# Table of Contents

Wind Modeling Handbook........................................................................................................... 1

Overview of the Wind Model method .................................................................................... 1

Clothe the structure in wind wall and roof panels ........................................................... 2
  Applying Wall Panels ......................................................................................................... 2
  Applying Roof Panels....................................................................................................... 3

Perform the gravity design................................................................................................... 3

Run the Wind Wizard ........................................................................................................... 3

Review the wind zones .......................................................................................................... 3

Define the wind loadcases ................................................................................................... 4

Review wind zone loads ........................................................................................................ 4

Combine the wind loadcases into design combinations ................................................. 4

Perform the static design ..................................................................................................... 4

EC1991 1-4 Wind Wizard .......................................................................................................... 4

Design Codes and References ............................................................................................. 5

Scope ....................................................................................................................................... 5

Limitations .............................................................................................................................. 6

  Geometry ............................................................................................................................ 6

  Loaded Areas...................................................................................................................... 6

  Wind Direction ................................................................................................................... 6

  Overall Loads ..................................................................................................................... 6

  Beneficial Loads ............................................................................................................... 6

  Singapore National Annex - Minimum Horizontal Loads ............................................ 7

  Wind loading on wall panels ............................................................................................ 7

  Wind loading on roof panels ............................................................................................ 7

  Additional wind loads ........................................................................................................ 8

Using the EC1991 1-4 Wind Wizard with BREVe data ....................................................... 8

  Data Source page ............................................................................................................... 8

  BREVe location page .......................................................................................................... 9

  Basic Data page ................................................................................................................ 11

  Roughness and Obstructions (BREVe) page ................................................................. 12

  Orography (BREVe) page ................................................................................................. 13

  Tall Neighbouring Structure page .................................................................................. 13

  Results (BREVe) page ....................................................................................................... 14

  Finishing the Wind Wizard .............................................................................................. 14
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A practical approach to setting the analysis type</td>
<td>44</td>
</tr>
<tr>
<td>Validity of the amplified forces method</td>
<td>45</td>
</tr>
<tr>
<td>Sway sensitivity assessment (Eurocode)</td>
<td>46</td>
</tr>
<tr>
<td>Calculation of the elastic critical load factor</td>
<td>46</td>
</tr>
<tr>
<td>Derivation of the kamp formula for concrete structures</td>
<td>47</td>
</tr>
<tr>
<td>How do I assess the worst elastic critical load factor for the building?</td>
<td>49</td>
</tr>
<tr>
<td>What are the twist results?</td>
<td>49</td>
</tr>
<tr>
<td>Modification Factors</td>
<td>50</td>
</tr>
<tr>
<td>Member second-order effects</td>
<td>50</td>
</tr>
<tr>
<td>Global imperfections</td>
<td>51</td>
</tr>
<tr>
<td>Member imperfections</td>
<td>51</td>
</tr>
<tr>
<td>Wind drift</td>
<td>51</td>
</tr>
<tr>
<td>Overall displacement</td>
<td>52</td>
</tr>
<tr>
<td>Solver Models Handbook</td>
<td>53</td>
</tr>
<tr>
<td>Solver models</td>
<td>53</td>
</tr>
<tr>
<td>Working Solver Model</td>
<td>53</td>
</tr>
<tr>
<td>Solver Model used for 1st Order Linear</td>
<td>53</td>
</tr>
<tr>
<td>3D Building Analysis model</td>
<td>54</td>
</tr>
<tr>
<td>Solver Model used for 1st Order Non Linear</td>
<td>55</td>
</tr>
<tr>
<td>Solver Model used for 2nd Order Linear</td>
<td>55</td>
</tr>
<tr>
<td>Solver Model used for 2nd Order Non Linear</td>
<td>56</td>
</tr>
<tr>
<td>Solver Model used for 1st Order Vibration</td>
<td>56</td>
</tr>
<tr>
<td>Solver Model used for 2nd Order Buckling</td>
<td>57</td>
</tr>
<tr>
<td>Solver Model used for Grillage Chasedown</td>
<td>57</td>
</tr>
<tr>
<td>Solver Model used for FE Chasedown</td>
<td>58</td>
</tr>
<tr>
<td>Solver Model used for Load Decomposition</td>
<td>59</td>
</tr>
<tr>
<td>Refresh Solver Model</td>
<td>60</td>
</tr>
<tr>
<td>Analysis objects</td>
<td>60</td>
</tr>
<tr>
<td>Solver models created for concrete members</td>
<td>61</td>
</tr>
<tr>
<td>Concrete column physical and solver models</td>
<td>61</td>
</tr>
<tr>
<td>Concrete column physical model</td>
<td>61</td>
</tr>
<tr>
<td>Concrete column solver elements</td>
<td>62</td>
</tr>
<tr>
<td>Concrete beam physical and solver models</td>
<td>63</td>
</tr>
<tr>
<td>Concrete beam physical model</td>
<td>63</td>
</tr>
<tr>
<td>Concrete beam solver elements</td>
<td>64</td>
</tr>
<tr>
<td>Rigid offsets</td>
<td>65</td>
</tr>
</tbody>
</table>
Rigid zones............................................................................................................................ 65
Application of Rigid Zones.............................................................................................. 66
Rigid Zones Example 1 - fixed ended beam ................................................................. 66
Rigid Zones Example 2 - pin ended beam ................................................................. 70
Solver models created for steel and other materials .................................................. 72
Steel column physical and solver models ..................................................................... 72
Steel column physical model ......................................................................................... 72
Steel column solver elements ......................................................................................... 72
Steel beam physical and solver models ........................................................................ 74
Steel beam physical model ............................................................................................. 74
Steel beam solver elements ........................................................................................... 76
Steel brace physical and solver models ........................................................................ 77
Steel brace physical model ............................................................................................. 77
Steel brace solver elements ........................................................................................... 78
Inactive steel braces ........................................................................................................ 78
Tension only and compression only braces .................................................................. 78
Input method for A and V Braces .................................................................................. 78
Steel parapet post physical and solver models .............................................................. 78
Solver models created for concrete walls ....................................................................... 79
Concrete wall physical model ......................................................................................... 79
Concrete wall solver model ............................................................................................ 80
Concrete wall openings and extensions ......................................................................... 84
Concrete wall openings ................................................................................................. 84
Limitations of wall openings ......................................................................................... 84
Analysis model applied to meshed wall panels with openings ..................................... 85
Alternative model for wall openings ............................................................................. 85
Concrete wall extensions ............................................................................................... 87
Use of concrete wall extensions .................................................................................... 87
Concrete wall extension examples ................................................................................. 88
Solver models created for bearing walls ..................................................................... 92
Bearing wall physical model ......................................................................................... 93
Bearing wall solver model ............................................................................................. 93
Solver models created for slabs .................................................................................. 95
Vertical alignment of slabs ......................................................................................... 95
Effect of slab openings on solver models ................................................................. 96
Slab on beams solver models ....................................................................................... 97
Table of Contents

- Flat slab solver models ................................................................. 97
- Precast solver models ................................................................. 97
- Steel deck solver models ............................................................ 97
- Timber deck solver models ......................................................... 97
- Composite slab solver models .................................................... 97
- Diaphragms and floor meshing .................................................... 98
  - Diaphragm types ...................................................................... 98
    - Rigid .................................................................................. 98
    - Semi-rigid ........................................................................ 99
  - Diaphragm constraint and mesh type configurations ............. 100
    - Diaphragm option .............................................................. 100
    - Decomposition .................................................................. 100
    - Mesh 2-way Slabs in 3D Analysis ...................................... 101
    - Summary of diaphragm constraint and mesh type configurations .... 101
    - Other slab properties affecting the solver models .............. 102
  - Mesh parameters ................................................................. 103
    - Slab Mesh ......................................................................... 103
    - Semi-Rigid Mesh .............................................................. 103
- Releases ..................................................................................... 103
  - Column Releases ................................................................. 103
  - Wall Releases ....................................................................... 104
  - Beam Releases ................................................................. 104
  - Brace Releases ................................................................. 105
- Supports ................................................................................... 105
  - Support degrees of freedom .................................................. 106
  - Non linear spring supports .................................................... 106
  - Partial fixity of column bases ............................................... 107
- Static Analysis and Design Handbook ...................................... 109
  - Definitions .......................................................................... 109
  - Summary of Static Analysis-Design Processes ...................... 110
  - Model validation ............................................................... 112
  - 3D pre-Analysis ................................................................. 113
    - Load decomposition .......................................................... 113
      - 1-way slab load decomposition ......................................... 114
      - 2-way slab load decomposition ......................................... 114
    - Global imperfections ....................................................... 115
Equivalent Lateral Force Method ................................................................. 128
Response Spectrum Analysis Method .......................................................... 128
  Summary of RSA Seismic Analysis Processes .......................................... 129
Seismic Drift ................................................................................................. 129
Limitations of Seismic Design ........................................................................ 130
  Specific limitations of steel seismic design ........................................... 130
Seismic Force Resisting Systems ................................................................. 131
  Available SFRS types ............................................................................ 131
    SFRS types included for steel members ........................................... 131
    SFRS types available for concrete members .................................. 131
    SFRS types excluded ........................................................................ 132
  Members allowed in the SFRS ............................................................... 132
  Assigning members to the SFRS ............................................................ 132
    Special Moment Frames - assigning connection types at steel beam ends 132
Validation of the SFRS ................................................................................ 133
  Auto design of SFRS members ............................................................. 133
Seismic Design Methods ................................................................................ 133
  Seismic analysis and conventional design ........................................... 134
    ELF seismic analysis and conventional design ................................ 134
    RSA seismic analysis and conventional design ............................ 134
  Seismic analysis and seismic design .................................................... 135
    ELF seismic analysis and seismic design ........................................ 135
    RSA seismic analysis and seismic design ....................................... 136
Eurocode EN1998-1:2004 Seismic Wizard ..................................................... 138
  Starting the Wizard ............................................................................... 138
  Site Specific Spectra ............................................................................. 138
  Base Information Page .......................................................................... 138
  Structure Regularity Page ...................................................................... 140
  Fundamental Period Page ...................................................................... 140
  Behaviour Factor Page .......................................................................... 141
  Seismic Inertia Combination Page ....................................................... 141
  Finishing the Seismic Wizard ............................................................... 142
Concrete Design Handbook .......................................................................... 143
  Design Concrete ................................................................................... 143
  Features common to concrete beam, column and wall design .......... 143
  Gravity or Static Design? ....................................................................... 143
Analysis types performed in the Design Concrete process ................................................................. 143
Pre-design considerations ........................................................................................................................ 144
Nominal cover ........................................................................................................................................ 144
Assume cracked ...................................................................................................................................... 145
Design parameters ................................................................................................................................. 146
Reinforcement Parameters ....................................................................................................................... 146
Design and detailing groups (concrete) .................................................................................................... 147
Why use concrete design and detailing groups? .................................................................................... 148
What happens in the group design process? ............................................................................................ 148
Concrete design group requirements ....................................................................................................... 149
Detailing group requirements .................................................................................................................. 150
Group management .................................................................................................................................. 151
How is grouped design and detailing de-activated for concrete members? ........................................ 152
Typical Design Concrete workflow ......................................................................................................... 152
Set up Pattern Loading ............................................................................................................................ 153
Set all beams columns and walls into autodesign mode .......................................................................... 154
Review beam and column design groups ............................................................................................... 154
Review beam, column and wall design parameters and reinforcement settings...................................... 155
Perform the concrete design .................................................................................................................... 155
Review the design status and ratios ........................................................................................................ 155
Create Drawings and Quantity Estimations ............................................................................................. 156
Print Calculations ...................................................................................................................................... 156
Reviewing Design Concrete and refining the design of individual members ........................................ 157
How do I view results for a single concrete member (without re-selecting steel)? ................................ 157
How do I re-select steel for a single concrete member and then view its results? ................................. 157
Features of concrete beam design ........................................................................................................... 158
Analysis types used for concrete beam design ....................................................................................... 158
Autodesign (concrete beam) .................................................................................................................... 158
Deflection control .................................................................................................................................... 159
Deflection control .................................................................................................................................... 159
Use of beam flanges .................................................................................................................................. 159
Longitudinal reinforcement ....................................................................................................................... 163
Bar layers .................................................................................................................................................. 163
Longitudinal Reinforcement Shapes Library ............................................................................................. 165
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Reinforcement Patterns Library</td>
<td>166</td>
</tr>
<tr>
<td>Longitudinal Reinforcement Regions</td>
<td>169</td>
</tr>
<tr>
<td>Relationship between Reinforcement Patterns and Design Regions</td>
<td>170</td>
</tr>
<tr>
<td>Shear reinforcement</td>
<td>172</td>
</tr>
<tr>
<td>Shear Reinforcement Shapes Library</td>
<td>172</td>
</tr>
<tr>
<td>Shear Reinforcement Patterns Library</td>
<td>173</td>
</tr>
<tr>
<td>Shear Reinforcement Regions</td>
<td>173</td>
</tr>
<tr>
<td>Features of concrete column design</td>
<td>174</td>
</tr>
<tr>
<td>Autodesign (concrete column)</td>
<td>174</td>
</tr>
<tr>
<td>Section</td>
<td>175</td>
</tr>
<tr>
<td>Slenderness</td>
<td>176</td>
</tr>
<tr>
<td>Stiffness</td>
<td>177</td>
</tr>
<tr>
<td>Load reductions</td>
<td>177</td>
</tr>
<tr>
<td>Stacks and reinforcement lifts</td>
<td>177</td>
</tr>
<tr>
<td>Column design forces</td>
<td>178</td>
</tr>
<tr>
<td>Features of concrete wall design</td>
<td>178</td>
</tr>
<tr>
<td>Autodesign (concrete wall)</td>
<td>178</td>
</tr>
<tr>
<td>Slenderness</td>
<td>179</td>
</tr>
<tr>
<td>Stiffness</td>
<td>179</td>
</tr>
<tr>
<td>Reinforcement</td>
<td>179</td>
</tr>
<tr>
<td>Load reductions</td>
<td>180</td>
</tr>
<tr>
<td>Stacks and reinforcement lifts</td>
<td>180</td>
</tr>
<tr>
<td>Wall design forces</td>
<td>181</td>
</tr>
<tr>
<td>Concrete slab design</td>
<td>181</td>
</tr>
<tr>
<td>Features of concrete slab analysis and design</td>
<td>181</td>
</tr>
<tr>
<td>Analysis types used for concrete slab design</td>
<td>181</td>
</tr>
<tr>
<td>Concrete slab load decomposition</td>
<td>182</td>
</tr>
<tr>
<td>Slab on beam idealized panels</td>
<td>182</td>
</tr>
<tr>
<td>Combined slab and patch reinforcement</td>
<td>184</td>
</tr>
<tr>
<td>Typical flat slab design procedure</td>
<td>185</td>
</tr>
<tr>
<td>Flat slab design example</td>
<td>185</td>
</tr>
<tr>
<td>Set up Pattern Loading</td>
<td>186</td>
</tr>
<tr>
<td>Design All</td>
<td>187</td>
</tr>
<tr>
<td>Consider Deflection (for Flat slabs)</td>
<td>187</td>
</tr>
<tr>
<td>Select a Level</td>
<td>188</td>
</tr>
<tr>
<td>Add Patches</td>
<td>188</td>
</tr>
</tbody>
</table>
Design Panels ................................................................................................................................. 189
Review/Optimise Panel Design ........................................................................................................ 190
Design Patches .................................................................................................................................. 191
Review/Optimise Patch Design ......................................................................................................... 191
Add and Run Punching Checks ........................................................................................................ 192
Create Drawings and Quantity Estimations .................................................................................... 192
Print Calculations ............................................................................................................................ 192

Typical slab on beams design procedure ...................................................................................... 193
Slab on beam design example .......................................................................................................... 193
Set up Pattern Loading .................................................................................................................... 194
Design All .......................................................................................................................................... 194
Select a Level ..................................................................................................................................... 195
Add Beam and Wall Top Patches ..................................................................................................... 195
Design Panels .................................................................................................................................. 196
Review/Optimise Panel Design ......................................................................................................... 197
Design Beam and Wall Patches ......................................................................................................... 198
Review/Optimise Beam and Wall Patch Design ............................................................................. 199
Create Drawings and Quantity Estimations .................................................................................... 199
Print Calculations ............................................................................................................................ 199

Interactive concrete member design .............................................................................................. 200
Interactive concrete beam design .................................................................................................... 200
How do I open the Interactive Beam Design Dialog? ................................................................... 200
Overview of the Interactive Beam Design Dialog ......................................................................... 200
How do I change the bar pattern? .................................................................................................... 203
Interactive concrete column design ................................................................................................ 203
How do I open the Interactive Column Design Dialog? .............................................................. 204
Overview of the Interactive Column Design Dialog ...................................................................... 204
How do I arrange bars in the Interactive Column Design Dialog? .............................................. 207
Interactive concrete wall design ...................................................................................................... 209
How do I open the Interactive Wall Design Dialog? .................................................................... 210
Overview of the Interactive Wall Design Dialog .......................................................................... 210

Working with large concrete models ............................................................................................. 213

Steel Design Handbook .................................................................................................................... 217
General design parameters ............................................................................................................... 217
Material type ..................................................................................................................................... 217
Autodesign (steel) ............................................................................................................................ 217
Table of Contents

Design Section Order ........................................................................................................218
  How do I view the list of sections in a design section order? ........................................218
  How do I specify that a section in the list should not be considered for design? ..........218
  How do I sort the listed sections by a different property? ...........................................218
  How do I specify that a section is ‘non-preferred’? .......................................................219
Size Constraints ..................................................................................................................219
Gravity only design ..............................................................................................................219
Design groups ......................................................................................................................220
  How is the ‘design using groups option’ activated? .......................................................220
  What happens in the group design process? .................................................................220
Instability factor ..................................................................................................................221
Steel beam design ...............................................................................................................221
  Steel beam scope..............................................................................................................221
Steel beam limitations and assumptions ..........................................................................222
Steel beam design properties ............................................................................................223
  Fabrication ......................................................................................................................223
  Section .............................................................................................................................224
  Restraints .......................................................................................................................224
  Web Openings to SCI P355 ............................................................................................224
  Deflection Limits .............................................................................................................227
  Camber ............................................................................................................................227
  Natural frequency ..........................................................................................................228
  Seismic............................................................................................................................228
Composite beam design .....................................................................................................228
  Composite beam scope .................................................................................................228
Composite beam loading ....................................................................................................229
  Construction stage loading ............................................................................................229
  Composite stage loading ...............................................................................................230
Concrete slab .......................................................................................................................231
  Precast concrete planks .................................................................................................231
  General limitations and assumptions............................................................................231
Concrete properties ............................................................................................................233
Loading .............................................................................................................................233
Shear Connectors ...............................................................................................................233
Longitudinal Shear .............................................................................................................233
Composite Moment of Inertia ...........................................................................................234
Composite beam design properties .......................................................... 234
  Properties common to composite and non-composite beams ................. 234
  Allow non-composite design ................................................................. 234
Restraints .................................................................................................. 235
Floor construction .................................................................................... 236
Effective width calculations ................................................................... 237
Metal deck ............................................................................................... 238
Stud strength ......................................................................................... 238
Transverse reinforcement ........................................................................ 238
Connector layout .................................................................................... 239
  Auto-layout for Perpendicular decks .................................................. 240
  Auto-layout for Parallel decks ............................................................ 241
  Manual Stud Layout .......................................................................... 242
Steel column design .............................................................................. 245
  Steel column scope .......................................................................... 245
  Limitations for sloping columns ......................................................... 246
Steel column design properties .............................................................. 246
  Simple Columns .............................................................................. 246
  Section ............................................................................................ 247
  Restraints ....................................................................................... 247
  Load Reductions ............................................................................ 248
  Splice and Splice offset ................................................................. 248
  Web Openings .............................................................................. 249
  Seismic .......................................................................................... 250
Steel brace design .................................................................................. 250
  Steel brace scope .......................................................................... 250
  Input method for A and V Braces ...................................................... 251
Steel brace design properties ................................................................. 251
  Section ............................................................................................ 251
  Compression .................................................................................. 251
  Tension .......................................................................................... 251
Steel truss design ................................................................................... 252
  Steel truss scope .......................................................................... 252
Steel truss design properties ................................................................. 252
Steel joist design .................................................................................... 253
  Standard types ............................................................................. 253
Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Special Joists</td>
<td>253</td>
</tr>
<tr>
<td>Joist Girders</td>
<td>253</td>
</tr>
<tr>
<td>Joist Analytical Properties</td>
<td>254</td>
</tr>
<tr>
<td>Performing steel structure design</td>
<td>254</td>
</tr>
<tr>
<td>Gravity design</td>
<td>254</td>
</tr>
<tr>
<td>Full design</td>
<td>254</td>
</tr>
<tr>
<td>Individual member design</td>
<td>255</td>
</tr>
<tr>
<td>How do I view the design results for the analysed section?</td>
<td>255</td>
</tr>
<tr>
<td>How do I quickly check an alternative section size?</td>
<td>255</td>
</tr>
<tr>
<td>How do I quickly design a new section size?</td>
<td>256</td>
</tr>
<tr>
<td>Foundation Design Handbook</td>
<td>257</td>
</tr>
<tr>
<td>Isolated foundation design</td>
<td>257</td>
</tr>
<tr>
<td>Overview of the isolated foundation analysis model</td>
<td>257</td>
</tr>
<tr>
<td>Association with member supports</td>
<td>257</td>
</tr>
<tr>
<td>Analysis types</td>
<td>257</td>
</tr>
<tr>
<td>Design forces and checks</td>
<td>258</td>
</tr>
<tr>
<td>Pad base and strip base design procedures</td>
<td>259</td>
</tr>
<tr>
<td>Pad base design example</td>
<td>259</td>
</tr>
<tr>
<td>Apply bases under supported columns</td>
<td>260</td>
</tr>
<tr>
<td>Auto-size bases individually for loads carried</td>
<td>261</td>
</tr>
<tr>
<td>Apply grouping to rationalize pad base sizes</td>
<td>262</td>
</tr>
<tr>
<td>Review/Optimise Base Design</td>
<td>264</td>
</tr>
<tr>
<td>Create Drawings and Quantity Estimations</td>
<td>264</td>
</tr>
<tr>
<td>Print Calculations</td>
<td>265</td>
</tr>
<tr>
<td>Pile cap design procedures</td>
<td>265</td>
</tr>
<tr>
<td>Pile cap design example</td>
<td>265</td>
</tr>
<tr>
<td>Apply pile caps under supported columns</td>
<td>266</td>
</tr>
<tr>
<td>Auto-size pile caps individually for loads carried</td>
<td>266</td>
</tr>
<tr>
<td>Apply grouping to rationalize pile cap sizes</td>
<td>268</td>
</tr>
<tr>
<td>Review/Optimise Pile Cap Design</td>
<td>269</td>
</tr>
<tr>
<td>Create Drawings and Quantity Estimations</td>
<td>269</td>
</tr>
<tr>
<td>Print Calculations</td>
<td>270</td>
</tr>
<tr>
<td>Mat foundation design</td>
<td>270</td>
</tr>
<tr>
<td>Features of the mat foundation analysis model</td>
<td>270</td>
</tr>
<tr>
<td>Analysis Types</td>
<td>270</td>
</tr>
<tr>
<td>Soil Structure Interaction</td>
<td>270</td>
</tr>
</tbody>
</table>
Soil Parameters ..............................................................................................................271
Pile Springs .....................................................................................................................272
Typical mat foundation design procedure...................................................................272
  Design the structure before supporting it on the mat.............................................274
  Create the mat, (either with ground springs, or discreet supports) ......................274
  Model validation .......................................................................................................275
  Perform the model analysis ....................................................................................276
  Check foundation Bearing Pressure and Deformations ........................................276
  Re-perform member design ....................................................................................277
  Open an appropriate view in which to design the mat ............................................278
  Add Patches ...............................................................................................................278
  Design Mats ...............................................................................................................279
  Review/Optimise Mat Design ...................................................................................279
  Design Patches ..........................................................................................................280
  Review/Optimise Patch Design ................................................................................281
  Add and Run Punching Checks ...............................................................................281
  Create Drawings and Quantity Estimations ............................................................283
  Print Calculations .....................................................................................................283
Typical piled mat foundation design procedure .............................................................283
  Design the structure before supporting it on the mat.............................................284
  Create the mat ..........................................................................................................285
  Define the pile catalogue .........................................................................................286
  Add piles to the mat .................................................................................................286
  Model validation .......................................................................................................288
  Perform the model analysis ....................................................................................288
  Perform the pile design .........................................................................................288
  Review the pile design status and ratios .................................................................288
  Perform the mat design ............................................................................................290
Vibration of Floors to SCI P354 Handbook ....................................................................291
  Introduction to Floor Vibration (P354) ....................................................................291
  Scope ..........................................................................................................................292
  Limitations and Assumptions ..................................................................................292
  Design Philosophy ....................................................................................................293
  General .......................................................................................................................293
  Dynamic Excitation .................................................................................................294
  Required Performance ............................................................................................295
### Table of Contents

- Provided Performance ...................................................................................................... 295
- Provided Performance .......................................................................................................... 296
- System Frequency ............................................................................................................. 296
  - Deflections ...................................................................................................................... 296
- Secondary Beam Mode ......................................................................................................... 298
- Primary Beam Mode .......................................................................................................... 299
- System Frequency .......................................................................................................... 299
- Limitations ...................................................................................................................... 299
- Modal Mass ........................................................................................................................ 299
- Mode Shape Factor ........................................................................................................... 301
- Resonance Build-up Factor .............................................................................................. 301
- Response Acceleration ...................................................................................................... 302
  - Low Frequency Floors ................................................................................................... 302
  - High Frequency Floors .................................................................................................. 303
- Response Factor ................................................................................................................ 303
- Vibration Dose Values ....................................................................................................... 303
- Input Requirements .............................................................................................................. 304
  - General ................................................................................................................................ 304
  - Data Derived from Tekla Structural Designer ............................................................... 305
  - Unit mass ........................................................................................................................ 305
  - Slab data .......................................................................................................................... 305
  - Secondary beam data ................................................................................................... 305
  - Primary beam data ........................................................................................................ 306
  - Floor plate data .............................................................................................................. 306
- User Input Data .................................................................................................................. 306
  - Secondary Beam Spacing ............................................................................................. 306
  - Proportion of Imposed Loads ...................................................................................... 306
  - Number of bays used to establish Modal mass ........................................................... 306
  - Mode Shape Factors ...................................................................................................... 306
  - Damping ratio ................................................................................................................. 306
  - Maximum corridor length ............................................................................................ 307
  - Walking Pace ................................................................................................................... 307
  - Resonance build-up factor ............................................................................................ 307
  - Required Response Factor ........................................................................................... 307
  - Vibration Dose Value (VDV) .......................................................................................... 307
- References .......................................................................................................................... 308
Wind Modeling Handbook

This handbook describes two approaches for defining wind loads in Tekla Structural Designer.

• The **Wind Model** method is the most comprehensive - requiring you to first ‘clothe’ the structure in wind and roof panels and then run the ‘Wind Wizard’. The wizard creates wind zone loads that are subsequently decomposed to the structure during analysis.

• Alternatively you might choose to **manually apply wind loads** (thus avoiding the requirement to construct a wind model). For this approach loads can either be applied directly to the structure as Panel, Member, or Structure loads; or they can be applied as **Simple Wind** loads, which are subsequently decomposed to the structure during analysis.

**Overview of the Wind Model method**

The basic steps required for this method are as follows:

1. Clothe the structure in wind wall and roof panels
2. Perform the gravity design
3. Run the Wind Wizard
4. Review the wind zones
5. Define the wind loadcases
6. Review wind zone loads
7. Combine the wind loadcases into design combinations
8. Perform the static design
Clothe the structure in wind wall and roof panels

The Wind Model calculations depend on the geometry and inter-connectivity of the wall panels and roof panels that envelope the building. You must therefore define the model together with its wall and roof panels before you run the Wind Wizard.

You can, should you wish, use Tekla Structural Designer purely for wind assessment – by setting up a model of consisting only of wall panels and roof panels (no members) the software can determine the wind loading on the building envelope.

In order to get the best results you should ensure that you define the largest possible sizes for the wind wall and roof panels. The results may be compromised if you define many small panels rather than one large one. (Particularly the calculation of the reference height can be unconservative.)

Applying Wall Panels

A single wall panel is determined to be a single planar surface. The outward face is vitally important for determining the wind direction relative to the wall, that is windward or leeward.

It is recommended that you check the outward faces are as you intend by ensuring they all shaded in the same colour (the one assigned to ‘Wind Wall - Front’ in Settings > Scene). The inward faces will all be shaded in a different colour. To correct any mistakes, choose the Reverse command (located on the Edit toolbar) and then click once on a wall panel to switch its direction. Note that connected wall panels are checked to ensure that the normal directions are consistent whenever automatic zoning is carried out, for example at the end of the Wind Wizard. If there is a problem it is indicated on the Wind tab of the Project Workspace, with affected panels being marked thus: (”).

Once a wall panel has been placed the following additional panel properties can be specified:

- **Rotation angle** - defines the span direction, 0° is horizontal and 90° is vertical.
- **Is a parapet wall** - you can indicate whether the wall panel is a parapet or not.

If a building face comprises a parapet above a wall, you should not attempt to model this as a single wall panel. It should be input as an upper and lower panel, with the upper panel being set as a parapet.

- **Gap** - where the gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funnelling. A zero gap value explicitly means ignore funnelling, for example where this building and the adjacent one are sheltered by upwind buildings.
• **Solidity** - If you set the wall panel as a parapet, then you also need to indicate the Solidity of the parapet. (Wall panels that are not parapets automatically adopt a solidarity of 1.0).

• **Decompose to** - for wall panels that are not parapets, you can indicate how the wall load is decomposed on to supporting members. See [Wind Model Load Decomposition](#).

To set this information as you require, select the wall panels and then use the Properties Window to make changes.

### Applying Roof Panels

A single roof panel is determined to be a single planar surface. The orientation of a roof panel is automatically determined when placed based upon the slope vector – the line of maximum roof slope.

Initially the roof type is set to ‘Default’. This is interpreted as Flat if the roof slope < 5 degrees, otherwise it is interpreted as Monopitch. You should select the roof panel and then use the Properties Window to adjust the roof type as necessary for all other situations (i.e. For Duopitch, Hip Main, Hip Gable or Mansard).

The span direction is also set in the Properties Window, this is defined as an angle, where 0° is parallel to the X axis and 90° is parallel to the Y axis.

### Perform the gravity design

We recommend that you perform an analysis and design at this stage for the gravity loading only, but this is not essential.

### Run the Wind Wizard

Once the model has been ‘clothed’ in wall panels and roof panels, the **Wind Wizard** (located on the **Load** toolbar) guides you through the process of intelligently ‘applying’ wind to the resulting building envelope.

The wizard uses databases where appropriate to determine the appropriate wind details for your structure location.

Having defined the wind directions in which you are interested, on completion of the wizard the appropriate wind zones on the roofs and walls of your structure are automatically calculated.

### Review the wind zones

The resulting wind model is accessed from the Project Workspace Wind tab. **Wind Views** can also be opened as required for each wind direction.

From here you can set the type of each roof to achieve the correct zoning, and can then tailor the zoning to account for particular features in more detail, if you so require.
Define the wind loadcases

The **Wind Loadcases** dialog (located on the **Load** toolbar) can then be used to automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases the appropriate wind pressures are calculated on each zone.

> It is assumed that the wind loads are developed to assess the overall stability of the structure and for member design. The wind loads have not been specifically developed for the design of cladding and fixings.

Review wind zone loads

Wind zones can be graphically displayed for each wind direction from the appropriate **Wind View**. Once the wind loadcases have been created you can also display the wind pressures and zone loads for each loadcase.

Combine the wind loadcases into design combinations

Combine the wind loadcases into design combinations in the usual way.

Perform the static design

Run a static design from the **Design** toolbar.

**EC1991 1-4 Wind Wizard**

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to BS EN 1991-1-4.

Once the wall and roof panels are in place, you use the **Wind Wizard** to define sufficient site information to calculate the peak wind velocity and velocity pressures for the required wind directions and heights around the building, (that is the Reference Heights ($z_e$ and $z_i$) for each wall panel or roof panel).

The wind velocity calculations are automated, the data source for the calculations is either:

- taken directly from the BREVe database which is based upon the Ordnance Survey data of Great Britain.
- Input directly for the worst case,
- Input directly for each direction.
Design Codes and References

Unless explicitly stated all calculations in the EC1991 1-4 Wind Wizard are in accordance with the relevant sections of EC1991 1-4(Ref. 3) and the chosen National Annex. It is essential that you have a copy of this code with you while assessing wind on any structure.

We would recommend having the following books to hand when using the software:

- Wind Loading - a practical guide to BS 6399-2 Wind Loads on buildings.(Ref. 7)

In addition, you may find the following book useful:

- Background information to the National Annex to BS EN 1991-1-4 and additional guidance. PD 6688 - 1-4:2009(Ref. 5)

Unless explicitly noted otherwise, all clauses, figures and tables referred to in this section of the handbook are from reference 3.

Scope

There is no guidance in the standard for anything other than a cuboid building. In order to develop a tool for engineers, we have extended this capability to address non-rectilinear buildings. It is therefore the user's responsibility to ensure that the wind loading generated by the software meets the needs of any building with a shape that is beyond the scope of BS EN 1991-1-4:2005.

The scope of EC1991 1-4 Wind Wizard encompasses:

- Enveloping the building with wall panels and roof panels is undertaken in Tekla Structural Designer in the normal manner. There is only limited validation of the envelope defined (for example connected wall panels must have consistent normal directions). The onus is on you to model the building shape as completely and as accurately as you determine necessary.
- Basic Wind Velocity and Peak Velocity Pressure is determined.
- Having defined wall panels and roof panels (defaults are standard wall, flat or monopitch roof depending on the slope), you are able to specify the type in more detail e.g. multi-bay, monopitch / duopitch etc.).
- The main wind parameters, are calculated for you but conservatively, (for example Crosswind Breadth, b, is determined for the enclosing rectangle of the whole building). Wherever possible other parameters are determined conservatively, but you are able to override the values should you need to.
- Given the above, zoning is semi-automatic, (not attempted for roofs with more than 4 sides which are defaulted to single conservative coefficient), with full graphical feedback.
- The software follows the UK NA(Ref. 3) recommendation that BS6399 roof zones and coefficients are used, including Mansard, Multipitch and Multibay roofs.
Load decomposition is fully automatic where valid, (wall panels and roof panels need to be fully supported in the direction of span).

**Limitations**

Throughout the development of the Wind Wizard extensive reference has been made to the and we consider it advisable that you are fully familiar with these before using the software.

In addition, because wind loading is complex and its application to general structures even more so, it is essential that you read and fully appreciate the following limitations in the software:

**Geometry**

You should seek specialist advice for building shapes that are not covered by the Standard 1991-1-4:2005.

- EC1 1-4 does not treat downwind re-entrant corners as special cases - see BS6399 Clause 2.4.3.1 c). So, they are ignored in the software and no warnings are given.
- EC1 1-4 does not handle stepped profiles, or inset storeys - see BS6399 clauses 2.4.4.2 and 2.5.1.7. Hence the software does not handle them automatically, but does generate warnings if such cases are detected - so you can manually edit the zones according to your engineering judgement.
- Open sided buildings are beyond scope.
- Free standing walls and sign boards are not considered.
- Canopies are not considered.
- Exposed members are not considered, for example lattices, trusses......
- Barrel-vault roofs and domes are not considered.
- Dominant Faces are not explicitly handled - Clause 7.2.9 (5). However, you can use Table 17 to calculate the necessary Cpi value or values and manually apply to a loadcase or individual zone loads.

**Loaded Areas**

The difference between the loaded area of wall panels and roof panels defined at the centre-line rather than the sheeting dimension is ignored.

**Wind Direction**

- All outward faces within 60 degs of being perpendicular to wind direction - loads applied as windward normal to face. All inside faces within 60 degs to wind direction - loads applied as leeward normal to face. All other faces considered as side.
- Orthogonal wind directions at the definition of the user.

**Overall Loads**
• Lack of correlation of pressures between the windward and leeward sides. For Overall loadcases, the software automatically reduces the windward and leeward wall pressures only. EC1 1-4 and the UK NA both suggest that the reduction “may” be applied to roofs as well.

• Division by Parts rule for “slender” buildings -Clause 7.2.2 and Figure 7.4 - not applied.

• Net pressure coefficients for vertical walls - UK NA Table NA.4. - not applied.

• Friction Forces - Clause 5.3 (3), equation 5.7 and Clause 7.5. During the “Update Zones” process, checks are performed to see if the effects can be disregarded, (Clause 5.3 (4)), and a ‘Friction needed’ warning is generated if not. When they cannot be disregarded you will need to manually model the friction forces as lateral loads in a separate loadcase and include them in your combinations.

Beneficial Loads

• Asymmetric and Counteracting Pressures and Forces - Clause 7.1.2 and NA.2.23. Beneficial loads are not automatically removed - instead you are able to flag individual loads to be reduced to zero.

Singapore National Annex - Minimum Horizontal Loads

• The Foreword to the Singapore National Annex to EN 1991-1-4 Wind Actions has a minimum horizontal load requirement (1.5% characteristic dead weight). Therefore if this National Annex has been applied, it is the users responsibility to check that this requirement has been met (by ensuring that the horizontal component of the factored wind load is greater.

Wind loading on wall panels

Automatic zoning applies to all wall panels subject to the limitations described below:

• Vertical Walls on rectangular buildings -Clause 7.2.2 - the assumption for wall wind forces is that the building is rectangular or close to being rectangular.

• Wall panels that are more than 15° from the vertical are outside the scope.

• Internal Wells are not covered by EC1 1-4 and in any case are not automatically identified but you can manually edit the zones to apply the roof coefficient or otherwise as you see fit - see BS6399 Clause 2.4.3.2a.

• EC1 1-4 does not specify how to treat recesses in side walls - see BS6399 clauses 2.4.3.2 b) and 2.4.3.3 and 3.3.1.5. So, they are ignored but warnings are given.

Wind loading on roof panels

• Automatic zoning only applies to all triangular roof panels and quadrilateral roof panels that are not concave, i.e. all of the internal angles < 180°

• Special care should be taken for winds blowing on duopitch with slopes that differ by more than 5°. If the wind is blowing on the steeper slope (that is that the less steep slope is downwind of ridge), the downwind slope should be set to be a flat roof with mansard at eaves for this wind direction.
• Mansard and Multipitch Roofs are not detected automatically, although certain special cases can be handled if you set the appropriate type manually - see EC1991 1-4 Wind Zones.

• BS 6399 Table 8 curved and mansard eaves - zones start from edge of horizontal roof.

• Roof Overhangs are not explicitly handled. It is suggested that you should define two separate roof panels - one forming the overhang and the other covering the inside of the building. For a small overhang, you can then manually define Cpi values to be the same as Cpe for the adjacent wall panel, (Clause 7.2.1 (3)). Reference 6, p45, implies that larger overhangs can be manually handled by using BS6399, Clauses 2.5.9.3 and 2.6.3, i.e. standard external coefficients for the top surface and Table 18 for the internal coefficients.

The only slight issue here is that there are two sets of edge zones which will occupy a slightly larger area than strictly necessary.

Additional wind loads
There may be situations when you perceive a need to manually define loads that cannot be determined automatically. You can do this by defining additional wind load cases to contain these loads and then include these with the relevant system generated loads in design combinations in the normal way.

Using the EC1991 1-4 Wind Wizard with BREVe data

Data Source page
Assuming you have chosen to specify the site data using BREVe Grid Ref data the remaining choices on the Data Source page are:

Consider Orography
If you check this box, then the orographic data, (either recovered by BREVe for the site or manually entered), is used to determine the Orography Factor $c_o$ as clause A.3. When calculating $c_{alt}$ the altitude of the upwind base of the orographic feature is used for each wind direction considered. Otherwise the orographic data is ignored, $c_o$ is 1.0 for all heights and $c_{alt}$ is the same for all directions, using the Site Altitude.

Consider Tall Neighbouring Structures
If the conditions in clause A.4 are met, then the wind loads need to be based on height $z_n$, see equation (A.14). With this box checked, you are able to enter sufficient data to check if this applies. If applicable, then $z_n$ will be used as the reference height for all wall panels and roof panels in the model.

Consider Obstructions
With this box checked, the obstruction data, (either defaulted by BREVe depending on the roughness category for the site or entered manually), is used to determine the Displacement Height, \( h_{\text{dis}} \) as (A15) in clause A.5. Otherwise the obstructions are ignored and \( h_{\text{dis}} \) is taken as zero.

**BREVe location page**

This page allows you to define the location of the site using the BREVe database, and also the orientation if known.

**Building details**

**Grid Ref.**

This shows the grid reference of the site which you have picked through BREVe, irrespective of the method you use to define the site location.

**Orientation of building known**

If you know the orientation of the building with respect to North, then you can define this information by checking this box. You can then define a value which relates the building direction axes of your *Tekla Structural Designer* model to geographic north.

**Orientation of North**

The orientation of North is defined using the same convention as is applied to the orientation of the Building Direction Arrows.

This can best be understood by reference to a couple of examples:

In the first example the building axes are aligned in the default directions (Dir 1 = 0° = Global X), and the orientation of North has been set to 315°.

The resulting relation between the building axes and North is as shown below:
In the second example the building direction has been input with Dir 1 = 30° and the orientation of North has been set to 250°.

In this case the building axes are related to North as shown below:
BREVe information

Using BREVe, there are 2 methods available for you to define the site location:

**Site By Ref...**
You can define the grid reference of the site.
You define this either as a national grid reference, or by specifying the Easting and Northing information for the site. There are several Internet based tools available which allow you to determine the Ordnance Survey grid reference from a postcode or given location, for example [www.streetmap.co.uk](http://www.streetmap.co.uk) or [www.multimap.co.uk](http://www.multimap.co.uk).

**Site By Map...**
You can pick the site from a Land / Town Map,
  - You can pick the site from a Orography Map.
  - You can pick the site from a ground roughness Category Map,

The site data is analysed fully by BREVe. Parameters are either set automatically but conservatively (Safe parameters within a 1 km square).

**Basic Data page**
This page allows you to define the site details when information is not available from the BREVe database, for instance if it is located outside of the UK.
Site Altitude, A

You need to enter the basic altitude that you want to use for the site directly. This is the altitude of your model's base.

Air Density

You need to enter air density at the site.

Ground Level

If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example a site datum may have been used rather than a building datum, then this field allows you to set the appropriate value so that the reference heights can be calculated correctly.

Fundamental Basic Wind Velocity ($v_{b,\text{map}}$) - Clause 4.2 and NA.2.4

The value required is defined as "the characteristic 10 minutes mean wind velocity, irrespective of wind direction and time of year, at 10m above ground level in open country terrain with low vegetation such as grass and isolated obstacles with separations of at least 20 obstacle heights", but is the value before the altitude correction is applied. The default is zero to force you to enter a value (valid range 1.0 to 1000 m/s).

Season Factor, $c_{\text{season}}$

Valid range 0.01 to 10.0 - default 1.0.

Probability Factor, $c_{\text{prob}}$

Valid range 0.01 to 10.0 - default 1.0.

Default Height for Internal Pressure ($z_i$)

Clause 7.2.9 (7) implies that all internal pressures should be calculated using a single reference height, ($z_i$), defaulting to the height of the structure. Leaving Use Building Height checked ensures that the value is automatically updated if the geometry of wall panels or roof panels changes.

Roughness and Obstructions (BREVe) page

The next page of the Wizard automatically defaults the data for ground roughness and obstructions for you.

Terrain Category

- **Sea** – this setting is for sites where the distance to sea is between 0 and 1 km, not for offshore sites. As the worst case must be for wind blowing across the sea, there is no need to specify data for upwind buildings or distance in town.
• **Country** – the worst case must be for wind blowing across open ground, there is no need to specify data for upwind buildings or distance in town.

• **Town** – for this category you need to specify data for upwind buildings and distance to the edge of the town, so the relevant fields are active. If you want to ignore obstructions, then you need to enter a zero value for \( h_{\text{ave}} \) For this category, the **Upwind distance from edge of town to site** can not be greater than the **Upwind distance from sea to site**.

### Orography (BREVe) page

If **Consider Orography** was checked, then the next page of the Wizard allows you to enter the data for Orography (topography) in each direction.

#### Orographic Feature (Clause A.3)

- **None** – no feature, i.e. \( c_o = 1.0 \).
- **2D Escarp** – Cliffs and Escarpments,
- **3D Hill** – Hills and Ridges.

#### Altitude of Upwind Base of Feature, \( A \)

This value is used to calculate \( C_{alt} \) instead of the Site Altitude because the Orography is significant.

\[
C_{alt} \text{ will be calculated at } z_e \text{ for each wall and roof panel, not } z_s.
\]

#### Effective Crest Height, \( H \) (Figures A.2 & A.3)

Effective height of the feature.

#### Length of Upwind Slope, \( L_u \) (Figures A.2 & A.3)

Actual length of the upwind slope in the wind direction.

#### Length of Downwind Slope, \( L_d \) (Figures A.2 & A.3)

Actual length of the downwind slope in the wind direction.

#### Horizontal Distance to Crest, \( x \) (Figures A.2 & A.3)

Distance upwind or downwind from the crest to the building site.

### Tall Neighbouring Structure page

For all methods, , if **Consider Tall Neighbouring Structure** was checked, the penultimate page allows you to determine if tall neighbouring structures affect the design of this structure. (Clause A.4)
Otherwise, the wizard will proceed directly to the Results page and the actual heights of wall and roof panels are used throughout.

The parameters, Height of Tall Neighbour, $h_{\text{high}}$, Largest Horizontal Dimension of Tall Neighbour, $d_{\text{large}}$ and Distance to Tall Neighbour, $x$ are all as described on Figure A.4 of the code.

**Average Height of Neighbours, $h_{\text{ave}}$ (Figure A.4)**

The default is calculated from the BREVe data or the values entered for the Roughness & Obstructions. If **Override calculated dimension** is unchecked, then the value will be updated whenever the wizard is run, otherwise the user-value is used.

**Height of this structure, $h_{\text{low}}$ (Figure A.4)**

The field is for information only - difference between top of highest wall / roof panel and ground level in the model.

**Results (BREVe) page**

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

Initially there are 4 orthogonal wind directions relative to the *Tekla Structural Designer* axes, (not geographical North), but you are able to update this using the **Dir.** buttons and / or changing the direction value as required. (Note : Minimum 1° difference between directions). At least one direction must be defined.

You are able to use the **Details...>>** button to obtain additional information, including the values of intermediate factors used in the calculations.

**BREVe Data**

BREVe determines the parameters required to calculate $q_p(z)$ for each height in the building at 30° intervals, (0° to 330°).

For each required wind direction the worst case $q_p$ is used for each height, based on splitting the difference to the next direction, with a maximum of ±45 degrees. Within these ranges $q_p$ is not interpolated.

Theoretically, it is possible for a quadrant to use different 30° directions for each height, so the critical wind direction is not displayed in the summary.

The Vortex view shows the Peak Velocity Pressures calculated for each reference height for each 30° sector.

**Finishing the Wind Wizard**

When you click **Finish**, the **Wind Wizard** generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no have been encountered.
Using the EC1991 1-4 Wind Wizard with other data

Data Source page

Assuming you have chosen to enter the site data manually (Other), then the you can choose to enter one set of Worst-Case data or different values for each direction to be considered.

The remaining choices on the Data Source page are:

Consider Orography

If you check this box, then the orographic data, (manually entered), is used to determine the Orography Factor $c_o$ as clause A.3. When calculating $c_{alt}$, the altitude of the upwind base of the orographic feature is used for each wind direction considered. Otherwise the orographic data is ignored, $c_o$ is 1.0 for all heights and $c_{alt}$ is the same for all directions, using the Site Altitude.

Consider Tall Neighbouring Structures

If the conditions in clause A.4 are met, then the wind loads need to be based on height $z_n$, see equation (A.14). With this box checked, you are able to enter sufficient data to check if this applies. If applicable, then $z_n$ will be used as the reference height for all wall panels and roof panels in the model.

Consider Obstructions

With this box checked, the obstruction data, (either defaulted by BREVe depending on the roughness category for the site or entered manually), is used to determine the Displacement Height, $h_{dis}$ as (A15) in clause A.5. Otherwise the obstructions are ignored and $h_{dis}$ is taken as zero.

Basic Data page

This page allows you to define the site details when information is not available from the BREVe database, for instance if it is located outside of the UK.

Site Altitude, A

You need to enter the basic altitude that you want to use for the site directly. This is the altitude of your model's base.

Air Density

You need to enter air density at the site.

Ground Level

If for some reason, the level 0.0m in the Tekla Structural Designer model does not correspond to the ground level, for example a site datum may have been used rather than a building datum, then this field allows you to set the appropriate value so that the reference heights can be calculated correctly.
**Fundamental Basic Wind Velocity \( v_{b,\text{map}} \) - Clause 4.2 and NA.2.4**

The value required is defined as "the characteristic 10 minutes mean wind velocity, irrespective of wind direction and time of year, at 10m above ground level in open country terrain with low vegetation such as grass and isolated obstacles with separations of at least 20 obstacle heights", but is the value before the altitude correction is applied. The default is zero to force you to enter a value (valid range 1.0 to 1000 m/s).

**Season Factor, \( c_{\text{season}} \)**

Valid range 0.01 to 10.0 - default 1.0.

**Probability Factor, \( c_{\text{prob}} \)**

Valid range 0.01 to 10.0 - default 1.0.

**Default Height for Internal Pressure (\( z_i \))**

Clause 7.2.9 (7) implies that all internal pressures should be calculated using a single reference height, \( z_i \), defaulting to the height of the structure. Leaving **Use Building Height** checked ensures that the value is automatically updated if the geometry of wall panels or roof panels changes.

**Roughness and Obstructions (Other - Worst Case) page**

If you select the **Other - Worst Case Data Source**, then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself.

**Terrain Category**

- **Sea** – this setting is for sites where the distance to sea is between 0 and 1 km, not for offshore sites. As the worst case must be for wind blowing across the sea, there is no need to specify data for upwind buildings or distance in town.
- **Country** – the worst case must be for wind blowing across open ground, there is no need to specify data for upwind buildings or distance in town,
- **Town** – for this category you need to specify data for upwind buildings and distance to the edge of the town, so the relevant fields are active. If you want to ignore obstructions, then you need to enter a zero value for \( h_{\text{ave}} \). For this category, the **Upwind distance from edge of town to site** can not be greater than the **Upwind distance from sea to site**.

**Roughness and Obstructions (Other - Data for each Direction) page**

If you select the **Other - Data for each Direction Data Source**, then the next page of the Wizard allows you to enter the data for ground roughness and obstructions yourself. However, most of the data is then dependent on the wind direction, so you must also make your choice of wind directions on this page.

**Direction**
Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but you are able to update these using the Dir. buttons and / or changing the direction value as required. (Note: Minimum 1° difference between directions). At least one direction must be defined.

Each row of the grid operates in a similar manner to the relevant fields of the “Roughness and Obstructions (Other - Worst Case) page”

**Orography (Other - Worst Case) page**

If Consider Orography was checked, then the next page of the Wizard for the Other - Worst Case Data Source allows you to enter the data for Orography.

**Orographic Feature (Clause A.3)**

- **None** – no feature, i.e. $c_o = 1.0$.
- **2D Escarp** – Cliffs and Escarpments,
- **3D Hill** – Hills and Ridges.

**Altitude of Upwind Base of Feature, A**

This value is used to calculate $C_{alt}$ instead of the Site Altitude because the Orography is significant.

$C_{alt}$ will be calculated at $z_e$ for each wall and roof panel, not $z_s$.

**Effective Crest Height, H (Figures A.2 & A.3)**

Effective height of the feature.

**Length of Upwind Slope, $L_u$ (Figures A.2 & A.3)**

Actual length of the upwind slope in the wind direction.

**Length of Downwind Slope, $L_d$ (Figures A.2 & A.3)**

Actual length of the downwind slope in the wind direction.

**Horizontal Distance to Crest, x (Figures A.2 & A.3)**

Distance upwind or downwind from the crest to the building site.

**Orography (Other - Data for each Direction) page**

The wind directions defined on the previous page are maintained and you are not able to update them.

Each row of the grid operates in a similar manner to the relevant fields of the Orography (BREVe) page.
Tall Neighbouring Structure page

For all methods, if Consider Tall Neighbouring Structure was checked, the penultimate page allows you to determine if tall neighbouring structures affect the design of this structure. (Clause A.4)

Otherwise, the wizard will proceed directly to the Results page and the actual heights of wall and roof panels are used throughout.

The parameters, Height of Tall Neighbour, \( h_{\text{high}} \), Largest Horizontal Dimension of Tall Neighbour, \( d_{\text{large}} \) and Distance to Tall Neighbour, \( x \) are all as described on Figure A.4 of the code.

Average Height of Neighbours, \( h_{\text{ave}} \) (Figure A.4)

The default is calculated from the BREVe data or the values entered for the Roughness & Obstructions. If Override calculated dimension is unchecked, then the value will be updated whenever the wizard is run, otherwise the user-value is used.

Height of this structure, \( h_{\text{low}} \) (Figure A.4)

The field is for information only - difference between top of highest wall / roof panel and ground level in the model.

Results page

The final page of the Wizard is a summary of the results - peak velocity pressure ranges.

Initially there are 4 orthogonal wind directions relative to the Tekla Structural Designer axes, (not geographical North), but except for the Other - Data for Each Direction method, you are able to update this using the Dir. buttons and / or changing the direction value as required. (Note : Minimum 1° difference between directions). At least one direction must be defined.

You are able to use the <<Details>> button to obtain additional information, including the values of intermediate factors used in the calculations.

Other - Worst Case Data

The calculation of \( q_p \) is very similar to the BREVe Method, (see above), except that the worst case data has been entered by you, and this page allows you to enter your own values for \( C_{\text{dir}} \).

As there is no data for each 30° sector, the Vortex view only shows the Peak Velocity Pressures calculated for each reference height for each Fastrak direction.

Other - Data for each Direction

The calculation of \( q_p \) is very similar to the BREVe Method, (see above), except that the data has been entered by you for each Fastrak direction only so a direct calculation can be performed instead of taking the worst case over a range of sectors. Also this page allows you to enter your own values for \( C_{\text{dir}} \).
As there is no data for each 30° sector, the Vortex view only shows the Peak Velocity Pressures calculated for each reference height for each Fastrak direction.

**Finishing the Wind Wizard**

When you click Finish, the Wind Wizard generates the wind zones for the entire building for each of the specified wind directions.

Before moving on you should take a moment to inspect the Wind Model status on the Project Workspace> Status tab, in order to check that no have been encountered. **EC1991 1-4 Wind Zones**

**EC1991 1-4 Wind Zones**

At the end of the Wind Wizard, the system creates default zones for all the walls and roof panels for each of the defined wind directions.

If any errors have occurred in this process, a red cross appears next to Pressure Zones in the Project Workspace.
Wind Model Loadcases

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate wind loadcases manually. This is achieved via the Wind Loadcases dialog (accessed from the Load toolbar).

The Auto button on the dialog provides various options to control the number of loadcases before they are generated (its intention being to prevent generation of very large numbers of loadcases). Alternatively you can create loadcases manually using the Add button.

You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard load cases so you can include them in combinations in the normal manner.

Try to use engineering judgement to identify the critical loadcases so that the number of load combinations that need to be considered can be minimized.

EC1991 1-4 - Creating Wind Loadcases

It is not practical to automatically determine critical combinations and thus required wind loadcases, therefore you control the generation of appropriate Wind Loadcases manually. This is achieved via the Wind Loadcases dialog (accessed from the Load toolbar.

The Auto button on the dialog creates a default set of loadcases in each of the directions, i.e.

- -0.3 for Cpi with -ve roof Cpe; not Overall
- -0.3 for Cpi with +ve roof Cpe; not Overall
- +0.2 for Cpi with -ve roof Cpe; not Overall
- +0.2 for Cpi with +ve roof Cpe; not Overall
- Overall with zero for Cpi; -ve roof Cpe

Alternatively you can create loadcases manually using the Add button. You then specify which direction the loadcase will be created for and set default values for all the zone loads generated in the loadcase.

Once generated these loadcases are standard load cases so you can include them in combinations in the normal manner.
Try to use engineering judgement to identify the critical loadcases so that the number of load combinations that need to be considered can be minimized.

**Wind Loadcases dialog**

As well as specifying which direction the loadcase will be created for, this dialog allows you to set default values for all the zone loads generated in the loadcase.

### Fields

**Structural Factor - Automatically calculate separate $c_s$ and $c_d$ factors**

The UK NA states that the Structural Factor, $c_sc_d$ may be separated into a size factor $c_s$ and a dynamic factor $c_d$, i.e. it is still acceptable to apply Clause 6.2 (1 a) to d) and set $c_sc_d = 1$, or use Annex D. Where **Structural Factor - Automatically calculate separate $c_s$ and $c_d$ factors** is checked, $c_d$ is calculated using Figure NA.9 and $c_s$ using Table NA.3.

**Structural Damping**

The $\delta_s$ value is used to determine the dynamic factor $c_d$. It is only visible if the Separate Factors box is checked. See Table F.2.

**Name**

The loadcase name is auto generated from the other input parameters, but it can be edited if required.

**Direction**
The direction the loadcase will be created for is selected from the droplist.

**Overall**
You are able to flag if the loadcase is to be specifically used for examining the overall behaviour of the structure by checking this box.

It may be necessary for you to create a second copy of the loadcase with this box unchecked if the loadcase is also used for designing elements.

**b+h**
When designing elements, (beams, columns, braces etc), Table NA.3 in the UK NA implies that b and h should be the width and height respectively of an element. Due to the nature of the loads in the program, it is not practical to do this automatically, and so you should specify a value to be used in the loads generated for this loadcase (default 5.0m).

If separate factors are not to be used, (i.e. use combined $c_s c_d$), then this value is redundant.

If separate factors are to be used, but **Overall** is checked on the row, then the b+h cell is marked inactive. In this case, for each wall and roof, the program calculates b+h using b & h from the zone properties for the relevant wind direction.

**Use +ve C_{pe}**
Where 2 sets of coefficients are given in a BS6399 Table for roof zones, this field indicates if the negative or positive $C_{pe}$ value is to be used.

**C_{pi}**
Default Internal Pressure Coefficient (-0.3, 0.0, +0.2 or other value) - to be calculated by you from Clause 7.2.9.

**Buttons**

Click this button to add a single wind loadcase.

Click this button to delete a wind loadcase.

Click this button to create a default set of loadcases in each of the directions.

**Wind Model Load Decomposition**

**Roof Panel Load Decomposition**
The direction of the one way decomposition of the wind zone loads to roof members is determined by the span direction (i.e. rotation angle) of the roof panel.

All types of elements within the wind wall plane (except bracing and cold rolled members) are considered during the load decomposition.
Wind Wall Panel Load Decomposition

Decomposition Options

Wind zone loads are decomposed on to the structure according to the setting of the ‘Decompose to’ wind wall property, which can be set to:

• Members,
• Nodes,
• Rigid Diaphragms

To demonstrate the effect of the different Decompose to’ settings, consider the braced steel frame clad in wind walls shown below:
For wind direction 0 the wind model produces Zone Loads as follows:

The related topics below illustrate how the above zone loads are decomposed for the three different options.

**Decompose to Members**

Decomposition to members is similar to the roof panel decomposition, the direction of the one way decomposition being specified by the span direction of the wind wall panel. All types of elements within the wind wall plane are considered except bracing and cold rolled members.

Decomposition to members allows the generation of UDL's on portal stanchions and gable posts without the need to model side rails.

So for the model shown in Decomposition Options, choosing ‘decompose to members’ would produce the following loads:
Decomposition to Members (rotation angle 0 degrees)

Decomposition to Members (rotation angle 90 degrees)

**Decompose to Nodes**

Decomposition to nodes is the default setting and results in nodal loads on the supporting members. This setting is generally appropriate to avoid lateral loads on simple beams.

All types of elements within the wind wall plane are considered except bracing and cold rolled members.

The initial decomposition is the same as for members, with the direction of the one way decomposition being specified by the span direction of the panel. A second decomposition stage is then undertaken to convert the member loads to nodal loads:

- Full/partial UDLs and VDLs on elements (lengths of beams/columns between nodes) are distributed back to nodes as if the elements were simply supported at either end.
- The final nodal load is the sum of all incoming element loads.

For the model shown in [Decomposition Options](#), choosing ‘decompose to nodes’ would produce the following loads:
In the above example, when the rotation angle is 0 degrees some of the nodal loads are applied directly to supports.

**Decompose to Rigid Diaphragms**

Decomposition to rigid diaphragms does not need to consider the span direction of the wall panel (so the rotation angle is irrelevant).

It is particularly useful for flat-slab structures, as the alternative ‘member’ or ‘node’ decomposition methods require supporting members that may not exist in the model.
All rigid diaphragms within the wind wall height are considered for decomposition irrespective of whether they are physically connected to the wind wall.

For the model shown in Decomposition Options, choosing ‘decompose to rigid diaphragms’ would produce the following loads:

Decomposition to Diaphragms - each Zone Load is decomposed as a separate point load on the diaphragm

**Validation of Panels set to Rigid Diaphragm Decomposition**

The following validation checks are performed for wind wall panels set to decompose to rigid diaphragms:

- Each panel must be rectangular
- The top level of each panel must align with a rigid diaphragm
- Each panel may be sub-divided into zones, but only by horizontal lines
- Unlike for ‘member’ or ‘node’ decomposition, each panel does **not** need to have supporting members along its edges.

**Alternative decomposition methods for complex models**

For complex models the placement of wind walls required by the Wind Model method can be a time consuming operation.

In such situations (provided the model has suitable rigid diaphragms) the following alternatives offer quicker and simpler ways to apply approximate wind loads:
• use engineering judgement to clothe the structure with an arrangement of Simplified Wind Wall Panels around its bounding box, set the wind walls to decompose to rigid diaphragms, and proceed with the Wind Model method
• don't apply wind walls - consider Application of manual wind loads instead

**Simplified Wind Wall Panels**

In theory, because wind walls set to decompose to rigid diaphragms don't have to be physically connect to the diaphragms, even a very complex model could be clothed with just four simplified wind walls defined along its bounding box, as illustrated below:
Simplified Wind Walls around the 'bounding box'

Zone Loads for the Simplified Wind Wall Model
Zone Loads Decomposed to Rigid Diaphragms

Such an approach could be used to rapidly establish approximate wind loads (based on the rectangular building envelope) which could then be refined at a later stage if necessary.

If this method is adopted you are strongly advised to review the wind zones that are formed, and the resulting decomposed loads to ensure they meet your expectations - it will not give good results for all models.

If the decomposed loads produced by the above approach are not satisfactory, you might decide to take greater control and consider Application of manual wind loads instead.

Application of manual wind loads

This approach provides a quick means to apply wind to the structure, without requiring you to create a wind model.

In order to manually apply wind loads, you must first create a loadcase for them and set its load type to Wind.

Panel, Member, and Structure loads can then be manually applied in this loadcase as required. You can also apply 'Simple Wind' loads in the same loadcase.

'Manual' wind loadcases are created from the ‘Loadcases’ dialog not from the ‘Wind Loadcases’ dialog (which is available for 'Wind Model' wind loadcases only).
Simple Wind

Overview of Simple Wind

There is a simple process to follow when you want to apply Simple Wind loads. The basic steps are detailed below.

1. Define the structure ensuring that slabs have been created with the diaphragm option set to rigid as opposed to semi-rigid.

2. In order access Simple Wind, you must first create and then select the loadcase into which the Simple Wind loads are to be added.

3. Click Simple Wind on the Load ribbon to define the wind loads to be applied in the selected wind loadcase.

4. Combine the wind loadcases with the other loadcases you have defined for your structure to create the design combinations you need to consider.

5. Perform the analysis and design of the structure.

Simple Wind application and decomposition

Application

Simple Wind loads require rigid diaphragms to exist within the width of the load.

Each Simple Wind load is applied over a defined width and height onto the structure.

- The load defaults to being uniform over the full height (as shown above), but levels can be inserted to cater for a stepped loading profile if required. The inserted levels do not have to coincide with actual building levels.
- The top of the load should align with a rigid diaphragm - if not a validation error is generated - this is the case to ensure the loading is distributed as correctly as possible.
- Only the load within the building profile is considered for decomposition.
Decomposition

Simple Wind loads are decomposed to point loads on rigid diaphragms only. They are not decomposed to semi-rigid diaphragms.

- All rigid diaphragms on the top or bottom level or anywhere in-between are considered, with the area load being divided between the levels before it is decomposed.

- The loads are then decomposed to the diaphragms at each level in proportion to the width of each diaphragm. Each load being applied as a nodal load in the direction of the Simple Wind load at the mid point of the projected load.

- If there are no suitable diaphragms on the top level, the load is applied at the next level down.
• Similarly if the 'Ignore diaphragms on lowest level' box is checked on the Simple Wind Loading dialog or there are no suitable diaphragms on the bottom level, the load is applied at the next level up.

If for some reason there are diaphragms at the ground level, then you may decide to check the 'Ignore diaphragms on lowest level' box in order to ensure no load is lost directly to the foundations.

Limitations of wind decomposition to diaphragms

Irrespective of whether loads are input via the 'Wind Model' method, or via 'Simple Wind' loads, certain building shapes need extra consideration when rigid diaphragm load decomposition is applied.

As illustrated by the following examples, buildings containing discreet towers (and thus containing disconnected rigid diaphragms) are a particular concern.

Wind load perpendicular to disconnected diaphragms

In this example wind load is applied in the Global Y direction, perpendicular to the disconnected diaphragms that exist at the second floor level.

Disconnected diaphragms at second floor level

An issue arises when a wind load is applied in such a way that it has to be decomposed to both diaphragms. Such a load could be applied either via a Simple Wind load, or a wind wall panel:
Irrespective of the method used to apply it, the area load within the building profile is shared between levels prior to decomposition.
The load is then decomposed to the diaphragms at each level in proportion to the width of each diaphragm.

In this case because both diaphragms the second floor level are of equal width, the load is shared equally between them. This is not satisfactory as more of the load should have been applied to the left hand diaphragm in this case.

The workaround varies according to the method of loading, but basically involves replacing the original load with separate loads in each bay:
Simple Wind Workaround - Load re-applied as 3 Simple Wind Loads

Wind Wall Workaround - Load re-applied via 3 Wind Wall Panels

In both the above cases, the load is then decomposed as originally intended.
Wind load parallel to disconnected diaphragms

In this example although the two blocks are now the same height, another issue arises when the wind load is applied in the Global X direction, i.e. parallel to the disconnected diaphragms at the second floor level:

Disconnected diaphragms at second floor level

The issue arises because one diaphragm is ‘hidden’ from the applied load by the other diaphragm. The issue occurs irrespective of whether the load is applied via a Simple Wind load, or a wind wall panel:
Load applied via a Simple Wind Load

Load applied via a Wind Wall Panel

The area load within the building profile is first shared between levels prior to decomposition.

At the second floor level, instead of all the load being decomposed to the diaphragm facing the wind; because it is decomposed in proportion to the width of each diaphragm at that level, it ends up being shared equally to both.
To avoid the load being shared equally:

- if using wind panels - you would need to decompose to members or nodes instead of to diaphragms
- if using Simple Wind loads - there is no workaround, you would have to manually input the loads instead.

References


9. BREVe software package version 3. *Copyright © 2009 CSC (UK) Ltd; BRE Ltd; Ordnance Survey.*
Introduction to stability requirements

The analysis and design process has to allow for the differences between a theoretical mathematical model of a building and a more realistic representation. For example, buildings are not truly vertical when first built nor do they remain so when subject to load. These are called Stability Requirements and are from four sources:

1. **Global second-order effects** to allow for deformation of the structure under load,

2. **Member second-order effects** to allow for deformation of the members under load,

3. **Global imperfections** - additional second order effects due to the structure not being built plumb and square,

4. **Member imperfections** - additional second order effects due to initial lack of straightness of the member.

There are various methods of allowing for each of these and they can be different for steel and concrete. There is also some variation based on country code.

It will be found in the foregoing that,

- Global second-order effects can be ignored when the building is 'non-sway' - the opposite being 'sway sensitive',
- Member second-order effects can be ignored when the member is 'non-slender' (concrete) - the opposite being 'slender' - or is intrinsically allowed for in the design equations (steel),
- Global imperfections are provided for by Equivalent Horizontal Forces
- Member imperfections are allowed for directly in design (concrete) or is intrinsically allowed for in the design equations (steel).

**Global second-order effects**

Global second-order effects can be ignored when the building is 'non-sway', but must be considered if the building is 'sway sensitive'.

**Choice of analysis type (Eurocode)**

First or second order analysis?
You have the choice of three analysis types on the Analysis page of the Design Options dialog. These are,

- First-order (Elastic) analysis,
- Amplified forces ($k_{amp}$) method (uses first-order analysis),
- Second-order analysis.

**First-order (Elastic) analysis**

For both steel and concrete, first-order analysis is only acceptable providing second order effects are small enough to be ignored - see: [When should a building be classed as sway sensitive?](#)

**Amplified forces ($k_{amp}$) method**

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (basically regular frameworks with $\alpha_{cr} > 3$ - see ). Full second-order analysis is preferred for steelwork and since it is not precluded by EC2 it can be used for concrete.

The amplified forces method is described differently in EC3 compared to EC2, whilst the presentations are different, they are both based on the amplifier, $k_{amp}$ given as,

$$k_{amp} = \frac{1}{1 - 1/\alpha_{cr}}$$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of $k_{amp}$. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination).

**Second-order analysis**

Full second-order analysis is more widely applicable for steelwork structures and since it is not precluded by EC2 it can be used for concrete.

The accuracy of determination of the second-order effects for concrete structures is dependent upon a reasonable estimation of the concrete long term properties. This is a significant issue for both the amplified forces method and second-order analysis. It is therefore important that appropriate member type specific modification factors have been specified - see [Modification Factors](#).

- [Calculation of the elastic critical load factor](#)

**A practical approach to setting the analysis type**

Unless $\alpha_{cr}$ is greater than 10 (in which case second-order effects can be ignored), it is essential that your final design utilizes one of the second-order analysis approaches. During the initial sizing process you may however choose to run a first-order analysis. Proceeding in this way you can obtain sections and an overall building performance with which you are satisfied, before switching to one of the P-∆ analysis methods.
The following approach to setting the analysis type is suggested:

1. On the Analysis page of the Design Options dialog, initially set the analysis type to First-order analysis.

2. Perform **Design All (Gravity)** using first-order analysis in order to size members for the gravity loads.

3. Once the members are adequately sized for the gravity combinations obtain a figure for the building’s elastic critical load factor, $\alpha_{cr}$ (See: How do I assess the worst elastic critical load factor for the building?)

4. If the $\alpha_{cr}$ that has been determined is greater than 10 you can continue to perform Design All (Static) with the analysis type set to First-order analysis.

5. If $\alpha_{cr}$ is less than 10 you will need to proceed with one of the second-order approaches - and if it is very low (i.e. less than 2.0) some remodelling is required:
   
   • Either, refine the design until $\alpha_{cr}$ is greater than 2.0 to make the structure suitable for a final design using the full second-order approach, (which is the only method permitted if the structure contains non-linear members such as tension only braces),
   
   • or, in order to use the amplified forces approach, refine the design further until $\alpha_{cr}$ is greater than 3.0.

6. When a suitable $\alpha_{cr}$ has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate.

7. With the analysis type still set to full second-order, or the amplified forces method, perform **Design All (Static)**

**If full second-order analysis is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.**

**If you use the ‘Second-order analysis - Amp. forces method’ be aware that EC3 classes certain structures outside of its scope (see Validity of the amplified forces method). Such structures would need to be refined during gravity sizing until the elastic critical load factor is at least greater than 2.0, so that the full second-order approach can be used for the full design.**

First or second order analysis?

Validity of the amplified forces method
EC3 Clause 5.2.2 (6)B lists limitations on the applicability of the Amp. Forces method. It is therefore your responsibility when selecting this method to ensure all of the following:

• all storeys have a similar distribution of vertical load
• all storeys have a similar distribution of horizontal load
• all storeys have a similar distribution of frame stiffness with respect to the applied storey shear forces

Also according to clause 5.2.1 (4)B limitation:

• roof slope shallow - not steeper than 1:2 (26 degs)
• axial compression in beams or rafters - $N_{cr} / N_{ed} \leq 11.1$

**Amplified forces (kamp) method**

**Sway sensitivity assessment (Eurocode)**

Susceptibility to second order effects is a general characteristic and is not material specific, it has just been presented differently in EC3 and EC2:

• In EC3 a building can be considered as ‘non-sway’ when the elastic critical load factor $\alpha_{cr} \geq 10$, else the building is ‘sway sensitive’ and (global) second-order effects must be taken into account.

• In EC2 the definition is slightly different - it does not use the terms ‘non-sway’ and ‘sway sensitive’. Rather it simply defines when second-order effects are small enough to be ignored. The principle is given in Clause 5.8.2 (6) which states that they can be ignored if they are less than 10% of the corresponding first order effects. Because of the way in which the amplification factor, $k_{amp}$ is calculated the change point is at an $\alpha_{cr}$ of 11 not 10. (See: ).

However, the intent is the same in both cases and so in *Tekla Structural Designer* $\alpha_{cr} \geq 10$ is taken as the change point. In any event, you are not restricted in your choice of analysis type irrespective of the value of $\alpha_{cr}$ (it is your call, although we will warn you about it).

---

*A suggested approach for assessing sway sensitivity and considering global second order effects is given in the topic: A practical approach to setting the analysis type*

---

**Calculation of the elastic critical load factor**

The elastic critical load factor, $\alpha_{cr}$ is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Equivalent Horizontal Forces (EHF) are determined. These consist of 0.5% of the vertical load at each column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the EHF loadcases the deflection at each storey node in every
column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of \( \alpha_{cr} \) for that combination as follows,

\[
\alpha_{cr} = \frac{h}{200 \times \delta_{EHF}}
\]

Where

- \( h \) = the storey height
- \( \delta_{EHF} \) = the storey drift in the appropriate direction (1 or 2) for the particular column under the current combination of loads

Note that within each column’s properties, a facility is provided to exclude particular column stacks from the drift check calculations to avoid spurious results associated with very small stack lengths.

**Derivation of the kamp formula for concrete structures**

EC2 provides two specific approaches to determine the change point below which second-order effects are small enough to be ignored:

The first specific approach is contained in **Clause 5.8.3.3** which provides a pass/fail criterion to check whether the global second-order effects may be ignored. It is given as,

\[
F_{VEd} = k_1 \times \frac{n_s}{n_s + 1.6} \times \frac{\Sigma(E_{cd} \times I_c)}{L^2}
\]

where

- \( F_{VEd} \) = the total vertical load (on ‘braced’ and ‘bracing’ members)
- \( k_1 \) = a factor that allows for cracking in the concrete of the LLRS and is a Nationally Determined Parameter (NDP)
- \( n_s \) = number of storeys
- \( E_{cd} \) = the design value of the modulus of elasticity of the concrete
- \( I_c \) = the second moment of area of the uncracked bracing members
- \( L \) = the total height of the building

However, the above approach has a number of restrictions in its application and as a result it is not applied in Tekla Structural Designer.

The second specific approach is given in **Annex H**.

The method given in Annex H.1.2 is the background for the more limited method given in Clause 5.8.3.3 as described above, but it does not apply where there is significant shear deformation in the LLRS e.g. for shear walls with significant openings, hence again it is not considered in Tekla Structural Designer.
Instead, recourse is made to determining the level of second-order effect using Annex H.2. Using this approach, by rearranging Equation H.8 it is possible to provide a 'stability coefficient' $1/\alpha_{cr}$ which can be applied as the change point between non-sway and sway sensitive structures.

$$F_{HEd} = \frac{F_{H0Ed}}{1-F_{H1Ed}/F_{H0Ed}}$$  \hspace{1cm} \text{Equation H.8}

Where:

$F_{H1Ed}$ = fictitious horizontal force, giving the same bending moments as vertical load $N_{VEd}$ acting on the deformed structure, with deformation caused by $F_{H0Ed}$ (first order deformation), and calculated with nominal stiffness values according to 5.8.7.2

Considering how this definition of $F_{H1Ed}$ might apply to an imaginary cantilever of height, $h$, we arrive at:

1. The moment due to $F_{H1Ed}$ is the same as that due to the vertical load $N_{VEd}$, so:

   $$F_{H1Ed} \times h = N_{VEd} \times \delta$$

   which can be rearranged to:

   $$F_{H1Ed} = \frac{(N_{VEd} \times \delta)}{h}$$

2. Substituting for $F_{H1Ed}$ in Equation H.8, we have:

   $$F_{HEd} = \frac{F_{H0Ed}}{1 - \frac{(N_{VEd} \times \delta)}{F_{H0Ed} \times h}}$$

3. By defining $k_{amp} = F_{HEd}/F_{H0Ed}$ the above can be rearranged to:

   $$k_{amp} = \frac{1}{1 - \frac{(N_{VEd} \times \delta)}{(F_{H0Ed} \times h)}}$$

4. Now, the EC3 Equation 5.2 for the elastic critical buckling load is:

   $$\alpha_{cr} = \frac{H_{Ed}/V_{Ed} \times h}{\delta_{HEd}}$$

   which, when re-expressed in the terminology used in H.2 becomes:

   $$\alpha_{cr} = \frac{F_{H0Ed}}{N_{VEd} \times h/\delta_{HEd}}$$

   and when further rearranged becomes:
\[
\frac{1}{\alpha_{cr}} = \frac{(N_{VEd} * \delta_{HEd})}{(F_{HOEd} * h)}
\]

5. Hence \(\frac{1}{\alpha_{cr}}\) can be substituted into the above equation for \(k_{amp}\) so that we arrive at the more well-known formula for amplification:

\[
k_{amp} = \frac{1}{1 - \frac{1}{\alpha_{cr}}}
\]

Strictly, the watershed for concrete structures should be at a \(k_{amp}\) factor of 1.1 (amplification of no more than 10% due to second-order effects). Setting \(k_{amp}\) to be 1.1 and rearranging gives \(\alpha_{cr} \geq 11\) i.e. a stability coefficient \(\leq 0.0909\) not 0.1

It is important to note that the resulting values of \(\alpha_{cr}\) and \(k_{amp}\) are very dependent upon the analysis properties that are used and the you therefore need to carefully consider the modification factors you choose to apply via the Analysis Options.

**How do I assess the worst elastic critical load factor for the building?**

To determine the sway sensitivity for the building as a whole, the worst stack (storey) in the worst column throughout the building in both directions has to be identified - this can be done as follows:

1. On completion of the analysis, open a Review View and select Tabular Data from the Review toolbar.

2. Select ‘Sway’ from the View Type drop list on the Review toolbar.

3. The elastic critical load factor in both directions (\(\alpha_{dir1}\) & \(\alpha_{dir2}\)) is tabulated for each column in the building.

4. Make a note of the smallest elastic critical load factor from all of the columns in either direction - this is the \(\alpha_{cr}\)

If there are a lot of columns in the building - in order to quickly determine the smallest elastic critical load factor in each direction, simply click on the \(\alpha_{dir1}\) header until the columns are arranged in increasing order of \(\alpha_{dir1}\), then repeat for \(\alpha_{dir2}\).

\(\alpha_{cr}\)

**What are the twist results?**

A ‘measure’ of twist is also tabulated for each column - this indicates the degree to which if you push the column one way, how much it moves orthogonally as well. If you
Engineers Handbooks (EC)

have a building where the 'lateral load resisting system' is not well dispersed then pushing one way can cause significant movement in the other direction.

The twist is reported as a ratio of: distance moved in the direction of loading/absolute distance moved.

| **EG** | When a column node moves in X and Y then the 'total' deflection is SQRT(\(\delta_x^2 + \delta_y^2\)) in other words the diagonal of the triangle and not either of the sides. So if a node moves say 10mm in X and 2mm in Y, its diagonal i.e. absolute deflection in this plane is SQRT(100 + 4) = 10.198. Hence its twist is what it should have been with just X loading i.e. 10mm divided into what it actually moved i.e. 10.198. So Twist = 1.0198. |

**Modification Factors**

You specify the modification factors to be applied for each of the different materials from the Modification Factors page of the Analysis Options dialog. (which is located on the Analyse toolbar).

These factors also vary according to the member types, (and in the case of concrete members whether they are cracked or not).

For concrete members in particular, design codes can require that analysis stiffness adjustment factors are applied since the appropriate properties to use in analysis are load and time dependent.

For various other investigations it is also possible that you will want to apply an adjustment to material properties. One suggested example is the assessment of structures subject to corrosion. Another classic requirement in this regard relates to torsion, it is common engineering practice to assume that if it will work without assuming any torsional strength, then torsion can be ignored.

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.

**Member second-order effects**

These are dealt with as part of the design of members.

For steel this is incorporated in the design routines for all members (beams, columns, braces).

Similarly for concrete, much of the calculation is carried out as part of the design. However, in order to assess the 'effective length' of the member (columns and walls) the
incoming members at the top and bottom of the column stack or wall panel are identified and their properties established.

**Global imperfections**

These are typically represented by the application of Equivalent Horizontal Forces.

The formula to calculate the global imperfections (using EHF) is the same for both steel and concrete and is applied in the same manner. It does require some user intervention to provide building height and number of columns to consider. These user inputs cause adjustment of the base value of imperfection of 0.5% of the vertical load and this can be a different adjustment in the two orthogonal directions (Direction 1 and Direction 2). For example, the adjustment factor might give an EHF of 0.4% in the X-direction and 0.3% in the Y-direction.

In all cases it is the adjusted value (0.4% and 0.3% in the example above) that is submitted for analysis.

**Member imperfections**

These are dealt with as part of the design of members.

For steel structures designed to Eurocodes, member imperfections are intrinsically included in the design routines for all members (beams, columns, braces). Apart from one explicit requirement, carrying out the design is all that is necessary.

The explicit requirement is from Clause 5.3.2 (6) in which member imperfection should be included as part of the analysis when the frame is sway sensitive and the axial force in members with moment connections is above a certain limit. If this situation arises *Tekla Structural Designer* issues a warning.

For concrete structures designed to Eurocodes, the explicit calculations are carried out as part of the design.

**Wind drift**

Wind Drift checks can be specified for columns of all materials - they can be applied either to all stacks, or selected stacks only (by checking the appropriate boxes in the column properties).

For those stacks to which the check has been applied, the lateral drift in each direction (i.e. the difference between top and bottom deflection of the stack) is determined for each wind load case and wind service combination. This drift is then compared against a user-defined limit (the default is 1/300 of the storey height, in line with Eurocode 3.
recommendations, but you are free to specify a limit of your choice). Different limits can be applied to different stacks if required.

The checks are performed by running any of the **Design (Gravity)**, or **Design (Static)** commands from the Design ribbon.

Any stack failures are flagged in the **Design - Wind Drift** branch of the Status Tree in the Project Workspace.

- Double clicking on a failing column in this list causes it to be highlighted in the current view.
- Double clicking on the Wind Drift heading at the top of the list causes all failing columns to be highlighted.

Full details for all columns checked are presented in a table accessed from the Review ribbon.

These details can be included in printed output by adding the **Analysis>Wind Drift** chapter to your model report.

The shape of the column wind drift displacements can be viewed graphically in the Results View using the Sway Drift and Storey Shear Ribbon group “X” and “Y” (direction) buttons when a Wind Load case is selected.

### Overall displacement

By expanding the Design branch of the Project Workspace Status Tree, you are able to review the maximum positive and negative overall displacement results from the 3D Building Analysis for both Strength and Service combinations.

Displacements can also be viewed graphically in the Results View by using the buttons on the Deflections group.
This handbook considers the solver models created during analysis and describes specific properties and modelling techniques related to them.

**Solver models**

If you have performed more than one analysis type on the structure, then (providing the geometry and loading have not changed between runs), each solver model and set of results is retained. You can show results for each analysis type without having to re-analyse.

Changes to mesh size or uniformity do not constitute a change in the geometry. Hence, if different meshes have been applied for each analysis, you can review the different solver models by opening a **Solver View** and then choosing the model required from the right click menu.

**Working Solver Model**

The **Working Solver Model** shows the model in its form prior to any analysis.

Although 1D elements and solver nodes are displayed, 2D elements are not (even when you have chosen to mesh slabs/walls). This is because 2D elements are only formed at the point of analysis.

**Solver Model used for 1st Order Linear**

This solver model is in the form of a **3D Building Analysis model**.

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order linear analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.
3D Building Analysis model

The 3D building analysis model consists a mix of 1D analysis elements and FE meshes as follows:

- beams and columns are modelled as 1D analysis elements
- walls are either ‘mid-pier’ analysis elements, or FE meshes
- slabs (optionally) form rigid diaphragms in floors
- 1-way slabs have their loads decomposed on to supporting members at a preliminary stage of the analysis.

Results are still displayed for the ‘old’ solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.
• 2-way slabs are (typically) not meshed, in which case they will also have their loads decomposed on to supporting members at a preliminary stage of the analysis - see: [Load decomposition](#).

• 2-way slabs (optionally) can be meshed
  - Recommended for special cases, typically where slabs participate in the lateral load stability system, e.g. transfer slabs
  - Supports are user defined

2-way slabs meshed

Optionally you can choose to mesh all 2-way slabs – making a fully meshed model (both walls and floors) possible.

This is generally not recommended as it will inevitably increase the model size, (and potentially the time to solve for large models), although it might be considered that a fully meshed model behaves more ‘correctly’ where slabs are considered to be part of the lateral load resisting system of the structure.

It is more likely that you will choose to mesh specific floor levels only (e.g. transfer levels), keeping other levels unmeshed.

 Solver Model used for 1st Order Non Linear

This solver model is in the form of a 3D Building Analysis model.

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order non-linear analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

---

Results are still displayed for the ‘old’ solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

---

 Solver Model used for 2nd Order Linear

This solver model is in the form of a 3D Building Analysis model.

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 2nd order linear analysis.
If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

Results are still displayed for the ‘old’ solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

**Solver Model used for 2nd Order Non Linear**

This solver model is in the form of a 3D Building Analysis model.

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 2nd order non-linear analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

Results are still displayed for the ‘old’ solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

**Solver Model used for 1st Order Vibration**

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 1st order vibration analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.
Results are still displayed for the ‘old’ solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 2nd Order Buckling

Any FE meshes in this solver model are formed using the mesh parameters in place for the most recent run of 2nd order buckling analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

Results are still displayed for the ‘old’ solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for Grillage Chasedown

In grillage chasedown a 3D sub model is formed for each floor and the columns connected to it.
The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

Two-way slabs are only meshed in grillage sub-models at those levels where they have been set as meshed for 3D Building Analysis. For all other slabs, load decomposition is carried out prior to the analysis.

**Solver Model used for FE Chasedown**
FE chasedown is similar to grillage chasedown, with 3D sub models being formed at each level; the one difference being that in the FE chasedown the two-way slabs are always meshed.

**Solver Model used for Load Decomposition**

At each level, (provided you have not checked the option to Mesh 2-way Slabs in 3D Analysis), a solver model is created solely for the purpose of decomposing slab and panel loads back on to the supporting members. As these load decomposition models are only used during the pre-analysis stage, by default they are not retained.
However, if you want to examine the load decomposition model used at a given level this is possible by editing the level properties prior to analysis and checking the 'keep solver model' property.

Solver Model used for Load Decomposition

**Refresh Solver Model**

The solver model currently displayed is updated to reflect changes that have occurred in the working solver model since the last analysis. The 2D element mesh is also updated to reflect the current mesh parameters. Previous analysis results are also erased for this solver model.

**Analysis objects**

A separate solver model is created for each analysis type performed, each solver model consisting of analysis objects with properties that reflect those specified in the physical model.

<table>
<thead>
<tr>
<th>Object</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver element</td>
<td>A 1D analysis object created between two solver nodes.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Releases</strong> are applied at the end of a solver element where these have</td>
</tr>
<tr>
<td></td>
<td>been specified in the physical model</td>
</tr>
<tr>
<td></td>
<td>• <strong>Rigid offsets</strong> are applied at the ends of solver elements where</td>
</tr>
<tr>
<td></td>
<td>required in order to make connections to other solver elements.</td>
</tr>
<tr>
<td></td>
<td>• <strong>Rigid zones</strong> are optionally applied at the ends of solver elements</td>
</tr>
<tr>
<td></td>
<td>to more accurately model the zone where two concrete members connect.</td>
</tr>
</tbody>
</table>

⚠️ Two solver elements are created from each physical member (column stack, or beam span) so that a solver
node exists at mid span/stack for the p-delta analysis.

<table>
<thead>
<tr>
<th>Solver element 2D</th>
<th>Meshes of 2D finite elements are created in the solver models where they have been specified for concrete walls and 2 way spanning slabs.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver node</td>
<td>Solver nodes are created at defined points in a solver model on the basis of solver element and finite element connectivity.</td>
</tr>
<tr>
<td></td>
<td>Solver nodes are created at:</td>
</tr>
<tr>
<td></td>
<td>• The ends and solver elements</td>
</tr>
<tr>
<td></td>
<td>• The corners of finite elements</td>
</tr>
<tr>
<td>Rigid diaphragm</td>
<td>By default these are automatically created in one way and two way slabs - in this type of diaphragm all the solver nodes in the plane of the diaphragm are constrained to move together in the plane of the diaphragm.</td>
</tr>
<tr>
<td>Semi-rigid diaphragm</td>
<td>Optionally created from in one way and two way slabs - in this type of diaphragm all the solver nodes in the plane of the diaphragm are seed nodes of a quadrilateral/triangular finite element mesh.</td>
</tr>
</tbody>
</table>

**Solver models created for concrete members**

**Concrete column physical and solver models**

**Concrete column physical model**

A single concrete column can be created over several storey 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 storey) does not have any significance for analysis or design purposes, but is ultimately important for detailing purposes.

The physical location of the column is determined from the alignment snap points and offsets specified in the column properties, and the insertion point(s) picked.

For concrete columns the alignment snap points and offsets are structurally significant as they will also have an effect on the Concrete column solver elements.

**Concrete column solver elements**

The solver elements for each column stack are always located at the stack centroid - thus they do not necessarily coincide with the insertion line used to position the column originally. If the centroid position shifts from one stack to the next a 'rigid offset' is created automatically to connect the solver elements. Similar rigid offsets are also created as required to connect incoming beams into the column centroids.
To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content check ‘1D Elements> Geometry & RigidOffset’ and ‘Solver Nodes> Geometry’.

As a consequence of this method of modeling, you are freer to simplify the grid layout in order to create the structure more effectively, and then employ column offsets to position each column exactly, knowing that during analysis the program automatically assumes the column is located at its centroid as shown in the plan view.

**Concrete beam physical and solver models**

**Concrete beam physical model**

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 determined from the alignment snap points and offsets specified in the beam properties, and the insertion points picked.
For concrete beams:
- The minor snap points and offsets are structurally significant and have an effect on the Concrete beam solver elements.
- The major snap points and offsets are not structurally significant.

Concrete beam solver elements

In the minor direction beam solver elements are always located at the centre of each beam section - as beam widths or minor offsets may vary, this may result in the introduction of lateral ‘rigid offsets’ to make the connection between spans.

In the major direction beam solver elements are created at the same level as the insertion line used to position the beam.
Rigid offsets

Rigid offsets are automatically applied to the start and end of solver elements as required to ensure that the analysis model is properly connected.

This will occur whenever Concrete beam solver elements or Concrete column solver elements are not co-linear. A typical example of this occurs when concrete edge beams are aligned to be flush with the face of the supporting columns, as shown below:

![Diagram of rigid offsets in a concrete structure]

To see rigid offsets: open a Solver View, and then in Scene Content ensure that ‘1D Elements> RigidOffsets’ is checked.

Rigid zones

Design codes allow engineers to assume parts of concrete beams / columns are rigid, leading to more efficient designs.

Tekla Structural Designer uses Rigid Zones to cater for this where columns and beams are connected and where beams are connected to other beams.
Columns can have rigid zones when they are the supporting or supported member, but beams will only have rigid zones when they are the supported member.

**Application of Rigid Zones**

In most situations in order to get an efficient design you would want rigid zones to be applied - however a **Rigid zones not applied** option is also provided in Model Settings for cases where you don’t want to use them.

- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the position of releases in analysis model is affected, and member start and end points for design are also adjusted.

When applied, rigid zones are created at concrete column/beam connections. The proportion of the zone which is modelled as rigid (the thick black line shown below) is specified as a percentage, the remaining portion of the rigid zone (the red line inside the rigid zone) remains elastic. The proportion of the rigid zone that is rigid can vary between 0 - 100%

![Diagram of rigid zone application](image)

As shown above, the elastic portion of the rigid zone is aligned with the central span solver element.

There is a significant difference between Rigid Zones Not Applied and Rigid Zones Applied with 0% rigidity. The total elastic length of a member is the same in the two models, but the position of releases and start/end of design members will be different.

Rigid zones should not be confused with rigid offsets which are used to ensure that the analysis model is properly connected, i.e. it is possible to have rigid offsets in the model even if rigid zones are turned off.

**Rigid Zones Example 1 - fixed ended beam**

Consider the following 2 span beam supported on columns and loaded with a udl:
Rigid Zones Not Applied

The analysis model is simply constructed from the solver elements with rigid offsets applied as necessary to connect the beam solver elements to the column solver elements.
The resulting beam bending moment diagram is as follows:

![Beam Bending Moment Diagram](image)

**Rigid Zones Applied (default 50%)**

Rigid Zones are activated via Model Settings, and this is also where the percentage of rigidity of the zones can be specified. Initially for this example it will be left at the default of 50%.

The revised solver model is as shown below, note the rigid zones that have been formed where the columns and beams connect:

![Revised Solver Model](image)

The beam bending moment diagram for the revised model is as shown below.
The above diagram was generated from a Results View to illustrate that a ‘gap’ is formed in the diagram where there are rigid zones. It should be noted that when the same result is displayed in a Load Analysis View the gap is removed, leaving only the non-rigid length of the member displayed.

We might expect the extra stiffness introduced at the supports to increase the hogging moments and reduce the sagging moments, however because the element end forces are now reported at the rigid zone boundaries (rather than the ends of the solver elements) - in this example the main effect is that the hogging moment over the central column support is substantially reduced.

**Rigid Zones Applied (100%)**

To investigate the effect of the percentage rigidity an additional run is made with the percentage rigidity increased to 100%. The bending moment diagram that results is shown below:

As expected the extra stiffness introduced at the supports increases the hogging moments and reduce the sagging moments in comparison to the run at 50%.

**Rigid Zones Applied (0%)**

If the percentage rigidity is reduced to 0% the bending moment is as shown below:
If this result is compared to the run in which rigid zones were not applied, it is clear that although the sagging moments are identical, the hogging moments that are reported are not the same. This is because, although the total elastic length of a member is the same in the two models, the position of the start and end of design members is different (being taken at the rigid zone boundaries when applied).

**Rigid Zones Example 2 - pin ended beam**

When rigid zones are applied to a pin ended member, the end release is shifted from the end of the solver element to the rigid zone boundary.

To illustrate this effect the model illustrated in [Rigid Zones Example 1 - fixed ended beam](#) is modified to have pinned connections introduced at the two remote ends of the beam.

**Rigid Zones Not Applied**

The analysis model is constructed from the solver elements with rigid offsets applied to connect the beam and column solver elements. Releases are formed at the two remote ends of the beam solver elements.
The beam bending moment diagram is as follows:

Rigid Zones Applied (0%)

For comparison, rigid zones are then introduced, (with 0% rigidity in order to keep the total elastic length of the beams the same in both models).

Because the releases are moved to the rigid zone boundaries, this has the effect of reducing the moments in the beams.
Solver models created for steel and other materials

Steel column physical and solver models

Steel column physical model

A single steel column can be created over several storey 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

![Diagram showing major snap and offset]

Minor snap and offset

![Diagram showing minor snap and offset]

The alignment snap points and offsets have no effect on the Steel column solver elements.

Steel column solver elements
Each steel column solver element is created between its insertion points. Its position **NOT** being affected by major and minor snap points or offsets.

**Column solver model example**

Consider the two stack column shown in the Structure View below. The same alignment (major snap: bottom, minor snap: left) has been applied to both stacks, but the section size reduces. Consequently, although the insertion lines for the two stacks are co-linear, the centroids of the two sections **are not** co-linear.

In the Solver View it can be seen that the solver elements for each stack **are** co-linear, (coinciding with the insertion lines).
This is different to the approach adopted for a Concrete column solver elements (in which the alignment snap points and offsets are structurally significant).

Steel beam physical and solver models

Steel beam physical model

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.

The alignment snap points and offsets and the construction level type have no effect on the Steel beam solver elements.

**Steel beam solver elements**

Each steel beam solver element is created between its insertion points. Its position **NOT** being affected by:

- major and minor snap points and offsets
- the level 'type' specified in the construction level dialog
- the section size
This is a different approach to that adopted for a Concrete beam solver elements.

To see solver elements and solver nodes: open a Solver View, and then in Scene Content check ‘1D Elements> Geometry’ and ‘Solver Nodes> Geometry’.

Steel brace physical and solver models

Steel braces are single span members with pinned end connections, that are only able to resist axial compression and tension.

Element loads can not be applied directly to the brace itself and moments due to self weight loading are ignored.

Steel brace physical model
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 Steel brace solver elements.

**Steel brace solver elements**

Each steel brace solver element is created between its insertion points. Its position NOT being affected by:

- major and minor snap points and offsets
- the section size

**Inactive steel braces**

Individual steel braces can be made inactive in the solver model by unchecking the ‘Active’ box 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.

**Input method for A and V Braces**

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

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

**Steel parapet post physical and solver models**

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.

 Solver models created for concrete walls

Concrete wall physical model

Concrete walls can be created over several storey 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 storey) does not have any significance for analysis or design purposes, but is ultimately important for detailing purposes.

The physical location of the wall is a determined from the alignment and offsets specified in the wall properties, and the insertion points picked.

For walls the alignment and offsets are not structurally significant as they will have no effect on the solver model.
Concrete wall solver model

The types of solver element created will depend on whether a meshed or mid-pier model is selected.

For both model types, horizontal 1D ‘wall beam’ solver elements are introduced mainly to collect slab mesh nodes and line elements. For meshed walls this allows the wall meshing to be independent of slab meshing.

For meshed walls the type of 2D solver element used will depend on whether the wall mesh type is set to quad dominant, quad only, or tri only.

1D and 2D solver elements for each wall panel are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified.

The different varieties of concrete wall solver model are illustrated in the formed for a 2 storey wall with a thicker panel at the lower level.

Midpier concrete wall solver model

2 storey midpier wall with a thicker panel at the lower level:
To see solver elements, solver nodes and 2D elements: open a Solver View, and then in Scene Content check ‘1D Elements> Geometry’, ‘2D Elements> Geometry’ and ‘Solver Nodes> Geometry’.

Meshed concrete wall solver model (quad only)

2 storey quad only meshed wall with a thicker panel at the lower level:
Meshed concrete wall solver model (tri only)

2 storey tri only meshed wall with a thicker panel at the lower level:
**Meshed concrete wall solver model (quad dominant)**

2 storey quad dominant meshed wall with a thicker panel at the lower level:
Concrete wall openings and extensions

Concrete wall openings

Limitations of wall openings

1. If you have specified a door or window opening in a wall panel you must model the wall using FE elements, otherwise a ‘Walls with openings have a mid-pier’ validation error is displayed and the analysis will not proceed.

2. Assuming the wall has been modelled using FE elements, the analysis will still not proceed if you have applied a wind wall panel over the top of the wall. In this situation a ‘Panel is not surrounded by load carrying members’ validation error is displayed. This error can only be cleared by deleting the openings from the affected walls.

3. Given that the analysis has been able to complete; a ‘Panel contains openings - these are ignored in design’ warning will always be issued when a wall containing openings is designed. When you encounter this warning, as well as taking stock of the design implications; you need also to consider if the analysis model is...
appropriate, as potentially it may not reflect your original intention. In certain situations the Alternative model for wall openings may prove to be a better solution.

Analysis model applied to meshed wall panels with openings

If an opening is introduced in a meshed wall, the properties of the ‘lintel’ wall beam directly above the opening are automatically adjusted in order to prevent the panel being unrealistically stiff. The adjustments that are applied are as follows:

- wall beam properties in the lintel use the lintel depth ($h_2$), rather than the panel depth ($h_1$)
- wall beam nodes in the lintel are removed from the slab diaphragm

Modeling in this way the lintel becomes less stiff resulting in improved wall results when subject to gravity and lateral loading.

Alternative model for wall openings

If the presence of an opening would form a beam like strip above or below the opening, you are advised to create separate wall panels to each side of the opening and then model the strip between the panels with a connecting beam (‘coupling beam’).

This method can be demonstrated by considering the below example, consisting of a two storey wall with a large opening at each level.

If the openings were to be created as a window and door the resulting model would be as shown:
However, by separating the wall into discrete panels and inserting coupling beams you obtain an alternative model as below:
Such an idealisation enables the panels either side of the openings to be designed for their respective forces and enables the strips between the openings to be designed as beams.

Of course, this approach will require some additional detailing, but that would have been the case anyway had the openings been added and subsequently ignored by the design.

**Concrete wall extensions**

**Use of concrete wall extensions**

Wall extensions (End 1/End 2) are applied in order to remove physical overlaps with adjoining walls and columns without compromising the integrity of the underlying analysis model.

Negative extensions can be created automatically where appropriate. Extensions can also be defined manually if required, in which case they can be input with either positive or negative values:

- A positive extension extends the wall length beyond its insertion point.
- A negative extension trims the wall back from the insertion point.

The example below shows the effect of a positive extension at end 1 and a negative extension at end 2.
Although the length of the wall used in the analysis model ($L_{wall}$) is unchanged, the wall length that is used in the design, quantity reporting and drawings changes to $L_{wall,d}$.

**Concrete wall extension examples**

The view below illustrates some examples where wall extensions can be applied.
1. Where wall end does not match architectural grid - not created automatically.

   Although this case could be catered for by using construction lines, it is both quicker to create and easier to edit by manually applying wall extensions.

2. Where wall end overlaps a column - a negative extension can be applied automatically.

3. Where two wall ends meet - a negative extension can be applied automatically.

4. Where a wall end meets another wall part way along its length - a negative extension can be applied automatically.

**Wall and column overlap**

Consider case 2 above, where the wall ends overlap the columns.
If negative wall extensions are not automatically applied you will see an overlap of the wall with the columns (as shown on the left). Potentially you could attempt to ‘fix’ this by creating extra construction lines or grids on the faces of the columns and then reinsert the wall between the faces. Although this looks better, the analysis model shown below is poor as the wall panels are dis-continuous and poorly connected to the columns.

If negative wall extensions are employed instead, the analysis model is much better.
Wall overlaps with another wall

Now consider cases 3 and 4 in the case study, where two walls overlap.

If negative extensions are not automatically applied you will see the overlap of walls (as shown above left). The main problem with this is that from a design point of view the concrete overlaps would result in a duplication of reinforcement in the overlapped areas.
Potentially you could attempt to ‘fix’ this by creating extra construction line or grid on the right hand face of the vertical wall and then reinsert the horizontal walls to this new line (as shown above right). Although this looks better, the analysis model shown below is very poor. The wall panels are completely disconnected from each other, this model will not resist lateral load in anything like the same way.

However by once again employing negative wall extensions, the overlaps are removed from the design whilst still retaining the correct analysis model.

**Solver models created for bearing walls**
Bearing walls are intended for modelling walls that take only vertical compressive load downwards, but which make no contribution to lateral stiffness - for example unreinforced masonry walls.

**Bearing wall physical model**

Bearing walls can be created over several storey heights. A single wall is created with a uniform thickness between the base and top level.

The physical location of the wall is determined from the alignment specified in the bearing wall properties, and the insertion points picked.

> For bearing walls the alignment is not structurally significant, as it has no effect on the solver model.

**Bearing wall solver model**

Bearing walls are modelled using a series of vertical 'wall column' and horizontal 'wall beam' solver elements, as indicated in the figure below. The beams have pinned ends and are placed at the top of the wall spanning between the columns. The next panel above is pinned to the one below and similarly the lower end of a column is pinned to a supporting beam. At the lowest level the column is 'fixed' to a pinned support.
Members supported by the wall either (fortuitously) bear directly on one of the wall columns or on one of the wall beams at the head of the wall. All wall columns and wall beams in an individual bearing wall are given properties automatically by Tekla Structural Designer, based on the width of the bearing wall with which they are associated.

For bearing walls that are defined between other vertical column members e.g. steel columns, the wall columns at the edge of the panel are omitted and the associated wall beam is connected to the steel column (for example) and the adjacent wall column - see figure below.

Wall columns at the edge of the panel are also omitted when it is defined between concrete walls.

Irrespective of whether the wall spans between other vertical column members or not - any load applied to the wall beam at the edge of the panel is shared between the end
column and the first internal column. This can result in some load being 'lost' directly into the supports.

Load transfer in the bearing wall model is not the same as it would be in for example, a masonry wall. A point load applied at the top of a masonry wall would result in a distributed load on any beam supporting the masonry wall, whereas in a bearing wall the supporting beam would be subjected to a pair of point loads, (or possibly even a single point load if the applied load coincides exactly with a wall column location).

Self weight of the bearing wall is concentrated in the wall beams so seismic weight is concentrated at the top of the wall and not split between the floor above and below.

**Solver models created for slabs**

**Vertical alignment of slabs**

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

Individual slab items within a slab adopt the same thickness as the parent slab unless a slab depth override is applied. This override can be applied to all slab types apart from precast.

When an override has been applied you can also apply a vertical offset in order to model a slab step.

**Effect of vertical offsets and changes in slab thickness on solver models**

Vertical offsets and changes in slab thickness are not structurally significant in so far as they have no effect on the vertical alignment of the analysis mesh, or diaphragm constraint relative to the top of the slab, (the actual mesh properties would however reflect changes in slab thickness).

Consider the following flat slab example:
Vertical offsets have been applied to lower some slab panels
• Some slabs have been thickened
• A drop panel has been applied to thicken the slab around a column.

The resulting FE chasedown solver model is formed from a mesh of 2D elements that are all in the same plane. However the properties of the 2D elements take account of the different flat slab thicknesses that exist.

Beam solver elements and slab meshes can only be offset vertically from one another by being defined in different construction levels.

**Effect of slab openings on solver models**

Openings can be created within individual slab items of all types.

For two-way spanning slabs:
• Any loads placed inside openings are not applied to the panel
• FE meshing takes account of openings

For one-way spanning slabs:
• Openings are ignored
**Slab on beams solver models**

Slab on beams can adopt one-way or two-way decomposition. Their diaphragm constraint can be set as rigid, semi-rigid, or none.

The solver models created depend on the above settings and also on whether the option to ‘Mesh 2-way slabs in 3D Analysis’ has been selected - see: [Summary of diaphragm constraint and mesh type configurations](#).

**Flat slab solver models**

Flat slabs always adopt two-way decomposition. Their diaphragm constraint can be set as rigid, semi-rigid, or none.

The solver models created depend on the above settings and also on whether the option to ‘Mesh 2-way slabs in 3D Analysis’ has been selected - see: [Summary of diaphragm constraint and mesh type configurations](#).

**Precast solver models**

Precast slabs always adopt one-way decomposition. Their diaphragm constraint can be set as rigid, semi-rigid, or none.

The solver models created depend on the above diaphragm constraint setting - see: [Summary of diaphragm constraint and mesh type configurations](#).

**Steel deck solver models**

Steel decks can adopt one-way or two-way decomposition. Their diaphragm constraint can be set as rigid, semi-rigid, or none.

The solver models created depend on the above settings and also on whether the option to ‘Mesh 2-way slabs in 3D Analysis’ has been selected - see: [Summary of diaphragm constraint and mesh type configurations](#).

**Timber deck solver models**

Timber decks always adopt one-way decomposition. Their diaphragm constraint can be set as rigid, semi-rigid, or none.

The solver models created depend on the above diaphragm constraint setting - see: [Summary of diaphragm constraint and mesh type configurations](#).

**Composite slab solver models**

Composite slabs always adopt one-way decomposition. Their diaphragm constraint can be set as rigid, semi-rigid, or none.
The solver models created depend on the above diaphragm constraint setting - see: Summary of diaphragm constraint and mesh type configurations.

Diaphragms and floor meshing

Diaphragm types

In a typical building lateral resistance is provided at a few discrete points and it is assumed that applied lateral loads will be distributed to the lateral load resisting systems via floor rigid diaphragm action, (or semi-rigid diaphragm action if a more flexible distribution is required).

For slabs the diaphragm type is controlled by the Diaphragm option slab property, which can be set as:

- Rigid
- Semi-rigid
- None

For roof panels the diaphragm type is controlled by the Include in diaphragm roof property, which when checked creates a semi-rigid diaphragm.

Rigid

In Tekla Structural Designer rigid diaphragm action is achieved using nodal constraints in the translational X and Y directions and about Z torsionally.

A nodal constraint maintains exact relative positioning of all nodes that it constrains, i.e. the distance between any two nodes constrained in a rigid diaphragm will never change, therefore no axial load will develop in any member that lies in the plane of the diaphragm between any two constrained nodes.
In the Solver View:

- **within the rigid diaphragm boundary**: solid nodes are constrained; hollow nodes have been manually excluded and are therefore unconstrained.

- **outside the rigid diaphragm boundary**: both solid nodes and hollow nodes are unconstrained.

**Semi-rigid**

Semi-rigid diaphragm action is achieved using a mesh of semi-rigid 2D elements with user defined properties.

The flexibility of the resulting diaphragm and consequently the distribution of the lateral load into the supports is manually controlled by adjusting the slab property **Divide Stiffness By**.
Diaphragm constraint and mesh type configurations

The slab mesh in a particular solver model is created from 2D elements; either shell or semi-rigid elements are used, depending on the following properties:

- Diaphragm option
- Decomposition
- Mesh 2-way Slabs in 3D analysis

**Diaphragm option**

Specified at the Slab level - this property can be set to:

- Rigid
- Semi-rigid
- None

How this affects the 2D element type used in each solver model is also dependant on the method of decomposition selected.

**Decomposition**

Specified at the Slab level - slabs can be set to either ‘Two-Way Spanning’ or ‘One-Way Spanning’, although for certain slab types the value is fixed as below:

- Composite Slab - One-Way only
• Precast Slab - One-Way only
• Slab on Beams - Two-Way or One-Way
• Flat Slab - Two-Way only
• Steel Deck - One-Way or Two-Way
• Timber Deck - One-Way only

One-way spanning slabs are unmeshed in all solver models unless the **Diaphragm option** is set to Semi-rigid, in which case they will be meshed with semi-rigid elements.

> Semi-rigid 1-way spanning slabs cannot be designed.

Two-way spanning slabs always adopt a mesh of shell elements for the FE chasedown and FE Load Decomposition solver models. However for the 3D Building Analysis and Grillage chasedown solver models they are:

• unmeshed - if the **Diaphragm option** is set to None,
• meshed with semi-rigid elements - if the **Diaphragm option** is set to Semi-rigid, or,
• meshed with shell elements - if the **Diaphragm option** is set to Rigid.

> Semi-rigid 2-way spanning slabs can be designed, but only using the results from the FE Chasedown, not the 3D Building Analysis or Grillage Chasedown.

**Mesh 2-way Slabs in 3D Analysis**

Two-way spanning slabs are always meshed with shell elements in FE Chasedown and Load decomposition solver models. When the **Mesh 2-way Slabs in 3D Analysis** property (specified in the **Level**, or **Slope** properties) is checked the same meshing parameters are then extended to the Grillage chasedown and 3D Building Analysis solver models.

**Summary of diaphragm constraint and mesh type configurations**

The configurations of mesh and nodal constraints applied to each solver model resulting from the different permutations of the ‘decomposition’, ‘diaphragm option’, and, ‘mesh 2-way slabs in 3D analysis’ properties are summarised in the table below.

<table>
<thead>
<tr>
<th>Decomposition</th>
<th>Diaphragm Option</th>
<th>Mesh 2-way slabs in 3D Analysis</th>
<th>FE Load Decomposition &amp; FE Chasedown Solver Models</th>
<th>Grillage Chasedown &amp; 3D Building Analysis Solver Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-way</td>
<td>None</td>
<td>Not Applicable</td>
<td>No mesh; no nodal constraints</td>
<td>No mesh; no nodal constraints</td>
</tr>
<tr>
<td>2-way</td>
<td>None</td>
<td>Semi-Rigid</td>
<td>No mesh; Nodal constraints</td>
<td>Semi-Rigid mesh; no nodal constraints</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
<td>------------</td>
<td>----------------------------</td>
<td>--------------------------------------</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Not Applicable</td>
<td>No mesh; Nodal constraints</td>
<td>Semi-Rigid mesh; no nodal constraints</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Yes</td>
<td>Shell Mesh; no nodal constraints</td>
<td>Shell Mesh; no nodal constraints</td>
</tr>
<tr>
<td></td>
<td></td>
<td>No</td>
<td>Shell Mesh; no nodal constraints</td>
<td>No mesh; no nodal constraints</td>
</tr>
<tr>
<td>Semi-Rigid</td>
<td>Yes</td>
<td>Shell mesh; no nodal constraints</td>
<td>Semi-Rigid mesh; no nodal constraints</td>
<td></td>
</tr>
<tr>
<td></td>
<td>No</td>
<td>Shell mesh; no nodal constraints</td>
<td>Semi-Rigid mesh; no nodal constraints</td>
<td></td>
</tr>
<tr>
<td>Rigid</td>
<td>Yes</td>
<td>Shell Mesh; Nodal constraints</td>
<td>Shell Mesh; Nodal constraints</td>
<td></td>
</tr>
<tr>
<td></td>
<td>No</td>
<td>Shell Mesh; Nodal constraints</td>
<td>No Mesh; Nodal constraints</td>
<td></td>
</tr>
</tbody>
</table>

**Other slab properties affecting the solver models**

**Rotation Angle**

Specified at the **Slab item** level, this property is used for the following where appropriate:

- Span direction for 1-way load decomposition
- To determine the 2D element local axes in the solver model
- Bar direction for Slab on Beam and Flat Slabs.

**Include in Diaphragm**

Specified at the **Slab item** level, this property is only active if the **Diaphragm option** is Semi-Rigid or Rigid. It has no effect on the shell mesh for 2-way spanning slabs.

- Semi-Rigid - excluded slab items are not meshed
- Rigid - internal nodes not considered in the nodal constraints

*Where 2 items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.*
Divide stiffness by

Specified at the Slab level - this property is only active if the Diaphragm option for the Slab is Semi-Rigid.

It is applied to the stiffness determined from the material properties and slab thickness in order to adjust semi-rigid diaphragm flexibility.

Mesh parameters

Slab Mesh

In the FE Chasedown and FE Load Decomposition solver models, slabs are meshed using 2D elements. The 2D element shape (triangular or quad) and the degree of mesh uniformity are specified in the Structure Properties. These parameters can be overridden for individual sub-models by setting different values in the respective Sub Model Properties.

Semi-Rigid Mesh

A semi-rigid mesh is created for slabs (both 1-way or 2-way spanning) that have the Diaphragm Option set as Semi-Rigid, and roofs that have the Include in Diaphragm property checked. The meshed elements are included in all solver models, including FE Load Decomposition.

Beam elements are not split by semi-rigid 2D element nodes.

The 2D element shape (triangular or quad) and the degree of mesh uniformity are specified in the Structure Properties. These parameters can be overridden for individual sub-models by setting different values in the respective Sub Model Properties.

Releases

End releases are applied by editing member properties in the physical model. They cannot be edited directly in solver views.

Column Releases

The fixity at the top and bottom of each column stack can be set as:

- **Free end** - only applicable to the top end of top-most stack and the bottom end of the bottom-most stack
- **Fixed** - in both directions (i.e. encastré, all degrees of freedom fixed)
- **Pinned** - in both directions (i.e. a pinned connection is created between the stack above and the stack below)
- **User defined** - (i.e. fixed in one direction but pinned in the other)
For columns of all materials apart from concrete, in addition to the above release options you are also able to apply a **torsional release** at either end by checking the appropriate box.

For gable posts only, an **axial release** can also be applied at the top of the post.

**User Defined**

The User defined option (i.e. pinned in one direction but fixed in the other) is not available in the Properties Window and can only be specified as follows:

1. Right click the column to display the context menu.
2. Choose **Edit**
3. From the Column Property Dialog open the Releases page and choose the stack to edit.
4. Uncheck the My or Mz degree of freedom at the desired end as required.

**Wall Releases**

Walls can be released about the minor axis at the top and bottom of each panel as follows:

- **Fixed** - Encastré, all degrees of freedom fixed.
- **Continuous (incoming members pinned)** - A fully fixed connection is created between the wall panel above and the wall panel below. Incoming members and incoming slabs are pinned to the wall.
- **Pinned** - A pinned connection is created between the wall panel above and the wall panel below.

*The ‘Pinned’ option should be used with caution as it may result in a mechanism during the analysis.*

**Beam Releases**

Releases at the two ends of a beam span can be set as follows:

- **Fully fixed (free end)** - Denotes a cantilever end. It is achieved by checking the ‘Free end’ box.
  (In a single span beam this box can only be checked if the opposite end is fully fixed.)
- **Pin** - Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.
• **Moment** - Major axis moment connection, and pinned about the minor axis.
• **Fully fixed** - Encastré, all degrees of freedom fixed.
• **Continuous** - This setting is automatically applied when a continuous beam is created and effectively creates a non-editable fully fixed connection between the spans of the continuous member. The connection can only be edited by splitting the beam.
• **User defined** - This setting appears if the connection is pinned for major axis bending (My released) but remains fixed for minor axis bending (Mz).

In addition to the above release options you are also able to apply a **torsional release** at either end by checking the appropriate box. Similarly an **axial release** can be applied to beams of all materials apart from concrete.

**User Defined**

The User defined option is not available in the Properties Window and can only be specified as follows:

1. Right click the beam to display the context menu.
2. Choose **Edit**
3. From the Beam Property Dialog open the Releases page.
4. Check the Mz and uncheck the My degree of freedom at the desired end as required.

**Brace Releases**

Braces can only be connected to supports or to the supporting structure via pinned connections. A torsional release can be applied at one end if required. If the brace connects into a beam (e.g. an A brace) an axial end release can be specified at one end to prevent vertical load from the beam being carried by the brace.

An option is provided to include force eccentricity moment.

**Supports**

Columns and walls have supports automatically placed underneath them, unless they are placed directly upon existing elements which provide a means of support (e.g. transfer beams or transfer slabs).

Supports can also be placed manually at other locations.

If the default supports are inappropriate, they can be changed, simply by box selecting the supports that require editing and then adjusting the support degrees of freedom displayed in the Properties Window.
Supports can be edited in both physical model views and solver model views.

**Support degrees of freedom**

Each support has six possible degrees of freedom:
- translational (Fx, Fy, and Fz)
- rotational (Mx, My, and Mz)

Wall supports and manually placed supports default to fully fixed, whereas column supports default to being rotationally free in Mx & My.

The options for a support that is rotationally free in Mx or My are:
- Release
- Spring Linear
- Spring Non-linear
- Nominally pinned
- Nominally fixed

The options for a support that is rotationally free in Mz are:
- Release
- Spring Linear
- Spring Non-linear

The options for a support that is translationally free in Fx, Fy, or Fz are:
- Release
- Spring Linear
- Spring Non-linear

**Non linear spring supports**

For non-linear supports, two spring stiffnesses are required one each for the positive and negative direction of action.

In addition an upper limit should be defined to set a cap on the force or moment that can be supported.

**Compression only ground spring**

A compression only ground spring would be defined translationally in z only as follows:
- Type: Spring Non-Linear
- Stiffness -ve: 0
- Fmax -ve (tension): 0
- Stiffness +e: your choice of ground spring stiffness value
Fmax +ve (compression): your choice of capacity

**Partial fixity of column bases**

Two additional types of rotational linear spring are provided to allow partial fixity to be modelled, these are:

- Nominally pinned
- Nominally fixed

These are specifically provided for supports under columns (of any material), but will result in a validation error if placed under walls (meshed or mid-pier), or if they are used for any other supports.

The support stiffness is based on the column properties \((E*I/L)\)

- \(E\) = Young's Modulus of the column
- \(I\) = relevant bending stiffness \((I_{xx} \text{ or } I_{yy})\) of the column
- \(L\) = distance from the support to the first column point (stack) that is on a Construction Level checked as a Floor in the Levels dialogue, i.e. combined length of all the stacks until a floor is found.

Where no Floor has been defined above a support then \(L\) is taken as total length of column.

Partial fixity spring stiffness is thus calculated as follows for each of the two bending releases \(M_x\) and \(M_y\):

- Nominally pinned (spring stiffness) - \(x\% \times 4 \times E*I/L\) (default \(x\% = 10\%\))
- Nominally fixed (spring stiffness) - \(x\% \times 4 \times E*I/L\) (default \(x\% = 100\%\))
Since the spring stiffness is dependent upon stack height and column stiffness (E and I), the spring stiffness will change if any changes are made to column stack height, column E or I values.

In addition, since for steel, Auto Design can change the column size (and hence I value) the spring stiffness will change with any change in column size.
This handbook describes the automated processes that take place when you perform a static analysis and design of your building.

**Definitions**

Various terms referred to in the analysis-design processes are listed below:

<table>
<thead>
<tr>
<th>Term</th>
<th>Head Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alpha-crit ($\alpha_{cr}$)</td>
<td>Eurocode</td>
<td>The factor by which the design loading would have to be increased to cause instability in a global mode. Alpha-crit is determined from a sway stability analysis and used to determine the sway sensitivity of the structure the Design Options dialog.</td>
</tr>
<tr>
<td>Amplified Forces Method</td>
<td>Eurocode</td>
<td>Using this method, second order sway effects due to vertical loads are calculated by amplifying horizontal loads as per Clause 5.2.2 (5)B.</td>
</tr>
<tr>
<td>Braced</td>
<td>All</td>
<td>The member contributes little or no stiffness to the lateral load resisting system (LLRS).</td>
</tr>
<tr>
<td>Bracing</td>
<td>All</td>
<td>The member is part of the LLRS.</td>
</tr>
<tr>
<td>Drift</td>
<td>All</td>
<td>The absolute horizontal deflection of a column or the relative deflection of two floors within a building when it is usually called 'interstorey drift'.</td>
</tr>
</tbody>
</table>
| **Equivalent Horizontal Forces** | **Eurocode** | **These are used for two purposes:**  
· for the calculation of acr  
· to represent frame imperfections  
EHF’s are calculated as 0.5% of vertical Dead and Imposed loads. These are sometimes referred to as Notional Loads. |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>First-order analysis</strong></td>
<td>All</td>
<td>Linear elastic analysis that takes no account of the effect on the forces due to deformations of the structure.</td>
</tr>
<tr>
<td><strong>Non slender</strong></td>
<td>All</td>
<td>The member length is sufficiently small that flexural (strut) buckling is unlikely to occur, second-order effects (P- δ) are small enough to ignore and, in the limit, the full squash load can be realized.</td>
</tr>
<tr>
<td><strong>Non-sway</strong></td>
<td>All</td>
<td>The global second-order effects (P- Δ) are small enough to be ignored.</td>
</tr>
</tbody>
</table>
| **Second-order (P-Delta) analysis** | Eurocode | Analysis that takes into account the effect on the forces due to deformations of the structure. Either:  
· by using the Amplified Forces Method, or  
· by a rigorous method using a two step iterative approach |
| **Slender**                   | All         | The member length is of such magnitude that member second-order effects (P- δ) must be taken into account and flexural buckling will be the failure mode. |
| **Sway sensitive**            | All         | The global second-order effects are significant and must be taken into account. |

**Summary of Static Analysis-Design Processes**

Members can be pre-sized for gravity combinations by running Design (Gravity), but a design that satisfies the code requirements can only be achieved by running Design (Static).

Irrespective of whether you run Design Steel (Static), Design Concrete (Static), or Design All (Static), the same basic steps are required:
The below analysis-design processes run from start to finish without user intervention - you should therefore ensure ‘Design Options’ are configured correctly before they are initiated.

<table>
<thead>
<tr>
<th>No.</th>
<th>Process</th>
<th>Description</th>
<th>Exclusions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Model validation</td>
<td>Run to detect any design issues which might exist.</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>3D pre-Analysis</td>
<td>A number of pre-processes are undertaken as necessary in preparation for the full analysis:</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Load decomposition</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Global imperfections</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Load reductions</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Pattern loading</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3D Building Analysis</td>
<td>A traditional frame analysis of the entire 3D model, with an option to mesh floors.</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>Grillage Chasedown Analysis</td>
<td>Requirements: Only performed if concrete members exist.</td>
<td>Not performed for:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>For a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Building Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.</td>
<td>• Design Steel (Gravity)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Design Steel (Static)</td>
</tr>
<tr>
<td>5</td>
<td>FE Chasedown Analysis</td>
<td>Requirements: Only performed if two-way slabs exist.</td>
<td>Not performed for:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>For a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and</td>
<td>• Design Steel (Gravity)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Design Steel (Static)</td>
</tr>
</tbody>
</table>
### Model validation

Validation is a check on your structure which must be performed before it can be 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. Two types of validation message can be displayed.

**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.

In the combined analysis-design process, model validation is followed by **Load decomposition**

### 3D pre-Analysis

Pre-analysis consists of a number of steps, the actual number being dependant on the specific model that has been defined.

In the combined analysis-design process, pre-Analysis is followed by **3D Building Analysis**

### Load decomposition

Load decomposition of slab loads on to supporting members is performed where necessary, prior to analysis. It is not restricted to beam and slab models, as it is also useful for decomposing flat slab loads onto columns.

Whether decomposition is performed or not will depend on the analysis model, the slab properties and the mesh setting.

- 1 way slab loads are always decomposed for each analysis model considered.
- 2 way slab loads are decomposed for 3D building analysis and grillage chasedown models unless the option to **Mesh 2-way Slabs in 3D Analysis** has been applied.
- 2 way slab loads are not decomposed for the FE chasedown model
1-way slab load decomposition

The loads on one-way slabs are always decomposed prior to analysis, (irrespective of the analysis model type).

2-way slab load decomposition

In the 3D building analysis and grillage chasedown models, whether the loads on two-way slabs are decomposed or not will depend on the mesh setting:

- By default 2-way slabs are not meshed in the building model, in which case the loads on two-way slabs are always decomposed prior to analysis.
- If the ‘Mesh 2-way slabs in 3D Analysis’ option has been checked, it is then not necessary to decompose the loads on two-way slabs prior to analysis.

FE chasedown is the only analysis type for which 2-way slab load decomposition is never required prior to analysis.

In those cases where two-way slab decomposition is required, a separate decomposition model is formed at each floor level.
A sophisticated FE (rather than yield-line) model is applied, which caters for irregular slabs, openings and any loading.

**Global imperfections**

Equivalent notional horizontal loads are determined and applied to cater for global imperfections (additional second order effects due to the structure not being built plumb and square). These loads are also used in seismic design to establish the base shears.

Following a first-order analysis of all gravity loadcases, the forces at the nodes at the top/bottom of each column stack/wall panel are resolved vertically. A proportion of the vertical load is determined which gives the value of the horizontal load at each point. The proportion is code dependent.

These horizontal loads are applied to the nodes in a particular direction (Direction 1 or Direction 2 or both) as specified in an individual design combination.

**Load reductions**

The imposed load reductions are established for use in subsequent column design, (and when the Head Code is set to ACI/AISC beam design also).

**Pattern loading**
If combinations of pattern load exist then the pattern loading is automatically generated prior to analysis.

**3D Building Analysis**

A traditional frame analysis of the complete structure is always the first analysis performed. This analysis generates a first set of results for the design of beams, columns and walls.

**First Order or Second Order Analysis**

You can control whether a first, or a second order 3D Building Analysis is run by making the appropriate selection on the Analysis page of the Design Options dialog. The actual options that are presented will vary depending on the design code being worked to.

**Linear or Non Linear Analysis**

The choice of linear or non-linear analysis is made automatically:

- if the model has entirely linear properties a linear analysis is performed,
- else if any non-linear properties are detected a non-linear analysis is performed.

In the combined analysis-design process, 3D Building Analysis is followed by **Griﬄage Chasedown Analysis**

**Griﬄage Chasedown Analysis**

We know from experience that 3D building analysis on its own does not give the gravity results engineers have traditionally used or want - staged construction analysis reduces but doesn't resolve this. Therefore, the combined analysis design process will also automatically undertake a griﬄage chasedown analysis. (provided concrete beams exist).

The **Solver Model used for Griﬄage Chasedown** emulates a traditional analysis and establishes an alternative second set of design forces for beams, columns and walls.

It is important to note however that the chasedown analysis procedure is only valid for gravity loads. The chasedown analysis results for any lateral loading case (wind / seismic ELF / EHF) or from the direct analysis of any combination that includes a lateral loading case are not valid.

Therefore in order to generate the design forces mentioned above, the chasedown analysis results are merged with the building analysis combination results as follows:

1. Start with the building analysis combination result
2. Identify all gravity cases used in the combination and the relevant load factor
3. For each included gravity loadcase:
4. For results with Imposed load reduction, subtract the relevant % of the chasedown result for each reducible loadcase.

Following this procedure means that chasedown analysis of lateral loading cases or combinations is not required.

This procedure is only applied to beam, column, and wall-line results, but not to 2D nodal results. For this reason it is not possible to calculate or display 2D element chasedown results for combinations that include lateral load cases.

In the combined analysis-design process, grillage chasedown analysis will either be followed by FE Chasedown Analysis (if it is required), or by an assessment of Sway sensitivity.

**FE Chasedown Analysis**

The Solver Model used for FE Chasedown is generated as part of the combined analysis and design process if the model contains flat slabs, or slabs on beams - the results from this analysis being required for the design of these slabs.

The same results can also be used to generate a third set of design forces for the chosen member types, (provided you have chosen in the Design Options dialog to design the concrete beams, columns, or walls for FE Chasedown results).

It is important to note however that the chasedown analysis procedure is only valid for gravity loads. The chasedown analysis results for any lateral loading case (wind / seismic ELF / EHF) or from the direct analysis of any combination that includes a lateral loading case are not valid.

Therefore in order to generate the design forces mentioned above, the chasedown analysis results are merged with the building analysis combination results as follows:

1. Start with the building analysis combination result
2. Identify all gravity cases used in the combination and the relevant load factor
3. For each included gravity loadcase:
   - Subtract the 1st order linear building analysis result multiplied by the relevant load factor
Add the chasedown result multiplied by the relevant load factor.

4. For results with Imposed load reduction, subtract the relevant % of the chasedown result for each reducible loadcase.

Following this procedure means that chasedown analysis of lateral loading cases or combinations is not required.

- This procedure is only applied to beam, column, and wall-line results, but not to 2D nodal results. For this reason it is not possible to calculate or display 2D element chasedown results for combinations that include lateral load cases.

A significant consideration when opting to design for the FE chasedown results is that the slabs will tend to carry a significant proportion of the load direct to the columns.

Consequently, for beam design in particular, it is unlikely that an FE chasedown could result in a more critical set of design forces than would be already catered for by the Grillage chasedown.

- If duplicate levels have been specified in the Construction Levels dialog separate sub models are created and analysed for the source and every duplicate level. This ensures that the increasing load carried by the vertical members in the lower sub models is catered for. In turn this can cause small differences in the analysis results (and consequently the design) for these sub models.

In the combined analysis-design process, FE chasedown analysis is followed by an assessment of Sway sensitivity.

**Sway sensitivity**

In order to determine whether the structure is 'non-sway' or 'sway sensitive', the elastic critical load factor is calculated.

If the structure is determined as 'non-sway', first-order analysis results can be used for both steel and concrete design.

If it is 'sway sensitive' then (global) second-order effects must be taken into account, either by:

- amplified forces method (uses first-order analysis),
- second-order analysis.

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (regular frameworks with \( \alpha_{cr} > 3 \)). Full second-order analysis is preferred for steelwork and can also be used for concrete.
In the combined analysis-design process, sway sensitivity is followed by Member design.

**Member design**

The final step in the combined analysis-design process is member design for all members for all available sets of design forces.

**Steel Member Design Forces**

The 3D Building Analysis results are the only results set used in steel member design.

**Concrete Member Design Forces**

Up to three sets of analysis results will exist for concrete member design as follows:

- **3D Building Analysis** results will always be used to design all beams, columns and walls.
- **Grillage Chasedown** results will exist for gravity loadcases if the model contains any concrete beams, in which case they will also be used to design all beams, columns and walls.
- **FE Chasedown** results for gravity loadcases will also exist if the model contains 2-way spanning slabs.

Concrete beams can be designed for this set of results by checking the ‘Design Beams for FE Chasedown analysis results’ box under **Design > Design Options > Concrete > Beam > General Parameters**.

Columns and walls can also be designed for this set of results by checking similar boxes on their respective General Parameters pages.

**Reset Autodesign**
On completion of your chosen design process, the original member design mode assigned to each member can either be retained or updated. (For example, you might typically reset auto-designed steel members into check mode if they have a pass status.) The action that is taken is controlled via Design Options > Autodesign.

In the combined analysis-design process, member design is automatically followed by Design Review.

### Design Review

On completion of the combined-analysis and design process the Review View and Review toolbar open automatically.

In this view a colour coded version of the model is displayed so that design status and various other parameters can be graphically interrogated and/or modified.

### Comparison of solver models used in Static Analysis-Design

The following table summarises the three analysis models used in the design process:

<table>
<thead>
<tr>
<th>Examples / When useful?</th>
<th>3D Building Analysis</th>
<th>Grillage Chasedown</th>
<th>FE Chasedown</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• Gravity and Lateral analysis (Notional/Wind/Seismic)</td>
<td>• ‘Beam &amp; Slab’ buildings</td>
<td>• Flat slab and ‘Beam &amp; Slab’ buildings</td>
</tr>
<tr>
<td>Special Features</td>
<td>• Pattern loading</td>
<td>• Mimics traditional design approach (sub-frame analysis)</td>
<td>• Mimics traditional design approach (isolated floor analysis)</td>
</tr>
<tr>
<td></td>
<td>• Automatic EC2 sway sensitivity assessment and sway amplification</td>
<td>• Pattern loading</td>
<td>• Slab Pattern loading</td>
</tr>
<tr>
<td></td>
<td>• Automatically centralised analysis wires (improved rigid offsets / rigid zones)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>• Option to mesh slabs in the 3D analysis</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Benefits</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
<td></td>
</tr>
<tr>
<td>• Member Design considers sway and differential axial deformation effects.</td>
<td>• Member design based on traditional sub frame is considered simultaneously with the 3D Building Analysis</td>
<td>• Member design based on traditional sub frame is considered simultaneously with the 3D Building Analysis</td>
<td></td>
</tr>
<tr>
<td>• Caters for slabs that contribute to the lateral load resisting system</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Analysis Model</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>3D model of entire building:</td>
<td>series of 3D sub models:</td>
<td>series of 3D sub models:</td>
</tr>
<tr>
<td>• either meshed 2-way slabs,</td>
<td>• all column and wall stacks immediately above and below the sub-model</td>
<td>• all column and wall stacks immediately above and below the sub-model</td>
</tr>
<tr>
<td>• or, slab loads decomposed to beams</td>
<td>• either meshed 2-way slabs,</td>
<td>• all 2-way slabs meshed</td>
</tr>
<tr>
<td></td>
<td>• or, slab loads decomposed to beams</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Analysis Method</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Whole model in one pass</td>
<td>Each sub model sequentially from top to bottom – chasing member loads down</td>
<td>Each sub model sequentially from top to bottom – chasing member loads down</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Analysis Type</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>• First order</td>
<td>First order</td>
<td>First order</td>
</tr>
<tr>
<td>• First order - ( K_{ampl} )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• Second order - ( P-D )</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Supports</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>External supports as defined by the user</td>
<td>Ends of members above/below each sub model are automatically supported</td>
<td>Ends of members above/below each sub model are automatically supported</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Loading</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Gravity and Lateral Loads</td>
<td>Gravity Loads only</td>
<td>Gravity Loads only</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Forces for design</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>RC Slab</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Yes– All Combs</td>
<td>No forces</td>
<td>Yes – All Gravity load</td>
</tr>
<tr>
<td></td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
</tr>
<tr>
<td>-------------------------</td>
<td>-----------------</td>
<td>------------------------------</td>
</tr>
<tr>
<td><strong>RC Beam</strong></td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
</tr>
<tr>
<td><strong>RC Column</strong></td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
</tr>
<tr>
<td><strong>RC Wall</strong></td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
</tr>
<tr>
<td><strong>Steel/Composite Members</strong></td>
<td>Yes – All Combs except patterns</td>
<td>Not required</td>
</tr>
<tr>
<td><strong>Foundation design</strong></td>
<td>Yes – All Combs except patterns</td>
<td>Yes – All Gravity load cases</td>
</tr>
</tbody>
</table>
This handbook describes *Tekla Structural Designer*’s seismic analysis and design capability, which is available both for ASCE7 and Eurocode EN1998-1.

**Introduction to Seismic Analysis and Design**

The below topics provide a simplified outline of how seismic analysis and design forces are determined for a building.

**Definitions**

Various terms used in *Tekla Structural Designer*’s seismic processes are described below:

**Code Spectra**

The spectra specified in a country's loading and design codes for use in ELF and RSA analysis.

**Site Specific Spectra**

User defined spectra for ELF and RSA which are required for locations which use another country's loading and design codes where the code spectra are not relevant.

**Base Shear Combination**

Also referred to as the **Effective Seismic Weight Combination** or the **Seismic Inertia Combination**. This combination is used for Vibration Analysis, and in the calculation of base shears, during the Seismic Analysis Process. This combination is created and modified by the Seismic Wizard only.

**RSA Seismic Combination**

These combinations are created by the Combination Generator at the end of the Seismic Wizard, but can also be modified in the standard Combination dialog. They consist of 3 kinds of loadcases: Static, RSA Seismic and RSA Torsion. The Effective Seismic Weight Combination is not included in this category of combination.

**Static Loadcase**
Standard loadcases, e.g. ‘Self weight - excluding slabs’, ‘Dead’, etc., and derived cases for NHF/EHF, but no patterns.

**RSA Seismic Loadcase**

Two loadcases, i.e. ‘Seismic Dir1’ and ‘Seismic Dir2’, which cannot be edited. These are created at the end of the Seismic Wizard, being derived from information supplied in the Seismic Wizard and the results of the Vibration Analysis. No actual loads are available for graphical display.

**RSA Torsion Loadcase**

These cases can be generated by the Seismic Wizard and are regenerated whenever RSA Seismic Combinations are modified.

**Fundamental Period (T)**

Separately for Dir 1 & Dir 2, this is either defined in the Seismic Wizard, (user value or calculated), or determined in the Vibration Analysis for the Seismic Inertia Combination.

**Level Seismic Weight**

For each relevant level, this is the sum of the vertical forces in nodes on that level, for the Seismic Inertia Combination.

**Effective Seismic Weight**

This is the sum of the level seismic weights for all relevant levels for the Seismic Inertia Combination.

**Seismic Base Shear**

The base shear is calculated separately for Dir 1 & Dir 2, for the Seismic Inertia Combination.

**Square root of Summation of Square (SRSS)**

The SRSS formula for combining modes in RSA is as follows:

\[
\lambda = \sqrt{\left( \sum_{k=1}^{n} (\lambda_k^2) \right)}
\]

\(\lambda\) = Absolute value of combined ‘response’
\(\lambda_k\) = ‘response’ value for Relevant Mode \(k\)
\(n\) = Number of Relevant Modes

**Complete Quadratic Combination (CQC)**

The CQC formula for combining modes in RSA is as follows:
\[ \lambda = \sqrt{\sum_{i=1}^{n} \left( \lambda_i \sum_{j=1}^{n} (\rho_{ij} \lambda_j) \right)} \]

\(\lambda\) = Absolute value of combined ‘response’

\(\lambda_i\) = ‘response’ value for Relevant Mode \(i\)

\(\lambda_j\) = ‘response’ value for Relevant Mode \(j\)

\(n\) = Number of Relevant Modes

\(\rho_{ij}\) = Cross modal coefficient for \(i\) & \(j\)

**Cross Modal Coefficient**

This co-efficient is used in the CQC method for combining modes in RSA.

\[ \rho_{ij} = \frac{8\zeta^2 (1 + \beta)\beta^{1.5}}{(1 - \beta^2)^2 + 4\zeta^2 (1 + \beta)^2} \]

\(\zeta\) = modal damping ratio

IBC/ASCE assumed = 5% (ASCE Figs 22-1 to 6)

EC8 assumed = 5% where \(q\) accounts for the damping in various materials being different to 5% (EC8 Cl 3.2.2.5)

IS codes the user can define the level of damping and this is accounted for in the above equation.

\(\beta\) = Frequency ratio = \(\omega_i / \omega_j\)

\(\omega_i\) = Frequency for Relevant Mode \(i\)

\(\omega_j\) = Frequency for Relevant Mode \(j\)

**Overview**

All seismic codes work in a similar manner from the loading viewpoint with relatively minor differences in terminology and methodology.

It is worth noting at the start that seismic analysis determines a set of forces for which it is expected (statistically) that if those forces are designed for and other design precautions taken (additional seismic design) then in the event of an earthquake the structure may well suffer extensive damage but will not collapse and for some categories of building should actually remain serviceable.
In *Tekla Structural Designer* a seismic wizard gathers all the information together to setup the requirements for a seismic analysis.

From this information a number of things are determined:

- The Seismic Inertia combination to determine the seismic base shear in the building
- The natural frequencies of the building in two horizontal directions
- The combination of the gravity and other lateral forces with the seismic load cases

Earthquakes load a building by a random cyclic acceleration and deceleration of the foundations. These are in both horizontal directions (Dir1 and Dir2) but can also be in a vertical direction too. This ground acceleration excites the building in its natural and higher frequencies.

As a result if the building is

- in an area of low seismic acceleration, low in height and poses limited risk to life then a gross approximation can be used in analysis - assuming a % of gravity loading is applied horizontally to the building to represent the earthquake (US codes 1%, Australian codes 10%).
- in an area of moderate to low seismic acceleration, medium to low in height and does not house a significant number of people - the predominant mode excited is the 1st mode of vibration. An equivalent lateral force (ELF) approximation can be used that applies static horizontal loading distributed up the building to mimic the shape of the 1st mode of vibration in a static analysis.
- anything else, in an area of high acceleration, tall in height and could be holding many people or be critical post-earthquake then a ‘more representative’ analysis method of Response Spectrum Analysis (RSA) should be used. This analysis is based upon a vibration analysis considering all mode shapes of vibration in the two horizontal directions in which typically 90% of the structure's mass is mobilised.

The results from the chosen method of seismic analysis are used in combination with other gravity and lateral load cases to design both normal members and those members in seismic force resisting systems (SFRS). These latter members need additional design and detailing rules to ensure they resist the seismic forces that they have to resist during an earthquake.

---

**Seismic Wizard**

In *Tekla Structural Designer* the Seismic Wizard sets up the information required for seismic analysis - the main parameters to be input being:

- Ground acceleration - strength of the earthquake
- The Importance Class of the building - the use to which the building is being put - typically
• I= very minor, farm and temporary buildings,
• II= general buildings occupied by people,
• III = buildings occupied by a large number of people
• IV = critical buildings with a post-disaster function eg hospitals, police stations, fire stations and buildings along access route to them)

• The ground conditions upon which the building is founded (typically Hard Rock, Rock, Shallow soil, Deep Soil, Very Soft Soil)
• Building height - for low buildings the first mode of vibration is totally dominant in taller buildings other modes become significant
• Plan and vertical irregularities in the building

From this input the Seismic Wizard determines the elastic design response spectrum to be used for the building.

Additionally the Wizard sets up the Seismic Inertia combination - the combination of loads likely to be acting on the building when the earthquake strikes.

Related topics
• Eurocode EN1998-1:2004 Seismic Wizard

Vertical and Horizontal Irregularities

There are typically 5 types of horizontal irregularity and 5 types of vertical irregularity - all are defined to pick up structures that have lateral framing systems and shapes in plan that will preclude the structure naturally developing a simple first mode of vibration. Since this is a basic assumption of ELF - the presence of these irregularities may preclude the use of ELF.

Torsion

When a structure's centre of mass at a level does not align with the position of the centre of rigidity then torsion is introduced in the structure at that level when an earthquake excites the structure. To account for this, there are three types of torsion potentially applied to levels with non-flexible diaphragms during a seismic analysis
• Inherent torsion - in a 3D analysis when the centre of mass and centre of rigidity at a level do not align, this is taken account of automatically
• Accidental torsion - to allow for the ‘miss positioning’ of loads in a structure, an additional eccentricity of usually 5% of the structure's width in all relevant directions - this is accounted for with a torsion load case in the analysis
• Amplified accidental torsion - structures with certain SDCs and certain horizontal irregularities require an amplified accidental torsion to allow for extra effects
Amplified accidental torsion is beyond scope in the current release of Tekla Structural Designer.

**Vibration Analysis**

Using the Seismic Inertia combination, a vibration analysis is run for two purposes:

- the natural frequencies of the building in two directions are determined to assist with the calculation of the seismic base shear that in turn is used to determine the distribution of applied loads up the building for an ELF analysis
- the frequencies and mode shapes of the building are determined that need to be included in an RSA analysis so that typically 90% of the mass in the building is mobilised during the RSA analysis

**Equivalent Lateral Force Method**

The ELF method assumes that the first mode shape is the predominant response of the structure to the earthquake.

Based on the natural frequency and the Seismic Inertia combination, a total base shear on the structure is determined and this is then set up as a series of forces up the structure at each level (in the shape of an inverted triangle) and these deflect the structure in an approximation to the shape of the first mode.

The resulting seismic load cases are combined with the correct combination factors with the other gravity and lateral load cases in the **seismic combinations** to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

**Response Spectrum Analysis Method**

The RSA method uses a set of vibration modes that together ensure that the mass participation is typically 90% in the structure in a particular direction.

The response of the structure is the combination of many modes that correspond to the ‘harmonics’. For each mode, a response is read from the design spectrum, based on the modal frequency and the modal mass, and they are then combined to provide an estimate of the total response of the structure.

Combination methods include the following:

- **Square root of Summation of Square (SRSS)**
- **Complete Quadratic Combination (CQC)** - a method that is an improvement on SRSS for closely spaced modes
As a result of the combination methods (SRSS and CQC), the resulting seismic load cases are without sign and so they are applied with the correct combination factors both + and - around the ‘static’ results of the other gravity and lateral load cases in the seismic combinations to give the design forces and moments which are used in both the conventional design of all steel and concrete members, and the seismic design of any steel or concrete members that have been identified as part of a seismic force resisting system.

**Summary of RSA Seismic Analysis Processes**

RSA Seismic Analysis (1st or 2nd order) is run as a stand-alone analysis from the Analyse toolbar, or as part of the Design (RSA) process. In the latter, the use of 1st order or 2nd order is set for the static analysis is set via Design Options > Analysis.

The process consists of the following steps:

<table>
<thead>
<tr>
<th>No.</th>
<th>Process</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Model Validation</td>
<td>Run to detect any design issues which might exist.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This is similar to standard model validation but also checks:</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Seismic Inertia Combination must exist</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• At least one RSA Seismic Combination must exist including at least one RSA Seismic Loadcase.</td>
</tr>
<tr>
<td>2</td>
<td>Vibration Analysis</td>
<td>A 1st Order Vibration analysis for the Seismic Inertia Combination only, which returns the standard results for that analysis type, but also the fundamental periods for directions 1 &amp; 2.</td>
</tr>
<tr>
<td>3</td>
<td>Pre-Analysis for Seismic</td>
<td>Performs calculations for RSA Torsion Loadcases.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>The seismic weight and seismic torque are both calculated at this stage.</td>
</tr>
<tr>
<td>4</td>
<td>Static Analysis</td>
<td>1st Order Linear or 2nd Order Linear analysis is performed for all RSA Seismic Combinations and all their relevant loadcases, i.e. this includes Static Loadcases, but does not include RSA Seismic and RSA Torsion Loadcases.</td>
</tr>
<tr>
<td>5</td>
<td>RSA Analysis</td>
<td>A set of results is generated for a sub-set of vibration modes for each RSA Seismic Loadcase.</td>
</tr>
<tr>
<td>6</td>
<td>Accidental Torsion Analysis</td>
<td>Analysis of any RSA Torsion Loadcases that exist.</td>
</tr>
</tbody>
</table>

**Seismic Drift**
Seismic drift is assessed on a floor to floor horizontal deflection basis and there are limits for acceptability of a structure - if too 'floppy' then the structure may fail the seismic drift check and the RSA will not progress as a result.

The building's overall seismic drift status is displayed in the Design branch of the Status Tree in the Project Workspace.

Full details for all columns are presented in a table accessed from the Review ribbon.

These details can be included in printed output by adding the Analysis>Seismic Drift chapter to your model report.

**Limitations of Seismic Design**

The following limitations apply:

- Where seismic design and detailing is required this is only supported in Tekla Structural Designer for the ACI/AISC Head Code.
- It is up to the user to assess whether framing is split horizontally or vertically, whether system specific requirements need to be assessed - like mixed system moment frames, whether diaphragms are rigid or flexible - in all instances, the user will need to make the necessary adjustments for the situation in hand. The software does not handle these situations automatically.
- Linear vibration with non-linear element properties - currently the vibration analysis is limited to a linear model so all non-linear elements are set to be linear.
- ELF can be run as 1st or 2nd order analyses however vibration and RSA can only be run as 1st order.
- Structures with linear members and supports are run using linear analyses. Structures with non-linear supports and/or members are run as non-linear in ELF but linear in vibration and RSA.
- We do not consider any of the standard methods for structurally accommodating seismic actions - e.g. base isolators, damping systems.
- We do not consider more accurate methods of analysis like time history analysis. As a result there are some situations with very tall buildings and very irregular buildings that Tekla Structural Designer does not cater for.
- Diaphragms - rigid and semi-rigid diaphragms (meshed floors) are available and it is the user's responsibility to ensure they are modelled suitably. Rigid diaphragms are only allowed in limited circumstances and, so called, 'flexible diaphragms' can be modelled as semi-rigid diaphragms with extremely low stiffness. Force transfer into and out of the diaphragm is not checked.
- Collector elements - no checks included.
- Non-structural elements - no checks included.

**Specific limitations of steel seismic design**

- Coincident V & A braces giving X type are beyond scope
• Various other requirements not checked
  • e.g. V & A braces are restrained at their intersection
  • e.g. tension braces resist between 30% and 70% of total horizontal force
  • e.g. forces in restraining members not checked
• Connections are not designed

Seismic Force Resisting Systems

Available SFRS types

The seismic design requirements for a particular member are based upon which Seismic Force Resisting System (SFRS) the member forms part. Hence, in Tekla Structural Designer you can set appropriate members as part of one of the following systems:

SFRS types included for steel members

Moment Frame Systems
• Special Moment Frames (SMF)
• Intermediate Moment Frames (IMF)
• Ordinary Moment Frames (OMF)

Braced Frame and Shear Wall Systems
• Ordinary Concentrically Braced Frames (OCBF)
• Special Concentrically Braced Frames (SCBF)

Other Seismic Frame Type
• Not specified - this is a 'catch all' for all other types- but no seismic design performed.

SFRS types available for concrete members

Moment Frame Systems
• Special Moment Frames (SMF) - no seismic design performed.
• Intermediate Moment Frames (IMF)
• Ordinary Moment Frames (OMF)

Walls
• Special reinforced concrete structural wall
• Intermediate precast structural wall
• Ordinary reinforced concrete structural wall

Other Seismic Frame Type
• Not specified - this is a 'catch all' for all other types- but no seismic design performed.
SFRS types excluded

Everything else
• e.g. Eccentrically Braced Frames

Members allowed in the SFRS

The following member types are allowed to be part of a SFRS in Tekla Structural Designer
• Steel columns
• Steel beams
• Steel braces
• Concrete columns
• Concrete beams
• Concrete walls

The following member types are not allowed to be part of a SFRS in Tekla Structural Designer
• Any timber, cold-formed, general
• All other ‘Characteristics’, e.g. steel joists, truss members, purlins
• Composite members
• Plated, Westok, Fabsec, concrete filled, concrete encased – selectable but no design (i.e. only rolled)

Assigning members to the SFRS

The choice of members to be part of a SFRS is entirely the engineer’s responsibility.
• It is expected that all members in a frame are assigned to the SFRS.
• The assigned members should be specified to act in building Direction 1 or Direction 2
• Direction 1 and Direction 2 are mutually exclusive for all members and thus a column, for example, cannot be in more than one SFRS.

Special Moment Frames - assigning connection types at steel beam ends

For SFRS comprising of steel SMF it is necessary to ensure that the beams fail before the columns. To this end, an assessment of plastic moment capacity is made at each floor. The moment capacity is dependent upon the position of the plastic hinge, typically \((d_{\text{col}} + d_{\text{beam}})/2\). These locations can be selected appropriate to each beam end either in the beam properties.

Options are provided as follows:
• Plastic hinge position at start
• Plastic hinge position at end

Either, \( \frac{(d_{col} + d_{beam})}{2} \) (default)
or, \( \frac{d_{col}}{2} + L \)
\( L = 0 \) (default) in

Validation of the SFRS

There is only a small amount of validity checking for an SFRS that can be performed automatically; it remains in large part the user’s responsibility to ensure that each SFRS is defined appropriately.

The following validation conditions are however detected:

1. Any A or V brace in a Seismic Force Resisting System must have the A or V as vertically released. A warning is provided in validation if this is not the case.

2. X type bracing is defined as more than one V or inverted V (A) type brace pair on the beam. When more than one A or V brace pair is detected, the additional checks required by AISC 341-05 and AISC 341-10 given in Section 8.3 are out of scope. This situation is not detected during validation, but it is identified in the seismic design, so that the beam is given a ‘beyond scope’ status.

3. The use of K braces is not allowed in AISC 341. An error is provided in validation.

4. Tension only braces were permitted to the 05 version but had no additional requirements. In the 10 version they are only are allowed for OCBF. Thus, an error is provided in validation when a tension only brace is set as part of a SCBF and the code is the 10 version. (The same validation is also applied to compression only braces.)

5. If seismic loadcases are included in combinations and there is not at least one member assigned to each of Direction 1 and Direction 2 then a warning is issued.

Auto design of SFRS members

For members in a SFRS, auto-design only applies to conventional (not seismic) design.

Seismic Design Methods
For those regions categorised as ‘low seismicity’ it is acceptable to assume ‘ductility class low’ applies. Under these conditions the results of a seismic analysis can be fed into ‘conventional’ design.

Certain conditions (e.g. ‘high seismicity’) necessitate that a ‘seismic’ design is performed - additional design and detailing requirements have to be applied in this situation.

The additional design and detailing requirements of ‘seismic’ design are only supported in Tekla Structural Designer for the ACI/AISC Head Code.

Seismic analysis and conventional design

ELF seismic analysis and conventional design

The overall modelling, analysis and conventional design process using the ELF method is summarised as follows:

1. Modelling
   - No additional seismic modelling requirements
   - There is no need to assign members to a SFRS

2. Loading and Analysis
   - Run the Seismic Wizard to:
     - Determine building height to the highest level and adjust it if required
     - Set the ground type
     - Select ELF method of analysis (note some vertical or horizontal irregularities can prevent the use of this method)
     - Set up the relevant seismic combinations

3. Static Design
   - Run the Design (Static):
     - the results of the ELF seismic combinations are fed into the design and considered in the same way as other combinations.

4. Calculation Output
   - A Seismic Design Report is available
   - Drift limitations are checked

RSA seismic analysis and conventional design

The overall modelling, analysis and conventional design process using the RSA method is summarised as follows:
1. Modelling
   - No additional seismic modelling requirements
   - There is no need to assign members to a SFRS

2. Loading and Analysis
   Run the **Seismic Wizard** to:
   - Determine building height to the highest level and adjust it if required
   - Set the ground type
   - Select RSA method of analysis
   - Set up the relevant seismic combinations

3. Static Design
   Run the **Design (Static)**:
   - Results of the static combinations are fed into conventional design.

4. RSA Seismic Design
   Run the **Design (RSA)**:
   - Results of the RSA seismic combinations are fed into conventional design and considered in the same way as the static combinations.
   - No additional seismic design is required

5. Calculation Output
   - A Seismic Design Report is available
   - Drift limitations are checked

---

**Seismic analysis and seismic design**

*The additional design and detailing requirements of ‘seismic‘ design are only supported in Tekla Structural Designer for the ACI/AISC Head Code.*

The overall modelling, analysis and design process to be followed when seismic design is required depends on the analysis procedure (ELF or RSA) that you have chosen to perform.

The seismic design requirements vary depending upon the ‘sophistication’ of the SFRS. For example OMF have less stringent requirements than SMF.

**ELF seismic analysis and seismic design**

The overall modelling, analysis, conventional design and seismic design process using the ELF method is summarised as follows:
1. Modelling

- Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type. These members will be designed and detailed according to the seismic provisions.

All other members are then automatically classed as secondary seismic members - these are not part of the seismic action resisting system - their strength and stiffness against seismic actions is neglected and no special seismic design required. They are designed for the gravity loading when subject to the seismic conditions with due allowance for P-delta effects.

2. Loading and Analysis

Run the **Seismic Wizard** to:

- Determine building height to the highest level and adjust it if required
- Set the ground type
- Select ELF method of analysis (note some vertical or horizontal irregularities (set by the user) can prevent the use of the lateral force method)
- Set up a vibration combination
- Set up the relevant seismic combinations

3. Static Design

Run **Design (Static)** to:

- Conventionally design all members for all non-seismic (gravity and lateral) combinations
- Conventionally design all members for all seismic combinations in the same way as the other combinations.
- Perform additional seismic design for the seismic combinations for those members assigned to a SFRS

4. Calculation Output

- A Seismic Design Report is available
- Drift limitations are checked

**RSA seismic analysis and seismic design**

The overall modelling, analysis, conventional design and seismic design process using the RSA method is summarised as follows:

1. Modelling
• Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system, and assign them an SFRS direction and SFRS type. These members will be designed and detailed according to the seismic provisions.

All other members are then automatically classed as secondary seismic members - these are not part of the seismic action resisting system – their strength and stiffness against seismic actions is neglected and no special seismic design required. They are designed for the gravity loading when subject to the seismic conditions with due allowance for P-delta effects.

2. Loading and Analysis
Run the Seismic Wizard to:
• Determine building height to the highest level and adjust it if required
• Set the ground type
• Select RSA method of analysis
• Set up a vibration combination
• Set up the relevant seismic RSA combinations

3. Static Design
Run the Design (Static) to:
• Conventionally design all members for all non-seismic (gravity and lateral) combinations

4. Vibration Analysis
• At this point it is recommended that you run a 1st order vibration analysis in order to confirm the model converges on a solution, (until it is able to do so, it is pointless proceeding with a full RSA Seismic Design).

5. RSA Seismic Design
Run Design (RSA) to:
• Conventionally design all members for all RSA seismic combinations in the same way as the other combinations.
• Perform additional seismic design for the RSA seismic combinations for those members assigned to a SFRS

6. Calculation Output
• A Seismic Design Report is available
• Drift limitations are checked
Eurocode EN1998-1:2004 Seismic Wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

The seismic loading wizard is set up for the base Eurocode, the UK NA, and the drafts of the Singapore NA and the Malaysia NA.

Starting the Wizard

To initiate the wizard:

1. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information.

Site Specific Spectra

Choose the analysis procedure to be user, either:

- Code Spectra
  Choose this option to use the EC Horizontal Design Spectrum appropriate to the currently selected National Annex.

- Site Specific Spectra
  Choose this option to define a user defined spectrum. (Required for locations which use another country's loading and design codes where the code spectra are not relevant).

Refer to the 'Site Specific Spectra for ELF and RSA' Reference Guide for details of the EC default spectra and the curve equations that are used for the site specific spectra.

Base Information Page

The Structural Ductility Class is determined on this page.

Structure details

- Height to the highest level
  This field defaults to the structure height (calculated from the base to the highest point on the structure)
• **Ignore seismic in floor (and below)**
  Only floors above this level are considered when the seismic inertia combination is determined.

• **Number of storeys**
  This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above

**Ground acceleration**

• **Region**
  For Malaysia NA: Peninsular Malaysia, Sarawak, Sabah

• **Reference Peak Ground Acc, \( a_{gr} \)**
  (Ref PD6698:2009)

• **Design Ground Acc, \( a_g \)**
  \( (a_{gr} \times \gamma) \)

**Importance & Ground**

• **Importance class**
  For Base Eurocode and UK NA: I- IV as defined in BS EN 1998-1:2004 - Table 4.3.
  For Singapore NA: Ordinary or Special

• **Ground type**
  For Base Eurocode and UK NA: A-E
  For Malaysia NA: Rock, Stiff Soil, Flexible Soil
  For Singapore NA: C, D, S1
  (A - Rock, B - Very dense soil/gravel/clay, C - Deposits of dense/medium dense soil, D - Loose to medium soil, E - Surface alluvium. Ref BS EN 1998-1:2004 - Table 3.1.)

• **Importance Factor, \( \gamma \)**
  This is automatically derived from the occupancy class.

• **Spectrum type**
  For Base Eurocode and UK NA: Spectrum Type - 1 or 2 BS EN 1998-1:2004 - Cl 3.2.2.2.1(P).
  For Malaysia NA: Spectrum Type - 1
  For Singapore NA: (No choice)

• **Lower bound factor, \( \beta \)**
  This is automatically derived.

• **Upper limit of the period of the constant spectral acceleration branch, \( T_c \)**
  This is automatically derived.

• **Structural Ductility Class**
  Low, Medium or High - for whole building (not directional)
  If \( a_g \leq 0.78 \text{ m/s}^2 \) or if \( a_g \times S \leq 0.98 \text{ m/s}^2 \) - Structure suitable for Low Seismicity (where \( S \) is the soil factor)
  If not - Structure suitable for Medium or High Seismicity

• **Site natural period, \( T_S \) (Malaysia NA only)**
  This is only required if the ground type is Flexible Soil.
• **Elastic response spectral displacement, \( S_{DR} \) (Malaysia NA only)**
  This is only required if the ground type is Flexible Soil.

**Structure Regularity Page**

For Medium and High structural ductility this page is used to indicate any irregularities in plan or elevation. (For Low structural ductility only the irregularities in elevation are displayed.)

**Structure Plan Regularity - Cl 4.2.3.2**

Check the appropriate boxes to define any plan irregularities

**Structure Elevation Regularity - Cl 4.2.3.3**

Check the appropriate boxes to define any elevation irregularities

Having specified any irregularities, you then choose the analysis procedure.

**Analysis procedure to be used**

- Use Equivalent Lateral Force Procedure
- Use Modal Response Spectrum Analysis

![Warning]

*Based on the irregularities you have defined, one or both of the above methods may be unavailable.*

Any additional information, assumptions or warnings applicable to the selected method are displayed for information.

**Fundamental Period Page**

This page is used to determine the fundamental period, \( T_{Dir 1} \) and \( T_{Dir 2} \)

**Fundamental Period Definition**

- **Use approx fundamental period \( T_A \)**
  (If RSA was selected on the previous page this option is greyed out.)
- **User defined fundamental period**
  (If RSA was selected on the previous page this option is greyed out.)
- **Use vibration analysis**
  Vibration analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.

**Fundamental Period Dir 1 and Dir 2**
• **Structure Type**  
  Select from Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures (ref BS EN 1998-1 Cl 4.3.3.2.2)

• **Approx fundamental period, $T_A$**  
  This is automatically derived.

• **Fundamental period, $T_{Dir 1}$, $T_{Dir 1}$**  
  Depending on your choice of definition this will either be taken as $T_A$, or it can be a user defined value, or it will be calculated from the vibration analysis.

### Behaviour Factor Page

For Low Ductility Class, $q = 1.5$, (but the value can be changed by the user) - all other fields on the page are greyed out and cannot be changed.

For Medium or High Ductility Classes, this page is used to determine the Behaviour Factor in direction 1 and 2.

• **Ductility Class**  
  Medium or High.

• **Structure Type**  
  Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures.

• **Frame Type**  
  This options displayed here depend on the structure type selected above.

• **$\alpha_u/\alpha_l$**  
  This is a user defined multiplication factor - for the structure

• **User defined q**  
  For Medium or High Ductility Classes, check this box in order to edit the calculated q value.

• **Behaviour Factor, q**  
  This is automatically derived, but you are given the facility to edit the calculated value.

### Seismic Inertia Combination Page

This page is used to set up the load cases for the Seismic Inertia Combination.  

You should include those load cases that you want to contribute to the effective seismic weight of the structure.

> This ‘Seismic Inertia Combination’ is used to develop the seismic design loading and is classed as a vibration mass combination for the vibration analysis. It is not used in any other analysis of the structure.
**Finishing the Seismic Wizard**

When you click **Finish**, the Seismic Wizard generates the seismic load cases to be applied to the structure.

The load is applied to the structure at each level as determined for the relevant direction.
Concrete Design Handbook

This handbook provides a general overview of *Tekla Structural Designer* in the context of its application to concrete structure design.

Refer to the Reference Guides for details of the specific concrete calculations that are performed for each design code.

**Design Concrete**

The **Design Concrete** commands on the Design ribbon can be run in order to design or check all concrete beams, columns and walls in the model.

Concrete slabs and foundations involve a more interactive process and thus have their own separate design commands.

**Features common to concrete beam, column and wall design**

**Gravity or Static Design?**

- **Design Concrete (Gravity)** can be run (as a precursor to the static design) in order to concentrate on sizing members for the designated gravity combinations only - any lateral stability issues are temporarily ignored.
- **Design Concrete (Static)** is run to size members for all active combinations (gravity and lateral).

For concrete models in *Tekla Structural Designer*, adopting the above two-stage design approach is often not required. Typically you can forego the (optional) gravity only design and go straight to the (essential) static design.

**Analysis types performed in the Design Concrete process**

To generate the sets of design forces required for the beam, column, and wall design; up to three separate analyses are automatically performed:
3D building analysis

This analysis type is always performed.

Depending on the code being designed to, there are different methods of 3D building analysis. The available choices are specified via Design Options > Analysis.

Grillage chasedown analysis

This analysis type is also always performed.

FE chasedown analysis

In addition, concrete beams, columns, and walls can (optionally) be designed for a third set of design forces established from an FE chasedown analysis. This set of forces is activated in the Design Options, (via the General Parameters for each member type).

Pre-design considerations

For a concrete structure, the following settings and options in particular should be considered before running the design:

1. **Grouping** - decide if you want to make use of Design and detailing groups (concrete).

2. **Concrete Design Options** - check the concrete Design Options are appropriate for your design.

3. **Member properties and Autodesign settings** - review the design related properties that have been assigned to individual members.

   In particular review the Autodesign settings as these control whether the reinforcement in each member will be designed or checked.

If a member is in a group, then if at least one member of the group is set to autodesign the whole group will be auto-designed.

Nominal cover

The nominal cover for each member is specified in the member properties.

For beams and columns

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.
For walls

For 1 layer of reinforcement, the vertical bar is on the centre-line of the wall thickness, the face of the horizontal bar is closest to the critical concrete face.

For 2 layers of reinforcement, the horizontal bars are placed outside the vertical bars at each face.

The nominal concrete cover is measured to the face of the horizontal bar or any link/containment transverse reinforcement that may be present.

**Eurocode: Minimum Cover**

You are required to set a minimum value for the nominal cover, $c_{nom,w}$ which is then checked against the nominal limiting cover, $c_{nom,lim}$ which depends on the diameter of the reinforcement plus an allowance for deviation, $\Delta c_{dev}$ (specified in Design Options).

Generally, the allowance for deviation, $\Delta c_{dev}$ is a NDP. (Refer BS EN 1992-1-1:2004 cl 4.4.1.3 (1)P). The recommended value is 10mm, but under strict controls it can be reduced to 5mm.

**Assume cracked**

This setting is specified for each member under the ‘Design control’ heading in the member properties.

Assuming concrete sections are cracked has a direct affect on the analysis; smaller Modification Factors are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway sensitivity calculations are also influenced by this assumption.
Design parameters

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load. 

\[ \frac{SLS}{ULS} = \frac{(1.0G_k + \psi_2Q_k)}{(factored G_k + factored Q_k*IL reduction)} \]

If Qk is taken as 0 then:

\[ \frac{SLS}{ULS} = \frac{1}{1.25} = 0.8 \]

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and \(\psi_2 = 0.3\)):

For 50%IL reduction,

\[ \frac{SLS}{ULS} = \frac{1 + 0.3}{1.25 + 1.5*0.5} = 0.65 \]

For no IL reduction,

\[ \frac{SLS}{ULS} = \frac{1 + 0.3}{1.25 + 1.5} = 0.47 \]

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

Reinforcement Parameters
The reinforcement parameters common to concrete members are specified in Design Options > Concrete > Reinforcement Parameters.

**Eurocode: Reinforcement Anchorage Length Parameters**

**Max. Bond Quality Coefficient**

Acceptable input range 0.5 to 1.0

In the bond stress calculation (Cl 8.4.2), the bond quality coefficient $\eta_1$ can be either 1.0 or 0.7 depending on section depth. Where 0.7 is used the bond strength is reduced and laps are extended.

Specifying a maximum of 1.0 for the Bond Quality Coefficient allows the coefficient to vary between 0.7 and 1.0 as required, hence lap lengths will vary accordingly.

Some users may prefer to specify a maximum of 0.7 (which actually fixes the coefficient at 0.7), the effect is to standardise on the use of extended lap lengths throughout. Further conservatism can be introduced in all lap lengths by using a value as low as 0.5.

**Plain Bars Bond Quality Modifier**

Acceptable input range 0.1 to 1.0

In the EC2 Cl 8.4.2 bond stress calculation, there is no factor relating to the rib type of reinforcement, and no guidance on what adjustments if any should be made for plain bars.

In *Tekla Structural Designer* a factor “T” has been introduced (as in BS8110) to allow for this adjustment. It is the users responsibility to enter a suitable value for plain bars. (Until further guidance becomes available, we would suggest that as per BS8110 a value of 0.5 would be reasonable.)

**Type-1 Bars Bond Quality Modifier**

Acceptable input range 0.1 to 1.0

In the EC2 Cl 8.4.2 bond stress calculation, there is no factor relating to the rib type of reinforcement, and no guidance on what adjustments if any should be made for Type 1 bars.

In *Tekla Structural Designer* a factor “T” has been introduced (as in BS8110) to allow for this adjustment. It is the users responsibility to enter a suitable value for Type 1 bars. (Until further guidance becomes available, we would suggest that as per BS8110 a value of 0.8 would be reasonable.)

**Design and detailing groups (concrete)**
Why use concrete design and detailing groups?

Concrete beams and columns and isolated foundations are automatically put into groups for two reasons:

**For editing purposes** - individual design groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.

* A fixed set of rules are used to determine the automatic member grouping: for example beams must be of similar spans, columns must have the same number of stacks, bases must be of similar lengths in X and Y, and similar depths etc. The same rules also constrain manual group editing.

**For design and detailing purposes** - to reduce the processing time and also reduce the volume of output created.

In a manual process, the Engineer might select a number of sufficiently similar members to form a ‘design group’ to carry out a single design that is sufficient for all members in the group. Using this single set of design results, they would then create sub-groups of the members in the ‘design group’ to produce a set of output details for each of these sub-groups.

In *Tekla Structural Designer*, concrete design groups are analogous to the manually created ‘design groups’ described above. Concrete detailing groups are analogous to the sub-groups.

* Grouped design and detailing is optional and can be deactivated if required.

What happens in the group design process?

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

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

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

Concrete beam design groups

- A beam element may be in only one design group.
- Design groups may be formed from single span or multi-span continuous beams.
- All beam elements in the group must have an identical number of spans.
- For each individual span all beam elements in the group must have an identical cross section, including flange width where appropriate, and span length.
- All beam elements in the group must have identical material properties and nominal cover.
- All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.

Concrete column design groups

- A column element may be in only one design group.
- All column elements in the group must have an identical number of stacks.
- For each individual stack all column elements in the group must have an identical cross section, and stack length.
- All column elements in the group must have identical material properties and nominal cover.

Pad base design groups

- A pad base may be in only one design group.
- Each base in the group must have an identical cross section and depth.
- Each base in the group must have identical eccentricities in X and Y.
- Each base in the group must have identical material properties and nominal cover.

Pile cap design groups

- A pile cap may be in only one design group.
- Each pile cap in the group must have an identical cross section and depth.
- Each pile cap in the group must have identical eccentricities in X and Y.
- Each pile cap in the group must have identical material properties and nominal cover.
**Detailing group requirements**

Each parent design group is sub-divided into one or more detailing groups.

Although there can be a ‘1 to 1’ relationship between a design group and a detailing group, in practice there will often be a ‘1 to many’ relationship as each design group is likely to require several detailing groups to allow for differences in the connected geometry.

Detailing groups are formed for the different concrete member types based on the following rules:

**Concrete beam detailing groups**

- A detailing group may be associated with only one parent design group.
- A beam element may be in only one detailing group.
- Detailing groups may be formed from single span or multi-span continuous beams.
- All beam elements in the group must have an identical number of spans.
- The cross section, including flange width where appropriate, span length and material properties in span
- ’i’ of all beam elements in the group must be identical.
- All beam elements in the group must have identical plan offsets.
- All beam elements in the group must be co-linear or be non-co-linear with identical degrees of non-co-linearity between spans.
- All beam elements in the group must have identical inclinations.
- The support types and sizes, including the attached structure above and below the beam element, must be identical in all beam elements in the group however different support types and sizes in individual multi-span continuous beams are acceptable i.e. support $i$ in beam element $j$ must be identical to support $i$ in all other beam elements in the group but supports $i$ and $i+1$ in beam element $j$ may be different.

**Concrete column detailing groups**

- A detailing group may be associated with only one parent design group,
- A column may only be in one detailing group,
- All columns in the detailing group must have an identical number of stacks,
- All columns in the group must have an identical cross-section, rotation and alignment/snap levels/offsets in stack ‘i’. In a multi-stack column, the cross-section may be different in each stack, i.e. the cross-section in span ‘i’ may be different to that in span ‘j’.
- Stack ‘i’ and stack ‘i+1’ must be co-linear for all columns, OR must be non-co-linear with an identical degree of non-co-linearity for all columns. The exact inclination must be the same for stack ‘i’ in all columns.
- At every level each column is considered to be either ‘internal’ or ‘external’ (depending on if it has beams framing into it on all four sides, or not). These settings
do not have to be identical for columns to be in the same group, but only if you have checked the option: ‘Provide ties through floor depth for internal columns’ in Design Options > Concrete > Column > Detailing Options.

**Pad base detailing groups**
- A detailing group may be associated with only one parent design group.
- A base may be in only one detailing group.
- The attached column cross-section above the base must be identical for all bases in the group however different support types are acceptable.

**Pile cap detailing groups**
- A detailing group may be associated with only one parent design group.
- A pile cap may be in only one detailing group.
- The attached column cross-section above the base must be identical for all pile caps in the group however different support types are acceptable.

**Group management**

**Automatic Grouping**
Concrete beams and columns are grouped automatically.

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

**Manual/Interactive Grouping**
After assessing the design efficiency of each group, you are able to review design groups and make adjustments if required from the Groups tab of the Project Workspace.

Detailing groups cannot be edited manually.

---

*When manually adding members to a group, the order in which they are added will incrementally affect the average length within the group, (which is then compared to the maximum length variation). Therefore, if members are not being added as you expect, try adding them in a different order.*

---

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

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

Model Editing and Group Validity Checks
When new beam elements are created when a ‘split’ or ‘join’ command is run the resulting beam elements are automatically placed in existing design and detailing groups [or new groups created].

How is grouped design and detailing de-activated for concrete members?

1. Click Design > Options... ( )
2. Click ‘Design Groups’
3. Uncheck the box adjacent to each concrete member type for which you want to de-activate design grouping.

Typical Design Concrete workflow
The following example illustrates the typical process to analyse and design all the beams, columns and walls in a concrete structure.
The example has been broken down into the following main steps:

1. **Set up Pattern Loading**
2. **Set all beams columns and walls into autodesign mode**
3. **Review beam and column design groups**
4. **Review beam, column and wall design parameters and reinforcement settings**
5. **Perform the concrete design**
6. **Review the design status and ratios**
7. **Create Drawings and Quantity Estimations**
8. **Print Calculations**

**Set up Pattern Loading**

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via **Update Load Patterns** on the Load ribbon.
Set all beams columns and walls into autodesign mode

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

Review beam and column design groups

Provided that the concrete beam and column options are checked in Design Options> Design Groups, the design groups shown in the Groups tab of the Project Workspace will be applied in the beam and column autodesign processes.

Groups will initially have been established for members sharing the same geometry, but you should consider reviewing and amending them if required.
**Review beam, column and wall design parameters and reinforcement settings**

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

**Perform the concrete design**

Concrete beams, columns and walls are designed by running **Design Concrete (Static)** from the Design ribbon.

- **Unknown**
- **Beyond Scope**
- **Error**
- **Warning**
- **Fail**
- **Pass**

> Reinforcement is designed, but member sizes are not changed during this process.

**Review the design status and ratios**

You can display the Design Status and Ratios from the **Review View** in order to determine if any remodelling is required.
In this example a number of members are not heavily loaded so you could consider resizing or designing them interactively.

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

**Create Drawings and Quantity Estimations**

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

**Print Calculations**

Create a model report that includes the member design calculations that have been performed. (The default Member Design Calcs report includes these along with design calculations for other member types in the model).
Reviewing Design Concrete and refining the design of individual members

A number of tools are available to assist the post-design review:

1. **Review View** - use the various tools on the Review toolbar to get an overall picture of the design results.

2. **Check Member** - to view detailed results for individual concrete members.
   See: [How do I view results for a single concrete member (without re-selecting steel)?](#)

3. **Design Member** - to quickly reselect reinforcement for an individual member, (without having to re-perform the entire structure design).
   See: [How do I re-select steel for a single concrete member and then view its results?](#)

   ‘Design Member’ is intended for individual member design, other members in the same design group are NOT updated with the revised reinforcement. If a member is in a group, then if at least one member of the group is set to autodesign the whole group will be auto-designed.

4. **Interactive Design** - if required, use to actively control the reinforcement selected for an individual member.
   See: [Interactive concrete member design](#)

**How do I view results for a single concrete member (without re-selecting steel)?**

1. Select the member for which you want to view results.
2. Hover the cursor over the member until its outline is highlighted, then right click.
3. From the context menu select **Check Member**.

The results dialog is displayed from where all the detailed calculations can be viewed.

**How do I re-select steel for a single concrete member and then view its results?**

1. Select the member for which you want to view results.
2. Ensure the **Autodesign** setting is active.
3. At the same time, review the Reinforcement and other settings.
   (For beams in particular, ensure you have the correct top and bottom bar pattern selected.)
4. Hover the cursor over the member until its outline is highlighted, then right click.
5. From the context menu select **Design Member**.
Steel is re-selected for the member and then the results dialog is displayed from where all the detailed calculations can be viewed.

**Features of concrete beam design**

**Analysis types used for concrete beam design**

Concrete beams are designed for a set of design forces obtained from the 3D building analysis plus a second set of design forces obtained from a grillage chasedown analysis. In addition, the beams can (optionally) be designed for a third set of design forces established from an FE chasedown analysis.

**Autodesign (concrete beam)**

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

*If concrete beams have been set to be designed using [Design and detailing groups (concrete)](https://www.autodesk.com), then if at least one member of the group is set to autodesign the whole group will be auto-designed.*

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

**Select bars starting from**

This option only appears if **Autodesign** is selected. It sets the autodesign start point for both longitudinal bars and links.

The options are:
- Minima (default)
- Current bars

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

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

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

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

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

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

Deflection control

Deflection control

Tekla Structural Designer controls deflection by comparing the calculated limiting span/effective depth ratio $L/d$ to the maximum allowable value $(L/d)_{\text{max}}$.

The parameters listed below directly affect the deflection calculations and hence require consideration:

1. Increase reinforcement if deflection check fails
   
   • If this beam property (located under the ‘Design Control’ heading) is unchecked and the check fails, then the failure is simply recorded in the results
   • If it is checked, then the area of tension reinforcement provided is automatically increased until the check passes, or it becomes impossible to add more reinforcement.

2. Consider flanges
   
   • Checking this beam property (located under the ‘Design Control' heading) can assist in satisfying the deflection check.

Use of beam flanges

A beam element or beam line is initially created in the model with a rectangular cross-section.
The beam properties can then be edited to take account of flanges arising from adjoining slabs, making the following beam shapes possible:

These shapes have common features which are shown in the figure below:

\[ h = \text{overall depth including the depth of the slab} \]
\[ h_f = \text{depth of slab} \]
\[ b_w = \text{width of beam} \]
\[ b_{\text{eff}1} = \text{flange width side 1} \]
\[ b_{\text{eff}2} = \text{flange width side 2} \]
\[ b_{\text{eff}} = \text{flange width} \]
\[ = b_{\text{eff}1} + b_w + b_{\text{eff}} \]

**Use of Flanged Beams**

Flanged beam properties can be specified under the **Design Control** heading in beam properties.
Typically, flanged beams can be either ‘T’ shaped with a slab on both sides of the beam or ‘I’ shaped with a slab on only one side of the beam.

The characteristic behaviour of flanged beams which can be made use of in design is the fact that the major axis bending resistance of the member is enhanced by the presence of adjoining concrete slabs which serve to increase the area of the compression zone activated during major axis bending.

This effectively raises the position of the neutral axis thereby increasing the lever arm of the longitudinal tension reinforcement and reducing the quantity of reinforcement required.

Modelling and Design Choices

The combinations of beam modelling and design choices that are allowed are:

• If a beam is modelled and analysed with a rectangular cross-section it may be designed as a rectangular beam or as a flanged beam at the choice of the user.
• If a beam is modelled and analysed as a flanged beam it can only be designed as a flanged beam.

Curved in plan flanged beams can be modelled and designed however for such beams the flange width must be set manually.

If a slab is present, the beam automatically validates the slab as a potential candidate for being a beam flange using a number of criteria, the main ones being:

• the slab can be on one or both sides of the beam but
  • it must extend for a distance ≥ the slab depth from the vertical face of the beam and
  • it must extend for the full span length of the beam
• the slab must be a reinforced concrete slab
• if there are slabs on both sides of the beam, they may be of different depths and these depths may vary along the length of the beam

The effective width of any valid slab on each side of the beam, \( b_{\text{eff,i}} \), is calculated and the results that are appropriate at the mid-span length point are displayed in the Properties window.

Effective Width of flanges

The effective width of the compression flange is based on \( L_0 \), the distance between points of zero bending moment.

For flanged beams the following values of \( L_0 \) are to be used;

For a simply supported beam \( L_0 = L \)

For a continuous beam, the value of \( L_0 \) may be obtained using the following simplified rules;

End span of a continuous beam with a pinned end support \( L_0 = 0.85*L \)
End span of a continuous beam with a fixed end support \( L_0 = 0.70L \)

Internal span of a continuous beam \( L_0 = 0.70L \)

where
\[ L = \text{the clear length of the span under consideration} \]

The effective flange width, \( b_{eff} \), is given by;
\[ b_{eff} = b_w + \sum b_{eff,i} \]

where
\[ b_{eff,i} = \text{the effective width of the flange on side } i \text{ of the beam} \]
\[ = \text{MIN}[0.2L_0, b_i(0.2b_i + 0.1L_0)] - O_w \]

where
\[ L_0 = \text{the distance between points of zero moment as defined above} \]
\[ b_i = 0.5\text{the clear distance between the vertical faces of the supports for the valid concrete slab on side } i \text{ of the beam or from the vertical face of the beam to the centreline of any supporting steel beam} \]
\[ b_w = \text{the width of the beam} \]
\[ O_{wi} = \text{the user specified allowance for an opening} \]

If the slab thickness varies on each side of the beam, the thinner value is used in calculating the beam properties.

This relationship is illustrated below

The above calculation for \( b_{eff} \) is also used for ‘I’ beams with a slab on only one side although in this case, \( b_1 \) or \( b_2 \) as appropriate is = 0.

Adjacent Beams not Parallel

For beams that are not parallel, the effective width of the flange will vary along the length of the beam and the value used in element design calculations is the
minimum width that occurs in the distance between the points of zero moment i.e. the previously calculated L₀ length.

**Holes/Openings in Calculated Effective Flange Width**

The presence of holes or openings in slabs can have an impact on the effective width of the slab used in the element design; indeed, in some circumstances it may mean that the beam cannot be designed as flanged.

However, as it is difficult to identify holes or openings in slabs that are within the calculated effective flange width - such openings are ignored in the automatic calculation of the effective flange width. Where such holes or openings exist you should therefore manually adjust the flange width to take account of them. This is achieved using the **Allowance for openings left/right** parameters in the beam properties.

**Longitudinal reinforcement**

In order to determine the design forces for the bending checks user defined longitudinal reinforcement regions must first be established.

**Longitudinal Reinforcement Shapes Library**

**Bar layers**

Designed longitudinal reinforcement is positioned in the top and bottom of the beam and can be tension reinforcement or compression reinforcement.
The longitudinal reinforcement in the top and bottom of a beam can consist of 1 to ‘n_{L}’ parallel layers with the layer nearest to the top or bottom surface of the beam being Layer 1.

The number and diameter of bars in each layer can vary but bars in different layers must be vertically aligned. This is to ensure that there is adequate space to allow the concrete to be poured and properly compacted around the bars.
Longitudinal reinforcement in the side of the beam is only provided in beams with a depth greater than a certain value as follows:

\[ h \geq 1000 \text{ mm} \quad \text{EC2} \]

**Longitudinal Reinforcement Shapes Library**

The common basic Shapes of bars used for the purposes of providing longitudinal reinforcement in beams are shown in the table below.

In the first release, only the Shapes listed in the table are available for selection.

<table>
<thead>
<tr>
<th>BS8666 Shape Code</th>
<th>Bar Shape</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td></td>
<td>Straight bar</td>
</tr>
<tr>
<td>26</td>
<td></td>
<td>Single crank</td>
</tr>
<tr>
<td>46</td>
<td></td>
<td>Double crank</td>
</tr>
</tbody>
</table>
### Longitudinal Reinforcement Patterns Library

There are three Standard Patterns for top reinforcement, \( SPT_1 \), \( SPT_2 \) and \( SPT_3 \) and two Standard Patterns for bottom reinforcement, \( SP_{B1} \) & \( SP_{B2} \) as illustrated in the figures below.

#### Standard Patterns of Top Reinforcement

<table>
<thead>
<tr>
<th>Number</th>
<th>Pattern</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td></td>
<td>Standard bob</td>
</tr>
<tr>
<td>34</td>
<td></td>
<td>Standard bob with crank</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>Extended bob</td>
</tr>
<tr>
<td>34</td>
<td></td>
<td>Extended bob with crank</td>
</tr>
<tr>
<td>21</td>
<td></td>
<td>U bar</td>
</tr>
<tr>
<td>99</td>
<td></td>
<td>U bar with crank</td>
</tr>
</tbody>
</table>
The bars used in the Standard Top Patterns are:

1. Straight bar extending to approximately 25% of each span (end points of this bar are determined by the design region settings)
2. Straight bar extending to approximately 10% of each span (end points of this bar are determined by the design region settings) – if required by the design
3. Double cranked bar lapped with bar (1)
4. Straight bar running approximately from face to face of beam supports
5. Single cranked bar running from centre span to centre span with the option to merge bars if they are the same size and number to extend the bar over several spans
6. Bob bar
7. Bob bar

**Standard Patterns of Bottom Reinforcement**
The bars used in the Standard Bottom Patterns are:

1. Bar with a bob at each end
2. Straight bar with a length approximately 70% of span – if required by the design
3. Single cranked bar extending over several spans or over one span only and lapped within a support – with bob if it continues over an end span.
4. Straight bar
5. Straight bar running approximately from face to face of beam supports
6. Straight bar
7. Bob bar

Modified versions of the above standard patterns are applied for use in single spans and in cantilever spans where no backspan beam is present.

For short span beams, it becomes uneconomic and impractical to lap bars in beams. These facts coupled with the anchorage lengths that are required make the use of multiple design regions for the longitudinal reinforcement unnecessary. To cater for this a short span beam length can be defined in Design Options > Beam > Reinforcement Settings and the bar patterns adopted for such short spans are as shown below:

Standard Patterns of Reinforcement for Short Span Beams
**Longitudinal Reinforcement Regions**

**Design Check Regions for Bending**

When considering the longitudinal steel in the top and bottom of the beam, the design checks are performed in a specified number of regions that are symmetrically placed about the centre of the beam. The regions are specified as user defined proportions of the clear span of the beam, expressed as a percentage; the number of regions being initially governed by the choice of longitudinal bar pattern.

**Top Regions**

Three standard patterns are available for defining the top regions:

- Standard Top 1 – maximum of 3 regions
- Standard Top 2 – maximum of 5 regions
- Standard Top 3 – maximum of 6 regions

To provide flexibility, once a pattern has been selected you are able to vary the percentage region widths, with the range of each being $0% \leq T_i \leq 100\%$ and with $\sum T_i = 100\%$.

For example the Standard Top 2 pattern initially consists of 5 regions, $T_1, T_2, T_3, T_4,$ and $T_5$:

By varying the percentage region widths a number of possibilities can be catered for:

- 1 Region  $T_1=T_5 = 0\%; T_2=T_4 = 0\%; T_3 = 100\%$
- 3 Regions $T_1=T_5 = 0\%; T_2=T_4 = 100\%-T_3\%; 0\% < T_3 < 100\%$
- 5 Regions $T_1=T_5 > 0\%; T_2=T_4 > 0\%; T_3 > 0\%$

In each top region, the maximum negative bending moment within the region is determined for design purposes.

**Bottom Regions**
Two standard patterns are available for defining the bottom regions:

Standard Bottom 1 – maximum of 3 regions

Standard Bottom 2 – maximum of 3 regions

Similar to the top patterns, once a pattern has been selected you are able to vary the percentage region widths, with the range of each being $0\% \leq B_i \leq 100\%$ and with $\sum B_i = 100\%$.

This enables the following single region, or three possibilities:

1 Region  $B_1 = B_3 = 0\%; B_2 = 100\%$

3 Regions  $B_1 = B_3 = \text{RANGE}(0\%-25\%); B_2 = \text{RANGE}(50\%-100\%)$

In each bottom region, the maximum positive bending moment within the region is determined for design purposes.

**Regions for Cantilevers**

The standard patterns for cantilevers are edited and applied in the same way as the standard patterns for continuous spans. Up to 3 regions can be defined for the top, but only a single region exists for the bottom.

These regions are illustrated below

The design value of the bending moment used for the design in a region is the maximum factored bending moment arising in the region under consideration.

**Relationship between Reinforcement Patterns and Design Regions**
There is a close link between the reinforcement patterns and the design regions. After selecting a Standard Reinforcement Pattern, you can then choose the length of each design region. The number of regions adopted will dictate the bars that are used to reinforce the beam and likewise, the selection or de-selection of particular bars will dictate the design regions used.

The selection process of Standard Reinforcement Patterns and Design Regions is:

1. Choose a Standard Reinforcement Pattern for the top reinforcement (from the Standard Pattern Setup droplist)
2. Select the bars that are to be used
3. Set the length of the resulting design regions
4. Repeat the selection process for the bottom reinforcement.

The bar selection and design region lengths are inextricably linked. If a bar is not selected then the design region has zero length.

It is important that the bar and design region selection is carried out in an orderly manner and that the selections are made in the correct order.

This relationship between bars and design regions is best illustrated using Standard Patterns SP_{T1} and SP_{B1} as an example (for an internal span) as shown in the diagram below.

Considering the top reinforcement first, if the user opts to de-select Bar 2 then design regions T_4 and T_5 will be zero length and the user will then select a length for T_1.

Likewise for the bottom reinforcement, de-selecting Bars 7 and 11 will set designs region B_2 and B_3 to zero length.

If all the available bars are selected in this example then the bars used to provide the area of reinforcement required by the design in each design region will be;

Design Region T_1 (& T_5) : Bar 1 + Bar 2
Design Region T_2 (& T_4) : Bar 1
Design Region T_3 : Bar 3
Design Region B₁ (& B₃): Bar 8 + Bar 11
Design Region B₂: Bar 7 + Bar 8

The above approach is extended for all the Standard Patterns.

**Shear reinforcement**

**Shear Reinforcement Shapes Library**

Vertical shear reinforcement is provided in the form of links which can be single or multiple with 1 (▁) or 2 (▁▁ or ▁▁) vertical legs.

The common basic shapes of bars used for the purposes of providing shear reinforcement in beams are shown in the table below.

In the first release, only the shapes listed in the table are available for selection.

**Shear Reinforcement Typical Shapes**

<table>
<thead>
<tr>
<th>BS8666 Shape Code</th>
<th>Link/Stirrup Shape</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>51</td>
<td><img src="image" alt="Closed Link/Stirrup" /></td>
<td>Closed Link/Stirrup</td>
</tr>
<tr>
<td>47</td>
<td><img src="image" alt="Open Link/Stirrup" /></td>
<td>Open Link/Stirrup</td>
</tr>
<tr>
<td>21</td>
<td><img src="image" alt="Top Closer Link/Stirrup" /></td>
<td>Top Closer Link/Stirrup</td>
</tr>
<tr>
<td>99</td>
<td><img src="image" alt="Single Leg Link/Stirrup" /></td>
<td>Single Leg Link/Stirrup</td>
</tr>
</tbody>
</table>
Shear Reinforcement Patterns Library

There are three Standard Patterns for shear reinforcement, *Closed*, *Open* and *Torsion*. However, you are initially only offered a choice of 2 patterns, Closed or Open.

The Standard Patterns for shear reinforcement are:

**Closed**
Closed links (shape code 51) with additional double leg (shape code 51) or single leg links (shape code 99) if required by the design. The design dictates the number of vertical legs required. You can choose if single leg links are acceptable.

**Open**
Open links with top closers (shape codes 47 and 21) with additional double leg (shape code 47) or single leg links (shape code 99) if required by the design. The design dictates the number of vertical legs required. You can choose if single leg links are acceptable.

**Torsion**
Torsion links (shape code 63) for the outer link with closed links (shape code 51) or single leg links (shape code 99) as internal links if required by the design. Note that this is not a user option but is determined by the design.

Shear Reinforcement Regions

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 and analysis types, $V_{zi}$, is determined and this maximum value used to determine the shear reinforcement required in that region.

The lengths of the shear regions are subject to user selection and may be either:
Optimised
This option is only valid when the maximum positive shear from all combinations and analysis types occurs at one end of a beam and the maximum negative shear from all combinations and analysis types occurs at the other end of the beam. If this situation does not exist then this option is not allowed and the ‘Fixed Proportions’ method will be used.

In this case in the central region $S_2$, shear reinforcement is provided to meet the maximum of the minimum code requirement or minimum 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.

In this method, region $S_2$ is defined as being that part of the beam in which the minimum amount of shear reinforcement is acceptable.

Or

Fixed proportions
In this case the regions are defined as fixed proportions of the clear span [face to face length] of the beam expressed as a percentage $S_1\%$, $S_2\%$ and $S_3\%$ with the default values for $S_1$ and $S_3$ being $\text{MAX}(0.25\times L, 2\times h)$ and that for $S_2$ being $(L-S_1-S_3)$.

In cantilevers, the regions are as shown below.

In all cases, the range of each region is $0\% \leq S_i \leq 100\%$ and $\sum S_i = 100\%$.

Features of concrete column design

Autodesign (concrete column)
The design mode for each column is specified in its properties.

If concrete columns have been set to be designed using Design and detailing groups (concrete), then if at least one member of the group is set to autodesign the whole group will be auto-designed.
• When Autodesign is selected an iterative procedure is used to design both the longitudinal bars and links. This applies the spacing maximisation method which attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link spacing.

• When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.

Select bars starting from

This option only appears if Autodesign is selected. It sets the autodesign start point for both longitudinal bars and links.

The options are:

• Minima (default)
• Current bars

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

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

Section

Shape

The following column shapes are allowed:

• Rectangular
• Circular
• L-shape
• T-shape
• C-shape
• Elbow with uniform thickness
• Trapezium
• Parallelogram
• Regular polygon (between 3 and 8 sides)

Breadth and Depth
When defining a section with two axes of symmetry (e.g. a rectangular section) you should ensure that the longer dimension is input as the depth and the shorter dimension as the breadth (as shown below left).

A rotation can then be applied if required, in order to orientate the column correctly in the model.

When the section is designed the ‘major axis’ calculations will then relate to bending about the strong axis and the ‘minor axis’ calculations to bending about the weak axis.

If the breadth and depth has been transposed during input (as shown above right), ‘major axis’ would then relate to bending about the weak axis and ‘minor axis’ would relate to bending about the strong axis.

Holes

Rectangular or circular holes can be placed in rectangular and circular columns when the section is being defined, they can’t be placed in other column shapes.

Reinforcement link arrangements such as double links, triple links and cross-links are not designed intelligently to account for holes.

Slenderness

The significant parameter within the slenderness criteria is a choice of how the column is contributing to the stability of the structure.

- bracing - provides lateral stability to the structure.
- braced - considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the ‘User input value’ option, or it is calculated in accordance with the requirements of the selected design code.
**Stiffness**

When determining the effective length, if no effective beams are found at the end of a stack, *Tekla Structural Designer* considers whether there is a flat slab restraining the stack at that end. The ‘Use slab for calculation...’ (upper/lower, major/minor) options are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for a slab on beams. If a slab on beams is found, this then acts as a restraint at the position provided the ‘Use slab for calculation...’ option has been selected, as is the case for flat slabs.

**Load reductions**

To cater for additional floors that are carried by the column that have not been included in the model an ‘Assume extra floors supported’ column property is provided. This allows you to specify how many extra floors are carried by the column. These are then taken into account when determining any reduction percentage to apply.

Reductions are only applied to those imposed load cases that have had the Reductions box checked on the **Loading dialog**. The reduction percentage for the number of floors carried is shown in **Model Settings**.

Reductions are not applied to inclined columns, (only vertical ones).

The floors that define the stacks can be designated either as ‘to be’ or ‘not to be' included in the determination of the imposed load reductions through ‘Count floor as supported' check boxes for each level in the column properties. This feature enables what appears to be a roof to be counted as a floor, or conversely allows a mezzanine floor to be excluded from the number of floors considered for any particular column. The moments from fixed ended beams framing into a column are never reduced.

**Stacks and reinforcement lifts**

A column may contain one or more reinforcement lifts, each of which may contain one or more stacks.

For the first release, reinforcement lifts are restricted to contain only one stack.

A reinforcement lift is defined as a height of column between two levels anywhere in the building throughout which the cross-section of the column and the reinforcement arrangement is constant. For the cross-section to be constant, all aspects of the shape,
dimensions and rotation must be identical. Reinforcement changes in one stack in the reinforcement lift apply to all stacks in the reinforcement lift.

It is only the longitudinal reinforcement which has to be constant throughout a reinforcement lift - the shear links may be different in stacks within a reinforcement lift.

During design, an arrangement is created for the top stack in the reinforcement lift which passes the design checks. This arrangement is then applied to all stacks in the reinforcement lift. The next stack down is then designed using the applied reinforcement arrangement. If the initial arrangement does not pass, the new arrangement which does pass is applied to all stacks and design continues with the new arrangement starting at the top stack. Eventually, the limits of bar size and spacing are reached or the design passes for all stacks - design then stops.

A stack is defined as a height of column between consecutive floor levels. Reinforcement is designed by stack - although this may be part of a larger reinforcement lift. Essentially the stack is considered in isolation during the design of the reinforcement but is then considered as part of a reinforcement lift. All stacks are part of reinforcement lifts, even if the reinforcement lift is only one stack high. Therefore, longitudinal reinforcement is constant throughout a stack.

**Column design forces**

Six design forces (axial, torsion, major and minor moment, major and minor shear) are determined at the top and bottom of each stack for each combination for one or more sets of analysis results.

Intermediate loads are allowed. The moments due to imperfections and slenderness effects are added appropriately, and the result checked against the minimum design moment as appropriate.

**Features of concrete wall design**

**Autodesign (concrete wall)**

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

- When Autodesign is selected an iterative procedure is used to determine the reinforcement. A spacing maximisation method is applied for both longitudinal bars and links. This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link spacing.

- When Autodesign is not selected (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient.
This option only appears if **Autodesign** is selected. It sets the autodesign start point for both longitudinal bars and links.

The options are:
- Minima (default)
- Current bars

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

---

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

---

**Slenderness**

The significant parameter within the slenderness criteria is the choice of how the wall is contributing to the stability of the structure.

- In-plane (major) direction, a wall is usually considered to be a bracing member.
- Out-of-plane (minor) direction, a wall is usually considered to be braced by other stabilizing members.

The second slenderness parameter is the effective length factor, which is either input directly by choosing the ‘User input value’ option, or it is calculated in accordance with the requirements of the selected design code.

**Stiffness**

When determining the effective length, if no effective beams are found at the end of a stack, **Tekla Structural Designer** considers whether there is a flat slab restraining the stack at that end. The ‘Use slab for calculation...’ (upper/lower, major/minor) options are used to indicate whether any such slab should be considered as a restraint.

If there are no effective beams and there is no flat slab, the program looks for a slab on beams. If a slab on beams is found, this then acts as a restraint at the position provided the ‘Use slab for calculation...’ option has been selected, as is the case for flat slabs.

**Reinforcement**
The wall properties: ‘Reinforcement layers’, ‘Form’ and ‘Include end zones’ can be combined as required in order to obtain a range of reinforcement patterns, e.g:

- Single layer, using mesh reinforcement
- Two layers, using mesh reinforcement
- Single layer, using loose bars
- Two layers, using loose bars
- End zones, with a single layer of mesh in the mid zone
- End zones, with two layers of mesh in the mid zone
- End zones, with a single layer of loose bars in the mid zone
- End zones, with two layers of loose bars in the mid zone

### Load reductions

To cater for additional floors that are carried by the wall that have not been included in the model an ‘Assume extra floors supported’ wall property is provided. This allows you to specify how many extra floors are carried by the wall. These are then taken into account when determining any reduction percentage to apply.

> Reductions are only applied to those imposed load cases that have had the Reductions box checked on the **Loading dialog**. The reduction percentage for the number of floors carried is shown in **Model Settings**.

Levels can be designated either as ‘to be’ or ‘not to be’ included in the determination of the imposed load reductions through ‘Count floor as supported’ check boxes for each level in the wall properties. This feature enables what appears to be a roof to be counted as a floor, or conversely allows a mezzanine floor to be excluded from the number of floors considered for any particular wall. The moments from fixed ended beams framing into a wall are never reduced.

### Stacks and reinforcement lifts

A wall may contain one or more reinforcement lifts, each of which may contain one or more stacks.

> For the first release, reinforcement lifts are restricted to contain only one stack.

A reinforcement lift is defined as a height of column between two levels anywhere in the building throughout which the cross-section of the wall and the reinforcement arrangement is constant. For the cross-section to be constant, all aspects of the shape, dimensions and rotation must be identical. Reinforcement changes in one stack in the reinforcement lift apply to all stacks in the reinforcement lift.
It is only the longitudinal reinforcement which has to be constant throughout a reinforcement lift - the shear links may be different in stacks within a reinforcement lift.

During design, an arrangement is created for the top stack in the reinforcement lift which passes the design checks. This arrangement is then applied to all stacks in the reinforcement lift. The next stack down is then designed using the applied reinforcement arrangement. If the initial arrangement does not pass, the new arrangement which does pass is applied to all stacks and design continues with the new arrangement starting at the top stack. Eventually, the limits of bar size and spacing are reached or the design passes for all stacks - design then stops.

A stack is defined as a height of wall between consecutive floor levels. Reinforcement is designed by stack - although this may be part of a larger reinforcement lift. Essentially the stack is considered in isolation during the design of the reinforcement but is then considered as part of a reinforcement lift. All stacks are part of reinforcement lifts, even if the reinforcement lift is only one stack high. Therefore, longitudinal reinforcement is constant throughout a stack.

Wall design forces

Six design forces (axial, torsion, major and minor moment, major and minor shear) are determined at the top and bottom of each stack for each combination for one or more sets of analysis results.

Intermediate loads are allowed. The moments due to imperfections and slenderness effects are added appropriately, and the result checked against the minimum design moment as appropriate.

Walls can be loaded laterally, but are always considered to span vertically.

Horizontal moments that may develop in a meshed wall are ignored in the design.

Concrete slab design

Features of concrete slab analysis and design

Analysis types used for concrete slab design

Flat Slabs and 2-way spanning slabs on beams are both designed for moments derived from an FE Chasedown Analysis; if you have elected to mesh 2-way slabs in the analysis, the 3D Building Analysis results are also considered.

Design of slab panels that have their decomposition property specified as ‘one-way’ is beyond scope - see: Concrete slab load decomposition
Concrete slab load decomposition

A flat slab panel always uses two-way load decomposition.

A slab on beams panel can either be specified to use one-way or two-way decomposition - however if it is specified as one-way it cannot then be designed in Tekla Structural Designer.

One-way load decomposition in Tekla Structural Designer is a simple procedure that does not determine slab design forces. When a slab's decomposition is set as one-way it is assumed that it is some form of precast slab (presumably designed by safe load tables).

It should be noted that any in-situ slab is capable of two-way decomposition:

• When a slab is set as two-way it will only effectively span in 2 directions if its proportions and support conditions mean that there will be a two-way effect.
• For example - If a slab that has a span of 6 units in one direction and 50 units in the other is set to two-way decomposition, then although it is two-way the FE analysis will still inherently take the load one-way.

When a slab panel is specified with two-way decomposition, a general FE based approach is always used to determine the design forces:

• The worst design moment (per unit width) is found in each direction of the slab - if the design moment is zero in one of the directions then the analysis has shown that the slab is effectively spanning one-way and the supplied reinforcement in this secondary direction will be selected to suit the minimum requirements of secondary reinforcement.
• Note that this FE based approach inherently caters for point loads, line loads, openings, etc and for the possibility of variable adjacent span lengths in a continuous ‘1-way’ slab (and of course it can still be applied to the simple case of a ‘1-way’ slab with a uniform UDL applied and uniform span lengths).

Slab on beam idealized panels

In order to perform ‘span - effective depth’ checks for irregular shaped beam and slab panels, they have to be converted to idealized four sided rectangular panels.

To do this, the enclosing lengths of the panel in X and Y are first determined, (local X being defined by the panel ‘rotation angle’):

\[ \text{Lx-enc} = \text{maximum overall length of the panel measured parallel to local X} \]
\[ \text{Ly-enc} = \text{maximum overall length of the panel measured perpendicular to local X} \]

A user specified adjustment ratio is then applied to these lengths to determine the adjusted lengths. Conservatively the adjustment ratio defaults to 1.0 in both directions.
In situations where the panel does not have 4 sides, (such as the one shown below), some engineering judgement might be required when deciding on appropriate values of the adjustment ratios in each direction.

The resulting idealized panel with dimensions in X and Y is illustrated below.

**Edge Category**

For the span-effective depth check, the edge categories in each direction have to be manually assigned to the idealized slab panel. The three edge categories being:

- Unsupported
- Continuous Support
- Dis-continuous Support (default)
**Combined slab and patch reinforcement**

Slabs panels can potentially have 4 layers of background reinforcement, (however any of the below layers/directions can be set to “none” if required).

- Top of slab
  - x dir reinforcement
  - y dir reinforcement
- Bottom of slab
  - x dir reinforcement
  - y dir reinforcement

In addition, rectangular reinforcement patches can be applied to slab panels:

- column patch - at column stack heads
- beam patch - along beams
- wall patch - along walls
- panel patch - at the panel centre
  - typically positioned centrally - but not restricted to this location and also not restricted to existing purely within one panel
  - can 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, the user has the option to use the sum of the background + patch reinforcement - if reasonably aligned.

Note that patches may overlap on the plan view, and there is no restriction on this, even patches relating to the same layer of reinforcement are allowed to overlap. This situation is handled conservatively during design by simply ignoring the overlap.
**Typical flat slab design procedure**

In *Tekla Structural Designer* an interactive design approach is required for flat slab design because the patch and panel design are inter-dependant.

The below example illustrates this approach, which typically consists of the following steps:

1.  **Set up Pattern Loading**

2.  **Design All** - to establish analysis results

3.  **Consider Deflection (for Flat slabs)**

4.  **Select a Level** (or sub-model) to be designed and within that level:
   
   a.  **Add Patches**
   
   b.  **Design Panels**
   
   c.  **Review/Optimise Panel Design**
   
   d.  **Design Patches**
   
   e.  **Review/Optimise Patch Design**
   
   f.  **Add and Run Punching Checks**

5.  Move to next level or sub-model and repeat step 4.

6.  **Create Drawings and Quantity Estimations**

7.  **Print Calculations**

**Flat slab design example**

A simple flat slab model as shown below is used in order to demonstrate the techniques involved in the slab design process.
Note that there is a transfer level at the first floor:

**Set up Pattern Loading**

If necessary you should consider manually splitting and joining slab panels to facilitate management of the pattern loading process.
By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to meshed slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via **Update Load Patterns** on the Load ribbon.

![Diagram](image)

**Design All**

Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either **Design Concrete** or **Design All** are run.

Typically these moments are taken from the FE chasedown model results - as each floor is analysed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Building Analysis model results will also be considered.

---

*It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however:*

- you may choose to mesh them to cater for the possibility of un-braced flat slab design.
- more likely, you may do so to deal with significant transfer slabs - i.e. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

---

**Consider Deflection (for Flat slabs)**
Slab deflections are obtained by reviewing the 2D deflection contours for the FE Chasedown results in the Results View. Corresponding deflections for the 3D Building Analysis will only be available if you have elected to mesh 2-way slabs in the analysis.

By viewing the deflection results for combinations based on ‘service’ rather than ‘strength’ factors the stiffness adjustments that you apply do not need to account for load factors.

The default adjustments are dependent on the design code. For design to EC2 the default adjustment factor applied is 0.2.

The default adjustment factors can be edited from the Analyse ribbon by selecting Options > Modification Factors > Concrete.

Span/Relative Deflection ratios should be determined between appropriate points in the slab in order to check the slab thickness is sufficient.

**Select a Level**

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the ‘3D geometry’ of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

---

**Add Patches**

This is an interactive process - requiring a certain amount of engineering judgement.

---
Typically you should expect to work on this one floor at a time, (making use of multiple views when creating patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mx moments on in one view on the left and My moments in a second view on the right, as below:

By doing this, it is possible to see how patches extend over the peaks.

Typically, at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimisation stage.

In a ‘slab on beam’ situation, you may want to add beam and wall patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to ‘none’ and the panel design should still pass.

**Design Panels**

*Panel design is dependent on the areas of patches (patch areas which are excluded from panel design) - hence patches should be added before panels are designed.*

To design multiple slab panels, either:

1. From the Design ribbon run **Design Slabs** in order to design or check all the panels in the model - by default newly created panels will all be in ‘auto-design’ mode - so reinforcement is selected automatically.

   or

2. In the 2D View of the floor which you want to design right click and choose either **Design Slabs** or **Check Slabs**. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large
buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.

These right click options operate on the same basis as the options for beams and columns:
- **Design Slabs** will re-design the slabs (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Slabs** will check the current reinforcement in the slabs regardless of the current autodesign setting.

**Review/Optimise Panel Design**

Once again it is suggested that you use split **Review Views** to examine the results as indicated below.

The view on left shows **Slab Design Status**, the view on right shows **Slab Reinforcement** in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a flat slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right click option to design panels unless you really want to.).

**Auto-design will select the smallest allowable bars at the minimum centres necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a ‘minimum spacing (slab auto design)’ = 150mm.**
After using the **Review View** update mode to standardise reinforcement you can then run **Check Slabs** from the right click menu to check the revised reinforcement.

**Remember:**
- Consider swapping between **Status** and **Ratio** views - if utilisations all < 1 but some panels failing then problems are to do with limit checks. The **Ratio** view is better for helping you focus on the critical panels.
- During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

**Design Patches**

Having established and rationalied the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

1. From the Design ribbon run **Design Patches** in order to design or check all the patches in the model - by default newly created patches will all be in ‘auto-design' mode - so reinforcement is selected automatically.

   or

2. In the 2D View of the floor which you want to design right click and choose either **Design Patches** or **Check Patches**. Working in this way restricts the design or checking to the patches in the current view.

---

*These right click options operate on the same basis as the options for beams and columns:*

- **Design Patches** will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Patches** will check the current reinforcement in the patches regardless of the current autodesign setting.

---

**Review/Optimise Patch Design**

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimised)
- Wall patches - can the width be adjusted (minimised)
- Column patches - Is the size reasonable - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
• Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.

• Now click **Slab Reinforcement** in the **Review View** to review and standardise the patch reinforcement. For instance in column patches this might include forcing the spacing of the slab reinforcement to be matched (if the slab has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

**Add and Run Punching Checks**

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added over the entire floor, or structure by windowing it. You can then select any check and review the properties assigned to it. Internal/edge/corner locations are automatically determined (with a user override if you require). Once added click **Design Punching Shear** and the checks are done and status is shown as:

• Pass - if no shear reinforcement is needed
• Warning - if shear reinforcement is needed
• Fail - if it is impossible to achieve required capacity by adding share reinforcement
• Unknown - if check not run yet
• Beyond scope or error - if for example the centroid of the column/wall lies outside the slab

**Create Drawings and Quantity Estimations**

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

**Print Calculations**

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).
Typical slab on beams design procedure

In *Tekla Structural Designer* an interactive design approach is required for 2-way spanning slab on beam design because the patch and panel design are inter-dependant.

---

*Design of slab panels that have their decomposition property specified as ‘one-way’ is beyond scope - see: Concrete slab load decomposition*

---

The below example illustrates the design approach, which typically consists of the following steps:

1. **Set up Pattern Loading**
2. **Design All** - to establish analysis results
3. **Select a Level** (or sub-model) to be designed and within that level:
   a. **Add Beam and Wall Top Patches**
   b. **Design Panels**
   c. **Review/Optimise Panel Design**
   d. **Design Beam and Wall Patches**
   e. **Review/Optimise Beam and Wall Patch Design**
4. Move to next level or sub-model and repeat step 4.
5. **Create Drawings and Quantity Estimations**
6. **Print Calculations**

Slab on beam design example

A simple slab on beam model as shown below is used in order to demonstrate the techniques involved in the slab design process.
Note that not all the slab panels are rectangular.

**Set up Pattern Loading**

By default, only beam loads, and slab loads that have been decomposed on to beams, are patterned. Loads applied to meshed slabs should be manually patterned using engineering judgement; this is achieved on a per panel basis for each pattern load by toggling the loading status via **Update Load Patterns** on the Load ribbon.
Analysis is required to establish the moments to be used in the slab design - this analysis is automatically performed when either Design Concrete or Design All are run.

Typically these moments are taken from the FE Chasedown model results - as each floor is analysed individually this method mimics the traditional design approach.

If you have elected to mesh 2-way slabs in the analysis, the 3D Building Analysis model results will also be considered.

> It is NOT suggested that it should be standard practice to mesh 2-way slabs, for slabs that are not needed to participate in the lateral load analysis it is better not to mesh in 3D, however:
> - you may choose to mesh them to cater for the possibility of un-braced flat slab design.
> - more likely, you may do so to deal with significant transfer slabs - i.e. a shear wall supported by a slab, (there is no need to create a system of fictitious beams).

Select a Level

In models with distinct floor levels you should use 2D Views to work on the floor design one level at a time. Occasionally the ‘3D geometry’ of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

> When working in a 2D View use the right click menu to design or check slabs and patches: this saves time as only the slabs and patches in the current level are considered.

> Running Design Slabs or Design Patches from the Design ribbon will take longer as it considers all the slabs and patches in the model.

Add Beam and Wall Top Patches

You may optionally want to add beam and wall ‘top surface’ patches to cover the full extent of non-zero top bending moments. By doing so you can then set the top reinforcement in the slab panels to ‘none’ and the panel design should still pass.

This is an interactive process - requiring a certain amount of engineering judgement.

Typically you should expect to work on this one floor at a time, (making use of multiple views when creating beam patches as discussed below).

It is suggested that you add patches in the Results View while looking at relevant moment contours - For example you might (after selecting the FE chasedown result type) have Mdx top moments on in one view on the left and Mdy top moments in a second view on the right, as below:
By doing this, it is possible to see how patches extend over the moment contours.

It is suggested that at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this should be reviewed/resolved at the patch design optimisation stage.

**Design Panels**

> Slab on beams panel design takes account of any beam or wall patches (by excluding the patch areas from panel design) - hence patches should be added before panels are designed.

To design multiple slab panels, either:

1. From the Design ribbon run **Design Slabs** in order to design or check all the panels in the model - by default newly created panels will all be in ‘auto-design’ mode - so reinforcement is selected automatically.

   or

2. In the 2D View of the floor which you want to design right click and choose either **Design Slabs** or **Check Slabs**. Working in this way restricts the design or checking to the slab panels in the current view. This will be a much faster option on large buildings and avoids time being wasted designing slab panels at levels where patches have not yet been set up.
These right click options operate on the same basis as the options for beams and columns:
- **Design Slabs** will re-design the slabs (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Slabs** will check the current reinforcement in the slabs regardless of the current autodesign setting.

3. If you chose to set the top reinforcement in the slab panels to ‘none’ but the design detects that top reinforcement is required, the affected panels will fail. In this situation you should increase the widths of the adjacent beam or wall patches before checking or designing the slab panels once again.

4. When panels are being designed (as opposed to checked), the design does not currently automatically increase reinforcement to satisfy deflection, in which case the panels might fail. In this situation you could manually increase the reinforcement until deflection is satisfied.

Adding reinforcement to resolve deflection issues can prove effective when designing to BS codes, however it is a less likely to be effective when designing to Eurocodes.

As part of the design process a span-effective depth check is performed in the appropriate span direction. If the span is the same in both directions, the more ‘continuously supported’ direction is checked, or if both are equally supported the direction which is set to be the outer bar layer is checked.

The span-effective depth check takes into account the edge category at the start and end of the span direction being checked, (the program default is to treat each edge as dis-continuous), therefore if the edges are in reality continuously supported, setting them as such will improve the result.

Non-rectangular panels are converted to idealized rectangular ones in order to perform the span-effective depth check calculations. See: Slab on beam idealized panels

**Review/Optimise Panel Design**

**Review Views** can be employed to examine the results and once again it is suggested that you use split views as indicated below.
The view on left shows **Slab Design Status**, (with slab patches turned off in Scene Content to assist clarity), the view on right shows **Slab Reinforcement** in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.

If the selections are unacceptable you may need to review the design settings. e.g - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a slab, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right click option to design panels unless you really want to.).

*Auto-design will select the smallest allowable bars at the minimum centres necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a ‘minimum spacing (slab auto design)’ = 150mm.*

After using the **Review View** update mode to standardise reinforcement you can then run **Check Slabs** from the right click menu to check the revised reinforcement.

Remember:

- Consider swapping between **Status** and **Ratio** views - if utilisations all < 1 but some panels failing then problems are to do with limit checks. The **Ratio** view is better for helping you focus on the critical panels.

**Design Beam and Wall Patches**
Having established and rationalised the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design any beam or wall patches that you may have defined.

To do this, either:

1. From the Design ribbon run **Design Patches** in order to design or check all the patches in the model - by default newly created patches will all be in 'auto-design' mode - so reinforcement is selected automatically.

   or

2. In the 2D View of the floor which you want to design right click and choose either **Design Patches** or **Check Patches**. Working in this way restricts the design or checking to the patches in the current view.

   *These right click options operate on the same basis as the options for beams and columns:*
   - **Design Patches** will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
   - **Check Patches** will check the current reinforcement in the patches regardless of the current autodesign setting.

**Review/Optimise Beam and Wall Patch Design**

At this stage the patch sizes can be reviewed:

- Beam patches - can the width be adjusted (minimised)?
- Wall patches - can the width be adjusted (minimised)?
- Having got the sizes sorted out and the patches re-designed, swap them out of auto-design mode.

- Now click **Slab Reinforcement** in the **Review View** to review and standardise the patch reinforcement.

**Create Drawings and Quantity Estimations**

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

**Print Calculations**

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).
Interactive concrete member design

Interactive concrete beam design

The completely automatic design processes, Design Concrete (Static), Design All (Static) etc. are complemented by the program’s interactive beam design facility. This allows you to interact with the beam design to override the design results arising from the auto-design process.

Generally you are advised to perform interactive member designs only after the Design All process has been carried out. In this way multiple analysis models will have been considered to arrive at the forces being designed for.

The Interactive Beam Design Dialog displays a limited selection of the relevant critical design results including bar details and allows you to make changes to the number, size and spacing (for links only) of the selected bars.

After making changes, you are able to see the effect on the displayed results – you then have the option of cancelling or accepting the changes.

How do I open the Interactive Beam Design Dialog?

1. Right click the member you want to design interactively and select Interactive Design... (Static or RSA as required) from the context menu that is displayed.

   The Interactive Beam Design Dialog opens, displaying results for the existing reinforcement.

Overview of the Interactive Beam Design Dialog

When the dialog is opened, the current reinforcement and check results are shown for each beam in the beam line.

When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of every single change you make to the reinforcement.

The Interactive Beam Design Dialog consists of the following areas:
Span Summary

A tree view displays the design status of each span and the associated utilisation ratio. Click a particular span in the summary to display or edit its design in the tabbed pages.

Longitudinal Bars tab

Bar Selection Table

Used for editing the longitudinal bars into the beam.

- Each row in the table is labelled with a specific ‘bar number’ (taken from the standard patterns applied to the beam in the Properties Window); these represent bar locations within the beam.
- Two different bar sizes can be defined in each row, the only restriction being that the second bar must always be smaller than the first.
- The number of bars of each size is defined using the ‘Count’ field.
- When bars are joined to the adjacent span, changing those bars within this span has the effect of changing those bars in the adjacent span, as they are effectively the same bar. (This is only done when the spans are “matching” in terms of their alignment and dimensions.)

Bar Pattern Layout

This is a schematic diagram representing the top and bottom patterns assigned to the beam.
**Design Summary Table**

The table displays critical results for each of the design regions from all combinations:

- Area of reinforcement required, $A_{s, reqd}$
- Area of reinforcement provided, $A_{s, prov}$
- Reinforcement area utilisation ratio
- Smallest clear spacing between bars
- Minimum required reinforcement area, $A_{s, min}$

**Deflection check**

This checks the actual span: effective depth ratio against the limiting span: effective depth ratio.

**Links tab**

**Link Selection Table**

Specifies the number of link legs, size and spacing in each of the regions.

**Optimise Button**

This calculates the optimum length of the central region given the reinforcement that you have selected. The button is not be visible when the beam is in a design group with other beams, and is also not visible when the span is a cantilever.
Link Design Summary Table
The table displays the most critical result from all combinations:
- Region length
- Link area over spacing required for shear, $A_{sw,reqd/s}$
- Link area over spacing required for torsion, $A_{sw,t,reqd/s}$
- Link area provided, $A_{sw,prov}$
- Link utilisation ratio

Buttons
- **OK**
  Closes the dialog and saves the current design
- **Cancel**
  Closes the dialog without saving changes
- **Check**
  Opens the Results dialog displaying the detailed results for the current design
- **Detail Drawing**
  Creates a detail drawing for the selected member
- **Drawing Options**
  Opens the DXF Export Preferences dialog

**How do I change the bar pattern?**

1. If the Interactive Beam Design Dialog is open, click **Cancel** to close it.
2. If necessary, re-select the beam to be designed.
3. Change the **Top** and **Bottom longitudinal bar pattern** in the **Properties Window** as required.
4. Hover the cursor over the beam until its outline is highlighted, then right click.
5. From the context menu select **Interactive Design...**

The Interactive Beam Design Dialog opens and reinforcement is automatically re-selected for the beam based on the new bar pattern.

**Interactive concrete column design**

The completely automatic design processes Design Concrete (Static), Design All (Static) etc. are complemented by the program’s interactive column design facility. This allows you to interact with the column design to override the design results arising from the auto-design process.
Generally you are advised to perform interactive member designs only after the Design All process has been carried out. In this way multiple analysis models will have been considered to arrive at the forces being designed for.

The Interactive Column Design Dialog displays a limited selection of the relevant critical design results including bar details and allows you to make changes to the number, size and spacing (for ties only) of the selected bars. Interaction diagrams are also displayed for the current design.

After making changes, you are able to see the effect on the displayed results – you then have the option of cancelling or accepting the changes.

How do I open the Interactive Column Design Dialog?

1. Right click the member you want to design interactively and select Interactive Design... (Static or RSA as required) from the context menu that is displayed.

   The Interactive Column Design Dialog opens, displaying results for the existing reinforcement.

Overview of the Interactive Column Design Dialog

When the dialog is opened, the current reinforcement and check results are shown for each stack.

When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of every single change you make to the reinforcement.

The Interactive Column Design Dialog consists of the following areas:
Stack Summary

A tree view displays the design status of each stack and the associated utilisation ratio. Click a particular stack in the summary to display or edit its design in the tabbed pages.

Longitudinal tab

All straight-edged cross sections have "Principal" bars located at shear tie corners. Between these, evenly spaced identical "Intermediate" bars can be located.

Circular sections have 6 or more evenly spaced bars around the edge of the section.

In the first release only one layer of reinforcement against any shear tie edge is permitted.

Principal bar size

Used to change the size of all principal bars (all must have the same size).

Intermediate bar size (not displayed for circular columns)

Used to change the size of all intermediate bars (all must have the same size).

Bar Location Table

Used for adding intermediate bars into the cross-section:

- Int. length - identifies the edge along which the bars are positioned
- Count - for changing the number of intermediate bars along the length
• Ctr spacing - the centreline spacing for the current number of bars along the length
• Status - indicates when the maximum bar spacing limit has been exceeded. (When the minimum bar spacing limit is exceeded this is displayed elsewhere in the Design Summary Table).

**Design Summary Table**
The table displays the most critical result from all combinations:
- Design and Resistance Moments and Moment Ratios
- Axial Force, Axial Resistance and Axial Ratios
- Smallest clear bar spacing
- Minimum area of steel
- Area of steel provided

**Cross-section**
The drawing displays:
- Exact bar positions (drawn to scale)
- Tie locations
- Section dimensions
- Principal bar labels

**Containment status**
This status is determined based on the requirements for bars being tied.

**Links tab**

**Use support region links**
Check the box to design support regions for the links.

**Link spacing**
specifies the link spacing (if support regions are applied two different spacings can be specified)

**Link size**
Used to change the size of link bars (all must have the same size).

**Link Design Summary Table**
The table displays the most critical result from all combinations:
- Link area over spacing required, major
- Link area over spacing required, minor
- Link area over spacing provided
- Link utilisation ratio

**Cross-section**
The drawing displays:
- Exact bar positions (drawn to scale)
- Link locations
- Section dimensions
- Principal bar labels

**Interaction diagram tab**

**N-M Interaction diagram**
Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination
- The curves for bending about the major axis are shown in red
- The curves for bending about the minor axis are shown in blue

**M-M Interaction diagram**
The diagram is different for each value of axial force, so only the diagram at the axial force of the critical combination is drawn - this is the combination with the highest $M_{Ed} / M_{res}$ ratio.

**Buttons**

**OK**
Closes the dialog and saves the current design

**Cancel**
Closes the dialog without saving changes

**Check**
Opens the Results dialog displaying the detailed results for the current design

**Detail Drawing**
Creates a detail drawing for the selected member

**Drawing Options**
Opens the DXF Export Preferences dialog

**How do I arrange bars in the Interactive Column Design Dialog?**

**Circular Columns**
Steel bars are arranged by modifying the bar size and count fields.

**Rectangular and Polyline Columns**

Principal bars exist at fixed locations; they are labelled with numbers in the cross-section. You can only change the principal bar sizes, not their locations.

Intermediate bars are the unnumbered bars in the cross-section. You can change both their size and number. They are defined in the bar location table by reference to the principal bars between which they lie.

<table>
<thead>
<tr>
<th>Int. length</th>
<th>Count</th>
<th>Ctrs spacing [mm]</th>
<th>Int. length</th>
<th>Count</th>
<th>Ctrs spacing [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-2</td>
<td>1</td>
<td>249.0</td>
<td>3-4</td>
<td>1</td>
<td>249.0</td>
</tr>
<tr>
<td>2-3</td>
<td>1</td>
<td>149.0</td>
<td>4-5</td>
<td>1</td>
<td>149.0</td>
</tr>
</tbody>
</table>
A count of ‘1’ for each intermediate length in the bar location table indicates that a single intermediate bar is positioned between each of the principal bars.

If the count was increased to ‘2’ for Int. length 1-2, but reduced to ‘0’ for Int. length 2-3, the following arrangement is achieved.

Note that Int. lengths 3-4 and 4-5 are adjusted automatically in the table to match.

<table>
<thead>
<tr>
<th>Int. length</th>
<th>Count</th>
<th>Ctr spacing [mm]</th>
<th>Int. length</th>
<th>Count</th>
<th>Ctr spacing [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-2</td>
<td>2</td>
<td>166.0</td>
<td>3-4</td>
<td>2</td>
<td>166.0</td>
</tr>
<tr>
<td>2-3</td>
<td>0</td>
<td>293.0</td>
<td>4-5</td>
<td>0</td>
<td>293.0</td>
</tr>
</tbody>
</table>

Link arrangements in rectangular and parallelogram sections have the following basic options:
- Single links,
- Double links,
- Triple links,
- Cross links.

Tie bars are used with these arrangements. Link arrangements in other section shapes use standard link positions with additional tie bars where required.

**Interactive concrete wall design**

The completely automatic design processes, Design Concrete (Static), Design All (Static) etc. are complemented by the program’s interactive wall design facility. This allows you to interact with the wall design to override the design results arising from the auto-design process.

Generally you are advised to perform interactive member designs only after the Design All process has been carried out. In this way multiple analysis models will have been considered to arrive at the forces being designed for.
The **Interactive Wall Design Dialog** displays a limited selection of the relevant critical design results including bar details and allows you to make changes to the number, size and spacing (for links only) of the selected bars. Interaction diagrams are also displayed for the current design.

After making changes, you are able to see the effect on the displayed results – you then have the option of cancelling or accepting the changes.

**How do I open the Interactive Wall Design Dialog?**

1. Right click the member you want to design interactively and select **Interactive Design...** (Static or RSA as required) from the context menu that is displayed.

   The Interactive Wall Design Dialog opens, displaying results for the existing reinforcement.

**Overview of the Interactive Wall Design Dialog**

When the dialog is opened, the current reinforcement and check results are shown for each stack.

When any of the editable fields are changed, the checks are re-run and the results are updated; enabling you to quickly see the effect of every single change you make to the reinforcement.

The **Interactive Wall Design Dialog** consists of the following areas:
A tree view displays the design status of each stack and the associated utilisation ratio. Click a particular stack in the summary to display or edit its design in the tabbed pages.

**Longitudinal tab**

**Use end-zones**
check in order to define end-zones.

**Principal bar size**
Used to change the size of all principal bars (all must have the same size).

**Intermediate bar size (not displayed for circular columns)**
Used to change the size of all intermediate bars (all must have the same size).

**End-zones Bar Location Table**
Used for specifying bars in the ‘end-zones’ when the end-zones option has been activated.
- Length - length of each end-zone
- Number of rows - the number of loose bars in each layer
- Vertical bar size - specifies the size to be checked
- Additional end row bars - the number of loose bars in each end face

**Panel (or Mid-zone) Bar Location Table**
Used for specifying bars in the ‘wall zone’
- Number of layers - (1, or 2)
- Reinforcement type - (loose bars, or mesh)
- Number of rows - the number of loose bars in each layer
- Vertical bar size/Mesh size/End row vertical bar size - specifies the size to be checked
- Additional end row bars - the number of loose bars in each end face

**Design Summary Table**
The table displays the most critical result from all combinations:
- Design and Resistance Moments and Moment Ratios
- Axial Force, Axial Resistance and Axial Ratios
- Smallest clear bar spacing
- Minimum area of steel
- Area of steel provided

**Cross-section**
The drawing displays:
- Exact bar positions (drawn to scale)
• Link locations
• Section dimensions

**Lateral tab**

**Use links**
Check the box to specify links.

**Use support region links**
Check the box to design support regions for the links.

**link spacing**
specifies the link spacing (if support regions are applied two different spacings can be specified)

**Link size**
Used to specify the size of link bars (all must have the same size).

**Horizontal bar size**
Used to specify the size of horizontal bars.

**Horizontal bar spacing**
Used to specify the vertical spacing of horizontal bars.

**Link Design Summary Table**
The table displays the most critical result from all combinations:
• Link area over spacing required, major
• Link area over spacing provided, major
• Link utilisation ratio, major
• Link area over spacing required, minor
• Link area over spacing required, minor
• Link utilisation ratio, minor

**Horizontal Reinforcement Summary Table**

**Cross-section**
The drawing displays:
• Exact bar positions (drawn to scale)
• Link locations
• Section dimensions

**Interaction diagram tab**
**N-M Interaction diagram**
Top, mid fifth and bottom moment results for each analysis type are plotted on the diagram for each combination

- The curves for bending about the major axis are shown in red
- The curves for bending about the minor axis are shown in blue

**M-M Interaction diagram**

The diagram is different for each value of axial force, so only the diagram at the axial force of the critical combination is drawn - this is the combination with the highest $\frac{M_{Ed}}{M_{res}}$ ratio.

**Buttons**

- **OK**
  Closes the dialog and saves the current design

- **Cancel**
  Closes the dialog without saving changes

- **Check**
  Opens the Results dialog displaying the detailed results for the current design

- **Detail Drawing**
  Creates a detail drawing for the selected member

- **Drawing Options**
  Opens the DXF Export Preferences dialog

**Working with large concrete models**

Although bigger models will typically need more RAM, you should note that it is the volume of model data and results data created during analysis and design that generates the demand for RAM, so by controlling the volume of data you are able to influence the speed of solution.

Your modelling and design choices can affect the volume of data produced; some of the more significant of these choices are discussed below:

**Don’t mesh concrete slabs in 3D building analysis**

Meshing is not necessary in 3D building analysis for traditional Beam and Slab models as these can use FE decomposition instead.

It is also not necessarily required for Flat Slab models, however, it does become required if you have a transfer slab. It is not critical unless the slab is part of the lateral resistance system.
By default the ‘Mesh 2-way slabs in 3D analysis’ option that controls this is not checked at any level; you should only check it at a given level when you have good reason to do so.

**Using coarser meshing in large models**

You should review and consider adjusting the Mesh Parameters as the defaults can often be conservative.

- Using coarser mesh parameters has no impact on sway or vibration frequencies.
- If you are not concentrating on slab design you can use very coarse meshes.

Ultimately it is your responsibility to be comfortable with the level of mesh refinement applied to the model. However we would definitely recommend using a courser mesh during design development and then perhaps consider refining a bit more at final design stage.

**Limit the number of load cases and combinations**

You can control the number of combinations created when running the Combination Generator.

In particular you should consider limiting the number of wind load cases and combinations.

Also:

- Don’t add wind loading during the initial design development.
- Don’t activate pattern load cases and combinations until you need to, probably only at the final design stage.

**Alternative design approach for large models**

For large models, rather than running ‘Design All’ you may be able to save time by running ‘Analyse All’ instead and then run a selective design, such as:

- Design by level
- Design by frame
- Design by group
- Design by sub-structure
- Design by member

**Effective use of Auto Design**

Although for the first design run you might choose to use ‘select bars starting from Minima’, on subsequent runs it is generally more efficient to use ‘select bars starting from Current’; this will run a check on the current steel provision and if inadequate, it will automatically re-select new steel bars to pass the design.

Check Design can also be very effective – you can turn off the Autodesign and then manually deal with any fails.
Design members for FE chedown analysis results

This is set in the design options for concrete and by default it is checked on for beams, columns and walls. However in a traditional Beam and Slab model it may not actually be necessary; it is generally not critical unless you have some unusual transfer level challenge. In large models you should therefore consider unchecking it.

Re-design columns (or beams) using previous analysis results

If you change the size of a member you can try out its design without being forced to re-analyse.

Basically you can make any edit you want that does not change the number of stacks in a column, (or spans in a beam); although the analysis results will be marked as out of date, you can still do a design for the changed member based on the old analysis results.

So using the list below as examples design can still be done in all the cases noted:

1. Changing column (or beam) size but retaining shape - designs ok (but see 3 and 4).
2. Changing column (or beam) shape - designs ok (but see 3 and 4).
3. Making column (or beam) smaller so that previously attached members no longer attach - this changes number of stacks - design beyond scope.
4. Making column (or beam) larger so that previously un-attached members now attach - this changes number of stacks (spans) - design beyond scope.
5. Adding / editing / deleting beams that attach to a column - design remains possible up to the point that it affects number of stacks - OK
6. Adding / editing / deleting flat slabs that attach to a column - design remains possible up to the point that it affects number of stacks - OK
7. Adding / deleting stacks or levels (or editing level properties) - Adjusting Levels designs ok because number of stacks is the same. If you add or remove stacks then design is beyond scope.

Model organisation

_Tekla Structural Designer_ has a number of features for organising the model than can each be used to increase efficiency:

- Grouping - one design is applied to all members in the group.
- Sub-structures - allow you to focus on specific areas of interest.
- Duplicate levels - generally save modelling time and reduce the volume of data.

When using duplicate levels, you can achieve further efficiency by designing slabs for a fine mesh at one level only, and then check the slabs at duplicates of the level using a courser mesh result.
Because meshing parameters are ‘sub model’, rather than ‘level’ based, to achieve this you would set coarse mesh parameters in the structure settings but then override them for an individual sub model.

Our general advice for duplicate levels is:

• For preliminary design set a coarse mesh for entire structure
• For final design where there are a lot of duplicate levels
  • possibly refine the mesh used for the entire structure a little
  • but for each set of duplicate levels, select one and adjust the relevant sub-model parameters to get a finer mesh.

Model complexity

Do not model every little architectural detail – especially not things like small holes in slabs and walls.

Design Options

Take control (get it right at the beginning!)
This handbook provides a general overview of Tekla Structural Designer in the context of its application to steel structure design.

Refer to the Reference Guides for details of the specific steel calculations that are performed for each design code.

General design parameters
A number of design parameters are common to the different steel member types - these are described in the topics below.

Material type
The material types supported depend on the code being designed to:
• ‘Steel’ is the only material permitted if designing to AISC 360.
• ‘Steel’ and ‘Cold formed’ materials are both permitted if designing to the Eurocode or BS 5950.

Autodesign (steel)
The design mode for each member is specified in the member properties.

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

When Autodesign is not selected (i.e. check mode), you assign your desired section size to the member and Tekla Structural Designer determines if the section is sufficient.

When Autodesign is selected the section type to be used is specified from a Design Section Order and Tekla Structural Designer attempts to automatically determine a suitable size.

The following controls can be applied to further limit the sections considered:
• Size Constraints
Design Section Order

A design section order is only applicable when Autodesign is checked.

The design process commences by starting with the smallest section in the chosen order file. Any section that fails any of the design conditions is rejected and the design process is then repeated for the next available section in the list.

On completion of the design process, the first satisfactory section from the Section Designation list is assigned to the member.

How do I view the list of sections in a design section order?

1. Edit the properties of the member.
2. Click the Design section order drop list and select <New\Edit>...
3. Choose a section order from the available list and then click Edit...

The sections contained within the chosen order file appear in the Sections in use list on the right of the page.

How do I specify that a section in the list should not be considered for design?

Only checked sections within the list are considered during the design process. Uncheck a section and it will no longer be considered.

Limiting the choice of sections by unchecking a section within an order file is a global change that affects ALL projects, (not just the currently open one). It is typically used therefore to eliminate unavailable or non-preferred sections from the design process. If design requirements for an individual member require section sizes to be constrained, (due to, for example depth restrictions), then the choice of sections should be limited instead by using Size Constraints, (as these only affect the current member).

How do I sort the listed sections by a different property?

While viewing the list of sections you can:

1. Select a property from the Sort By droplist
2. Click Sort to re-order by the chosen property.
Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

**How do I specify that a section is ‘non-preferred’?**

Some sections might be more expensive or difficult to obtain; you might therefore want other sections to be chosen in preference to them, (whilst still keeping them available). You can achieve this by moving the ‘non-preferred’ sections further down the design order list.

To move a section up or down the list:

- Highlight the section in the **Sections in use** list and then click **Move Up** or **Move Down** to promote or demote it.

Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

---

**Size Constraints**

Size Constraints are only applicable when **Autodesign** is checked. They allow you to ensure that the sections that **Tekla Structural Designer** proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm. If so you would simply enter this value as the Minimum width, and **Tekla Structural Designer** would not consider sections with flanges less than this width for the design of this beam.

**Gravity only design**

By checking/unchecking the **Gravity only** option members are set to be:

- Gravity only - designed for gravity combinations and seismic combinations
- Lateral and Gravity - designed for all combinations types - gravity, lateral and seismic

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

Engineering judgement will however be required when identifying members as being ‘gravity only’.

For example:

- if an inclined braced member connects to a simple/composite beam, axial force in the brace (from both gravity and lateral loads) puts the beam into bending and therefore the beam should be designed for both gravity and lateral loads.
potentially, simple beams in a sloping roof would also need to be designed for both gravity and lateral load

If a simple, or composite beam is identified to be designed for both gravity and lateral combinations, only the component of the lateral load that acts in the plane of the strong axis of the member is considered. Any axial loads, or loads in the weak axis are ignored. A warning is provided if the ignored loads exceed a preset limit.

Design groups

Steel members are automatically put into groups, primarily for editing purposes. In this way, individual groups can be selected and displayed graphically so that their properties can be changed as a group in the properties window.

A fixed set of rules are used to determine the automatic member grouping: for example beams must be of similar spans, columns must have the same number of stacks etc. The same rules also constrain manual group editing.

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

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

How is the ‘design using groups option’ activated?

1. Click Design > Options... (📷)
2. Click ‘Design Groups’
3. Check the box adjacent to each member type for which you want to apply design grouping.

What happens in the group design process?

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

• In each design group the first member to be designed is selected arbitrarily. A full design is carried out on this member and the section so obtained is copied to all other members in the group.
• These other members are then checked one by one to verify that the section is adequate for each and if this proves not to be the case, the section is increased as necessary and the revised section copied to all members in the group.

• This process continues until all members in the group have been satisfactorily checked.

• A final check design is then carried out on each group member. During this process peak and individual utilisations are established.

Instability factor

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

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

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

This multiplier can be applied to steel beams, composite beams and steel columns. It is defined in the properties window by checking the ‘Prevent out of plane instability’ box and then entering a suitable value in the ‘Instability factor’ field.

Steel beam design

Steel beam scope

Tekla Structural Designer allows you to analyse and design a structural steel beam or cantilever which may have incoming beams providing restraint and which may or may not be continuously restrained over any length between restraints. In addition to major axis bending, it also considers minor axis bending and axial loads.

In its simplest form a steel non-composite beam can be a single member between supports to which it is pinned.
It can also be a continuous beam consisting of multiple members that do not, with the exception of the remote ends, transfer moment to the rest of the structure.

Steel non-composite beams that share load with columns form part of a rigid moment resisting frame.

Steel non-composite beams can optionally be set as continuous; in which case all internal connections are considered continuous.

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

- Free end
- Moment connection
- Pin connection
- Fully fixed

Conditions of restraint can be defined in- and out-of-plane for compression buckling and top and bottom flange for lateral torsional buckling (LTB). It is upon these that the buckling checks are based.

A full range of strength and buckling checks are available. As mentioned above the buckling lengths are based on the restraints along the member. The effective lengths to use in the checks depend on the type of restraint, particularly at supports.

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

Each span of a continuous beam can be of different section size, type and grade. The entire beam can be set to automatic design or check design.

In check design mode web openings can be added and designed for.

**Steel beam limitations and assumptions**

The following limitations apply:

- continuous beams (more than one span) must be co-linear in the plane of the web within a small tolerance (sloping in elevation is allowed),
- only doubly symmetric prismatic sections (that is rolled or plated I- and H-sections), doubly symmetric hollow sections (i.e. SHS, RHS and CHS) and channel sections are fully designed,

  Westok cellular beams are excluded,
  - Fabsec beams (with or without openings) are excluded,

The following assumptions apply:
Steel Design Handbook

- All supports are considered to provide torsional restraint, that is lateral restraint to both flanges. This cannot be changed. It is assumed that a beam that is continuous through the web of a supporting beam or column together with its substantial moment resisting end plate connections is able to provide such restraint.
- If, at the support, the beam oversails the supporting beam or column then the detail is assumed to be such that the bottom flange of the beam is well connected to the supporting member and, as a minimum, has torsional stiffeners provided at the support.
- In the Tekla Structural Designer model, when not at supports, coincident restraints to both flanges are assumed when one or more members frame into the web of the beam at a particular position and the cardinal point of the centre-line model of the beam lies in the web. Otherwise, only a top flange or bottom flange restraint is assumed. Should you judge the actual restraint provided by the in-coming members to be different from what has been assumed, you have the flexibility to edit the restraints as required.
- Intermediate lateral restraints to the top or bottom flange are assumed to be capable of transferring the restraining forces back to an appropriate system of bracing or suitably rigid part of the structure.
- It is assumed that you will make a rational and ‘correct’ choice for the effective lengths between restraints for both LTB and compression buckling. The default value for the effective length factor of 1.0 may be neither correct nor safe.

Steel beam design properties

Fabrication
The following fabrication options are available:

Rolled
- A wide range of doubly symmetric rolled sections can be designed.

Plated
You can add your own plated sections and these can then be designed.

Westok cellular
- Design of Westok cellular beams is currently beyond scope.

Westok plated
- You can add Westok plated sections and these can then be designed.

Fabsec
- Design of Fabsec beams is currently beyond scope.
**Section**

Tekla Structural Designer will design steel non-composite beams for an international range of doubly symmetric I-sections, C-sections, rectangular and square hollow sections for many different countries and also for many specific manufacturers.

**Restraints**

Lateral and Strut restraints are determined from the incoming members described within the Tekla Structural Designer. The buckling checks are based on these.

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

**Continuously Restrained Flanges**

In the Properties Window you can independently set both the top and bottom flanges as continuously restrained.

By setting ‘Top flange cont. rest.’ and/or ‘Bottom flange cont. rest.’ to ‘Yes’ the relevant buckling checks are not performed during the design process.

**Web Openings to SCI P355**

When the Head Code has been set to Eurocode, Tekla Structural Designer adopts the following approach to web openings which is specific to the UK National Annex.

(The checking of beams with web openings to US codes is currently ‘Beyond Scope’.)

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

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

Openings can not be defined from the Properties Window, they can only be defined from the Properties Dialog, (by right clicking on the member and selecting Edit...)

As each web opening is added it is checked against certain geometric and proximity recommendations taken from Table 2.1 of SCI Publication P355 (see below).

**Guidance on size and positioning of openings**
The following general guidance on size and positioning of openings is taken from Table 2.1 Section 2.6 of the SCI Publication P355

These geometric limits should normally be observed when providing openings in the webs of beams. It should be noted that these limits relate specifically to composite beams and caution should be used in applying these limits to non-composite beams.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Circular Opening</td>
</tr>
<tr>
<td>Max. depth of opening:</td>
<td>&lt;= 0.8h</td>
</tr>
<tr>
<td>Min. depth of Tee,</td>
<td>&gt;= t_r + 30 mm</td>
</tr>
<tr>
<td>Min. depth of Top Tee:</td>
<td>As above</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Max. ratio of depth of Tees: h_b/h_t</td>
<td>&lt;= 3</td>
</tr>
<tr>
<td></td>
<td>&gt;= 0.5</td>
</tr>
<tr>
<td>Max. unstiffened opening length, l_o</td>
<td>-</td>
</tr>
<tr>
<td>Max. stiffened opening length, l_o</td>
<td>-</td>
</tr>
<tr>
<td>Min. width of web post:</td>
<td>&gt;= 0.3h_o</td>
</tr>
<tr>
<td>- Low shear regions</td>
<td>&gt;= 0.4h_o</td>
</tr>
<tr>
<td>- High shear regions</td>
<td></td>
</tr>
<tr>
<td>Corner radius of rectangular openings:</td>
<td>-</td>
</tr>
<tr>
<td>Min. width of end post, s_e:</td>
<td>&gt;= 0.5h_o</td>
</tr>
<tr>
<td>Min. horizontal distance to point load:</td>
<td>&gt;= 0.5h_o</td>
</tr>
<tr>
<td>- no stiffeners</td>
<td>&gt;= 0.5h_o</td>
</tr>
<tr>
<td>- with stiffeners</td>
<td>&gt;= 0.25h_o</td>
</tr>
</tbody>
</table>
* A high shear region is where the design shear force is greater than half the maximum value of design shear force acting on the beam.

Symbols used in the above table:

- \( h \) = overall depth of steel section
- \( h_o \) = depth of opening [diameter for circular openings]
- \( h_t \) = overall depth of upper Tee [including flange]
- \( h_b \) = overall depth of lower Tee [including flange]
- \( l_o \) = (clear) length of opening [diameter for circular openings]
- \( s_e \) = width of end post [minimum clear distance between opening and support]
- \( t_f \) = thickness of flange
- \( t_w \) = thickness of web
- \( r_o \) = corner radius of opening

In addition, the following fundamental geometric requirements must be satisfied.

\[
\begin{align*}
d_o &\leq 0.8*h \text{ for circular openings} \\
d_o &\leq 0.7*h \text{ for rectangular openings} \\
d_o &< 2 * (d_{oc} - t_t - r_t) \\
d_o &< 2 * (h - d_{oc} - t_b - r_b) \\
d_2 &< d_{oc} - d_o/2 - t_t - t/2 \\
d_2 &< h - t_b - d_{oc} - d_o - t/2 \\
l_o &< 2 * L_c \\
l_o &< 2 * (L - L_c) \\
L_s &< 2 * L_c \\
L_s &< 2 * (L - L_c)
\end{align*}
\]

where

- \( d_o \) = the depth of the web of the upper tee section measured from the underside of the top flange
- \( d_{oc} \) = the distance to the centre line of the opening from the top of the steel section
- \( d_2 \) = the distance from the edge of the opening to the centre line of the stiffener
- \( t_s \) = thickness of stiffener [constrained to be the same top and bottom]
- \( t_t \) = the thickness of the top flange of the steel section
- \( t_b \) = the thickness of the bottom flange of the steel section
- \( r_t \) = root radius at the top of the steel section
- \( r_b \) = root radius at the bottom of the steel section
- \( L_c \) = the distance to the centre line of the opening from the left hand support
- \( L \) = the span of the beam
Dimensional checks - The program does not check that openings are positioned in the best position (between 1/5 and 1/3 length for udl's and in a low shear zone for point loads). This is because for anything other than simple loading the best position becomes a question of engineering judgment or is pre-defined by the service runs.

Adjustment to deflections - The calculated deflections are adjusted to allow for the web openings

Deflection Limits

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

Deflections limits allow you to control the amount of deflection in both composite beams and steel beams by applying either a relative or absolute limit to the deflection under different loading conditions.

A typical application of these settings might be:

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

Camber

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

The amount of camber can be specified either:

• As a value
• As a proportion of span
• As a proportion of dead load deflection
• If this option is selected, the engineer should identify the combination to be used for the calculations by checking the ‘Camber’ box adjacent to the appropriate gravity combination on the Loading Dialog Combinations Page.
• If no combination is selected then the first gravity combination in the combination list is used.

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

A lower limit can be set below which the calculated camber is not applied, this ensures that impractical levels of camber are not specified.

**Natural frequency**

A natural frequency check can optionally be requested. When activated a simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse particularly for long span beams and does not consider the response side of the behaviour i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the designer has the option to perform a 1st Order Vibration Analysis.

**Seismic**

*Design of members in seismic force resisting systems is beyond the scope of the current release.*

**Composite beam design**

**Composite beam scope**

*Tekla Structural Designer* allows you to analyse and design steel beams acting compositely with concrete slabs created using profile steel decking.

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

The following are beyond scope:
  • continuous or fixed ended composite beams,
  • composite sections formed from hollow rolled sections,
  • composite sections where the concrete slab bears on the bottom flange.
  • the use of fibre reinforcement
Beams are designed for gravity loads acting through the web only. Minor axis bending and axial loads are not considered.

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

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

_Tekla Structural Designer_ will determine the size of beam which:

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

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

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

In check design mode web openings can be added and designed for

**Composite beam loading**

All loads must be positive since the beam is considered as simply supported and no negative moment effects are accommodated.

**Construction stage loading**

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

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

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

Having created the loadcases to be used at construction stage, you then include them, together with the appropriate factors in the dedicated Construction stage design combination. You can include or exclude the self-weight of the beam from this combination and you can define the load factors that apply to the self weight and to each loadcase in the combination.

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

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

Composite stage loading

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

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

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

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

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

You can define concrete slabs in both normal and lightweight concrete.

For design to EC4 you must comply with the following constraints:

• the slab depth must be between 90 and 500 mm,
• Normal concrete range C20/25 - C60/75 - See EC4 Clause 3.1(2),
• Lightweight concrete range LC20/22 - LC60/66 - See EC4 Clause 3.1(2),
• Minimum density for lightweight concrete 1750 kg/m³ - see EC4 Clause 6.6.3.1(1).

Concrete properties are obtained by reference to EN 1992-1-1, 3.1 for normal concrete and EN 1992-1-1, 11.3 for lightweight concrete. If normal concrete is specified, you are required to specify the type of aggregate used as this influences the value of Elastic Modulus.

The default concrete densities are as follows:

• normal concrete, wet - 2600 kg/m³,
• normal concrete, dry - 2500 kg/m³,
• lightweight concrete, wet - 2150 kg/m³,
• lightweight concrete, dry - 2050 kg/m³,

The default densities above allow for 0.5kN/m³ reinforcement; the wet densities also allow 1kN/m³ for water. The dry density of unreinforced concrete is taken from BS EN 1991-1-1 Annex A.

Precast concrete planks

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

General limitations and assumptions

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

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

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

A balanced condition is assumed during the construction stage and the top flange of the beam is treated as laterally restrained in construction. This condition should be evaluated against the particular application, if it is not suitable then it should be unchecked.
Both hollow core units and solid planks are assumed to act compositely only with perpendicular secondary beams and not primary beams parallel to the span of the PC units. Beams neither parallel nor perpendicular to the PC slab are termed angled and are also designed non-compositely.

**Precast unit**

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

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

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

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

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

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

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

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

**Steel Beam**

The following applies to steel beam sections:

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

  These recommendations can be reduced by decreasing the bearing - special provisions must be made after consultation with both the precast manufacturer and the steelwork providers*
- Web openings are ignored in design*
- No significant point loads are applied to the composite beam*
• Beam must not behave as a cantilever
• For solid precast units only the topping is to be included as the joints between the units may not be in good contact
• For hollow core units only the precast plank will be taken into account during design
  *A warning is issued in this case and subsequent design is carried out assuming that the engineer has justified the particular condition as safe.

**Bearing**

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

**Concrete properties**

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

**Loading**

**Slab self-weight**

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

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

**Significant Point Loads**

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

**Shear Connectors**

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

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

**Longitudinal Shear**

It is assumed the shear force is divided equally between the two sides of the beam flange.
The factors that influence the longitudinal shear capacity of your composite beam are:

• Concrete strength, slab depth and slab width – you cannot change these independently for the longitudinal shear check, since they apply equally to the entire composite beam design,

• The areas of Transverse and Other reinforcement which you provide in your beam

**Transverse**

Transverse reinforcement is designed to ensure $V_{Ed} \leq V_{Rd}$. Additional reinforcement to that detailed in design may be required for other purposes.

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

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

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

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

**Other**

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

**Composite Moment of Inertia**

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

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

**Composite beam design properties**

**Properties common to composite and non-composite beams**

The related topic links below describe those properties that are shared by composite and non-composite beams.

**Allow non-composite design**
Typically, at the outset you will know which beams are to be non-composite and which are to be composite and you will have specified the construction type accordingly. However, circumstances can arise in which a beam initially intended to be composite proves to be ineffective. Examples might be:

- very small beams,
- beams with a significant point load close to a support,
- beams where the deck is at a shallow angle to the beam, hence the stud spacing is impractical,
- beams where, for a variety of reasons, it is not possible to provide an adequate number of studs, and
- edge beams, where the advantages of composite design (e.g. reduced depth) are not so clear

Where *Tekla Structural Designer* is unable to find a section size which works compositely, you can ask for a non-composite design for the same loading. You will find that this facility is particularly useful when you right click on a key beam in the model in order to perform an individual member design.

**To invoke non-composite design:**

1. Select the composite beam(s) as required.
2. In the Properties Window check the ‘Allow non-composite design’ box.

**Restraints**

In the Properties Window you can independently set both the top and bottom flanges of a composite beam as continuously restrained.

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

The defaults are:

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

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

When not continuously restrained, Lateral and Strut restraints are determined from the incoming members described within the *Tekla Structural Designer*. The buckling checks are based on these.
By right clicking a member to edit its properties in the Property dialog, you are then able to edit the restraints. You can indicate continuously restrained sub-beams and also edit length factors.

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

**Floor construction**

**Deck Type, Angle and Condition**

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

- When specifying the slab item properties you will find that a wide range of profiled metal decks have been included for manufacturers from many countries. PC Planks are also available, but only for the EC Head Code.
- The slab item's rotation angle relative to the global X axis is used to set the profiled metal deck as spanning at any angle between 0° (parallel) and 90° (perpendicular) to the direction of span of the steel beam.

The beam's ‘condition’ is:

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

**Shear connector type**

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

\[ The \text{ use of channel connectors or Hilti™ connectors is currently beyond the program scope. } \]

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

**Effective Width**

The effective width can either be entered directly, or to have it calculated automatically proceed as follows:
1. First click on 'Calculate effective width' in the properties.
2. Then click the [...] box that appears next to 'Calculate'.

For details see: Effective width calculations

Effective width calculations

Checking the effective width used in the design

Tekla Structural Designer will calculate the effective width of the compression flange, \( b_{eff} \), for each composite beam as per Clause 5.4.1.2 of EC4.

It is taken as the smaller of:

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

These effective breadths are used in both strength and serviceability calculations.

Although the program calculates \( b_{eff} \), it is your responsibility to accept the calculated figure or alternatively to adjust it. Engineering judgement may sometimes be required.

For example consider the beam highlighted below:

The program calculates the effective width as the sum of:

- to the right of the beam, \( b_{eff(right)} = \) beam span/8
- to the left of the beam, \( b_{eff(left)} = \) one half of the shortest distance to the centreline of the adjacent diagonal beam

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

Minimum lap distance

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

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

Stud strength

The stud properties you can choose from are appropriate to the stud source.

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

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

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

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

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

Optimize shear interaction

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

Further details of stud optimization and the partial interaction rules are provided in the Steel Design Reference Guides. Refer to the Composite Beam Section in the guide appropriate to the code being designed to.

Transverse reinforcement

This reinforcement is be provided specifically to resist longitudinal shear.
Since the profile metal decking can be perpendicular, parallel or at any other angle to
the supporting beam the following assumptions have been made:

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

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

**Automatic transverse shear reinforcement design**

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

> The *Auto-select* option for designing transverse shear reinforcement is only
available when the beam is in auto-design mode.
If you are checking a beam, then you must specify the transverse shear
reinforcement that you will provide, and then check out this arrangement.

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

**Bar spacing as a multiple of stud spacing**

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

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

**Controlling the bar spacing directly**

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

**Connector layout**

When running in Auto-design mode you may not want to specify the stud layout at the
start of the design process. To work in this way check **Auto-layout** to have the program
automatically control how the stud design will proceed. When the beam is subsequently
designed **Auto-layout** invokes an automatic calculation of the required number of
studs, which is optimized to provide an efficient design.
'Auto layout' can actually be checked regardless of whether you are auto designing the beam size or checking it. The combination of 'Check' design with 'Auto layout' of studs can be used to assist you to rationalise your designs e.g. to force a beam to be the same size as others in the building but have Tekla Structural Designer determine the most efficient layout of studs.

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

**Auto-layout for Perpendicular decks**

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

**Uniform**

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

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

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

**Example:**

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

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

**Non-uniform**
If optimization has been checked (see [Optimize shear interaction](#)) studs are placed at suitable rib intervals (every rib, alternate ribs, every third rib etc.), in order to achieve sufficient interaction without falling below the minimum allowed by the code.

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

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

In this example the point of maximum moment occurs one third of the way along the span, this results in an asymmetric layout. If you prefer to avoid such arrangements you can check the box **Adjust layout to ensure symmetrical about centerline**. A redesign would then result in the symmetric layout shown below.

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

**Auto-layout for Parallel decks**

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

**Uniform**

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

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

**Non-uniform**
Engineers Handbooks (EC)

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

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

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

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

Manual Stud Layout

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

If you specify the stud spacing manually, then it is most important to note:
- the resulting design may not be the optimal design possible for the beam, or
- composite design may not be possible for the stud spacing which you have specified.

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

Manual layout for Perpendicular decks

For perpendicular decks, the dialog for manual layouts is as shown:
To use Quick layout, proceed in one of two ways:

- Choose to position groups in either every rib, or alternate ribs, then specify the number of studs required in the group and click **Generate**.

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

To use the Layout table:

- For each segment you should define the following parameters: **No. of connectors in length** and **No. of connectors in group; Group spacing x rib**.
• Your input for these parameters is used to automatically determine **Distance end 2** - this latter parameter can not be adjusted directly, hence it is greyed out.

• If required click **Insert** to divide the beam into additional segments. (Similarly **Delete** will remove segments). You can then specify a different stud layout for each segment.

• We would advise that having entered No. of studs in length, group and spacing and ignoring Distance ends 1 and 2 you click **Update**, this will automatically fill in the missing fields.

**Manual layout for Parallel decks**

For parallel decks, the dialog for manual layouts is as shown:
To use **Quick layout**, proceed in one of two ways:

- Choose to position groups at a set repeat distance, then specify the number of studs required in the group and click **Generate**.
- Alternatively: specify the total number of studs, then click **Generate** - the program calculates the repeat distance automatically, subject to the code limits.

To use the **Layout** table:

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

### Steel column design

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

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

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

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

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

Web openings cannot currently be designed for.

Limitations for sloping columns

The following limitations apply:

• the web of each stack of a sloping column must lie in the same plane,
• sloping general columns are limited to having either their web, or flanges in a vertical plane.
• eccentricity moments are not taken into account in design,
• there is no imposed load reduction.

Steel column design properties

For design purposes, in addition to the General design parameters, the column properties listed below should be considered:

Simple Columns

A steel column can be designated as a 'simple column' - in which case specific design rules are required.

A simple column should not have any applied loading in its length. Simple columns are subject to axial forces and moments due to eccentricity of beam reactions. In order to prevent end fixity moments you would have to manually pin the ends of the column.
The simple column design rules have not yet been implemented in Tekla Structural Designer: such columns are thus classed as ‘beyond scope’ when designed.

Section

Tekla Structural Designer can handle design for an international range of steel I-sections and also rectangular, square and circular hollow sections for many different countries and also for many specific manufacturers.

Restraints

Restraints to flexural and torsional buckling are determined from the incoming members described within the Tekla Structural Designer. The buckling checks are based on these.

Members framing into either Face A or C will provide restraint against major axis strut buckling. Members framing into either Face B or D will provide restraint against minor axis strut buckling. Tekla Structural Designer determines the buckling restraints and you cannot edit these.

You have complete control on the settings for the lateral torsional buckling (LTB) restraints to the flanges on Faces A and C. The default is blank so you are forced to decide whether a particular configuration of incoming members can provide LTB restraint.

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

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

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

To cater for additional floors that are carried by the column that have not been included in the model an ‘Assume extra floors supported’ column property is provided. This allows you to specify how many extra floors are carried by the column. These are then taken into account when determining any reduction percentage to apply.

Reductions are only applied to those imposed load cases that have had the Reductions box checked on the Loading dialog. The reduction percentage for the number of floors carried is shown in Model Settings.

The load reduction is only applied to vertical columns. It is not applied to inclined columns.

The floors that define the stacks can be designated either as ‘to be’ or ‘not to be’ included in the determination of the imposed load reductions through ‘Count floor as supported’ check boxes for each level in the column properties. This feature enables what appears to be a roof to be counted as a floor, or conversely allows a mezzanine floor to be excluded from the number of floors considered for any particular general column. The moments from fixed ended beams framing into a column are never reduced.

Splice and Splice offset

Splices are allowed at floor levels only and must be placed at changes of angle between two adjacent stacks and at changes of section size or type. A validation error will result if this is not the case. The splice can be given an offset from the floor level - the default of 500mm is considered not to be structurally significant. You must detail the splice to resist the applied forces and moments. The detail should provide continuity of stiffness and strength. Splices given considerable offset should take account of the P-δ moment at the position as well as the forces from the analysis.

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

Column Splice Report

Column splice connections are not designed in Tekla Structural Designer, however the steel column splice loads can be output to a report if required.

In this report factored forces are output at the top and bottom of each lift for every column in which a splice has been defined.

The splice offset from the floor level is taken into account when calculating these forces.
Eccentricity moments are also reported. These are determined using the vertical end reactions of incoming beams at each level applied at the connection eccentricities that have been specified in the column properties dialog. Again, the splice offset from the floor level is taken into account when calculating the eccentricity moments.

The below example illustrates the effect of the splice offset. The lower splice has been offset by 500mm, so the (Lift 1 Top and Lift 2 Bottom) loads are reported 500mm above the 1st floor level. The higher splice has not been offset, so the (Lift 1 Top and Lift 2 Bottom) loads are reported at the 2nd floor level.

**Web Openings**

In the current release of the program the design or checking of columns with web openings is 'Beyond Scope'.

If you need to provide access for services, etc., then you can add openings to a designed column and then check them.
You can define rectangular or circular openings and these can be stiffened on one, or on both sides.

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

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

The 'Quick-layout' process, which is activated using the check box on the Web openings dialog page, adds web openings which meet certain geometric and proximity recommendations taken from Table 2.1 of SCI Publication P355. The openings so created are the maximum depth spaced at the minimum centres recommended for the section size.

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

Seismic

Design of members in seismic force resisting systems is beyond the scope of the current release.

Steel brace design

Steel brace scope

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

Applied loading

The following points should be noted:

• Loads for the brace are derived from the building model.
• Element loads can not be applied directly to the brace itself.
• Imposed load reductions are not applied.
• Moments due to self weight loading are ignored.

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

**Input method for A and V Braces**

A and V Braces should be modelled 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 EHF (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 EHF are not correct.

**Steel brace design properties**

For design purposes, in addition to the General design parameters, the brace properties listed below should be considered:

**Section**

The design of steel braces is carried out for rolled I-sections, C-sections, T-sections, rectangular, square and circular hollow sections, angles, double angles, and flat sections.

**Compression**

Effective length factors are defined for each axis of buckling.

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

**Tension**

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

- Percentage value
- Effective net area
Steel truss design

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

Truss members can either be defined manually, or the process can be automated using the Truss Wizard. Irrespective of the method used the resulting truss members will be one of four types:
- Internal
- Side
- Bottom
- Top
Depending on the type, different design procedures are adopted.

Internal and Side Truss Members
The scope for internal and side truss members is the same as that for braces.
See:

Top and Bottom Truss Members
The scope for top and bottom truss members are the same as those for beams, with the exception that seismic forces are not designed for.
See:

Steel truss design properties
Depending on the truss member type, different design properties are adopted.

Internal and Side Truss Members
The properties for internal and side truss members is the same as that for braces.
See:

Top and Bottom Truss Members
The properties for top and bottom truss members are the same as those for beams.
See:
Steel joist design

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 can not support any other member.

Slab and roof loads are supported by steel joists and loads are distributed to them.

Standard types

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

- K series joists - open web, parallel chord steel joists - depths 8" to 30" with spans up to 60ft.
- 2.5 K series joist substitutes - a depth of 2.5in, intended to be used for spans up to 10ft.
- KCS series joists - K series adapted and specially designed for constant moment/shear along length (position of point loads become irrelevant).
- LH series joists - long span joists - depths 18" to 48" for clear spans up to 96ft.
- DLH series joists - deep long span joists - depths 52" to 72" for clear spans up to 144ft.

Special Joists

"SP" suffixes can be added to K, LH and DLH Series joists. Special Joists can handle 'non-uniform' loading situations. They will attract loads and participate in the 3D structural analysis, but they can not be checked or designed. Load diagrams for the relevant joist can be output to forward to the fabricator for designing.

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

**Joist Analytical Properties**

Steel joists must be simply supported and cantilever ends can not be defined. They cannot be released axially.

Only Joist Girders and SP joists are able to support members along their length.

The inertia and area values are taken directly from the Steel Joist Institute tables.

---

**Performing steel structure design**

*During the design process every steel member in the model will either be designed automatically, or checked - depending on their individual Autodesign (steel) settings. You should therefore ensure the Autodesign property has been set correctly before commencing.*

**Gravity design**

In large models you may prefer to adopt a two-stage design process in which a gravity design is performed in advance of the full design.

The gravity design stage enables you to design or check the simple beams, composite beams and Gravity only design/simple columns for the designated gravity combinations (this will include the Construction Stage combination). Other members will also be designed or checked for these combinations but the resulting section sizes are less useful and are likely to require increasing in later stages of the analysis/design process. This approach is intended to speed up the design process.

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

After the gravity design has been completed, by default all steel members are reset to check design mode. You may therefore decide to reset certain members to auto design e.g. columns and beams in 'moment frames'. In such cases, when the full design is performed member 'pre-sizing' will take place and for members resisting lateral loads this section size will be used if it is larger than that which resulted from the Gravity design.

**Full design**

All beams, columns and braces are designed or checked for all active combinations. (Gravity only design members will be designed or checked for the active gravity combinations).
Full design is initiated by clicking Design Steel (Static), or Design All (Static).

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

**Individual member design**

At the end of the structure design process the **Check Member** command can be used to view detailed results for individual steel members.

If at this stage you want to quickly investigate an alternative section for a specific member, you are able to do so without having to re-perform the entire structure design. Simply change the section size assigned to the member and view the results once more. Alternatively you put the member back into **autodesign** mode, modifying any of its other properties as required before using the **Design Member** command to redesign it in isolation (again without having to re-perform the entire structure design).

> Changing the section size or other properties associated with a member will invalidate the analysis and potentially the design status for the model (displayed in the Status Bar).

**How do I view the design results for the analysed section?**

Provided that the analysis status is valid, the design results are based on the current analysis and can be viewed as follows:

1. Hover the cursor over the member until its outline is highlighted, then right click.

2. From the context menu select **Check Member**

   The results dialog is displayed from where all the detailed calculations can be viewed.

**How do I quickly check an alternative section size?**

> Changing the section size will invalidate the analysis status for the model.

1. Select the member for which you want to view results.

2. Change the section size displayed in the Properties Window to that required.
3. Hover the cursor over the member until its outline is highlighted, then right click.

4. From the context menu select **Check Member**

The results dialog is displayed for the new section.

**How do I quickly design a new section size?**

```
Changing the section size will invalidate the analysis status for the model.
```

1. Select the member that you want to re-design.

2. Review and adjust the member properties as required and ensure the auto-design setting is active.

3. Hover the cursor over the member until its outline is highlighted, then right click.

4. From the context menu select **Design Member**

A new section is selected for the member and then the results dialog is displayed.
This handbook provides a general overview of Tekla Structural Designer in the context of its application to the foundation design.

The following isolated foundations can be designed:

- **Pad base** - an isolated foundation that supports a single column
- **Strip base** - an isolated foundation that supports a single wall
- **Pile cap** - an isolated piled foundation that supports a single column

In addition, the following mat foundations can be designed:

- **Mat foundation** - a foundation supporting multiple columns and walls on ground springs
- **Piled mat foundation** - a foundation supporting multiple columns and walls on pile supports.

**Isolated foundation design**

**Overview of the isolated foundation analysis model**

**Association with member supports**

Columns and walls typically have supports at their bases (apart from transfer columns/walls where the lower end resides on a beam, slab etc.). These supports can be set to be pinned/fixed/sprung or non-linearly sprung (compression/tension only) as required.

At any time during the modelling process, you can define isolated foundations (pad bases, strip bases and pile caps) which are associated with the above mentioned supports.

> Typically pad bases and pile caps can only support and be loaded by a single column, and strip bases can only support and be loaded by a single wall; however, if a ground beam is attached to the same support, loading from the beam will also be considered in the isolated foundation design.

**Analysis types**
Isolated foundations are designed using the results of up to three analysis types:

- 3D Building Analysis
- FE Chase-down Analysis
- Grillage Chase-down Analysis

**Design forces and checks**

**Forces acting on supports**

The following forces and moments on the supports are determined from the analysis of the active load combinations:

- Vertical force in direction Z
- Horizontal forces in directions Y and X
- Moments around X and Y axis

```
A torsional moment around the Z axis is also determined, but the base/pile cap is not designed for this in the current release.
```

**Foundation self weight**

The foundation self-weight is automatically calculated and applied as an added load, $F_{swt}$.

**Soil self weight**

The surcharge depth and soil unit weight that have been specified in the base/pile cap properties are used to determine the soil self weight. This is applied as an added load, $F_{soil}$.

```
In the current release, horizontal pressure caused by soil is not considered.
```

**Additional surcharge loads**

For isolated foundations user can apply additional surcharge loads: acting in the global Z direction.

- Permanent (dead) surcharge load
- Variable (live) surcharge load

**Design Forces**

To obtain the design forces, the loads due to foundation self weight, soil self weight and additional surcharge are added to the forces acting on the supports.
These design forces (axial load and bi-axial shear and moment) are then applied to the base/pile cap at the foundation level.

**Design Checks**

The foundation is treated as being a rigid base resting on the soil working up to a maximum bearing pressure. Tension is not able to develop, but uplift (zero pressure) is allowed.

Bottom reinforcement (both directions) is designed in accordance with the selected design code and a bearing pressure check is performed.

The checks performed as part of the design are as follows:

- bearing pressure check
- design for shear
- design for punching shear
- design for bending

When bases are placed at different levels and close to each other there is a potential risk that the lower base will be affected by the base pressure of the other one. A specific check is made for this and if they are too close a validation warning message is issued.

**Pad base and strip base design procedures**

The overall procedure is demonstrated in the following Pad base design example (but is basically the same for strip bases also).

The typical steps required are as follows:

1. **Apply bases under supported columns**
2. **Auto-size bases individually for loads carried**
3. **Apply grouping to rationalize pad base sizes**
4. **Review/Optimise Base Design**
5. **Create Drawings and Quantity Estimations**
6. **Print Calculations**

**Pad base design example**

The small concrete building model shown below will be used to demonstrate the base design process.
The model has already been designed prior to placing the bases.

**Apply bases under supported columns**

At this stage, as you are not aware of the individual base size and depth requirements; you can simply choose to place the bases where required, accepting the default size/depth offered.
Auto-size bases individually for loads carried

To obtain an idea of the range of potential sizes, bases should initially be designed individually for their respective loads, as follows:

1. Access Design Options to ensure that group design is turned off for Isolated Foundations.

2. Select the bases to be auto-sized and in the Properties Window and choose to auto-design both the size and depth; In this way the program establishes suitable base dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.

3. From the Foundations ribbon click Design Pad Bases.

   Each base will be sized accordingly (any that are not in auto-design mode will simply be checked).
4. With the auto-design options unchecked, you can then adjust individual base dimensions and re-check if required (by right clicking the base that has been edited and choosing **Check Member**).

The site boundary may impose restrictions on the positioning of an isolated foundation relative to the column/wall it supports. This restriction may result in a requirement for an offset base, this can be achieved by specifying the eccentricity required in the base properties.

**Apply grouping to rationalize pad base sizes**

> Grouping can only be applied to pad bases - not to strip bases.

Once pad bases have been sized individually, the designs can be rationalised by activating grouping, in order to obtain one design per group sufficient for all bases within the group.

This is done as follows:

1. Select a base that you want to be in a particular group.

2. In the Properties Window, ensure it is set to auto-design.
3. Right click on the same base and from the context menu choose ‘Create Property Set...’

4. Select all the other bases that you want to be in the same group.

5. In the **Properties Window**, click ‘Apply...’ to apply the property set you have just created to the selected bases.

Ensure you click ‘Apply’ from the **Properties Window** and not from the right click menu, otherwise the properties will only be applied to the last base selected.

6. From the Groups page of the Project Workspace, right click on ‘Pad Bases’ (under the Design branch) and choose ‘Regroup Members’ - this will put those bases that share similar properties into the same group.
7. Open the Design Options dialog, and from the Design Groups page check the option to design isolated foundations using groups.

8. Click Design Pad Bases - the results obtained will reflect the grouping that has been applied.

**Review/Optimise Base Design**

In the **Review View** you can examine the design efficiency by switching from **Foundations Status** to **Foundations Ratio**. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings in **Design Options> Concrete> Foundations**.

**Create Drawings and Quantity Estimations**

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

Create a model report that includes the concrete pad base design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Pile cap design procedures

The overall procedure is demonstrated in the following Pile cap design example.

The typical steps required are as follows:

1. **Apply pile caps under supported columns**
2. **Auto-size pile caps individually for loads carried**
3. **Apply grouping to rationalize pile cap sizes**
4. **Review/Optimise Pile Cap Design**
5. **Create Drawings and Quantity Estimations**
6. **Print Calculations**

Pile cap design example

The small concrete building model shown below will be used to demonstrate the pile cap design process.
The model has already been designed prior to placing the pile caps.

**Apply pile caps under supported columns**

> Before a pile cap can be placed, the Pile Type Catalogue must contain at least one pile type.

At this stage, as you are not aware of the individual pile cap size and depth requirements; you can simply choose to place pile caps where required, accepting the default size/depth offered.

**Auto-size pile caps individually for loads carried**
To obtain an idea of the range of potential sizes, pile caps should initially be designed individually for their respective loads.

Note that when piles are auto-designed the outcome will depend on the auto-design method selected; the pile cap size will either be based on the minimum number of piles required, or on the minimum pile capacity.

To individually size the pile caps:

1. Access Design Options > Concrete > Foundations > Isolated Foundations > Piles to choose the pile auto-design method required: (minimise pile capacity, or minimise number of piles).

2. Still in the Design Options, ensure that group design is turned off for Isolated Foundations.

3. Select the pile caps to be auto-sized and then in the Properties Window choose to auto-design both the piles and depth; In this way the program will establish suitable pile cap dimensions for you.

   Similarly, the reinforcement can be set to be auto-designed in the same manner.

4. From the Foundations ribbon click Design Pile Caps and all the pile caps set in auto-design mode will be sized accordingly. (Those not in auto-design mode will simply be checked).

At any point you can switch to a user defined arrangement, modify the pile cap configuration and have the design re-checked.

One example where you might choose a user defined arrangement is where the site boundary imposes restrictions on the positioning of the pile cap relative to the column/wall it supports. Switching to a user defined arrangement allows you to specify an eccentricity and create an offset pile cap.
Apply grouping to rationalize pile cap sizes

Once pile caps have been sized individually, the designs can be rationalised by activating grouping, in order to obtain one design per group sufficient for all pile caps within the group.

This is done as follows:

1. Select a pile cap that you want to be in a particular group.
2. In the Properties Window, ensure it is set to auto-design.
3. Right click on the same pile cap and from the context menu choose ‘Create Property Set...’
4. Select all the other pile caps that you want to be in the same group.

When applied moments are significant, be cautious when grouping pile caps where auto-design has initially determined different principal directions.

5. In the Properties Window, click ‘Apply...’ to apply the property set you have just created to the selected pile caps.
6. From the Groups page of the Project Workspace, right click on ‘Pile Caps’ (under the Design branch) and choose ‘Regroup Members’ - this will put those pile caps that share similar properties into the same group.

7. Open the Design Options dialog, and from the Design Groups page check the option to design isolated foundations using groups.

8. Click Design Pile Caps - the results obtained will reflect the grouping that has been applied.

---

**Review/Optimise Pile Cap Design**

In the Review View you can examine the design efficiency by switching from Foundations Status to Foundations Ratio. Note that the tool tip also indicates the base size and status as you hover over any base.

If the selections are unacceptable you may need to review the settings in Design Options> Concrete> Foundations.

---

**Create Drawings and Quantity Estimations**

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

Create a model report that includes the concrete pile cap design calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Mat foundation design

Features of the mat foundation analysis model

Analysis Types

Foundation mats are designed for the results of up to three analysis types:

- 3D Building Analysis
- FE Chase-down Analysis
- Grillage Chase-down Analysis

In each of the above analyses, mats are modelled as meshed 2-way slabs, either supported on ground bearing springs, or on discreet piled supports, or a combination of both.

In both the FE and grillage chase-down models the mat and first level above the mat are always combined in a single sub-model.

Soil Structure Interaction

When not supported by a mat, columns and walls typically have supports at their bases.

When a mat or piled mat is introduced these supports must be removed - as the mat now supports the whole building (either on ground bearing springs, or pile
springs). Consequently adding a mat means re-analysis and hence re-design of the whole building.

Inherent in the re-design is the inclusion of 'soil structure interaction' (or support settlement). In the past this has often been ignored, even though design codes suggest that it should be considered.

If you want to include design for situation without support settlement then you need to think about the workflow sequence or use runs with differing ground stiffness assumptions.

Note however that 'soil structure interaction' only affects the 3D analysis model results, and the chasedown results in the lowest sub-model. Members in all other sub-models are thus already being designed both with and without the affects of support settlement.

**Soil Parameters**

**Ground Bearing Springs**

Unless you have defined discreet piled supports, the mat will need to be supported on ground bearing springs.

**Allowable Bearing Pressure**

When using ground bearing springs, a check is performed as part of the design process to ensure the allowable bearing pressure you enter is not exceeded.

**Ground Stiffness - Type**

Both linear and non-linear ground springs can be defined, although in the majority of cases it is suggested that linear springs should suffice.

When linear spring are applied:

i. Allowable bearing pressures are checked  
ii. Uplift (tension) is checked  
iii. If no problems then linear springs are sufficient

When non-linear springs are applied:

i. You can have compression only  
ii. And also capped compression  
iii. Either way analysis takes longer

**Stiffness**

You are required to enter an appropriate stiffness for the soil conditions on site.

The actual value entered is your responsibility, but as a guide the table below illustrates the potential range of values that might be considered.
**Material** | **Lower Limit (kN/m³)** | **Upper Limit (kN/m³)**
--- | --- | ---
Loose Sand | 4,800 | 16,000
Medium Dense Sand | 9,600 | 80,000
Dense Sand | 64,000 | 128,000
Clayey Medium Dense Sand | 32,000 | 80,000
Silty Medium Dense Sand | 24,000 | 48,000
Clayey Soil (qa<200kPa) | 12,000 | 24,000
Clayey Soil (200<qa<800kPa) | 24,000 | 48,000
Clayey Soil (qa>800kPa) | 48,000 | 200,000

**Horizontal Support**

The degree of horizontal support provided by the ground springs can be set as:

- Fixed
- Free
- Spring (default 20% of vertical spring stiffness)

If set to ‘Free’ a mechanism will result unless you provide additional discreet supports.  

**Pile Springs**

In a piled mat, a spring is inserted into the solver model at the top of each pile. The spring stiffness is a user defined property that is specified in the Pile Catalogue.

**Vertical**

Either a linear, or a non-linear spring stiffness can be defined.

**Horizontal**

Horizontally you can either specify full fixity, or a linear spring.

**Typical mat foundation design procedure**
The following example illustrates the typical process to model and design a mat foundation.

For mat foundations the overall procedure is basically the same as that for flat slabs, with the exception that an extra check for mat bearing pressure is required.

The example has been broken down into the following main steps:

1. Design the structure before supporting it on the mat
2. Create the mat, (either with ground springs, or discreet supports)
3. Model validation
4. Perform the model analysis
5. Check foundation Bearing Pressure and Deformations
6. Re-perform member design
7. Open an appropriate view in which to design the mat and:
   a. Add Patches
   b. Design Mats
   c. Review/Optimise Mat Design
   d. Design Patches
   e. Review/Optimise Patch Design
   f. Add and Run Punching Checks
8. Create Drawings and Quantity Estimations
9. **Print Calculations**

**Design the structure before supporting it on the mat**

The model should already be designed and member sizing issues resolved prior to placing the mat foundation.

In order to retain the existing reinforcement design all members should be set to ‘check’ mode. Alternatively you might choose to ‘check and increase’ the reinforcement instead, (by leaving members in ‘autodesign’ mode with the option to select bars starting from ‘current’.)

**Create the mat, (either with ground springs, or discreet supports)**

Unless you have defined discreet piled supports, the mat will need to be supported on ground bearing springs. These are defined by specifying appropriate Soil Parameters in the mat properties.

You are required to manually specify the **Reduce** imposed loads by mat property. This could vary for individual mat panels, depending on the columns and walls attached to each panel, and the number of floors they are supporting.

If using an ‘area’ method of mat creation you will also need to specify the amount of slab overhang.
The remaining properties are the same as those used to define a typical concrete flat slab.

The ‘Mesh 2-way slabs in 3D Analysis’ option gets activated automatically for the level in which the mat is created, enabling the 3D analysis model to be supported on the ground springs.

In this example the minimum area method is used to create a mat with:

- Imposed loads reduced by 30%
- Default overhang
- Mat thickness 600mm
- Ground springs used
- Default allowable bearing pressures
- Default linear spring properties

Model validation

Once the mat has been placed it is worth running a validation check from the Model ribbon at this stage - specifically to check for potential conflicting supports.

A ‘Supports exist within area of Mat Foundation’ warning is issued if member supports conflict with ground springs. (This can be remedied by right clicking on the warning and choosing ‘Delete Items’).
Perform the model analysis

Analysis is required to establish the bearing pressures and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analysed by running **Design All (Static)**, and any seismic RSA combinations by running **Design All (RSA)**.

The above processes will also recheck all the member designs taking account of the effects of soil structure interaction.

In a large model, if you anticipate several analysis might be required to determine the size of mat required, you might find it more efficient to just run the analyses that are required from the Analyse ribbon then re-run the member design at a later stage.

Check foundation Bearing Pressure and Deformations

You can check the mat bearing pressure and 2D deflection contours from the Results View for each of the analysis types that have been performed before commencing the detailed design.
By viewing the 2D deflection results for combinations based on ‘service’ rather than ‘strength’ factors the stiffness adjustments that you apply (via Analysis Options> Modification Factors> Concrete) do not need to account for load factors.

The default stiffness adjustments are dependent on the design code. For design to EC2 the default adjustment factor applied is 0.2.

**Re-perform member design**

Depending on the steps that have been taken to establish the mat size, at this point the design condition reported in the Status Bar for the members will either be ‘Valid’ or ‘Out of Date’.

Unless the status is ‘Valid’ you should recheck the member designs (taking into account the effects of soil structure interaction) by clicking Design All (Static) from the Design toolbar.

> Similarly if an RSA design has previously been performed, but is now out of date Design All (RSA) should be re-run.

If any concrete members now fail, you can switch them back into ‘Autodesign’ mode, (ensuring the ‘Select bars starting from’ property is set to ‘Current’ rather than ‘Minima)
and then design again. This ensures that the bars that were satisfactory for the original design are only ever increased not reduced.

Open an appropriate view in which to design the mat

In models with mats at distinct floor levels you should use 2D Views to work on the mat design one level at a time. Occasionally the ‘3D geometry’ of a model may make it less easy to distinguish between individual floors, in which case it may be easier to design the floors one sub-model at a time.

When working in a 2D View use the right click menu to design or check mats and patches: this saves time as only the mats and patches in the current level are considered.
Running Design Mats or Design Patches from the Foundation ribbon may take longer as it considers all the mats and patches in the model.

Add Patches

This is an interactive process - requiring a certain amount of engineering judgement.
When placing patches under walls you can choose to place a single patch along the wall, an internal patch under the middle, or end patches at the wall ends (as shown above).

Typically at the initial patch creation stage you should make the patches a reasonable size and not be concerned if they are a bit too big - as this will be reviewed/resolved at the patch design optimisation stage.

Whilst the command to add Patches appears on both the Foundations and Design ribbons, the former defaults to creating patches on the bottom surface, the latter to the top surface. For mats the Foundations ribbon default will generally be correct.

Design Mats

Mat design is dependent on the areas of patches (patch areas which are excluded from mat design) - hence patches should be added before mats are designed.

To design multiple mats, either:

1. From the Foundations ribbon run Design Mats in order to design or check all the mats in the model (each according to their own autodesign setting), or,

2. If you have created mats at multiple levels you may prefer to work on one level at a time. To do this, open a 2D View of the level to be designed then right click and choose either Design Slabs or Check Slabs.

These right click options operate on the same basis as the options for beams and columns:
- Design Slabs will re-design slabs and mats (potentially picking new reinforcement) regardless of the current autodesign setting.
- Check Slabs will check the current reinforcement in slabs and mats regardless of the current autodesign setting.

Review/Optimise Mat Design

It is suggested that you use split Review Views to examine the results. You could arrange one view to show Mat Design Status, and then a second view to show Slab Reinforcement in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.
If the selections are unacceptable you may need to review the design settings. e.g. - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a mat, then set that as a minimum.

Once the selections are reasonable it is advisable to select all the panels and swap them out of auto-design mode (and after this point be careful not to use the right click option to design panels unless you really want to.)

---

Auto-design will select the smallest allowable bars at the minimum centres necessary to provide sufficient reinforcement area. Larger bars are only tried when the minimum permissible spacing is reached. Design options allow you to control the minimum acceptable bar sizes and spacings. It is important to use these settings so that the reinforcement established in the slabs is sufficiently spaced to allow introduction of additional reinforcement in the patches. For example you might set a ‘minimum spacing (slab auto design)’ = 150mm.

---

After using the Review View update mode to standardise reinforcement you can then run Check Slabs from the right click menu to check the revised reinforcement.

Remember:

• Consider swapping between Status and Ratio views - if utilisations all < 1 but some panels failing then problems are to do with limit checks. The Ratio view is better for helping you focus on the critical panels.

• During this process it will also make sense to be adding panel patches in which the reinforcement is set to none and strip is set to average. The purpose of this is to smooth out local peaks at the most critical locations which would otherwise dictate the background reinforcement level needed to get a pass status.

**Design Patches**

Having established and rationalised the slab panel reinforcement which sets the background levels of reinforcement, it is now logical to design the patches which will top up the reinforcement as necessary within the strips of each patch.

To do this, either:

1. From the Foundations ribbon run Design Patches in order to design or check all the foundation patches in the model - by default newly created patches will all be in ‘auto-design’ mode - so reinforcement is selected automatically, or,

2. In the 2D View of the level which you want to design right click and choose either Design Patches or Check Patches. Working in this way restricts the design or checking to the patches in the current view.
These right click options operate on the same basis as the options for beams and columns:
- **Design Patches** will re-design the patches (potentially picking new reinforcement) regardless of the current autodesign setting.
- **Check Patches** will check the current reinforcement in the patches regardless of the current autodesign setting.

**Review/Optimise Patch Design**

At this stage the patch sizes can be reviewed:
- Wall patches - can the width be adjusted (minimised)?
- Column patches - Is the size reasonable? - this relates to code guidance about averaging in columns strips - it is not reasonable to average over too wide a width. If in doubt safe thing to do is err on the side of caution and make them a bit smaller.
- Having got the sizes sorted out and the patches re-designed, swap them out of autodesign mode.
- Now click **Slab Reinforcement** in the **Review View** to review and standardise the patch reinforcement. For instance in column patches this might include forcing the spacing of the mat reinforcement to be matched (if the mat has H12-200, then in the patch don't add H12-125, swap to H16-200 - then there will be alternate bars at 100 crs in this strip of the patch.

**Add and Run Punching Checks**

Punching checks require slab reinforcement to be defined/known in order to determine punching capacities.

Punching checks can be added over the entire level, or structure by windowing it. You can then select any check and review the properties assigned to it.
Once added click **Design Punching Shear**

The checks are done and status is shown as:
- **Pass** - if no shear reinforcement is needed
- **Warning** - if shear reinforcement is needed
- **Fail** - if it is impossible to achieve required capacity by adding shear reinforcement
- **Unknown** - if check not run yet
- **Beyond scope or error** - if for example the centroid of the column/wall lies outside the mat
Create Drawings and Quantity Estimations

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

Print Calculations

Create a model report that includes the panel, patch, and punching check calculations that have been performed. (The default Building Design report includes these along with design calculations for other member types in the model).

Typical piled mat foundation design procedure

The following example illustrates the typical process to model and design piles in a piled mat foundation.
The example has been broken down into the following main steps:

1. Design the structure before supporting it on the mat
2. Create the mat
3. Define the pile catalogue
4. Add piles to the mat
5. Model validation
6. Perform the model analysis
7. Perform the pile design
8. Review the pile design status and ratios
9. Perform the mat design

**Design the structure before supporting it on the mat**
The model should already be designed and member sizing issues resolved prior to placing the piled mat foundation.

In order to retain the existing reinforcement design all members should be set to ‘check’ mode. Alternatively you might choose to ‘check and increase’ the reinforcement instead, (by leaving members in ‘autodesign’ mode with the option to select bars starting from ‘current’.)

**Create the mat**

As piled mats are typically designed on the basis that they are supported on the piles alone, in most situations you would generally uncheck the ‘Use Ground Bearing Springs’ option under Soil Parameters in the mat properties.

- The ‘Mesh 2-way slabs in 3D Analysis’ option gets activated automatically for the level in which the mat is created.

In this example the minimum area method is used to create a mat with:
- An overhang of 1.0m
- A mat thickness of 600mm
- The ‘Use Ground Bearing Springs’ option unchecked
Define the pile catalogue

The pile catalogue should be created prior to placing piles in the mat. You are required to define the pile properties in the catalogue manually.

In this example it is assumed this structure is stable horizontally so that the pile type horizontal restraint is left as ‘Fixed’.

In Tekla Structural Designer the only properties that directly influence the pile design are the compressive and tensile capacity and the horizontal and vertical stiffness.

Add piles to the mat

Although piles can be added individually, it is often more efficient to add multiple piles at regular spacings in the form of a pile array.

These can either be placed vertically or on an incline if required.

A preview of the array is displayed making it easier to adjust the array properties before placing the layout.
After piles have been placed as an array, you are then free to edit individual pile positions, or delete them as required.
Model validation

Once the piles have been placed it is worth running a validation check from the Model ribbon to check for potential conflicting supports.

Before the mat was placed the structure would have required supports underneath columns and walls for it to be designed. As support is now being provided by the piles the member supports are no longer required and should therefore be removed.

A ‘Supports exist within area of Mat Foundation’ warning is issued if such member supports exist. The conflict can be remedied by right clicking on the warning and choosing ‘Delete Items’.

Perform the model analysis

Analysis is required to establish the axial forces in the piles and the moments to be used for the mat design.

Gravity, lateral and seismic combinations can be analysed by running Analyse All (Static), and any seismic RSA combinations by running 1st or 2nd Order RSA Seismic.

‘Analyse All’ is run in preference to ‘Design All’ at this stage because member design is influenced by, and should therefore follow after the piled raft design.

Perform the pile design

The piles are checked (and the mat is designed) by running Design Mats from the Foundations ribbon.

The pile types/sizes are not changed during this process.

Review the pile design status and ratios
You can display the Pile Status and Pile Ratios from the **Review View** in order to determine if any remodelling of piles is required.

In this example a number of piles are not heavily loaded so you could consider deleting some of them, or switching the pile type that has been applied.

If you make any changes, to see their effect simply re-run **Analyse All** followed by **Design Mats** once more.

At any stage if you reapply a pile array to a mat any previously existing piles in the mat are removed.
Perform the mat design

The Design of the mat itself is described elsewhere in the Typical mat foundation design procedure
Vibration of Floors to SCI P354 Handbook

Introduction to Floor Vibration (P354)

With the advent of long span floors, multiple openings in webs, minimum floor depth zones etc. the vibration response of floors in multi-storey buildings under normal occupancy has increasingly become of concern to clients and their Engineers and Architects.

Detailed guidance on the subject is available through the SCI Publication P354 Design of Floors for Vibration: A New Approach (Ref. 2).

This handbook describes the method for the assessment of floor vibration in accordance with P354 that has been adopted in Tekla Structural Designer. The method seeks to establish, with reasonable accuracy, the response of the floor to dynamic excitation expected in offices of normal occupancy. This excitation is almost solely based on occupants walking. With appropriate design criteria, the approach is likely to be equally applicable to sectors other than offices.

The existing solution to checking this type of criterion - a simple calculation of the natural frequency of an individual beam - is felt in many cases to be insufficiently accurate. More importantly, such calculations do not consider two important factors,

- the natural frequency is only the 'response side' of the equation. The 'action' side of the equation is also important i.e. the dynamic excitation - this is the activity that might cause an adverse response from the floor. Walking, dancing and machine vibration are all on the 'action' side of the equation and are all very different in their potential effect.
- the natural frequency of an isolated beam is exactly that and takes no account of the influence (good or bad) of the surrounding floor components. In particular, with composite floors, the slabs will force other beams to restrict or sympathize with the beam under consideration.

The culmination of the calculations carried out by Tekla Structural Designer is a 'Response Factor'. It is important to note that this response factor,

- is not a truly real value of the response of the actual floor since the complex nature of real building layouts are idealized into standard 'cases'.
• is compared with certain limits that have been recommended by industry experts for a limited classification of building type. They are not arbitrary but are not absolute either (cf. calculated deflection and deflection limits).

• is relatively insensitive. That is, a twofold change in the response factor will only just be perceptible to the occupants (cf. logarithmic scale of sound power levels, dBa).

• could be over-conservative particularly for those buildings where tight requirements are imposed.

Notwithstanding the above, this approach is another tool at your disposal that could enable you to spot a problem before the floor is built and prevent the first steps of the client into his new building proving a disaster!

You should find that the check is simple to operate, but it will require you to make choices that may be unfamiliar to you. The purpose of this handbook is to assist you in becoming familiar with the requirements of the check and to assist you in making reasonable judgments regarding the input required.

**Scope**

The reference upon which Tekla Structural Designer's floor vibration check is based is the main limiting factor with regard to scope. This is SCI Publication P354 (Ref. 2). There are no doubt many other texts that deal with vibration problems in buildings, and indeed there is a British Standard dealing with the evaluation of human exposure to vibration in buildings, BS 6472: 1992 (Ref. 1). However this SCI publication has distilled this wider knowledge into readily usable design guidance that is specifically aimed at floors in multi-storey buildings of normal occupancy.

You are able to define an area on a particular floor level that is to be subject to the vibration response analysis and design. The layout of beams in real multi-storey buildings can be of almost any configuration. The methodology adopted in P354 is only applicable to regular structures which by and large have to be created from rectilinear grids. It is your responsibility to make an appropriate selection of the beams etc. that are to be the basic components of the idealized case.

As you proceed through the input making your selections, Tekla Structural Designer will, where it is possible to do so, interrogate the underlying model and retrieve the appropriate data. Once all the data has been assembled, you are then able to perform the check, after which a detailed set of results will be available for review. If you are unhappy with the outcome of your choices you can close the results window and make alternative selections by editing the Floor Vibration Check item properties.

**Limitations and Assumptions**

The scope is primarily defined by the reference design document (Ref. 2) but the following additional limitations and assumptions should be noted.
• The design guidance is based on composite floors acting compositely with the steel beams. It is unclear whether the design approach is directly applicable to non-composite construction.

• For simplicity and to avoid the necessity of Tekla Structural Designer having to identify all the beams in the area selected for vibration assessment, the component of the unit mass from the self-weight of the beams is ignored. This will lead to a slight inaccuracy in the participating mass that is conservative (more mass is advantageous). Note, however, that beam self-weight is included in the calculation of beam deflection but only when the self-weight loadcase is included in the load combination.

• Cantilever beams are excluded from the analysis.

• Cold formed sections are excluded.

Design Philosophy

General

The Engineer ensures the safety of building occupants by satisfying all design criteria at the Ultimate Limit State. Similarly, the health of building occupants is partly taken care of when deflection limits at the Serviceability Limit State are satisfied (although this Limit State does have other purposes than simply the health of occupants).

However, for floors that are subject to cyclic or sudden loading, it is the human perception of motion that could cause the performance of a floor to be found unsatisfactory. Such perception is usually related to acceleration levels. In most practical building structures, the reaction of the occupants to floor acceleration varies between irritation and a feeling of insecurity. This is based on the instinctive human perception that motion in a 'solid' building indicates inadequacy or imminent failure.

The working environment also affects the perception of motion. For busy environments, where the occupant is surrounded by the activity that is producing the vibrations, the perception of motion is reduced. In contrast, for quieter environments (such as laboratories and residential dwellings), where the source of vibration is unseen, the perception of motion is significantly heightened.

The design philosophy to ensure that the potential for such human response is minimized, has a number of facets,

• the **dynamic excitation** causing the vibration i.e. the disturbing force profile, which is force and time dependent. For the sorts of building and occupancy considered here, this is the act of walking.

• the **required performance**. This depends upon the type of environment. As discussed above this, in turn, depends upon the involvement of the occupant in the generation of the vibration and also on the nature of the occupancy. The latter is important for laboratories carrying out delicate work, or operating theatres, for example.
• the **provided performance**. This is the ‘Response Factor’ and is dependent on the system natural frequency and, more importantly, the participating mass. The latter is driven mainly by the selection of an area of floor that is reasonable and appropriate.

**Dynamic Excitation**

In a classical spring-mass system that includes a (viscous) damper, when a simple force is applied to the mass to extend (or contract) the spring, the mass moves up and down (oscillates). This movement is significant at first but eventually reduces to zero due to the resistance offered by the damper. In a floor system in a building,

• the mass is the self-weight of the floor and any other loading that is present for the majority of the time that the occupants could be exposed to vibration effects,

• the spring is the stiffness of the floor system, which will have a number of different component beams (secondary and primary) and the floor slab,

• the damper is provided by a number of elements that are able to absorb energy from the free vibration of the system. There will be energy absorbed,

  • within connections, since they behave ‘better’ than the ideal that is assumed
  • from losses due to the unsymmetrical nature of real buildings e.g. grid layout, and dispersion of loads from furnishings and contents
  • from components such as partitions that are out-of-plane of the vibration and interfere with the ‘mode’.

The determination of the contribution of each of these components as they affect real floor systems is given in detail in later sections. These describe the ‘response’ side of the floor system. In order to establish the required performance of the system the ‘input' must also be defined i.e. that event, events or continuum that is the ‘dynamic excitation’.

In the simple example described at the start of this section the 'input' was simply a force that caused a displacement to the system and was then released. This might be equivalent to a person jumping off a chair onto the floor. However, in the context of the concerns over the vibration of floors, it is not this sort of input that is of interest. The main concern is the excitation of the floor brought about by walking.

Unlike the simple example, walking produces loading that is cyclic. This loading can be idealized into a series of sine curves of load against time. Each curve is an exact multiple of the walking frequency called harmonics. When one of these harmonics of the cyclic loading coincides with the natural frequency of the floor system then resonance is set up. The consequence of resonance that is detected, and may disturb occupants, is the associated peak acceleration. For the first harmonic, the peak acceleration is dependent upon the applied force (the weight of one standard person multiplied by a factor, \( \alpha_n \)), the mass of the system (the self-weight of the floor plate plus other loading that could be considered as permanent), and the amount of damping in the system (the damping ratio, \( \zeta \)). The factor, \( \alpha_n \), is known as a Fourier coefficient and links the magnitude of the applied force in any harmonic of the walking function to the weight of one standard
person. It has been established experimentally for different activities and different activity frequencies.

Hence, the dynamic excitation of a floor is dependent upon the forcing function due to walking and its relationship to the natural frequency of the floor system. It is the level of the peak acceleration that this generates that is particularly important in determining the performance of the floor.

**Required Performance**

The required performance of a floor system is very dependent upon the potential response of humans. Human response is a very complex subject since there is no such thing as a 'standard human'. The perception of vibration will differ from person to person, their body mass varies significantly and the body's reaction will depend upon age, gender etc. The human response has been studied and the accepted wisdom is embodied in BS 6472: 1992, Guide to evaluation of human exposure to vibration in buildings (1 Hz to 80 Hz) (Ref. 1).

It may be remembered that it is the acceleration of the floor system that the human perceives. BS 6472: 1992 provides a series of curves one of which is the 'base' limit of (vertical) acceleration against frequency (of the floor). Within the practical range of frequencies dealt with, a single value of the 'base' limit on acceleration is given as 0.005 m/s². This single value holds

- down to 3 Hz but no floor should be allowed to have a system natural frequency below this value anyway
- up to 10 Hz. Such a large value would be unusual but beyond that point there is a simple linear relationship between the base limit of acceleration and the natural frequency within an extended but just practical range.

The accelerations acceptable for different use of buildings are described using the 'base' limits. Multiplying factors are used to increase the base acceleration limit according to the intended use of the building. The multiplying factors are referred to as 'response factors' in the SCI guidance. Thus the target acceleration of the floor under consideration is the root mean square acceleration multiplied by the response factor. This design condition is turned on its head to give a 'provided response factor' that is then compared with the 'required response factor'. The required response factor is the measure of the ‘Required performance’ and is given in the SCI guidance as,

- R = 8 for a workshop
- R = 8 for a general office
- R = 2 for a residential building during day time use

You should choose a required response factor based on both engineering judgement and the advice given in P354. In particular it may be noted that, 'changing R by a factor of 2 is equivalent only to the most marginal change to human perception'.

**Provided Performance**
It is in establishing the provided performance that most of the design calculations are required. The object of these calculations is to determine the 'required response factor'.

The start point is the calculation of the natural frequency of the floor system. This is established from the individual component frequencies for each of two possible shape modes, namely the Secondary Beam Mode and the Primary Beam Mode. The natural frequencies of the individual components can be adjusted to allow for boundary conditions e.g. two spans continuous. The fundamental frequency, $f_0$, is the lower value for the two modes considered. A minimum natural frequency is given in SCI P354 of 3.0 Hz.

Next the 'modal mass' is required. This is dependent upon the physical size of the floor plate selected and an effective width and/or length that is itself dependent on the natural frequency of the floor. The modal mass has by far the largest influence on the response factor provided.

The 'Resonance Build-up Factor' makes allowance for the time it takes for someone walking across the floor to begin to excite the floor - vibration is not instantaneous upon the first footfall. This has an upper limit of 1.0 and can be taken conservatively as 1.0.

The calculation requires the 'damping ratio' - this is a user input.

The resonance build-up factor, the damping ratio, the modal mass, and the weight of a 'standard person' along with an appropriate Fourier coefficient are used to calculate the peak acceleration.

The final determination of the response factor provided requires the 'root mean square' acceleration. The rms acceleration has two formulations depending upon the fundamental, system frequency. The response factor is a very simple calculation.

The design condition is simply,

$$ R_{\text{prov}} \leq R_{\text{reqd}} $$

**Provided Performance**

**System Frequency**

**Deflections**

For the primary beam, the base maximum simply supported deflection, $\delta_{PBSS}$, is derived from the TSD model with no allowance for boundary conditions.

For the secondary beam, the base maximum simply supported deflection, $\delta_{SBSS}$, is derived from the TSD model and the maximum deflection for a fixed end condition, $\delta_{SBFE}$, is calculated from,

$$ \delta_{SBFE} = m*b*L_{SB}^4/(384*E_{SB}*I_{SB}) + m*b*L_{SB}^2/(24*G*A_{y}) $$
Where

\( m \) = unit mass in kN/mm\(^2\)

\( b \) = secondary beam spacing in mm

\( L_{SB} \) = span of the secondary beam in mm

\( I_{SB} \) = the inertia of the secondary beam in mm\(^4\)

\( E_S \) = the steel modulus in kN/mm\(^2\)

\( G \) = the steel shear modulus in kN/mm\(^2\)

\( A_y \) = the major axis shear area in mm\(^2\)

For the slab, the base maximum deflection for a fixed end condition, \( \delta_{\text{SlabFE}} \), is calculated from,

\[
\delta_{\text{SlabFE}} = \frac{m \cdot L_{\text{Slab}}^4}{384 \cdot E_C \cdot I_{\text{Slab}}}
\]

Where

\( m \) = unit mass in kN/mm\(^2\)

\( L_{\text{Slab}} \) = span of the slab in mm

\( I_{\text{Slab}} \) = the inertia of the slab in mm\(^4\)/mm

\( E_C \) = the dynamic concrete slab modulus in kN/mm\(^2\)

\[
= \frac{E_s \cdot 1.1}{\alpha_{\text{short}}}
\]

These base, maximum simply supported deflections for both primary and secondary beams, \( \delta_{\text{SS}} \), derived from the TSD model, can be adjusted to cater for boundary conditions for 'two-span continuous' or 'three-span continuous' cases to give \( \delta_{\text{barSS}} \).

For 'two span continuous' the adjusted deflection is taken from P354 as,

\[
\delta_{\text{barSS}} = \text{MIN}\left(0.4 + k_M/k_S \cdot \left(1 + 0.6 \cdot \frac{L_S^2}{L_M^2}\right)/(1 + k_M/k_S), 1.0\right) \cdot \delta_{SS}
\]

Where

\( k_M \) = the 'stiffness' of the critical span selected by the user (primary or secondary beam as appropriate)

\[
= \frac{I_M}{L_M}
\]

\( k_S \) = the stiffness of the adjoining span selected by the user (primary or secondary beam as appropriate)

\[
= \frac{I_S}{L_S}
\]
\[ L_M = \text{the span of the critical span selected by the user (primary or secondary beam as appropriate)} \]

\[ L_S = \text{the span of the adjoining span selected by the user (primary or secondary beam as appropriate)} \]

\[ I_M = \text{the inertia of the critical span selected by the user (primary or secondary beam as appropriate)} \]

\[ I_S = \text{the inertia of the adjoining span selected by the user (primary or secondary beam as appropriate)} \]

For 'three span continuous' the adjusted deflection is taken from P354 as,

\[
\delta_{\text{barSS}} = \text{MIN}\left[(0.6 + 2 * k_M/k_S * (1 + 1.2 * L_S^2/L_M^2))/(3 + 2 * k_M/k_S), 1.0\right] \delta^{SS}
\]

Where

\[ k_M = \text{the 'stiffness' of the critical (middle) span selected by the user (primary or secondary beam as appropriate)} \]

\[ = I_M/L_M \]

\[ k_S = \text{the stiffness of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)} \]

\[ = I_S/L_S \]

\[ L_M = \text{the span of the critical (middle) span selected by the user (primary or secondary beam as appropriate)} \]

\[ L_S = \text{the span of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)} \]

\[ I_M = \text{the inertia of the critical (middle) span selected by the user (primary or secondary beam as appropriate)} \]

\[ I_S = \text{the inertia of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)} \]

**Secondary Beam Mode**

In this mode the primary beams form nodal lines (zero deflection) about which the secondary beams vibrate. The slab is assumed to be continuous over the secondary beams so a fixed end condition is used.

\[ \delta_{\text{SBmode}} = \delta_{\text{barSBSS}} + \delta_{\text{SlabFE}} \]

and
Primary Beam Mode

In this mode the primary beams vibrate about the columns as simply supported beams whilst the secondary beams and slabs are taken to be fixed ended

\[ \delta_{PBmode} = \delta_{barPBSS} + \delta_{SBFE} + \delta_{SlabFE} \]

and

\[ f_{PBmode} = \frac{18}{\sqrt{\delta_{PBmode}}} \]

System Frequency

The natural frequency of the system, \( f_0 \), is calculated from,

\[ f_0 = \text{MIN}\{ f_{SBmode}, f_{PBmode} \} \]

Limitations

The absolute minimum natural frequency of the floor system is limited to 3.0 Hz. Where the floor system frequency is below these limits the design fails.

Similarly, no single element within the floor structure should have a fundamental frequency less than 3.0 Hz. Three additional checks are therefore carried out and their results only published if there is a Fail. These checks are,

\[ f_{PBSS} = \frac{18}{\sqrt{\delta_{PBSS}}} \quad \text{must be } \geq 3 \text{ else the design Fails} \]
\[ f_{SBSS} = \frac{18}{\sqrt{\delta_{SBSS}}} \quad \text{must be } \geq 3 \text{ else the design Fails} \]
\[ f_{SlabFE} = \frac{18}{\sqrt{\delta_{SlabFE}}} \quad \text{must be } \geq 3 \text{ else the design Fails} \]

Modal Mass

The 'modal mass' is the effective mass participating in the vibration of the floor. In accordance with SCI P354, it is taken as the 'unit mass' multiplied by the effective plan area of the floor participating in the motion as given by,

\[ M = m \times L_{eff} \times S \]

Where

\[ m = \text{the unit mass in kg/m}^2 \]
\[ L_{eff} = \text{the effective floor length} \]
\[ S = \text{the effective floor width} \]
Where

\[ L_{\text{eff}} = 1.09 \times (1.10)^{n_y-1} \times \left( \frac{E \times I_{\text{SB}}}{m \times b \times f_0^2} \right)^{0.25} \text{ but } \leq n_y \times L_y \]

Where

- \( n_y \) = number of bays (\( \leq 4 \)) in the direction of the secondary beam span
- \( EI_{\text{SB}} \) = dynamic flexural rigidity of the composite secondary beam (in Nm² when \( m \) is in kg/m²)
- \( b \) = floor beam spacing (in m)
- \( f_0 \) = system, natural frequency from above
- \( L_y \) = span of the secondary beam (in m)

and

\[ S = \eta \times (1.15)^{n_x-1} \times \left( \frac{E \times I_{\text{Slab}}}{m \times f_0^2} \right)^{0.25} \text{ but } \leq n_x \times L_x \]

Where

- \( n_x \) = number of bays (\( \leq 4 \)) in the direction of the primary beam span
- \( EI_{\text{Slab}} \) = dynamic flexural rigidity of the slab (in Nm² when \( m \) is in kg/m²)
- \( f_0 \) = system, natural frequency from above
- \( L_x \) = span of the primary beam (in m)
- \( f_0 \) = frequency factor
- \( \eta \) =

\[ \begin{align*}
&= 0.5 \quad \text{for } f_0 < 5 \text{ Hz} \\
&= 0.21 \times f_0 - 0.55 \quad \text{for } 5 \text{ Hz} \leq f_0 \leq 6 \text{ Hz} \\
&= 0.71 \quad \text{for } f_0 > 6 \text{ Hz}
\end{align*} \]
Mode Shape Factor

As previously described, there are two main mode shapes which relate to the lowest frequencies - a secondary beam mode and a primary beam mode. The lowest frequency of the two modes is used and the mode shape factors is determined using the same mode.

There are two mode shape factors, $\mu_e$ at the point of excitation and $\mu_r$ at the point of response.

If the response and excitation points are unknown, or if a general response for the whole floor is required, $\mu_e$ and $\mu_r$ can conservatively be taken as 1.

TSD will not calculate the values of these mode shape factors, and will default to 1.0 but also gives you the option of providing values to be used.

Resonance Build-up Factor

The 'resonance build-up factor' makes an allowance for the time it takes for someone walking across the floor to begin to excite the floor - vibration is not instantaneous upon the first footfall. Hence, a 'walking time' is required and is calculated from the 'walking distance' (see: Maximum corridor length) divided by the 'walking velocity'.

First it is necessary to calculate the walking velocity as given by Equation 16 of SCI P354,
\[ V = 1.67 \times f_p^2 - 4.83 \times f_p + 4.5 \text{ for } f_p \text{ in the range 1.7 to 2.4 Hz} \]

Where

\[ f_p = \text{the pace (walking) frequency supplied by the user} \]

The resonance build-up factor is taken from Equation 37 of SCI P354,

\[ \rho = 1 - e^{(2\pi \zeta L_p f_p / V)} \]

Where

\[ \zeta = \text{the damping ratio} \]
\[ L_p = \text{the walking distance} \]
\[ V = \text{the walking velocity given above} \]

Note that the resonance build-up factor has an upper bound of 1.0 and may, conservatively be set to 1.0.

**Response Acceleration**

**Low Frequency Floors**

For system frequencies between 3 Hz and 10 Hz, the root mean square (rms) acceleration is calculated from,

\[ a_{w,rms} = \mu_e \times \mu_r \times 0.1 \times Q \times W \times \rho / (2 \sqrt{2} \times M \times \zeta) \]

Where

\[ \mu_e \& \mu_r = \text{mode shape factors} \]
\[ Q = \text{the person's weight taken as 745.6 N (76 kg)} \]
\[ M = \text{the modal mass (kg)} \]
\[ \zeta = \text{the damping ratio} \]
\[ \rho = \text{the resonance build-up factor} \]
\[ W = \text{the appropriate code-defined weighting factor for the human perception of vibrations, based on the fundamental frequency, } f_0 \]

\[ = f_0/5 \quad \text{for } 2 \leq f_0 < 5 \]
\[ = 1.0 \quad \text{for } 5 \leq f_0 \leq 16 \]
\[ = 16/f_0 \quad \text{for } f_0 > 16 \]
**High Frequency Floors**

For system frequencies greater than 10 Hz, the root mean square (rms) acceleration is calculated from the following expression, which assumes that the floor exhibits a transient response,

\[ a_{w,rms} = \frac{2\pi \mu_e \mu_r \cdot 185 \cdot Q \cdot W \cdot M \cdot f_0^{0.3} \cdot 700 \cdot \sqrt{2}}{\pi} \]

**Response Factor**

The 'base curves' in BS 6472: 1992 are given in terms of root mean square (rms) acceleration. The provided response factor is then calculated from,

\[ R_{prov} = \frac{a_{w,rms}}{0.005} \]

The 'required response factor', \( R_{reqd} \), is a user input and leads to the final design condition,

\[ R_{prov} \leq R_{reqd} \]

**Vibration Dose Values**

When the floor has a higher than acceptable response factor, the acceptability of the floor may be assessed by considering the intermittent nature of the dynamic forces. This is accomplished by carrying out a Vibration Dose Value (VDV) analysis.

This method calculates the number of times an activity (for example walking along a corridor) will take place during an exposure period, \( n_a \), from,

\[ n_a = \left( \frac{1}{T_a} \right) \cdot \frac{VDV}{0.68 \cdot a_{w,rms}} \]

where

\[ T_a = \begin{cases} \frac{L_p}{V} & \text{if } L_p \text{ is known} \\ \text{value supplied by user} & \text{if } L_p \text{ is not known} \end{cases} \]

\[ VDV = \text{VDV value supplied by user} \] [Typical values shown in table below]
Input Requirements

General

The simplified method for the analysis of the vibration of floors given in the SCI Publication P354, on which the Tekla Structural Designer check is based, is only applicable to regular structures which, by and large, are created from rectilinear grids.

Of course the floor layouts of 'real' multi-storey buildings are rarely uniform and Tekla Structural Designer therefore provides you with the opportunity to select the more irregular floor areas to be assessed with grids that are other than rectilinear.

In so far as the selection of the beams to be used in the analysis is concerned, only beams with Non-Composite or Composite attributes are valid for selection and, within these confines, you are able to:

- select a single beam
- select a beam span as critical plus an adjoining span (in a two or three span configuration)

In all cases, and subject to the above restrictions, which beams from the selected area of floor are chosen is entirely at your discretion and under your judgement, but it is expected that the beams chosen will be those that are typical, common or the worst case. Irrespective, Tekla Structural Designer will take these beams as those that form the idealized floor layout. There is no validation on what the you select (although there is some validation on which beams are selectable i.e. beams which have no slab for part of their length, beams from angle sections, beams with no adjoining span when a 2-span configuration is chosen, and beams with no adjoining span at both ends when a 3-span configuration is chosen will not be selectable).
Data Derived from Tekla Structural Designer

Note that, where appropriate, the derived data is for each design combination under SLS loads only.

Unit mass

The unit mass in kg/m² is used to establish the 'participating mass' of the floor - that is the mass of floor and its permanent loading that has to be set in motion during vibration of the floor. It is taken as the slab self-weight (and to be accurate, the beam self-weight), other permanent 'Dead' loads and the proportion of the 'Imposed' loads that can be considered as permanent. The latter is usually taken as 10% and, whilst this is the default, the value is editable since imposed storage loads, for example, would warrant a higher value.

The unit mass is obtained by summing all the loads (or the appropriate percentage in the case of imposed loads) that act over or in the selected area. This includes any blanket, area, line and spot loads that are present within the selected area. The component of any of these load types that lie outside of the selected area are ignored. Nodal loads directly on columns are also ignored. The total load is then divided by the area selected.

The slab self-weight will usually be in the Slab Dry loadcase - note that in the case of composite slabs this includes the weight of decking. The beam self-weight is in a separate protected loadcase. For simplicity this component of the unit mass is ignored. This leads to a slight inaccuracy in the participating mass that is conservative (more mass is advantageous).

Note that the use of imposed load reductions has no effect on the floor vibration check.

Slab data

If there are more than one set of slab attributes in the selected area then you have to choose which of these it is appropriate to use. From the designated slab attributes the following information/data is obtained,

- the un-transformed inertia in cm⁴ per metre width. For profiled decking this takes account of the concrete in the troughs and is independent of the direction of span of the decking. For pc slabs it is the gross inertia ignoring the formed cells.
- the short-term modular ratio for normal or lightweight concrete as appropriate.

If the designated slab attributes are for a 'generic' slab, then you are asked for the inertia and the dynamic modular ratio.

Secondary beam data

When these are non-composite beams, the inertia is obtained from the sections database. When these beams are of composite construction the inertia is the gross, uncracked composite inertia based on the dynamic modular ratio that is required. Steel joist inertias from the database are assumed to be 'gross' inertias of the chords and are editable. Following guidance contained in AISC Steel Design Guide 11
(Ref. 3), section 3.6, the gross steel joist inertia is factored by quantity $C_r$ and displayed as the 'effective' inertia in the results viewer.

The span of the critical/base beam and the adjoining beams is required.

The deflection of the critical beam under the permanent loads is required. To calculate this value, the deflection under the Dead loads and the appropriate percentage of the Imposed load deflection is summed.

**Primary beam data**

The same data is required as that for the secondary beams.

**Floor plate data**

The dimensions of the floor plate in the idealized cases are defined in one direction by the number of secondary beam bays and in the orthogonal direction by the number of primary beam bays. In practice, given that the idealized case may not attain, the floorplate dimensions are derived from the slab items you select as participating in the mass.

**User Input Data**

**Secondary Beam Spacing**

You must confirm the spacing of the secondary beams - an average value when the spacing is non-uniform.

**Proportion of Imposed Loads**

You are required to specify the proportion of the imposed loads that is to be used in the vibration analysis.

**Number of bays used to establish Modal mass**

You are required to specify the number of bays in the direction of the secondary beam span, $n_y$, and the number of bays in the direction of the primary beam span, $n_x$, that are to be used to establish the modal mass. The number of bays ranges from 1 to 4 for both directions.

**Mode Shape Factors**

You are required to specify the mode shape factors, $\mu_e$ and $\mu_r$, which are to be used in the evaluation of the root mean square response acceleration. The default value is 1.0 for both variables.

**Damping ratio**

Floors do not vibrate as a free mass but have some damping i.e. dissipation of the energy in the system. Four values of damping ratio are recommended in P354 as a percentage,

- 0.5%, for fully welded steel structures, e.g. staircases,
• 1.1%, for completely bare floors or floors where only a small amount of furnishings are present,
• 3.0%, for normal, open-plan, well-furnished floors (the default),
• 4.5%, for a floor where the designer is confident that partitions will be appropriately located to interrupt the relevant mode(s) of vibration i.e. the partition lines are perpendicular to the main vibrating elements of the critical mode shape.

Since an even higher damping ratio might be justified for storage floors for example, a range of up to 10% is offered.

**Maximum corridor length**

This is used in the calculation of the 'Resonance Build-up Factor' that makes an allowance for the time it takes for someone walking across the floor to begin to excite the floor - vibration is not instantaneous upon the first footfall. Hence, a 'walking time' is required and is calculated from the 'walking distance' (maximum corridor length) divided by the 'walking velocity'.

The designer will often not know, reliably, the maximum corridor length. The default is therefore taken as the longer of the floor plate dimensions.

If the designer does not wish to estimate the maximum corridor length or accept the default, then the Resonance Build-up Factor can be set to 1.0 by selecting ‘Not known’ for the maximum corridor length. This sets the Resonance Build-up Factor to 1.0.

**Walking Pace**

The walking frequency (pace) must be selected in the range 1.7 to 2.4 Hz. This range is equivalent to a walking velocity of 2.5 to 5.7 mph (4.0 to 9.1 kph). 'Walking' velocities less than and greater than this are achievable - slow walking 1.0 to 1.5 mph (1.6 to 2.4 kph) or running 6.0 to 12.0 mph (9.6 to 19.2 kph). However, the range of validity of the formula for calculating the walking velocity is given as that quoted. Thus any consequent value outside of the range 1.7 to 2.4 Hz is given a Warning that this is outside of the range given in Equation 16 of SCI P354. The default value is 1.8 Hz.

**Resonance build-up factor**

This is calculated data and has an upper bound of 1.0. However, you are able to specify that the calculations should use 1.0 perhaps because there is insufficient information at the time to make a more accurate and reliable estimate (see: Maximum corridor length). Setting the value to 1.0 is conservative.

**Required Response Factor**

You must enter the response factor that you expect the floor to achieve. This will be based on your engineering judgement and the advice given in P354. Tables 5.2 and 5.3 of that publication give a range of values with the common values being 2, 4, and 8.

**Vibration Dose Value (VDV)**

You have to specify the VDV value to be used if this analysis is performed.
References


3. AISC Steel Design Guide Series. 11: Floor Vibrations Due to Human Activity. AISC 2003 re-print.