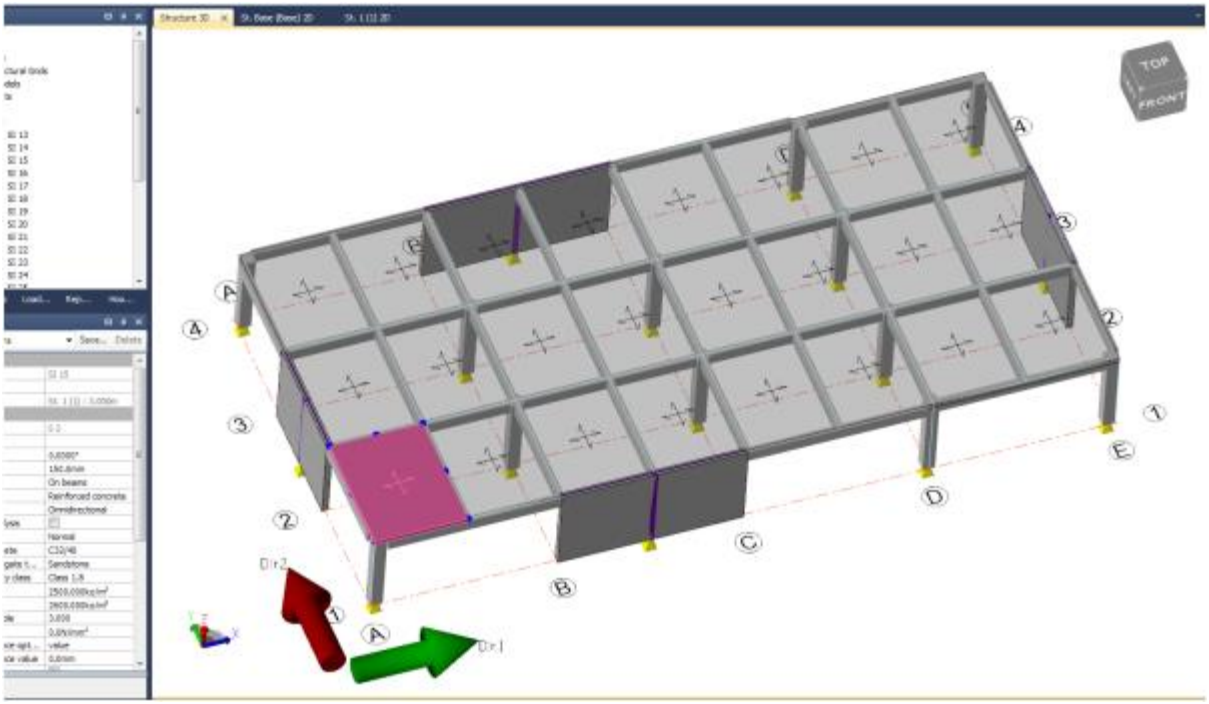
Openings of any shape can be created within slabs.

In flat slabs, at points where the slab is supported by columns, column drop panels can be applied to thicken the slab, (drop panels are effectively slab items with an increased thickness).

Overview of slab modeling

The Concept of Slabs and Slab Panels

Tekla Structural Designer stores slab data in the form of "parent slabs" each consisting of one or more slab items (or slab panels).



In the view above there are 24 slab items grouped together in one slab.

Some data is set at the "slab" level and is common to all panels, other data is set at the "slab panel" level.

In simple overview terms the data breakdown is as follows:

Slab Data:

- Thickness
- Vertical Offset
- Material Properties
- Analysis Settings
- General Design Settings

Slab Panel Data:

- Cover
- Reinforcement information
- Specific Design Settings

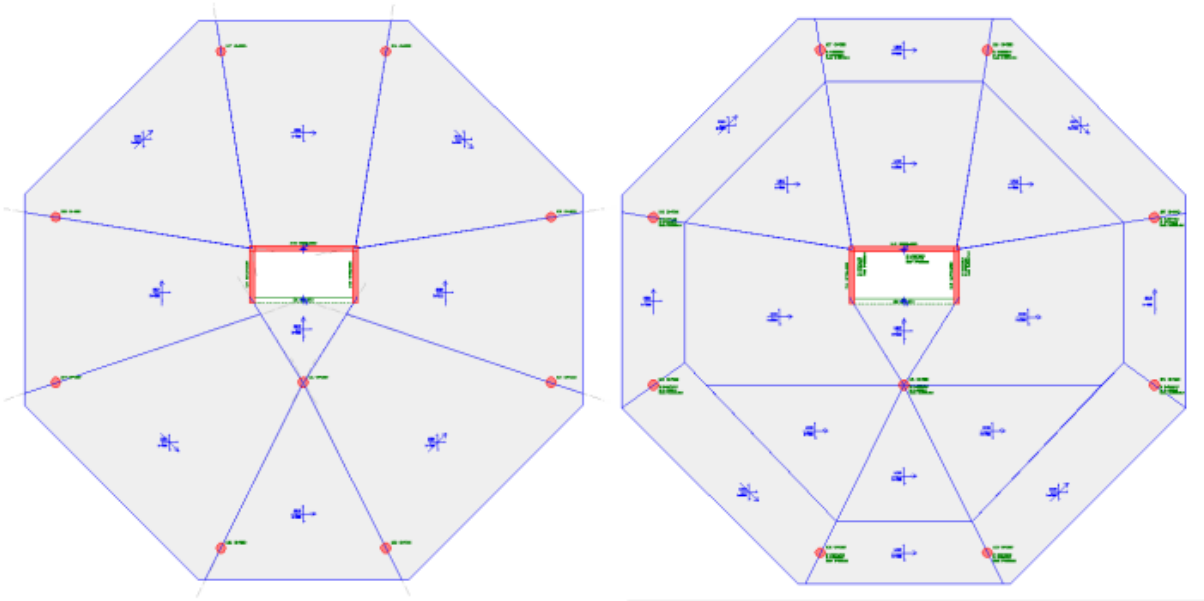
In modeling terms you are thus able to create slabs over a wide area, there is no reason at this stage to consider sub-sections of the slab - it is just one big expanse of slab.

When it comes to design you then need to conceptualize the slab as a series of design panels. Each design panel will have it's own design settings and it's own design results. Different reinforcement can be selected in different panels. You also have to consider pattern loading (some panels loaded and others not). When results are then presented in calculations and drawings you are able to specifically reference the design panels.

Panel sub-division

Regardless of how the slabs and panels are initially created, you are able to further divide (or re-form) them via the **Slab Split** and **Slab Join** commands. There are several reasons why you may choose to do this relating to refining geometry (adding steps), pattern loading, and panel design.

When it comes to flat slabs in particular, the way that slabs are split for the purposes of pattern loading is a matter of engineering judgement - the views below show a couple of options that 2 different engineers might both justifiably choose for the same slab perimeter.



Creating slab panels

•

How do I choose the slab type and specify its properties?

1. Click **Model**, then from the first **Slabs** drop list select the slab type required.
2. Define panel properties for the chosen slab type.

Slab Type	Property Set
Slab on Beams	slab on beams item properties set
Flat Slab	flat slab item properties set
Precast	precast item properties set
Steel Deck	steel deck item properties set
Timber Deck	timber deck item properties set

Composite Slab	composite slab item properties set

3. Before placing the panel confirm the **Slab** and **Select bays** properties (under the **General** heading) are set as required.

How do I specify the parent slab to which the panel belongs?

1. Under the **General** heading use the **Slab** property to specify that you want to either:
   - create a **New Slab**
   - automatically **add to an existing** parent slab
   - manually select the parent slab.
2. When the panel properties are defined it can then be placed either by bays or by picking points.

When using the **Add to existing** *option, at the point of slab item creation a check is made to find if there is a existing slab in the plane with same properties. If found - the new panel will be added to it (if two are found then the one that is physically closest is used). If not found - a new parent slab is created.*

The option to manually select the parent slab is only available in 2D Views, and then only after one or more panels have been placed in that view.

How do I create slab panels by bay?

This method is only available in 2D Views.

1. Under the **General** heading ensure that the **Select bays** property is checked.

   A "Create Slab Item: Select Bay" prompt appears and the bays in which slab items can be placed (i.e those that have members attached to all sides) are identified by shading.

If no bays are identified this could be due to invalid member intersections:
- Validate the model to detect and correct any beams with crossing centerlines.

2. To create the panel(s):

- Either click within the outline of a bay bounded by beams/walls to add an individual panel, or
- Drag a box to add panels into all bounded bays,

Dragging the box from left to right:
- places panels in those bays totally enclosed by the box.
Dragging the box from right to left:
- places panels in all bays that are either enclosed by, or cross the box.
Hold down the [Shift] key while dragging:
- creates a line instead of a box. Panels are placed in all bays that are cross the line.

How do I create slab panels by points?

This method is available in both 2D and 3D Views.

1. Under the **General** heading ensure that the **Select bays** property is cleared.
2. Pick the first point to define a corner of the panel.
3. Continue and pick the other points which define the panel outline.
4. When you get to the final point double-click to create the panel.

Modeling slab and mat openings

Simple Openings

Simple openings can be quickly defined from within existing slabs. Such openings are rectangular or circular in plan.

The Slab Opening command is located on the droplist in the Slabs group, it is only accessible when a 2D view is active.

Openings can

- cross more than one panel/slab
- be overlayed/joined to create openings which together have shapes other than rectangular
- cut across a stepped edge
- be applied to level and sloping slabs

Openings cannot

- be applied to one way spanning slabs
- Cannot reside within or cut a column drop

Irregular Openings

Alternatively, more complex openings can be created by using construction lines and constructing panels around an irregular shape.

How do I create a rectangular slab opening?

1. Open a 2D view of the level containing the slab or mat within which you want to create an opening.
2. Click **Model** > **Slab Opening** (located on the second **Slabs** drop list) or **Foundations** > **Mat Opening** (located on the first **Mats** drop list) as appropriate.
3. The slab opening and mat opening properties are displayed in the Properties Window.
4. Use the **Opening Type** property in this set to specify a **Rectangular** opening.
5. If required enter a rotation angle to rotate the opening on plan.
6. Click within the outline of an existing slab/mat panel to define a corner of the opening. (Press **<F2>** if required to enter its exact position.)
7. Drag to the diametrically opposite corner of the opening (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating the opening).
8. Click to define the opposite corner of the opening. (Again, press **<F2>** if required to enter its exact position.)

How do I create a circular slab opening?

1. Open a 2D view of the level containing the slab or mat within which you want to create an opening.
2. Click **Model** > **Slab Opening** (located on the second **Slabs** drop list) or **Foundations** > **Mat Opening** (located on the first **Mats** drop list) as appropriate.
3. The slab opening and mat opening properties are displayed in the Properties Window.
4. Use the **Opening Type** property in this set to specify a **Circular** opening.
5. Click within the outline of an existing slab panel to define the center of the opening. (Press **<F2>** if required to enter its exact position.)
6. Drag then click to define the radius of the opening. (Again, press **<F2>** if required to enter the exact radius.)

How do I delete a slab opening?

1. Open the **Slab Openings** branch of the Structure Tree.
2. Right-click over the **Name** of the slab opening you want to delete.
3. Click **Delete** from the context menu which appears.

Adding overhangs to existing slab or mat edges

The building objects that we have dealt with thus far are defined along the grid lines that we have created. For slabs this may not be entirely the case. Particularly for slabs the edge of the slab may extend beyond the grid line, either to the edge of the beam which supports the edge of the slab, or, around the perimeter of the building to meet the inside face of the cladding.

In order to cater for these common situations, *Tekla Structural Designer* allows you to define overhangs to the edges of a slab (full length or partial length). An overhang may extend across many panels in one slab. Any loads that you define over an overhang will be included in the total loading on your building. .

The Slab Overhang command is located on the droplist in the Slabs group, it is only accessible when a 2D view is active.

How do I add an overhang to a slab edge?

In order to define a slab overhang you must have already defined the slab to which it applies.

To create the overhang:

1. Open a 2D view of the level containing the slab for which you want to create an overhang.
2. Click **Model** > **Slab Overhang** (located on the second **Slabs** drop list) or **Foundations** > **Mat Overhang** (located on the first **Mats** drop list) as appropriate.
3. The slab overhang and mat overhang properties are displayed in the Properties Window from where you can enter the width of the overhang.
4. Click along the edge of an existing slab/mat panel to define the start point of the overhang.
   (Press **<F2>** if required to enter its exact position.)
5. Click along the same edge to define the end point of the overhang.
   (Press **<F2>** if required to enter its exact position.)

How do I add a curved overhang to a slab edge?

In order to define a curved slab overhang you must have already defined the slab to which it applies.

To create the overhang:

1. Open a 2D view of the level containing the slab for which you want to create an overhang.
2. Click **Model** > **Slab Overhang** (located on the second **Slabs** drop list) or **Foundations** > **Mat Overhang** (located on the first **Mats** drop list) as appropriate.
3. In the slab overhang and mat overhang properties, (displayed in the Properties Window), clear **Edge Affiliated**.
4. Enter the curvature of the overhang.
5. Enter the width of the overhang.
6. Click along the edge of an existing slab panel to define the start point of the overhang.
   (Press **<F2>** if required to enter its exact position.)

7. Click along the same edge to define the end point of the overhang.
   (Press **<F2>** if required to enter its exact position.)

Modeling column drops

In order to increase punching resistance, drop panels can be inserted within concrete slabs at points where they are supported by columns. Column drops are a slab thickening that can be above the slab, below the slab or both. A column drop is rectangular in plan and is aligned to the column axes.

The extent of the drop is limited by the extent of the slab boundary. (The cut back drop shape may not be rectangular).

How do I create a column drop?

In order to define a slab drop you must have already defined the concrete slab to which it applies.

To create the drop:

1. Click **Model**, then from the second **Slabs** drop list select **Column Drop**.
2. The column drop properties are displayed in the Properties Window from where you can enter the geometry of the drop.

The drop is always positioned centrally over the supporting column.

3. Click an existing column connected to a concrete slab panel to create a single drop panel, or box around multiple columns to create a series of drops.

For the drop panel to be inserted correctly, it is important that in the construction level dialog the setting out point "Type" for the slab level in question is set as S.S.L. not T.O.S.

Splitting and joining slabs and mats

Existing slab panels can be sub-divided into smaller panels using the **Slab Split** command.

Similarly they can be merged into larger panels using the **Slab Join** command.

The Slab Split and Slab Join commands are only accessible when a 2D view is active.

How do I split a slab?

1. Click **Model** > **Slab Split**, or **Foundations** > **Mat Split** as appropriate.
2. Hover the cursor over the slab edge where you want the split to start.

> *The points used to define the cut line can be outside the boundary of the slabs being split - they don't have to be on the slab edges - you can also use grid points or other "points of interest".*

3. Suggested split points along the edge are highlighted (the selected point being indicated by a red dot).
4. Click at the point indicated, or move the cursor further along the edge to identify other points at which it could be split.
5. Hover the cursor over another slab edge (either of the same, or another panel).

- Either click at the point required and continue to click subsequent points to continue splitting,
- or, double click to complete the split.

6. Any slab panels entirely crossed by the cut line are split along it.

How do I join slabs?

1. Click **Model** > **Slab Join**, or **Foundations** > **Mat Join** as appropriate.
2. Pick the first of the slab panels you wish to join, (the "Master" panel).
3. Pick the next slab panel.

   The two panels are joined to create a new panel which adopts the properties of the "Master" panel.

> *The panels to be joined must share a common edge.*

4. Pick additional panels as required to add to the "Master" panel, or press Esc when complete.

Slab steps

A step is modeled in *Tekla Structural Designer* as a panel to which a slab depth override and/or vertical offset is applied.

Steps can therefore be located anywhere in a slab that a panel can be created, the **Slab Split** and **Slab Join** commands are likely to prove useful in their creation.

The additional data needed to allow any panel to be stepped up or down is:

- Override slab depth (checked)
- Depth
- Vertical offset
  (+ve raises the panel surface and -ve drops it).

Vertical offsets **are not** *structurally significant as they have no effect on the analysis mesh position relative to the top of the slab in the solver model. The mesh properties would however reflect any change to the slab thickness.*

How do I modify the shape or size of a panel?

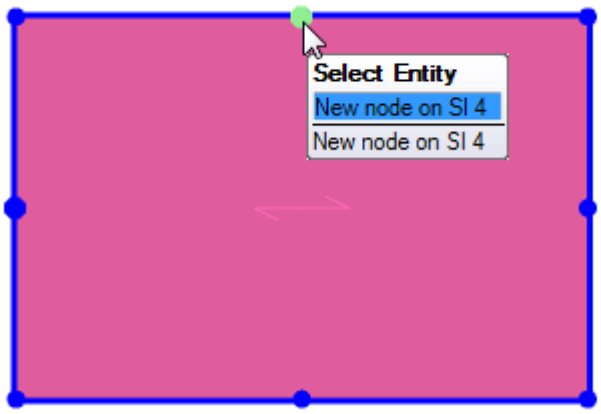
You can modify a panel in both 2D and 3D Views; either by moving its nodes, or its edges.

How do I modify a panel by moving a node?

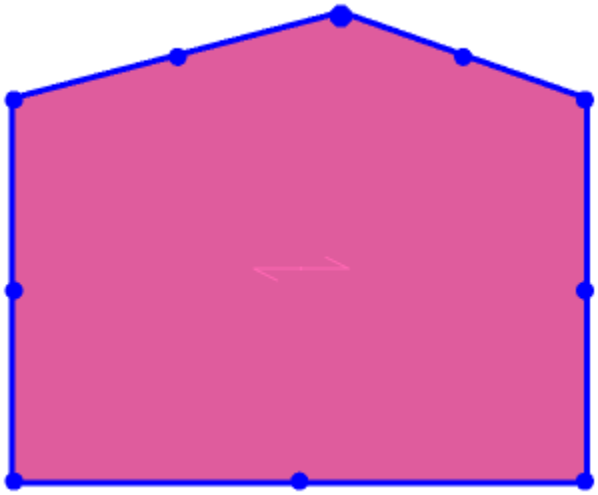
A panel can be modified by moving the nodes at its corners, or at the mid-points of the panel edges as follows:

1. Select the panel you want to modify.

2. Click a node to select it.



3. Move the mouse over an existing construction point or intersection, then click to reposition the node.



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

When panel nodes are moved, note that:

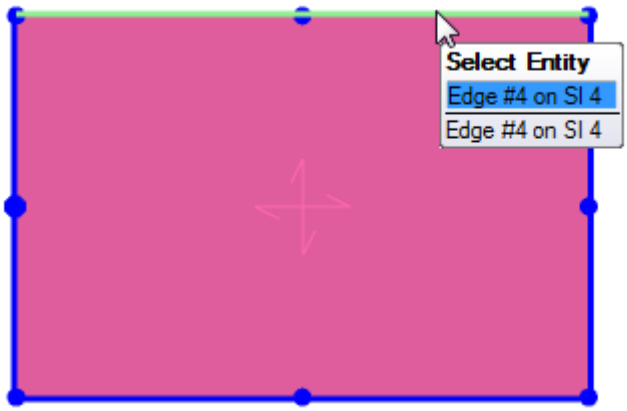
- When a node at the mid-point of an edge is moved, this becomes a new corner node (causing two new mid-point nodes to be automatically created).
- When a corner node is moved directly over an adjacent corner node, the original corner node is deleted.

How do I modify a panel by moving an edge?

A panel can be modified by moving any of its edges as follows:

1. Select the panel you want to modify.

2. Click an edge (away from the nodes) to select it.



3. Move the mouse over an existing construction point or intersection, then click to reposition the edge so that it passes through the point selected.

The new edge position must be in the same plane as the panel.

How do I apply curved edges to existing slab items?

1. Select the slab items you want to modify.
2. To apply the same degree of curvature to all the slab edges:
   - Locate the **All edges** property
   - Uncheck **Linear**
   - Enter the Curvature required as a chord offset, specify a positive value to curve inwards, a negative value to curve outwards.
3. To apply curvature to a specific edge:
   - Locate the Edge property (1,2,3 etc.) for the required edge
   - Uncheck **Linear**
   - Enter the Curvature required as a chord offset, specify a positive value to curve inwards, a negative value to curve outwards.
4. *T*he slab panel is redrawn with the specified curvature.

How do I delete an entire slab from my model?

1. Open the **Slabs** branch of the Structure Tree.
2. Right-click over the **Name** of the slab you want to delete.
3. Click **Delete** slab from the context menu which appears.
4. *Tekla Structural Designer* deletes the entire slab from your model.

Timber Member modeling

Timber sections can be modelled and analysed in Tekla Structural Designer but they are not designed.

If working to Eurocodes timber sections can be exported to Tekla Tedds for design.
Tekla Tedds is a separately purchasable product.

Modeling Timber Columns

How do I create a single timber column in a 2D View?

1. Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie.

2. Click **Model** > **Timber Column** ( )

3. The column will adopt properties from the currently displayed timber column properties set.

4. Check that the **Base Level** and the **Top Level** shown in the property set are correct, or adjust if necessary.

5. Check that the other properties shown in the property set are also correct, and again adjust if necessary.

6. Click the point where the column is to be placed.
(What are the points I can click to create a member?)

How do I create a series of timber columns in a 2D View?

1. Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie.

2. Click **Model** > **Timber Column** ( )

3. The column will adopt properties from the currently displayed timber column properties set.

4. Check that the Base Level and the Top Level shown in the property set are correct, or adjust if necessary.

5. Check that the other properties shown in the property set are also correct, and again adjust if necessary.

6. Move the cursor to one corner of an imaginary box which will encompass the grid intersection points at which you want to create columns.

7. Click and hold the left mouse button.

8. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating).

9. Once the rubber rectangle encompasses the grid intersection points at which you want to create columns, release the mouse button.

How do I create a single timber column in a Frame, or Structure View?

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

To create the column:

1. Click **Model** > **Timber Column** ( )

2. The column will adopt properties from the currently displayed timber column properties set.

3. Check that the properties shown in the property set are also correct or adjust if necessary.

4. Click the point where the column is to start.
(What are the points I can click to create a member?)

5. Click the point where the column is to end. *Tekla Structural Designer* creates the column between these two points.

How do I align a column to a specific angle, or an angled gridline?

Three options are provided for specifying the alignment of a new column - achieved by setting the **Rotation** property as follows:

- **0, 90, 180, 90** - aligns the column to the global axes
- **Angle** - aligns the column to the exact rotation angle you specify
- **Define** - aligns the column to the angle of any grid line you select

How do I create an inclined column?

An inclined column can only be created in a Frame, or Structure View.

In order to define the column you must have defined the construction levels between which the column will run and the grid points between which it will lie.

1. Click **Model** > **Timber Column** ( )

2. Check that the timber column properties set shown in the Properties Window are correct - adjust if necessary.

3. Click the point where the column is to start.
(What are the points I can click to create a member?)

4. Click the point where the column is to end. *Tekla Structural Designer* creates the column between these two points.

How do I create a cranked column?

A cranked column can only be created in a Frame, or Structure View.

In order to define the column you must have defined the construction levels between which the column will run and the grid points between which it will lie.

1. Click **Model** > **Timber Column** ( )

2. Check that the timber column properties set shown in the Properties Window are correct - adjust if necessary.

3. Click the point where the column is to start.

4. Press and hold the Ctrl key on the keyboard and click the next node of your cranked column.

5. Repeat step 3 until you have defined the penultimate node of your cranked column. Release the Ctrl key and click the point where the column is to end.

How do I modify the position of a single column stack?

A column stack's position can best be modified in either a Frame, or Structure View.

1. In Scene Content, ensure that the Grid & Construction Lines box is checked.
2. Select the column for which a stack is to be moved.

   The column is highlighted in one color, the column end nodes and middle node in another.
3. Select one end nodes for the column stack to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)
4. Click a grid or construction point to redefine the column stack end node position.

   The entire column is redrawn with the selected node moved to the new position.
5. Select the next end node. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)
6. Click a grid or construction point to redefine the end node position.

   The column is redrawn once more with the selected node moved to the new position.

How do I modify the position of an entire column?

Simply use the **Move** command located on the Edit ribbon.

Modeling Timber Beams

How do I create a single span timber beam?

1. Click **Model** > **Timber Beam** ( )
2. The beam will adopt properties from the currently displayed timber beam properties set. Review the property set and adjust as necessary.
3. Click where the beam is to start (Point 1).
   (What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

4. Click where the beam is to end (Point 2).

> *If you are prompted to pick another point (Point 3), this will be due to* **Continuous** *being selected in the beam property set. In order to create a single span beam simply click once again on Point 2, or press the Enter key.*

How do I create a series of single span timber beams?

In order to create beams using this method the floor or construction level must already contain the columns between which the beams will run. You must also use a 2D view of the floor or construction level to use this option.

1. Click **Model** > **Timber Beam** ( )
2. The beams will adopt their properties from the currently displayed timber beam properties set. Review the properties and adjust as necessary.
3. Move the cursor to one corner of an imaginary box which will encompass the columns between which you want to create beams.
4. Click and hold the left mouse button.
5. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating).
6. Once the rubber rectangle encompasses the columns between which you want beams creating, release the mouse button.
7. Beams are created between each adjacent pair of columns within the area you select.

How do I create a continuous timber beam?

1. Click **Model** > **Timber Beam** ( )
2. The beam will adopt its properties from the currently displayed timber beam properties set; ensure that the **Continuous** option is selected and adjust any other details as necessary.
3. Click where the beam is to start.
   (What are the points I can click to create a member?)

> *If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.*

4. Click the next node of your continuous beam.

If you are using a point along a beam or member, then click the beam or member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the beam.

5. Repeat step 4 until you have defined the penultimate node of your continuous beam.
6. Click where the beam is to end.
7. You are now prompted to pick another point, simply click once again on the same point, or press the Enter key in order to create the continuous beam.

You cannot define continuous beams which are curved either horizontally or vertically.

How do I create a curved timber beam?

1. Click **Model** > **Timber Beam** ( )
2. The beam will adopt its properties from the currently displayed timber beam properties set; ensure that the **Linearity** is set to **Curved Major** (if it is to curve vertically), or **Curved Minor** (if it is to curve horizontally), and an appropriate **Chord height** value is specified to define the curve.

You can control the direction in which horizontally curved beams curve. When you place the beam you select its start point and its end point. The beam always curves such that if you were to look along the theoretical line from the start point to the end point, then the curve on the beam will always lie to the right of that line.

For vertically curved beams, a negative chord height value can be used to reverse the curve direction.

3. Adjust any other details in the property set as necessary.
4. Click where the beam is to start (Point 1).
(What are the points I can click to create a member?)

If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

5. Click where the beam is to end (Point 2).

When defining beams which curve vertically, if you are using a 2D view of the floor or construction level, then the vertical projection of the beam is out of the plane of the floor or construction level that you are using. In this case you will not see the beam on the graphical display. You will need to change to a 3D view of the floor, construction level or the entire Structure in order to see them.

Horizontally curved beams always take the chord height defined in the property set, they do not curve automatically to fit on any curved grid line that you may have defined.

How do I modify the position of a beam?

You can modify an individual beam's position in both 2D and 3D views.

1. Select the beam to be moved.

   The beam is highlighted in one color, the beam end nodes and middle node in another.

2. Now select the beam end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

3. Click a grid or construction point to redefine the beam end node position.

   The beam is redrawn spanning to the new end position.

Modeling Timber Braces

Modeling Timber Braces

Tekla Structural Designer allows you to define braces to provide lateral stability to your structure. Alternatively you can specify rigid frames to achieve the same effect. You can also use both methods within a single structure should this be necessary.

How do I create a single timber brace?

1. Click **Model** > **Timber Brace** ( )

2. The brace will adopt properties from the currently displayed timber brace properties set. Review the property set and adjust as necessary.

3. Click where the brace is to start (Point 1).
   (What are the points I can click to create a member?)

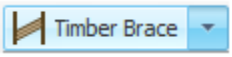
If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.

4. Click where the brace is to end (Point 2).

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

Although it is also possible to model the exact same brace arrangement using individual elements created using the simple 'Timber Brace' command, it is important to note that whilst the 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 EHF calculations with the result that the calculated EHFs are not correct.

How do I create an X, K, V or A brace?

1. Click **Model**, then from the 

drop list select the pattern required (X, K, V, or A).

2. The braces will adopt their properties from the currently displayed timber brace properties set. Review the properties and adjust as necessary.

3. Click to identify the bottom corner of the bay to be braced, (Point 1).

4. Click to identify the opposite bottom corner of the bay to be braced, (Point 2).

5. Click to identify the top corner of the bay to be braced, (Point 3). You will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the orientation is correct.

6. Click to identify the opposite top corner of the bay to be braced, (Point 4).

7. The brace pattern is created within the area you have selected.

How do I modify the position of a brace?

You can modify an individual brace's position in both 2D and 3D views.

1. Select the brace to be moved.

The brace is highlighted in one color, the brace end nodes in another.

2. Now select the end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to one of the members connecting to the node.)

3. Click a grid or construction point to redefine the brace end node position.

   The brace is redrawn spanning to the new end position.

Modeling Timber Trusses

Trusses are particular arrangements of members which *Tekla Structural Designer* calculates automatically for you. Once you have created a truss you can copy this throughout your model as necessary, and you can pick a truss and move it to the location you require.

How do I use the Timber Truss Wizard?

1. Click **Model** > **Timber Truss** ( )

2. Choose the truss shape from the list of standard truss patterns.

3. Click the truss Start Point.
   (What are the points I can click to create a member?)

> *If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.*

4. Click the truss End Point, then click **Next**

5. Specify the truss alignment parameters, then click **Next**

6. Specify the truss details, then click **Finish**

How do I use the Timber Space Truss Wizard?

1. Click **Model** > Timber Truss drop list.

2. Select **Space** from the drop list.

3. Choose either a linear or planar truss type and then specify the alignment and number of bays, then click **Next**

4. In the 2D or 3D View, click the truss location points:

   - for a linear truss - click the start and end points
   - for a planar truss - click the four corners,

5. Specify the height, then click **Next**

6. Define either a straight or curved truss geometry then click **Finish**

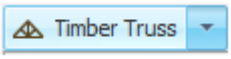
7. Specify the truss alignment parameters, then click **Next**

8. Specify the truss details, then click **Finish**

How do I define a Free Form Timber Truss?

A free form timber truss can be created by placing a series of truss members in the shape required.

1. Open a 2D Frame view in which the truss is to be created. (Note that free form trusses cannot be created in a 3D view, or 2D level view).

2. Click **Model** > 

drop list.

3. Select **Free Form** from the drop list.

4. Click where the first truss member is to start.

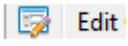
5. Click where the first truss member is to end.

6. Continue in the same way to pace each truss member until the truss geometry is complete.

7. Press **<Esc>** to finish.

How do I edit the geometry of an existing timber truss?

1. Hover the cursor over the truss to be edited so that it becomes highlighted.

2. Right-click and select 

... from the dropdown menu.

   The **Truss Wizard** is displayed.

3. Edit the properties as required and then click **OK**

How do I edit the section sizes, material grades and section orientations in an existing timber truss?

1. Hover the cursor over the truss to be edited so that it becomes highlighted.

2. Left click the truss to display the edit timber truss properties. Review the property set, adjust as necessary and then click **OK**

Panel modeling

Topics listed here relate specifically to modeling each of the different panel types found on the **Model** tab.

Modeling Roof Panels

Roofs are the medium via which loads placed on a sloping plane are decomposed back to the supporting structure. Area loads on roofs can act either vertically or normal to the roof plane.

How do I create a roof panel?

In order to define a roof panel you must have already defined the grid points with which to define its outline, and you must identify these in order.

To create the roof panel:

1. Click **Model** > **Roof Panel** ( )
2. Click the grid point where the panel is to start.
3. Click the grid point which identifies the next vertex of the panel.
4. Continue to click the grid points which define the perimeter of your panel until you reach the final point. You now have two options:
   - either double click this final point,
   - or single click the final point, and then click the first point of the panel again.
5. *Tekla Structural Designer* will create a roof panel between the points that you identified.

You cannot define a roof panel which does not lie in a single plane. If the points you define do not so lie, then Tekla Structural Designer will fail the panel during validation.

How do I edit the properties of a roof panel?

You can edit roof panel properties in both 2D and 3D Views.

1. Hover the cursor over the panel to be edited so that it becomes highlighted.
2. Left click to select it.

   The selected Roof panel properties are displayed in the Properties Window.

   If required, at this point you can add further roof panels to the current selection by holding the Ctrl key whilst clicking on each subsequent panel.
3. Edit these properties as required. Any changes are automatically applied to all the selected panels.

Modeling Wall Panels

Wall panels (also referred to as Wind Walls) are the medium via which loads calculated by the **Simple Wind Loading** generator and **Wind Wizard** are applied to your structure. Wall panels do not introduce any structural strength or stiffness of any kind to your structure. If you wish to introduce walls that resist gravity or lateral loads then you must model these by defining them as concrete walls.

How do I create a wall panel?

In order to define a wall panel you must have already defined the grid points which define the panel vertices, and you must identify these in order.

To create the wall panel:

1. Click **Model** > **Wall Panel** ( )
2. Click the grid point where the panel is to start.
3. Click the grid point which identifies the next vertex of the panel.
4. Continue to click the grid points which define the perimeter of your panel until you reach the final point. You now have two options:
   - either double click this final point,
   - or single click the final point, and then click the first point of the panel again.

The minimum number of grid points to define a wall panel is 3.

5. *Tekla Structural Designer* will create a wall panel between the points that you identified.

You cannot define a wall panel which does not lie in a single plane. If the points you define do not so lie, then Tekla Structural Designer will fail the panel during validation.

How do I edit the properties of a wall panel?

You can edit wall panel properties in both 2D and 3D Views.

1. Hover the cursor over the panel to be edited so that it becomes highlighted.
2. Left click to select it.

   The selected Wall panel properties are displayed in the Properties Window.

   If required, at this point you can add further wall panels to the current selection by holding the Ctrl key whilst clicking on each subsequent panel.
3. Edit these properties as required. Any changes are automatically applied to all the selected panels.

How do I create a wall panel with a parapet?

A wall panel with a parapet should be modeled in two parts:

- An ordinary wall panel is created up to the roof level.
- A second wall panel is created above the roof level which is then marked as a parapet.

Modeling in this way ensures the wind analysis correctly accounts for the parapet.

To create the two parts of the wall panel:

1. Open a Frame view in which the wall can be created.
2. Create the wall panel below the roof level as an ordinary wall panel.
3. Create the wall panel above the roof level as an ordinary wall panel. (A construction level may be required to define the top level of the parapet.)

To mark the upper wall panel as a parapet:

1. To ensure no existing wall panels are currently selected, press the Esc key.
2. Hover the cursor over the panel to be edited so that it becomes highlighted.
3. Left click to select it.

   The selected Wall panel properties are displayed in the Properties Window.

4. In the Properties Window select the check box to indicate the panel is a parapet wall.

How do I reverse a wall panel?

Wall panels are generally created so as to have their outer faces automatically pointing outwards. For certain complex building geometries, it may occasionally be necessary to alter the inner and outer face of the panel. Should this be necessary it is achieved using the **Reverse** option.

1. Click **Model** > **Reverse** ( ) (in the **Edit** group).
2. Click the wall panel to be reversed.

   The selected panel changes color to indicate it has been reversed.

Support, Analysis Element and Bearing Wall modeling

Modeling Supports

Supports allow you to constrain points in your structure vertically and rotationally. You can use supports to model connections to existing structures, so that you don't need to incorporate these in your current model. The fixity provided at an existing support can be changed by modifying the Support properties.

How do I create a single support?

Supports can only be placed at existing grid points.

To create the support:

1. Click **Model** > **Support** ( )
2. The support will adopt properties from the currently displayed Support properties. Review the property set and adjust as necessary.

3. Click the grid point where you want to create the support.

How do I create a rotated support using 3 Grid Points?

A local coordinate system can be applied to your supports, allowing the X and Y axes to be rotated about global Z.

1. Click **Model** > **Support** ( )
2. Before placing the support, ensure that the **3 Grid Points** option is selected in the Support properties.
3. Click the grid point where you want to create the support, then to define the support direction click a second point (to define the x direction) and click a third point (to define the y direction). Note that the z direction cannot be changed.

How do I create a spring support?

Linear and non linear supports are both created in a similar fashion.

1. Click **Model** > **Support** ( )
2. Before placing the support, edit its properties as follows:
   - Ensure that in the direction the spring is required to act, the degree of freedom is set to Free
   - Enter the stiffness properties as described in the Support properties.
3. Click the grid point where you want to create the support.

How do I edit the properties of supports?

You can edit support properties in both 2D and 3D views.

1. Hover the cursor over the support to be edited so that it becomes highlighted.
2. Left click to select it.

   The selected support's Support properties are displayed in the Properties Window.

   If required, at this point you can add further supports to the current selection by holding the Ctrl key whilst clicking on each subsequent support.
3. Edit these properties as required. Any changes are automatically applied to all the selected supports.

Modeling Analysis Elements

How do I create an analysis element?

1. Click **Model** > **Element** ( )

2. The member will adopt properties from the currently displayed Element properties. Review the property set and adjust as necessary.

3. Click where the member is to start (Point 1).
(What are the points I can click to create a member?)

> *If you are using a point along a member, then click the member first to see its points, then either click the point you want to use, or type the distance to the point from the start of the member.*

4. Click where the member is to end (Point 2).

How do I modify the position of an analysis element?

You can modify an individual analysis element's position in both 2D and 3D views.

1. Select the element to be moved.

   The element is highlighted in one color, the end nodes and middle node in another.

2. Now select the element end node that is to be moved. (Ensure the node is highlighted in the **Select Entity tooltip** when selecting, as opposed to any other member/element connecting to the node.)

3. Click a grid or construction point to redefine the element end node position.

   The analysis element is redrawn spanning to the new end position.

Modeling Bearing Walls

Bearing wall overview

Bearing Walls are used to provide resistance to vertical compressive loads (but not lateral loads) and to support certain other member types. Unreinforced masonry walls, for example, could be modelled as bearing walls.

Bearing Walls can be created over several story heights. A single wall is created with a uniform thickness between the base and top level.

The location of the wall is a determined from the alignment specified in the bearing wall properties, and the insertion points picked.

Bearing Walls do not perform the same function as Wall Panels, i.e. they do not act as a medium via which loads calculated by the Wind Wizard get applied to your structure. Therefore, in order for these wind loads to be applied you should create additional **Wall Panels** in the same physical locations as the **Bearing Walls**.

How do I create a bearing wall in a 2D View?

1. Ensure that you have defined the construction levels between which the wall will run and the grid points between which it will lie.

2. Click **Model** > **Bearing Wall** ( )

3. The wall will adopt properties from the Bearing wall properties set displayed in the Properties Window.

4. Check that the **Base Level** and the **Top Level** shown in the property set are correct, or adjust if necessary.

5. Check that the **Thickness** and other properties shown in the property set are also correct, and again adjust if necessary.

6. Click where the wall is to start (Point 1).
   (What are the points I can click to create a member?)

7. Click where the wall is to end (Point 2).

   Tekla Structural Designer creates the wall between the points clicked on.

How do I create a bearing wall in a Frame, or Structure View?

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

To create the wall:

1. Click **Model** > **Bearing Wall** ( )

2. The wall will adopt its properties from the *Bearing Wall properties* displayed in the Properties Window.

3. Check that the **Thickness** and other properties shown in the property set are also correct, and again adjust if necessary.

4. Click the first corner of the wall.

5. Either click:

- the opposite corner of the wall to create it in a single click, or
- the adjacent, and then the opposite corner of the wall to create it in using two clicks

  Tekla Structural Designer creates the wall between the points clicked on.

Sub Models

Structures can if required be sub-divided by horizontal planes between levels in order to create sub-models. Each sub-model can then be used to control the slab mesh parameters at the levels within it.

Until further sub-models have been introduced a structure will initially be treated as a single sub model, so that the same mesh parameters are applied globally to all meshed slabs.

Additional sub-models are created automatically (for every level specified as a Floor in the Construction Levels dialog) when either a Grillage chasedown or FE chasedown analysis is performed. They can also be defined manually from the Sub Models dialog.

The slab mesh parameters specified for a sub-model are then used in any analysis that requires the slabs to be meshed (i.e. load decomposition, building analysis with meshed floors, or FE chasedown analysis).

In both grillage and FE chasedown the analyses are performed one sub-model at a time. The topmost sub-model is analyzed first, its support reactions are then applied as loads for the analysis of the sub-model below. This sequence continues until all sub-models down to the foundation level have been analyzed.

For both of these analysis types you can if required edit the default support conditions applied to the sub-models.

Sub Model Characteristics

Definitions

The following definitions are applied:

- Sub-model - part of the 3D model between two horizontal sub model divide planes. Each sub model contains all members entirely between the two horizontal planes. For those columns, wall and braces severed by a divide plane, the stacks and brace length above the top plane are included in the sub-model as are the stacks and brace length below the lower plane.
- Sub-model divide planes - are horizontal planes that can be positioned (added, deleted or moved) in the 3D structure. These sub-model divide planes are notional and infinite. They are only permitted to cut through the structure where they only "sever"
  - Column stacks
  - Walls stacks
  - Steel Braces
- Sub model supports - the "artificial" supports as defined by the system for the column and wall stack ends and braces that pass through the divide planes
- Structure supports - the supports in the 3D structure as defined by the user
- Column and Wall stacks - the span length of a column or wall
- The volume of the sub-model - the 3D space that exists between any two adjacent sub-model divide planes

Basic rules of sub-models

There are a number of basic rules to assist understanding of sub-models.

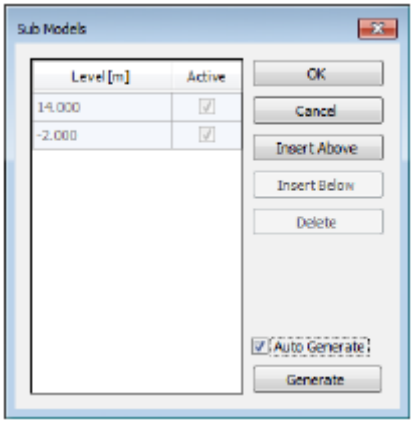
All sub-models when considered together form the complete structure - only some column stacks, some wall stacks and some braces (those split by sub-model divide planes) are in more than one sub-model. So:

- Every member in the 3D model is in at least one sub-model.
- A sub-model cannot contain a member already in a sub-model unless that member is a column, wall or brace divided by a sub-model divide plane.

- A sub-model must contain at least one beam member, one truss member or one slab panel.

Sub Models dialog

This dialog is accessed from the Structure Tree. It is used to split the structure into a continuous series of sub-models, working from the top of the building down to and including the foundations.



Fields

Level

Initially (before any analysis has been performed) only be two levels are displayed, one at a set distance (2m) above the top construction level and a second at the same distance below the base level. These cannot be changed. At this point there is a single sub-model comprising the whole structure.

Active

Only active levels act to divide the structure into sub-models. The top and bottom levels must always remain active, as there must always be at least one sub model.

Once intermediate levels have been inserted, you can choose to inactivate them if required, in which case the sub-model immediately above and the sub-model immediately below the level in question are merged into a single sub-model.

Auto Generate

If you have made any adjustments to the cutting planes, or cutting plane levels, or made any of them inactive you must also clear **Auto-Generate** - otherwise the changes you have made will be lost the next time the sub models are generated.

Buttons

Insert Above Click this button to insert a new level between the selected level and the one above. It defaults to being exactly half way between them but can be edited manually, provided it remains between the two.

Insert Below Click this button to insert a new level between the selected level and the one below. It defaults to being exactly half way between them but can be edited manually, provided it remains between the two.

Delete Deletes the selected level.

Generate This button can optionally be used to auto-generate default sub models for every level specified as a Floor in the Construction Levels dialog. You may choose to do this if you want to review the sub models prior to the first run of the analysis. It is optional as default sub models will be generated for you automatically when you run the analysis (provided the Auto Generate box on the dialog is checked). The Generate button can also be used to revert back to the default sub models at any time.

If you have made any adjustments to the cutting planes, cutting plane levels, or made any of them inactive you should avoid clicking the Generate button (even if the Auto Generate box is cleared) - as you will lose the edits that you have made.

Working with Sub Models

How do I open the Sub Models dialog?

To open from the Structure Tree:

- Double-click Sub Models

To open from the ribbon:

- Click **Model** > **Sub Models** ( )

How do I create Sub Models?

1. In the Sub Models dialog click **Generate** to automatically create the sub model levels, alternatively click **Insert Above** and/or **Insert Below** to manually create them.
2. For each new level if necessary modify the height above the base in the **Level** field.

How do I delete Sub Models?

1. In the Sub Models dialog select the sub model level to be removed then click **Delete**
2. Click **OK** to close the dialog.

How do I open a 3D view of an existing Sub Model?

1. In the Project Workspace, expand the **Sub Models** branch of the Structure Tree, then:
2. Double click a sub model
   (or right-click it to open a view of a particular type).

Sub Structures

If required, you can choose to place collections of elements into named sub-structures. This can prove useful in large models as individual sub-structures can then be differentiated by color and worked on in separate sub-structure views.

The purpose of sub structures is primarily to simplify modelling visualisation - the sub structures themselves are never analysed. They are distinct from, and not to be confused with Sub Models.

Sub Structure Characteristics

The basic features of sub structures are:

- Elements can exist in more than one sub structure.
- Not every element has to be in a sub-structure.
- Deleting a sub-structure does not delete the elements within it.

Working with Sub Structures

How do I create a new sub structure?

1. On the status bar, click **Review View**
2. Click **Review** > **Sub Structures**
3. In the **Review/Update** dialog box in the Properties Window select **Update Selected**
4. In the **Update Sub Structure** dialog box select **--New--**.
5. In the **Name** dialog box enter a name for the sub structure.
6. Choose a color for the substructure.
7. In the **Review View** click or box around the members to be included in the sub structure.
8. Change the Selection Mode to **only Remove**, or **Add or Remove**) if you need to remove members from the sub structure.

How do I add or remove existing elements in an existing sub structure?

1. On the status bar, click **Review View**
2. Click **Review** > **Sub Structures**
3. In the **Review/Update** dialog box in the Properties Window select **Update Selected**
4. In the **Update Sub Structure** dialog box select the sub structure to edit.
5. In the **Selection Mode** dialog box choose the edit operation required.

6. In the **Review View** click or box around the members to be edited.

How do I review all existing sub structures?

1. On the status bar, click **Review View**
2. Click **Review** > **Sub Structures**
3. In the **Review/Update** dialog box in the Properties Window select **Review All**

Each sub structure is shown in a different color in the display.

How do I open a 3D view of an existing sub structure?

1. In the Project Workspace, expand the **Sub Structures** branch of the Structure Tree, then:
2. Double click a sub structure
   (or right-click it to open a view of a particular type).

Measure commands

How do I Measure distances?

To measure the distance between any two points in the model:

1. Click **Model** > **Measure** ( )
2. Pick a node to define the start position.
3. Pick a second node to measure to.

The distance between the nodes is displayed on the current view - to clear the measurement press **[Esc]**.

How do I Measure Angles?

You can only measure angles in 2D Views:

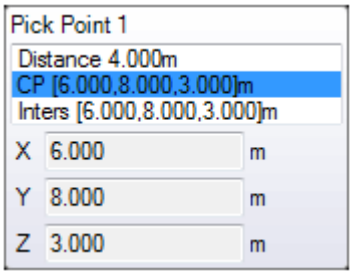
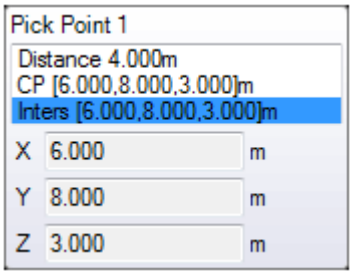
1. Click **Model** > **Measure Angle** ( )
2. Pick a node to define the arc center.
3. Pick a second node to define the start position.
4. Pick a third node to define the end position.

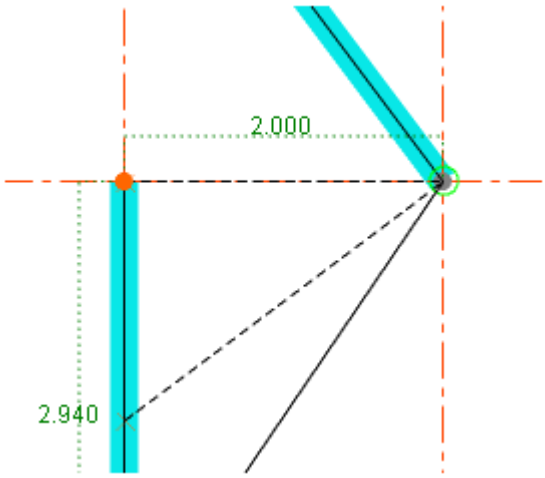
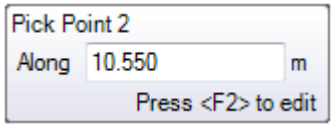
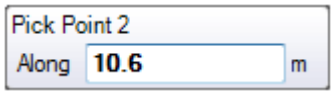
The clockwise angle between the start and end position is displayed on the current view - to clear the measurement press **[Esc]**.

What are the points I can click to create a member?

Beams, columns, braces, analysis elements etc. are collectively referred to as "members". In order to define any of these you must define the points between which they will lie.

The points that you click can be any combination of:

Point	Note
grid or construction points,	To pick a grid or construction point that lies part way along, or at the end of an existing member; simply move the cursor along the member to the point required. The dialog displays: • **CP** when it finds a grid or construction point at the end of a member.  • **Inters** when it finds a grid or construction point part way along a member  **To specifically use one or other of these points, press the down cursor key until the correct point is highlighted (as shown above) in the dialog.**
standard points on elements which you have already defined,	Position the cursor over an existing element and the standard points will automatically appear along it. These are at 0.250, 0.333, 0.500, 0.667, 0.750 of the span of the element.

special points on elements which you have already defined,	Special points are created automatically and allow you to create a member which is: • perpendicular to the first element that you picked, • perpendicular to the second element that you picked, • perpendicular to the X axis (that is whose ends lie at the same X distance from the origin of your model), • perpendicular to the Y axis (that is whose ends lie at the same Y distance from the origin of your model). Having picked the first point at the end of an existing element, to use one of these special points position the cursor over the second existing element and a dotted line indicates the perpendicular standard point location. 
a point at a specific distance (whose value you enter directly) along a member.	To pick a point at a specific distance, simply move the cursor along the element to the point required and click to specify it.  You can zoom in to achieve finer accuracy when moving along the element. Alternatively, by pressing <F2> you can type the exact distance directly and then press <Enter> to use it.  This option creates a grid intersection point over the top of the element. If you want to use the point again, then you can pick it as any other grid intersection point. Please note that this option does not create a new point along the element.

Model Validation

Your structure should be validated before it is analysed and designed. Validation checks all elements in your structure for a wide range of conditions. If any condition is not satisfied then *Tekla Structural Designer* tells you.

Validation is performed automatically whenever a design process is initiated from the Design ribbon, but it can also be run manually at any time.

The checks that are performed during validation can be set within Validation Settings.

At the end of the validation two types of message could be displayed in the Project Workspace Status Tree:

Errors

Error messages prevent the analysis from continuing until appropriate corrective action is taken.

Warnings

Although warning messages do not prevent the analysis process from continuing, it is very important that these messages are reviewed to decide whether any action is warranted.

How do I run model validation?

1. On either the **Model**, or **Load** tab, click **Validate** ( )

   The validation checks are performed and if any issues exist these are displayed as warning messages.

How do I control which conditions are considered during model validation?

1. Click **Home** > **Model Settings** ( )
2. On the **Validation** page check the validation conditions to be considered.

Edit commands

The Edit toolbar provides commands to copy, move, mirror and rotate the model (or a part of it).

It also has commands for splitting, joining, deleting and reversing the direction of members.

Edit toolbar

The **Edit** toolbar contains the following commands:

Button	Description

Copy	Copies existing selected objects to new locations in the model. See: Copying, moving and mirroring objects
Copy Loads	Copies existing selected loads to new locations or between loadcases. See: Copy Loads
Move	Moves existing selected objects to new locations in the model. See: Copying, moving and mirroring objects
Mirror	Make a reflected copy of existing selected objects about a given plane. See: Copying, moving and mirroring objects
Delete	This command is used to delete items.
Join	Joins members to make a longer continuous member. See: How do I Join Members (Make Continuous)?
Split	Splits existing continuous members. See: How do I Split Members?
Reverse	Reverses the front and back faces of a wind wall panel. The front of each wall should be facing outwards in order to correctly determine the wind direction relative to the wall. Also reverses the local X-axis of a beam. This can be useful for standardizing output. See: Reversing member axes and panel faces
Beam Lines	Joins existing concrete beams to make a continuous concrete beam. See: How do I Automatically Join All Concrete Beams (Make Continuous)?
Cutting Planes	Cutting Planes are useful tools for temporarily hiding parts of the model that obscure the area that you require to work on. By dragging a cutting plane so that it slices through your model everything that is on the positive side of the plane is hidden, leaving only items on the negative side still visible. See: Cutting Planes

Move Model	Moves the entire model to a new origin. See: How do I move the model to a new location?
Rationalize	Opens a dialog which allows you to: • Remove unused sloped planes and frames • Remove unused grid lines • Remove unused construction lines • Update all grid and construction lines to extend exactly 1'8" beyond the point they are required. See: Rationalizing the model
Move DXF Shadow	Moves the dxf shadow to a new origin. See: How do I move the dxf shadow?
Create Infills	Creates a pattern of infill members in the selected bay. See: Creating infill members

Related topics

- Commands on the ribbon toolbars

Copying, moving and mirroring objects

The **Copy**, **Move** and **Mirror** editing commands only becomes active after you have selected the objects that you want to edit.

How do I copy elements to a new location?

To copy existing elements to new locations in the model:

1. Select the members to be copied.
2. Click **Edit** > **Copy**

Alternatively you can select **Copy** *from the right-click context menu.*

3. In the **Copy Elements** dialog, you can either choose to clear elements from the set to be copied (if so required), or leave all elements selected in order to copy the entire selection.
4. Pick the Reference node.

5. In the Properties Window, select **Keep Elevation** if you want to copy the selected elements horizontally only (ignoring any difference in the elevation between the Reference node and the new position); alternatively clear **Keep Elevation** if you want to the copied elements to be moved both vertically and horizontally.

6. Pick the new position to move the members to.

7. The selected members are copied to the new location.

How do I copy and rotate elements to a new location?

To copy and rotate existing elements to new locations in the model:

1. Select the members to be copied.

2. Click **Edit** > **Copy**

Alternatively you can select **Copy** *from the right-click context menu.*

3. In the **Copy Elements** dialog, you can either choose to clear elements from the set to be copied (if so required), or leave all elements selected in order to copy the entire selection.

4. In the Properties Window enter the rotation required about the Z axis.

5. Pick the Reference node.

6. In the Properties Window, select **Keep Elevation** if you want to copy the selected elements horizontally only (ignoring any difference in the elevation between the Reference node and the new position); alternatively clear **Keep Elevation** if you want to the copied elements to be moved both vertically and horizontally.

7. Pick the new position to move the members to.

8. The selected members are copied to the new location and rotated.

How do I move elements to a new location?

To move elements to new locations in the model:

1. Select the members to move.

2. Click **Edit** > **Move** ( )

3. Pick the Reference node.

4. Pick the new position to move the members to.

5. The selected members are moved to the new location.

How do I move and rotate elements to a new location?

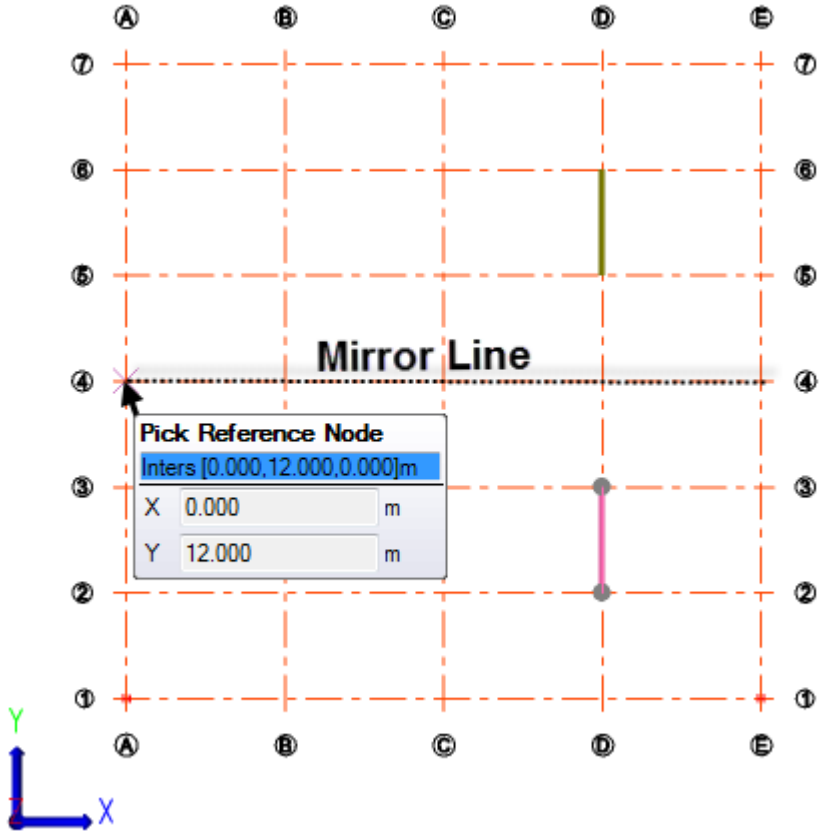
To move and rotate elements to new locations in the model:

1. Select the members to move.
2. Click **Edit** > **Move** ( )
3. In the Properties Window enter the rotation required about the Z axis.
4. Pick the Reference node.
5. Pick the new position to move the members to.
6. The selected members are moved and rotated to the new location.

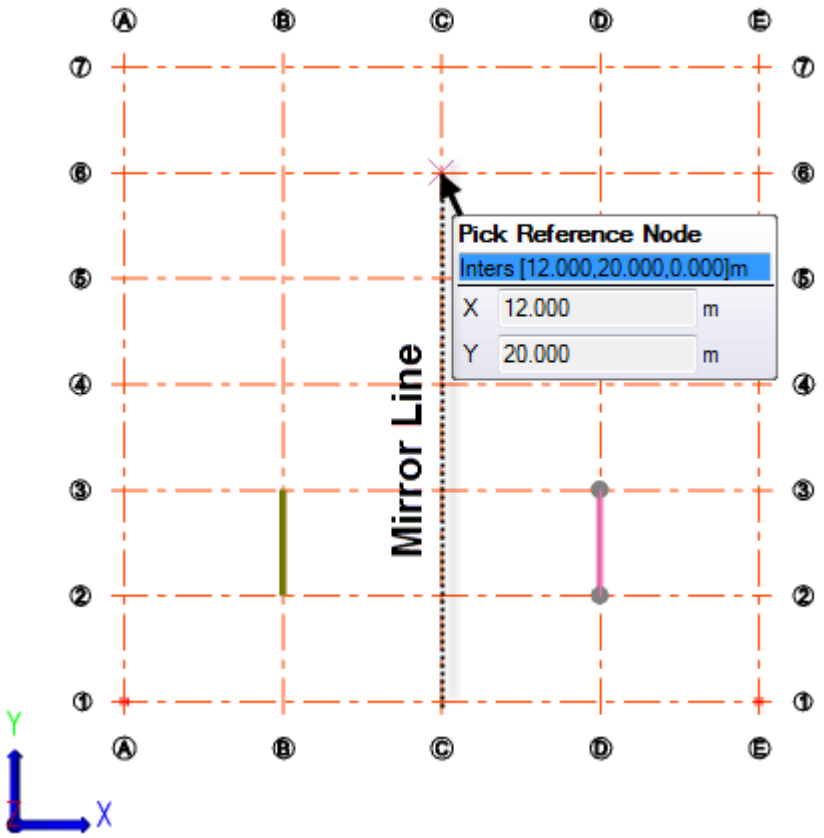
How do I mirror elements to a new location?

To copy existing elements by mirroring them about a specified plane:

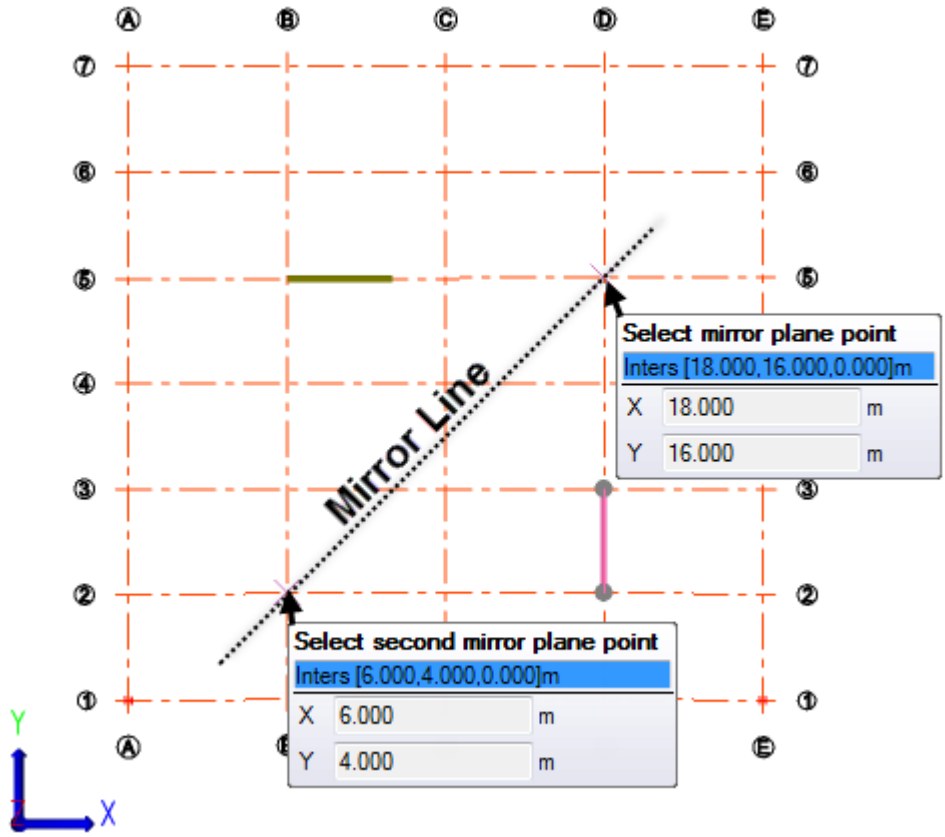
1. Select the members to be mirrored.
2. Click **Edit** > **Mirror** ( )
3. In the Properties Window select the mirror mode:
   - Along Y - mirrors about a global XZ plane defined by a single reference node



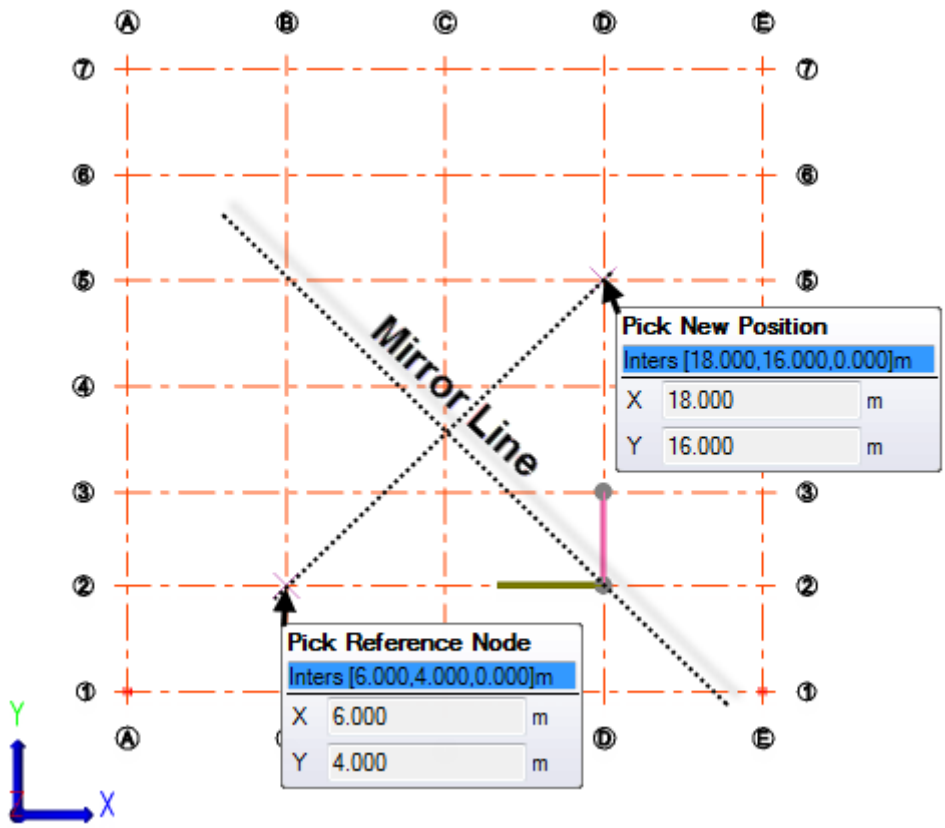
   - Along X - mirrors about a global YZ plane defined by a single reference node



- Pick Mirror Plane - mirrors about a Z plane defined by two reference nodes



- Pick Destination Point - mirrors about a Z plane perpendicular to two reference nodes



4. Pick the reference nodes.

5. The selected members are mirrored about the selected plane.

Copy Loads

The **Copy Loads** command can be used to copy member, panel and structure loads to different locations. It can also be used to copy loads between loadcases.

How do I copy all member loads from one span to another?

1. Click **Edit** > **Copy Loads**

2. In the Properties Window, select the **Copy Span Loading** mode.

3. In the Properties Window, choose to either copy loads in **Current loadcase** or **All loadcases**.

- If **Current loadcase** - you must also select the required loadcase from the Loading drop list.

A "Select loaded element span" prompt is displayed.

4. Click the span containing the member loads to be copied.

An "Apply loading onto span(s)" prompt is displayed.

5. Click a span that you want to apply the loads to.

The member loads are copied to the chosen span.

The source span and the destination span do not need to be the same length

6. Click additional spans as required to continue applying the loads.
7. Press <Esc> in order to select a different loaded element span to copy from, or press <Esc> twice to exit the command.

How do I copy just one member load if several are applied to the same span?

1. Click **Edit** > **Copy Loads**
2. In the Properties Window, select **Copy Member/Area Load** mode.

   A "Select load to be copied" prompt is displayed.
3. Click the member load to be copied.

   An "Apply load copy" prompt is displayed.
4. Click a span that you want to apply the load to.

   The member load is copied to the chosen span.

The source span and the destination span do not need to be the same length

5. Click additional spans as required to continue applying the load.
6. Press <Esc> in order to select a different load to copy, or press <Esc> twice to exit the command.

How do I copy panel area, level and slab loads?

1. Click **Edit** > **Copy Loads**
2. Select the required load case from the Loading drop list.
3. In the Properties Window, select Copy Member/Area Load mode.

   A "Select load to be copied" prompt is displayed.
4. Click the Area, Slab or Level load to be copied.

   An "Apply load copy" prompt is displayed.
5. Click a panel where the load is to be applied.

   The copied load is applied.

6. Click additional panels as required to continue applying the load.

7. Press <Esc> in order to select a different load to copy, or press <Esc> twice to exit the command.

How do I copy panel point, line and patch loads?

1. Click **Edit** > **Copy Loads**

2. Select the required load case from the Loading drop list.

3. In the Properties Window, select **Copy Plane Loads** mode.

   A "Select load to be copied" prompt is displayed.

4. Click the panel point, line or patch load to be copied.

   At this point you can add to the selection if required by clicking on additional loads; you can also remove a load from the selection by clicking on it once more.

   A red circle displays the original reference point for the selected loads.

5. To apply the loads either:

   - Click a panel node to define a new reference point at that node - the loads are applied at the same offset from the new reference point.
   - Click anywhere within a panel boundary at a different level to define a new reference point directly above/below the original reference point - the loads are applied at the same offset from the new reference point.

If you choose a new reference point that results in the loads being applied outside the panel area, they will not be applied to the model. In this situation a warning will be issued during validation

6. Click as required to define further reference points to continue applying the loads.

7. Press <Esc> in order to select different loads to copy, or press <Esc> twice to exit the command.

How do I copy structure loads?

1. Click **Edit** > **Copy Loads**

2. In the Properties Window, select **Copy Member/Area Load** mode.

   A "Select load to be copied" prompt is displayed.

3. Click the structure load to be copied.

   An "Apply load copy" prompt is displayed.

4. Click where you want to apply the load to.

   The structure load is copied to the new location.

5. Click additional locations as required to continue applying the load.

6. Press <Esc> in order to select a different load to copy, or press <Esc> twice to exit the command.

How do I copy loads to another loadcase?

1. Click **Edit** > **Copy Loads**

2. Select the loadcase to be copied from using the Loading drop list.

3. In the Properties Window, select **Copy Loads to another Loadcase** mode.

4. In the Properties Window, select the loadcase to **Copy loads to**.

   A "Select load to be copied" prompt is displayed.

5. Click the load to be copied.

   At this point you can add to the selection if required by clicking on additional loads; you can also remove a load from the selection by clicking on it once more.

6. To copy the loads to the loadcase displayed in the Properties Window press <Enter>

7. Press <Esc> in order to select a different load to copy, or press <Esc> twice to exit the command.

Joining and splitting members

The **Join** and **Split** commands are used for joining discontinuous members, and splitting continuous members of any material.

How do I Join Members (Make Continuous)?

Provided that the angle between two similar members is less than 45 degrees in both plan and elevation, you are able to join them to make a longer continuous member.

The "Join" command can be used to manually join concrete beams, even when the **Allow automatic join** *beam property is cleared. This is because this property only applies to the automatic joining that occurs during design process or when the "Beam Lines" command is run.*

1. Click **Edit** > **Join** ( )

2. Hover the cursor over the member that is to be joined to another member.

3. Both the original member and the member to which it will be joined become highlighted and the point at which they will be joined is indicated by a red dot. If the wrong end is being joined, move the cursor towards the other end of the member until the correct end is indicated.

The joined beams must have an end point in common. (If beams fail to join, they have probably been defined using different grid or construction lines at the end point in question).

4. Click the highlighted members to join them.
5. Pick additional members to join as required, or press **<Esc>** when complete.

How do I Split Members?

Members that have previously been joined can be split if required.

1. Click **Edit** > **Split** ( )
2. Hover the cursor over previously joined members that you want to split.
3. The member is highlighted and the point where it will be split is indicated by a red dot.

Only those members that are valid for splitting are highlighted when you hover the cursor over them.

4. Click the member to split it at the point indicated, or move the cursor further along the member to identify other points at which it could be split.

   If the member being split is a concrete beam, it is split into two separate beams; the first one having the **Allow automatic join end 2** beam property cleared, the second one having the **Allow automatic join end 1** beam property cleared. This prevents the two beams from being automatically made continuous once again when the model is designed.

Concrete Beam Lines

Concrete beam lines are formed automatically as part of the combined analysis and design process. However, if you would prefer to have greater control, you can choose to run the **Beam Lines** command manually. In this way you can verify that continuous beam lines are formed as you intend before proceeding with the design.

When run manually, the **Beam Lines** command is applied to all concrete beam members in the model, irrespective of whether they are selected or not.

Discontinuous concrete beam ends can only be joined if the **Allow automatic join** beam property is checked for the appropriate beam ends at the join.

The ends will then only be joined if all the following criteria are met:

- the angle in plan at which the two beams meet is less than the **Limiting join angle in plan** specified in **Model Settings > Beam Lines**,
- the angle in elevation at which the two beams meet is less than the **Limiting join angle in elevation** specified in **Model Settings > Beam Lines**,

- the amount by which the cross sectional areas of the two beams overlap is greater than the **Minimum section overlap** percentage specified in **Model Settings > Beam Lines**.

In addition, if either of the two beam ends being joined is pinned, they will not be joined unless the **Join pinned beam end** box is checked in **Model Settings > Beam Lines**.

How do I Automatically Join All Concrete Beams (Make Continuous)?

1. Click **Edit** > **Beam Lines** ( )

Reversing member axes and panel faces

How do I reverse the local x axis of a beam?

You can easily end up with beams running forwards and backwards if a consistent approach has not adopted during model creation when clicking on start and end nodes. This can result in confusing force diagrams in printed output.

The **Reverse** command can be used to manually standardize by flipping axes one beam at a time. In this way it is possible to force all beams to run left to right and bottom to top of a plan view.

To reverse a beam:

1. Click **Edit** > **Reverse** ( )
2. Click a beam to reverse its direction.

How do I reverse the outward face of a wind panel?

The front of each wall should be facing outwards in order to correctly determine the wind direction relative to the wall. To check this is the case, ensure that all the outward faces are displayed in the color assigned to "Wind Wall - Front". If a wall is facing in the wrong direction you should reverse it.

1. Click **Edit** > **Reverse** ( )
2. Click a wind wall panel to reverse its direction.

Cutting Planes

Six cutting planes initially form a cube around the extents of each model. By activating a cutting plane it can be moved inwards so that it slices through the model, everything to the positive side of the plane is temporarily hidden from view, making it easier to work on areas inside the model.

How do I activate or deactivate a cutting plane?

To activate a cutting plane:

1. Click **Edit** > **Cutting Planes** ( )
   Six (initially deactivated) cutting planes are shown.

2. Click a plane to activate it.
   The active cutting plane faces are shown in a different color. (By default blue indicates the positive side of the plane and red the reverse side).

To deactivate a cutting plane:

1. Click an active plane to deactivate it.
   Any part of the model that was previously hidden from view by the plane is re-displayed.

How do I move a cutting plane in order to hide part of the model?

When a cutting plane is active, a large arrow projects from it's center. The arrow is used to reposition the plane.

To move an active cutting plane:

1. Click the arrow at the center of the plane.
   The arrow is replaced by a line indicating the direction in which the plane can move.

2. Drag the plane to a new position, (or press <F2> to enter an exact distance).
   The cutting plane is redrawn at the new position; if it slices through the model, everything to the positive side of the plane is hidden from view.

How do I reshow the hidden part of the model?

Once a cutting plane has been positioned, there is no need to move it in order to reshow the hidden part of the model - all you need to do is deactivate the plane (by clicking on it once more).

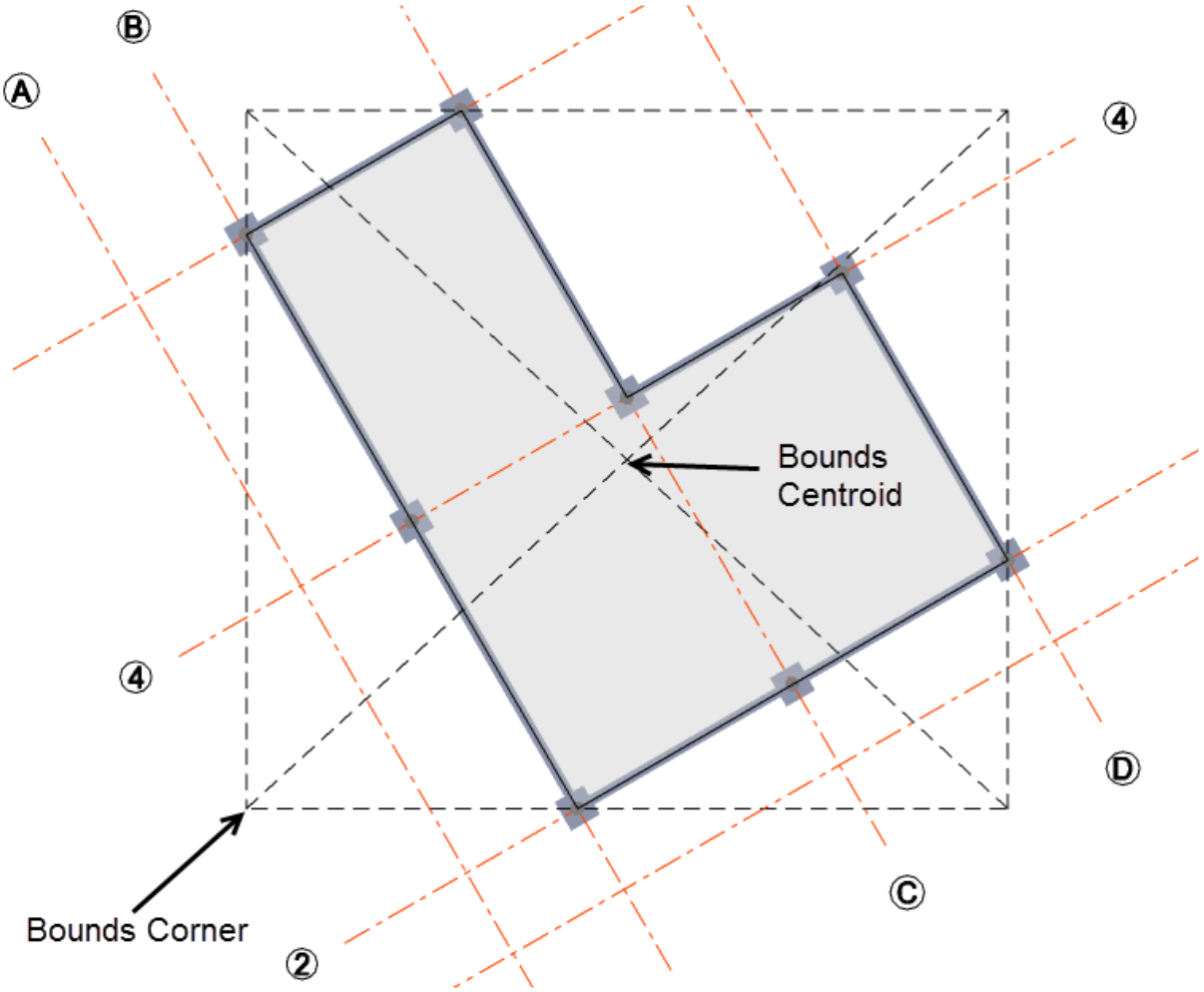
Moving the model, or the DXF shadow

The **Move Model** command is used to relocate the entire model to a new co-ordinate in the XY plane.

Similarly, the **Move DXF Shadow** command is used to relocate a dxf shadow, (this command only becomes available once a shadow has been imported).

How do I move the model to a new location?

In order to move the model you first have to decide which reference point to use. The reference points are determined by drawing an imaginary box aligned to X and Y around the extents of the model as shown below:



- Bounds Corner - equates to the lower left hand corner of the imaginary box.
- Bounds Centroid - equates to the centroid of the imaginary box.

1. Click **Edit** > **Move Model** ( )
2. Choose the reference point as either Bounds Corner or Bounds Centroid.
3. Enter the required target co-ordinate of the reference point.
4. Click **Move**

How do I move the dxf shadow?

1. Click **Edit** > **Move DXF Shadow**
2. Choose an existing point on the DXF Shadow as the reference point.
3. Click any other point to specify the new position of the reference point.

Rationalizing the model

The **Rationalize** command provides an efficient way to remove all unused sloped planes, frames, grids and construction lines from the model.

In addition, it can optionally update all grid and construction lines to extend only a short length (1'8") beyond the point they are required.

After running a summary of changes made is listed in the Process Window.

How do I remove unused sloped planes, frames, grids, and construction lines?

Sloped planes, frames, grids and construction lines are unused if they have no objects associated with them - provided they are unused they can be removed in a single operation as follows:

1. Click **Edit** > **Rationalize**
2. Select the check box(es) for the unused items you want to remove.

> **Planes** *removes unused sloped planes and unused frames also.*

3. Click **OK**

How do I update grid and construction lines to extend only a short length beyond where they are required?

1. Click **Edit** > **Rationalize**
2. Check **Shrink/Extend Grid & Construction Lines**
3. Click **OK**

> *All grids and construction lines in the model are updated to extend a fixed length, (*1'8" *) beyond the point they are required.*

Creating infill members

The **Create Infills** command can be used to rapidly place a pattern of infill members into selected bays in a Level, Sloped Plane, or Frame.

Only those bays with members attached to all sides can be selected.

How do I create infills?

To define the infill member properties and pattern:

1. Click **Edit** > **Create Infills** ( )

A "Select bay" prompt appears and the bays in which infills can be placed (i.e those that have members attached to all sides) are identified by shading.

2. Adjust the properties in the Properties Window -

Click **Define Beams** and choose:

- **By Number** - to specify the number of (equally spaced) members in the pattern
- **By max spacing**- to specify the spacing of members in the pattern

Click **Direction** and choose:

- **Perpendicular** - to have the members drawn perpendicular to the highlighted edge member
- **Parallel with left** - to have the members drawn parallel to the edge member that connects to **end 1** of the highlighted edge member
- **Parallel with right** - to have the members drawn parallel to the edge member that connects to **end 2** of the highlighted edge member

3. Define the **Element Parameters** as required.

If you have saved properties to a named Property Set, these can be recalled simply by selecting the set from the droplist at the top of the Properties Window.
See: How do I save properties to a named Property Set from the Properties Window?

To place the pattern in a single bay:

1. Hover the cursor over the required bay, adjacent to the required edge member.

A preview of the infill pattern (as it applies for the highlighted edge member) is displayed.

2. To change the orientation, either move the cursor to a different edge member, or adjust the general parameters in the Properties Window.

3. When both the preview and the Element Parameters are as required, click within the bay to create the pattern.

To place the pattern in multiple bays:

1. At the "Select bay" prompt drag a box to select the required bays.

A "Select reference bay" prompt appears.

2. Hover the cursor over one of the selected bays, adjacent to the required edge member.

A preview of the infill pattern (as it applies for the highlighted edge member) is displayed.

3. To change the orientation, either move the cursor to a different edge member, or adjust the general parameters in the Properties Window.

4. When both the preview and the Element Parameters are as required, click within the bay to create the pattern in the selected bays.

Loading Guide

Individual loads are applied to the model from the Load toolbar. The same toolbar is used for creating loadcases and combinations and also for accessing the Wind and Seismic Wizards.

Load toolbar

The groups listed below are displayed on the toolbar when a 2D or 3D View is active.
If any other view is active the "Panel Loads", "Member Loads", "Structure Loads" and "Validate" groups are hidden.

Structure group

The **Structure** group contains the following commands:

Button	Description
Loadcases	This command opens the **Loading** dialog at the Loadcases page. The same dialog is used to create either loadcases, combinations or envelopes. See: Working with Load Cases
Combination	This command opens the **Loading** dialog at the Combinations page. The same dialog is used to create either loadcases, combinations or envelopes. See: Working with Combinations
Envelope	This command opens the **Loading** dialog at the Envelopes page. The same dialog is used to create either loadcases, combinations or envelopes. See: Working with Envelopes

Update Load Patterns	If load patterns have been applied and the building geometry or loading is subsequently modified, to ensure the load patterns reflect theses changes you are required to update the pattern loads by clicking this button. See: Working with Load Patterns

Wind Load group

The **Wind Load** group contains the following commands:

Button	Description
Wind Wizard...	The Wind Wizard dialog is used for defining the information that is required in order to calculate the wind loading on the structure. See: How do I run the Wind Wizard? The Wind Wizard requires at least one wall panel or roof panel to have been defined before it becomes available.
Update Zones	If it is necessary to change the roof type of your structure, or if you alter your structure dimensionally, then changes to the existing wind zoning do not occur automatically. This is intentional, since you may wish to make subsequent alterations before you recalculate the zoning. Once you have completed your changes the Update Zones button is used to incorporate the changes and recalculate the zoning details. See: What happens if I make changes to my model?
Wind Loadcases	The Wind Loadcases dialog is used for defining the details of each wind loadcase. See: • How do I Define Wind Loadcases? • How do I add new Wind loadcases? • How do I delete Wind loadcases?
Delete Wind Model	Deletes the wind model data previously defined using the Wind Wizard. See: Working with the Wind Wizard
Decomposition	This command is used to decompose loads from 1-way and 2-way slabs onto supporting members. See: Working with the Wind Wizard

Snow Load group

The **Snow Load** group contains the following commands:

Button	Description
Snow wizard...	The Snow wizard dialog is used to automate snow loadcase generation. See: How do I run the Snow Wizard The Snow Wizard is not currently available for the Indian, Australian, or British Standard snow code variants.
Uniform Snow	Uniform Snow loads are applied manually. See: How do I manually apply a Uniform Snow load?
Valley Snow	Valley Snow loads are applied manually. See: How do I manually apply a Valley Snow load?
Local Drift Snow	Local Drift Snow loads are applied manually. See: How do I manually apply a Local Drift Snow load?
Update Snow Loads	See: Update Snow Loads
Delete Snow	Deletes the snow model data previously defined using the Snow Wizard. See: Delete Snow

Seismic Load group

The **Seismic Load** group contains the following command:

Button	Description
Seismic Wizard...	This wizard can be run to define the parameters required for determining the seismic loading and load cases on the structure. See: Working with Seismic Loads
Horizontal Spectrum	The Horizontal Spectrum is generated by the Seismic Wizard. This button displays the resulting spectrum for either direction 1 or direction 2, (the choice of direction being made in the Properties Window).

Delete Seismic	Deletes all the seismic data entered in the Seismic Wizard along with Horizontal Spectrum, the Seismic Loadcases and Seismic Load Combinations.

Decomposition group

The **Decomposition** group contains the following command:

Button	Description
Decomposition	This command is used to decompose loads from 1-way and 2-way slabs onto supporting members. See: How do I manually decompose slab loads for an individual construction level?

Panel Loads group

The **Panel Loads** group contains the following commands:

Button	Description
Point Load	This command is used to apply a point load to any panel. See: How do I create a point load?
Line Load	This command is used to a apply line load to any panel. See: How do I create a line load? Point Loads and Line loads can only be applied in 2D views (not 3D).
Patch Load	This command is used to apply a patch load to any panel. See: How do I create a patch load? Patch loads can only be applied in 2D views (not 3D).
Polygon Load	This command is used to apply a polygonal load to any panel. See: How do I create a polygonal load? Polygonal loads can only be applied in 2D views (not 3D).

Perimeter Load	This command is used to apply a load around the external perimeter of all slabs at a given level. See: How do I create a perimeter load?
Variable Patch Load	This command is used to apply variable patch load to any panel. See: How do I create a variable patch load? Variable patch loads can only be applied in 2D views (not 3D).
Area Load	This command is used to apply an area load to any panel. Area loads entirely cover an individual panel. See: How do I create an area load?
Variable Area Load	This command is used to apply a variable area load to any panel. Area loads entirely cover an individual panel. See: How do I create a variable area load?
Level Load	This command is used to apply a level load to all slabs at the current level. See: How do I create a level load?
Slab Load	This command is used to apply a slab load to all slab panels that constitute an individual slab. See: How do I create a slab load?

Member Loads group

The **Member Loads** group contains the following commands:

Button	Description
Full UDL	This command is used to apply a full UDL to a member (beam, column, brace etc.) See: How do I create a full UDL?
UDL	This command is used to apply a UDL to a member (beam, column, brace etc.) See: How do I create a partial length UDL or VDL?

VDL	This command is used to apply a VDL to a member (beam, column, brace etc.) See: How do I create a partial length UDL or VDL?
Trapezoidal Load	This command is used to apply a trapezoidal load to a member (beam, column, brace etc.) See: How do I create a trapezoidal load?
Point Load	This command is used to apply a point load to a member (beam, column, brace etc.) See: How do I create a point or moment load?
Moment Load	This command is used to apply a moment load to a member (beam, column, brace etc.) See: How do I create a point or moment load?
Torsion Full UDL	This command is used to apply a torsion full UDL to a member (beam, column, brace etc.) See: How do I create a torsion full UDL?
Torsion UDL	This command is used to apply a torsion UDL to a member (beam, column, brace etc.) See: How do I create a partial length torsion UDL or torsion VDL?
Torsion VDL	This command is used to apply a torsion VDL to a member (beam, column, brace etc.) See: How do I create a partial length torsion UDL or torsion VDL?

Structure Loads group

The **Member Loads** group contains the following commands:

Button	Description
Nodal Load	This command is used to apply a nodal load to a solver node. See: How do I create a nodal load?
Temperature Load	This command is used to apply a temperature load (a global rise in temperature) to individual elements/panels, selected elements/panels, or to all elements/panels. See: How do I create a temperature load?

Settlement Load	This command is used to apply a settlement load (a translation or a rotation) to a support. See: How do I create a settlement load?

Validate

Button	Description
Validate	This command is used to perform the model validity checks. See: Model Validation

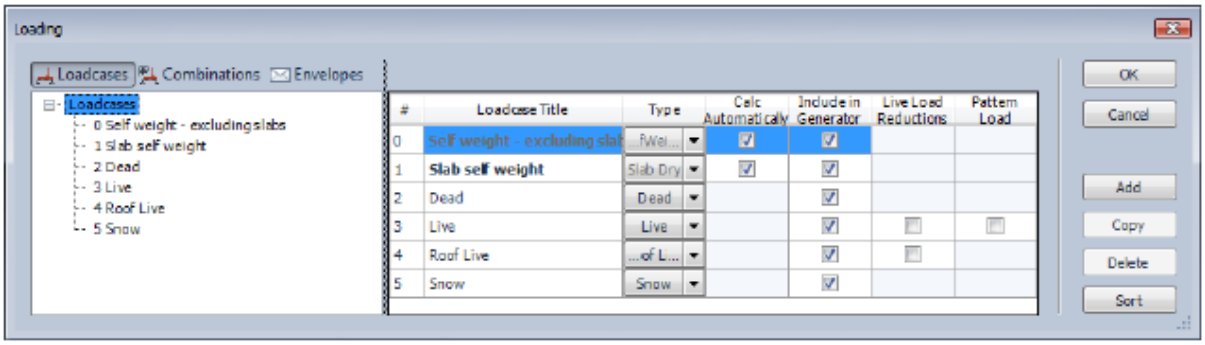
Working with Load Cases

Applied loads are defined within load cases, with each load case being assigned to one of the following load types:

- Self weight -excluding slabs
- Slab wet
- Slab dry
- Dead
- Live
- Roof Live
- Wind
- Snow
- Snow Drift
- Temperature
- Settlement
- Seismic

Loading Dialog Loadcases Page

This **Loading** dialog page is used to define loadcases.



Loadcases Pane

Within this pane you can:

- Click the **Loadcases** main branch to view a summary table of all the loadcases on the right hand side (as above).
- Click individual loadcases to view and edit settings specific to the selected loadcase.
- Click one of the loadcase name sub branches to edit the settings for the loadcase selected.

Loadcase Summary Table

Loadcase Title

Click to edit individual loadcase names.

Type

Select the type from the drop list.

Calc Automatically

When selected loads in the self weight loadcases are calculated automatically.

Include in Generator

When selected the loadcase will be included in the automatically generated combinations.

Live Load Reductions

When selected live load reductions will be applied in this loadcase.

Pattern Load

When selected pattern loading will be applied for this loadcase.

Individual Loadcases

Name

Click to edit the loadcase title.

Include in Generated Combinations

When selected the loadcase will be included in the automatically generated combinations.

Automatic Loading

When selected loads in the self weight loadcases are calculated automatically.

Prior to Brittle Finishes [%]

For Dead load types, specifies the percentage of dead load applied prior to brittle finishes.

Reductions

For live load types, select for reductions to be applied in this loadcase.

Pattern Load

When selected pattern loading will be applied for this loadcase.

Long Term [%]

For live load types, a proportion of the load is identified as **long term** (default 33%). This determines, for the Serviceability Limit State, what proportion of the load is used in conjunction with the long term modular ratio of concrete and what proportion with the short term modular ratio for the design of composite steel beams.

Category of imposed loads

For imposed load types, select the category from the drop list

Combination factors

For specifying psi factors appropriate to the loadcase.

Buttons

Loadcases Displays the Loadcases Pane and Loadcase Summary Table (as above).

Combinations Displays the Loading Dialog Combinations Page.

Envelopes Displays the Loading Dialog Envelopes Page.

OK Closes the dialog and saves changes.

Cancel Closes the dialog without saving changes.

Add Adds a new loadcase. See: How do I create load cases?

Copy Copies the selected loadcase.

Delete Deletes the selected loadcase.

Sort Renumbers all loadcases sequentially in the Loadcase Summary Table. (removing any gaps in the numbering).

How do I create load cases?

When you create a new model a **Self-weight - excluding slabs** load case is automatically created. You cannot access this load case as the loads it contains are automatically calculated from the objects in your structure.

Three further load cases are defined, although these are initially empty - you must define the loading in each of these manually. You will almost certainly need to create other load cases to contain the loads that your building must withstand.

1. Click **Load** > **Loadcases** ( )

   This will display the Loading Dialog Loadcases Page listing the current load cases which have been defined.

2. Click **Add** to create a new load case.

3. Enter the **Loadcase Title** and choose it's **Type**.

4. Choose whether it should be included when load combinations are automatically generated.

5. Click **OK** to create the load case.

6. Your new load case will now appear in the list of load cases available in the Loading drop list.

How do I indicate that reductions apply to live (imposed) load cases?

When you create a load case of type Live, an option is provided to allow for live load reductions to be calculated in accordance with the reduction percentages specified in Model Settings.

This option is not provided if the type is set to "Roof Live".

To activate reductions in an existing imposed load case:

1. Click **Load** > **Loadcases** ( )

   This will display the Loading Dialog Loadcases Page listing the current load cases which have been defined.

2. Select the live load case to which the reductions are to be applied.

   Provided that the Type is Live a **Reductions** check box is displayed on the dialog.

3. Select **Reductions**.

4. Click **OK** to close the dialog.

How do I renumber all loadcases?

When loadcases are deleted from the Loading dialog the remaining loadcases retain their original loadcase number.

To renumber the remaining loadcases in sequence (with no gaps):

1. Right-click the Loadcases branch in the Loading tab of Project Workspace.

2. From the right-click menu, choose Renumber.

How do I add loads into a load case?

Loads can be defined from either a 2D, or a 3D view (depending on the load type). Loads cannot be defined in a solver view, or a solver model data view.

Working with Combinations

Combinations allow you to assemble sets of load cases, applying the appropriate factors for the strength and service condition. These factors are specific to the design code that is being worked to.

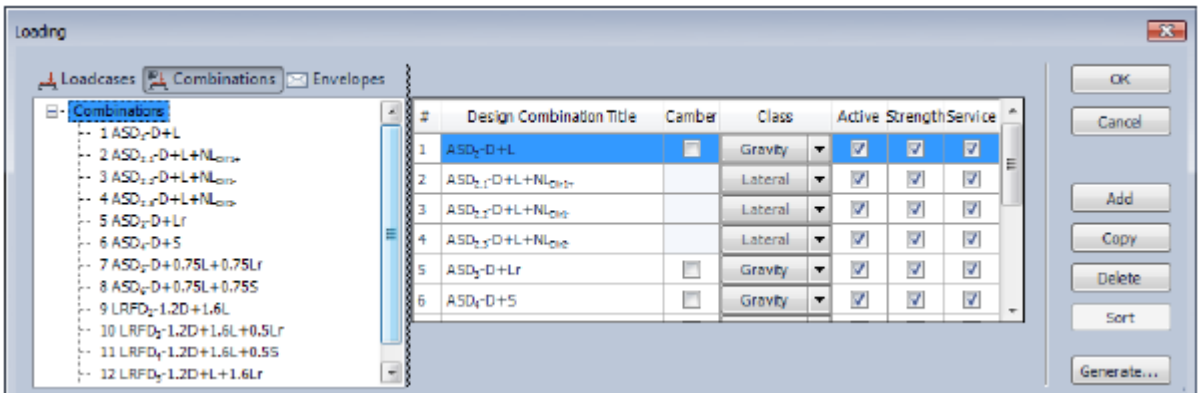
Combinations fall into five types with a number of options being available for each of these types:

Combination Type	Description	Active/Inactive	Strength	Service
Construction Stage	Only required for design of composite beams	Not applicable	Not applicable	Not applicable
Gravity	Consists of gravity loads only (Self Weight, Dead, Slab Dry, Slab Wet, Live, Roof Live and Snow)	On/Off	On/Off	On/Off
Lateral	In addition to gravity load contains lateral loads due to Notional Loads or Wind	On/Off	On/Off	On/Off
Seismic	Consists of gravity and/or lateral loads as well as seismic load cases	On/Off	On	Not applicable
Vibration Mass	Only required if a vibration analysis is performed	On/Off	Not applicable	Not applicable

- Active/Inactive - switches the combination on/off for analysis/design and vibration
- Strength - If the Strength column is not ticked and the combination is active, it is not assessed for design
- Service - If the Service column is not ticked and the combination is active, the combination is not assessed for deflection.

Loading Dialog Combinations Page

This **Loading** dialog page is used to define load combinations.



Combinations Pane

Within this pane you can:

- Click the **Combinations** main branch to view a summary table of all the combinations on the right hand side (as above).
- Click one of the combination name sub branches to edit the settings for the combination selected.

Combination Summary Table

Design Combination Title

Click to edit individual combination names.

Camber

If camber has been applied to any steel beams as a proportion of the deflection, this box is used to specify the combination on which the camber calculations should be based.

Class

Select the class from the drop list.

Active

Switches the combination on/off for analysis/design.

Strength

If the Strength box is not ticked and the combination is active, it is not assessed for design.

Service

If the Service box is not ticked and the combination is active, the combination is not assessed for deflection.

Buttons

Loadcases Displays the Loading Dialog Loadcases Page.

Combinations Displays the Combinations Page (as above).

Envelopes Displays the Loading Dialog Envelopes Page.

OK Closes the dialog and saves changes.

Cancel Closes the dialog without saving changes.

Add Adds a new combination. See: How do I create load combinations manually?

Copy Copies the selected combination.

Delete Deletes the selected combination.

Generate... Runs the load combination generator. See: How do I generate load combinations automatically?

How do I generate load combinations automatically?

The easiest way to create load combinations is to generate them automatically.

If a Construction Stage combination is required it should be created manually.

1. Click **Load** > **Combination** ( )

   This will display the Loading Dialog Combinations Page listing any current combinations which have been defined.

2. Click **Generate** to initiate the combination generator.

3. Specify the **Initial Parameters** as required then click **Next**

4. Depending on the number of combination types that were requested on the initial parameters page, one or more pages of combinations are generated. Review each of these by clicking the **Next/Previous** buttons and amend if required.

5. Click **Finish** to save the load combination.

6. Your new load combination will now appear in the list of combinations available in the Loading drop list.

7. To review the factors and options that have been applied, select the combination name from the list on the left side of the dialog.

How do I create load combinations manually?

Load combinations can be created manually if required, with a default factor being used for each load case as it is added to the combination.

1. Click **Load** > **Combination** ( )

   This will display the Loading Dialog Combinations Page listing any current combinations which have been defined.

2. Click **Add** to create a new combination.

3. Enter the **Combination Title** and choose its **Class**.

4. Choose whether it should be **Active**.

5. Choose if it is a **Strength** combination.

6. Choose if it is a **Service** combination.

7. Click the combination name in the left hand panel of the dialog to display the available loadcases.

8. Select each load case in turn to be included and click the right arrow button to copy it into the combination.

9. Click **OK** to save the load combination.

10. Your new load combination will now appear in the list of combinations available in the Loading drop list.

11. To review the factors and options that have been applied, re-display the **Loading** dialog then select the combination name from the list on the left side of the dialog.

How do I create a Vibration Mass combination?

1. Click **Load** > **Combination** ( )

   This will display the Loading Dialog Combinations Page listing any current combinations which have been defined.

2. Click **Add** to create a new combination.

3. Enter the **Combination Title** and choose **Vibration Mass** from the **Class** drop list.

4. Click the combination name in the left hand panel of the dialog to review/amend the loadcases to be included.

5. Click the **Applied mass** tab to set the directions to be considered, and if required, to specify the level below which mass can be ignored.

6. Click the **Second order effects** tab to define the amplifier to be applied.

7. Click **OK** to save the load combination.

8. Your new load combination will now appear in the list of combinations available in the Loading drop list.

How do I renumber all combinations?

When combinations are deleted from the Loading dialog the remaining combinations retain their original number.

To renumber the remaining combinations in sequence (with no gaps):

1. Right-click the Combinations branch in the Loading tab of Project Workspace.

2. From the right-click menu, choose **Renumber**.

Working with Envelopes

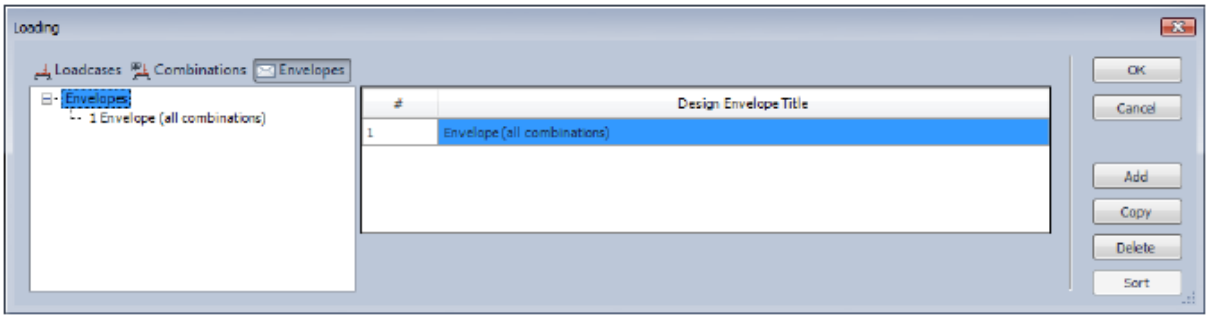
Envelopes allow you view analysis results for multiple combinations at the same time, with the maximum positive and negative values being displayed along each member from any combination included in the envelope.

If you have defined patterned load combinations, you only need to include the "base case" pattern combination in the envelope - all pattern combinations derived from the base case are automatically included.

Gravity, Lateral and Seismic combination classes can all be included in the same envelope. Seismic RSA combinations should not be included in envelopes as the result cannot currently be displayed.

Loading Dialog Envelopes Page

This **Loading** dialog page can be used to define envelopes.



Envelopes Pane

Within this pane you can:

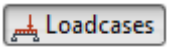
- Click the **Envelopes** main branch to view a summary table of all the envelopes on the right hand side (as above).
- Click one of the envelope name sub branches to edit the settings for the envelope selected.

Envelope Summary Table

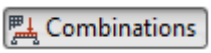
Design Envelope Title

Click to edit individual envelope names.

Buttons



Displays the Loading Dialog Loadcases Page.



Displays the Loading Dialog Combinations Page.

Displays the Envelopes Page (as above).

OK	Closes the dialog and saves changes.
Cancel	Closes the dialog without saving changes.
Add	Adds a new envelope. See: How do I create envelopes?
Copy	Copies the selected envelope.
Delete	Deletes the selected envelope.

How do I create envelopes?

Envelopes are created manually if required as follows:

1. Click **Load** > **Envelope** ( )

   This will display the Loading Dialog Envelopes Page listing any current envelopes which have been defined.

2. Click **Add** to create a new envelope.

3. Enter the **Envelope Title**.

4. Click the envelope name in the left hand panel of the dialog to display the available combinations.

5. Select each combination in turn to be included and click the right arrow button to copy it into the envelope.

6. Click **OK** to save the envelope.

7. Your new envelope will now appear in the list of envelopes available in the Loading drop list.

If you have defined patterned load combinations, you only need to include the "base case" pattern combination in the envelope - all pattern combinations derived from the base case are automatically included.

Gravity, Lateral and Seismic combination classes can all be included in the same envelope. Seismic RSA combinations should not be included in envelopes as the result cannot currently be displayed.

Working with Load Patterns

The factors for the loaded and unloaded spans when pattern loading is applied to live loads are specific to the design code being worked to.

Overview of Load Patterns

Basic Steps of the Load Patterning Process

The live loads patterning process in *Tekla Structural Designer* is applied as follows:

1. Firstly, you must set the individual live load cases to be patterned as required - these are the "fully loaded" pattern load cases.
   (See: How do I set an imposed load case to be patterned?)

2. Next, set the gravity combinations containing live load cases to be patterned as required - these are the "base case" pattern combinations.
   (See: How do I set a combination to be patterned?)

3. With patterns set as above, then after load decomposition the building analysis will automatically set up the pattern cases for concrete beams in Dir1 and Dir2 directions - 10 pattern cases (beams live loads on/off by span) for each pattern gravity combination - 5 for beams along Dir1 and 5 for beams along Dir2. Pattern cases are also set up for continuous steel beams if the option for this has been activated in Model Settings> Loading.
   (See: Pattern Load Cases)

4. By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to meshed slabs should be manually patterned using engineering judgement; this is achieved by toggling the loading status via **Update Load Patterns**.
   (See: How do I update load patterns?)

Clarification of the Slab and Beam Load Pattern Application Rules

Slab load patterning only applies where 2-way slabs have been meshed in the solver model, i.e.

- in **FE Chasedown Analysis**
- in **3D Analysis** where 2-way slabs are set as meshed

This means that the slab load pattern setting has no effect on:

- 1-way spanning slabs
- 2-way slabs not meshed in **3D Analysis** (because the slab loads are being decomposed to beams and walls prior to creation of the solver model)
- 2-way slabs in **Grillage Chasedown Analysis** (again, because the slab loads are being decomposed to beams and walls prior to creation of the solver model)

Consequently in **3D Analysis** and **Grillage Chasedown Analysis**:

- When a beam is set to "Full Load" it will receive the full decomposed load from adjacent unmeshed 2-way slabs irrespective of whether the slabs themselves are set to "Full Load" or "Min Load".
- When a beam is set to "Min Load" it will receive the min decomposed load from adjacent unmeshed 2-way slabs irrespective of whether the slabs themselves are set to "Full Load" or "Min Load".

How do I set an imposed load case to be patterned?

Load patterning is only applied to those live load cases in a combination that have individually been set to be patterned.

1. Click **Load** > **Loadcases** ( )

   This will display the Loading Dialog Loadcases Page.

2. Select the live loadcase name to be patterned from the list on the left side of the dialog.

   The factors and options that have been applied are displayed in the dialog.

3. Ensure the **Pattern Load** option is selected.

4. Click **OK**

How do I set a combination to be patterned?

Only Gravity combinations can consider pattern loading - lateral and seismic combinations do not consider pattern loading.

1. Click **Load** > **Combination** ( )

   This will display the Loading Dialog Combinations Page listing any current combinations which have been defined.

2. Select the combination name to be patterned from the list on the left side of the dialog.

3. The factors and options that have been applied are displayed in the dialog.

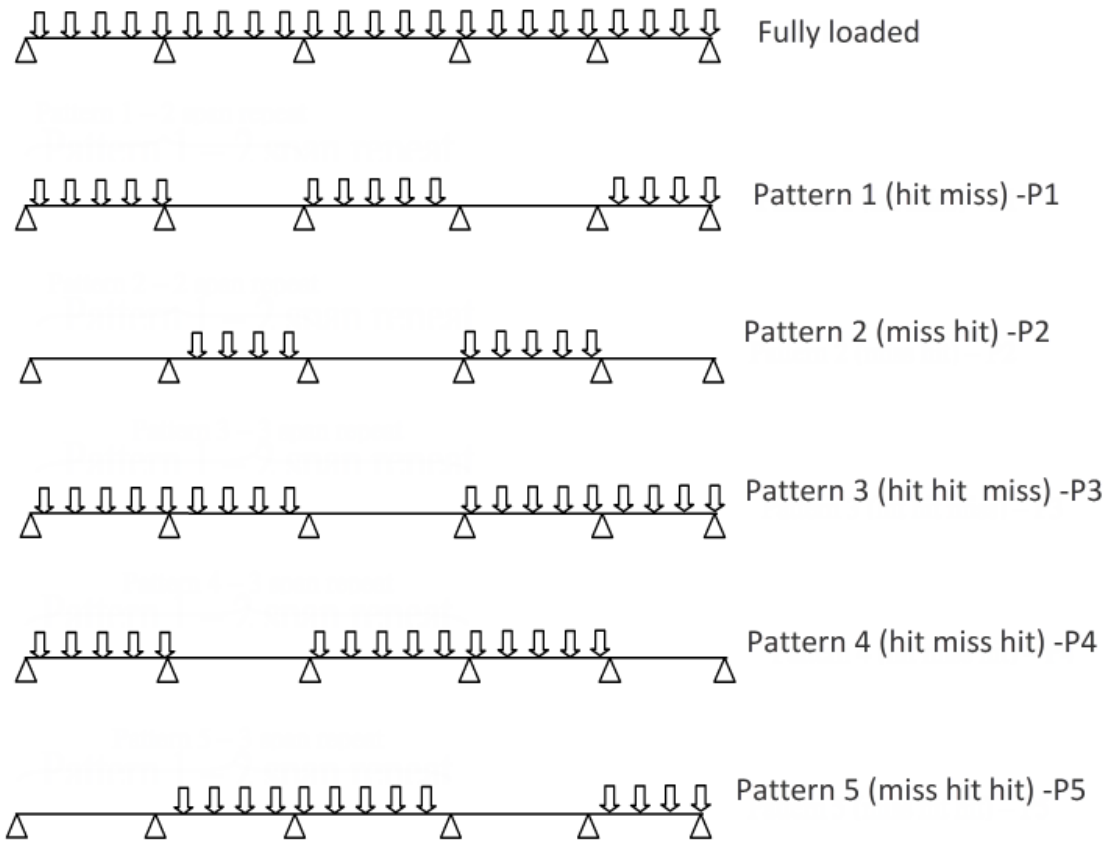
4. Ensure the Pattern option is selected.

5. Click **OK**.

You can, if you wish, use pattern loading for every gravity combination, however you should be aware that this could create many additional combinations.

Pattern Load Cases

The five load patterns are:



In effect, a pattern combination containing pattern live load cases results in 11 combinations - the "base case" combination and 10 pattern combinations derived from it.

If the building geometry or loading is subsequently modified, to ensure the load patterns reflect these changes you are required to update the pattern loads.

There are NOT separate pattern combinations for beams and slabs, there is just one set of 10 pattern combinations associated with each "fully loaded" pattern combination.

How do I update load patterns?

Beam load patterning is a completely automatic process; the only way you can influence whether a beam is subject to "full" or "min" load is by editing the continuous beam lines.

Conversely, slab load patterning is a completely manual process. Hence if slab load patterning is required, e.g. for slab design purposes, you are required to manually update the load patterns using engineering judgement.

To manually specify slab load patterning:

1. Click **Load** > **Update Load Patterns**
2. In the **Properties Window**, select each pattern in turn and review the loading status of the beams and slabs.

3. Click a slab to toggle its loading status.

Working with the Wind Wizard

A **Wind Wizard** is used to automate the wind modeling process. This uses databases where appropriate (depending on the wind code) to determine the appropriate wind details for your structure location and then calculates the appropriate wind loading details for it in accordance with the chosen wind code.

The Wind Wizard is not currently available for the AS:1170.2 wind code variant.

Having defined the wind directions in which you are interested, the appropriate wind zones on the roofs and walls of your structure are automatically calculated. You can set the type of each roof to achieve the correct zoning, and can then tailor the zoning to account for particular features in more detail, if you so require.

The wind modeling process can automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases the appropriate wind pressures are calculated on each zone. You can then combine these wind loadcases into design combinations in the usual way.

You will find that the determination of the wind speeds, the pressures and the zones is rigorous but the final wind loads adopted are your responsibility.

Running the Wind Wizard

The **Wind Wizard** is run to define the information required for the wind analysis of the structure. It uses this information to automatically determine the wind zoning and external pressure coefficients for the roofs and walls.

How do I run the Wind Wizard?

The Wind Wizard requires at least one wall panel or roof panel to have been defined before it becomes available.

The **Wind Wizard** guides you through the process of defining the information that is required in order to calculate the wind loading on the structure.

1. Click **Load** > **Wind Wizard...** ( )

   A Wind Wizard appropriate for the selected wind loading code will start, and you can use its pages to define the necessary information.

2. Run through the various pages of the Wind Wizard by clicking **Next>**. When you reach the end of the Wind Wizard the **Next>** button will change to **Finish**. Click this button to terminate the Wind Wizard.

After running the Wind Wizard the roof and wall zones can be reviewed for each wind direction.

How do I add a new wind direction?

Simply run the **Wind Wizard** again. You will find that all the existing details are maintained. You can simply add the new direction in the **Results** page.

You will also need to add new wind loadcases and design combinations to incorporate the wind loading for the new direction into your calculations.

How do I delete a wind direction?

Simply run the **Wind Wizard** again. You will find that all the existing details are maintained. You can delete the line for the direction you no longer require in the **Results** page.

You will also need to update your wind loadcases and design combinations to remove the details for the wind direction you have removed from your calculations.

How do I delete the entire wind model?

Occasionally you may require to delete the entire wind model and start the wind modeling process from scratch.

1. Click **Load** > **Delete Wind**

All the previously defined wind directions and wind loadcases are removed.

Reviewing wind zones and wind zone loads

After the wind model has been established by the wind wizard, you can graphically display the wind zones and loading that apply for a particular wind direction by opening the appropriate **Wind View**.

Zone Loads toolbar

The **Zone Loads** toolbar contains the following commands:

Button	Description
Wind Direction	This control is used to switch between the Wind Direction displayed in the active Wind View. See How do I open a Wind View?
Wind Zones	Graphically displays the Wind Zone information for the current direction. See How do I view the Wind Zones?
Zone Loads	Graphically displays the Zone load information for the current direction. See How do I view the Wind Zone Loads?

Roof Type	Graphically displays the Roof Type.
Std. Table No.	Graphically displays the Standard Table No.
Crosswind Breadth	Graphically displays the Crosswind Breadth.
Multibay	Graphically displays Multibay information.
Name	Graphically displays the wind zone name.
-ve Cpe	Graphically displays the -ve Cpe values for the zones.
+ve Cpe	Graphically displays the +ve Cpe values for the zones.
Net Pressure	Graphically displays the net pressure in zones.
Area	Graphically displays the area in zones on net pressure diagrams.
Applied Load	Graphically displays the applied load in zones on net pressure diagrams.
Cpi	Graphically displays the Cpi in zones on net pressure diagrams.
Cpe	Graphically displays the Cpe in zones on net pressure diagrams.
Beneficial (Head Code Eurocode or BS)	Zone loads that have been manually set to "Beneficial" are labelled accordingly and shown with reduced net pressures.

How do I open a Wind View?

Wind Views are only available after the **Wind Wizard** *has been run to establish the wind model.*

In order to view the wind zones and loading that apply for a particular wind direction you have to first of all open the appropriate **Wind View**.

1. On the **Wind** tab of the **Project Workspace**, right-click the **Wind Direction** in which you are interested.

2. Click **Open view** from the context menu.

   This simultaneously opens the chosen wind direction view and switches to the **Zone Loads** ribbon from where you can review wind zone data graphically.

How do I view the Wind Zones?

1. Open a Wind Direction view which shows the details for the wind direction in which you are interested.
2. Click **Show** > **Wind Zones**
3. The Wind Direction view shows the zones that are applied to the structure for this wind direction.

How do I edit a Wind Zone?

1. Open a Wind Direction view which shows the details for the wind direction in which you are interested.
2. From the Wind tab of the **Project Workspace**, expand the **Wall Zones** or **Roof Zones** branch for the wind direction in question.
3. In the required branch, right-click the panel containing the zone to be edited and choose **Edit Zones...**
4. Initially the dialog shows the automatically calculated values for the particular wind zone. You can change these on a zone-by-zone basis by removing the check against **Std.**
5. Make the edits as required in the dialog and click OK.

How do I view the Wind Zone Loads?

Once you have defined your wind loadcases you can view the wind zone loading that is generated.

1. Open a Wind Direction view which shows the details for the wind direction in which you are interested.
2. Click **Show** > **Zone Loads**
3. In the Loading drop list pick the particular wind loadcase in which you are interested from the list of wind loadcases.
4. The Wind Direction view shows the loads that are applied to the structure for this loadcase in this wind direction.

Related topics

- How do I Define Wind Loadcases?

How do I change a Wind Zone Load?

To change a wind zone load:

1. Open a Wind Direction view which shows the details for the wind direction in which you are interested.
2. Click **Zone Loads** from the **Show** group.
3. In the Loading drop list pick the particular wind loadcase in which you are interested from the list of wind loadcases.
4. The Wind Direction view shows the loads that are applied to the structure for this loadcase in this wind direction.
5. Click the particular zone that you want to change and you will see the **Wind Load Zone Data** dialog.
6. Initially this dialog shows the automatically calculated values for the particular wind zone. You can change these on a zone-by-zone basis should this prove necessary.
7. In order to make changes to the automatically calculated values, you must first remove the check against **Default**. The values used to determine the load and the **Beneficial** load box (see below) then become active.
8. Enter the details that you require for these values, Once these settings meet with your approval click **OK** and the zone will be updated to show the result of your changes.

   Beneficial

   When working to Eurocodes, by checking this box the nett pressure for such loads is reduced to zero (see Clause 7.1.2 (2) Note b.), or is reduced if working to BS6399-2.

   When working to BS6399-2, by checking this box the nett pressure for such loads is reduced by 40%.

What happens if I make changes to my model?

If it is necessary to change the roofs or walls of your structure, either because you change the face of a wall, or the type of a roof, or if you alter your structure dimensionally, then changes to the existing wind zoning do not occur automatically. This is intentional, since you may wish to make subsequent alterations before you recalculate the zoning. Once you have completed your changes it is simple to incorporate them and recalculate the zoning details.

1. Make the changes that you require to your model. If you can see existing zoning information, then this information will be removed for any walls or roofs you modify.
2. To reinstate the zoning,

   - Click **Load** > **Update Zones** ( )

3. The wind zoning calculations run in the background. once these are complete the new zoning layout for your structure will be shown.

If you have defined your own zone layout for any roof or wall, and you make changes to that roof or wall, then the existing zones will be maintained. It is your responsibility to update these appropriately.

Wind loadcase definition

It is not practical to automatically determine critical combinations and thus required wind loadcases. Thus you control the generation of wind loadcases manually.

How do I Define Wind Loadcases?

Once you have defined the basic wind data for your model, and calculated the wind zoning using the **Wind Wizard**, you can then define the wind loadcases you want to investigate.

1. Click **Load** > **Wind Loadcases**

2. Use the Wind Loadcases dialog to **Add** the details of each wind loadcase individually, or alternatively if you want to selectively generate standard wind loadcases for the wind directions you defined in the Wind Wizard, then click **Auto**

If you want to use the Auto option, then you must do so before you have defined any other wind loadcases. Once a wind loadcase exists Auto is unavailable.

How do I add new Wind loadcases?

Once you have defined wind loadcases it is an easy matter to create additional ones if you require them.

1. Click **Load** > **Wind Loadcases**

2. In the **Wind Loadcases** dialog click **Add** to create a line for your new loadcase in the table of wind loadcases.

3. Enter the appropriate details directly into this line.

4. Repeat steps 2 and 3 for each new loadcase you wish to create.

5. Once your list of loadcases is complete click **OK** to close the **Wind Loadcases** dialog.

6. Don't forget to update / augment your design combinations to take account of your new wind loadcases.

How do I delete Wind loadcases?

Once you have defined wind loadcases it is an easy matter to delete ones that you no longer require.

1. Click **Load** > **Wind Loadcases**
2. In the **Wind Loadcases** dialog click the line relating to the wind loadcase you want to delete.
3. Click **Delete** to permanently delete this wind loadcase.
4. Repeat steps 2 and 3 for each loadcase you wish to delete.
5. Once your list of loadcases is complete click **OK** to close the **Wind Loadcases** dialog.
6. Don't forget to update your design combinations to take account of the wind loadcases you have deleted.

Working with Simple Wind

The **Simple Wind** loading method is an alternative to using the **Wind Wizard.** It allows you to input wind loads directly without having to construct a wind model.

In order access Simple Wind, you must first create and then select the loadcase in which the simple wind loads are to be added.

How do I manually create a loadcase for Simple Wind?

1. Click **Load** > **Loadcases**
2. Use the Loadcases dialog to **Add** a new loadcase, give it an appropriate name and ensure the **Type** is set as **Wind**, then click **OK**

How do I create a Simple Wind Load?

1. Ensure you are in a 3D Structure View, not a 2D View.
2. In the Loading toolbar, select a manually created loadcase of type Wind, (i.e not one that has been automatically created by the Wind Wizard).

   The Simple Wind command should now be available.
3. Click **Load** > **Simple Wind**
4. Click to define the start point of the load width.
5. Click to define the end point of the load width.

   The **Simple Wind Loading** dialog is displayed - this is used to define a wind load profile within the width specified, up the height building.

   The initial default enables you to specify a single area load from the lowest level of the building, up to the highest level that contains a rigid diaphragm.

6. Enter the area load required.

7. To create a stepped profile up the height of the building, use the buttons on the right of the dialog to specify the levels at which the load changes, (these do not have to coincide with actual levels in the building), then enter the load required at each level.

8. Once your load is complete click **OK** to close the dialog.

The load does not have to begin at the lowest level in the building. If you have one or more levels below ground, you can simply adjust the value of the lowest level in the dialog.

How do I edit Simple Wind load vertical properties?

The Simple Wind dialog can be redisplayed for an existing simple wind load as follows:

- Double click anywhere within the graphical representation of the simple wind load.

How do I edit Simple Wind load width?

1. Select the Simple Wind Load to be edited.

   The load becomes highlighted, and the two nodes that define the load width are shown.

2. Select one of the two nodes, then click a new location for the node.

   The load is regenerated between the two nodes.

Working with the Snow Wizard

For both the ASCE7 and Eurocode snow codes variants a **Snow Wizard** can be used to automate snow loadcase generation.

The wizard is run in order to set up:

- basic snow data
- snow loadcases
- snow loads within the balanced (undrifted) snow loadcases

Snow loads in drift snow loadcases are not set up automatically and must therefore be manually applied.

The Snow Wizard is not currently available for the Indian, Australian, or British Standard snow code variants.

How do I run the Snow Wizard

The **Snow Wizard** consists of two pages: the first page is used to define the basic data for the chosen snow loading code, the second page is used to set up the actual number of loadcases to be generated.

1. Click **Load** > **Snow Wizard...**

   A Snow Wizard appropriate for the selected snow loading code will start.

2. On the first page define the basic data, then click **Next>**

3. On the second page specify the loadcases to be generated and the number of drift directions to be considered.

4. Click **Finish** to complete the Snow Wizard.

After running the Snow Wizard the loadcases that have been set up can then be reviewed.

Manually applying Snow Loads to Snow Loadcases

While the loads in the balanced (or undrifted) snow loadcases are created automatically, the loads in the drift cases have to be applied manually as Uniform Snow, Valley Snow, or Local Drift Snow loads.

Snow loads that are not appropriate for the currently selected loadcase are unavailable.

How do I manually apply a Uniform Snow load?

1. Select the required snow loadcase in the Loading drop list.

2. From the Snow Load group on the ribbon, select **Uniform Snow**

3. Review the load details displayed in the Properties Window and adjust as necessary.

Note the Windward/Leeward option for Uniform Loads in the Unbalanced cases.

4. Click anywhere on the relevant roof panel to apply the load.

How do I manually apply a Valley Snow load?

1. Select the required snow loadcase in the Loading drop list.

2. From the Snow Load group on the ribbon, select **Valley Snow**

3. Select a roof panel that identifies the Left (L) roof panel of the valley in which the load is to be applied, (the Right (R) panel is automatically determined).

The valley load is automatically calculated.

If required a user override can be applied to the calculated load by selecting the load and selecting **Override load value** in the Properties Window.

If you rerun the Snow Wizard any user overrides that have been applied are maintained.

How do I manually apply a Local Drift Snow load?

It is not necessary to have walls in order to use drift loads - you can create a drift against a completely free edge of a roof panel if required.

1. Select the required snow loadcase in the Loading drop list.
2. From the Snow Load group on the ribbon, select **Local Drift Snow**
3. In the Properties Window choose the Drift type: (**Step**, or **Parapet or projection**).
4. In the Properties Window define the appropriate height and roof lengths for the drift.
5. Review the other load details displayed in the Properties Window and adjust as necessary.
6. Hover the cursor over the required roof panel edge along which the drift will act. The closest appropriate edge becomes highlighted.
7. Pick the load start position on the panel.
8. Pick the load end position.

If required a user override can be applied to the calculated load by selecting the load and selecting **Override load value** in the Properties Window.

If you rerun the Snow Wizard any user overrides that have been applied are maintained.

How do I apply a user override to a snow load?

User overrides can be applied if required for snow load values and drift lengths.

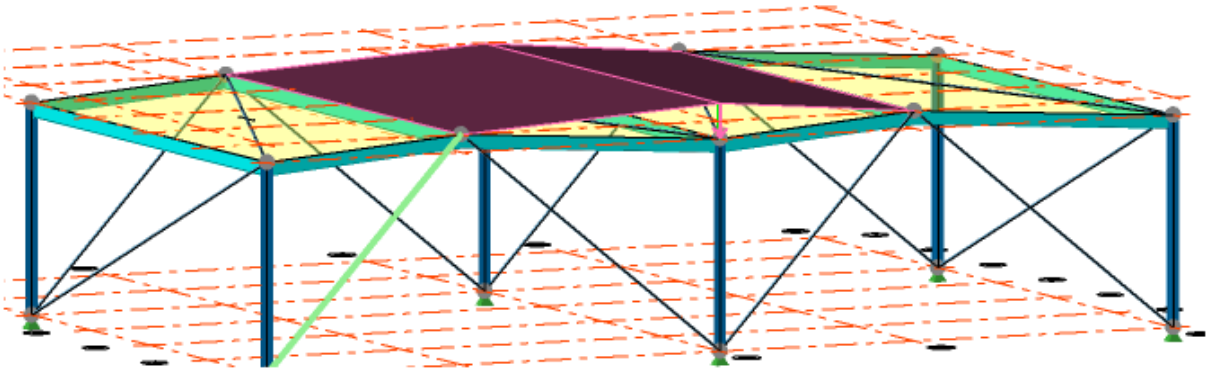
1. Select the snow load.

2. In the Properties Window check the appropriate override box.
3. Enter the override value.

If you rerun the Snow Wizard any user overrides that have been applied are maintained.

A note about the graphical display of Snow Loads

In the graphic displays it is the loading **magnitude** rather than the **depth** of loading that is depicted. This should be borne in mind when reviewing the load that has been calculated on structures such as the low pitched valley model shown below:



Although the above appears to indicate snow load rising above the ridges, because it is the **magnitude** being depicted this is not actually the case.

Update Snow Loads

Having run the Snow Wizard, if the roof geometry is subsequently modified the snow loads would need to be recalculated. Click **Update Snow Loads** on the Loads ribbon to achieve this.

Delete Snow

Occasionally you may require to delete the entire snow model and start the process from scratch. This is done as follows:

- Click **Load** > **Delete Snow**

The basic data that was previously entered in the Snow Wizard is re-set and the loadcases set up by the Snow Wizard are deleted.

Working with Seismic Loads

A **Seismic Wizard** is provided to enable the definition of all the parameters required to determine the seismic loading and load cases so that a seismic analysis can be run on the structure.

The Seismic Wizard is not currently available for the AS:1170.4 code variant.

Running the Seismic Wizard

The **Seismic Wizard** guides you through the process of defining the information required for the seismic analysis of the structure. It uses this information to the seismic loading (a force and moment, applied to every floor and construction level).

How do I run the Seismic Wizard?

1. Click **Load** > **Seismic Wizard...** ( )

   A wizard appropriate for the selected loading code will start, and you can use its pages to define the necessary information.

2. Run through the various pages of the wizard by clicking **Next>**. When you reach the end the **Next>** button will change to **Finish**. Click this button to terminate the Seismic Wizard.

3. After running the Seismic Wizard the Combination Generator is displayed to allow you to set up the seismic load combinations.

Related topics

- ASCE7 Seismic Wizard
- UBC 1997 Seismic Wizard
- Eurocode EN1998-1:2004 Seismic Wizard

How do I display the Horizontal Design Spectrum?

After running the Seismic Wizard the Horizontal Design Spectrum can be viewed.

1. Click **Load** > **Horizontal Design Spectrum**

   You can switch between the Dir 1 and Dir 2 spectrum via the Properties Window.

How do I delete Seismic Loads?

Occasionally you may require to delete the seismic loads and start the seismic modeling process from scratch.

1. Click **Load** > **Delete Seismic**

All the previously defined seismic loads are removed.

Load Decomposition

The way in which load decomposition is performed (if at all) depends on the how the slabs are modeled and how they are spanning.

For 1-way spanning slabs:

- slab loads are always decomposed directly on to supporting members before 3D analysis is performed.

For 2-way spanning slabs:

- load decomposition is not required at those levels where the option **mesh 2-way slabs in 3D Analysis** is applied.
- at other levels, before 3D analysis can be performed, any loads applied to 2-way slabs have to be decomposed back on to the supporting members.
- to achieve this, a separate FE load decomposition is carried out automatically prior to the 3D analysis.

Where load decomposition has occurred, the resulting decomposed loads (rather than the original slab loads) get applied to the 3D analysis model.

For verification purposes you can elect to display the decomposed loads in the 3D views.

Although load decomposition is carried out automatically, (when you click to analyze the structure), the **Decomposition** command provides an optional way to perform the same task manually - you may elect to do this in order to manually check that loads are being decomposed as you intend prior to running the analysis. In the context of big/complex models this can be a useful time saver, particularly as manual decomposition can be carried out on a level by level basis.

How do I manually decompose slab loads for an individual construction level?

1. Open a 2D view of the level required and display it in 3D.

Related topics

- How do I display a 2D view in 3D?

2. Click **Load** > **Decomposition** ( )

   Provided the relevant Scene Content settings are activated, (i.e. **Slabs** > **Mesh** and **Loading** > **Decomposed**), you should then see an FE mesh generated within the slab panels and the resulting decomposed loads applied to the beams.

How do I manually decompose slab loads for all the required levels?

1. Open the Structure 3D view.

2. Click **Load** > **Decomposition** ( )

How do I view the decomposed loads (either with, or without load values)?

1. Open a 3D view physical view of the model, (or a 2D view displayed in 3D).
2. In Scene Content expand the **Loading** group and select **Decomposed**.
3. To view the decomposed load values click in the cell to the right of the **Decomposed** option and select both **Geometry** and **Text**.
4. To view the decomposed load without values click in the cell to the right of the **Decomposed** option and clear **Text**.
5. Select the load case from the Loading drop list.

Decomposed loads do not exist at levels where the option **mesh 2-way slabs in 3D Analysis** *has been applied.*
At levels where this is not the case you can view the decomposed loads, (but you cannot see any shell results from the FE load decomposition).

Working with Panel Loads

Panel loads are used to apply loads to slab panels (a.k.a "slab items"), roof panels and wall panels as follows:

- **Point**, **Line, Patch** and **Polygonal Loads** can be applied anywhere within individual or across multiple slab, roof or wall panels. They can only be applied in 2D views (not 3D).
- **Area Loads** entirely cover a slab, roof or wall panel.
- **Slab Loads** entirely cover all slab panels in a parent slab.
- **Level Loads** entirely cover all parent slabs in a level.

A parent slab can consist of slab panels that are physically separated from each other, however they must be on the same level.

How do I create a point load?

1. Open a 2D view of the level in which you want to apply the load.
2. Select an appropriate load case from the Loading drop list.
3. Click **Load** > **Point Load** ( ) (in the **Panel Loads** group).
4. Review the load details displayed in the **Properties Window** and adjust as necessary.

5. Click a reference node from which the load position can be offset - this can be the start/end point of any member at the level.

> *To select a reference node,* **Points** *must be switched on in* **Scene Content***.*

6. Click again to specify the load position graphically, or press **<F2>** to enter the position via the keyboard.

> *If the slab/panel is moved by manually selecting & re-positioning the slab/panel nodes then the point load does not move with the slab/panel.*
>
> *However, if any of the grid lines defining the reference node are moved then the load will move also.*

How do I create a line load?

1. Open a 2D view of the level in which you want to apply the load.
2. Select an appropriate load case from the Loading drop list.
3. Click **Load** > **Line Load** ( )
4. Review the load details displayed in the **Properties Window** and adjust as necessary.
5. Click a reference node from which the load position can be offset - this can be the start/end point of any member at the level.

> *To select a reference node,* **Points** *must be switched on in* **Scene Content***.*

6. Click to specify the load start position graphically, or press **<F2>** to enter the position via the keyboard.
   Note that this is an offset (X, Y) from the chosen reference node.
7. Click, or press **<F2>** to specify the load end position.
   Note that when entered via keyboard this is an offset (X, Y) from the load start position.

How do I create a patch load?

1. Open a 2D view of the level in which you want to apply the load.
2. Select an appropriate load case from the Loading drop list.

3. Click **Load** > **Patch Load** ( )

4. Review the load details displayed in the **Properties Window** and adjust as necessary.

5. Click a reference node from which the load position can be offset - this can be the start/end point of any member at the level.

> *To select a reference node,* **Points** *must be switched on in* **Scene Content***.*

6. Click to specify a corner position of the load graphically, or press **<F2>** to enter the corner position via the keyboard.

7. Click, or press **<F2>** to specify the size.
(Specified as the offset dimensions from the corner position.)

8. Click, or press **<F2>** to define the rotation.

How do I create a polygonal load?

1. Open a 2D view of the level in which you want to apply the load.

2. Select an appropriate load case from the Loading drop list.

3. Click **Load** > **Polygon Load**

4. Review the load details displayed in the **Properties Window** and adjust as necessary.

5. Click a reference node from which the load position can be offset - this can be the start/end point of any member at the level.

> *To select a reference node,* **Points** *must be switched on in* **Scene Content***.*

6. Click to specify the load start position graphically, or press **<F2>** to enter the position via the keyboard.
Note that this is an offset (X, Y) from the chosen reference node.

7. Click to specify a corner position of the load graphically, or press **<F2>** to enter the corner position via the keyboard.

8. Continue to specify the remaining corner positions in the same way.

9. When the polygon is complete press **<Enter>** (or, click once more on the first corner defined) to create the load.

Once a polygonal load has been created, its shape can subsequently be graphically edited by:
- selecting the load,
- clicking on one of its vertices,
- clicking again to move the vertex to a new position

How do I create a perimeter load?

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. Select an appropriate load case from the Loading drop list.
3. Click **Load** > **Perimeter Load**
4. Review the load details displayed in the **Properties Window** and adjust as necessary, note in particular:
   - **Create as line loads** - creates the load as series of separate line loads along each external edge. (Leave cleared to created as a single load object.)
   - **Ignore openings** - creates the load around external perimeters only but not around internal opening perimeters.
5. Click any slab or mat item.

   The load is created around the external perimeter of every continuous area of slabs at the identified level.

How do I create a variable patch load?

Variable patch loads are not just restricted to quadrilaterals - they can be any polygonal shape.

1. Open a 2D view of the level in which you want to apply the load.
2. Select an appropriate load case from the Loading drop list.
3. Click **Load** > **Variable Patch Load** ( )
4. Review the load details displayed in the **Properties Window** and adjust as necessary. (At this point you are only required to specify three load values to describe the variable load, their locations are picked graphically at a later stage.)
5. Click a reference node from which the load position can be offset - this can be the start/end point of any member at the level.

To select a reference node, **Points** *must be switched on in* **Scene Content**.

6. Click to specify the load start position graphically, or press **<F2>** to enter the position via the keyboard.
   Note that this is an offset (X, Y) from the chosen reference node.

7. Click to specify a corner position of the load graphically, or press **<F2>** to enter the corner position via the keyboard.

8. Continue to specify the remaining corner positions in the same way.

9. When the polygon is complete press **<Enter>** (or, click once more on the first corner defined).

10. Click one of the corners of the polygon to pick the position of Load 1.

11. Click another corner to pick the position of Load 2.

12. Click another corner to pick the position of Load 3.

Once a variable patch load has been created, its shape can subsequently be graphically edited by:
- selecting the load,
- clicking on one of its vertices,
- clicking again to move the vertex to a new position.

How do I create an area load?

1. Open a 2D view of the level in which you want to apply the load.

2. Select an appropriate load case from the Loading drop list.

3. Click **Load** > **Area Load** ( )

4. Review the load details displayed in the **Properties Window** and adjust as necessary.

5. Click the panel to which the load will be applied.

How do I create a variable area load?

1. Open the 3D view, frame view or sloped plane view in which you want to apply the load.

Variable Area Load can only be applied to non-horizontal slab and roof panels, as well as wall panels and concrete walls - hence they can't be applied in a 2D view.

2. Select an appropriate load case from the Loading drop list.
3. Click **Load** > **Variable Are Load** ( )
4. Review the load details displayed in the **Properties Window** and adjust as necessary.
5. Click the panel to which the load will be applied.

How do I create a slab load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Slab Load** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Click any slab panel within the parent slab in order to apply the load to all panels within the parent slab.

How do I create a level load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Level Load** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Click any slab panel in order to apply the load to all slabs within the level.

How do I edit a load?

1. Select the load to be edited.
2. Review the load details displayed in the **Properties Window** and adjust as necessary.

How do I delete a load?

1. Ensure the load case containing the load is displayed in the Loading drop list.
2. Click **Delete** ( ) on the **Quick Access toolbar**.
3. Select the load to be deleted and click to delete it.

Working with Member Loads

Member Loads are used to apply loads to one dimensional members (beams, columns, braces etc.)

How do I create a full UDL?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Full UDL** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Click anywhere along the element to apply the load.

How do I create a partial length UDL or VDL?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **UDL** ( ), or **VDL** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Hover the cursor over the element until it is displayed with a number of special points along it.
5. Pick the load start position either by clicking on one of the special points, or by clicking at any given distance along the element, or by pressing **<F2>** to enter the position by the keyboard.
6. Click again, or press **<F2>** to specify the load length.

How do I create a trapezoidal load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Trapezoidal Load** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Hover the cursor over the element until it is displayed with a number of special points along it.
5. Pick the load position either by clicking on one of the special points, or by clicking at any given distance along the element, or by pressing **<F2>** to enter the position by the keyboard.

The load position defines the point at which the trapezoidal load (which is symmetrical) reaches its maximum intensity.

How do I create a point or moment load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Point Load** ( ), or **Moment load** ( ) (on the **Member Loads** panel).
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Hover the cursor over the element until it is displayed with a number of special points along it.
5. Pick the load position either by clicking on one of the special points, or by clicking at any given distance along the element, or by pressing **<F2>** to enter the position by the keyboard.

How do I create a torsion full UDL?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Torsion Full UDL** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Click anywhere along the element to apply the load.

How do I create a partial length torsion UDL or torsion VDL?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Torsion UDL** ( ), or **Torsion VDL** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Hover the cursor over the element until it is displayed with a number of special points along it.
5. Pick the load start position either by clicking on one of the special points, or by clicking at any given distance along the element, or by pressing **<F2>** to enter the position by the keyboard.
6. Click again, or press **<F2>** to specify the load length.

Working with Structure Loads

The following **Structure Loads** can be applied:

- Nodal load - applied at solver node locations.
- Temperature load - a global rise in temperature applied to individual elements/panels, selected elements/panels, or to all elements/panels .
- Settlement load - a translation or a rotation applied to a support (in the support UCS system).

How do I create a nodal load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Nodal Load** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Pick the load position by clicking on a node.

How do I create a temperature load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Temperature Load** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Click anywhere along the element to apply the load.

How do I create a settlement load?

1. Select an appropriate load case from the Loading drop list.
2. Click **Load** > **Settlement Load** ( )
3. Review the load details displayed in the **Properties Window** and adjust as necessary.
4. Pick the load position by clicking on a supported node.

Analysis Guide

An extensive range of 1st and 2nd order analyses can be performed from the Analyze toolbar.

Analyze toolbar

The Analyze toolbar contains the following commands:

Button	Description
Options	Opens the Analysis Options dialog. See: Analysis Options
1st Order Linear	Runs a linear static analysis. This analysis type is suitable for structures where secondary effects are negligible. Any nonlinear springs or nonlinear elements present are constrained to act linearly. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: No • Material: No See: How do I run a 1st order linear analysis?
1st Order Non-linear	Runs a nonlinear analysis with loading applied in a single step. This analysis type is suitable for structures where secondary effects are negligible and nonlinear springs/elements are present. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: No • Material: Yes See: How do I run a 1st order non-linear analysis?

1st Order Vibration	Runs an unstressed vibration analysis to determine the structure's natural frequencies. The structure is assumed to be in an unstressed state and nonlinear elements are constrained to act linearly. Nonlinearity Included: • Geometric: No • Material: No See: How do I run 1st order vibration analysis?
2nd Order Linear	Runs a 2-stage P-Delta analysis. This analysis type is suitable for structures where secondary effects are of comparable magnitude to primary effects. Any nonlinear springs or nonlinear elements present are constrained to act linearly. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: Yes • Material: No See: How do I run a 2nd order linear analysis?
2nd Order Non-linear	Runs a nonlinear analysis with loading applied in a single step. This analysis type is suitable for structures where secondary effects are of comparable magnitude to primary effects and nonlinear springs/elements are present. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: Yes • Material: Yes See: How do I run a 2nd order non-linear analysis?
2nd Order Buckling	Runs a linear buckling analysis to determine a structure's susceptibility to buckling. The stressed state of the structure is determined from linear analysis; therefore nonlinear elements are constrained to act linearly. Loadcases and Combinations to be considered in the analysis can be pre-selected. Nonlinearity Included: • Geometric: Yes • Material: No See: How do I run 2nd order buckling analysis?

1st Order RSA Seismic	Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis. Nonlinearity Included: • Geometric: No • Material: No See: How do I run a 1st order RSA seismic analysis?
2nd Order RSA Seismic	Runs Modal Response Spectrum Analysis to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis. Nonlinearity Included: • Geometric: Yes • Material: No See: How do I run a 2nd order RSA seismic analysis?
Tabular Data	Displays the analysis model data in spreadsheets for review/editing. See: Tabular data (Solver Model Data)
Mesh Slabs	Slabs are meshed automatically for FE Load Decomposition, FE Chasedown, and also if specified in the 3D Analysis. This command is only used if you need to mesh slabs manually. See: Working with FE meshed slabs
Update Wall Beams	Wall Beams are created automatically as internal 1D elements within walls in the analysis model. Their analytical properties are determined automatically from the geometry of the wall. If the model has been changed wall beams are updated automatically when you run the analysis. This command is only used if you need to update wall beams manually. See: Working with FE meshed walls
Analyse All (Static)	Runs all the analyses required (for all loadcases and combinations) to enable design to proceed: • **3D Analysis** - (either first or second order, as specified in Design Options > Analysis) • **Grillage Chasedown** • **FE Chasedown** Once analysis is complete, individual members can be checked or designed.

	See: How do I run FE chasedown and Grillage chasedown analysis?
Update	Diaphragms are updated to reflect the latest model changes every time the model is analysed or designed. They can also be updated manually after the model has been edited by clicking the Update. See: Working with rigid and semi-rigid diaphragms The above command is only available in Solver Views. It is not displayed in the Solver Model Data View.
Result Type (1st drop list in the Result Type group)	This drop list is used to select the Analysis for which tabular results are to be displayed in the Solver Model Data View. See: Tabular data (Solver Model Data) This drop list is only displayed when the Solver Model Data View is made active (by clicking **Tabular Data**).
Mode (2nd drop list in the Result Type group)	If the Result Type is set to 1st Order Vibration, or 2nd Order Buckling, and an appropriate loadcase or combination is selected in the Loading drop list, the Mode drop list becomes active. It is used to select the vibration mode for which results are to be displayed: See: Tabular data (Solver Model Data) This drop list is only displayed when the Solver Model Data View is made active (by clicking **Tabular Data**).
View Type (the drop list in the View Type group)	This drop list is used to select the type of data to be displayed in the **Solver Model Data** View for the chosen Result Type. See: Tabular data (Solver Model Data) This drop list is only displayed when the Solver Model Data View is made active (by clicking **Tabular Data**).

Related topics

- Commands on the ribbon toolbars

Analysis Options

How to apply and manage Analysis Options

To modify analysis options in the current project

1. Click **Analyze** > **Options...** ( )
2. Review and edit the settings as required.
3. If you change any of the settings, click:

- **OK** - to apply the changes directly to the current project, or
- **Save...** - to save the changes back to the active settings set (to act as defaults for future projects), or
- **Cancel...** - to cancel the changes

You can also click:

- **Load...** - to revert to the analysis options specified in the active settings set.

To modify analysis option defaults for future projects

1. Click **Home** > **Settings** ( )
2. In the **Analysis Options** page of the dialog, select the settings set to be updated, then review and edit the settings as required.

You can update any settings set simply by selecting it from the droplist, it does not need to be active.

3. If you change any of the settings, click:

- **OK** - to save the changes to the selected settings set (to act as defaults for future projects when that set is active), or
- **Cancel** - to cancel the changes

1st Order Non-Linear Options

Convergence Criteria

Maximum number of iterations

default = 100

Tolerance
default = 0.0001

Relative
default = on

Relaxation Factors

Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either zero or tension force. The process is entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances.

Use relaxation factors for tension only elements
default cleared

Minimum relaxation factor
default = 0.1

Maximum relaxation factor
default = 0.5

If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence.

Relaxation factors are not activated by default as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

2nd Order Non-Linear Options

The options available are identical to those for 1st order non-linear.

1st Order Vibration Options

Extraction Method

Jacobi
An iterative transformation method used to calculate all Eigen values and Eigen Vectors. Good for small models but unsuitable for medium to large models

Subspace
An iterative simultaneous vector method to calculate the smallest Eigen values and corresponding Eigen Vectors. 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. See: http://www.ecs.umass.Ledu/~polizzi/feast/ for more information.

Automatic

Initially uses Subspace to find the lowest modes. If the stopping criteria (either mass, or number of modes) is not met FEAST is then automatically used to find higher modes until the stopping criteria is fulfilled.

Stopping Criteria

Stopping criteria prevent analysis continuing forever. If either of these criteria are met the analysis will not look for any more modes

Modes

Automatic number of modes

If this option is selected you then specify the mass participation required in each direction. You can optionally specify an initial number of modes, (which 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 specify the total number of modes required, (default 10).

Jacobi Settings

Max number of sweeps

A sweep is a transformation of every off-diagonal in the global matrices. This sets the maximum number of sweeps allowed.

Sweep tolerance

At the end of each sweep values are checked against the previous sweeps results. If the difference is less than this tolerance the result is converged, and the analysis is complete.

Subspace Settings

Maximum number of iterations

The number of iterations to perform.

Tolerance

At the end of each iteration values are checked against the previous iterations results. If the difference is less than this tolerance the result is converged, and the analysis is complete.

FEAST

Initial search range

Specifies the initial range of values FEAST will search for Eigen values in.

Overestimation multiplier

Within each range specifies the initial guess for the subspace dimension. to be used: an overestimate of the predicted number of modes in the range.

Maximum modes in range

The maximum allowable 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 this it will no longer be split, even if the maximum number of modes is greater than that allowed.

2nd Order Buckling Options

Maximum number of iterations

default = 1000

Tolerance

default = 0.00001

Max number of sweeps

default = 50

Sweep tolerance

default = 1.0E-12

Modes

default = 10

Show negative buckling factors

default = no

Extraction method

The Jacobi method is more suited for small models and Subspace method more suited for large models. Choose Automatic for the program to determine the most appropriate method for your structure.

1st Order Seismic Options

Extraction Method

Jacobi

An iterative transformation method used to calculate all Eigen values and Eigen Vectors. Good for small models but unsuitable for medium to large models

Subspace

An iterative simultaneous vector method to calculate the smallest Eigen values and corresponding Eigen Vectors. 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. See: http://www.ecs.umass.Ledu/~polizzi/feast/ for more information.

Automatic

Initially uses Subspace to find the lowest modes. If the criteria (either mass, or number of modes) FEAST is then automatically used to find higher modes until the stopping criteria is fulfilled.

Stopping Criteria

Stopping criteria prevent analysis continuing forever. If either of these criteria are met the analysis will not look for any more modes

Modes

Initial number of modes

In order to speed up the analysis process you can specify an initial number of modes you expect to be required to achieve the required participation, (this should be close to the actual number required; if you enter too few, or too many, the analysis may take longer.)

Mass participation for RSA

you specify the mass participation required in each direction. (If this isn't achieved before the stopping criteria apply, the RSA analysis will still be performed but a warning will be issued.

Min. Mass participation for RSA

If the minimum participation isn't achieved before the stopping criteria apply, the RSA analysis is not performed.

The parameters in the stopping criteria supersede both the number of modes and mass percentage.

Jacobi Settings

Max number of sweeps

A sweep is a transformation of every off-diagonal in the global matrices. This sets the maximum number of sweeps allowed.

Sweep tolerance

At the end of each sweep values are checked against the previous sweeps results. If the difference is less than this tolerance the result is converged, and the analysis is complete.

Subspace Settings

Maximum number of iterations

The number of iterations to perform.

Tolerance

At the end of each iteration values are checked against the previous iterations results. If the difference is less than this tolerance the result is converged, and the analysis is complete.

FEAST

Initial search range

Specifies the initial range of values FEAST will search for Eigen values in.

Overestimation multiplier

Within each range specifies the initial guess for the subspace dimension. to be used: an overestimate of the predicted number of modes in the range.

Maximum modes in range

The maximum allowable 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 this it will no longer be split, even if the maximum number of modes is greater than that allowed.

Modal Combination Method

To determine the representative maximum "response" of interest for a loadcase, the relevant values for each Relevant Mode are combined by using the method specified. Note that once modes have been combined the relative signs are lost.

CQC

Suitable for models where modes are closely spaced or well-spaced

SRSS

Suitable only for models where modes are well-spaced

Modification Factors

Different factors can be applied for each of the different materials in each of the different solver models in order to adjust the following properties:

- E - Youngs 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/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)

These factors also vary according to the member types, (and in the case of concrete members whether they are cracked or not).

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 these factors, analysis must be repeated.

Concrete slab (Flat Slab, Foundation Mat, Beam and Slab) modification factors only apply to solver models in which the slab adopts a "Shell" mesh; they do not get applied in solver models where the slab adopts a "Semi-Rigid" mesh.

Meshing

These parameters can be adjusted in order to fine tune the meshes that are produced in slabs and walls. The default settings are generally appropriate, but they could potentially be adjusted if 2D element quality errors are being created during validation.

Composite Steel Beams

Inertia used for loading analysis

Allows the engineer to specify the inertia to be used in the global analysis of the model for a composite steel beam:

- Steel beam (default)
- Long term composite
- Short term composite

US Head Code:

When long or short term inertia is selected, this only applies to the following sections:

- Rolled symmetric I sections (without web openings)

For all other sections the standard steel beam inertia is used regardless of the analysis option selected.

EC Head Code:

When long or short term inertia is selected, this only applies to the following sections:

- Symmetric I rolled sections (without web openings)
- I plated sections (without web openings)
- Westok plated sections (without web openings)
- Fabsec sections (without web openings)

For all other sections the standard steel beam inertia is used regardless of the analysis option selected.

BS Head Code:

When long or short term inertia is selected, this only applies to the following sections:

- Symmetric I rolled sections (without web openings)
- Asymmetric I rolled sections (without web openings)
- I plated sections (without web openings)
- Westok plated sections (without web openings)
- Fabsec sections (without web openings)

For all other sections the standard steel beam inertia is used regardless of the analysis option selected.

Aus Head Code:

Design of composite beams is not currently supported for the Australian Head Code.

IS Head Code:

Design of composite beams is not currently supported for the Indian Head Code.

Working with FE meshed slabs

At those levels where 2-way spanning slabs exist, FE meshing is applied as follows:

- If a level has the option **Mesh 2-way Slabs in Analysis** cleared, then the slabs at that level are only meshed in order to allow load decomposition to occur prior to analysis. In the analysis itself the slabs are not meshed. This is the default setting.
- If a level has the option **Mesh 2-way Slabs in Analysis** selected, then load decomposition is not performed for the 2-way slabs at that level and they are meshed in the analysis.
- 2-way spanning slabs are always meshed in the FE Chasedown that occurs as part of the static design process.

In all of the above situations, the meshes are created according to the defined mesh parameters.

How do I specify whether a level uses meshed slabs in analysis or FE load decomposition?

A level based setting is used to control whether 2 way slabs are meshed in the building analysis, or FE load decomposition is carried out instead.

You can choose to use FE meshed slabs at certain levels only and FE decomposition at the remaining levels if you so require.

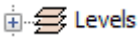
To use meshed 2 way slabs in building analysis at all levels:

1. Highlight Levels in the Structure Tree.
2. In the Properties Window, check **Mesh 2-way Slabs in 3D Analysis**.

To use FE decomposed slab loads at all levels:

1. Highlight Levels in the Structure Tree.
2. In the Properties Window, clear **Mesh 2-way Slabs in 3D Analysis**.

To use meshed 2 way slabs at specific levels and FE decomposed slab loads at others:

1. Expand 

in the Structure Tree.

2. Click each construction level in turn and select/clear **Mesh 2-way Slabs in 3D Analysis** as required.

How do I control the mesh parameters to be used?

Mesh parameters are held as properties of the structure so that initially the same mesh parameters are applied globally on all floors.

The parameters are specified as follows:

1. Select the Structure branch in the Structure Tree.

2. You will see the Structure properties in the Properties Window.

3. Adjust the **Shell Mesh Size** and **Shell Uniformity Factor** as required.

Although the default size and uniformity (3' 6" and 50%) are likely to be conservative, the degree of mesh refinement applied remains the users responsibility.

To optimize solution time consider using a coarser mesh during design development before switching to a more refined mesh at the final design stage.

Different mesh parameters can be applied at specific floors, by introducing additional sub models.

How do I apply different mesh parameters at different levels?

1. If you require different mesh parameters at a specific level, you will firstly need to introduce an additional sub model at that level.
(See: How do I create Sub Models?)

2. Once the new sub model has been created, expand the Sub Models branch in the Structure Tree and select it.

3. You will see the properties of the selected sub model in the Properties Window.

4. Check the **Override Model's** box.

5. Adjust the shell mesh size and uniformity as required.

How do I review the slab mesh prior to running the analysis?

1. Open a **Solver View**.
2. Right-click anywhere in the view and choose **Solver Models** from the context menu.
3. From the sub-menu choose the solver model appropriate to the analysis to be run.

The slab mesh will be displayed if it is applicable to the Solver Model selected.

The slab mesh is not displayed in the **Working Solver Model** *as this shows the model in its form prior to any analysis and 2D elements are only formed at the point of analysis.*

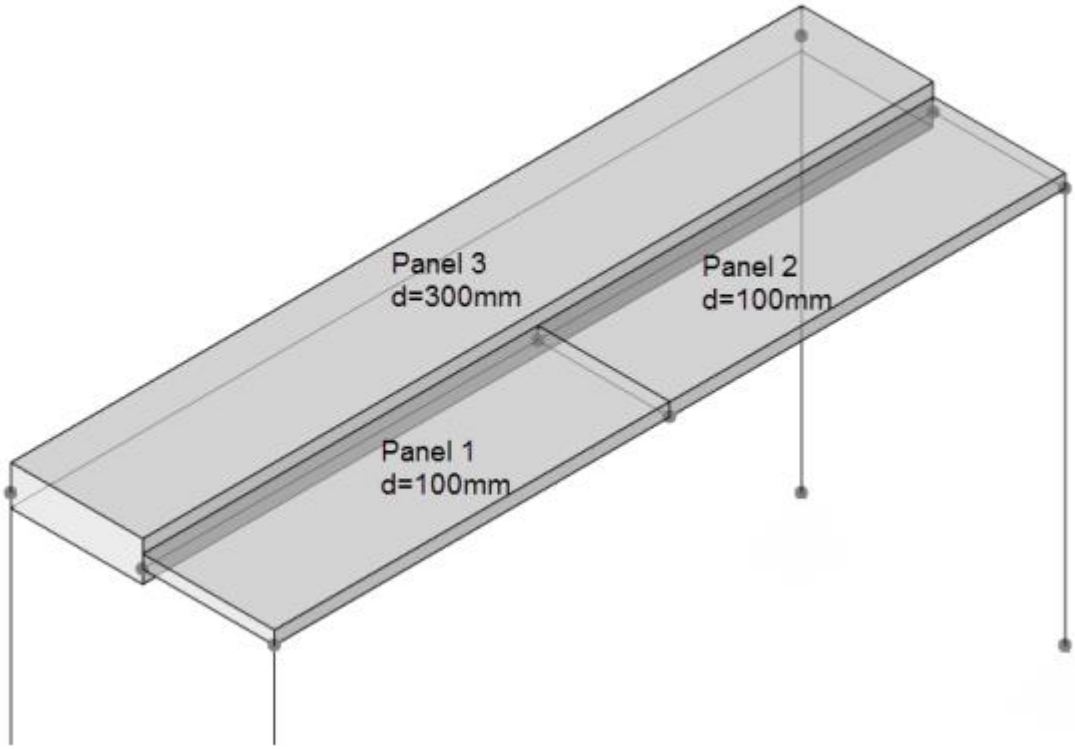
Slab mesh groups

To facilitate meshing, slab panels and features are gathered together automatically into "mesh groups" and meshed as a single entity. Mesh groups cannot be edited directly.

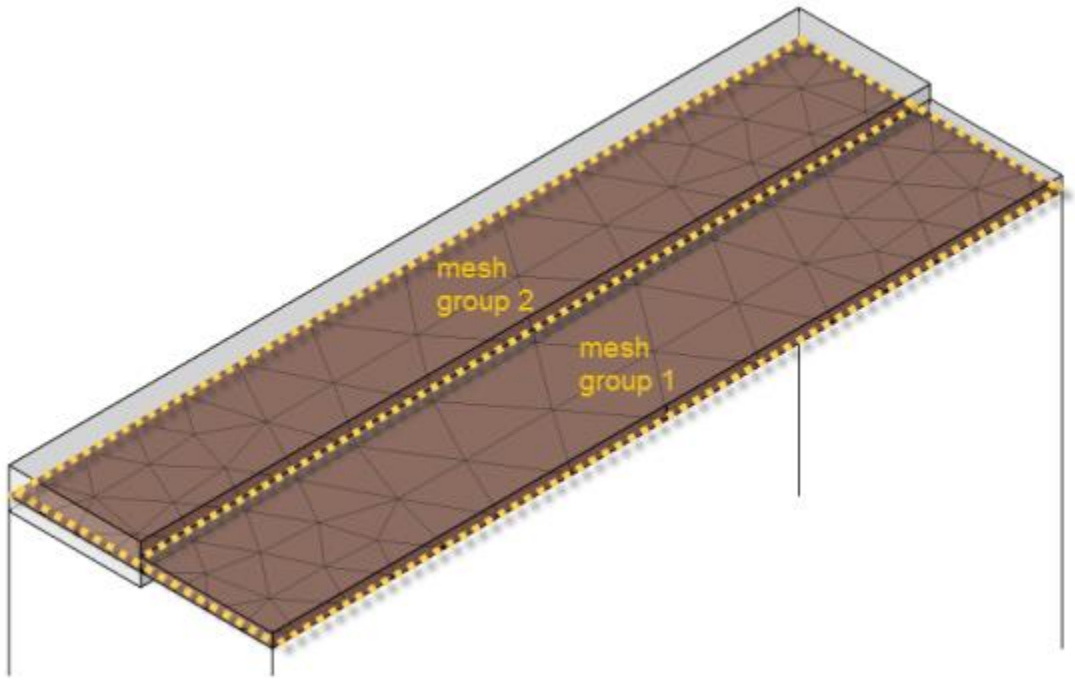
A mesh group contains one or more panels with identical analysis attributes. Since panel thickness is a key analysis attribute, by definition a slab step (or column drop) will produce an additional mesh group.

Example of mesh groups at a slab step

Consider the following three slab panels with thicknesses as shown:



Although there are 3 panels, there are only two slab depths, so only two mesh groups are required:



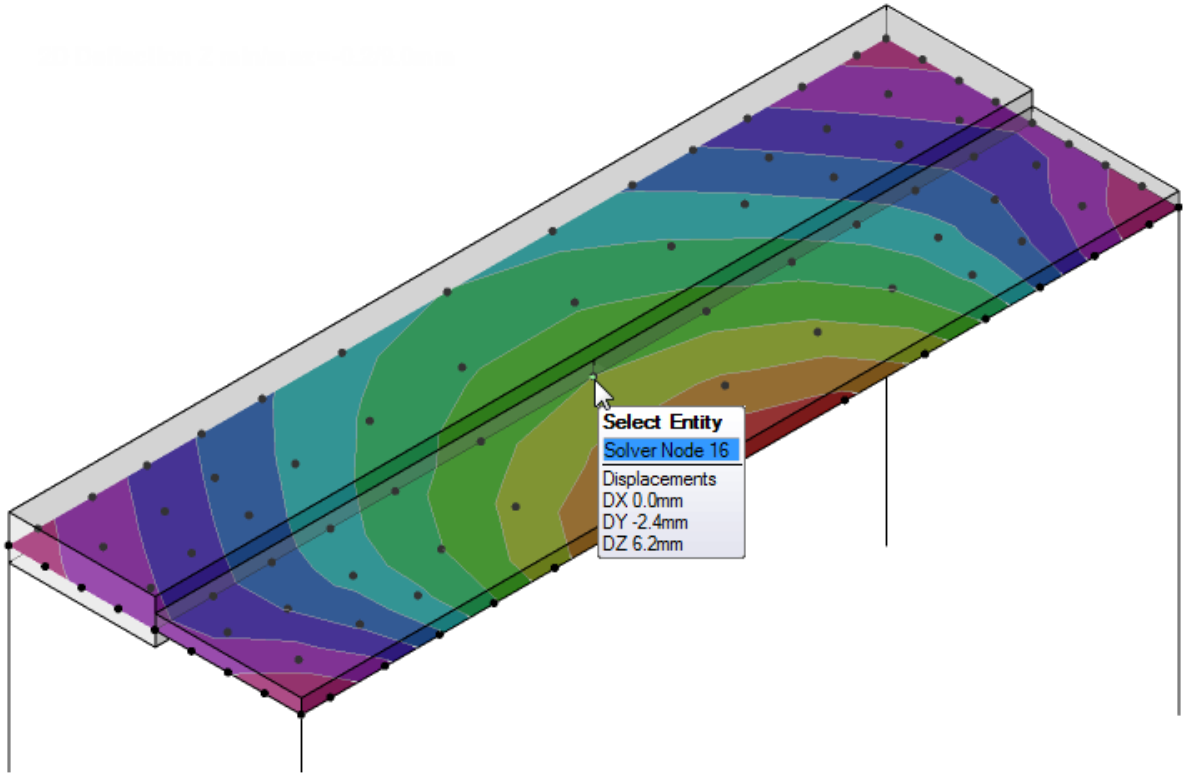
Related topics

- Discontinuity of force contours at slab steps (and column drops)

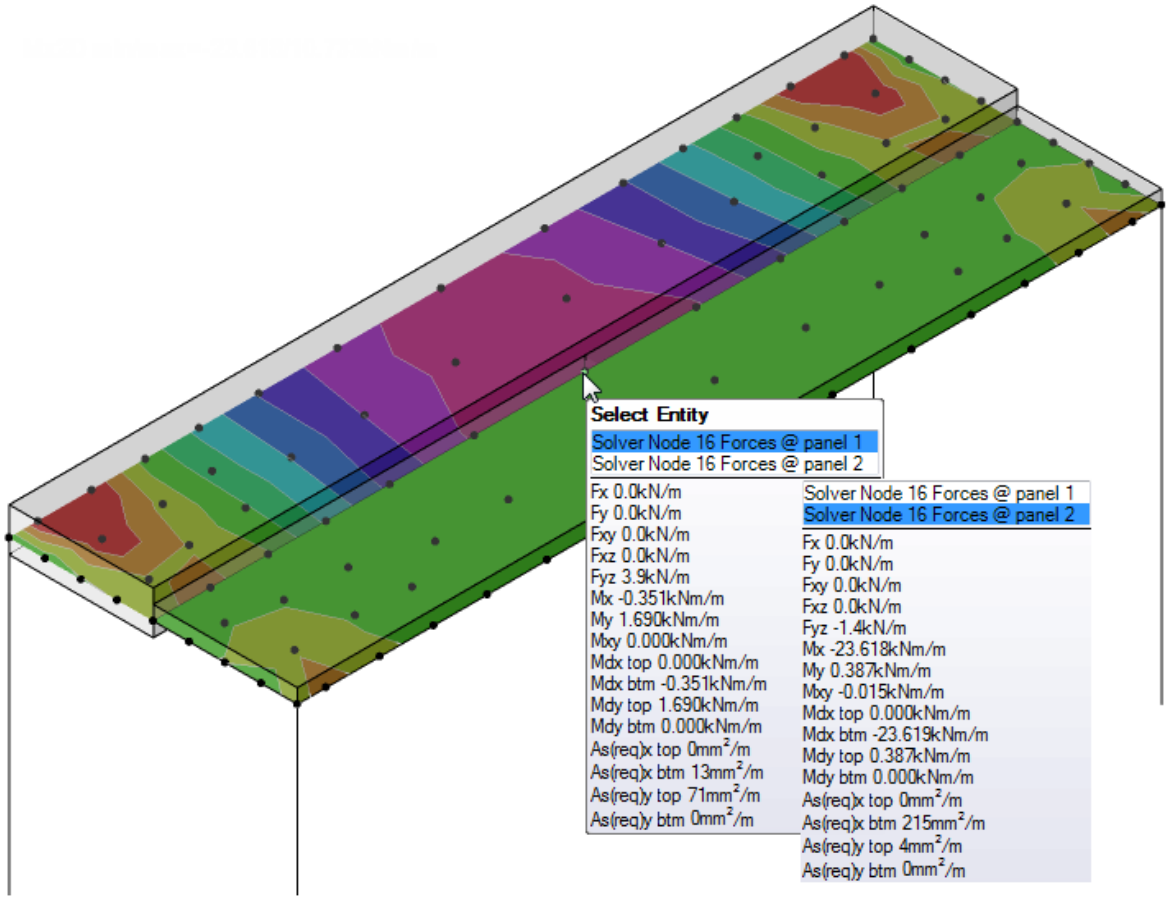
Discontinuity of force contours at slab steps (and column drops)

As a consequence of slab panels to either side of a step being placed into different mesh groups, the solver nodes along the boundary are shared by both groups. Each node on the boundary reports a single value of deflection, but two values of force, (one for each group) - hence there will be a discontinuity of force contours along the boundary.

Deflection contours (no discontinuity):



Moment contours (discontinuity along boundary):



This force discontinuity is a genuine result - the slabs share the same curvature at the step and have the same elastic modulus, so the moment must be directly proportional to the inertia of each panel.

Other programs may choose to average the value across the boundary when generating the contours, but the approach adopted by Tekla Structural Designer is to be preferred since averaging would result in an unrealistically high design of the thinner slab.

Mesh group boundary warnings

Meshing may fail or produce undesirable results when there is challenging mesh group boundary geometry. In this case warnings that point towards the source of meshing issues are provided.

Examples of possible warning triggers are:

- short edges
- distance between a hole and an edge
- small area enclosed by mesh group

Working with FE meshed walls

By default, (unless they are edited to use a mid-pier model), concrete walls adopt an FE meshed model when the building is analyzed.

Meshed walls default to the model's mesh parameters, but these can be overridden to allow a user defined mesh to be applied to an individual wall.

How do I control the global wall mesh parameters to be used?

Mesh parameters are held as properties of the structure so that initially the same mesh parameters are applied globally to all meshed walls.

The parameters are specified as follows:

1. Select the Structure branch in the Structure Tree.
2. You will see the Structure properties in the Properties Window.
3. Use the **Wall mesh type** property to control the shape of mesh applied:

Used to specify the wall mesh type:

- Quad only
- Tri only
- Quad dominant

4. Adjust the **Wall mesh horizontal size** and **Wall mesh vertical size** as required.

How do I override the global wall mesh parameters for an individual wall?

If you require different wall mesh parameters to be applied to specific walls, this is achieved by editing the wall properties.

1. Select the walls to be edited.
2. In the **Properties Window** use the **Wall mesh type** property to control the shape of mesh applied.
3. Adjust the **Wall mesh horizontal size** and **Wall mesh vertical size** as required.

How do I review the wall mesh prior to running the analysis?

1. Open a **Solver View**.
2. Right-click anywhere in the view and choose **Solver Models** from the context menu.
3. From the sub-menu choose the solver model appropriate to the analysis to be run.

The wall mesh will be displayed if it is applicable to the Solver Model selected.

The wall mesh is not displayed in the **Working Solver Model** *as this shows the model in its form prior to any analysis and 2D elements are only formed at the point of analysis.*

Working with rigid and semi-rigid diaphragms

Diaphragms are formed in every slab panel of a parent slab when the **Diaphragm option** in the slab properties is set to **Rigid**, or **Semi-Rigid**.

- When a rigid diaphragm is specified, nodal constraints are automatically applied to the associated slab nodes.
- When a semi-rigid diaphragm is specified, a mesh of semi-rigid 2D elements is created within the slab.

How do I activate rigid diaphragm action within a slab?

By default slabs are initially created with rigid diaphragm option already set.

You can review and edit this setting for specific slabs as follows:

1. From the Slabs branch of the Structure Tree, select the slab which contains the diaphragm.
2. In the Properties Window, set the **Diaphragm option** as required.

Setting the **Diaphragm option** *in the properties of a Slab Item has the same effect as setting it in the properties of the parent Slab. Either way all Slab Items in the Slab adopt the new Diaphragm option setting.*

How do I activate semi-rigid diaphragm action within a slab?

Semi-rigid diaphragms can be formed in both one-way and two-way spanning slabs, their properties can be reviewed and edited by selecting the slab in which they are contained:

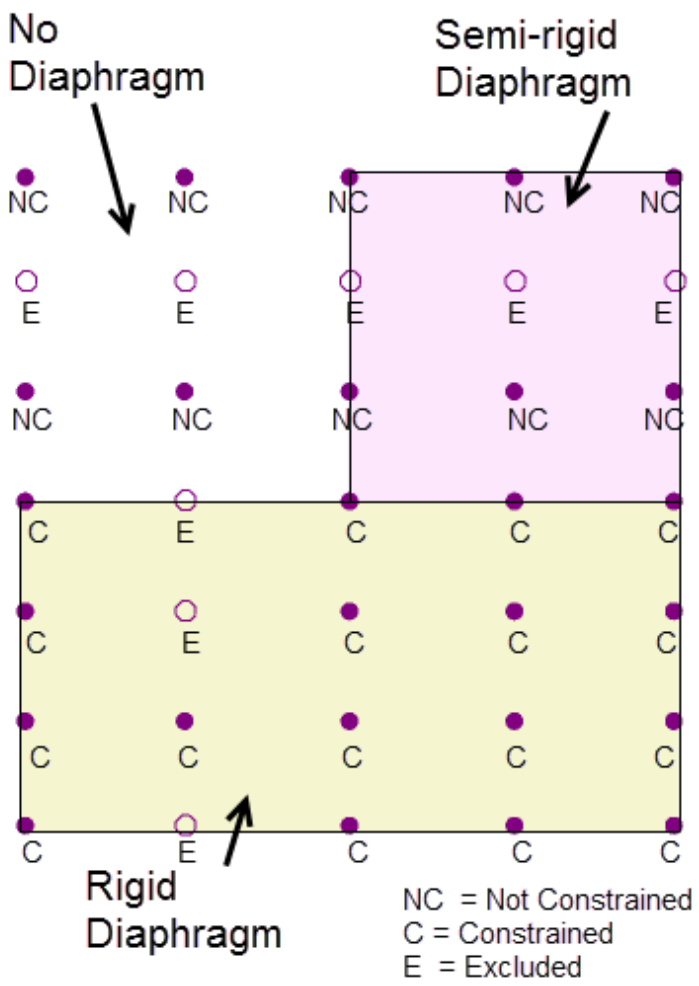
1. From the Slabs branch of the Structure Tree, select the slab
2. In the Properties Window set the **Diaphragm option** to Semi-rigid
3. If required use **Divide stiffness by** to adjust the flexibility of the diaphragm.
4. The meshing parameters that are adopted for the semi-rigid diaphragms are controlled in the Structure properties. (Displayed by selecting the "Structure" branch in the Structure Tree).

Setting the **Diaphragm option** *in the properties of a Slab Item has the same effect as setting it in the properties of the parent Slab. Either way all Slab Items in the Slab adopt the new Diaphragm option setting.*

How do I identify which nodes are constrained by rigid diaphragms?

In order to see which nodes are constrained by diaphragms you must firstly open a **Solver View**. (See How do I open a Solver View?).

In the Solver View, rigid and semi-rigid diaphragms are represented as different colored shaded planes**.**



Nodes constrained by a rigid diaphragm can be identified as follows:

- Firstly they must lie within, or be on the edge of, the shaded rigid diaphragm plane.
- Secondly, they must not be excluded nodes (i.e they should be solid, rather than hollow).

The following are therefore **not** constrained by a rigid diaphragm:

- Solid nodes that lie outside the shaded rigid diaphragm plane.
- Excluded (i.e. hollow) nodes.

How do I intentionally exclude individual nodes from a rigid diaphragm?

1. In order to exclude specific nodes from a rigid diaphragm you must firstly open a **Review View**.

2. Click **Review** > **Diaphragm On\Off**

3. Nodes are then displayed as follows:

- Included nodes - these are only constrained if they lie within, or on the edge of a rigid diaphragm. If they are outside the rigid diaphragm they are not constrained by it.
- Excluded nodes - these are always unconstrained.

4. Click a node to toggle its state between included and excluded.

How do I intentionally exclude a slab panel from a diaphragm?

By default, a rigid diaphragm is formed in all the individual slab panels within a slab, however it is then possible to indicate that specific slab panels should be excluded as follows:

1. In the Structure View, select the slab panel to be excluded.
2. In the Properties Window, clear **Include in diaphragm**.

> *Individual slab panels can also be excluded graphically using the 'Diaphragm On|Off command in the Review View. Clicking a slab panel toggles its inclusion in the diaphragm.*

Running an analysis

How do I run 1st order linear or non-linear analysis?

How do I run a 1st order linear analysis?

1. Click **Analyze** > **1st Order Linear**
2. In the **Select loading** dialog, choose the combinations and loadcases to be analyzed, then click **OK**.

   At the end of the analysis the active view switches to a **Results View** and the tab switches from **Analysis** to **Results** - ready for reviewing the results graphically.

How do I run a 1st order non-linear analysis?

1. Click **Analyze** > **Options** ( )
   Review the 1st order non-linear convergence criteria and adjust if required.
2. Click **Analyze** > **1st Order Non-linear**
3. In the **Select loading** dialog, choose the combinations and loadcases to be analyzed, then click **OK**.

   At the end of the analysis the active view switches to a **Results View** and the tab switches from **Analysis** to **Results** - ready for reviewing the results graphically.

How do I run 2nd order linear or non-linear analysis?

How do I run a 2nd order linear analysis?

1. Click **Analyze** > **2nd Order Linear**

2. In the **Select loading** dialog, choose the combinations and loadcases to be analyzed, then click **OK**.

   At the end of the analysis process the active view is switched to a **Results View** and the active tab is switched to **Results** - ready for reviewing the results graphically.

How do I run a 2nd order non-linear analysis?

1. Click **Analyze** > **Options** ( )
   Review the 2nd order non-linear convergence criteria and adjust if required.

2. Click **Analyze** > **2nd Order Non-linear**

3. In the **Select loading** dialog, choose the combinations and loadcases to be analyzed, then click **OK**.

   At the end of the analysis process the active view is switched to a **Results View** and the active tab is switched to **Results** - ready for reviewing the results graphically.

How do I run 1st order vibration analysis?

Note that this type of analysis requires an active Vibration Mass combination.
(How do I create a Vibration Mass combination?)

1. Click **Analyze** > **Options** ( )
   Review the 1st order vibration analysis options and adjust if required.

2. Click **Analyze** > **1st Order Vibration**

   At the end of the analysis process the active view is switched to a **Results View** and the active tab is switched to **Results** - ready for reviewing the results graphically.

How do I run 2nd order buckling analysis?

1. Click **Analyze** > **Options** ( )
   Review the 2nd order buckling options and adjust if required.

2. Click **Analyze** > **2nd Order Buckling**

3. In the **Select loading** dialog, choose the combinations and loadcases to be analyzed, then click **OK**

   At the end of the analysis process the active view is switched to a **Results View** and the active tab is switched to **Results** - ready for reviewing the results graphically.

How do I run 1st or 2nd order seismic analysis?

How do I run a 1st order RSA seismic analysis?

1. Click **Analyze** > **Options** ( )
   Review the 1st order seismic analysis options and adjust if required.

2. Click **Analyze** > **1st Order RSA Seismic**

   At the end of the analysis process the active view is switched to a **Results View** and the active tab is switched to **Results** - ready for reviewing the results graphically.

How do I run a 2nd order RSA seismic analysis?

1. Click **Analyze** > **Options** ( )
   Review the seismic analysis options and adjust if required.

2. Click **Analyze** > **2nd Order RSA Seismic**

   At the end of the analysis process the active view is switched to a **Results View** and the active tab is switched to **Results** - ready for reviewing the results graphically.

How do I run FE chasedown and Grillage chasedown analysis?

These analyses are run (in addition to the 1st or 2nd order **3D Analysis**) by selecting Analyze All (Static).

They are also be run when required as part of the combined analysis and design process:

- **Grillage Chasedown Analysis** is performed if one or more concrete members exist.
- **FE Chasedown Analysis** is performed if two-way slabs exist, or by user option (i.e. if in the **Design Options** dialog you have opted to design the concrete beams, columns, or walls for FE Chasedown results).

Both these analyses are run for loadcases only and not for combinations.

How do I run a Analyze All (Static)?

1. Click **Analyzse** > **Analyzse All (Static)**

All the analyses required to enable the static design to proceed are performed for all loadcases and combinations as follows:

- **3D Analysis** - (either first or second order, as specified in Design Options > Analysis)
- **Grillage Chasedown** - if one or more concrete members exist.
- **FE Chasedown** - if two-way slabs exist, or by user option.

At the end of the above analyses the active view switches to a **Review View** and the tab switches from **Analysis** to **Review** - ready for individual members to be checked or designed.

Graphical display of the solver model (Solver View)

The **Solver Model** used for each analysis type can be viewed in 2D or 3D from an appropriate **Solver View**.

The following entities are not part of the Solver Model: grid and construction lines, dimensions, slabs and slab openings, wind wall and roof panels. Consequently they are never displayed in Solver Views.
Conversely, when diaphragms exist in the Solver Model, they are only displayed in Solver Views, but are not displayed in the other view types.

How do I open a Solver View?

To open a Solver View as a new view:

1. Duplicate an existing 2D or 3D view by right-clicking its tab and selecting **Duplicate View** from the right-click menu.
2. Change the view type of the newly opened duplicate view:
   - either by right-clicking its tab and selecting **Solver View** from the right-click menu,
   - or by clicking the **Solver View** button on the **Status Bar**.

To change an existing view to a Solver View:

1. Select an appropriate existing 2D or 3D view.
2. Change the view type of the view:
   - either by right-clicking its tab and selecting **Solver View** from the right-click menu,
   - or by clicking the **Solver View** button on from the **Status Bar**.

How do I display the Solver Model used for a particular analysis type?

1. Open a **Solver View**.
2. Right-click anywhere in the view and choose **Solver Models** from the context menu.
3. From the sub-menu choose the solver model required.

How do I see Solver Node and Solver Element properties?

1. Open a **Solver View**
2. Display the appropriate **Solver View**
3. Select the node or element required.
4. The selected node or element's properties are displayed in the **Properties Window**.

Tabular data (Solver Model Data)

The **Solver Model Data View** provides a tabular means to view the raw analysis data and certain results for the different analysis types that have been performed.

Tabular display of Node and Element Data

How do I view tabulated node co-ordinates and degrees of freedom?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **View Type** droplist choose the **Nodes**

How do I view tabulated solver element properties?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **View Type** droplist choose the **Elements**

Tabular display of Reactions, Deflections and End Forces

How do I view tabular results for support reactions?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the analysis type.

Support reactions can be viewed for all analysis types apart from 1st order Vibration and 2nd order Buckling.

3. From the **View Type** droplist choose **Nodal Forces**
4. Select the load case, combination, or envelope required from the Loading drop list.

How do I view tabular results for nodal deflections?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the analysis type.

Nodal deflections can be viewed for all analysis types apart from 1st order Vibration and 2nd order Buckling.

3. From the **View Type** droplist choose **Nodal Delections**

4. Select the load case, combination, or envelope required from the Loading drop list.

5. If a combination has been selected, from the **Result Type** droplist choose between **Strength Factors** and **Service Factors**

How do I view tabular results for solver element end forces?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the analysis type.

> *Element end forces can be viewed for all analysis types apart from 1st order Vibration and 2nd order Buckling.*

3. From the **View Type** droplist choose **Element End Forces**

4. Select the load case, combination, or envelope required from the Loading drop list.

> *The results displayed are the "raw" analysis results; the Element End Forces results have not had any axial load reductions applied.*

What does the asterisk next to certain nodes signify in the Element End Forces table?

When displaying the table of element end forces, if you have modeled rigid arms then you may see asterisks against the start or end nodes of certain element numbers.

If the node number has an asterisk next to it, it signifies that the results are actually output at the end of the rigid arm rather than at the node itself.

How do I view tabular results for wall lines?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the analysis type.

> *Wall line forces can be viewed for all analysis types apart from 1st order Vibration and 2nd order Buckling.*

3. From the **View Type** droplist choose **Wall Lines**

4. Select the load case, combination, or envelope required from the Loading drop list.

Tabular display of Vibration and Buckling Analysis results

How do I view tabular results for mode shapes?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the analysis type.

> *Mode shapes can only be viewed for 1st order Vibration and 2nd order Buckling analysis types.*

3. From the **View Type** droplist choose **Mode Shape**
4. Select the loading required from the Loading drop list.

How do I view the summed mass for vibration mass combinations?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the **1st Order Vibration** analysis type.
3. From the **View Type** droplist choose **Summed Mass**

How do I view the dynamic masses for vibration mass combinations?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the **1st Order Vibration** analysis type.
3. From the **View Type** droplist choose **Dynamic Masses**

How do I view the active masses by node?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the **1st Order Vibration** analysis type.
3. From the **View Type** droplist choose **Active Masses by Node**

How do I view the total masses by node?

1. Click **Analyze** > **Tabular Data**

A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the **1st Order Vibration** analysis type.
3. From the **View Type** droplist choose **Total Masses by Node**

How do I view vibration frequencies and modal masses?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the **1st Order Vibration** analysis type.
3. From the **View Type** droplist choose **Vibration Frequencies**
4. Select the load combination required from the Loading drop list.

How do I view buckling factors?

1. Click **Analyze** > **Tabular Data**

   A Solver Model Data View opens in a new tab.

2. From the **Result Type** droplist choose the **2nd Order Buckling** analysis type.
3. From the **View Type** droplist choose **Buckling Factors**
4. Select the load case or combination required from the Loading drop list.

Graphical display of the analysis results (Results View)

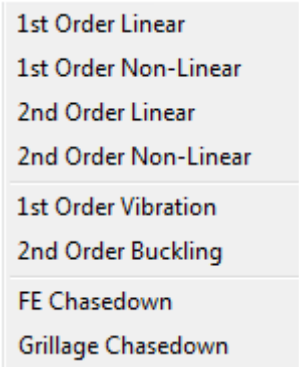
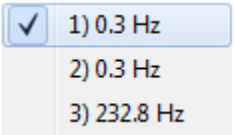
Once you have defined and analyzed your model you can review the results of the analysis graphically in 2D or 3D **Results Views** using the Results toolbar.

Results toolbar

Result Type group

The Result Type group contains two drop lists and two buttons:

Button	Description

Analysis Type (1st drop list)	Use the **Analysis Type** drop list to choose which analysis type to display results for: 
Mode (2nd drop list)	If the **Analysis Type** is set to 1st Order Vibration, or 2nd Order Buckling, and an appropriate loadcase or combination is selected in the Loading drop list, the **Mode** drop list becomes active. It is used to select the vibration mode for which results are to be displayed: 
Reduce Axial Force	When axial forces are displayed, click this button to display the reduced values. This only applies to imposed loadcases which have been defined with **Reductions** applied, (and to any combinations which include the same loadcases).
SLS Deflections	When deflections are displayed for a combination, click this button to display them using the serviceability (as opposed to the strength) load factors.

Scale Settings group

Button	Description

1D Forces Scale	Use the slider to increase/decrease the diagram scales for 1D force results.
2D Forces Scale	Use the slider to increase/decrease the diagram scales for 2D force results.
Deflections Scale	Use the slider to increase/decrease the diagram scales for deflection results.

1D Results group

Click these controls to display analysis results for 1D elements: beams, columns, trusses etc. (Walls modeled using the mid-pier option also fall into this category.)

Button	Description
1D Results	Click this button to activate the display of the required 1D Result (chosen from the drop list below).
1D Results (drop list)	Use the **1D Results** drop list to choose which result type to display: • Axial Force - in local x • Shear Major - along local z • Shear Minor - along local y • Torsion - about local x • Moment Major - bending about local y • Moment Minor - bending about local z • Moment Biaxial - bending about local y and z

Deflections group

Click these controls to display deflection results for 1D members: beams, columns, trusses etc. (Walls modeled using the mid-pier option also fall into this category.)

Whole structure absolute global deflections are shown which are relative to the undeformed model.

Button	Description
Deflections	Click this button to activate the display of the required Deflection Result (chosen from the drop list below).

Deflections (drop list)	Use the **1D Results** drop list to choose which result type to display: • X - model deflections in the global X • Y - model deflections in the global Y • Z - model deflections in the global Z • Dir 1- model deflections in the building direction 1 • Dir 2 - model deflections in the building direction 2 • Total - total (resultant) model deflections

Notional Horizontal Forces

Button	Description
EHF	Displays the calculated equivalent horizontal forces for the selected combination.
Seismic	Displays the calculated equivalent lateral forces/torsions for the selected loadcase or combination.

2D Results group

Click these controls to display FE contours for two dimensional elements (i.e. FE slabs and FE walls).

Note that contours are always in the panel axis system and based on the shell nodal forces.

Button	Description
2D Results	Click this button to activate the display of the required 2D Result (chosen from the drop list below).

2D Results (drop list)	Use the **2D Results** drop list to choose which result type to display: • Fx - Axial Force in panel x axis • Fy - Axial Force in panel y axis • Fxy - Complementary in-plane shear • Fxz - Shear in panel z axis in the panel xz plane • Fyz - Shear in panel z axis in the panel yz plane • Mx - Bending along panel x axis • My - Bending along panel y axis • Mxy - Plate torsional moment • Mdx top - Wood Armer top bending along panel x axis • Mdx bottom - Wood Armer bottom bending along panel x axis • Mdy top - Wood Armer top bending along panel y axis • Mdy bottom - Wood Armer bottom bending along panel y axis • Bearing pressure

AsReq group

Click these controls to display FE contours for two dimensional elements (i.e. FE slabs and FE walls).

Note that contours are always in the panel axis system and based on the shell nodal forces.

Button	Description
As Req	Click this button to activate the display of the required As Req (chosen from the 1st drop list below).
As Req (1st drop list)	Use this **As Req** drop list to choose which result type to display: • x top - Required steel area contours in the top face of the slab along panel x axis (Wood-Armer effects included) • x bottom - Required steel area contours in the bottom face of the slab along panel x axis (Wood-Armer effects included) • y top - Required steel area contours in the top face of the slab along panel y axis (Wood-Armer effects included) • y bottom - Required steel area contours in the bottom face of the slab along panel y axis (Wood-Armer effects included)

As Req (2nd drop list)	Use this to choose whether to display values or the pass/fail status for the result type selected from the 1st drop list. • Value - contours are displayed as values • Pass/Fail - pass/fail threshold contours are displayed

2D Deflections group

Click these controls to display FE deflection contours for two dimensional elements (i.e. FE slabs and FE walls).

Whole structure absolute global deflections are shown which are relative to the undeformed model.

Button	Description
2D Deflections	Click this button to activate the display of the required 2D Deflection Result (chosen from the drop list below).
2D Deflections (drop list)	Use the **1D Results** drop list to choose which result type to display: • X - 2D element deflections in the global X • Y - 2D element deflections in the global Y • Z - 2D element deflections in the global Z • Total - total (resultant) 2D element deflections

2D Strip Results group

After using the **Create Strip** command in a 2D view to define your strips, click the 2D Strip Results button then choose the result from the drop list to see it displayed along the strip - (best displayed in a 3D view).

Button	Description
Create Strip	Click **Create Strip** to place a strip across a mesh. Results for the strip can then be displayed and used for design purposes. **Create Strip** is only available when a 2D view is active.
2D Strip Results	Click this button to activate the display of the required 2D Strip Result (chosen from the drop list below).

2D Strip Results (drop list)	Use the drop list to choose which result type to display: • Deflection - displays a Deflection diagram along the strip • Shear - displays a Shear Force diagram along the strip • Moment - displays a Moment diagram along the strip • Design Moment - displays a Design Moment diagram along the strip • AsReq - displays an Area of Steel Required diagram along the strip

2D Wall Results group

FE Wall results can be collated and displayed on **Result Lines** that are created automatically within FE walls. (The resulting display being similar to that for 1D results in mid-pier walls.)

Button	Description
2D Wall Results	Click this button to activate the display of the required 2D Wall Result (chosen from the drop list below).
2D Wall Results (drop list)	Use the drop list to choose which result type to display: • Axial Force - displays an Axial Force diagram along the result line • Shear Major - displays a Shear Force Major diagram along the result line • Shear Minor - displays a Shear Force Minor diagram along the result line • Torsion - displays a Torsion diagram along the result line • Moment Major - displays a Moment Major diagram along the result line • Moment Minor - displays a Moment Minor diagram along the result line

Text group

Click these controls to display numerical values on the diagrams for the selected result.

Button	Description

Deflection	Displays the *Deflection* value on deflection diagrams.
Axial Force	Displays the *Axial Force* value on axial force diagrams.
Shear	Displays the *Shear Force* value on shear force diagrams.
Support Reaction	Displays the *Reaction* value on support reaction and beam end reaction diagrams.
Torsion	Displays the *Torsion* value on torsion diagrams.
Moment	Displays the *Moment* value on moment diagrams.

Cross checking the sum of reactions against the load input

Once an analysis has been performed the Loading Tree can be used to quickly check that the **Total Reaction** from each analysis equates to the **Total Load on Structure**. This allows you to quickly establish that none of the applied load has gone missing.

The status of each loadcase and combination is indicated as follows:

✓ - **Total Reaction** in equilibrium with the **Total Load on Structure**

✗ - **Total Reaction** not in equilibrium with the **Total Load on Structure**

? - **Total Reaction** not available

Loadcase Properties

When you select a loadcase in the Loading Tree, the following properties are displayed in the Properties Window.

General

This section provides summations (in global X, Y & Z) of the different load types applied to the structure, from these the total applied load is determined.

The total reaction from the **3D Analysis** result is also reported.

Property	Description

Member Loads	Sum of all loads applied that have been as Member Loads to the structure.
Nodal Loads	Sum of all loads applied that have been as Nodal Loads to the structure.
Total NHF Dir 1	Sum of NHFs in Dir 1.
Total NHF Dir 2	Sum of NHFs in Dir 2.
Decomposable Loads	Sum of all loads applied as Panel Loads to the structure (prior to load decomposition).
1 way Decomp Results	After load decomposition - this is the sum of loads decomposed on to members from one way spanning panels.
2 way Decomp Results	After load decomposition - this is the sum of loads decomposed on to members from two way spanning panels.
Total User Applied Load	Sum of all loads applied to the structure (prior to load decomposition).
Total Load on Structure	Sum of the 1 way and 2 way Decomp results.
Total Reaction	Sum of reactions from the **3D Analysis**.

The reported **Total User Applied Load** should be compared to the **Total Load on Structure**, then the reported **Total Load on Structure** should also be cross checked against the **Total Reaction**.

FE Chasedown

Property	Description
Total Reaction	Overall sum of reactions from the FE Chasedown Analysis.
[-] Each Submodel	
Load from above	Sum of all vertical load applied to this sub model from the sub model directly above it.

Load Applied	Sum of vertical load applied within the sub model.
Reaction	Sum of reactions from the analysis of the FE sub model.

The **Total User Applied Load** (from the General section) should be cross checked against the **Total Reaction**.

In addition, for each sub-model the **Load Applied** when added to the **Load from above** should equate to the **Reaction**.

Grillage Chasedown

Property	Description
Total Reaction	Overall sum of reactions from the Grillage Chasedown Analysis.
[-] Each Submodel	
Load from above	Sum of all vertical load applied to this sub model from the sub model directly above it.
Load Applied	Sum of vertical load applied within the sub model.
Reaction	Sum of reactions from the analysis of the grillage sub model.

The **Total User Applied Load** (from the General section) should be cross checked against the **Total Reaction**.

In addition, for each sub-model the **Load Applied** when added to the **Load from above** should equate to the **Reaction**.

Displaying 1D and 2D results, deflections and reactions

Once the analysis type, and the loadcase, combination, or envelope have been chosen, the result can be displayed simply by selecting the required effect from the appropriate group on the **Results** ribbon.

- 1D groups show results for 1D elements - i.e. beams, columns, trusses etc, and also for walls modeled using the mid-pier option.
- 2D groups show results for 2D elements - i.e. FE slabs and FE walls.

How do I choose which analysis to see the results for?

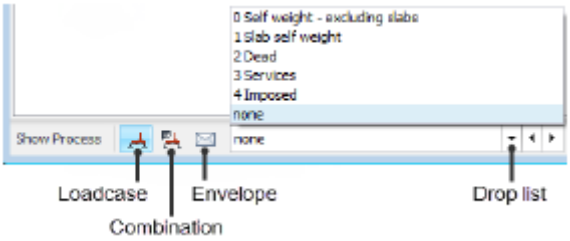
When several analysis types have been performed, the results of each one are held separately, hence there is no need to re-perform a particular analysis to recall its results.

1. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.

2. Select the diagrams to display for this analysis type as required.

How do I choose the loadcase, combination, or envelope to see the results for?

1. First click the **Loadcase**, **Combination**, or **Envelope** button on the **Loading drop list**.



2. Then select the specific loadcase, combination, or envelope name required from the drop list.

How do I display Support Reactions?

1. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.

2. Select the load case or combination to display from the Loading drop list.

3. From the **Reactions** group choose

   - Click **Support Reactions**
   - Choose the reaction required from the drop list.

4. From the **Text** group choose **Reaction.**

How do I display 1D Results and 1D Deflections?

1. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.

2. Select the load case or combination to display from the Loading drop list.

To view 1D Results:

- From the **1D Results** group click **1D Results** then from the drop list underneath it choose the result required

To view 1D Deflections:

- From the **Deflections** group click **Deflections** then from the drop list underneath it choose the deflection required

To show values on the diagrams:

- From the **Text** group choose the effects required

How do I display 2D Results and 2D Deflections?

1. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.
2. Select the load case or combination to display from the Loading drop list.

To view 2D Results:

- From the **2D Results** group click **2D Results** then from the drop list underneath it choose the result required

To view 2D Deflections:

- From the **2D Deflections** group click **2D Deflections** then from the drop list underneath it choose the deflection required

How do I display AsReq contours?

AsReq contours can either be displayed as:

- values, or,
- as a pass/fail threshold

Displaying as a pass/fail threshold helps to visualize minimum patch sizes when optimizing panel and patch reinforcement as it highlights the specific areas in which the existing reinforcement is not sufficient.

1. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.
2. Select the load case or combination to display from the Loading drop list.
3. In the **AsReq** group choose the effect that you want to be displayed from the first drop list.

To view AsReq contours as values:

- From the second drop list in the **AsReq** group choose **Value**

To view AsReq contours as values:

- From the second drop list in the **AsReq** group choose **Pass/Fail**

What are 2D Wall Results and how do I display them?

By post-processing the FE meshed wall nodal analysis results, it is possible to determine forces and moments along Result Lines automatically created within the walls - these results can then be used for design purposes.

The sign convention for Result Line output follows that of the mid-pier wall model.

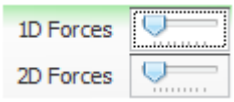
To view the 2D Wall Results:

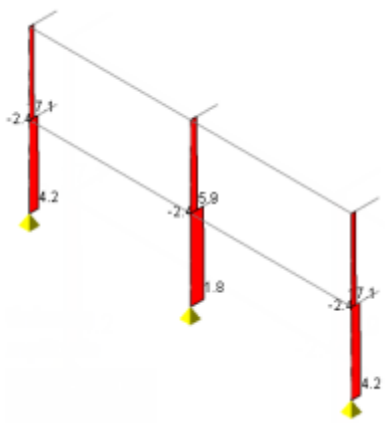
1. Open a 3D View containing the walls to be viewed.
2. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.
3. Select the load case or combination to display from the Loading drop list.
4. From the **2D Wall Results** group click **2D Wall Results** then from the drop list underneath it choose the result required
5. *Tekla Structural Designer* will display the chosen effect on the Result Line.

How do I increase the amplitude of the diagrams?

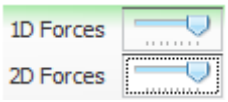
The Forces and Deflections sliders on the settings pane are used to adjust the diagrams.

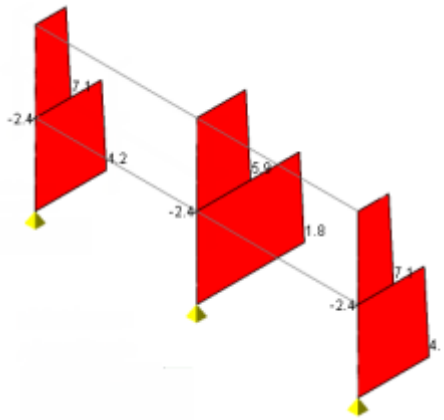
Minimum Amplitude

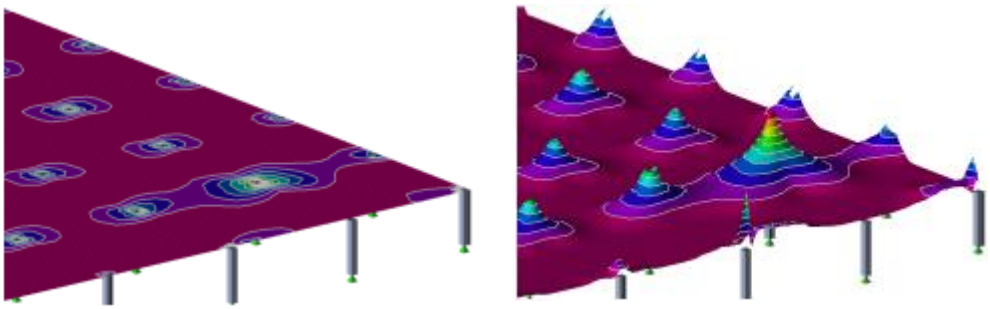




Maximum Amplitude



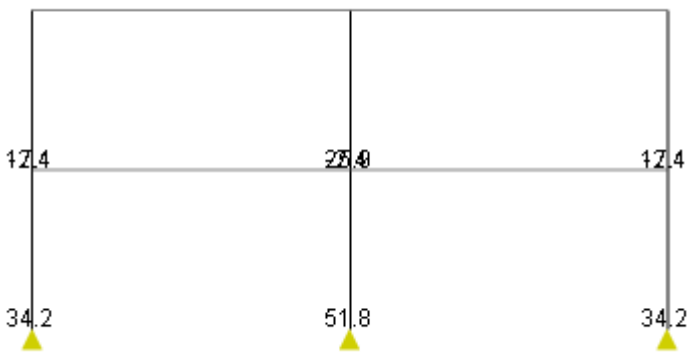




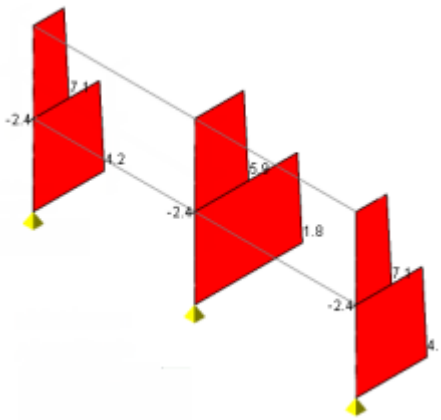
How do I see a 3D display of the results in a 2D view?

Because the diagrams are plotted on each element in the planes in which they act, when you are working in a 2D view you will need switch on an isometric display to see the out of plane forces.

2D Frame View



2D Frame View displayed isometrically



1. If the 2D view is currently displayed in plan, the 3D/2D toggle button in the Status Bar at the bottom right of the screen will be labeled **3D**.
2. Click the **3D/2D** toggle button.
3. The 2D view is now displayed isometrically (and the 3D/2D toggle button changes to **2D**.
4. To change back to a plan view click the **3D/2D** toggle button once more.

How do I customise the display of 2D contours?

By default all contour diagrams comprise of 10 evenly sized contours, each accounting for 10% of the total range. You can increase or decrease the number of contours, and also change the size and the color of individual contours.

1. Click **Home** > **Settings** ( )

2. The "active" settings set is displayed on the **Settings Sets** page - in order to change the contour configuration for the current work session this is the set that should be edited.

3. Expand the **Scene** page and click **Contours**.

- Click **Split** to add new contours
- Click **Delete** to remove contours
- Edit the **Size** of each contour as required.
- You can also change the **Color** of each contour.
- To revert to the default contour configuration click **Reset**

Displaying Mode Shapes

Mode shapes can be displayed for 1st Order Vibration, 2nd Order buckling, 1st Order RSA Seismic and 2nd Order RSA Seismic analyses.

How do I display mode shapes?

When several analysis types have been performed, the results of each one are held separately, hence there is no need to re-perform a particular analysis to recall its results.

1. On the **Results** ribbon, select the analysis type required (either 1st Order Vibration, 2nd Order buckling, 1st Order RSA Seismic, or 2nd Order RSA Seismic) from the drop list in the **Result Type** group.

2. Select the loadcase or combination required from the **Loading drop list**.

For RSA Seismic analysis only loadcases should be selected, as mode shapes are not applicable for combinations and hence cannot be viewed.

3. Select the mode required from the **Mode** drop list in the **Result Type** group.

4. Select the diagram to be displayed.

Displaying Notional Forces and Seismic Equivalent Lateral Forces

EHF

(Applicable to Eurocode only) To see the magnitude of EHF in a particular combination:

1. Select the combination required from the **Loading drop list**.

2. On the **Results** ribbon, click **EHF** in the **Notional Horizontal Forces** group.

Notional Loads

(Applicable to United States (ACI/AISC) only) To see the magnitude of notional loads in a particular combination:

1. Select the required combination which includes notional loads from the **Loading drop list**.
2. On the **Results** ribbon, click **NL** in the **Notional Horizontal Forces** group.

NHF

(Applicable to BS and Australian codes only) To see the magnitude of NHF in a particular combination:

1. Select the combination required from the **Loading drop list**.
2. On the **Results** ribbon, click **NHF** in the **Notional Horizontal Forces** group.

Seismic

To see the magnitude of equivalent lateral forces in a seismic loadcase or combination:

1. Select the required seismic loadcase or combination from the **Loading drop list**.
2. On the **Results** ribbon, click **Seismic** in the **Notional Horizontal Forces** group.

Displaying RSA Seismic Results

Result Type

When the Result Type is set to 1st or 2nd Order RSA Seismic, the results that can be displayed depend on the type of the currently selected Loadcase or Combination.

Mode Shapes

Mode Shapes can be displayed for:

- RSA Seismic Loadcases
  - Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options
  - Plus all relevant modes for this loadcase
- Effective Seismic Weight Combination
  - List of all modes returned by the Vibration Analysis

Mode Shapes are not displayed for:

- RSA Torsion Loadcases
- Static Loadcases included in the RSA Seismic Combination
- RSA Seismic Combinations

1D Element Results

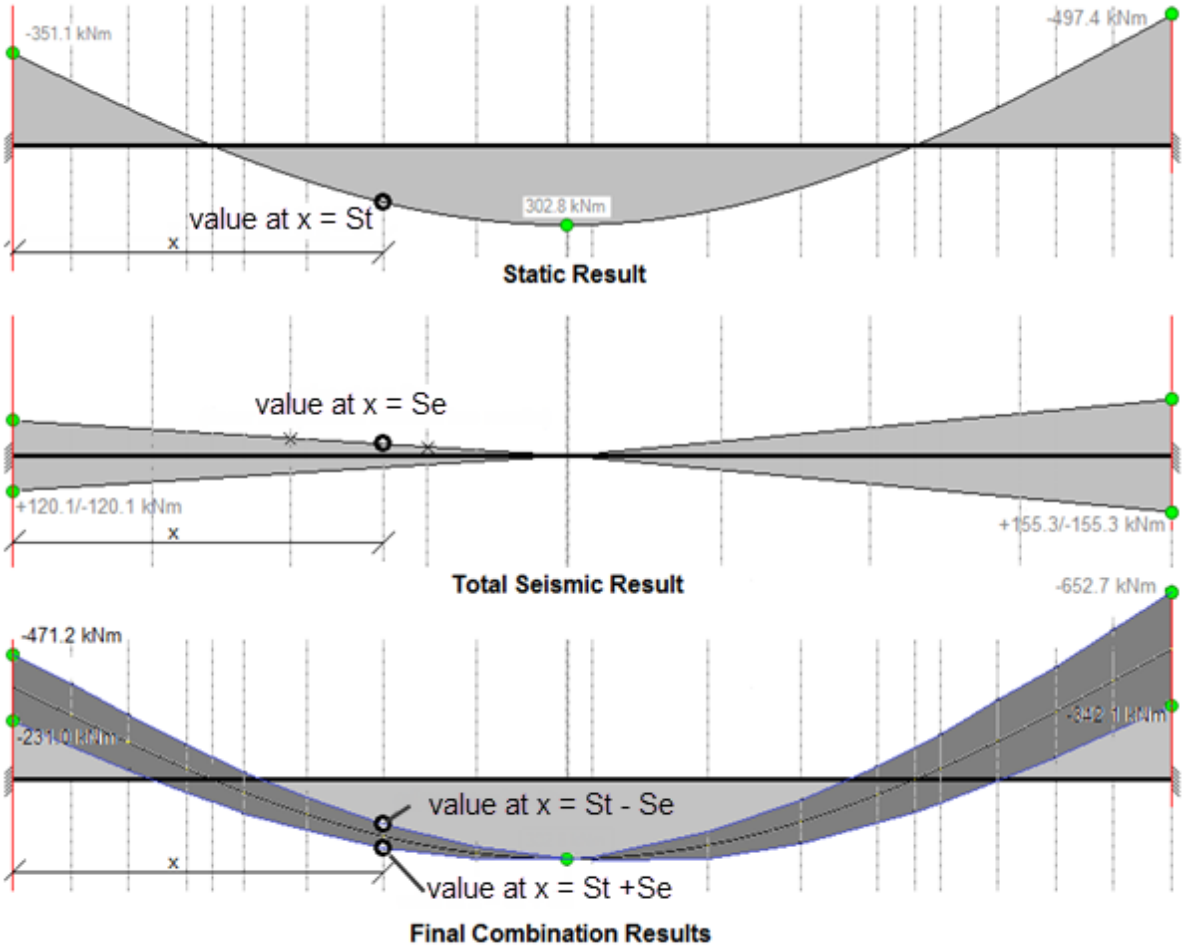
1D Element Results are displayed as follows:

- RSA Seismic Loadcases
  - Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options Absolute values are determined at various points along each member and then

plotted on both the positive and negative side of the diagram, (so that the diagrams are always symmetrical about the base line).

- All relevant modes
  A standard enveloped diagram is displayed

- RSA Torsion Loadcases - displayed as per 1st order linear analysis
- Static Loadcases included in the RSA Seismic Combination - displayed as per 1st order linear analysis
- Effective Seismic Weight Combination - not available
- RSA Seismic Combinations
  An envelope is drawn showing the seismic results above and below the static result.
  - Base line is through the static values
  - Top line is static value + seismic value
  - Bottom line is static value - seismic value

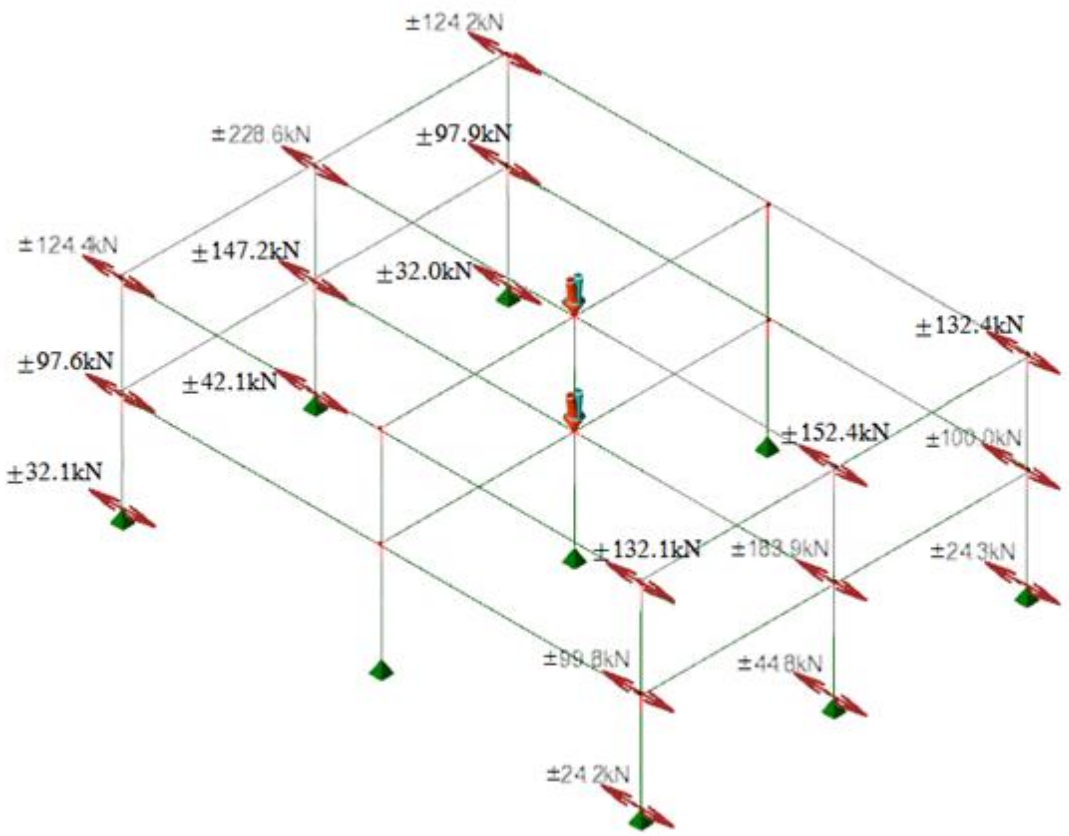


Story Shear

Story Shears are displayed as follows:

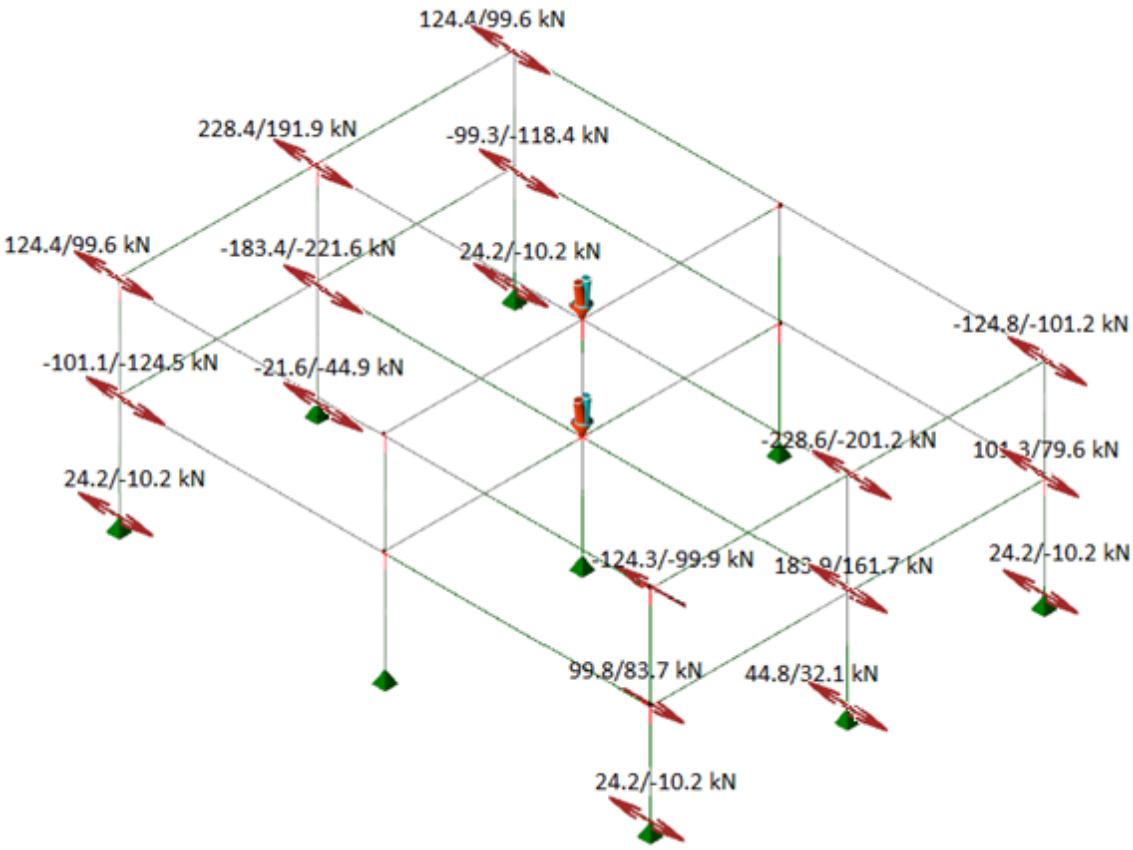
- RSA Seismic Loadcases

- Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options
  Absolute values are determined at each position of interest and the result is then shown as both positive and negative.



- All relevant modes
  A standard diagram with a single value at each point of interest is displayed

- RSA Torsion Loadcases - displayed as per 1st order linear analysis
- Static Loadcases included in the RSA Seismic Combination - displayed as per 1st order linear analysis
- Effective Seismic Weight Combination - not available
- RSA Seismic Combinations
  The diagram displays two values at each point of interest:
  - static value + seismic value
  - static value - seismic value.



Support Reactions

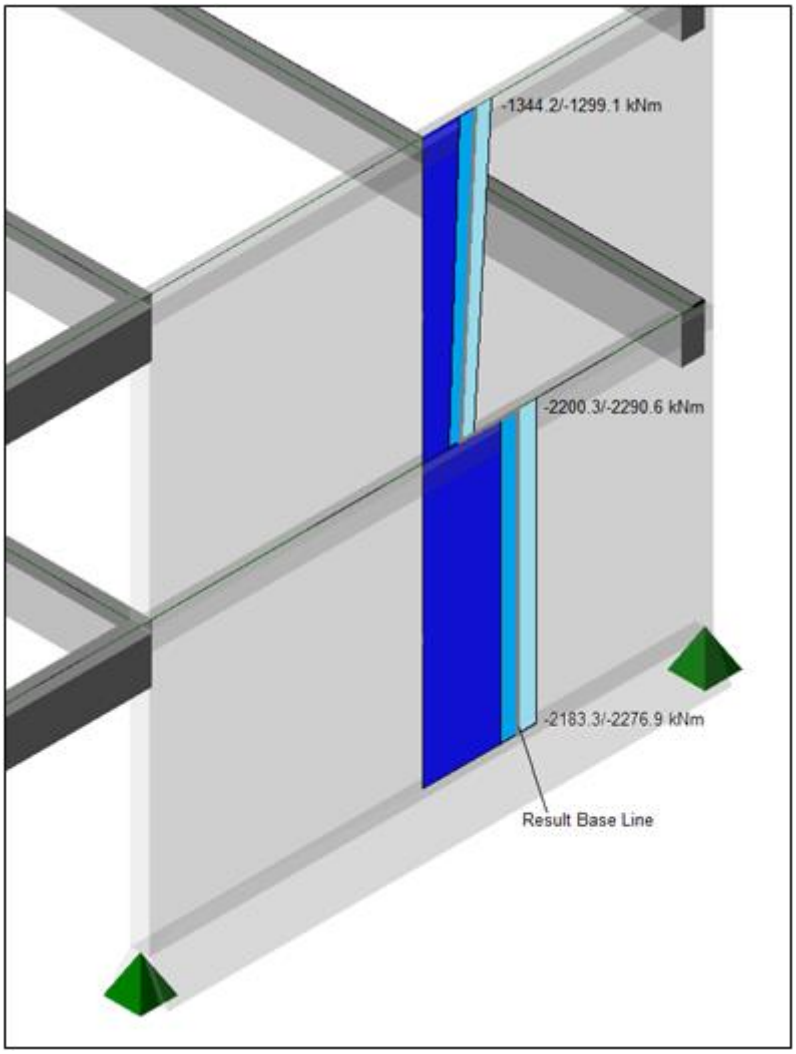
Support Reactions are displayed as follows:

- RSA Seismic Loadcases
  - Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options Absolute values are determined at each support and the result is then shown as both positive and negative.
  - All relevant modes - a standard diagram is displayed
- RSA Torsion Loadcases - a standard diagram is displayed
- Static Loadcases included in the RSA Seismic Combination - a standard diagram is displayed
- Effective Seismic Weight Combination - not available
- RSA Seismic Combinations
  The diagram displays two values at each support:
  - static value + seismic value
  - static value - seismic value.

2D Wall Results

2D Wall Results are displayed as follows:

- RSA Seismic Loadcases
  - Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options Absolute values are determined at points along the wall line and then plotted on both the positive and negative side of the diagram, (so that the diagrams are always symmetrical about the wall line).
  - All relevant modes - a standard diagram is displayed
- RSA Torsion Loadcases - a standard diagram is displayed
- Static Loadcases included in the RSA Seismic Combination - a standard diagram is displayed
- Effective Seismic Weight Combination - not available
- RSA Seismic Combinations
  An envelope is drawn showing the seismic results above and below the static result.
  - Base line is through the static values
  - Line above is static value + seismic value
  - Line below is static value - seismic value

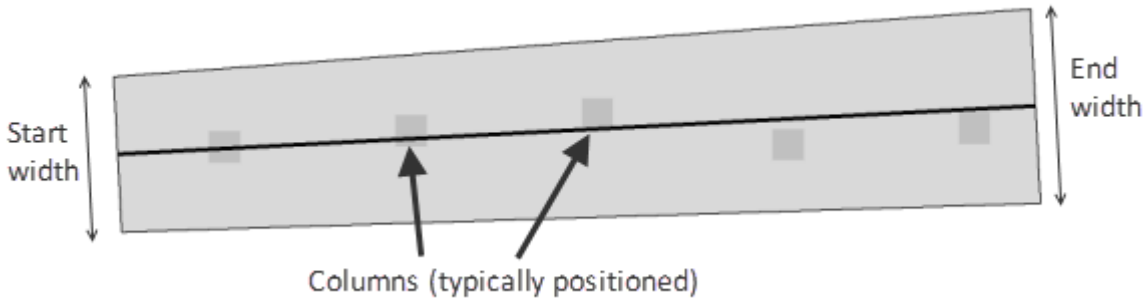


Working with 2D Strips and displaying Strip Results

Result Strip Overview

User defined Result Strips can be placed across 2D element meshes. From these strips, force and moment results are determined from the shell/plate/membrane nodal analysis results - these can then be used for design purposes, (typically for slab design).

Engineering judgement is required when positioning the strips to ensure suitable design forces are obtained. By default they have parallel edges, but tapering strips can also be defined as shown below:



How do I create a Result Strip?

You must first open a 2D view of the FE mesh where the strip is to be placed and then display the Results tab.

To display the Results tab:

1. In the **Status Bar**,

- Click **Results View**

To create the strip:

1. Click **Results** > **Create Strip** ( )
2. The Result Strip property set is displayed in the **Properties Window**. Adjust the properties in this set to specify:
    - The start and end width of the strip
    - The result type (Average, Centerline, or Maximum)
    - The number of stations per metre along the strip
    - The number of points per metre across the strip at each station

3. Click a point where the strip is to start.
4. Click a 2nd point where the strip is to end.

   (Neither start or end points have to match nodes in the mesh.)
5. *Tekla Structural Designer* will create a strip between the points that you identified.
6. Either continue to place further strips, or if done, press [Esc] to exit.

How do I delete a Result Strip?

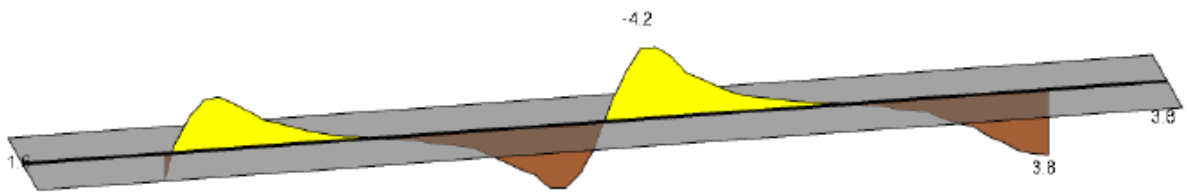
To be able to delete a strip you must first ensure that **Result Strips** *are switched on in Scene Content.*

1. Open a View containing the strips to be deleted.
2. Click **Delete**
3. Click the strip to be deleted.

How do I view the results for a Result Strip?

Once a Result Strip has been defined in the model you can obtain results for it without having to re-run the analysis (providing an analysis has been run previously):

1. Open a 3D View containing the strips to be viewed.
2. Select the analysis type required from the drop list in the **Result Type** group of the Results tab.
3. Select the load case or combination you want to display should be selected from the Loading drop list.
4. From the **2D Strip Results** group on the Results tab, choose the effect that you want to be displayed.
5. *Tekla Structural Designer* will display the chosen effect on the strip along with the maximum positive and negative values (calculated in accordance with the method specified in the strip properties).



How are the strip results calculated?

Along the strip center line there are a user defined number of **stations**. At each station there is a transverse line with a user defined number of **points** along it.

Final results are always given by **station** and obtaining them may or may not use **points**.

There are three alternative ways to calculate the results:

- ***Normal*** - the results on the center of the Result Strip are calculated at each station

  The shell elements local to each station are considered and a weighted average force is calculated based on the distance of the element nodes from the station.

  This is repeated for all stations along the center line of the strip to give the results for the strip.
- ***Maximum*** - results on the transverse line across the strip are calculated for each station along the strip.

  The shell elements local to each point are considered and a weighted average force is calculated based on the distance of the element nodes from the point. The maximum result across the strip from all points is taken as the result for the station on the strip center line. Note that maximum values include nodes within the strip.

  The values calculated at points are always weighted averages of results at adjacent nodes - hence they are always less than the peak nodal values.

  This is repeated for all stations along the center line of the strip to give the results for the strip.
- ***Average*** - average over strip width

  The results are obtained in the same way as for the maximum above but in this case are averaged to give the results for each station.

  This is repeated for all stations along the center line of the strip to give the results for the strip.

All forces in the results are rotated to be in the axis system of the Result Strip.

Displaying analysis results for individual members (Loading Analysis View)

The Loading Analysis toolbar is used to view the loading and analysis result diagrams for individual members.

It is activated by right-clicking on a member and selecting **Open Load Analysis View** from the context menu that is displayed.

Loading Analysis toolbar

The **Loading Analysis** toolbar contains the following commands:

Button	Description
Refresh Loading	If any changes have been made to the structure while the loading view has remained open, you must click **Refresh Loading** in order to update the display accordingly.
Result Type (drop list)	Use the drop list to choose the analysis type for which results are to be displayed.
Axial	View the element loading and results for axial and torsion.
Major	View the element loading and results for the major axis.
Minor	View the element loading and results for the minor axis.
Next	Move the cursor to the next set of results along the element.
Prev	Move the cursor to the previous set of results along the element.

How do I open a Loading Analysis View?

1. Right-click the member you want to view and select **Open Load Analysis View** from the context menu that is displayed.

2. Select the load case or combination you want to display from the Loading drop list.

3. Select the analysis type for which you want to see the results from the **Result Type** drop list on the ribbon.

4. If displaying results for a load combination - select whether to view the results based on Strength or Service Factors.

5. Select the axis type to view (Axial, Major, or Minor) from the **Direction** group on the ribbon.

   The results are displayed accordingly and can then be further controlled using the Properties Window.

Displaying RSA Seismic Results in a Loading Analysis View

Loading Analysis Views for 1st or 2nd Order RSA Seismic result types use the same rules as those applied to multi-member Results Views for the same result types, i.e. as follows:

RSA Seismic Loadcases

- Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options Absolute values are determined at various points along each member and then plotted

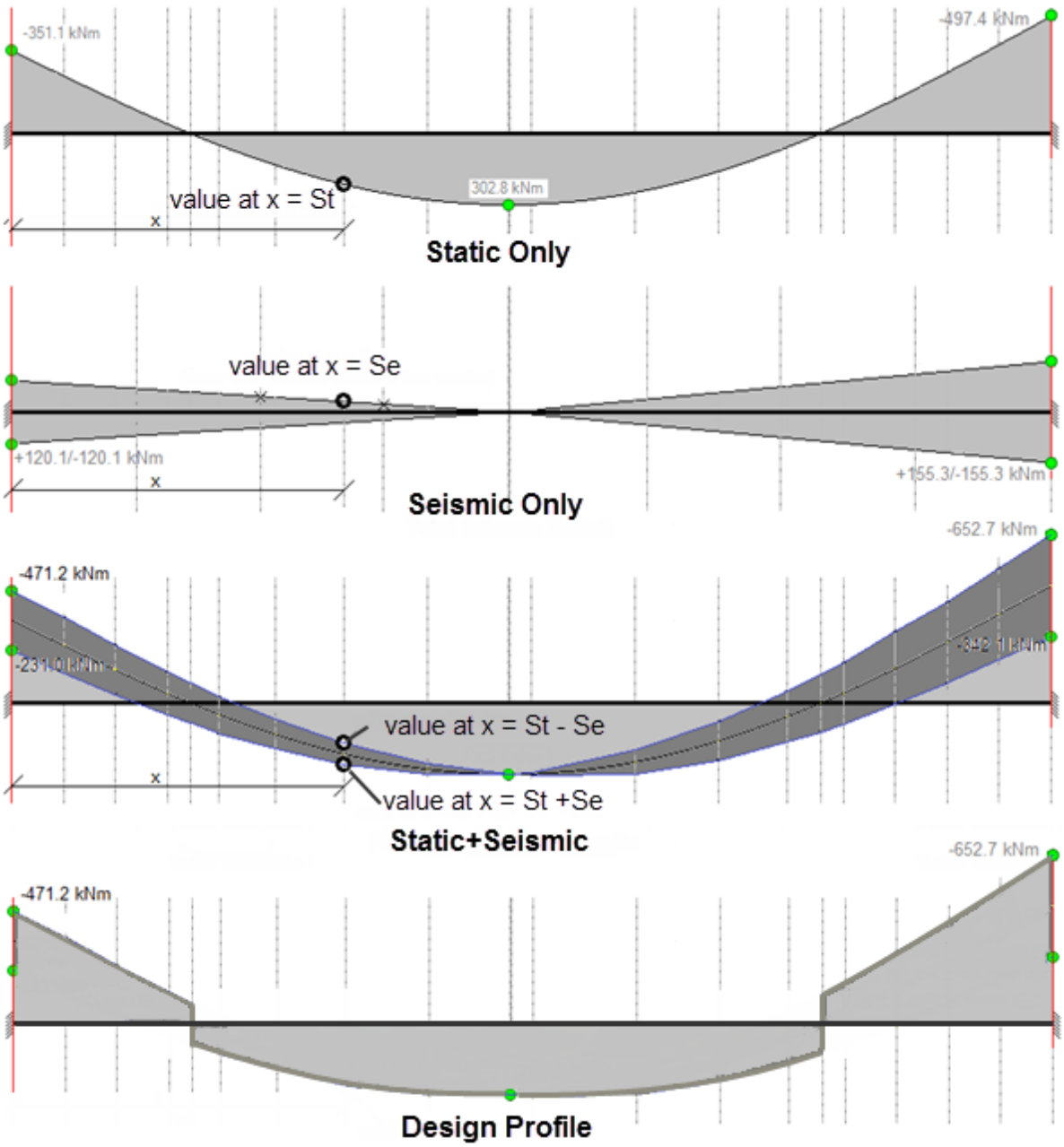
on both the positive and negative side of the diagram, (so that the diagrams are always symmetrical about the base line).

- All relevant modes - a standard enveloped diagram is displayed
- RSA Torsion Loadcases - displayed as per 1st order linear analysis
- Static Loadcases included in the RSA Seismic Combination - displayed as per 1st order linear analysis
- Effective Seismic Weight Combination - not available

RSA Seismic Combinations

A droplist is provided to allow you to view:

- Design Profile - derived from the Static+Seismic result, the Design Profile is always plotted on the same side of the base line as the Static Only result
- Static Only - displayed as per 1st order linear analysis
- Seismic Only - absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram
- Static+Seismic - an envelope is drawn showing the seismic results above and below the static result.
    - Base line is through the static values
    - Top line is static value + seismic value
    - Bottom line is static value - seismic value



Loading Analysis View Properties for Columns

The properties displayed in the Properties Window vary depending on the element type being viewed, for columns the properties are as follows:

Property	Description
General	
Distance	The distance along the member at which the results are displayed.

	For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.
Axial force above	The axial force in the column immediately above the cross section at the distance specified. (This property is only displayed if the Axial direction is selected.)
Axial force below	The axial force in the column immediately below the cross section at the distance specified. (This property is only displayed if the Axial direction is selected.)
Axial force reduced above	The axial force in the column immediately above the cross section at the distance specified, taking into account imposed load reductions. (This property is only displayed if the Axial direction is selected.)
Axial force reduced below	The axial force in the column immediately below the cross section at the distance specified, taking into account imposed load reductions. (This property is only displayed if the Axial direction is selected.)
Torsion moment above	The torsion in the column immediately above the cross section at the distance specified. (This property is only displayed if the Axial direction is selected.)
Torsion moment below	The torsion in the column immediately below the cross section at the distance specified. (This property is only displayed if the Axial direction is selected.)
Shear above	The major or minor shear force immediately above the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Shear below	The major or minor shear force immediately below the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)

Moment above	The major or minor moment immediately above the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Moment below	The major or minor moment immediately below the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Ecc. Moment above	The major or minor moment due to eccentricity immediately above the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Ecc. Moment below	The major or minor moment due to eccentricity immediately below the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Relative deflection	The relative deflection in the major or minor direction at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Load above	The applied distributed load in the major or minor direction immediately above the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Load below	The applied distributed load in the major or minor direction immediately below the cross section at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Force	The applied point load in the major or minor direction at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Moment	The applied moment in the major or minor direction at the distance specified. (This property is only displayed if the Major or Minor direction is selected.)
Show axial force	If cleared, the axial force diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)

Show axial force reduced	If cleared, the axial force diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show torsion moment	If cleared, the torsion diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show loading	If cleared, the loading diagram is removed from the view. (This property is only displayed if the Major or Minor direction is selected.)
Show shear	If cleared, the shear diagram is removed from the view. (This property is only displayed if the Major or Minor direction is selected.)
Show moment	If cleared, the moment diagram is removed from the view. (This property is only displayed if the Major or Minor direction is selected.)
Show dimensions	If cleared, the dimensions are removed from the view. (This property is only displayed if the Axial direction is selected.)
Show extremes	If cleared, the max and min values are removed from the view.

Loading Analysis View Properties for Beams

The properties displayed in the Properties Window vary depending on the element type being viewed and the **Result Type** selected, for beams the properties for the different Result Types are as follows:

Result Type: Axial

Property	Description
General	
Distance	The distance along the member at which the results are displayed. For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.

Axial force left	The axial force in the beam immediately to the left of the cross section at the distance specified.
Axial force right	The axial force in the beam immediately to the right of the cross section at the distance specified.
Torsion moment left	The torsion in the beam immediately to the left of the cross section at the distance specified.
Torsion moment right	The torsion in the beam immediately to the right of the cross section at the distance specified.
Relative angle of twist	The relative angle of twist (due to torsion) in the beam cross section, at the distance specified.
Angle of twist derivative	A droplist allowing the selection of the derivative of angle of twist: • First • Second • Third (This property is displayed for "Open" sections only).
Angle of twist derivative left	The relative angle of twist (due to torsion) in the beam cross section, immediately to the left of the distance specified. (This property is displayed for "Open" sections only).
Angle of twist derivative right	The relative angle of twist (due to torsion) in the beam cross section, immediately to the right of the distance specified. (This property is displayed for "Open" sections only).
Show axial force	If cleared, the axial force diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show torsion moment	If cleared, the torsion diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show relative angle of twist	If cleared, the relative angle of twist diagram is removed from the view.
Show angle of twist derivative	If cleared, the angle of twist derivative diagram is removed from the view. (This property is displayed for "Open" sections only).
Show dimensions	If cleared, the dimensions are removed from the view.

Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor, Major Principal, Minor Principal

Property	Description
General	
Distance	The distance along the member at which the results are displayed. For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.
Shear left	The major or minor shear force immediately to the left of the cross section at the distance specified.
Shear right	The major or minor shear force immediately to the right of the cross section at the distance specified.
Moment left	The major or minor moment immediately to the left of the cross section at the distance specified.
Moment right	The major or minor moment immediately to the right of the cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load left	The applied distributed load in the major or minor direction immediately to the left of the cross section at the distance specified.
Applied load right	The applied distributed load in the major or minor direction immediately to the right of the cross section at the distance specified.
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.

Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflection diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Analysis results sign conventions

Tekla Structural Designer adopts the standard convention for engineering / CAD software that lower case x, y, z represent local coordinate systems whereas upper case X, Y, Z represent the global coordinate system.

Introduction

Axis Systems

The following axis systems are relevant in the software:

- Global Coordinate System - the global XYZ axis system within which all other systems exist
- Building Directions 1 and 2 - the principle axes of the building - Dir 1 being rotated at an angle to global X in the horizontal plane.
- User Coordinate System - a local coordinate system defined by the system or the user
- 1D Member Local Coordinate System - the local coordinate system applicable to all 1D members - beams, columns, braces etc
- Mid-pier Wall Coordinate System - the local coordinate system applicable to walls modeled using the mid-pier option
- 2D Member Local Coordinate System - the local coordinate system applicable to all 2D members - walls and slabs
- Result Line Coordinate System - the local coordinate system applicable to Result Lines
- Result Strip Coordinate System - the local coordinate system applicable to Result Strips
- Foundation Reaction Coordinate System - the local coordinate system applicable to foundations

General

The global (XYZ) and local (xyz) axis systems all follow the right hand rule.

- x-axis = pointing index finger
- y-axis = crooked middle finger
- z-axis = extended thumb

And the directions of +ve rotation

- +ve rotation about x: the y-axis moves toward the z-axis.
- +ve rotation about y: the z-axis moves toward the x-axis.
- +ve rotation about z: the x-axis moves toward the y-axis.

Object Orientation

Tekla Structural Designer takes account of an object's orientation when displaying the analysis results. Therefore, to apply the sign convention correctly you need to know which is end 1 and which is end 2 for beams/walls and you also need to know which is Face A for columns.

To determine the orientation

In the Members group within Scene Content, if you switch the **Direction** option on, then *Tekla Structural Designer* shows an arrow on all beams, walls and columns.

- The arrow points from the start to the end of beams/walls and is drawn along the top face.
- The arrow points from the bottom to the top of columns along Face A. Looking down from the top of a column, Face B, C, and D then follow in the clockwise direction.

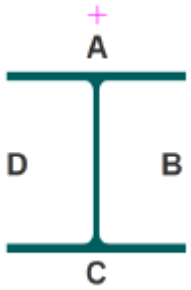
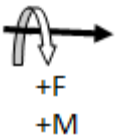
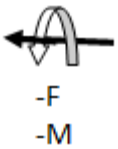


Diagram Conventions

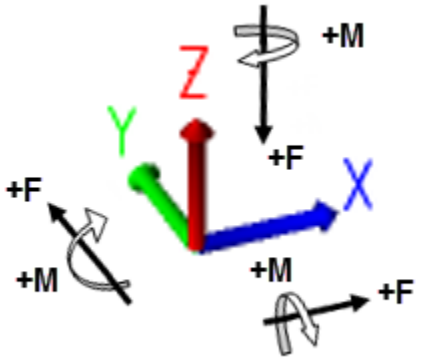
All arrows should point in the direction of the force or moment - and so are reversed for -ve forces and moments - eg



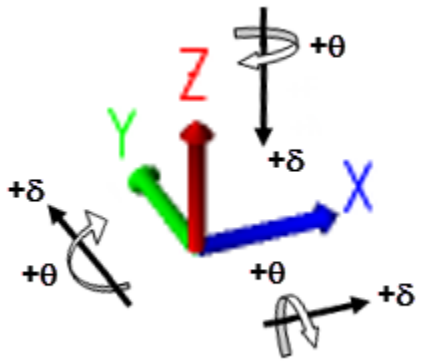


Global Coordinate System

Global Axis System and Applied Load Directions



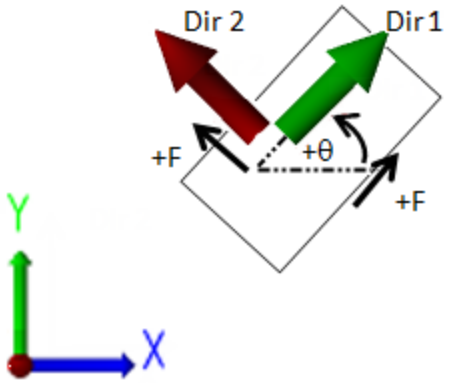
Resulting Deflection Directions



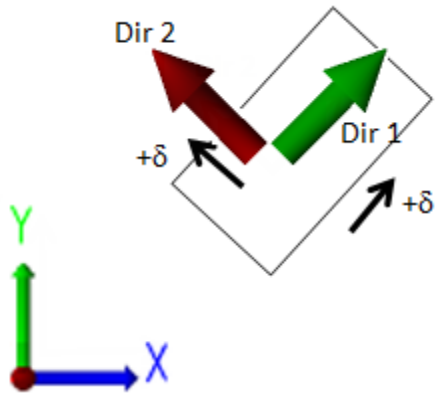
Building Directions 1 and 2

Building Directions and Applied Load Directions

Global axes (+ve Z vertically up) and angle between X and Dir 1 = θ where θ is +ve in RH rule about Z



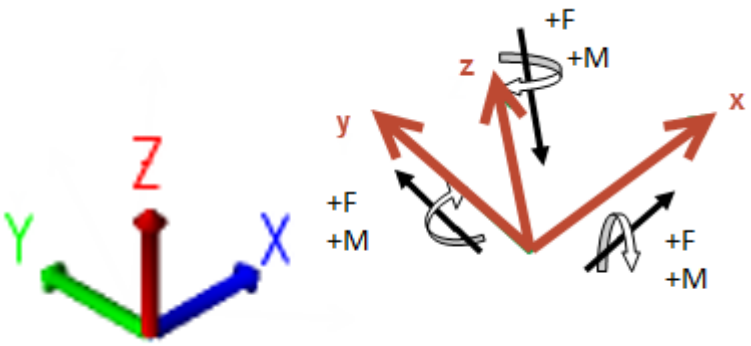
Resulting Deflection Directions



User Coordinate System

UCS Axis System and Applied Load Directions

A UCS can be at any angle to the Global Coordinate System

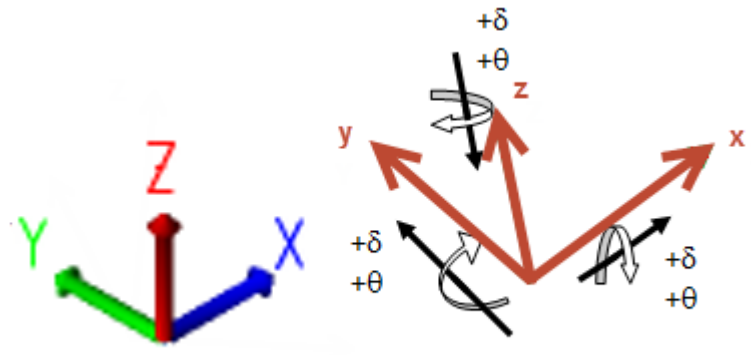


Every support is given a UCS. Automatically created supports under certain objects default to the method below. All other supports default to the global coordinate system.

- Support under a single column/wall rotates the foundation forces to align with the y/z axes of the column/wall

- Support under a mat foundation - uses the global coordinate system

Resulting Deflection Directions

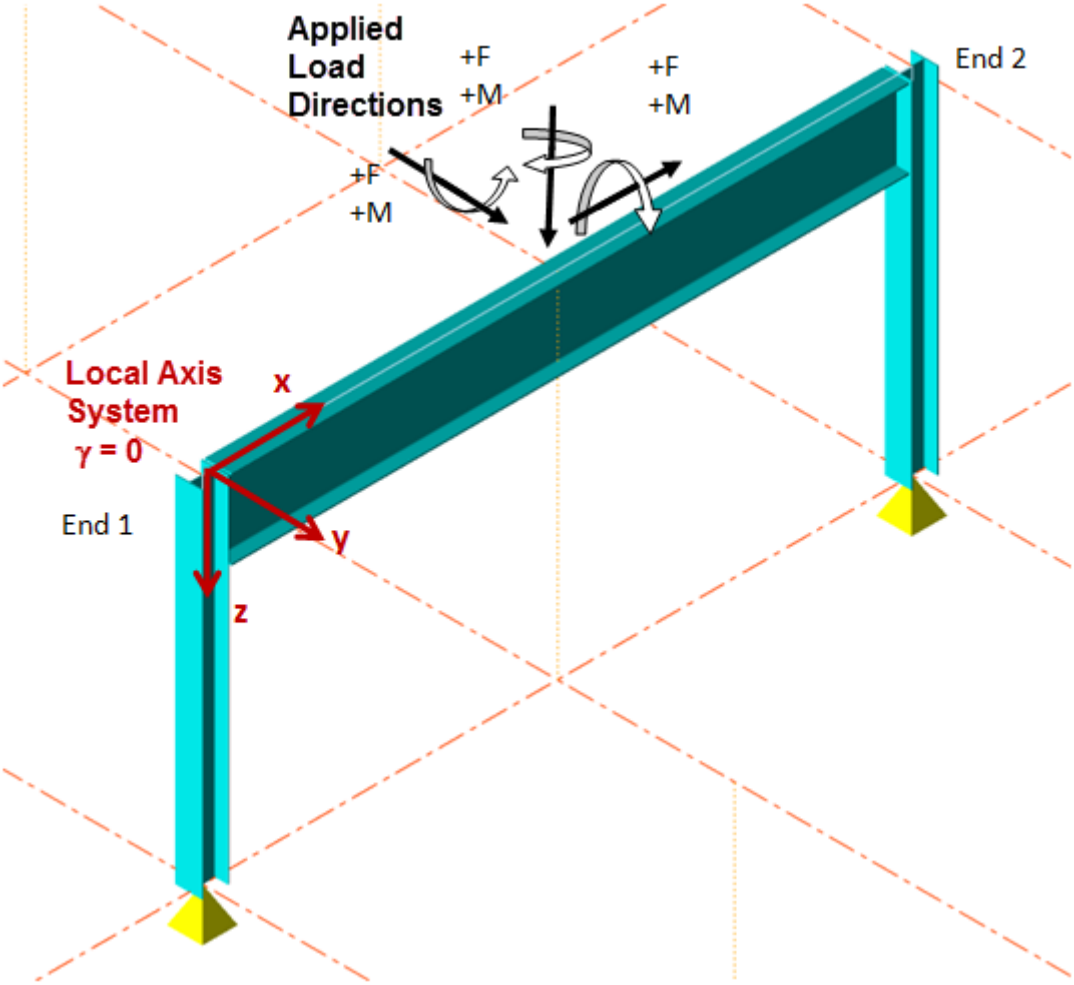


1D Member Local Coordinate System

General case for 1D members

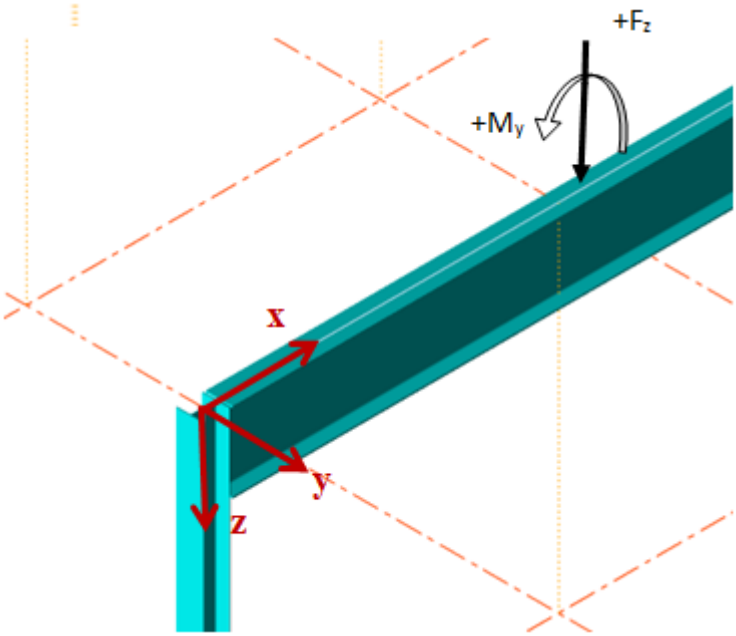
Local Axis System and Local Applied Load Directions

- Local x along member - end 1 to end 2
- When rotation angle γ = 0,
- Local z lies in the plane created by the local x axis and the global Z axis. The global Z component of the local z axis is always negative
- Local y according to RH rule
  - γ = +ve clockwise rotation of y and z about x looking towards +ve x

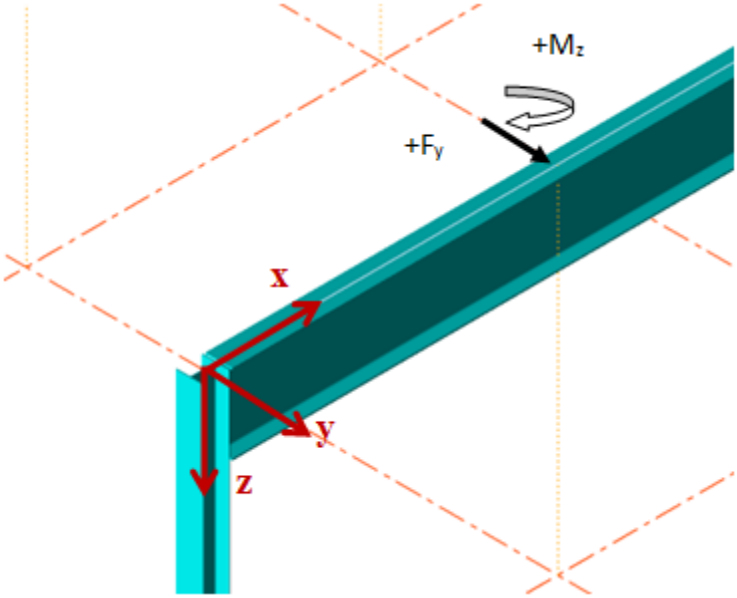


With the local axis aligned as shown above, it can be seen that:

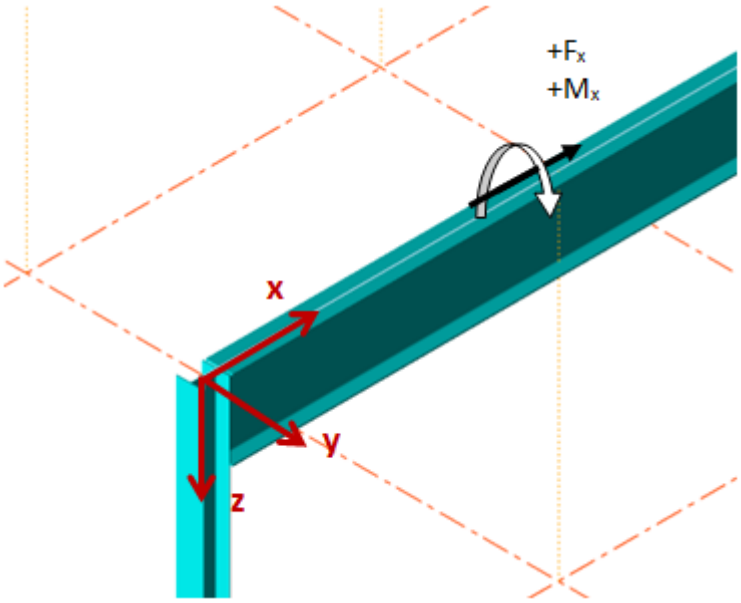
- Major direction loads act in the local xz plane; positive Major direction Force (Fz), and Moment (My) are shown below:



- Minor direction loads act in the local xy plane; positive Minor direction Force (Fy), and Moment (Mz) are shown below:



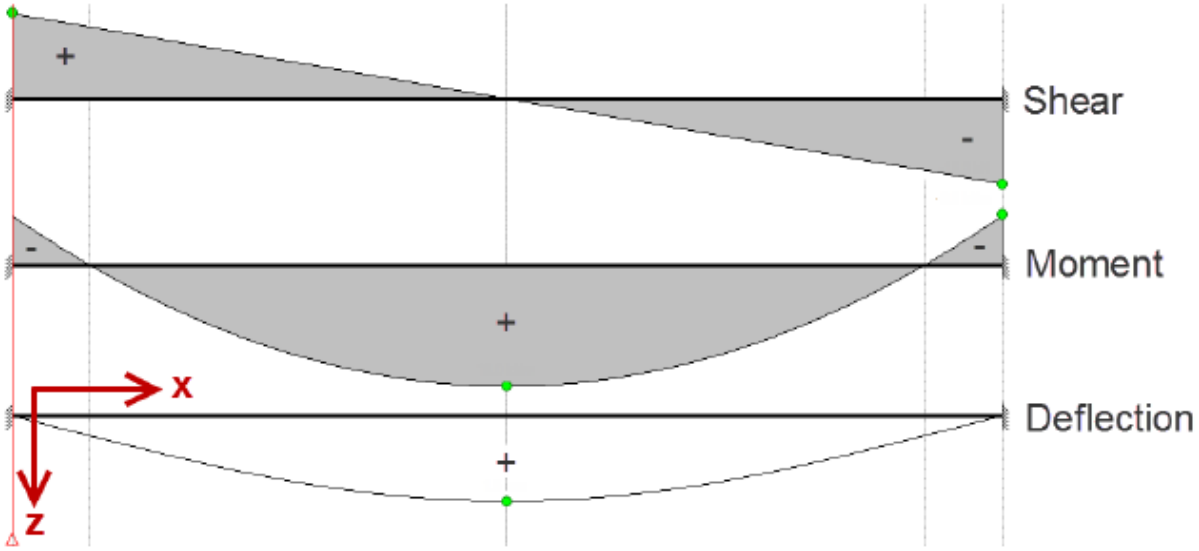
- Positive Axial Force (Fx) acts in the local x direction, and positive Axial Moment (Mx) acts clockwise about local x as shown below:



Result Axis System and Directions

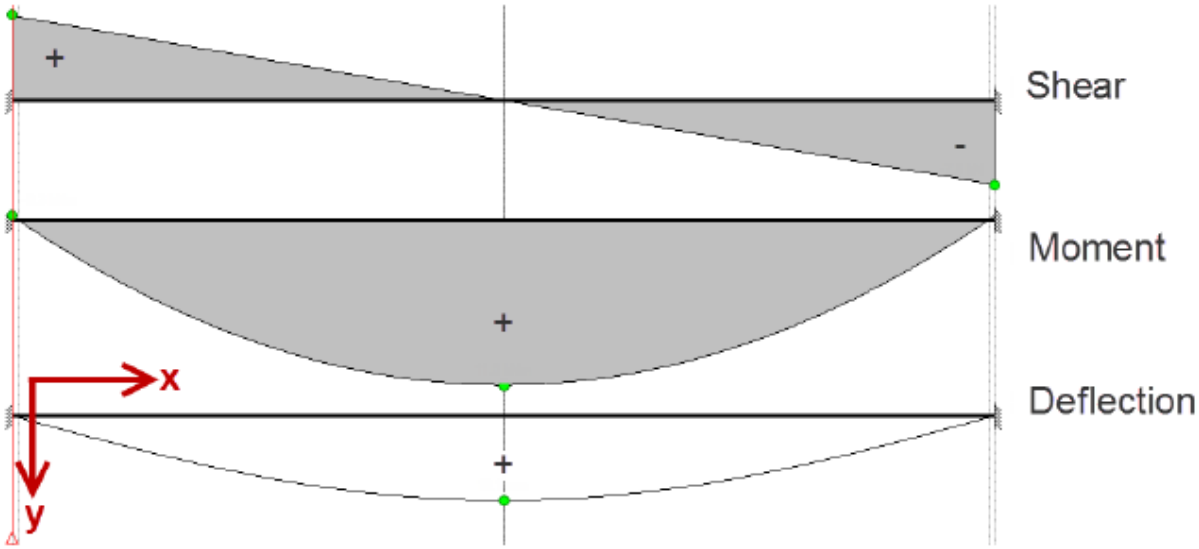
In the Major Axis:

- Moment Major = bending about y
- Shear Major = shear along z

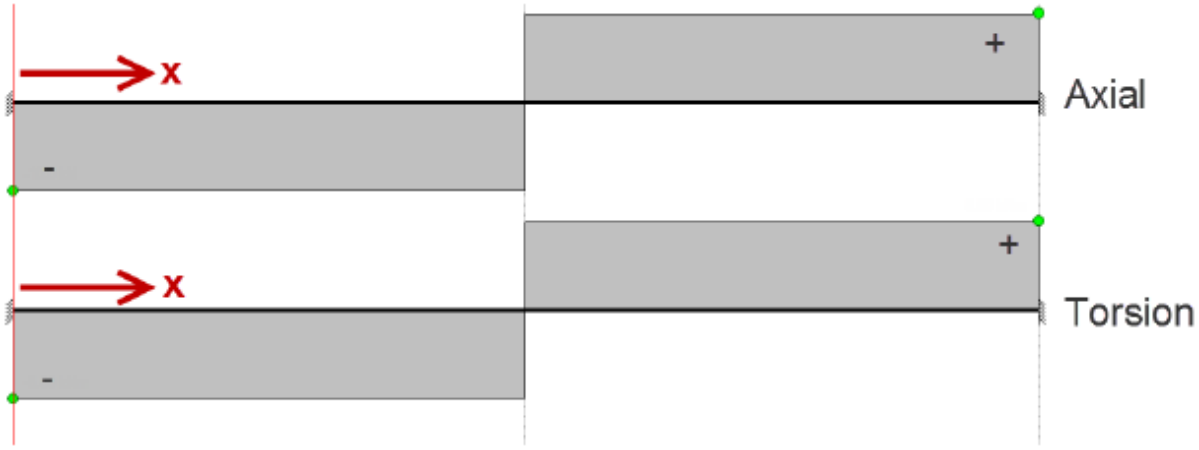


In the Minor Axis:

- Moment Minor = bending about z
- Shear Minor = shear along y



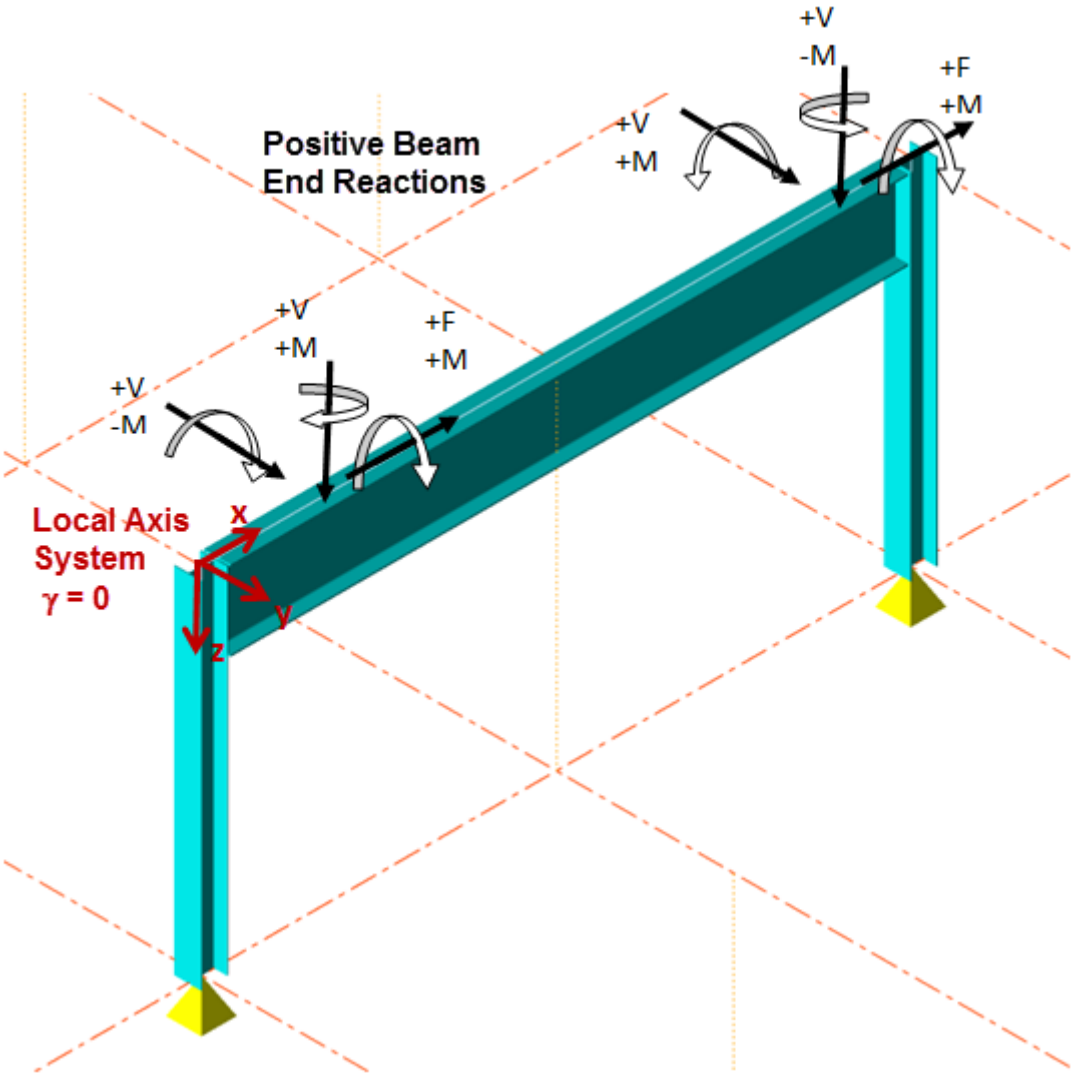
In the axial direction



Member End Forces and Directions

Member End Forces are the forces applied to the rest of the structure by the member.

Positive beam end reactions act as shown for a beam with its Local Axis System indicated below:

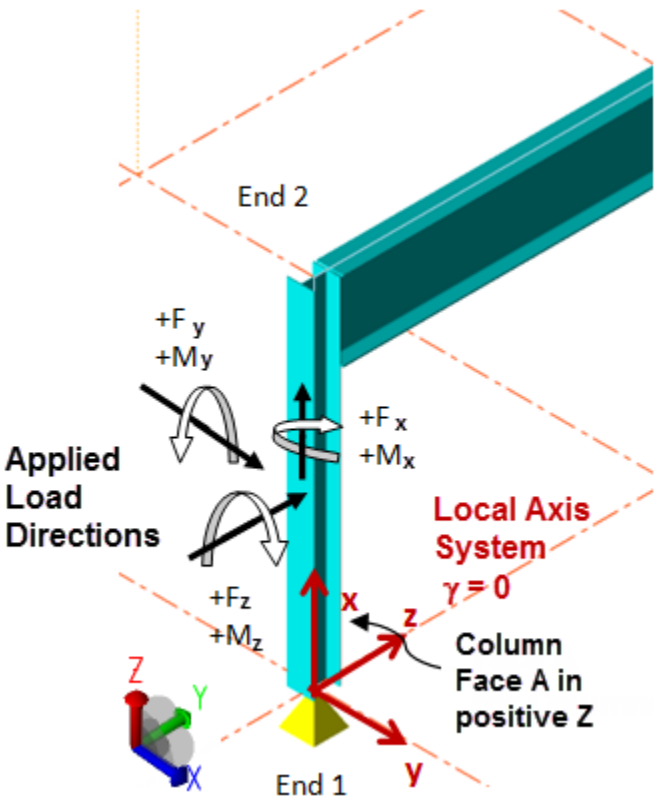


Special case for 1D members

Local Axis System and Applied Load Directions

Local x aligns with global Z (i.e vertical)

- When rotation angle γ = 0,
- Local y aligns with global X
- Local z according to RH rule
  - γ = +ve clockwise rotation of y and z about x looking towards +ve x



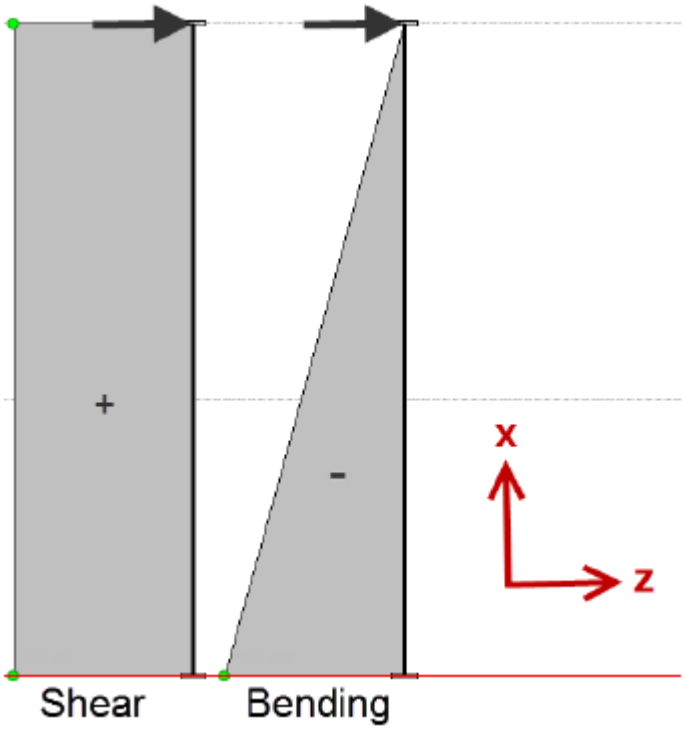
Applied Force directions are as follows:

- Major direction loads act in the local xz plane; positive Major direction Force (Fz), and Moment (My) are as shown above.
- Minor direction loads act in the local xy plane; positive Minor direction Force (Fy), and Moment (Mz) are as shown above.
- Positive Axial Force (Fx) acts in the local x direction, and positive Axial Moment (Mx) acts clockwise about local x as shown above.

Result Axis System and Directions

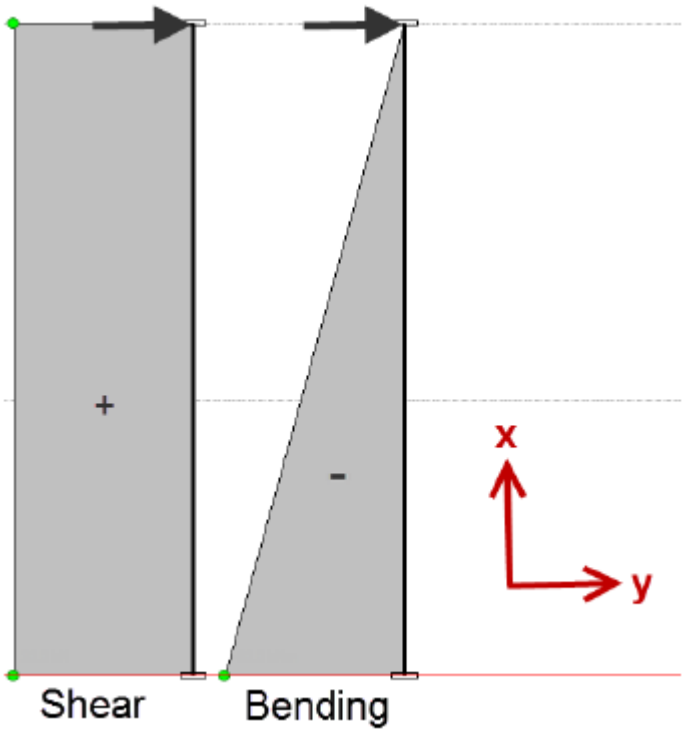
In the Major Axis:

- Moment Major = bending about y
- Shear Major = shear along z

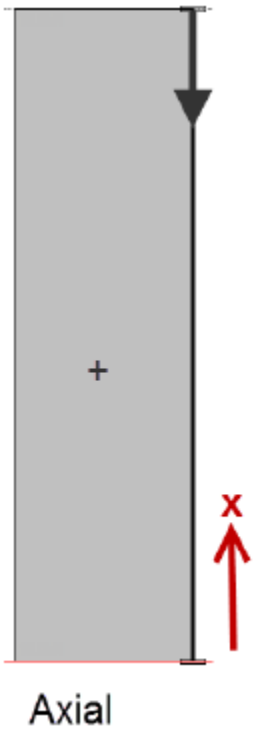


In the Minor Axis:

- Moment Minor = bending about z
- Shear Minor = shear along y



In the axial direction



2D Member Local Coordinate System

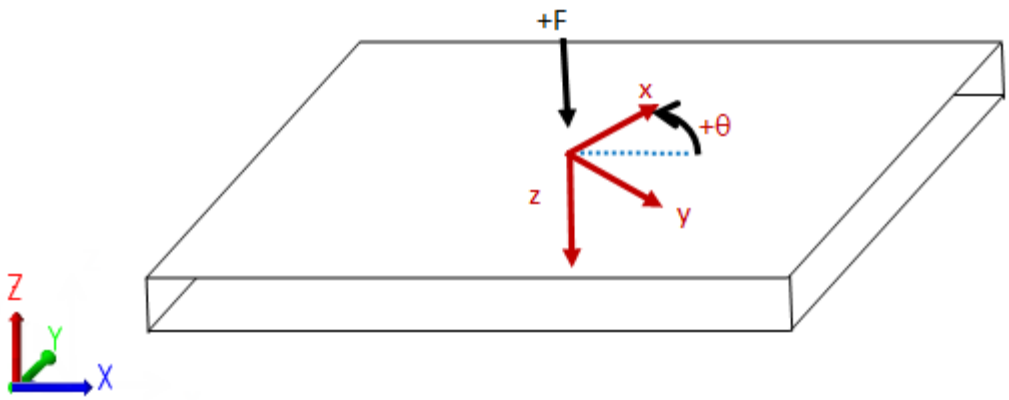
Horizontal Panel Local Axis System and Applied Load Directions

Horizontal panel local axes are -

Local z is normal to the plane of the panel

When θ = 0,

- Local x is in the plane of the panel, aligned with global X and +ve in +ve global X
- Local y is in the plane of the panel and follows the RH rule
  - θ = +ve clockwise rotation of x and y about z looking towards +ve z

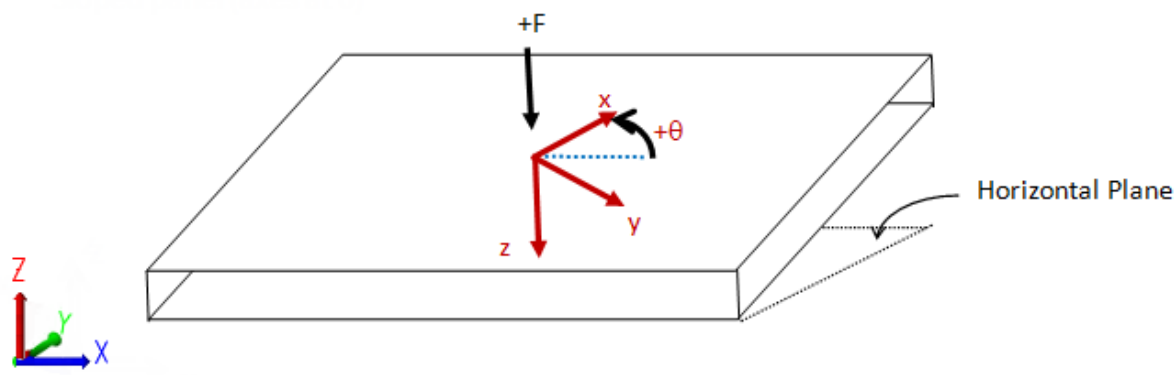


Vertical and Sloped Panel Local Axis System and Applied Load Directions

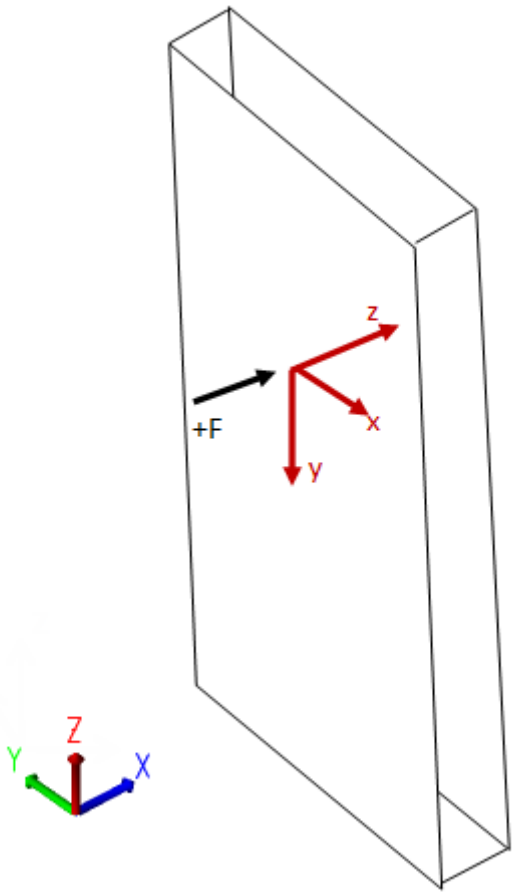
Vertical and Sloped panel local axes are -

- Local z is normal to the plane of the panel
- When θ = 0,
- Local x is in the plane of the panel and in a horizontal plane
- Local y is in the plane of the panel and follows the line of greatest slope of the plane (+ve in the direction of +ve global)
- θ = +ve clockwise rotation of x and y about z looking towards +ve x

Sloped panel (axes at θ)

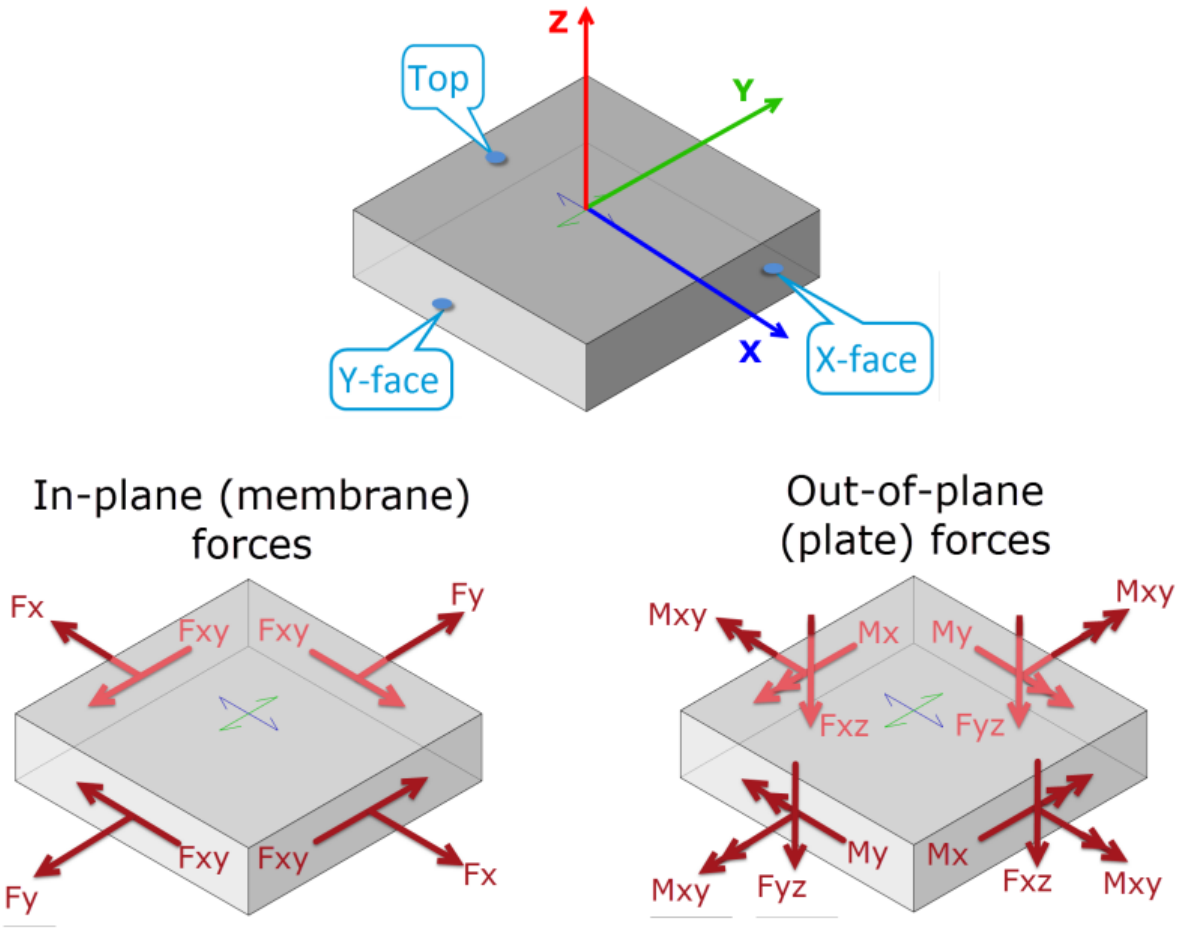


Vertical panel (axes when θ = 0)



2D Member Forces Sign Convention

The diagram below illustrates the forces and the panel and 2D element axis system (for results).



The arrows show positive force directions. The double-arrow convention is used for moments - the moment is around the double-arrow, positive being clockwise when looking in the arrow direction. The forces act on a 'face' cut (anywhere) in the FE mesh perpendicular to the force direction. Thus for example; Mx acts on the X-face which is perpendicular to the X axis and is moment resulting from spanning in the X direction; Fxz is the out-of-plane shear force acting on the X-face, and so on. The Wood Armer Design moments, denoted by the suffix 'd', act in the same manner as the unprocessed moments without this suffix; thus Mdx acts in the same manner as Mx and so on. The design moments are further classified into top and bottom components (hogging and sagging respectively) for the slab design process.

You may note that the positive Z-axis direction (up) follows the right-hand rule and is not the same as that for the 2D Member Local Coordinate System. The reason for this apparent contradiction is that the 2D member local co-ordinate system for the applied load directions shows the positive applied load direction convention which - for Z only - is opposite to that of the Global and 2D element axes.

The sign convention for 2D Member forces is not the same as that of 1D elements.

- For moments; a positive moment creates tension in the Top surface of the shell. Hence 'hogging' moment for a slab (e.g. moment over a supporting column) is positive while

'sagging' (span) moment is negative. The conventions for wall results are exactly the same, so understanding the axis directions they can be interpreted in the same way.

- For axial load (Fx and Fy); compression is negative.
- We are not usually concerned about the sign of shear, however, for completeness; out-of-plane shear (Fxz and Fyz) is positive when shear is such that moment is increasing in the positive X or Y direction.

Result Line Coordinate System

Centered on the centroid of the cut section

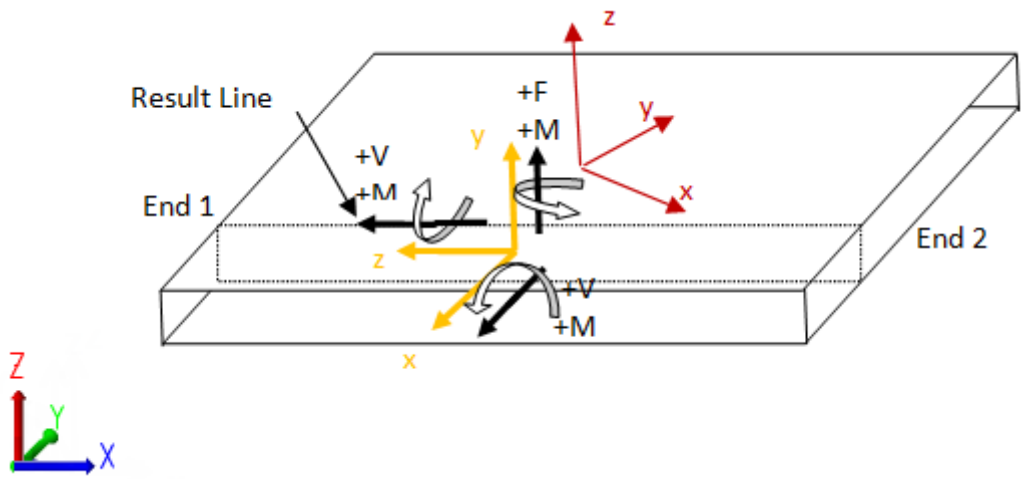
- z axis lies along the Result Line (+ve - end 2 to end 1)
- y axis normal to plane of mesh (generally +ve in the +ve Z direction, special case +ve x towards +ve Z)
- x axis follows the RH rule and lies in the mesh - x is therefore perpendicular to the cut line

The results from a Result Line are exactly as that for a mid-pier model when the cut is horizontal and the cut direction matches the direction required.

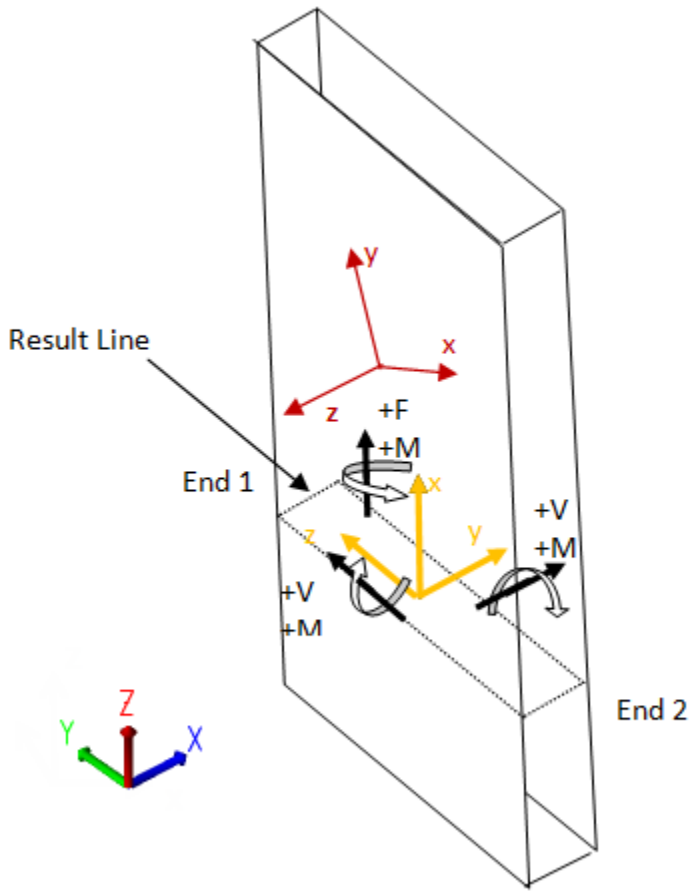
Result Axis System

- In the Major Axis = bending about y and shear along z
- In the Minor Axis = bending about z and shear along y
- Axial and torsion = force in x and bending about x

General Case



Special Case



Result Strip Coordinate System

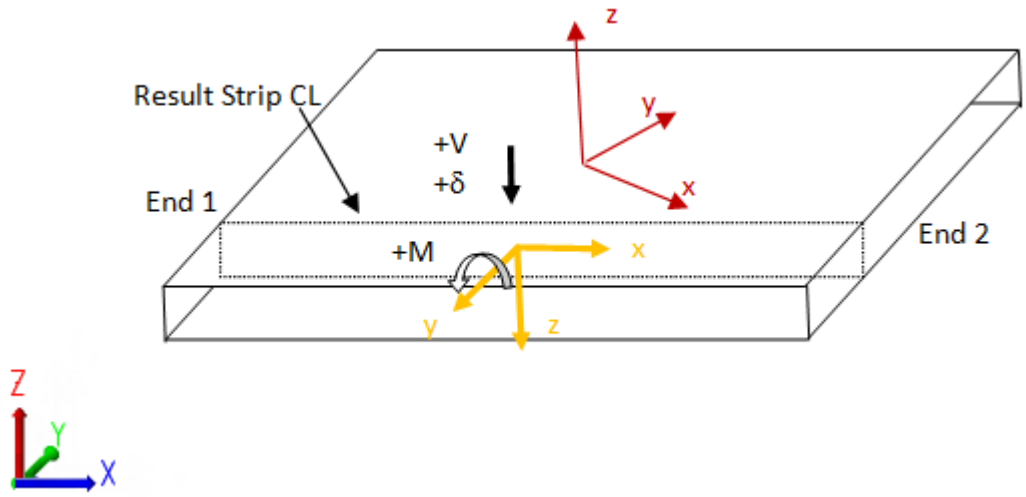
Centered at each station along the strip center line (single or several continuous strips)

- z axis normal to plane of mesh (generally +ve in the -ve Z direction, special case +ve x towards +ve Z)
- x axis lies along the Result Strip (+ve - end 1 to end 2)
- y axis lies along the transverse line to the strip and follows the RH rule - y is therefore perpendicular to the strip line

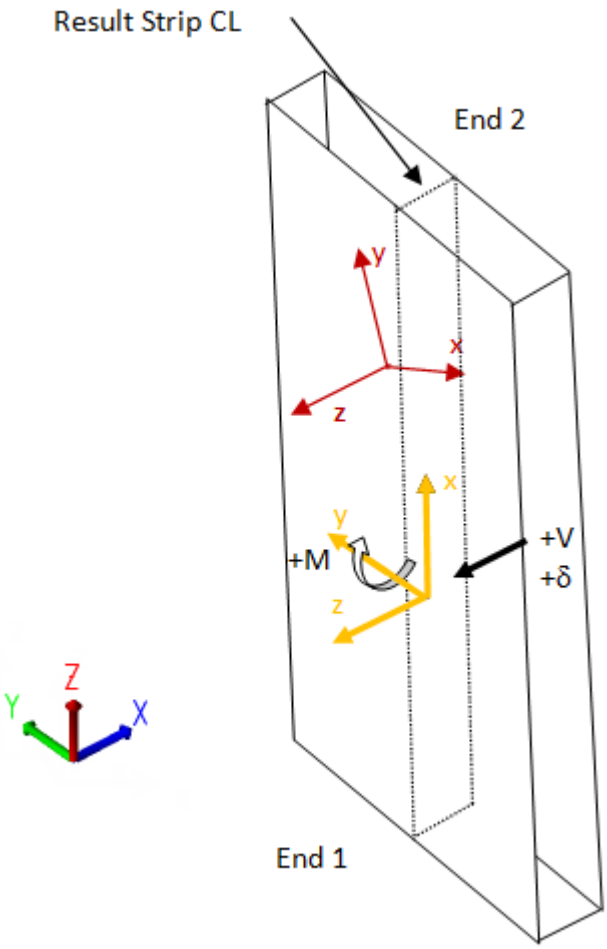
Result Axis System

- Deflection - in the z axis
- Out of plane moment about the y axis
- Shear in the z axis

General Case



Special Case

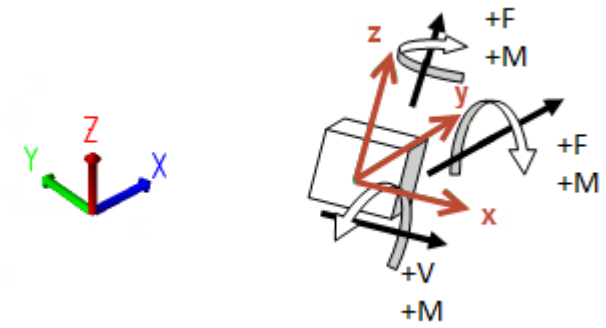


Foundation Reaction Coordinate System

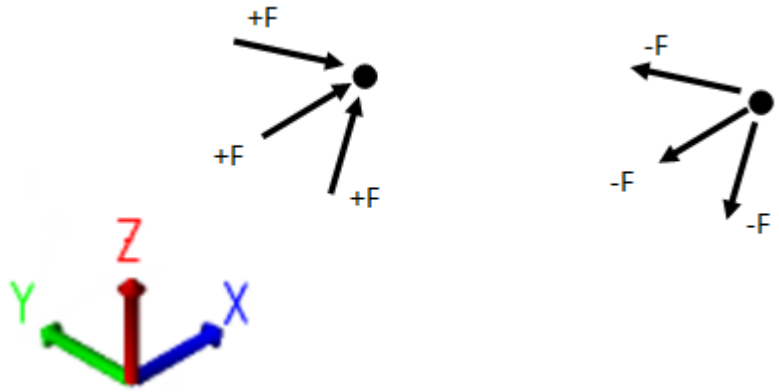
Result Axis System and Directions

The foundation reaction coordinate system is aligned with the coordinate system for the support node whether that is the Global Coordinate System or a User Coordinate System.

Reactions are the forces applied to the structure by the foundation.



Reactions



Design Guide

Combined analysis and design is performed from the Design toolbar by running:

- Design Steel - to design all steel beams and columns
- Design Concrete - to design all concrete beams, columns and walls
- Design All - to design all steel beams and columns; concrete beams, columns and walls

By selecting Gravity, Static, or RSA you control which classes of combination are considered.

Slabs require a certain amount of user interaction and are therefore designed separately by running:

- Design Slabs - to design all slab panel areas that lie outside patches
- Design Patches - to design all slab panel areas that lie inside patches
- Design Punching Shear - to design all punching check items (Not available for Australia Head Code)

Floor vibrations can also be checked by running:

- Check Floor Vibration - to establish the response of the floor to dynamic excitation

Design toolbar

The **Design** toolbar contains the following commands:

Button	Description
Options	Opens the Design Options dialog. See Design Options
Validate	Validates the model for any design issues which might exist. See Model Validation
Design Steel (Gravity)	Initiates an analysis-design process for steel members for gravity combinations only. See Design Steel (Gravity)
Design Steel (Static)	Initiates a complete static analysis-design process for all steel members in the structure. See Design Steel (Static)

Design Steel (RSA)	Initiates a complete RSA analysis-design process for all steel members in the structure. See Design Steel (RSA)
Design Concrete (Gravity)	Initiates an analysis-design process for all concrete members in the structure for gravity combinations only. See Design Concrete (Gravity)
Design Concrete (Static)	Initiates a complete static analysis-design process for all concrete members in the structure. See Design Concrete (Static)
Design Concrete (RSA)	Initiates a complete RSA analysis-design process for all concrete members in the structure. See Design Concrete (RSA)
Design All (Gravity)	Initiates an analysis-design process for all members in the structure for gravity combinations only. See Design All (Gravity)
Design All (Static)	Initiates a complete static analysis-design process for all members in the structure. See Design All (Static)
Design All (RSA)	Initiates a complete RSA analysis-design process for all members in the structure. See Design All (RSA)
Patch Column (on drop list)	Creates a column patch which adopts the specified create column patch properties set See How do I create a column patch?
Patch Beam (on drop list)	Creates a beam patch which adopts the specified create beam patch properties set See How do I create a beam patch?
Patch Wall (on drop list)	Creates a wall patch which adopts the specified create wall patch properties set See How do I create a wall patch?
Patch Panel (on drop list)	Creates a panel patch which adopts the specified create panel patch properties set See How do I create a panel patch?

Design Patches	Designs or checks slab reinforcement in all slab areas inside patches. See Slab design, patches and punching shear
Design Slabs	Designs or checks slab reinforcement in all slab areas that are not inside patches. See Slab design, patches and punching shear
Add Check	Used to add a punching shear check around a column or wall perimeter. See How do I create a Punching Check item?
Design Punching Shear	Performs the punching shear checks. See Perform Punching Shear Checks
Add Check	Used to add a floor vibration check for a slab area. See How do I create a Floor Vibration Check item?
Check Floor Vibration	Performs the floor vibration checks. See How do I check vibration for all Floor Vibration Check items?

Related topics

- Commands on the ribbon toolbars

Design Options

How to apply and manage Design Options

To modify design options in the current project

1. Click **Design** > **Options...** ( )
2. Review and edit the settings as required.
3. If you change any of the settings, click:
   - **OK** - to apply the changes directly to the current project, or
   - **Save...** - to save the changes back to the active settings set (to act as defaults for future projects), or

- **Cancel...** - to cancel the changes

You can also click:

- **Load...** - to revert to the design options specified in the active settings set.

To modify design option defaults for future projects

1. Click **Home** > **Settings** ( )
2. In the **Design Options** page of the dialog, select the settings set to be updated, then review and edit the settings as required.

> *You can update any settings set simply by selecting it from the droplist, it does not need to be active.*

3. If you change any of the settings, click:

- **OK** - to save the changes to the selected settings set (to act as defaults for future projects when that set is active), or
- **Cancel** - to cancel the changes

Design Options-General

General

Use multi-core processors for design

leave this option selected to take advantage of multi-core processing during the design phase.

Run chase-downs concurrently

leave this option selected to take advantage of multi-core processing during the chase-down analysis phase.

> *The above options are only displayed when the design settings are accessed from the Settings dialog on the Home toolbar.*

Gravity Design

Fix column nodes horizontally

When selected, this option automatically applies lateral translational fixed supports to column nodes for the **3D Analysis** performed during the gravity design processes. This is useful at the early design stage when lateral systems have not been established. The

automatic lateral supports can remove unwanted lateral displacements that may prevent the analysis from completing and/ or produce unrealistic forces.

The 'Fix Column nodes horizontally' setting does not apply to:
- Column nodes that are in a rigid floor diaphragm,
- Analysis run as a separate process,
- Analyse All (Static),
- 3D Analysis performed during Design...(Static).

Design Options-Analysis

Analysis

The Analysis options on this page allow you choose whether a first or second order **3D Analysis** is performed.

For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design.

Solve lateral loadcases in isolation if non-linear analysis is required during design

In non-linear analysis, lateral loadcases will often fail to solve at all or take a long time to converge when considered in isolation - this is especially true for compression only foundation mats. Hence it is suggested that this check box is left cleared.

If you need to see the results for the individual lateral loadcases you can either run the analysis directly from the Analyses ribbon, (or if you know that the analysis does not fail, you can select this check box.)

Stability Coefficient Tolerance

If very small deflections were to be used in the calculation of the Stability Coefficient. then potentially very high Stability Coefficients could erroneously be reported.

To prevent this, the Stability Coefficient Tolerance provides you with a means to control the value of deflection that can safely be ignored. setting the analysis to first-order If the second order drift is less than the tolerance defined here (default stack height/10000), the Stability Coefficient is returned as 'N/A' with a note to say that the 'Drift is small enough to be ignored'.

Reduced Stiffness Factor

This factor is only exposed once the analysis is set to second-order. For correct design to the AISC Specification using the DAM, this should be set to 0.8.

As an alternative to setting the analysis to first-order to explore the reason for any second order analysis failure, it is possible to alter this factor. If you set it to a value of say 10, this will stiffen both the Modulus of elasticity (E) and the shear modulus of elasticity (G) by a factor of 10 in the second order analysis. Although the results will not be able to be used for a valid design, it should now be possible to run the analysis to see which member might fail

a design and hence be the cause of the analysis instability. This factor can then be reduced towards 0.8 for further investigation.

•

Design Options-Concrete-Reinforcement Parameters

Reinforcement anchorage length parameters

These parameters are used in the calculation of the ultimate bond stress, from which the anchorage lengths are determined.

Eurocode:

Plain Bars Bond Quality Modifier
Allowable range 0.1 - 1.0; default 0.5

Type-1 Bars Bond Quality Modifier
Allowable range 0.1 - 1.0; default 0.8

Type-2 Bars Bond Quality Modifier
Allowable range 0.1 - 1.0; default 1.0

ACI:

Plain Bars Bond Quality Modifier
Allowable range 0.1 - 1.0; default 0.5

Deformed Bars Bond Quality Modifier
Allowable range 0.1 - 1.0; default 1.0

Design Options-Concrete-Beam

This page has several sub-pages for setting up the options that are used in the beam design.

Concrete - Beam - Reinforcement Settings

This page is used to set limits on the ranges and spacings of bars used in the design.

General

Country
The available bar sizes and properties are taken from the Materials - Reinforcement database according to the country selected.

Longitudinal bars

Minimum bar size
Sets the minimum allowable bar size that can be used in the design process.

Maximum bar size
Sets the maximum allowable bar size that can be used in the design process.

Minimum side bar size

Sets the minimum allowable side bar size that can be used in the design process.

Minimum top steel clear spacing

Sets the minimum allowable top steel clear spacing that can be used in the design process.

Minimum bottom steel clear spacing

Sets the minimum allowable bottom steel clear spacing that can be used in the design process.

Maximum tension steel spacing

Sets the maximum allowable tension steel spacing that can be used in the design process.

Maximum compression steel spacing

Sets the maximum allowable compression steel spacing that can be used in the design process.

Use single bars when beam width <=

Single bars are only permitted in beams less than the width specified.

Steel overstrength factor (ACI only)

This factor is used in seismic design to account for the reinforcement steel yielding overstrength.

Short Span

Short Span maximum length

Spans smaller than the value set here are treated as short spans. Support bars of short spans are merged with the span bars.

Concrete - Beam - Detailing Options

Settings on this page relate to the detail that is produced, but have no effect on the design.

Longitudinal Bars

Use same size bars in multilayer arrangements

Select to use the same bar size in each layer.

Use same number of bars in each layer

Select to use the same number of bars in each layer.

Cut Length

Cut length rounding increment

Cut lengths are rounded to the value specified.

Longitudinal Bar Curtailment

Merge identical longitudinal bars if appropriate

Merging is only possible in some of the standard bar patterns - top bar patterns 1 and 2 don't have any bars that can merge.

In other patterns, bars of the same size, number and position can be merged, provided that the total length of the merged bar does not exceed the max allowable bar length.

The max bar length can be verified from **Home > Materials > Reinforcement** *by clicking an available bar size and then clicking the* **View...** *button.*

Extend top longitudinal support bars symmetrically

Extends the support bars symmetrically to both spans based on the larger effective span length; but only if the spans vary by less than the percentage specified.

Extend top longitudinal support bars by anchorage length

Adds an anchorage lengths to the calculated extension lengths.

Eurocode:

Min anchorage length {value} x dia

The minimum anchorage length as a multiple of bar diameter is controlled with this setting.

ACI:

Min anchorage length of plain bars {value} x dia

The minimum anchorage length as a multiple of bar diameter is controlled with this setting (only used when the rib type is 'Plain').

End Support Curtailment

Extend top span bars to end support

Extends the top span bars of the first and last spans to the end supports.

Use 'U' bars at end support if appropriate

Under certain conditions this option replaces the top and bottom bars at the end support region with 'U' bars. The bars that are joined/merged to create the 'U' bars depend on the top and bottom patterns chosen for the beam.

The anchorage lengths for the resulting 'U' bars are taken as the lengths required for the pair of bars that made the 'U' bar.

Extend and bob all bars for supporting member widths up to

This setting is provided to control the extension/application of end bobs within the supporting member. It allows anchorage lengths to be reduced in situations where they would otherwise extend too far into the supporting member. In the ordinary situation where the supporting members are perpendicular narrow beams the bars are bobbed in the normal way. In situations where the supporting members are wider than the entered value reduced anchorages result (as for anchorage into in-plane walls).

The setting is applied at each end support as follows:

1. Calculate required anchorage length
2. Determine support width

If anchorage length	>	(support width - cover):	bob bars

Else if support width	<	entered value for this setting:	extend and bob bars
Else:			Provide straight anchorage

Links

Select same bar size in support region and in the span

Select to use the same bar size in the support and span regions.

Select symmetrical link in support region

Select to use same link arrangement (bar size and spacing) in both supports.

Concrete - Beam - Top Longitudinal Bar Pattern

This page is used to configure the patterns used for the top bars.

Standard Pattern Setup

Longitudinal Bar Pattern

This drop list is used to select a pattern to be viewed/modified in the Design Options dialog.

Default Pattern

Longitudinal Default Pattern

This drop list is used to select the pattern that gets applied to new beams when they are first created.

The 'Default Pattern' cannot be used to change the pattern associated with existing beams. This can only be achieved by editing the beam properties.

Continuous Span + Cantilever (Backspan)

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to continuous spans and cantilever backspans. An interactive diagram is also displayed which updates as changes are made.

Bar check boxes

Selected bars that are also dimmed are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your preference.

Region Fields

The regions in which bars are applied are defined as a percentage of the span length.

Single Span

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to single spans. An interactive diagram is also displayed which updates as changes are made.

Cantilever

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to cantilevers. An interactive diagram is also displayed which updates as changes are made.

Concrete - Beam - Bottom Longitudinal Bar Pattern

This page is used to configure the patterns used for the bottom bars.

Standard Pattern Setup

Longitudinal Bar Pattern

This drop list is used to select a pattern to be viewed/modified in the Design Options dialog.

Default Pattern

Longitudinal Default Pattern

This drop list is used to select the pattern that gets applied to new beams when they are first created.

The 'Default Pattern' cannot be used to change the pattern associated with existing beams. This can only be achieved by editing the beam properties.

Continuous Span + Cantilever (Backspan)

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to continuous spans and cantilever backspans. An interactive diagram is also displayed which updates as changes are made.

Bar check boxes

Selected bars that are also dimmed are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your preference.

Region Fields

The regions in which bars are applied are defined as a percentage of the span length.

Bar Lapping

- Inside Support - select this option to lap the bottom bars in the middle of the support region.
- Outside Support - select this option to lap the bottom bars at the face of the support region.

Single Span

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to single spans. An interactive diagram is also displayed which updates as changes are made.

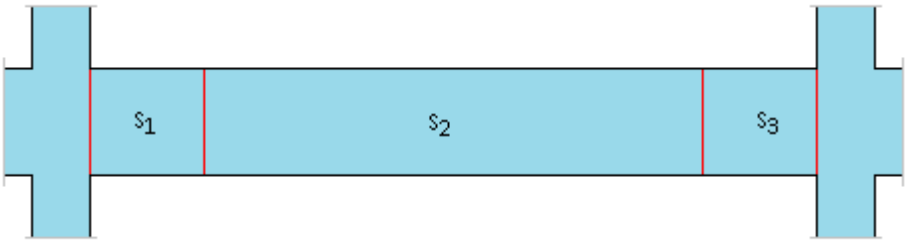
Cantilever

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to cantilevers. An interactive diagram is also displayed which updates as changes are made.

Concrete - Beam - Link Settings

Shear Design Regions - Normal

When considering shear, the design shear checks are performed in each of 3 regions S_1 , S_2 and S_3 as shown below. In each region, the maximum vertical shear from all load combinations is determined and this maximum value used to determine the shear reinforcement required in that region.



Region S_1 , S_2 , S_3

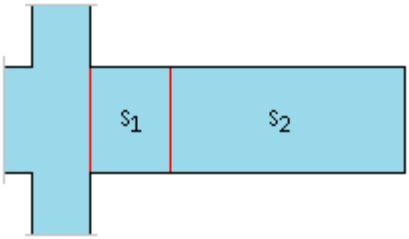
The regions are defined as fixed proportions of the clear span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically.

Link Type

Select either open, or closed links from the drop list.

Shear Design Regions - Cantilever

In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2 as shown below.



Link bars

This tab is used to define how the selected Longitudinal Bar Pattern gets applied to single spans. An interactive diagram is also displayed which updates as changes are made.

Minimum bar size

Sets the minimum allowable bar size that can be used in the design process.

Maximum bar size

Sets the maximum allowable bar size that can be used in the design process.

Minimum spacing

Sets the minimum allowable link spacing that can be used in the design process.

Maximum spacing

Sets the maximum allowable link spacing that can be used in the design process.

Spacing increment

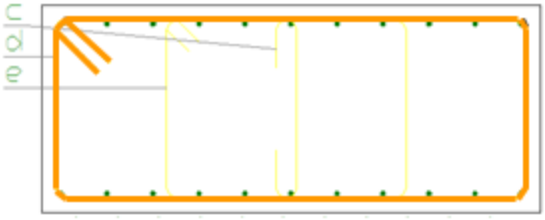
The designed link spacings are multiples of this value.

Maximum link leg spacing across the beam

This value is used to determine if single links, double links, or more are required, depending on the width of the beam.

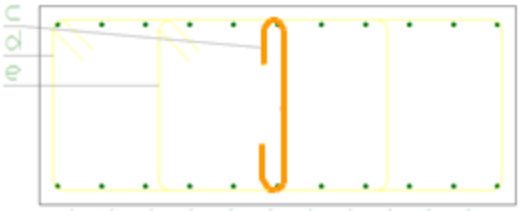
Use single outside link

This option uses a single outside link, with additional links added as required, (as per link 'd' shown dotted below).



Accept single leg internal link

This option allows the use of single leg internal links, (as per link 'c' shown dotted below).



Optimize link design regions where possible

In this case in the central region S_2 , shear reinforcement is provided to meet the minimum of the code requirement or user preference whilst in regions S_1 and S_3, designed shear reinforcement is required.

The position and length of region S_2 is determined from considerations of the shear resistance of the concrete cross-section and this then enables the lengths of regions S_1 and S_3 to be determined.

Concrete - Beam - General Parameters

Eurocode:

Reinforcement anchorage length parameters

These parameters are used in the calculation of the ultimate bond stress, from which the anchorage lengths are determined.

Maximum Bond Quality Coefficient

Allowable range 0.5 - 1.0;default 0.7.

Aggregate

Maximum nominal aggregate size

This value is used in the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Allowance for deviation

This value is used in the calculation to determine the limiting nominal concrete cover, $c_{nom, lim}$

- $c_{nom, lim} = c_{min} + \Delta c_{dev}$

ACI:

Fixity

Fixity Coefficient β_1

This coefficient is applied to the maximum positive moment in the beam span (excluding support positions) to set a user defined minimum design moment for beam support regions.

Allowable range 0.0 - 1.0; default 0.25

Deflection

Long term deflection period

Allowable range 3 - 60 months; default 60

Time at which brittle finishes are introduced

Allowable range 1- 6 months; default 1

All Codes:

FE Chasedown

Design Beams for FE Chasedown analysis results

Provided **FE Chasedown Analysis** has been performed (which requires the model to contain two way spanning slabs) this option can be used to specify that beams are designed for the forces obtained from this analysis in addition to the forces obtained from the **3D Analysis** and **Grillage Chasedown Analysis**.

Effective flanges calculation

Tolerance on rectilinearity

The calculation of the effective width is only carried out for concrete beams if they lie within the tolerance on rectilinearity set here. The default tolerance is 15 degrees; at greater angles you will be prompted to enter the effective width manually.

Design Options-Concrete-Column

This page has several sub-pages for setting up the options that are used in the column design.

Concrete - Column - Reinforcement Layout

This page is used to set limits on the ranges and spacings of bars used in the design.

General

Country

The available bar sizes and properties are taken from the Materials - Reinforcement database according to the country selected.

Vertical bars

Minimum bar size

Sets the minimum allowable bar size that can be used in the design process.

Maximum bar size

Sets the maximum allowable bar size that can be used in the design process.

Minimum bar spacing

Sets the minimum allowable bar spacing that can be used in the design process.

Maximum bar spacing

Sets the maximum allowable bar spacing that can be used in the design process.

Minimum reinforcement ratio

Sets the minimum allowable area of reinforcement as a ratio of the concrete section area.

Maximum reinforcement ratio

Sets the maximum allowable area of reinforcement as a ratio of the concrete section area.

Steel overstrength factor (ACI only)

This factor is used in seismic design to account for the reinforcement steel yielding overstrength.

Link bars

Minimum bar size

Sets the minimum allowable link size that can be used in the design process.

Maximum bar size

Sets the maximum allowable link size that can be used in the design process.

Minimum bar spacing

Sets the minimum allowable link spacing that can be used in the design process.

Maximum bar spacing

Sets the maximum allowable link spacing that can be used in the design process.

Spacing increment

The designed link spacings are multiples of this value.

Confinement reinforcement

These options are used to limit the range of confinement link options that are applied to rectangular sections, depending upon the aspect ratio of the section.

Aspect ratio change point

The value entered here determines which rectangular sections are considered to have a low aspect ratio and which are considered to have a high aspect ratio.

Use double links for low aspect ratio

Select to use double links where applicable in preference to single links for sections with a low aspect ratio.

Use double links for high aspect ratio

Select to use double links where applicable in preference to single links for sections with a high aspect ratio.

Use triple links for low aspect ratio

Select to use triple links where applicable in preference to double or single links for sections with a low aspect ratio.

Use cross links for low aspect ratio

Select to use cross links where applicable in preference to triple, double or single links for sections with a low aspect ratio.

Concrete - Column - Detailing Options

Settings on this page relate to the detail that is produced, but have no effect on the design.

Longitudinal Bars

Cut length rounding increment

Cut lengths are rounded to the value specified.

The overall steel bar cut length is stored in the Material data on the Home tab. For existing bars it can be verified by clicking **Materials > Reinforcement**. *Then click an available bar size and click the* **View...** *button.*

Kicker dimension

The height of column kicker cast above the slab level.

Join identical bars where possible

When this option is selected, bars are merged only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.

Links

Number of span region links required to use separate regions

The column design may have utilized span regions to economize on the link requirements, however, if these only exist over a short length you may prefer to standardize the link spacing on the detail. This setting allows you to specify the minimum number of span region links that are necessary for a span region to be indicated on the detail.

Provide links through full foundation depth

Select to draw links through the full depth of the foundation.

Foundation penetration depth of links

If the **Provide links through full foundation depth** option is cleared this field becomes accessible; allowing for the penetration depth of links into the foundation to be specified.

Provide links through floor depth for internal columns

- For columns restrained by flat slabs - links are always provided through the floor depth irrespective of this setting.
- For columns restrained by beam and slab - links are always provided through the beam depth for edge columns but are only provided for internal columns when the option is selected. When cleared, links are provided up to the soffit of the shallowest beam depth.

Concrete - Column - General Parameters

General

Bar sizes no smaller than stack above

Select to ensure that bar sizes do not reduce in lower stacks.

Match bar position to stack above

Select to have the starting arrangement for longitudinal bars match the arrangement of the bars in the stack above, if the section geometry matches.

Increase main bar size preferentially

Select to have the corner bars increased in preference to the intermediate bars. All bars start off at the same size (unless the initial bar size is driven by the current arrangement or the stack above), but when the check fails the corner bars will be increased in size if all bar sizes are the same, otherwise the intermediate bars will be increased in size. This means that when the final design is produced, either all bars will be the same size or the corner bars will be one size larger than the intermediate bars.

Alternatively, if you require all bar sizes to be the same, clearing this option results in all bar sizes being increased together.

Reinforcement anchorage length parameters (Eurocode only)

These parameters are used in the calculation of the ultimate bond stress, from which the anchorage lengths are determined.

Maximum Bond Quality Coefficient

Allowable range 0.5 - 1.0.

Aggregate

Maximum nominal aggregate size

This value is used in the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Allowance for deviation (Eurocode only)

This value is used in the calculation to determine the limiting nominal concrete cover, $c_{nom,\ lim}$

$$c_{nom,\ lim} = c_{min} + \Delta c_{dev}$$

FE Chasedown

Design Columns for FE Chasedown analysis results

Provided an FE Chasedown has been performed (which requires the model to contain two way spanning slabs) this option can be used to specify that columns are designed for the forces obtained from the FE Chasedown analysis in addition to the forces from the other analyses that have been performed.

Design Options-Concrete-Wall

This page has several sub-pages for setting up the options that are used in the wall design.

Concrete - Wall - Reinforcement Layout

This page is used to set limits on the ranges and spacings of bars used in the design.

General

Country

The available bar sizes and properties are taken from the Materials - Reinforcement database according to the country selected.

Vertical bars

Minimum bar size

Sets the minimum allowable bar size that can be used in the design process.

Maximum bar size

Sets the maximum allowable bar size that can be used in the design process.

Minimum bar spacing

Sets the minimum allowable bar spacing that can be used in the design process.

Maximum bar spacing

Sets the maximum allowable bar spacing that can be used in the design process.

Spacing increment

The designed bar spacings are multiples of this value.

Minimum reinforcement ratio

Sets the minimum allowable area of reinforcement as a ratio of the concrete section area.

Maximum reinforcement ratio

Sets the maximum allowable area of reinforcement as a ratio of the concrete section area.

Vertical bars in end zone

Minimum bar size

Sets the minimum allowable bar size that can be used in the design process.

Maximum bar size

Sets the maximum allowable bar size that can be used in the design process.

Minimum reinforcement ratio
Sets the minimum allowable area of reinforcement as a ratio of the end zone area.

Maximum reinforcement ratio
Sets the maximum allowable area of reinforcement as a ratio of the end zone area.

Horizontal bars

Minimum bar size
Sets the minimum allowable bar size that can be used in the design process.

Minimum reinforcement ratio
Sets the minimum allowable area of reinforcement as a ratio of the end zone area.

Link/horizontal bars

Minimum bar spacing
Sets the minimum allowable bar spacing that can be used in the design process.

Maximum bar spacing
Sets the maximum allowable bar spacing that can be used in the design process.

Spacing increment
The designed bar spacings are multiples of this value.

Link/confinement bars

Minimum bar size
Sets the minimum allowable bar size that can be used in the design process.
Maximum bar size
Sets the maximum allowable bar size that can be used in the design process.

Confinement bars in end zone

Minimum bar size
Sets the minimum allowable bar size that can be used in the design process.
Maximum bar size
Sets the maximum allowable bar size that can be used in the design process.

Substitute loose bars if mesh inadequate
Select to use additional loose bars in the end zones when the mesh is inadequate.

Concrete - Wall - Detailing Options

Settings on this page relate to the detail that is produced, but have no effect on the design.

Longitudinal Bars

Cut length rounding increment
Cut lengths are rounded to the value specified.

The overall steel bar cut length is stored in the Material data on the Home tab. For existing bars it can be verified by clicking **Materials > Reinforcement**. *Then click an available bar size and click the* **View...** *button.*

Kicker dimension

The height of column kicker cast above the slab level.

Join identical bars where possible

When this option is selected, bars are merged only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.

Links

Number of span region links required to use separate regions

The wall design may have utilized span regions to economize on the link requirements, however, if these only exist over a short length you may prefer to standardize the link spacing on the detail. This setting allows you to specify the minimum number of span region links that are necessary for a span region to be indicated on the detail.

Provide links through full foundation depth

Select to draw links through the full depth of the foundation.

Foundation penetration depth of links

If the **Provide links through full foundation depth** option is cleared this field becomes accessible; allowing for the penetration depth of links into the foundation to be specified.

Provide links through floor depth for internal walls

- For walls restrained by flat slabs - links are always provided through the floor depth irrespective of this setting.
- For walls restrained by beam and slab - links are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When cleared, links are provided up to the soffit of the shallowest beam depth.

Concrete - Wall - General Parameters

Reinforcement anchorage length parameters

These parameters are used in the calculation of the ultimate bond stress, from which the anchorage lengths are determined.

Maximum Bond Quality Coefficient

Allowable range 0.5 - 1.0.

Aggregate

Maximum nominal aggregate size

This value is used in the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Allowance for deviation

This value is used in the calculation to determine the limiting nominal concrete cover, $c_{nom, lim}$

$$c_{nom, lim} = c_{min} + \Delta c_{dev}$$

FE Chasedown

Design Columns for FE Chasedown analysis results

Provided an FE Chasedown has been performed (which requires the model to contain two way spanning slabs) this option can be used to specify that walls are designed for the forces obtained from the FE Chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Design Options-Concrete-Slab

This page has two sub-pages for setting up the options that are used in the slab and flat slab design.

Concrete - Slab - Reinforcement Layout

This page has two tabs for setting limits on the range and spacing of bars for slabs on beams and flat slabs.

General

Country

The available bar sizes and properties are taken from the Materials - Reinforcement database according to the country selected.

Principal/secondary bars

Minimum loose bar size

Sets the minimum allowable loose bar size that can be used in the design process.

Maximum loose bar size

Sets the maximum allowable loose bar size that can be used in the design process.

Minimum clear spacing

Sets the minimum allowable clear bar spacing that can be used in the design process.

Minimum spacing (slab auto-design)

Sets the minimum allowable clear bar spacing used in the panel auto-design process. As this only applies to panels, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more readily pass by reducing min spacing check failures in the patches.

Maximum principal bar spacing distance

Sets the maximum allowable bar spacing as a distance.

Maximum principal bar spacing x slab depth

In addition to the above maximum distance - this sets the maximum allowable bar spacing as a function of the slab depth.

Maximum secondary bar spacing distance

Sets the maximum allowable bar spacing that can be used in the design process. Maximum principal bar spacing x slab depth.

Maximum secondary bar spacing x slab depth

In addition to the above maximum distance - this sets the maximum allowable bar spacing as a function of the slab depth.

Bar spacing increment

The bar spacings are multiples of this value.

Maximum bar length

Sets the maximum length of bar.

Bar length increment

The bar lengths are multiples of this value.

Slab edge clearance

Sets the clearance from the reinforcement to the slab edge.

Reinforcement

Auto selection of outer bars

Select to have the bars in the outer layer selected automatically.

Make bob for top steel of cantilevers

Select to apply a bob to the end of the top steel in cantilevers.

Use mesh where possible

Select to use mesh where possible.

Concrete - Slab - Patches

This page has separate tabs for setting default properties for each path type.

Reinforcement

Use mesh where possible

Select to use mesh where possible.

Patch Size

Width, Lx, Ly

Sets the default patch size to be used.

Strip Proportions

Edge strip width as proportion of patch width

for column patches this ratio controls how the patch width is split between edge and center strips.

Center strip width

for beam patches this sets the width of the center strip.

Concrete - Slab - General Parameters

Reinforcement anchorage length parameters

These parameters are used in the calculation of the ultimate bond stress, from which the anchorage lengths are determined.

Maximum Bond Quality Coefficient

Allowable range 0.5 - 1.0.

Cover and Spacing check parameters

Maximum nominal aggregate size

This value is used in the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Allowance for deviation (Eurocode only)

This value is used in the calculation to determine the limiting nominal concrete cover, $c_{nom,\ lim}$

$$c_{nom,\ lim} = c_{min} + \Delta c_{dev}$$

Detailing

Allowance for additional detailing bars

The total mass of bars for slabs/mats shown in material listing reports and on slab/mat detailing drawings will be increased by the percentage specified here.

Design Options-Concrete Punching Shear

This page sets up the parameters for punching shear stud rail design.

General

Country

The available bar sizes and properties are taken from the Materials - Reinforcement database according to the country selected.

Reinforcement bars

Minimum bar size

Sets the minimum allowable bar size that can be welded to the rail in the design process.

Maximum bar size

Sets the maximum allowable bar size that can be welded to the rail in the design process.

Diagonal spacing between rails

Sets the maximum allowable diagonal spacing for radial layouts.

Spacing along rail

Minimum spacing

Sets the minimum allowable bar spacing along each rail.

Maximum spacing

Sets the maximum allowable bar spacing along each rail.

Spacing increment

The bar spacing adopted will be a multiple of this value.

Spacing to the first reinforcement line

Minimum spacing

Sets the minimum spacing from the column face to the first reinforcement line.

Maximum spacing

Sets the maximum spacing from the column face to the first reinforcement line.

Spacing increment

The spacing from the column face will be a multiple of this value.

Auto-design

Method

Auto-design can either be set to minimise the bar size, or minimise the number of rails.

Starting number of studs per face

this is the minimum number of studs per face that will be used when auto-designing from minima.

Diagonal spacing between rails (for circular arrangements)

Maximum spacing

Sets the maximum diagonal spacing between the outermost studs in the last critical perimeter.

Spacing between rails in the Y direction

Minimum spacing

Sets the minimum allowable rail spacing in Y.

Maximum spacing

Sets the maximum allowable rail spacing in Y.

Spacing increment

The rail spacing in Y adopted will be a multiple of this value.

Spacing between rails in the Z direction

Minimum spacing

Sets the minimum allowable rail spacing in Z.

Maximum spacing

Sets the maximum allowable rail spacing in Z.

Spacing increment

The rail spacing in Z adopted will be a multiple of this value.

Design Options-Concrete Isolated Foundations

This page has several sub-pages for setting up the options that are used in pad base and pile cap design.

Concrete - Isolated Foundations - Reinforcement Layout

This page has two tabs for setting limits on the range and spacing of bars for pad bases and pile caps.

General

Country

The available bar sizes and properties are taken from the Materials - Reinforcement database according to the country selected.

Principal/secondary bars

Minimum loose bar size

Sets the minimum allowable loose bar size that can be used in the design process.

Maximum loose bar size

Sets the maximum allowable loose bar size that can be used in the design process.

Minimum clear spacing

Sets the minimum allowable clear bar spacing that can be used in the design process.

Maximum principal bar spacing

Sets the maximum allowable principal bar spacing as a distance.

Maximum secondary bar spacing

Sets the maximum allowable secondary bar spacing as a distance.

Bar spacing increment

The bar spacings are multiples of this value.

Reinforcement

Use mesh where possible

Select to use mesh where possible.

Concrete - Isolated Foundations - Foundation Size

This page has two tabs for setting the size parameters used in the design process for pad bases and pile caps.

Minimum depth

Sets the minimum allowable depth.

Maximum depth

Sets the maximum allowable depth.

Auto-design depth increment

Sets the depth increment.

Minimum side length

Sets the minimum allowable side length.

Maximum side length under columns

Sets the maximum side length under columns as a distance.

Maximum side length ratio under columns

Sets the maximum side length ratio under columns.

Rounding increment for footing dimensions and pile spacing

The overall footing size is rounded to this value.

Default 3-pile cap shape

the default shape when 3 piles are used can be set to either triangular or rectangular.

Concrete - Isolated Foundations - General Parameters

Reinforcement anchorage length parameters (Eurocode only)

These parameters are used in the calculation of the ultimate bond stress, from which the anchorage lengths are determined.

Maximum Bond Quality Coefficient

Allowable range 0.5 - 1.0.

Cover and Spacing check parameters

Maximum nominal aggregate size

This value is used in the calculation to determine the minimum clear horizontal distance between individual parallel bars.

Allowance for deviation (Eurocode only)

This value is used in the calculation to determine the limiting nominal concrete cover, $c_{nom,\ lim}$

$$c_{nom,\ lim} = c_{min} + \Delta c_{dev}$$

Pad Base Shear Reactions at Pinned Bases (ACI/AISC and BS head codes only)

Ignorable percentage of passive pressure

Default of 1% is typically a negligible load level. (if you don't want to ignore small shear forces at pinned bases set this value to zero).

Pad Base Foundation bearing capacity (Eurocode only)

Use presumed bearing capacity method (EN 1997-1 cl 6.5.2.4)

If this option is selected the bearing pressure is checked for serviceability combinations using the presumed bearing resistance specified in the pad base properties (no 'STR' and 'GEO' capacities).

Pile Cap Design parameters

Tolerance for pile position

The allowable deviation of the pile position from its original plan position which is used in the design moment calculations.

Concrete - Isolated Foundations - Piles

Minimum and maximum number of piles for pile caps under columns

These parameters apply when a pile cap under a column has been set to auto design the piles.

Minimum spacing option

Sets whether the minimum spacing check uses a set value, or a multiple of the pile width/circumference.

Minimum spacing of piles

Sets the minimum spacing (either as a value, or as multiple of the pile width/circumference).

Minimum pile edge distance

Minimum distance from the pile face to edge of foundation.

Pile auto-design method

When a pile cap has been set to auto design, you can choose to use the smallest number of high capacity piles or more piles of lowest capacity.

Design Options-Concrete Mat Foundations

The options displayed here are identical to those applied to slabs.

Design Options-Composite Beams

Effective flange width calculation

Tolerance on rectilinearity

The calculation of the effective width is only carried out for composite beams if they lie within the tolerance on rectilinearity set here. The default tolerance is 15 degrees; at greater angles you will be prompted to enter the effective width manually.

Design Options-Design Forces

Ignore Forces Below

A full 3D analysis may expose small forces that are normally ignored in the design of members. The options for ignore forces below on the Design Forces page of the Design Options dialog simply provide you with a way of setting negligible/nominal force levels with which you are comfortable. When the small forces from the 3D analysis are below the specified threshold levels they are ignored so that design can proceed automatically. If the

forces are above these limits, then you will be warned during the design process but the forces will still be ignored.

Concrete Beams

Torsion force % of concrete resistance

For Eurocode design only, this defines a user specified ignorable torque limit. If the maximum torsional moment acting on the beam is less than this limit then design for torsion is not performed.

This setting is only applicable for design to Eurocodes. For all other Head Codes it is not used and can be ignored.

Design Options-Design Groups

Check those member types for which grouped design is to be applied.

When grouped, only one member in the group is designed. This design is then copied to the remaining members in the group so that they can be checked. Any failing member in the group is redesigned, and the process repeated until all members pass, or a design is not possible.

Design Options-Autodesign

This option controls what happens to individual steel and concrete member autodesign settings, and isolated foundation autodesign settings at the end of the design process.

The first two options need little explanation:

- **Always** - The autodesign setting is automatically cleared at the end of the design process - putting every member into check mode.
- **Never** - The autodesign setting (either selected, or cleared) is always retained at the end of the design process.

The third option **When check status is at worst** makes the change from autodesign conditional upon the design status as follows:

- **Pass** - The autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass
- **Warning** - The autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, or Warning
- **Fail** - The autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, Warning, or Fail
- **Invalid** - The autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, Warning, Fail or Invalid
- **Beyond Scope** - The autodesign setting is only automatically cleared at the end of the design process for members with design status: Pass, Warning, Fail, Invalid or Beyond Scope.

The most practical use of the **When check status is at worst** option would be to set it to **Pass** and start with all members in autodesign mode. At the end of the first design run

passing members would be set to check mode, allowing you to focus on the remaining members still in autodesign mode.

Design Options-Design Warnings

Concrete Seismic Warnings

Warnings used as guidance during the design of a structure to US seismic provisions are in effect by default. These can be de-activated as found useful.

The following selection is available:

Specified concrete grade is too low

- Compliance with ACI 318-08 and ACI 318-11 Section 21.1.4.2, "(..) f'c, shall not be less than 3000 psi."
- Applies to normal weight reinforced concrete columns and beams part of Special Moment Frames
- Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls
- Prompts the user to increase the concrete grade of the member

Specified longitudinal reinforcement grade too high

- Compliance with ACI 318-11 Section 21.1.5.2, "(..) reinforcement resisting earth-quake induced flexure, axial force, or both, shall comply with ASTM A706, Grade 60.";
- Applies to normal weight reinforced columns and beams part of Special Moment Frames
- Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls
- Prompts the user to use lower longitudinal reinforcement grades

Column at the end of the beam is not in the same seismic frame

- Used to validate if the frame of which the beam is part off is correctly set-up as a seismic frame
- Applies to normal weight reinforced concrete beams part of a seismic force resisting system
- Applies to both SRFS type and direction
- Checks both ends of each beam span
- Prompts the user to set up a consistent SFRS type and direction between all members of the frame

Beam at the end of the column is not in the same seismic frame

- Used to validate if the frame of which the column is part off is correctly set-up as a seismic frame
- Applies to normal weight reinforced concrete columns part of a seismic force resisting system
- Checks the top of each column stack
- Prompts the user to set up a consistent SFRS type and direction between all members of the frame

Beam support region size is too small

- Compliance with ACI 318-08 and ACI 318-11 Section 21.3.4.2, "(..) hoops shall be provided over lengths not less than 2h measured from the face of the supported member toward the midspan"
- Applies to normal weight reinforced concrete beams part of Intermediate and Special Moment Frames
- Comes into effect if the length of the end shear design regions is smaller than the requirement
- Prompts the user to review the shear design regions - Design options

High seismic to conventional demand on non-SFRS beams / High seismic to conventional demand ratio of non-SFRS columns / High seismic to conventional demand of non-SFRS walls

- Used to recognize when a member has a significant contribution to the building's lateral force resistance but has not been assigned to the SFRS with the required ductility properties
- Specifically apply to each member type
- The highest non-seismic design ratio from bending and shear design is obtained
- The highest seismic design ratio from bending and shear design is obtained
- Comes into effect when the ratio between seismic and non-seismic values is higher than a given threshold (default = 1.0)
- The maximum seismic/non-seismic ratio threshold that triggers the warning is set through the "Seismic to conventional minimum demand ratio" field on the same page
- Prompts the user to either include the member in the SFRS or review the structure layout to reduce the contribution of the member to the lateral force resistance

Non-SFRS design requirements applicable in SDC D-F

- Compliance with ACI 318-08 and ACI 318-11 Sections 21.13.1, "Requirements of section 21.13 apply to members not designated as part of the seismic-force-resisting system in structures assigned to SDC D, E and F"
- Applies to normal weight reinforced concrete beams, column and walls not included in the SFRS
- Comes into effect in Seismic Design Category, SDC D or above
- Prompts the user to apply code prescribed requirements on top of current design

Steel Static Warnings

Additional notional loads required when alpha Pr/Py > 0.5, C2.3

- Compliance with 7.3 (3) of AISC 360-05 and C2.3 of AISC 360-10 requires the stiffness of certain members to be reduced. When the ratio of the required axial compressive force (Pr) and the axial yield strength (Py = Fy Ag) is > 0.5 then this stiffness must be reduced further
- As an alternative which is the approach adopted in Tekla Structural Designer, an additional notional load of 0.001 Yi can be applied instead
- When the program identifies that the ratio of Pr/Py exceeds 0.5 a warning is given to alert the designer
- If this additional notional load has been dealt with by the designer then this warning is no longer relevant and this option allows you to switch off the warning

Concrete Seismic

Seismic to conventional minimum demand ratio

This ratio is used to set the threshold above which the 'High seismic to conventional demand' warnings come into effect.

Design Options-Steel Joists

The loading pattern on each joist is assessed and classified as Uniform (or near uniform), Equivalent Uniform or Non-Uniform taking into account user defined tolerances.

Uniform load tolerance

if the calculated percentage load tolerance is less than this value, the load is classified as 'Uniform'.

Equivalent load tolerance

if the calculated percentage load tolerance is greater than this value, the load is classified as 'Non-uniform'.

If 0% equivalent load tolerance is specified then only pure uniform loading is accepted.

Maximum eccentricity of zero shear

In the equivalent uniform load method the position of the point of zero shear relative to the centre span point of the joist is determined. If this is located outside this maximum eccentricity of zero shear value then the load is classified as 'Non-uniform'.

Deflection increase to allow for shear

Deflections due to live load are calculated using the following effective inertia value:

$I_{effective} = I_{gross} / (1 + R)$

where

R = the deflection increase to allow for shear effects (defaulted to 15%) and applied to KCS only.

Design Options-Wind Drift

A Wind Drift check is performed on all columns (apart from those with 'Wind drift check' deselected in their properties), for all combinations containing wind load case(s).

Check wind cases only

- With this setting on (default): the Wind Drift check only considers the effects of the wind load case(s) in each wind combination.
- With this setting off: the check considers the effects of all load cases in wind combinations (which would include drift induced by gravity loads).

Static Analysis & Design

Member design modes

Before you start the analysis and design process, every design member in your model will be set into one of two possible modes:

- **Check design mode** - you assign your desired section size (steel members) or section size and reinforcement (concrete members) and the program then determines if the section/reinforcement is sufficient.
- **Autodesign mode** - you select the desired section type and the program then automatically determines a suitable size of the chosen section type (steel members); or you assign your desired section size and the program then automatically determines a suitable reinforcement configuration (concrete members).

You can quickly review (and update) the mode applied to every member in the model using the 'Auto|Check design' toggle on the Review toolbar.

Once the autodesign\check design modes have been set appropriately you can then begin the member design process.

Your choice of which analysis-design process to run will be affected by the materials used (steel, concrete, or both) and also the combination types that you want to consider (gravity, or all) - steel structures in particular may require an initial 'gravity design'.

Do I run Design Steel, Design Concrete or Design All?

Your choice will depend on the materials used in the model.

In simple terms:

- if your model consists of steel members only, you can run the design from the **Analysis & Design Steel** group on the Design toolbar. (i.e. Design Steel (Gravity) and Design Steel (Static)
- if your model consists of concrete members only, you can run the design from the **Analysis & Design Concrete** group on the Design toolbar. (i.e. Design Concrete (Gravity) and Design Concrete (Static)
- if your model consists of a mix of both concrete and steel members, you can run the design from the **Analysis & Design** group on the Design toolbar. (i.e. Design All (Gravity) and Design All (Static)

For structures that are mostly steel but have a few concrete members: instead of running Design All, you could run Design Steel (in order to focus on the steel design) before switching to Design Concrete for the remaining members. In this way during the steel design phase you are not running grillage and FE chasedown analyses when they are not required .

The processes that get initiated reflect your choice as follows:

Design Steel

- performs a **3D Analysis**,
- does not perform **Grillage Chasedown** or **FE Chasedown** analysis,

- designs/checks all steel elements and shear walls,
- does not design/check concrete beams or columns.

Design Concrete

- performs a **3D Analysis** and a **Grillage Chasedown** analysis,
- may also be required to perform an **FE Chasedown** analysis,
- designs/checks all concrete beams, columns and shear walls,
- does not design/check steel elements.

Design All

- performs a **3D Analysis** and a **Grillage Chasedown** analysis,
- may also be required to perform an **FE Chasedown** analysis,
- designs/checks all concrete beams, columns and shear walls,
- design/checks all steel elements.

Do I run Design...(Gravity), or Design...(Static)?

Design (Gravity)

Although your final design should consider all combinations, designing for gravity combinations can be a useful way to rapidly pre-size those members in the structure that are not subjected to lateral loads.

Designing members for gravity combinations is a technique more likely to be employed for steel rather than concrete structures.

This technique involves the use of first-order analysis on a limited set of design combinations.

- Design Steel (Gravity) **-** rapid gravity sizing of the majority of steel members
- Design Concrete (Gravity) **-** design of concrete members for gravity combinations only
- Design All (Gravity) **-** design of all steel and concrete members for gravity combinations only

Because lateral systems may not have been established at this stage, those column nodes that are not in a rigid floor diaphragm are by default fixed horizontally. (This default can be changed if required in **Home > Settings > Design Options > General**)

Design (Static)

Once initial member sizes have been adequately sized, you should make a Sway sensitivity assessment (ACI/AISC), as this can affect the Choice of analysis type (ACI/AISC) used in the final static design.

This final static design is invoked by selecting the **Design (Static)** process applicable to the type of material involved:

- Design Steel (Static) **-** full design of all steel members
- Design Concrete (Static) **-** full design of all concrete members

- Design All (Static) **-** full design of all steel and concrete members

For each of the above, **3D Analysis** is performed (first, or second order as specified in **Design Options > Analysis**) for all active combinations to establish a set of design forces.

If the model contains concrete members, **Grillage chasedown** and potentially **FE chasedown** are performed to establish additional sets of design forces.

All members are checked or designed for the appropriate design requirements. Gravity members are checked for gravity combinations, lateral members are checked for all combinations. Only active combinations are checked.

Design Steel (Gravity)

Performs analysis of all active gravity load combinations and then designs or checks steel members.

Overview:

- A **3D Analysis** is run (excluding pattern loading) to establish a set of design forces for the steel members.
- For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/ or produce unrealistic forces (controlled via Settings> Design Options> General on the Home ribbon).
- Each 'gravity only' steel member is then either checked, or designed - according to its autodesign setting - for gravity combinations only.
- Remaining steel members set to auto-design are designed for gravity combinations, to give them a reasonable start size prior to consideration of the lateral load combinations. (Remaining steel members set to check-design are not checked.)
- On completion of the gravity sizing process all steel members are set to check-design mode. At this point it is possible that the lateral members are under-sized (having been designed for the gravity combinations only) so it is recommended that you reset them to auto-design mode before continuing the design process.
- No design forces are established for concrete elements in a multi-material building.

How do I run Design Steel (Gravity)?

1. Prior to running the design you should have assigned appropriate **Autodesign** settings to the steel members:

   - Autodesign 'on' - new section sizes will be designed
   - Autodesign 'off' - existing section sizes will be checked

2. You should also be aware of the **Design Options** ( ) that are in place, in particular:

   - Design Options-Analysis - which sets the analysis type to be run,
   - Design Options-Design Forces - which sets the levels at which forces can be ignored,
   - Design Options-Autodesign - which controls how Autodesign settings will be reset after the design

3. To run the design:

Click **Design Steel (Gravity)**

4. At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design Steel (Static)

Performs analysis of all active static load combinations (gravity, lateral, and ELF seismic) then designs or checks every steel member.

Overview:

- A first-order analysis of all unfactored loadcases is carried to establish Serviceability Limit State requirements such as deflections.
- EHF's are determined for every active combination in which they have been included.
- Having established the EHF's, their contributions to frame deflections are determined using a first-order analysis. α_{cr} values are also established.
- **3D Analysis** is performed (first, or second order as specified in Design Options > Analysis) for all active combinations to establish a set of design forces.
- All steel members set to autodesign mode are designed for the appropriate design requirements. Gravity members are designed for gravity combinations, lateral members are designed for all combinations. Only active combinations are checked. If section sizes are changed the analysis-design cycle is repeated.
- All steel members set to check mode are checked for the appropriate design requirements. Gravity members are checked for gravity combinations, lateral members are checked for all combinations. Only active combinations are checked.
- At the end of the process, all autodesign settings are typically cleared - putting every member into check mode. You can change this if required via Design Options-Autodesign.

How do I run Design Steel (Static)?

1. Prior to running the design you should have assigned appropriate **Autodesign** settings to the steel members:

   - Autodesign 'on' - new section sizes will be designed
   - Autodesign 'off' - existing section sizes will be checked

2. You should also be aware of the **Design Options** ( ) that are in place, in particular:

   - Design Options-Analysis - which sets the analysis type to be run,
   - Design Options-Design Forces - which sets the levels at which forces can be ignored,
   - Design Options-Autodesign - which controls how Autodesign settings will be reset after the design

3. To run the design:

   Click **Design Steel (Static)**

4. At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design Concrete (Gravity)

Performs analysis of all active gravity load combinations and then designs or checks every concrete member (but not concrete slabs).

Overview:

- **3D Analysis** is run to establish a set of design forces for the concrete members.
- For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the analysis from completing and/ or produce unrealistic forces (controlled via Settings> Design Options> General on the Home ribbon).
- **Grillage chasedown** analysis is performed to establish a second set of design forces.
- If the model contains flat slabs, or slabs on beams, **FE chasedown** analysis is also performed. The results being (optionally) used to establish a third set of design forces for concrete beams, columns and walls.
- Every concrete member is then either checked, or designed - according to its autodesign setting - for gravity combinations only.
- No design forces are established for steel elements in a multi-material building.

How do I run Design Concrete (Gravity)?

1. Prior to running the design you should have assigned appropriate **Autodesign** settings to the concrete members:

   - Autodesign 'on' - new reinforcement will be designed
   - Autodesign 'off' - existing reinforcement will be checked

2. You should also review the Design Options that are in place - to do this:

   Click **Design** > **Options** ( )

3. To run the design:

   Click **Design Concrete (Gravity)**

4. At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design Concrete (Static)

Performs analysis of all active static load combinations (gravity, lateral, and ELF seismic) then designs or checks every concrete member.

Overview:

- A first-order analysis of all unfactored loadcases is carried to establish Serviceability Limit State requirements.
- EHF's are determined for every active combination in which they have been included.
- Having established the EHF's, their contributions to frame deflections are determined using a first-order analysis. α_{cr} values are also established.
- **3D Analysis** is performed (first, or second order as specified in Design Options > Analysis) for all active combinations to establish a set of design forces.
- **Grillage chasedown** analysis is performed to establish a second set of design forces.
- If the model contains flat slabs, or slabs on beams, **FE chasedown** analysis is also performed. The results being (optionally) used to establish sets of design forces for concrete beams, columns and walls.
- All concrete members set to auto-design mode are designed for the appropriate design requirements. Only active combinations are checked.
- All concrete members set to check mode are checked for the appropriate design requirements. Only active combinations are checked.
- At the end of the process, all autodesign settings are typically retained - however you can change this if required via Design Options-Autodesign.

How do I run Design Concrete (Static)?

1. Prior to running the design you should have assigned appropriate **Autodesign** settings to the concrete members:

   - Autodesign 'on' - new reinforcement will be designed
   - Autodesign 'off' - existing reinforcement will be checked

2. You should also review the Design Options that are in place - to do this:

   Click **Design** > **Options** ( )

3. To run the design:

   Click **Design Concrete (Static)**

4. At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design All (Gravity)

Performs analysis of all active gravity load combinations and then designs or checks every member (but not concrete slabs).

Overview:

- **3D Analysis** is run - excluding pattern loading - to establish a set of design forces for the steel and concrete members.
- For the above analysis, lateral translational fixed supports can optionally be applied to column nodes in order to remove unwanted lateral displacements that may prevent the

analysis from completing and/ or produce unrealistic forces (controlled via Settings> Design Options> General on the Home ribbon).

- **Grillage chasedown** analysis is performed to establish a second set of design forces (for concrete members only).
- If the model contains flat slabs, or slabs on beams, **FE chasedown** analysis is also performed. The results being (optionally) used to establish a third set of design forces for concrete beams, columns and walls.
- Each member is then either checked, or designed - according to its autodesign setting - for gravity combinations only.
- On completion of the gravity sizing process all steel members are set to check-design mode. At this point it is possible that the lateral members are under-sized (having been designed for the gravity combinations only) so it is recommended that you reset them to auto-design mode before continuing the design process.

How do I run Design All (Gravity)?

1. Prior to running the design you should have assigned appropriate **Autodesign** settings to all members:

   - Autodesign 'on' - new section sizes will be designed for steel members and new reinforcement will be designed for concrete members
   - Autodesign 'off' - existing section sizes will be checked for steel members and existing reinforcement will be checked for concrete members

2. You should also review the Design Options that are in place - to do this:

   Click **Design** > **Options** ( )

3. To run the design:

   Click **Design All (Gravity)**

4. At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design All (Static)

Performs analysis of all active static load combinations (gravity, lateral, and ELF seismic) then designs or checks every member (but not concrete slabs).

Overview:

- A first-order analysis of all unfactored loadcases is performed to establish Serviceability Limit State requirements such as deflections.
- EHF's are determined for every active combination in which they have been included.
- Having established the EHF's, their contributions to frame deflections are determined using a first-order analysis. α_{cr} values are also established.
- **3D Analysis** is performed (first, or second order as specified in Design Options > Analysis) for all active combinations to establish a set of design forces.

- **Grillage chasedown** analysis is performed to establish a second set of design forces (for the design of concrete members only).
- If the model contains flat slabs, or slabs on beams, **FE chasedown** analysis is also performed. The results being (optionally) used to establish sets of design forces for concrete beams, columns and walls.
- All steel members set to auto-design mode are designed for the appropriate design requirements. Gravity members are designed for gravity combinations, lateral members are designed for all combinations. Only active combinations are checked. If section sizes are changed the analysis-design cycle is repeated.
- All steel members set to check mode are checked for the appropriate design requirements. Gravity members are checked for gravity combinations, lateral members are checked for all combinations. Only active combinations are checked.
- All concrete members set to auto-design mode are designed for the appropriate design requirements. Only active combinations are checked.
- All concrete members set to check mode are checked for the appropriate design requirements. Only active combinations are checked.
- At the end of the process, all autodesign settings are typically cleared for steel members but retained for concrete members - you can change this if required via Design Options-Autodesign.

How do I run Design All (Static)?

1. Prior to running the design you should have assigned appropriate **Autodesign** settings to all members:

   - Autodesign 'on' - new section sizes will be designed for steel members and new reinforcement will be designed for concrete members
   - Autodesign 'off' - existing section sizes will be checked for steel members and existing reinforcement will be checked for concrete members

2. You should also review the Design Options that are in place - to do this:

   Click **Design** > **Options** ( )

3. To run the design:

   Click **Design All (Static)**

4. At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

How can I check if Autodesign is on or off?

1. Select the members for which you want to review or change the **Autodesign** property.

2. Check the current **Autodesign** setting shown in the **Properties Window** is correct - adjust if necessary.

   - Autodesign 'on' - new reinforcement will be designed
   - Autodesign 'off' - existing reinforcement will be checked

Alternatively, edit the member to do the same thing:

1. Highlight the member and right-click it to display the context menu.
2. Choose **Edit** from the context menu.
3. Review the **Automatic design** setting in the dialog - adjust if necessary.

RSA Analysis & Design

RSA load combinations should be set up in advance of RSA Design by running the Seismic Wizard (choosing the option to use Modal Response Spectrum Analysis). The Seismic Wizard is not currently available for the AS:1170.4 code variant.

Design Steel (RSA)

Performs analysis of all active RSA load combinations and then designs steel members.

Overview:

- Validation - standard model and analysis validation plus additional seismic specific checks are performed
- 1st Order Vibration Analysis - performed for the Effective Seismic Weight Combination only, which determines the fundamental periods for directions 1 & 2
- Pre-Analysis for Seismic - (seismic weight and seismic torque calculations)
- Static Analysis - 1st Order Linear or 2nd Order Linear for all RSA Seismic Combinations and all their relevant loadcases, i.e. included Static Loadcases, but not RSA Seismic or and RSA Torsion Loadcases.
- RSA Analysis - a set of results are generated for a sub-set of vibration modes for each RSA Seismic Loadcase. The results for each mode are then combined into a single set
- Accidental Torsion Analysis - nodal loads are generated for each RSA Torsion Loadcase, these are then analysed using 1st Order Linear analysis.
- All steel members set to autodesign mode are designed taking the current size that has been determined from static design as a starting point. If section sizes are changed the analysis-design cycle is repeated.
- All steel members set to check mode are checked for the appropriate design requirements.

How do I run Design Steel (RSA)?

1. First, check the **Autodesign** property is correctly set for all members:
    - Autodesign 'on' - new section sizes will be designed
    - Autodesign 'off' - existing section sizes will be checked

(How can I check if Autodesign is on or off?)

2. Next, review the Design Options and adjust if required:

   - Click **Design** > **Options** ( )

3. Then:

   - Click **Design Steel (RSA)**

   At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design Concrete (RSA)

Performs analysis of all active RSA load combinations and then designs concrete members.

Overview:

- Validation - standard model and analysis validation plus additional seismic specific checks are performed
- 1st Order Vibration Analysis - performed for the Effective Seismic Weight Combination only, which determines the fundamental periods for directions 1 & 2
- Pre-Analysis for Seismic - (seismic weight and seismic torque calculations)
- Static Analysis - 1st Order Linear or 2nd Order Linear for all RSA Seismic Combinations and all their relevant loadcases, i.e. included Static Loadcases, but not RSA Seismic or and RSA Torsion Loadcases.
- RSA Analysis - a set of results are generated for a sub-set of vibration modes for each RSA Seismic Loadcase. The results for each mode are then combined into a single set
- Accidental Torsion Analysis - nodal loads are generated for each RSA Torsion Loadcase, these are then analysed using 1st Order Linear analysis.
- All concrete members set to autodesign mode are designed taking the current reinforcement that has been determined from static design as a starting point.
- All concrete members set to check mode are checked for the appropriate design requirements.

How do I run Design Concrete (RSA)?

1. First, check the **Autodesign** property is correctly set for all members:

   - Autodesign 'on' - new reinforcement will be designed for concrete members
   - Autodesign 'off' - existing reinforcement will be checked for concrete members

   (How can I check if Autodesign is on or off?)

2. Next, review the Design Options and adjust if required:

   - Click **Design** > **Options** ( )

3. Then:

- Click **Design Concrete (RSA)**

At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Design All (RSA)

Performs analysis of all active RSA load combinations and then designs all members.

Overview:

- Validation - standard model and analysis validation plus additional seismic specific checks are performed
- 1st Order Vibration Analysis - performed for the Effective Seismic Weight Combination only, which determines the fundamental periods for directions 1 & 2
- Pre-Analysis for Seismic - (seismic weight and seismic torque calculations)
- Static Analysis - 1st Order Linear or 2nd Order Linear for all RSA Seismic Combinations and all their relevant loadcases, i.e. included Static Loadcases, but not RSA Seismic or and RSA Torsion Loadcases.
- RSA Analysis - a set of results are generated for a sub-set of vibration modes for each RSA Seismic Loadcase. The results for each mode are then combined into a single set
- Accidental Torsion Analysis - nodal loads are generated for each RSA Torsion Loadcase, these are then analysed using 1st Order Linear analysis.
- All steel members set to autodesign mode are designed taking the current size that has been determined from static design as a starting point. If section sizes are changed the analysis-design cycle is repeated.
- All concrete members set to autodesign mode are designed taking the current reinforcement that has been determined from static design as a starting point.
- All steel and concrete members set to check mode are checked for the appropriate design requirements.

How do I run Design All (RSA)?

1. First, check the **Autodesign** property is correctly set for all members:

   - Autodesign 'on' - new section sizes will be designed for steel members and new reinforcement will be designed for concrete members
   - Autodesign 'off' - existing section sizes will be checked for steel members and existing reinforcement will be checked for concrete members

   (How can I check if Autodesign is on or off?)

2. Next, review the Design Options and adjust if required:

   - Click **Design** > **Options** ( )

3. Then:

   - Click **Design All (RSA)**

At the end of the analysis-design process the active view switches to a **Review View** and the tab switches from **Design** to **Review** - ready for reviewing the design graphically.

Slab design, patches and punching shear

The overall slab design process comprises the following steps:

- Create Patches - slab patches are placed over columns, beams, walls or panels as required,
- Perform Slab Design - slab panel areas that lie outside the patch areas are designed,
- Perform Patch Design - patches are designed,
- Add Punching Shear Checks - punching shear checks are placed as required (Not available for Australia Head Code),
- Perform Punching Shear Checks - punching shear checks are performed for the whole model or individually (Not available for Australia Head Code).

Patch Creation

What is a patch?

Patches are used during the design of concrete slabs as a way of managing the physical and design data. They each define a rectangular area of slab within which FE Analysis results are collected enabling a design to be performed. Design moments are calculated along **Result Strips** embedded within each patch. Depending on the patch type it can contain up to 6 result strips, catering for up to 3 strips of reinforcement in each of two perpendicular directions.

The following patch types are available:

- column patch - at column stack heads
- beam patch - along beams
- wall patch - along walls
- panel patch - at a specified position within the panel boundary
  - not restricted to a 'centralised' position and also not restricted to existing purely within one panel
  - might also be positioned under loads

These patches are either in the top or the bottom of the slab and may or may not have reinforcement defined in them. If no reinforcement is defined then the background reinforcement is used. If reinforcement is defined then for the top/bottom, x/y direction, you have the option to use the sum of the background + patch reinforcement - if reasonably aligned.

Note that patches may overlap on the plan view, 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.

How do I create a column patch?

1. Click **Design**

   Then from the **Patch drop list** ( ) select **Patch Column**

2. The create column patch properties set is displayed.

   - Use the **Lx** and **Ly** properties to specify the patch size.
   - Use the **Surface** property to specify the layer.
   - Select the **Autodesign** property in order to have the reinforcement automatically designed.
   - Only select the **Consider patch surface moments only** property if you don't want to consider the opposite surface.

Situations when you need to select this option should be very rare. (Typically when the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face - when the other patch is designed additional reinforcement would be provided at that surface, so the surface check is not required for the original patch.)

To place the patch over a specific column:

1. Click a column node within the slab. *Tekla Structural Designer* will add a patch to the selected column.

To create multiple patches by windowing:

1. Move the cursor to one corner of an imaginary box which will encompass the columns for which you want to define patches.

2. Click and hold the left mouse button.

3. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating patches).

4. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create patches to all columns which are totally within the rubber rectangle.

How do I create a beam patch?

1. Click **Design**

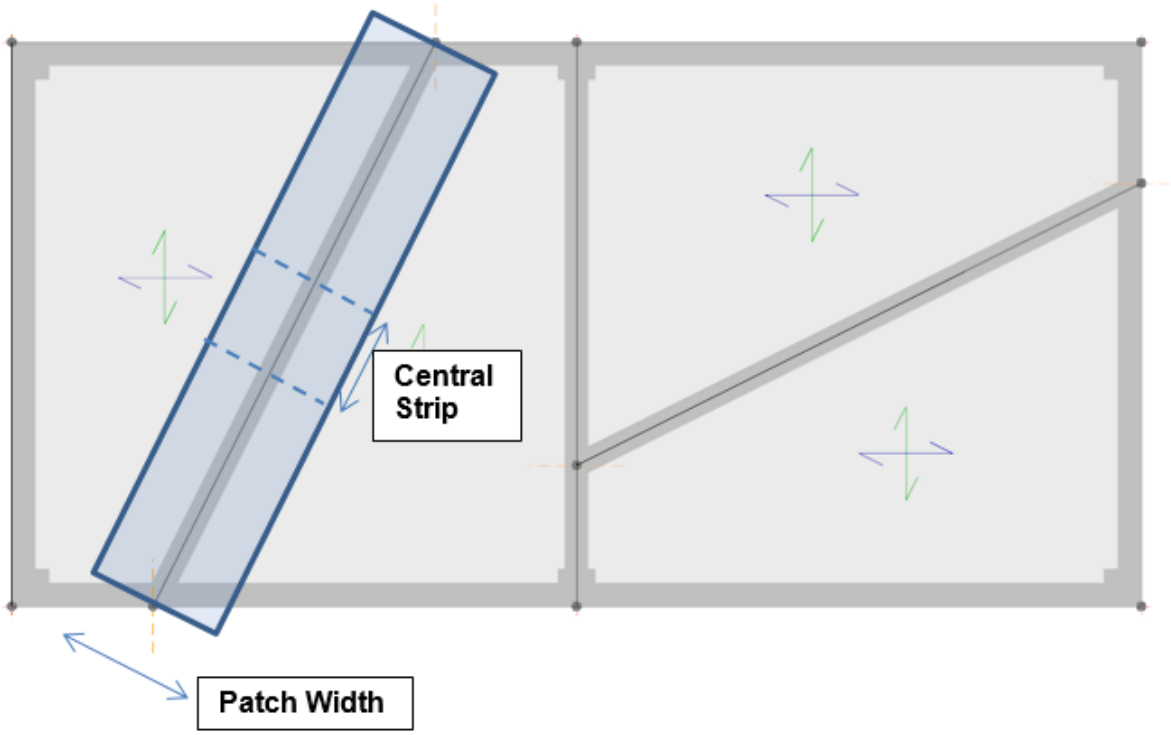
   Then from the **Patch drop list** ( ) select **Patch Beam**

2. The create beam patch properties set is displayed.

   - Use the **Surface** property to specify the layer.

- Select the **Autodesign** property in order to have the reinforcement automatically designed.
- Define the required **Patch Width** perpendicular to the beam span.
- Define the required **Center Strip Width.**
  The two end strips are recalculated accordingly and cannot be edited.
- Only select the **Consider patch surface moments only** property if you don't want to consider the opposite surface.

Situations when you need to select this option should be very rare. (Typically when the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face - when the other patch is designed additional reinforcement would be provided at that surface, so the surface check is not required for the original patch.)



To place the patch over a specific beam span:

1. Click the required beam span. *Tekla Structural Designer* will add a patch centerd on and orientated to the beam center line.

To create multiple patches by windowing:

1. Move the cursor to one corner of an imaginary box which will encompass the beams for which you want to define patches.

2. Click and hold the left mouse button.

3. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating patches).

4. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create patches to all beams which are totally within the rubber rectangle.

How do I create a wall patch?

1. Click **Design**

   Then from the **Patch drop list** ( ) select **Patch Wall**

2. The create wall patch properties set is displayed.

   - Use the **Surface** property to specify the layer.
   - Select the **Autodesign** property in order to have the reinforcement automatically designed.
   - Only select the **Consider patch surface moments only** property if you don't want to consider the opposite surface.

Situations when you need to select this option should be very rare. (Typically when the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face - when the other patch is designed additional reinforcement would be provided at that surface, so the surface check is not required for the original patch.)

3. Define the required **Patch Width** perpendicular to the wall.

To place a single patch along a specific wall:

1. Choose the **Create Mode**

   - Single Patch Along Wall

2. Click the required wall. *Tekla Structural Designer* will add a patch centerd on and orientated to the wall center line.

To place an internal patch and two end patches along a specific wall:

1. Choose the **Create Mode**

   - Internal with End Patches

2. Click the required wall. *Tekla Structural Designer* will add one internal patch and two end patches centerd on and orientated to the wall center line.

To place an end patch at one end of a specific wall:

1. Choose the **Create Mode**
   - End Patch at Wall End
2. Click near to the required end of the wall. *Tekla Structural Designer* will add a patch at the end of the wall closest to the point clicked on.

To place an internal patch part way along a specific wall:

1. Choose the **Create Mode**
   - Internal Patch
2. Move the cursor along the required wall and then click to define the required start point.
3. Move the cursor further along the wall and click again to define the required end point.
4. *Tekla Structural Designer* will add a patch between the two points clicked on.

To create patches along multiple walls by windowing:

1. Choose one of the following **Create Modes** as required:
   - Single Patch Along Wall
   - Internal with End Patches
2. Move the cursor to one corner of an imaginary box which will encompass the walls for which you want to define patches.
3. Click and hold the left mouse button.
4. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating patches).
5. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create patches to all walls which are totally within the rubber rectangle.

How do I create a panel patch?

1. Click **Design**

   Then from the **Patch drop list** ( ) select **Patch Panel**
2. The create panel patch properties set is displayed.
   - Use the **Lx** and **Ly** properties to specify the patch size.
   - Use the **Surface** property to specify the layer.

- Select the **Autodesign** property in order to have the reinforcement automatically designed.
- Only select the **Consider patch surface moments only** property if you don't want to consider the opposite surface.

Situations when you need to select this option should be very rare. (Typically when the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face - when the other patch is designed additional reinforcement would be provided at that surface, so the surface check is not required for the original patch.)

To place the patch at the centroid of a specific panel:

1. Ensure **Create Patch at Centroid** is selected.
2. Click anywhere within the required panel. *Tekla Structural Designer* will add a patch at the panel centroid.

To place the patch elsewhere within a specific panel:

Ensure **Create Patch at Centroid** is cleared.

- To define the patch by its corners, leave **Define corner points** selected and click to select the first, and then the second corner of the patch. *Tekla Structural Designer* will add a patch at the chosen position.
- To define the patch by its center, clear **Define corner points** and click where you want the center of the patch to be within the required panel. *Tekla Structural Designer* will add a patch at the chosen position.

To create multiple centroid patches by windowing:

1. Ensure **Create Patch at Centroid** is selected.
2. Move the cursor to one corner of an imaginary box which will encompass the panels within which you want to define patches.
3. Click and hold the left mouse button.
4. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating patches).
5. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create patches at the centroids of all the panels that are totally within the rubber rectangle.

How do I edit the properties of an existing patch?

1. Hover the cursor over the patch to be edited.
   The **Select Entity tooltip** should appear.

2. Press the **<down arrow>** cursor key until the patch name is highlighted in the **Select Entity tooltip**.

3. Press **<Enter>**
   One of the below property sets should be displayed, (appropriate to the type of patch selected).

   - edit column patch properties set
   - edit beam patch properties set
   - edit wall patch properties set
   - edit panel patch properties set

4. Edit the properties as required.

   - Use the **Lx** and **Ly** properties to specify the patch size.
   - Use the **Surface** property to specify the layer.
   - Select the **Autodesign** property in order to have the reinforcement automatically designed.
   - Use the **Consider Strips** property to specify the directions of reinforcement to be designed.
   - Use the **Reinforcement** property to specify the type and direction of reinforcement to be designed.
   - Define the required **Width** of each strip.
   - Specify whether each strip is to be designed for the average, or maximum **Design Force** within it.

How do I resize an existing patch?

1. Hover the cursor over the patch to be resized.
   The **Select Entity tooltip** should appear.

2. Press the **<down arrow>** cursor key until the patch name is highlighted in the **Select Entity tooltip**.

3. Press **<Enter>**

4. Hover the cursor the patch edge, or patch corner to be adjusted.
   The **Select Entity tooltip** should appear.

5. Press the **<down arrow>** cursor key until the Edge #, or Node # is highlighted in the **Select Entity tooltip**.

6. Press **<Enter>**

7. Click and drag to move the selected edge or corner.

Design Slabs

How do I design or check all slab panels in the model?

1. From the **Design** ribbon, click **Design Slabs** ( )

   All panels in the model are designed or checked automatically (in accordance with their individual 'auto-design' settings).

How do I check an individual slab panel?

1. Hover the cursor over the top of the panel to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the slab panel name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Check Panel**.

   The results of the check are displayed in a dialog on screen.

How do I design an individual slab panel?

1. Hover the cursor over the top of the panel to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the slab panel name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Design Panel**.

   The results of the design are displayed in a dialog on screen.

Design Patches

How do I design or check all patches in the model?

1. From the **Design** ribbon, click **Design Patches** ( )

   All patches in the model are designed or checked automatically (in accordance with their individual 'auto-design' settings).

How do I check an individual patch?

1. Hover the cursor over the top of the patch to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the patch name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Check Slab Patch**.

The results of the check are displayed in a dialog on screen.

How do I design an individual patch?

1. Hover the cursor over the top of the patch to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the patch name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Design Slab Patch**.

   The results of the design are displayed in a dialog on screen.

Add Punching Shear Checks

Located on the **Design** toolbar, the **Add Check** ( ) command can be used to apply punching shear checks to:

- concrete columns (with and without drops) and walls supporting a flat slab
- concrete columns and walls supported by a flat slab or foundation mat
- concrete columns (with and without drops) and walls through a flat slab
- point loads on a flat slab or foundation mat

How do I create a Punching Check item?

Punching Check items, (the objects to which the punching shear information and calculations are attached), are created using the **Add Check** command. They can be created in both 2D and 3D Views.

1. Click **Design**

   Then click **Add Check** ( )

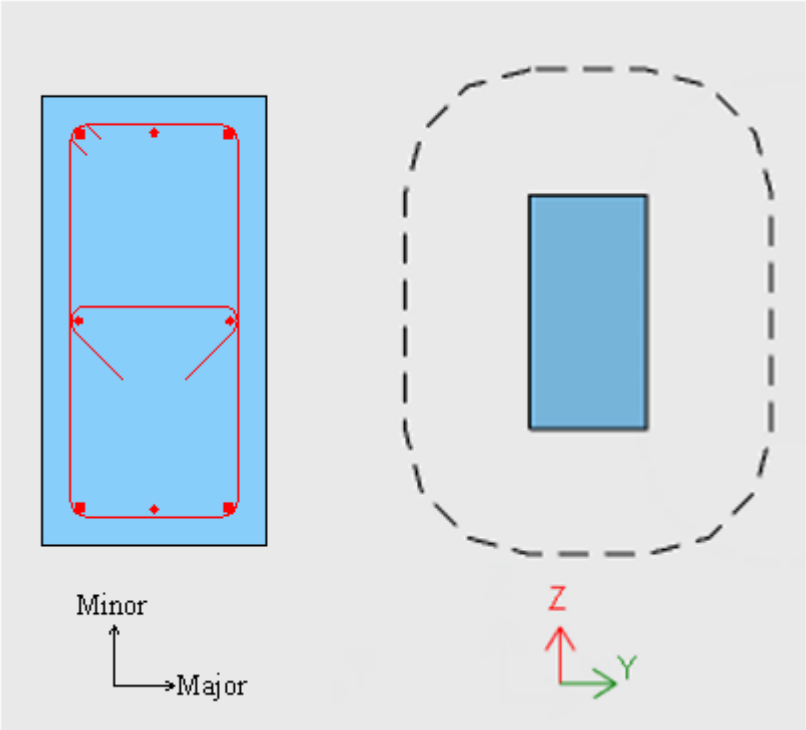
2. The create punching check properties set is displayed
   - check that the location for the slab tension reinforcement is correctly specified, then set the remaining properties as required.
   (The Point Load properties are irrelevant unless you are adding the check to a point load position).

3. To create the check, either:

- Hover the cursor over the required column node, wall node, or point load, until a 'Pick Punch Check Location' tool tip appears, then click to add the Punching Check item at the selected node, or
- Click and drag a box to add Punching Check items to all potential check locations within the box.

Punching Check axis orientation

When a Punching Check is applied to a **column**, the check Y and Z axes are automatically orientated to align with the column Major and Minor axes. (The two axis systems are locked together; if the column is rotated, the punching check axes also rotate).



When a Punching Check is applied to a **point load**, you must manually define the punching check Z axis orientation by specifying the 'Point Load Orientation' in relation to Global Y.

Point Load Orientation (along with Point Load Breadth, and Depth) are not applicable when the check is applied to a column.

How do I specify stud rail reinforcement?

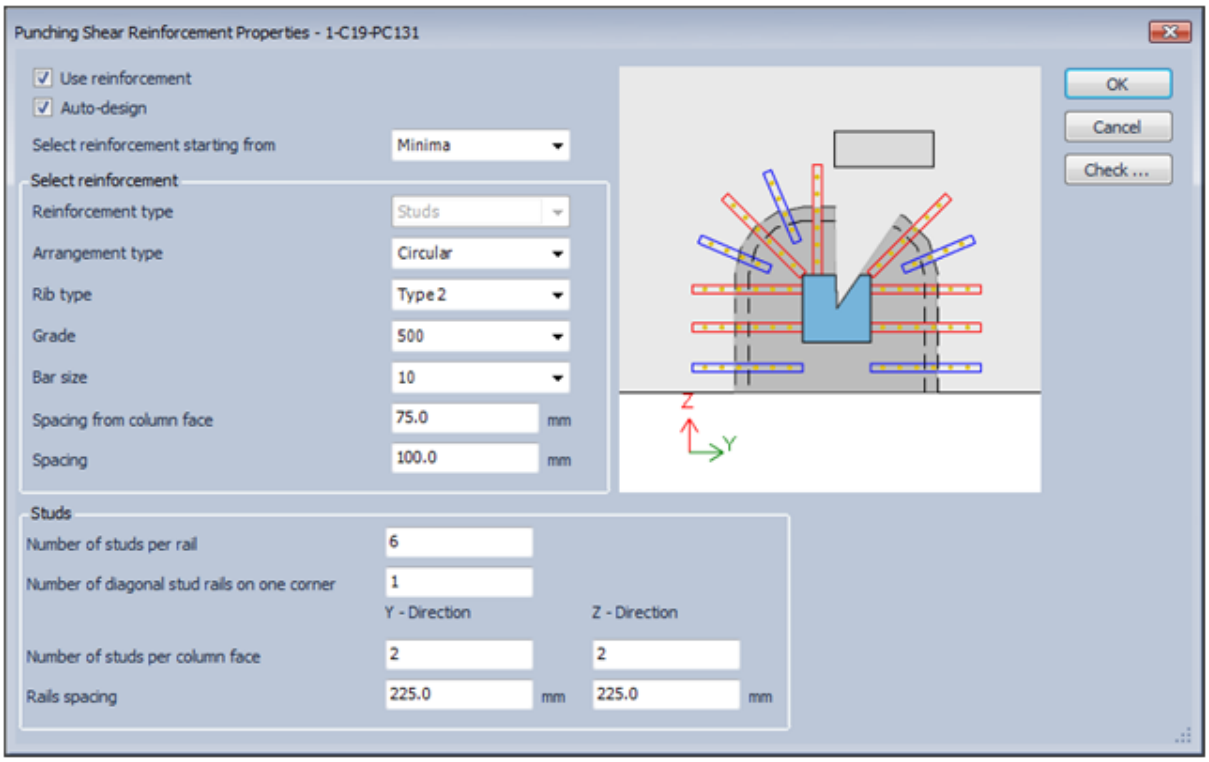
For checks at columns or under slab point loads; provided the Head Code is Eurocode or ACI, then an orthogonal or circular arrangement of stud rails can be specified and then either checked or designed as required.

1. Create a punching check item.

2. In the create punching check properties set, select the option to use reinforcement - choose whether to autodesign the reinforcement, then set the reinforcement properties as required.

Once the check has been placed, by right-clicking on the punching check item you can choose Edit Reinforcement in order to graphically modify the punching reinforcement arrangement.

Overview of the Edit Reinforcement Dialog



Graphic

The graphic updates to reflect the defined reinforcement and also indicates other items specific to the check location:

- Column section
- Punching shear check local axis
  - Using the YZ system assumes that the vertical force is positive upwards following the right hand rule
  - This is also the local axis system for the column elements so it is easier to relate
- Slab openings
- Slab opening dead-zones
  - Interruption in perimeters and reinforcement
- Slab edges
- Stud rail reinforcement
  - Design rails in red: Rails considered for design
  - Detailing rails in blue: Rails which are considered for detailing purposes only
- Perimeters
  - Dashed lines around the column section
- Critical section
  - Darker region inside the perimeters conditioned by slab edges and opening dead-zones

Fields

Use reinforcement

Select in order to apply a default punching reinforcement arrangement that can then either be checked, or used as the starting reinforcement for an auto-design.

You would only choose to clear the check box when specifying a new check if you want to perform an auto-design but starting from Minima.

Auto-design

When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.

Select reinforcement starting from

This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is 'on'.

- Minima - removes the current arrangement and begins with the minimum allowed bar size.
- Current - auto-design commences from the current bar arrangement.

Select reinforcement

Reinforcement type

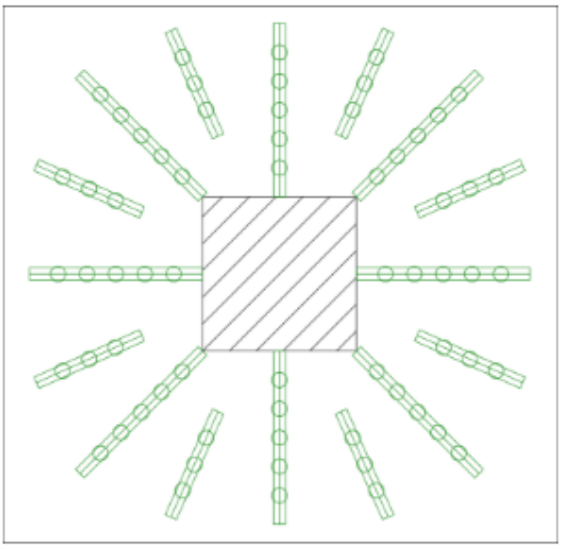
In the current release only Stud reinforcement is considered.

Arrangement type

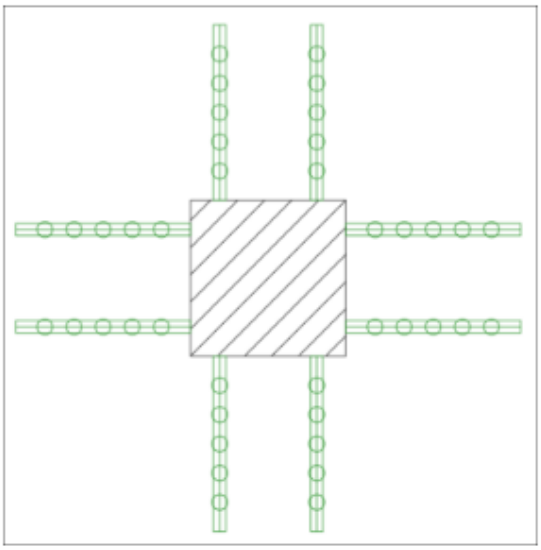
The options are:

- Orthogonal
- Circular

Circular arrangement



Orthogonal arrangement



Rib type

Specifies the reinforcement rib type.

Grade

The reinforcement grades that are available here are set from the Materials button on the Home ribbon.

Bar size

The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.

Spacing from column face

Defines the spacing of the first bar in each rail from the column face.

Spacing

Defines the spacing between bars along each rail.

Studs

Number of studs per rail

Number of studs on each rail.

Number of diagonal stud rails on one corner

The number of stud rails adjacent to each corner of the column.

(This property is only displayed when the 'Arrangement Type' is Circular)

Number of studs per column face

The number of stud rails adjacent to the column face in the local Y and Z directions.

Rails spacing

Spacing between rails in the local Y and directions.

Buttons

OK

Closes the dialog and saves the current reinforcement.

Cancel

Closes the dialog without saving changes.

Check

Opens the Results dialog displaying the detailed results for the current design.

How do I edit the properties of an existing Punching Check item?

Using the Properties Window:

1. Hover the cursor over the punching check item to be edited.
   The **Select Entity tooltip** should appear.

2. Press the **<down arrow>** cursor key until the punching check item name is highlighted in the **Select Entity tooltip**.

   - Press **<Enter>**
     The edit punching check properties set for the item should be displayed in the Properties Window.

3. Edit the properties as required.

Using the Edit Reinforcement Dialog

(This option is not available for checks applied to walls)

1. Hover the cursor over the punching check item to be edited.

2. Right-click and choose **Edit Reinforcement**

The properties are displayed graphically in a dialog - this method is recommended as it provides:

- an interactive view of the reinforcement arrangement
- a convenient 'Check' button to check the arrangement
-

Floor Vibration

The **Add Check** ( ) command in the Floor Vibration Group can be used to apply a floor vibration check over a user defined rectangular or polygon shaped slab area.

In order to create the check a primary beam, secondary beam, and critical slab item have to be identified; associated data also has to be specified before the **Check Floor Vibration** command can be used to perform the calculation.

How do I create a Floor Vibration Check item?

Floor Vibration Check items, (the objects to which the floor vibration check information and calculations are attached), are created using the **Add Check** command. They can only be created in 2D Views.

1. Click **Design**

   Then from the Floor Vibrations group click **Add Check** ( )

2. The Floor Vibration Check Properties set is displayed, in the set:

   - Choose to define a Rectangular or Polygon area of slab,
   - If rectangular - accept the angle of 0 degrees if the area is orthogonal to the global axes, or enter the angle at which the rectangle is to be drawn.

3. To create the check area, either:

   - If rectangular - click once to define the first corner, then drag, then click a second time to define the opposite corner
   - If polygon - click as required to define the polygon, (ensuring that you close the shape by clicking on the first point a second time).

4. The prompt changes to 'Create Vibration Check: PrimaryBeam'

   - Hover the cursor over the required beam, until it becomes highlighted, then click to select it.
     (the beam reference should appear in the Properties window)

5. The prompt changes to 'Create Vibration Check: SecondaryBeam'

   - Hover the cursor over the required beam, until it becomes highlighted, then click to select it.
     (again, the beam reference should appear in the Properties window)

6. The prompt changes to 'Create Vibration Check: CriticalSlab'

- Hover the cursor over the required slab, until it becomes highlighted, then click to select it.
  (the slab reference should appear in the Properties window)
- Provided that a composite slab item has been identified as the critical slab, the critical slab item properties are automatically defined, for other slab item types you are required to input these values manually.

7. The prompt changes to 'Create Vibration Check: ConfirmCreate'

- Before confirming the check, you should review the check data in the Properties Window.

8. Once all the properties are displayed as required, left click anywhere in the Scene View in order to create the check.

- The primary beam, secondary beam and critical slab are shown highlighted and the prompt changes to allowing you to continue creating further checks if required:
- If no further checks are required, press [Esc] to exit from the command.

How do I create a check which considers two or three adjoining spans?

The process is identical to that required for single spans, except that in the Primary, and/or Secondary Beam Adjoining Spans droplist you should select Two Span, or Three Span as required.

If the Two Span option is selected:

- When you hover the cursor to select the beam, only beams of two or more spans are available to be highlighted,
- When a beam is highlighted, note that it is the beam directly under the cursor that will be nominated as the 'critical beam', the second highlighted beam will be nominated as the 'adjoining beam'.
- Note also that the second highlighted beam (the 'adjoining beam') will be the beam closest to the cursor position, hence to highlight the adjoining beam at a particular end of the critical beam, simply move the cursor toward that end of the critical beam.

If the Three Span option is selected:

- When you hover the cursor to select the beam, only beams of three or more spans are available to be highlighted,
- When a beam is highlighted, note that it is the beam directly under the cursor that will be nominated as the 'critical beam', the second highlighted beam will be nominated as the 'adjoining beam'.
- Note also that the second highlighted beam (the 'adjoining beam') will be the beam closest to the cursor position, hence to highlight the adjoining beam at a particular end of the critical beam, simply move the cursor toward that end of the critical beam.

How do I edit the properties of an existing Floor Vibration Check item?

In order to select the check for editing, ensure that Floor Vibration Checks have been set to be displayed in Scene Content.

1. Hover the cursor over the slab area where the Floor Vibration Check item exists.
   The **Select Entity tooltip** should appear.

2. Press the **<down arrow>** cursor key until the floor vibration check item name is highlighted in the **Select Entity tooltip**.

3. Press **<Enter>**
   The Floor Vibration Check Properties set for the item should be displayed.

4. Edit the properties as required.

How do I check vibration for all Floor Vibration Check items?

1. Click **Design**

   Then click **Check Floor Vibration** ( )

   Floor vibration checks are performed for all Floor Vibration Check items in the model.

How do I check floor vibration for an individual Floor Vibration Check item?

1. Hover the cursor over the top of the Floor Vibration Check item to be checked.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the Floor Vibration Check item name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Check Floor Vibration**.

   The results of the check are displayed in a dialog on screen.

Foundations Guide

The Foundations toolbar contains the commands necessary to model and design both isolated and mat foundations.

Foundations toolbar

The **Foundations** toolbar contains the following commands:

Button	Description
Options	Opens the Design Options dialog
Catalogue	Opens the Pile Type Catalogue.
Pad Base Column	Creates a pad base under a column. See How do I create a pad base column?
Strip Base Wall	Creates a strip base under a wall. See How do I create a strip base wall?
Pile Cap Column	Creates a pile cap under a column. See How do I create a pile cap?
Design Pad Bases	Performs the design of all pad bases in the model. See How do I design or check all pad and strip bases in the model?
Design Pile Caps	Performs the design of all isolated foundations in the model. See How do I design or check all pile caps in the model?
Minimum Area	Creates a minimum area mat foundation under selected columns/walls. See How do I create a minimum area or rectangular mat?

Rectangular	Creates a rectangular mat foundation under selected columns/walls. See How do I create a minimum area or rectangular mat?
Strip	Creates a strip mat foundation. See How do I create a strip mat?
Area	Creates an area mat foundation. See How do I create an area mat?
Bays	Creates a mat foundation within an enclosed area defined by grids. See How do I create a mat within bays?
Column Drop (on drop list)	Creates a column drop which adopts the specified column drop properties
Mat Opening (on drop list)	Creates a mat opening which adopts the specified slab opening and mat opening properties
Mat Overhang (on drop list)	Creates a mat overhang which adopts the specified slab overhang and mat overhang properties
Mat Split	This command is used to split an existing mat. See: How do I split a slab?
Mat Join	This command is used to join an existing mat. See: How do I join slabs?
Pile	Creates a single pile which adopts the specified create pile properties set
Pile Array	Creates a pile array which adopts the specified create pile properties set
Design Mats	Designs or checks reinforcement in all mats that are not inside patches. See Design Mats
Design Patches	Designs or checks slab reinforcement in all slab areas inside patches. See Perform Patch Design

Patch Column (on drop list)	Creates a column patch which adopts the specified create column patch properties set See How do I create a column patch?
Patch Beam (on drop list)	Creates a beam patch which adopts the specified create beam patch properties set See How do I create a beam patch?
Patch Wall (on drop list)	Creates a wall patch which adopts the specified create wall patch properties set See How do I create a wall patch?
Patch Panel (on drop list)	Creates a panel patch which adopts the specified create panel patch properties set See How do I create a panel patch?
Design Punching Shear	Performs the punching shear checks. See Perform Punching Shear Checks
Add Check	Used to add a punching shear check around a column or wall perimeter. See How do I create a Punching Check item?

Related topics

- Commands on the ribbon toolbars

Foundation Options

Pile Type Catalogue

In order to use piles, you must first set up a catalogue of pile types.

How do I set up the pile type catalogue?

1. Click **Foundations > Catalogue** ( )
2. The Pile Catalogue is displayed - from here you can add, edit or delete individual pile types.
3. Click **Add...** to enter the Pile Type Properties

Pad and Strip Base modelling

- **pad base** - an isolated foundation that supports a single column
- **strip base** - an isolated foundation that supports a single wall

How do I create a pad base column?

1. Click **Foundations > Pad Base Column** ( )
2. The Pad Base Column Properties set is displayed - specify the properties as required.

To place a pad base under a specific column:

1. Click anywhere on the required column. *Tekla Structural Designer* will add a pad base under the column.

To create multiple pad bases by windowing:

1. Move the cursor to one corner of an imaginary box which will encompass the columns under which you want to define bases.
2. Click and hold the left mouse button.
3. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating bases).
4. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create bases under all columns which are totally within the rubber rectangle.

How do I create a strip base wall?

1. Click **Foundations > Strip Base Wall** ( )
2. The Strip Base Wall Properties set is displayed - specify the properties as required.

To place a strip base under a specific wall:

1. Click anywhere on the required wall. *Tekla Structural Designer* will add a strip base under the wall.

To create multiple strip bases by windowing:

1. Move the cursor to one corner of an imaginary box which will encompass the walls under which you want to define bases.
2. Click and hold the left mouse button.
3. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating bases).

4. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create bases under all walls which are totally within the rubber rectangle.

Pile Cap modelling

- **pile cap** - an isolated piled foundation that supports a single column

How do I create a pile cap?

Before a pile cap can be placed, the Pile Type Catalogue must contain at least one pile type.

1. Click **Foundations > Pile Cap Column** ( )
2. The Pile Cap Column Properties set is displayed - specify the properties as required.

To place a pile cap under a specific column:

1. Click anywhere on the required column. *Tekla Structural Designer* will add a pile cap under the column.

To create multiple pile caps by windowing:

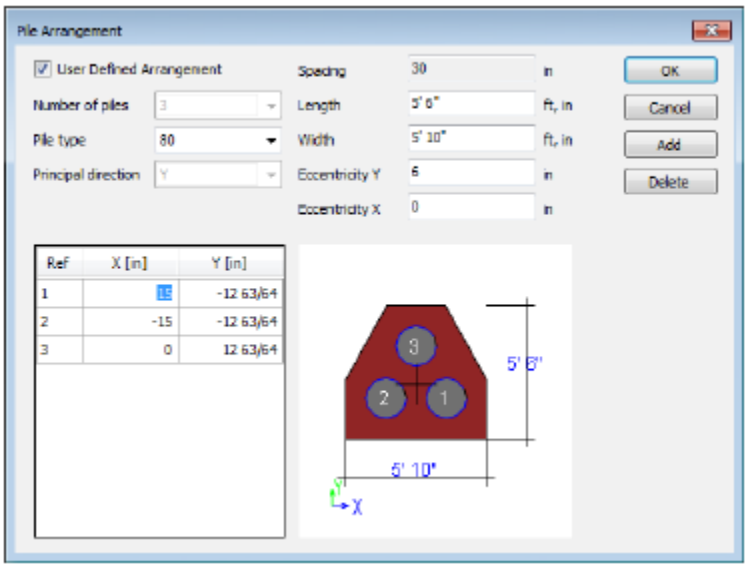
1. Move the cursor to one corner of an imaginary box which will encompass the columns under which you want to define pile caps.
2. Click and hold the left mouse button.
3. Drag to the diametrically opposite corner of the box (you will see a rubber rectangle on the screen which follows your mouse movements and helps you to check the area in which you are creating bases).
4. Once the rubber rectangle encompasses the creation area release the mouse button and *Tekla Structural Designer* will create pile caps under all columns which are totally within the rubber rectangle.

User Defined Pile Arrangement

The Pile Arrangement dialog is used to modify the pile arrangement in a pile cap.

If required a 'user defined' arrangement can be specified, (which can be particularly useful for checking a pile cap where the pile positions on site do not exactly match the originally specified arrangement).

Pile Arrangement dialog



Fields

User Defined Arrangement

When selected, you have full control to define the pile configuration you require; when cleared, the number of piles specified are arranged at the defined spacing with the pile cap length and width being determined accordingly.

Number of piles

For standard arrangements you can select the number of piles required from the drop list. For user defined arrangements use the 'Add' button instead.

Pile type

Select an existing type from the catalog, or create a new pile type.

Principal direction

The pile cap is aligned to the X or Y direction of the support underneath the column.

Spacing

The pile spacing determines the arrangement, but does not apply to user defined arrangements.

Length

Length of the pile cap

Width

Width of the pile cap

Eccentricity Y

Eccentricity in Y of the pile cap centroid from the centroid of the column it supports.

Eccentricity X

Eccentricity in X of the pile cap centroid from the centroid of the column it supports.

Pile Positioning Table (only editable for user defined arrangements)

Ref

The individual pile references as displayed on the accompanying diagram.

X

Eccentricity in X of the pile centroid from the centroid of the pile cap.

Y

Eccentricity in Y of the pile centroid from the centroid of the pile cap.

Buttons

Add Adds a new pile to the Pile Positioning Table.

Delete Deletes the pile currently highlighted in the Pile Positioning Table.

How do I open the Pile Arrangement dialog?

1. Select the pile cap to be edited.
2. In the Pile Cap Properties, click 'Pile Arrangement'
3. Click the ... button located at the right side of the 'Pile Arrangement' property box.
4. The Pile Arrangement dialog is displayed.

How do I create a user defined pile arrangement?

1. Select the pile cap to be edited.
2. Open the Pile Arrangement dialog.
3. Click 'User Defined Arrangement'.
4. If the selected pile cap had 'auto-design piles' turned on, you will be prompted to turn it off - click **Yes** to do this and proceed.
5. Specify the pile configuration as required - when the preview picture displays the arrangement as required, click **OK**.

Isolated Foundation design

Before bases and pile caps can be designed you need to have run an analysis to establish the base design forces - either by running **Analyse All (Static)**, or by running any of the design processes (**Design Concrete**, **Design All** etc.).

Once this is complete, isolated foundations can then be batch designed from the toolbar, or designed individually from the right-click context menu. Grouping can be employed to rationalize the designs produced.

How do I design or check all pad and strip bases in the model?

- Click **Foundations > Design Pad Bases**

All pad and strip bases in the model are designed or checked automatically (in accordance with their individual 'auto-design' settings). All active Static and RSA combinations are considered.

How do I design or check all pile caps in the model?

- Click **Foundations > Design Pile Caps**

  All pile caps in the model are designed or checked automatically (in accordance with their individual 'auto-design' settings). All active Static and RSA combinations are considered.

How do I check an individual isolated foundation?

1. Hover the cursor over the top of the foundation to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the foundation name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Check Member**.

   The results of the check are displayed in a dialog on screen.

How do I design an individual isolated foundation?

1. Hover the cursor over the top of the foundation to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the foundation name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Design Member**.

   The results of the design are displayed in a dialog on screen.

Mat Foundation modelling

Mat foundations support multiple columns and walls - they can be ground bearing, supported on piles, or both. The following different tools for creating mat foundations are provided:

- **minimum area** - creates an overhanging polygonal mat so as to minimise the area required
- **rectangular** - creates an overhanging rectangular mat by one of two methods
  - at a specified angle to the global axes, or
  - smallest rectangular area (aligned to the global axes)
- **strip** - creates a constant width mat along a series of points that are not restricted to being in the same line
- **area** - creates an overhanging polygonal mat by identifying points around its outline
- **bay** - creates a mat with no overhang by clicking within a closed grid area

- **pile** - creates a individual piles underneath existing mats
- **pile array** - creates an array of piles underneath existing mats

How do I create a minimum area or rectangular mat?

1. Click **Foundations > Minimum Area** ( ), or **Rectangular** ( ),
2. The foundation mat properties set is displayed - specify the properties as required.

To create a mat in a 3D view:

1. Click anywhere on the first column or wall to be supported.
2. Continue clicking on subsequent columns or walls as required.
3. Either click one of the supported members, or press **<Enter>** to create the mat.

To create a mat in a 2D view:

1. To select multiple walls - click and drag **from right to left** to encompass those walls under which you want to define the mat.
2. To select multiple columns - click and drag **from left to right** to encompass those columns under which you want to define the mat.
3. To add individual columns and wall - click each member individually.
4. Either click one of the supported members, or press **<Enter>** to create the mat.

How do I create a strip mat?

1. Open a 2D view at the base level where the strip is to be placed.
2. Click **Foundations > Strip**( )
3. The foundation mat properties set is displayed - specify the strip width and other properties as required.
4. Pick the strip start point
5. Click subsequent points to define the strip as required.
6. Either click once again on the last point, or press **<Enter>** to create the strip.

How do I create an area mat?

1. Click **Foundations > Area** ( )

2. The foundation mat properties set is displayed - specify the properties as required.
3. Pick a construction point at a corner of the area.
4. Pick subsequent construction points for each remaining corner.
5. Either click once again on the last point, or press **<Enter>** to create the mat.

How do I create a mat within bays?

1. Click **Foundations > Bays**
2. The foundation mat properties set is displayed - specify the properties as required.
3. Select a bay by either by clicking within an enclosed area defined by grids; or by dragging around the area required.

How do I create a piled mat?

A piled mat can either be supported on piles alone, or by a combination of ground springs and piles.

Piles can either be placed individually at specific locations within the mat, or an array of equally spaced piles can be created.

How do I specify if a piled mat is ground bearing?

1. Define the slab mat using any of the existing mat creation methods.
2. In the slab item properties, review the Soil Parameters that have been applied.

   Either clear **Use Ground Bearing Springs** if the mat is to be supported on piles alone, or leave it checked to support the mat by a combination of ground springs and piles.

How do I place an individual pile in a mat?

1. Click **Foundations > Pile**
2. In the 2D or 3D View, click the slab mat in which the pile is to be placed.
3. If defining an inclined pile; specify the pile direction in the pile properties window.
4. In the 2D or 3D View, select the pile position.

How do I place a pile array in a mat?

1. Click **Foundations > Pile Array**
2. Specify the Pile Array Properties to define the pile type, spacing and, for raking piles the pile direction.

You can graphically preview the array by hovering the cursor over a mat.

3. Click an individual mat, or box around multiple mats to apply the pile array.

   The array is displayed on the mats that have been selected.

4. If required, continue to add further mats to the selection, or to finish selecting, click once more on one of the mats that have already been selected.

5. Select the reference point from which piles in the array are laid out.

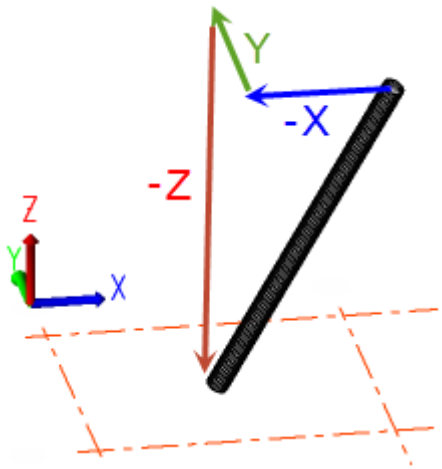
   The piles are placed in the mat. At this point you can either press Esc to finish the command, or you can continue to place the same array on additional mats.

How do specify the pile direction of an inclined pile?

The pile direction is specified in the pile properties window in using one of the following methods:

To specify the direction as a vector:

Whilst any values can be entered, the Z value of the vector must be a negative number, as this ensures that the highest point of the pile is at the node where it connects to the mat foundation.



1. Enter the X component of the vector, (or leave it as 0 if you don't want the pile to slope in the global X direction).

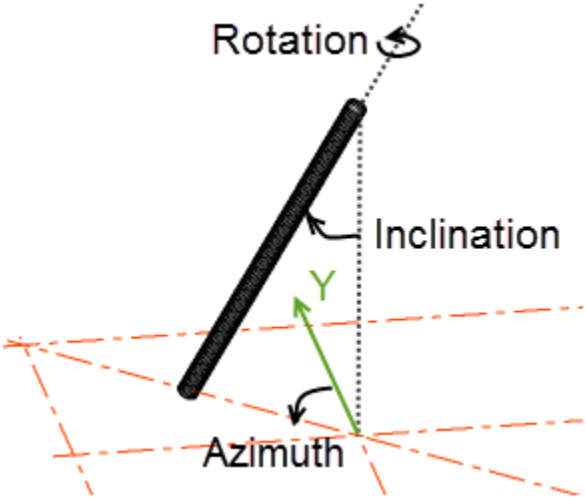
2. Enter the Y component of the vector, (or leave it as 0 if you don't want the pile to slope in the global Y direction).

3. Enter the Z component of the vector, (must be negative).

EG

To specify the direction as an angular measurement:

The direction can only be entered in this way when a pile is first positioned, (when a pile is edited its direction can only be defined as a vector).



1. In the pile properties window, check the 'Direction by Angles' box.
2. Enter the inclination of the pile from vertical.
3. Enter the azimuth, (i.e. the as a horizontal angle measured clockwise about positive global Z).
4. Enter the rotation about the piles longitudinal axis. (Since the pile is only checked for axial capacity, the rotation angle has no effect on the pile checks, but you could still use this to vary how the moment and shear force values are shown.)

EG

Mat design, patches and punching shear

Since a mat foundation is simply a type of slab, once it has been created, a similar design process can be followed:

- Create Patches - patches are placed under columns, beams, walls or in panels as required.
- Design Mats - mat panel areas that lie outside the patch areas are designed, (if the mat is supported on piles these are also designed),
- Perform Patch Design - patches are designed,
- Add Punching Shear Checks - punching shear checks are placed under columns supported by the mat as required,
- Perform Punching Shear Checks - punching shear checks are performed.

Design Mats

Once any required patches have been placed, you can determine the top and bottom reinforcement requirements in the mats that lie outside of the patch areas. (Because patch areas are excluded from panel design, patches should be placed before mats are designed.)

Before the mats can be designed you have to run an analysis to establish the design moments - either by running **Analyse All (Static)**, or by running any of the design processes (**Design Concrete**, **Design All** etc.).

All active Static and RSA combinations are considered in the design process.

How do I design or check all mats in the model?

1. Click **Foundations**

   Then click **Design Mats** ( )

   All mats in the model are designed or checked automatically (in accordance with their individual 'auto-design' settings).

How do I check all mats in a single floor?

1. Open a 2D View of the floor to be designed.
2. Right-click and choose **Check Slabs**.

   All slab panels in the floor have their reinforcement checked (regardless of their individual 'auto-design' settings).

How do I design all mats in a single floor?

1. Open a 2D View of the floor to be designed.
2. Right-click and choose **Design Slabs**.

   All mats in the floor are designed - potentially picking new reinforcement, (regardless of their individual 'auto-design' settings).

How do I check an individual mat?

1. Hover the cursor over the top of the mat to be designed.
   The **Select Entity tooltip** should appear.
2. If necessary press the **<down arrow>** cursor key until the mat name is highlighted in the **Select Entity tooltip**.
3. Right-click and from the context menu pick **Check Panel**.

   The results of the check are displayed in a dialog on screen.

How do I design an individual mat?

1. Hover the cursor over the top of the mat to be designed.
   The **Select Entity tooltip** should appear.

2. If necessary press the **<down arrow>** cursor key until the mat name is highlighted in the **Select Entity tooltip**.

3. Right-click and from the context menu pick **Design Panel**.

   The results of the design are displayed in a dialog on screen.

Design Review Guide

The **Review View** opens automatically at the end of the structure design process, it can also be accessed at any time by clicking in the Status Bar.

In this view by selecting from the Review toolbar, you can graphically display the design status for members or slabs; you can also interrogate, and/or modify a variety of model parameters and properties.

In addition, by clicking (Tabular Data) from this toolbar, you can open a **Review Data** view, from where by making appropriate selections from the Review Data toolbar you can display tables of Sway Results, Design Summaries and Material Lists.

Review toolbar

The **Review** toolbar contains the following commands:

Button	Description
Design Type (1st drop list)	Use the **Design Type** drop list to choose which design type to display results for: Static, RSA, or Combined.
Status	Each member is color coded to indicate its design status (Pass, Fail, Warning, Error, Beyond Scope, Unknown.) See Member Design Status
Ratio	Each member is color coded to indicate its design utilisation ratio. See Member Design Ratio
Depth Ratio	Each beam is color coded to indicate its span to depth utilisation ratio. See Depth Ratio
Foundation Design Status	Each foundation is color coded to indicate its design status (Pass, Fail, Warning, Error, Beyond Scope, Unknown.) See Foundation Design and Pile Design Status

Foundation Design Ratio	Each foundation is color coded to indicate its design utilisation ratio. See Foundation Design and Pile Design Ratios
Piles Status	Each pile is color coded to indicate its design status (Pass, Fail, Warning, Error, Beyond Scope, Unknown.) See Foundation Design and Pile Design Status
Piles Ratio	Each pile is color coded to indicate its design utilisation ratio. See Foundation Design and Pile Design Ratios
Slab/Mat Design Status	Each slab panel is color coded to indicate its design status (Pass, Fail, Warning, Error, Beyond Scope, Unknown.) for the selected condition chosen from the droplist. See Slab/Mat Design Status
Slab/Mat Design Ratio	Each slab panel is color coded to indicate its design utilisation ratio (for the selected condition chosen from the droplist). See Slab/Mat Design Ratio
Slab/Mat Design droplist	By default the overall slab design status or ratio is displayed, however by selecting a different option from the droplist the status or ratio can be displayed for a specific condition instead (Reinforcement, Top X, Top Y, Bottom X, Bottom Y, Span Depth.) See Slab/Mat Design droplist
Connections Status	Each connection is color coded to indicate its design status (Pass, Fail, Warning, Error, Beyond Scope, Unknown.) See Connection Design Status
Connections Ratio	Each connection is color coded to indicate its design utilisation ratio. See Connection Design Ratio
Auto\Check Design	Each member is color coded to indicate its Autodesign setting (On, or Off). Clicking on a member toggles its setting. See Auto\Check Design
Diaphragm On\Off	Each diaphragm node is color coded to indicate its setting (Excluded, or Included). Clicking on a diaphragm node toggles its setting. See Diaphragm On\Off

Fixed\Pinned	Each member is color coded to indicate its end fixity setting (N/A, Pinned, Fixed, Moment, Mixed, Cantilever). Clicking on a member toggles its fixity setting between those that are applicable. See Fixed\Pinned
BIM Status	Each member is color coded to indicate its BIM status. See BIM Status
Slab Reinforcement	Each slab or mat is color coded to indicate the reinforcement provided in the layer/direction indicated in the Properties Window. Clicking on a slab updates its reinforcement to match what you have set in the Properties Window. See Slab Reinforcement
Steel	Each steel member is color coded to indicate its section and grade. Clicking on a steel member updates its section and/or grade to match what you have set in the **Properties Window**. See Material Grade and Section
Copy Properties	After firstly selecting a parameter in the Properties Window you are able to copy it from a designated source member to valid target members. See Copy Properties
Report Filter	This command is only accessible once member filters have been defined. Each member is color coded to indicate if it is included in the currently selected filter. Clicking a member toggles its inclusion status. See Report Filter
Sub Structures	Each member is color coded to indicate if it is included in any Sub Structures. By making selections in the Properties Window, new sub structures can be created and existing ones edited. See Sub Structures
Concrete Beam Flanges	Each concrete beam is color coded to indicate its if flanges are considered and flange widths determined. See Use of beam flanges

Column Splices	Potential splice locations in steel columns are color coded to indicate where splices exist. See Column Splices
Property Sets	Each member or panel is color coded to indicate if it has had a saved Property Set applied to it. See Property Sets
UDA	Each member and panel is color coded to indicate the value of the chosen attribute in the Properties Window. See UDA
Show/Alter State	Provides access to a wider range of Show/Alter State commands via the Properties Window. Some Show/Alter State commands are repeated in the toolbar; others can only be accessed via this button. See Show/Alter State
Braced \Bracing (via Show/Alter State button)	Each concrete wall and column is color coded to indicate its braced setting (Braced, Bracing) in the direction indicated in the Properties Window. Clicking on a concrete wall or column toggles its setting between those that are applicable. See Braced/Bracing
Deflection Limits (via Show/Alter State button)	Allows you to review deflection limits or copy the specified deflection limits from one steel beam to others. See Deflection Limits
Gravity Only (via Show/Alter State button)	Each steel beam and column is color coded to indicate where its gravity only setting. See Gravity Only
Punch Check Position (via Show/Alter State button)	Each punching check item is color coded to indicate its Loaded Perimeter position (Corner, Edge Z, Edge Y, Internal). Clicking on a punching check item or boxing around it toggles its setting. See Punch Check Position
Restrained \Unrestrained (via Show/Alter State button)	Each member is color coded to indicate its restraints setting (N/A, Unrestrained, Partially Restrained, Fully Restrained). Clicking on a member toggles its setting between those that are applicable. See Restrained\Unrestrained

SFRS Direction (via Show/Alter State button)	Each member is color coded to indicate the SFRS direction. Clicking on a member updates it to match what you have set in the Properties Window. See SFRS Direction
SFRS Type (via Show/Alter State button)	Each member is color coded to indicate the SFRS type. Clicking on a member updates it to match what you have set in the Properties Window. See SFRS Type
Shear connectors (via Show/Alter State button)	Clicking on a target composite beam updates its shear connectors to match those of the source beam. See Shear Connectors
Size Constraints (via Show/Alter State button)	Allows you to review size constraints for steel members or copy the specified size constraints from one steel member to others. See Size Constraints
Steel (via Show/Alter State button)	In **Review** mode each steel member is color coded to indicate its grade. In **Copy** mode clicking on a steel member updates its grade to match that of a source member. See Material Grade
Steel section (via Show/Alter State button)	In **Review** mode each steel member is color coded to indicate its section. In **Copy** mode clicking on a steel member updates its section to match that of a source member. See Section
Stud auto layout (via Show/Alter State button)	Clicking on a composite beam updates its stud auto layout setting. See Stud auto layout
Transverse Reinforcement (via Show/Alter State button)	Clicking on a target composite beam updates its transverse reinforcement to match those of the source beam. See Transverse reinforcement
Web openings (via Show/Alter State button)	Clicking on a target member updates its web openings to match those of the source member. See Web openings
Westok openings (via Show/Alter State button)	Clicking on a target member updates its web openings to match those of the source member. See Westok openings

Tabular Data	Displays the Review Data toolbar which allows spreadsheets to be displayed for further review. See Working with the Review Data view

Review Data toolbar

The **Review Data** toolbar contains the following groups:

Button	Description
View Type (drop list)	Choose the spreadsheet to be displayed by selecting from this drop list: • How do I display Sway Results in a Review Data table? • How do I display a Design Summary in a Review Data table? • How do I display a Material List in a Review Data table?
Material Type	Choose the material type to be displayed, (only one material at a time).
Characteristic	Choose the member type to be displayed, (only one member at a time).
Construction	For steel members only - choose the construction type to be displayed.
Fabrication	Choose the fabrication type to be displayed. The choices available depend on the material and characteristic selections.
Content	For the Material List spreadsheet for concrete only - Choose the content to be displayed.
Filter (drop list)	Use the drop lists to apply group based or geometric filters to the data.

Working with the Review View

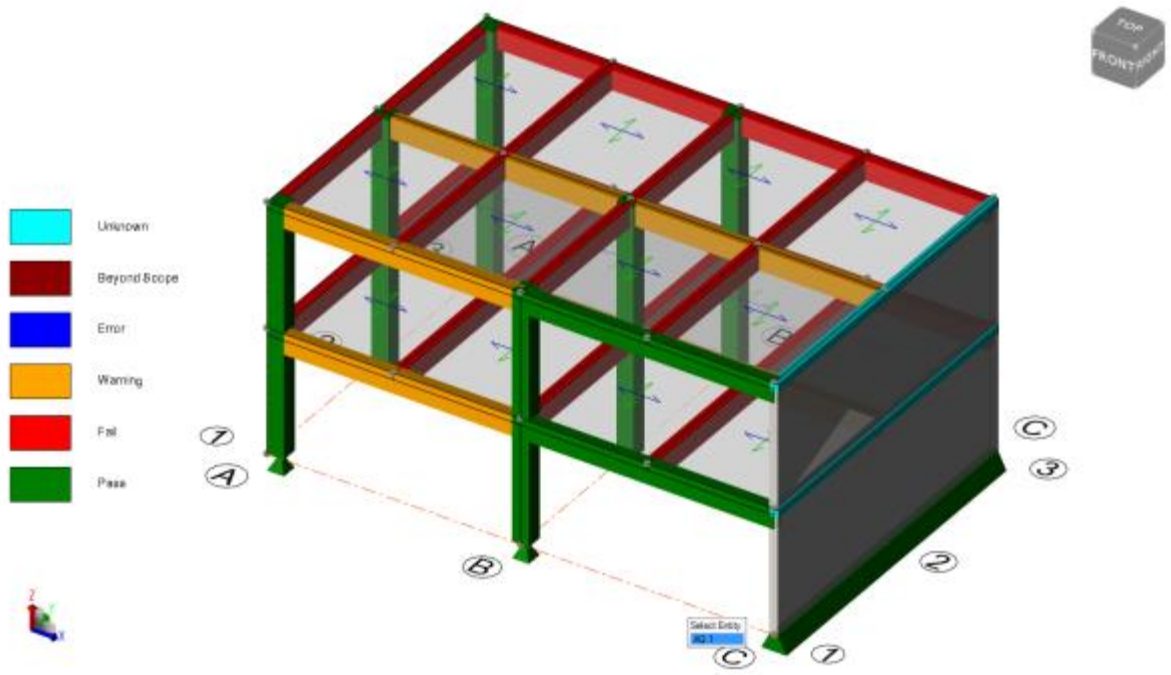
Setting the Design Type to Review

The **Design Type** drop list on the Review toolbar enables you to focus on a specific design type (Static, RSA, or Combined) when reviewing the member, foundation and slab designs.

Reviewing Member Design

Member Design Status

Color codes are used to graphically display the design status of each member.

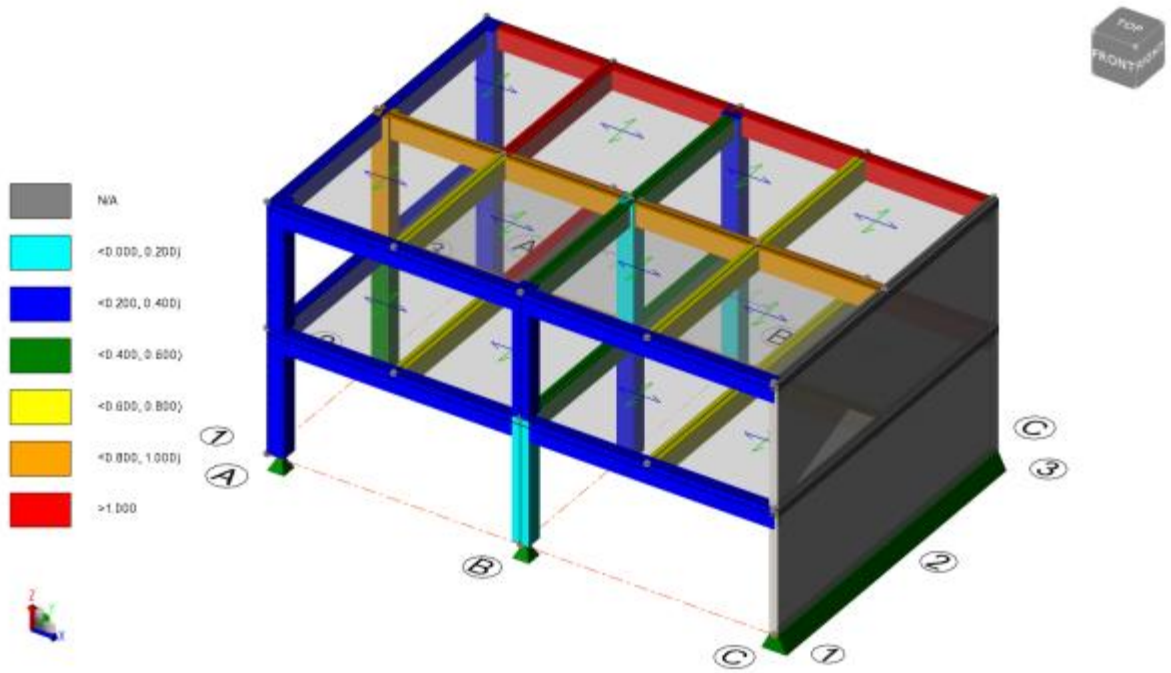


The following classifications are applied:

Status	Description
Unknown	Prior to being designed, the member status is set to unknown.
Beyond Scope	Design of this member is beyond the scope of the program.
Error	The member cannot currently be designed because an error has occurred.
Warning	Although the member has passed the design checks, one or more warnings have been issued. The user should review these warnings before deciding if any further action is required.
Fail	The member has failed one or more design checks.
Pass	The member has passed all design checks.

Member Design Ratio

Color codes are used to graphically display the design ratio of each member.



The 'N/A' colour code (which is assigned if there is no ratio, or if it is smaller than the lowest band), indicates those members either beyond scope or which have yet to be designed.

Depth Ratio

All steel and concrete beams are color coded to indicate their span to depth utilisation ratios.

Reviewing Foundation and Pile Design

Foundation Design and Pile Design Status

Color codes are used to graphically display the design status.

The following classifications are applied:

Status	Description
Unknown	Prior to being designed, the status is set to unknown.
Beyond Scope	Design of this foundation/pile is beyond the scope of the program.
Error	The foundation/pile cannot currently be designed because an error has occurred.
Warning	Although the design checks have passed, one or more warnings have been issued. The user should review these warnings before deciding if any further action is required.

Fail	The design checks have failed.
Pass	All design checks have passed.

Foundation Design and Pile Design Ratios

Color codes are used to graphically display the design ratios.

The 'N/A' colour code (which is assigned if there is no ratio, or if it is smaller than the lowest band), indicates those foundations or piles either beyond scope or which have yet to be designed.

Reviewing Slab Design

Slab/Mat Design Status

Color codes are used to graphically display the design status of each slab panel.

The following classifications are applied:

Status	Description
Unknown	Prior to being designed, the slab panel status is set to unknown.
Beyond Scope	Design of this slab panel is beyond the scope of the program.
Error	The slab panel cannot currently be designed because an error has occurred.
Warning	Although the slab panel has passed the design checks, one or more warnings have been issued. The user should review these warnings before deciding if any further action is required.
Fail	The slab panel has failed one or more design checks.
Pass	The slab panel has passed all design checks.

Slab/Mat Design Ratio

Color codes are used to graphically display the design ratio of each slab panel.

The 'N/A' colour code (which is assigned if there is no ratio, or if it is smaller than the lowest band), indicates those slab panels either beyond scope or which have yet to be designed.

Slab/Mat Design droplist

The droplist is used in combination with the Slab Design Status and Ratio buttons to filter the review information.

- Overall - the governing Top X, Top Y, Bottom X, Bottom Y, or span depth result is displayed
- Reinforcement - the governing Top X, Top Y, Bottom X, or Bottom Y result is displayed
- Top X - the result is only displayed for top reinforcement in the X direction
- Top Y - the result is only displayed for top reinforcement in the Y direction
- Bottom X - the result is only displayed for bottom reinforcement in the X direction
- Bottom Y - the result is only displayed for bottom reinforcement in the Y direction
- Span Depth - the span to depth result is displayed

Show and Alter State

Show/Alter State

The **Show/Alter State** group of commands collectively provide an efficient means to graphically review and/or edit various model properties. For each of these commands entities are color coded in accordance with a legend, the entity color changing to reflect changes as they are made.

Most of the **Show/Alter State** commands include a Filter feature in the Properties Window to control both the visibility of entities by their current setting, and also which are 'active' (i.e. those which can be edited) by their type and characteristic (where appropriate). Inactive but visible entities are shaded a light transparent grey for which there is an opacity control also in the Properties Window.

Auto\Check Design

This command provides a means to:

- graphically alter member autodesign settings,
- or by changing the Attribute, graphically alter stud auto layout settings (see: Stud auto layout)

Autodesign	
Options	
Mode	• Toggle - clicking on a member/slab switches auto design on/off • SetOn - clicking on a member/slab only sets auto design on SetOff - clicking on a member/slab only sets auto design off Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Filter	

Show	• All - all members are shown • On- only members with autodesign on are shown • Off - only members with autodesign off are shown
Entity Type	Select from the droplist to filter the display to a particular entity type
Characteristic	Select from the droplist to filter the display to a particular characteristic
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To alter autodesign settings:

1. Open a **Review View**
2. Click **Review > Auto\Check Design**

   Members and slabs are color coded to show their autodesign setting.

3. Choose the appropriate mode in the Properties Window, then either:

- Click an individual member to toggle/set its Autodesign setting.
- Drag a box from left to right to toggle/set the Autodesign setting for all members totally enclosed by the box.
- Drag a box from right to left to toggle/set the Autodesign setting for all members that are either enclosed by the box, or cut by the box perimeter.

BIM Status

This command provides a means to graphically assess the BIM Status, and also to exclude members and panels from the import/export process.

BIM status	
Filter	
Show	Select from the droplist to filter the display to a particular BIM status
Entity Type	Select from the droplist to filter the display to a particular entity type
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

BIM Status Values:

Status	Description
New	If newly created in *Tekla Structural Designer*
Imported	If created externally
Exported	If exported
Updated	If modified by an import
Deleted	If the member appears to have been deleted in the other system
Mixed	If the status varies from span to span in multi-span beams, or lift to lift in multi-lift columns.
Excluded	Indicates those parts of the model that have been excluded from the BIM process.
Not modified	Indicates those parts of the model that have not been modified by the last BIM import.

To include/exclude members and panels from the BIM process:

1. Open a **Review View**
2. Click **Review > BIM Status**

- Click an individual member\panel to exclude it, click it once more to re-include it.
- Drag a box from left to right to toggle the exclude setting for all members totally enclosed by the box.
- Drag a box from right to left to toggle the exclude setting for all members that are either enclosed by the box, or cut by the box perimeter

Braced/Bracing

This command provides a means to graphically assess and modify the slenderness settings in each direction for concrete columns and walls.

Braced/Bracing	
Filter	
Show	• All - all members are shown • Bracing - only 'Bracing' members are shown • Braced - only 'Braced' members are shown
Entity Type	Select from the droplist to filter the display to a particular entity type

Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable
Options	
Direction	• Direction1 - assess and modify slenderness settings in direction 1 • Direction2 - assess and modify slenderness settings in direction 2
Entire Element	• Checked - changes are applied to all stacks • Cleared - changes are applied to the individual stack clicked on

To modify slenderness settings:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Braced\Bracing**
4. In the Properties Window, choose the direction under consideration.

   Members are color coded to show their slenderness setting in the chosen direction.

- Clicking on an individual concrete column or wall toggles its slenderness setting between the braced and bracing.
- Dragging a box from left to right toggles the restraint setting for all concrete columns and walls totally enclosed by the box.
- Dragging a box from right to left toggles the restraint setting for all concrete columns and walls that are either enclosed by the box, or cut by the box perimeter.

If you required to toggle an individual stack, rather than all stacks; clear **Entire element** in the Properties Window.

Column Splices

This command provides a means to graphically assess and modify splice positions within steel columns.

To modify column splices:

1. Open a **Review View**
2. Click **Review** > **Column Splices**

   All potential splice locations are color coded to indicate if they are on or off.

3. Click a potential splice location to toggle its setting between on and off, or in a 2D View click-and-drag a window to toggle multiple splices in one go.

Concrete Beam Flanges

This command provides a means to graphically review the flanges of concrete beams.

To review concrete beam flanges:

1. Open a **Review View**
2. Click **Review** > **Concrete Beam Flanges**

   Each concrete beam is color coded to indicate its if flanges are considered and flange widths determined.

Copy Properties

This command provides a means to graphically copy a specified element parameter (e.g. web openings, connectors or transverse reinforcement) from a source member to other valid target members.

To copy properties:

1. Open a **Review View**
2. Click **Review** > **Copy Properties**
3. In the Properties Window, select the parameter of the member you want to be copied (e.g. web openings, connectors or transverse reinforcement).
4. Next click the source member that contains the property to be copied. The color coding should update accordingly:
   - Source - this color identifies the member that was clicked on. (If this is incorrect, press [Esc] in order to reselect.)
   - Same as Source - this color identifies those members in which the selected parameter already matches the Source.
   - Valid Target - this color identifies those members to which it is possible to copy the selected parameter.
   - NA - this color identifies those members to which it is not possible to copy the selected parameter.
   - Either click an individual target member, or box around a series of target members to copy the selected parameter to them.

Deflection Limits

Graphically review specified deflection limits, and copy the limits from one steel beam to others.

Deflection Limits	
Options	

Mode	• Copy - copy specified deflection limits from one steel beam to others • Review - graphically review the deflection limits Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time)
By load type	• Checked - provides options for selecting an individual load type and load limit • Cleared - limits for all load types
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To copy deflection limits:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Deflection Limits**
4. Set the Mode to **Copy**
5. Either leave **By Load type** cleared if you want to copy the limits for all load types, or check it if you want to specify an individual load type.

   If you specify an individual load type you can also choose to copy the absolute value, the span over factor, or both.
6. Select a valid source beam.
7. Select a beam to copy the limits to.
8. Continue copying as required, or press ESC when done.

To review deflection limits:

1. Set the Mode to **Review**
2. Select the load type to be reviewed.

Diaphragm On\Off

The 'Include in diaphragm' setting for slab items, roofs and nodes can be toggled graphically.

Diaphragm

Options	
Mode	• Toggle - clicking on a slab/roof/node switches it between included/excluded • Include - clicking on a slab/roof/node includes it • Exclude - clicking on a slab/roof/node excludes it Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Filter	
Show	• All - all slabs/roofs/nodes are shown • Included- only included slabs/roofs/nodes are shown • Excluded - only excluded slabs/roofs/nodes are shown
Entity Type	Select from the droplist to filter the display to a particular entity type
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To display the current diaphragm settings:

1. Open a **Review View**
2. Click **Review > Diaphragm On\Off**

   Slab items, roofs and nodes are color coded to show their 'Include in diaphragm' setting.

   The Properties Window shows the currently selected 'toggle' mode.

To toggle the diaphragm settings for slab items or roofs:

First set the 'Entity type' in the Properties Window to 'Slab Item' or 'Roof' then either:

- Click an individual panel.
- Drag a box from right to left to toggle the setting for all panels that are either enclosed by the box, or cut by the box perimeter.
- Drag a box from left to right to toggle the setting for all internal panels totally enclosed by the box. (Panels at the edge of the diaphragm are not toggled when dragging left to right.)

To include/remove solver nodes from the diaphragm:

First set the 'Entity type' in the Properties Window to 'Solver Node' then either:

- Click an individual node to change its state (according to the selected mode).

- Drag a box to change the state for all nodes enclosed by the box (according to the selected mode).

Fixed\Pinned

This command provides a means to graphically assess and modify the end fixity for all members in the model.

Fixed/Pinned	
Options	
Mode	• Toggle - clicking on a member toggles its end fixity settings between the types that are valid for the member • Set On- clicking on a a member sets its end fixity to fixed or pinned, according to the Fixity Type specified in the Properties Window Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To toggle end fixity:

1. Open a **Review View**
2. Click **Review > Fixed\Pinned**

- Clicking on an individual member toggles its end fixity settings between the types that are valid for the member.
- Dragging a box from left to right toggles the end fixity settings for all members totally enclosed by the box.
- Dragging a box from right to left toggles the end fixity settings for all members that are either enclosed by the box, or cut by the box perimeter.

Where the end fixity is shown as 'Mixed' this indicates that the fixity at end 1 differs from that at end 2. 'Mixed' end fixity can only be specified by editing the member properties directly.

Gravity Only

This command provides a means to graphically assess and modify the 'Gravity Only' setting for beams and columns.

Gravity only

Options	
Mode	• Toggle - clicking on a member toggles its 'gravity only' setting • Set On- clicking on a member sets 'gravity only' on • Set Off- clicking on a member sets 'gravity only' off TIP! Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Filter	
Show	• All - all members are shown • On- only 'gravity only' members are shown • Off - only members that are not 'gravity only' are shown
Characteristic	Select from the droplist to filter the display to a particular characteristic
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To graphically change 'gravity only' settings:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Gravity Only**

All members are color coded to indicate their 'Gravity Only' setting.

- Clicking on a steel beam or column toggles its Gravity Only' setting between on and off.
- Dragging a box from left to right toggles the on/off setting for all steel beams and columns totally enclosed by the box.
- Dragging a box from right to left toggles the on/off setting for all steel beams and columns that are either enclosed by the box, or cut by the box perimeter.

Property Sets

This command provides a means to graphically review or apply property sets to members, slab items and roof panels.

Property Sets	
Review/Update	Choose to review or apply property sets

Show	• All - all members are shown • On- only 'gravity only' members are shown • Off - only members that are not 'gravity only' are shown
Entity Type	Select from the droplist to filter the display to a particular entity type
Characteristic	Select from the droplist to filter the display to a particular characteristic
Property set	When applying property sets, choose a property set from this list (which is filtered by the selected entity type/characteristic).

Punch Check Position

This command provides a means to graphically modify the assumed punching check position for determining the loaded perimeter when the perimeter is close to a free edge.

How do I modify the assumed punch check position in the Review View?

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Punch Check Position**
4. Click or box around punching check items to change their positions:
   - Click a punch check item on a Z edge to toggle its position between 'Edge Z' and 'Internal'.
   - Click a punch check item on a Y edge to toggle its position between 'Edge Y' and 'Internal'.
   - Click a punch check item on a corner to toggle its position between 'Corner', 'Edge Z' 'Edge Y' and 'Internal'.
   - Clicking an internal punch check item does not toggle its position.
   - Dragging a box from left to right toggles the position setting (as described above) for all punch check items totally enclosed by the box.
   - Dragging a box from right to left toggles the position setting (as described above) for all punch check items that are either enclosed by the box, or cut by the box perimeter.

Report Filter

This command provides a means to graphically review and modify any member filters that have been defined.

The Report Filter command remains unavailable until a member filter has been defined. (A members filter is defined by clicking **Members** *in the* **Filters** *group on the* **Report** *toolbar).*

1. Open a **Review View**
2. Click **Review** > **Report Filter**

- Clicking on an individual member toggles its inclusion in the selected member filter between 'on' and 'off'.
- Dragging a box from left to right toggles the inclusion setting for all members totally enclosed by the box.
- Dragging a box from right to left toggles inclusion setting for all members that are either enclosed by the box, or cut by the box perimeter.

Restrained\Unrestrained

This command provides a means to graphically assess and modify the lateral restraint settings for all steel beams and columns in the model.

Restrained\Unrestrained	
Options	
Mode	• Toggle - clicking on a member toggles it between fully restrained and unrestrained • Set On- clicking on a member sets it to fully restrained • Set Off- clicking on a member sets it to unrestrained Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Filter	
Show	Select from the droplist to filter the display to a members with a particular restraint condition
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To graphically change restraint settings:

1. Click **Review > Show/Alter State**

2. In the Properties Window, click the **Attributes** list, and then click **Restrained/unrestrained**

   Members are color coded to show their restraint setting.

3. To modify lateral restraint settings, either:

- Click an individual member to toggle its restraint setting between the types applicable to the member.
- Drag a box from left to right to toggle the restraint setting for all members totally enclosed by the box.
- Drag a box from right to left to toggle the restraint setting for all members that are either enclosed by the box, or cut by the box perimeter.

SFRS Direction

This command provides a means to graphically review and modify the direction of any seismic force resisting systems (SFRS) in the model.

SFRS Direction	
Options	
Mode	• Review Direction -shows which members/walls are in which SFRS direction • Update Direction - clicking on a member/wall includes it in the SFRS direction • Toggle Direction - clicking on a member/wall toggles it between each SFRS direction setting. • Remove from SFRS - clicking on a member/wall excludes it from the SFRS Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Direction	• Direction1 - review/update SFRS in direction 1 • Direction2 - review/update SFRS in direction 1
Entire Element	• Checked - changes are applied to all stacks/spans • Cleared - changes are applied to the individual stack/span clicked on
Filter	
Entity Type	Select from the droplist to filter the display to a particular entity type
Scene	

Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To graphically review/change SFRS Direction settings:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **SFRS Direction**

   Members and walls are color coded to show their 'SFRS Direction' setting.

4. Select the required mode in the Properties Window.
   - Review Direction - shows which members/walls are in which SFRS direction
   - Update Direction - clicking on a member/wall includes it in the SFRS direction shown in the Properties Window. (Either the entire member, or the span/stack in accordance with the 'Entire element' check box setting)
   - Toggle Direction - clicking on a member/wall toggles it between each SFRS direction setting. (Either the entire member, or the span/stack in accordance with the 'Entire element' check box setting)
   - Remove from SFRS - clicking on a member/wall excludes it from the SFRS
5. Click the members to change.

SFRS Type

This command provides a means to graphically review and modify the types of seismic force resisting systems (SFRS) in the model.

SFRS Type	
Options	
Mode	• Review Type -shows which members/walls are in which SFRS type • Update Type - clicking on a member/wall changes its SFRS type • Remove from SFRS - clicking on a member/wall excludes it from the SFRS TIP! Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Select direction	When updating the type: • Checked - the direction is changed to the direction specified in the Properties Window • Cleared - when the type is updated the original direction is maintained

Direction	When updating the type: • Direction1 - sets the SFRS to direction 1 • Direction2 - sets the SFRS to direction 1
Entire Element	• Checked - changes are applied to all stacks/spans • Cleared - changes are applied to the individual stack/span clicked on
Filter	
Entity Type	Select from the droplist to filter the display to a particular entity type
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To graphically review/change SFRS Type settings:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **SFRS Type**

   Members and walls are color coded to show their 'SFRS Type' setting.

4. Select the required mode in the Properties Window.
   - Review Type - shows which members/walls are in which SFRS type
   - Update Type - clicking on a member/wall includes it in the SFRS type and direction shown in the Properties Window. (Either the entire member, or the span/stack in accordance with the 'Entire element' check box setting)
   - Remove from SFRS - clicking on a member/wall excludes it from the SFRS
5. Click the members to change.

Shear Connectors

Composite beam shear connector data can be copied from a source beam to other composite beams as follows:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Shear Connectors**

   Composite beams that can act as a source for the shear connector data are color coded in the Review View.

4. Click a valid source beam.

5. Click an a valid target beam to update its shear connector data to match the source.

Size Constraints

Graphically review specified size constraints for steel beams, columns, and braces, and copy the constraints from one steel member to others.

Deflection Limits	
Options	
Mode	• Copy - copy specified size constraints from one steel member to others • Review - graphically review the size constraints Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
By constraint type	• Checked - provides options for selecting an individual constraint type to review or copy • Cleared - when copying, all size constraints are copied
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To copy size constraints:

1. Open a **Review View**

2. Click **Review > Show/Alter State**

3. In the Properties Window, click the **Attributes** list, and then click **Size Constraints**

4. Set the Mode to **Copy**

5. Either leave **By constraint type** cleared if you want to copy all the constraint types, or check it if you only want to copy an individual constraint type.

6. Select a valid source steel member.

7. Select a steel member to copy the constraints to.

8. Continue copying as required, or press ESC when done.

To review size constraints:

1. Set the Mode to **Review**

2. Select the size constraint to be reviewed.

Slab Reinforcement

This command provides a means to graphically review and modify slab panel and patch reinforcement, allowing you to update the bar size and spacing applied in each layer and direction.

> *If you arrange your screen to have two Review Views open side by side - you can then use the first of these to modify the slab reinforcement whilst displaying the design status in the second. With this arrangement, each change you make to the reinforcement immediately updates the design status.*

1. Open a **Review View**
2. First click **Review** > **Auto\Check Design** and ensure that the slab panels you intend to modify have Autodesign off.
3. Then click **Review** > **Slab Reinforcement**

To modify slab reinforcement:

1. In the Properties Window, select the Slab Reinforcement you want to modify, (Panel, or Patch).
2. Select the Reinforcement Direction you want to modify, (BarsX, BarsY or Mesh).
3. Select the Slab Layer you want to modify, (Top or Bottom).
4. If modifying bars, select the Bar Parameters you want to modify, (Bar Size, Spacing, or Bar Size & Spacing).
5. Use the Apply drop list to select the Bar Parameters that you want to apply.
6. Click an individual slab to update the reinforcement to that specified.
7. Use **Review** > **Status** to check that the updated reinforcement is sufficient.

To graphically copy reinforcement between panels or patches:

1. In the Properties Window, select the Slab Reinforcement you want to modify, (Panel, or Patch).
2. Select the Reinforcement Direction you want to modify, (BarsX, BarsY or Mesh).
3. Select the Slab Layer you want to modify, (Top or Bottom).
4. Click the panel or patch containing the reinforcement to be copied.
5. Click the panels or patches to which you want to apply the reinforcement.

> *To change the source of the reinforcement being copied press [Esc] and select a different panel or patch.*

6. Use **Review** > **Slab Design Status** to check that the updated reinforcement is sufficient.

Stud auto layout

The stud auto layout setting for composite beams can be toggled graphically.

Stud auto layout	
Options	
Mode	• Toggle - clicking on a beam switches auto layout on/off • Set On- licking on a beam only sets auto layout on • Set Off- clicking on a beam only sets auto layout off Shortcut: **Ctrl [M]** cycles through the Modes, (provided that you have nothing highlighted in the Properties Window at the time).
Filter	
Show	Select from the droplist to filter the display to a members with a particular auto layout condition
Scene	
Opacity N/A	Reduces the opacity of elements to which the command is not applicable

To graphically modify the stud auto layout settings:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Stud auto layout**

   Composite beams are color coded to show their auto layout setting.

   The Properties Window shows the currently selected mode.

   - Toggle - clicking on a beam switches auto layout on/off
   - SetOn - clicking on a beam only sets auto layout on
   - SetOff - clicking on a beam only sets auto layout off

4. Choose the appropriate mode in the Properties Window, then either:

- Click an individual beam to toggle/set its auto layout setting.
- Drag a box from left to right to toggle/set the auto layout setting for all beams totally enclosed by the box.

- Drag a box from right to left to toggle/set the auto layout setting for all beams that are either enclosed by the box, or cut by the box perimeter.

Sub Structures

This command provides a means to graphically create, review and modify sub structures for modeling purposes.

These can prove useful in large models as individual sub-structures can then be differentiated by color and worked on in separate sub-structure views.

1. Open a **Review View**
2. Click **Review** > **Sub Structures**

Transverse reinforcement

Composite beam transverse reinforcement can be copied from a source beam to other composite beams as follows:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Transverse reinforcement**

   Composite beams that can act as a source for the transverse reinforcement data are color coded in the Review View.
4. Click a valid source beam.
5. Click a valid target beam to update its transverse reinforcement data to match the source.

UDA

This command provides a means to graphically review and modify the values of attributes that have been attached to the model.

Web openings

Web openings can be copied from a source member to other members as follows:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Web Openings**

   Members that can act as a source for the web opening data are color coded in the Review View.
4. Click a valid source member.
5. Click a valid target member to update its web openings data to match the source.

Westok openings

Westok openings can be copied from a source member to other members as follows:

1. Open a **Review View**
2. Click **Review > Show/Alter State**
3. In the Properties Window, click the **Attributes** list, and then click **Westok Openings**

   Members that can act as a source for the Westok opening data are color coded in the Review View.
4. Click a valid source member.
5. Click a valid target member to update its Westok openings data to match the source.

Working with the Review Data view

Clicking the Tabular Data button displays the Review Data view from where Sway Results, Design Summaries and Material Lists can be displayed in tables.

How do I display a Design Summary in a Review Data table?

1. To display a Design Summary in a Review Data table you must first open a Review View.
2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)
3. From the drop list on the **View Type** toolbar group choose **Design Summary**.
4. Make selections from the other groups on the toolbar in order to filter to the specific results required.

How do I display Sway Results in a Review Data table?

1. To display Sway Results in a Review Data table you must first open a Review View.
2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)
3. From the drop list on the **View Type** toolbar group choose **Sway**.
4. Make selections from the other groups on the toolbar in order to filter to the specific results required.

How do I display Storey Shear in a Review Data table?

1. To display Storey Shear in a Review Data table you must first open a Review View.
2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)

3. From the drop list on the **View Type** toolbar group choose **Storey Shear**

4. From the Loading drop list choose the loadcase or combination required.

How do I display Drift Results in a Review Data table?

Drift Results are only relevant to models in which the United States (ACI/AISC) Head code has been selected.

1. To display Drift Results in a Review Data table you must first open a Review View.

2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)

3. From the drop list on the **View Type** toolbar group choose **Drift**.

4. Make selections from the other groups on the toolbar in order to filter to the specific results required.

How do I display Seismic Drift Results in a Review Data table?

1. To display Seismic Drift Results in a Review Data table you must first open a Review View.

2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)

3. From the drop list on the **View Type** toolbar group choose **Seismic Drift**.

4. Make selections from the other groups on the toolbar in order to filter to the specific results required.

How do I display Wind Drift Results in a Review Data table?

1. To display Wind Drift Results in a Review Data table you must first open a Review View.

2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)

3. From the drop list on the **View Type** toolbar group choose **Wind Drift**.

4. Make selections from the **Material Type** and **Fabrication** groups to identify the column results required.

5. From the Loading drop list choose the wind loadcase or combination to be checked.

How do I display Connection Resistance checks in a Review Data table?

The connection resistance check compares the maximum vertical shear from the gravity combinations against a pre-defined resistance for the supporting member.

This check requires you to have specified a set of resistances for each section serial size, composite|non composite, for the current head code. These are specified from the Materials dialog on the Home ribbon.The resistances are saved to the database and can subsequently be used for checking other structures.

1. To display Connection Resistances in a Review Data table you must first open a Review View.
2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)
3. From the drop list on the **View Type** toolbar group choose **Connection Resistance**
4. Make selections from the other groups on the toolbar in order to filter to the specific results required.

How do I display a Material List in a Review Data table?

The material list displayed in a Review Data table is not suitable for printing - to obtain a formatted list that can be output you should display a Material Listing Report instead.

1. To display a Material List as a Review Data table you must first open a Review View.
2. Then click **Review** > **Tabular Data**

   (A Review Data View opens in a new tab.)
3. From the drop list on the **View Type** toolbar group choose **Material List**.
4. Make selections from the other groups on the toolbar in order to filter to the specific results required.

EG

For example, to display the loose bar estimate in a flat slab you would choose:
- 'Concrete' from the 'Material Type' group
- 'Slabs' from the 'Characteristic' group
- 'Loose Bars' from the 'Content' group.

Reporting Guide

A wide range of reports which can be tailored to meet your specific requirements can be created from the Report toolbar.

Report toolbar

The 'Contents' and 'Filters' groups described below are always displayed, but the 'Appearance', 'Navigation' and 'Export' groups are hidden until a Report view is active.

Contents group

The **Contents** group contains the following commands:

Button	Description
Select	Use the drop list to select the report to be created. The chosen report can then be viewed by clicking **Show Report**.
Model Report...	Opens a dialog which can be used to: • setup a new **Model Report**, • edit and filter the content for an existing **Model Report**.
Member Report...	Opens a dialog which can be used to: • setup a new **Member Report**, • edit and filter the content for an existing **Member Report**.
Show Report	Displays the selected Model Report (i.e. the one that is shown in the **Select** drop list) on screen. Clicking **Show Report** also activates other groups on the toolbar to enable: • the appearance of the report to be controlled, • navigation around the report, • export of the report.

Filters Group

The **Filters** group contains the following commands:

Button	Description
Levels	Opens a dialog for creating a new **Model Filter** which can be used to output selected levels only.
Frames	Opens a dialog for creating a new **Model Filter** which can be used to output selected frames only.
Planes	Opens a dialog for creating a new **Model Filter** which can be used to output selected planes only.
Loadcases	Opens a dialog for creating a new **Loading Filter** which can be used to output selected loadcases only.
Combinations	Opens a dialog for creating a new **Loading Filter** which can be used to output selected combinations only.
Members	Opens a dialog for creating a new **Model Filter** which can be used to output selected members only.
Design Groups	Opens a dialog for creating a new **Model Filter** which can be used to output selected design groups only.
Trusses	Opens a dialog for creating a new **Model Filter** which can be used to output selected trusses only.
Portal Frames	Opens a dialog for creating a new **Model Filter** which can be used to output selected portal frames only.

Appearance group

Use the commands in this group to control the appearance of the currently displayed report.

Button	Description
Settings	Opens the **Settings** dialog which can be used to configure paragraph styles and other report display options.

Page Setup	Opens the **Page Setup** dialog which can be used to configure page size and orientation.
Edit Header	Opens the **Document headers/footers** dialog which can be used to configure the layout and content of the page header and footers.
Edit Footer	Opens the **Document headers/footers** dialog which can be used to configure the layout and content of the page headers and footers.

Navigation group

Use the commands in this group to move through the currently displayed report.

Button	Description
Previous Page, First Page, Next Page, Last Page	Use the commands in this group to move around the currently open report.
Report Index	Displays the report index in the **Project Workspace**. This can be used to navigate through the report.

Export group

The commands in this group can be used to export the currently open report to other programs.

Button	Description
PDF	Exports the report to a pdf document.
Word	Exports the report to a word document.
Excel	Exports the report to an Excel document.
Tedds	Exports the report to a Tedds document.

Reports and Report Terminology Explained

This topic provides background information to improve your understanding of the report creation process and the terminology involved.

Model Reports

Model Reports are used to set up the printed output for the entire structure, (or a part of it). These reports are configured by choosing specific output categories ('chapters') from a list of all the available output categories. You can include entire chapters, or just those headings within the chapter you require. The resulting subset of chapters is referred to as a **Report Structure**. Filters can then be applied to individual headings in the Report Structure to limit the output that is produced.

Model Report chapters include:

- **Structure**
  Structure data under headings such as 'Loadcases', 'Wind Data' etc.
- **Analysis**
  Analysis model properties and results.
- **Concrete, Steel, Timber, Cold Formed, Cold Rolled, General Material**
  - Member Reports for each of the different member types. Each of these can be set up to contain as much or as little output as you require. This is achieved by configuring a separate **Member Report** for each member type that you have included.
  - Design Summary Tables can also be included for those members that can be designed.
- **Beam End Forces, Bracing Forces, Foundation Reactions**
  Chapters for specific sets of analysis results.
- **Picture**
  A 3D view of the whole structure, or by applying a Model Filter you can choose selected sub structures in 3D, or 2D views of selected frames or levels.
  Applied loads can also be displayed on the picture by applying a Loading Filter.
- **View**
  The current display in any 2D or 3D Scene View can be saved at any time as a **View Configuration**. Each of these view configurations can then be included in the Model Report as required by including a separate View chapter for each configuration required.
- **Analysis Diagram**
  A 3D diagram of the whole structure, or by applying a Model Filter you can choose selected sub structures in 3D, or 2D diagrams of selected frames or levels.
  The analysis results to be displayed are configured by applying a Loading Filter and choosing the Analysis Method.
- **Material Listing**
  Tabulated quantities of materials.
- **Revision History**
  The revision history that has been recorded in the Project Wiki.

Member Reports

Member Reports are not output directly from the Report toolbar, they are only output as part of the Model Report, (or, by right-clicking on an individual member and clicking 'Report for member').

They are set up in a similar way to the Model Report itself with chapters including:

- **Picture**
  This is the 3D view you obtain by right-clicking on a member and clicking 'Open member view'.
- **Drawing**
  This is the dxf you obtain by right-clicking on a member and clicking 'Generate Detail Drawing'.
- **Loading**
  This is the table you obtain by right-clicking on a member and clicking 'Show Member Loading'.
- **Force and Deflected Shape Diagrams**
  These are the diagrams you obtain by right-clicking on a member and clicking 'Open load analysis view'.

Available Styles

A number of sample reports are available for selection from the **Available Styles** list. These reports serve as templates and can be modified to suit the model in question.

If there isn't already a report that can be customized to meet your needs you can add further reports to the list.

If you start a new project, the same default reports will be available, but they are reset to the default report structures. It is not currently possible to save customized reports from the Report Contents dialog in order to apply them to other projects.

Active Model Report

The active model report is simply the default report shown on the drop list on the Report toolbar - unless a report view is already active, (in which case the view's report is shown n the drop list instead.)

The active model report can be specified by clicking the '>> Active' button in the Report Contents dialog for Model Reports.

Active Member Report

When you right-click a member in a 2D or 3D View and choose Report for Member; the member report that has been specified as being active is the one that is generated.

The active member report can be specified by clicking the '>> Active' button in the Report Contents dialog for Member Reports.

Active and Inactive Chapters

When the Report Contents dialog is displayed each chapter in the current report structure can be marked as either active, or inactive. Only those chapters that are marked as active get included when the report is generated.

Filters

Filters can be employed to limit the amount of output produced, different filters being applicable for different data types:

- **Model Filters** - By default output is produced for the entire structure, but if required it can be filtered for selected levels, frames, or planes. For certain data types (e.g. Foundation Reactions) output can be filtered for selected beams, columns or walls. If user defined attributes (UDAs) have ben applied, these can also be used to filter the output.
- **Loading Filters** - By default output is produced for all loadcases and all combinations, but if required it can be filtered for selected loadcases and combinations.

Creating Reports

How do I configure a Model Report?

1. Click **Report** > **Model Report...** ( )

2. Choose a report as follows:

- To choose an existing report, use the Available Styles list.
- To add a new report, click Add, then type a name for the new report in the **Active Style** box.

3. Review the Report Structure and modify it if required.
(See: How do I modify the Report Structure?)

4. To limit the amount of output to selected levels, frames, planes, or sub structures only; apply a model filter.
(See: How do I apply a filter?)

5. For 'Loadcases' and 'Combinations' sub-chapters you can further limit the output by applying a loading filter.
(See: How do I apply a filter?)

6. If you have created specific view configurations of the model and want to include these in the report you must include a separate View chapter for each view configuration required.

7. If the Report Structure includes any member chapters (Beams, Columns, Walls etc.):

- for each chapter ensure the appropriate Member Report style is selected.
(See: How do I select the Member Report style to use in the Model Report?)
- ensure the selected Member Report style for each chapter is configured as required.
(See: How do I configure a Member Report?)

8. Click **OK**

9. Click **Show Report** ( )

The chosen report should now be displayed in a new window from where it can be reviewed.

If it contains loading analysis views of individual members (e.g. force diagrams or pictures) these are displayed for each member type according to the Member Report style that was selected above.

When the model report is displaying the information as required it can then printed. (See: How do I display a Model Report?)

If you have included a **Drawing** *in the Report Structure, you should specify appropriate settings for it:*

- Right-click the Drawing in the Report Structure

- Click **Settings...**

- Select the Drawing Type and Style from the drop lists

- Specify the Scale and Text Block Spacing

How do I display a Model Report?

1. Select the report to display from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )

How do I configure a Member Report?

1. Click **Report** > **Member Report...** ( )
2. Choose the member type required from the drop list.
3. Choose a report style for the member type.
   - To choose an existing report, use the Available Styles list.
   - To add a new report style, click Add, then type a name for the new report in the **Active Style** box.
4. Review the Report Structure for this style and modify it if required. (See: How do I modify the Report Structure?)
5. To control the level of output, apply settings and loading filters where applicable as required. (See: How do I apply a filter?)
6. Click **OK**

If you have included a **Drawing** *in the Report Structure, you should specify appropriate settings for it:*

- Right-click the Drawing in the Report Structure

- Click **Settings...**

- Select the Drawing Type and Style from the drop lists

- Specify the Scale and Text Block Spacing

How do I display an individual Member Report?

Firstly ensure you have configured the member report style and made it active.
(See: How do I configure a Member Report?) and
How do I select the Member Report style to use in an individual Member Report?)

Then:

1. Hover the cursor over the member until its outline is highlighted, then right-click.
2. From the context menu select **Report for member**

The report for the chosen member is displayed in a report view.

How do I select the Member Report style to use in the Model Report?

Each member type can have a number of 'member report styles' available, (each being configured to produce a different level of output). You therefore need to specify which one to use if the member type is included in the Model Report.

With the required Model Report selected in the Report Contents dialog; to select the Member Report Style to be used in the Model Report:

1. Locate the specific member chapter, (Beams, Columns etc.) in the report structure.
2. Right-click the chapter and select the required member report style from the 'Style' option.

How do I select the Member Report style to use in an individual Member Report?

To control the report that gets generated when you right-click a member in a 2D or 3D View and choose 'Report for Member', the active member report needs to be set correctly.

1. Click **Report** > **Member Report...** ( )
2. Choose the member type required from the drop list.
3. In the list of available styles, one will be marked '(active)'.

- if this is the style you want to use simply click OK to close the dialog,
- or, choose a different style and click >> Active, then click OK.

How do I modify the Report Structure?

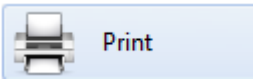
Having selected a report from the **Available Styles** list you can modify its structure as follows:

1. Drag the chapters and options to be included from the left to right. To permanently remove unwanted chapters and options drag them back from right to left.
2. Re-arrange the report order as required by dragging selected chapters up or down.
3. To exclude a specific chapter from the report, whilst maintaining the report structure; make it inactive by right-clicking on the chapter and clearing **Active**. (It can subsequently be re-introduced if required by rechecking **Active**.)
4. Right-click each chapter to specify any filtering requirements. (See: How do I apply a filter?)
5. Click **OK**

To aid the report structuring process, an option is provided to display the structure as either a Flat, or a Hierarchical layout.

How do I print a Report?

The report displayed in the active Report View is printed as follows:

1. Click the File menu
2. Click 

Filtering Reports

What are the different types of filter?

There are two main categories of filter that can be applied to chapters in the Report Structure, each chapter having different filters available according to its content.

Model Filters are available to enable selective output based on:

- Selected levels
- Selected frames
- Selected planes
- Selected groups
- Selected members
- Selected trusses
- Selected portal frames
- Selected levels
- Selected sub structures
- Selected UDAs

Loading Filters are available to enable selective output based on:

- Selected loadcases
- Selected combinations
- Selected envelopes

How do I create and edit a filter?

Filters can be created and edited either directly from the **Report** toolbar as described below; or at the same time as they are applied to a chapter in the Report Structure, by right-clicking on the chapter and then clicking **Edit\New...** from the relevant filter menu.

1. Click the type of filter required from the **Filters** group on the **Report** toolbar.

A dialog is displayed listing any existing filters of the chosen type.

2. Click Add, then enter a name for the new filter.

3. In the **Selected items** area of the dialog, select the check boxes as required to define the content of the filter.

4. Click **OK**

If a Members filter is created, it can subsequently be reviewed and edited graphically via the **Report Filter** *button on the* **Review** *toolbar.*

5. Once a filter has been created, it then has to be applied to a specific report.

How do I apply a filter?

Filters are applied to reports as follows:

1. Click **Report** > **Model Report...** , or **Member Report...** to open the appropriate **Report Contents** dialog.

2. Choose the report to be filtered.

In the **Report Structure** *any chapters that can be filtered will always show the currently applied filter in blue text.*

3. Right-click a chapter and choose the required filter type and filter name from the right-click menu.

If the available filters do not meet your requirements, choose 'Edit/New...' from the right-click menu, then:

1. Click Add

2. Enter a name to describe the filter.
3. Select the check boxes as required under **Selected Items** to define the filter requirements.
4. Click **OK**

If a filter type is not applicable for the selected chapter it will be unavailable.

Members filters specifically, can subsequently be reviewed and edited graphically via the **Report Filter** *button on the* **Review** *toolbar.*

Formatting Reports

Tekla Structural Designer allows you to customize the appearance of the reports in a number of ways.

Report Settings

These settings are used to control (for the settings set selected at the top of the page) the appearance of the reports. The various options allow you to configure paragraph styles, page margins and numbering, tables, headers/footers and other options.

Unlike other "settings set" settings, changes to report settings are instantly applied to the current work-session when you click OK to close the Settings dialog (provided the edits were made to the active set).

How do I apply report settings?

1. Click **Report** > **Settings** ( )
2. Use the **Report** sub-pages to format the report.

   The various options allow you to set:

   - **Styles** - customize the paragraph styles to be applied to the different areas of the report.
   - **Page Options** - specify page margins and numbering.
   - **Table Options** - control the appearance of tables.
   - **Document Options** - control display of header/footer and other options.
   - **Picture Fonts** - control the appearance of fonts used in pictures and force diagrams.

3. Having configured the report format as required, click **OK** to apply it to the current project.

The specified report settings are applied to the current project and continue to apply in future work-sessions for all models.

Report Settings-Styles

Report Styles

The list displays the styles that are used in reports.

The selected style's appearance can modified by adjusting the font, color and other parameters as required

Report Settings-Page Options

Page margins, numbering and frame

Adjust as required.

Report Settings-Table Options

Table style, border and width

Adjust as required.

Report Settings-Document Options

Header/footer

Show document header

Select to include the header at the top of each report page.

Show border in document header

Select to draw a border around each field cell in the header.

Show document footer

Select to include the footer at the bottom of each report page.

Show border in document footer

Select to draw a border around each cell in the footer.

Show document field description in line with the value

Select to display the field descriptions in line with the field values in each cell, or leave cleared to display the field values on a new line.

Underline document field cell

Select to underline each field cell value.

Image width

Percentage of page width

Specify the width for all images that are included in reports.

Paragraphs

Spacing

Specify the paragraph spacing in reports.

Page breaks

Start each item on a new page

Select to start each chapter of the report on a new page.

Start each member report on a new page

Select to start each member on a new page.

Report Settings-Picture Fonts

If you have included any pictures in your reports, you can use these settings to control the fonts that are used within them.

Headers and Footers

Default page header and footer layouts are provided, but you can also create and save your own layouts as required.

How do I enter company details (logo, address etc.) in the header?

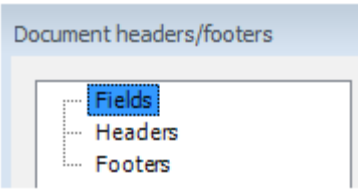
1. Click **Show Report** ( ) on the **Report** toolbar

2. Click **Edit Header** ( )

   Assuming you have not already created your own header layouts, there will only be a 'Default Header' layout available, (you can rename this if you wish, by over-typing the displayed name).

   Note that the default layout already contains a field for the address.

   For each of the following operations, ensure 'Fields' remains highlighted at the top left of the dialog.



First select the Fields to be included:

1. Highlight 'Address' in the Available fields list, then click the **Edit...** button (the one below the Available fields list).

   The 'Address' field comprises up to 11 individual fields, only those listed in the 'Included fields' will be shown in the header.

2. In the Field properties area of the dialog, move fields between 'Included' and 'Available' as required.

Next define the text to be displayed:

1. In the left hand Available fields list, highlight the first text field (e.g. Company Name) that appears as an 'Included field'.
2. In the Field properties area, type in the text that you want to appear for this field.
3. Highlight the next text field (e.g. Address Line 1) that appears as an 'Included field'.
4. Type in the text that you want to appear for this field.
5. Repeat until all the text fields included in the address have been defined.

Finally select the logo:

By default there is only one image field (Company Logo) available.

1. In the Available fields list, highlight 'Company Logo'.
2. In the Field properties area, click the **[...]** button to browse to the image you want to use as the logo.

Large logos are automatically resized to fit within the column width in which they are placed. However, it should be noted that using physically smaller images saves memory and reduces the time taken to export reports. For the same reason it is recommended that logos are created in PNG format rather than BMP format.

If you want to include additional images in the header/footer simply click 'Add Image Field' to create extra place holders for each one.

3. Click 'Headers' at the top left of the dialog to re-display the layout to ensure the 'Company Logo' field is assigned to the appropriate cell in the header.
4. If further images are required return to the 'Fields' pane, click Add Image Field and repeat the above process.

How do I enter project specific details (job ref, structure etc.) in the header?

1. Click **Project Wiki** ( ) on the **Home** toolbar
2. On the Project Summary page of the Project Wiki dialog, type the text that you want to appear for each field that appears in the header.

How do I create a new header layout?

To customize report headers and footers you must first display a report on screen, then:

1. Click **Edit Header** ( )

   The **Document headers/footers** dialog is displayed allowing you to create new headers and footers, or modify existing ones.

2. Click **New**

   A default header is displayed in the Current layout area.

3. In the 'Name' field, (optionally) give the new layout a more descriptive name.

The default header can be reconfigured to suit your requirements as follows:

To remove an existing field to create an empty cell:
Move the cursor over the Current Layout area and the Field under the cursor will be highlighted in red, click the field to remove it and create an empty cell.

To place a field into an empty cell (or replace an existing field):
Drag the field required from the 'Available fields' list into the cell.

To join multiple cells into a single merged cell:
Left click and drag over the cells to be joined.

(If you subsequently require to unjoin previously joined cells you can do so from the right-click menu.)

To modify the field alignment within a cell:
Right-click the required cell in the layout and set the horizontal and vertical alignment options as required.

To insert or remove rows or columns in the layout, or change the alignment:
Right-click over the layout and select the command required from the context menu.

To change the column widths in the layout:
Click **Edit...** under the Current Layout and adjust the widths as required.

To review the new layout:

Once the changes have been made click **OK** to re-display the report and review their effect.

Exporting Reports

How do I export a report to PDF?

1. Set up your report so that it contains the information you require.

2. Click **PDF** ( )
   (in the **Export** group).

3. Click **OK**.

How do I export a report to Word?

1. Set up your report so that it contains the information you require.

2. Click **Word** ( )
   (in the **Export** group).

3. Click **OK**.

How do I export a report to Excel?

1. Set up your report so that it contains the information you require.

2. Click **Excel** ( )
   (in the **Export** group).

3. Click **OK**.

How do I export a report to Tedds?

1. Set up your report so that it contains the information you require.

2. Click **Tedds** ( )
   (in the **Export** group).

3. Click **OK**.

Example Reports

A number of standard reports are installed by default. Although these are unlikely to match your exact needs, in many cases they can serve as templates which can then be edited to suit your individual requirements.

Solver Model Data Report

By default this report will include a picture of the model and then list the following tables of analysis model input data:

- Nodes
- Node support Details
- Elements
- Element-Members
- Element Properties
- contains to PDF?

How do I show a Solver Model Data Report?

1. Select **Solver Model Data** from the drop list on the **Report** toolbar.

2. Click **Show Report** ( )

3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Building Loading Report

By default this report will include a picture of the model and then list the following tables of loading related input data:

- Action Codes
- Resistance Codes
- Combinations
- Wind Data

How do I show a Building Loading Report?

1. Select **Building Loading** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Building Analysis Checks Report

By default this report will list the following tables:

- Loadcase Summary
- Analysis Drift Results
- Analysis Sway Results

How do I show a Building Analysis Checks Report?

1. Select **Building Analysis Checks** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Building Design Report

By default this report displays the design results for the building at a summary level.

How do I show a Building Design Report?

1. Select **Building Design** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Material Listing Report

By default the material listing report is displayed for all members in the building.

How do I show a Material Listing Report?

1. Select **Material Listing** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Beam End Forces Report

This report is typically generated for steel or timber beams in order to provide connection design forces in one of the following configurations:

- End 1/End 2
- End 1/End 2 + Coincident
- Higher End Only
- Higher End Only + Coincident (initial default)

Choose an **End 1/End 2** configuration if you intend to design the connections at both ends of each beam independently, otherwise choose **Both Ends** to standardise your connection designs and employ the same connection at both ends of the beam where possible.

For each of the above, if you choose **+ Coincident,** the maximum and minimum for each of the 6 forces is reported in a separate row in the table along with the coincident forces in the other directions, (potentially generating many more pages of output).

What data is output in each Beam End Forces Report configuration?

Six columns of forces, (Fx, Fy, Fz, Mx, My, Mz) are output for each configuration as follows:

End 1/End 2

For each end of each steel beam: A single row of data is output for those loadcases/combinations/envelopes in the chosen loading filter:

- Fx, Fy, Fz, Mx, My, Mz,

End 1/End 2 + Coincident

For each end of each steel beam: A separate row is output for each loadcase, combination, or envelope (according to the loading filter applied) for the maximum and the minimum of each of the 6 forces, along with the coincident forces in the other directions.

Potentially up to 12 rows of data could theoretically be output for loadcase, combination, or envelope as follows:

- maximum: Fx, (+ coincident Fy, Fz, Mx, My, Mz)
- minimum: Fx, (+ coincident Fy, Fz, Mx, My, Mz)
- maximum: Fy, (+ coincident Fx, Fz, Mx, My, Mz)
- minimum: Fy, (+ coincident Fx, Fz, Mx, My, Mz)
- maximum: Fz, (+ coincident Fx, Fy, Mx, My, Mz)
- minimum: Fz, (+ coincident Fx, Fy, Mx, My, Mz)
- maximum: Mx, (+ coincident Fx, Fy, Fz, My, Mz)

- minimum: Mx, (+ coincident Fx, Fy, Fz, My, Mz)
- maximum: My, (+ coincident Fx, Fy, Fz, Mx, Mz)
- minimum: My, (+ coincident Fx, Fy, Fz, Mx, Mz)
- maximum: Mz, (+ coincident Fx, Fy, Fz, Mx, My)
- minimum: Mz, (+ coincident Fx, Fy, Fz, Mx, My)

The loadcases/combination/envelope names in which these maximums/minimums occur are also reported.

Higher End Only

For each steel beam: Two rows of data are output for the chosen loading filter:

- max (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz,
- min (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz.

Loadcase/combination/envelope names are not included for this configuration as they could vary for each of the 6 values.

Higher End Only + Coincident (initial default)

For each steel beam: Potentially up to 12 rows of data could theoretically be output for the loading filter applied. A separate row is output for the maximum and the minimum of each of the 6 forces, along with the coincident forces in the other directions.

- max (end 1, end 2): Fx, (+ coincident Fy, Fz, Mx, My, Mz)
- min (end 1, end 2): Fx, (+ coincident Fy, Fz, Mx, My, Mz)
- max (end 1, end 2): Fy, (+ coincident Fx, Fz, Mx, My, Mz)
- min (end 1, end 2): Fy, (+ coincident Fx, Fz, Mx, My, Mz)
- max (end 1, end 2): Fz, (+ coincident Fx, Fy, Mx, My, Mz)
- min (end 1, end 2): Fz, (+ coincident Fx, Fy, Mx, My, Mz)
- max (end 1, end 2): Mx, (+ coincident Fx, Fy, Fz, My, Mz)
- min (end 1, end 2): Mx, (+ coincident Fx, Fy, Fz, My, Mz)
- max (end 1, end 2): My, (+ coincident Fx, Fy, Fz, Mx, Mz)
- min (end 1, end 2): My, (+ coincident Fx, Fy, Fz, Mx, Mz)
- max (end 1, end 2): Mz, (+ coincident Fx, Fy, Fz, Mx, My)
- min (end 1, end 2): Mz, (+ coincident Fx, Fy, Fz, Mx, My)

The loadcases/combination/envelope names in which these maximums/minimums occur are also reported.

How do I change the default Beam End Forces Report configuration?

1. Click **Report > Model Report**
2. Select **Beam End Forces** from the Available Styles list.
3. Expand the **Beam End Forces** group under Chapters and Options.
4. Drag the required configurations in/out of the Report Structure as required.
5. Click OK

How do I show a Beam End Forces Report?

1. Select **Beam End Forces** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Bracing Forces Report

For steel buildings provides the bracing design forces for all loadcases and combinations.

How do I show a Bracing Forces Report?

1. Select **Bracing Forces** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Foundation Reactions Report

This report summarizes the foundation design forces and is available in two configurations:

- Foundation Reactions (initial default)
- Foundation Reactions + Coincident

By choosing the **Foundation Reactions** configuration, 6 forces are output (in the support's local co-ordinate system) for the selected loadcases, combinations, or envelopes.

Source	Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
						F_{vert} [kip]	F_{major} [kip]	F_{minor} [kip]	M_{major} [kip ft]	M_{minor} [kip ft]	M_{tor} [kip ft]
C19	SUP 9/11	0.0000	C19 (30"x30")	0.0000	1 ASD_2-D+L	176.2	0.0	0.0	0.5	0.2	0.0
					2 $ASD_{2.1}$ -D+L+NL $_{Dir1+}$	176.2	0.0	-0.2	0.6	-1.8	0.0
					3 $ASD_{2.2}$ -D+L+NL $_{Dir1-}$	176.2	0.0	0.2	0.4	2.2	0.0
					4 $ASD_{2.3}$ -D+L+NL $_{Dir2+}$	176.2	-0.3	0.0	2.7	0.3	0.0
					5 $ASD_{2.4}$ -D+L+NL $_{Dir2-}$	176.2	0.2	0.0	-1.7	0.1	0.0
					6 ASD_3-D+Lr	139.0	0.0	0.0	0.3	0.2	0.0
					7 ASD_5 -D+0.75L+0.75Lr	166.9	0.0	0.0	0.4	0.2	0.0
					8 $LRFD_2$-D+1.6L	198.5	0.0	0.0	0.6	0.2	0.0
					9 $LRFD_3$ -1.2D+1.6L+0.5Lr	226.3	0.0	0.0	0.6	0.3	0.0
					10 $LRFD_4$ -1.2D+1.6L+S	226.3	0.0	0.0	0.6	0.3	0.0
					11 $LRFD_5$ -1.2D+L+1.6Lr	204.0	0.0	0.0	0.5	0.2	0.0
					12 $LRFD_6$ -1.2D+L+1.6S	204.0	0.0	0.0	0.5	0.2	0.0

If you choose the **Foundation Reactions + Coincident** configuration, the maximum and the minimum value of each of the 6 forces for the selected loading is output along with the coincident forces that exist in the other directions for the loadcase/combination/envelope in which the max/min occurs.

For the same selection of load combinations as above, by switching to the Foundation Reactions + Coincident configuration, only six rows of data are required to output all the maximum values (highlighted in red) and minimum values (highlighted in blue).

Source	Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
						F_{vert} [kip]	F_{major} [kip]	F_{minor} [kip]	M_{major} [kip ft]	M_{minor} [kip ft]	M_{tor} [kip ft]
C19	SUP 9/11	0.0000	C19 (30"x30")	0.0000	2 $ASD_{2.1}$ -D+L+NL $_{Dir1+}$	176.2	0.0	-0.2	0.6	-1.8	0.0
					3 $ASD_{2.2}$ -D+L+NL $_{Dir1-}$	176.2	0.0	0.2	0.4	2.2	0.0
					4 $ASD_{2.3}$ -D+L+NL $_{Dir2+}$	176.2	-0.3	0.0	2.7	0.3	0.0
					5 $ASD_{2.4}$ -D+L+NL $_{Dir2-}$	176.2	0.2	0.0	-1.7	0.1	0.0
					6 ASD_3-D+Lr	139.0	0.0	0.0	0.3	0.2	0.0
					9 $LRFD_3$ -1.2D+1.6L+0.5Lr	226.3	0.0	0.0	0.6	0.3	0.0

Using the 'Foundation Reactions + Coincident' option, the table will never contain more than 12 rows (one each for the maximum and minimum of each of the 6 forces) - irrespective of the number of loadcases or combinations that have been selected.

How do I show a Foundation Reactions Report?

1. Select **Foundation Reactions** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Seismic Design Report

By default this report will list the following tables:

- Seismic Loading Summary
- Analysis Seismic Drift Results

How do I show a Seismic Design Report?

1. Select **Seismic Design** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Member Design Report

By default this report displays the design results for each member at a summary level.

How do I show a Member Design Report?

1. Select **Member Design** from the drop list on the **Report** toolbar.
2. Click **Show Report** ( )
3. If the resulting report is displaying too little, or too much information, you can adjust the content by re-configuring the report.

Setting up the Report Page Header and Footer

Default page header and footer layouts are provided, but you can also create and save your own layouts as required.

Topics in this section

- How do I enter company details (logo, address etc.) in the header?
- How do I enter project specific details (job ref, structure etc.) in the header?
- How do I create a new header layout?

Related video

- Report Headers and Footers

Drawing Guide

Drawings of individual levels and frames are created directly from the Draw toolbar, whilst single member drawings are created for selected members from the right-click menu.

Alternatively drawings can be created in batch via Drawing Management.

Draw toolbar

The **Draw** toolbar contains the following commands:

Button	Description
Edit...	Opens the Model Settings dialog at the Drawings page. See Drawing Settings
Drawing Management...	Opens a dialog for the generation and laying out of multiple drawings on to a single drawing sheet. The dialog can also be used to manage drawing revisions. See Drawing Management
Schedule Management...	Opens a dialog for the generation of schedules on drawing sheets. The dialog can also be used to manage schedule revisions. See Schedule Management
Beam Schedule	Opens a dialog to configure and then create a beam schedule. See Concrete Member Schedules
Column Schedule	Opens a dialog to configure and then create a column schedule. See Concrete Member Schedules
Wall Schedule	Opens a dialog to configure and then create a wall schedule. See Concrete Member Schedules

General Arrangement	Opens a dialog to configure and then create a general arrangement drawing. See Planar Drawings
Beam End Forces	Opens a dialog to configure and then create a beam end forces drawing. See Planar Drawings
Column Splice Loads	Opens a dialog to configure and then create a column splice loads drawing. See Planar Drawings
Foundation Reactions	Opens a dialog to configure and then create a foundation reactions drawing. See Planar Drawings
Loading Plan	Opens a dialog to configure and then create a loading plan drawing. See Planar Drawings
Slab/Mat Detailing	Opens a dialog to configure and then create a slab or mat detail drawing. See Planar Drawings
Foundation Layout	Opens a dialog to configure and then create a foundation layout drawing. See Foundations

Overview of Drawings

In *Tekla Structural Designer* there are five main drawing categories, each containing a number **Drawing Variants** (see table below).

Each variant has a number of specific Drawing Options available to further configure the drawing appearance.

Each variant contains a set number of **Drawing Layers**. These are switched on or off as required. Whether each layer is on or off is controlled by the Layer Configuration that is selected. You can either use the default layer configurations or add your own.

The appearance of each layer (color, linestyle, font etc.) is controlled by the Layer Style. Again, you can either use the default layer styles or add your own.

Drawing Options, Drawing Layers and Layer Styles are all managed via Drawing Settings.

Drawing Variants

Category	Drawing Variant	Description

Planar Drawings	General Arrangement	General Arrangement (GA) of 2D levels and frames.
	Foundation Reactions	A GA that also contains support reactions (to assist foundation design).
	Loading Plan	A GA that also contains applied loads for the selected loadcase.
	Beam End Forces	A GA that also contains forces at the ends of steel beams (to assist steel connection design).
	Column Splice Loads	A Frame GA that also contains the splice loads at splice locations in steel columns (to assist steel connection design).
Member Details	Concrete Beam Detail	Beam reinforcement in elevation and section for each span. A reinforcement quantity table can optionally be included.
	Concrete Column Detail	Column reinforcement in elevation and section. A reinforcement quantity table can optionally be included.
	Concrete Wall Detail	Wall reinforcement in elevation and section. A reinforcement quantity table can optionally be included.
	Non-concrete Beam Detail	Individual details for non-concrete beams.
	Non-concrete Column Detail	Individual details for non-concrete columns.
Concrete Member Schedules	Concrete Beam Schedule	Tabular data created by building, by floor, or by selected beams. The information shown in the schedule is based on the design groups.
	Concrete Column Schedule	A table containing cross sections through each stack for the selected columns. A reinforcement quantity table is optionally included.
	Concrete Wall Schedule	A table containing cross sections through each stack for the selected walls. A reinforcement quantity table is optionally included.

Slabs and Mats	Slab/Mat Layout	Slab panel, patch and punching shear reinforcemei requirements. A quantity table for the reinforcement displayed wi a detailing allowance added is optionally included.
	Punching Check Detail	Punching shear reinforcement details for individual punching check locations.
Foundations	Isolated Foundation Detail	Individual foundation details in plan, with options t show the detail in cross section and a reinforcemei quantities table.
	Foundation Layout	A GA at foundation level showing isolated bases, p layouts, and mats. Isolated foundation details, a reinforcement quantities table and an isolated foundation schedu can optionally be included.

Drawing Settings

How to apply and manage Drawing Settings

To modify drawing settings in the current project

1. Click **Draw** > **Edit...** ( )

   (alternatively you could also click **Home** > **Model Settings** ( ) > **Drawings**)

2. Review and edit the settings as required.

3. If you change any of the settings, click:

- **OK** - to apply the changes directly to the current project, or
- **Save...** - to save the changes back to the active settings set (to act as defaults for future projects), or
- **Cancel...** - to cancel the changes

You can also click:

- **Load...** - to revert to the drawing settings specified in the active settings set.

To modify drawing setting defaults for future projects

1. Click **Home** > **Settings** ( )

2. In the **Settings Sets** page of the dialog select the settings set to be updated.

You can update any settings set simply by selecting it from the droplist, it does not need to be active.

3. In the **Drawings** page of the dialog, review and edit the settings as required.

4. If you change any of the settings, click:

- **OK** - to save the changes to the selected settings set (to act as defaults for future projects when that set is active), or
- **Cancel** - to cancel the changes

Export Preferences

This page is only displayed when the drawing settings are accessed from the Settings dialog on the Home toolbar.

Drawing Variant

Use the drop list to select the drawing variant to be configured.

Minimum Text Block Spacing

The value entered here is used to adjust the distance between independent drawing blocks.

Caution should be applied when adjusting this value - if it is set too large, text labels can be displaced away from the objects to which they refer.

Scale

Controls the scale of the drawing.

How do I add a new drawing scale?

If the drawing scale you want to use is not listed you can add new scales as follows:

1. Click **Home** > **Settings**

2. On the **Drawings** page select the settings set to be edited from the drop list.

3. Enter the scale required then click **Add**

   The new scale is shown in the **Available Scales** list and will apply to all projects that use this settings set.

5. Click **OK**

Layer Configuration

A layer configuration controls which layers are displayed when the drawing is created.

Several default layer configurations have been pre-loaded in the **Settings Sets** - these can be modified if required from the Settings dialog. Alternatively if you only want to change layer configurations in the current project this can be done by clicking **Draw > Edit**

When editing layer configurations via either of the above methods, you should first select the main drawing category on the left side of the dialog, the following fields are then displayed:

Drawing Variant
Use the drop list to select the drawing variant to be configured.

Available Configurations
The available configurations that are displayed depend on the drawing variant selected. You can **Add** a new configuration and set up the layers within it accordingly. Alternatively you can **Add Copy...** in order to copy an existing configuration and modify it to suit your particular requirements.

Active Configuration - Name
Having selected one of the available configurations, you can edit its name if required.

Active Configuration - Layers
Having selected one of the available configurations, you can then proceed to specify the layers that are to be displayed for it when the drawing is created.

If you access and edit layer configurations in the current project by clicking Draw > Edit, you should consider saving the changes to a Settings Set so that they can be applied to other projects also.

How do I add, modify, copy or remove a layer configuration?

1. Click **Draw** > **Edit...** ( )
   This opens the Model Settings at the **Drawings** page.
2. Expand the **Layer Configurations** page and select a drawing category sub-page (Planar Drawings, Member Details, or Member Schedules).
3. From the dropdown list choose the **Drawing Variant** required, then:

To add a new configuration:

1. Click **Add...**
2. Enter a name for the new layer configuration.
3. Select/clear the layers as required.
4. Click **OK**

To modify an existing configuration:

1. Select the available configuration to be modified.
2. Select/clear the layers as required.
3. Click **OK**

To copy an existing configuration:

1. Select the available configuration to copy from.
2. Click **Copy**
3. Enter a name for the new layer configuration.
4. Select/clear the layers as required.
5. Click **OK**

To remove an existing configuration:

1. Select the available configuration to be removed.
2. Click **Remove**
3. Click **OK**

Layer Style

In order to control how a drawing is displayed (line types, fonts, colors etc.) you will need to apply a drawing **Style**.

Several default layer styles have been pre-loaded in the **Settings Sets** - these can be modified if required from the Settings dialog. Alternatively if you only want to change layer styles in the current project this can be done by clicking **Draw > Edit**

When editing layer styles via either of the above methods, you should first select the main drawing category on the left side of the dialog, the following fields are then displayed:

Drawing Variant

Use the drop list to select the drawing variant to be configured.

Available Styles

The available styles that are displayed depend on the drawing variant selected. You can **Add** a new style and set up the layers within it accordingly. Alternatively you can **Add Copy...** in order to copy an existing style and modify it to suit your particular requirements.

Active Style - Name

Having selected one of the available styles, you can edit its name if required.

Active Style - Layers

Having selected one of the available styles, you can then proceed to specify the appearance of layers within it when the drawing is created.

How do I add, modify, copy or remove a drawing style?

1. Click **Draw** > **Edit...** ( )
   This opens the Model Settings at the **Drawings** page.
2. Expand the **Layer Styles** page and select a drawing category sub-page (Planar Drawings, Member Details, or Member Schedules).
3. From the dropdown list choose the **Drawing Variant** required, then:

To add a new style:

1. Click **Add...**
2. Enter a name for the new drawing style.
3. Configure the colors, line types, fonts etc. for each of layers as required.
4. Click **OK**

To modify an existing style:

1. Select the available style to be modified.
2. Modify the colors, line types, fonts etc. as required.
3. Click **OK**

To copy an existing style:

1. Select the available style to copy from.
2. Click **Copy**
3. Enter a name for the new drawing style.
4. Modify the colors, line types, fonts etc. as required.
5. Click **OK**

To remove an existing style:

1. Select the available style to be removed.
2. Click **Remove**
3. Click **OK**

Drawing Options

You must expand the Options page and select the sub-page for an individual drawing variant in order to see the applicable options for that particular variant.

Options - Planar Drawings

After selecting the appropriate Planar Drawings sub-page (General Arrangement, Foundation Reactions, Loading Plan, Slab/Mat Detail, or Beam End Forces), use the menu tabs to configure the options as required.

General tab

Hatching

Show columns and walls above the level as hatched

Select to hatch columns and walls that continue above the current level.

Show transfer column and walls as cross hatched

Select to cross hatch transfer columns and walls.

Reaction Values (Foundation Reactions and Beam End Forces Variants only)

Current Loading Values

values are displayed for the loadcase or combination currently selected in the Loading drop list.

Max/Min Values by Loadcase/Combination

max and min values are displayed for each loadcase/combination.

Max Absolute Value by Loadcase/Combination

the max absolute value is displayed for each loadcase/combination.

Force & Moment Values (Foundation Reactions and Beam End Forces Variants only)

Factor by

for the current drawing variant, the values displayed are factored by the amount specified.

A factor can be applied to the forces and moments that are displayed if required.

Moment Values, Shear Force Values and Axial Force Values (Foundation Reactions and Beam End Forces Variants only)

None

values are not displayed.

Only greater than

moments/forces are only displayed if the absolute value is greater than the value specified.

All
values are always displayed, (even if zero).

Enveloped Reaction Values (Beam End Forces Variant only)

Include coincident forces
when a beam end forces drawing is created for a loading envelope and this option is selected, further options are presented allowing you to choose the max/min values and coincident forces to display.

Beams tab

Grouped Beam Labelling

These two options only apply when the beams have been designed using groups.

Use detail group name
Select to use the detail group name, or clear to use the design group name in the label.

Include the beam name
Select to include the beam name in the label for grouped beams.

Beam Labelling Position

Above, Inside, Below
Sets the location of the beam label in relation to the beam.

Beam Attributes

Show beam size in parentheses
check to place brackets around the beam size in the label.

Camber

Append camber to section
displays the amount of camber specified for steel beams.

Camber prefix
specifies the camber prefix used in the label.

Composite Properties

Stud separator
for composite beams the number of studs are displayed inside the chosen separator.

Columns tab

Grouped Column Labelling

These two options only apply when the columns have been designed using groups.

Use detail group name
Select to use the detail group name, or clear to use the design group name in the label.

Include the column name
Select to include the column name in the label for grouped columns.

Column Labelling Position

The column size is either included at the right of the label, or on the line below the label.

Column Attributes

Show column size in parentheses
check to place brackets around the column size in the label.

Steel Columns

2x scale for steel columns
Select to draw steel columns at double the drawing scale.

Walls tab

Wall Labelling Position

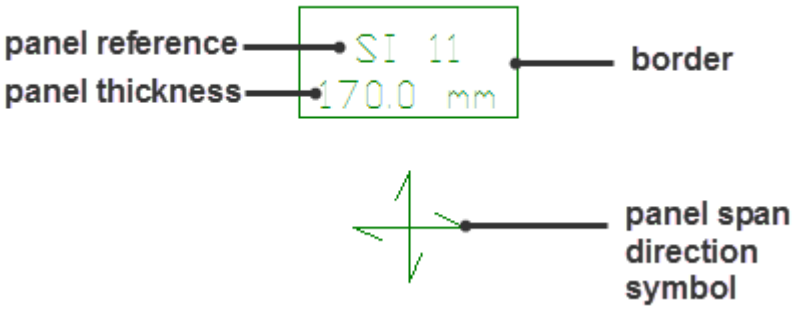
Above, Inside, Below
Sets the location of the wall label in relation to the wall.

Wall Attributes

Show wall size in parentheses
check to place brackets around the wall size in the label.

Slabs and Mats tab

Panel Labeling



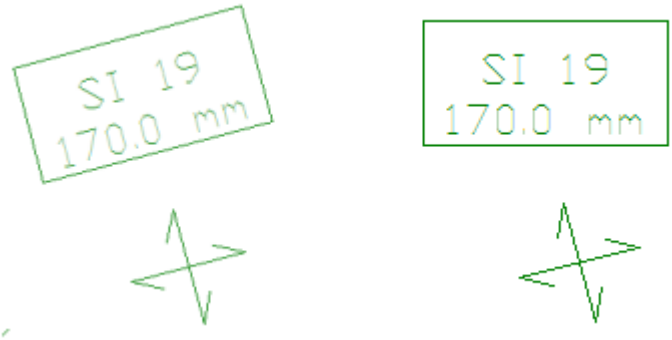
Include panel reference
Select to include the slab panel reference in the label.

Include panel thickness
Select to include the slab panel thickness in the label.

Include surface offset (if non zero)
Select to include any surface offset that has been applied to the slab panel in the label.

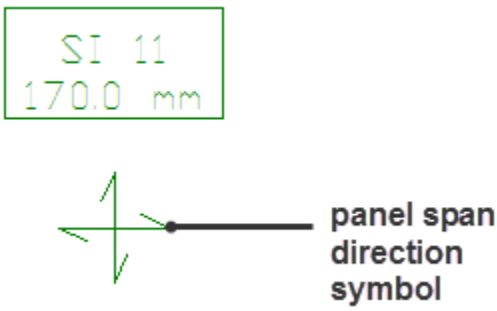
Include border around label
Select this check box to add a border around the label.

Align label to panel reinforcement
Select this check box to align the label to the slab panel span direction.
- When selected the label is displayed as below left
- When cleared the label is displayed as below left



Slab/Mat Geometry

Include panel span direction symbol
Select to include the symbol



Loads tab

(Loading Plan Variant only)

Display size

Width of line/UDL/VDL loads on plan
Each of these load types is drawn as a hatched rectangle of fixed width when drawn in plan - this setting controls the width.

Height of line/UDL/VDL loads on elevation

Each of these load types is drawn as a hatched rectangle of fixed height when drawn in elevation - this setting controls the height.

Point load marker size

This setting controls the drawn size of point loads.

Options

Show dimensions for panel loads

Select to include the dimensions of panel loads.

Include dimensions to reference points

Select to include the dimensions from panel loads to the reference points that were used to set them out.

Show dimensions for member loads

Select to include the dimensions of member loads.

Options - Member Details

These are used to further control the appearance of the different drawing categories.

Beam Detail - Content

These settings are used to control the **content** of the beam detail drawings.

Grouped Beams tab

Show grouped beam number

Provided the beams in the model have been arranged into detailing groups - checking this option causes the Detailing Group to be used as the member label instead of the beam reference.

If detailing groups have not been used, this option has no effect - the beam reference is always used as the member label.

Levels tab

Show span levels

Select to show the span levels on the elevation.

Cross sections tab

Spans

Choose which cross sections to display from the drop list:

- None
- First Span
  (i.e. for **multi-span beams**, cross-sections are not shown for 2nd and subsequent spans)
- All Spans

Positions

If cross sections are displayed, choose where they are to be positioned from the drop list:

- Span Only
- Support
- Support and Span

Bar annotation

Choose the cross section annotation from the drop list:

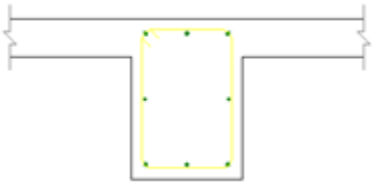
- None
- Standard
- IStructE

Display bar marks

For Standard and IStructE annotation you have the option to display bar marks in the cross section labels.

Show slab lines in section

Select to display slab lines in sections, as shown below:



When the option is cleared the slab lines are not displayed:

Bar Labels tab

Show bar marks in elevation

Select to include bar marks in the bar labels on the elevation. They will also be displayed on the cross-sections provided the **Cross-sections** > **Display bar marks** option is also checked.

If this option is cleared, bar marks are not displayed on cross-sections - irrespective of the **Cross-sections** > **Display bar marks** option.

Show steel bar layer information

Select to show steel bar layer information, (**B1**, **B2**, **T1**, **T2** etc.)

Dimensions tab

Laps

Where laps exist these are either dimensioned, not dimensioned, or the dimension is replaced by a standard label (**TL**) according to the drop list selection.

Anchorage lengths

Where anchorage lengths are required these are either dimensioned, not dimensioned, or the dimension is replaced by a standard label (**AL**) according to the drop list selection.

Axes

Axes are either not shown, shown above the beam with dimensions, or shown below the beam with dimensions.

Additional bottom span bar positioning dimensions

When optional 2nd span bars have been employed, these can be dimensioned from the face of the support by checking this option.

Support region length

Where different stirrup regions has been employed along a beam span, the length of the support regions is dimensioned on the elevation by checking this option.

First and last stirrups

Select to add dimensions from the face of the supports to the first and last stirrups.

Support columns and clear spans

Select to add dimensions showing the width of each supporting column and the clear beam span between supports.

Beam section

Select to dimension the beam depth and width on the cross section.

Slabs in beam sections

Select to dimension the slab depth on the cross section.

Quantities tab

Show reinforcement quantities table

select this check box to include reinforcement quantity tables on the drawings.

Beam Detail - Style

These settings are used to control the **style** of the beam detail drawings.

Beam Labels tab

Print beam labels below the detail

Select to show the beam label centrally below each span. When cleared the label is positioned immediately above each span.

Underline beam labels

Select to underline the beam label on the elevation.

Label every span of multi-span beams

Select to repeat the beam label below the detail for every beam span.

Cross sections tab

Section label style

Choose the label naming style to be applied to the cross sections from the drop list.

Restart section labels in each beam line

When multiple beam lines are displayed on the same drawing sheet, select this check box to restart the section labels for each line.

Add beam name label as prefix to section labels

Select to prefix each section label with the beam name.

Longitudinal Bars tab

Draw bar groups in same elevated layer at different levels

Select to draw the bars displaced vertically (although they are in the same elevated layer), to enhance the display.

Display elevated bobs at same position as shifted

Select to draw bobs displaced vertically (when they are bent at the same position), to enhance the display.

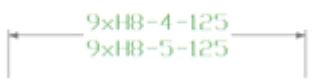
Display only a single side bar in detail

If multiple side bars are required in each face, when this option is selected only a single side bar is drawn full length. When cleared, all side bars are drawn full length.

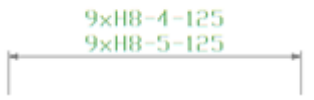
Stirrups/Links tab

Draw stirrup labels in line

Select to show stirrup labels in line on the elevation:



When cleared the labels are drawn above the line.



Print stirrup labels inside beam

Select to show the stirrup labels inside the beam; clear to show below the beam.

Stirrup label distance from bottom edge

When stirrup labels are positioned inside the beam this setting is used to control their vertical position.

Dimensions tab

Draw lap dimensions outside the detail
Select to draw lap dimensions outside the detail.

Lap and anchorage rounding increment
This option is used to control the rounding increment of lap and anchorage dimensions.

Column Detail - Content

These settings are used to control the **content** of the column detail drawings.

Grouped Columns tab

Show grouped column number
Select to display column groups

Levels tab

Show levels
Select to label the construction levels

Cross sections tab

Show sections
Select to show cross sections through each stack

Bar annotation
Choose the cross section annotation from the drop list (None, Standard, IStructE).

Display bar marks
For Standard and IStructE annotation you have the option to display bar marks in the cross section labels

Dimensions tab

Laps
Where laps exist these are either dimensioned, not dimensioned, or given a standard label according to the drop list selection

Support region length
Select to dimension the support regions on the elevation

Levels
Select to add dimensions between levels

Grid line offsets
Select to add dimensions from the grid to the column face on the elevation

Connecting elements and clear heights

Select to add dimensions connecting elements and clear heights

Column section

Select to add column dimensions on the section

Column Detail - Style

These settings are used to control the **style** of the column detail drawings.

Column Labels tab

Underline column labels

Select to underline the column label on the elevation

Cross sections tab

Section label style

Choose the label style to be applied to the cross sections from the drop list

Restart section labels in each column line

When multiple columns are displayed on the same drawing sheet, select this check box to restart the section labels for each column

Add column name label as prefix to section labels

Select to as a prefix each section label with the column name

Longitudinal Bars tab

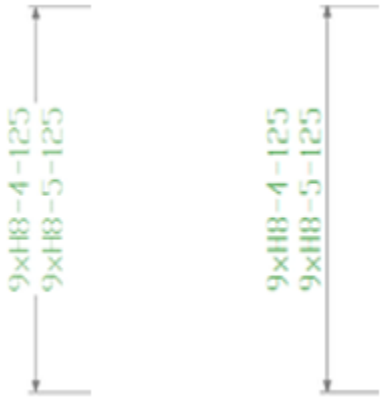
Show hidden bar ends

Select to show hidden bar ends

Ties/Links tab

Draw tie labels in line

Select to show tie labels in line on the elevation, (as below left), or leave cleared to display above the line (as below right):



Dimensions tab

Lap dimension rounding increment
This option is used to control the rounding increment of lap dimensions

Wall Detail - Content

These settings are used to control the **content** of the wall detail drawings.

Levels tab

Show levels
Select to label the construction levels

Cross sections tab

Show sections
Select to show cross sections through each stack

Bar annotation
Choose the cross section annotation from the droplist (None, Standard, IStructE).

Display bar marks
For Standard and IStructE annotation you have the option to display bar marks in the cross section labels

Dimensions tab

Laps
Where laps exist these are either dimensioned, not dimensioned, or given a standard label according to the droplist selection

Support region length
Select to dimension the support regions on the elevation

Levels
Select to add dimensions between levels

Grid line offsets
Select to add dimensions from the grid to the column face on the elevation

Connecting elements and clear heights
Select to add dimensions connecting elements and clear heights

Wall section
Select to add wall dimensions on the section

Wall Detail - Style

These settings are used to control the **style** of the wall detail drawings.

Wall Labels tab

Underline wall labels

Select to underline the wall label on the elevation

Cross sections tab

Section label style

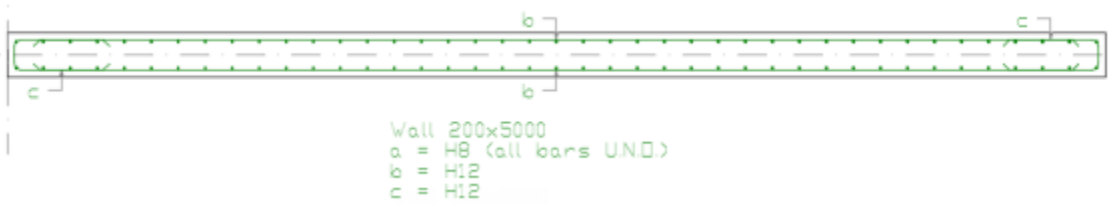
Choose the label style to be applied to the cross sections from the drop list.

Add wall name label as prefix to section labels

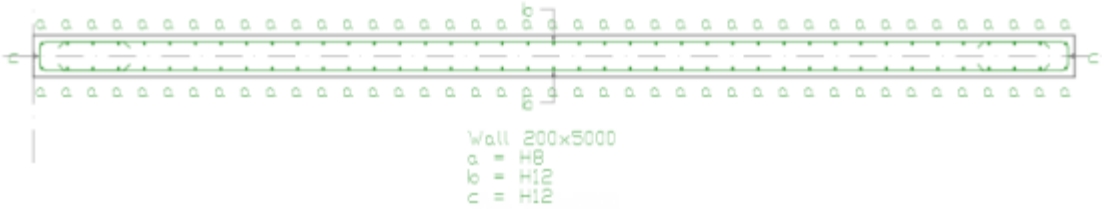
Select to as a prefix each section label with the wall name.

Don’t label typical panel bars

When selected the typical bars are not labelled in the cross section:



When cleared every bar is labelled.



Horizontal & Vertical Bars tab

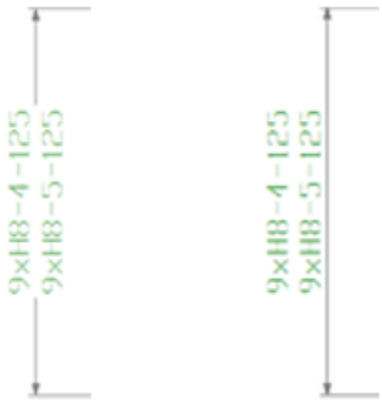
Show hidden bar ends

Select to show hidden bar ends

Horizontal Bars and Ties/Links tab

Draw tie labels in line

Select to show tie labels in line on the elevation, (as below left), or leave cleared to display above the line (as below right):



Dimensions tab

Lap dimension rounding increment
This option is used to control the rounding increment of lap dimensions

Options - Member Schedules

Beam Schedule Options

These settings are used to control the appearance of the concrete beam schedule.

General tab

Texts

Item
Select an item in the list to see the text label that will applied to it in the schedule.

Text
If required you can edit the text to be displayed in the schedule for the selected item.

Columns

Size Column Format
Choose whether to display the width of the beam or its height first in the size column.

Use single column for size
Check to display both the width and height in a single column.

Use single column for bottom bars
Check to display the bottom bars in a single column.

Omit top middle bars column
Check to omit the top middle bars from the schedule.

Use single column for stirrups
Check to display the stirrups in a single column.

Show only design group name
Check to display only the design group name in the mark column.

Bar Key tab

Reference
Every reference that can potentially appear in the bar bending details table is listed. Hover the cursor over a reference to see the bar and its associated note.

Use Custom Name
Select the check box against a reference in order to apply a custom name.

Custom Name
any text entered here will replace the original reference in the bar bending details table.

Column Schedule Options

These settings are used to control the content of the concrete column schedules.

General

Show grouped column number
Select to show column groups

Include starter bars
Select to include starter bars

Show reinforcement quantities table
Select to include the quantities table

Cross sections

Bar annotation
Choose the cross section annotation from the drop list (Standard, IStructE).

Display bar marks
Select to display bar marks in the cross section labels.

Show outline of stack below
Select to show the outline of the stack below.

Dimension column section
Select to add column dimensions on the section.

Dimension levels
Select to add dimensions between levels.

Wall Schedule Options

These settings are used to control the content of the concrete wall schedules.

General

Include starter bars
Select to include starter bars

Show reinforcement quantities table
Select to include the quantities table

Cross sections

Bar annotation
Choose the cross section annotation from the drop list (Standard, IStructE).

Display bar marks
Select to display bar marks in the cross section labels.

Show outline of panel below
Select to show the outline of the panel below.

Dimension wall section
Select to add wall dimensions on the section.

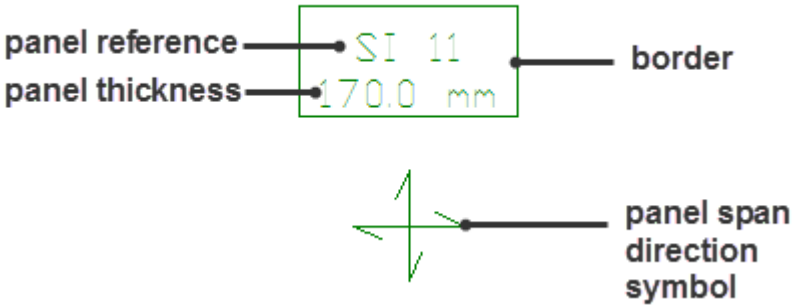
Dimension levels
Select to add dimensions between levels.

Options - Slabs and Mats

Layout

Slabs and Mats tab

Panel Labeling



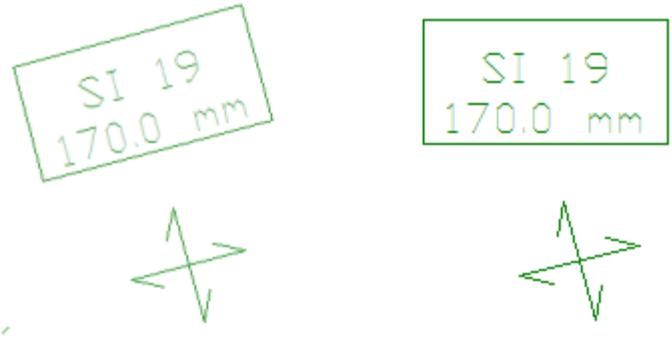
Include panel reference
Select to include the slab panel reference in the label.

Include panel thickness
Select to include the slab panel thickness in the label.

Include surface offset (if non zero)
Select to include any surface offset that has been applied to the slab panel in the label.

Include border around label
Check this to add a border around the label.

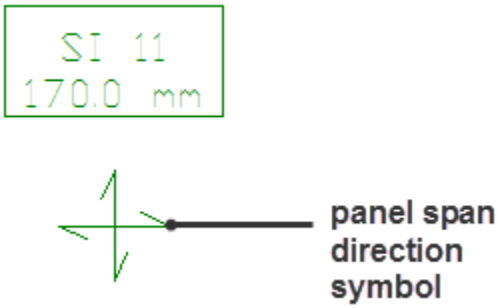
Align label to panel reinforcement
Check this to align the label to the slab panel span direction.
- When checked the label is displayed as below left
- When cleared the label is displayed as below left



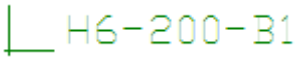
Slab/Mat Geometry

Include panel span direction symbol

Select to include the symbol



Reinforcement Display



Extend loose bar panel reinforcement lines across full panel

Leave this option cleared to display the loose bar reinforcement as above, otherwise the bars are drawn across the entire panel.

Reinforcement Labelling

Always show main bar layer for rectangular mesh

If the mesh is not a square mesh it is normal practice to put the main bars in the outer layer, (no text is required on the drawing in this situation). However, when this is not the case this is indicated on the drawing by adding B2 (if bottom mesh) or T2 (if top mesh) aligned to the main bar direction.

Anchorage rounding increment

Specifies the rounding value applied to the anchorage length.

If a square mesh (i.e. if it does have the same size and spacing of bars in both directions) is applied then a square mesh symbol is used which indicatively shows bars equally spaced in both directions.
If the mesh is not a square mesh (i.e. if it does not have the same size and spacing of bars in both directions) then a rectangular mesh symbol is used which indicatively shows bars in both directions but with closer spacing for the more closely spaced bars in the mesh.

Patches tab

General

Show patches with no reinforcement

If a patch has no reinforcement specified you can choose not to show it on the drawing by clearing this option.

Anchorage Length

You are given the option to either draw the bars with full anchorage lengths, or to draw curtailed (indicative) anchorage lengths instead.

Punching Shear tab

Show punching reinforcement details

Select to include a detail to the side of the layout showing the punching shear reinforcement provided.

Don't show area of steel requirement where rails have been designed

When checked: the area of steel requirement is shown for punching check items where rails are not provided (e.g. walls and column drops), but is not shown elsewhere.

When cleared: the area of steel requirement is shown for all punching check items.

Hide punching reinforcement on the main layout

Select to hide the punching reinforcement on the main layout. It will still be shown in the detail (provided that the above 'Show punching reinforcement details' is checked).

Punching Check Detail - Content

These settings are used to control the **content** of the beam detail drawings.

Dimensions tab

Show column to first stud spacing

Select to add a dimension from the column face to the first stud.

Show stud spacing

Select to add dimensions showing the stud spacings along the rail.

Show rail spacing
Select to add dimensions showing the rail spacings.

Include schematic showing stud dimensions
Select to add a schematic showing the stud width and height above the rail.

Quantities tab

Show reinforcement quantities table
Select to include reinforcement quantity tables on the drawings.

Punching Check Detail - Style

These settings are used to control the **style** of the beam detail drawings.

General tab

Show column as hatched
Select to hatch the column.

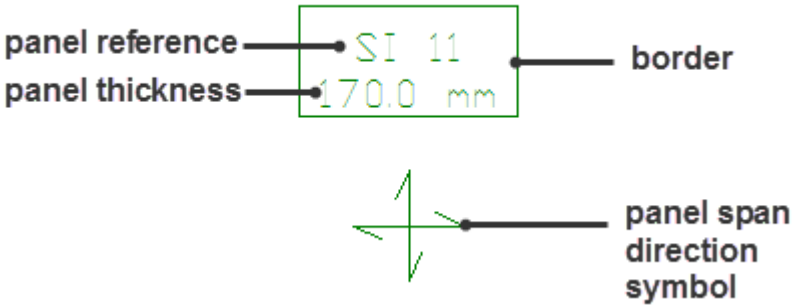
Underline punching check label
Select to underline the punching check label on the detail.

Options - Foundations

Foundation Layout

Slabs and Mats tab

Panel Labeling



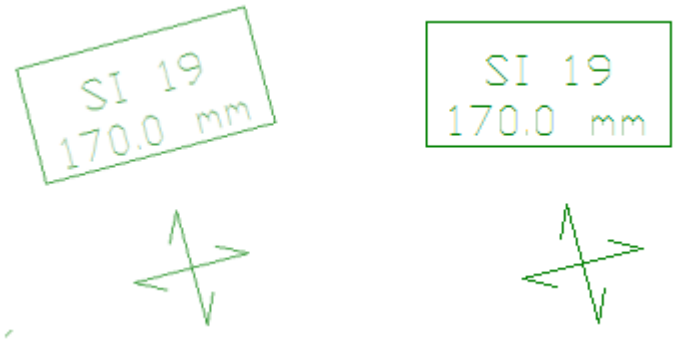
Include panel reference
Select to include the slab panel reference in the label.

Include panel thickness
Select to include the slab panel thickness in the label.

Include surface offset (if non zero)
Select to include any surface offset that has been applied to the slab panel in the label.

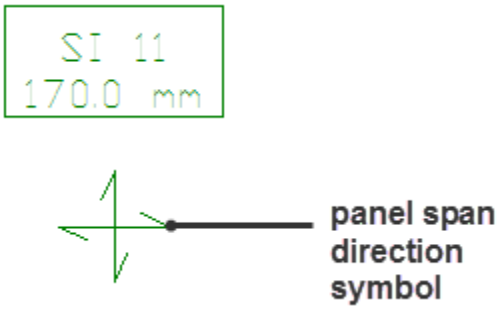
Include border around label
Check this to add a border around the label.

Align label to panel reinforcement
Check this to align the label to the slab panel span direction.
- When checked the label is displayed as below left
- When cleared the label is displayed as below left



Slab/Mat Geometry

Include panel span direction symbol
Select to include the symbol



Mats

Show pile location table
Select to include a table showing the pile locations.

Show pile references
Select to show the pile references on the layout.

Isolated Foundations tab

Drawing Content

Show pile type name
Select to include the pile type names in the footing attributes displayed for each pile cap on the layout.

Show foundation details

Select to include pad/strip base and pile cap details to the side of the layout (showing the designed reinforcement).

Show reinforcement quantities table

Select to include the reinforcement quantities table for the pad/strip bases and pile caps shown on the layout.

Show pad/strip base schedule

Select to include the reinforcement schedule for the pad/strip bases on the layout.

Show pile cap schedule

Select to include the reinforcement schedule for the pile caps on the layout.

Show allowable pile capacity

Select to include the allowable pile capacity table for the piles under pile caps on the layout.

Dimensions on plan

Show overall dimensions

Select to include the overall dimensions of the pad/strip bases and pile caps on the layout.

Show pile spacings

Select to include the pile cap pile spacing dimensions on the layout.

Grouped Foundation Labelling

Include the foundation name

Select to include the pad/strip base or pile cap name in the footing mark for grouped footings on the layout.

Labelling Position

Above/Below

controls where the footing mark and attributes are displayed for isolated foundations on the layout.

Isolated Foundation Detail-Content

These settings are used to control the **content** of the base detail drawings.

Grouped Foundations: Show number of foundations in group

Select to display the number of bases/pile caps in the group

Cross sections: Positions

Choose 'None' to exclude, or choose 'Length' for a length-wise section or 'Length and Width' to display cross sections in both directions.

Pile Labels: Show pile type name

Select to include the pile type name in the label.

Dimensions: Show overall dimensions

Select to include the overall dimensions of the base/pile cap.

Dimensions: Show overall dimensions

Select to include dimensions of the column/wall being supported.

Dimensions: Show pile spacings

For pile caps, select this check box to include dimensions from center to center of each pile.

Quantities: Show reinforcement quantities table

Select to show the reinforcement quantities table for the base.

Isolated Foundation Detail-Style

These settings are used to control the **style** of the column detail drawings.

General: Show supported member as hatched

Select to hatch columns and walls that are supported on the base pile cap.

General: Underline foundation label

Select to underline the foundation label on the detail

Cross-sections: Section label style

Choose the label style to be applied to the cross sections from the drop list

Cross-sections: Restart labels in each foundation

When multiple foundations are included on the same drawing, checking this option causes the labelling to restart from the beginning on each section.

Planar Drawings

The following drawing variants fall into this category:

General Arrangement

General Arrangement drawings show the member layouts for 2D levels and frames.

Beam End Forces

Beam End Force drawings are similar to General Arrangements but also display the forces at the ends of steel beams for the purpose of connection design.

Column Splice Loads

This drawing should be created from a Frame View in order to display the splice loads at splice locations in steel columns.

Foundation Reactions

Foundation Reaction drawings include support reactions to assist foundation design.

Loading Plan

Loading Plan drawings show the applied loads for individual loadcases.

Prior to creating any of these drawings you should ensure that the Options - Planar Drawings have been configured to meet your requirements.

How do I create a General Arrangement drawing?

1. Open a 2D scene view displaying the part of the model to be included on the drawing. For example a particular construction level, frame, or sloped plane.

2. Click **Draw** > **General Arrangement** ( )

The above button is only available when the current view is displayed in 2D. If the button is dimmed, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

3. In the DXF Export Preferences dialog choose the Layer Configuration and Layer Style required.

4. Specify the drawing **scale**.

5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).

6. Either accept the automatic file name, or enter the name directly.

7. Click **OK**

How do I create a Beam End Forces drawing?

Beam End Force drawings are typically created to assist connection design in steel structures. These forces are not relevant in concrete structures and therefore do not get displayed for concrete beams.

1. Open a 2D Results View displaying the part of the model to be included on the drawing. For example a particular construction level, frame, or sloped plane.

2. From the **Loading drop list** select the loadcase or combination to be displayed.

3. Click **Draw** > **Beam End Forces** ( )

The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

4. In the DXF Export Preferences dialog choose the Layer Configuration and Layer Style required.

5. Specify the drawing **scale**.

6. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).

7. Either accept the automatic file name, or enter the name directly.

8. Click **OK**

How do I create a Column Splice Loads drawing?

Column Splice drawings are typically created to assist connection design in steel structures. These forces are not relevant in concrete structures.

1. Open a Frame view containing the steel frame for which you want to see the column splice loads.

2. In the **Status Bar** click **Results View**

3. From the **Loading drop list** select the loadcase or combination to be displayed.

4. Click **Draw** > **Column Splice Loads**

The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

5. In the DXF Export Preferences dialog choose the Layer Configuration and Layer Style required.

6. Specify the drawing **scale**.

7. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).

8. Either accept the automatic file name, or enter the name directly.

9. Click **OK**

How do I create a Foundation Reactions drawing?

1. Open a 2D Results View displaying the part of the model to be included on the drawing. (Typically the base construction level).

2. From the **Loading drop list** select the loadcase or combination to be displayed.

3. Click **Draw** > **Foundation Reactions** ( )

The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

4. In the DXF Export Preferences dialog choose the Layer Configuration and Layer Style required.
5. Specify the drawing **scale**.
6. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
7. Either accept the automatic file name, or enter the name directly.
8. Click **OK**

How do I create a Loading Plan drawing?

1. Open a 2D scene view displaying the part of the model to be included on the drawing. For example a particular construction level, frame, or sloped plane.
2. From the **Loading drop list** select the loadcase to be displayed.
3. Click **Draw** > **Loading Plan** ( )

The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

4. In the DXF Export Preferences dialog choose the Layer Configuration and Layer Style required.
5. Specify the drawing **scale**.
6. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
7. Either accept the automatic file name, or enter the name directly.
8. Click **OK**

DXF Export Preferences dialog

From this dialog you can specify the drawing location and control both its content and appearance

Use automatic file name

Leave the box checked in order to create the drawing with an automatically generated file name in the same folder as the model. Uncheck to specify an alternative file name and location.

Layer Configuration

The Layer Configuration drop list is used to specify which layers are included in the dxf.

Layer Style

The Layer Style drop list is used to specify the appearance of the text and lines in the included layers.

Scale

Controls the scale of the drawing.

Minimum Text Block Spacing

The **Minimum Text Block Spacing** is used in order to adjust the distance between independent drawing blocks.

Caution should be applied when adjusting this value - if it is set too large, text labels can be displaced away from the objects to which they refer.

Member Details

The following drawing variants fall into this category:

Concrete Beam Detail

This drawing displays the beam reinforcement in elevation and section for each span. A reinforcement quantity table can optionally be included.

Concrete Column Detail

This drawing displays the column reinforcement in elevation and section. A reinforcement quantity table can optionally be included.

Concrete Wall Detail

This drawing displays the wall reinforcement in elevation and section. A reinforcement quantity table can optionally be included.

Non-concrete Beam Detail

Non-concrete beam detail drawings are used to display individual steel beam details.

Non-concrete Column Detail

Non-concrete column detail drawings are used to display individual steel beam details.

Prior to creating any of these drawings you should ensure that the Options - Member Details have been configured to meet your requirements.

How do I create a concrete beam detail?

Prior to creating your beam detail drawings you should ensure that both the Beam Detail - Content and the Beam Detail - Style options have been configured to meet your requirements.

To create a beam detail:

1. Hover the cursor over the beam to be detailed until its outline becomes highlighted.
2. Right-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

How do I create a concrete column detail?

Prior to creating your column detail drawings you should ensure that both the Column Detail - Content and the Column Detail - Style options have been configured to meet your requirements.

To create a column detail:

1. Hover the cursor over the column to be detailed until its outline becomes highlighted.
2. Right-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.

7. Click **OK**

How do I create a concrete wall detail?

Prior to creating your column detail drawings you should ensure that both the Wall Detail - Content and the Wall Detail - Style options have been configured to meet your requirements.

To create a wall detail:

1. Hover the cursor over the wall to be detailed until its outline becomes highlighted.
2. Right-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

How do I create a non concrete beam detail?

Prior to creating your beam detail drawings you should ensure that both the Beam Detail - Content and the Beam Detail - Style options have been configured to meet your requirements.

To create a beam detail:

1. Hover the cursor over the beam to be detailed until its outline becomes highlighted.
2. Right-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

How do I create a non concrete column detail?

Prior to creating your column detail drawings you should ensure that both the Column Detail - Content and the Column Detail - Style options have been configured to meet your requirements.

To create a column detail:

1. Hover the cursor over the column to be detailed until its outline becomes highlighted.
2. Right-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

Slabs and Mats

The following drawing variants fall into this category:

Slab/Mat Layout

Slab and mat detail drawings are used to convey slab panel reinforcement and patch reinforcement requirements, (which may take the form of either loose bars or mesh).

The drawings also include a quantity table for the reinforcement displayed with a detailing allowance added.

An option is provided to also show punching check reinforcement details at the side of the main layout.

Punching Check Detail

This drawing shows an individual punching check detail in plan, with an option to show the reinforcement quantities table

How do I create a Punching Check Detail drawing?

To create a punching check detail:

1. Hover the cursor over the punching check to be detailed until its outline becomes highlighted.
2. Right-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.

4. Specify the drawing **scale**.

5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).

6. Either accept the automatic file name, or enter the name directly.

7. Click **OK**

How do I create a Slab or Mat Layout drawing?

1. Open a 2D scene view displaying the slabs/mats to be included on the drawing.

2. Click **Draw** > **Slab/Mat Detailing** ( )

The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

The DXF Export Preferences dialog is displayed.

3. Choose the Layer Configuration and Layer Style required.

4. Specify the drawing **scale**.

5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).

6. Either accept the automatic file name, or enter the name directly.

7. Click **OK**

Foundations

The following drawing variants fall into this category:

Isolated Foundation Detail

This drawing shows an individual foundation detail in plan, with options to show:

- the detail in cross section
- reinforcement quantities table

Foundation Layout

This drawing shows the piling layout and also the layout of isolated foundations, with options to show:

- isolated foundation details
- reinforcement quantities table
- isolated foundation schedule
- pile location table

How do I create a Base or Pile Cap Detail drawing?

Prior to creating your base detail drawings you should ensure that both the Isolated Foundation Detail-Content and the Isolated Foundation Detail-Style options have been configured to meet your requirements.

To create a base/pile cap detail:

1. Hover the cursor over the base/pile cap to be detailed until its outline becomes highlighted.
2. Right-clickRight-click and select **Generate Detailing Drawing**... from the context menu to display the DXF Export Preferences dialog.
3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

How do I create a Foundation Layout drawing?

1. Open a 2D scene view displaying the piles/bases/pile caps to be included on the drawing.
2. Click **Draw** > **Foundation Layout** ( )

The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.

The DXF Export Preferences dialog is displayed.

3. Choose the Layer Configuration and Layer Style required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

Drawing Management

Although individual drawings can be created as and when required, it is often more efficient to create a batch of drawings in a single operation. This becomes an almost essential requirement when working in large models.

The **Drawing Management...** command facilitates this process allowing you to:

- Select the drawing variant
- Either manually add a drawing sheet, or generate a series of drawing sheets
- Select the frames, levels or members for which drawings are to be created
- Arrange the drawings on the drawing sheet, (either in a linear or grid arrangement)
- For load dependant drawing variants, select the loadcases/combinations
- Create drawing revisions
- View the revision history
- Reset bar marks on concrete detail drawings (in order to remove gaps in the bar mark numbering)

How do I add new drawings and specify their content?

For all drawing variants this can be done manually as follows:

1. Click **Draw** > **Drawing Management...**
2. Specify the **drawing variant**.
3. Click **Add**
4. Enter a **Name** for the new drawing sheet
5. Click **Content...**
6. To choose from a list of only those items not already placed on a drawing: pick 'Show unassigned only', else pick 'Show all'
7. Drag the items to be included from the left pane to the right pane of the Drawing Content dialog.
8. Click **OK**

For concrete beam and column detail variants the drawings can also be generated automatically:

1. Click **Draw** > **Drawing Management...**
2. Select either the **Concrete Beam Detail** or **Concrete Column Detail** drawing variant as required.
3. Click **Generate**

   Drawings are created with their content automatically generated: one drawing will contain containing typical beams or columns, additional drawings are also created which contain any ungrouped beams or columns

4. If necessary you can click the available drawings in order to rename them.

How do I specify the layout?

1. Select the drawing name from the available drawings list.
2. Click **Layout...**
3. Choose the direction and arrangement of the layout as required
4. Click **OK**

For load dependant drawing variants, how do I specify the loading?

1. Select the drawing name from the available drawings list.
2. Click **Loading...**
3. Choose the loadcases and combinations required
4. Click **OK**

How do I view a drawing?

1. Select the drawing name from the available drawings list.
2. Click **View Drawing...**
3. In **DXF Export Preferences** choose the drawing **type** and **style** required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Click **OK**

How do I consolidate the bar marks used on concrete detail drawings?

Each bar geometry that has ever been used in the model has an associated mark - and this information is not automatically deleted. That means if a model is designed and some of the bars fall out of use, their bark mark assignment is still retained. Consequently there is the potential for gaps in the bar marks and bar marks starting at high numbers. To avoid this a **Reset ALL Marks** button is provided.

It is envisaged that this consolidation feature is used as follows:

Phase 1 - During initial design development you would want to continually consolidate. In this stage things can change a lot and the bar marks used will climb quickly leaving lots of gaps. Nothing is being issued so there is no problem consolidating.

Phase 2 - At later stages (once information starts to be issued/shared with others) it becomes increasingly preferable not to consolidate.

To consolidate the bar marks on **all** drawing sheets:

1. Click **Draw** > **Drawing Management...**
2. Select the **Concrete Beam Detail** or **Concrete Column Detail** drawing variant as required.

3. Click **Reset ALL Marks**

How do I apply a revision to a drawing?

To apply a revision:

1. Click **Draw** > **Drawing Management...**
2. Select the drawing
3. Click **Create Revision...**
4. Enter a **revision name**
5. Enter the revision note.
6. Click **OK**

How do I view the revision history of a drawing?

To view revision history:

1. Click **Draw** > **Drawing Management...**
2. Select the drawing
3. Click **History...**

Concrete Member Schedules

Concrete Member Schedules

The following drawing variants fall into this category:

Concrete Beam Schedule

A beam schedule (not to be confused with a bar bending schedule) is a particular form of output which is generated by the Engineer in which the reinforcement is listed on a span by span basis. The beam schedule is used by other parties (such as a specialist detailing firm or a contractor) to produce the bar bending schedules that are necessary for the construction to proceed.

Beam schedules are created by building, by floor or by selected beams. The information shown in the schedule is based on the design groups.

Beam schedules are created in dxf format even though they don't include graphical information, so that they can be added to the beam detail drawings.

Concrete Column Schedule

Column schedules display a cross section through each stack for the selected columns. A reinforcement quantity table is optionally included.

Concrete Wall Schedule

Wall schedules display a cross section through each stack for the selected walls. A reinforcement quantity table is optionally included.

> *Prior to creating any of these drawings you should ensure that the Options - Member Schedules have been configured to meet your requirements.*

How do I create a concrete beam schedule?

1. Open a 2D scene view displaying the beams to be included on the schedule. For example a particular construction level, frame, or sub model.
2. Click **Draw** > **Beam Schedule** ( )

> *The above button is only available when the current view is displayed in 2D. If the button is grayed out, check that you are not in a Structure view or in a 2D scene view displayed in 3D.*

3. In the DXF Export Preferences dialog choose the drawing **type** and **style** required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Either accept the automatic file name, or enter the name directly.
7. Click **OK**

How do I create a concrete column schedule?

1. Open a 3D View or Frame View displaying the columns to be included on the schedule.
2. Click **Draw** > **Column Schedule** ( )
3. Select the columns to be included, then click **OK**
4. In the DXF Export Preferences dialog choose the drawing **type** and **style** required.
5. Specify the drawing **scale**.
6. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
7. Either accept the automatic file name, or enter the name directly.
8. Click **OK**

How do I create a concrete wall schedule?

1. Open a 3D View or Frame View displaying the walls to be included on the schedule.
2. Click **Draw** > **Wall Schedule** ( )
3. Select the walls to be included, then click **OK**
4. In the DXF Export Preferences dialog choose the drawing **type** and **style** required.
5. Specify the drawing **scale**.
6. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
7. Either accept the automatic file name, or enter the name directly.
8. Click **OK**

Schedule Management

Although individual schedules can be created as and when required, it is also possible to create a batch of schedules in a single operation.

The **Schedule Management...** command facilitates this process allowing you to:

- Select the drawing variant
- Add the drawing
- Select members to be included
- Arrange the drawing layout, (either in a linear or grid arrangement)
- Create revisions
- View the revision history
- Reset bar marks (in order to remove gaps in the bar mark numbering)

How do I add new drawings and specify their content?

1. Click **Draw** > **Schedule Management...**
2. Specify the **drawing variant**.
3. Click **Add**
4. Enter a **Name** for the new drawing
5. Click **Content...**
6. To choose from a list of only those items not already placed on a drawing: pick 'Show unassigned only', else pick 'Show all'
7. Drag the members to be included from the left pane to the right pane of the Drawing Content dialog.
8. Click **OK**

How do I specify the layout?

1. Select the drawing name from the available drawings list.
2. Click **Layout...**
3. Choose the direction and arrangement of the layout as required
4. Click **OK**

How do I view a drawing?

1. Select the drawing name from the available drawings list.
2. Click **View Drawing...**
3. In **DXF Export Preferences** choose the drawing **type** and **style** required.
4. Specify the drawing **scale**.
5. If required modify the **Minimum Text Block Spacing,** (in order to adjust the distance between independent lines of text).
6. Click **OK**

How do I consolidate the bar marks?

Each bar geometry that has ever been used in the model has an associated mark - and this information is not automatically deleted. That means if a model is designed and some of the bars fall out of use, their bark mark assignment is still retained. Consequently there is the potential for gaps in the bar marks and bar marks starting at high numbers. To avoid this a **Reset ALL Marks** button is provided.

It is envisaged that this consolidation feature is used as follows:

Phase 1 - During initial design development you would want to continually consolidate. In this stage things can change a lot and the bar marks used will climb quickly leaving lots of gaps. Nothing is being issued so there is no problem consolidating.

Phase 2 - At later stages (once information starts to be issued/shared with others) it becomes increasingly preferable not to consolidate.

To consolidate the bar marks on **all** drawings:

1. Click **Draw** > **Schedule Management...**
2. Select the **Concrete Beam**, **Column** , or **Wall Schedule** drawing variant as required.
3. Click **Reset ALL Marks**

How do I apply a revision to a drawing?

To apply a revision:

1. Click **Draw** > **Drawing Management...**
2. Select the drawing
3. Click **Create Revision...**

4. Enter a **revision name**
5. Enter the revision note.
6. Click **OK**

How do I view the revision history of a drawing?

To view revision history:

1. Click **Draw** > **Drawing Management...**
2. Select the drawing
3. Click **History...**

Object Properties

The topic links below list the properties for the different object types that can be selected in Scene Views, or the Project Workspace.

Topics in this section

- Structure, Level, Frame and Slope Properties
- Steel, Concrete and Timber Member Properties
- Slab and Mat Properties
- Roof Panel and Wall Panel Properties
- Support, Element and Bearing Wall Properties
- Patch, Punching Check, Result Strip and Check Line Properties
- Isolated Foundation and Pile Properties
- Sub Model Properties
- Solver Model Properties

Structure, Level, Frame and Slope Properties

Topics in this section

- Structure Properties
- Level Properties
- Frame Properties
- Slope Properties

Structure Properties

When Structure is selected in the Structure Tree, properties for defining the building's principal direction and default meshing are displayed:

Property	Description
General	

Building Direction Rotation	All buildings are set out in Global X, Y and Z coordinates in *Tekla Structural Designer*. However, a building may have its "principal axes" orientated at an angle to the global axes (in plan). There are certain processes that require this knowledge in order to orientate themselves correctly for the building - these include • Slenderness calculations • Direction of continuous beams • Equivalent horizontal forces/notional loads • Seismic loading • Natural frequencies • Deflection results • Drift results The default (0 degrees) aligns the building direction 1 arrow with the global X axis and the direction 2 arrow with the global Y axis. Entering a positive value rotates the Building Direction arrows clockwise about positive Global Z. A negative value rotates anti-clockwise. The limiting values are +45 degrees and -45 degrees. (If you enter larger values they will be capped at these limits). The building direction arrows are always at 90 degs to each other.
Show Building Direction Arrows	• Checked - the building direction arrows are displayed in all the 2D and 3D Views
Building Direction Labels	Choose the labels to be used for the building direction arrows. • Dir 1/2 • Dir H/V • Dir X/Y
Consequence class (Head Code Finland (Eurocode))	The choice of consequence class sets additional factor k_{Fl} used in strength combinations. • CC3 (k_{Fl} =1.1) • CC2 (k_{Fl} =1.0) • CC1 (k_{Fl} =0.9)
Reliability class (Head Code Sweden (Eurocode))	The choice of reliability class sets additional factor γ_d used in strength combinations. • RC3 (γ_d =1.0) • RC2 (γ_d =0.91) • RC1 (γ_d =0.83)

Meshing	
Shell Mesh Size	Specifies the shell mesh size for two way spanning slabs. (Default 1.000m or 3′ 6″). *Although the above default is likely to be conservative, the degree of mesh refinement applied remains the users responsibility.* To optimize solution time consider using a coarser mesh during design development before switching to a more refined mesh at the final design stage.
Shell Uniformity Factor	Specifies the shell uniformity factor for two way spanning slabs. (Default 50%). *Although the above default is likely to be conservative, the degree of mesh refinement applied remains the users responsibility.* To optimize solution time consider using a coarser mesh during design development before switching to a more refined mesh at the final design stage.
Semi-Rigid Mesh Size	Specifies the mesh size for roof panels, and slabs when modelled as semi-rigid diaphragms. (Default 1.000m or 3′ 6″).
Semi-Rigid Uniformity Factor	Specifies the semi-rigid uniformity factor for roof panels, and slabs when modelled as semi-rigid diaphragms. (Default 100%)
Semi-Rigid Mesh Type	Specifies the semi-rigid mesh type (QuadDominant, QuadOnly, or Triangular) for roof panels, and slabs when modelled as semi-rigid diaphragms.
Wall Mesh Horizontal Size	Specifies the horizontal mesh size for all meshed walls in the model - but can be overridden in individual wall properties. (Default 1.000m or 3′ 6″)
Wall Mesh Vertical Size	Specifies the vertical mesh size for all meshed walls in the model - but can be overridden in individual wall properties. (Default 1.000m or 3′ 6″)
Wall Mesh Type	Used to specify the mesh type for all meshed walls in the model, (but can be overridden in individual wall properties): • Quad only • Tri only • Quad dominant

Level Properties

When a level is selected in the Structure Tree the following properties are displayed:

Name	Description
General	
Level	The height of the construction level above the base level.
Floor	By setting a construction level to be a Floor you are indicating that it is a major level in the building. Floor levels are used to determine items such as your inter story height and positions from which column splices are laid out. If a level is not set to be a floor then no live load reductions will be accounted for in the beams at that level, or in the columns supporting that level. There will certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs, • K Bracing - you require a construction level for the intermediate bracing connection points, • steps in the building floor levels. Where you define a level which is clearly not a floor, then you should not check the floor box.
Type	Select the type from the drop down menu: T.O.S, S.S.L, or F.F.L
Mesh Slabs in 3D Analysis	• Checked - the slabs are meshed in the 3D Analysis. The Sub Model in which the Level is contained determines the mesh parameters that are applied. • Cleared - the slabs are not meshed in the 3D Analysis.
Short Name	Each construction level should be given a unique reference. Typically this might be a storey number, 1, 2 , 3 etc.
Long Name	Each construction level can also be given a name to further assist identification. 'First Floor', or 'Mezzanine' etc.
Name	Automatically generated from the short and long name. By default this will be used as the name in the Structure Tree.
User Name	You can use this to replace the name displayed in the **Structure Tree**.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting

Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified level • Off = Grid & construction lines are always off for the specified level, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting *This setting does* **not** *control the display of* **Architectural Grids** *which have a separate checkbox in Scene Content.*
Decomposition	
Keep solver model	The is recreated each time the analysis is performed - this setting simply controls whether you will be able to review the model between analyses: • Checked - the solver model is retained to allow you to visually review the meshing etc. • Cleared (Default) - the solver model is discarded immediately after analysis has been performed.

Frame Properties

When a frame is selected in the Structure Tree the following properties are displayed:

Name	Description
Name	The automatically generated name for the frame is derived from the grid line selected.
User Name	Can be used to override the automatically generated name if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in frame views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified frame • Off = Grid & construction lines are always off for the specified frame, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting

	This setting does **not** *control the display of* **Architectural Grids** *which have a separate checkbox in Scene Content.*
Visible	Controls whether the frame view can be opened or not.

Slope Properties

When a slope is selected in the Structure Tree the following properties are displayed:

Name	**Description**
Type	Select the type from the drop down menu: T.O.S, S.S.L, or F.F.L.
Mesh Slabs in 3D Analysis	• Checked - the slabs in the Sloped Plane are meshed in the 3D Analysis. The Sub Model in which the plane is contained determines the mesh parameters that are applied. • Cleared - the slabs in the Sloped Plane are not meshed in the 3D Analysis.
Name	The automatically generated name for the slope.
User Name	Can be used to override the automatically generated name if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in slope views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified slope • Off = Grid & construction lines are always off for the specified slope, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting *This setting does* **not** *control the display of* **Architectural Grids** *which have a separate checkbox in Scene Content.*

Visible	Controls whether the slope view can be opened or not.
Decomposition	
Keep solver model	The is recreated each time the analysis is performed - this setting simply controls whether you will be able to review the model between analyses: • Checked - the solver model is retained to allow you to visually review the meshing etc. • Cleared (Default) - the solver model is discarded immediately after analysis has been performed.

Steel, Concrete and Timber Member Properties

Topics in this section

- Steel Member Properties
- Concrete Member Properties
- Timber Member Properties
- Cold Formed Member Properties
- Cold Rolled Properties
- Definition of rotation angle and gamma angle

Steel Member Properties

Topics in this section

- steel column properties set
- steel beam properties set
- composite steel beam properties set
- steel brace properties set
- steel joist properties set
- portal frame properties set
- gable post properties set
- parapet post properties set
- edit steel column properties set
- edit steel beam properties set
- edit composite steel beam properties set
- edit steel brace properties set
- edit steel joist properties set
- edit gable post properties set

- edit parapet post properties set
- edit steel truss properties set

edit steel column properties set

The following properties are displayed when you select one or more existing steel columns.

For an individual column the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the column.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the column belongs.
Top Level	Define the vertical position of the column top. (This property is only displayed in 2D Floor Views)
Base Level	Define the vertical position of the column base. (This property is only displayed in 2D Floor Views)
Plane	Indicates the grid along which the column is placed.
Characteristic	Define the basic member type.
Material type	Define the material for the column.
Construction	Define type of construction. Composite column construction is beyond the scope of the current release for design.
Fabrication	Define the type of fabrication. Plated, concrete filled and concrete encased fabrication, is beyond the scope of the current release for design.
Autodesign	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)

Design section order	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Gravity only	Indicates if the column is designed for gravity combinations only. • Checked - designed for gravity combinations • Cleared - designed for gravity and lateral combinations See: Gravity only design
Rotation	Define the rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Define alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the column from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the column from the snap point in the minor axis.
Assume extra floors supported	The number of extra floors assumed to be supported by the column. See: Load reductions
Simple Column	Indicates if the column is to be designed as a simple column. See: Simple columns

[-] All stacks	
Section	The section initially applied to the column that is created. This will be replaced by a new section from the design section order if the member is auto-designed.
Grade	Define the steel grade of the column. The available grades are set from the Materials button on the Home ribbon.
Encasing section	The encasing section. (This property is only displayed for concrete encased fabrication).
Concrete class	Define the concrete grade for concrete filled, or concrete encased columns. The concrete grades that are available are set from the Materials button on the Home ribbon. (This property is only displayed if the fabrication type is set to concrete filled, or concrete encased).
Face A cont. rest.	Indicates continuous restraint to Face A. See: Steel column restraints
Face C cont. rest.	Indicates continuous restraint to Face C. See: Steel column restraints
[+] Releases	Define the end releases. See: Column releases
[+] Size constraints	Size Constraints are only applicable when Autodesign is checked. They allow you to impose upper and lower limits to the section depth and width. See: Size Constraints
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.

Lambda crit check or drift check	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Wind drift check	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
[+] Stack	See also **[-] All Stacks**
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
Splice	Indicates splice positions.
[+] Level	
[+] Load Reductions Count the floor as being supported (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the imposed load reductions. See: Load reductions
[+] Load Reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] Eccentricity	Define eccentric beam end connections to the column faces.

[+] Seismic	
In a seismic force resisting system	Define if the column is part of a seismic force resisting system and if so, the SFRS type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release. See: Steel columns in seismic force resisting systems.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit steel beam properties set

The following properties are displayed when you select one or more existing steel beams.

For an individual beam the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the beam.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the beam belongs.
Plane	Indicates the level at which the beam is placed.
Characteristic	Define the basic member type.
Material type	Define the material of the beam.
Construction	Define the type of construction: • Non composite • Composite - the beam acts compositely with a concrete slab

Fabrication	Define the type of fabrication: • Rolled • Plated • Westok cellular • Westok plated • Fabsec Westok cellular and Fabsec are beyond the scope of the current release for design.
Autodesign	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process See: Autodesign (steel)
Design section order	Specifies the order file from which the designed sections will be selected. See: Design Section Order
Gravity only	Indicates if the beam is designed for gravity combinations only. • Checked - designed for gravity combinations • Cleared - designed for gravity and lateral combinations See: Gravity only design
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the beam from the snap point in the major axis.

Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the beam from the snap point in the minor axis.
[-] All Spans	
Section	The section initially applied to the beam that is created. This will be replaced by a new section from the design section
Grade	Define the steel grade of the beam. The available grades are set from the Materials button on the Home ribbon.
Linearity	Indicates if the beam is straight or curved. Design of curved steel beams is beyond the scope of the current release.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
Top flange cont. rest.	Define if the top flange is continuously restrained. Restraints and their factors can also be managed from the **Steel Beam Properties Dialog** by right-clicking on the beam and selecting Edit from the context menu.
Bottom flange cont. rest.	Define if the bottom flange is continuously restrained. Restraints and their factors can also be managed from the **Steel Beam Properties Dialog** by right-clicking on the beam and selecting Edit from the context menu.
[+] Releases	Define the end releases. See: Beam releases
[+] Load reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] Deflection limits	Define the deflection limits for specific load types. See: Steel beam deflection limits

[+] Torsion (Head Code ACI/AISC)	Used to specify if the member should be checked for torsion in accordance with Design Guide 9 (DG9), and also to apply a rotational limit if required. See: Steel beam torsion
[+] Camber	Used to specify a camber to the beam if required. See: Steel beam camber
[+] Natural frequency	Used to specify a natural frequency limit to the beam if required. See: Composite beam natural frequency
[+] Size constraints	Define upper and lower limits to the section depth and width when it is being autodesigned. See: Size Constraints
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.
[-] Seismic	
In a seismic force resisting system	Indicates if the beam is part of a seismic force resisting system and if so, the SFRS type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release. See: Steel beams in seismic force resisting systems
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).
[+] Span 1, Span 2 etc.	In a multi-span beam properties can be entered for a specific span, over-riding those defined at the **All spans** level. See: [-] All Spans

edit composite steel beam properties set

The following properties are displayed when you select one or more existing composite steel beams.

For an individual beam the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	Description
[-] General	
Name	The automatically generated name for the beam.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the beam belongs.
Plane	Indicates the level at which the beam is placed.
Characteristic	Define the basic member type.
Material type	Define the material of the beam.
Construction	Define the type of construction: • **Composite beam**- the beam acts compositely with a concrete slab
Allow non-composite design	Select to allow the beam to be designed non-compositely when a composite design cannot be achieved. See: Allow non-composite design
Fabrication	Define the type of fabrication: • Rolled • Plated • Westok cellular • Westok plated • Fabsec Westok cellular and Fabsec are beyond the scope of the current release for design.
Autodesign	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process See: Autodesign (steel)
Design section order	Specifies the order file from which the designed sections will be selected. See: Design Section Order

Gravity only	Indicates if the beam is designed for gravity combinations only. • Checked - designed for gravity combinations • Cleared - designed for gravity and lateral combinations See: Gravity only design
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the beam from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the beam from the snap point in the minor axis.
[-] Spans (All/Individual)	
Section	The section initially applied to the beam that is created. This will be replaced by a new section from the design section
Grade	Define the steel grade of the beam. The available grades are set from the Materials button on the Home ribbon.

Linearity	Indicates if the beam is straight or curved. Design of curved steel beams is beyond the scope of the current release.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
Top flange cont. rest.	Define if the top flange is continuously restrained. Restraints and their factors can also be managed from the **Steel Beam Properties Dialog** by right-clicking on the beam and selecting Edit from the context menu..
Bottom flange cont. rest.	Define if the bottom flange is continuously restrained. Restraints and their factors can also be managed from the **Steel Beam Properties Dialog** by right-clicking on the beam and selecting Edit from the context menu.
[+] Releases	Define the end releases. See: Beam releases
[+] Load reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] Deflection limits	Define the deflection limits for specific load types. See: Steel beam deflection limits
[+] Camber	Used to specify a camber to the beam if required. See: Steel beam camber
[+] Natural frequency	Used to specify a natural frequency limit to the beam if required. See: Composite beam natural frequency
[+] Size constraints	Define upper and lower limits to the section depth and width when it is being autodesigned. See: Size Constraints
Floor construction	Used to specify the shear connector type. See: Floor construction
Metal deck	Used to specify the minimum lap distance. See: Metal deck

Stud strength	Used to specify the stud properties. See: Stud strength
Connector layout	Used to specify the stud layout. See: Connector layout
Transverse reinforcement	Used to specify the transverse reinforcement. See: Composite beam transverse reinforcement
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.
[-] Seismic	
In a seismic force resisting system	Indicates if the beam is part of a seismic force resisting system and if so, the SFRS type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release. See: Steel beams in seismic force resisting systems
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

For an individual beam the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

edit steel brace properties set

The following properties are displayed when you select one or more existing steel braces.

For an individual brace the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the brace.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the brace belongs.

Plane	Indicates the level at which the brace is placed.
Characteristic	Define the basic member type.
Material type	Define the material of the brace.
Autodesign	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process See: Autodesign (steel)
Design section order	Specifies the order file from which the designed sections will be selected. See: Design Section Order
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
[+] Alignment	
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the brace from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the brace from the snap point in the minor axis.
[-] All spans	

Section	The section initially applied to the brace that is created. This will be replaced by a new section from the design section
Grade	Define the steel grade of the brace. The available grades are set from the Materials button on the Home ribbon.
Compression only	Specify if the brace is compression only.
Tension only	Specify if the brace is tension only.
Relaxation factor	Only displayed if the brace is tension only (default 0.0). Any value entered is not used unless you also check the 'Use relaxation factors for tension only elements' box in Analysis Options. It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See: Relaxation Factors
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Releases	Define the end releases. See: Brace releases
Compression (Head Code BS/EC)	Expands to allow the definition of effective length factors. See: Steel brace in compression
Tension	Expands to allow the definition of net area of the section for tension checks. See: Steel brace in tension
[+] Load reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] Size constraints	Define upper and lower limits to the section depth and width when it is being autodesigned. See: Size Constraints
[-] Seismic	

In a seismic force resisting system	Indicates if the brace is part of a seismic force resisting system and if so, the SFRS type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release. See: Steel beams in seismic force resisting systems
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit steel joist properties set

The following properties are displayed when you select one or more existing steel joists.

For an individual joist the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the joist.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the joist belongs.
Plane	Indicates the level at which the joist is placed.
Characteristic	Steel joist
Material type	Define the material of the joist.
Steel joist type	Define the joist type.
Autodesign	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process See: Autodesign (steel)
Design section order	Specifies the order file from which the designed sections will be selected. See: Design Section Order

Gravity only	Indicates if the joist is designed for gravity combinations only. • Checked - designed for gravity combinations • Cleared - designed for gravity and lateral combinations See: Gravity only design
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the joist from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the joist from the snap point in the minor axis.
[-] All spans	
Section	The section applied to the joist that is created.
Grade	The steel grade of the joist. The available grades are set from the Materials button on the Home ribbon..
[+] Releases	Define the fixity and torsional load releases at each end.
[+] Load reductions KLL	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions

[+] Deflection limits	Define the deflection limits for specific load types. See: Steel beam deflection limits
[+] Size constraints	Define upper and lower limits to the section depth and width when it is being autodesigned. See: Size Constraints
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.

edit gable post properties set

The following properties are displayed when you select one or more existing steel gable posts.

For an individual post the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the column.
User Name	Can be used to override the automatically generated name if required.
Top Level	Define the vertical position of the column top. (This property is only displayed in 2D Floor Views)
Base Level	Define the vertical position of the column base. (This property is only displayed in 2D Floor Views)
Plane	Indicates the grid along which the column is placed.
Characteristic	Define the basic member type.
Material type	Define the material for the column.
Construction	Define type of construction.
Fabrication	Define the type of fabrication.

Autodesign	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Design section order	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Gravity only	Indicates if the column is designed for gravity combinations only. • Checked - designed for gravity combinations • Cleared - designed for gravity and lateral combinations See: Gravity only design
Rotation	Define the rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Define alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the column from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the column from the snap point in the minor axis.

Assume extra floors supported	The number of extra floors assumed to be supported by the column. See: Load reductions
Simple Column	Indicates if the column is to be designed as a simple column. See: Simple columns
[-] All stacks	
Section	The section initially applied to the column that is created. This will be replaced by a new section from the design section order if the member is auto-designed.
Grade	Define the steel grade of the column. The available grades are set from the Materials button on the Home ribbon.
[+] Releases	Define the end releases. See: Column releases
Deflection limits	Define the relative and absolute deflection limits to be applied for wind loads.
[+] Size constraints	Size Constraints are only applicable when Autodesign is checked. They allow you to impose upper and lower limits to the section depth and width. See: Size Constraints
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.
Lambda crit check or drift check	By default gable posts are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular gable posts from these calculations.
Wind drift check	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Stacks' to exclude the

	entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
[+] Seismic	
In a seismic force resisting system	Define if the column is part of a seismic force resisting system and if so, the SFRS type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release. See: Steel columns in seismic force resisting systems.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit parapet post properties set

The following properties are displayed when you select one or more existing parapet posts.

For an individual post the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the post.
User Name	Can be used to override the automatically generated name if required.
Top Level	Define the vertical position of the post top. (This property is only displayed in 2D Floor Views)
Base Level	Define the vertical position of the post base. (This property is only displayed in 2D Floor Views)
Group	The name of the group to which the post belongs.
Plane	Indicates the grid along which the post is placed.
Characteristic	Define the basic member type.

Material type	Define the material for the post.
Construction	Define type of construction.
Fabrication	Define the type of fabrication.
Autodesign	This setting is ignored - Parapet posts are not designed in Tekla Structural Designer.
Design section order	This setting is ignored - Parapet posts are not designed in Tekla Structural Designer.
Rotation	Define the rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Define alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the column from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the column from the snap point in the minor axis.
Assume extra floors supported	The number of extra floors assumed to be supported by the post. See: Load reductions
[-] All stacks	

Section	The section initially applied to the column that is created. This will be replaced by a new section from the design section order if the member is auto-designed.
Grade	Define the steel grade of the column. The available grades are set from the Materials button on the Home ribbon.
[+] Releases	Define the end releases. See: Column releases
[+] Size constraints	Size Constraints are only applicable when Autodesign is checked. They allow you to impose upper and lower limits to the section depth and width. See: Size Constraints
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.
Lambda crit check or drift check	By default gable posts are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular gable posts from these calculations.
Wind drift check	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
[+] Seismic	
In a seismic force resisting system	Define if the column is part of a seismic force resisting system and if so, the SFRS type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release. See: Steel columns in seismic force resisting systems

[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs)

edit steel truss properties set

Once a truss has been created its properties can be displayed in the **Properties Window** by left clicking on the truss.

Property	**Description**
Name	The automatically generated name for the truss is derived from the grid line selected.
User Name	Can be used to override the automatically generated name if required.
Autodesign	• Cleared - the specified section sizes are checked during the design process. • Checked - the section sizes are selected automatically during the design process. See: Autodesign (steel)
Section	The section size of the truss member.
Grade	The steel grade of the truss member. The available grades are set from the Materials button on the Home ribbon.
Design section order	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.

Concrete Member Properties

Topics in this section

- concrete wall properties set
- concrete column properties set

- concrete beam properties set
- edit concrete wall properties set
- edit concrete wall span properties set
- edit concrete column properties set
- edit concrete beam properties set

edit concrete wall properties set

The following properties are displayed in the **Properties Window** when you select one or more existing concrete walls.

For an individual wall the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the wall.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	• Meshed Shear Wall • Mid-pier Shear Wall • Bearing Wall Mid-pier shear walls and bearing walls must be rectangular in a vertical plane. Meshed shear walls can be vertical or sloping.
Fabrication	• Reinforced • Precast Design of precast members is beyond scope in the current release.
Autodesign	• Cleared - the specified reinforcement will be checked during the design process. • Checked - reinforcement will be designed during the design process. See: Autodesign (concrete column)

Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See: Autodesign (concrete column)
Assume extra floors supported	Enter the number of extra floors supported.
Rotation (mid-pier only)	• 0° - wall spans horizontally • 90° - wall spans vertically
AutomaticGenerate Support	• Cleared - a support will only be created if the 'Generate support' property is checked. • Checked - a support will only be created if no members/slabs capable of providing support exist under the wall. See: Automatic support generation
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
Plane	Indicates the grid along which the wall is placed.
[+] All Panels	See: edit concrete wall span properties set
[-] Wall support	

Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
[-] Reinforcement	
Include end zones	• Cleared - the wall is designed without end zones of reinforcement • Checked - the wall is designed with end zones of reinforcement
Reinforcement country	Specifies the country from which the reinforcement properties belong.
Form	• Bar • Mesh
[-] Wall Zone	
Rib type - horizontal & vertical (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - horizontal & vertical (Head Code ACI)	• Plain • Deformed

Class - horizontal & vertical	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Rib type - confinement (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - confinement (Head Code ACI)	• Plain • Deformed
Class - confinement	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
[-] End Zone	
Rib type - horizontal & vertical (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - horizontal & vertical (Head Code ACI)	• Plain • Deformed
Class - horizontal & vertical	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Rib type - confinement (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - confinement (Head Code ACI)	• Plain • Deformed
Class - confinement	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
[+] Panel 1,2,3 etc.	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

	See: edit concrete wall span properties set
[+] Count the floor as being supported (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the imposed load reductions. See: Load reductions
[+]Restrained	Used to indicate at which levels the wall is restrained. Only levels with a connecting member are listed.
[-] Meshing	**(meshed walls only)**
Override Model's	Select this check box to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Used to specify the wall mesh type: • Quad only • Tri only • Quad dominant
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit concrete wall span properties set

The following properties are displayed in the **Properties Window** when you select an individual panel within an existing concrete wall.

Property	**Description**
[-] General	
Grade	The concrete grades that are available here are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.

Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Assume cracked	Cracked concrete sections have different analytical properties to uncracked concrete sections. See: Assume cracked
End 1 extension	The amount the wall is extended or trimmed back from end 1. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Concrete wall extensions
End 2 extension	The amount the wall is extended or trimmed back from end 2. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Concrete wall extensions
Reinforcement Layers	Number of layers of reinforcement to be used in the wall: • 1 • 2
[-] Releases	
Minor Top	• Fixed • Pinned • Continuous (incoming members pinned) The 'Continuous' option is only available for FE meshed walls. See: Wall releases

Minor Bottom	• Fixed • Pinned • Continuous (incoming members pinned) The 'Continuous' option is only available for FE meshed walls. See: Wall releases
[-] Design parameters	
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Relative humidity (RH) (Eurocode only)	Entered as a percentage.
Age of loading (Eurocode only)	Age at which loading is applied.
[-] Confinement reinforcement	
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
[-] Slenderness	
Major (Minor)	• Braced • Bracing
Effective length factor direction Major (Minor)	• Calculated • User input value
[-] Stiffness	

Use slab for calculation (upper major/ minor, lower major/minor)	For the unrestrained length calculation: • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See: Clear height
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
[-] Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.

edit concrete column properties set

The following properties are displayed in the **Properties Window** when you select one or more existing concrete columns.

For an individual column the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the column.
User Name	Can be used to override the automatically generated name if required.
Group	The automatically generated name for the column group.
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the grid along which the column is placed.
Concrete type	• Normal • Lightweight

Characteristic	***Column***
Material type	***Concrete***
Construction	***Concrete column***
Fabrication	• Reinforced • Precast Design of precast members is beyond scope in the current release.
Autodesign	• Cleared - the specified reinforcement will be checked during the design process. • Checked - reinforcement will be designed during the design process. See: Autodesign (concrete column)
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See: Autodesign (concrete column)
Rotation	The rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Assume extra floors supported	The number of extra floors assumed to be supported by the column.
[-] All stacks	
Section	The section applied to the column that is created. See: Section (concrete column)

Concrete class	The concrete grades that are available here are set from the Materials button on the Home ribbon.
Linearity	Although beams can be curved, columns are restricted to straight
[-] Releases	
Free end top	When this check box is selected - defines a free end at the top of the top-most stack of the column (available for the top-most stack only).
Fixity top	The fixity at the top of individual stacks can be set to: • Fixed • Pinned If you require to pin one direction but fix the other you can do this from the Column Property Dialog. See: Column releases
Fixity bottom	Same as 'Fixity top'.
Free end bottom	When this check box is selected - defines a free end at the bottom of the bottom-most stack of the column (available for the top-most stack only).
[-] Alignment	
Major snap level	Alignment of the major properties: • Top • Center • Bottom
Major offset	Used to offset the column from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the column from the snap point in the minor axis.
[-] Design control	

Assume cracked	Cracked concrete sections have different analytical properties to uncracked concrete sections. See: Assume cracked
Apply rigid zones	Unless cleared, rigid zones are automatically created at the connection between the column and any connecting beams. See: Application of rigid zones
[-] Design parameters	
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Relative humidity (RH) (Eurocode only)	Entered as a percentage.
Age of loading (Eurocode only)	Age at which loading is applied.
[-] Confinement reinforcement	
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
[+] Slenderness	
Direction 1 (Direction 2)	The bracing classification is set as either 'Braced' or as 'Bracing' in each of Building Directions 1 and 2. Whether Direction 1 or 2 applies to the design calculations in the column's major or minor direction then depends on the column's rotation angle.
Effective length factor direction 1 (direction 2)	The bracing effective length factor is set as either 'Calculated' or as 'User Input Value' in each of Building Directions 1 and 2. Whether Direction 1 or 2 applies to the design calculations in the column's major or minor direction then depends on the column's rotation angle.
[+] Stiffness	

Use slab for calculation (upper major/ minor, lower major/minor)	For the unrestrained length calculation: • if a slab exists at a stack end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next stack instead. See: Clear height
Lambda crit check or drift check	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Wind drift check	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
[-] Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

[-] Reinforcement	
Rib type - vertical (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - vertical (Head Code ACI)	• Plain • Deformed
Class - vertical	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Rib type - confinement (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - confinement (Head Code ACI)	• Plain • Deformed
Class - link(tie)/confinement	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
[-] Stack 1,2,3 etc.	In a multi-stack column properties can be entered for a specific stack, over-riding those defined at the **All stacks** level. See: [-] All stacks
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Level	
Count the floor as being supported (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the imposed load reductions. See: Load reductions
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions

edit concrete beam properties set

The following properties are displayed in the **Properties Window** when you select one or more existing concrete beams.

For an individual beam the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the column.
User Name	Can be used to override the automatically generated name if required.
Group	The automatically generated name for the beam group.
Plane	Indicates the level at which the beam is placed.
Concrete Type	• Normal • Lightweight
Characteristic	***Beam***
Material type	***Concrete***
Construction	***Concrete beam***
Fabrication	• Reinforced • Post tensioned • Precast Design of precast and post tensioned beams is beyond scope in the current release.
Autodesign	• Cleared - the specified reinforcement will be checked during the design process. • Checked - reinforcement will be designed during the design process. See: Autodesign (concrete beam)

Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement.
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). • Degrees0 • Degrees90 • Degrees180 • Degrees270 • Angle See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Allow automatic join end 1 (end 2)	When this check box is selected - the end in question will be automatically joined to a suitable connecting concrete beam end during design process or when the 'Beam Lines' command is run, (providing the Beam Lines limiting criteria specified in Model Settings are met.)
[-] All Spans	
Section	The section applied to the beam that is created.
Concrete class	The concrete grades that are available here are set from the Materials button on the Home ribbon.
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Maximum facet error

[-] Releases	
Free end 1 (Free end 2)	When this check box is selected - defines a cantilever end. See: Beam releases
Fixity end 1 (Fixity end 2)	• Moment • Pin • Fully fixed See: Beam releases
Torsional load release end 1 (end 2)	Check one end to define a torsional release.
[-] Load Reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[-] Alignment	
Major snap level	Alignment of the minor properties: • Top • Center • Bottom
Major offset	Used to offset the beam from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the beam from the snap point in the minor axis.
[-] Deflection limits	
Limit for immediate live load deflection (ACI only)	Input is as a proportion of span length.

Limit for total deflection affecting sensitive finishes (ACI only)	Input is as a proportion of span length.
Calculate total deflection at design time (AS only)	This option influences how the deflection is calculated. See **Reference Guides > Concrete Design - AS 3600 > Beam Design** for further details.
Calculate deflection after installation of finishes (AS only)	This option influences how the deflection is calculated. See **Reference Guides > Concrete Design - AS 3600 > Beam Design** for further details.
[-] Design control	
Structure supporting sensitive finishes (ACI only)	This option influences the deflection check that is performed.
Increase reinforcement if deflection check fails (Eurocode BS and IS only)	Select in order to increase the reinforcement during the auto-design process if the deflection check fails.
Permissible increase in reinforcement	Specify the max percentage increase in reinforcement that is allowed in order to satisfy the deflection check.
Consider flanges	Select in order to consider flanges in the concrete beam design calculations - once checked additional fields are displayed for specifying an allowance for openings. Flange dimensions can only be calculated by editing the beam once it has been positioned and slabs have been defined. (In this case a 'Calculate flanges' button is also displayed, this can be clicked in order to automatically calculate the flange dimensions based on the adjoining slabs.) See: Use of beam flanges
Include flanges in analysis	This property is only displayed when the 'Consider flanges' option has been selected. Select this check box to use flanged beam properties when the analysis is performed. See: Use of beam flanges

Isolated beam (ACI only)	This property is only displayed when the 'Consider flanges' option has been selected. Select this check box in order to apply ACI 318 clause 8.12.4. When the check is performed, if the flange geometry does not meet the requirements the flanges are ignored. See: Use of beam flanges
User defined flange (left/right)	This property is only displayed when 'Include flanges in analysis' has been checked. If you clear this check box the flange depth and the effective flange width are determined automatically. If you select the check box, two new properties are displayed for defining 'Flange width' and 'Flange depth'. See: Use of beam flanges
Assume cracked	Cracked concrete sections have different analytical properties to uncracked concrete sections. See: Assume cracked
[-] Design parameters	
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface. Different values of nominal cover can be specified to the beam edges, sides and ends.
[-] Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.

[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).
[-] Reinforcement	
Rib type - longitudinal (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - longitudinal (Head Code ACI)	• Plain • Deformed
Class - longitudinal	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Rib type - links (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type - stirrups (Head Code ACI)	• Plain • Deformed
Class- links(stirrups)	The available classes are specified in the Materials dialog on the Home ribbon.
Top longitudinal bar pattern	Choose from one of the three standard patterns (which can be setup in Design Options) to control the top bar arrangement when the beam is auto-designed. See: Longitudinal reinforcement (concrete beam)
Bottom longitudinal bar pattern	Choose from one of the two standard patterns (which can be setup in Design Options) to control the bottom bar arrangement when the beam is auto-designed. See: Longitudinal reinforcement (concrete beam)
[+] Span 1, Span 2 etc.	In a multi-span beam properties can be entered for a specific span, over-riding those defined at the **All spans** level. See: [-] All Spans

Timber Member Properties

Topics in this section

- Timber property assumptions
- timber column properties set
- timber beam properties set
- timber brace properties set
- edit timber column properties set
- edit timber beam properties set
- edit timber brace properties set
- edit timber truss properties

Timber property assumptions

The engineer shall verify that the properties of the selected grades are those that are characteristic of the material, are obtained from up-to-date manufacturer's data and are applicable to the analysis model. Analysis Modification Factors must also be confirmed to be correct.

(Head Code Eurocode)

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.
- The glulam grade being used complies 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.

(Head Code ACI/AISC)

The following assumptions have been made:

- The moisture content of the structural composite lumber or glulam section 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.
- The glulam section will be loaded perpendicular to the wide faces of the laminations (bending about the x-x axis).
- The glulam grade being used complies with the characteristic properties outlined in AWC NDS supplement 2012.
- 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.

edit timber column properties set

The following properties are displayed when you select one or more existing timber columns.

For an individual column the properties can be displayed in a dialog by right-clicking and selecting **Edit**.`

Property	**Description**
[-] General	
Name	The automatically generated name for the column.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the column belongs.
Top Level	Define the vertical position of the column top. (This property is only displayed in 2D Floor Views)
Base Level	Define the vertical position of the column base. (This property is only displayed in 2D Floor Views)
Plane	Indicates the grid along which the column is placed.
Characteristic	**Column**
Material type	**Timber**
Autodesign	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Design section order	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Rotation	The rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.

Major snap level	Define alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the column from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the column from the snap point in the minor axis.
Assume extra floors supported	The number of extra floors assumed to be supported by the column. See: Load reductions
[-] All stacks	
Section	The section applied to the column that is created.
Grade	Define the timber grade of the column. The available grades are set from the Materials button on the Home ribbon.
[+] Releases	Define the end releases. See: Column releases
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Level	

[+] Load Reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Timber Columns

edit timber beam properties set

The following properties are displayed when you select one or more existing timber beams.

For an individual beam the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the beam.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the beam belongs.
Plane	Indicates the level at which the beam is placed.
Characteristic	**Beam**
Material type	**Timber**
Fabrication	• Timber • Glulam • Structural Composite Lumber
Autodesign	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Design section order	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.

Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the beam from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the beam from the snap point in the minor axis.
[-] All spans	
Section	The section initially applied to the beam that is created.
Grade	Define the timber grade of the beam. The available grades are set from the Materials button on the Home ribbon.
Linearity	Indicates if the beam is straight or curved.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Releases	Define the end releases. See: Beam releases
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.

[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Timber Beams

edit timber brace properties set

The following properties are displayed when you select one or more existing timber braces.

For an individual brace the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	Description
[-] General	
Name	The automatically generated name for the brace.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the brace belongs.
Plane	Indicates the level at which the brace is placed.
Characteristic	**Brace**
Material type	**Timber**
Fabrication	• Timber • Glulam • Structural Composite Lumber
Autodesign	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Design section order	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.

[+] Alignment	
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the brace from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the brace from the snap point in the minor axis.
[-] All spans	
Section	The section applied to the brace.
Grade	Define the timber grade of the brace. The available grades are set from the Materials button on the Home ribbon.
Compression only	Specify if the brace is compression only.
Tension only	Specify if the brace is tension only.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Releases	Define the end releases. See: Brace releases
[+] Load reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions

[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Timber Braces

edit timber truss properties

The following properties are displayed when you select one or more existing timber trusses:

Property	Description
General	
Name	The automatically generated name for the truss is derived from the grid line selected.
User Name	Can be used to override the automatically generated name if required.
Autodesign	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Truss Top Members	
Section	Section size.
Grade	The timber grades that are available here are set from the Materials button on the Home ribbon.
Design section order	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Rotation	The default (Degrees0) aligns the major properties with the global Z axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Truss Bottom Members	
Section	Section size.
Grade	The timber grades that are available here are set from the Materials button on the Home ribbon.

Design section order	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Truss Internals	
Section	Section size.
Grade	The timber grades that are available here are set from the Materials button on the Home ribbon.
Design section order	Not Applicable: timber sections cannot be designed or checked in *Tekla Structural Designer*.
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Timber Trusses

Cold Formed Member Properties

Topics in this section

- create cold formed column properties set
- create cold formed beam properties set
- create cold formed brace properties set
- edit cold formed column properties set
- edit cold formed beam properties set
- edit cold formed brace properties set

- edit cold formed truss properties

create cold formed column properties set

The following properties are displayed when you create a cold formed column.

Property	Description
Top Level	Define the vertical position of the column top. (This property is only displayed in 2D Floor Views)
Base Level	Define the vertical position of the column base. (This property is only displayed in 2D Floor Views)
Continuous	Indicates if all stacks have the same properties. • Checked - all stacks have the same properties • Cleared - allows splices to be positioned; enabling different stacks to have different properties. (This property is only displayed in Structure, or Frame Views)
Characteristic	**Column**
Material type	**Cold formed**
Stress relieved (Head Code AS only)	Indicates whether cold formed member is stress relieved or not.
Use Automatic Grouping	Indicates if group names are created automatically: • Checked - Group names are created automatically • Cleared - Group names can be entered manually
Rotation	The rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify • Define - aligns the column to the angle of any grid line you select For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.

Grade	Define the steel grade of the column. The available grades are set from the Materials button on the Home ribbon.
Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Section	The section applied to the column that is created.
Major Alignment	Define alignment of the major properties: • Top • Center • Centroid • Bottom
Minor Alignment	Define alignment of the minor properties: • Left • Center • Centroid • Right
[+] Releases	Define the end releases. See: Column releases
[+] Instability Factor	Define if out of plane stability is prevented.

Prevent out of plane instability	
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

create cold formed beam properties set

The following properties are displayed when you create a cold formed beam.

Property	**Description**
Continuous	Define if the beam is single or multi-span. • Cleared - creates a single span beam • Checked -creates a multi-span continuous beam.
Characteristic	**Beam**
Material type	**Cold formed**
Stress relieved (Head Code AS only)	Indicates whether cold formed member is stress relieved or not.
Linearity	Indicates if the beam is straight or curved.
Use Automatic Grouping	Indicates if group names are created automatically: • Checked - Group names are created automatically • Cleared - Group names can be entered manually
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Grade	Define the steel grade of the beam. The available grades are set from the Materials button on the Home ribbon.

Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Section	The section applied to the beam that is created.
Major Alignment	Alignment of the major properties: • Top • Center • Centroid • Bottom
Minor Alignment	Alignment of the minor properties: • Left • Center • Centroid • Right
[+] Releases	Define the end releases. See: Beam releases
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.

[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

create cold formed brace properties set

The following properties are displayed when you create a cold formed brace.

Property	Description
Type	Define the brace pattern to be applied.
Characteristic	**Brace**
Material type	**Cold formed**
Stress relieved (Head Code AS only)	Indicates whether cold formed member is stress relieved or not.
Use Automatic Grouping	Indicates if group names are created automatically: • Checked - Group names are created automatically • Cleared - Group names can be entered manually
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Grade	Define the steel grade of the brace. The available grades are set from the Materials button on the Home ribbon.
Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.

Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Section	The section initially applied to the brace that is created.
Major Alignment	Alignment of the major properties: • Top • Center • Centroid • Bottom
Minor Alignment	Alignment of the minor properties: • Left • Center • Centroid • Right
[+] Releases	Define the end releases. See: Brace releases
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit cold formed column properties set

The following properties are displayed when you select one or more existing cold formed columns.

For an individual column the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the column.

User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the column belongs.
Top Level	Define the vertical position of the column top. (This property is only displayed in 2D Floor Views)
Base Level	Define the vertical position of the column base. (This property is only displayed in 2D Floor Views)
Plane	Indicates the grid along which the column is placed.
Characteristic	**Column**
Material type	**Cold formed**
Stress relieved (Head Code AS only)	Indicates whether cold formed member is stress relieved or not.
Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Rotation	The rotation of the member around its local x-axis. • 0, 90, 180, -90 - aligns the column to the global axes • Angle - aligns the column to the exact rotation angle you specify For vertical columns the default (Degrees0) aligns local y with the global X axis. See: Rotation angle

Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Define alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the column from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the column from the snap point in the minor axis.
Assume extra floors supported	The number of extra floors assumed to be supported by the column. See: Load reductions
[-] All stacks	
Section	The section applied to the column that is created.
Grade	Define the steel grade of the column. The available grades are set from the Materials button on the Home ribbon.
[+] Releases	Define the end releases. See: Column releases
[+] Size constraints	Size Constraints are only applicable when Autodesign is checked. They allow you to impose upper and lower limits to the section depth and width. See: Size Constraints
[+] Instability Factor Prevent out of plane instability	Define if out of plane stability is prevented.

Lambda crit check or drift check	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Wind drift check	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Level	
[+] Load Reductions Count the floor as being supported (Head Code EC, BS or AS)	If selected, the floor will be treated as supported when calculating the imposed load reductions. See: Load reductions
[+] Load Reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Steel Columns and Cold Formed Columns

edit cold formed beam properties set

The following properties are displayed when you select one or more existing cold formed beams.

For an individual beam the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the beam.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the beam belongs.
Plane	Indicates the level at which the beam is placed.
Characteristic	**Beam**
Material type	**Cold formed**
Stress relieved (Head Code AS only)	Indicates whether cold formed member is stress relieved or not.
Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.

Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the beam from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the beam from the snap point in the minor axis.
[-] All spans	
Section	The section initially applied to the beam that is created.
Grade	Define the steel grade of the beam. The available grades are set from the Materials button on the Home ribbon.
Linearity	Indicates if the beam is straight or curved.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Releases	Define the end releases. See: Beam releases
[+] Instability Factor	Define if out of plane stability is prevented.

Prevent out of plane instability	
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Steel Beams and Cold Formed Beams

edit cold formed brace properties set

The following properties are displayed when you select one or more existing cold formed braces.

For an individual brace the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the brace.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the brace belongs.
Plane	Indicates the level at which the brace is placed.
Characteristic	**Brace**
Material type	**Cold formed**
Stress relieved (Head Code AS only)	Indicates whether cold formed member is stress relieved or not.
Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)

Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Rotation	Define the rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
[+] Alignment	
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the brace from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the brace from the snap point in the minor axis.
[-] All spans	
Section	Define the section applied to the brace.

Grade	Define the steel grade of the brace. The available grades are set from the Materials button on the Home ribbon.
Compression only	Specify if the brace is compression only.
Tension only	Specify if the brace is tension only.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Releases	Define the end releases. See: Brace releases
[+] Load reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Steel Braces and Cold Formed Braces

edit cold formed truss properties

The following properties are displayed when you select one or more existing cold formed trusses.

Property	Description
Name	The automatically generated name for the truss is derived from the grid line selected.
User Name	Can be used to override the automatically generated name if required.

Autodesign (Head Code EC, BS or AS)	• Cleared - the specified section will be checked during the design process. • Checked - sections from the design section order will be considered during the design process. See: Autodesign (steel)
Autodesign (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Design section order (Head Code EC, BS or AS)	The design order file from which a section size will be selected if Autodesign is employed. See: Design Section Order
Design section order (Other Head Codes)	Not Applicable: cold formed sections cannot be designed or checked.
Section	The section size of the truss member.
Grade	The steel grade of the truss member. The available grades are set from the Materials button on the Home ribbon.
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Steel Trusses

Cold Rolled Properties

Topics in this section

- create cold rolled section properties set
- edit cold rolled section properties set

create cold rolled section properties set

The following properties are displayed when you create any type of cold rolled section.

Property	Description
Element Parameters	
Characteristic	The chosen type of cold rolled section.
Material type	***Cold rolled***
Construction	The chosen type of cold rolled section.
Fabrication	***Cold rolled***
Linearity	Straight
Use Automatic Grouping	• Cleared - Group names can be entered manually • Checked - Group names are created automatically
Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Grade	The steel grades that are available here are set from the Materials button on the Home ribbon.
Autodesign	Not Applicable: cold rolled sections cannot be designed or checked in *Tekla Structural Designer*.
Design section order	Not Applicable: cold rolled sections cannot be designed or checked in *Tekla Structural Designer*.
Section	The cold rolled section size.
Major Alignment	Alignment of the major properties: • Top • Center • Centroid • Bottom

Minor Alignment	Alignment of the minor properties: • Left • Center • Centroid • Right
[+] Releases	Define the end releases. See: Beam releases
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit cold rolled section properties set

The following properties are displayed when you select one or more existing cold rolled sections.

For an individual section the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the member.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs.
Plane	Indicates the level at which the member is placed.
Characteristic	The chosen type of cold rolled section.
Material type	***Cold rolled***
Construction	The chosen type of cold rolled section.
Fabrication	***Cold rolled***
Autodesign	Not Applicable: cold rolled sections cannot be designed or checked in *Tekla Structural Designer*.
Design section order	Not Applicable: cold rolled sections cannot be designed or checked in *Tekla Structural Designer*.

Rotation	The rotation of the member around its local x-axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane). See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Major snap level	Alignment of the major properties: • Top • Center • Centroid • Bottom
Major offset	Used to offset the brace from the snap point in the major axis.
Minor snap level	Alignment of the minor properties: • Left • Center • Centroid • Right
Minor offset	Used to offset the brace from the snap point in the minor axis.
[-] All spans	
Section	The cold rolled section size.
Grade	The available grades are set from the Materials button on the Home ribbon.
Gamma Angle	Derived from the rotation angle, this value defines the orientation of the major axis of the section. See: Gamma angle
[+] Releases	Define the end releases. See: Brace releases
[+] Load reductions KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See: Load reductions

[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Slab and Mat Properties

Topics in this section

- Slabs Properties
- Mat properties
- Openings, Overhangs and Column Drops
- Rotation angle for panels

Slabs Properties

Topics in this section

- slab properties
- slab on beams item properties set
- flat slab item properties set
- precast item properties set
- steel deck item properties set
- timber deck item properties set
- composite slab item properties set
- edit slab on beams item properties set
- edit flat slab item properties set
- edit precast item properties set
- edit steel deck item properties set
- edit timber deck item properties set
- edit composite slab item properties set

slab properties

These properties are displayed by selecting an existing parent slab in the Slabs branch of the Structure Tree.

Property	Description
[-] Slab General	

Name	The automatically generated name for the slab.
User Name	Can be used to override the automatically generated name if required.
Slab type	Define the slab type.
Deck type	Define the slab type.
Decomposition	Depending on the slab type either: • One-way • Two-way
[-] Slab parameters, Design parameters etc.	The remaining parameters that are displayed depend on the slab item - refer to the individual slab item properties.

edit slab on beams item properties set

The following properties are displayed when you select one or more existing slab on beams.

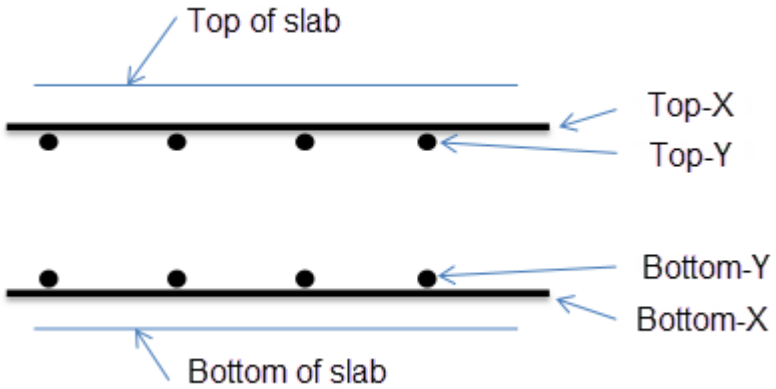
For an individual slab the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the slab. See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.

Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant
Auto-design	For panels in **Auto-design** mode, $A_{s,prov}$ is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Select bars starting from	This option controls the starting point for auto-design procedures. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement.
Plane	Indicates the level at which the slab item is placed.
[-] Slab general	
Name	The parent slab name.
Slab type	***Slab on Beams***
Deck type	• Reinforced concrete • Post tension Design of post tensioned slabs is beyond scope in the current release.
Decomposition	• One-way • Two-way Design of one way spanning slabs is beyond scope in the current release.
[-] Slab parameters	
[+] Slab properties	
Depth	Specifies the slab thickness.

Concrete type	• Normal • Lightweight
Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • None • Semi-Rigid • Rigid
[+] Design parameters	
Permanent load ratio (Head Code Eurocode)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width (Head Code Eurocode)	• 0.2 • 0.3 • 0.4
[-] Design parameters	
Adjustment ratio direction X	This factor is applied to the enclosing length X in order to manually adjust the X direction span in the span-effective depth check. See: Slab on beam idealized panels
Adjustment ratio direction Y	This factor is applied to the enclosing length Y in order to manually adjust the Y direction span in the span-effective depth check. See: Slab on beam idealized panels

Enclosing length X	The automatically calculated span length in the X direction. See: Slab on beam idealized panels
Enclosing length Y	The automatically calculated span length in the Y direction. See: Slab on beam idealized panels
Adjusted length X	The adjusted span length in the X direction. See: Slab on beam idealized panels
Adjusted length Y	The adjusted span length in the Y direction. See: Slab on beam idealized panels
Edge category start X	The assumed support condition at the start of the span in the X direction.
Edge category end X	The assumed support condition at the end of the span in the X direction.
Edge category start Y	The assumed support condition at the start of the span in the Y direction.
Edge category end Y	The assumed support condition at the end of the span in the Y direction.
Average stiffness ratio (Head Code ACI/AISC)	In the current release of Tekla Structural Designer the average stiffness ratio is a user defined value. (Default value = 1.0). The way in which the minimum thickness is calculated directly depends on this value.
[-] Live load reduction	
Reduce live loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. For mats, the bearing pressures are reduced as well.

[+] Reinforcement	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to "none" if required).  Select/clear **Outside layer in X direction** as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)
[-] All edges	
Linear	Uncheck in order to specify all edges as curved.
Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.
[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.
Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit flat slab item properties set

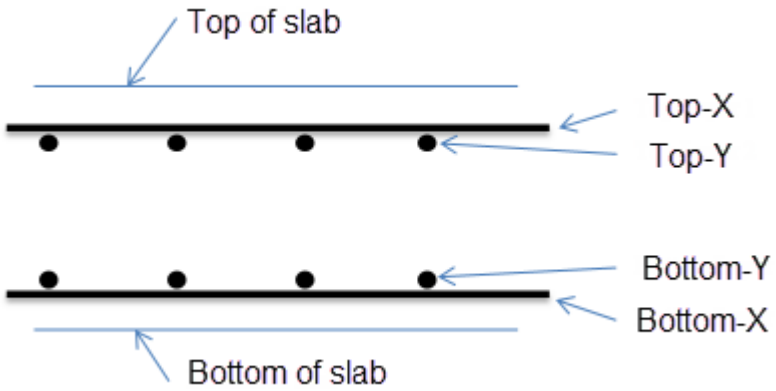
The following properties are displayed when you select one or more existing flat slab.

For an individual slab the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	

Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the slab. See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant
Auto-design	For panels in **Auto-design** mode, $A_{s,prov}$ is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Select bars starting from	This option controls the starting point for auto-design procedures. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement.
Plane	Indicates the level at which the slab item is placed.
[-] Slab general	
Name	The parent slab name.
Slab type	***Slab on Beams***

Deck type	• Reinforced concrete • Post tension Design of post tensioned slabs is beyond scope in the current release.
Decomposition	• Two-way
[-] Slab parameters	
[+] Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • None • Semi-Rigid • Rigid
[+] Design parameters	
Permanent load ratio (Head Code Eurocode)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.

Maximum crack width (Head Code Eurocode)	• 0.2 • 0.3 • 0.4
[-] Live load reduction	
Reduce live loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. For mats, the bearing pressures are reduced as well.
[+] Reinforcement	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to "none" if required).  Select/clear **Outside layer in X direction** as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)
[-] All edges	
Linear	Uncheck in order to specify all edges as curved.

Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.
[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.
Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit precast item properties set

The following properties are displayed when you select one or more existing precast slab.

For an individual slab the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	Description
[-] General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of the slab. Different angles can be specified for different panels within the slab. See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.

Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant
Auto-design	Design of precast slabs is beyond scope in the current release.
Plane	Indicates the level at which the slab item is placed.
[-] Slab general	
Slab type	**Precast**
Deck type	**Precast concrete planks**
Decomposition	**One-way**
[-] Slab parameters	
Depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Wet weight per area	Defines the overall slab wet weight per unit area (PC plank + topping)
Dry weight per area	Defines the overall slab dry weight per unit area (PC plank + topping)

Short term elastic modulus	Specifies the short term elastic modulus
Long term elastic modulus = Ecm divided by...	Factor by which the short term elastic modulus is divided to obtain the long term modulus
Diaphragm option	For new slabs only, sets the default diaphragm action for all slab items within the new slab. • None • Semi-Rigid • Rigid
[+] Decking Properties	
Country, Manufacturer, Reference, Gauge	Define properties of precast slab.
[+] Topping details	
Topping	Specifies type of topping: • None • Non-Structural • Structural
Depth	Specifies the depth of topping (if topping other than 'None' is selected).
[-] All edges	
Linear	Uncheck in order to specify all edges as curved.
Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.
[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.

Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit steel deck item properties set

The following properties are displayed when you select one or more existing steel deck.

For an individual deck the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of the deck. Different angles can be specified for different panels within the slab.See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant
Plane	Indicates the level at which the slab item is placed.
[-] Slab general	
Slab type	**Steel deck**
Deck type	**Steel plate**

Decomposition	• One-way • Two-way
[-] Slab parameters	
Depth	Specifies the slab thickness.
Diaphragm option	Sets the default diaphragm action for all slab items within the new slab. • None • Semi-Rigid • Rigid
[+] Decking Properties	
Self weight density	Define the self weight of the deck.
[+] Material properties	Define the steel properties
[-] All edges	
Linear	Uncheck in order to specify all edges as curved.
Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.
[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.
Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit timber deck item properties set

The following properties are displayed when you select one or more existing timber deck.

For an individual deck the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	Description
[-] General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of the deck. Different angles can be specified for different panels within the slab. See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant
Plane	Indicates the level at which the slab item is placed.
[-] Slab general	
Slab type	**Timber deck**
Deck type	**Timber**
Decomposition	**One-way**
[-] Slab parameters	
Depth	Specifies the slab thickness.

Diaphragm option	Sets the default diaphragm action for all slab items within the new slab. • None • Semi-Rigid • Rigid
[+] Decking Properties	
Self weight density	Define the self weight of the deck.
[+] Material properties	Define the timber properties
[-] All edges	
Linear	Uncheck in order to specify all edges as curved.
Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.
[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.
Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

edit composite slab item properties set

The following properties are displayed when you select one or more existing composite slab.

For an individual slab the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the slab item.

User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the slab. See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the slab item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant
Auto-design	Design of composite slabs is beyond scope in the current release.
Plane	Indicates the level at which the slab item is placed.
[-] Slab general	
Slab type	***Composite slab***
Deck type	***Profiled metal decking***
Decomposition	***One-way***
[-] Slab parameters	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class	Specifies the concrete grade.
Dry density	Specifies the dry concrete density

Wet density	Specifies the wet concrete density
Short term elastic modulus (Head Code AISC or EC)	Define the short term elastic modulus
Long term elastic modulus = Ecm divided by... (Head Code AISC or EC)	Factor by which the short term elastic modulus is divided to obtain the long term modulus
Short term ratio (Head Code BS)	Define the short term modular ratio
Long term ratio (Head Code BS)	Define the long term modular ratio
Diaphragm option	Set the default diaphragm action for all slab items within the new slab. • None • Semi-Rigid • Rigid
Ponding allowance option	If required, choose either value or percentage and then specify actual ponding allowance accordingly.
Apply ponding allowance to construction stage	Check to apply ponding allowance to construction stage.
Apply ponding allowance to composite stage	Check to apply ponding allowance to composite stage.
[+] Decking Properties	Define the properties of the profiled metal deck
[+] Reinforcement in slab for crack control or fire requirements	Define the reinforcement in slab for crack control or fire requirements

[-] All edges	
Linear	Uncheck in order to specify all edges as curved.
Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.
[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.
Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Mat properties

Topics in this section

- Foundation Mat properties
- foundation mat properties set
- edit foundation mat properties set

Foundation Mat properties

These properties are displayed by selecting an existing parent slab in the Foundation Mats branch of the Structure Tree.

Property	Description
[-] Slab General	
Name	The automatically generated name for the slab.
User Name	Can be used to override the automatically generated name if required.
Slab type	Foundation mat

Deck type	• Reinforced concrete • Post tension Design of post tensioned slabs is beyond scope in the current release.
Decomposition	Two-way
[-] Slab parameters	The slab parameters that are displayed depend on the mat item - refer to the individual mat item properties.

edit foundation mat properties set

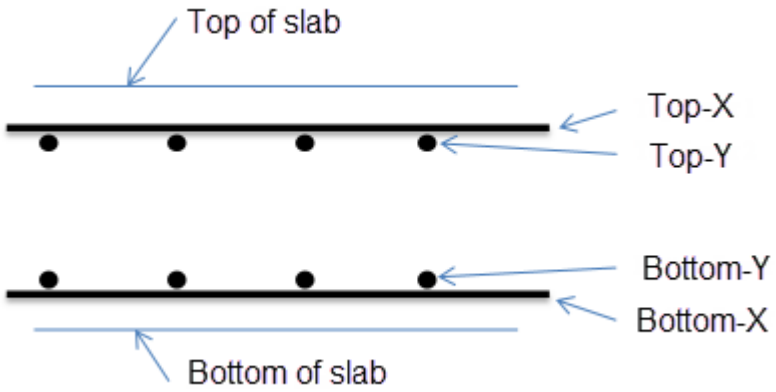
The following properties are displayed when you select one or more existing mat.

For an individual mat the properties can be displayed in a dialog by right-clicking and selecting **Edit**.

Property	**Description**
[-] General	
Name	The automatically generated name for the mat item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the mat. See: Rotation angle for panels
Include in diaphragm	If this option is cleared, the mat item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. Vertical offsets are not structurally significant

Auto-design	For panels in **Auto-design** mode, $A_{s,prov}$ is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Plane	Indicates the level at which the mat is placed.
[-] Slab general	
Name	The parent foundation mat name.
Slab type	***Foundation mat***
Deck type	• Reinforced concrete • Post tension Design of post tensioned slabs is beyond scope in the current release.
Decomposition	• Two-way
[-] Slab parameters	
[+] Slab properties	
Depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density

Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • None • Rigid
[+] Design parameters	
Permanent load ratio (Head Code Eurocode)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width (Head Code Eurocode)	• 0.2 • 0.3 • 0.4
[-] Soil parameters	
Use Ground Bearing Springs	Select to specify that the mat is ground bearing.
Allowable Bearing Pressure	The allowable bearing pressure is only required when the 'Use Ground Bearing Springs' option is selected.
[+] Ground Stiffness	The following properties are only displayed when the 'Use Ground Bearing Springs' option is selected.
Type	• Spring Linear • Spring Non-Linear
Stiffness	Only displayed when the Type is set to 'Spring Linear'.
Stiffness -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Tension limit -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Stiffness +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Compression limit +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Horizontal Support	• Fixed • Free • Spring

% of vertical spring stiffness	This field is displayed when the Horizontal Support is set to 'Spring'.
[-] Live load reduction	
Reduce live loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. For mats, the bearing pressures are reduced as well.
[+] Reinforcement	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to "none" if required).  Select/clear **Outside layer in X direction** as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)
[-] All edges	
Linear	Uncheck in order to specify all edges as curved.
Curvature	Defines the amount of curvature to all edges. Only displayed when **Linear** is cleared.

[-] Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify a curved edge.
Curvature	Defines the amount of curvature. Only displayed when **Linear** is cleared.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Openings, Overhangs and Column Drops

Topics in this section

- slab opening and mat opening properties
- slab overhang and mat overhang properties
- column drop properties

slab opening and mat opening properties

This command is used to create an opening in an existing slab or deck panel with properties as specified in the Properties Window.

Property	Description
General	
Opening Type	From the drop list choose: • Rectangular • Circular
Rotation Angle	For rectangular openings, this is used to specify the rotation of the opening in plan.
Center	X and Y coordinates to the center of the opening
Width and Height	For rectangular openings, the size of the opening.
Radius	For circular openings, the size of the opening.

Related topics

- Modeling slab and mat openings

slab overhang and mat overhang properties

This command is used to create an overhanging slab or deck panel. The overhang will adopt the properties specified in the Properties Window.

Property	Description
General	
Edge Parallel	Clear this check box in order to specify an overhang with a curved edge.
Curvature	This property only exists for overhangs which are not set as parallel. It is used to specify the maximum extent of the circular curve.
Tapered	Select this check box if the width of the overhang varies along the length supported edge.
Width1	The width of the overhang.
Width2	For tapered overhangs, the width of the overhang at end 2.

Related topics

- Adding overhangs to existing slab or mat edges

column drop properties

This command is used to create a column drop in a concrete slab panel. The drop dimensions are specified in the Properties Window.

Property	Description
General	
Depth	The overall depth of the drop measured from the top of the slab.
Plan Breadth	The breath of the drop.
Plan Width	The width of the drop.

For the drop panel to be inserted correctly, it is important that in the construction level dialog the setting out point 'Type' for the slab level in question is set as S.S.L. not T.O.S.

Related topics

- Modeling column drops

Rotation angle for panels

The rotation angle is used to control the orientation of the panel span.

Panels defined by a level

When the rotation angle = 0

- Span direction aligns with global X

When the rotation angle = 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

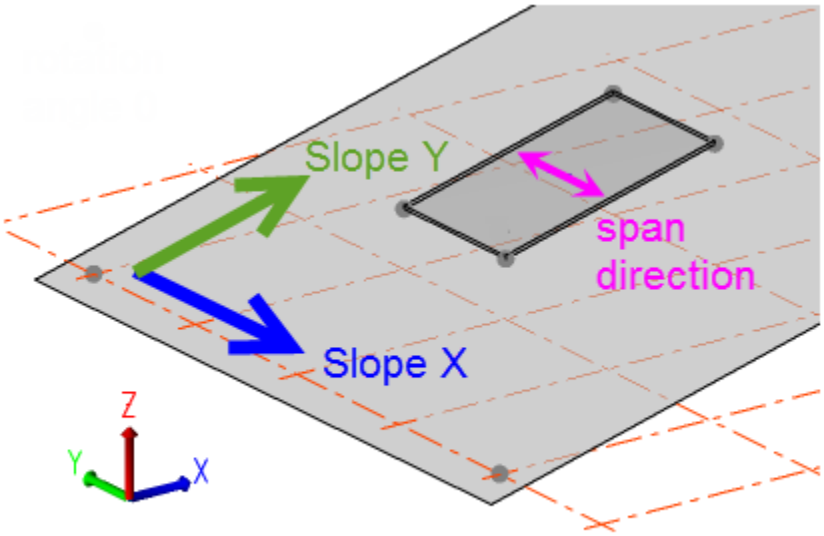
- Span direction is vertical

Panels defined by a sloped plane

In a sloped plane positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

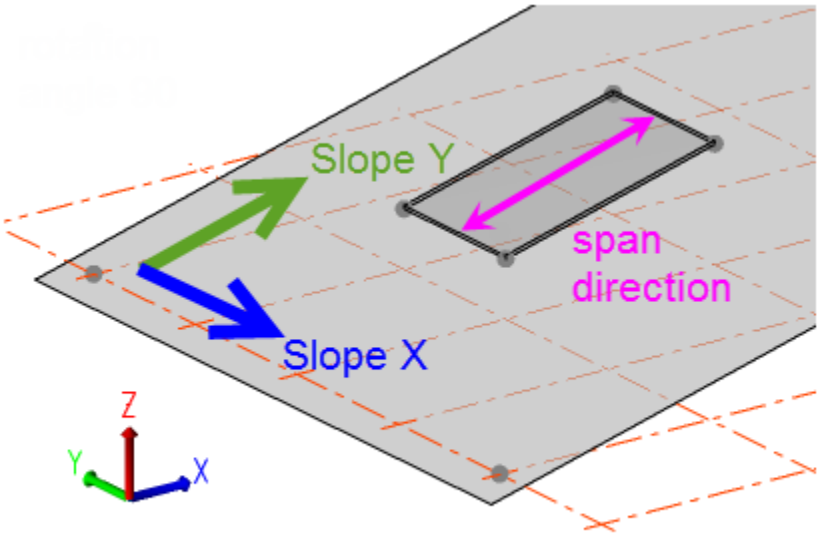
When the rotation angle = 0

- Span direction aligns with X direction of the slope (i.e. perpendicular to the slope, as shown below)



When the rotation angle = 90

- Span direction aligns with Y direction of slope (i.e. parallel to the slope, as shown below)



Roof Panel and Wall Panel Properties

Topics in this section

- Roof panel properties
- Wall panel properties

Roof panel properties

The 'Create Roof' command is used to create a roof panel.

Once created, the panel properties can then be edited in the Properties Window:

Property	Description
General	
Plane	Describes the level at which the panel was placed.
Rotation angle	This value determines the following where appropriate: • Span Direction for 1-way load decomposition • Orientation of semi-rigid 2D elements in the Solver Model See: Rotation angle for panels
Include in diaphragm	When this check box is selected, the roof panel is meshed to form a diaphragm of semi-rigid 2D elements.

Thickness	Roof panel thickness. Only displayed/applicable if 'include in diaphragm' is checked.
Youngs Modulus	Youngs Modulus. Only displayed/applicable if 'include in diaphragm' is checked.
Shear Modulus	Shear Modulus. Only displayed/applicable if 'include in diaphragm' is checked.
Temperature coefficient	Temperature coefficient. Only displayed/applicable if 'include in diaphragm' is checked.
Divide Stiffness by	Used to adjust the roof panel stiffness. Only displayed/applicable if 'include in diaphragm' is checked.
Decompose only wind and snow	Unless this box is checked a 'Slab/roof overlap' validation error will be produced if the roof panel overlaps a slab panel (for example on a flat-roofed building). This is because gravity loads cannot be decomposed by both panels types simultaneously. Checking the box resolves the validation issue as it ensures the roof panel is only used to decompose wind loads, and snow loads applied to roof panels by the Snow Wizard.
RoofType	The 'Default' option treats pitched roofs as monopitch. The droplist allows you to more accurately specify the roof type as 'Duopitch', 'Mansard', 'Hip' etc. If the type is changed after the wind model has already been established, you will also have to run 'Update Zones' to reinstate the zoning.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Roof Panels

Wall panel properties

The 'Create Wind Wall' command is used to create a wall panel.

Once created, the panel properties can then be edited in the Properties Window:

Property	Description
General	
Plane	Describes the plane in which the panel was placed.
Rotation angle	Describes the panel span direction as an angle, 0° is horizontal and 90° is vertical.
IsParapet	Check to indicate the panel is to be treated as a parapet in the wind analysis.

After the Wind Wizard has been run, extra panel properties are then appended as follows:

Property	Description
General	
Gap	Where the funneling gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funneling. A zero gap value explicitly means ignore funneling, for example where this building and the adjacent one are sheltered by upwind buildings.
Solidity	If you indicate that the wall panel is a parapet, then you also need to indicate the Solidity of the parapet. (Walls that are not parapets automatically adopt a solidarity of 1.0).
Status	Indicates whether the panel is valid or not.
Decompose to	Loads can be set to be decomposed to: • Members • Nodes (Default) • Rigid Diaphragms See: Wind Wall Panel Load Decomposition
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs)

Related topics

- Modeling Wall Panels

Support, Element and Bearing Wall Properties

Topics in this section

- Support properties
- Element properties
- Bearing wall properties
- Definition of rotation angle and gamma angle

Support properties

The 'Create Support' command is used to apply additional supports to the model.

When a support is first created, it's properties will be taken from those displayed in the Properties Window at that particular time.

You should edit the properties to suit the support being created:

Property	Description
General	
Name	(This property is only displayed when editing existing supports) The name displayed here is automatically created based on the grid point location.
User Name	(This property is only displayed when editing existing supports) You can enter a user name to replace the automatically created name if required.
Plane	(This property is only displayed when editing existing supports) Describes the level at which the support was placed.
3 Grid Points	(This property is only displayed when creating new supports) • Cleared - support properties are defined in accordance with the global coordinate system. • Checked - a user defined coordinate system is applied to the support. (After clicking where you want to create the support, the second click defines the x direction and the third click defines the y direction.)
Fx, Fy, Fz	The translational degrees of freedom can be set as either Free, or Fixed in each direction.
Mx, My, Mz	The rotational degrees of freedom can be set as either Free, or Fixed in each direction.

Angles	
Inclination, Azimuth and Rotation	When creating new supports, the angles are calculated automatically depending on the placement method (3 Grid Points on/off). When editing existing supports, the angles can be edited in order to redefine the direction in which the support acts.
Translational stiffness x, y, and z	
Type	In order to define a translational spring in a particular direction, the translational degree of freedom in the same direction must first be set to Free. The available types are then: • Release - (i.e. zero translational stiffness) • Spring Linear • Spring Non-linear
Stiffness	• Spring Linear - a single stiffness value is entered, which acts in both the positive and negative directions. • Spring Non-linear - two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction..
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).
Rotational stiffness x, y, and z	
Type	In order to define a rotational spring in a particular direction, the rotational degree of freedom in the same direction must first be set to Free. The available options are then: • Release - (i.e. zero rotational stiffness) • Spring Linear • Spring Non-linear • Nominally Pinned • Nominally Fixed
Stiffness	• Spring Linear - a single stiffness value is entered, which acts in both the positive and negative directions.

	• Spring Non-linear - two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.
Stiffness Percentage	• Nominally Pinned - 10% (i.e. 10% * 4 EI/L) • Nominally Fixed - 100% (i.e. 100% * 4 EI/L) In the above equations, L is the length from the base of the column to the level of the next column stack that is denoted as a floor, or it is the distance to the top of the column if this is shorter.
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).

Related topics

- Modeling Supports

Element properties

The following properties are displayed when you create a new analysis element..

Analysis elements are not designed.

Property	Description
[-] Element Parameters	
Characteristic	***Analysis element***
Material type	• Steel • Concrete • Timber • General material • Cold formed • Cold rolled The material is only displayed when the element type is 'Beam', 'Truss', 'Tension only', or 'Compression only'.

Element type	• Beam • Truss - (used for braces & truss internals) • Tension only - (e.g. tension only brace) • Compression only - (e.g. compression only brace) • Linear axial spring • Linear torsional spring • Non-linear axial spring • Non-linear torsional spring
Use Automatic Grouping	• Cleared - Group names can be entered manually • Checked - Group names are created automatically
Rotation	The rotation of the element around its local x-axis. The default (Degrees0) aligns the element properties with the global Z axis, (provided that the element has not been specifically defined within an incline plane). • Degrees0 • Degrees90 • Degrees180 • Degrees270 • Angle See: Rotation angle
Rotation Angle	To enter an angle directly, set the above Rotation to 'Angle'.
Grade	The grades that are available here are set from the Materials button on the Home ribbon.
Ax	Specifies the area of the section.
A parallel to minor	Specifies the shear area parallel to the minor axis.
A parallel to major	Specifies the shear area parallel to the major axis.
Ix	Specifies the torsional inertia.
I major	Specifies the major axis inertia.
I minor	Specifies the minor axis inertia.
[+] Releases	Define the end releases. See: Beam releases

[+] Instability Factor Prevent out of plane instability	• Define if out of plane stability is prevented. This property is not displayed when the material is set as concrete.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

- Modeling Steel Beams and Cold Formed Beams

Bearing wall properties

The following properties are displayed when you create a bearing wall

Property	Description
[-] General	
Top Level	Specifies the top level for the wall. (When creating a new member, this property is only displayed in 2D Floor Views)
Base Level	Specifies the bottom level for the wall. (When creating a new member, this property is only displayed in 2D Floor Views)
AutomaticGenerate Support	• Cleared - a series of point supports will only be created if the 'Generate support' property is checked. • Checked - a series of point supports will only be created if no members/slabs capable of providing support exist under the wall. See: Automatic support generation
Generate support	• Cleared - no supports are created under the wall. • Checked - a series of point supports are created under the wall. When supports are created, their degrees of freedom are as specified in the 'Wall support' area of the bearing wall properties.
Thickness	The thickness of the wall.

Alignment	Alignment of the wall: • Front • Back • Middle
Material type	• General • Timber • Concrete
Grade	The grade for the selected material.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels. See: User Defined Attributes (UDAs).

Related topics

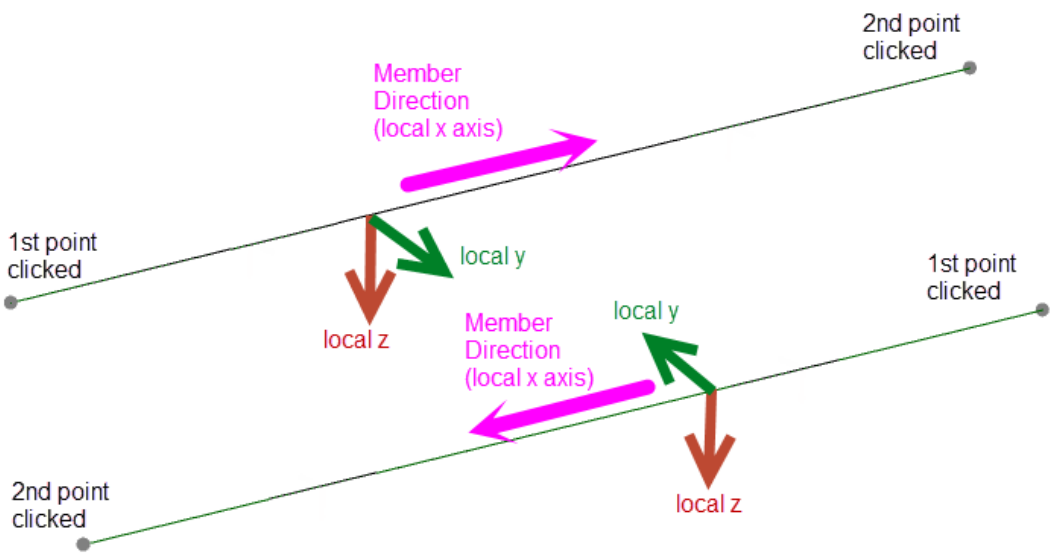
Definition of rotation angle and gamma angle

Rotation angle
• Gamma angle

Rotation angle

The rotation angle is used to control the orientation of the member's local z axis.

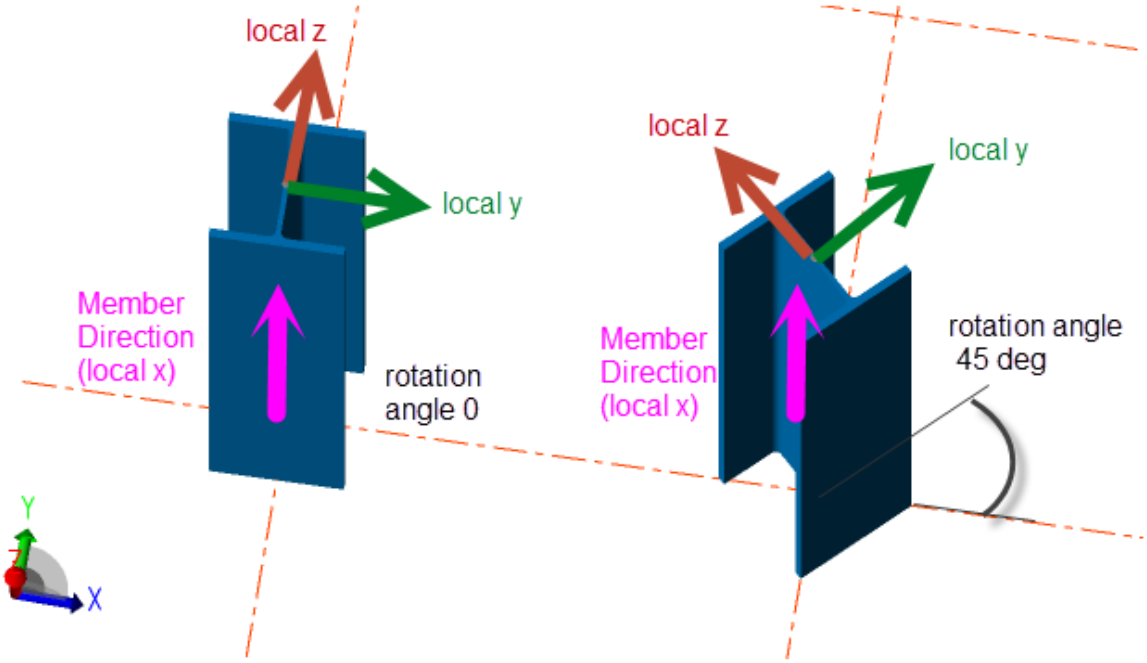
It is measured about the member's local x axis, which is itself defined by the line between the first and second point clicked when defining the member.



Entering a positive rotation angle results in a clockwise rotation of y and z about x looking towards positive x.

The local z axis is therefore orientated by the rotation angle as follows, but noting that special cases exist when the local x axis of the member is vertical, or when the member is defined within a sloped plane.

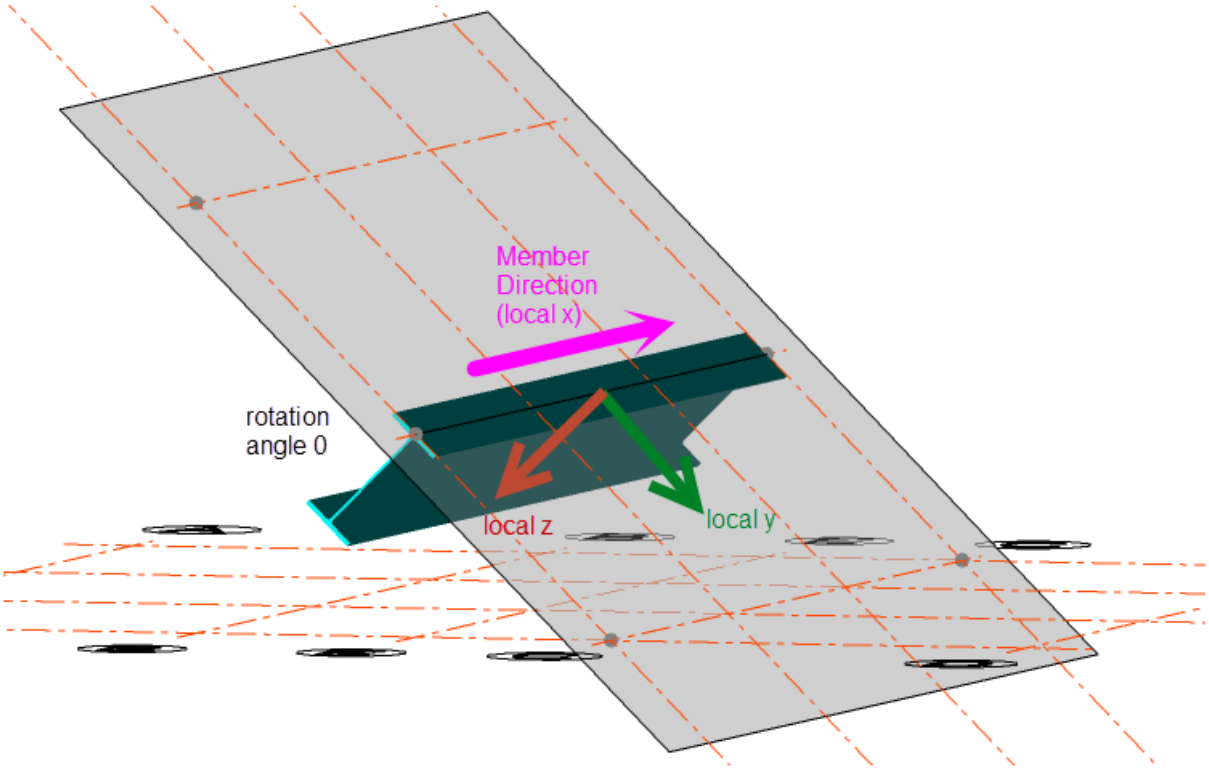
For vertical members only



When the rotation angle = 0

- Local y aligns with global X
- Local z according to the 'right hand rule'.

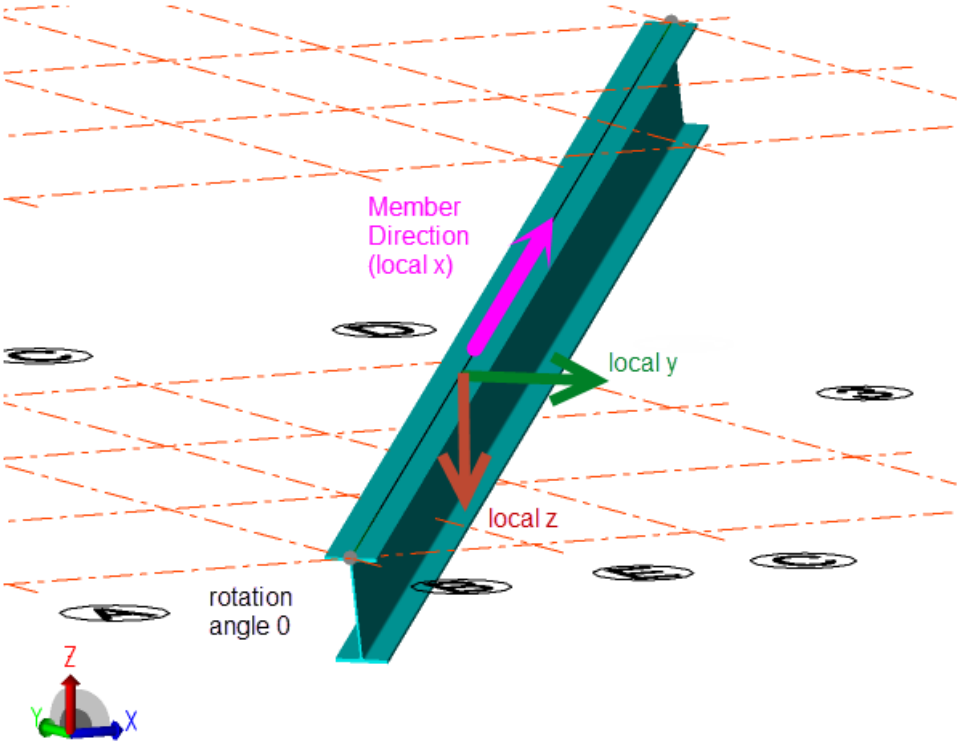
Members defined by a sloped plane



When the rotation angle = 0

- Local z is perpendicular to the sloped plane. The global z component of the local z axis is always negative
- Local y according to the 'right hand rule'.

For all other members



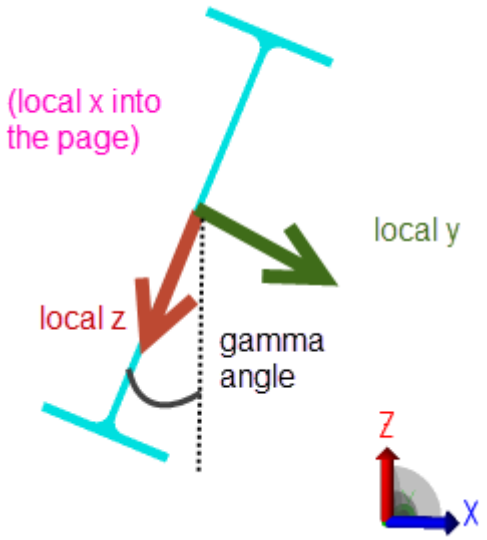
When the rotation angle = 0

- Local z lies in the plane created by the local x axis and the global z axis. The global z component of the local z axis is always negative
- Local y according to the 'right hand rule'.

Gamma angle

Gamma angle

The gamma (γ)angle is the angle between the local z axis and the plane created by the local x axis and the global Z axis.



You do not enter the gamma angle directly as it is derived for you from the rotation angle, (the rotation angle being easier to work with and visualise, particularly when modeling in sloped planes).

In the majority of cases the gamma angle will have the same value as the rotation angle, although this is not always the case - a hip rafter in a steel portal frame being one example where they could differ.

Rotation angle

Patch, Result Strip and Punching Check Properties

result strip properties

To display the properties for a Result Strip you must first select it, (either in a View, or in the Structure Tree).

To select a strip in a View you must ensure that 'Result Strips' are switched on in Scene Content.

The following strip properties are then displayed:

Property	Description
General	
Name	The automatically generated name for the strip.
User Name	Can be used to override the automatically generated name if required.
Start Width	The total strip width at the first point picked when creating the strip.
End Width	The total strip width at the second point picked when creating the strip.
Result Type	Determines how the strip result is calculated: • Average • Centreline • Maximum
Number of Stations	The number of stations per metre along the strip.
Number of Points	The number of points per metre across the strip at each station.

Related topics

- Working with 2D Strips and displaying Strip Results

Isolated Foundation and Pile Properties

Pad Base Column Properties

The properties required to create this type of pad base are described in the table below.

Property	Description
General	

Foundation Type	Isolated Pad Base (not editable).
Auto-design depth	• When **Autodesign** is selected an iterative procedure is used to determine the depth. If the shear design fails the depth is increased until either the check passes, or the maximum depth specified in Design Options has been reached. • When **Autodesign** is not selected (i.e. check mode), the existing depth is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. When **Design Member** on the right-click context menu is used to design an individual base, the depth will be checked or designed according to the auto-design depth setting.
Auto-design size	• When **Autodesign** is selected an iterative procedure is used to determine the base size. If the bearing check fails the size is increased until either the check passes, or the maximum side length specified in Design Options has been reached. • When **Autodesign** is not selected (i.e. check mode), the existing size is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. When **Design Member** on the right-click context menu is used to design an individual base, the size will be checked or designed according to the Auto-design size setting.
Select size/depth starting from	This option only appears when either **Autodesign depth** or **Autodesign size** is selected. It sets the autodesign start point for the depth, or size, or both. The options are: • Minima (default) • Current Selecting 'Minima' removes the current depth/size and begins with the minimum allowed depth/size specified in **Design Options.** For both options the auto-design depth increment is that specified in **Design Options> Foundations Isolated Foundations> Foundation Size**

Autodesign reinforcement	This setting applies to top and bottom reinforcement, but reinforcement in either location can still be set to none. If enabled, auto-design of reinforcement occurs after the base size and depth have been established. • When **Autodesign reinforcement** is enabled an iterative procedure is used to determine the reinforcement. If the bending design fails the reinforcement size gets increased and spacing gets decreased until these checks pass. Iterations continue until either a satisfactory bar configuration has been achieved, or the maximum bar size and minimum spacing specified in Design Options has been reached. At this point the depth gets increased and the above procedure is repeated. If the smallest bar at largest spacing is insufficient then the next bar size at max spacing is used. If the largest bar is reached the spacing gets reduced and the smallest bar is used. Then the bar size gets increased until the largest bar size reached. • When **Autodesign reinforcement** is disabled (i.e. check mode), the existing reinforcement provision is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their reinforcement auto-designed. When **Design Member** on the right-click context menu is used to design an individual base the reinforcement is always designed (irrespective of the Autodesign reinforcement setting).
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if **Autodesign reinforcement** is selected. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size specified in Design Options. • Current - the auto design commences from the current bar arrangement.
Foundation Parameters	
Eccentricity in Y direction	Specifies the eccentricity of the pad in the Y direction.
Eccentricity in X direction	Specifies the eccentricity of the pad in the X direction.
Rotation angle	Specifies the rotation of the base relative to that of the support (which is controlled by the rotation angle specified for column).
Shape	Specifies where the base shape in plan: • Square • Rectangular

Length in Y direction	Specifies the size of the pad in the Y direction.
Length in X direction	Specifies the size of the pad in the X direction.
Depth	Specifies the depth of the pad.
Concrete class	The concrete grade. The concrete grades that are available are set from the Materials button on the Home ribbon.
Sliding in Y prevented / Sliding in X prevented	If sliding is prevented by some other means (e.g. if attached to a floor slab in one or other direction) then this can be indicated by selecting the appropriate direction. TIP: If the direction is not clear make the base non-rectangular so that length in X and hence X-direction is clearer. Selecting this option sets the shear force for the particular direction to zero. This affects the sliding check, but actually affects all checks because there is no additional overturning moment developed due to shear at top of base.
Overturning about X prevented /Overturning about Y prevented	This option only appears when the **Sliding in Y prevented** / **Sliding in X prevented** option is selected. Selecting this option indicates that overturning is also prevented by some other means for the selected direction, which sets the support moment for that direction to zero. This affects all checks because there is no overturning moment considered.
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. See: Design parameters
Maximum crack width	• 0.2 • 0.3 • 0.4
Reinforcement	
Type	• Mesh • Loose Bars • None
Rib type (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2

Rib type (Head Code ACI)	• Plain • Deformed
Bar type	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size, spacing, Mesh type etc.	The actual reinforcement provided in each of the layers is indicated here.
Top, Bottom, Side cover	Nominal cover to reinforcement.
Soil Parameters	
Surcharge depth	Surcharge depth
Soil unit weight	Soil unit weight
Permanent surcharge load	Permanent surcharge load
Variable surcharge load	Variable surcharge load
Characteristic friction angle	Characteristic friction angle
Design shear strength of soil angle (ACI & BS Head Codes)	Design shear strength of soil angle
Allowable bearing capacity (ACI & BS Head Codes)	Allowable bearing capacity
Bearing capacity A1 - STR (EC Head Code)	Allowable bearing capacity A1. Only displayed and used when the 'presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)' is unselected in Design Options.
Bearing capacity A2 - GEO (EC Head Code)	Allowable bearing capacity A2. Only displayed and used when the 'presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)' is unselected in Design Options.

Presumed bearing resistance (EC Head Code)	The presumed bearing resistance is only displayed and used when the 'l bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)' is selected in Design
[+] UDA	A customizable list of the attributes that can be applied to individual me panels. See: User Defined Attributes (UDAs).

Strip Base Wall Properties

The properties required to create this type of pad base are described in the table below.

Property	**Description**
General	
Foundation Type	Isolated Pad Base (not editable).
Auto-design depth	• When **Autodesign** is selected an iterative procedure is used to determine the depth. If the shear design fails the depth is increased until either the check passes, or the maximum depth specified in Design Options has been reached. • When **Autodesign** is not selected (i.e. check mode), the existing depth is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed.
Auto-design size	• When **Autodesign** is selected an iterative procedure is used to determine the base size. If the bearing check fails the size is increased until either the check passes, or the maximum side length specified in Design Options has been reached. • When **Autodesign** is not selected (i.e. check mode), the existing size is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed.

Select size/depth starting from	This option only appears when either **Autodesign depth** or **Autodesign size** is selected. It sets the autodesign start point for the depth, or size, or both. The options are: • Minima (default) • Current Selecting 'Minima' removes the current depth/size and begins with the minimum allowed depth/size specified in **Design Options.** For both options the auto-design depth increment is that specified in **Design Options> Foundations Isolated Foundations> Foundation Size**
Autodesign reinforcement	This setting applies to top and bottom reinforcement, but reinforcement in either location can still be set to none. If enabled, auto-design of reinforcement occurs after the base size and depth have been established. • When **Autodesign reinforcement** is enabled an iterative procedure is used to determine the reinforcement. If the bending design fails the reinforcement size gets increased and spacing gets decreased until these checks pass. Iterations continue until either a satisfactory bar configuration has been achieved, or the maximum bar size and minimum spacing specified in Design Options has been reached. At this point the depth gets increased and the above procedure is repeated. If the smallest bar at largest spacing is insufficient then the next bar size at max spacing is used. If the largest bar is reached the spacing gets reduced and the smallest bar is used. Then the bar size gets increased until the largest bar size reached. • When **Autodesign reinforcement** is disabled (i.e. check mode), the existing reinforcement provision is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their reinforcement auto-designed.
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if **Autodesign reinforcement** is enabled. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size specified in Design Options. • Current - the auto design commences from the current bar arrangement.
Foundation Parameters	
Eccentricity in Y (along wall)	Specifies the eccentricity of the pad in the Y direction.
Eccentricity in X (across wall)	Specifies the eccentricity of the pad in the X direction.

Rotation angle	Specifies the angle of the base about global Z.
Shape	Not editable
Width	Specifies the width of the base.
Extension beyond wall end	Specifies the amount the base projects beyond both wall ends.
Depth	Specifies the depth of the base.
Concrete class	The concrete grade. The concrete grades that are available are set from the Materials button on the Home ribbon.
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. See: Design parameters
Maximum crack width	• 0.2 • 0.3 • 0.4
Reinforcement	
Type	• Mesh • Loose Bars • None
Rib type (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type (Head Code ACI)	• Plain • Deformed
Bar type	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size, spacing, Mesh type etc.	The actual reinforcement provided in each of the layers is indicated here.
Top, Bottom, Side cover	Nominal cover to reinforcement.

Soil Parameters	
Surcharge depth	Surcharge depth
Soil unit weight	Soil unit weight
Permanent surcharge load	Permanent surcharge load
Variable surcharge load	Variable surcharge load
Characteristic friction angle	Characteristic friction angle
Design shear strength of soil angle (ACI & BS Head Codes)	Design shear strength of soil angle
Allowable bearing capacity (ACI & BS Head Codes)	Allowable bearing capacity
Bearing capacity A1 - STR (EC Head Code)	Allowable bearing capacity A1
Bearing capacity A2 - GEO (EC Head Code)	Allowable bearing capacity A2
[+] UDA	A customizable list of the attributes that can be applied to individual me panels. See: User Defined Attributes (UDAs).

Pile Cap Column Properties

The properties required to create a pile cap are described in the table below.

Property	**Description**
General	
Foundation Type	Isolated Pile Cap (not editable).

Auto-design depth	• When **Autodesign** is selected an iterative procedure is used to determine the depth. If the shear design fails the depth is increased until either the check passes, or the maximum depth specified in Design Options has been reached. • When **Autodesign** is not selected (i.e. check mode), the existing depth is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one pile cap in the group is set to have its depth auto-designed the whole group will have their depth auto-designed.
Auto-design piles	**Autodesign piles** automatically determines the quantity and size of piles. • Cleared - the specified number of piles at their specified positions will be checked during the design process. • Checked - the number of piles and their positions under the pile cap will automatically determined during the design process. When this check box is selected, the actual procedure used will depend on the **Pile auto-design method** that has been specified in **Design Options > Concrete Foundations > Isolated Foundations Piles** • The **Minimise number of piles** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded the pile with the next smallest load capacity in the catalogue is selected and the process starts again. This is repeated until the pile loading check passes. • The **Minimise pile capacity** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded one pile is added to the pile group and the process starts again. This is repeated until the pile loading check passes.
Select size/depth starting from	This option only appears when either **Autodesign depth** or **Autodesign size** is selected. It sets the autodesign start point for the depth, or size, or both. The options are: • Minima (default) • Current Selecting 'Minima' removes the current depth/size and begins with the minimum allowed depth/size specified in **Design Options.** For both options the auto-design depth increment is that specified in **Design Options> Foundations Isolated Foundations> Foundation Size**

Autodesign reinforcement	This setting will always be enabled if either **Auto-design depth** or **Auto-design piles** are enabled. It applies to top and bottom reinforcement, but reinforcement in either location can still be set to none. • When **Autodesign reinforcement** is enabled an iterative procedure is used to determine the reinforcement. If the bending design fails the reinforcement size gets increased and spacing gets decreased until these checks pass. Iterations continue until either a satisfactory bar configuration has been achieved, or the maximum bar size and minimum spacing specified in Design Options has been reached. At this point the depth gets increased and the above procedure is repeated. If the smallest bar at largest spacing is insufficient then the next bar size at max spacing is used. If the largest bar is reached the spacing gets reduced and the smallest bar is used. Then the bar size gets increased until the largest bar size reached. • When **Autodesign reinforcement** is disabled (i.e. check mode), the existing reinforcement provision is retained and *Tekla Structural Designer* determines if it is sufficient. In **Design Options** - if Isolated Foundations have been set to be designed using groups, then if at least one pile cap in the group is set to have its depth auto-designed the whole group will have their reinforcement auto-designed
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if **Autodesign reinforcement** is enabled. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size specified in Design Options. • Current - the auto design commences from the current bar arrangement.
Shape	This option controls the pile cap shape when **Auto-design piles** is enabled. • Square • Rectangular
Foundation Parameters	
Geometry type	Specifies where the pile cap shape: • Cuboid Tapered, Pedestal and Stepped geometries are currently beyond scope.
Eccentricity in X direction	Specifies the eccentricity of the pile cap in the X direction.
Eccentricity in Y direction	Specifies the eccentricity of the pile cap in the Y direction.
Rotation angle	Specifies the rotation of the pile cap relative to that of the support (which is controlled by the rotation angle specified for column).

Length in Y direction	Specifies the size of the pile cap in the Y direction.
Length in X direction	Specifies the size of the pile cap in the X direction.
Depth	Specifies the depth of the pile cap.
Concrete class	The concrete grade. The concrete grades that are available are set from the Materials button on the Home ribbon.
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. See: Design parameters
Maximum crack width	• 0.2 • 0.3 • 0.4
Piles	
Pile type	Pile types that have previously been specified in the Pile Catalogue are available for selection. You can choose <New...> to add additional types.
User defined arrangement	Specifies the eccentricity of the pile cap in the X direction.
Pile arrangement	Opens the Pile arrangement dialog for specifying the number of piles, and spacing, pile type and principal direction.
Number of piles	Specifies the number of piles.
Principal direction	Specifies the principal direction.
Pile spacing	The dimension between pile centers.
Shape	For three piles only, you can choose to specify either a triangular. or rectangular pile cap.
Reinforcement	
Type	• Mesh • Loose Bars • None

Rib type (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type (Head Code ACI)	• Plain • Deformed
Bar type	The reinforcement grades that are available here are set from the Mater button on the Home ribbon.
Bar size, spacing, Mesh type etc.	The actual reinforcement provided in each of the layers is indicated here
Top, Bottom, Side cover	Nominal cover to reinforcement.
Soil Parameters	
Surcharge depth	Surcharge depth
Soil unit weight	Soil unit weight
Permanent surcharge load	Permanent surcharge load
Variable surcharge load	Variable surcharge load
[+] UDA	A customizable list of the attributes that can be applied to individual me and panels. See: User Defined Attributes (UDAs).

Pile Type Properties

The following properties are required when adding pile types to the Pile Catalogue.

Property	Description
Data	
Name	User specified description.
Installation type	• Driven • Bored • CFA

Shape	• Square • Circular
Dimension	Represents 'side length' for square piles, or 'diameter' for circular piles.
Length	Pile length
Embedment	Length of embedment.
Load transfer type	• End-bearing • Friction
Pile Capacity	
Axial compressive resistance	Enter the compressive capacity. For Eurocode design both a GEO and STR resistance are requested.
Axial tensile resistance	Enter the tensile capacity. For Eurocode design both a GEO and STR resistance are requested.
Vertical	
Linearity	Specify the vertical spring type: • Linear • Non-linear
Vertical spring stiffness	For a "linear" spring type only: • Enter the vertical spring stiffness
Vertical compression spring stiffness	For a "Non-linear" spring type only: • Enter the vertical compression spring stiffness
Vertical compression limit	For a "Non-linear" spring type only: • Enter the vertical compression limit
Vertical tension spring stiffness	For a "Non-linear" spring type only: • Enter the vertical tension spring stiffness
Vertical tension limit	For a "Non-linear" spring type only: • Enter the vertical tension limit

Horizontal	
Horizontal restraint	• Fixed • Spring linear
Horizontal spring stiffness	For a 'Spring linear' type only: • Enter the horizontal spring stiffness

Pile Properties

create pile properties set

The following properties are displayed when you create a pile:

Property	**Description**
[-] General	
Pile type	From the drop list choose an existing pile type from the pile catalogue, or <New>... to add a new pile type to the catalogue. If you click <New>, the Edit Pile Type dialog is displayed to allow the Pile Type Properties to be defined.
Direction by Angles	This property is provided to allow you to define 'raking piles' - meaning piles that are not vertical but are inclined. • Cleared - the pile direction is defined in accordance with the global coordinate system. • Checked - a user defined coordinate system is applied to the pile. (After clicking where you want to create the pile, the second click defines the x direction and the third click defines the y direction.)
Direction	Defines the direction of the pile.
X, Y, Z	When not defining by angles, the pile direction is defined by an {X, Y, Z} vector. The default direction vector of {0,0,-1} meaning vertically down. See: How do specify the pile direction of an inclined pile?
Inclination, Azimuth and Rotation	If defining the direction by angles, angular measurements are used instead of vectors. See: How do specify the pile direction of an inclined pile?

edit pile properties set

The following properties are displayed when you select one or more existing pile:

Property	Description
[-] General	
Name	The automatically generated name for the pile.
User Name	Can be used to override the automatically generated name if required.
Pile type	From the drop list choose an existing pile type from the pile catalogue, or <New>... to add a new pile type to the catalogue. If you click <New>, the Edit Pile Type dialog is displayed to allow the Pile Type Properties to be defined.
Offset	The offset is measured in global X and Y relative to the mat.
Position	The pile position is measured relative to the global origin.
Direction	Defines the direction of the pile.
X, Y, Z	When a pile is edited the pile direction is given as an {X, Y, Z} vector. The default direction vector of {0,0,-1} meaning vertically down. See: How do specify the pile direction of an inclined pile?

Pile Array Properties

The following properties are displayed when you create a pile array:

Property	Description
[-] General	
Layout	From the drop list choose to define either a rectangular, or hexagonal array of piles.
Pile type	From the drop list choose an existing pile type from the pile catalogue, or <New>... to add a new pile type to the catalogue. If you click <New>, the Edit Pile Type dialog is displayed to allow the Pile Type Properties to be defined.
Spacing X, Spacing Y, Spacing	• In a rectangular array: 'Spacing X' and 'Spacing Y' define the spacing between piles in each direction. • In a hexagonal array: 'Spacing' defines the spacing between each pile and its nearest neighbouring piles.

Angle	The angle about global Z at which the array is positioned.
Min distance to Edge	Only piles at a greater distance to the edge than this value will be positioned.
Direction by Angles	This property is provided to allow you to define 'raking piles' - meaning piles that are not vertical but are inclined. • Cleared - the pile direction is defined in accordance with the global coordinate system. • Checked - a user defined coordinate system is applied to the pile. (After clicking where you want to create the pile, the second click defines the x direction and the third click defines the y direction.)
Direction	Defines the direction of the pile.
X, Y, Z	When not defining by angles, the pile direction is defined by an {X, Y, Z} vector. The default direction vector of {0,0,-1} meaning vertically down. See: How do specify the pile direction of an inclined pile?
Inclination, Azimuth and Rotation	If defining the direction by angles, angular measurements are used instead of vectors. See: How do specify the pile direction of an inclined pile?

Sub Model Properties

The following properties associated with a sub model are displayed in the Properties Window when it is selected in the Structure Tree.

Name	**Description**
Override Model's	Select this check box in order to override the Structure meshing properties in the current sub model.
Shell Mesh Size	Defines the shell mesh size for two way spanning slabs in the sub model.
Shell Uniformity Factor	Defines the shell mesh uniformity factor for two way spanning slabs in the sub model.
Semi-Rigid Mesh Size	Defines the semi-rigid mesh size when slabs are modelled as semi-rigid diaphragms in the sub model.
Semi-Rigid Uniformity Factor	Defines the semi-rigid uniformity factor when slabs are modelled as semi-rigid diaphragms in the sub model.

Semi-Rigid Mesh Type	Defines the semi-rigid mesh type (QuadDominant, QuadOnly, or Triangular) used in the sub model.

Solver Model Properties

Solver Node properties

When in a Solver View, solver node properties are displayed in the **Properties Window** as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

General	
Fx, Fy, Fz	These define the translational degrees of freedom at the node.
Mx, My, Mz	These define the rotational degrees of freedom at the node.
Coordinate	Defines the node location.
P-Delta	This property cannot be edited.
Exclude from Diaphragm	Check to remove the node from the diaphragm.
Diaphragm #	Specifies the diaphragm number to which the node is connected.

Solver Element properties

When in a Solver View, solver element properties are displayed in the **Properties Window** as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

General	
Active	When this is set to False the solver element is inactive in the analysis. Only certain member types (braces, analysis elements) can be made inactive.
Type	The type of the solver element
Fabrication	The fabrication type of the solver element.
Construction	The construction type of the solver element.

Material	The solver element material.
Gamma angle	Defines the element orientation about its local x axis. When gamma = 0, the local z lies in the plane created by the local x axis and the global z axis.
Length	The solver element length.
Start Releases	
Fx, Fy, Fz	These define the translational end releases at end 1.
Mx, My, Mz	These define the rotational end releases at end 1.
End Releases	
Fx, Fy, Fz	These define the translational end releases at end 2.
Mx, My, Mz	These define the rotational end releases at end 2.

Solver Element 2D properties

When in a Solver View, solver element 2D properties are displayed in the **Properties Window** as shown below. None of these properties can be directly edited in the Solver View.

General	
Type	The 2D element type will be **Shell** or **Semi-rigid** depending on the Slab Type, Decomposition and Diaphragm Options that were been set.
Thickness	The 2D element thickness is derived from a different property depending on the slab type: • Composite Slab; Precast Slab; Slab on Beams; Flat Slab - Overall Depth • Steel Deck; Timber Deck - Thickness
Orientation	The 2D element orientation in the solver model follows the rotation angle defined for the slab item to which it belongs.
DivideStiffnessBy	This property applies to semi-rigid elements only. It adjusts the stiffness determined from the material properties in order to control semi-rigid diaphragm flexibility.
CrackedOption	yes/no

Glossary

2

2D view: view that displays objects in a two dimensional plane

3

3D view: view that displays objects three dimensionally

A

analysis model: structural model that is created from a physical model and used for analyzing structural behavior and subsequently for design

analysis part: analysis model object that is a representation of a building object in an analysis model

architectural grid: modeling aid that represents a three-dimensional complex of horizontal and vertical planes

B

background reinforcement: The overall panel reinforcement in the slab top and bottom in directions x and y. The background level of reinforcement can be "none".

beam: linear building object in a mainly horizontal position

BIM: process of modeling and communicating the structure of a building in detail to benefit the entire construction life cycle. Building information modeling facilitates the exchange and use of building information in a digital format.

building object: object that represents something that will exist in the real building or be closely related to it

C

column: linear building object in a mainly vertical position

construction line: construction object that represents a line between two points

construction point; CP: point at the end of a building object

D

diaphragm: analysis model object that connects more than two nodes that move with exactly the same rotation and translation

drawing: building contract document that shows, in graphic or pictorial form, the design, location, and dimension of the elements of a project

drawing object: object that is represented in a Tekla Structural Designer drawing

E

element intersection; Elm Inters: point at the intersection of two building objects

entity: Collective term for any object used in the modelling process. For example, model objects and modelling aids are both entities.

G

global coordinate system: coordinate system that reflects the entire space of a Tekla Structural Designer model . A tripod representing positive global X, Y and Z lies at the bottom left of each 3D view.

grid line: single line that visualizes a single grid plane on a view plane

grid point: point at the intersection of two grid lines

H

highlight: to emphasize a single object so that it stands out. Right click menu commands operate on highlighted (not selected) objects

I

intersection; Inters: point at the intersection of two grid or construction lines

L

load: model object that represents a force or system of forces carried by a structure or a part of a structure

load combination: set of loadcases multiplied by their partial safety factors

loadcase: set of loads that are caused by the same action and to which the user wants to refer collectively

Loading Analysis View: window that is used for reviewing the forces and moments in an individual member

local coordinate system: coordinate system that applies to an individual building object

M

mesh group: Interconnecting panels and slab features with sufficient common properties are gathered together automatically into 'mesh groups'. Each mesh group is meshed as a single entity in the meshing process.

model: pattern of an object, a system, or a process that exists or will exist in the real world

model object: Entity that is represented in a Tekla Structural Designer model. A model object is either created in a model or imported into it. For example, an individual steel beam, concrete wall or support is a model object.

Model Settings: the settings applied to the current Tekla Structural Designer model

modeling aid: Entity that represents information that is only relevant in building a model. For example, grids, points, construction lines, frames, planes, and reference drawings are modeling aids.

N

node: analysis model object that Tekla Structural Designer creates at a defined point of an analysis model based on analysis part connectivity

O

object: collection of human and computer interpretable data that is needed to model, manufacture, and construct a structure

P

patch: A special rectangular area of slab at a location in a slab at which design will be performed. The patch may or may not have additional reinforcement defined in its area.

physical model: structural model with a direct or indirect counterpart in the structure to be built

pick: to click one or more points in a model in order to execute an action using those points

point: modeling aid that represents a determined place in a coordinate system

Properties Window: a dialog box in which the properties related to an object can be given values (for example, Grid Line Properties).

punching shear: The punching basic control perimeter surrounds a load or loaded object that might punch through a slab with a shear failure and hence is a location at which a punching shear check is to be performed.

R

reinforced concrete: concrete structure which contains reinforcement designed on the assumption that the concrete and reinforcement act together in resisting forces

reinforcement mesh; mesh: reinforcement that represents a mesh of steel bars in two perpendicular directions In Tekla Structural Designer, the reinforcement mesh bars in one direction are called main bars and reinforcement mesh bars perpendicular to them are called crossing bars.

reinforcement; reinforcement object: building object that represents steel bars that are cast into the concrete in such a manner that the steel and the concrete act together in resisting forces Reinforcement types are reinforcing bars (also referred to as rebars) and reinforcement meshes,

reinforcing bar; rebar: reinforcement that represents a steel bar used to increase the tensile strength of concrete

report: model output that is represented as a list of information for the entire Tekla Structural Designer model, or selected objects, Reports react to modifications in the Tekla Structures model.

result strip: A result strip is a line from point A to point B in the plane of a slab which has a width. The results for bending, shear and deflection are calculated at stations along the length of the strip centre line and displayed to you and used in slab design. .

Results View: view that is used for reviewing the analysis results

Review View: view that is used for reviewing the design model and design status

S

select: to choose one or more objects in order to execute a command directed at them

Settings; Settings Set: a set of defaults typically configured for a particular geographic region

slab: A grouping of slab panels with the same general properties. The panels can either be connected or separated from each other, however they must be on the same level. Each panel in a slab must have certain attributes the same (e.g. slab type and material) but can have other attributes that differ (e.g. thickness and the direction of reinforcement).

slab panel; slab item: An individual area of slab, often within a single bay, that can have specific properties (e.g.reinforcement) that can differ from other panels within the slab. It is also an "on/off" area for pattern loading. Slab design is performed by panel.

slope; sloped plane: construction object that represents a plane that is not perpendicular to the global axes

Solver View: view that is used for reviewing the underlying analytical model

Structural View: view that is used for modeling purposes

T

Tekla Structural Designer drawing: drawing that includes the output of the selected information in a Tekla Structural Designer model or a part of the model and other information related to the model and the project Tekla Structural Designer drawings react to modifications in the Tekla Structural Designer model and are updated accordingly

Tekla Structural Designer model: model that is built with Tekla Structural Designer and that represents a structure to be constructed, containing information needed to analyse and design the structure, and other information related to the project

U

user coordinate system; UCS: local coordinate system defined by the user

V

view: representation of a Tekla Structural Designer model or a part of it, which is displayed inside the Tekla Structural Designer window

W

Wind View: view that is used for reviewing wind zones and zone load details