Trimble

Tekla Structural Designer 2021
Release Notes

May 2021

Contents

2

1 Tekla Structural Designer 2021 release notes

Welcome to Tekla Structural Designer 2021!

Select your headcode/region from the list below to see the many new features and improvements that apply to you in this version:

1.1 US

General & Modeling

- New Embodied Carbon Calculation & Design Efficiency Assessment (page 9)
- New Fireproofing (page 25)
- New Automatic Backups (page 29)

Loading

- New Open Structure Wind Load (page 32)
- New Equipment Load Entities (page 37)

Analysis & Results

- New Partially Cracked Concrete Walls & Members (page 43)

Design

- Issues with associated bulletins (page 9)
- New Steel Column Base Plate Connection Modeling & Design - Eurocode and USA Head Codes (page 47)
- Enhanced Sway/ Drift/ Wind Drift & Seismic Drift Check Control and Reporting - All Head Codes (page 60)
- New Reinforced Concrete Design to ACI 318-2019 - US Head Code (page 72)

Other Enhancements & Fixes

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.2 Eurocode

General & Modeling

Loading

Analysis & Results

Design

Other Enhancements & Fixes

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.3 BS

General & Modeling

- New Embodied Carbon Calculation & Design Efficiency Assessment (page 9)
- New Fireproofing (page 25)
- New Automatic Backups (page 29)

Loading

- New Equipment Load Entities (page 37)

Analysis & Results

- New Partially Cracked Concrete Walls & Members (page 43)

Design

- Enhanced Sway/ Drift/ Wind Drift & Seismic Drift Check Control and Reporting - All Head Codes (page 60)
- Foundation Design - New Lateral Load Check for Pile Caps and Piled Mats (page 76)

Other Enhancements & Fixes

- Other Enhancements and Fixes (page 79)

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.4 India

General & Modeling

- New Embodied Carbon Calculation & Design Efficiency Assessment (page 9)
- New Fireproofing (page 25)
- New Automatic Backups (page 29)

Loading

- New Open Structure Wind Load (page 32)
- New Equipment Load Entities (page 37)

Analysis & Results

- New Partially Cracked Concrete Walls & Members (page 43)

Design

Other Enhancements & Fixes

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.5 Australia

General & Modeling

Loading

Analysis & Results

Design

Other Enhancements & Fixes

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.6 Compatibility

We suggest that you complete any unfinished models using your current version of Tekla Structural Designer.

This version is not backwards compatible. When you create or save a model in Tekla Structural Designer 2021, you cannot open it in older versions due to database differences.

We suggest that Tekla Structural Designer 2021 is only installed on systems meeting these hardware recommendations.

If you are upgrading from a version earlier than the latest release 2020 SP6 (version 20.0.6.29 released Jan 2021) you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

1.7 Issues with associated bulletins

- [TSD-8930] - Composite Beam Design - US Head Code - this issue relates to the design of composite beams for the United States (AISC/ACI) Head code and the AISC 360/341 resistance code. In some circumstances an incorrect overall pass design status could be reported when the Connector Resistance check governed design and reported a warning or fail status. It is believed that the latter would be a relatively rare occurrence since, for this to happen, the issue described in bulletin PBTSD-2102-02 must also occur. For more information please see Product Bulletin PBTSD-2102-01.
  - This issue is fixed in this release.

- [TSD-8759] - Composite Beam Design - US Head Code - this issue relates to the design of composite beams for the United States (AISC/ACI) Head code and the AISC 360/341 resistance code. In some circumstances an incorrect pass status could be reported for the Flexure check when the Connector Resistance check failed. When the related issue detailed in bulletin PBTSD-2102-01 also occurred, an incorrect overall passing status could potentially be incorrectly displayed. However, it is believed that this would be a relatively rare occurrence. For more information please see Product Bulletin PBTSD-2102-02.
  - This issue is fixed in this release.

1.8 New Embodied Carbon Calculation & Design Efficiency Assessment

With an exciting further development of recent features to assess the Carbon Impact of design, this release adds an automated and comprehensive new built-in calculation of *Embodied Carbon* quantities, together with powerful new options for graphical review/ optimization and reporting of this aspect of design. For information about recently added related features see the 2020 SP6 release note sections Review View - Design Status Enhancements and Material List Enhancements (material quantity improvements detailed in this are directly used in the new Embodied Carbon calculation) and the help topic on the Export to One Click LCA feature (released in 2020 SP5).

For more background on Embodied Carbon calculation see this Institute of Structural Engineering guide "How to calculate embodied carbon". With reference to this, the new calculation in Structural Designer essentially deals with stages A1 to A3 for the construction products in a project (commonly termed the 'Product stage" or "cradle to gate" stage), over which the design engineer has direct influence and which typically account for some 50% of all Embodied Carbon in a building project life cycle. The picture directly below shows an example of the new graphical review capability for Embodied Carbon together with just a small sample of the reporting options; the **Overview** report giving totals for the entire model (both by entity and grouped by plane) and just part of a **Detail** report for steel composite beams.

Here, and also in this related video, we will highlight some key aspects of the settings and workflow for this feature.

Embodied Carbon for Member : All
kgCO$_2$e/m
Show highest 50.00% embodied carbon
Show utilization below 0.700

- ■ 260.7 (3)
- ■ 211.8 (14)
- ■ 173.0 to 179.4 (6)
- ■ 154.8 (6)
- ■ 147.1 (5)
- ■ 134.9 to 139.5 (7)
- ■ 127.7 (5)
- ■ 113.2 to 117.6 (9)
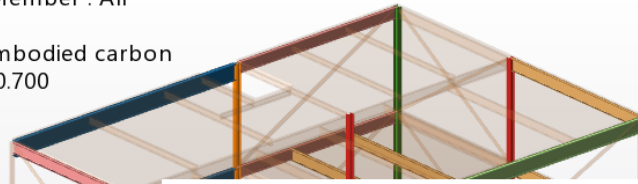
Embodied Carbon Overview

Construction Type ▼	Embodied Carbon Mass [kgCO$_2$e]
Steel Columns Rolled	14144
Steel Braces Rolled	3438
Steel Beams Rolled Non-Composite	64946
Steel Beams Rolled Composite	8876
Composite Slabs	54527
Total	**145932**

Embodied Carbon Overview

| Reference	Level [m] ▼	Embodied Carbon Mass [kgCO$_2$e]	Floored Area [m²]	Embodied Carbon Mass per Floored Area [kgCO$_2$e/m²]
St. 3 (Roof)	12.000	38580	223	173.198
St. 2 (2nd Floor)	8.000	44885	223	201.503
St. 1 (1st Floor)	4.000	44885	223	201.503
St. Base (Base)	0.000	0	0	-
FRM 10	-	0	-	-
FRM 11	-	0	-	-
Other	-	17582	-	-
Total	-	**145932**	-	-

Embodied Carbon Detail

Member Reference	Material	Quantity	Unit	Carbon Factor	Embodied Carbon Mass [kgCO$_2$e]
SB 3/A/3-3/3/Ba - 1	Steel	862.91	kg	UK rolled open sections: British Steel EPD (2.450)	2114
SB 3/A/3-3/3/Ba - 1	Connection End 1	21.57	kg	UK rolled open sections: British Steel EPD (2.450)	53
SB 3/A/3-3/3/Ba - 1	Connection End 2	21.57	kg	UK rolled open sections: British Steel EPD (2.450)	53
SB 3/A/3-3/3/Ba - 1	Coating	19.4	m²	Standard steel coating (0.200)	4
SB 3/A/3-3/3/Ba - 1	Shear Connectors	66.00		Basic Steel Connectors (1.500)	99
SB 3/A/3-3/3/Ba - 1	Reinforcement class 500	0.00	kg	Reinforcement UK CARES Sector Average EPD (0.760)	0
SB 3/3/Ba-3/C/3 - 1	Steel	345.24	kg	UK rolled open sections: British Steel EPD (2.450)	846
SB 3/3/Ba-3/C/3 - 1	Connection End 1	8.63	kg	UK rolled open sections: British Steel EPD (2.450)	21
SB 3/3/Ba-3/C/3 - 1	Connection End 2	8.63	kg	UK rolled open sections: British Steel EPD (2.450)	21
SB 3/3/Ba-3/C/3 - 1	Coating	10.1	m²	Standard steel coating (0.200)	2
SB 3/3/Ba-3/C/3 - 1	Shear Connectors	22.00		Basic Steel Connectors (1.500)	33
SB 3/3/Ba-3/C/3 - 1	Reinforcement class 500	0.00	kg	Reinforcement UK CARES Sector Average EPD (0.760)	0
SB 3/3/Ba-3/4/Ba - 1	Steel	179.97	kg	UK rolled open sections: British Steel EPD (2.450)	441

- **Embodied Carbon Factors** (ECF) - at the heart of this new feature is the Embodied Carbon Factors (ECF) definitions dialog shown below, which lists the factors and controls how they are assigned to the entities in the model. The ECF value is the key value required for the calculation, being the mass of embodied carbon per unit mass or area of a material or surface e.g. kgCO$_2$e/kg. The ECF dialog is accessed from the new "Embodied Carbon Factors" button on the Home tab of the Ribbon. Assignment of ECFs to the model entities and the resulting Embodied Carbon calculation is entirely automatic and updates in real-time following edits to the list/ factors. Automated ECF assignment is controlled first by the order of the factor list then by the criteria set for each individual factor.
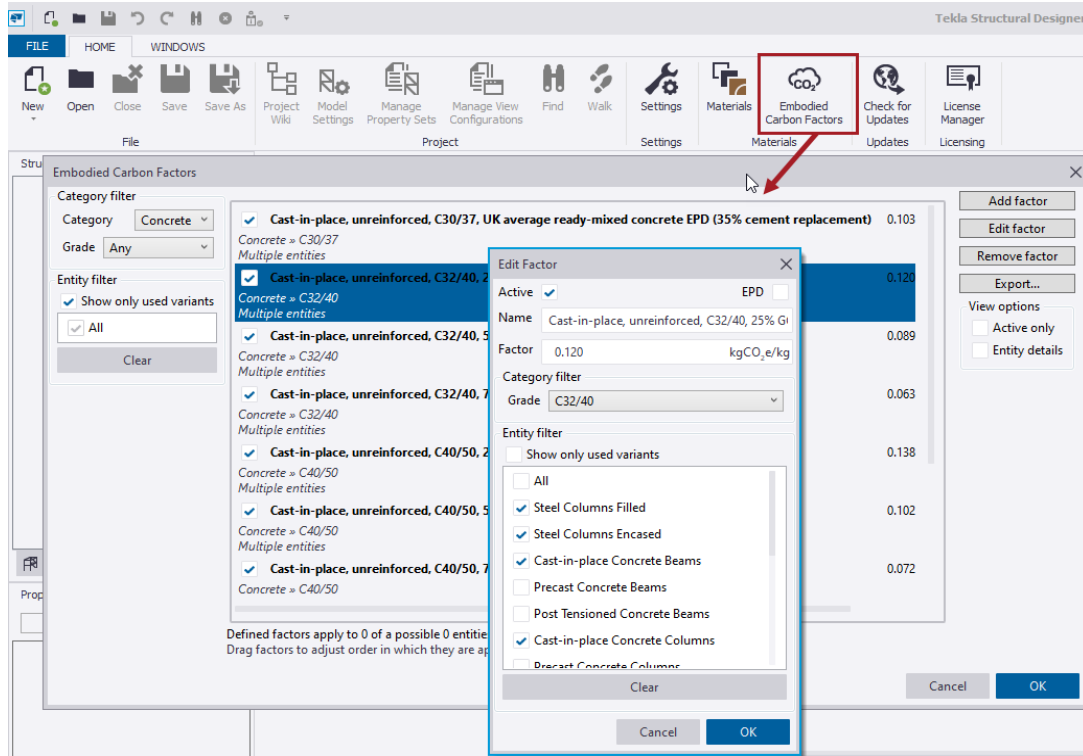
- This feature is applicable to all Head Codes & Countries. Some sample ECF definitions have been added to the Settings sets for all countries/ codes for the following categories in general; Concrete, Metal Deck, Reinforcement, Steel, Timber. Note the following:

  - We would note that there is potentially considerable variability/ uncertainty in ECFs - for example for Concrete, many mix variations are possible for every strength grade and all could be unique. Hence the sample ECF's are included principally for guidance and are not a complete set - the engineer is advised to review, edit and add to them as they need. To establish ECF values you can for example consult an *Environmental Product Declaration* (EPD) where available from construction product suppliers/ manufacturers.

  - **Existing Installations** - for an existing installation, your current settings sets (from the latest currently installed version) will be copied to this release. Hence, to use the sample ECFs, it is necessary to first import the new 2021 Settings sets for your preferred country/ countries into global settings. New models created with these new settings set(s) active will then contain the new ECF definitions. For existing models the

ECFs can be loaded via Home > Embodied Carbon Factors > Load...
(from the currently active global settings).

- If you wish to add the ECF definitions to your own settings set(s), you can do this as follows:

    - Step 1 - import a new 2021 settings set, set it to be Active, OK this then create a new model - this will contain the sample ECF definitions.

    - Step 2 - open Home > Settings and set your settings set to be Active.

    - Step 3 - open the ECF dialog and click the [Save...] button, you can then confirm the load the ECF definitions into your settings set from the model as shown in the picture below.
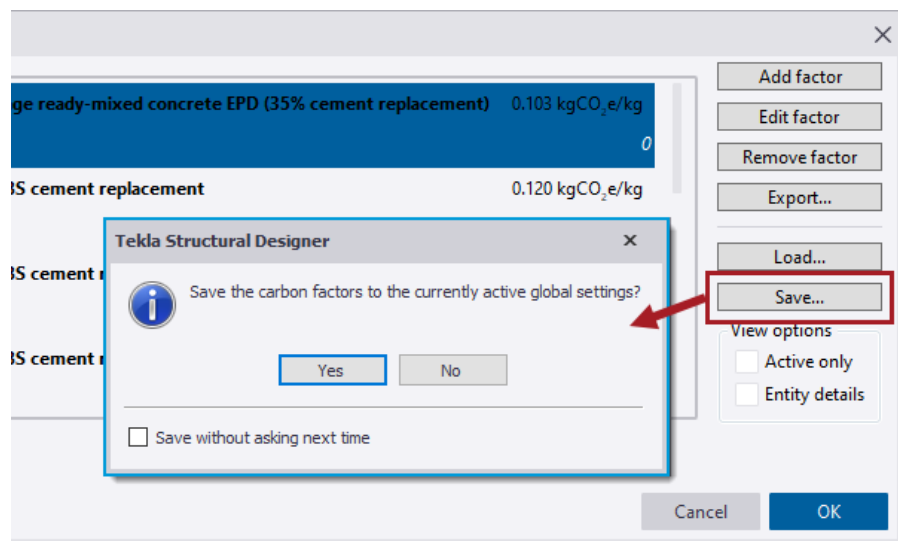
- **Steel Connections Allowance** - note that there is also a new model setting for the Embodied Carbon calculation to make an allowance for the additional amount within steel connections. This is found in both Global and Model Settings > Embodied Carbon > Steel connections. There are separate settings for the allowances for Pin-ended (default 5%) and Fixed-ended connections (default 10%).

- **ECF Definition & Assignment** - key aspects of the ECF definition and assignment process are:

    - Open the ECF dialogue to review the current assignment of ECFs which is constantly active, starting at the top of the list of ECFs and working down, so the top factor has first priority and so on. The list order can be changed simply by dragging and dropping factors with the mouse. The assignment of factors updates automatically as edits are made to the list order, and the number of entities to which an ECF is assigned to is

given at the right of each factor entry - e.g. 60 for the top "UK rolled hollow sections..." ECF shown in the picture below.

- Once an entity in the model has a factor from the list assigned the process stops for that entity <u>and category</u>, hence there is no duplication of factor assignment within a category where lower (in the list) ECF definitions are also applicable.

   - It is important to note here that there is not a one-to-one relationship between model entities and ECFs - for example a reinforced concrete member will have different ECFs assigned for both its concrete material and its rebar.

- Factors can also be activated/ inactivated via the checkbox at the left of each factor entry.
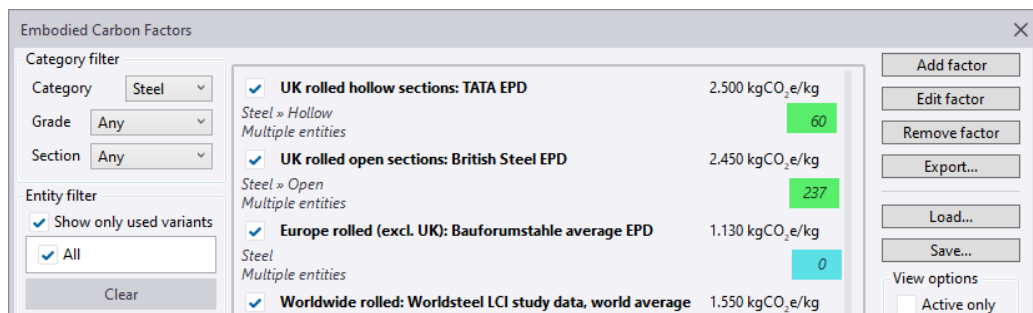
- To use the ECF dialog you first select the main Category you wish to view/ edit at the top left e.g. "Steel" in the picture above. You can then review for this category the ECF list order, description (which can be any variation of text and numbers etc)value, and the number of assigned entities where applicable.

   - The list of factors in the main dialog can be further controlled by additional filters below the main Category, those available being dependent on the selected category - e.g. by a Grade filter for the material categories and/ or by a Country filter for categories such as Reinforcement and Metal deck.

   - Next, via the Add/ Edit factor options, you can review/ edit the description and/ or value for ECFs and add additional ones. You can also set the Category and Entity *filters* which control the grades and/ or entity types to which the factor applies. For example you could

set a factor for the Concrete category that applies only to the specific grade C32/40 and only concrete columns and beams.

- The category and entity filters available in the Edit Factors dialog are dictated by the main Category filter set at the top left of the dialog - e.g. "Concrete", "Steel", "Cladding" etc. Thus for example only steel entity types are listed in the Edit Factor dialog's Entity filter list when "Steel" is set as the main dialog Category.

- **Review, Adjustment & Reporting** - while the key result of focus is the total Embodied Carbon quantity for the whole model (as shown in the Embodied Carbon Overview Table in the picture above), the engineer will also likely wish to drill down to see how this is calculated and identify areas/objects with high carbon impact that can potentially be improved. A good deal of the development focus for the new features has gone into this aspect. The engineer can review the Embodied Carbon results as follows:

  - **Graphical Review/ Adjustment** - Review View > Show/Alter state now has powerful new commands to rapidly review and edit the embodied carbon result and settings; "Carbon Factors" and "Utilization and embodied carbon". These are activated via the new "Embodied Carbon" button in the Show/Alter state group of the Review ribbon and then controlled by their Properties Window settings.
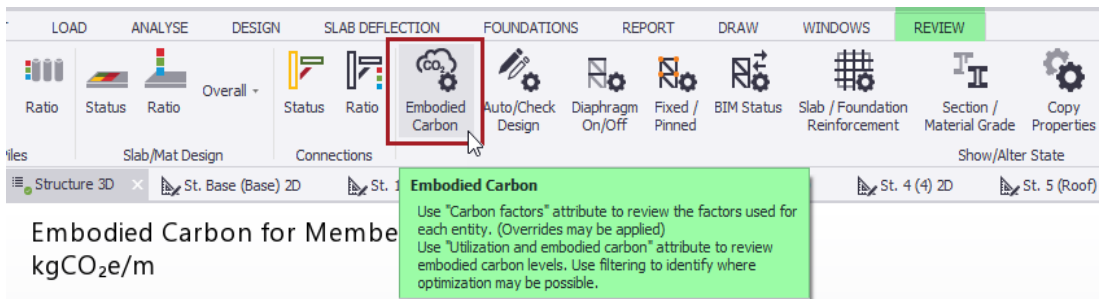
  - **Carbon factors** - the new Carbon Factors attribute has both Review and Override modes. In the Review mode you can see the distribution of carbon factors throughout the model via a colour coded view. The view legend lists the factor values with the number of entities for each being shown adjacent to the factor value in brackets, as shown in the picture below. Here you can see that different factors are assigned based on the steel section shape; the hollow sections having a slightly higher ECF of 2.5 than the open

sections (see picture of ECF's above). The cursor tooltip also lists the ECF value for a selected member.

- Note also the Carbon Source category and Entity type filters which allow you to restrict the view to specific categories/ entity types e.g. Steel Members as shown in the view below.
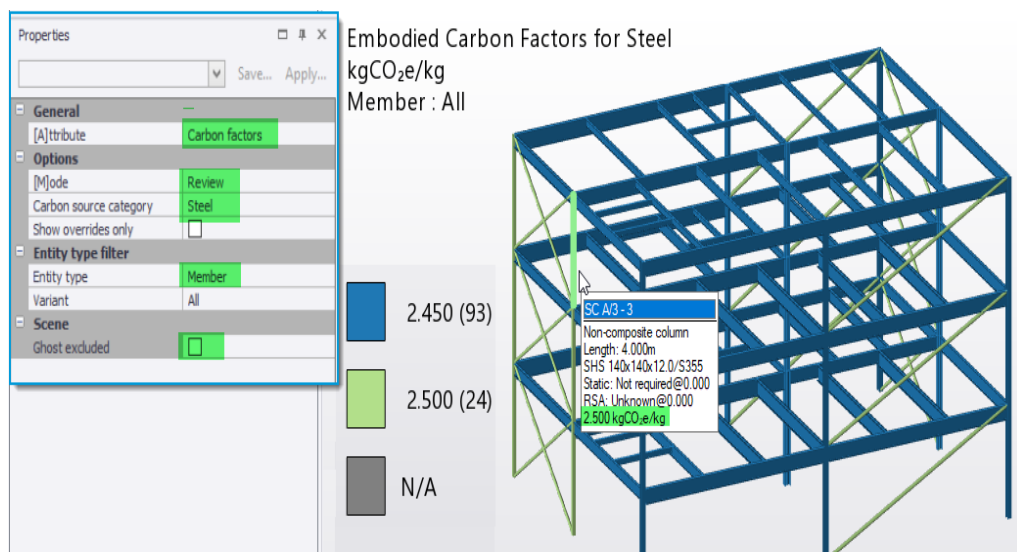
- You can also use the "Set override" mode to directly apply carbon factors as shown in the picture below - the legend and color coding adjust appropriately to the current override value to show entities with values above/ below it, with the override value set and with other override values > and < this. Overrides can also be removed with the "Remove override" mode.

  - Note that we recommend Set override is used to make limited changes to ECFs rather than for the majority of the model - managing and application of ECFs is most effectively done via ECF definitions which are automatically assigned for you as discussed above. Setting overrides is primarily recommended for entities that require different factors but are indistinguishable in terms of their data (and so cannot be assigned automatically) such as Roof/ Wind panels (in the "Cladding" category).

- Overrides also appear in the ECF dialogue as 'Custom' values where they can be edited or deleted. Deletion removes the override for all entities linked to the factor.
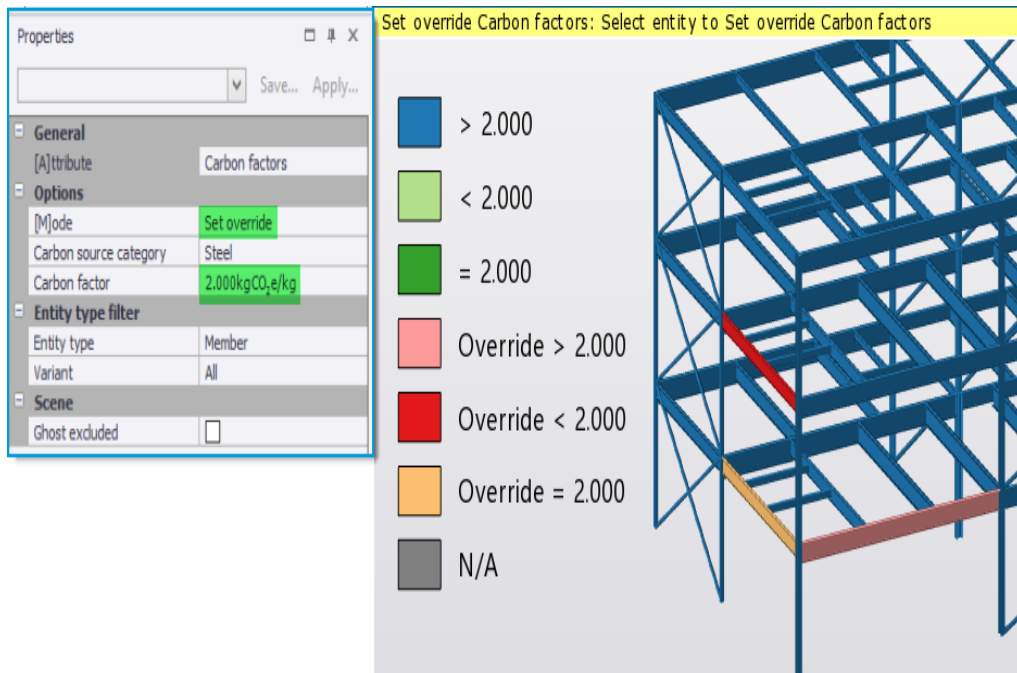
- **Utilization and embodied carbon** - this mode enables you to see the distribution of the amount of embodied carbon for each entity. Again you can use the category and entity type filters to tailor the view to what you want to focus on - e.g. Reinforcement in members as shown below.
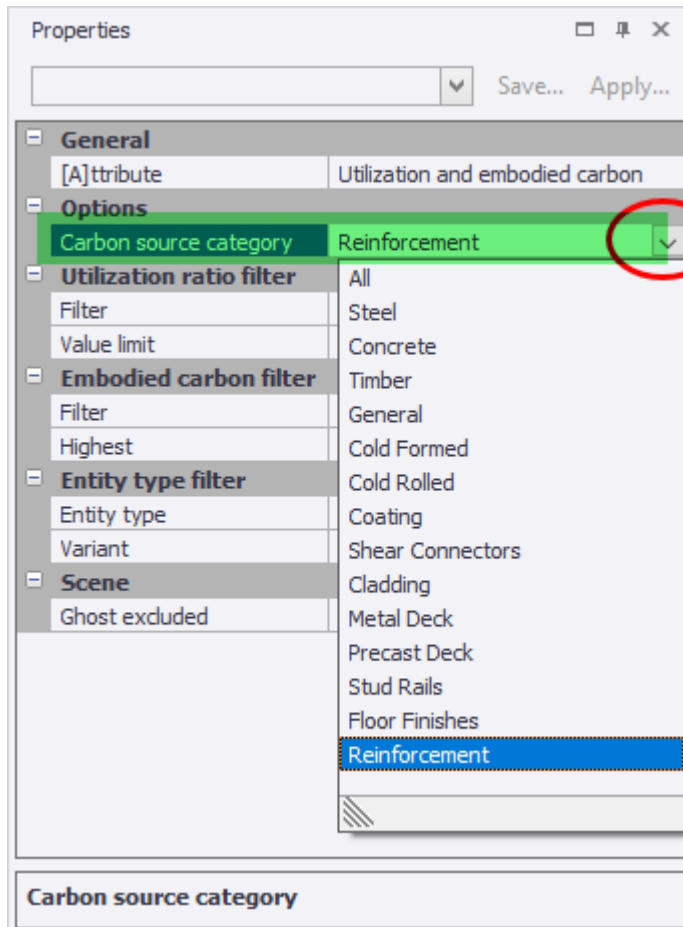
- It is important to note here again, as we did earlier, that there is not a one-to-one relationship between entities and ECFs - for example a reinforced concrete member will have different ECFs assigned for both its concrete material and its rebar, both of which will contribute to the member's total embodied carbon quantity. Hence the facility to restrict the view to specific categories.

Embodied Carbon for Member : All
kgCO₂e/m
Reinforcement

■	24.7 to 45.9 (14)
■	19.7 to 20.9 (20)
■	17.6 to 17.9 (24)
■	17.5 (48)
■	13.8 to 15.7 (10)
■	12.0 to 13.1 (14)
■	10.9 to 11.3 (18)
■	9.9 to 10.6 (28)
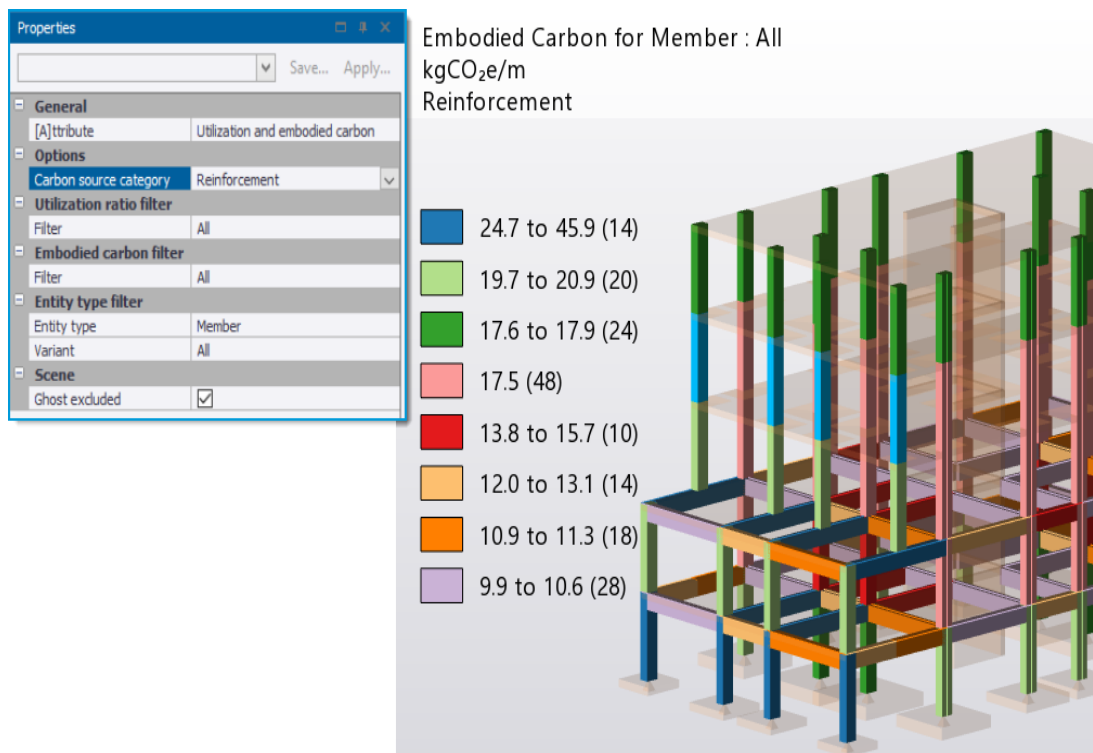
- The Utilization and embodied carbon attribute also features additional filters for Utilization ratio (UR) and Embodied carbon, both of which allow you to set thresholds for the visible entities; either above/ below an absolute value or a highest or lowest % proportion. You can use these filters singly, or simultaneously for example to focus on entities with both a low UR and high embodied carbon by setting the UR filter to "Value below" with a value limit = 0.5 and the Embodied carbon filter to "Highest X%" and Highest = 50%, as shown in the picture below.

- **Tabular Review & Reporting** - a comprehensive set of tables for the Embodied Carbon results is added in Review View > Tabular Data. These give both a brief overview and summary of model totals and a potentially highly detailed table for individual entities allowing the engineer to dig right into the details of the calculations. The new report tables ``Embodied Carbon Overview" and "Embodied Carbon Details" are available from the drop list of tabular results at the top left of the Review Data Ribbon tab, as shown in the picture below. Embodied

Carbon Mass values are also included in a new column added to every material list (Including the associated report item).

- Filtering - as for the existing tabular data reports, you can control the new table content using the Ribbon filter buttons for Material Type, Characteristic and Construction etc.

- Sorting - the tables also feature automated sorting by selected column header - simply click the column header you wish to sort by, then click it again to reverse the sort order, or click another header to sort by that.

- Reporting - for rapid reporting, the currently configured table can also be exported directly to Excel by clicking the Export Excel Ribbon button. The Overview tables have associated report items and so can also be included in a Report (see below).

- Locating - as with the Material List tables, you can locate in the model the entity or entities associated with a row in the tables simply by double-clicking the row. This will automatically activate the Structure 3D view, then select, highlight and arrow the associated entity/ entities. Together with the filtering and sorting facilities this enables the engineer to rapidly home in on and locate entities with high Embodied Carbon.

- **Embodied Carbon Overview** - this features two additional button controls below the drop list "Group by Plane" and "Toggle Selection". With *Group by Plane* active, Embodied Carbon totals are given for each construction plane (commonly the floors). When off the total are grouped by Characteristic e.g. Slabs, Beams, Columns etc (see the first picture in this section above for how this looks). *Toggle Selection* turns on/off all of the Ribbon filter buttons with a single click, so the engineer can quickly set them all on to see the full model results, or set them all off to begin a specific filter such as Material = Concrete and Characteristic = Beams.

- **Embodied Carbon Detail**- this table has two levels controlled by the "Summary Only" ribbon button which works in a similar manner to the Material List tables;

   - "Summary Only" on (the default) gives a brief summary table giving the Embodied Carbon Mass quantity for each entity together with their reference (for continuous beam/ columns/

walls a reference and value are given for each span/ stack/ panel).

- Set the button off to see the Detailed level which gives a much higher level of detail for every entity included in the table, including Embodied Carbon quantities for the individual parts and surfaces of each entity appropriate to the entity type. So e.g. for concrete beams, for each span it gives quantities and Embodied Carbon values for; Concrete, Coating and Reinforcement. For steel composite beams it gives values for; Steel, Coating, End 1 & 2 Connections (allowance), Shear Connectors and (Transverse) Reinforcement.

Double-click row to locate in Structure 3D View

- **Reports** - new report chapters are added for the Embodied Carbon factors and both Overview result tables options of by Plane and Construction Type

discussed above. The Overview tables are also included in a new default Embodied Carbon report style (for new models started in this release).

Tekla Structural Designer

Project		Job Ref.	
Structure		Sheet no.	
			Page1
Calc. by	Date	Chk'd by	Date	App'd by
jekni	01/03/2021		01/03/2021	

Embodied Carbon Overview by Plane

Reference	Level [m]	Embodied Carbon Mass [kgCO$_2$e]	Floored Area [m^2]	Embodied Carbon Mass per Floored Area [kgCO$_2$e/m^2]
St. 5 (5)	20.000	45547	384	118.611
St. 4 (4)	16.000	45548	384	118.615
St. 3 (3)	12.000	45548	384	118.615
St. 2 (2)	8.000	45548	384	118.615
St. 1 (1)	4.000	45548	384	118.615
St. Base (Base)	0.000	0	0	
FRM 3		8334		
FRM 4		8334		
FRM B		5573		
FRM C		5575		
Other		25506		
Total		281060		

Embodied Carbon Overview by ConstructionType

Construction Type	Embodied Carbon Mass [kgCO$_2$e]
Cast-in-place Concrete Slabs/Mats	179898
Cast-in-place Concrete Beams	47842
Cast-in-place Concrete Walls	27815
Cast-in-place Concrete Columns	25506
Total	281060

Further details on the above topics are also given in the new Help topic Embodied carbon workflow.

1.9 New Fireproofing

Fireproofing is the application of fire resistant insulation material to the surface of structural members (as illustrated in the picture below) in the form of; protective paints, thin/thick film coating sprays, fire boards etc. This release adds the facility to apply such fireproofing to all 1D members of the model with the definition of its distribution, thickness and density. The fireproofing increases both the weight and surface area of members. The additional member self-weight is automatically calculated, reported and included in the

analysis, and the resulting enlarged surface of the member is also considered by the new Open Structure Wind Loading (page 32).

For the Eurocode Head Code, the fire resistance check settings are now incorporated into those for Fireproofing - for more about the fire resistance check see this Help Topic and the release note Steel Design - New Fire Resistance Check to Eurocode EN 1993-1-2, Cl 4.2.4 - All NA's.

- A new Fire proofing Property settings group is added as shown below left. For members with fireproofing applied, enable the "Protected" option and you can then specify all the required fireproofing details of; Shape (Contour or Hollow encasement as illustrated below right - Contour on the left and Hollow the right), thickness and density. The new properties are available for all 1D member types/ materials (with the exception of; analysis elements, DELTABEAM and FABSEC Beams)

  - For the Eurocode and steel beams, if you also enable the fire resistance check - which is optional - you can also set the Thermal conductivity and Specific heat values used by this.

  - Note that the calculated additional self weight is displayed at the bottom of the new properties. This updates automatically to reflect changes to the section size and/ or fireproofing values.
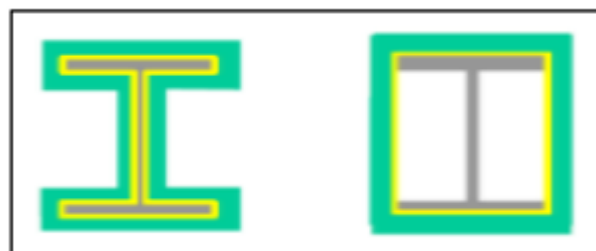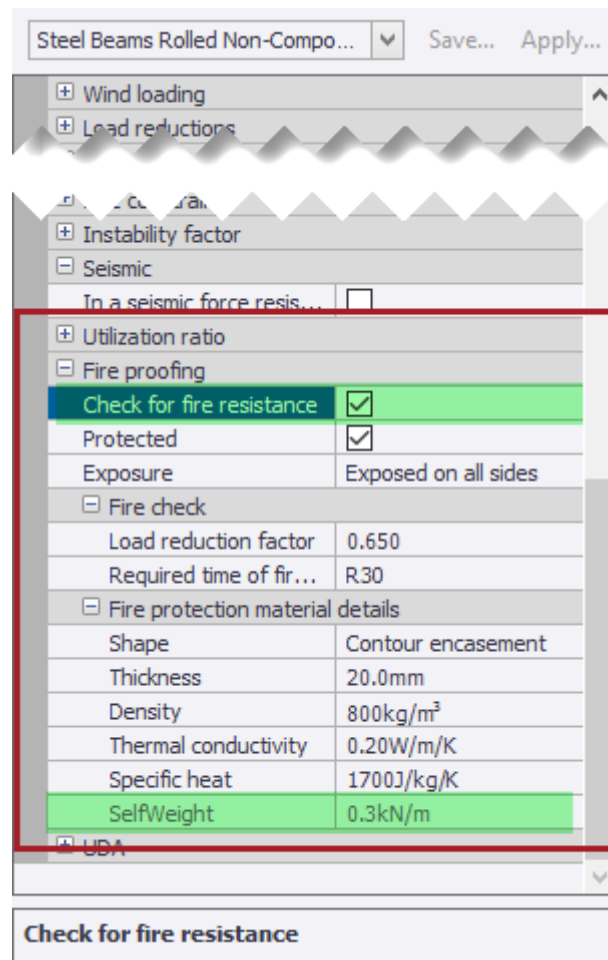
- Review View - a new "Fire proofing" Attribute is added to the Review View > Show/Alter State options, enabling rapid graphical review and editing of the Fire proofing and resistance check settings.

  - Review Mode - in this mode you can see a color-coded view + legend showing the distribution of fireproofing and fire resistance check settings throughout the model, as shown in the picture below.
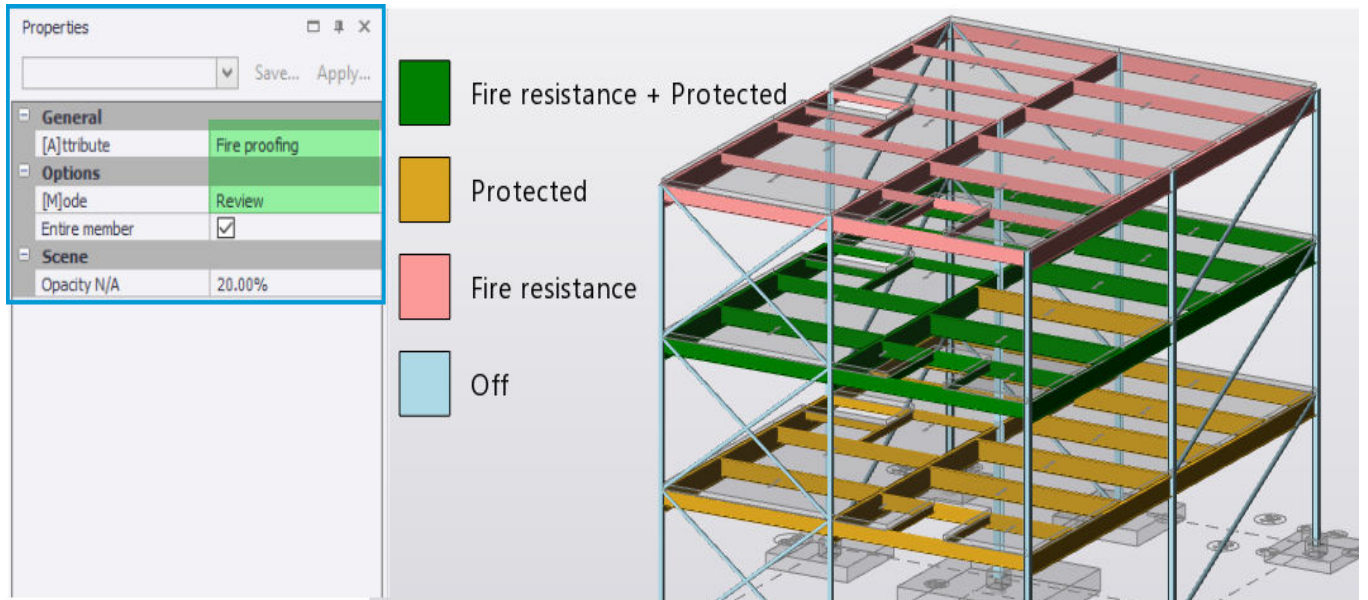
- Toggle/ Set On/Off Modes - using these you can quickly Toggle On/Off either fire protection or the fire resistance check using the mouse for single or multiple entities. In Set On mode for "check for fire resistance" you can set on the check and optionally set the exposure level (all sides or 3 sides) while for "Protected" you have options to specify all the protection properties of; shape, thickness and density.
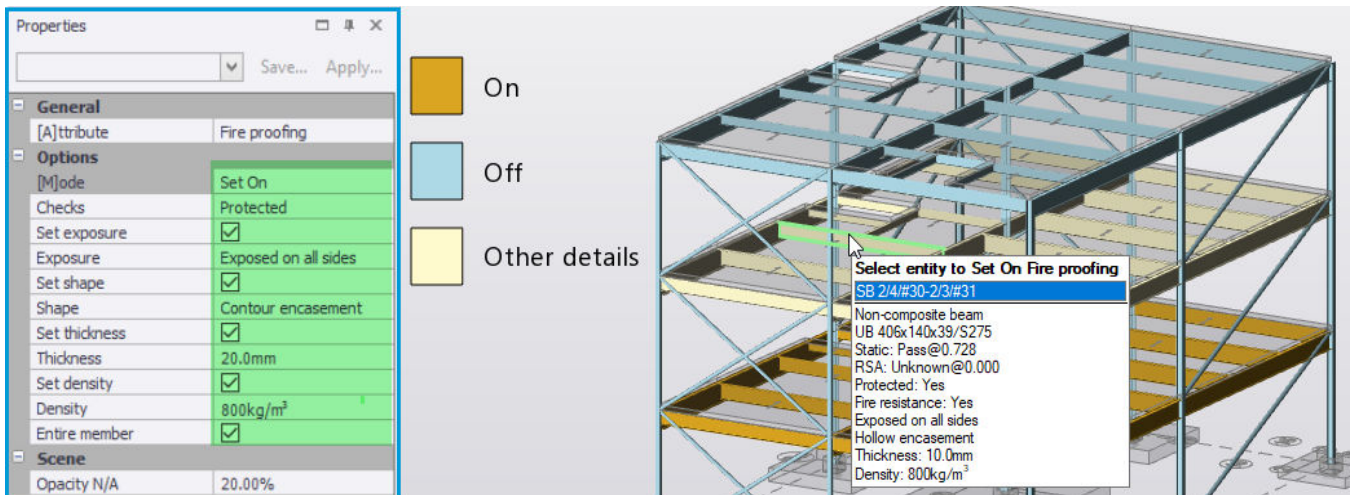
1.10 New Automatic Backups

While Tekla Structural Designer has always had an *Autosave* feature which provided additional file security to cater for unexpected PC/ program problems this had some limitations. These are addressed in this release with the addition of a comprehensive new Backup facility. This ensures that, whatever happens, there can be previous versions of the model to revert to. Another benefit is an automatic history of the model (limited by the configuration settings - see below).

- The Backup configuration settings are in the Home > Settings > Autorecover group as shown in the picture below. Via these you can:

  - Enable/ Disable the Backup process via the **Enabled** check box - default = enabled.

  - Set the time **Interval** for backup creation - default = 30 minutes

  - Define the backup **Location** directory - the default is your Windows user profile Documents directory. Click the link to browse to your preferred location (both local and network locations are allowed).

  - Specify the **Maximum number of backups per project** - default = 10 (when the limit is reached the older backups start to be automatically deleted).
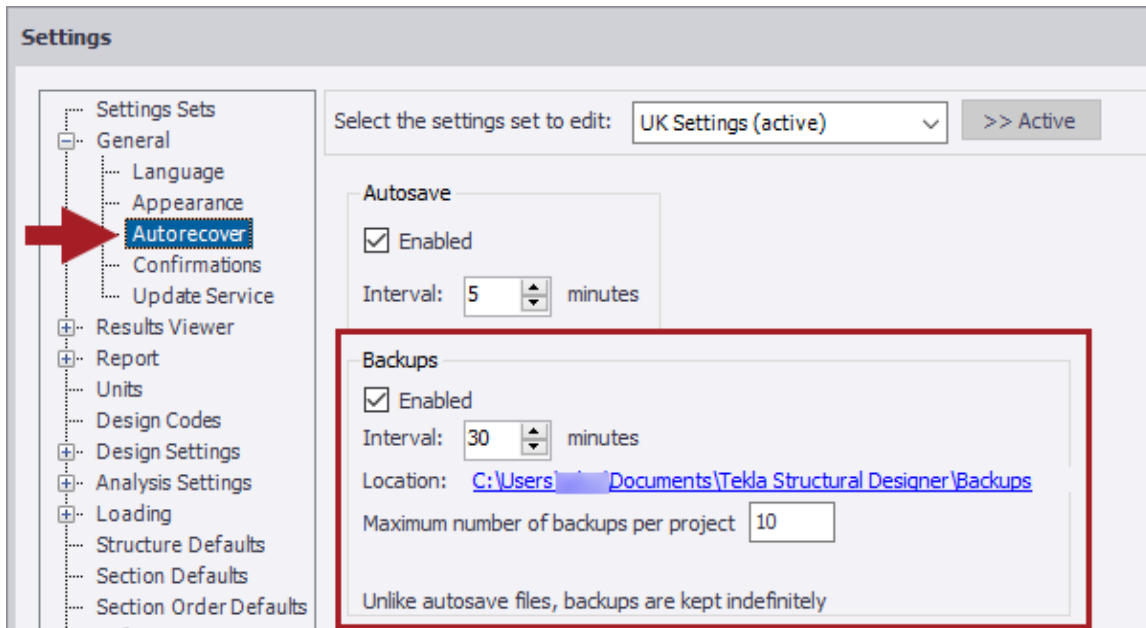
- Backup File List - the list of backup files for the currently opened model is accessed from the new "Model Backups" option in the File menu. This opens a new dedicated Model Backups dialog listing all the current backup files.
  - Note that it includes the unique *Model ID* which is used for the model's specific backup directory name.
  - Click the Backup path to open the backup directory in Windows Explorer.
  - The date and time of each backup file is added to the end of the file name. The backup files are saved with read-only permission
    - Similar to autosaves, backups are core saves of the model - they are only created if, at the end of an interval, there have been changes since last backup.

Click path to open backup directory in Windows Explorer
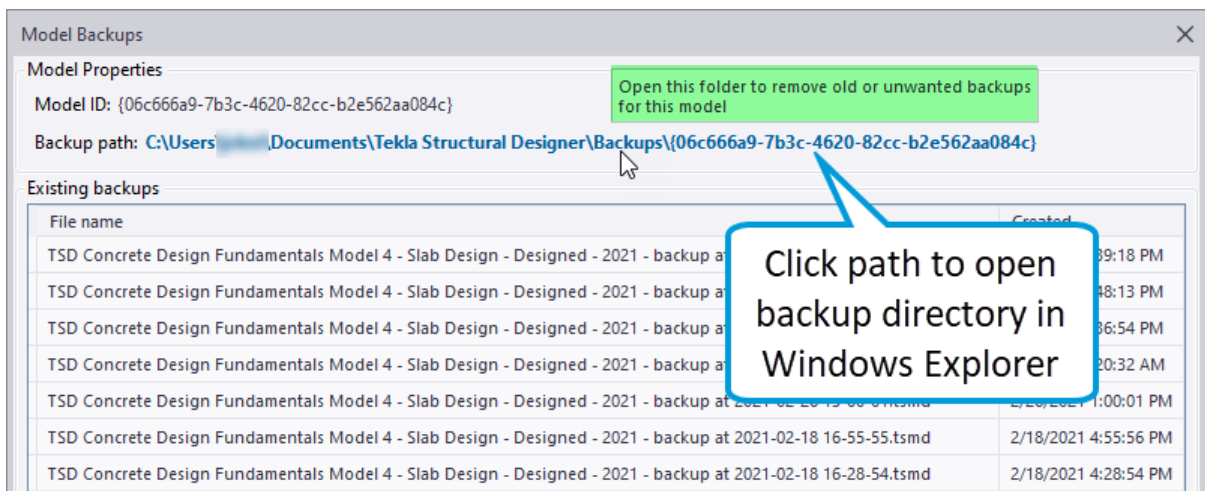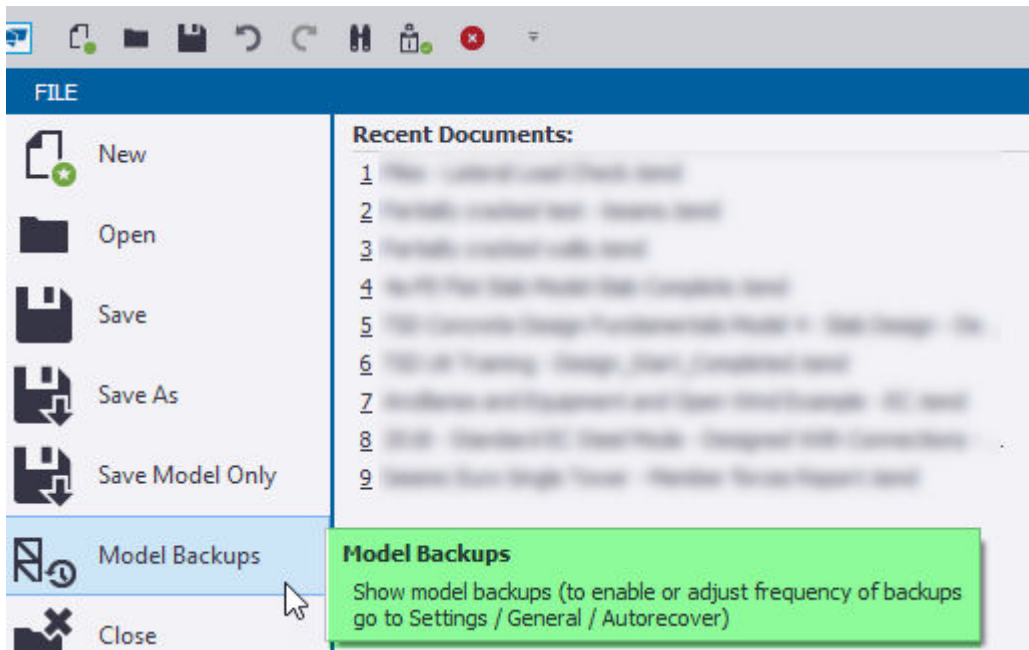
- Using Backups for Recovery:

  - Manual Recovery - this can be accomplished from the Model Backups dialog or directly from Windows Explorer for numerous eventualities/ reasons.

  - Handled Exceptions - in a process that has been in place for some time, when a program error occurs a dialog is displayed when possible,

prompting the user to select from a list of file Revert options - these depending on when the error occurs and what previously saved/autosaved files exist. Backup files can now be selected for reversion in this eventuality:

• Where they exist, Backup files are now added to the list of options via a new "Revert to automatic back up this model" radio button option, this includes a drop list of existing backup files from which you can select the preferred file.

1.11 New Open Structure Wind Load

This new feature adds a quick and effective way to apply wind load especially for Industrial Structures which typically are not clad, though it can be used for any un-clad structure or part. Application is simple and straightforward - the existing Wind Wizard is used in essentially the same manner as currently, other than there being no requirement to 'clad' the model with wall and roof panels. The feature can also be used in conjunction with wind wall and roof panels and has a host of user-defined settings, making it very flexible and applicable to a wide range of structures and geometries. The method of calculation for open structure wind loads we have developed follows both general principles and guidance from the United States Loading code ASCE7 and the ASCE document '*Wind Loads For Petrochemical And Other Industrial Facilities*' (adapted for all loading codes). For more information on using the feature and its load calculations, see the new Help Topic .

As shown in the picture below, the applied loads can be reviewed both via the decomposed loads in the Structure View (enable Text for Loading > Decomposed in the Scene Content settings to see values) and in the Wind View (for a selected direction and loadcase). In the Wind view, the principal wind load values of shape factor, wind pressure and applied wind force are reported in the cursor tooltip of a selected entity. All the applied loads are fully considered in analysis and design. For detailed information about how the loads are calculated see the new Help Topic .

Wind View

Select Entity
SB 3/B/1-3/B/2 - 1
Non-composite beam
Length: 8.000m
UB 406x178x54/S355
Static: Unknown@0.000
RSA: Unknown@0.000
$C_f = 1.600$
Wind pressure = 1.200 kN/m²
Wind force = 6.2 kN
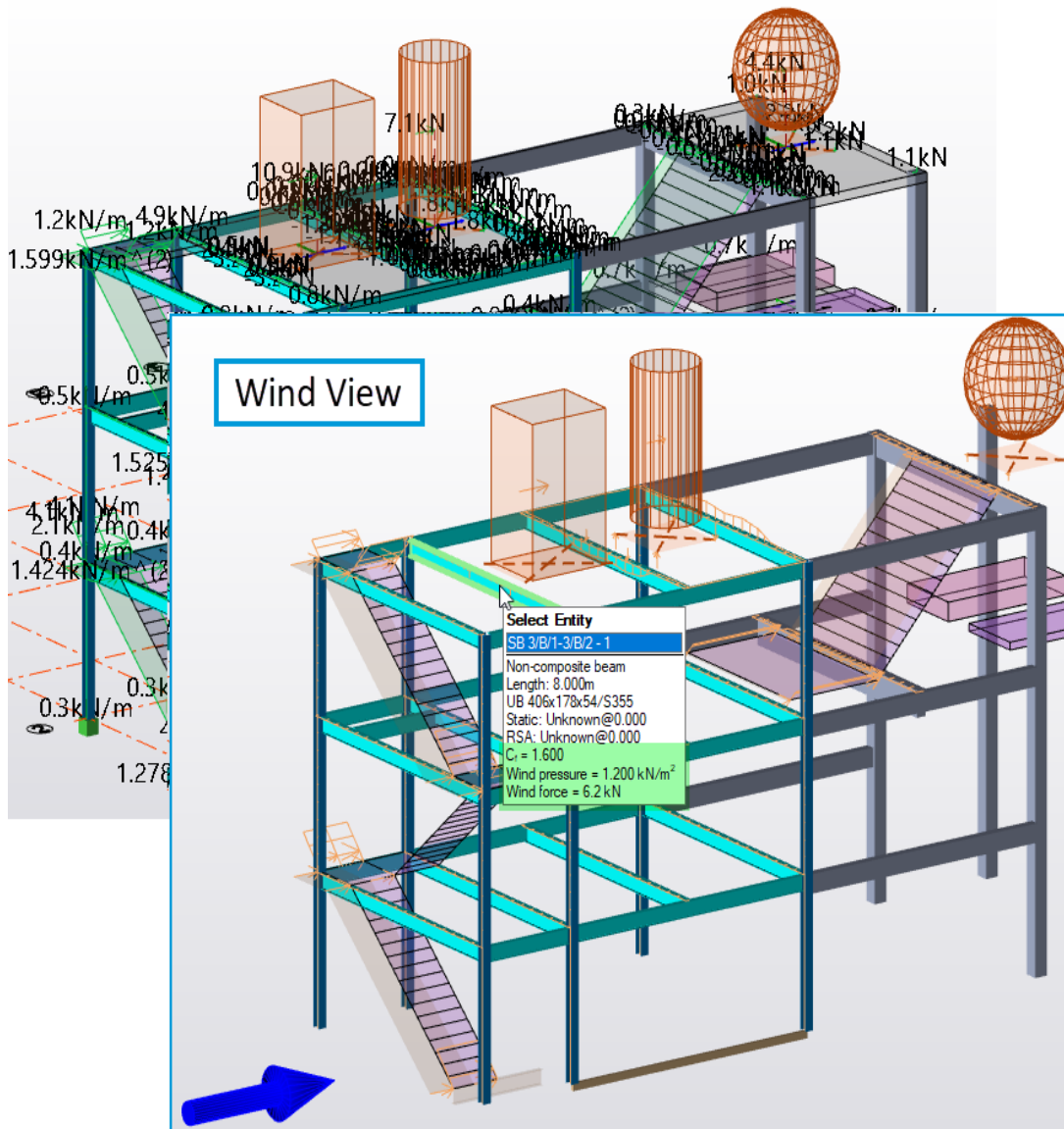
- **Scope**:
  - Loaded Entities - Open Structure Wind Loads are generated on the following entities: Structural Members (beams, columns etc), Line Ancillaries, Area Ancillaries, Equipment (new for this release - see below).
  - Head Codes - applies to the United States, Eurocode (all countries) and Indian Head codes. It is not currently available for the British Standard BS 6399-2, or Australian AS:1170.2 wind code variants.
- **Assumptions & Limitations**:
  - All wind loads are assumed to act perpendicular to the members, equipment and ancillaries.
  - Applicable for all 1D structural members irrespective of their characteristic except; Analysis elements, DELTABEAM, FABSEC beams.
  - Are not applied on structural walls (use wall panels over these to apply load).
  - Shielding effect is not considered.
  - Does not consider frictional wind forces on any surfaces.
- **Workflow** - application of Open structure wind loads essentially requires two simple steps; 1) specify which entities the wind load will apply to 2) run the Wind Wizard and enable the new "Apply Open Structure Wind Load" It's as simple as that! In more detail:
  - Review/ specify the entities to which the Wind Load is to be applied. This is controlled by a new Wind loading > "Apply open structure wind load" member/ entity property as shown in the picture below - when this is enabled the wind loading is applied.

  - You can carry out this step most effectively in Review View > Show/Alter State using the new "Wind Loading" attribute which enables rapid graphical review and editing of the setting. Simply Toggle or Set On/Off the setting by individual mouse click and/or Window cursor operations (see picture below).

- The "Apply open structure wind load" property is enabled by default for Ancillaries and Equipment and is off for structural members.

  - Note that for continuous members it can be applied on a per span/ stack basis.

- You can additionally review/ edit - via entity properties - the listed wind loading parameters Shape factor, Cf and Effective area XY and XZ factors. See below for more on these.

- Run the Wind Wizard and check on the new "Apply Open Structure Wind Load" - from this point forward the process of completing the Wind Wizard and adding Wind loadcases is unchanged. Following this, the open wind loads calculation and application is entirely automated.
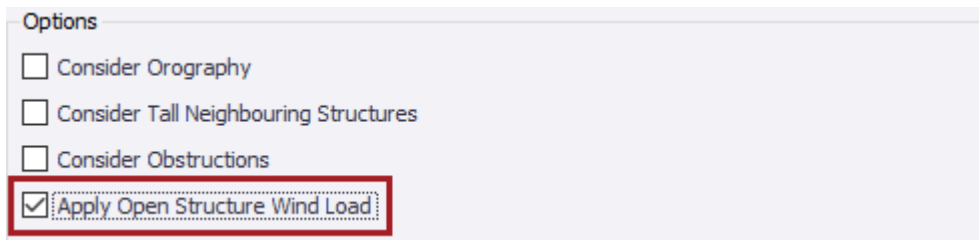
- Additional aspects:

  - Review View - as mentioned above, Review View > Show/Alter State includes a new "Wind Loading" attribute enabling rapid graphical review and editing of the "Apply open structure wind load" property via Toggle or Set On/Off modes.
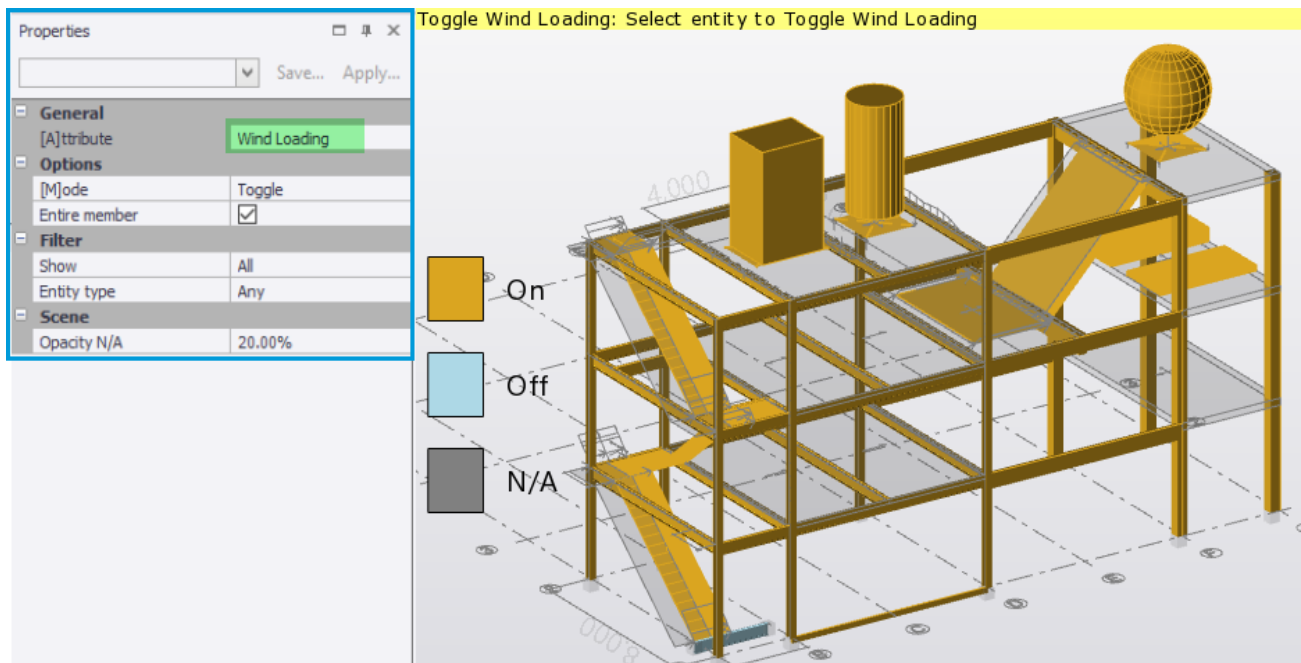
- Shape factor (C_f) and Effective area factors - new Global and Model settings are added to set the default values for these, accessed via Global/ Model Settings > Loading > Wind Loading as shown in the picture below. There are also Area/Length values for some ancillaries such as Walkways/ catwalks both with and without guardrails. Note the following:

  - The default C_f factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.

  - The settings also have columns showing the Effective Area equations so you can see how these values are calculated.

  - Factor values set/changed in Model Settings are applied to all entities in the model. You can also set the value for an individual entity via its properties.

Entity Type	Shape Factor, C_f	Effective Area$_{XY}$	Factor$_{XY}$	Effective Area$_{XZ}$	Factor$_{XZ}$	Area Per Length$_{XZ}$ [m²/m]
Member	1.800	1.000 × B × L	1.000	1.000 × D × L	1.000	
Walkways/catwalks, with guardrail	2.000	1.000 × W × L	1.000	0.245 × L		0.245
Walkways/catwalks, without guardrail	2.000	1.000 × W × L	1.000	0.000 × L		0.000
Ladders with cage	2.000	1.000 × W × L	1.000	0.230 × L		0.230
Ladders without cage	2.000	1.000 × W × L	1.000	0.152 × L		0.152
Access platforms, with guardrail	2.000	1.000 × W × L	1.000	0.245 × L		0.245
Access platforms, without guardrail	2.000	1.000 × W × L	1.000	0.000 × L		0.000
Operating platforms (storage/laydown), with guardrail	2.000	1.000 × W × L	1.000	0.245 × L		0.245
Operating platforms (storage/laydown), without guardrail	2.000	1.000 × W × L	1.000	0.000 × L		0.000
Operating platforms (standard), with guardrail	2.000	1.000 × W × L	1.000	0.245 × L		0.245
Operating platforms (standard), without guardrail	2.000	1.000 × W × L	1.000	0.000 × L		0.000
Stairs/landing	2.000	0.500 × W × L	0.500	1.000 × H1 × L	1.000	
Lines of Pipework	0.700	1.000 × W × L	1.000	1.000 × (D + 0.	1.000	
Lines of Cable tray	2.000	1.000 × W × L	1.000	1.000 × (H + 0.	1.000	
Other	2.000	1.000 × W × L	1.000	1.000 × H × L	1.000	

1.12 New Equipment Load Entities

New for this release *Equipment Load Entities* are similar in concept and operation to the recently added *Line and Area Ancillaries* , for more about which see the 2020 SP4 Release notes section New Line and Area Ancillaries Loading. They enable the definition of three dimensional objects for the purpose of load application for such items as retaining tanks, plant and machinery etc. While developed principally with industrial structures in mind, they may have wider application for tanks, plant etc acting on any type of structure. As shown in the picture below, since (like Ancillaries) they represent physical objects, they are applied via a new button added to the "Walls &

Loading Objects" group of the Model tab of the Ribbon. They can be of three principal shapes; Cuboid, Cylinder (Vertical & Horizontal) and Sphere.

They are also fully considered and loaded by the New Open Structure Wind Loading (page 32) - For detailed information about how loads are calculated and applied to them see the new Help Topic Wind load on open structures calculations.
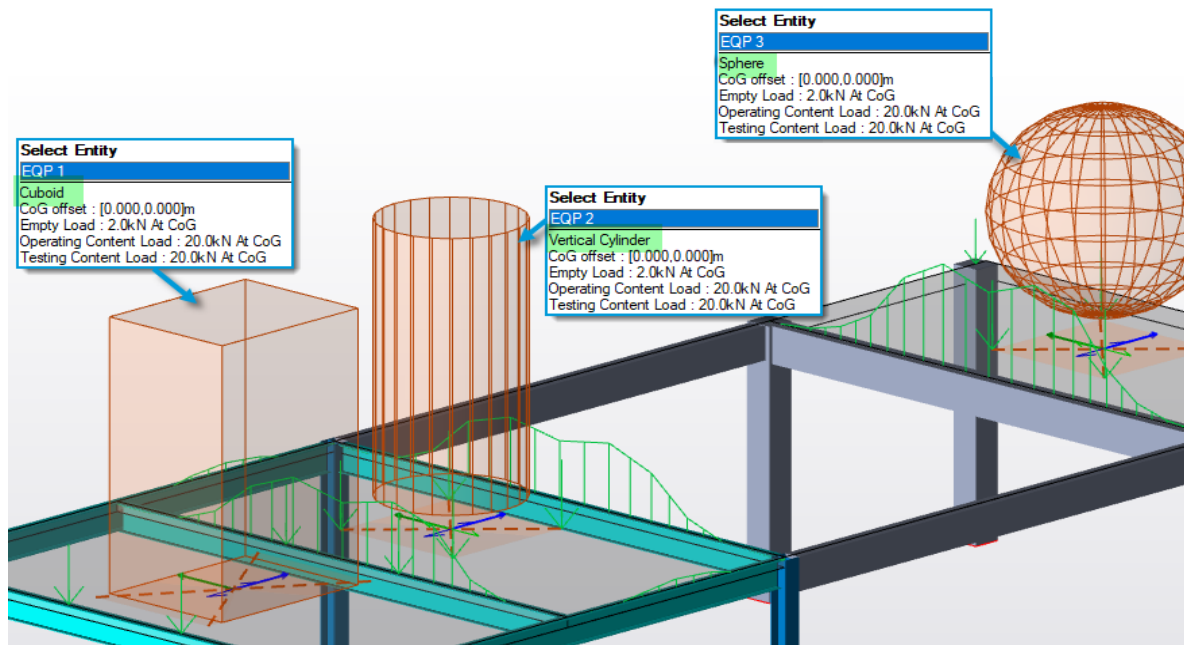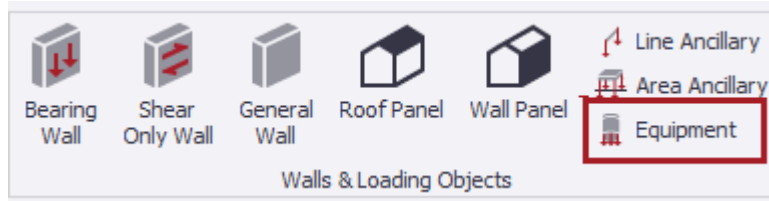
Key aspects of the new Equipment Loads are:

- The Equipment entities do not form any part of the analysis model - only the loads they apply take part in the analysis and are fully considered in design.

- Definition - after activating the Model ribbon command, you create an Equipment Load entity by clicking its "Support points" in turn, clicking twice on the final point, just like creating a panel as shown in the picture below. Any number of points forming any shape can be selected though, just as for panels, they must all lie in the same plane.
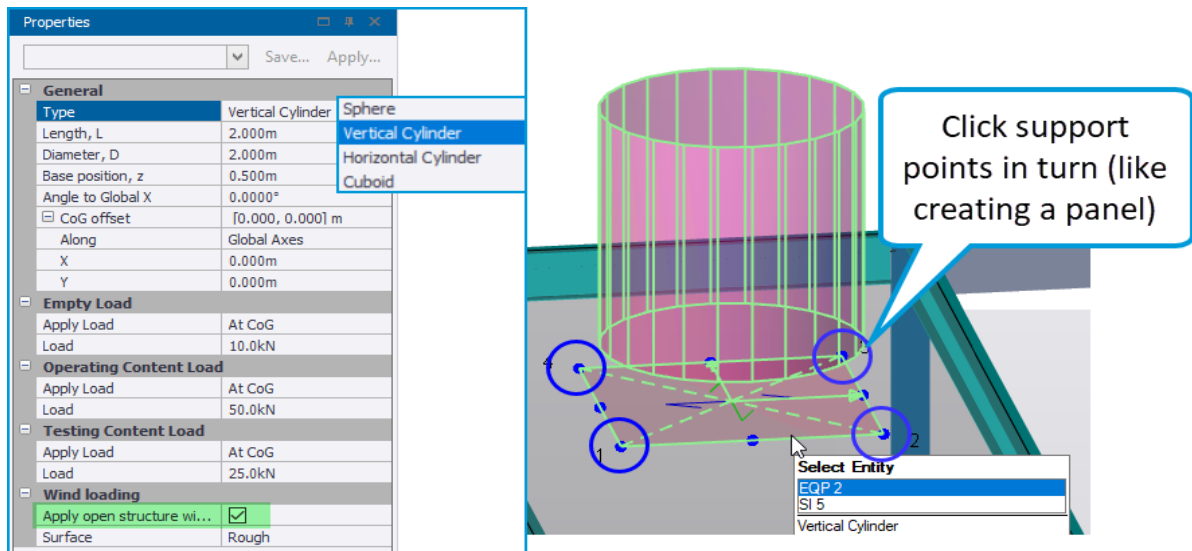
- Prior to creating, select the Equipment type in the Properties Window and enter the required parameters (which default to zero) for its size, base position, z (vertical distance above support points plane), lateral offsets (if any), loads and load application option - either "At CoG" (center of gravity) or "At supports" . Just as for any physical entity, all these settings can be reviewed/ edited via the Properties Window after creation by selecting existing Equipment entities. Note the option to "Apply open structure wind loads" which defaults to on.

- The support points for an existing Equipment entity can subsequently be manually adjusted just as you can the vertices of any panel; select the Equipment entity, click a support point (single click, not click-and-drag), move the cursor to another point and click again to move it there.

Adjustment points are also added between each support point, enabling the addition of further support points, as shown in the picture above.

- Support point load calculation and decomposition - the support points define a "panel" which is used to calculate (arithmetically) the support point loads by assuming it acts as a rigid body and the load is applied at the Equipment CoG. The load decomposition is potentially in two stages as for Ancillaries; in the first stage the calculated loads are applied to the support points, in the second phase these are decomposed - where necessary - again by whatever supports these. For example in the case of a supporting two-way slab - as shown in the picture below - this 2nd stage decomposition will be in a two-way manner to the members supporting the slab producing VDL's and point loads etc as appropriate.

  - Note that the calculation of the support point loads for lateral loads (from Open Structure Wind Loads) accounts for the vertical offset of the centre of gravity (CoG) of the Equipment entity, producing "push/pull" point loads and thus an overturning moment on the supporting structure.
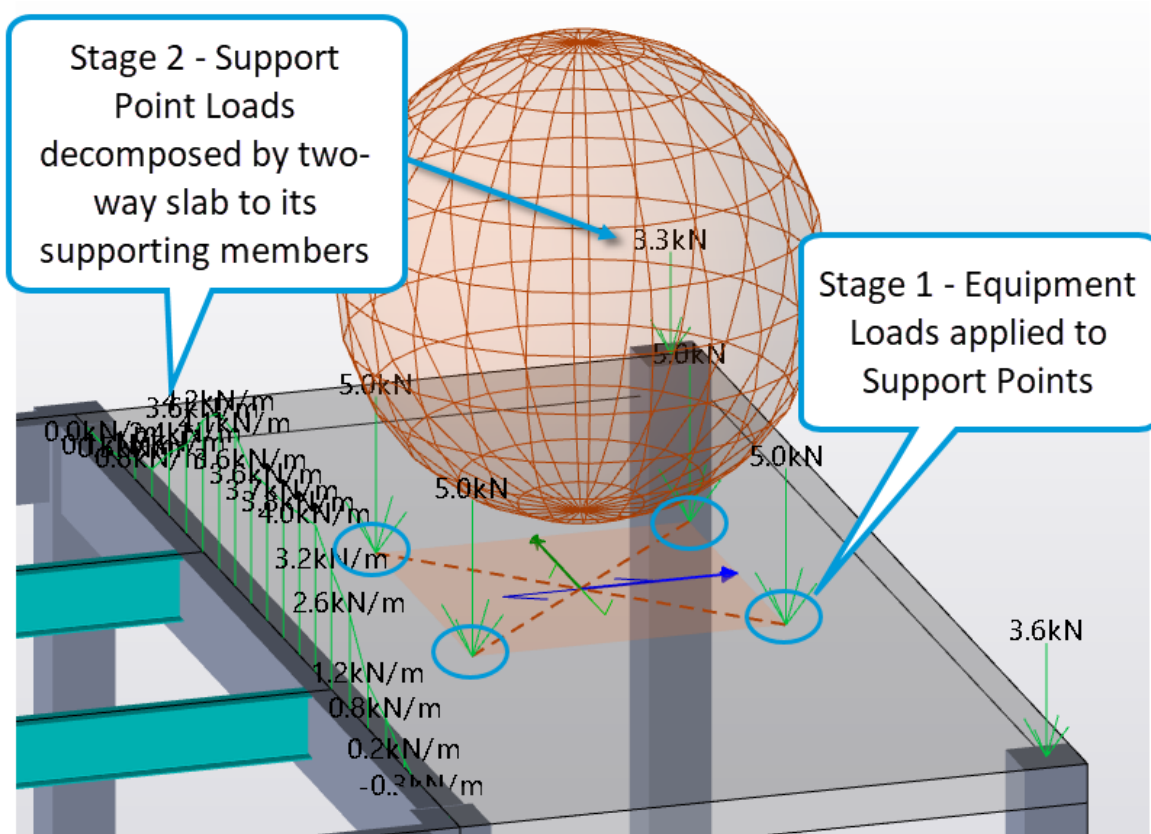
- If you do not wish to use automatically calculated support point loads, you can override this by selecting Apply load = "At Supports" instead of the

default "At CoG". As shown in the picture below, you can then directly enter your own load values for each support point.
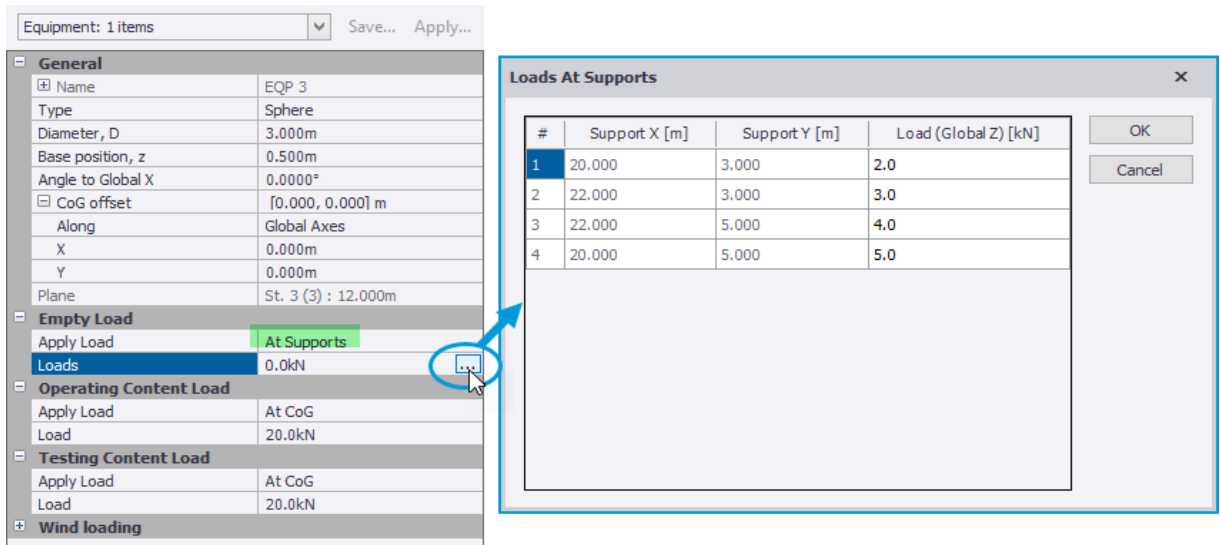
- You can also manually apply additional point loads - as dead/imposed and in horizontal or vertical directions - to Equipment Entities using the new "Equipment" load option in the Structure Loads group of the Load tab of the Ribbon. The loads can be applied either to the Equipment CoG or Support points.
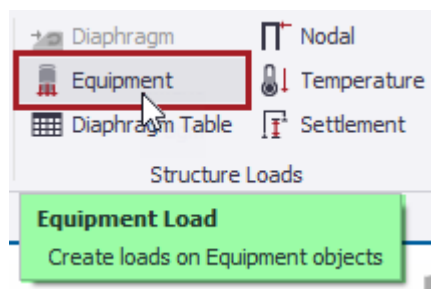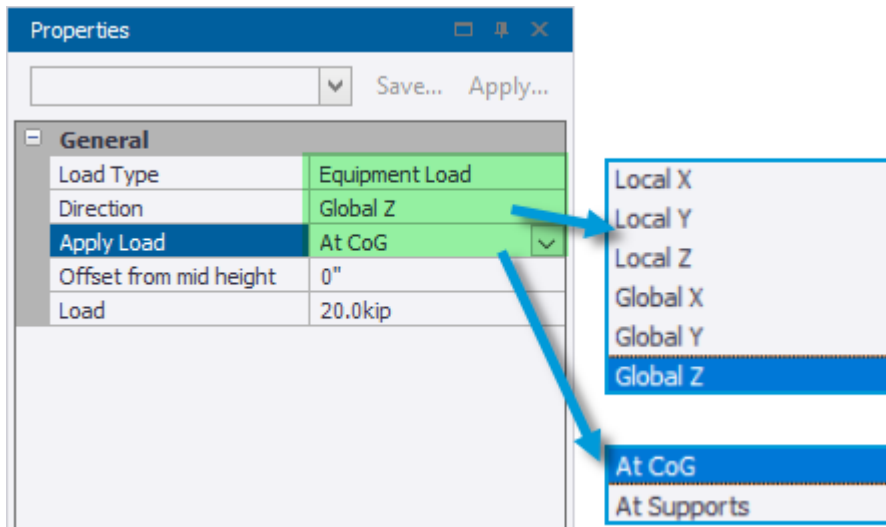
- Load Cases - dedicated Equipment loadcases are automatically created and removed (just like for ancillaries) as Equipment entities are added/deleted and can be included in combinations in the usual manner.

An additional setting in Global/ Model Settings > Loading > General controls the load **Type** for the Equipment Operating Content and Testing Content loadcases which can be set to either Dead or Imposed (similar to pipework ancillaries).

#	Loadcase Title	Type		Calc Automatically	Include in Generator
0	**Self weight - excluding slabs**	SelfWeight	∨	☑	☑
1	**Slab self weight**	Slab Dry	∨	☑	☑
2	Dead	Dead	∨		☑
3	Services	Dead	∨		☑
4	Live	Live	∨		☑
6	Roof Live	Roof Live	∨		☑
7	**Equipment Empty**	Dead	∨		☑
8	**Equipment Operating Content**	Dead	∨		☑
9	**Equipment Testing Content**	Dead	∨		☑
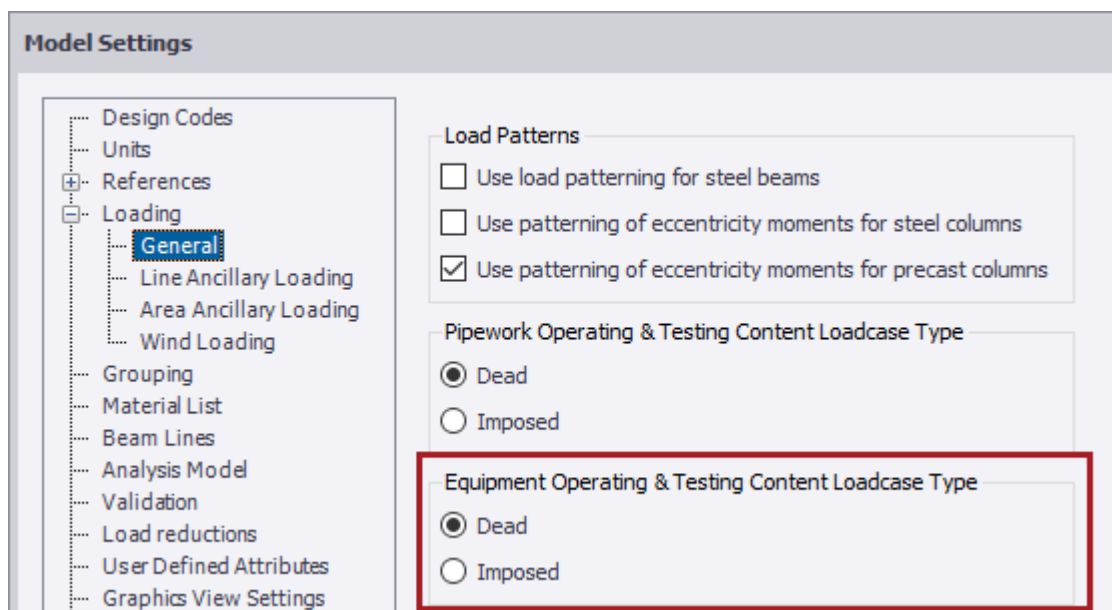
1.13 New Partially Cracked Concrete Walls & Members

This new feature is an addition to existing features of especial use in the design of tall buildings, for which the engineer needs to particularly assess and modify the stiffness of the lateral force resisting system - commonly composed of concrete shear walls - to control lateral displacements. The stiffness of concrete members is determined in large part by their state of cracking - which can vary between the 'extreme' conditions of entirely uncracked and fully cracked. All previous releases have had the option to specify the (fully) cracked and uncracked conditions for concrete members (beams, columns and walls) and control their stiffness accordingly and separately via Modification Factors. Note the default values of these **for the Building Analysis** assume the loading is long term and should be modified when considering short term loading - for more see this TUA article: Where do the default values for cracked and uncracked properties of concrete come from?

This release adds a further refinement to modeling the effect of cracking on stiffness with a new **partially cracked** option for the intermediate condition between (fully) cracked and uncracked which is applicable to beams, columns and walls (meshed). The partially cracked state is set by a simple degree of cracking % for which the stiffness modification factor is automatically calculated and applied. For more see this Help topic:

- **Tall Building Companion Features** - as mentioned, this feature adds to and can be considered in the context of a number of more recently added features targeted particularly at the design of tall buildings. These are:
  - Dynamic Analysis - Wind Tunnel Data Report Generation
  - Design for high strength concrete grades (Eurocode) - for more on this see the Design section in Tekla Structural Designer 2019i Release Notes
  - Stresses in 2D elements
  - 2D In-Plane Stress Contouring useful for assessing extent of cracking of Concrete Walls
  -
  -

- Key Aspects of the new feature are:
  - **Scope** - the feature is applicable to:
    - Wall type: Meshed shear wall (not mid-pier), Material = Concrete, Fabrication = Cast in place & precast
    - Beams - Material = Concrete, Fabrication = Cast in place, Post tensioned & precast
    - Columns - Material = Concrete, Fabrication = Cast in place & precast
  - **New Property Settings** - the properties for walls, beams and columns are adjusted for the new feature; the "Assume Cracked" setting is now listed in the "Analysis & design control" group (previously "Design Control" for beams and columns) and a further option "Partially" is added. When this is selected you can enter the degree of cracking as a % from zero (fully uncracked) up to 100% (fully cracked).

Walls

Properties		□ ⫟ ⟩
Cast-in-place Concrete Walls: 1 items	∨	Save... Apply.
Reinforcement layers	2	
⊞ Releases		
⊟ Analysis & design control		
Assume cracked	Yes	∨
⊞ Design parameters	Yes	
⊞ Sway and drift checks	No	
⊞ Confinement reinforcement	Partially	
⊞ Slenderness		
⊟ Analysis & design control		
Assume cracked	Partially	⟱
Percentage	50.00%	

Beams & Columns

Properties		□ ⫟ ⟩
Cast-in-place Concrete Beams: 1 items	∨	Save... Apply..
⊞ Load reductions		
⊟ Analysis & design control		
Structure supporting sensitiv...	☑	
Increase reinforcement if def...	☐	
Permissible increase in reinfor...	0.00%	
Consider flanges	☐	
Ignore lateral instability	☐	
Assume cracked	Yes	∨
⊞ Design parameters	Yes	
Nominal cover beam top edge	No	
Nominal cover beam bottom edge	Partially	
Nominal cover section side	30.0mm	
Nominal cover beam ends	30.0mm	

- **Partially cracked modification factor** - the partially cracked Modification Factor ($ModFactor_{Partially\ Cracked}$) is automatically calculated by linear interpolation for the specified percentage (**x%**) between the factors set for uncracked ($ModFactor_{unCracked}$) and cracked ($ModFactor_{Cracked}$) as follows:

  - $ModFactor_{Partially\ Cracked} = ModFactor_{unCracked}$ - **x%** * ($ModFactor_{unCracked}$ - $ModFactor_{Cracked}$)

- • Thus for example for the default short term (Modal Analysis & RSA) concrete meshed wall modification factors of cracked = 0.5 and uncracked = 1.0, a Partially cracked value of 50% gives a modification factor = 1.0 - 0.5*(1.0 - 0.5) = 0.75.

  - Member stiffnesses are then automatically adjusted by the interpolated partially cracked modification factor values, these being calculated and applied for every property pair for which different values are specified for cracked and uncracked (specific to the factor values currently set in the model), thus:

    - Meshed Walls - the interpolated factor applies to properties E, G and/ or t as appropriate in the same manner as the cracked/ uncracked factors.

    - Beams & Columns - the interpolated factor applies to the section properties as appropriate in the same manner as the cracked/ uncracked factors.

- **Review View > Show/Alter State** - the controls for the "Assume cracked" attribute are enhanced to allow review, application and editing

of the new partially cracked setting as illustrated below. While the pictures show the review/ settings for walls, the functionality for concrete beams and columns is the same.

- Mode options are adjusted - a new "Review" mode is added (in addition to Review wall stresses) and the editing options are now "Toggle" and "Set On".

- Review Mode - In Review Mode the color-coding and legend are enhanced to show the distribution of applied partially cracked values. With this mode active, the cursor tooltip also includes the Partially cracked % value and calculated *EI* values for major and minor axis inertia*.

  - *Note that the *EI* values are for illustrative purposes only, should you wish to compare them with your own calculated values - the *analytical* adjustment of stiffness is made by application of the partially cracked modification factors as discussed above.
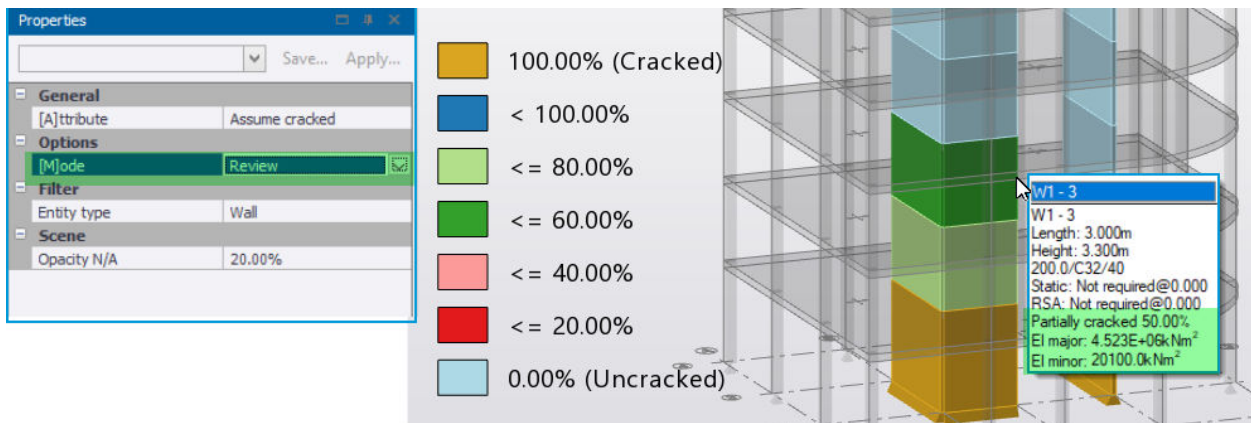
- Set On Mode - when enabled a new "Assume cracked" option is displayed with the edit options of "Yes", "Partially" and "No". With "Partially" selected, you can check on the "Set Percentage" option and enter your desired % value. This can then be applied graphically to the model using mouse operations. As shown below, the color-coding and legend adjust in relation to the current % value to show you where it is currently applied, the distribution of other % values and Uncracked/ Cracked.
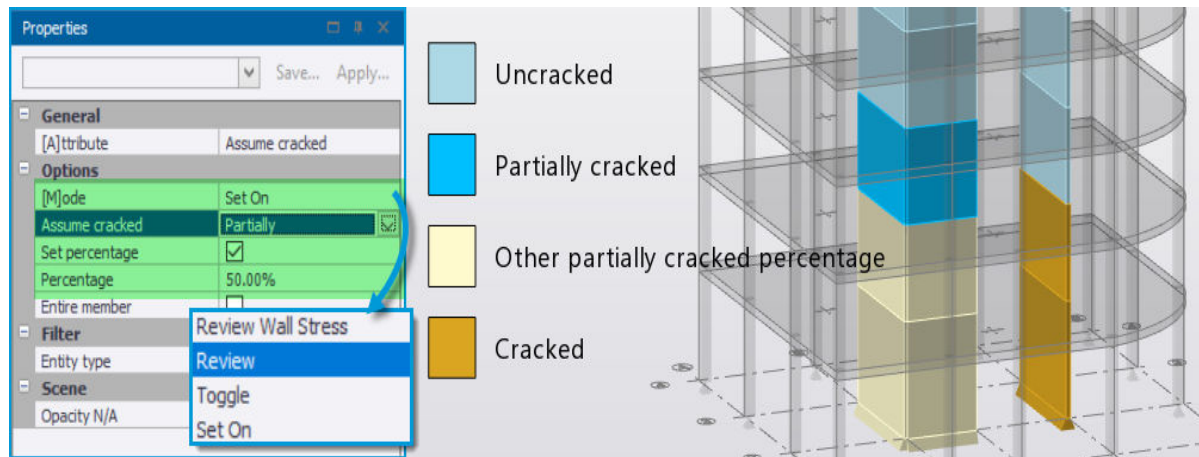
1.14 New Steel Column Base Plate Connection Modeling & Design - Eurocode and USA Head Codes

In this release it is now possible to explicitly model and design steel column Base Plate Connections to the Eurocode and USA Head Codes within Tekla Structural Designer. The design is carried out and reported fully within the program, with no use of Tekla Connection Designer (TCD), as shown in the

picture directly below. For full details of this new feature see the Help topic and this related video.
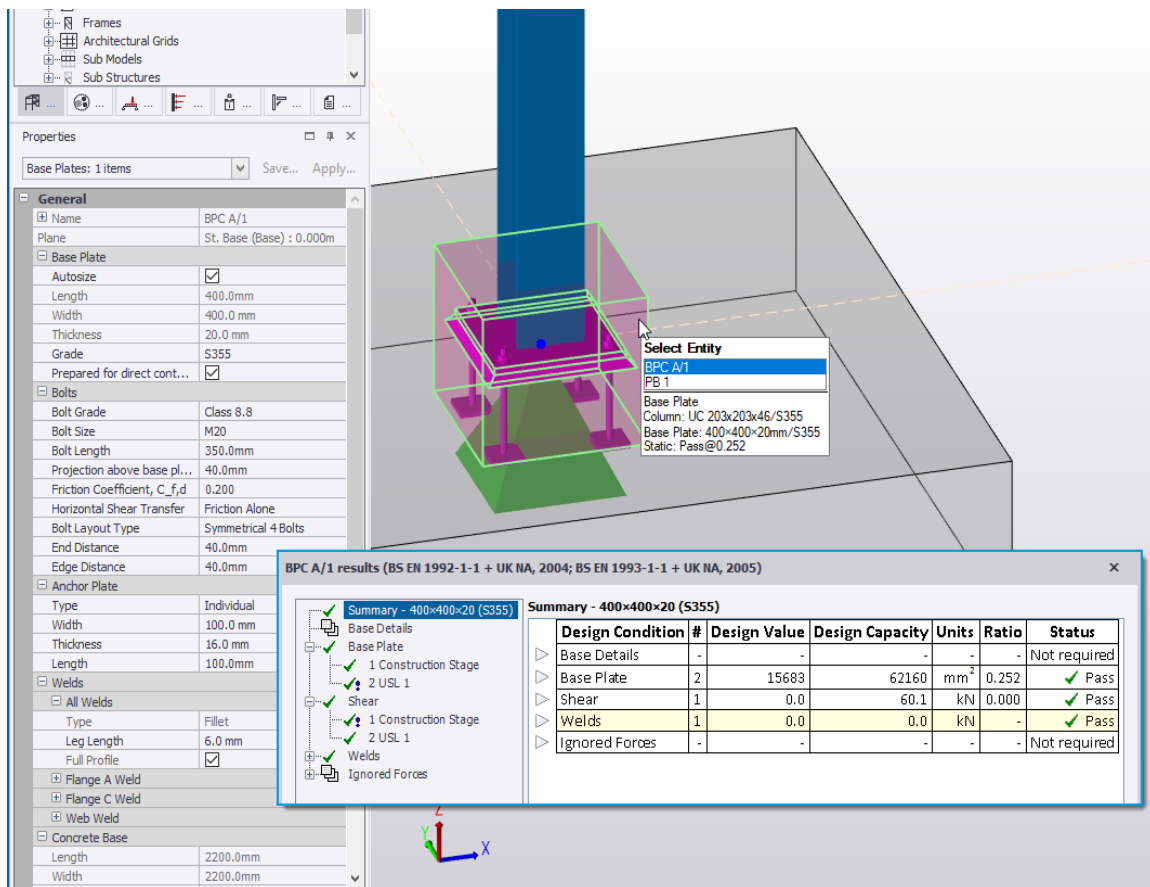
- **New/ changed operation** - base plates will be added (or can be created) in this release for steel columns for all Head Codes and will be listed in the Structure Tree as described below. Other changes are dependent on the Head Code as follows:

  - Eurocode - base plate connections are no longer listed in the Connections tree (after running the Update Connections command/ for existing models in which this has been done). Additionally, for the UK/ Irish Eurocode NA, they can no longer be exported to TCD for design (TCD can still however be used on its own if you wish).

  - United States - base plate connections are no longer listed in the Connections tree (after running the Update Connections command/ for existing models in which this has been done).

  - British Standards - as currently, base plate connections can be added but are not designed directly in Tekla Structural Designer. They can be designed in Tekla Connection Designer (TCD) - either integrated within Tekla Structural Designer or via export. New in this release is that they

are listed in both the Structure Tree and the Connections Tree. For more information see the Help topic:

- Australia & India - as currently, base plates cannot be checked in Tekla Structural Designer or designed in TCD.

- **Modeling** - to enable base plate connections definition, Bolts/ Rods and Weld data is added to the database in this release (see picture below). The bolt/ rod data includes all the required dimensions and design strengths required for design, while the weld data is in terms of the leg length size and, for the USA Head Code, the electrode strength. You can add to and/ or make your own named copies of bolt/ rod sizes and add your own welds data should you need to.

  - Note that base plates and anchor plate Width & Thickness are taken from the existing Plate Dimensions in the Sections Database.
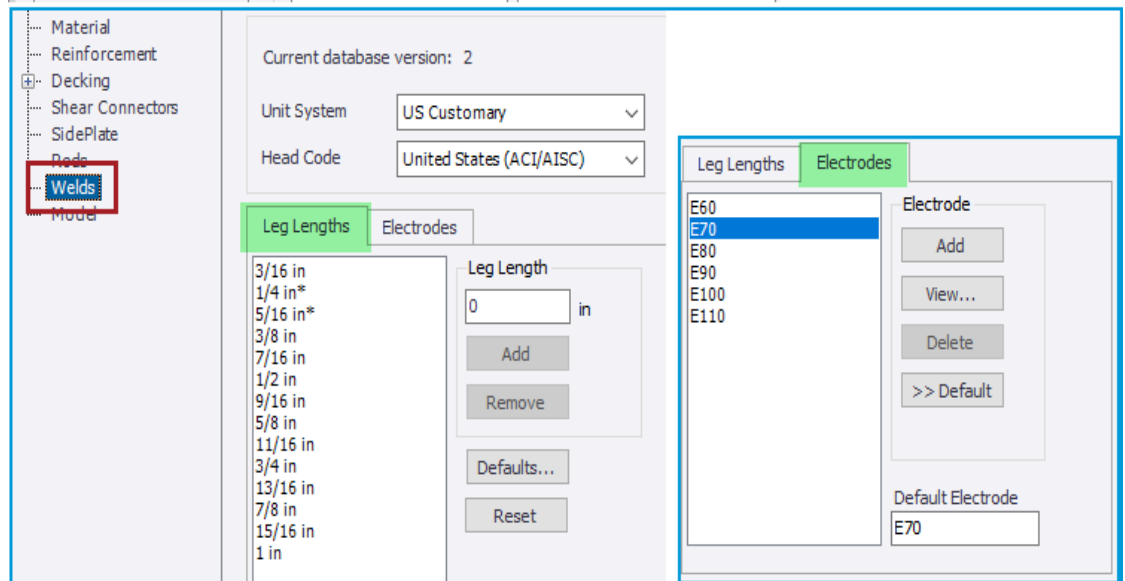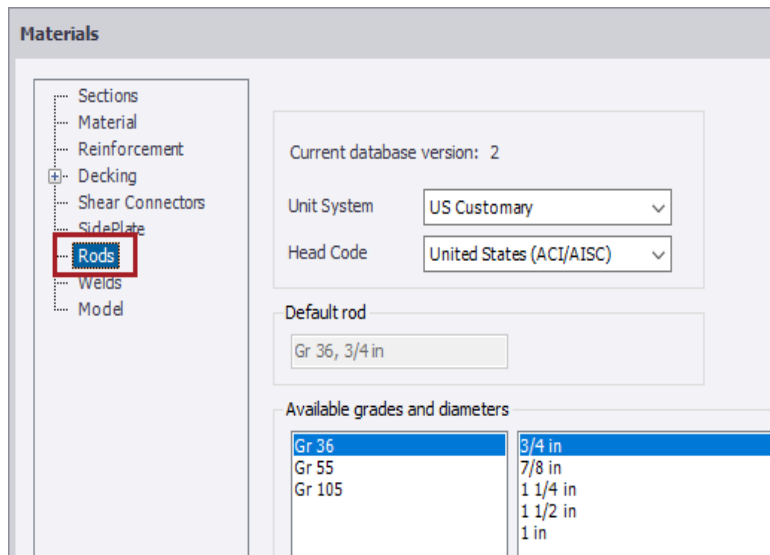
- **Creation** - base plate entities are now automatically added on creation of a new steel column when it requires a support. Base plates can also

be manually created using the new Add Base Plate button on the Design tab of the ribbon (final button on right of ribbon).
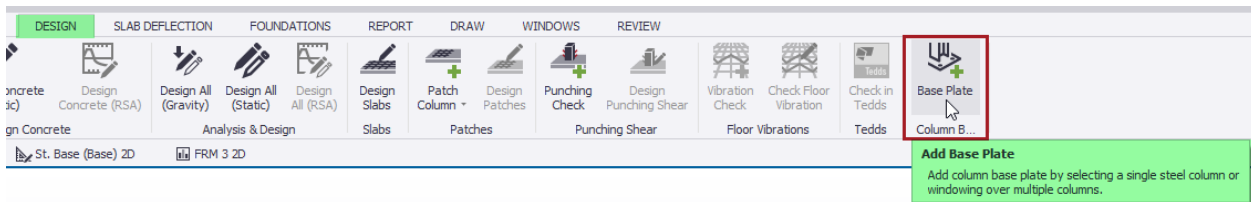
- Adding base plates works like adding pad bases - you can click on either the column or its support, or window over a selection of multiple columns/ supports. A column which has no support cannot be selected for application.
  - When a pad base (spread footing) also exists at a column support, the base plate connection's concrete base properties are automatically populated from this.
- Existing Models - the process and workflow for existing models created in prior releases depends on the scenario, of which there are three:
  - No base plate connections exist - you can add them manually as you need as described above.
  - Base plate connections already exist (the *Update Connections* process has been run previously) but have not been sized/ designed (using TCD) - when opened in this release, the existing base plate connections are automatically listed in the Structure Tree (see below) and their properties populated. They will no longer be listed in the Connections Window of the Project Workspace (Eurocode and USA Head Codes). The "Autosize" property for them will be off and you can enable it if you wish (recommended - see below) to obtain initial sizing.
  - Sized/ designed base plate connections already exist (Eurocode Head Code models only) - the same process described above occurs and again the base plate "Autosize" property will be off - you can leave it in this state to retain all the sized connection properties. The base plate's design status will be reset and you can then run a check to update this (see below).
- Base plates can be selected, moved, copied and deleted - either singly or a selection - just like other entities. Also like other entities, they have a full set of properties which can be accessed for review/ editing both via a dedicated dialog - as shown below - and via the Properties Window for a single or multiple selection.
  - Since they are associated with a column, when selecting a column to move/copy/delete the base plate, as default, will be selected also (just as a column support is) - you can then choose to only move/

copy/delete the column by de-selecting the Base Plate from the associated operation dialog.
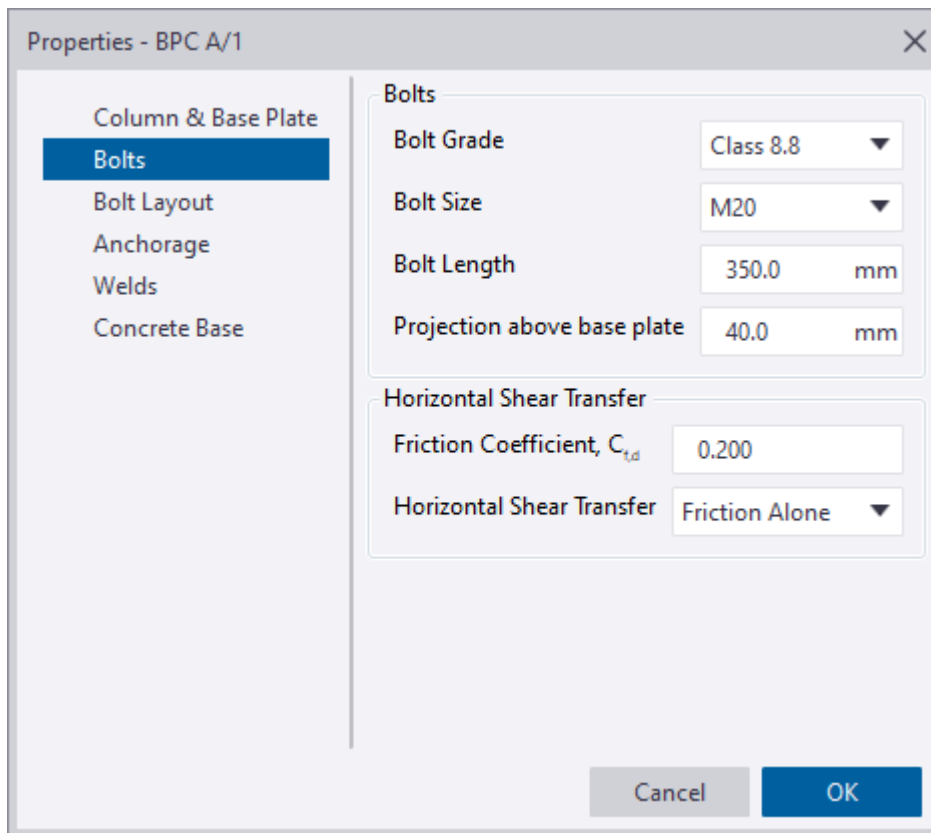
- Base plates are also automatically listed in the "Members" group of the Structure Tree, grouped by their column section size. As well as selecting base plates here, it is also possible to run a check design of them via the right-click context menu as shown in the picture below.
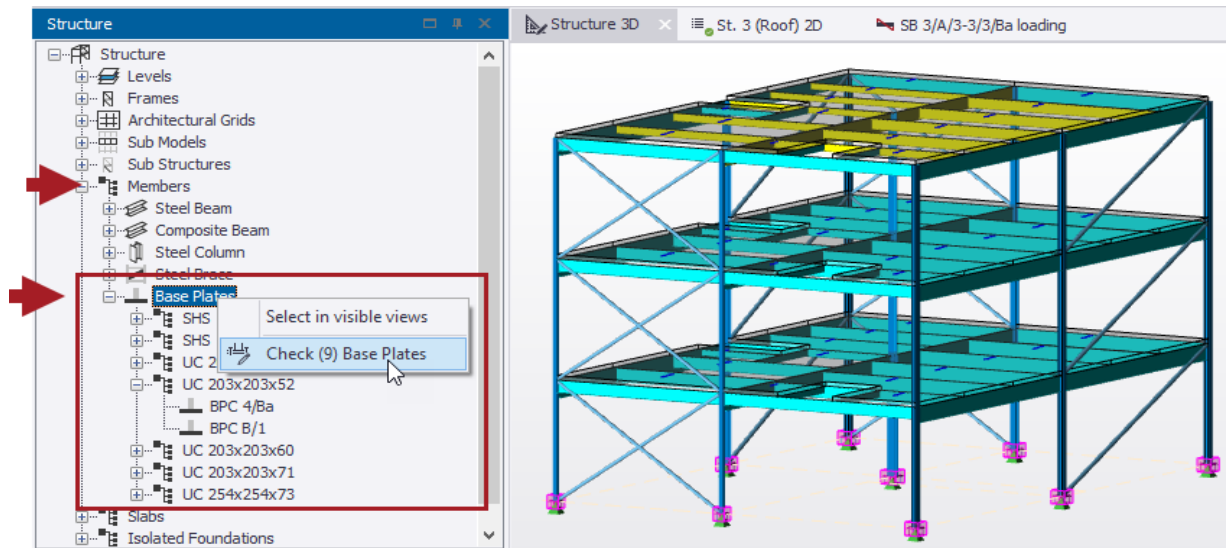
- **Autosize** - this key property of base plates is enabled by default when they are created. This uses simple geometric rules to produce an initial valid and sensible base plate size and configuration for the column's section size. When the column's section size is changed - for example by an auto-design - the base plate is adjusted accordingly.

  - The autosized base plate is not guaranteed to be adequate so this should not be confused with auto-design. The engineer will typically need to manually configure the connections - in terms of plate size and thickness, bolt size and layout and weld sizes etc - then run a check to assess if they are adequate.

- **Design Process** - as mentioned above, the design process is essentially manual configuration and check-design. Once the column section sizes have been established with autosize enabled, this setting can then be turned off and you can make further manual edits to the base plate properties should you wish to. All base plates can then be checked in a single process via the new "Check Base Plates" button on the Design Ribbon. You can also run design for some or all base plates from the Structure tree as shown in the picture above, as well as for individual base plates via the right-click context menu.

  - Property Sets - note that Property sets can be created for/ from and applied to base plates, just like with other entities in Tekla Structural

Designer - so you can use these to rationalize your design for similar/typical base plate connections.

- Review View - base plates design results are available with other members via the main *Design* group of the Review View Ribbon, (not the Connections ribbon group), so you can quickly visually review their design statuses and ratios. The recently added (in 2020 SP6) Review View Entity type filter for *Connections* can be enabled to focus the view only on base plate design results as shown in the picture below. For more about this feature see Review View - Design Status Enhancements
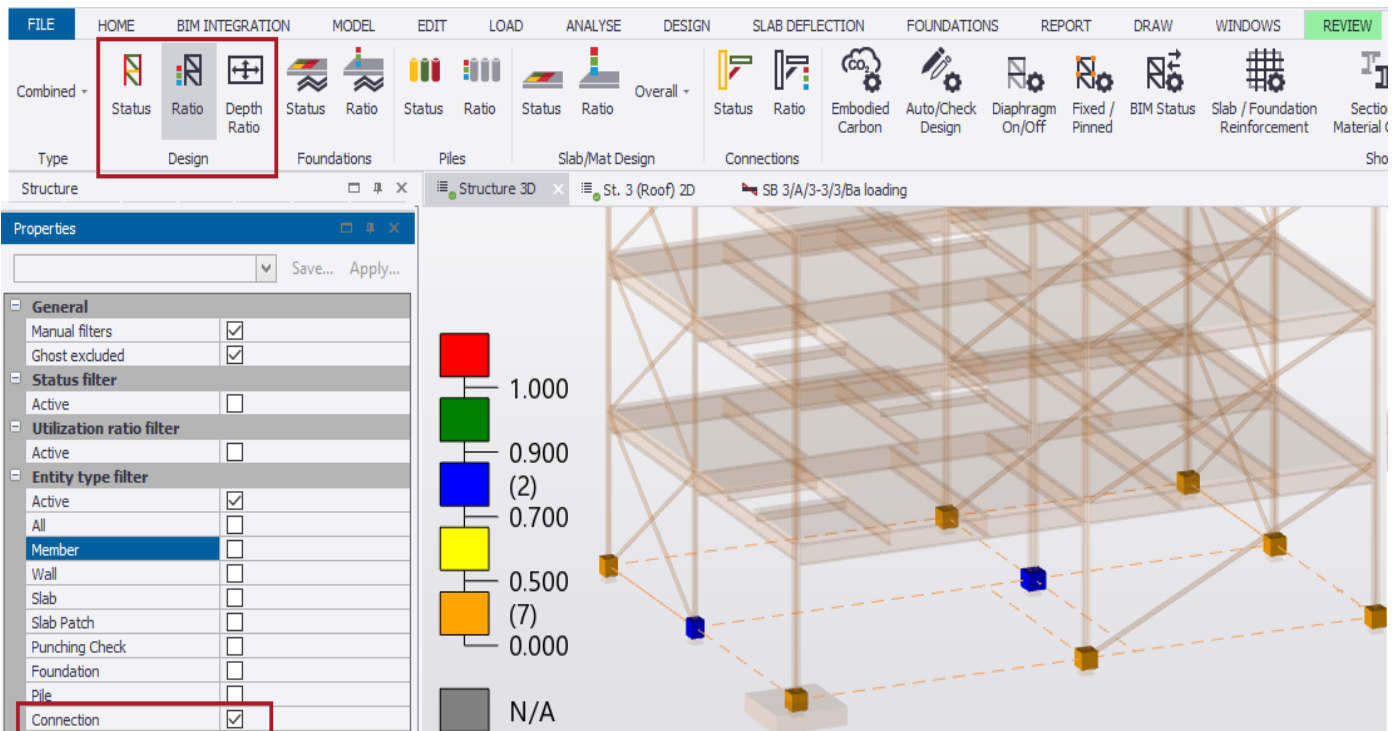
- **Design Scope & References -** these are as follows depending on the Head Code:

  - Eurocode - the design covers all Eurocode Country NA variants and the checks follow the SCI design guide 'Green Book' P358 so the scope is:

    - Connection type/ configuration - Simple (i.e. no applied moment) column bases only for I, H or hollow section columns and steel

grades S235, S275, S355 & S460 (the default grade for base plate is S355). No eccentricity of the column to the base plate is allowed (ebpz = 0).

- Design forces*; axial compression and major axis shear (Warning or Beyond scope messages will be issued where these conditions are not met). All load combinations are considered except EQU, GEO, and SEIS combinations from RSA.

- Design checks - Base plate size (Effective Area Method), Shear and Welds capacity

  - *Note that, unlike for the previous export of column base plates to TCD, the minimum design forces (Design > Settings > Design Forces) are not considered for design within Tekla Structural Designer hence all non-zero forces from analysis are considered.

- United States - design checks are per AISC DG1, AISC 360 (05, 10 & 16) and ACI 318 (08, 11, 14 & 19) and the scope is:

  - Connection type/ configuration - Simple (i.e. no applied moment) column bases only for W, M or hollow section columns for all steel grades (default grade for base plate is A36-36).

  - Design forces; axial compression only. All load combinations are considered except seismic from RSA.

  - Design checks - Base plate Bearing Strength and Plate thickness.

- **Drawings & Reports:**

  - Drawings: Base plates are now included in the following drawings (*anchor plates applicable to Eurocode only):

    - GA drawing - the following information is included; Base Plates, Bolts/Rods, Base plate & bolt/rod details*. Note that the 2x column scaling option applies to base plates also. See picture below for an example (Eurocode).

    - Non-concrete column drawings - both In elevation at the bottom of each column view (minor & major) and In plan (x4 scale in order to clearly show bolt layouts) with the following information included; Base plate, Bolts/Rods, Grout, Welds, Anchor plates*, Base plate, bolt/rod, grout & anchor plate details*. See picture below for an example (Eurocode).

    - New Dedicated Base Plate Drawing - a dedicated Base Plate Member Drawing as been added with the following information shown in elevation & plan where applicable: Base plate, Bolts/Rods, Grout, Welds, Anchor plates*, Concrete base, Base plate, bolt/rod, grout, concrete base & anchor plate details*

    - To control these new items and drawing, new associated layers have been added to Draw > Settings which are on as default in the

settings sets for this release. Layer styles and configuration can be adjusted as required.
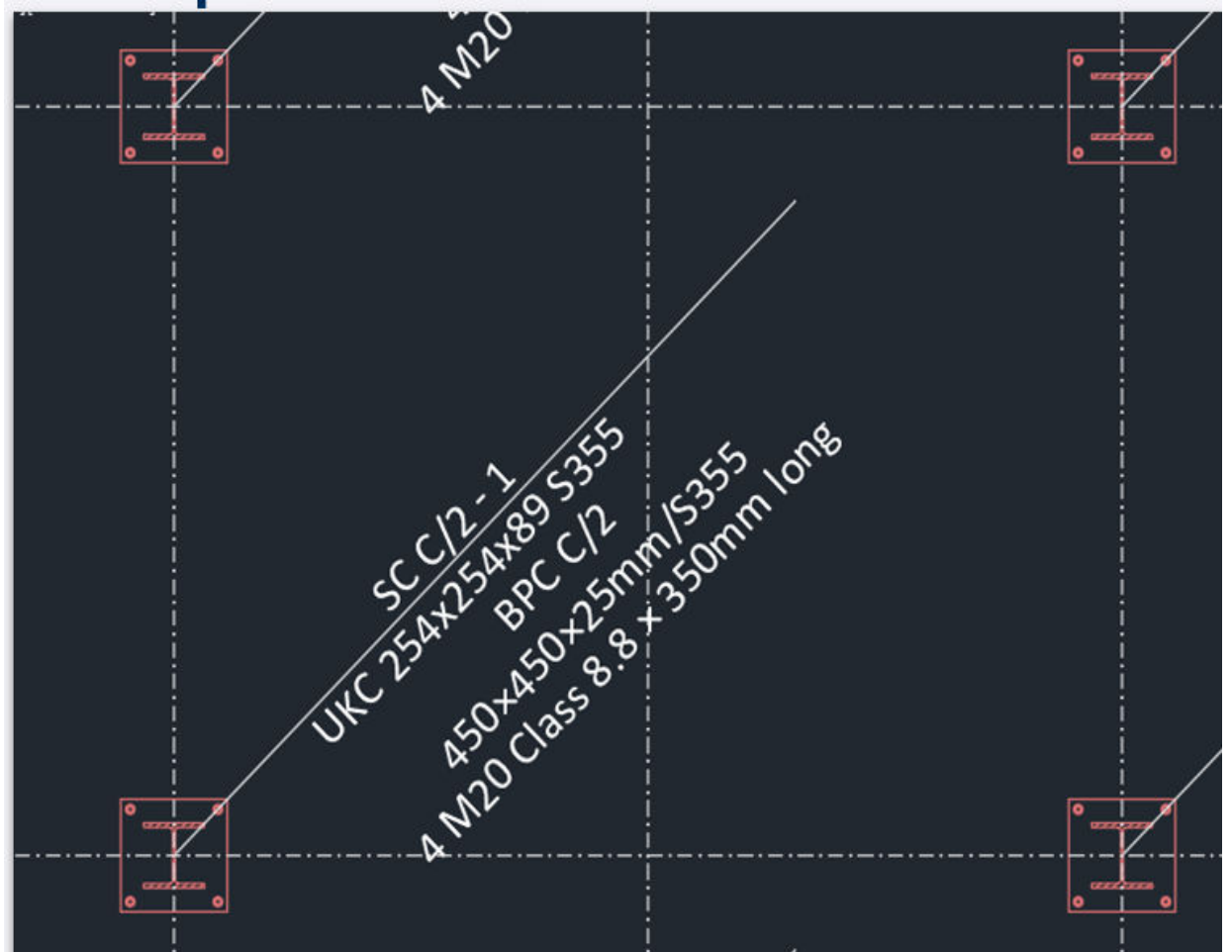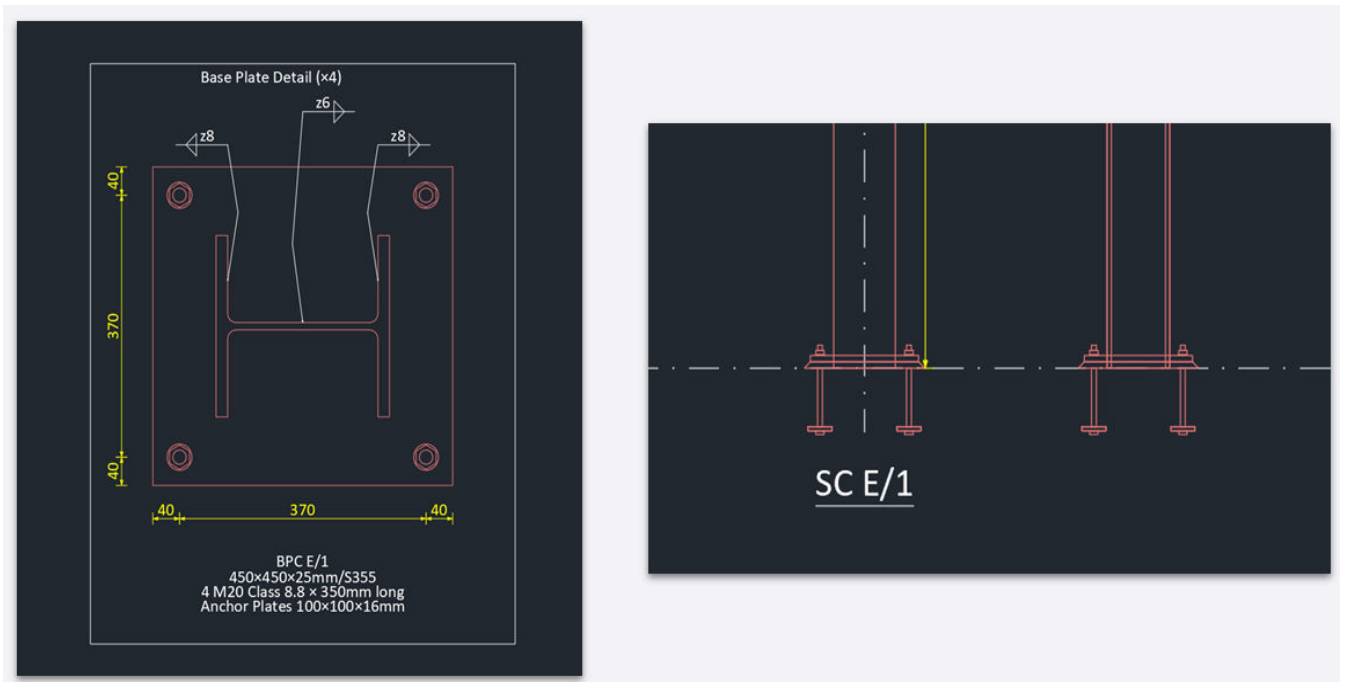
- Reports - base plates output can now be included in reports as follows:

  - Model Report - new *Base Plates* and *Base Plate Design Summary* report items have been added to Model Report > Chapters and options. The Base Plates item adds the Member Report for all base plates in the model (see below) and the Summary produces a simple condensed table with a single row for each base plate giving the main dimensions, number of bolts, UR and overall design status as shown in the picture below.

New Steel Column Base Plate Connection Modeling & Design - Eurocode and USA Head Codes
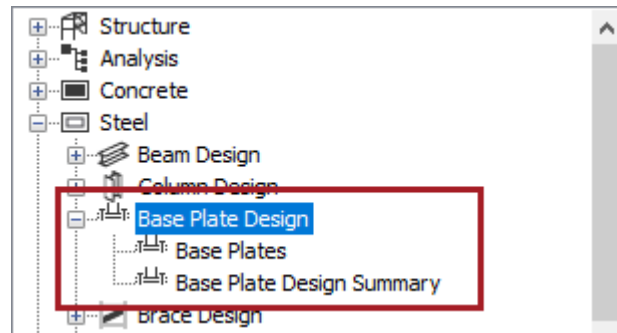
Base Plate Design

Base Plate Design Summary

Static

Base Plate	Length [mm]	Width [mm]	Thickness [mm]	No. of bolts	Utilization	Status
BPC A/1	400.0	400.0	20.0	4	0.252	✔ Pass
BPC B/1	400.0	400.0	20.0	4	2.628	⚠ Warning
BPC C/1	400.0	400.0	20.0	4	0.256	✔ Pass
BPC A/3	300.0	300.0	15.0	4	0.788	✔ Pass
BPC C/3	400.0	400.0	20.0	4	17.465	⚠ Warning
BPC A/4	300.0	300.0	15.0	4	0.398	✔ Pass
BPC C/4	400.0	400.0	20.0	4	0.261	✔ Pass
BPC 4/Ba	400.0	400.0	20.0	4	0.372	✔ Pass
BPC 3/Ba	450.0	450.0	20.0	4	0.728	✔ Pass

- Member Report - a new Member report has been added for base plates which can be configured and generated in the same manner as other member reports, both individually and included in the model report. The Member Report dialog now includes a "Base plate" member type which has default Design and Design Summary styles as for other member types and these are configured in the same manner. The report includes the new Base plate detail drawing (by default) and an example of the output this gives is shown below.

Project	Training Course	Job Ref.	123
Structure	Model 1	Sheet no.	Page 1/2
Calc. by	jekni	Date 01/03/2021	Chk'd by	Date 23/11/2015	App'd by	Date

BPC A/1
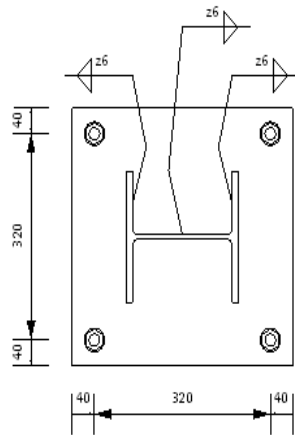
BPC A/1
400×400×20mm/S355
4 M20 Class 8.8 × 350mm long
Anchor Plates 100×100×16mm
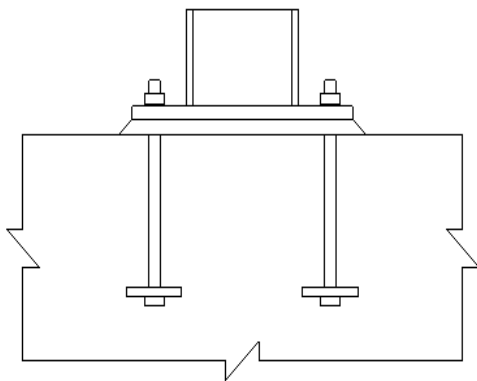
Column UC 203x203x46/S355
Grout Space 25mm
Concrete 2200×2200×500mm/C32/40

Project	Training Course	J
Structure	Model 1	S
Calc. by	jekni	Date 01/03/2021	Chk'd by	Date 23/11/2015	A

Static & RSA

Summary - 400×400×20 (S355)

Design Condition	#	Design Value	Design Capacity	Units	Ratio	Status
Base Details	-	-	-	-	-	Not required
Base Plate	2	15683	62160	mm^2	0.252	✓ Pass
Shear	1	0.0	60.1	kN	0.000	✓ Pass
Welds	1	0.0	0.0	kN	-	✓ Pass
Ignored Forces	-	-	-	-	-	Not required

Static & RSA

Base Plate

2 USL 1 - Critical

Effective Area Method

Required area, A_{reqd} = 15683 mm^2

Effective area, A_{eff} = 62160 mm^2

Ratio = 0.252

✓ Pass

Shear

1 Construction Stage - Critical

Design shear force, V_{Ed} = 0.0 kN

Design friction resistance, $F_{f,Rd}$ = 60.1 kN   EN 1993-1-8: 2005 Cl 6.2.2(6)

Ratio = 0.000

✓ Pass

Welds

Flange A - Critical

1 Construction Stage - Critical

Flange thickness, t_f = 11.0 mm

Leg length, s = 6.0 mm

Minimum leg length = 6.0 mm

✓ Pass

- Material List - base plate quantities can also be included in the Material List Report with the following information included (controlled by the Material List > Options > Steel settings); Plates, Bolts and Anchor Plates*.

1.15 Enhanced Sway/ Drift/ Wind Drift & Seismic Drift Check Control and Reporting - All Head Codes

The control settings and reporting for the Sway/ Drift/ Wind & Seismic Drift Checks have been thoroughly updated and enhanced for this release in line with customer feedback. All these checks assess the relative lateral deflections of successive levels - resulting from lateral loads of various kinds - and so their results and reporting requirements are similar. Tekla Structural Designer has always performed these checks very thoroughly in line with the design code intent by assessing the deflections of all columns and walls (where not specifically excluded) on a stack/panel basis and also reported the results on this basis. While this does follow the code requirements, they generally describe these checks in terms of *levels* - rather than stacks/panels - and with one check per level rather than many. Hence the sway results as previously presented were potentially less in line with the engineer's expectations, over-complex and difficult to efficiently assess. This is addressed in this release as follows:

- **Common Changes** - all the Sway/ Drift/ Wind Drift & Seismic Drift check results in all reporting locations are now presented in terms of *levels* rather than column/ wall stacks/ panels.

  - **Level Check Control** - there is now a single control in Level Properties "Check for drift" as shown below left. This is only listed for levels set as floors and when enabled removes the level from the check reporting for all sway/ drift checks (although the checks are still performed for stacks/ panels above/ below it).

  - **Status Window Sway/ Drift Results** - the check Warning and Fails status for these results in the Design Status group are now grouped by Level as shown below right - each level group lists the individual columns/ walls with the status for that level. This can indicate immediately the level(s) where sway/ drift is a problem while the previous result display did not.

- **Accessing Sway/Drift Review Views** - the Review Data Table and Show/ Alter State Views can now be opened directly from the Status Window as shown in the picture below - just right-click over a Warning/ Fail message in the sway/ drift results tree and select to open either its associated Review Data Table or Show/Alter View from the context menu. The Show/Alter State Views automatically open with the Review Mode activated.
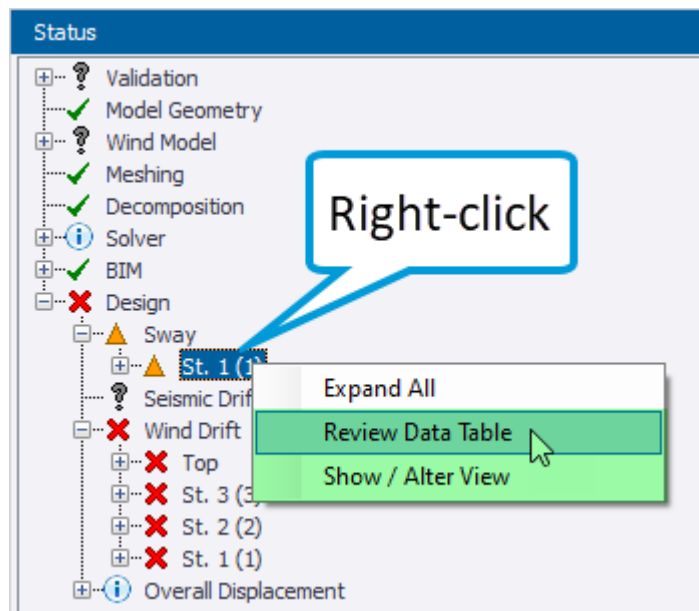
- **Sway/ Drift Review Data Tables** - these are similarly improved as shown in the pictures below comparing the 2021 (Eurocode) Sway results to those in previous releases. The results are now reported by level, producing a more compact results table where problematic levels are immediately apparent. While the picture below shows the (Eurocode) Sway results table, all Sway/ Drift, Wind Drift and Seismic tables for all Head Codes are similarly improved.

    - Similar to the Material List and Embodied Carbon tables, the sway/ drift result tables now have two levels - *Summary Only* and *Detailed* - controlled by the "Summary Only" toggle button directly below the table drop list at the top right of the Review Data ribbon tab. The Summary Only option (default) lists only the critical columns/ walls stacks/ panels for each level. Set the "Summary Only" toggle button to see results for all columns/ walls stacks/ panels (sorted by level).

    - In a similar manner to the Material List and Embodied Carbon tables, you can also now double-click on a row in the sway/ drift review tables to locate the associated column/ wall stack/ panel in the model - doing this will automatically switch to the 3D Structure view, select and arrow it.

    - Note that results for both columns and walls can now be viewed simultaneously in a single table where previously they could only be viewed separately.

2021										
Critical Sway										
Level ▼	Ref.	Stack	Combination Dir 1	$\alpha_{Dir\,1}$	Combination Dir 2	$\alpha_{Dir\,2}$	Combination Dir 1/2	Twist	Status	Details
Top	SC A/4	4	2 STR$_1$-1.35G+1.5Q+1.5RQ	18.940	2 STR$_1$-1.35G+1.5Q+1.5RQ	356.499	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.000	✓ Pass	Details...
St. 3 (3)	SC B/1	3	2 STR$_1$-1.35G+1.5Q+1.5RQ	13.977	2 STR$_1$-1.35G+1.5Q+1.5RQ	89.352	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.000	✓ Pass	Details...
St. 3 (3)	SC A/1	3	2 STR$_1$-1.35G+1.5Q+1.5RQ	13.980	2 STR$_1$-1.35G+1.5Q+1.5RQ	88.429	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.000	✓ Pass	Details...
St. 2 (2)	SC A/5	2	2 STR$_1$-1.35G+1.5Q+1.5RQ	10.353	2 STR$_1$-1.35G+1.5Q+1.5RQ	47.488	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.004	✓ Pass	Details...
St. 2 (2)	SC A/1	2	2 STR$_1$-1.35G+1.5Q+1.5RQ	10.517	2 STR$_1$-1.35G+1.5Q+1.5RQ	47.488	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.006	✓ Pass	Details...
St. 1 (1)	SC A/5	1	2 STR$_1$-1.35G+1.5Q+1.5RQ	3.512	2 STR$_1$-1.35G+1.5Q+1.5RQ	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.003	⚠ Warning	Details...
St. 1 (1)	SC A/1	1	2 STR$_1$-1.35G+1.5Q+1.5RQ	3.516	2 STR$_1$-1.35G+1.5Q+1.5RQ	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.004	⚠ Warning	Details...

Critical Sway											
Ref.	Combination Dir 1	Stack Dir 1	$\alpha_{Dir\,1}$ ▲	Combination Dir 2	Stack Dir 2	$\alpha_{Dir\,2}$	Combination Dir 1/2	Twist	Status	Details
SC C/5	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.512	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	49.169	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.007	⚠ Warning	Details...
SC A/5	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.512	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.004	⚠ Warning	Details...
SC B/5	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.512	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	44.392	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.006	⚠ Warning	Details...
SC C/4	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.513	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	49.169	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.001	⚠ Warning	Details...
SC B/4	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.513	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	44.392	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.001	⚠ Warning	Details...
SC A/4	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.513	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.001	⚠ Warning	Details...
SC C/3	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.515	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	49.169	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.000	⚠ Warning	Details...
SC B/3	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.515	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	44.392	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.000	⚠ Warning	Details...
SC A/3	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.515	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.000	⚠ Warning	Details...
SC C/2	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.516	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	49.169	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.005	⚠ Warning	Details...
SC A/2	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.516	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.003	⚠ Warning	Details...
SC B/2	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.516	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	44.392	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.004	⚠ Warning	Details...
SC A/1	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.516	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	40.461	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.006	⚠ Warning	Details...
SC B/1	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	3.516	2 STR$_1$-1.35G+1.5Q+1.5RQ	1	44.392	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	1.007	⚠ Warning	Details...

- **Show/Alter State Views** - the Review Mode view for these is improved to give results on an individual stack/ panel basis (for all Combinations), again giving a much clearer indication of where sway/ drift problems are located as shown in the picture below. Where previously the entire column/ wall was assigned the critical status/ color, now each stack/ panel is assigned its own individual status/ color. This is controlled by the "Entire Member" Mode option which is enabled by default. You can check this off to see a view similar to previous versions with the entire column/ wall having the status/ color of the critical stack/ panel. While the picture below shows an example (Eurocode) Sway results view, the views for all Sway/ Drift, Wind Drift and Seismic results for all Head Codes are similarly improved.

  - The legend is also expanded with more statuses including "Not Required" for stacks/ panels for which the sway/ drift check has been turned off.

  - The following additional improvements are made for the Toggle and Set On/ Off modes used to set the sway/ drift check on/off for selected members. Again this applies to all sway/ drift checks and Head Codes.

    - These modes can now be used for walls where previously only columns were available.

    - An Entity Type filter has been added to specify which entities can be selected for the current operation with options of; All, Columns, Walls.

Pass
Warning
Fail
N/A

Previously

SC B/2
Non-composite column
UKC 305x305x97/S355
Static: Pass@0.973
RSA: Unknown@0.000
α_{Dir1} 3.516
α_{Dir2} 44.392
Sway check: Warning
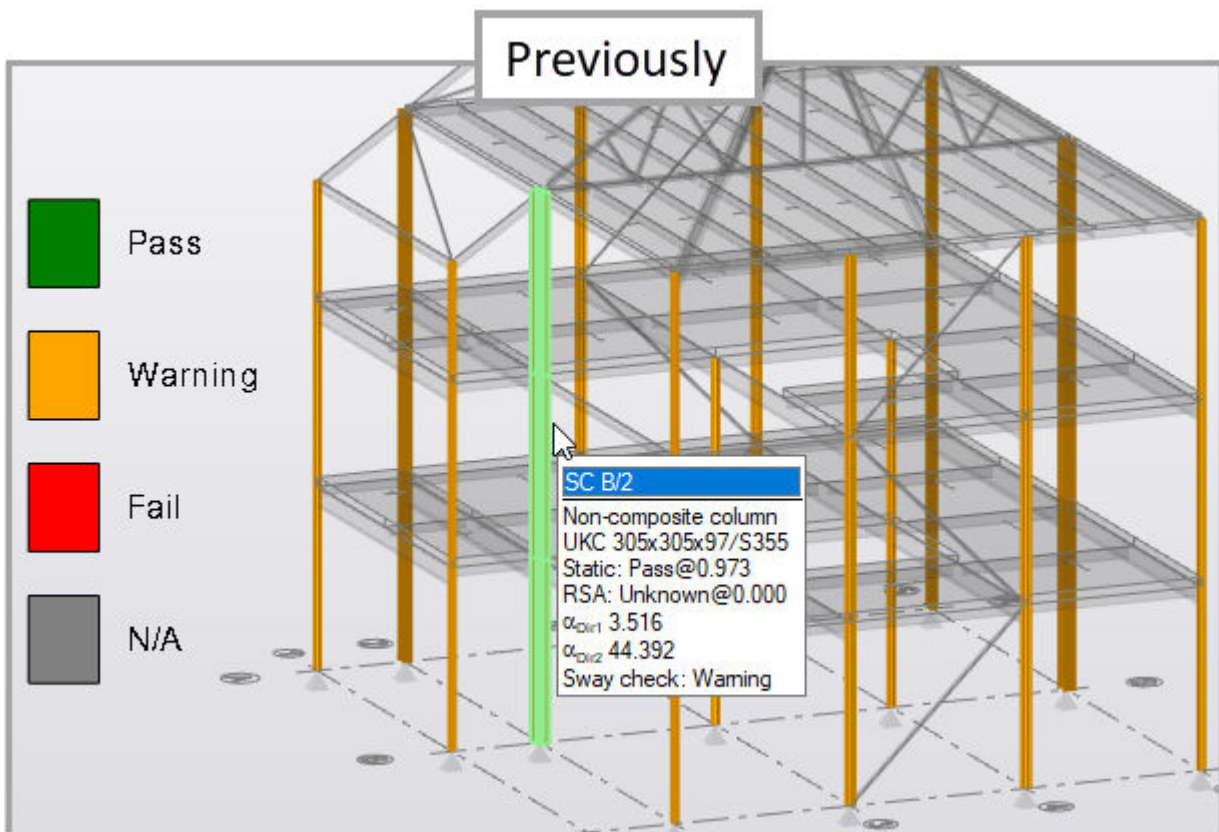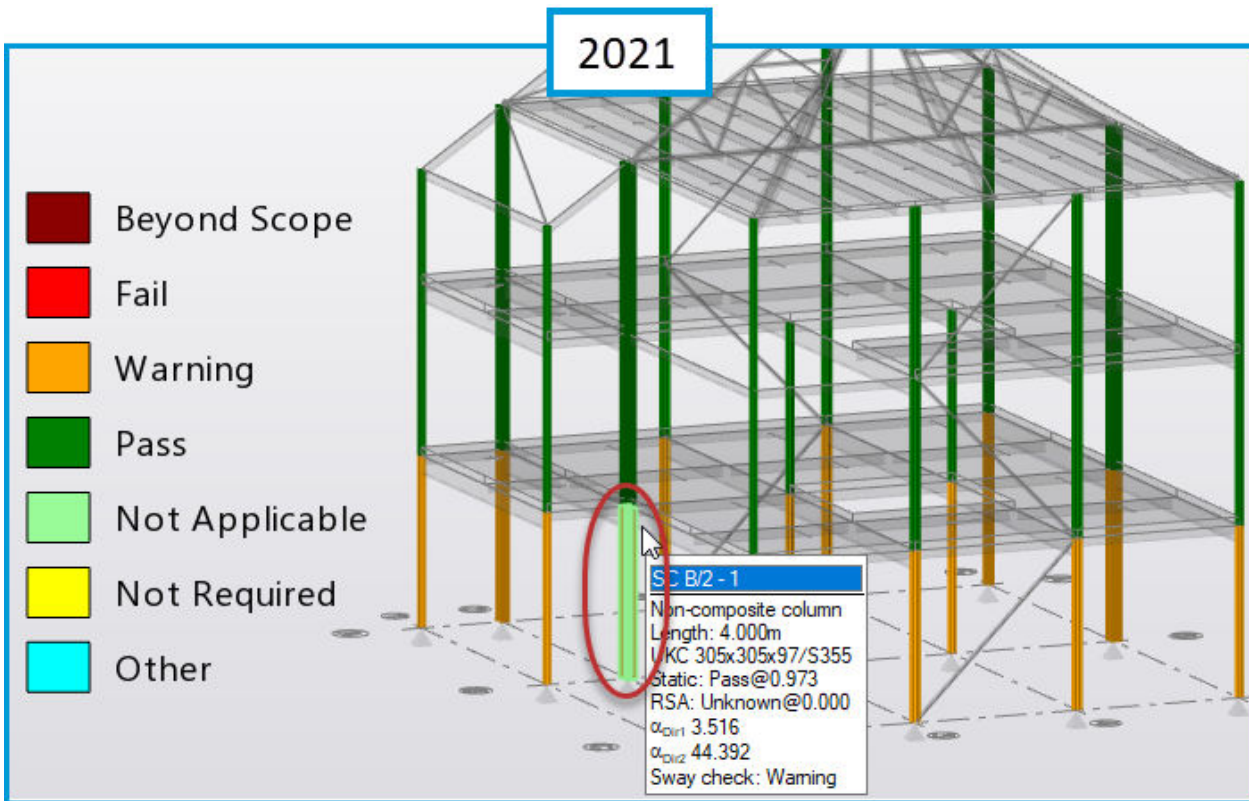
- **Wind Drift** - some specific improvements are made to the settings and checks for Wind drift, again in line with customer feedback.

  - **Wind Drift Limit** - a new Wind drift ratio limit value control is added in Design Settings > Sway & Drift Checks as shown below. This allows the engineer to set the drift limit for the entire model in a single step, where previously this was set on an individual column/ wall and span/ stack basis.
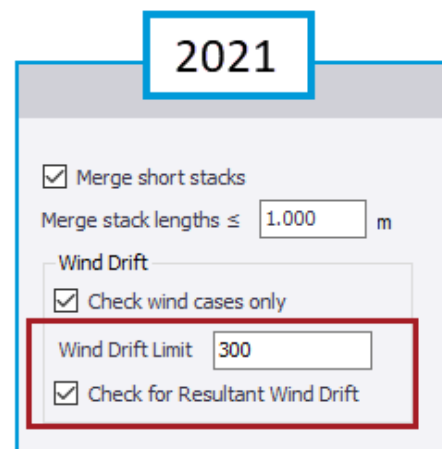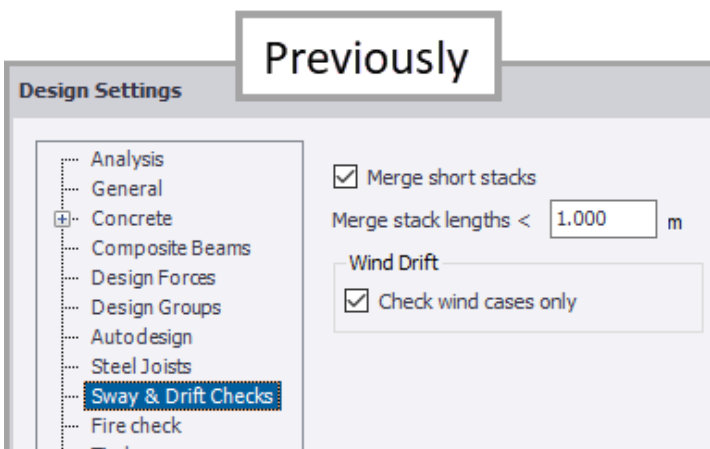
- The column/ wall properties are changed accordingly also - in place of the previous limit value, there is now an "Override…" checkbox. This is off by default, in which case the value set in Design Settings is used. When enabled you can enter your own limit value - again on an individual column/ wall and span/ stack basis.
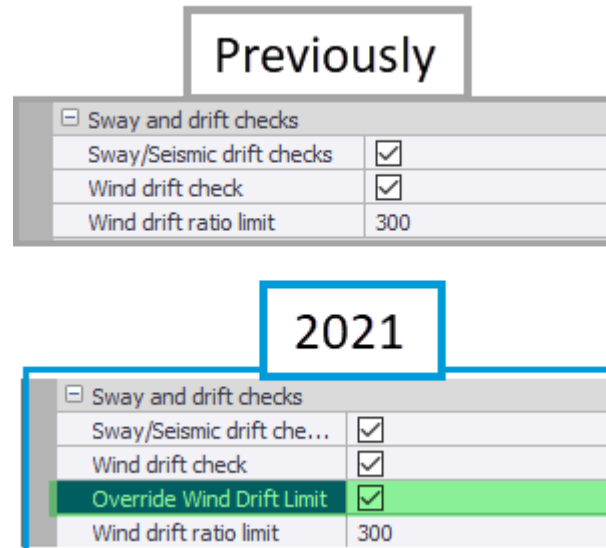
- The Review View > Show/Alter State Wind Drift controls for the "Set On" mode are also enhanced accordingly; options are added to enable the override option and set the limit value for selected members. The view color-coding and legend update to reflect this operation for the current drift limit value to show; members with the current override applied, those with other override values and the default (set in Design options), as shown in the picture below.
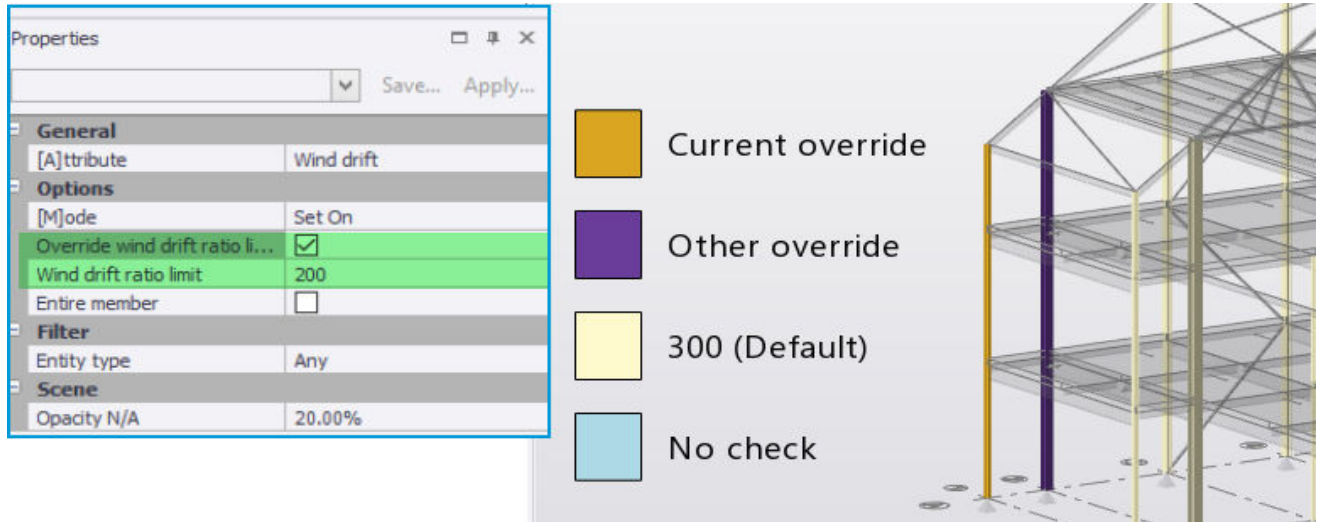
- **Resultant Wind Drift** - to cater for wind loads/ resulting displacements being at some intermediate angle w.r.t. the building directions, a new Design setting has also been added to set the Wind Drift check to consider *Resultant* (i.e. square root of sum squared) drift values, rather than values for each building direction separately as previously, as also shown in the picture above right. This is on by default and, in conjunction with the enhancement detailed above, produces a much more immediately useful and compact results table, as illustrated in the picture below. Simply set the new setting off should you wish to see check results for each direction separately.

  - Note that you can also now select individual load combinations to review their results via the combinations drop list at the bottom right of the program window as shown below. By default this is set to "All Combinations", giving the critical results considering all combinations. In recent previous releases this was the default setting and could not be changed to view individual combination results.

  - The check details report the derivation of the resultant deflection and drift values when this setting is enabled.

2021

Wind Drift								
Level ▼	Ref.	Stack	Combination	Resultant Deflection [mm]	Resultant Drift [mm]	Ratio Limit	Ratio	Status	Details
Top	SC A/4	4	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	105.9	24.5	300.000	108.721	✖ Fail	Details...
St. 3 (3)	SC A/1	3	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	81.5	18.0	300.000	148.468	✖ Fail	Details...
St. 2 (2)	SC C/5	2	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	63.3	19.0	300.000	210.654	✖ Fail	Details...
St. 1 (1)	SC A/1	1	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	44.5	44.5	300.000	89.803	✖ Fail	Details...

7 STR$_{6.1}$-1.35G+1.5Q+1.5ψ_0S+1.5ψ_0W+EHF$_{Dir1+}$ - Stack 3

Deflection at top in Dir 1, $\Delta_{Dir1(top)}$	40.7	mm
Deflection at top in Dir 2, $\Delta_{Dir2(top)}$	0.1	mm
Resultant deflection at top, $\Delta_{DirResultant(top)}$	40.7	mm
Deflection at bottom in Dir 1, $\Delta_{Dir1(bottom)}$	31.7	mm
Deflection at bottom in Dir 2, $\Delta_{Dir2(bottom)}$	0.1	mm
Resultant deflection at bottom, $\Delta_{DirResultant(bottom)}$	31.7	mm
Resultant story drift, $\delta_{DirResultant}$	9.0	mm
Story height, h	4.000	m
Wind drift ratio	444.417	
Wind drift limit	300.000	
Status	✓ Pass	

13 STR$_{8.3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1-}$
14 STR$_{8.4}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir2-}$
15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$
16 STR$_{9.2}$-G+1.5W+EHF$_{Dir2+}$
17 STR$_{9.3}$-G+1.5W+EHF$_{Dir1-}$
18 STR$_{9.4}$-G+1.5W+EHF$_{Dir2-}$

General

✓ Show Process

All Combinations
All Combinations

Wind Drift												
Reference	Combination	Stack No.	Deflection Dir 1 [mm]	Deflection Dir 2 [mm]	Drift Dir 1 [mm]	Drift Dir 2 [mm]	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2	Details	
SC A/1	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	-0.1	44.5	0.1	89.803	53955.818	✖ Fail	✔ Pass	Details...	
SC A/2	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	-0.1	44.5	0.1	89.860	53955.818	✖ Fail	✔ Pass	Details...	
SC A/3	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	-0.1	44.5	0.1	89.974	53955.818	✖ Fail	✔ Pass	Details...	
SC A/4	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.4	-0.1	44.4	0.1	90.088	53955.818	✖ Fail	✔ Pass	Details...	
SC A/5	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.3	-0.1	44.3	0.1	90.203	53955.818	✖ Fail	✔ Pass	Details...	
SC B/1	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	0.0	44.5	0.0	89.803	126586.250	✖ Fail	✔ Pass	Details...	
SC B/2	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	0.0	44.5	0.0	89.860	126586.250	✖ Fail	✔ Pass	Details...	
SC B/3	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	0.0	44.5	0.0	89.974	126586.250	✖ Fail	✔ Pass	Details...	
SC B/4	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.4	0.0	44.4	0.0	90.088	126586.250	✖ Fail	✔ Pass	Details...	
SC B/5	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.3	0.0	44.3	0.0	90.203	126586.250	✖ Fail	✔ Pass	Details...	
SC C/2	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	0.1	44.5	0.1	89.860	29126.338	✖ Fail	✔ Pass	Details...	
SC C/3	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.5	0.1	44.5	0.1	89.974	29126.338	✖ Fail	✔ Pass	Details...	
SC C/4	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.4	0.1	44.4	0.1	90.088	29126.338	✖ Fail	✔ Pass	Details...	
SC C/5	11 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$	1	44.3	0.1	44.3	0.1	90.203	29126.338	✖ Fail	✔ Pass	Details...	

- **Reports** - all the associated report items for the sway/ drift checks have also been updated accordingly to encompass the improvements detailed above. It is thus now possible to obtain much more succinct and clear sway/ drift reports.

  - The configuration settings for the Wind Drift item are also changed to reflect the enhancements as shown below - the options for reporting only critical stack/ combination are removed as no longer required.

Previously — Wind Drift Settings

Output
- ⦿ All columns/walls
- ○ Most critical columns/walls only

☑ Report only critical stack
☑ Report only critical combination

[OK] [Cancel]

2021 — Wind Drift Settings

Output per level
- ○ All columns/walls
- ⦿ Most critical columns/walls only

[OK] [Cancel]

	Project			Job Ref.	
Tekla Structural Designer	Structure			Sheet no.	
					Page 1/1
	Calc. by	Date	Chk'd by	Date	App'd by	Date
	paupa	04/03/2021		14/11/2014		14

Analysis

Wind Drift

Non-wall Elements

Level	Ref.	Stack	Combination	Resultant Deflection [mm]	Resultant Drift [mm]	Ratio Limit	Ratio	Status
Top	SC A/4	4	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	105.9	24.5	300.000	108.721	✖ Fail
St. 3 (3)	SC A/1	3	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	81.5	18.0	300.000	148.468	✖ Fail
St. 2 (2)	SC C/5	2	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	63.3	19.0	300.000	210.654	✖ Fail
St. 1 (1)	SC A/1	1	15 STR$_{9.1}$-G+1.5W+EHF$_{Dir1+}$	44.5	44.5	300.000	89.803	✖ Fail

Analysis

Sway

First-order linear

Level	Ref.	Stack	Combination Dir 1	$\alpha_{Dir\,1}$	Combination Dir 2	$\alpha_{Dir\,2}$	Combination Dir 1/2	Twist	Status
St. 3 (3)	SC B/1	3	2 STR$_1$ -1.35G+1.5Q+1.5RQ	13.977	2 STR$_1$ -1.35G+1.5Q+1.5RQ	89.352	15 STR$_{9.1}$ -G+1.5W+EHF$_{Dir1+}$	1.000	✔ Pass
St. 3 (3)	SC A/1	3	2 STR$_1$ -1.35G+1.5Q+1.5RQ	13.980	2 STR$_1$ -1.35G+1.5Q+1.5RQ	88.429	15 STR$_{9.1}$ -G+1.5W+EHF$_{Dir1+}$	1.000	✔ Pass
St. 2 (2)	SC A/5	2	2 STR$_1$ -1.35G+1.5Q+1.5RQ	10.353	2 STR$_1$ -1.35G+1.5Q+1.5RQ	47.488	15 STR$_{9.1}$ -G+1.5W+EHF$_{Dir1+}$	1.004	✔ Pass
St. 2 (2)	SC A/1	2	2 STR$_1$ -1.35G+1.5Q+1.5RQ	10.517	2 STR$_1$ -1.35G+1.5Q+1.5RQ	47.488	15 STR$_{9.1}$ -G+1.5W+EHF$_{Dir1+}$	1.006	✔ Pass
St. 1 (1)	SC A/5	1	2 STR$_1$ -1.35G+1.5Q+1.5RQ	3.512	2 STR$_1$ -1.35G+1.5Q+1.5RQ	40.461	15 STR$_{9.1}$ -G+1.5W+EHF$_{Dir1+}$	1.003	⚠ Warning
St. 1 (1)	SC A/1	1	2 STR$_1$ -1.35G+1.5Q+1.5RQ	3.516	2 STR$_1$ -1.35G+1.5Q+1.5RQ	40.461	15 STR$_{9.1}$ -G+1.5W+EHF$_{Dir1+}$	1.004	⚠ Warning

1.16 New Reinforced Concrete Design to ACI 318-2019 - US Head Code

ACI 318-2019 is added to the Resistance Codes available for USA Concrete Design. The scope essentially matches that of the existing 2008, 11 and 14 versions for Static and loading, and applies to all the following concrete elements; Reinforced Concrete Beams, Columns and Walls; Reinforced Concrete Slab Panels, Slab Patches and isolated Punching Checks; Reinforced Concrete Spread and Strip Footings, Pile Caps and Mat Foundations. Design for Seismic loading is covered for; Beams, Columns and Walls. There are many detail changes incorporated in the update in accordance with the new code version - for full details of these please see the new Help Topic and its subtopics. We include just a few highlights and items of note here.

- General:

  - Clause 20.2.2.4 of ACI-318-19 makes a revision to reinforcement strength, limiting it to 100 ksi. Accordingly a new 100 ksi reinforcement grade with common bars sizes has been added to the Materials database.
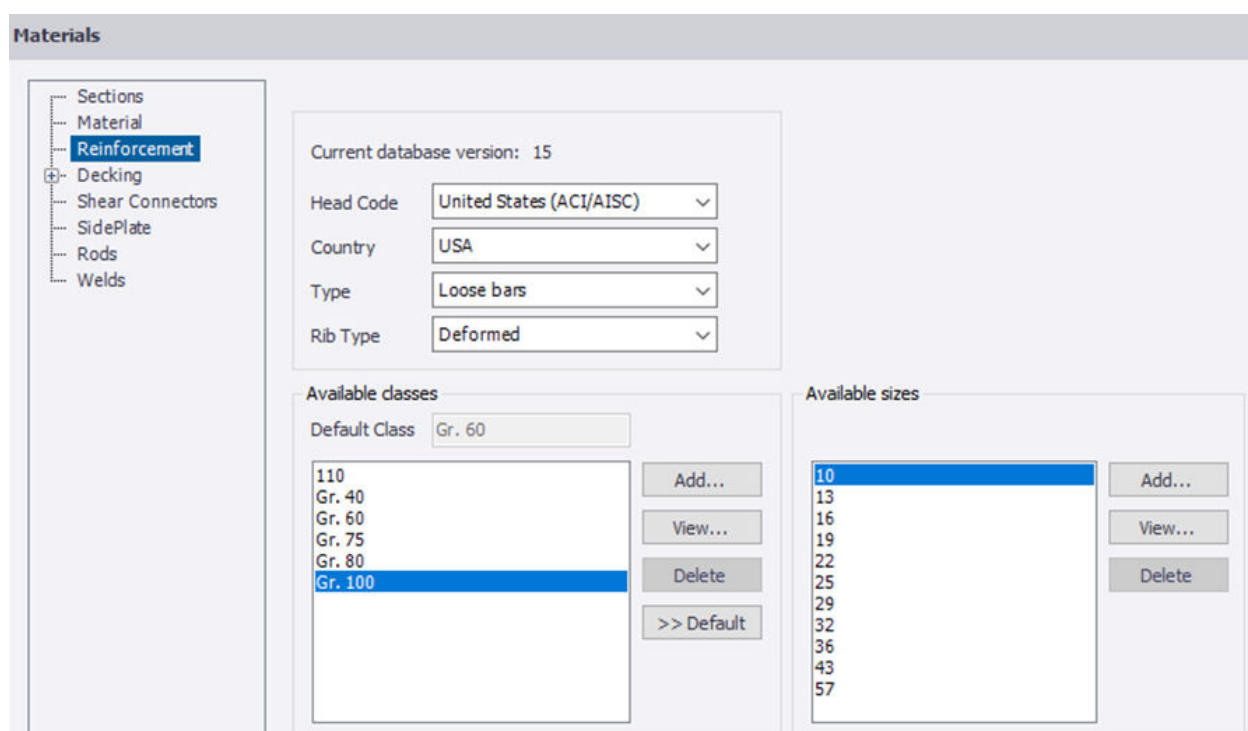
- As per Table 21.2.2 - For tension controlled sections the strain limit is revised from 0.004 to ϵ_{ty} + 0.003. This is reflected in updated design calculations wherever required in the calculation of flexure checks.

- Clause 19.2.1.1 has added a minimum compressive strength (f_{ck}) check.

1B6 - 4 250x500 - Longitudinal Bars - Bottom: 0.844 - 4.781 - 3D Building Analysis - 2 LRFD$_2$-1.2D+1.6L

Largest applied positive moment in region	$M_{u,region}$ = **128.3** kNm
Design moment in region	$M_u = |M_{u,region}|$ = **128.3** kNm
Strength reduction factor	ϕ = **0.900** ACI 318-19: Section 21.2
Effective depth in region	d = **437.8** mm
Width of compression face	b = **250.0** mm
Nominal strength coefficient of resistance	$R_n = M_u / (\phi \times b \times d^2)$ = **2.974** N/mm^2
Stress block depth factor	β_1 = **0.804** ACI 318-19: Section 22.2.2.4.3
Concrete compressive strength	f'_c = **34.474** N/mm^2
Steel yield strength	$f_{y,user}$ = **413.7** N/mm^2
Allowable steel yield strength	$f_y = MIN[\,f_{y,user}, 690\ MPa\,]$ = **413.7** N/mm^2 ACI 318-19: Table 20.2.2.4(a)
ω_t	$\omega_t = 0.85 \times \beta_1 \times 0.003 / (0.006 + (f_y/E_s))$ = **0.254**
Limiting strength coefficient of resistance	$R_{nt} = \omega_t \times (1 - (0.59 \times \omega_t)) \times f'_c$ = **7.444** N/mm^2 Notes on ACI 318-11 Section 10.3.4
$R_n \leq R_{nt}$: compression reinforcement not required	
Required tension reinforcement ratio	$\rho = MIN[\,0.85 \times (f'_c / f_y) \times (1 - \sqrt{1 - (2 \times R_n / (0.85 \times f'_c))}),\ \omega_t \times f'_c / f_y\,]$ = **0.008** Notes on ACI 318-11 Section 7 Eq.(3)
Required compression steel area for bending in region	A'_s = **0** mm^2
⚠ Required tension steel area for bending in region	$A_s = \rho \times b \times d$ = **831** mm^2

1B15 - 1 300x600 - Concrete Compressive Strength

Concrete compressive strength	f'_c = **15.000** N/mm^2
Minimum compressive strength of normal weight concrete	f'_c = **17.000** N/mm^2 ACI 318-19: Section 19.2.1.1
⚠ Warning	

- Concrete Column Design - Clause 22.5.5.1 introduces revised shear equations while Clause 22.5.1.10 adds a new biaxial shear check; column design is updated accordingly to these.
  - Clause 22.5.5.1 changes - account for size effect and for low longitudinal reinforcement ratios for members without shear reinforcement. The design equations are simplified (8 Equations → 3 Equations). The

changes of this clause are also reflected in revised sectional shear checks for other entities such as walls, slabs and foundations.

Table 22.5.5.1—V_c for nonprestressed members

Criteria		V_c	
$A_v \geq A_{v,min}$	Either of:	$\left[2\lambda\sqrt{f_c'} + \dfrac{N_u}{6A_g}\right]b_w d$	(a)
		$\left[8\lambda(\rho_w)^{1/3}\sqrt{f_c'} + \dfrac{N_u}{6A_g}\right]b_w d$	(b)
$A_v < A_{v,min}$		$\left[8\lambda_s\lambda(\rho_w)^{1/3}\sqrt{f_c'} + \dfrac{N_u}{6A_g}\right]b_w d$	(c)

22.5.1.10 The interaction of shear forces acting along orthogonal axes shall be permitted to be neglected if (a) or (b) is satisfied.

(a) $\dfrac{v_{u,x}}{\phi v_{n,x}} \leq 0.5$ \hfill (22.5.1.10a)

(b) $\dfrac{v_{u,y}}{\phi v_{n,y}} \leq 0.5$ \hfill (22.5.1.10b)

22.5.1.11 If $\dfrac{v_{u,x}}{\phi v_{n,x}} > 0.5$ and $\dfrac{v_{u,y}}{\phi v_{n,y}} > 0.5$ then Eq.

22.5.1.11) shall be satisfied.

$$\frac{v_{u,x}}{\phi v_{n,x}} + \frac{v_{u,y}}{\phi v_{n,y}} \leq 1.5 \hspace{2em} (22.5.1.11)$$

Shear resistance utilization ratio in major direction	$V_{u,major} / (\phi \times V_{n,major} / 2) = $ **1.211** ACI 318-19: Section 22.5.1.10(a)
Shear resistance utilization ratio in minor direction	$V_{u,minor} / (\phi \times V_{n,minor} / 2) = $ **1.363** ACI 318-19: Section 22.5.1.10(b)
Biaxial shear resistance utilization ratio	$(1 / 1.5) \times (V_{u,major} / (\phi \times V_{n,major}) + V_{u,minor} / (\phi \times V_{n,minor})) = $ **0.858** ACI 318-19: Section 22.5.1.11

✔ Pass

- Clause 15.4 is implemented which adds a joint shear check for static gravity loads, in line with that for seismic design.
- Seismic - the revisions of Clauses 20.2.2.4, and 19.1.1 to reinforcement strength/ yield strength and minimum compressive strength check respectively are also implemented in seismic design for beams, columns

and walls as appropriate. Design specific to the entity type is also updated in several areas accordingly:

- Beams - design implements the following clauses; Cl. 18.6.3.1, 18.8.2.3 and 18.6.4.4.

- Columns - design implements the following clauses; Cl. 18.7.3.1, 18.8.4.3, 18.7.5.3, 18.8.3.1, 18.7.5.5 and 18.4.3.3

- Walls - design implements the following clauses; Cl. 20.2.2.4, 19.2.1.1, 18.10.3.1 18.10.6.4(e) and 18.10.6.5

1.17 Foundation Design - New Lateral Load Check for Pile Caps and Piled Mats

This release adds the ability to check the piles of Pile Caps and Piled Mats for lateral loads (previously only vertical loads were checked). This enhancement applies to the following Head Codes; Eurocode (all countries), United States (ACI), India and British Standards.

- New Settings:

    - Pile Catalogue - you can now specify the lateral pile capacity as shown in the picture below.

- Design Settings - the lateral load check is activated by a new Concrete Design setting "Check piles for lateral load" (default is off) as shown in the picture below. There are separate settings for Isolated Foundations (Pile caps) and Mat Foundations in Concrete > Cast-in-place > Foundations.
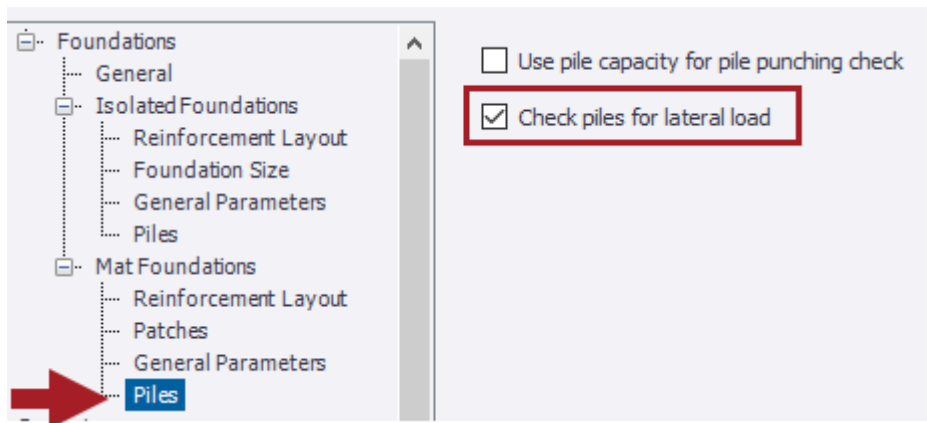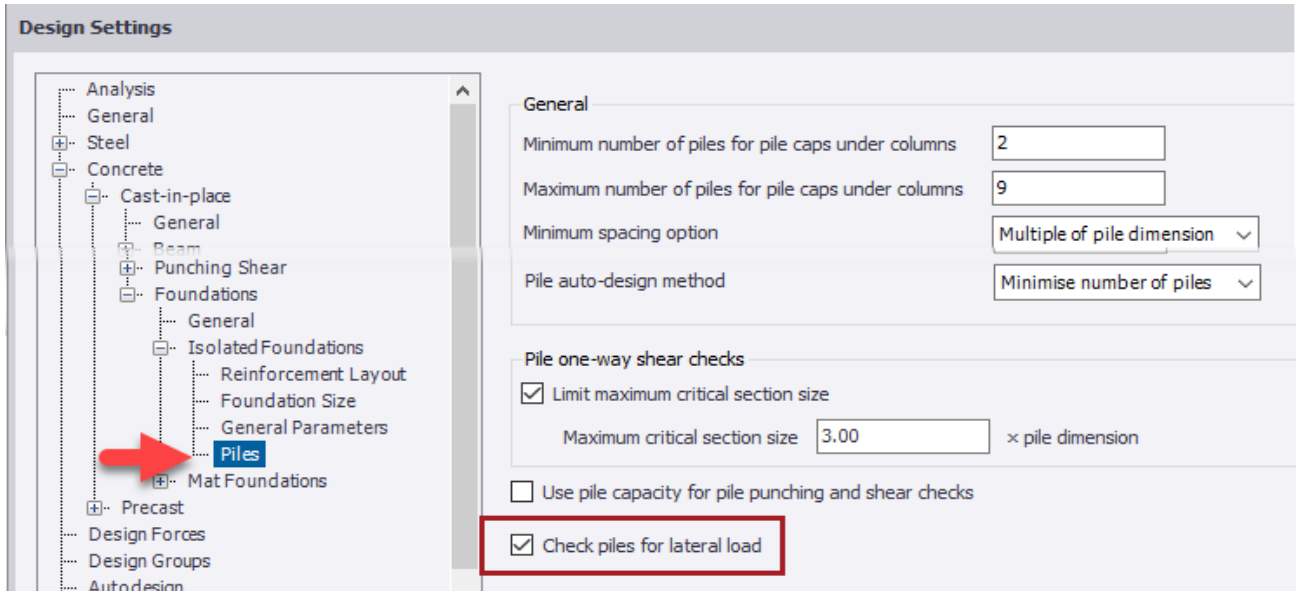
- Design Checks - the design details are enhanced to display the results of the new checks:

  - Pile Caps - the new "Pile Lateral Capacity" result group has both a summary table giving the results for the governing combination and check details for all checked combinations for all analysis types.

PC 1 results

Pile Lateral Capacity - Summary

	Lateral Resistance
Analysis	3D Building Analysis
Combination	2 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$
H$_F$	8.0 kN
H$_{Rd}$	20.0 kN
Ratio	0.401

PC 1 results

Pile Lateral Capacity - 3D Building Analysis - 2 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$

▷ Pile 1 : 8kN < 20kN ✔ Pass
▷ Pile 2 : 8kN < 20kN ✔ Pass
▷ Pile 3 : 8kN < 20kN ✔ Pass
▷ Pile 4 : 8kN < 20kN ✔ Pass

- Pile Mats - all individual pile design details include a new "Pile Lateral Capacity" giving the check details for the governing combination for each analysis type.

SI 1 results

P 5 - Pile Lateral Capacity - First-order linear

Lateral load in X direction		H$_{F,X}$ = -7.6 kN
Lateral load in Y direction		H$_{F,Y}$ = 0.2 kN
△ Lateral load in pile		H$_F$ = $\sqrt{H_{F,X}^2 + H_{F,Y}^2}$ = 7.7 kN
Critical combination		2 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$
Combination type		STR
△ Pile design lateral resistance		H$_{R,d}$ = 20.0 kN
H$_{R,d}$ ≥ H$_F$		✔ Pass

1.18 Design Using Tekla Portal Frame Designer - Improvements for Class 4 Haunches - Eurocode

For those designing steel portal frames to the Eurocode (UK and Irish NA's), there is a new companion release of Tekla Portal Frame Designer which features significant enhancements to the classification of haunches, in accordance with customer feedback. The Eurocode introduced tighter rules for section classification on the Class 3/ Class 4 boundary which resulted in many haunches that were adequate to BS 5950 being Class 4 at the deep end to the Eurocode. This resulted in a 'Fail' because Class 4 sections are not dealt with in Tekla Portal Frame designer. A number of improvements to the classification rules and equations are made in the new release to mitigate the occurrence of Class 4 haunches. For full details please see the Tekla Portal Frame Design 21 Release notes and Improvements to classification of haunched rafters to EC3. Briefly the enhancements are:

- Haunch Web Slenderness - calculation of the slenderness ratio c/t_w now includes the rolled section root radius thus reducing slenderness e.g. for a 405x140x46 section c/t_w reduces from 56.0 to 53.0

- Strength Check - Strength is checked at five positions with the first position now accounting for the end plate thickness.

- Classification of moment + axial - the combined compressive stress is 'normalised' to f_y, by increasing the bending stress while axial stress remains constant.

- Class 4 Design Status - when the haunch web is still Class 4 this is accorded a Warning rather than Fail status (note this applies only to the haunch web not that of the rafter). The design process is completed assuming the section is Class 3 - the designer can discuss this circumstance with the checking authority.

- Cross-section Strength Checks - classification now includes the more favourable provisions of Cl. 5.5.2(9) in which calculation of ε is based on the actual stress rather than that of grade S235.

1.19 Other Enhancements and Fixes

General & Modeling

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability. An example of this is given directly below:

    - [TSD-8916] - Solver model - the meshing of wall panels is improved to better handle connection to supporting columns and beams at corner nodes which, in some circumstances, could produce errors preventing analysis and design completion. For the circumstance of the wall corner node being inside a column section, the connection is now made in preference to the column, avoiding any beam rigid zones also within the column section.

        - Note that due to these changes, some existing models with rigid zones turned off will have the existing analyses invalidated automatically on opening in this release so these analyses must be re-run.

- [TSD-6878] - Views - Selection Display - the display of selected objects is improved to aid visualization and modeling as illustrated in the picture below. Where selected objects were previously alway opaque (below left) - thus obscuring the graphical information within their area - they can now be transparent (below right) allowing you to clearly see entities and grid/construction lines when, for example, making graphical edits to such entities as panels (of all types).

    - Selection transparency is controlled by a new setting Home > Settings > Scene > Graphics > Selection Style with option of "Transparent + Outlines" (default) and "Opaque" (previous behaviour).

    - Slabs - additionally, when selecting slab items, only the individual slab is now colored with the selection color, rather than the entire slab as previously.
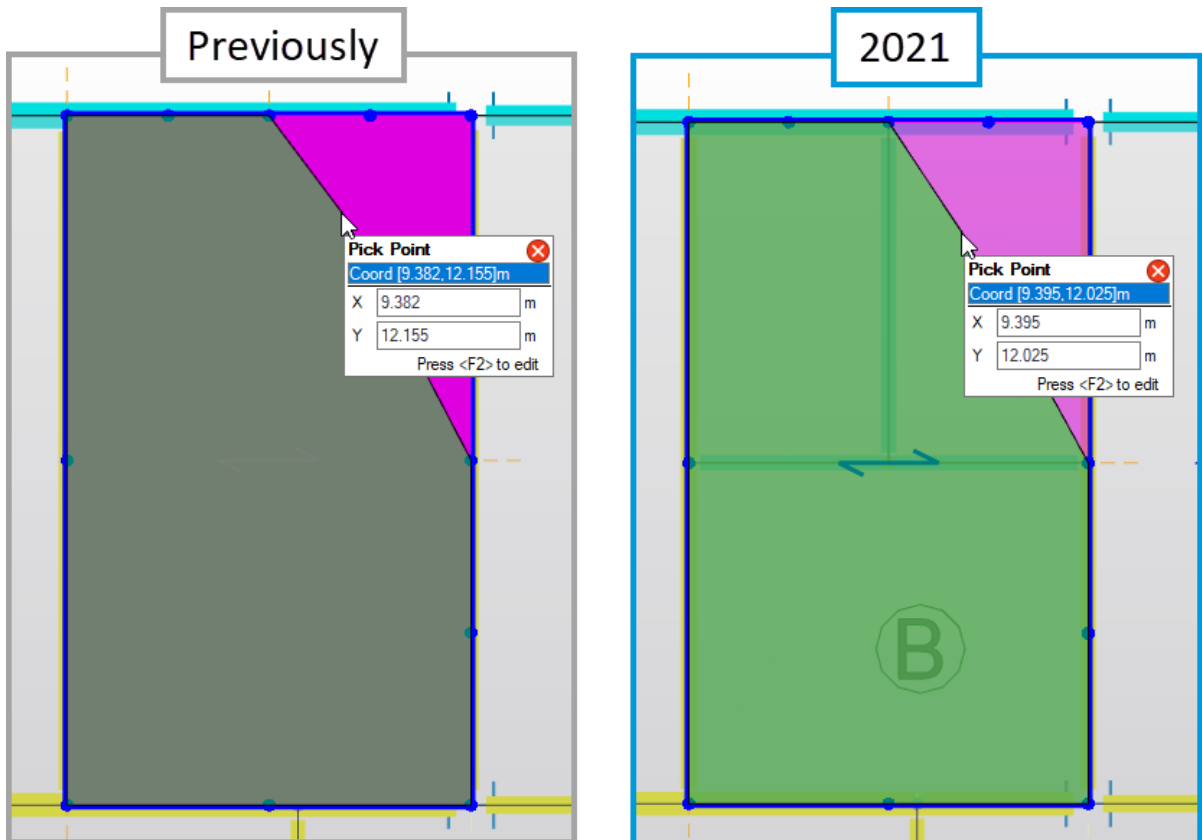
- [TSD-7659 & 7660] - Wall Panels - a new General 'Rectangular' property is added to wall properties (listed at the top Properties window during wall creation) the default for which is on. This facilitates the modeling of rectangular walls - when active you need only click on any two opposite corners of the panel shape to create a wall in a 3D view. The new setting should also help prevent the unintentional modeling of non-rectangular walls (by inadvertently selecting a slightly incorrect construction point/ intersection during modeling).

  - The new setting applies to all wall panel types as follows; Concrete Meshed walls, Concrete Mid-pier walls (can only be rectangular so the setting cannot be disabled for these), General (meshed) walls, Wind walls.

- Note that the new setting also applies when graphically adjusting the corner nodes of wall panels as illustrated in the picture below - after you have selected a wall panel then one its corner nodes, the Properties Window changes to show only the Rectangular edit 'mode' option. This will be checked on by default for a wall created with the setting on, and off otherwise. You can then enable/ disable this setting as you require - when enabled the wall shape remains rectangular and only a single node needs to be moved to another point to adjust the wall height and/ or width, the top of the wall remaining horizontal.
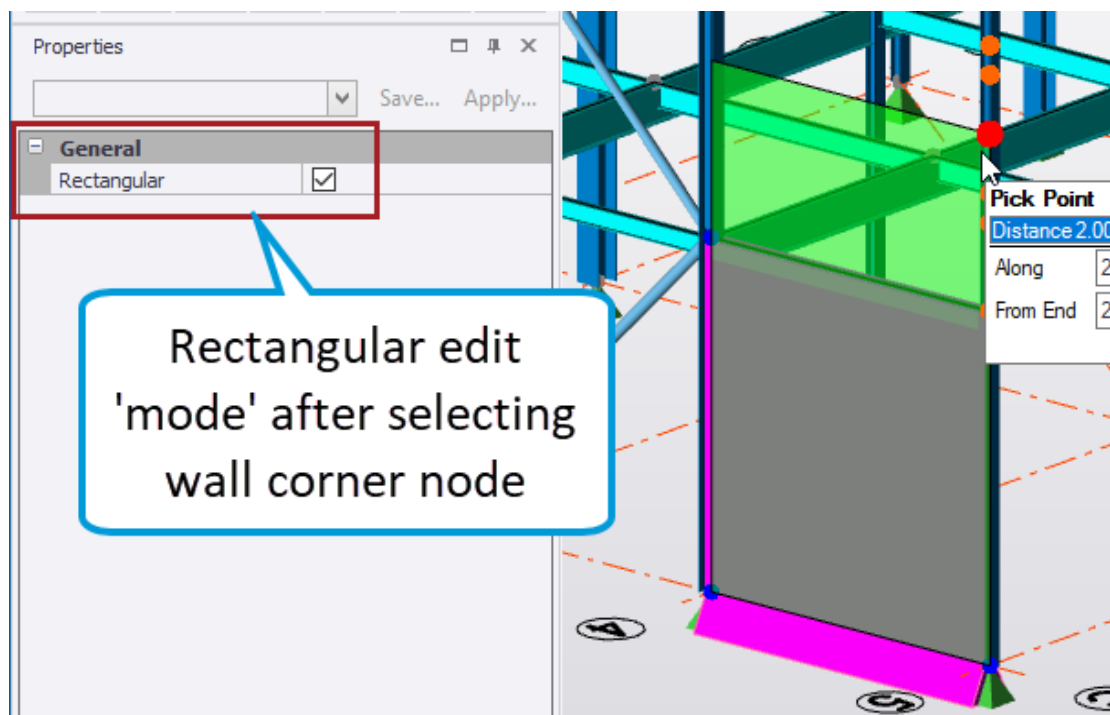
- [TSD-7837 & 8545] -Member & Wall Properties - Height/ Length information - the Properties Window data and entity tooltip information for selected members/ walls are enhanced to include the length/ height of selected entities, so you can quickly see this without needing to search in properties or measure it. The new information is added in a number of locations as detailed and illustrated below:

  - Columns and walls - the stack/ panel height is shown in brackets adjacent to each stack/ panel properties group (continuous columns/ walls) and also in the properties for each stack/ panel or the single stack/ panel.

  - Beams - the span length is shown in brackets adjacent to each span properties group (continuous beams) and also in the properties for each span or the single span.

- Tooltip - the information is also added to the cursor tooltip displayed when it is over an entity in a view; for walls the Length and Height is now included and for line elements the Length is included.
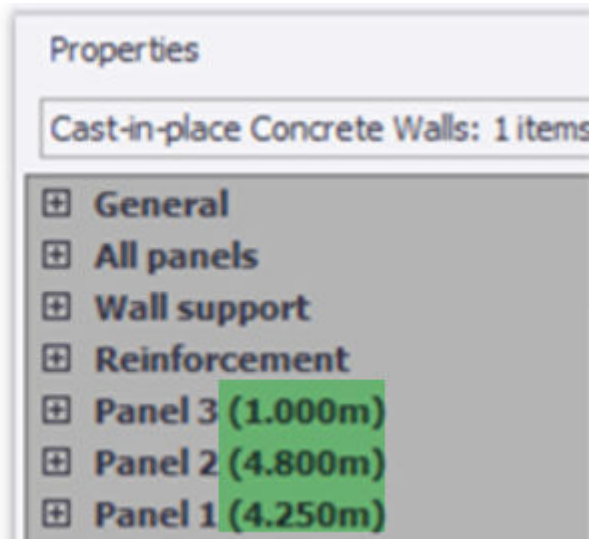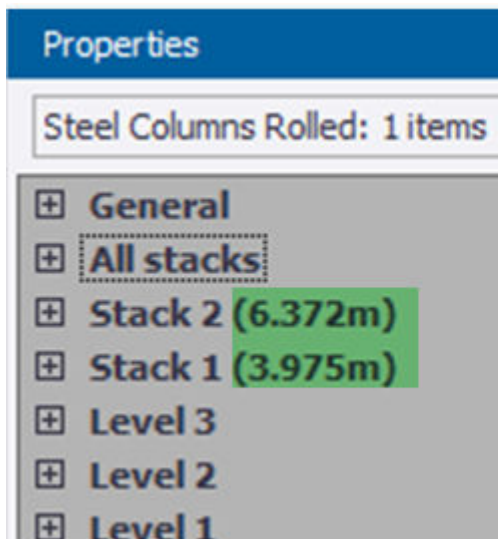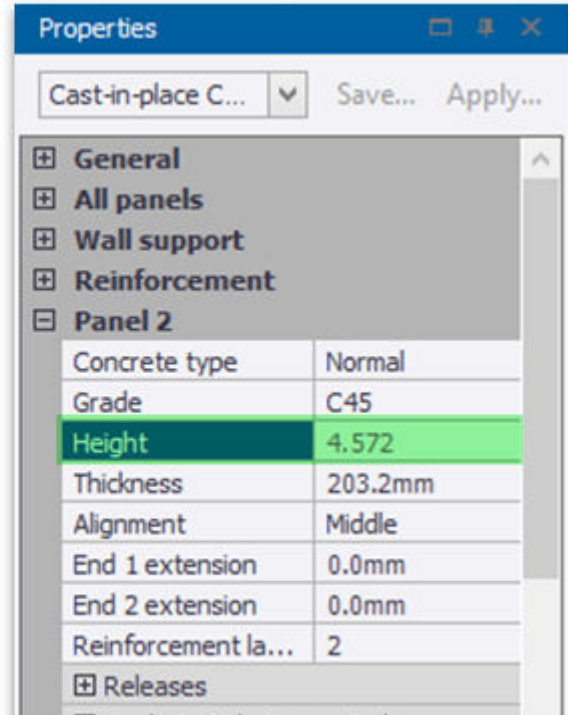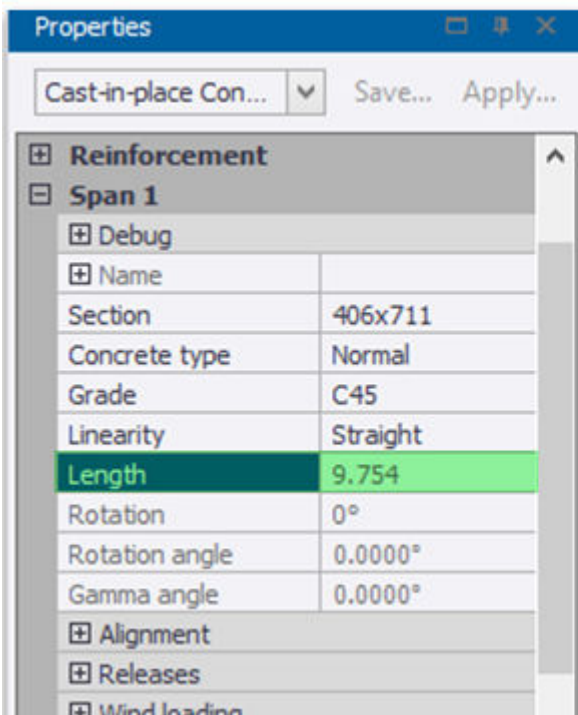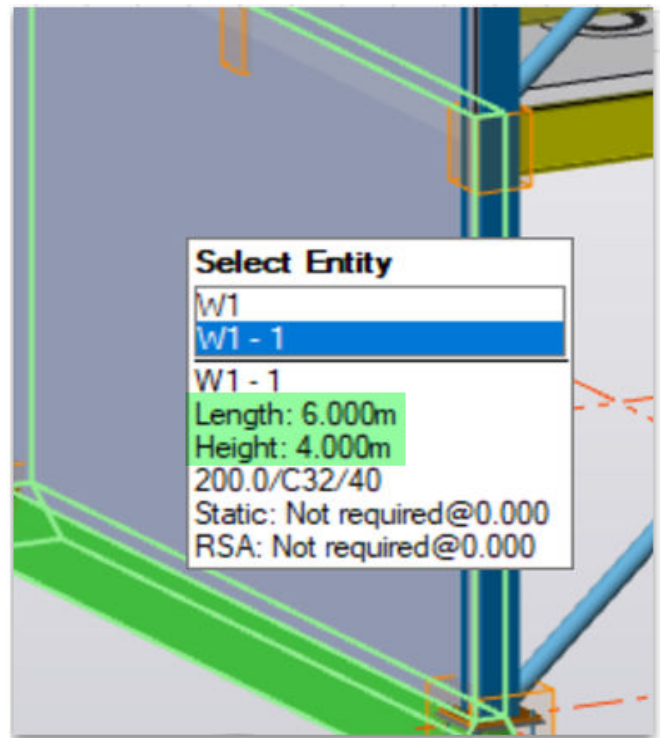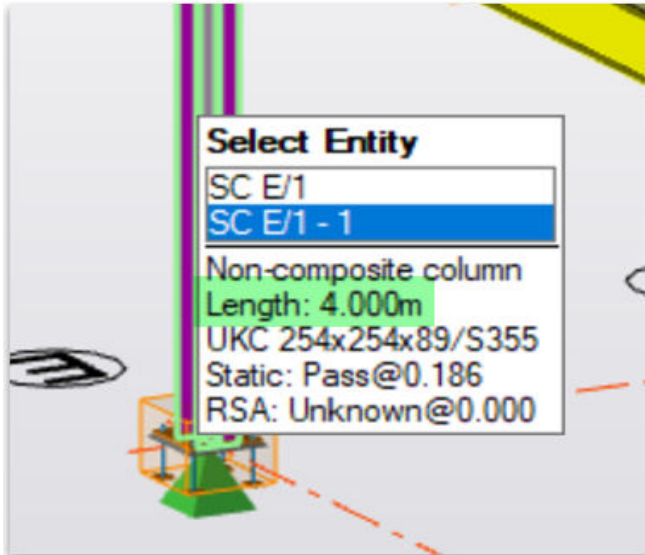
- [TSD-7722] - Slabs - Direction - to make it clearer and aid with modeling of slabs, the direction angle property of two-way spanning slabs is now termed "X direction", as illustrated in the picture below. This also shows how the X direction corresponds with the span direction icon for two-way slabs; the blue icon arrow shows the X direction, the angle being measured from the horizontal on plan.

  - The text displayed for the property when it is selected now also explains its significance; "X direction - Used for analysis and reinforcement alignment".
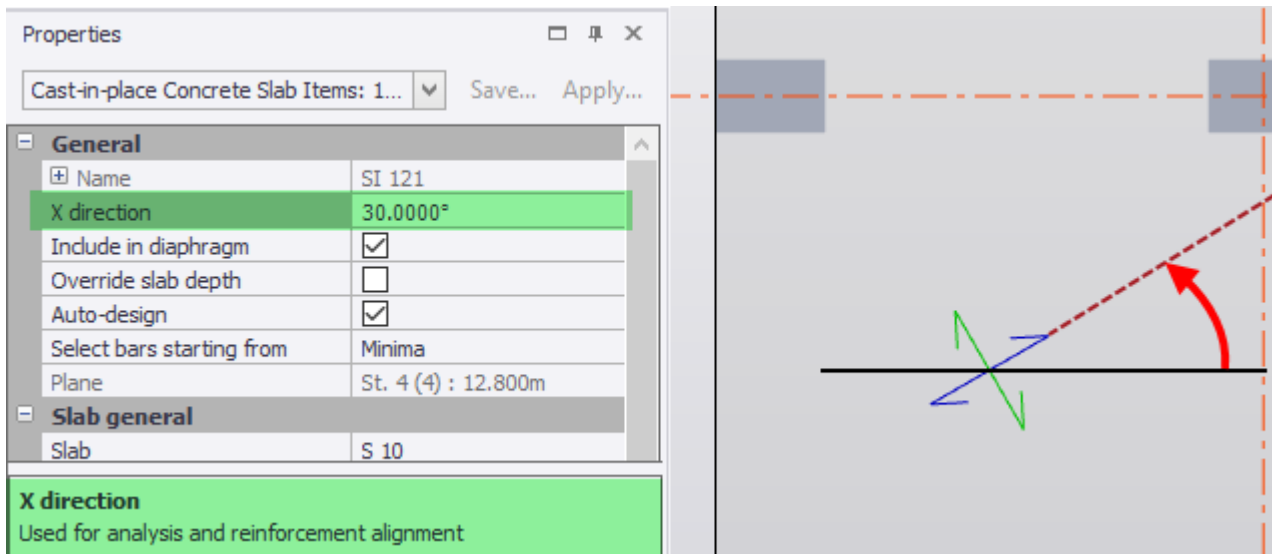
- [TSD-7722] - Slabs and Wall/ Roof Panels - the span direction of these can now easily be graphically edited, in both 2D and 3D views, using the mouse as follows (applies to both one-way and two-way spanning and all slab/ wall/ roof panel types):

  - Step 1 - select the slab/ panel item.

  - Step 2 - select the span direction icon.

  - Step 3 - the Properties Window now lists the direction mode options of "Align with edge" and "Define by 2 points" - select the desired mode.

  - Step 4 - select either a slab edge (of any slab item) or two points, depending on the selected mode, and the slab direction will automatically be set to the appropriate angle.

Select slab then span direction icon
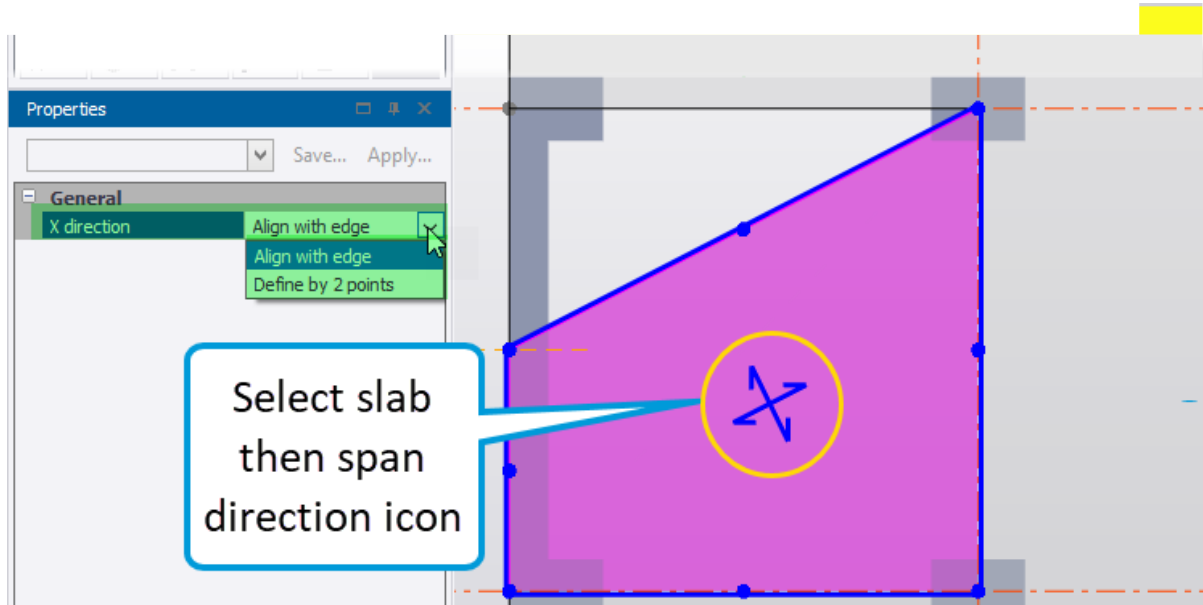
- [TSD-7636] - Steel Cellular Beams - Cellular beams have been added to the list of steel section types available for Thailand when this country is selected in the Select Section dialog. This allows users to build their cellular beam section database specific to this country, making them readily available during modeling when selecting section sizes.

Interoperability

- **BIM Integration** - Tekla Structures -there have been significant improvements to integration with Tekla Structures for the 2021 releases, especially for the transfer of rebar design intent information from Tekla Structural Designer to Tekla Structures. For optimum integration install both this release and Tekla Structures 2021 on the same computer. For more information please see the Tekla Structures 2021 Help Topic Tekla Structural Designer Import and Export. Please note the following:

  - For optimum performance of rebar transfer to Tekla Structures, ensure you set the following In Structural Designer; in Model Settings > Structural BIM > Export > Continuous Objects ensure the following are enabled:

    - Separate objects for each stack > Concrete columns
    - Separate objects for each span > Concrete beams
    - Separate objects for each panel > Shear walls
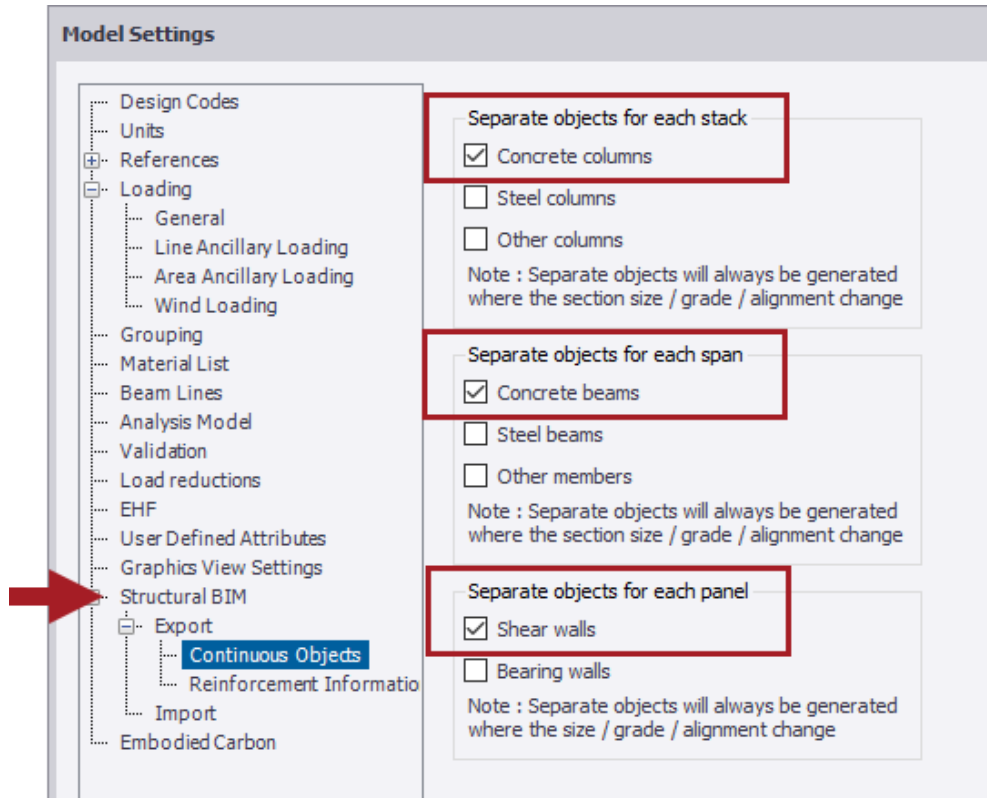
- **Remoting API** - first featured in the 2020 release, the Tekla Structural Designer Remoting API provides an interface for different kinds of applications to interact with the analysis model and results in Tekla Structural Designer. For more about using the API please see the getting started page in the Tekla Developer Centre. The API is further enhanced in this release as follows:

  - In terms of data extraction it provides access to:

    - Forces, displacements, camber and ecc. moments from member loading analysis at zero, max and any other position.

    - Additional results (wall and result lines, vibration and buckling analysis, seismic RSA).

    - Foundation reactions for RSA seismic and envelopes.

    - Construction points, point groups, levels, frames, slopes, slab items, slab openings, slab patches, slabs, roofs, structural walls, wind walls, supports, piles.

    - Architectural grids and construction lines.

  - In terms of model creation it allows:

    - Model and horizontal construction planes.

    - Architectural grids and construction lines.

    - Construction points.

- Analysis elements.
- It is also now possible to access the API remotely through a LAN or WAN.

Performance

- [TSD-4597] - Load Cases/ Combinations dialog - when making changes to the checkbox settings (such as "Active" for example) for a selection of a large number of combinations, the update process could take a significant amount of time in previous releases, during which it seemed the program was not responding (potentially up to several minutes where there were 200+ combinations). The time taken in such a case is now reduced to around 1s or less. This is a general improvement which applies to all similar list activation/ deactivation options within the program, but is likely to be most apparent in the load combinations dialog where a large combination list is relatively common.

Loading

- [TSD-7779] - Ancillaries - Imposed Load Cases - for the Eurocode (all Country NAs), the ψ and ϕ factors of the automatically created Ancillary Imposed load cases now have the following default values (which were previously zero); $\psi_1 = 1.0$, $\psi_2 = 0.9$, $\psi_3 = 0.8$, $\phi = 1.0$. This also applies to those of the new Equipment Load entities introduced in this release.

  - The engineer may wish to use different ψ / ϕ factors for certain Ancillary/ Equipment loads. This is also now enabled in this release by the addition of a new property setting for the Imposed Loadcase with options of 'default' and a list of the existing other imposed loadcases, as illustrated in the picture below for an Area Ancillary (also applies to Line Ancillaries and Equipment loads). This operates as follows:

    - When set to "default" the operation is as previously - the imposed loads are included in the automatically created imposed loadcases (which will be created if they do not already exist).

    - Otherwise you can select from the list of existing imposed loadcases to include the Ancillary/ Equipment imposed loads in one of these (the automated imposed cases will not be created if they do not already exist).

    - For Pipework Ancillaries and Equipment loads, the new Loadcase option is only listed when the setting for their Operating & Testing Content Loadcase Type is set to "Imposed" in Model Settings > Loading > General.
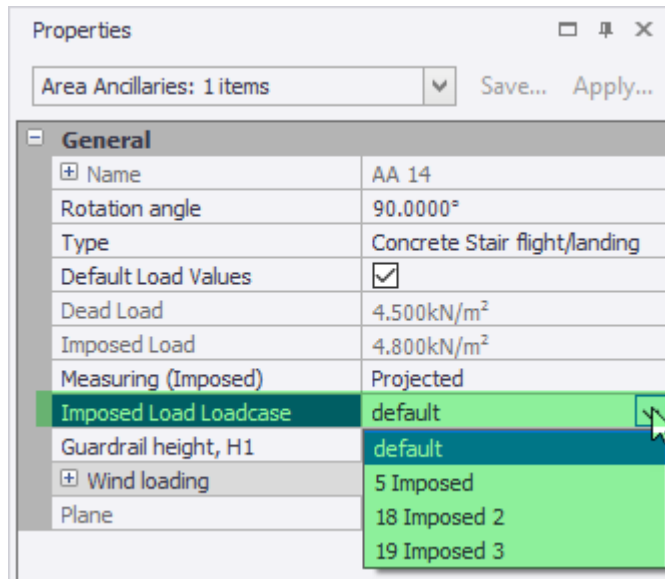
- [TSD-8425] - Snow Loading - US Head Code - a new property for snow drift loading "Do not apply balanced load" has been added to improve the application of multiple drift loads to the same roof panel. Previously it could be difficult to correctly size subsequent drift loads after the first drift - which included the balanced load - was defined. The operation of the new setting is as follows:

  - New models/ drift loads - the first drift applied to a roof panel has the balanced load applied so the new setting is disabled. For any subsequent drift loads applied to the same roof panel, the setting is automatically enabled so it does not have the balanced load applied but uses that of the first drift to determine its drift load and extent. This is illustrated in the picture below.

  - Existing models - you can enable the new setting for existing subsequent drifts and turn off Balanced load "Override load value" setting to correct these if necessary.
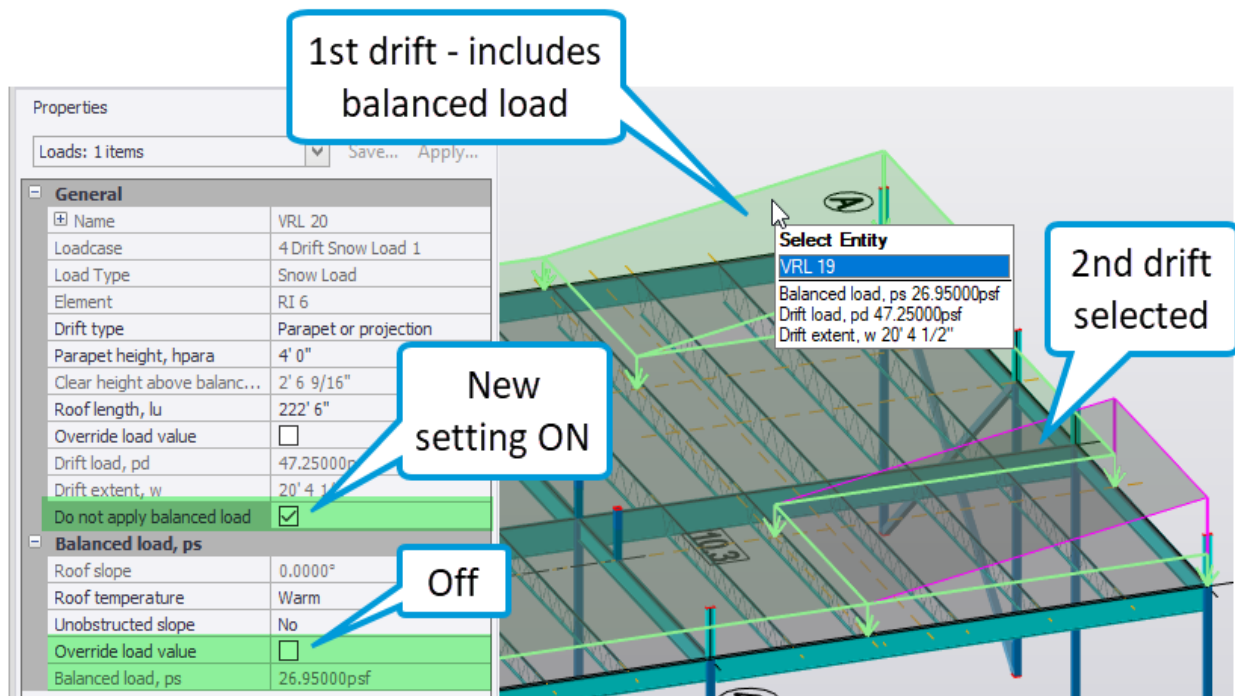
Validation

- [TSD-6123] - Validation Status - an inappropriate Validation status "Validation is outdated" message could be issued following certain operations that did not affect the status. This applies to the following operations and is fixed in this release; Slab design, Slab patch design, Punching check design, Report content editing (either model or element), Pad base design, Pile cap design, Design Mat.

- [TSD-7596] - Wall Panels - a new validation check has been introduced in this release for the shape/ orientation of wall panels to flag cases where they are not quite rectangular and/ or vertical, since such situations are probably due to modeling errors. When this situation occurs a new validation warning is now issued; "Wall panel almost but not exactly rectangular and vertical". The warning will be issued in the following cases:

  - A side edge that is between 1mm and 50mm out of true vertical, or

  - A top/bottom edge that is between 1mm and 50mm out of true horizontal.

- Note that a new 'Rectangular' property is also added to wall properties (default 'on') to help prevent the unintentional modeling of non-rectangular - see the details for issue TSD-7659 & 7660 in the **General & Modeling** for more on this.

Design - General

- [TSD-8468] - Review View - Design Utilization Ratio (UR) - the legend banding and colors are adjusted to improve the clarity of the view as follows:
    - Legend banding is changed so that everything below 50% is in single group (accessed via Home > Settings > Scene > Utilization Ratio)
    - The colors are changed such that green indicates a 'good' UR condition (applies to UR > 0.9)
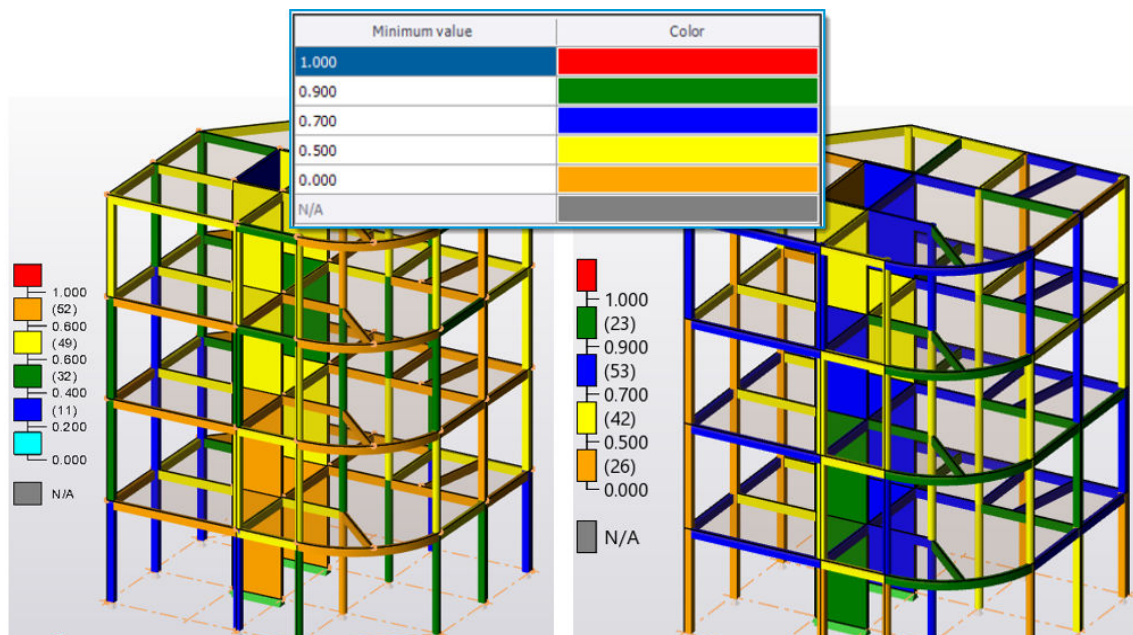
- [TSD-3382] - Interactive Concrete Design - Column & Wall Tie/ Link Spacing - All Head Codes - the design summary table in the shear design related tab of the interactive design dialogue (titled "Ties" or "Links" for columns, depending on the language set, and "Lateral" for walls) for reinforced concrete columns and walls has been enhanced to now report required areas of reinforcement and reinforcement utilization ratios where these were missing previously. The enhancement also ensures that the design Pass/Fail status is consistent between the check details and the interactive design dialogue.

- [TSD-7238] - Pile Caps - All Head Codes - a new iterative analytical method of distributing loads to multiple support points assuming rigid body behavior has been developed for this release and is now used to calculate the pile loads for isolated foundation Pile caps. This is expected to produce

somewhat different pile load distributions for asymmetric pile layouts to the method used in previous releases*.

- *We would note that all practical analytical methods of calculating the distribution of pile forces assuming (infinitely) rigid behavior are approximate. The previous method followed well-established and logical engineering principles to give a reasonable and realistic distribution for the majority of pile layouts, but had acknowledged limitations, particularly for unsymmetrical pile layouts. The new method addresses the limitations for such layouts and will give different/ improved results when these circumstances apply.

Design - Head Code US

- [TSD-8360] - Seismic Design - Steel - it is now possible to use structural hollow section columns in an ordinary moment frame (OMF) SFRS, in line with the requirements of AISC 341. Note the following:

  - This applies to rectangular, square and circular hollow sections.

  - Use of hollow section columns is still beyond scope for IMFs and SMFs.

  - Seismic classification is not applicable and Flexural torsion under axial compression is not checked.

Design - Head Code Eurocode

- [TSD-8802 & 8869] - Steel Design - Partial Safety Factors - making steel design more flexible, it is now possible to set your own user defined partial safety factor γ values. As shown in the picture below, this is enabled by the new setting Design > Steel > General > "Use user defined partial safety factors''. The settings are also found in Global Settings > Design for the Eurocode country Settings Sets where they can be set for all new models. When enabled, you can enter your own partial safety factor γ values to be used in steel design. This enhancement applies to all Eurocode Countries and NA's.

  - As part of this change, the Design settings for Composite Beams and Steel Joists have been grouped, together with the new Steel > General settings discussed above, in a new main "Steel" settings group. This change applies to all Head Codes.
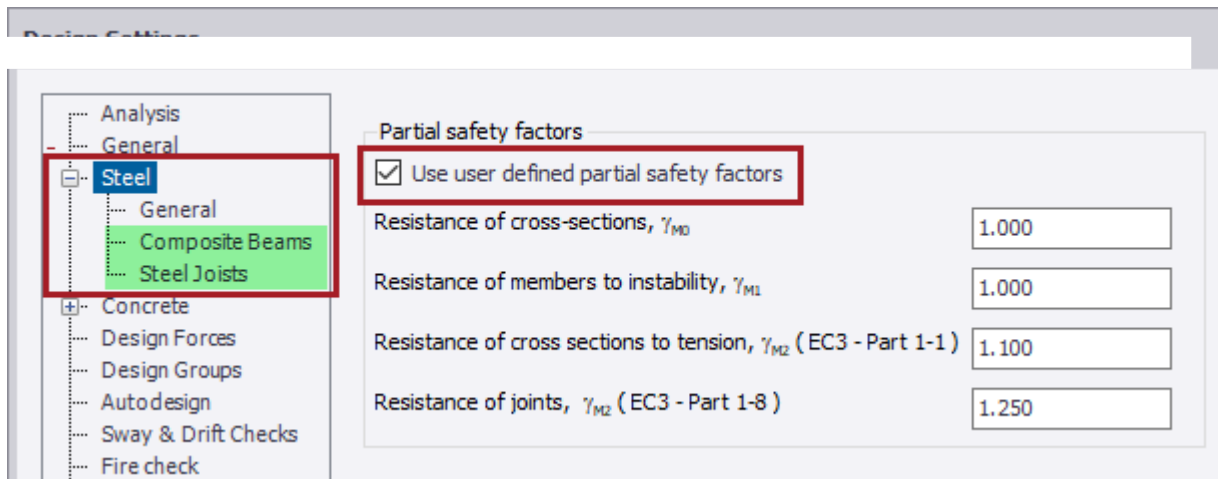
Reports

- [TSD-5371] - View Items - an issue is fixed in this release with the display in reports of consecutive view items with applied saved Wind View Configurations which caused the item to not display with an error "The image could not be generated!".

NOTE The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

1.20 Release notes: Tekla Structural Designer 2021 SP1

This release will upgrade your Tekla Structural Designer installation to version number 21.1.0.33 and should be installed to ensure optimum function of the program. It includes a number of enhancements and issue resolutions as detailed below.

If you are upgrading from a version earlier than the immediately prior release 2021 (version 21.0.0.116 released March 2021), you can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes

- Tekla User Assistance Service Pack release notes

- Tekla Downloads

Licensing & Installation

Licensing:

- No new license is required for this version.

- **Client License Service Version** - the Tekla Structural License Service V3.3.1.0 update should be installed with this release on clients*. This update of the license service is released to coincide with the release of Tekla Structural Designer 2021 SP1 because if this is installed but the License Service V3.3.1.0 update is not installed, then you may experience a blank sign-in window when attempting to sign-in to your Trimble Identity account from Tekla Structural Designer.

  - *This update does NOT need to be installed on Sentinel RMS license servers.

- **License Server Version** - for Sentinel RMS Server licensing, the latest version of the Tekla Structural License Service (v3.3.0) (incorporating Sentinel RMS 9.7) **must** be installed on your license server to be compatible with this release. Licensing will not function correctly if this is not the case. For more on this requirement please see the TUA article Tekla Analysis & Design 2021 Releases & Sentinel RMS Server Licensing - the License Server must be updated.

Installation:

- This service pack requires Tekla Structural Designer 2021 first release (version 21.0.0.116) to be installed and will update your current version.

- **Integration:**

  - **Tekla Structures** - if you wish to integrate with Tekla Structures you should install both this release and Tekla Structures 2021 for optimum performance.

    - Note that there have been significant improvements to integration - especially for rebar - in the 2021 release of Tekla Structures. For more information about this please see the Tekla Structures Help Topic Tekla Structural Designer Import and Export.

  - **Tekla Tedds** - to use Design using Tekla Tedds, for optimum performance you should install Tekla Tedds 2021. This can be obtained from Tekla Downloads.

  - **Tekla Portal Frame and Connection Designer 21** - if you wish to integrate this release with Tekla Portal Frame Designer/ Tekla Connection Designer - or use these products on their own - you should install Service Pack 1 for Tekla Portal Frame Designer 21/ Tekla Connection Designer 21. For more information about this release see their Release notes. The installation for this can be obtained from Tekla Downloads.

- **Autodesk Revit©** - the Tekla Structural Designer Integrator for Autodesk Revit© 2021 (version 7.0) was released on 1st July 2020 and updated to version 7.01 in September 2020. The installation and update are available in Tekla Downloads. If you are using Autodesk Revit© 2021, you can install this to perform BIM integration with Tekla Structural Designer.

  - All fixes and enhancements included in the 2021 release and update were also included in updates for the Integrators specific to the other currently supported Revit© versions (2018, 2019, 2020). For more information see the Tekla Structural Designer Integrator September 2020 updates. If you are performing BIM integration with any of these Revit© versions, we recommend you update to the latest version of the associated Integrator.

- **Previous Versions and file compatibility** - files from all previous versions can be opened in this release, however note that, once saved, they cannot then be opened in a previous release. If you wish to retain this option we therefore recommend using the File > Save As... option to save a copy of the file in the previous release and retain the original.

Highlights

- Imposed/ Live Load Reductions - Review View Enhancements - All Head Codes (page 113)

- Steel Beam Design - Enhanced Camber Settings & New Review View Attribute (page 120)

- Steel Column Base Plate Design - Enhanced Scope for USA Head Code (page 128)

- New Overall Wind Drift Check - All Head Codes (page 132)

General & Modeling

- A number of additional fixes which are not detailed explicitly here are also made to improve general performance and stability.

- Embodied Carbon: - a number of further enhancements are made to the Embodied Carbon calculation features as detailed below.

  - **New Embodied Carbon eLearning course** - to learn more about this novel and powerful feature we highly recommend you view the new eLearning for it here; Embodied carbon assessment in Tekla Structural Designer (eLearning).

  - [TSD-9226 and 9227] - Review View - the legend colors for the Embodied Carbon Review View > Show/Alter state > "Utilization and embodied carbon" attribute have been adjusted and enhanced to more clearly

display the distribution of embodied carbon in the model. As shown in the picture below, the colors now graduate from red (highest embodied carbon) to green (lowest), and include more gradations.

- In addition, the information text at top left of the Review View window now includes the Overall Embodied Carbon value (for both Carbon Factors and Utilization and embodied carbon options), the number of selected entities in the current view, their amount of Embodied carbon and the % of the total.

Embodied Carbon for Member : All
$kgCO_2e/m$

■	98 to 119 (14)
■	93 to 94 (20)
■	91 (24)
■	89 to 90 (52)
■	86 to 87 (8)
■	85 (12)
■	84 (22)
■	83 (24)
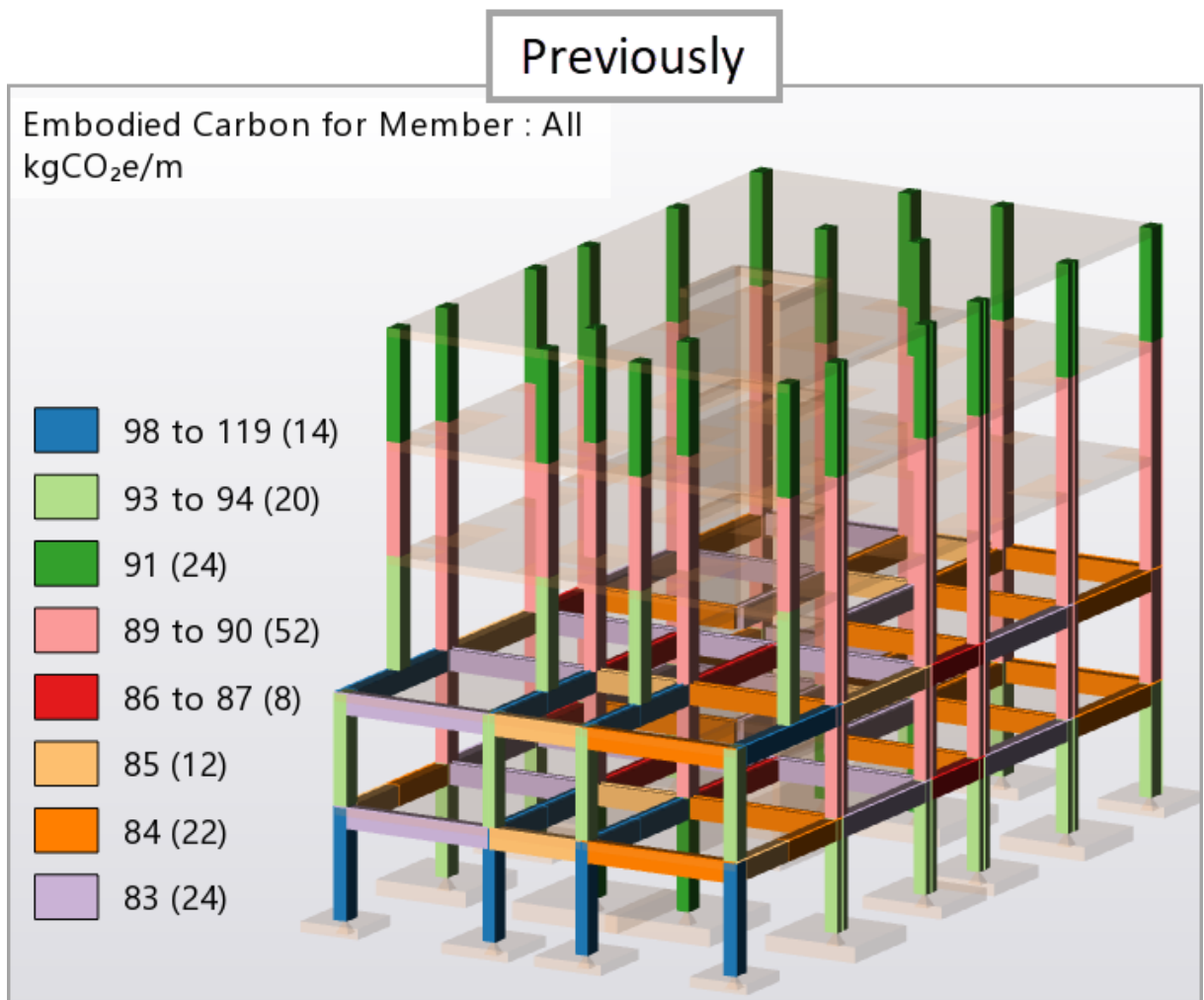
Embodied Carbon Totals:
  Overall: 213930 $kgCO_2e$
  176 Selected Entities : 54282 $kgCO_2e$ = 25.37% of total
Current Entity Filters:
  Members: Member : All
$kgCO_2e/m$

2021 SP1

■ (red)	116 to 119 (4)
■ (orange-red)	104 (4)
■ (orange)	98 (6)
■ (orange)	94 (4)
■ (amber)	93 (16)
■ (yellow)	91 (24)
■ (yellow-green)	90 (48)
■ (light green)	89 (4)
■ (green)	86 to 87 (8)
■ (green)	85 (12)
■ (dark green)	84 (22)
■ (dark green)	83 (24)
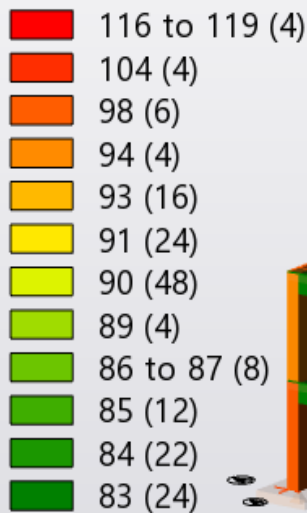
- Note the following:
  - The legend colors are controlled by new settings in Home > Settings > Scene > Colors > "Gradient Legend" as shown in the picture below.
  - When the displayed content requires fewer values the red and green extremes remain.
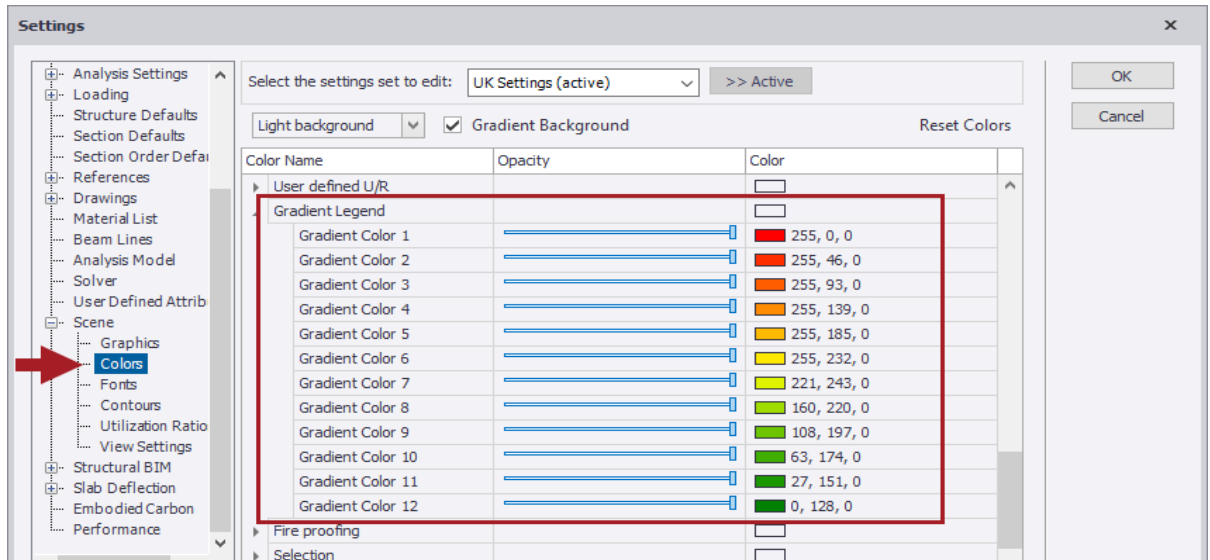
- [TSD-9213] - Embodied Carbon Factors Dialog - the count of entities to which a particular carbon factor is assigned given in the Embodied Carbon factors dialog is corrected to be consistent with that given in the Review View Legend for the Carbon Factors attribute, as shown in the picture below. Previously, the value in the Embodied Carbon factors dialog also counted the steel member connection allowance and so was commonly considerably larger.

Embodied Carbon - Overall Total : 146823 kgCO₂e
Embodied Carbon Factors for Steel
kgCO₂e/kg
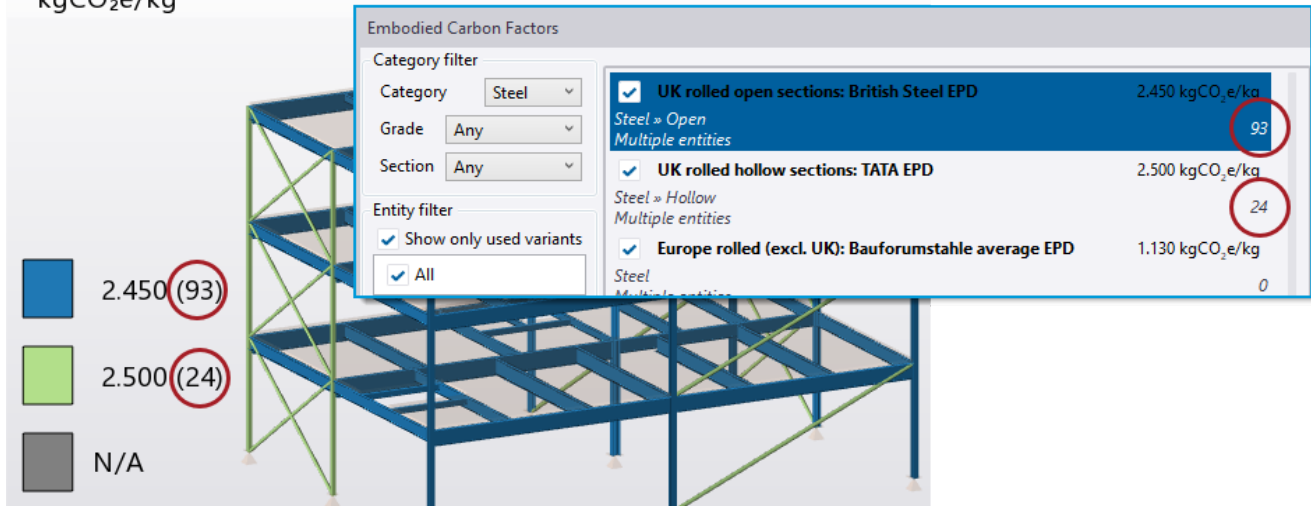
- [TSD-4951] - User Defined Attributes (UDA's) - UDA's have multiple uses - see for example the Help Topic . In this release, UDA's assigned to members can be automatically copied to duplicate floors, facilitating their application especially in larger models.

  - This feature is enabled via a new checkbox for each UDA titled "Linked" in the User Defined Attributes Settings (both Global and Model) as shown in the picture below (default setting is disabled). Simply check it on to enable the automatic copying to duplicate levels of specific UDA's.
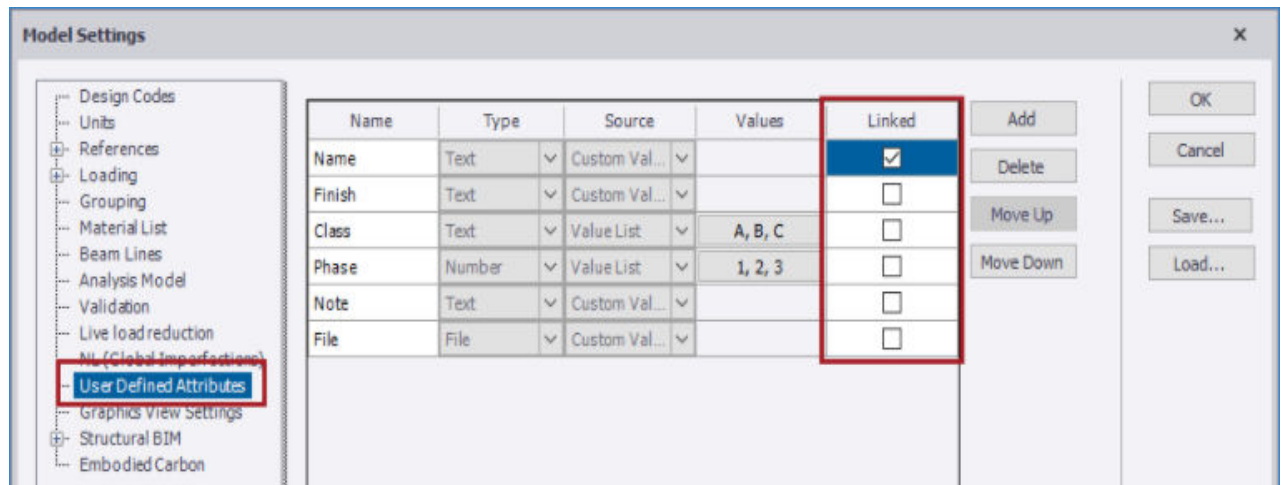
- Grid Lines:
  - [TSD-9328] - Grid Lines in 3D Views - Architectural grids have been made visible in all 3D Review Views, and are now excluded from selection and ghosting by default, which helps navigate the model when using the Review views. Please note the following:
    - This change means that if you wish to select and/ or edit the grid lines using the 3D structure view, you must now enable Scene Content > "Grid & Construction Lines" and "Show grids in 3D view" for a Level in the Structure Tree to make the grid lines selectable.
    - The functionality in 2D views is unchanged.
  - [TSD-4926] - Renumbering - the automated renumbering of Gridlines has been enhanced with additional controls. To use Renumber, select 'Renumber' from the right-click context menu for the "Architectural Grid" group of the Structure tree. At this point a new configuration dialog "Renumber Grid" opens, allowing you to specify how the renumbering will work. This has options of:
    - Detect Name type from direction (default ON) - when enabled there are two naming options for Vertical and Horizontal direction lines. When off there is only one
    - Direction - allows you to select the direction you want the renumbering to work in.
    - Line name type - Alphanumeric or Numeric - and Start value
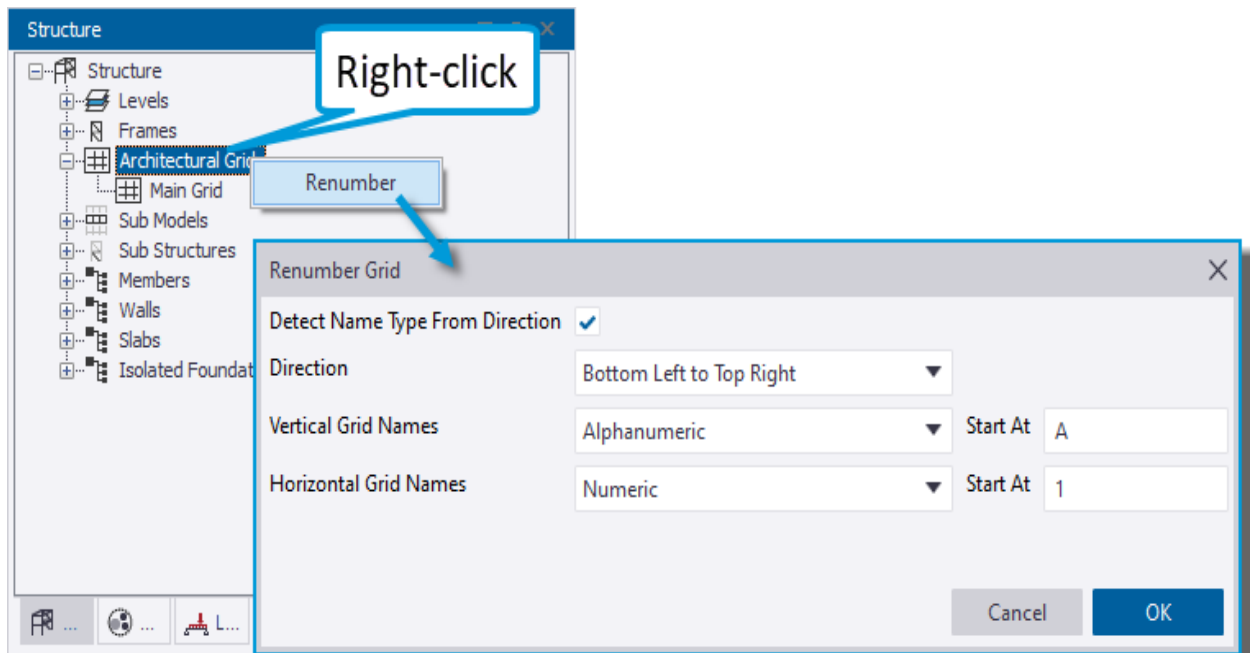
- [TSD-4944] - Grid Line Creation - when adding grid lines, two new options "Next Vertical Grid Name" and "Next Horizontal Grid Name" have been added to the Properties in the 'Grid Line Naming' group. These allow you to specify grid line name and type prior to its creation.
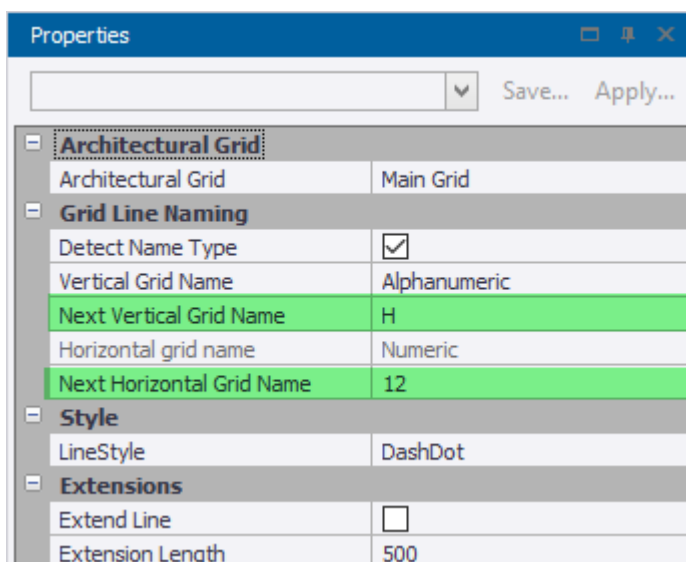

- Member Properties:
  - [TSD-9129] - Columns & Walls - the options 'Assume extra floors supported' and 'Count floor as being supported' present in the property

grid and/or member properties dialog box of column/wall are no longer listed for the USA and Australia Head Codes as they are not relevant to these. These options do not play any role in the calculations of Live/ imposed load reduction where this is based on the tributary area method.

- [TSD-8326] - Braces - in previous releases it was possible to enter invalid values for the properties Tension > Net area > Percentage value and Tension > Net area > Effective net area. This is now prevented in this release - a validation message giving the acceptable range of input is now displayed when an invalid value is entered.

  - Existing files - when opened in this release these values are checked and when invalid input exists a Validation Error is issued in the Status tree listing affected braces.

Interoperability

- [TSD-9235] - BIM Integration - Embodied Carbon - the Embodied carbon results can now be exported to external BIM applications as follows:

  - Tekla Structures or Revit - the Embodied carbon results are now automatically included in the Structural BIM Export CXL file which is produced by the Tekla Structures/ Revit Export options on the BIM Integration tab of the ribbon - they can then be displayed in Tekla Structures/ Revit as attributes as illustrated below.
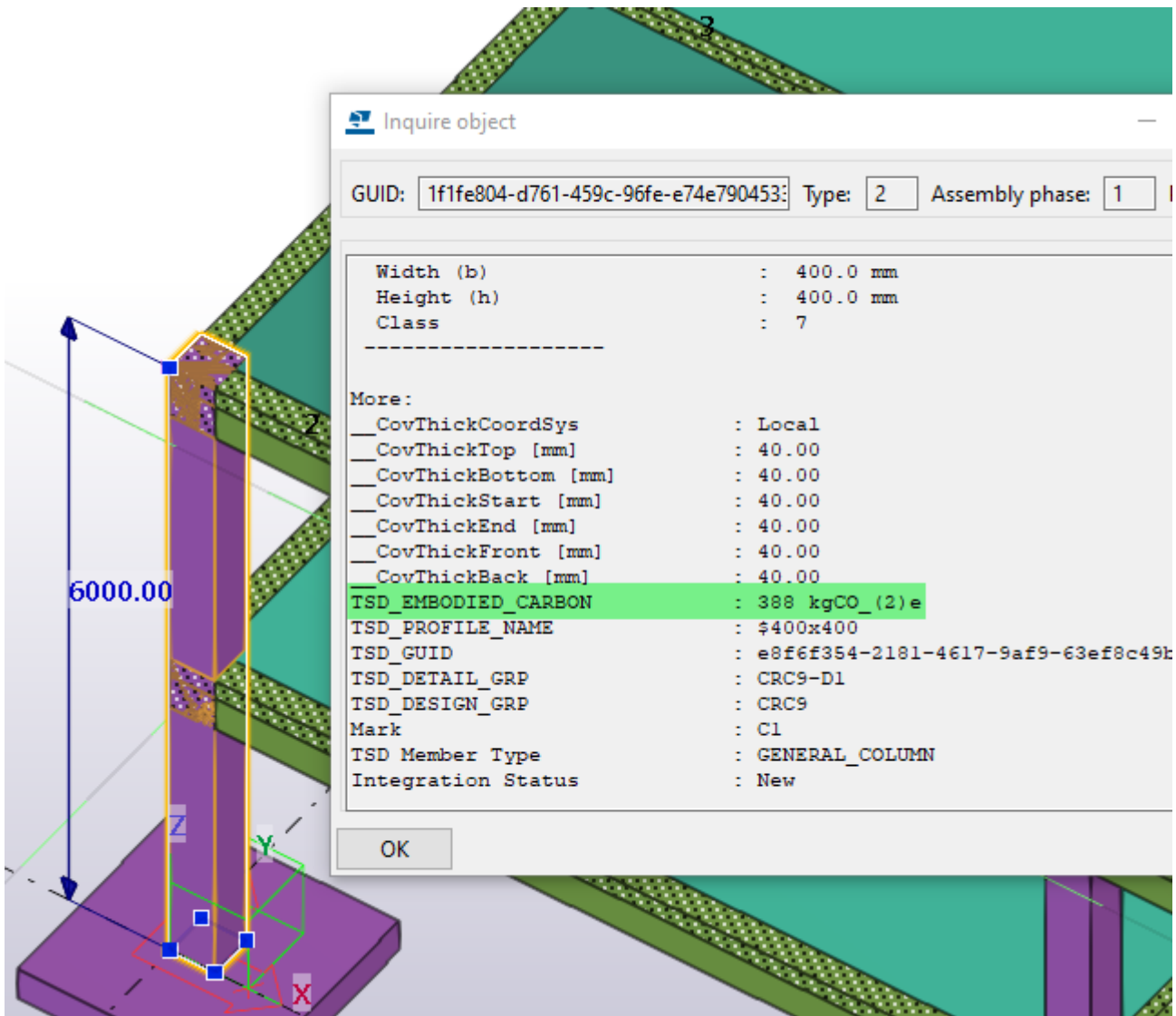
- IFC - the values are also included in the IFC export.
- Trimble Connect - uploading an IFC of the model to Trimble Connect will allow these values to be seen in the Properties when viewing the IFC model in Trimble Connect as illustrated below.
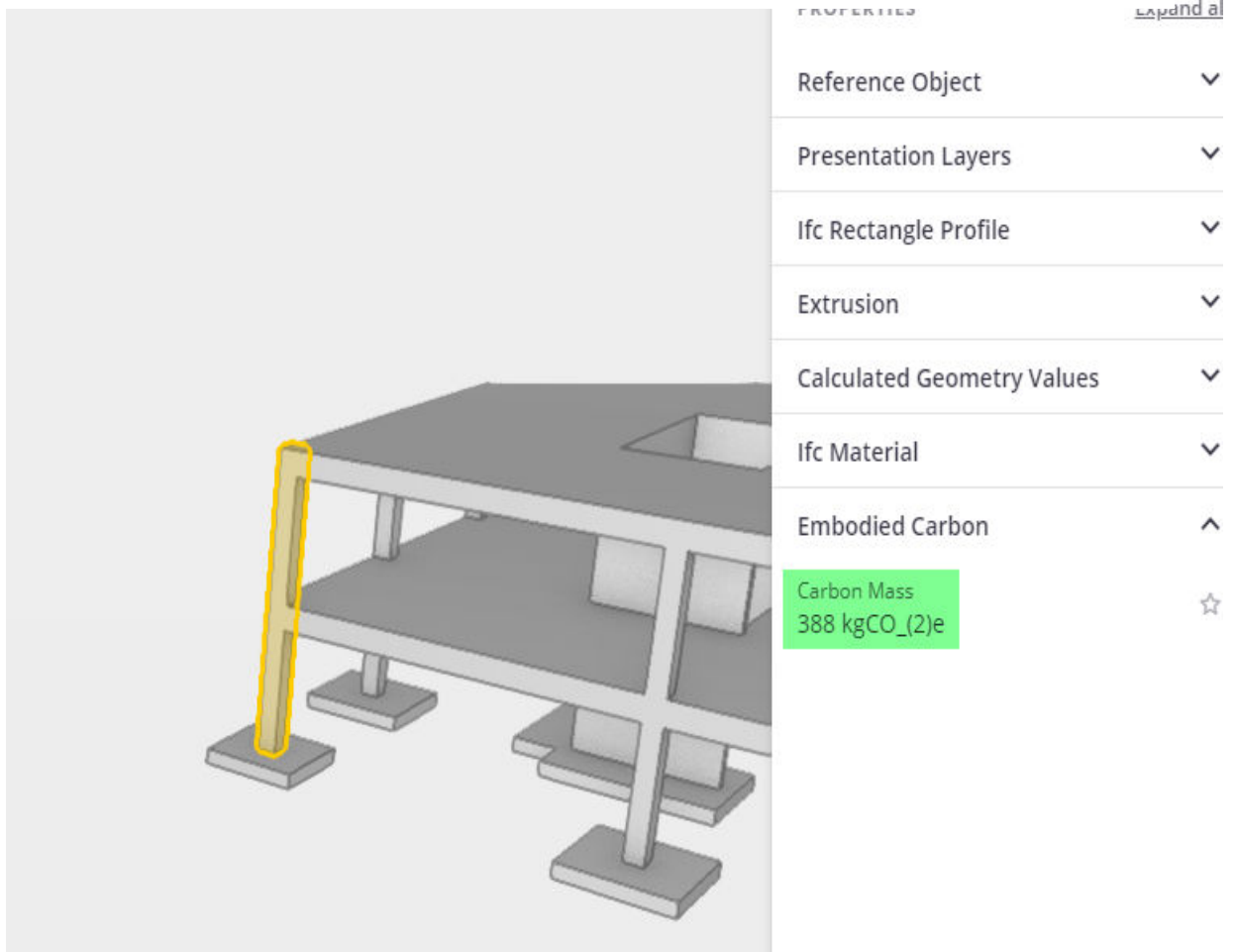
- In all cases, the values are exported for members, walls, slabs and foundations. One value per object is exported - the total resulting embodied carbon of all constituent materials of the object.

- [TSD-9219] - Grasshopper Live Link - Embodied Carbon - Embodied Carbon results and Rebar Mass per Volume can now be returned to Grasshopper using the Concrete Material Quantities and Steel Material Quantities components. These components operate with filters as previously, and

new output parameters give the total Embodied Carbon Mass of objects included in the filter, or the overall Rebar Mass per Volume.

- As well as reporting these values, this allows the user to optimize the Structural Designer model for these parameters using the optimization tools in Grasshopper.

For more about the Grasshopper Link see the TUA article Grasshopper - Tekla Structural Designer Live Link (GH-TSD). The link can be downloaded from the Tekla Warehouse here.

Loading

Highlights

- Imposed/ Live Load Reductions - Review View Enhancements - All Head Codes (page 113)

Other Loading Enhancements and Fixes

- [TSD-9144] - Equivalent Horizontal Forces (EHF) - Eurocode - it is now possible to override the calculated value of ϕ in the Model Settings > EHF dialog by selecting the new "Override" option as shown in the picture below. This enhancement is applicable to the Eurocode Head Code (all countries and NAs).

  - When the option is enabled you can enter you own values for ϕ_x and/ or ϕ_y and these are used in place of calculated values in the calculation of EHFs

  - In addition, In Model Settings the formerly titled "EHF" heading has been renamed to "EHF (Global Imperfections)" for clarity.
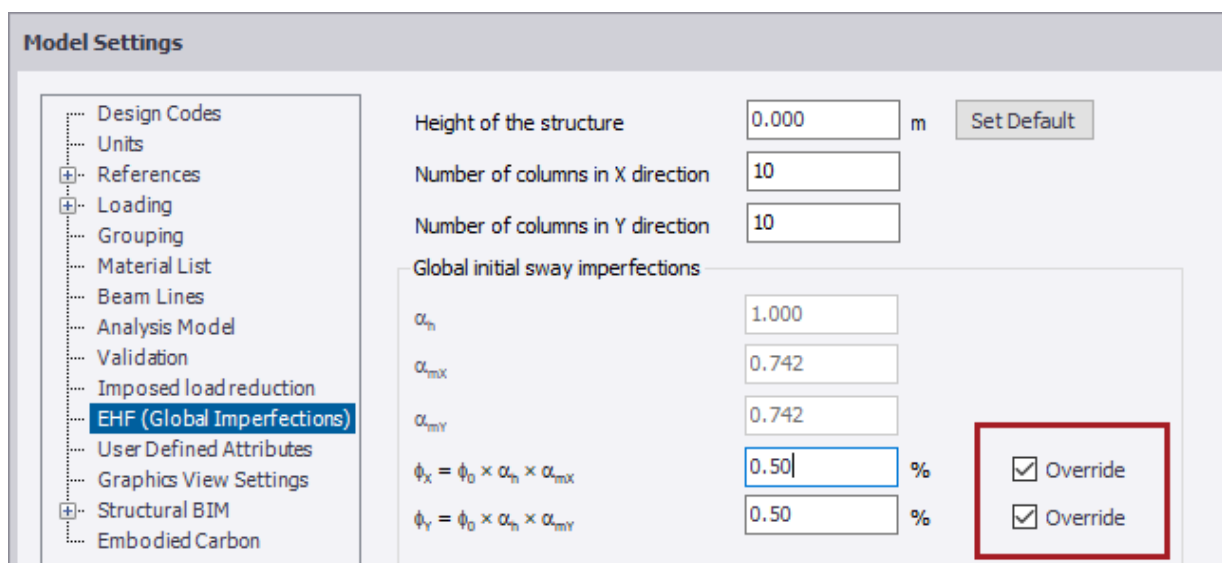
- [TSD-9284] - Notional Loads/ Horizontal Forces (NL/ NHF) - for Head Codes other than the Eurocode, a new "NL/NHF(Global Imperfections)" Setting is now included in Model Settings, as shown in the picture below (illustrates this for the USA Head Code). This includes override options for NL/ NHF parameters for each building direction - typically the proportion of gravity loads used to calculate the NL/ NHF's.
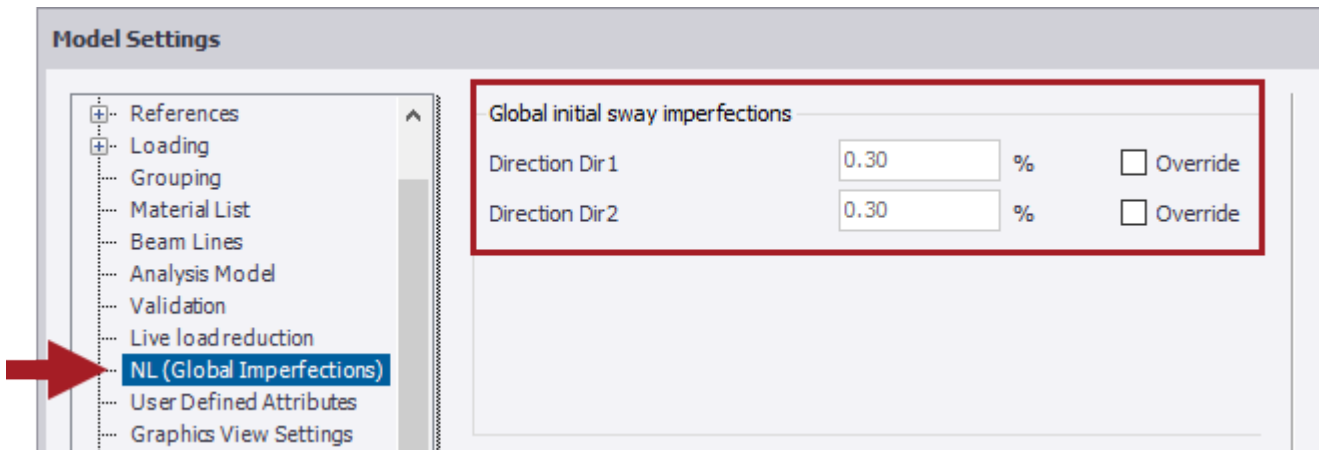
- [TSD-6477] - Diaphragm Loads - Creation - application of Diaphragm loads is now enabled in a 3D View. To facilitate this and their general application, the following new option and enhancements are added:
   - When in a 3D View, a new option "Select Diaphragm Level" is listed in the Properties window. Load application then operates as follows:
        - When enabled - application points will be displayed in all the diaphragm levels as you move the cursor over available points in the model. You then specify the load location in two steps: 1) first click an application point at any level - this 'locks' the application point and displays the available points at each diaphragm level intersected by a vertical line at the point - then 2) click the point on the level you wish the load to be applied - it is then created.
        - When disabled - the load is applied with a single click on the desired application point.
            - When clicking on a point that is not in a diaphragm level - for example a point along a grid line in the base level - the load is created at a position projected vertically to the nearest diaphragm level.
    - Snapping to members, grid and construction lines:
        - The application point can now also snap to points along a member when you move the cursor over one, in a similar manner to the application of member loads. Just as with applying member loads, the available snap points are; end, mid, 1/3 and 1/4 points. You can also apply the load at any point along the member - either by simply

clicking as you move the cursor along it, or press F2 on your keyboard to enter the exact distance along the member. This behavior applies to both 3D and 2D views.

- The application point can also snap to points along grid and construction lines in just the same manner as for members - again in both 3D and 2D views.

- [TSD-9160] - Model Settings - Live Load Reductions - USA Head Code - an issue is fixed in this release in which edits to the Live Load Reduction % limit values in Model Settings > Live Load Reductions were potentially not stored correctly when the model was saved. This specifically affected the following % limit values; "Roof live Beams", "Live walls supporting 1 floor", "Roof live Columns" and "Live walls supporting 2 or more floors" limits.

  - For existing models it is recommended that these limit values are reviewed and corrected if necessary then, when saved in this release, they will be stored correctly.

- [TSD-9239] - Ancillaries & Equipment - Operating & Testing Content Load Case type - for both Pipework Ancillaries (Line & Area) & Equipment load entities, the loadcase Type (either Dead or Live/ Imposed) for the automatically created Operating & Testing Content Load cases is specified in Model Settings > General (default Dead). This setting dictates the combination factor assigned to these loadcases so, if it is changed, existing combinations must be updated. To cater for this circumstance, a warning is added in this release to this effect when such a change is made as shown in the picture below.
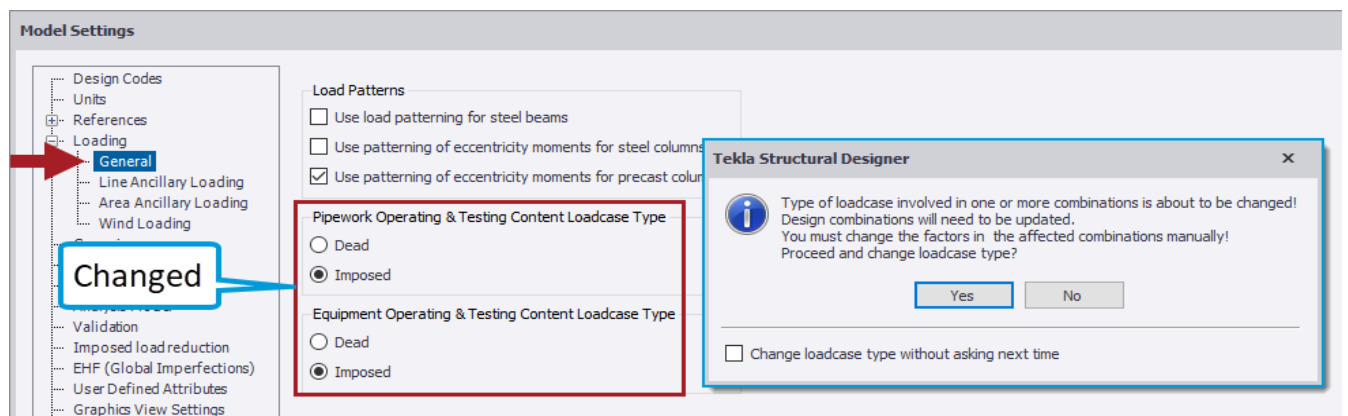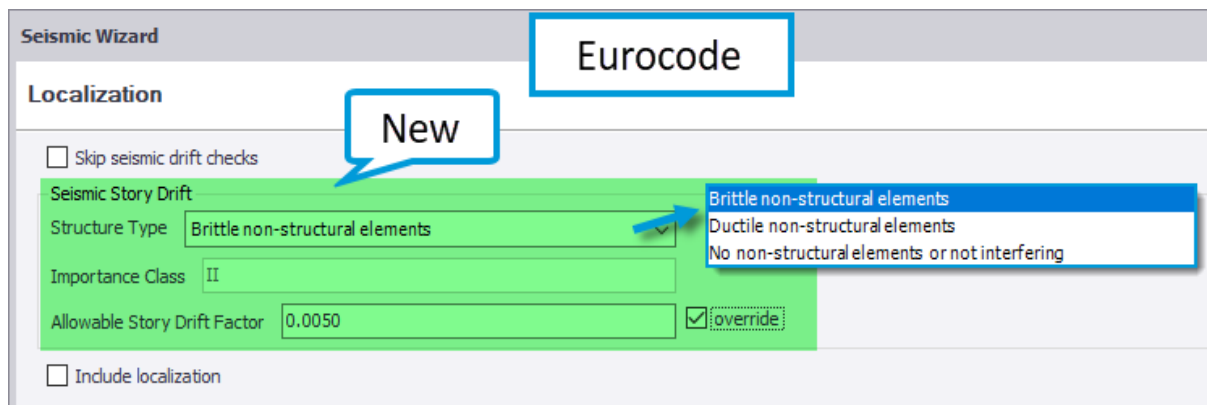
Design

Highlights

- Imposed/ Live Load Reductions - Review View Enhancements - All Head Codes (page 113)
- Steel Beam Design - Enhanced Camber Settings & New Review View Attribute (page 120)
- Steel Column Base Plate Design - Enhanced Scope for USA Head Code (page 128)
- New Overall Wind Drift Check - All Head Codes (page 132)

General Design Enhancements and Fixes

- [TSD-9039] - Steel Column Base Plates - Review Data - a Design Summary table has been added for Steel Column Base Plates. To view this summary, activate the Review View > Tabular Data Window, select the "Design Summary" option then enable the new "Base Plates'' Characteristic button as shown in the picture below. This enhancement applies to both the USA and Eurocode Head Codes. In common with other design summaries you can do all of the following:

  - Export the Table directly to Excel via the Excel Export button,

  - Click the "Results…" button at the end of a row to open the design details dialog.

  - Double-click a row to locate that particular base plate in the 3D Structure view.

Member Reference	Column Section	Column Grade	Plate Length [mm]	Plate Width [mm]	Plate Thickness [mm]	Plate Grade	No. Bolts	Utilization	Status	Results
BPC A/1	UC 203x203x46	S355	400.0	400.0	20.0	S355	4	0.252	✓ Pass	Results...
BPC B/1	UC 203x203x52	S355	400.0	400.0	20.0	S355	4	0.000	⚠ Warning	Results...
BPC C/1	UC 203x203x46	S355	400.0	400.0	20.0	S355	4	0.256	✓ Pass	Results...
BPC A/3	SHS 140x140x12.0	S355	300.0	300.0	15.0	S355	4	0.788	✓ Pass	Results...
BPC C/3	UC 203x203x60	S275	400.0	400.0	20.0	S355	4	0.000	⚠ Warning	Results...
BPC A/4	SHS 140x140x10.0	S355	300.0	300.0	15.0	S355	4	0.398	✓ Pass	Results...
BPC C/4	UC 203x203x71	S275	400.0	400.0	20.0	S355	4	0.261	✓ Pass	Results...
BPC 4/Ba	UC 203x203x52	S275	400.0	400.0	20.0	S355	4	0.372	✓ Pass	Results...
BPC 3/Ba	UC 254x254x73	S275	450.0	450.0	20.0	S355	4	0.728	✓ Pass	Results...

- [TSD-8591] - Seismic Story Drift - it is now possible to override the previously hardcoded Allowable Story Drift Factor for the Seismic Drift check. New settings for this are added to the Seismic Wizard on the Localization page, as illustrated in the pictures below for the USA and Eurocode Head Codes. This enhancement applies to all Head Codes which feature Seismic Analysis.

  - To enter your own override value, select the Structure Type option, enable the override checkbox then enter your desired factor value - this will then be used to calculate the allowable story drift. As shown below, the Structure type options where listed depend on the model Head Code.
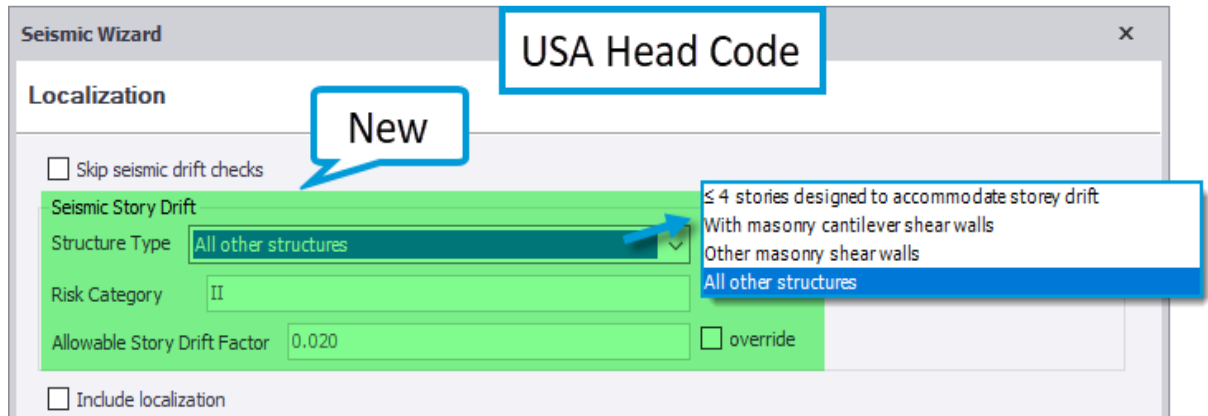
- [TSD-8799] - Steel Design - Auto-design Constraints - it is now possible to set the Size constraints "Min depth" and "Min width" as low as zero. Previously the lowest limit allowed was 50mm / 2inch which could cause problems with assigning acceptable sections during design in some circumstances.

  - The default Min depth and Min width values remain unchanged.

Eurocode Design Enhancements and Fixes

- [TSD-8542] - Steel Beam Design - Web Openings - for two circumstances when closely spaced web openings are such that the Web Posts check is beyond scope, the following explanatory messages are now issued where previously there was none (other than the Beyond Scope status):

  - Circular openings too closely spaced - "Closely spaced circular openings - spacing, s_{o} is less than 0.4 \times sum of the radii"

- Adjacent circular and rectangular openings too closely spaced - "Closely spaced openings - One opening is rectangular, one is circular and the spacing, s_{o} is less than 0.5 \times (rectangular opening depth + circular opening diameter)"

Reports and Drawings

- Report Styles

    - [TSD-7879] - Ancillary and Equipment Loads - a new report style 'Industrial Structure Loading' has been added to report the details of the following loading elements (which are typically - though not exclusively - used for Industrial structures); Line Ancillaries, Area Ancillaries & Equipment.

        - The new report consists of tables for each of these load entities, listing their principal properties and values; Name, Type, Location, Area, Loadcase Title and Applied load.

        - Note that existing models will not include the new default style, however you can easily create it as shown in the picture below by 1) Adding a new style (naming it as you like) then 2) dragging the new "Industrial Structure Loading" Chapter into this.
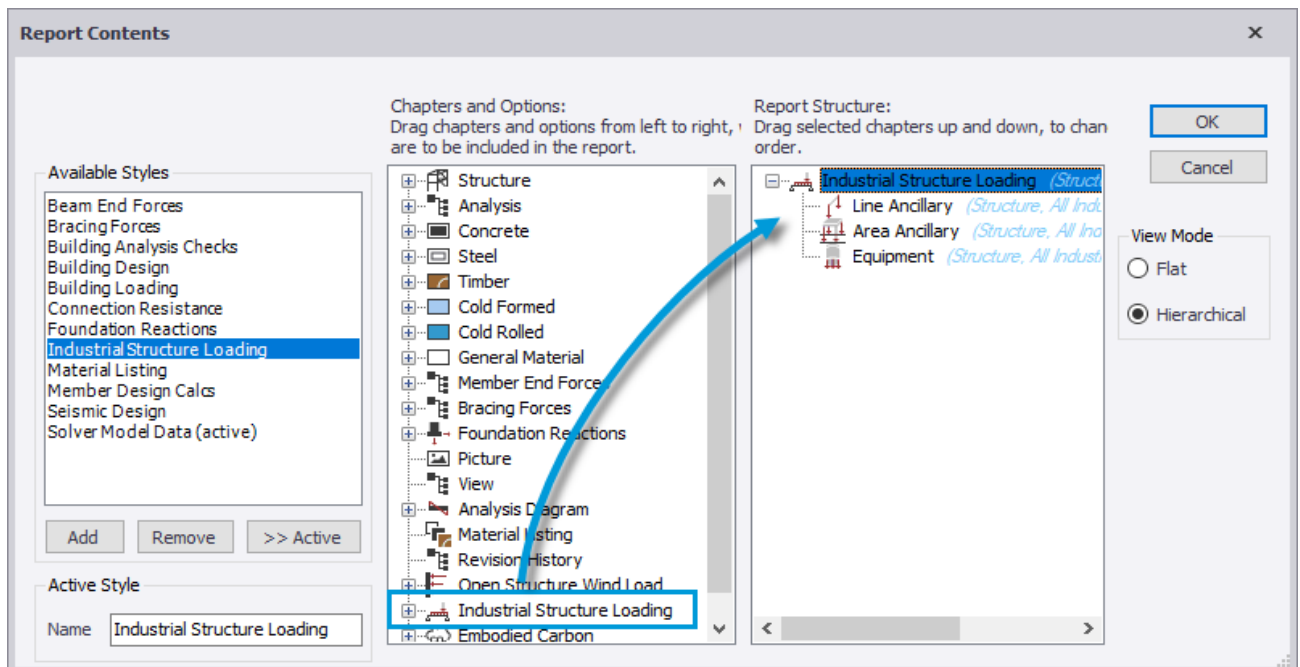
- [TSD-4951] - the default "Building Analysis checks" report style is renamed to"Building Analysis & Drift Checks". In addition the "Wind drift" option is added to this style, together with the "Loadcase summary" and "Sway / Drift" options as appropriate to the Head Code.

  - Note that existing models will not include these amendments, however you can easily adjust the existing "Building Analysis checks" style to make these same changes should you wish to.
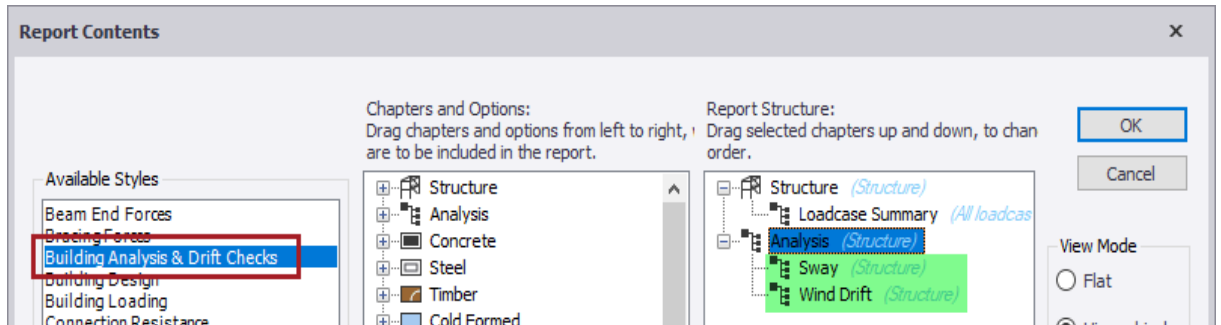

Notes

The number in brackets before an item denotes an internal reference number. This can be quoted to your local Support Department should further information on an item be required.

Imposed/ Live Load Reductions - Review View Enhancements - All Head Codes

While Tekla Structural Designer has always featured a sophisticated automated approach to the calculation of Imposed/ Live Load Reductions (as described in the Help Topic), the process of applying these reductions is quite simple (as described in this Help topic) and fully within the engineer's control.

The engineer may like to review automatically applied reductions and there are some situations where automatic reduction is not possible and/ or engineering judgment may be required, hence some manual review/ adjustment is necessary. To facilitate this, the Review View > Show Alter state is now enhanced with a new dedicated attribute for Imposed/ Live Load Reductions (I/LLR), enabling rapid and efficient graphical review and control of reduction settings and values.

An example is illustrated in the picture below (Eurocode) showing the new Review mode for the Imposed/ Live Load Reductions Show/Alter state Attribute.
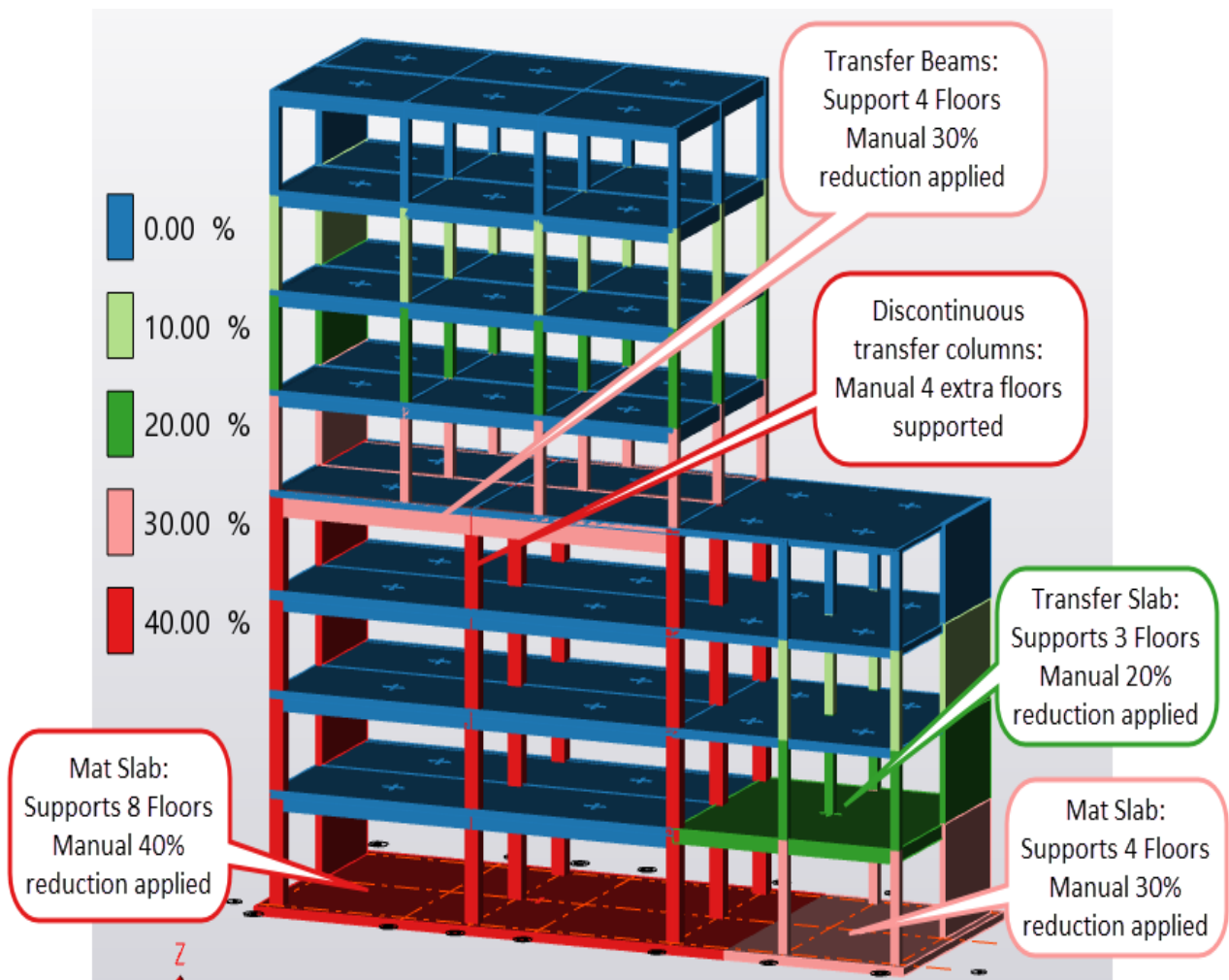
- In the above example, the following manual ILR adjustments are made:

  - Transfer beams in the fourth floor - logically these would have at least the same reduction as the columns they support, which themselves support four floors - hence at least 30%. A manual 30% reduction is applied

  - Discontinuous transfer columns - these also support four floors above so logically would have the same reduction as the continuous columns to either side supporting the same number of floors directly. A manual 4 extra floors supported is applied (entire member), producing an automatic 40% reduction for the full height.

- Transfer Slab in the first floor lower section of the building - logically this would have at least the same reduction as the column it supports, which itself supports three floors - so at least 20%. A manual 20% reduction is applied

- Mat slabs - following a similar rationale, the panel under the taller section of the building is set to 40% reduction (matching the columns/ walls it supports) and under the lower section to 30% reduction.

The operation and settings available for Imposed/ Live Load Reductions depend on the model Design Code, which dictates the fundamental method of calculating reductions; by a tributary area approach (USA & Australian Head Codes) or by the number of supported floors (other Head Codes).

Some key aspects/ differences specific to Head Codes are noted briefly below:

- In common with other attributes, there are two principal modes for the new I/LLR attribute; **Review** and **Set**:

- **Review Mode** - this allows review of the following I/LLR attributes via automated color coding and legend:

  - Applied live load reduction factor (applicable to all Head Codes)

  - Reduce Imposed/ Live load by (applicable to all Head Codes) - the reduction factor expressed as a % reduced by e.g. reduction factor of 0.6 = reduced by 40%.

  - Count the floor as supported (applicable to all Head Codes except USA and Australia)

  - Number of floors carried (applicable to all Head Codes except USA and Australia)

  - KLL value (applicable to USA Head Code only)

- Using the Review mode for example for the USA Head Code, you can easily review all the automatically calculated reduction factors (from tributary areas) as well as the assigned default KLL factors as illustrated in the pictures below. The tooltip of a selected entity also includes the principal I/LLR data.
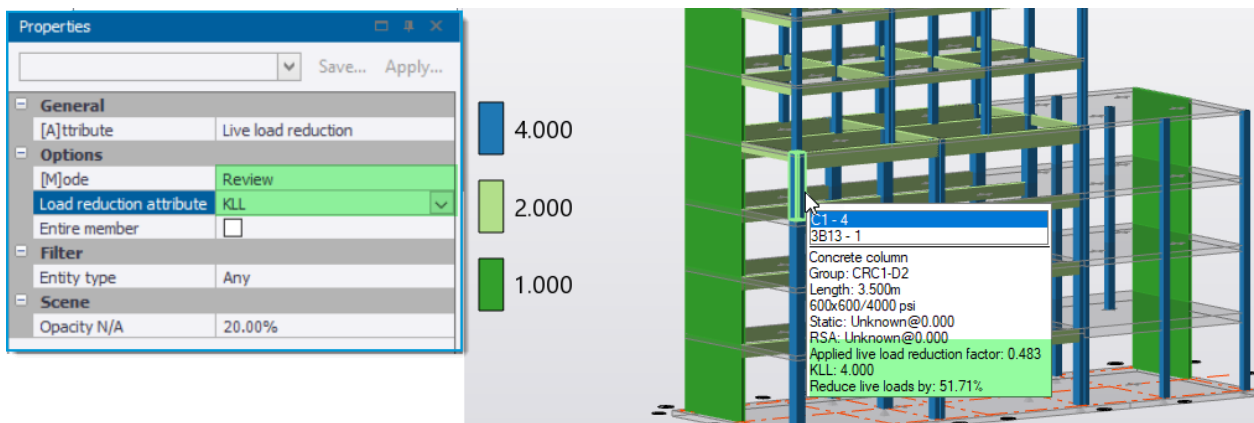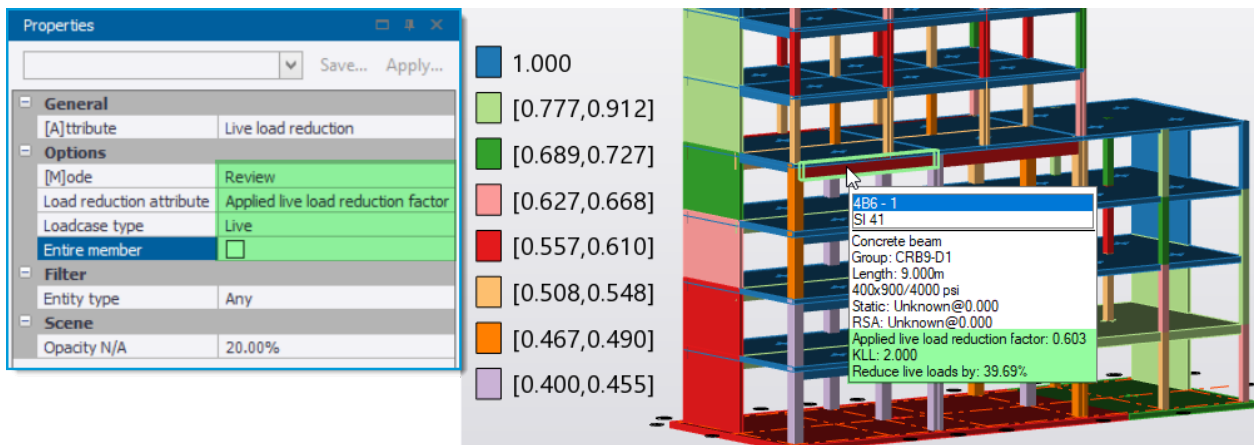
- **Set Mode** - allows you to graphically edit the I/LLR settings as follows:

  - **Reduce imposed loads by** - using this you can manually apply specified reduction % values using mouse operations as illustrated in the picture below. As this shows, you can use two adjacent 3D Structure views - one in Review mode and one in Set mode - to simultaneously view the applied reductions as a guide in one while manually making adjustments in the other.

    - With the "Entire member" option enabled, the value is applied to the entirety of continuous entities such as beams, columns and walls. When off it can be applied to individual spans/ stacks/ panels. You can also enable an entity type filter to apply the operation only to beams for example.
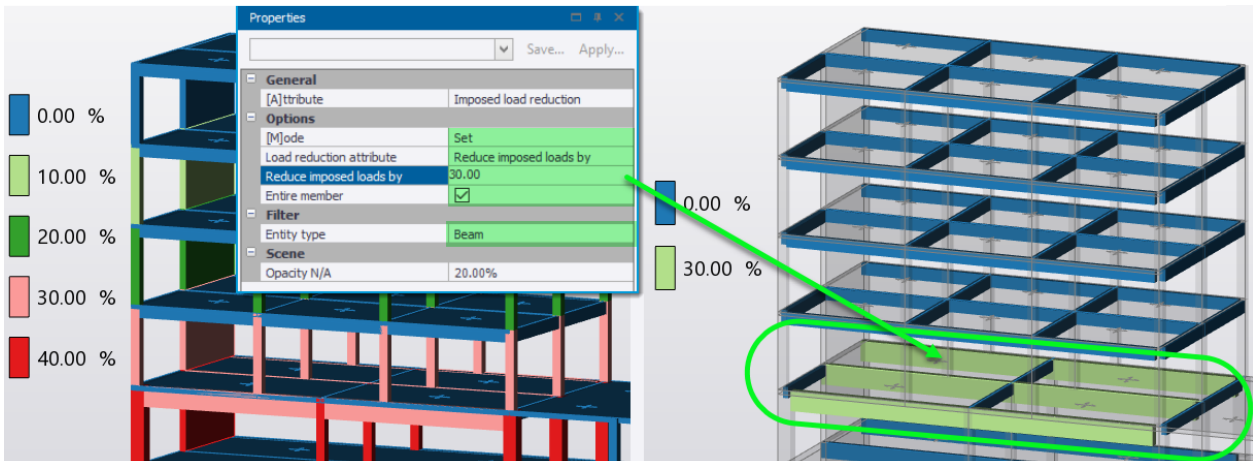
- **Assume extra floors supported** - using this you can manually apply a specified number of extra supported floors as illustrated in the picture below for transfer columns for example. The automatically calculated reductions are then updated accordingly. This control has the same additional "Entire member" and entity filter options.
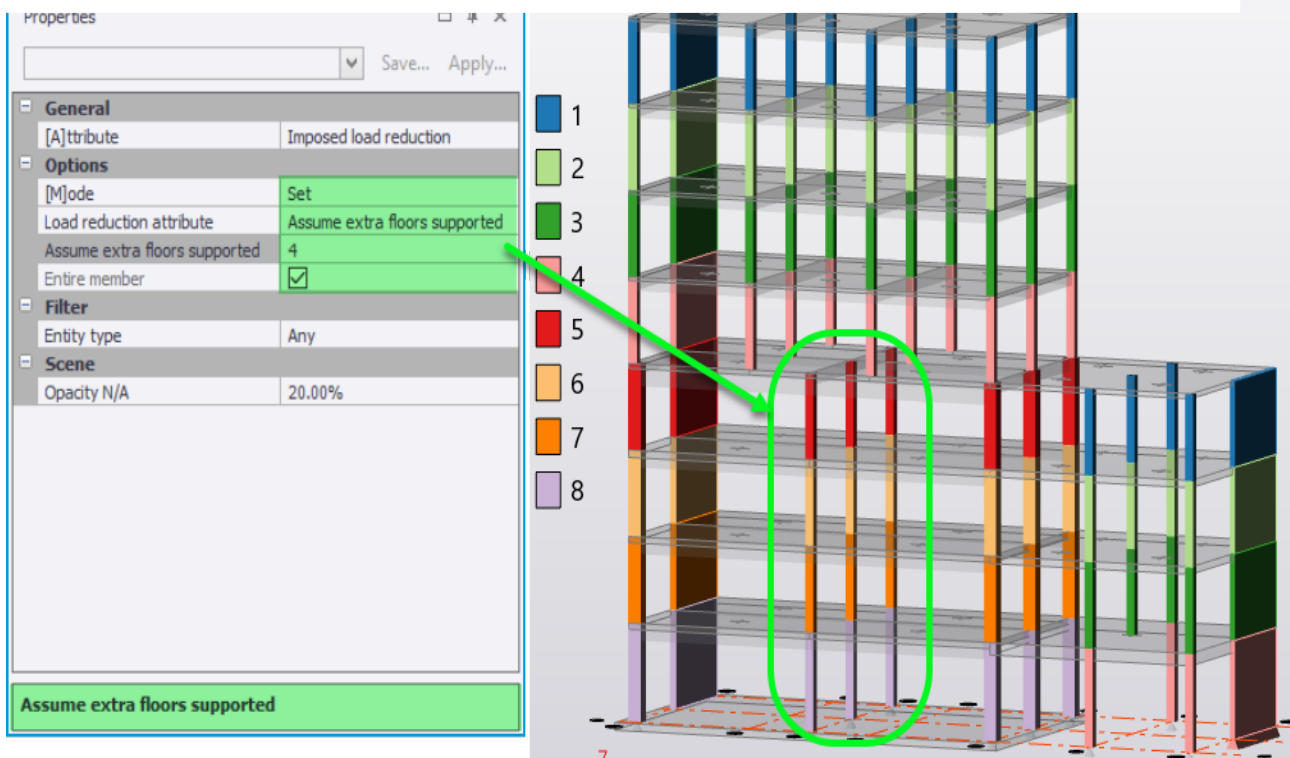
- **Count the floor as being supported** - using this you can manually edit the "Count the floor as being supported" Level property for columns and walls as illustrated in the pictures below. Note that this property defaults to on for all stacks/ panels for all levels set to be a floor that a column/ wall intersects and hence are listed in the column/ wall properties.

  - To change this setting, uncheck the "Count the floor as being supported" option and also the "Entire member" option to edit individual level settings, then click on a column stack/ wall panel to apply. Note the following:

    - Only stacks/ panels with their setting opposite that of the current "Count the floor as being supported" setting in the Attribute Properties can be selected for editing - the cursor tooltip displays the current settings at the upper/ lower ends of a selected stack/ panel for selectable entities.

- Since the setting applies to the **ends** of stacks/ panels, it works in a similar manner to editing member end releases using the Fixed/ Pinned Show/ Alter state command; you can click near the end of a stack/ panel to apply the setting only to the level for that end, or in the middle to apply it to the levels at both ends.
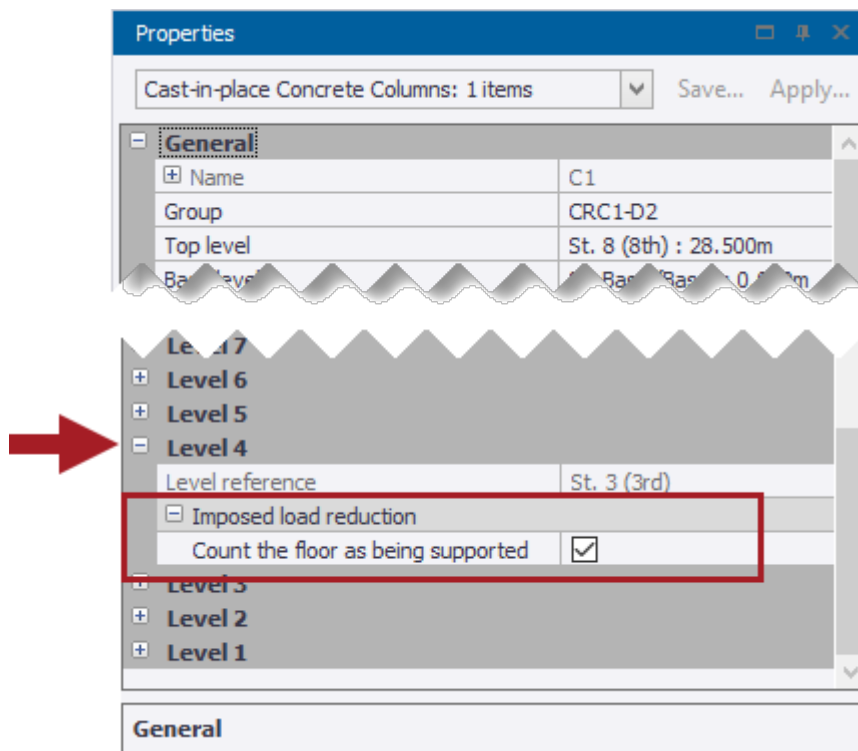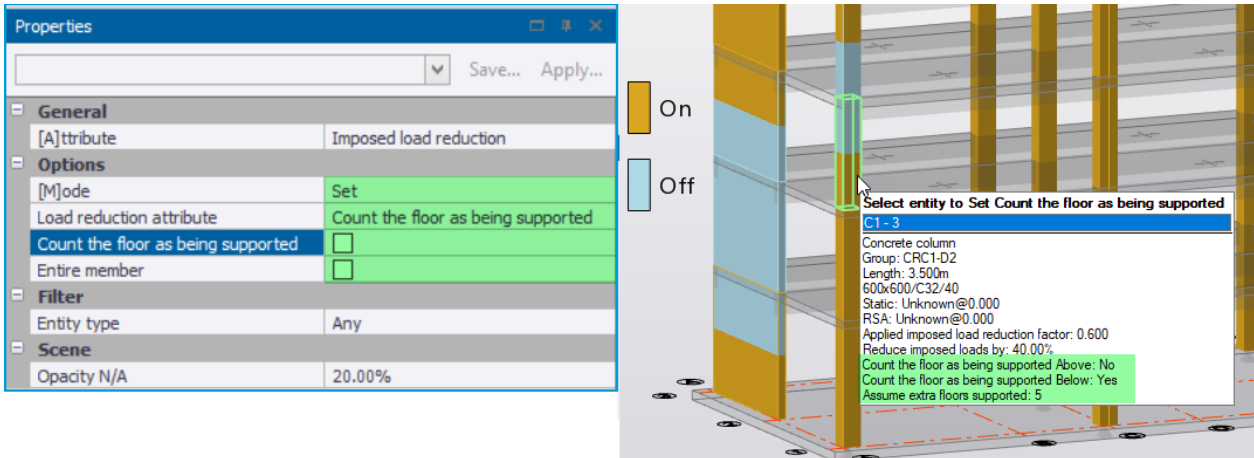
- **KLL** - USA Head Code only - using this you can manually edit the Live Load reduction factor KLL for entities.

Steel Beam Design - Enhanced Camber Settings & New Review View Attribute

Previous Tekla Structural Designer releases have featured the option to apply camber to steel beams - both composite and non-composite - including automatic camber calculation and application based on a number of criteria. For more about this see the Composite Beam deflections including camber TUA article. Following customer feedback, In this release the settings and processes for camber have been thoroughly reviewed and enhanced to give increased flexibility and make the workflow more intuitive.

In addition the Review View > Show Alter state is enhanced with a new dedicated attribute for Camber, enabling rapid and efficient graphical review and control of its settings and values. Camber values are also now included in the beam design summary tables and associated report items. These enhancements apply to all Head Codes that feature steel and steel composite beam design.

Property	Value
General	
[A]ttribute	Camber
Options	
[M]ode	Review
Entire member	☑
Filter	
Entity type	Any
Scene	
Opacity N/A	20.00%
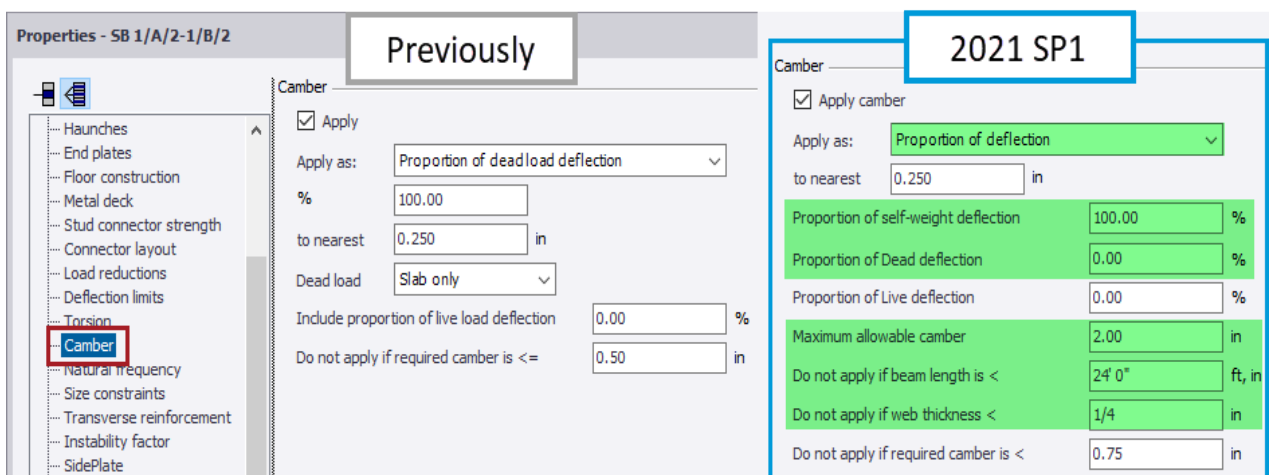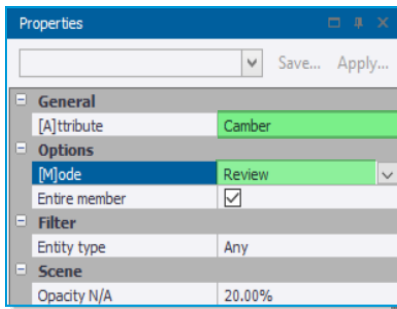
- 🟩 Absolute
- 🟧 Proportion of span
- 🟥 Proportion of deflection
- 🟦 Off

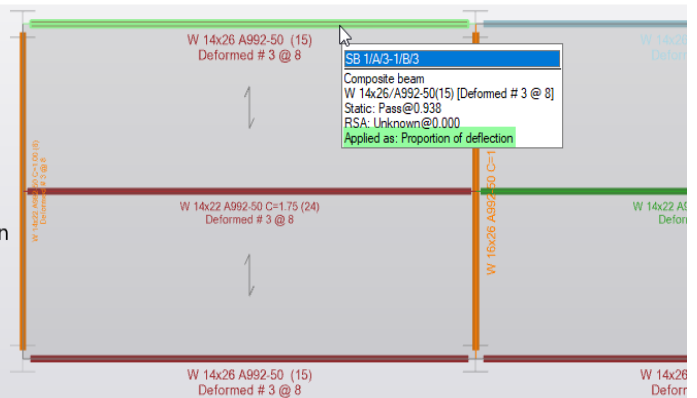**Review Data**

| Member Reference	Group Ref.	Span Ref.	Section	Grade	Length [ft, in]	No. Connectors	Camber [in]	Utilization	Status	Results
SB 1/C/1-1/C/3	SBRC3	1	W 14x22	A992-50	20' 0"	8	0.00	0.883	✓ Pass	Results...
SB 1/B/1-1/B/3	SBRC3	1	W 16x26	A992-50	20' 0"	28	0.75	0.984	⚠ Warning	Results...
SB 1/A/1-1/A/3	SBRC3	1	W 12x22	A992-50	20' 0"	8	0.50	0.973	✓ Pass	Results...
SB 1/A/1-1/B/1	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...
SB 1/B/1-1/C/1	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...
SB 1/A/2-1/B/2	SBRC2	1	W 16x26	A992-50	30' 0"	15	0.75	0.947	✓ Pass	Results...
SB 1/B/2-1/C/2	SBRC2	1	W 14x22	A992-50	30' 0"	24	1.50	0.997	⚠ Warning	Results...
SB 1/A/3-1/B/3	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...
SB 1/B/3-1/C/3	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...

Camber Settings Enhancements

As in previous releases, camber can either be off, or applied as one of three options; *Value* (an absolute dimension), *Proportion of span* (automatically calculated and applied as a proportion of the span length) or *Proportion of deflection* (automatically calculated and applied as a proportion of deflection resulting from the applied loads).

- The principal enhancements and changes to operation are:
  - **Apply camber: Proportion of deflection** - this option is renamed from "Proportion of dead load deflection" since it also has the option to include a proportion of Live/ Imposed loads. The following additional changes are made:
    - The previous "Dead load" setting - with options of "Slab only" and "All dead" - is removed.
    - In its place, there are now separate settings for the proportion of self-weight deflection (which includes the self weight of the slab + the steel beam section) and Dead deflection (applies to other dead loadcases, which are assumed to be applied post-composite action)*.
      - *Note that, similar to previous behavior, both these options are automatically disabled and set = 100% when the proportion of Imposed deflection is set to be greater than > 0% (this being the default value). The rationale for this behavior is that all the dead loads will have already been applied by the time the imposed loads are.
    - These changes improve the clarity and simplicity of defining what factors affect the camber calculation and how. They also improve flexibility in enabling different percentages to be set for selfweight and applied dead load
  - **Maximum allowable camber -** a new criteria that applies to all of the camber application options; Value, Proportion of span, Proportion of deflection. Key aspects of this option are:
    - It allows the user to pre-set the maximum amount of camber that is allowed for the member.
    - For a camber value above this limit there will be an explanatory warning in the beam check results.
    - The auto-design process will look for sections in the design list that fulfill this limit.
  - **Do not apply if beam length is <** - a new criteria that applies to all of the camber application options; Value, Proportion of span, Proportion of deflection. Key aspects of this option are:
    - Allows the user to define the minimum beam length for camber to be applied.
    - Camber is not applied to beams of length equal or below this limit.
  - **Do not apply if web thickness** < - a new criteria that applies to all types of the camber application options; Value, Proportion of span, Proportion of deflection. Key aspects of this option are:
    - Allows the user to define the minimum section web thickness that camber should be applied for.

- For camber values at or below this limit there will be an explanatory warning in the beam check results.

- The auto-design process will only consider sections in the design order list that comply with this limit.

- **Do not apply if required camber is <** - the operation of this option is essentially unchanged, however the ***condition*** for application of the limit is amended to less than (<) the limiting value rather than less than or equal to (<=) as previously . Thus for example, with the limit set = 20mm, a calculated camber of 20mm would be applied where previously it was not.

  - For existing models the original limit value is therefore automatically increased accordingly to match the previous behavior as closely as possible*. The value of the increase is the same as the "to nearest" value for Head Codes that default to metric units and half this value for those that default to US-customary units.

    - *Despite this adjustment, it is still possible that due to this change of condition, camber may not be applied in this release in some cases where it was previously. Thus we recommend that the camber settings and results are carefully reviewed in this release - this process should now be much simplified by the addition of the new review view attribute for camber.

Review View > Show/Alter State - Camber

A new dedicated Review View "Camber" attribute is added to those for Show/ Alter state, enabling quick and easy review and editing of all the camber settings. As shown in the picture below, this is selected at the top of the Properties Window when Show/ Alter state is activated on the Review Ribbon and has the following Modes; Review, Set On, Set Off, Toggle and Copy.
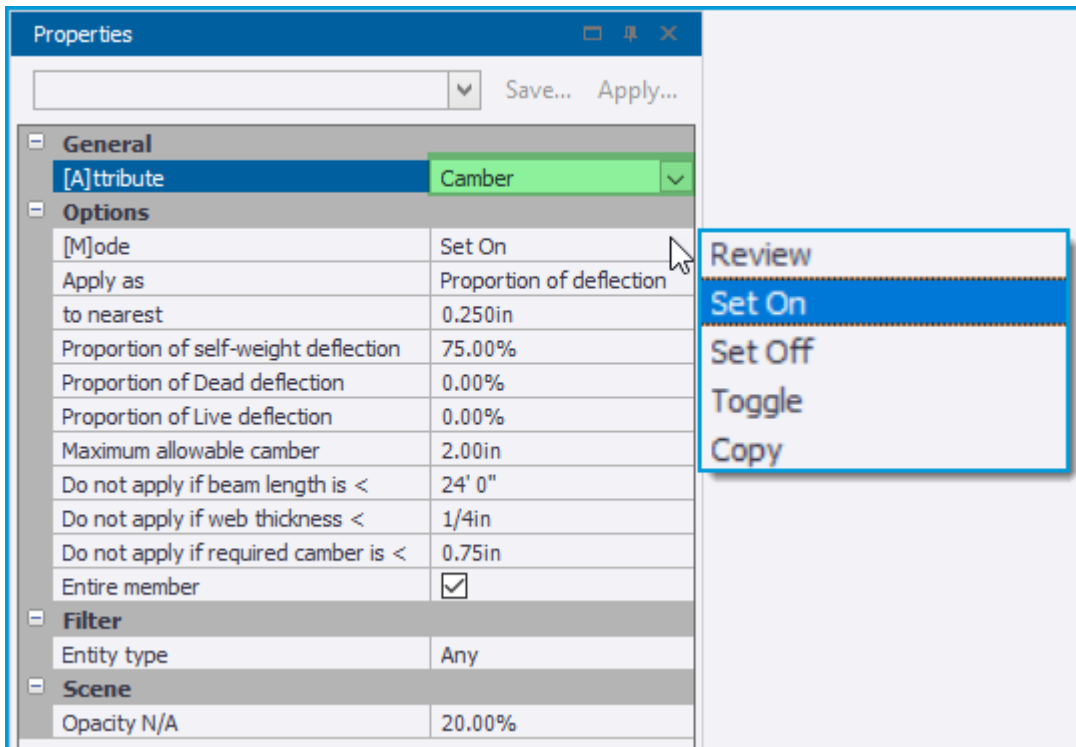
- **Review Mode** - this displays a color coded view and associated legend showing the distribution of the different camber settings; Value, Proportion of span, Proportion of deflection and Off.

- **Set On Mode** - with this mode selected you then select the "Apply as"option from the following; Value, Proportion of span, Proportion of deflection.

  - You can then input your desired settings/ values in the properties window and apply these graphically to beams in the model using mouse operations . The color coding and legend indicate which beams have; the current settings applied, Camber off or Other settings. These will update automatically following changes to the current properties. For beams with "Other details", the cursor tooltip displays all their current settings when selected by moving the cursor over them, as shown in the picture below

- **Set Off Mode** - turns camber fully off for selected beam(s):
  - Members in view are color coded by camber status (On / Off).
  - Member tooltip includes the full set of camber properties currently applied.
  - The tooltip of members with camber applied includes the camber calculation method.

- **Toggle Mode** - sets camber on or off for selected beam(s) depending on their current state:
  - Members in view are color coded by camber status (On / Off).
  - If the member being turned on has had camber properties previously defined, these will be persisted, otherwise the defaults are used.
  - The tooltip of members with camber applied includes the camber calculation method.
- **Copy Mode** - using this you can copy all the camber settings from one beam to one or multiple other beams.
  - To use this you first click the **Source** beam as indicated in the picture below - other beams in the view are then automatically color coded to indicate which are the same as the source and which are valid target beams to which the source beam properties can be copied. You can then use mouse operations to select the target beam(s).
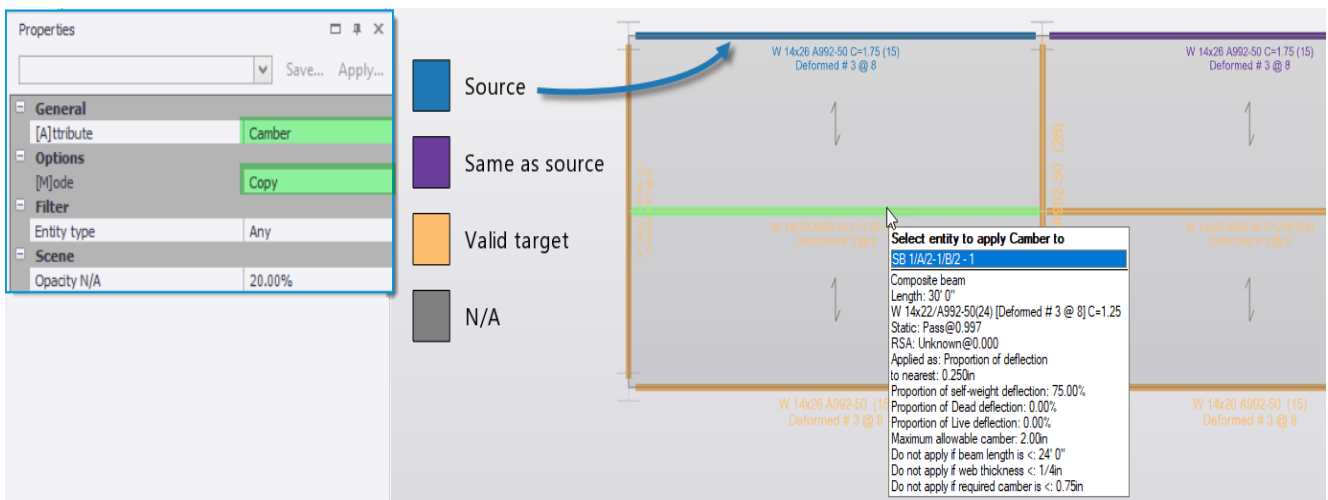
Review View > Tabular Data, Reports and Drawings

A new column giving the applied camber value for each beam is also added to the Tabular data > Design summary table for steel/ composite beams and the associated report items.

Although not new for this release, note that the camber value is also automatically included in drawings as illustrated in the picture below showing a General Arrangement Drawing - the camber value is included in the Steel Beam Attributes text in this manner for example "**C=1.00**".

Member Reference	Group Ref.	Span Ref.	Section	Grade	Length [ft, in]	No. Connectors	Camber [in]	Utilization	Status	Results
SB 1/C/1-1/C/3	SBRC3	1	W 14x22	A992-50	20' 0"	8	0.00	0.883	✓ Pass	Results...
SB 1/B/1-1/B/3	SBRC3	1	W 16x26	A992-50	20' 0"	28	0.75	0.984	⚠ Warning	Results...
SB 1/A/1-1/A/3	SBRC3	1	W 12x22	A992-50	20' 0"	8	0.50	0.973	✓ Pass	Results...
SB 1/A/1-1/B/1	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...
SB 1/B/1-1/C/1	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...
SB 1/A/2-1/B/2	SBRC2	1	W 16x26	A992-50	30' 0"	15	0.75	0.947	✓ Pass	Results...
SB 1/B/2-1/C/2	SBRC2	1	W 14x22	A992-50	30' 0"	24	1.50	0.997	⚠ Warning	Results...
SB 1/A/3-1/B/3	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...
SB 1/B/3-1/C/3	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass	Results...

Tekla Structural Designer

| Project		Job Ref.	
Structure		Sheet no.	
			Page
Calc. by	Date	Chk'd by	Date	App'd by
jekni	20/04/2021		18/10/2019	

Steel

Beam Design

Beam Design Summary

Static

Member Reference	Group Ref.	Span	Section	Grade	Length [ft, in]	No. Connectors	Camber	Utilization	Status
SB 1/C/1-1/C/3	SBRC3	1	W 14x22	A992-50	20' 0"	8	0.00	0.883	✓ Pass
SB 1/B/1-1/B/3	SBRC3	1	W 16x26	A992-50	20' 0"	28	0.75	0.984	⚠ Warning
SB 1/A/1-1/A/3	SBRC3	1	W 12x22	A992-50	20' 0"	8	0.50	0.973	✓ Pass
SB 1/A/1-1/B/1	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass
SB 1/B/1-1/C/1	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass
SB 1/A/2-1/B/2	SBRC2	1	W 16x26	A992-50	30' 0"	15	0.75	0.947	✓ Pass
SB 1/B/2-1/C/2	SBRC2	1	W 14x22	A992-50	30' 0"	24	1.50	0.997	⚠ Warning
SB 1/A/3-1/B/3	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass
SB 1/B/3-1/C/3	SBRC2	1	W 12x22	A992-50	30' 0"	15	1.00	0.759	✓ Pass
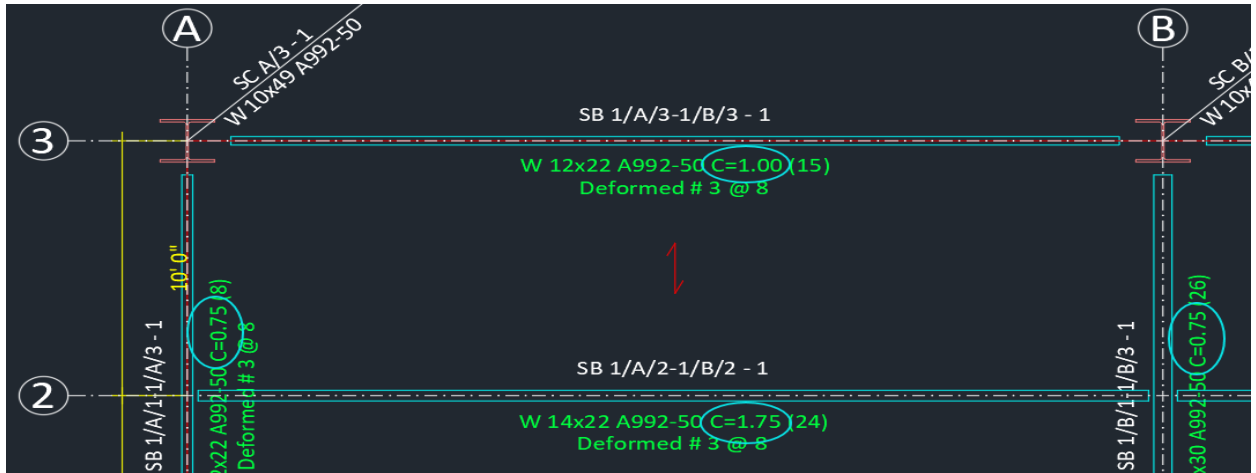
Steel Column Base Plate Design - Enhanced Scope for USA Head Code

The scope of Steel Column Base Plate Design checks is significantly enhanced in this release for the USA Head Code, making it applicable to a wider range of loading conditions. Major axis shear loading is now checked for the shear resistance of the rods and/ or concrete and the welds are now also checked as shown in the picture below. In addition, all the applied forces are reviewed and beyond scope and/ or Ignored forces warnings issued as appropriate for cases that are not considered by the design. For full details of the checks performed, please see the updated Help Topic .
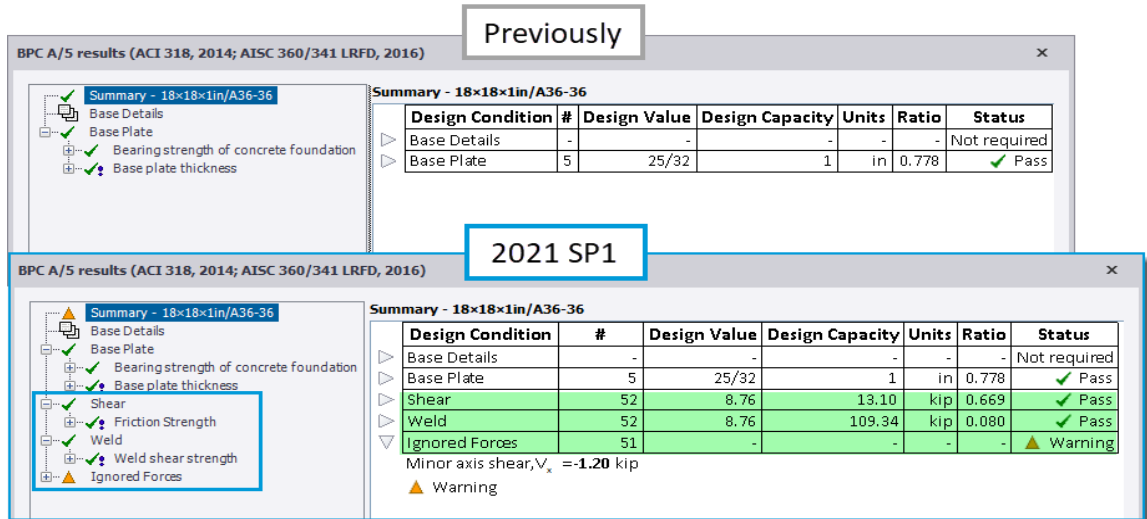
- The enhanced and new checks are as follows for applied Major axis shear load - both with/ without Positive vertical load.

  - Base Plate

    - Bearing strength of concrete foundation.

    - Base plate thickness.

  - Shear

    - Friction Strength

    - Shear strength of rods

    - Concrete anchorage strength for shear forces on rods

    - Concrete pry-out strength of anchor rod in shear

  - Weld

    - Weld shear strength

- **Shear Check** - the shear capacity calculations performed are dependent on the *Horizontal Shear Transfer* option selected in the settings page/ group for

"Rods" as shown in the picture below left. This has options of; *Friction Alone* (default), *Shear on Rods Alone* and *Friction and Shear on Rods*. The picture below right shows an example of the check details for the shear strength of rods with the *Friction and Shear on Rods* option selected. Below we give some brief details of checks performed and their references - for fuller details please see the updated Help Topic .
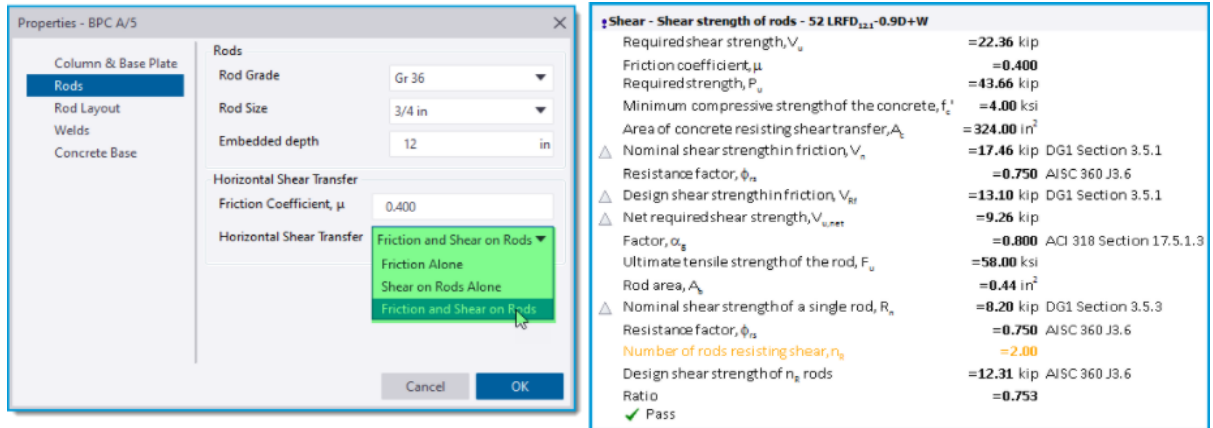
- **Friction Alone** - for this the shear strength check is calculated in accordance with ACI 318 and considers the full (major) shear force. Note the following:

  - Since ACI 318 only considers LRFD requirements, this check is not performed for ASD load combinations.

  - The default for μ, the coefficient of friction between the base plate and concrete, is taken as 0.4 per ACI 349-01 (section RB.6.1.4), 349-06 and -13 (section RD.6.1.4).

- **Shear on Rods Alone** - checks for this option are divided into the following three separate parts and consider the full (major) shear force: 1) Shear strength of rods 2) Concrete anchorage strength for shear forces on rods 3) Concrete pry-out strength of anchor rod in shear. Note the following:

  - The anchor rod strength is calculated in accordance with AISC 360 and AISC Design Guide 1 for both LRFD and ASD load combinations.

  - For the concrete strength checks - these are calculated in accordance with ACI 318 and since this only considers LRFD

requirements these checks are not performed for ASD load combinations.

- **Friction and Shear on Rods** - when this option is selected, the three Shear on Rods checks consider a net shear force i.e. the remaining (major axis) shear force not taken by frictional resistance alone.

  - Checks for Friction and Shear on Rods are divided into the same three separate parts as for Shear on Rods Alone described above, with the addition of a Friction calculation at the start of each of the three Shear on Rods parts that derives the net (major) shear force

- **Weld Check** - the welds are checked for one design condition: shear.

  - Weld Shear Strength

    - For base plates with thickness <= 3/4 in (19 mm), the default weld leg length is 1/4 in (6mm) and for all other base plate thicknesses the default weld leg length is 5/16 in (8 mm). These defaults can be adjusted in the database [Home > Materials > Welds > Defaults].

    - AISC 360-10 and -16 stipulates that, when the length of the weld exceeds 300 times the leg size, w, the effective length shall be taken as 180w (section J.2b.(d).(3)), and this is similarly applied when AISC 360-05 has been selected for steel design.

Weld - Weld shear strength - 44 LRFD$_{10.5}$-1.2D+L+0.5S+W

Required shear strength, V_u	= 26.46 kip
Actual length of weld, L_f	= 7 55/64 in
Weld leg length, w	= 5/16 in
Factor, β	= 1.000 AISC 360 J2.2b
⚠ Effective weld length, $L_{w,eff}$	= 7 55/64 in
⚠ Total effective weld length, L_w	= 15 45/64 in
Effective area of the weld, A_w	= 3.47 in^2
Web electrode classification number, F_{EXX}	= 70.00 ksi
Angle of loading, θ	= 0.0000 °
Weld strength, F_w	= 42.00 ksi AISC 360 Eqn J2.5
⚠ Nominal strength of the fillet weld between the web and the base plate in shear, R_{nws}	= 145.78 kip AISC 360 J2.4
Resistance factor for weld strength, ϕ_w	= 0.750 AISC 360 J2.4
Thickness, t_w	= 11/32 in
Yield Strength = MIN[$F_{y.column}$, $F_{y.plate}$]	= 36.00 ksi
⚠ Nominal strength of the parent material in shear yielding, R_{pws}	= 115.36 kip AISC 360 J4.2
Resistance factor, ϕ_{ps}	= 1.000 AISC 360 J4.2
Design capacity = MIN[$\phi_w \times R_{nws}$, $\phi_{ps} \times R_{pws}$]	= 109.34 kip AISC 360 J2.4
Ratio	= 0.242
✓ Pass	

- **Beyond Scope/ Ignored forces** - all the forces applied to a base plate connection are now fully reviewed and compared to those considered by the design. Where they are beyond scope and/ or ignored, an appropriate

status is issued in the check details and reflected in the Review View > Design Status and connection entity tooltip. The picture below is an example of the check details showing this.

- Beyond Scope - this status is issued for applied negative vertical load (uplift) or major axis moment. No design checks are performed for combinations where either or both of these are present.

- Ignored forces - this applies to minor axis moment and shear; where their value is greater than a given limit*, they are displayed in the check details together with a Warning status. Where all other forces are within scope, all the design checks are carried out and the engineer can make a judgement on their validity.

  - *The given limits are defined on the Design Forces page of the Design Settings dialog available from the Design tab on the ribbon.

New Overall Wind Drift Check - All Head Codes

In line with customer requests, a new Overall Wind Drift check option is added in this release. This checks the maximum deflection at the top of the structure relative to its total height against a specified limit. When activated, the check is fully automated and is carried out/ updated when any of the following processes are run; Analyze > Analyze All (Static) or 3D Only (Static); any of the Design Steel/ Concrete/ All (Static) options of the Analysis & Design groups of the Design Ribbon.

- The check is enabled by a new "Check for Overall Wind Drift" option in the "Sway & Drift Checks" Design Settings, as shown in the picture below. When

this is enabled, the desired drift limit value can be entered (default 500). By default this new setting is enabled for new models but not for existing models.
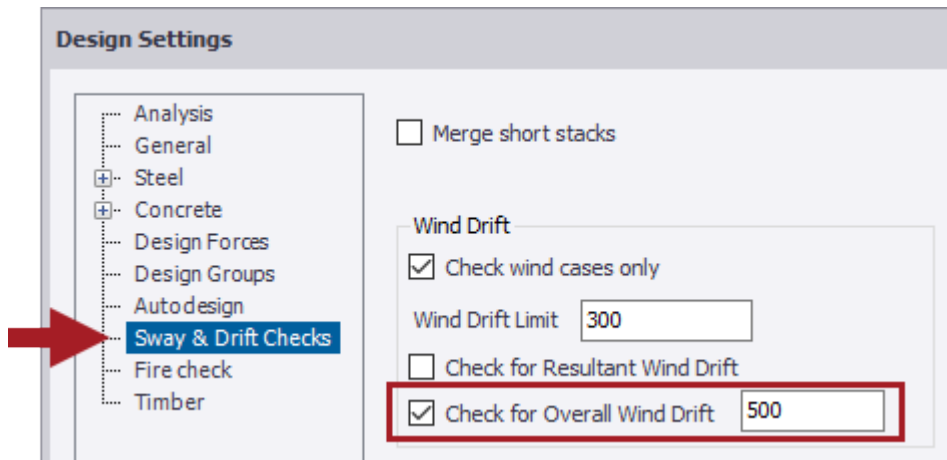
- Key aspects of the check undertaken are as follows:
  - When the "Check for Resultant Wind Drift" option is enabled, a single maximum resultant deflection value is checked, otherwise the deflection values in Dir1 and Dir2 are considered separately and the maximum of these is checked.
  - The height of the structure considered by the check is the distance between highest and lowest levels with the "Check for drift" level setting enabled.
- **Check Results** - The results of the check are reported in the "Design" group of the Status tree under the heading "Overall Wind Drift" as shown in the picture below. This reports; the governing deflection value, the combination it occurs for and the node it occurs at.
  - You can double click the message to locate and highlight the node in the Solver View.
  - You can also right click over the message and select the "Details" option from the resulting context menu to see full details of the check, as shown below.

Solver Node 13 - 3 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$

Load combination	3 STR$_{8.1}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF$_{Dir1+}$
Maximum deflection, Δ_{max}	124.0 mm
Height between highest level and lowest level active drift check, H	50.200 m
Overall wind drift ratio, H / Δ_{max}	404.925
Overall wind drift limit	500
Status	✖ Fail