

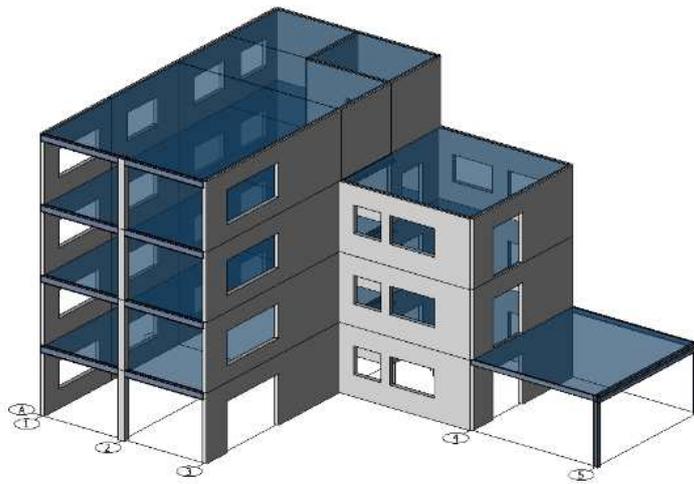


FEM-Design modelling guide for PREFAB

Made for version: PREFAB 25.0.1

Edited by:

Casper E. Hougaard (2025.10.27)



Disclaimer:

This document may contain errors or inaccuracies. StruSoft assumes no responsibility for the use of the information herein. Users must apply their own professional engineering judgment and verify that the modelling approach is appropriate for their specific project.



FEM-Design models for PREFAB

Table of contents

- Table of contents..... 1**
- Check-list..... 2**
 - Essential..... 2*
 - Recommended..... 2*
 - Specific for New optimized plastic redistribution 2*
- Essentials..... 3**
 - Edge-connections..... 3*
 - End-point behavior 4*
 - Point-connections 5*
 - Minimum edge-division number..... 6*
 - Storey..... 7*
 - Axis 8*
 - Assign loads to structure 9*
- Recommendations 10**
 - Supporting structures for covers..... 10*
 - Anchorage behavior 11*
 - Edge-connections representing the desired behavior..... 12*
 - “Special conversion method” for covers 13*
- Specific for New optimized plastic redistribution 14**
 - Supported structural elements 14*
 - Analysis setting..... 15*
- Design checks 16**
 - Wall design checks..... 17*
 - Plastic limits..... 17*
 - Concrete wall specific checks 18*
 - Truss design checks..... 20*
 - Plastic limits..... 20*
 - Steel truss specific design checks..... 21*



Check-list

Essential

When modelling for PREFAB, there are a few things necessary to ensure good results. Here's a basic check-list of the must-haves.

- Apply Edge-connections on all edges interacting with other structural elements (i.e., shells & supports).
- Apply End-point behavior set to free on all edge-connections with multiple edges meeting in a corner.
- Use Point-connections in all places where bar elements interact with shell elements.
- Set minimum edge-division number to 3.
- Use storey in the model.
- Use axis in the model
- Use assign to structure for all loads (Surface loads on covers don't need to be assigned).

Recommended

Here are some non-essential but highly recommended points to run through.

- Set the correct supporting structures for covers.
- Ensure anchorage has the desired behavior (i.e., only tension).
- Does the applied edge-connections represent the desired behavior?
- Check if "Special conversion method" is necessary for your load distribution.

Specific for New optimized plastic redistribution

If you intend to use the New optimized plastic redistribution, here are recommended checks.

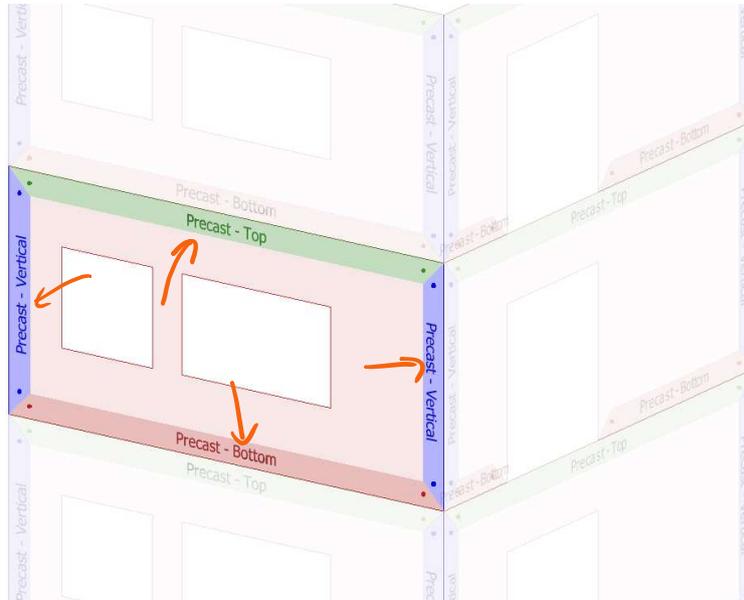
- Model contains only shell elements, trusses and covers.
- Don't waste time on NLE/PL analysis, PREFAB needs only linear-elastic load combination results



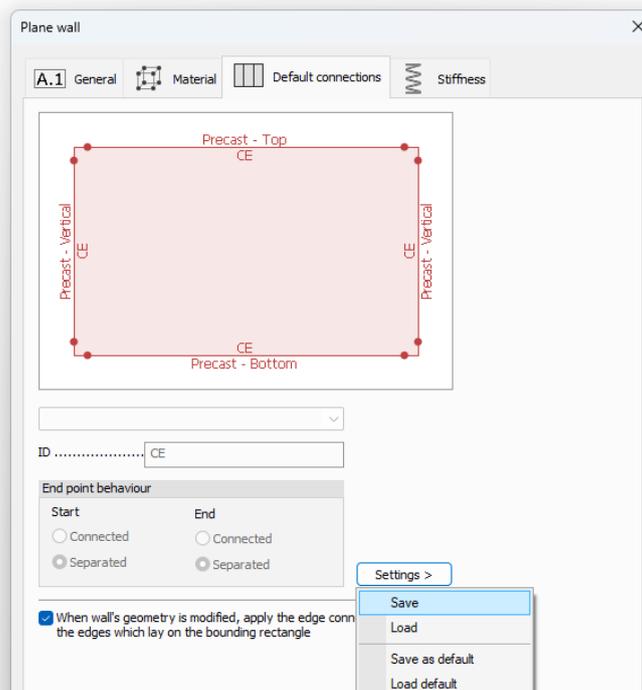
Essentials

Edge-connections

For PREFAB, it's important to apply edge-connections on all edges interacting with other elements. If they aren't applied, PREFAB can't read the forces and will most likely not be able to find equilibrium.

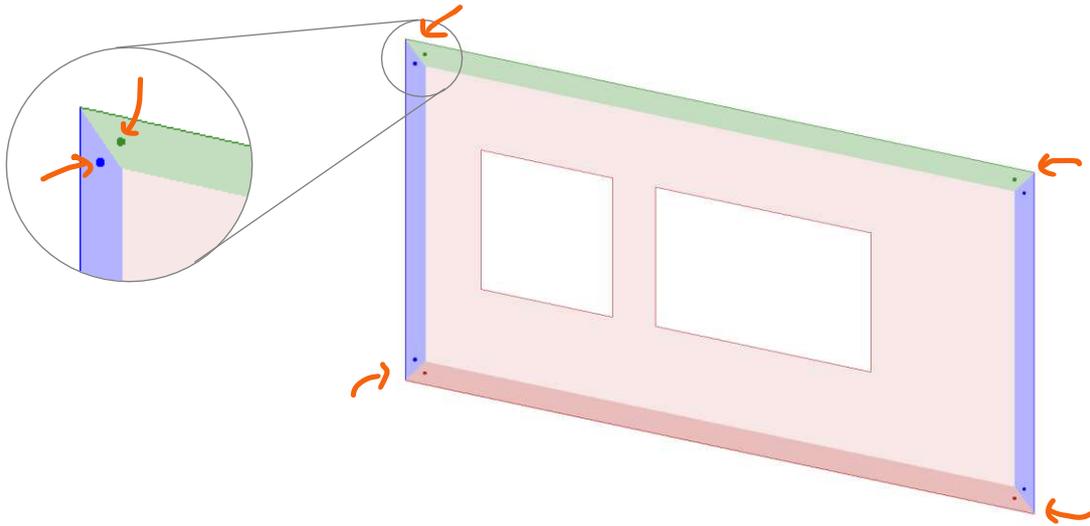


You can apply the edge-connection in the **Default connections** tab. Define your desired connections and save the setup as a preset.

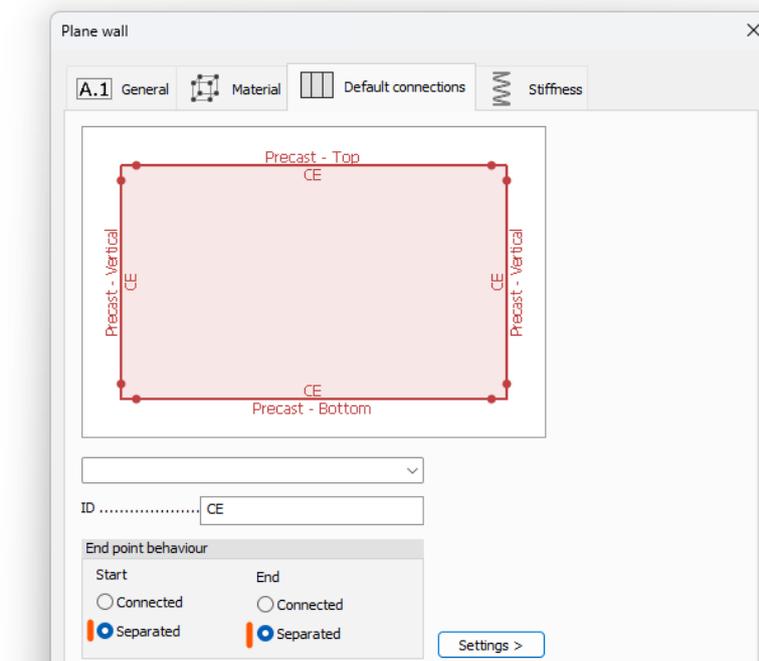


End-point behavior

This is a finite element technical necessity. To ensure correct results for edge-connection forces always apply **separated end-point behavior**. This is visually represented by small dots and the edge of the edge-connection. If they aren't separated, PREFAB will in many cases warn about equilibrium errors.

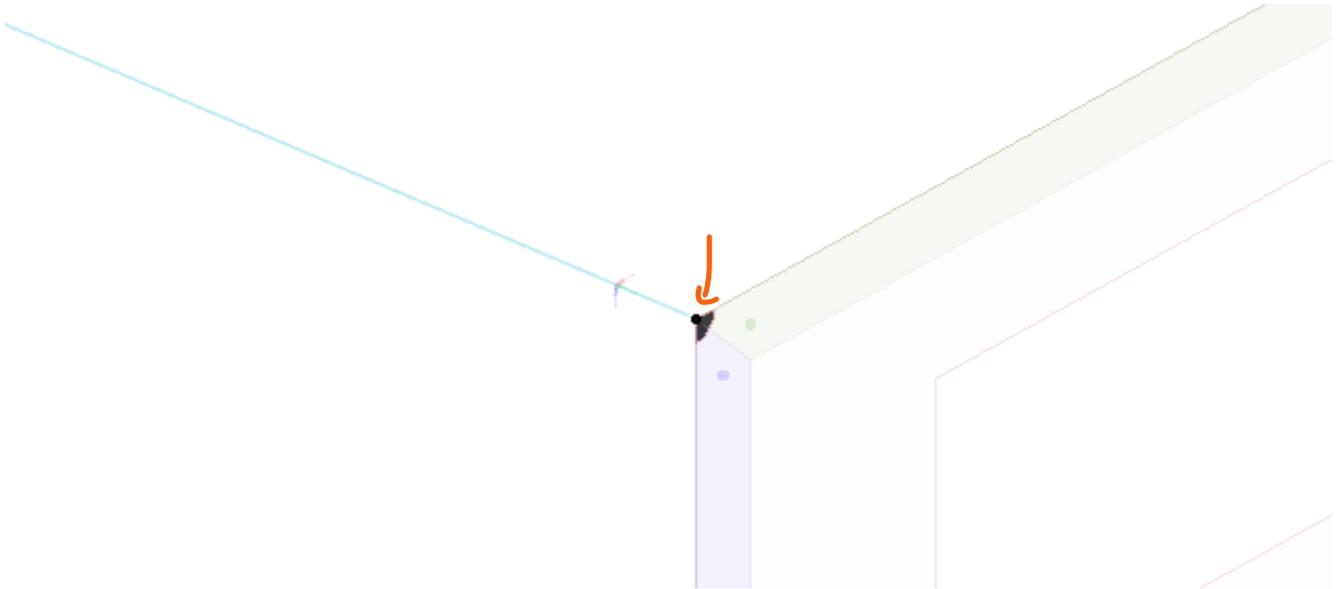


They can be applied in the Default connections tab and saved to the setup with the correct edge-connection types.

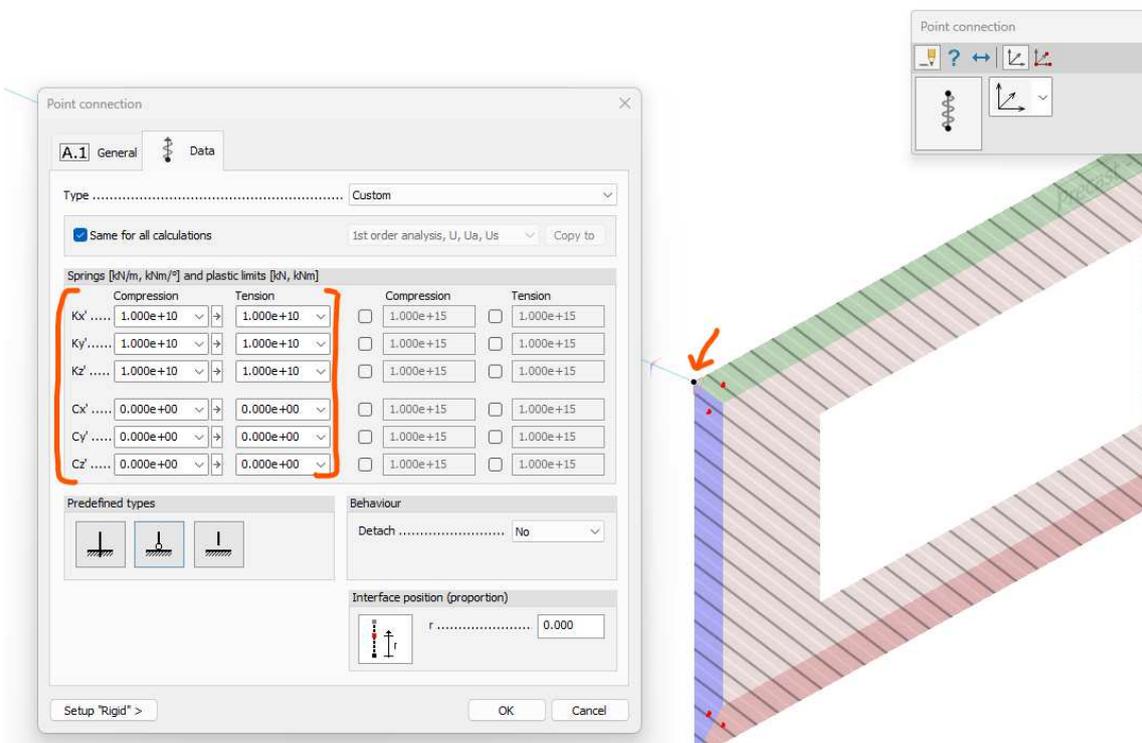


Point-connections

For PREFAB to get the correct information about the interaction between bar elements (both columns and bars) it's important to apply **point-connections**.



Apply them by setting the desired possibilities of interaction. Define the wall as the Master object, and place the second point on top of the master. (If you can't see the tool, set User-interface to **Full**).

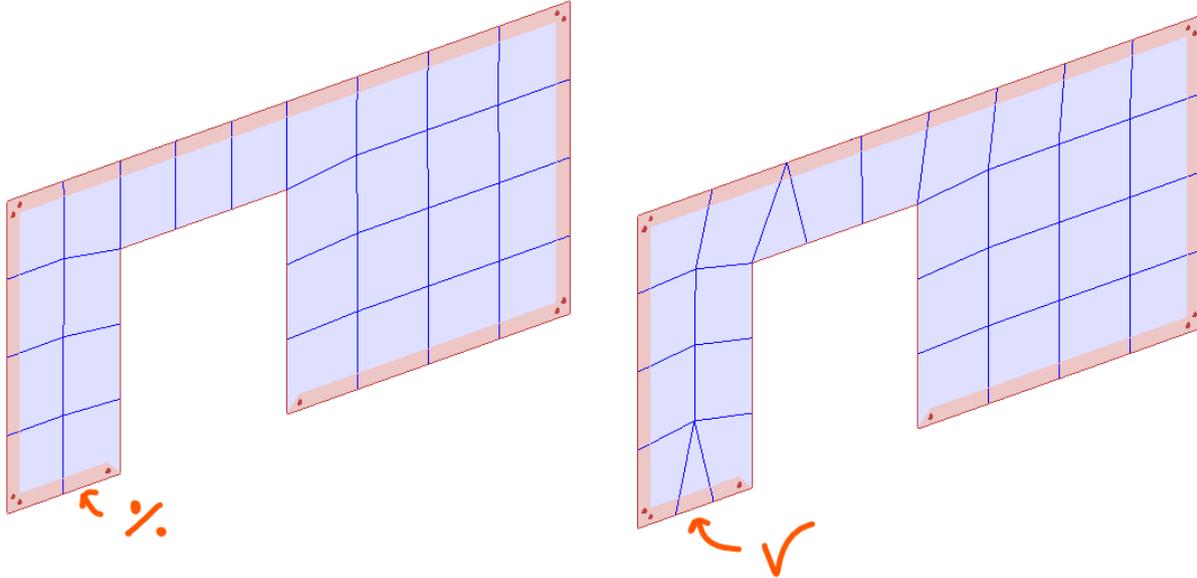


Use shortcut: *ctrl + shift + P* in PREFAB to unlock a secret

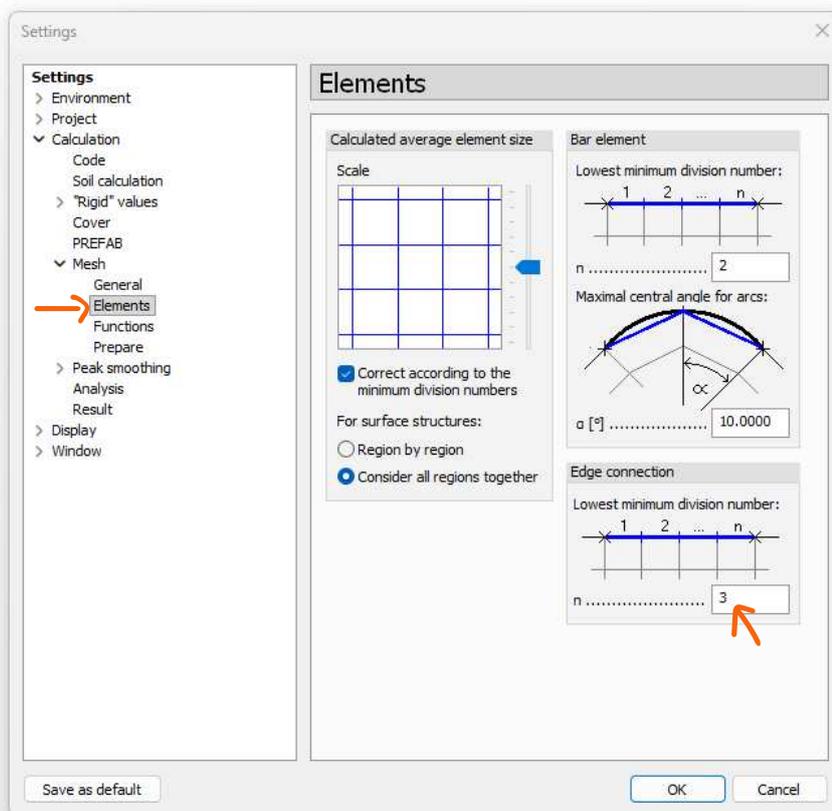


Minimum edge-division number

To help ensure good results from FEM-Design and thereby improving the quality of the imported data to PREFAB, apply a minimum of 3 divisions of the finite element at all edges.

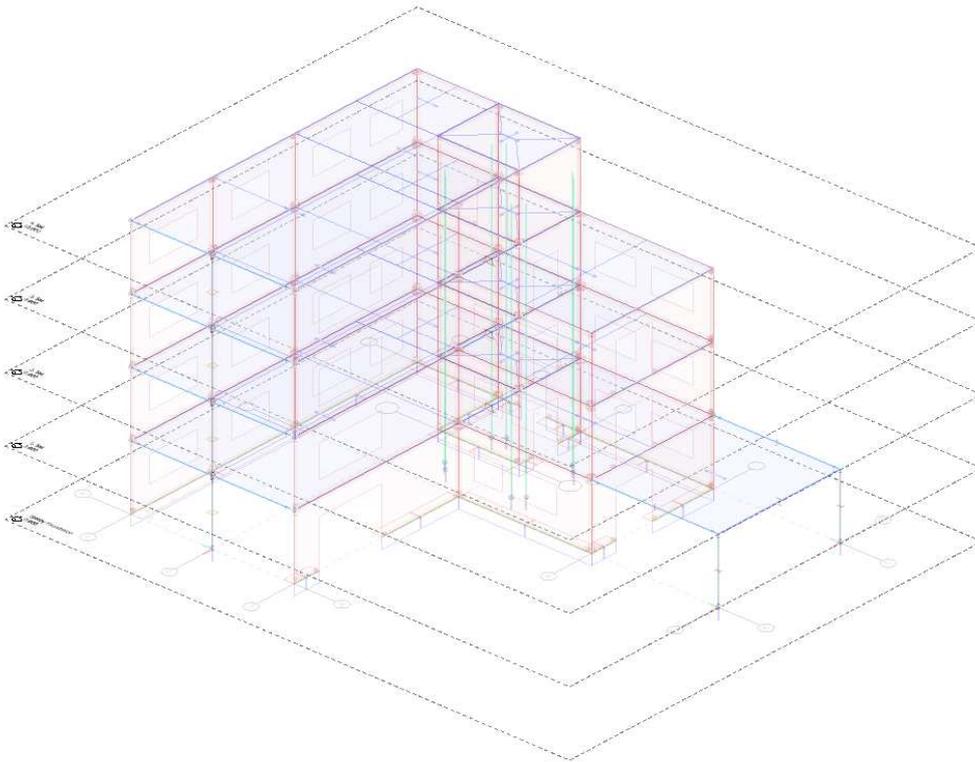


You can manually adjust the division number of edges in the Finite elements tab, or go to the settings menu, and change the minimum to be a default of 3 (This is the recommended method).

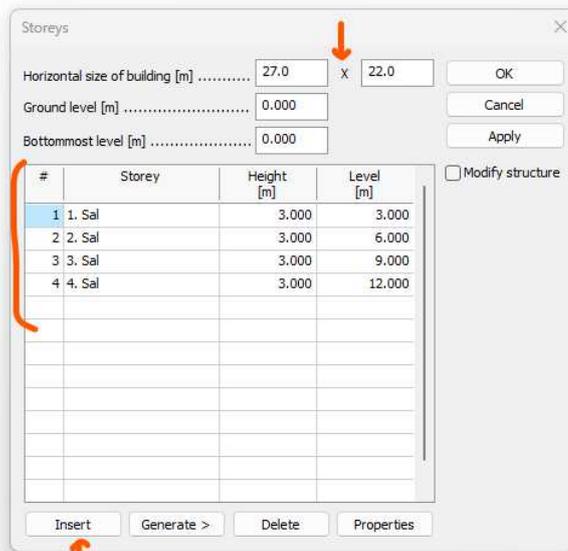
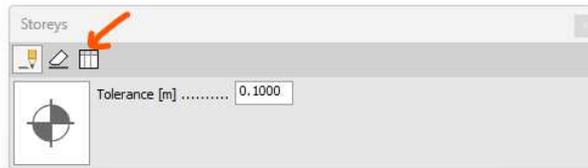


Storey

To help with the overview, 3D navigation and exporting options, **storeys** need to be applied to the model.

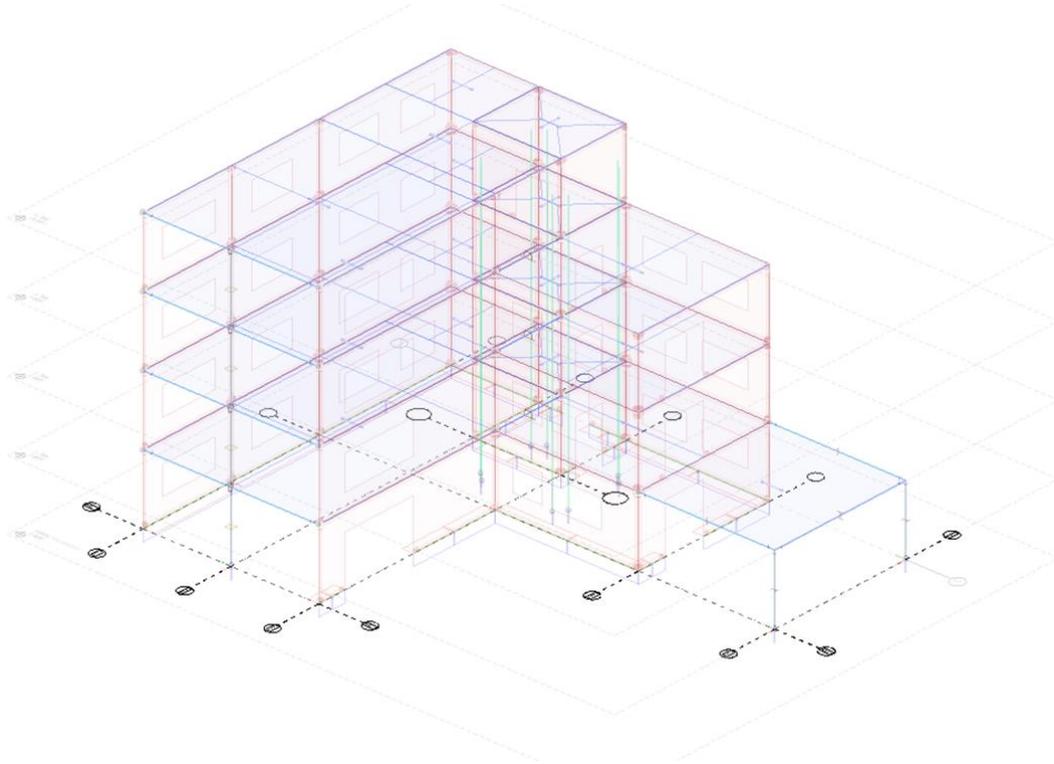


Apply these using the **Storeys** tool, and *edit in table* option.

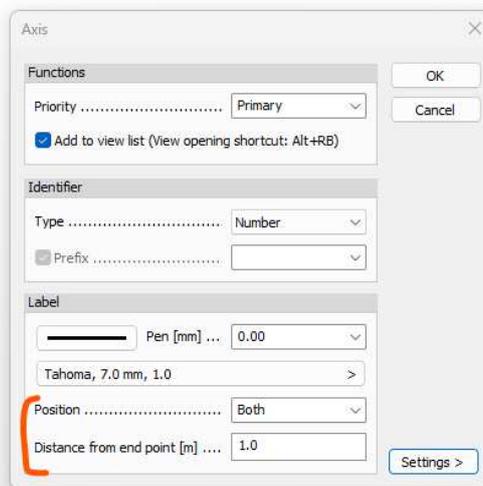
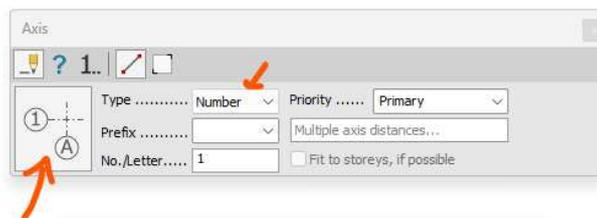


Axis

To help with the overview, 3D navigation and exporting options, **axes** need to be applied to the model.

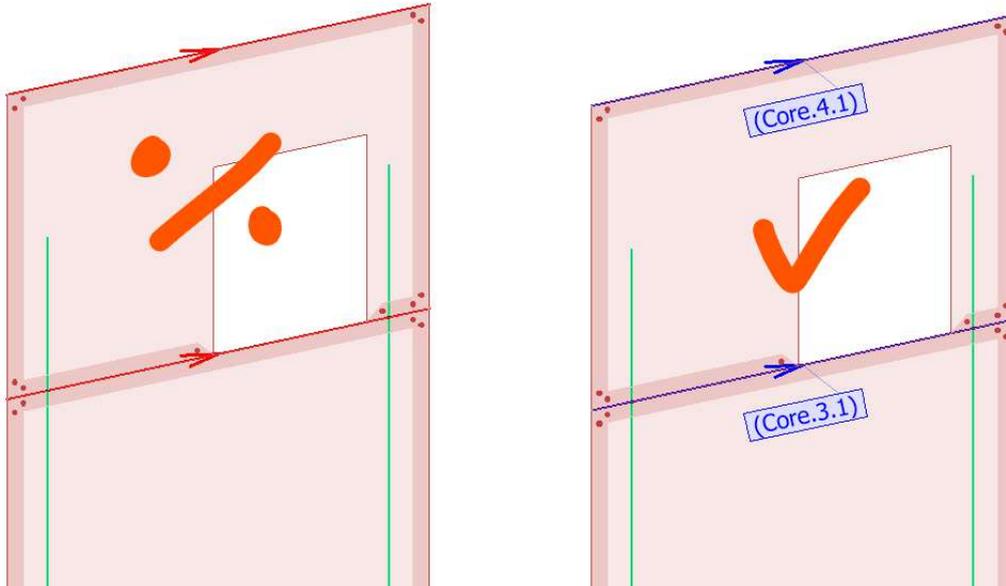


Apply these by using the **Axis** tool and draw them in the appropriate places. For any walls not placed on an axis plane, a custom axis will be created by PREFAB named after the closest existing axis.

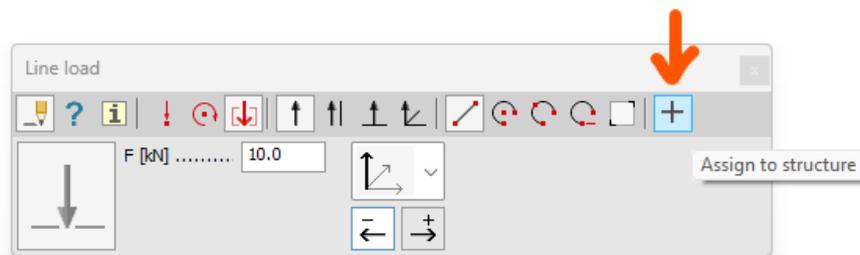


Assign loads to structure

To avoid ambiguous loads being applied to an undesired element, use the assign to structure option for your loads. It's important for all loads placed at the intersection between structural elements. But in general, it is recommended to always use. In some cases, it is not necessary, i.e. surface loads on covers.



To assign loads to the structure, you can activate the function when defining them (Please remember you need to then first select the structural element before you define the load).



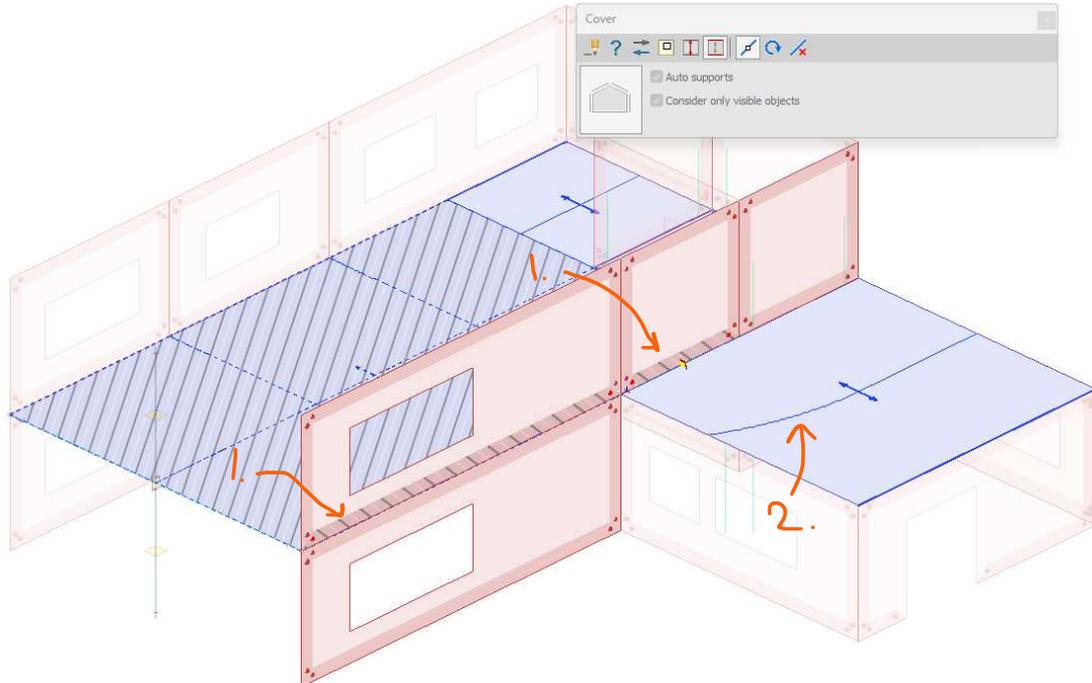
For any loads already defined, you can use the Assign loads to structure tool.



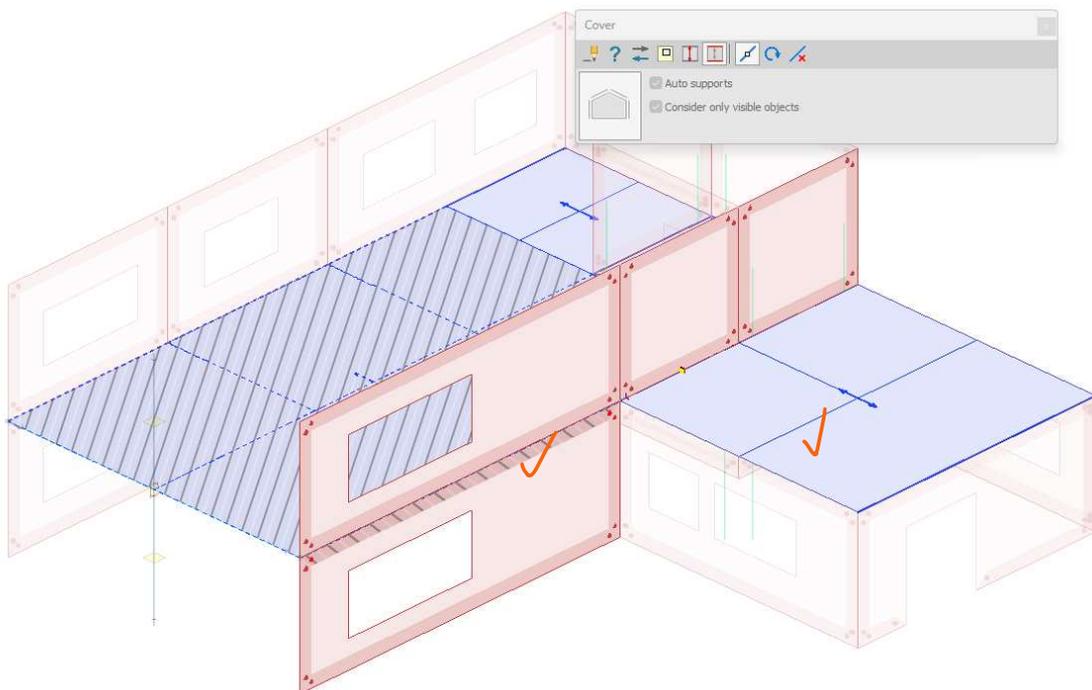
Recommendations

Supporting structures for covers

To ensure correct conversion of loads, make sure the covers have the correct supporting structures. As an example, (1) Wrong supporting structures on one side. (2) Missing supporting structure.

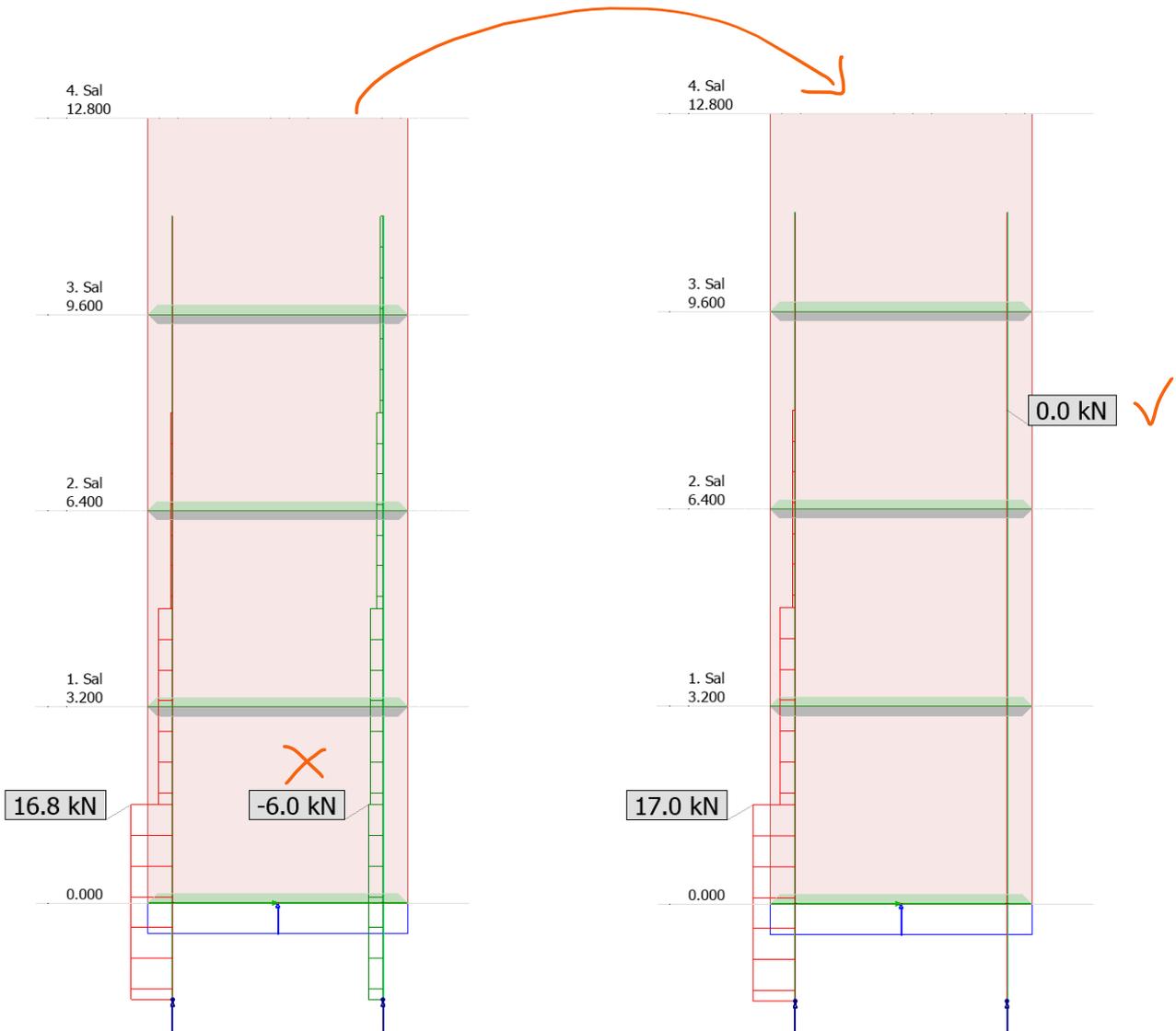
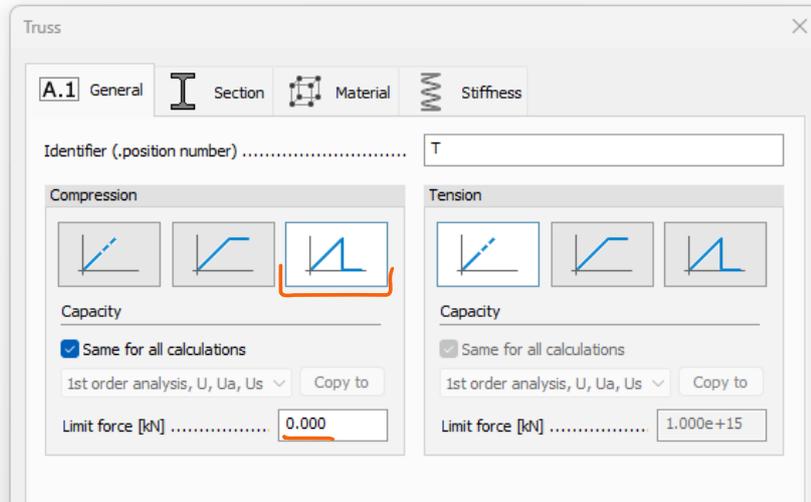


Use the manual supporting structures option in the Cover dialogue to modify/correct a covers supporting structures. Pay attention to the highlighted objects, and the load distribution line.



Anchorage behavior

To ensure correct behavior of anchorage (trusses), make sure to consider the settings for Compression/Tension. For non-compression, a good solution is to set it to *brittle* with the limit force as 0 kN. Remember this is non-linear behavior and it's only applied in load combination analysis, not load cases.





Edge-connections representing the desired behavior

For the elements it's important for the correct structural behavior to consider in each local axis, whether:

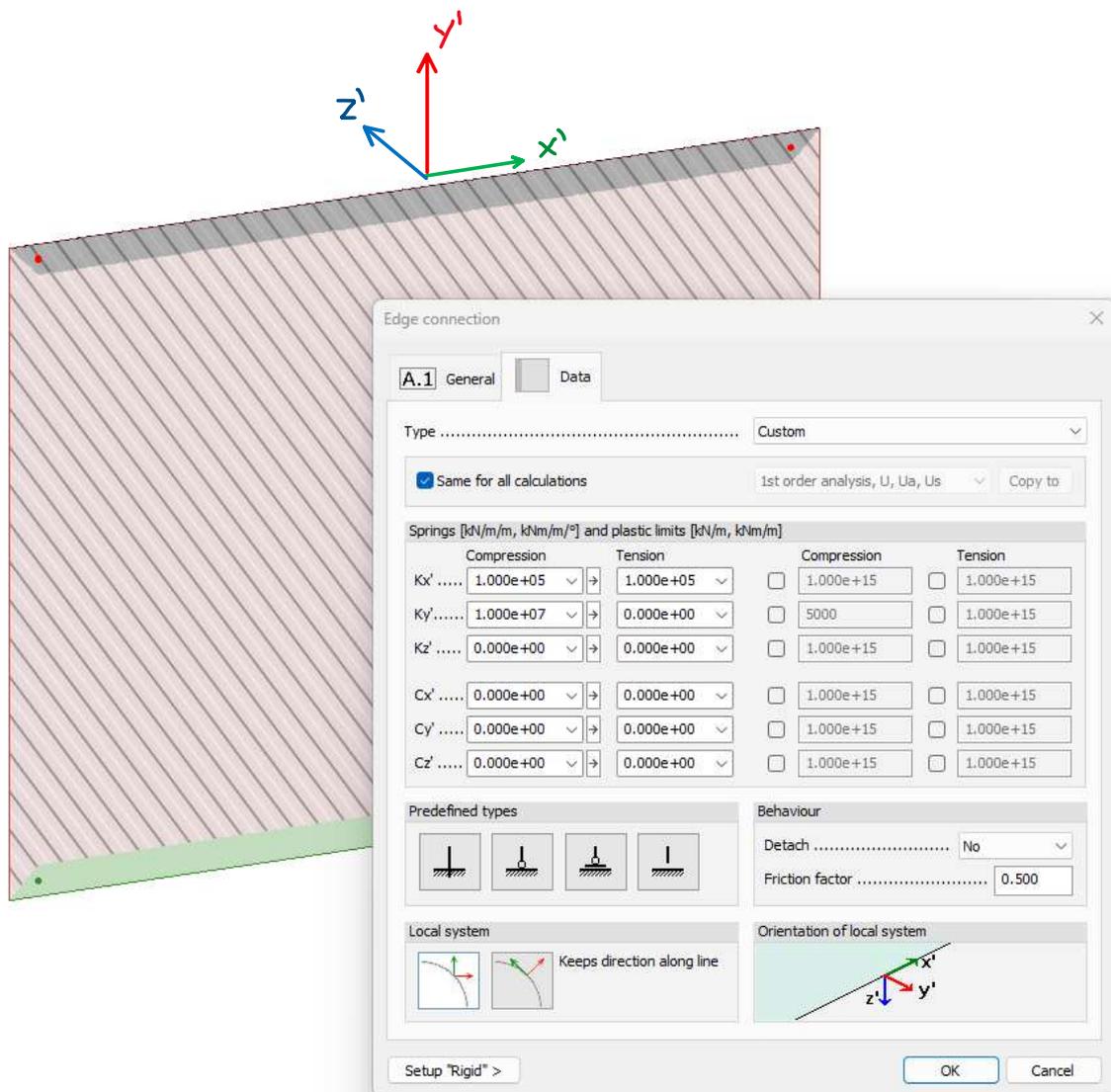
1. It's able to handle tension/compression in that direction
2. The stiffness represents the joint type -> connection forces give good results

On the element below, the local axes are drawn on the top edge.

- y' is always perpendicular to the edge
- x' is always along the edge
- z' is always perpendicular to the plane of the shell

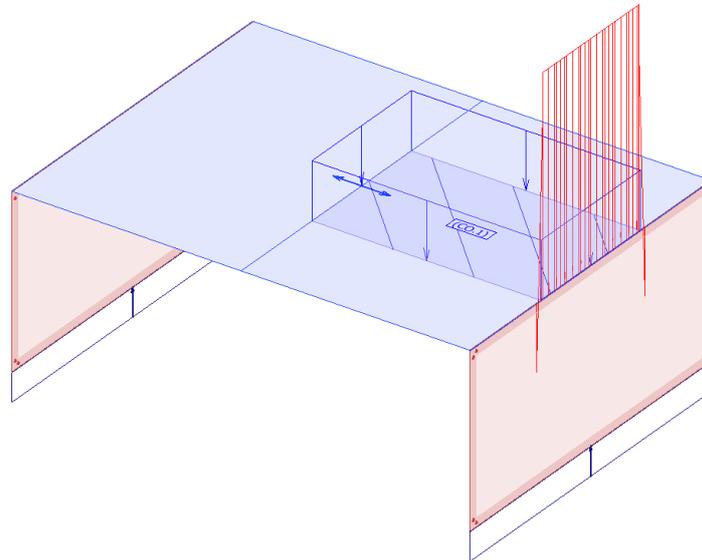
Tip: Turn on local system in the display settings as a help.

If for one of the directions, the edge as an example can't handle any forces at all if there's a displacement in a specific direction, use the Detach feature.

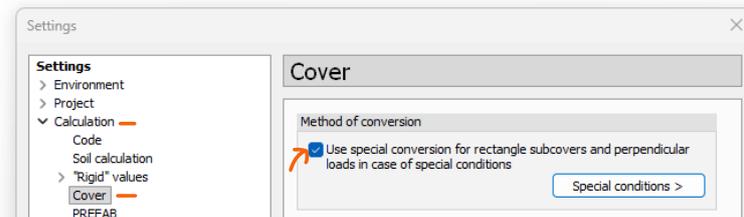


“Special conversion method” for covers

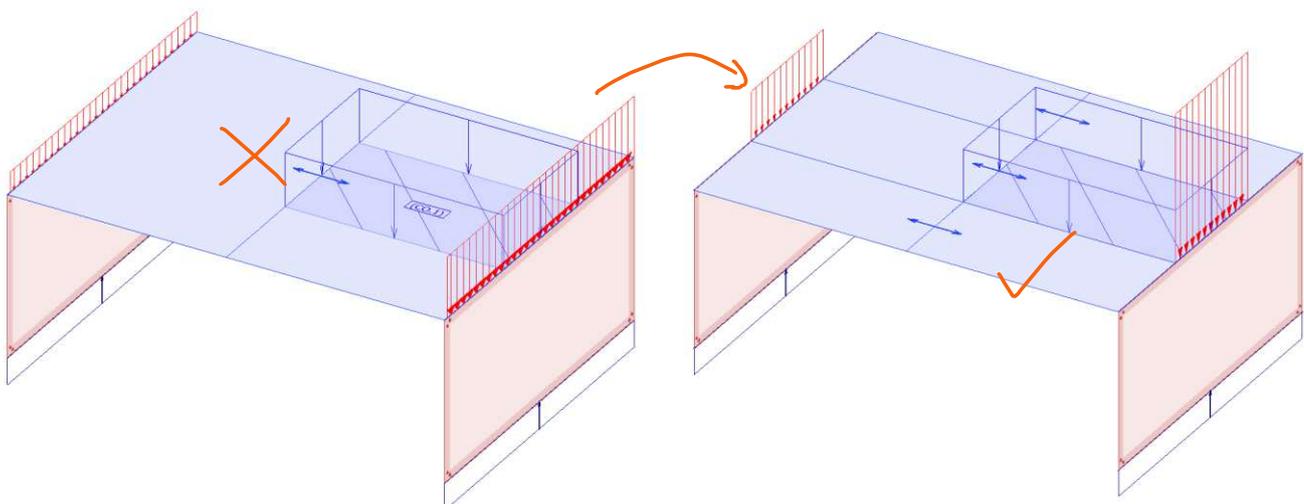
For covers it’s important to know that they function using a very basic rule. Any load inside a load distribution zone goes to the zones supporting structure. This means that in a case like below, where all the surface load is placed on the right side of the cover, all the load goes to the corresponding wall.



To avoid this, there’s an option to use what’s called “special conversion method” in the settings for covers:



With this activated, for regular single span covers the distribution will be like a single span beam. But the resultant will be distributed along the full length of the supporting structure, hence it’s recommended to split the cover in case the loading is not full length. After the split, you’ll get the result on the right.



Tip: Special conversion will be activated on ALL covers in the model, so read the Special conditions, and as always pay close attention to the result “Converted loads” from the analysis to verify the distribution.

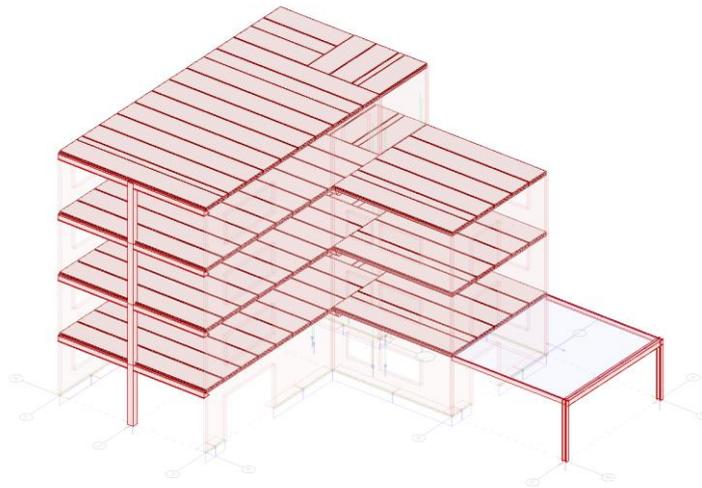
Specific for New optimized plastic redistribution

Supported structural elements

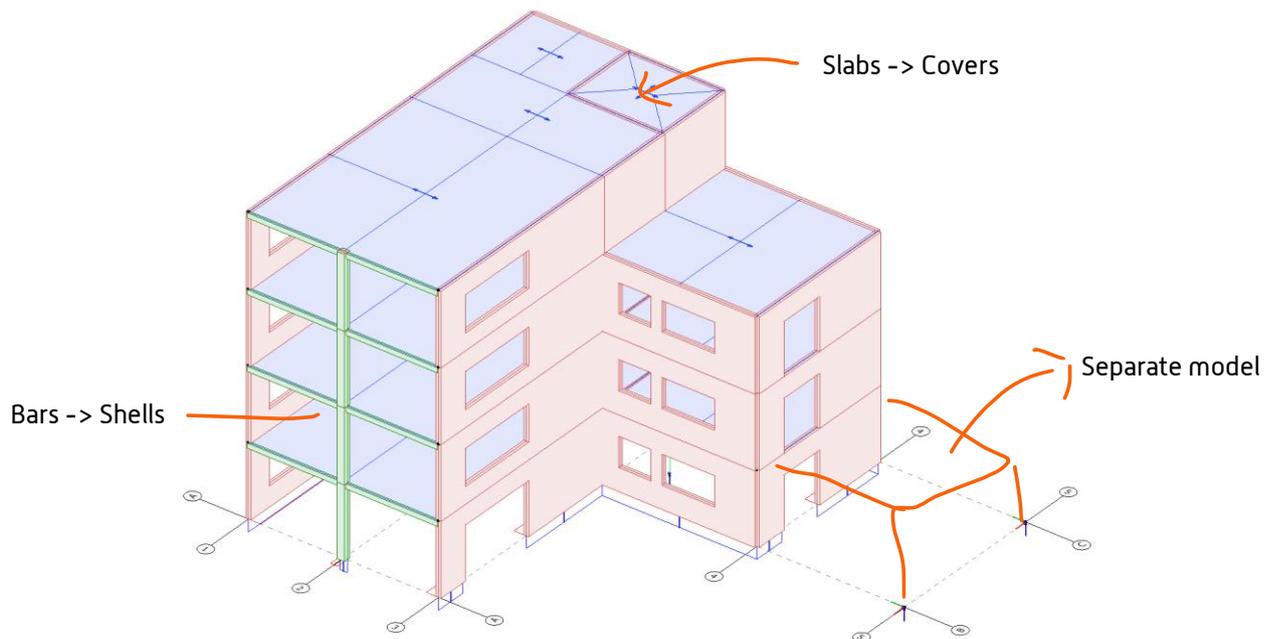
The New optimized plastic redistribution options for PREFAB are at the moment only available if the FEM-Design model contains nothing but:

1. Plane walls
2. Trusses (Must only go from element to element, so no passing through multiple walls)
3. Line support

This means slabs, bar elements, points support etc. can't be in the model.



As a solution, slabs need to be modelled as covers. Columns/beams can either be solved in a separate model with reactions converted to loads or modelled as shells with careful consideration of edge-connections.



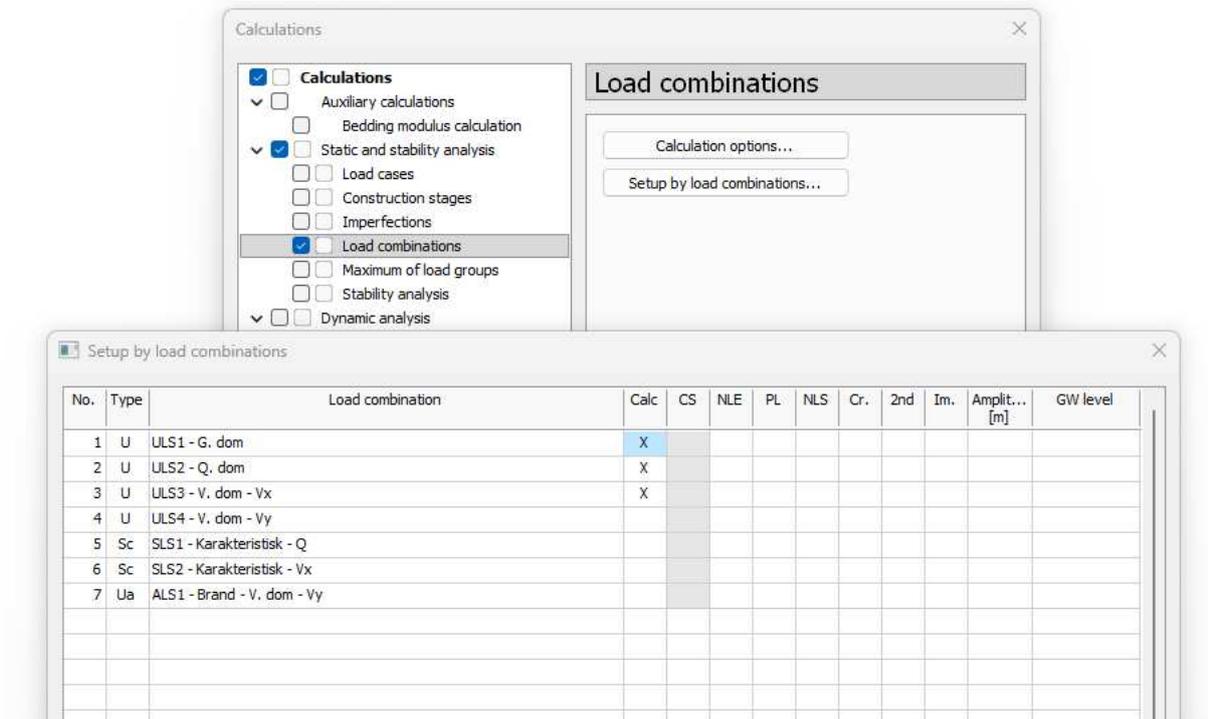
NOTE: It's on the development plan to support bar elements.



Analysis setting

PREFAB doesn't need the non-linear results from the FEM-Design analysis in order to do the new plastic redistribution, so it's highly advised to save your time by going to *Setup by load combinations...* and deselecting NLE and PL (they're on by default).

You can also control which load combinations are read by PREFAB using the Calc setting.





Design checks

As a default, PREFAB runs checks for plastic limits, or design check for **shell elements of concrete** and **steel trusses**. The design checks are run on import and will vary depending on the chosen calculation method. Where it's included in the PDF print can be specified. An example of the utilization check on import:

```

Converting results
Simplifying FEM-Design results
ULS1 - G. dom utilization 16 %
ULS2 - Q. dom utilization 22 %
ULS3 - W. dom - Vind X+ - G. gunst utilization 19 %
ULS4 - W. dom - Vind Y+ - G. gunst utilization 147 %
    
```

Here's a quick overview of the design checks:

Walls

- Compression (Min. of Eurocode in case of concrete or Plastic limit on edge)
- Tension (Plastic limit on edge)
- Shear (Min. of Eurocode in case of concrete or Plastic limit on edge)
- Sliding

Trusses

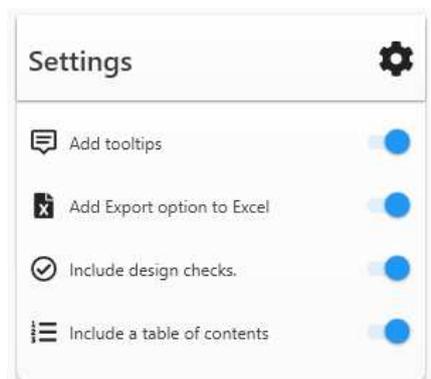
Using *Based on FEM-Design*:

- Tension of steel (Min. of Eurocode or Plastic limit)
- Compression of steel (Min. of Eurocode or Plastic limit)

Using *New optimized plastic redistribution*:

- Tension and shear of steel (Combined check with min. of Eurocode or Plastic limit)

The design checks can be toggle on/off for the PDF:





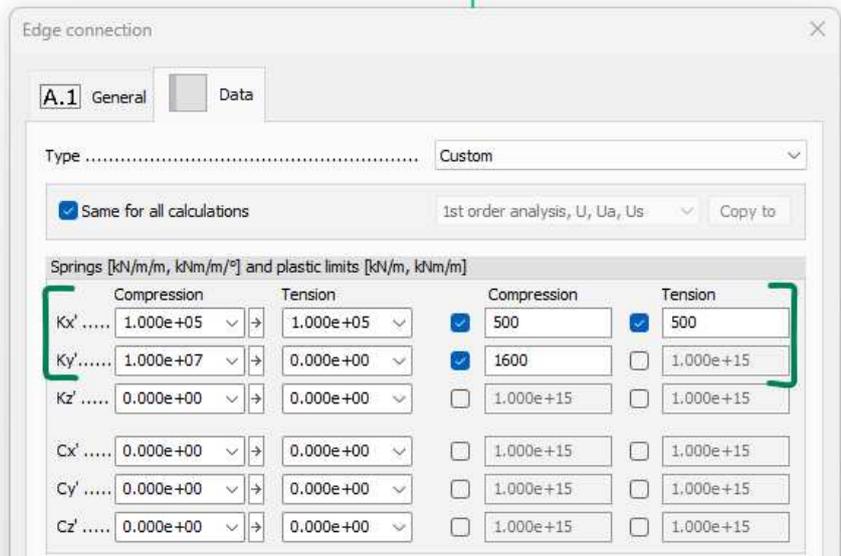
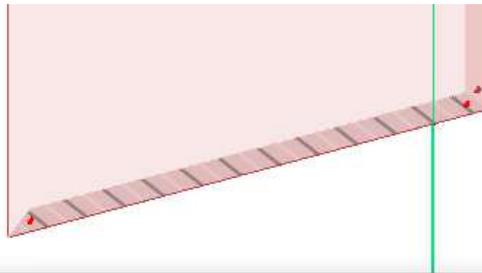
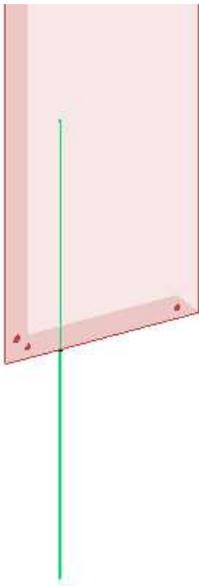
Wall design checks

Plastic limits

As a rule, PREFAB will always compare the connection forces to the plastic limits defined in the edge-connection properties. If a stiffness value is set to 0, it will be seen as a plastic limit of 0 by PREFAB.

As an example, in the following, for the selected edge-connection PREFAB will check to see if

- **Shear V_{PL}** (Kx' – Both directions) – 500 kN/m
- **Compression C_{PL}** (Ky' - Compression) – 1600 kN/m
- **Tension limit T_{PL}** (Ky' - Tension) – 0 kN/m *since stiffness is set to 0.*



NOTE: This is not connected to the material used, and it only checks this for Kx' and Ky'.



Concrete wall specific checks

NOTE: We support National Annex for Norway, Denmark, and Sweden. The chosen Annex in FEM-Design will be used by PREFAB.

Tension

If K_y' tension = 0 – No tension allowed (Will show 1000% utilization)

Springs [kN/m/m, kNm/m²] and plastic limits [kN/m, kNm/m]							
	Compression		Tension			Tension	
Kx'	1.000e+06	→	1.000e+06	↓	<input type="checkbox"/>	500	<input type="checkbox"/>
Ky'	1.000e+06	→	0.000e+00	↓	<input type="checkbox"/>	1600	<input type="checkbox"/>
Kz'	0.000e+00	→	0.000e+00	↓	<input type="checkbox"/>	1.000e+15	<input type="checkbox"/>

If K_y' tension > 0 – Tension is allowed. **There's no concrete check. If a plastic limit is applied the utilization is:**

$$u = \frac{T_{Ed}}{T_{PL}}$$

Springs [kN/m/m, kNm/m²] and plastic limits [kN/m, kNm/m]							
	Compression		Tension			Tension	
Kx'	1.000e+06	→	1.000e+06	↓	<input type="checkbox"/>	500	<input type="checkbox"/>
Ky'	1.000e+06	→	1.000e+06	↓	<input type="checkbox"/>	1600	<input checked="" type="checkbox"/>
Kz'	0.000e+00	→	0.000e+00	↓	<input type="checkbox"/>	1.000e+15	<input type="checkbox"/>

Compression

The compression capacity is simply calculated as:

$$C_{Rd} = t \cdot \alpha_{cc} \cdot \frac{f_{ck}}{\gamma_c}$$

Where:

t = thickness set in FEM-Design

f_{ck} = from the material set in FEM-Design

γ_c = from the material U | Ua, Us setting in FEM-Design

α_{cc} = from the material setting in FEM-Design

The utilization is calculated as:

$$u = \frac{C_{Ed}}{\min(C_{Rd}, C_{PL})}$$

Library material		Material	Default connections	Stiffness
Name	C25/30			
Compressive strength - f _{ck} [N/mm²]	25.0			
γ _c (U Ua,Us)	1.45	1.00		



Shear of concrete

The shear capacity of concrete is simply calculated as:

$$V_{Rd} = t \cdot 0.5 \cdot v \cdot \frac{f_{ck}}{\gamma_c}$$

Where:

t = thickness set in FEM-Design

f_{ck} = from the material set in FEM-Design

γ_c = from the material U | Ua, Us setting in FEM-Design

v = effectiveness factor – depends on NA:

For DK:NA

$$v = \max \left[\left(0.7 - \frac{f_{ck}}{200} \right), 0.45 \right]$$

For all others

$$v = 0.6 \cdot \left(1 - \frac{f_{ck}}{250} \right)$$

The utilization is calculated as:

$$u = \frac{V_{Ed}}{\min(V_{Rd}, V_{PL})}$$

Sliding

When **detach** is activated on an edge, a combined shear and compression check is activated to check for sliding. The compression and shear of concrete is checked individually as explained already.

The utilization is simply calculated as:

$$u = \frac{V_{ed}}{C_{ed} \cdot \mu}$$

Where:

μ = friction factor from the edge set in FEM-Design



In case $u > 100\%$ PREFAB will tell the necessary mechanical anchorage needed to keep it from sliding.



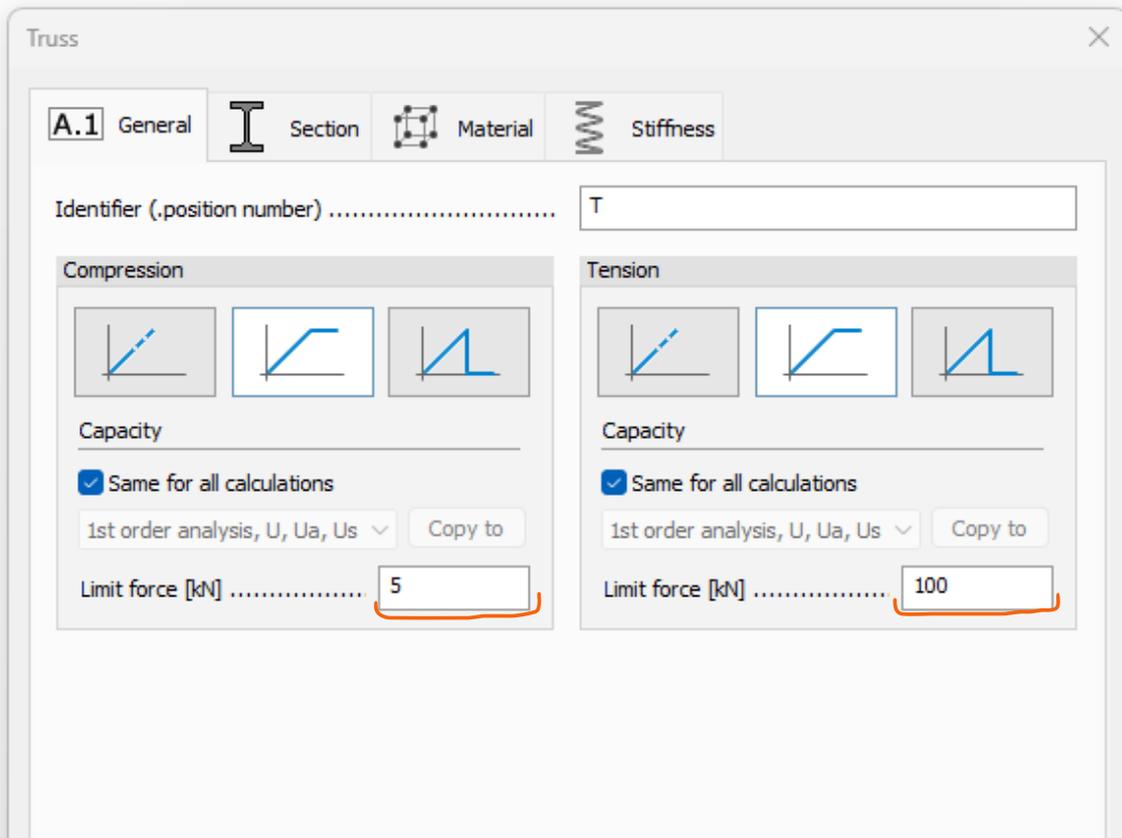
Truss design checks

Plastic limits

As a rule, PREFAB will always compare the normal forces to the plastic limits defined in the truss properties.

As an example, in the following, for the selected truss PREFAB will read the plastic limits as:

- **Compression** C_{PL} - 5 kN
- **Tension** T_{PL} - 100 kN

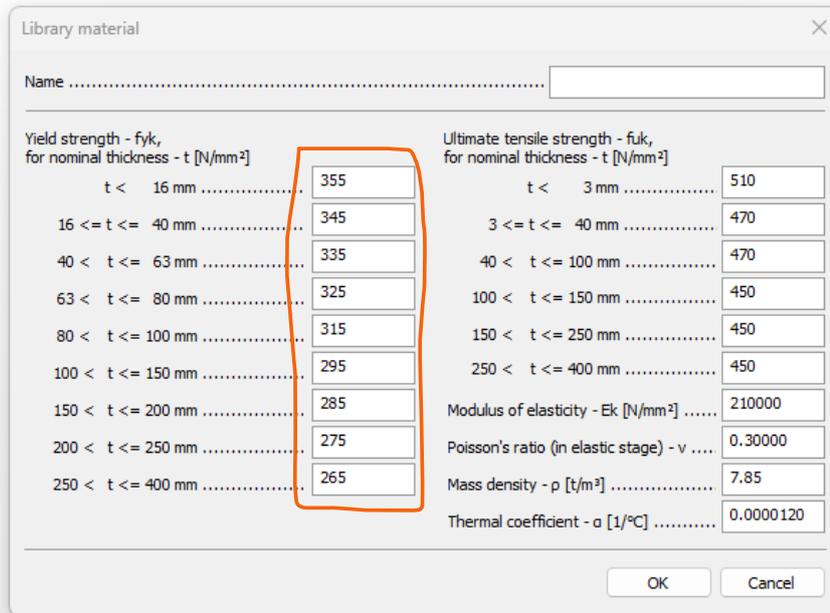




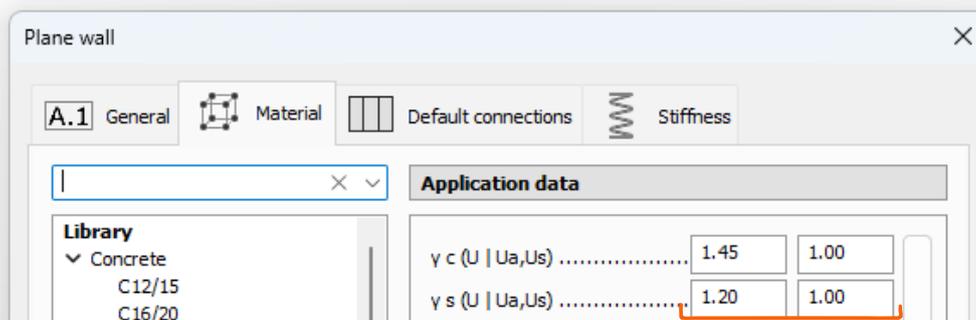
Steel truss specific design checks

General information

When PREFAB gets the yield capacity of the truss element it takes the smallest value defined for the material in FEM-Design. You can create a new material and specify the desired yield capacity in all the fields highlighted below.



For the design value, PREFAB reads the maximum γ_s of the connected shell elements.





Based on FEM-Design: Tension/Compression of steel

The tension capacity is simply calculated as:

$$T_{Ed} = \frac{f_{yk}}{\gamma_s} \cdot A_s$$

And the utilization:

$$u = \frac{T_{Ed}}{\min(T_{Rd}, T_{PL})}$$

The compression capacity is simply calculated as:

$$C_{Rd} = 0.8 \cdot \frac{f_{yk}}{\gamma_s} \cdot A_s$$

And the utilization:

$$u = \frac{C_{Ed}}{\min(C_{Rd}, C_{PL})}$$

New optimized redistribution: Tension and shear of steel

For the optimized redistribution, shear transfer will be used if the element can't utilize the friction from compression between the elements alone. In this case a combined tension and shear check is performed.

The utilization is simply calculated as:

$$u = \frac{T_{Ed}}{\min(T_{Rd}, T_{PL})} + \frac{V_{Ed}}{\min(T_{Rd}, T_{PL}) \cdot \mu}$$