# Table of Contents

The Wind Modeling Process ........................................................................................................ 1

Introduction to Wind Modeling ............................................................................................... 1

Design Codes and References ............................................................................................. 1

Scope ....................................................................................................................................... 1

Limitations .............................................................................................................................. 2

Geometry ............................................................................................................................ 3

Loaded Areas ...................................................................................................................... 3

Wind Direction .................................................................................................................... 3

Code Specific Limitations .................................................................................................. 3

Additional wind loads ........................................................................................................ 4

Load decomposition on to concrete walls ..................................................................... 4

Basic Steps of the wind modeling process ........................................................................ 6

Applying Wall Panels and Roof Panels ................................................................................... 7

Applying Wall Panels ............................................................................................................. 7

Applying Roof Panels ............................................................................................................. 8

BS 6399-2 Wind Wizard ............................................................................................................ 8

Using the BS6399-2 Wind Wizard with BREVe data .......................................................... 9

Method page ...................................................................................................................... 9

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

Using the BS6399-2 Wind Wizard with other data .......................................................... 13

Method page .................................................................................................................... 13

Other Location page .......................................................................................................... 13

Other Standard Wind Data page ................................................................................... 14

Other Hybrid Wind Data page ....................................................................................... 14

Results page ......................................................................................................................... 15

Finishing the Wind Wizard .............................................................................................. 17

Creating Wind Zones on the Building ................................................................................... 17

BS6399-2 Wind Zones ............................................................................................................ 17

Basic Geometry ................................................................................................................ 17

Wall Zones ........................................................................................................................ 19

Roof Zones ......................................................................................................................... 20

User Modification of Zones ............................................................................................... 22

Creating Wind Loadcases .............................................................................................. 22
Load Decomposition on to Roof and Wall Panels .............................................................. 23
    Roof Panels......................................................................................................................... 23
    Wall Panels.......................................................................................................................... 23
References................................................................................................................................ 23

Stability Requirements .......................................................................................................... 25
    Introduction to stability requirements .................................................................................. 25
    Global second-order effects ................................................................................................. 25
        Choice of analysis type (BS).............................................................................................. 25
            First-order (Elastic) analysis.......................................................................................... 26
            Amplified forces (kamp) method.................................................................................... 26
            Full second-order analysis............................................................................................. 26
        Sway sensitivity assessment (BS) ..................................................................................... 27
            Twist................................................................................................................................. 28
        First-order or second-order analysis? (BS)......................................................................... 28
        Modification Factors........................................................................................................... 29
    Member second-order effects............................................................................................... 30
    Global imperfections............................................................................................................ 30
    Member imperfections........................................................................................................... 30

Static Analysis and Design Handbook ................................................................................ 31
    Summary of Static Analysis-Design Processes.................................................................... 31
        Definitions............................................................................................................................ 33
        Model validation ............................................................................................................... 34
        Load decomposition.......................................................................................................... 34
            1-way slab load decomposition...................................................................................... 35
            2-way slab load decomposition..................................................................................... 35
        Global imperfections......................................................................................................... 36
        Pattern Loading.................................................................................................................... 36
        3D Building Analysis........................................................................................................... 37
            First Order or Second Order Analysis........................................................................... 37
        Grillage Chasedown Analysis ........................................................................................... 37
        FE Chasedown Analysis....................................................................................................... 37
        Sway sensitivity................................................................................................................... 38
        Member design.................................................................................................................... 38
            Steel Member Design Forces.......................................................................................... 39
            Concrete Member Design Forces.................................................................................... 39
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reset Autodesign</td>
<td>39</td>
</tr>
<tr>
<td>Design Review</td>
<td>40</td>
</tr>
<tr>
<td>Solver models created by Static Analysis-Design</td>
<td>40</td>
</tr>
<tr>
<td>Solver models created by Static Analysis-Design</td>
<td>40</td>
</tr>
<tr>
<td>3D Building Analysis model</td>
<td>41</td>
</tr>
<tr>
<td>2-way slabs meshed</td>
<td>42</td>
</tr>
<tr>
<td>Grillage Chasedown model</td>
<td>42</td>
</tr>
<tr>
<td>FE Chasedown model</td>
<td>43</td>
</tr>
<tr>
<td>Comparison of solver models used in Design (Static)</td>
<td>44</td>
</tr>
<tr>
<td>Seismic Analysis and Design Handbook</td>
<td>47</td>
</tr>
<tr>
<td>Definitions</td>
<td>47</td>
</tr>
<tr>
<td>Base Shear Combination</td>
<td>47</td>
</tr>
<tr>
<td>RSA Seismic Combination</td>
<td>47</td>
</tr>
<tr>
<td>Static Loadcase</td>
<td>47</td>
</tr>
<tr>
<td>RSA Seismic Loadcase</td>
<td>47</td>
</tr>
<tr>
<td>RSA Torsion Loadcase</td>
<td>47</td>
</tr>
<tr>
<td>Fundamental Period (T)</td>
<td>48</td>
</tr>
<tr>
<td>Level Seismic Weight</td>
<td>48</td>
</tr>
<tr>
<td>Effective Seismic Weight</td>
<td>48</td>
</tr>
<tr>
<td>Seismic Base Shear</td>
<td>48</td>
</tr>
<tr>
<td>Square root of Summation of Square (SRSS)</td>
<td>48</td>
</tr>
<tr>
<td>Complete Quadratic Combination (CQC)</td>
<td>48</td>
</tr>
<tr>
<td>Cross Modal Coefficient</td>
<td>49</td>
</tr>
<tr>
<td>Introduction to Seismic Analysis and Design</td>
<td>49</td>
</tr>
<tr>
<td>Overview</td>
<td>49</td>
</tr>
<tr>
<td>Seismic Wizard</td>
<td>50</td>
</tr>
<tr>
<td>Vertical and Horizontal Irregularities</td>
<td>51</td>
</tr>
<tr>
<td>Torsion</td>
<td>51</td>
</tr>
<tr>
<td>Vibration Analysis</td>
<td>51</td>
</tr>
<tr>
<td>Equivalent Lateral Force Method</td>
<td>52</td>
</tr>
<tr>
<td>Response Spectrum Analysis Method</td>
<td>52</td>
</tr>
<tr>
<td>Summary of RSA Seismic Analysis Processes</td>
<td>53</td>
</tr>
<tr>
<td>Seismic Drift</td>
<td>53</td>
</tr>
<tr>
<td>Limitations</td>
<td>54</td>
</tr>
<tr>
<td>Seismic Design Methods</td>
<td>54</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Longitudinal Reinforcement Regions</td>
<td>78</td>
</tr>
<tr>
<td>Relationship between Reinforcement Patterns and Design Regions</td>
<td>79</td>
</tr>
<tr>
<td>Shear reinforcement</td>
<td>81</td>
</tr>
<tr>
<td>Shear Reinforcement Shapes Library</td>
<td>81</td>
</tr>
<tr>
<td>Shear Reinforcement Patterns Library</td>
<td>82</td>
</tr>
<tr>
<td>Shear Reinforcement Regions</td>
<td>82</td>
</tr>
<tr>
<td>Concrete column design</td>
<td>83</td>
</tr>
<tr>
<td>Concrete column design properties</td>
<td>83</td>
</tr>
<tr>
<td>Autodesign (concrete column)</td>
<td>83</td>
</tr>
<tr>
<td>Section</td>
<td>84</td>
</tr>
<tr>
<td>Releases</td>
<td>85</td>
</tr>
<tr>
<td>Slenderness</td>
<td>86</td>
</tr>
<tr>
<td>Stiffness</td>
<td>86</td>
</tr>
<tr>
<td>Load reductions</td>
<td>86</td>
</tr>
<tr>
<td>Stacks and reinforcement lifts</td>
<td>87</td>
</tr>
<tr>
<td>Column design forces</td>
<td>88</td>
</tr>
<tr>
<td>Concrete wall design</td>
<td>88</td>
</tr>
<tr>
<td>Concrete wall design properties</td>
<td>88</td>
</tr>
<tr>
<td>Autodesign (concrete walls)</td>
<td>88</td>
</tr>
<tr>
<td>Wall extensions</td>
<td>89</td>
</tr>
<tr>
<td>Releases</td>
<td>90</td>
</tr>
<tr>
<td>Slenderness</td>
<td>90</td>
</tr>
<tr>
<td>Stiffness</td>
<td>90</td>
</tr>
<tr>
<td>Reinforcement</td>
<td>91</td>
</tr>
<tr>
<td>Load reductions</td>
<td>91</td>
</tr>
<tr>
<td>Restrictions on wall geometry</td>
<td>91</td>
</tr>
<tr>
<td>Stacks and reinforcement lifts</td>
<td>92</td>
</tr>
<tr>
<td>Limitations of wall openings</td>
<td>92</td>
</tr>
<tr>
<td>Recommended alternative model for wall openings</td>
<td>93</td>
</tr>
<tr>
<td>Wall extension examples</td>
<td>95</td>
</tr>
<tr>
<td>Wall design forces</td>
<td>99</td>
</tr>
<tr>
<td>Flat slab design</td>
<td>100</td>
</tr>
<tr>
<td>Summary of the flat slab design process</td>
<td>100</td>
</tr>
<tr>
<td>Flat slab design example</td>
<td>101</td>
</tr>
<tr>
<td>Set up Pattern Loading</td>
<td>101</td>
</tr>
</tbody>
</table>
## Engineers Handbooks (BS)

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design All .........................................................................................................................</td>
<td>102</td>
</tr>
<tr>
<td>Consider Deflection (for Flat slabs) ..................................................................................</td>
<td>103</td>
</tr>
<tr>
<td>Select a Level ..................................................................................................................</td>
<td>103</td>
</tr>
<tr>
<td>Add Patches ......................................................................................................................</td>
<td>103</td>
</tr>
<tr>
<td>Design Panels ...................................................................................................................</td>
<td>104</td>
</tr>
<tr>
<td>Review/Optimise Panel Design ............................................................................................</td>
<td>105</td>
</tr>
<tr>
<td>Design Patches ..................................................................................................................</td>
<td>106</td>
</tr>
<tr>
<td>Review/Optimise Patch Design ............................................................................................</td>
<td>106</td>
</tr>
<tr>
<td>Add and Run Punching Checks .............................................................................................</td>
<td>107</td>
</tr>
<tr>
<td>Create Drawings and Quantity Estimations ..........................................................................</td>
<td>107</td>
</tr>
<tr>
<td>Slab on beam design .........................................................................................................</td>
<td>107</td>
</tr>
<tr>
<td>Summary of the slab on beams design process ....................................................................</td>
<td>108</td>
</tr>
<tr>
<td>Slab on beam design example ..............................................................................................</td>
<td>108</td>
</tr>
<tr>
<td>Set up Pattern Loading ......................................................................................................</td>
<td>109</td>
</tr>
<tr>
<td>Design All .........................................................................................................................</td>
<td>109</td>
</tr>
<tr>
<td>Select a Level ...................................................................................................................</td>
<td>110</td>
</tr>
<tr>
<td>Add Beam and Wall Top Patches .........................................................................................</td>
<td>110</td>
</tr>
<tr>
<td>Design Panels ...................................................................................................................</td>
<td>111</td>
</tr>
<tr>
<td>Review/Optimise Panel Design ............................................................................................</td>
<td>112</td>
</tr>
<tr>
<td>Design Beam and Wall Patches ............................................................................................</td>
<td>113</td>
</tr>
<tr>
<td>Review/Optimise Beam and Wall Patch Design .....................................................................</td>
<td>114</td>
</tr>
<tr>
<td>Create Drawings and Quantity Estimations ..........................................................................</td>
<td>114</td>
</tr>
<tr>
<td>Performing concrete structure design ................................................................................</td>
<td>114</td>
</tr>
<tr>
<td>Pre-design considerations ..................................................................................................</td>
<td>114</td>
</tr>
<tr>
<td>Gravity design ..................................................................................................................</td>
<td>115</td>
</tr>
<tr>
<td>Full design .......................................................................................................................</td>
<td>115</td>
</tr>
<tr>
<td>Reviewing concrete structure design ................................................................................</td>
<td>115</td>
</tr>
<tr>
<td>How do I view results for a single concrete member (without re-selecting steel)? ..........</td>
<td>116</td>
</tr>
<tr>
<td>How do I re-select steel for a single concrete member and then view its results? ..........</td>
<td>116</td>
</tr>
<tr>
<td>Interactive concrete member design ..................................................................................</td>
<td>116</td>
</tr>
<tr>
<td>Interactive concrete beam design .....................................................................................</td>
<td>116</td>
</tr>
<tr>
<td>How do I open the Interactive Beam Design Dialog? .......................................................</td>
<td>117</td>
</tr>
<tr>
<td>Overview of the Interactive Beam Design Dialog ................................................................</td>
<td>117</td>
</tr>
<tr>
<td>How do I change the bar pattern? .....................................................................................</td>
<td>120</td>
</tr>
<tr>
<td>Interactive concrete column design ..................................................................................</td>
<td>120</td>
</tr>
<tr>
<td>Interactive concrete column design</td>
<td>120</td>
</tr>
<tr>
<td>-----------------------------------</td>
<td>-----</td>
</tr>
<tr>
<td>How do I open the Interactive Column Design Dialog?</td>
<td>121</td>
</tr>
<tr>
<td>Overview of the Interactive Column Design Dialog</td>
<td>121</td>
</tr>
<tr>
<td>How do I arrange bars in the Interactive Column Design Dialog?</td>
<td>124</td>
</tr>
<tr>
<td>Interactive concrete wall design</td>
<td>126</td>
</tr>
<tr>
<td>Interactive concrete wall design</td>
<td>126</td>
</tr>
<tr>
<td>How do I open the Interactive Wall Design Dialog?</td>
<td>127</td>
</tr>
<tr>
<td>Overview of the Interactive Wall Design Dialog</td>
<td>127</td>
</tr>
<tr>
<td>Steel Design Handbook</td>
<td>133</td>
</tr>
<tr>
<td>Steel Design Handbook</td>
<td>133</td>
</tr>
<tr>
<td>General design parameters</td>
<td>133</td>
</tr>
<tr>
<td>Autodesign (steel)</td>
<td>133</td>
</tr>
<tr>
<td>Design Section Order</td>
<td>134</td>
</tr>
<tr>
<td>How do I view the list of sections in a design section order?</td>
<td>134</td>
</tr>
<tr>
<td>Size Constraints</td>
<td>134</td>
</tr>
<tr>
<td>Design groups</td>
<td>134</td>
</tr>
<tr>
<td>How is the ‘design using groups option’ activated?</td>
<td>135</td>
</tr>
<tr>
<td>What happens in the group design process?</td>
<td>135</td>
</tr>
<tr>
<td>Instability factor</td>
<td>135</td>
</tr>
<tr>
<td>Steel beam design</td>
<td>136</td>
</tr>
<tr>
<td>Steel beam scope</td>
<td>136</td>
</tr>
<tr>
<td>Steel beam limitations and assumptions</td>
<td>137</td>
</tr>
<tr>
<td>Steel beam design properties</td>
<td>138</td>
</tr>
<tr>
<td>Fabrication</td>
<td>138</td>
</tr>
<tr>
<td>Section</td>
<td>138</td>
</tr>
<tr>
<td>Releases</td>
<td>139</td>
</tr>
<tr>
<td>Restraints</td>
<td>139</td>
</tr>
<tr>
<td>Web Openings to SCI P068</td>
<td>140</td>
</tr>
<tr>
<td>Deflection Limits</td>
<td>141</td>
</tr>
<tr>
<td>Camber</td>
<td>141</td>
</tr>
<tr>
<td>Natural frequency</td>
<td>142</td>
</tr>
<tr>
<td>Seismic</td>
<td>142</td>
</tr>
<tr>
<td>Composite beam design</td>
<td>142</td>
</tr>
<tr>
<td>Composite beam scope</td>
<td>142</td>
</tr>
<tr>
<td>Composite beam loading</td>
<td>143</td>
</tr>
</tbody>
</table>
# Table of Contents

- Steel truss scope ................................................................. 161
- Steel truss design properties ............................................... 162
- Steel joist design ............................................................... 162
  - Standard types ................................................................. 163
  - Special Joists ................................................................. 163
  - Joist Girders ................................................................. 163
  - Joist Analytical Properties ............................................... 163
- Performing steel structure design ........................................ 163
  - Gravity design ............................................................... 164
  - Full design ................................................................. 164
- Individual member design .................................................. 164
  - How do I view the design results for the analysed section? ........................................ 165
  - How do I quickly check an alternative section size? ..................................................... 165
  - How do I quickly design a new section size? ................................................................. 165
- Vibration of Floors to SCI P354 ............................................ 167
  - Introduction to Floor Vibration (P354) .................................. 167
  - Scope .............................................................................. 168
  - Limitations and Assumptions ............................................. 168
  - Design Philosophy .......................................................... 169
    - General ........................................................................ 169
    - Dynamic Excitation .................................................... 170
    - Required Performance ................................................ 171
    - Provided Performance .................................................. 171
  - Provided Performance ..................................................... 172
  - System Frequency .......................................................... 172
    - Deflections .................................................................... 172
    - Secondary Beam Mode ............................................... 174
    - Primary Beam Mode .................................................... 174
    - System Frequency ....................................................... 175
    - Limitations ................................................................... 175
  - Modal Mass ................................................................. 175
  - Mode Shape Factor ....................................................... 177
  - Resonance Build-up Factor ............................................ 177
  - Response Acceleration .................................................. 178
    - Low Frequency Floors .................................................. 178
The Wind Modeling Process

Introduction to Wind Modeling

Design Codes and References

Unless explicitly stated all calculations in the BS 6399-2 Wind Modeller are in accordance with the relevant sections BS 6399-2:1997 incorporating Amendment 1 and corrigendum No. 1. (Ref. 4) It is essential that you have a copy of this code with you while assessing wind on any structure.

Your attention is particularly drawn to BS6399-2:1997 – Clause 1.1. **For building shapes which are not covered by the Standard you will need to seek specialist advice.**

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)

Unless explicitly noted otherwise, all clauses, figures and tables referred to in this handbook are from reference 4.

Scope

*Tekla Structural Designer* can assess and apply wind loading to your model in advance of analysis and design.

It can determine:

- main wind parameters
- zone wind pressures on wall and roof panels
- pressure coefficients for each zone
- wind pressures on each zone
- wind loads and loadcases/combinations for your structure

The model has to be ‘clothed’ in wall panels and roof panels. Wind is ‘applied’ intelligently to this building envelope within the scope below and the limitations clearly laid out in the next section.
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.

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 the main, BS6399-2:1997 addresses rectilinear buildings. In order to develop a tool for engineers, we have extended this capability to address non-rectilinear buildings using the standard method. For more information, please refer to reference 7 (section 2.5.3.2.4, page 82 and 2.5.4.3.3 pages 89-90).

The scope of BS 6399-2 Wind Modeller 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.
- Choice of method:
  - BS6399-2:1997 - Standard Method - Standard effective wind speeds with standard pressure coefficients,
- Basic Wind Speed and Dynamic 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.
- 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 Modeller 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**

- Open sided buildings are beyond scope.
- Free standing walls and sign boards are not considered.
- Parapets and free-standing canopies are not considered.
- Exposed members are not considered, for example lattices, trusses......
- Barrel-vault roofs and domes are not considered.
- Dominant Openings are not explicitly handled – Clause 2.6.2. However, you can use Table 17 to calculate the necessary $C_{pl}$ 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.

**Code Specific Limitations**

**Beneficial Loads**

- No automatic reduction is made for beneficial load. When you edit the Zone Load Data for a wind direction, having generated wind load cases, there is an option to allow for beneficial loads.

**Wind loading on wall panels**

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

- Wall panels that are more than 15° from the vertical are outside the scope – Clause 2.4.1.5.
- The inset storey clause 2.4.4.2 b) is not implemented. You can edit the zones manually according to your engineering judgement to include zone E if you consider this necessary.
Engineers Handbooks (BS)

- Wall panels of internal wells are not automatically identified – Clause 2.4.3.2a. You can manually edit the zones to apply the roof coefficient to the wall panels.

**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°
- The inset storey clauses 2.5.1.7 a) and b) are not implemented. In clause a) the software sets H, and H equal conservatively. You are obviously able to edit the zones manually according to your engineering judgement to include the further zones indicated in Figure 18 should you consider this necessary.
- It should be noted that in Table 8 for curved and mansard eaves, the zones start from edge of horizontal roof and not from the edge of the feature.
- 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, However, you can manually apply the relevant roof type, apex type and bay position parameters for each appropriate wind direction to match the requirements of Figure 22 and Figure 23 - see Creating Wind Zones on the Building.
- 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. You can then define Cpi values manually to either have the same coefficient as the adjacent wall, (Clause 2.5.8.2 Small Overhangs), or as an open sided building (Clause 2.6.3).

**Additional wind loads**

There may be situations when you perceive a need to manually define loads that can not 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.

**Load decomposition on to concrete walls**

All wall panel loads are decomposed into loads on columns. In a building that contains unmeshed concrete walls, the analytical model of the wall consists partly of a ‘mid-pier’ vertical column at the centre of the wall, hence wind wall panel loads will be decomposed onto the mid-pier column.

This decomposition on to the mid pier column could in certain cases result in an averaging of the wind pressure profile that removes the localised pressure increase at the corners of the building.

The example below illustrates the problem and provides an alternative model as a workaround:
The Wind Modeling Process

**Physical model of concrete wall**

Although not shown here, wind wall panels are also added to all four faces of the building.

**Wind zones**

The zones are generated on the wind wall panel faces.

**Resulting stepped wind pressure on wind wall panel faces**

Highest pressure occurs in Zone A, lesser pressure exists in other zones.

**Wind pressure decomposed on to the concrete wall**

Stepped pressure gradient is averaged over the face of the wall and then decomposed on to the ‘mid-pier’ column at it’s centre. Hence only a single point load is applied at each floor level.
Basic Steps of the wind modeling process

There is a simple process to follow when you want to model wind loads. The basic steps are detailed below.

1. Define the structure including all the wall panels and roof panels that will carry the wind loading.

   In order to get the best results from wind modelling you should ensure that you define the largest possible sizes for the wall and roof elements. The results you obtain may be compromised if you define many small elements rather than one large one.

   In complex models you must ensure the outward face of each wall is set correctly, as this is important for determining the wind direction relative to the wall. The front and back faces of the wind walls are assigned different colours (controlled via Settings > Scene) but if you are struggling to distinguish between them you can also switch on the Local Axes for Wind Panels via Scene Content. If the outward face of a wall is set incorrectly it can be reversed from the Edit toolbar.

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

3. Run the Wind Wizard to define the information required for the wind analysis of the structure. The wind wizard automatically determines the wind zoning and external pressure coefficients for the roofs and walls.
4. Confirm the wind zones - If necessary review the Roof Properties and use the **Roof Type** options to change the roof type if required - if the type is changed you will also have to run **Update Zones**. (See Reviewing wind zones and wind zone loads in the User Guides).

5. Define the wind loadcases you want to consider. **Tekla Structural Designer** can calculate details for standard loadcases automatically. You can easily define the information for other loadcases yourself.

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

7. Perform the analysis and design of the structure.

---

**Applying Wall Panels and Roof Panels**

All the calculations for wind 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 can start to calculate the wind loading.

Whilst defining the model's wall and roof panels, it is essential that you define the largest planar surfaces possible for these if you want to get the best results from the software. If you ignore this advice, then 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 gives 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 solidity of 1.0).
- **Decompose to Member** - for wall panels that are not parapets, you can indicate how the wall load is decomposed on to supporting members. See Load Decomposition on to Roof and Wall Panels.

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.

**BS 6399-2 Wind Wizard**

To access this configuration of the Wind Wizard the Wind Loading Code has to be set to BS 6399-2.

Once the wall and roof panels are in place, you use the Wind Wizard to define sufficient site information to calculate the effective wind speeds and dynamic pressures for the required wind directions and heights around the building, (that is the Reference Height \((H_r)\) for each wall panel or roof panel).

The wind speed 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.

It should be noted that BS6399-2:1997 recommends that the Standard Method requires assessment of orthogonal load cases for wind directions normal to the faces of the building. The wizard permits you to create wind load for any wind direction and thus it is up to you to create those loads for the directions most appropriate to your structure.

**Using the BS6399-2 Wind Wizard with BREVe data**

**Method page**

This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

There are two calculation methods available:

- **Standard**, which uses standard effective wind speeds with standard pressure coefficients,
- **Hybrid**, which uses directional effective wind speeds with standard pressure coefficients.

Assuming you have are going to specify the site data using BREVe Grid Ref data there are two options for the source of the wind data:

- **BREVe - UK National Grid Ref.**
- **BREVe - Irish Grid Ref.**

**BREVe location page**

This page allows you to define the location of the site using the BREVe database, and to define various options to be considered in the wind analysis. Once you have retrieved the data for a site from the BREVe database you can edit these to take account of your local knowledge of the site.

**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.

**Site Altitude, A**

You are able to override the altitude determined by BREVe by entering a value directly here.

**Air Density**

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

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.

If you want to use the Hybrid method, then you must know and define the building orientation.

For the Standard method, the orientation is not essential. If you don't define the building's orientation then North is not shown in graphics views and all the $S_\alpha$ values are set to 1.0.

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:
Consider Topography

If you check this box, then BREVe uses the topographic data it recovers for the site and determines the Altitude Factor $S_a$ in accordance with Clause 2.2.2.2.3. Otherwise the topographic data is ignored and $S_a$ is calculated in accordance with Clause 2.2.2.2.

In theory the topography could be significant for some directions and not for others.

Consider Obstructions

With this box checked, BREVe uses the obstruction data it recovers for the site and determines the Effective Height $H_e$ as defined in Clause 1.7.3.3. Otherwise the obstructions are ignored and $H_e$ is taken as $H_r$ - see Clause 1.7.3.2.

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 or 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).

Next
Click **Next** to go to the .

**Using the BS6399-2 Wind Wizard with other data**

**Method page**
This page allows you to specify the method that you want to use to calculate the wind loading on the building, and the source of the wind data.

There are two calculation methods available:
- **Standard**, which uses standard effective wind speeds with standard pressure coefficients,
- **Hybrid**, which uses directional effective wind speeds with standard pressure coefficients.

The remaining topics in this section assume you have chosen to enter the site data manually (Other).

**Other Location 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.

**Altitude**
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 in model**

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.

**Other Standard Wind Data page**

If you select the Standard Method and Other Data Source, then the next page of the Wizard allows you to enter the wind data yourself.

**Basic Wind Speed**

You need to enter the basic wind speed at the site.

**Ground Roughness**

The following settings are available:

- **Sea** – this setting is for sites where the distance to sea is between 0 and 1 km, (see Clause 1.7.2), it is not for offshore sites.

- **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 $H_o$.

  For this category, the ‘Upwind distance from edge of town to site’ can not be greater than the ‘Upwind distance from sea to site’.

**Consider Topography / Altitude Factor, $S_a$**

When this box is checked, you need to use your own topographic data and determine the Altitude Factor $S_a$ in accordance with Clause 2.2.2.2.3. Otherwise $S_a$ is calculated in accordance with Clause 2.2.2.2.2 and you are not able to override it.

**Season factor**

You need to enter the season factor (default 1.0).

**Probability factor**

You need to enter the probability factor (default 1.0).

**Next**

Click Next to go to the .

**Other Hybrid Wind Data page**
If you select the Hybrid Method and Other 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 [Other Standard Wind Data page](#).

**Consider Topography / Altitude Factor, $S_a$**

[reference 8](#) (section 4.10, page 26) essentially recommends using the Standard Method approach to topography even for the Hybrid Method. So, when calculating the Terrain and Building Factor, $S_b$, we ignore the effects of topography, that is we take $S_h = 0$.

When the box is checked, you need to use your own topographic data and determine the Altitude Factors $S_a$ as defined in Clause 2.2.2.2.3. Otherwise $S_a$ is calculated as defined in Clause 2.2.2.2.2 and you are not able to override it.

| In theory the topography could be significant for some directions and not for |
| Season factor |
| You need to enter the season factor (default 1.0). |

| Probability factor |
| You need to enter the probability factor (default 1.0). |

**Next**

Click **Next** to go to the Results page.

**Results page**

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

**BREVe Standard Method**

Initially this method creates 4 orthogonal wind directions relative to the *Tekla Structural Designer* axes, (not geographical North). Except for the Hybrid Method with Other Data,
you can update the wind directions either by using the ‘Dir.’ buttons or by changing the
direction value as required.

Separately, for each relevant parameter of the Standard Method, BREVe determines the
worst case over all its 30° sectors. If the orientation of the building is not known, then
$S_d$ is taken as 1.0 for all directions. Otherwise we determine the worst case $S_d$ for each
direction. You can not override the system value in either case.

The worst case $S_d$ is based on splitting the difference to the next direction, with a
minimum of ±15° and maximum of ±45°. Within these ranges $S_d$ is interpolated.

For each reference height in the model, we then calculate the site wind speed ($V_s$ using
equation 8) and thus the effective wind speed ($V_e$ using equation 12) and the dynamic
pressure ($q_s$ using equation 1) for each direction. When calculating actual loads on walls
and roofs, we use the $q_s$ value for the relevant reference height, but the Results page
only shows the maximum values for each direction.

The Vortex view shows the effective wind speed calculated for each reference height for
each 30° sector. Since a single worst case value is used for each parameter, the speeds
for different sectors only differ due to $S_d$ provided that the orientation of the building is
known.

**BREVe Hybrid Method**

In this case, BREVe uses the directional method to determine the parameters required
to calculate $V_s$ using equation 8, for each height in the building at 30° intervals, (0° to
330°) taking the diagonal dimension ‘a’ as the default 5.0m. (The size effect factor is
applied when determining individual loads). We then use equation 27 to determine $V_e$
and equation 16 for $q_s$.

For each required wind direction the worst case $V_e$ is used for each height, based on
splitting the difference to the next direction, with a maximum of ±45 degrees. Within
these ranges $V_e$ 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 effective wind speed calculated for each reference height for
each 30° sector.

**Other Standard Method**

The calculation of $V_e$ and $q_s$ are very similar to the BREVe Standard 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 $S_d$.

As there is no data for each 30° sector, the Vortex view only shows the effective wind
speed calculated for each reference height for each direction.

**Other Hybrid Method**
The calculation of $V_e$ and $q_e$ are be very similar to the BREVe Hybrid Method, (see above), except that the data has been entered by you for each 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 $S_d$.

As there is no data for each 30° sector, the Vortex view only shows the effective wind speed calculated for each reference height for each 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.

**Creating Wind Zones on the Building**

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.

**BS6399-2 Wind Zones**

**Basic Geometry**

The basic building geometry is assessed as follows:

- Reference Height ($H_r$) – is taken as the difference between highest point on wall or roof panel and ground level.
- Wall height ($H$) – is taken as the difference between highest and lowest points on the wall panel.

These definitions apply to wall panels without parapets and the actual parapets. Wall panels with parapets above them will take their highest point from the parapet. See the diagram below.
• Roof height \( (H) \) – is taken as the difference between highest point on wall or roof panel and ground level. This definition does not handle the upper roof of inset storey but is conservative and only affects the scaling dimension, \( b \) - see Clause 2.5.1.7.

• The Building Breadth, \( B \) is calculated from the smallest enclosing rectangle around the whole building (considered over all roof and wall panels only) for the given direction. You can override the calculated value in case the *Tekla Structural Designer* model does not include the whole building.
Wall Zones

Wall Type

We assess each wall panel to determine if it is a windward, leeward or side wall. We classify the type of wall dependent on $\theta$:

- $\theta \leq 60$ deg – Windward,
- $\theta \geq 120$ deg - Leeward,
- Other walls are classed as Side.

Windward Walls
Windward walls have a single zone, Table 5 is used with interpolation for D/H.

**Leeward Walls**

Leeward walls have a single zone, Table 5 is used.

**Side Walls**

Side walls are assessed for recesses (narrow or wide), irregular flushed faces, downwind re-entrant corners. In all cases, side walls have the relevant number of zones. Table 5 is used.

**Roof Zones**

Roof zones are automatically generated where possible for each wind direction. In essence each roof panel is assessed in its own right based on its properties. The interconnectivity of touching roof panels is not generally considered.

**Direction**

Internally the roof slope vector (line of maximum slope) is determined from the normal vector, with its direction always giving a positive slope angle, i.e. the roof slope vector must always point up the slope.

We calculate the angle between the wind direction and projection of roof slope vector onto horizontal plane (θ in range -180° to +180°).

**Automatic Zoning**

Automatic zoning normally only applies to all triangular roof panels and quadrilateral roof panels that are not concave, that is that all of the internal angles < 180°. However, additionally, it only applies to Hip Gable roofs if they are triangular, and Hip Main roofs if they are quadrilateral. Further, Downwind Slope Hip Gables must not have 2 upwind corners.
Dimensions
All zone dimensions are specified in plan.

Flat Roofs
See BS 6399 Clause 2.5.1, Figure 16 and Table 8.

Monopitch Roofs
See BS 6399 Clause 2.5.2.3, Figure 19 and Table 9.

Duopitch Roofs
See BS 6399 Clause 2.5.2.4, Figure 20 and Table 10.

Hip Gable
See BS 6399 Clause 2.5.3, Figure 21 and Table 11.

Hip Main
See BS 6399 Clause 2.5.3, Figure 21 and Table 11.

Mansard Roofs
If you manually set the connected roof types to Mansard, then the program will correctly identify the special cases in BS6399 Figures 17c, 22a and 22b, and use the correct tables and values. See BS 6399 Clauses 2.5.1.6.2 & 2.5.4

Multi-bay Roofs
We allow you to interpret BS 6399 Clause 2.5.5 and Figure 23 as you think appropriate and manually define the roof types and sub-types accordingly. You also have the ability to manually set the multi-bay position for each roof panel for each wind direction:

- Not Multi-Bay - for this wind direction (conservative default),
- Upwind Bay – first bay of many for this wind direction,
- Second Bay – for this wind direction,
- Third or more Bay – for this wind direction.

Where the reduction applies, the values of all coefficients are reduced according to Table 12.

**Non-Automatic Zoning**

Where automatic zoning does not apply, the system creates a single zone covering the entire roof as follows:

- Flat – \( B \),
- Monopitch – \( B \),
- Duopitch – \( B \) for upwind, \( F \) for downwind, \( B \) for side,
- Hip Gable – \( B \) for upwind, \( G \) for downwind, \( I \) for side,
- Hip Main – \( B \) for upwind, \( F \) for downwind, \( I \) for side.

**User Modification of Zones**

This is not possible in the first release of the software.

Initially the expectation is that only “Expert” users may want to make changes to the actual zone layouts or other data.

Whenever you edit the zones for a wall or roof item, please note that the zone layout will not be updated to reflect changes elsewhere in the model, you must make any necessary changes yourself.

**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 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.

**Load Decomposition on to Roof and Wall Panels**

**Roof Panels**

The direction of the one way decomposition of the wind zone loads to roof members is as specified by the span direction of the roof panel. All types of elements (except bracing and cold rolled members) are considered during the load decomposition.

**Wall Panels**

Wall load decomposition depends on the setting of the ‘DecomposeToMember’ wall panel parameter:

- The default setting for this parameter is ‘No’ and results in nodal loads on the supporting members. This setting is generally appropriate to avoid lateral loads on simple beams.
- Setting ‘DecomposeToMember’ to ‘Yes’ allows the generation of UDL’s on portal stanchions and gable posts without the need to model side rails.

Irrespective of the setting of this parameter, the initial decomposition of wind zone loads to wall members is similar to the roof decomposition. Again all types of elements are considered except bracing and cold rolled members.

If ‘DecomposeToMember’ is set to ‘No’ a second decomposition stage is undertaken:

- 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.

| This second stage is always performed if the members supporting the wall are concrete walls, irrespective of the ‘DecomposeToMember’ setting. |

**References**


9. BREVe software package version 3. Copyright © 2009 CSC (UK) Ltd; BRE Ltd; Ordnance Survey.
Stability Requirements

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 Notional 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 (BS)**
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 certain criteria are met - see [Sway sensitivity assessment (BS)](BS).

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

Both steel and concrete can use the amplified forces method to determine second-order effects although for steel this does have restrictions on use (regular frameworks with $\lambda_{cr} > 4$).

If the amplified forces method is selected you must also indicate which formula to use for determining the amplification factor,

If the structure is clad and if the stiffening effect of cladding is not taken into account explicitly:

$$k_{amp} = \frac{\lambda_{cr}}{(1.15\lambda_{cr} - 1.5)}$$

If the structure is unclad or clad with a direct allowance made for the stiffening effect:

$$k_{amp} = \frac{\lambda_{cr}}{\lambda_{cr} - 1}$$

During the design process for both steel and concrete members, the member end forces from the analysis of the lateral loadcases are amplified by the 'appropriate' value of $k_{amp}$. Since the analysis is first-order this is carried out as part of summing the load effects from each loadcase (multiplied by their appropriate load factor given in the design combination). The 'appropriate' value is the worse of $k_{amp,Dir1}$ and $k_{amp,Dir2}$ based on the worst value of $\lambda_{cr}$ for all stacks in the building.

The $k_{amp}$ results are summarised for each column in both directions. These can be viewed as follows:

1. Open a Review View, and select Tabular Data from the Review toolbar.
2. Select 'k_{amp}' from the View Type drop list on the Review toolbar.
3. The $k_{amp}$ results in both directions are tabulated for each column in the building.

**Full second-order analysis**

Full second-order analysis is not restricted to regular frameworks, but requires $\lambda_{cr} > 2$.

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

**Sway sensitivity assessment (BS)**

In order to determine whether a building is 'non-sway' or 'sway sensitive', *Tekla Structural Designer* calculates the elastic critical buckling load factor, $\lambda_{cr}$. It is calculated in the same manner for steel and concrete. The approach adopted is that for each loadcase containing gravity loads (Dead, Imposed, Roof Imposed, Snow) a set of Notional Horizontal Forces (NHF) are determined. It uses 0.5% of the vertical load at the column node applied horizontally in two orthogonal directions separately (Direction 1 and Direction 2). From a first order analysis of the NHF loadcases the deflection at each storey node in every column is determined for both Direction 1 and Direction 2. The difference in deflection between the top and bottom of a given storey (storey drift) for all the loadcases in a particular combination along with the height of that storey provides a value of $\lambda_{cr}$ for that combination as follows,

$$\lambda_{cr} = \frac{h}{200 \times \delta_s}$$

Where

- $h$ = the storey height
- $\delta_s$ = 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 lambda crit check calculations to avoid spurious results associated with very small stack lengths.*

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. 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 ($\lambda_{Dir1}$ & $\lambda_{Dir2}$) is tabulated for each column in the building.
4. Make a note of the smallest $\lambda$ value from all of the columns in either direction.

*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 $\lambda_{Dir1}$ header until the columns are arranged in increasing order of $\lambda_{Dir1}$, then repeat for $\lambda_{Dir2}.*
In BS 5950-1:2000 a building can be considered as 'non-sway' when $\lambda_{cr} \geq 10$ else it is 'sway sensitive' and (global) second-order effects must be taken into account.

Note however that you are not restricted in your choice of analysis type irrespective of the value of $\lambda_{cr}$ (it is your call, although we will warn you about it).

---

**Twist**

A ‘measure’ of twist is also tabulated for each column - this indicates the degree to which if you push the column one way, how much it moves orthogonally as well. If you have a building where the 'lateral load resisting system' is not well dispersed then pushing one way can cause significant movement in the other direction.

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

---

**First-order or second-order analysis? (BS)**

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

Note: If the rigorous second-order ($P-\Delta$) analysis approach is used during the initial sizing process, you may find that it can be more sensitive to parts of your model that lack stiffness.

The following approach to setting the analysis type is suggested:

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

2. Perform Design All (Gravity) using first-order analysis in order to obtain a figure for $\lambda_{cr}$

3. Either:
• refine the design if necessary until $\lambda_{cr}$ 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 - see below),

• or, in order to use the amplified forces ($k_{amp}$) approach, refine the design further until $\lambda_{cr}$ is greater than 4.0.

4. When a suitable $\lambda_{cr}$ has been achieved change the analysis type to the full second-order, or the amplified forces method as appropriate. (If the $k_{amp}$ approach is selected you must also indicate which formula to use for determining the amplification factor, This will depend on whether the structure is clad or not and if the cladding is taken into account explicitly or not.)

5. Apply lateral loads to the structure.

6. With the analysis type still set to the full second-order, or the $k_{amp}$ approach perform Design All (Static).

If you use the $k_{amp}$ approach be aware that BS5950-1:2000 classes certain structures outside the scope of this method. e.g. tied portals, or structures containing tension only braces. For such structures, you would need to refine the design during gravity sizing (step 3 above) until $\lambda_{cr}$ is at least greater than 2.0, and then use the full second-order analysis approach for the full design (steps 4-6 above).

**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 .

Member imperfections
This handbook describes the automated processes that take place when you perform a static analysis and design of your building.

**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:

<table>
<thead>
<tr>
<th>No.</th>
<th>Process</th>
<th>Description</th>
<th>Exclusions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td><strong>Model validation</strong></td>
<td>Run to detect any design issues which might exist.</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td><strong>Load decomposition</strong></td>
<td>Decomposition of slab loads on to supporting members.</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td><strong>Global imperfections</strong></td>
<td>Notional Horizontal Forces are determined to cater for global imperfections (additional second order effects due to the structure not being built plumb and square). They are also used in seismic design to establish the base shears.</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td><strong>Pattern Loading</strong></td>
<td>Continuous Beam Lines are formed and then load patterns established.</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td><strong>3D Building Analysis</strong></td>
<td>A traditional frame analysis of the entire 3D model, with an option to mesh floors.</td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>-------------------------</td>
<td>----------------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td><strong>Grillage Chasedown Analysis</strong></td>
<td>Requirements: Only performed if concrete members exist. For a series of 3D sub models each containing the members between two horizontal planes but with no 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. Not performed for: • Design Steel (Gravity) • Design Steel (Static)</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td><strong>FE Chasedown Analysis</strong></td>
<td>Requirements: Only performed if two-way slabs exist. 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 wall design. Not performed for: • Design Steel (Gravity) • Design Steel (Static)</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td><strong>Sway sensitivity</strong></td>
<td>Sway sensitivity is checked and design forces amplified automatically if required.</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td><strong>Member design</strong></td>
<td>Member design is performed according to the selected analysis-design process. • Steel members and concrete walls only • Concrete members and walls only • Steel and concrete members and walls For: • Gravity combinations only, or • All combinations Not performed: • Flat slab design • Slab on beam design</td>
<td></td>
</tr>
</tbody>
</table>
At the end of the combined analysis-design process the pass/fail status and utilisation ratio of each beam, column and wall can be reviewed graphically.

**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>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>
<tr>
<td>Elastic critical buckling load factor</td>
<td>BS</td>
<td>Determined from a sway stability analysis and used to determine the sway sensitivity of the structure.</td>
</tr>
<tr>
<td>First-order analysis</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>Non slender</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>Non-sway</td>
<td>All</td>
<td>The global second-order effects (P-Δ) are small enough to be ignored.</td>
</tr>
<tr>
<td>Notional Horizontal Forces</td>
<td>BS</td>
<td>NHF's are calculated as 0.5% of vertical Dead and Imposed loads. These are sometimes referred to as Notional Loads.</td>
</tr>
</tbody>
</table>
### P-Delta analysis

**BS**

Analysis that allows for the presence of second-order effects - referred to in this documentation as Second-order analysis, either:
- by using the Amplified Forces Method, (Kamp)
- 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.

---

**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*.

---

**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 the grillage chasedown model and are also decomposed in the 3D building analysis model 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 Grillage Chasedown Analysis model, the loads on two-way slabs are always decomposed prior to analysis.

In the 3D Building Analysis model, 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.

In the combined analysis-design process, load decomposition is followed by **Global imperfections**

**Global imperfections**

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.

In the combined analysis-design process, equivalent horizontal force application is followed by **Pattern Loading**

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

In the combined analysis-design process, pattern loading is followed by 3D Building Analysis.

**3D Building Analysis**

A traditional frame analysis of the complete structure is always the first analysis performed. The 3D Building Analysis model used 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.

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

**Grillage 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 grillage chasedown analysis. (provided concrete beams exist).

The Grillage Chasedown model emulates a traditional analysis and generates an alternative second set of design forces for beams, columns and walls.

Grillage analysis loadcase results are used in the design combinations as follows:

- Gravity loadcase results in each combination are retained.
- Lateral loadcase results are discarded (as they are invalid) and replaced by those obtained from the 3D Building Analysis.

In the combined analysis-design process, grillage chasedown analysis is (optionally) followed by FE Chasedown Analysis.

**FE Chasedown Analysis**

An FE Chasedown model 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).
FE Chasedown loadcase results are used in the design combinations as follows:

- Gravity loadcase results in each combination are retained.
- Lateral loadcase results are discarded (as they are invalid) and replaced by those obtained from the 3D Building Analysis.

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.

In the combined analysis-design process, FE chasedown analysis is followed by 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.
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.

**Solver models created by Static Analysis-Design**

There are three potential analyses that can be run during the Analysis-Design process:

- **3D Building Analysis** - for the entire 3D model, either with meshed or non meshed
  floors (in the later case load distribution distributes loads to members).

  **3D Building Analysis is performed for ALL buildings.**

- **Grillage Chasedown** - for a series of 3D sub models each containing the members
  between two horizontal planes, either with meshed or non meshed floors (in the
  latter case load distribution distributes loads to members). The complete series of
  models is chased down from top to bottom of the structure so loads are carried
  from the level above to the level below.

  **Grillage Chasedown is performed for any building containing one or more RC
  Member (Beam, Column or Wall).**
  **Grillage chasedown analysis is only valid for gravity loadcases, therefore in
  grillage chasedown design combinations any lateral loadcase results are
  replaced with those obtained from the 3D Building Analysis.**

- **FE Chasedown including both flat slab and beam and slab models** - 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 of
  the structure so loads are carried from the level above to the level below.

  **FE Chasedown is performed for any building containing two-way spanning slabs.**
  **FE chasedown analysis is only valid for gravity loadcases, therefore in FE**
chasedown design combinations any lateral loadcase results are replaced with those obtained from the 3D Building Analysis.

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
- slabs (typically) not meshed, so have their loads decomposed on to supporting members at a preliminary stage of the analysis
- 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.

Although this will inevitably increase the model size, (and potentially the time to solve for large models), 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.

You may also choose to mesh specific floor levels only (e.g. transfer levels), keeping other levels unmeshed.

Grillage Chasedown model

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.
In each of the sub models slab load decomposition is performed using the same FE load decomposition model used for unmeshed slabs in 3D Building Analysis.

**FE Chasedown model**

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 slabs are meshed.
Comparison of solver models used in Design (Static)

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>• Gravity and Lateral analysis (Notional/Wind/Seismic)</td>
<td>• ‘Beam &amp; Slab’ buildings</td>
<td>• Flat slab and ‘Beam &amp; Slab’ buildings</td>
<td></td>
</tr>
<tr>
<td>Special Features</td>
<td>Benefits</td>
<td>Analysis Model</td>
<td>Analysis Method</td>
</tr>
<tr>
<td>------------------</td>
<td>----------</td>
<td>----------------</td>
<td>-----------------</td>
</tr>
<tr>
<td>• Pattern loading • Automatic EC2 sway sensitivity assessment and sway amplification • Automatically centralised analysis wires (improved rigid offsets / rigid zones) • Option to mesh slabs in the 3D analysis</td>
<td>• Member Design considers sway and differential axial deformation effects. • Caters for slabs that contribute to the lateral load resisting system</td>
<td>3D model of entire building: • either meshed 2-way slabs, • or, slab loads decomposed to beams</td>
<td>Whole model in one pass</td>
</tr>
<tr>
<td>• Mimics traditional design approach (sub-frame analysis) • Pattern loading</td>
<td>• Member design based on traditional sub frame is considered simultaneously with the 3D Building Analysis</td>
<td>series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • no FE slabs</td>
<td>Each sub model sequentially from top to bottom – chasing member loads down</td>
</tr>
<tr>
<td>• Mimics traditional design approach (isolated floor analysis) • Slab Pattern loading</td>
<td>• Member design based on traditional sub frame is considered simultaneously with the 3D Building Analysis</td>
<td>series of 3D sub models: • all column and wall stacks immediately above and below the sub-model • no FE slabs</td>
<td>Each sub model sequentially from top to bottom – chasing member loads down</td>
</tr>
<tr>
<td>Component</td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
<td>Options</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>------------------------------------------------------</td>
<td>-------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>RC Beam</td>
<td>Yes – All Combs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RC Column</td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
<td>Options</td>
</tr>
<tr>
<td>RC Wall</td>
<td>Yes – All Combs</td>
<td>Yes – All Gravity load cases</td>
<td></td>
</tr>
<tr>
<td>Steel/Composite Members</td>
<td>Yes – All Combs except patterns</td>
<td>Not required</td>
<td>Not req</td>
</tr>
<tr>
<td>Foundation design</td>
<td>Yes – All Combs except patterns</td>
<td>Yes – All Gravity load cases</td>
<td>Yes – All</td>
</tr>
</tbody>
</table>
Seismic Analysis and Design Handbook

This handbook describes Tekla Structural Designer’s seismic analysis and design capability.
• Eurocode EN1998-1:2004 Seismic Wizard

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

**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:

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

\(\lambda\) = 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:

\[
\lambda = \sqrt{\left( \sum_{i=1}^{n} \sum_{j=1}^{n} \rho_{ij} \lambda_i \lambda_j \right)}
\]

\(\lambda\) = 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
\[ \rho_{ij} = \text{Cross modal coefficient for i & j} \]

**Cross Modal Coefficient**

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

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

\[ \zeta = \text{modal damping ratio} \]

- 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.

\[ \beta = \text{Frequency ratio} = \frac{\omega_i}{\omega_j} \]

\[ \omega_i = \text{Frequency for Relevant Mode i} \]

\[ \omega_j = \text{Frequency for Relevant Mode j} \]

**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.

**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. These latter members need additional design and detailing rules to ensure they resist the seismic forces that they have to resist during an earthquake.

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

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 steel and concrete seismic design of certain specific members.

**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

When the base shear from RSA is < X% of the base shear (code dependent, 85% in the US) from ELF then the RSA forces and drifts are factored up to the X% level as defined in the code. This ‘equates’ the two methods giving RSA a small advantage over ELF but recognising that there will be differences in the distribution of forces and deflections up the structure.

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 steel and concrete seismic design of certain specific members.

**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. This is similar to standard model validation but also checks: • Seismic Inertia Combination must exist • 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>As 1st Order Vibration for the Seismic Inertia Combination only, including returning all 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. 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 included 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**

All structures are assessed for seismic drift - this is done using the ELF approach and so even if RSA is requested, an ELF analysis will be run.

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.
Limitations

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.

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.

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
- 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
- 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 when seismic design is required can be summarised as follows:

1. Modelling

• Identify the primary seismic members, i.e. those members that are part of the seismic force resisting system. 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
• Select ELF or RSA 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 the Design (Static) to:

• Design the non-seismic combinations
• Only if the ELF method was selected - the seismic combinations are also designed in the same way as the other 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

If the RSA method was selected in the Seismic Wizard run Design (RSA) to:
• Perform an additional analysis/design associated with RSA Seismic. The design is a final check design on all members in the structure for the seismic and seismic drift combinations.

6. Calculation Output

• A Seismic Design Report is available
• Drift limitations are checked

7. Limitations and Exclusions

• The limitations for high seismicity regions apply.

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 only set up for the base Eurocode, the UK NA and the draft Singapore NA.

Topics in this section
• Starting the Wizard
• Base Information Page
• Structure Regularity Page
• Fundamental Period Page
• Behaviour Factor Page
• Seismic Inertia Combination Page
• Finishing the Seismic Wizard
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.

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

- **Reference Peak Ground Acc, \( a_{gR} \)**
  (Ref PD6698:2009)

- **Design Ground Acc, \( a_{g} \)**
  \( (a_{gR} \times \gamma_I) \)

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 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, \( \gamma_I \)**
  This is automatically derived from the occupancy class.

- **Spectrum type**
  For Base Eurocode and UK NA: Spectrum Type - 1 or 2 BS EN 1998-1:2004 - CI
3.2.2.2.1(P).
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 $ag \leq 0.78 \text{ m/s}^2$ or if $ag \times S \leq 0.98 \text{ m/s}^2$ - Structure suitable for Low Seismicity (where $S$ is the soil factor)
  If not - Structure suitable for Medium or High Seismicity

**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

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, $T_{Dir1}, T_{Dir1}$**
  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 Analysis and Design Handbook

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.

General design parameters (concrete)

A number of properties and design parameters are common to the different concrete member types - these are described in the topics below.

Autodesign (concrete)

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

If the member type has been set to be designed using Concrete design groups, 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 determine the reinforcement.
- 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.
Concrete design groups

Concrete beams and columns 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 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 (which are created automatically and cannot be edited) are analogous to the sub-groups.

Grouped design and detailing is optional and can be deactivated if required.

Concrete Beam Design Group Definition

Concrete beam design grouping is based on the following rules:

• 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 Group Definition

Concrete column design grouping is based on the following rules:

• 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.

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.

Detailing groups
Each parent design group is sub-divided into one or more detailing groups. These use the design output from the relevant parent design group to create and hold the detailing output relevant to the detailing group.

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.

Concrete Beam Detailing Group Definition
For the purpose of reinforcement detailing beam elements are formed into unique detailing groups based on the following rules:
• 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 Group Definition

For the purpose of reinforcement detailing columns are formed into unique detailing groups based on the following rules:

• 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.

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.

**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.

**Nominal cover**

**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 horz bar or any link/containment transverse reinforcement that may be present.

**Assume cracked**

Assuming concrete sections are cracked has a direct affect on the analysis; smaller Modification Factors are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway sensitivity calculations are also influenced by this assumption.

**Reinforcement Parameters**

The reinforcement parameters which are common to all the concrete member types are specified in Design Options > Concrete > Reinforcement Parameters.

**Concrete beam design**

**Concrete beam design properties**

For design purposes, in addition to the Design Options-Concrete-Beam settings and General design parameters (concrete), the beam properties listed below should be considered:

**Autodesign (concrete beam)**

When Autodesign has been 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.

**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.
There is no need to rationalise the arrangement of shear links. These can vary in size, spacing and number from region to region without having any impact on adjoining regions.

**Deflection control**

**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:

- **T Beam**
- **Γ Beams**
- **Isolated Beam**

These shapes have common features which are shown in the figure below:

\[
\text{Flange width, } b_{\text{eff}}
\]

where

\[
h = \text{overall depth including the depth of the slab}
\]
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 'Γ' 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.

**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 \( L_0 \) 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.

**Releases**

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

<table>
<thead>
<tr>
<th>Release</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fully fixed (free end)</td>
<td>Denotes a cantilever end. It is achieved by checking the ‘Free end’ box.</td>
</tr>
<tr>
<td></td>
<td>(In a single span beam this box can only be checked if the opposite end is fully fixed.)</td>
</tr>
<tr>
<td>Pin</td>
<td>Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.</td>
</tr>
<tr>
<td>Moment</td>
<td>Major axis moment connection, and pinned about the minor axis.</td>
</tr>
<tr>
<td>Fully fixed</td>
<td>Encastré, all degrees of freedom fixed.</td>
</tr>
<tr>
<td>User defined</td>
<td>This setting appears if the connection is pinned for major axis bending (My released) but remains fixed for minor axis</td>
</tr>
<tr>
<td>Continuous</td>
<td>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. (If a pin were to be introduced at an internal position then there would be two beams, hence you cannot edit this setting.)</td>
</tr>
</tbody>
</table>

In addition to the above release options you are also able to apply an axial or torsional release by checking the appropriate box.

**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 ‘nL’ 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:

\[ h \geq 750 \text{ mm} \quad \text{BS8110} \]

**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>
There are three Standard Patterns for top reinforcement, $S_{PT1}$, $S_{PT2}$ and $S_{PT3}$ and two Standard Patterns for bottom reinforcement, $S_{PB1}$ & $S_{PB2}$ as illustrated in the figures below.

**Standard Patterns of Top Reinforcement**

<table>
<thead>
<tr>
<th>11</th>
<th>Standard bob</th>
</tr>
</thead>
<tbody>
<tr>
<td>34</td>
<td>Standard bob with crank</td>
</tr>
<tr>
<td>11</td>
<td>Extended bob</td>
</tr>
<tr>
<td>34</td>
<td>Extended bob with crank</td>
</tr>
<tr>
<td>21</td>
<td>U bar</td>
</tr>
<tr>
<td>99</td>
<td>U bar with crank</td>
</tr>
</tbody>
</table>

**Longitudinal Reinforcement Patterns Library**

There are three Standard Patterns for top reinforcement, $S_{PT1}$, $S_{PT2}$ and $S_{PT3}$ and two Standard Patterns for bottom reinforcement, $S_{PB1}$ & $S_{PB2}$ as illustrated in the figures below.
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:

4. Bar with a bob at each end
7. Straight bar with a length approximately 70% of span – if required by the design
8. Single cranked bar extending over several spans or over one span only and lapped within a support – with bob if it continues over an end span.
9. Straight bar
10. Straight bar running approximately from face to face of beam supports
11. Straight bar
14. Bob bar

Modified versions of the above standard patterns are applied for use in single spans and in cantilever spans where no backspan beam is present.

For short span beams, it becomes uneconomic and impractical to lap bars in beams. These facts coupled with the anchorage lengths that are required make the use of multiple design regions for the longitudinal reinforcement unnecessary. To cater for this a short span beam length can be defined in Design Options > Beam > Reinforcement Settings and the bar patterns adopted for such short spans are as shown below:

Standard Patterns of Reinforcement for Short Span Beams
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 \( \text{SP}_{T1} \) and \( \text{SP}_{B1} \) 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 $B_1$ ($\& B_3$): Bar 8 + Bar 11  
Design Region $B_2$: 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 $\text{MAX}(0.25*L, 2*h)$ and that for $S_2$ being $(L-S_1-S_3)$.

In cantilevers, the regions are as shown below.

![Diagram showing regions S1 and S2 in a cantilever beam]

In all cases, the range of each region is $0\% \leq S_i \leq 100\%$ and $\sum S_i = 100\%$.

**Concrete column design**

**Concrete column design properties**
For design purposes, in addition to the Design Options-Concrete-Column settings and General design parameters (concrete), the column properties listed below should be considered:

**Autodesign (concrete column)**
When **Autodesign** has been 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.

**Increase Principal Bar Sizes Preferentially**

One of the following two options must be selected:

- Yes
- No

Selecting ‘yes’ results in the corner bars being increased in size before the intermediate bars, i.e. increased in preference. All bars will start off at the same size (unless the initial bar size is driven by the current arrangement or the stack above), but when the check fails the corner bars will be increased in size if all bar sizes are the same, otherwise the intermediate bars will be increased in size. This means that when the final design is produced, either all bars will be the same size or the corner bars will be one size larger than the intermediate bars.

Alternatively, if you require all bar sizes to be the same, selecting ‘no’ results in all bar sizes being increased together.

**Match Bar Sizes and Positions to Stack Above**

Selecting this option for a column stack determines that the starting arrangement for longitudinal bars will match the arrangement of the bars in the stack above if the section geometry matches.

The selection of this option is only relevant when the shape, dimensions, rotation and eccentricity of the given stack match the stack above exactly. When running auto-design for a stack, if all of these criteria are met and this option is selected, then the program will set all longitudinal bar positions to match those of the stack above and set all bars to the same size as the equivalent bars in the stack above.

**Section**

**Shape**

The following column shapes are allowed:

- Rectangular
- Circular
- L-shape
- T-shape
- C-shape
- Elbow with uniform thickness
- Trapezium
Concrete Design Handbook

- 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.

**Releases**

Columns can be released about the both axes at the top and bottom of each stack as follows:

<table>
<thead>
<tr>
<th>Release</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed</td>
<td>Encastré, all degrees of freedom fixed.</td>
</tr>
</tbody>
</table>
**Pinned**

A pinned connection is created between the stack above and the stack below.

⚠️ This option should be used with caution as it may result in a mechanism during the analysis.

---

**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.
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. Also, floors can be designated to not have their imposed loads reduced, for example if they are storage or plant floors. In this case the full loading on that floor will be used in determining the reactions onto the 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.

**Concrete wall design**

**Concrete wall design properties**

For design purposes, in addition to the Design Options-Concrete-Wall settings and General design parameters (concrete), the wall properties listed below should be taken into account:

**Autodesign (concrete walls)**

You are required to select whether each wall is in ‘autodesign’ mode or ‘check’ mode. Selecting check mode ensures that the existing reinforcement arrangement is not changed.

When a wall is first created, it will have no reinforcement arrangement so the ‘autodesign’ option must be selected until such time as a reinforcement arrangement has been found. Only when a design has been run (and no errors occurred in the design process) and a valid (but not necessarily passing) reinforcement arrangement has been returned does the ‘check’ option become selectable.

**Autodesign Starting Point**

This option only appears if ‘Autodesign’ is selected. It applies to both longitudinal bars and links.

The options are:
- Minimum bars (default)
- Current bars

Selecting ‘minimum bars’ removes the current arrangement and begins with the minimum allowed bar size from the order list.

**Autodesign for concrete walls**
Auto-design applies the spacing maximisation method for both longitudinal bars and links. This attempts to return a solution with the largest possible longitudinal bar spacing and largest possible link spacing.

**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 $L_{wall,d}$. 
Releases

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

<table>
<thead>
<tr>
<th>Release</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed</td>
<td>Encastre, all degrees of freedom fixed.</td>
</tr>
</tbody>
</table>
| Pinned  | A pinned connection is created between the wall panel above and the wall panel below.  
This option should be used with caution as it may result in a mechanism during the analysis. |
| 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. |

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. Also, floors can be designated to not have their imposed loads reduced, for example if they are storage or plant floors. In this case the full loading on that floor will be used in determining the reactions onto the wall. The moments from fixed ended beams framing into a wall are never reduced.

**Restrictions on wall geometry**

Each wall panel is a 4 sided rhomboid:

- The sides may not be vertical
- The bottom or top may not be horizontal
• The plane of the wall panel may not be vertical. Although it must be of uniform slope, without any crank in the cross section.
• The slope must be constant in the panel length
• All of the panels in a particular wall either follow the plane of the wall, or there may be a change of slope at the panel intersection
• Thickness is measured perpendicular to the face of the wall
• Curved walls in plan or elevation are beyond scope

**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.

**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. Even 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 particular you need to be aware that currently the inertia of the wall beam element formed along the top of each panel does not get modified to take account of the opening.) In certain situations the alternative method of modelling the openings as described below may prove to be a better solution.

**Recommended 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.

**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 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.

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.

**Flat slab design**

Flat Slabs are 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.

An interactive design approach is adopted, which is particularly necessary in the case of flat slab design as the patch and panel design are inter-dependant.

**Summary of the flat slab design process**

Concrete slab design is an interactive process typically consisting 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.

**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.

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.

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 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.

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.

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.

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 tends to end up selecting small bars at close centres in preference to larger bars at wide centres. If you are not careful this will result in impossibly small spacing in the patches. The workaround to this is to set a wide min clear spacing (something like 150mm) for the initial panel design. Then, once the panel reinforcement is optimised and switched out of auto-design, adjust the design setting down to something like 75mm before doing the patch design.
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 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 **Check 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.

---

**Slab on beam design**
Slabs on beams are 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.

An interactive design approach is adopted as the patch and panel design are inter-dependant.

**Summary of the slab on beams design process**

Concrete slab on beams design is an interactive process typically consisting 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.

**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).

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 less likely to be effective when designing to Eurocodes.

**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 the left shows **Slab Design Status**, (with slab patches turned off in Scene Content to assist clarity), the view on the right shows **Slab Reinforcement** in the panels. Note that the tool tip indicates all panel/patch reinforcement as you hover over any panel.
If the selections are unacceptable you may need to review the design settings. e.g. - if 8mm dia bars are selected and you would just never use anything less than a 10mm bar in a 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 tends to end up selecting small bars at close centres in preference to larger bars at wide centres. If you are not careful this will result in impossibly small spacing in the patches. The workaround to this is to set a wide min clear spacing (something like 150mm) for the initial panel design. Then, once the panel reinforcement is optimised and switched out of auto-design, adjust the design setting down to something like 75mm before doing the patch design.

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 autodesign 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.

**Performing concrete structure design**

**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 Concrete design groups.
   
   Adjust Design Options\Design Groups accordingly.

2. **Concrete Design Options** - check the concrete Design Options are appropriate for your design.
   
   Adjust the various pages under Design Options\Concrete accordingly.

3. **Member properties and Autodesign settings** - review the design related properties that have been assigned to individual members.
   
   In particular review the Autodesign setting as this controls 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.

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 concrete beams, columns and walls for the designated gravity combinations.

Gravity design is initiated by clicking Design Concrete (Gravity).

Typically this step can be by-passed by proceeding straight to Design (Static), although some users may find it an efficient means to progress the design of large models.

Full design

All concrete beams, columns and walls are designed or checked for all active combinations.

Full design is initiated by clicking Design Concrete (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.

Reviewing concrete structure design

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?](#)
‘Check Member’ and ‘Design Member’ ignore the auto-design setting of the member. 
In addition because ‘Design Member’ is intended for individual member design, other members in the same design group are NOT updated with the revised reinforcement. 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. 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.)
3. Hover the cursor over the member until its outline is highlighted, then right click.
4. 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.

**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 Design... 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, $A_{s,\text{reqd}}$
- Area of reinforcement provided, $A_{s,\text{prov}}$
- Reinforcement area utilisation ratio
- Smallest clear spacing between bars
- Minimum required reinforcement area, $A_{s,\text{min}}$

**Deflection check**
This checks the actual span: effective depth ratio against the limiting span : effective depth ratio.

**Links tab**

![Image of Link Selection Table]

**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, $A_{swt,reqd/s}$
• 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 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 **Design...** 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 $\frac{M_{Ed}}{M_{res}}$ 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

**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>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>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.

![Diagram showing bar arrangement](image)

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>298.0</td>
<td>4-5</td>
<td>0</td>
<td>298.0</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**

**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 Design... 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:
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
**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 $M_{Ed} / M_{res}$ 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
Steel Design Handbook

This handbook provides a general overview of *Tekla Structural Designer* in the context of its application to hot rolled 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.

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
- Deflection Limits
- Camber (in the case of beams)
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. Only checked sections within this list are considered during the design process.

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).

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.

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.

| This multiplier is applied to prevent unwanted behaviour in the analysis model. While the analysis results may be affected by this adjustment, there is no amplification of the minor axis inertia in the design of the member. |

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:

- 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 to 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

For design purposes, in addition to the General design parameters, the beam properties listed below should be considered:

Fabrication
• Section
• Releases
• Restraints
• Web Openings to SCI P068
• Deflection Limits
• Camber
• Natural frequency
• Seismic

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.

### Releases

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

<table>
<thead>
<tr>
<th>Release</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free end</td>
<td>as in a cantilever</td>
</tr>
<tr>
<td>Pin</td>
<td>pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally</td>
</tr>
<tr>
<td>Moment</td>
<td>major axis moment connection, and pinned about the minor axis.</td>
</tr>
<tr>
<td>Fully fixed</td>
<td>encastré, all degrees of freedom fixed</td>
</tr>
</tbody>
</table>

All internal connections in multi-span beams are considered continuous - if a pin were to be introduced at an internal position then there would be two beams, hence you cannot edit this setting.

At the remote ends of the beam you are responsible for specifying appropriate release conditions.

In addition to the above release options you are also able to apply an axial or torsional release by checking the appropriate box.

### 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.
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 SCI Publication P068.

**Guidance on size and positioning of openings**

We advise you to comply with the following positional recommendations for web openings:

- Web openings are designed using the bending moment and vertical shear values at the side of the opening where the moment is lower,
- Openings should preferably be positioned at the mid-height of the section. If not, the depth of the upper and lower sections of web should differ by not more than a factor of two,
- Openings should not be located closer to the support than two times the beam depth or 10% of the span whichever is the greater,
- The best location for any opening is between 1/5 and 1/3 of the span from a support in uniformly loaded beams, or in lower shear zone of beams subject to point loads,
- Openings should be not less than the beam depth, D, apart,
- Unstiffened openings should not generally be deeper than 0.6D or longer than 1.5D,
- Stiffened openings should not generally be deeper than 0.7D or longer than 2D,
- Point loads should not be applied at less than D from the side of the adjacent opening.

**Dimensional checks** - The program does not check that openings are positioned in the best position (between 1/5 and 1/3 length for udls 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

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.

**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.

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.
- 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 use of channel connectors or Hilti™ connectors is currently beyond the program scope.*

---

Only 19 mm diameter studs with 100 and 125 nominal height (95 and 120 as welded height) are offered. These 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, \( b_e \), for each composite beam as per section 4.6 of BS 5950 : Part 3 : Section 3.1 : 1990.

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

- beam span/8 – span taken as the centre to centre of supports
- one half of the distance to the centre line of the adjacent beam (for slabs spanning perpendicular)
- 40% of the distance to the centre line of the adjacent beam (for slabs spanning parallel)
- the distance to the edge of the slab
Although the program calculates $b_{e}$, 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, $b_{e\text{\{right\}}} = \text{beam span}/8$
- to the left of the beam, $b_{e\text{\{left\}}} = \frac{1}{2}\text{ of the shortest distance to the centre line of the adjacent diagonal beam}$

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

**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.

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 cannot 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**

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 cannot be achieved this is indicated on the design summary thus: “Not able to design stud layout”.

**Manual Stud Layout**

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

**End releases**

The top and bottom of each stack in the general column can be either pinned or fixed which means that the member is pinned about both the major and minor axes in the first case, or fixed about both axes in the second.

Additionally a torsional load release can be specified at the top or bottom of each stack (but not both).

**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. Also, floors can be designated to not have their imposed loads reduced, for example if they are storage or plant floors. In this case the full loading on that floor will be used in determining the reactions onto the column. The moments from fixed ended beams framing into a column are never reduced.
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 SCI Publication P068. 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 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 calculations with the result that the calculated 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.

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.

**Compression**

Effective length factors are defined for each axis of buckling.

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

**Head code: BS 5950-1:2000**

If either an Angle (single or double) or Channel or Tee section has been selected as the brace member then a 'Connection at each end' property box appears which allows selection of an appropriate Clause from Table 25 of the BS code to describe the connection type.

The default Clauses from Table 25 are as follows:

- **Single Angle**: 4.7.10.2a
- **Double Angle**: 4.7.10.3d
- **Channel**: 4.7.10.4b
- **Tee**: 4.7.10.5b

**Notes:**

1. Clauses 4.7.10.2b, 4.7.10.2c, 4.7.10.3b, 4.7.10.3c, 4.7.10.3d and 4.7.10.3e only apply to Bolted connections so in these cases 'Bolted' should be selected in the 'Connection' property box. ('Bolted' is the default connection.)

2. For Angle (single and double), Channel and Tee sections the Effective Length Factors will be auto-completed according to the connection Clause selected but these Factors can be changed if required. For Angle (single and double) sections the length $L_v$ is always assumed to be $L/3$ and the Effective Length Factor $v$ will act on this $L_v$.

3. For single Angle sections the longer leg is assumed to be the connected element unless 'Short attached' is checked on the Size page of the dialog.

4. For double unequal Angle sections, whichever leg is not the back-to-back leg is assumed to be the connected element when connection Clauses 4.7.10.3a and 4.7.10.3b are selected, and vice versa with Clauses 4.7.10.3c, 4.7.10.3d and 4.7.10.3e
**Tension**

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

- Percentage value
- Effective net area

**Head code: BS 5950-1:2000**

Brace tension capacity is calculated according to section type as follows:

A. Hollow sections (CHS, RHS, SHS): Gross area capacity

B. Angle, Channel, Tee sections: Reduced effective net area capacity (Clauses 4.6.3.1 and 4.6.3.2)

C. I/H and any other sections: Effective net area capacity (Clause 4.6.1)

**Notes:**

1. For section types B and C listed above, an Effective net area ($A_e$) should be entered either as a percentage of the gross area ($A_g$) or as an absolute value. The default is to use 100% of $A_g$ and an absolute value can not be used if autodesign is also selected.

2. The 'Geometry' property needs to be selected manually if autodesign is also selected, otherwise the 'Geometry' property does not appear visible.

**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
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 can modify other properties associated with the member and use the Design Member command to redesign it in isolation (again without having to re-perform the entire structure design).
Check Member and Design Member ignore the autodesign setting of the member. (Check Member always checks the existing section size and Design Member always selects a new size irrespective of whether autodesign is on or off.)

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.
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.
Vibration of Floors to SCI P354

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.

**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
Engineers Handbooks (BS)

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, \( \delta_{\text{PBSS}} \), is derived from the TSD model with no allowance for boundary conditions.

For the secondary beam, the base maximum simply supported deflection, \( \delta_{\text{SBSS}} \), is derived from the TSD model and the maximum deflection for a fixed end condition, \( \delta_{\text{SBFE}} \), is calculated from,

\[
\delta_{\text{SBFE}} = \frac{m \times b \times L_{\text{SB}}^4}{(384 \times E_s \times I_{\text{SB}})} + \frac{m \times b \times L_{\text{SB}}^2}{(24 \times G \times A_y)}
\]

Where

\[
m = \text{unit mass in kN/mm}^2
\]

\[
b = \text{secondary beam spacing in mm}
\]
\[ L_{SB} = \text{span of the secondary beam in mm} \]
\[ I_{SB} = \text{the inertia of the secondary beam in mm}^4 \]
\[ E_S = \text{the steel modulus in kN/mm}^2 \]
\[ G = \text{the steel shear modulus in kN/mm}^2 \]
\[ A_y = \text{the major axis shear area in mm}^2 \]

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
\[ m = \text{unit mass in kN/mm}^2 \]
\[ L_{\text{Slab}} = \text{span of the slab in mm} \]
\[ I_{\text{Slab}} = \text{the inertia of the slab in mm}^4/\text{mm} \]
\[ E_C = \text{the dynamic concrete slab modulus in kN/mm}^2 \]
\[ = E_s \times 1.1/\alpha_{\text{short}} \]

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,
\[ \delta_{\text{barSS}} = \text{MIN} \left[ \frac{0.4 + k_M/k_S \times (1 + 0.6 \times L_S^2/L_M^2)}{1 + k_M/k_S}, 1.0 \right] \times \delta_{\text{SS}} \]

Where
\[ k_M = \text{the 'stiffness' of the critical span selected by the user (primary or secondary beam as appropriate)} \]
\[ = I_M/L_M \]
\[ k_S = \text{the stiffness of the adjoining span selected by the user (primary or secondary beam as appropriate)} \]
\[ = I_S/L_S \]
\[ L_M = \text{the span of the critical span selected by the user (primary or secondary beam as appropriate)} \]
\[ L_S = \text{the span pf the adjoining span selected by the user (primary or secondary beam as appropriate)} \]
For 'three span continuous' the adjusted deflection is taken from P354 as,

\[ \delta_{\text{barSS}} = \min \left( \frac{(0.6 + 2 \times k_M/k_S \times (1 + 1.2 \times L_s^2/L_M^2))/(3 + 2 \times k_M/k_S)}{1.0}, 1.0 \right) \times \delta_{\text{SS}} \]

Where

- \( k_M \) = the 'stiffness' of the critical (middle) span selected by the user (primary or secondary beam as appropriate)
  \[ k_M = \frac{I_M}{L_M} \]
- \( k_S \) = the stiffness of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)
  \[ k_S = \frac{I_S}{L_S} \]
- \( L_M \) = the span of the critical (middle) span selected by the user (primary or secondary beam as appropriate)
- \( L_S \) = the span of the adjoining (outer) span selected by the user (primary or secondary beams as appropriate)
- \( I_M \) = the inertia of the critical (middle) span selected by the user (primary or secondary beam as appropriate)
- \( I_S \) = 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

\[ f_{\text{SBmode}} = 18/\sqrt{\delta_{\text{SBmode}}} \]

**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.
\[ \delta_{\text{PBmode}} = \delta_{\text{barPBSS}} + \delta_{\text{SBFE}} + \delta_{\text{SlabFE}} \]

and

\[ f_{\text{PBmode}} = \frac{18}{\sqrt{\delta_{\text{PBmode}}}} \]

**System Frequency**

The natural frequency of the system, \( f_0 \), is calculated from,

\[ f_0 = \text{MIN}\{ f_{\text{SBmode}}, f_{\text{PBmode}} \} \]

**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,

\[ f_{\text{PBSS}} = \frac{18}{\sqrt{\delta_{\text{PBSS}}}} \text{ must be } \geq 3 \text{ else the design Fails} \]

\[ f_{\text{SBSS}} = \frac{18}{\sqrt{\delta_{\text{SBSS}}}} \text{ must be } \geq 3 \text{ else the design Fails} \]

\[ f_{\text{SlabFE}} = \frac{18}{\sqrt{\delta_{\text{SlabFE}}}} \text{ must be } \geq 3 \text{ else the design Fails} \]

**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,

\[ M = m \times L_{\text{eff}} \times S \]

Where

\[ m = \text{the unit mass in kg/m}^2 \]

\[ L_{\text{eff}} = \text{the effective floor length} \]

\[ S = \text{the effective floor width} \]

Where

\[ L_{\text{eff}} = 1.09 \times (1.10)^{n_y-1} \times (E^* L_{SB}/(m^* b^* f_0^2))^{0.25} \text{ but } \leq n_y \times L_y \]

Where
\[ n_y = \text{number of bays (≤ 4) in the direction of the secondary beam span} \]
\[ E_{S_{SB}} = \text{dynamic flexural rigidity of the composite secondary beam (in Nm}^2 \text{ when m is in kg/m}^2) \]
\[ b = \text{floor beam spacing (in m)} \]
\[ f_0 = \text{system, natural frequency from above} \]
\[ L_y = \text{span of the secondary beam (in m)} \]

and
\[ S = \eta^*(1.15)^{n_x-1}*(E*I_{Slab}/(m^*f_0^2))^{0.25} \text{ but } n_x*L_x \]

Where
\[ n_x = \text{number of bays (≤ 4) in the direction of the primary beam span} \]
\[ E_{I_{Slab}} = \text{dynamic flexural rigidity of the slab (in Nm}^2 \text{ when m is in kg/m}^2) \]
\[ f_0 = \text{system, natural frequency from above} \]
\[ L_x = \text{span of the primary beam (in m)} \]
\[ \eta = \text{frequency factor} \]

\[ \eta = \begin{cases} 
0.5 & \text{for } f_0 < 5 \text{ Hz} \\
0.21f_0 - 0.55 & \text{for } 5 \text{ Hz} \leq f_0 \leq 6 \text{ Hz} \\
0.71 & \text{for } f_0 > 6 \text{ Hz} 
\end{cases} \]
Figure 1: Definition of variables used to establish effective modal mass

**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,
\[ V = 1.67 \times f_p^2 - 4.83 \times f_p + 4.5 \]

for \( f_p \) in the range 1.7 to 2.4 Hz

Where

\( f_p \) = 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

\( \zeta \) = the damping ratio

\( L_p \) = the walking distance

\( V \) = 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,

\[ a_{w,rms} = \mu_e \times \mu_r \times 0.1 \times Q \times W \times \rho / (2 \times \sqrt{2} \times M \times \zeta) \]

Where

\( \mu_e \) & \( \mu_r \) = mode shape factors

\( Q \) = the person’s weight taken as 745.6 N (76 kg)

\( M \) = the modal mass (kg)

\( \rho \) = the damping ratio

\( W \) = the appropriate code-defined weighting factor for the human percept of vibrations, based on the fundamental frequency, \( f_0 \)

\[ \begin{align*}
W &= f_0/5 \\
&= 1.0 \\
&= 16/f_0
\end{align*} \]

for \( 2 \leq f_0 < 5 \)

for \( 5 \leq f_0 \leq 16 \)

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,

\[
a_{w,rms} = 2\pi \mu_e \mu_r * 185*Q*W / (M*fo^{0.3}*700*\sqrt{2})
\]

**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,

\[
R_{prov} = a_{w,rms} / 0.005
\]

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,

\[
na = \frac{(1/T_a)*(VDV/(0.68*a_{w,rms}))^4}{T_a} = \frac{L_p}{V} \text{ if } L_p \text{ is known OR }
\]

\[
= \text{ value supplied by user if } L_p \text{ is not known}
\]

\[VDV = \text{ VDV value supplied by user [Typical values shown in table below]}\]
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
Engineers Handbooks (BS)

(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.