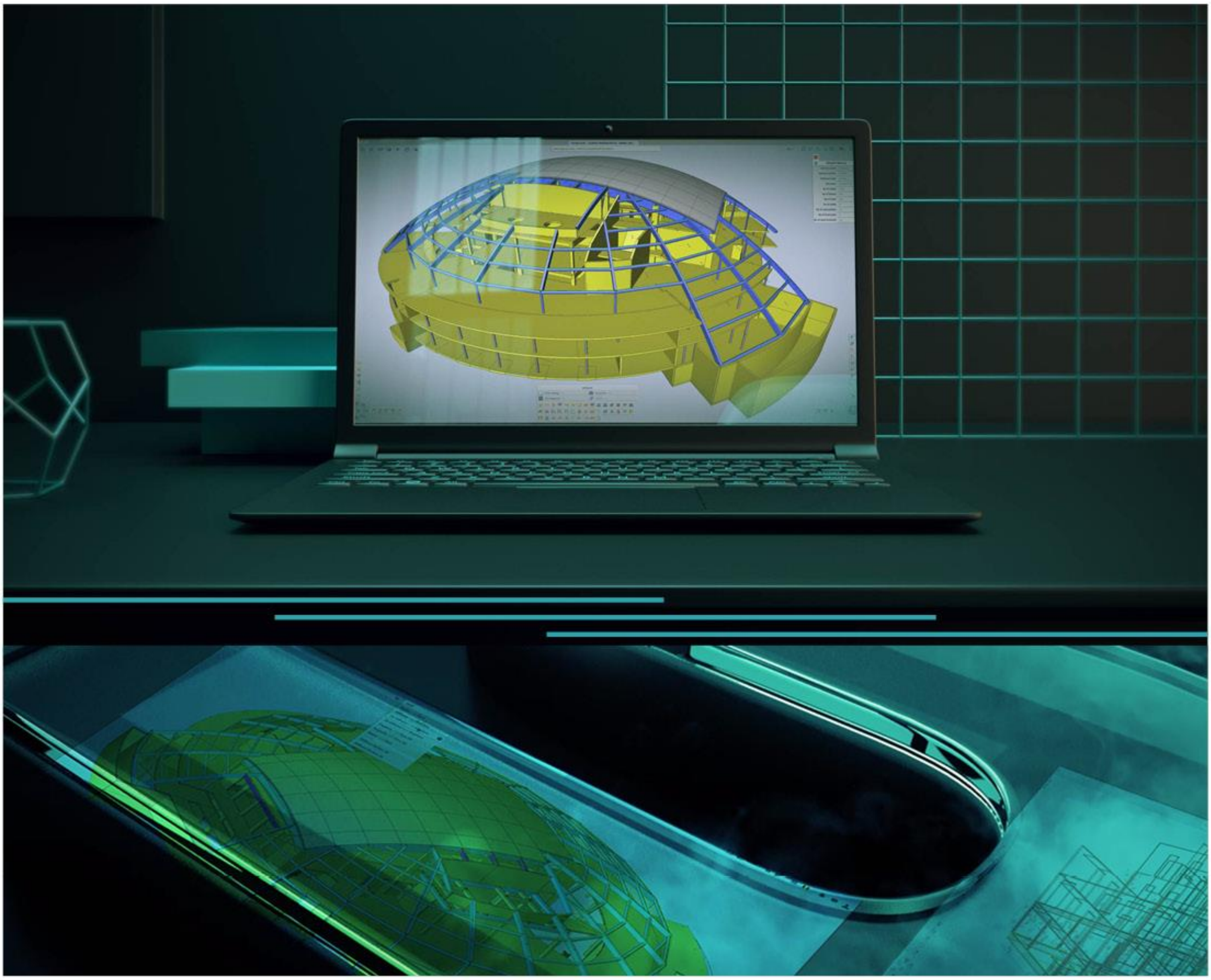


Advanced Expert Trainig

Pre-tensioning (Precast)

Table of Contents

Introduction

All discussed topics are available in the **Expert Edition** or the **Precast Edition** of SCIA Engineer.

Most of the functionalities used in this tutorial are currently only available in the “V16 and older” PPE in the 32-bit version of SCIA engineer.

Only the functions used in 1.1 Input geometry and prestress have already been transferred to the default PPE in the 64-bit version and can thus be used in the new GUI.

Because only one subchapter is possible in the new GUI, we will execute the entire tutorial in SCIA 21.1 32-bit.

In this training document an example will be worked out for a precast beam element which is part of a bridge deck.

In the first chapter the example will be worked out from scratch. The input of pre-tensioning, a time dependent construction stage analysis (TDA) and some EN checks of the concrete menu will be treated.

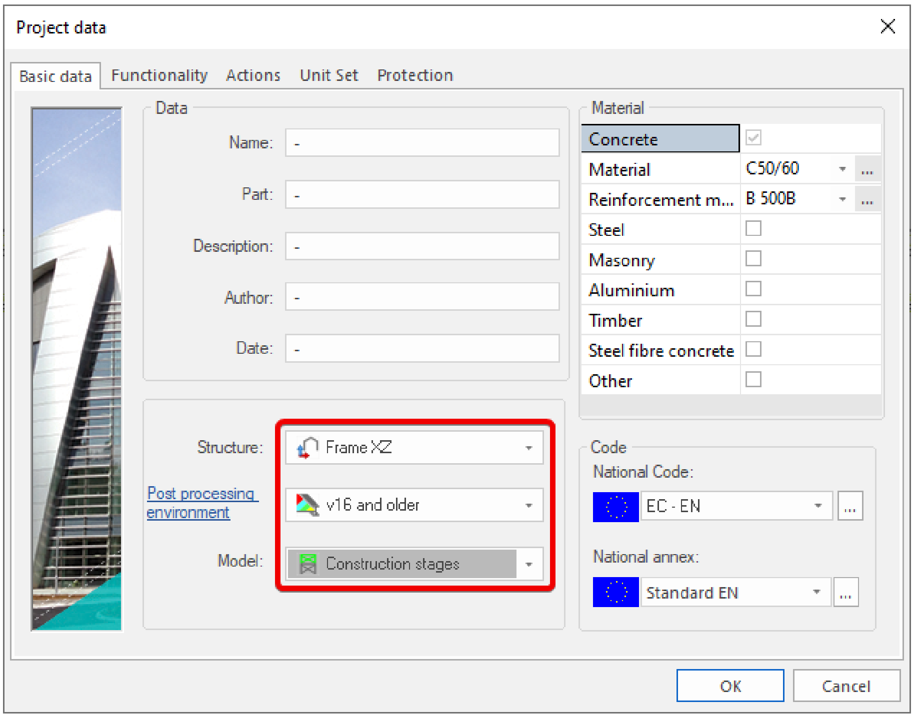
In the last chapter will be illustrated that the 1D beam model could be created directly from a 2D plate model in which the load distribution is analyzed. This can be done through the 2D->1D upgrade functionality.

Chapter 1: Pre-tensioned precast beam

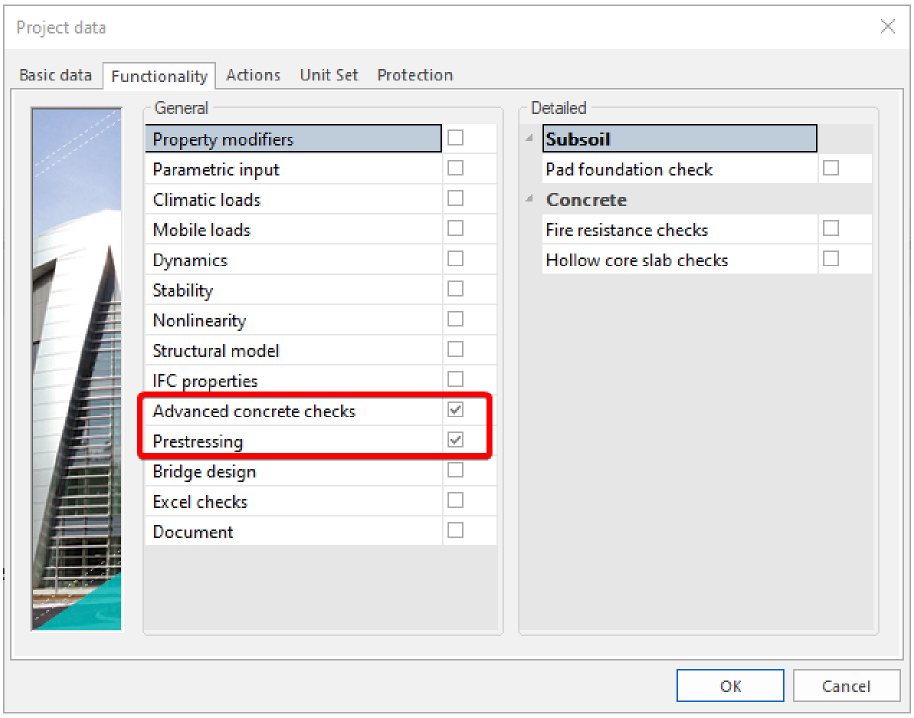
1.1. Input geometry and prestress

1.1.1. Project settings

In the project data a frame XZ environment and a construction stages model will be chosen and we will use the v16 and older post processing environment.

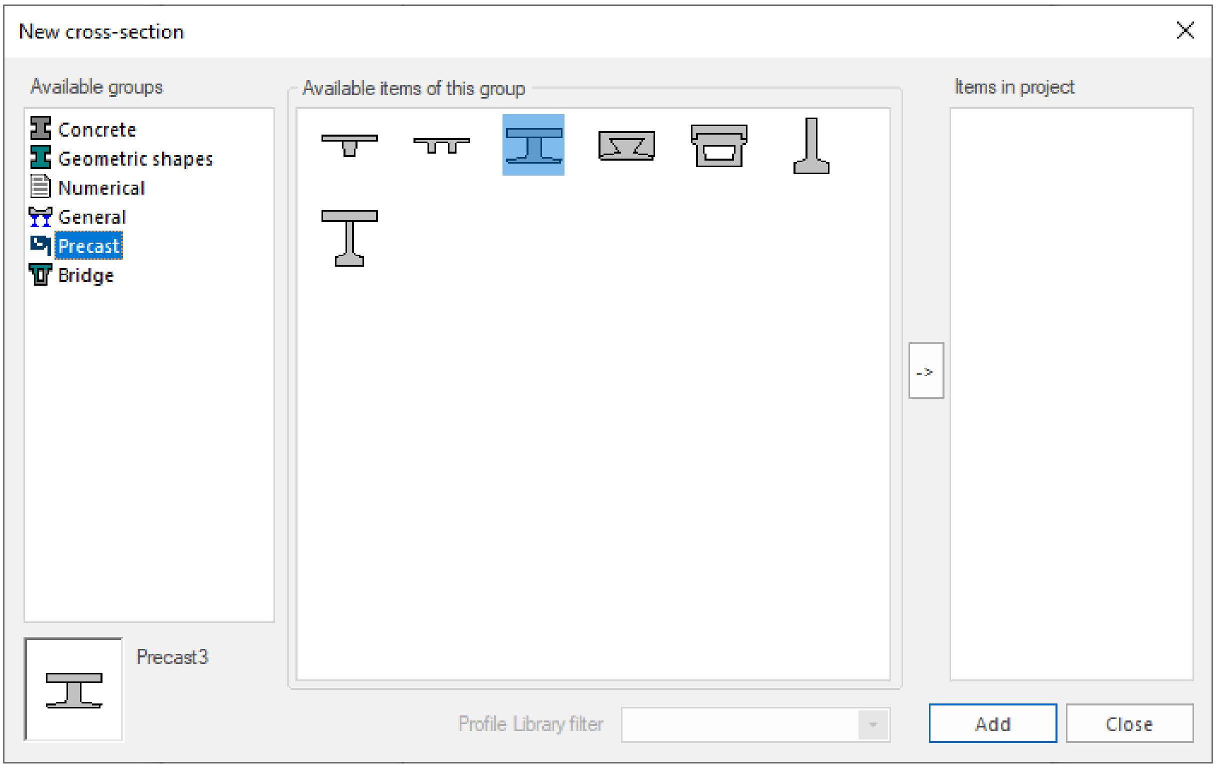


The functionalities “Advanced concrete checks” and “Prestressing” will be ticked on.



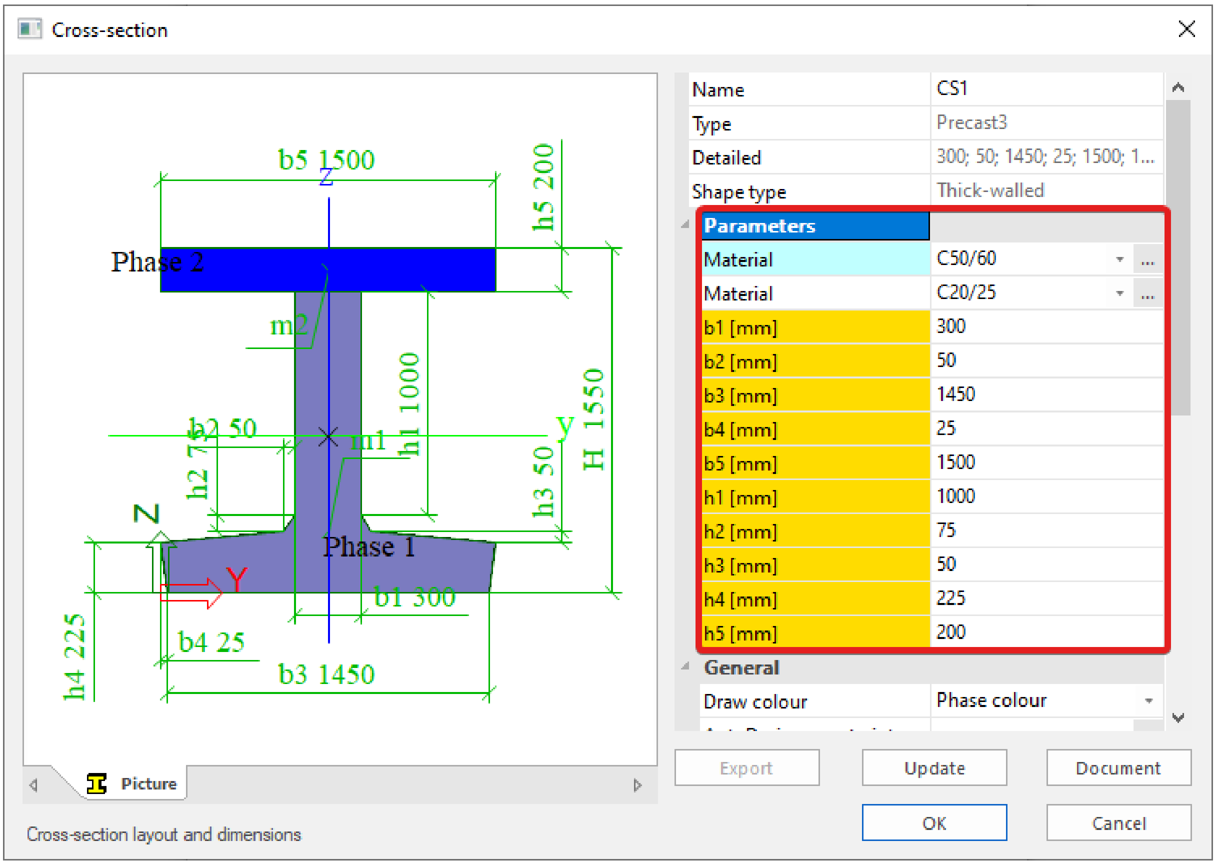
1.1.2. Input geometry

In the Scia Engineer profile library we will find some parameterized prefab profiles. In this example we will choose for the Prefab3 profile type.



Next to this list of predefined types of prefab profiles, the user also has the possibility to create his own type of profile by using the ‘General’ cross-section.
The General cross-section contains a cross-section editor which allows the user to draw a shape himself by using the tools in the editor menu or by importing a dwg/dxf file.

In this example, we will use the prefab I-beam with an in situ cast topping.
The following parameters are selected for this exercise:



Next, insert a 30 m long beam with this cross-section and add support at both ends.

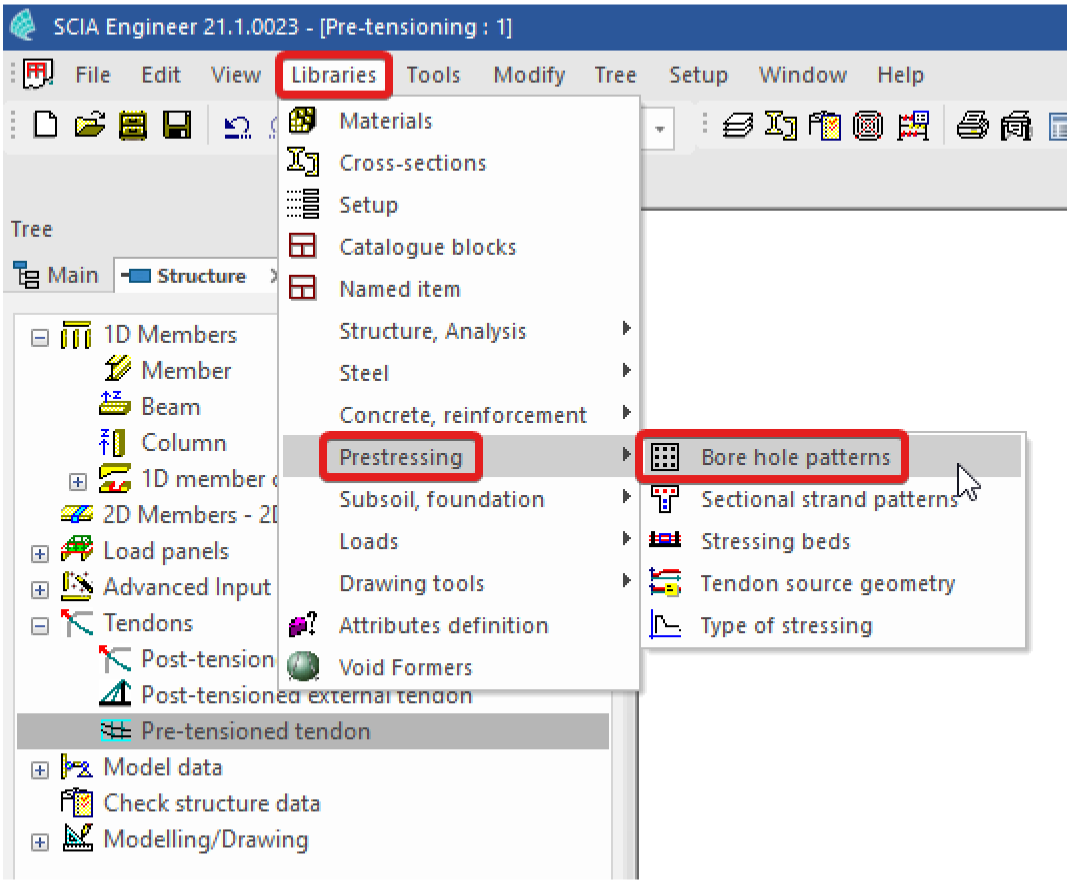
1.1.3. Load cases + loading

Before we continue with the input of the pre-tensioned tendons, it is useful to set up the load cases and their corresponding loads. To do this, use the following table as a guideline.

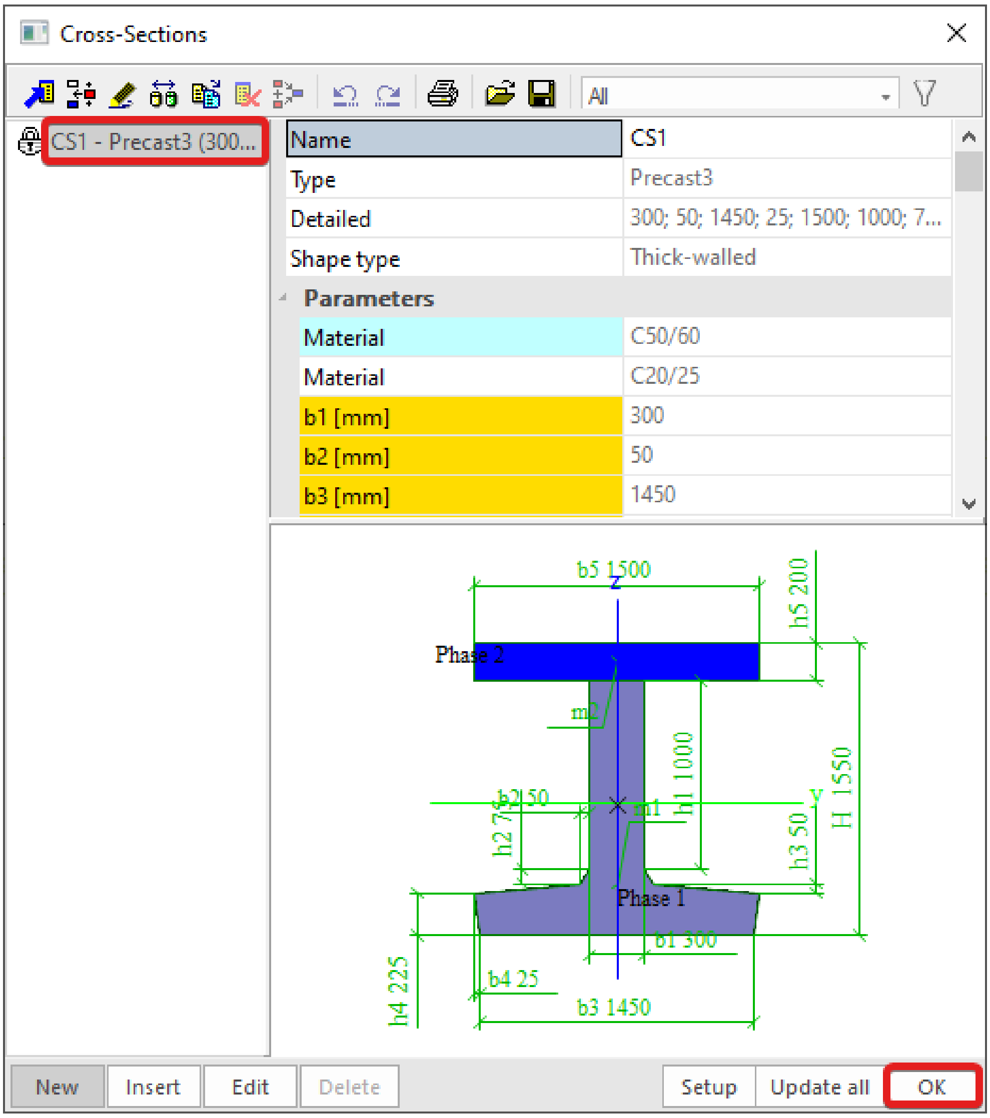
Name	Description	Action type	Load group	Load type	Duration	Load
LC1	Prestress	Permanent	LG1	Prestress	/	Automatic
LC2	Placing of concrete beam	Permanent	LG1	Self weight	/	Automatic
LC3	Weight of in situ topping	Permanent	LG1	Standard	/	-7,5 kN/m
LC4	Finishes	Permanent	LG1	Standard	/	-10 kN/m
LC5	Service load	Variable	LG2	Static	Long	-20 kN/m
LC6	Empty (check after 50 years)	Permanent	LG1	Standard	/	/

1.1.4. Bore hole pattern

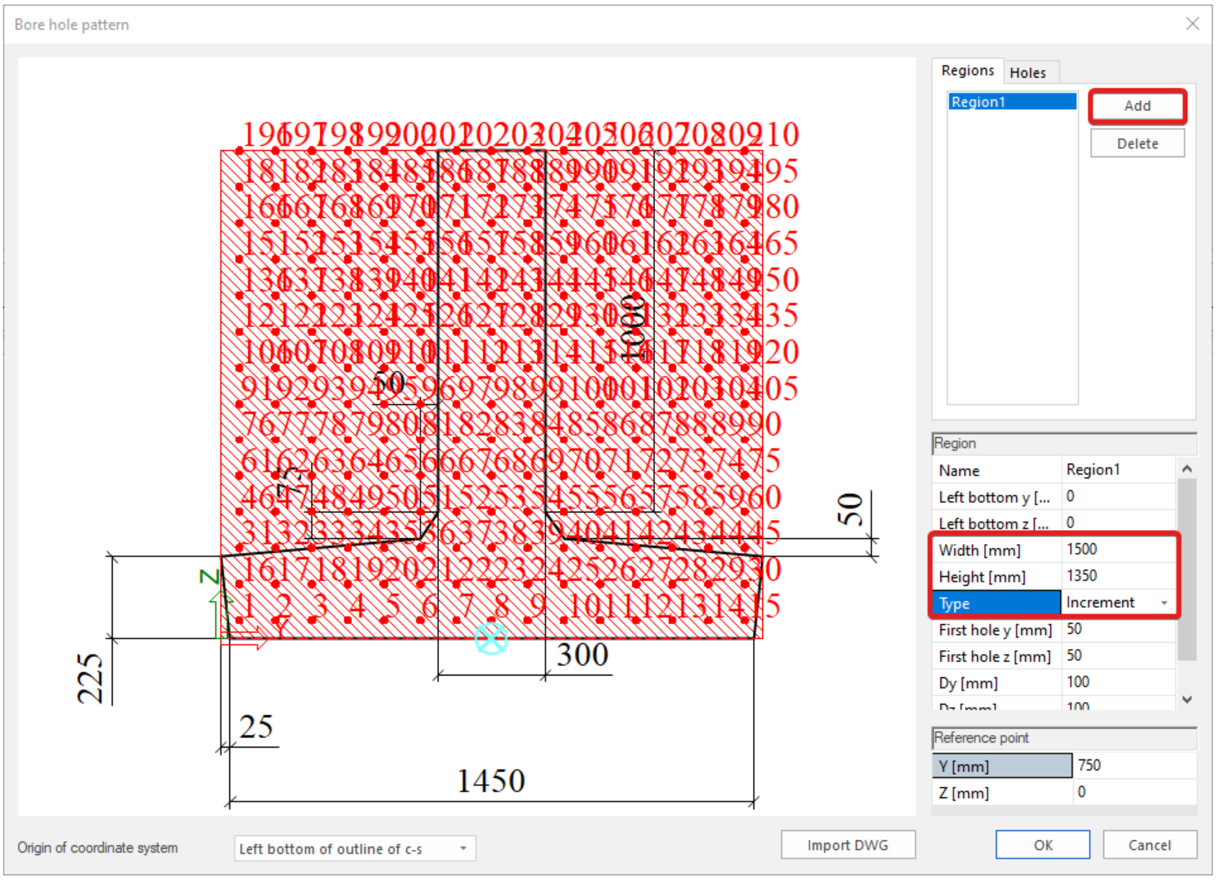
Before we enter the pre-tensioned tendons, we will have to create a bore hole pattern. We access the bore hole database automatically when we want to add pre-tensioned tendons and no bore hole patterns are found, or via Libraries as shown below.



If we enter the database for the first time, there will be no patterns and we will receive the opportunity to define a new bore hole pattern on a profile. We choose of course the earlier defined cross-section.



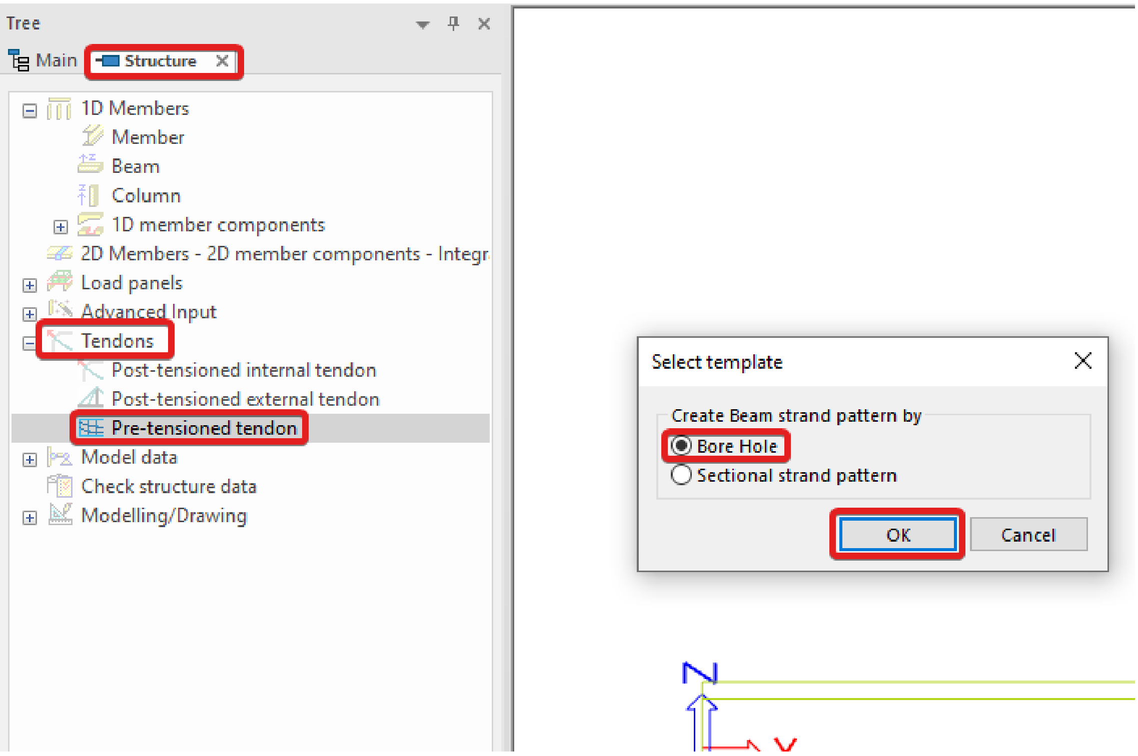
The definition of the bore hole pattern can be done by means of the input of individual holes or a whole region. In this example a whole region over the cross-section will be created by adding a region and adjusting its width and height. Finally select Increment as its type.
(Notice that the program automatically detects that no bore hole pattern can be defined in the in situ concrete pressure zone.)

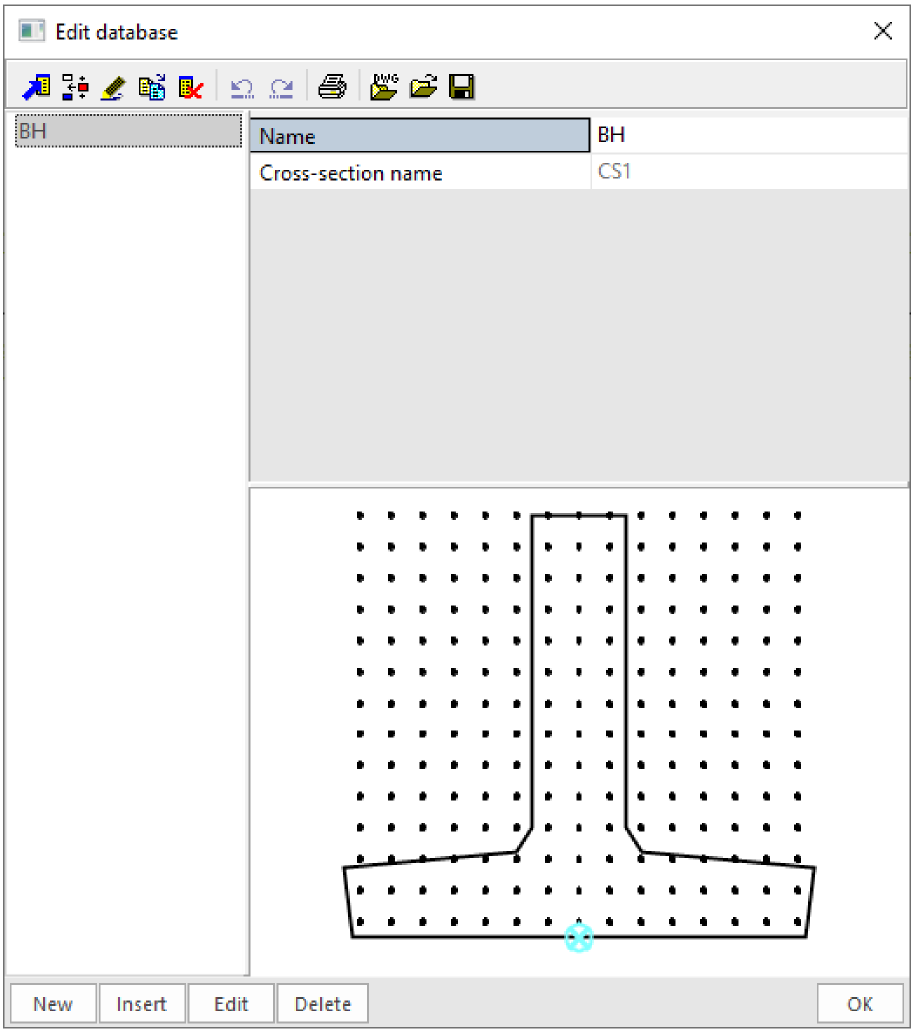


This borehole profile could be stored to use in other projects.

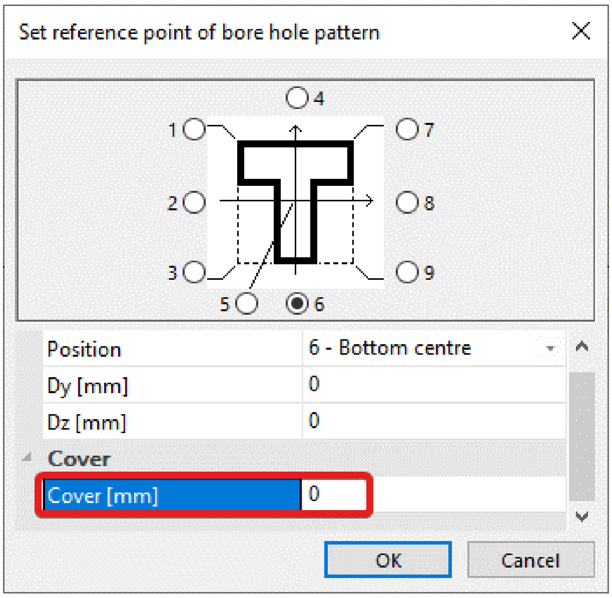
1.1.5. Pre-tensioned tendons

The input of the pre-tensioned tendons is done in the structure menu. We will start with the creation of a bore hole pattern:

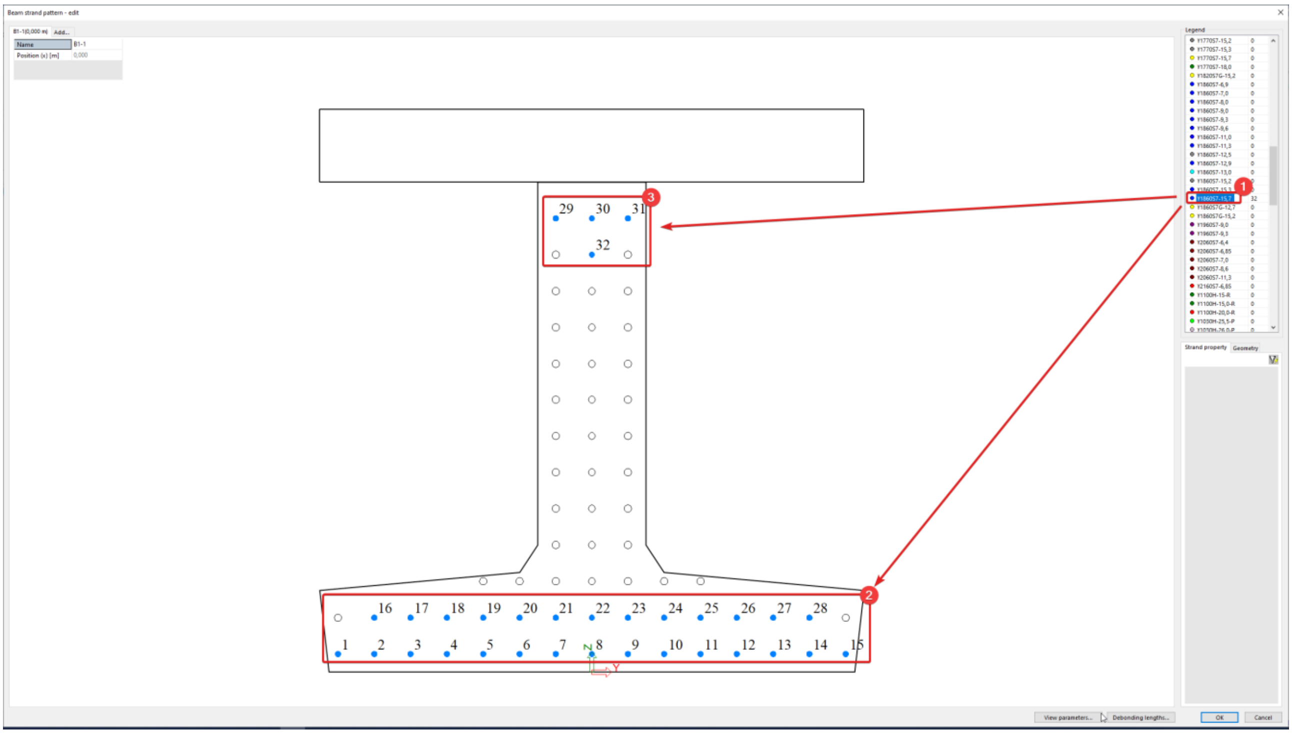




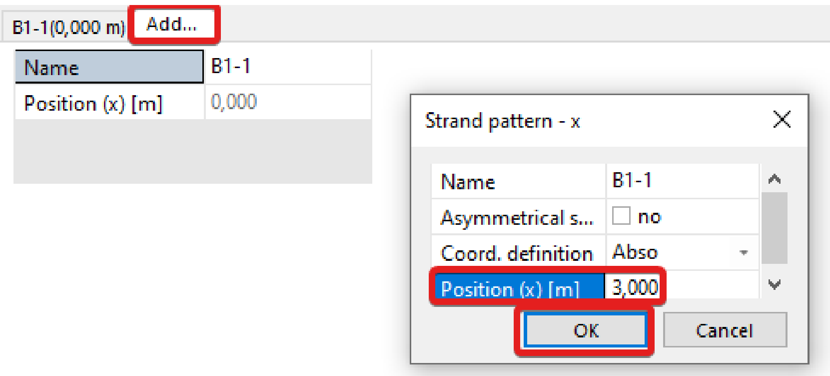
Now we will use this bore hole pattern to define a strand pattern.
As reference point for the input of the bore hole pattern, a cover of 0mm will be applied because the cover is already taken into account in the definition of the bore holes.



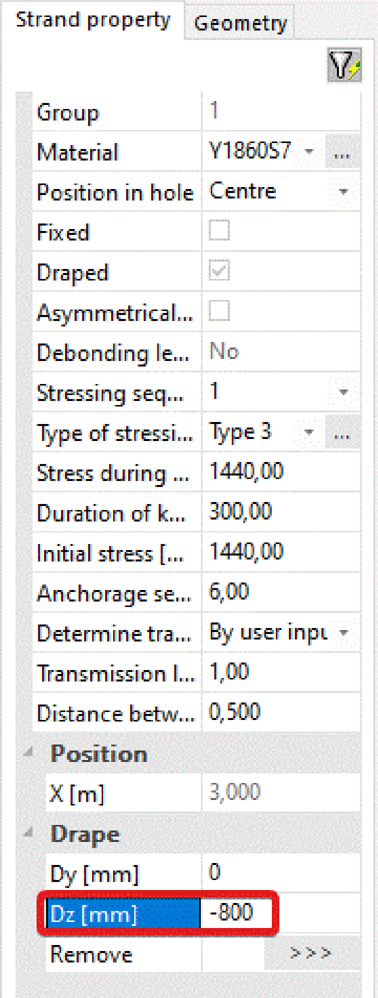
The beam strand pattern is shown below. Choose Y1860S7-15,7 as the strand quality and assign this to the 32 positions as shown below.
(You have to select the strands one by one and you have to reselect the strand quality if you miss click.)



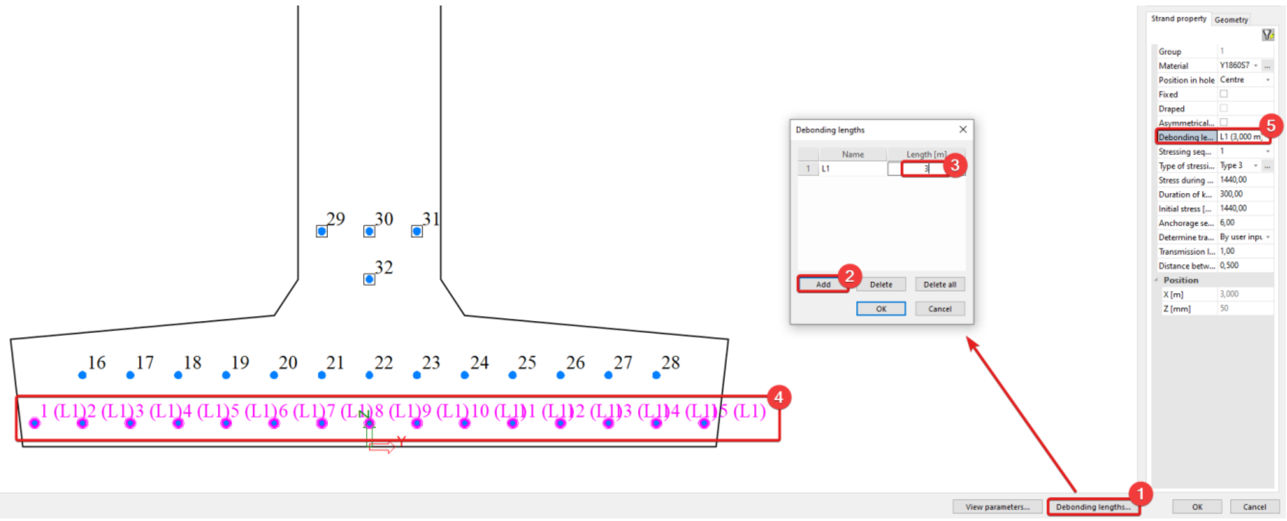
The top strands are bent so they will join the bottom strands at a distance of 3m from the support. To achieve this an extra section at 3m is created and on this section a drape of 800mm is given to the top strands. You can find the add-button on the top right side of the window.



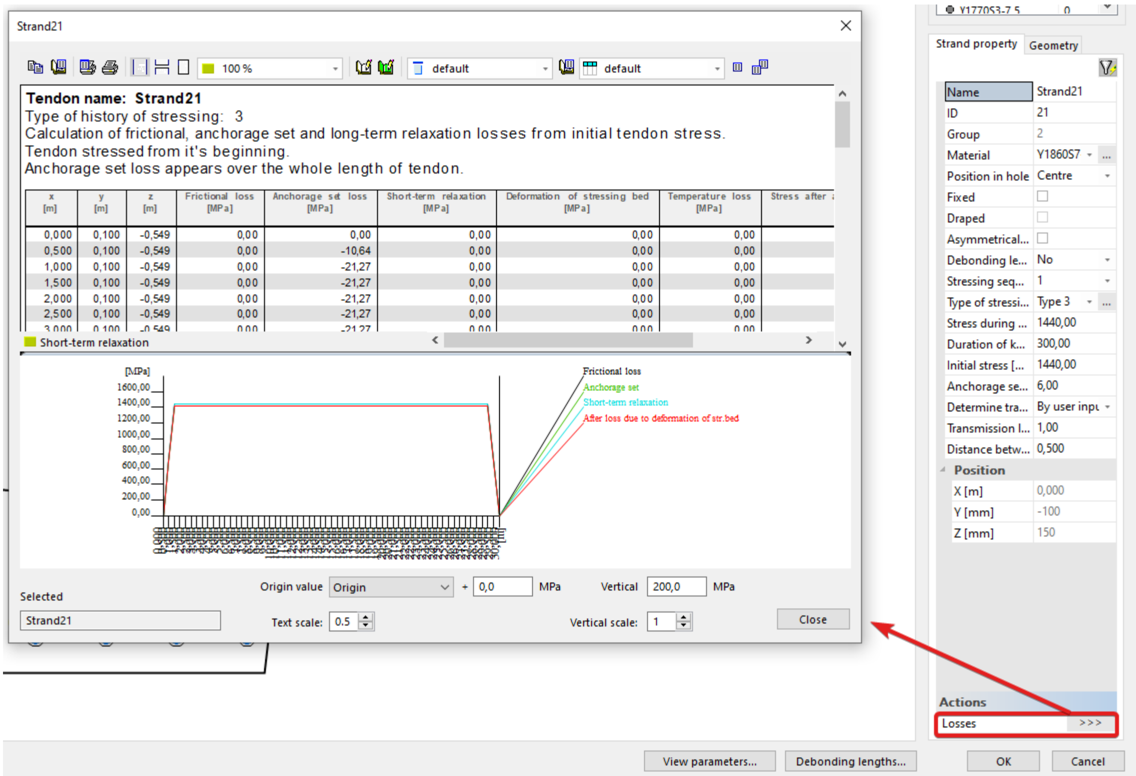
Select the four strand at the top of the beam by holding ctrl and edit their Drape (Dz) in the properties window on the right.



Another measure that could be taken to avoid that in the begin section a tension stress at the top fibre would occurs the debonding of some strands over the first meters. In this example the strands on the bottom row are all given a debonding length of 3m

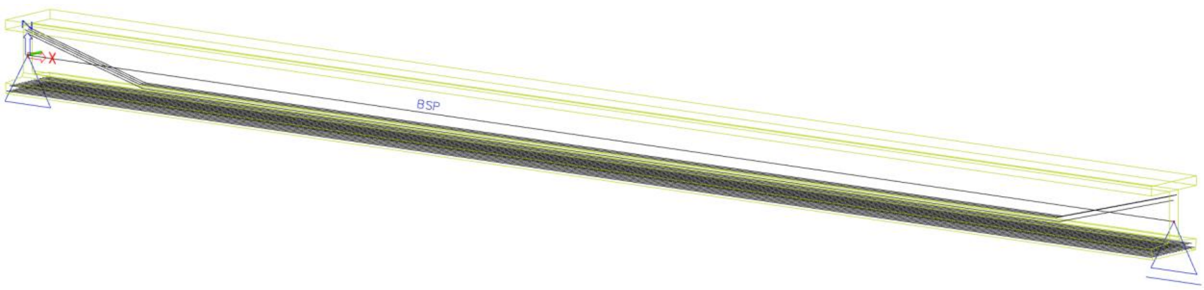


It is already possible to have a look at the stress after transfer that is put to the beam before the construction stage analysis is run. This can be accessed when one strand is selected via the action menu below the properties of a strand.

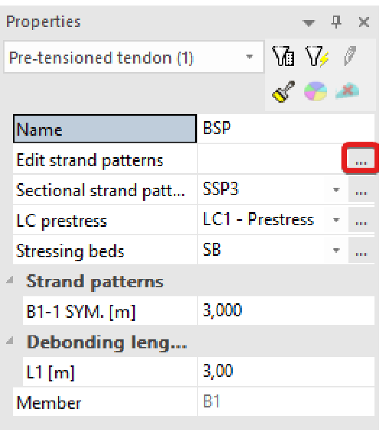


In this example a transmission length of 1m is entered. Also no additional losses like an anchorage set loss or a short term relaxation loss are taken into account. They could be defined by means of the advanced parameters, but they are not treated in this example.

The final strand pattern will look like the following.



To edit the properties afterwards, the beam strand pattern or “BSP” can be selected. Trough the property menu the user can click on edit strand pattern to edit the strand pattern at any time.



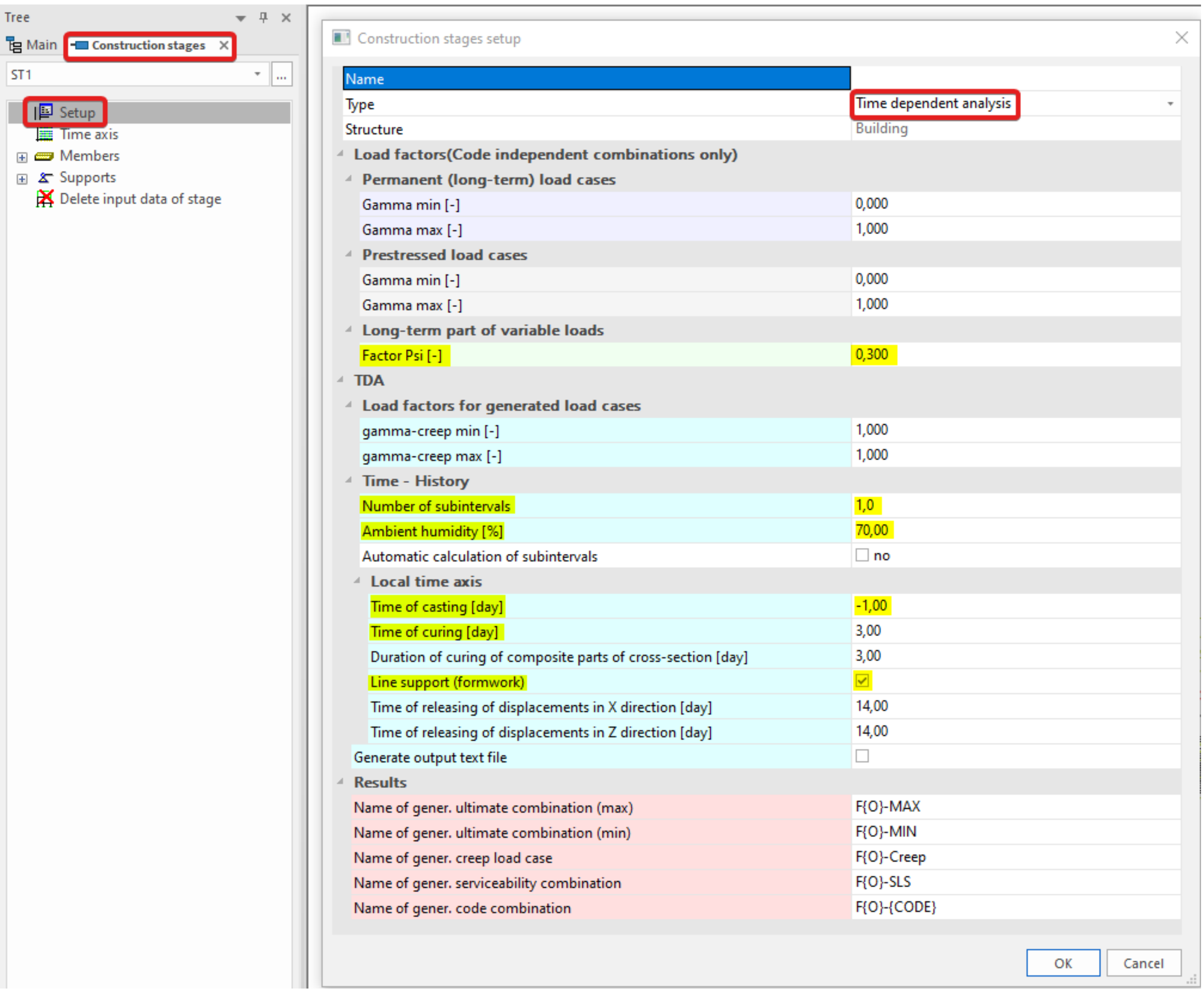
1.2. TDA Construction stages (and loading)

The construction stages in this project will not be normal linear construction stages, but they will be used for time dependant analysis. This means that also a time should be assigned to each construction stage.

It is easier to create the load cases before the stages, because for every stage at least one permanent load case is required. We already did this in 1.1.3 Load cases, so we can now enter the stages.

1.2.1. Construction stages setup

The configuration window for a time dependent analysis will automatically will open when you select Construction stages for the first time or can be accessed via Construction stages > Setup as shown below. Select “Time dependent analysis” as the “Type”.



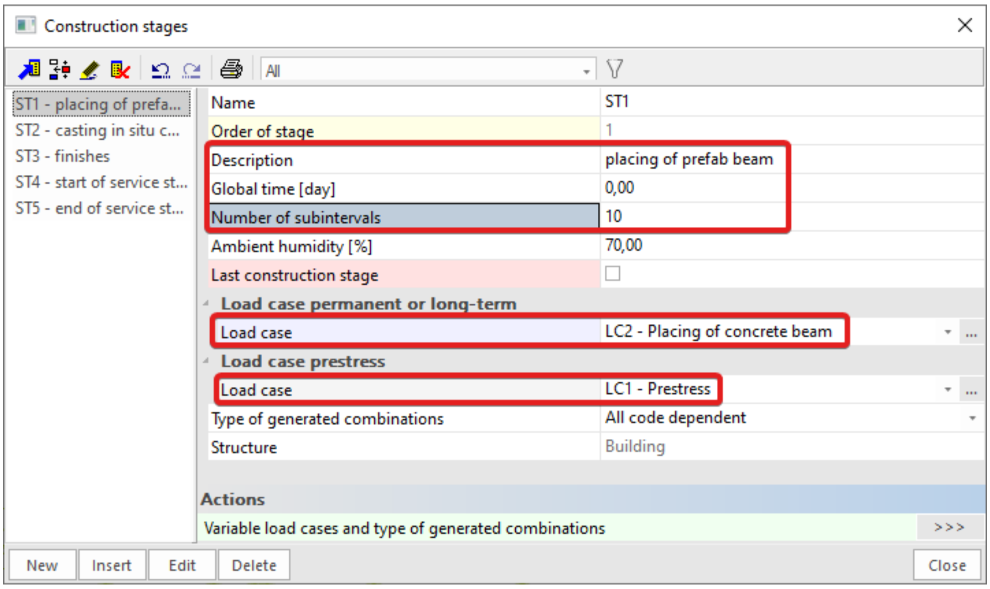
The most important properties are:

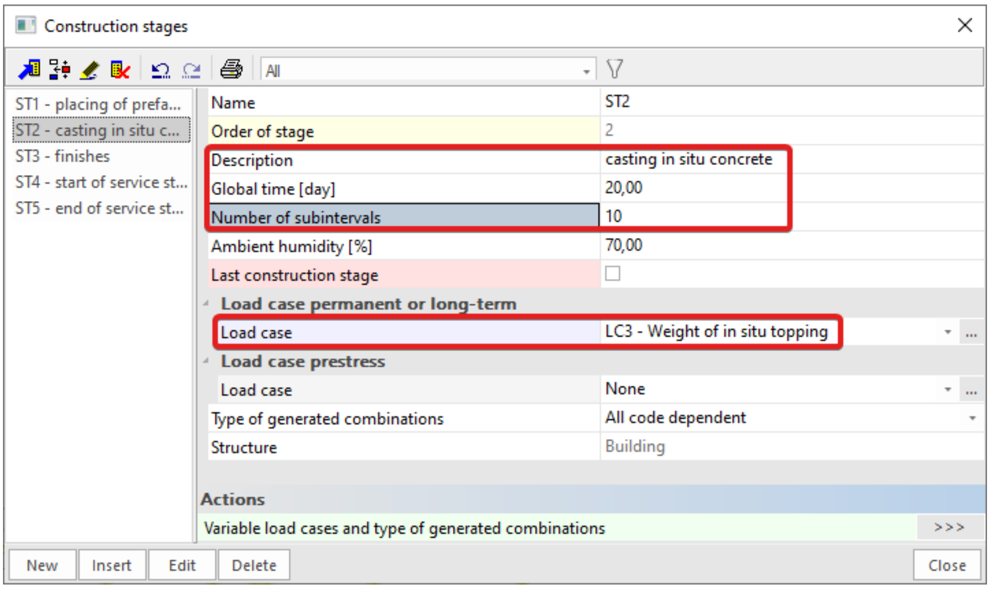
- Long term part of variable load: Factor Psi [-]
  This implies that variable loads of long term are also taken into account for the time dependant analysis of the long term losses, but then only for a certain percentage. This is only valid for variable load cases of long term duration. Variable load cases of short term duration are not taken into account for the long term losses, permanent load cases are taken into account for the full 100%. The value of Psi is a user input value. The value depends on the type of loading. The code could be used for guide values.
  In this example Psi is taken as 0,3.

- Number of subintervals:
  The number of subintervals that is defined between two construction stages defines the number of time-nodes which is created between two stages for the time dependent analysis. The TDA solver is a numeric method which is more precise if more time nodes are defined. The more time discretisation, the more precise the results will be, but this will take a longer calculation time. The number of subintervals can be defined between each stage individually. Between two stages with a higher loss to be expected, an increase of the number of subintervals could result in more precision. A number of 10 subintervals is usually a good guide value, but we will leave it at 1 for the general settings.
- Ambient humidity [%]
  This property has an influence on the shrinkage. The higher the ambient moisture, the lower the shrinkage.
  The default value of 70% is kept in this example.
- The local time axis introduces a few extra time nodes on the timeline for which no construction stage was created.
  - Time of casting: the default here is taken is -1 day, because the introduction of an element that has the stiffness of day 0 will cause problems in the solver, because the solver cannot treat elements with zero stiffness. To be more precise, the value of -1 day could be changed by -0,1 day, but not more than 1 decimal is accepted.
  - Time of curing: this is a special treatment that has positive influence on shrinkage.
  - Line support (formwork): This property assumes that the concrete is still in its formwork over the first days, so that after casting the stiffness of the concrete can be built up. Note that during this period, there will be no displacement of the structure at all, also the reaction forces on the formwork cannot be viewed. If that is required the formwork should be modeled by means of temporary supports.

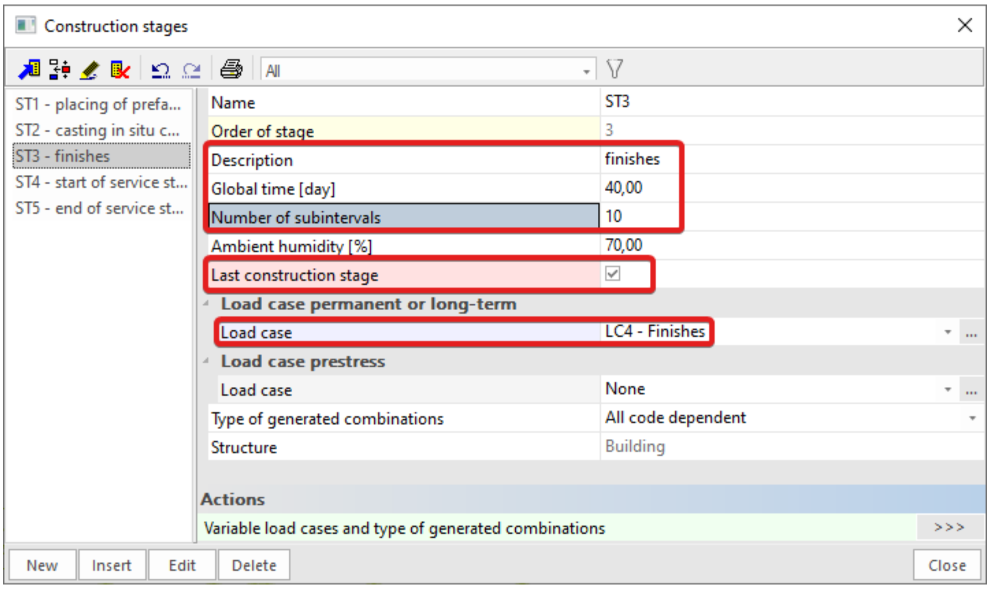
1.2.2. Adding construction stages

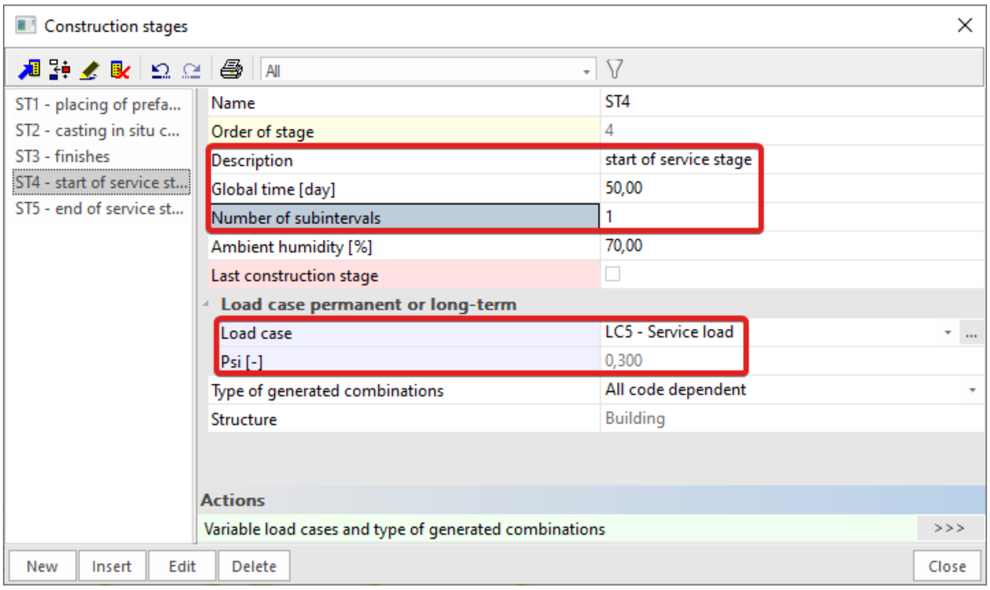
The properties of each construction stage used in this example are shown below. The important properties are marked in red.





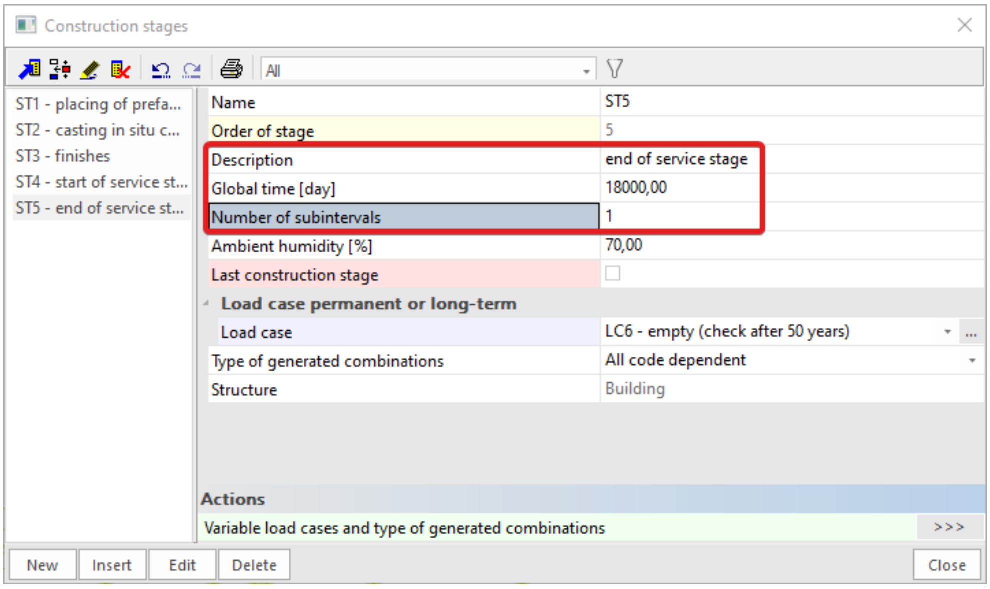
Note that for stage 3 (shown below) is the last construction stage. This means that after this stage, it is allowed to add service loads of type long term variable.





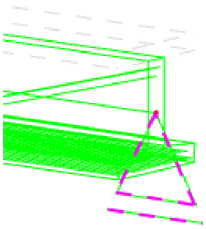
The Psi factor of 0,3 in stage 4 shown above, will only influence the participation of LC5 to the long term losses calculated by TDA solver. In the generated combinations, this load case will be considered without this reduction factor, so this reduction factor is only for the calculation of losses, not for the concrete checks.

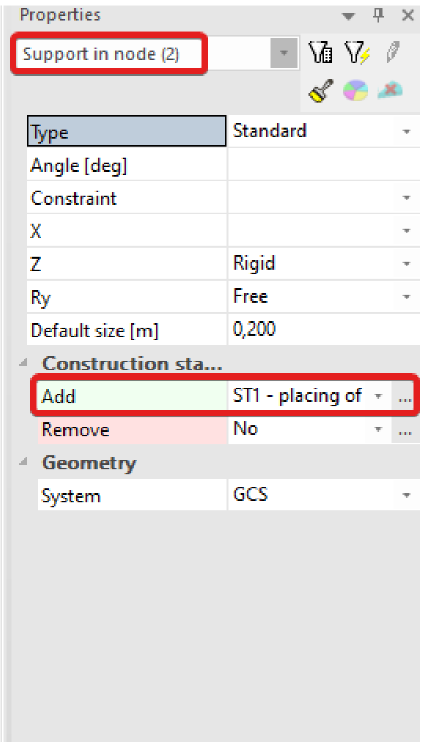
A control stage at 50 years (18000 days) is created to take into account the long term losses.

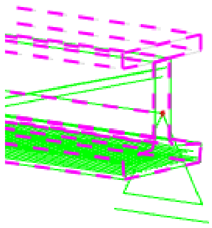


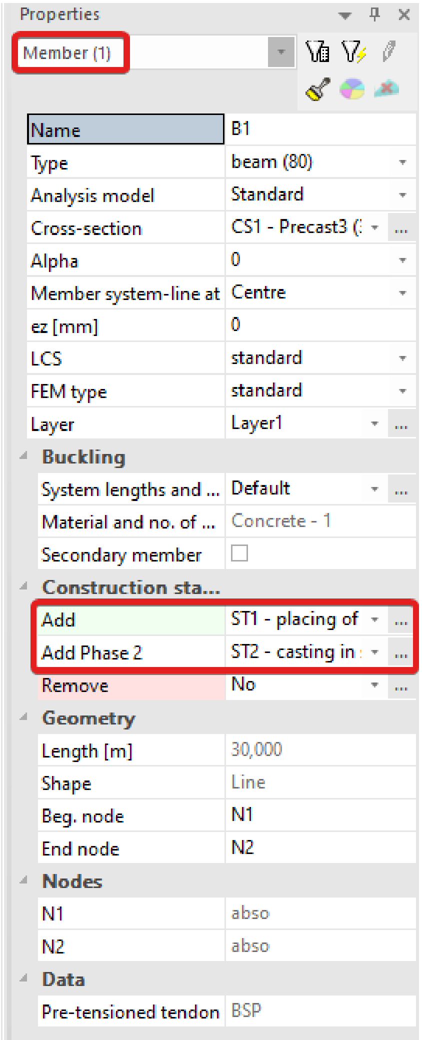
1.2.3. Defining boundary conditions

For the actual definition of the stages, we will add the precast beam and its supports in stage 1.
And the in situ concrete is added in stage 2. We do this in the properties window of both the support and the beam as shown in the two images below.

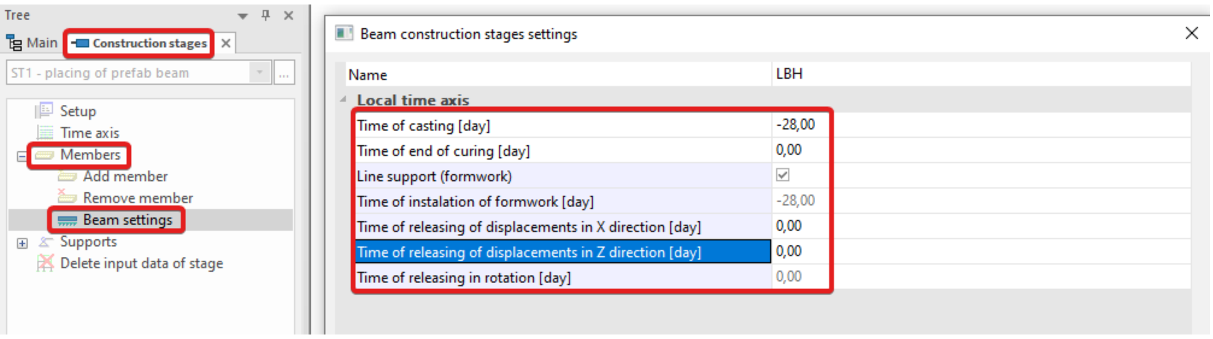








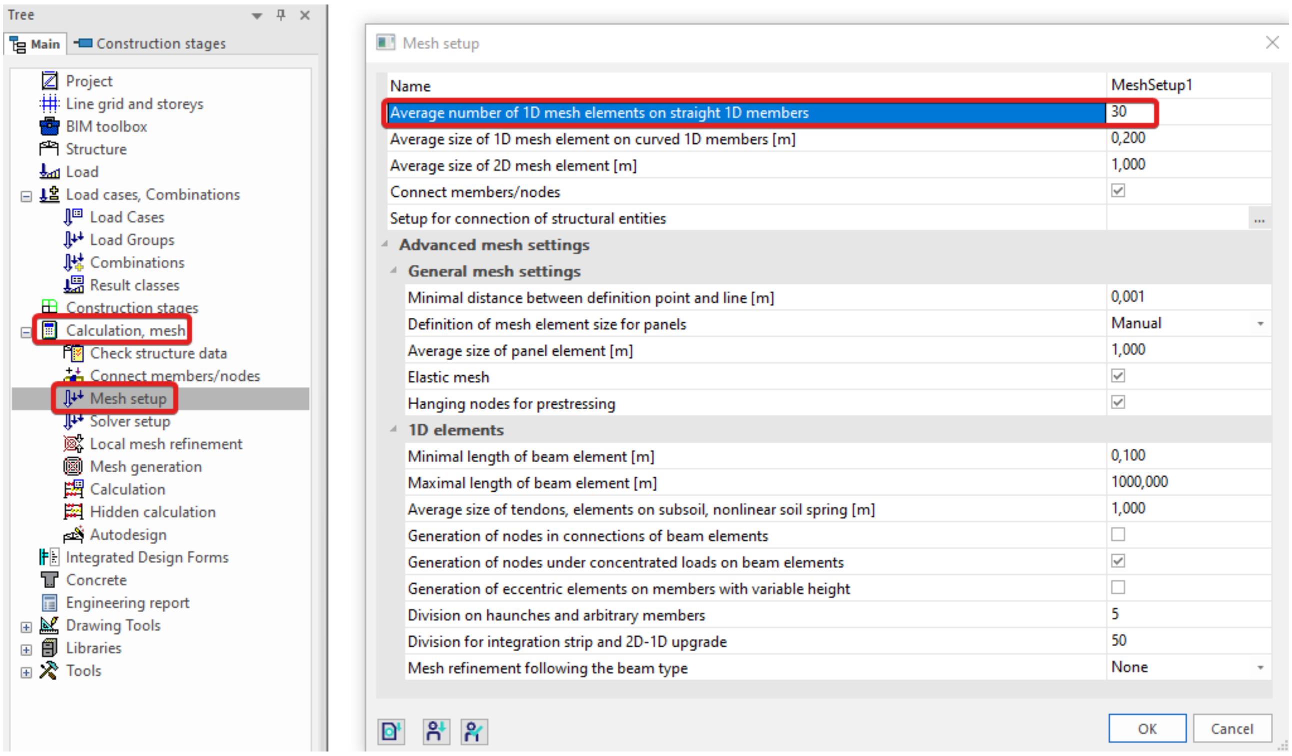
The last point to take care off is the age of the concrete. Since the precast beam which is added in stage 1, is assumed to have full stiffness at the moment of placing, we will define an age of this precast beam of 28 days through local beam settings.



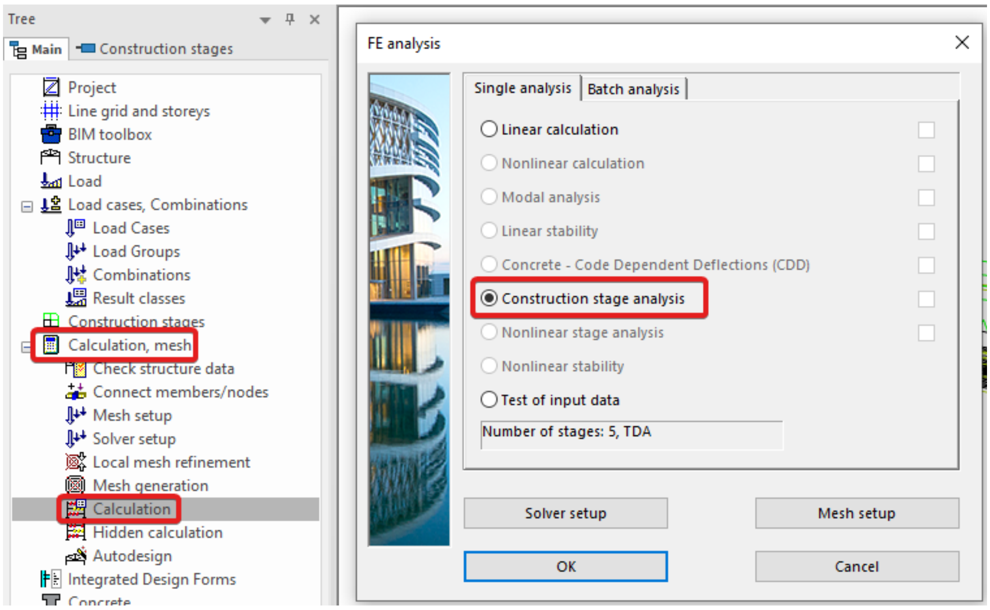
Note that this will not influence the default age of the in situ concrete which is added fresh (age 0) in stage 2.

1.3. Calculation

Before running the TDA analysis, the beam and the patterns are meshed. We will divide the beam into 30 mesh elements (on average) and the strands will have an average mesh length of 1m.



It is important to run the Construction Stage Analysis, and not just the linear calculation. (If the linear calculation is run, everything defined in construction stages will be ignored.)



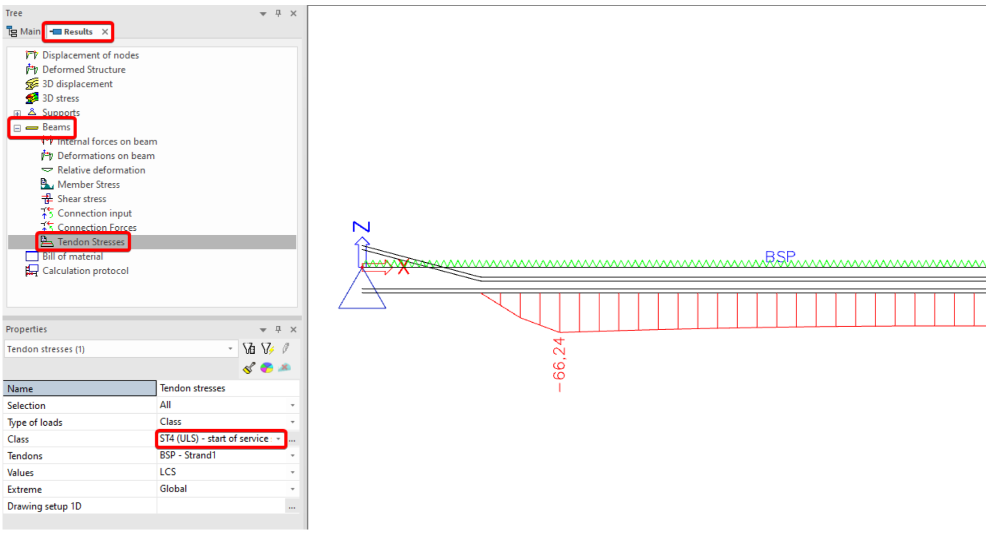
1.4. Results

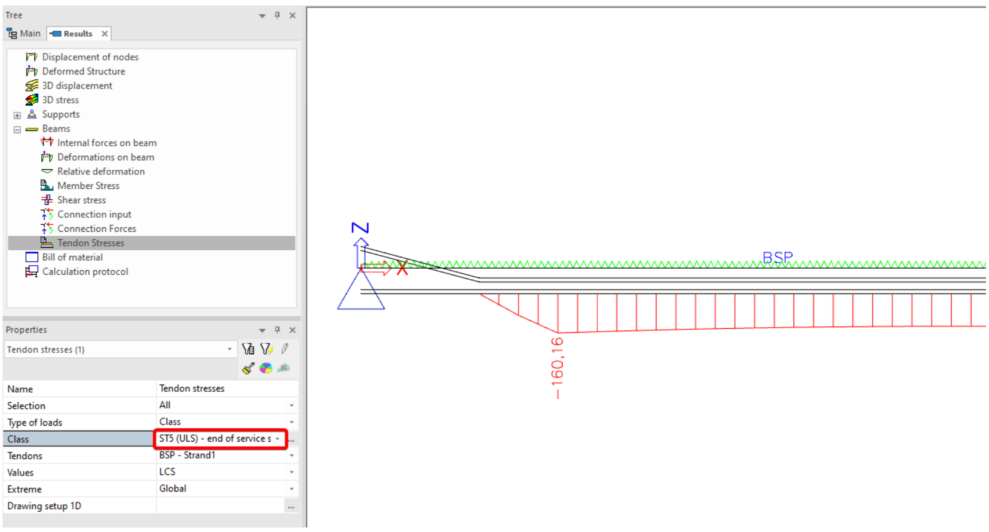
In the results menu, we will have a look at the tendon stresses, the deformations and the internal forces in the different stages.

1.4.1. Tendon stresses

The tendon stresses in the results menu show us the losses that are calculated after time dependant analysis. The immediate losses were already calculated before running the time dependent analysis. The initial stress (-immediate losses) results in the stress after transfer (SAT).The LED losses are the losses caused by elastic deformation of the concrete. These losses could also be calculated by a linear construction stage analysis. The LCS losses are the losses caused by creep, shrinkage and long term relaxation. These LCS losses are only calculated by TDA analysis.

Below the LCS losses at the beginning and at the end of the service stage are shown:



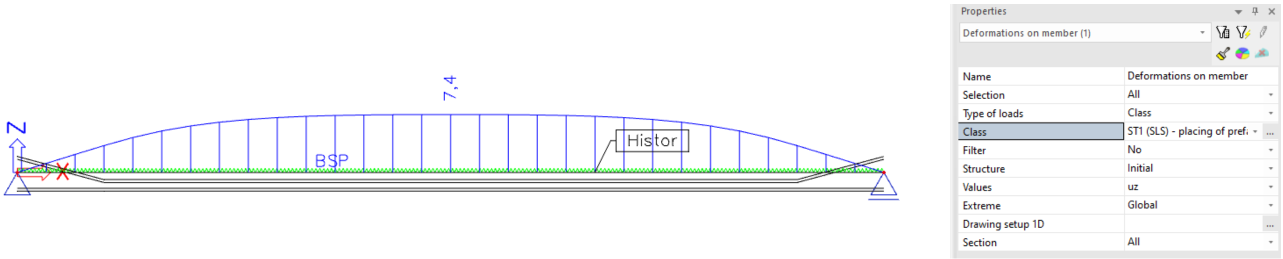


There is clearly an increase in LCS losses between the beginning and the end of the service stage.

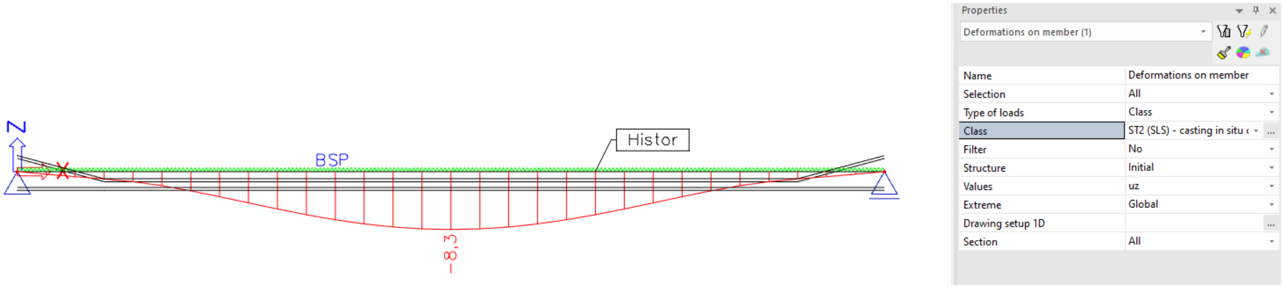
1.4.2. Deformations

The deformation of the pretensioned prefab beam in the different construction stage is also given in the results menu. The results for the deformation in het 5 construction stages are shown below.

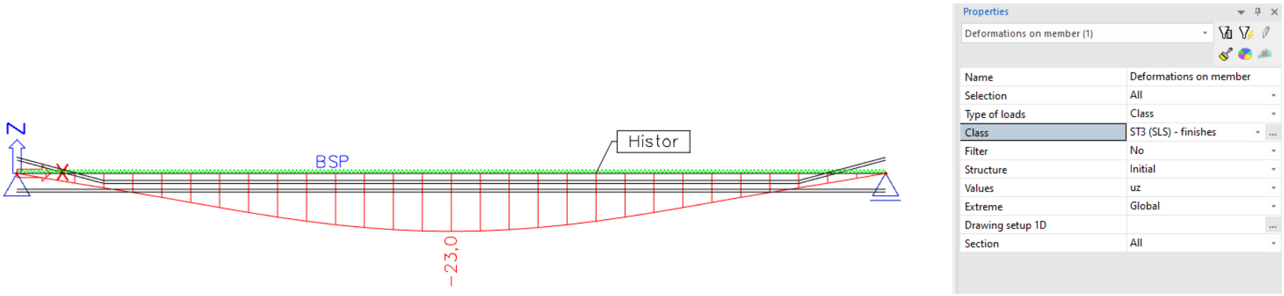
Stage 1:



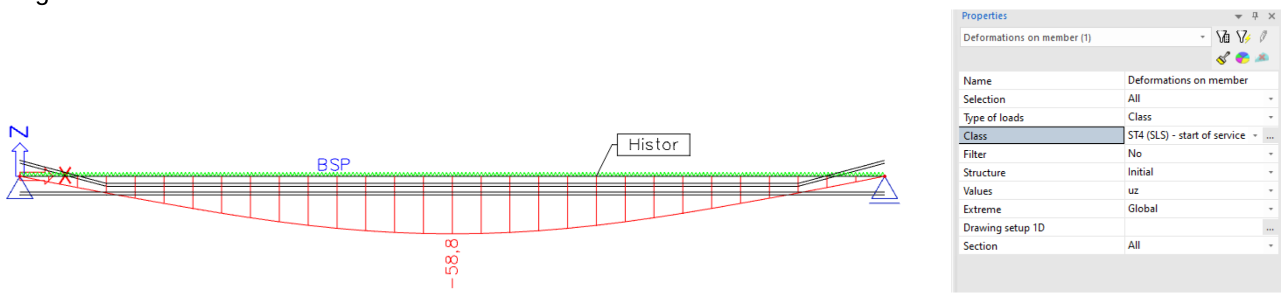
Stage 2:



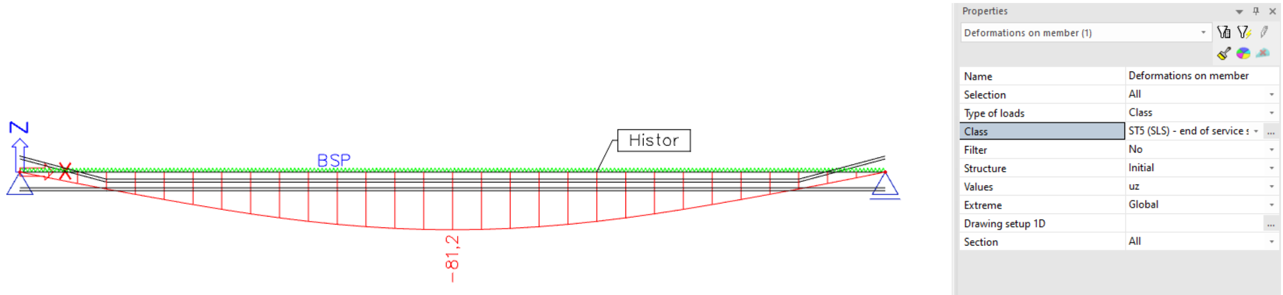
Stage 3:



Stage 4:

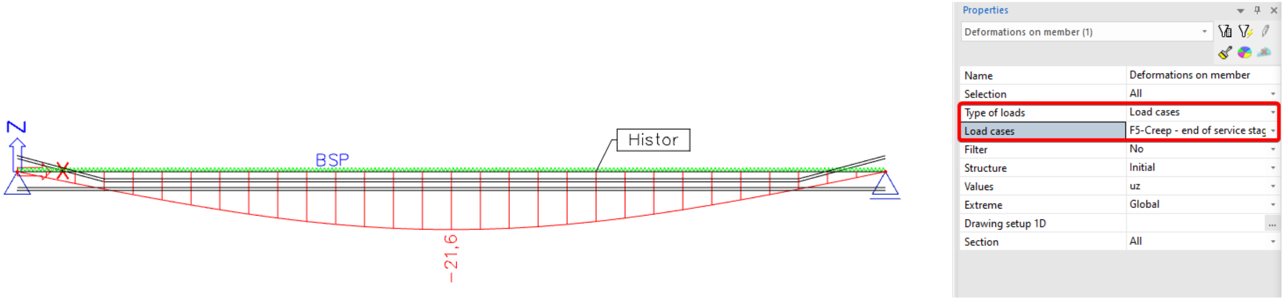


Stage 5:



There is a difference in deformation between stage 4 and stage 5.The increase in deformation is caused by creep, because the load case assigned to stage 5 was an empty dummy load case. The effect of creep (and more in general all long term losses) is stored by the TDA analysis in a special load case F-creep. The deformation for the load case F5-Creep contains the extra deformation:

F5-Creep:

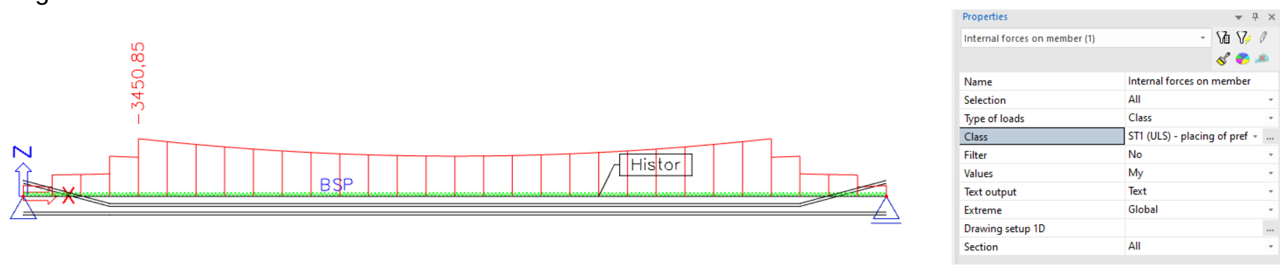


Note that also a relative deformation could be given for checking. This relative deformation could be checked manually or by means of an imbedded excel check.

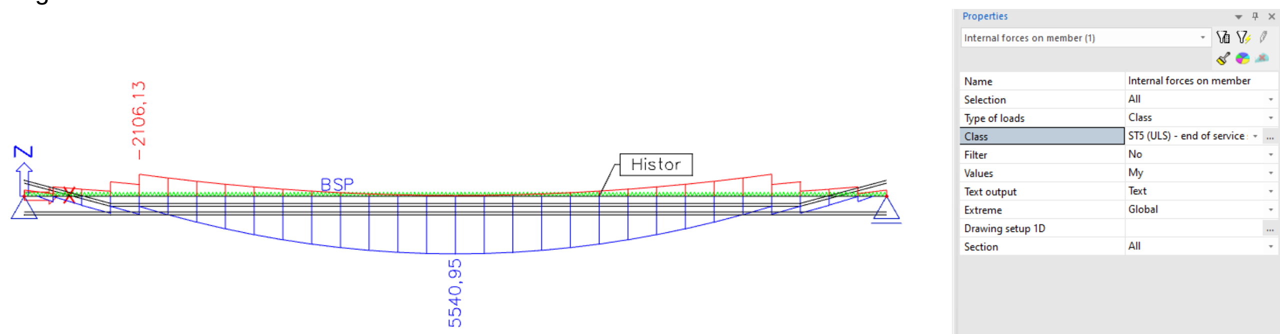
1.4.3. Internal forces

The (envelope) moment lines for stage 1 and stage 5 are shown below.

Stage 1:



Stage 5:

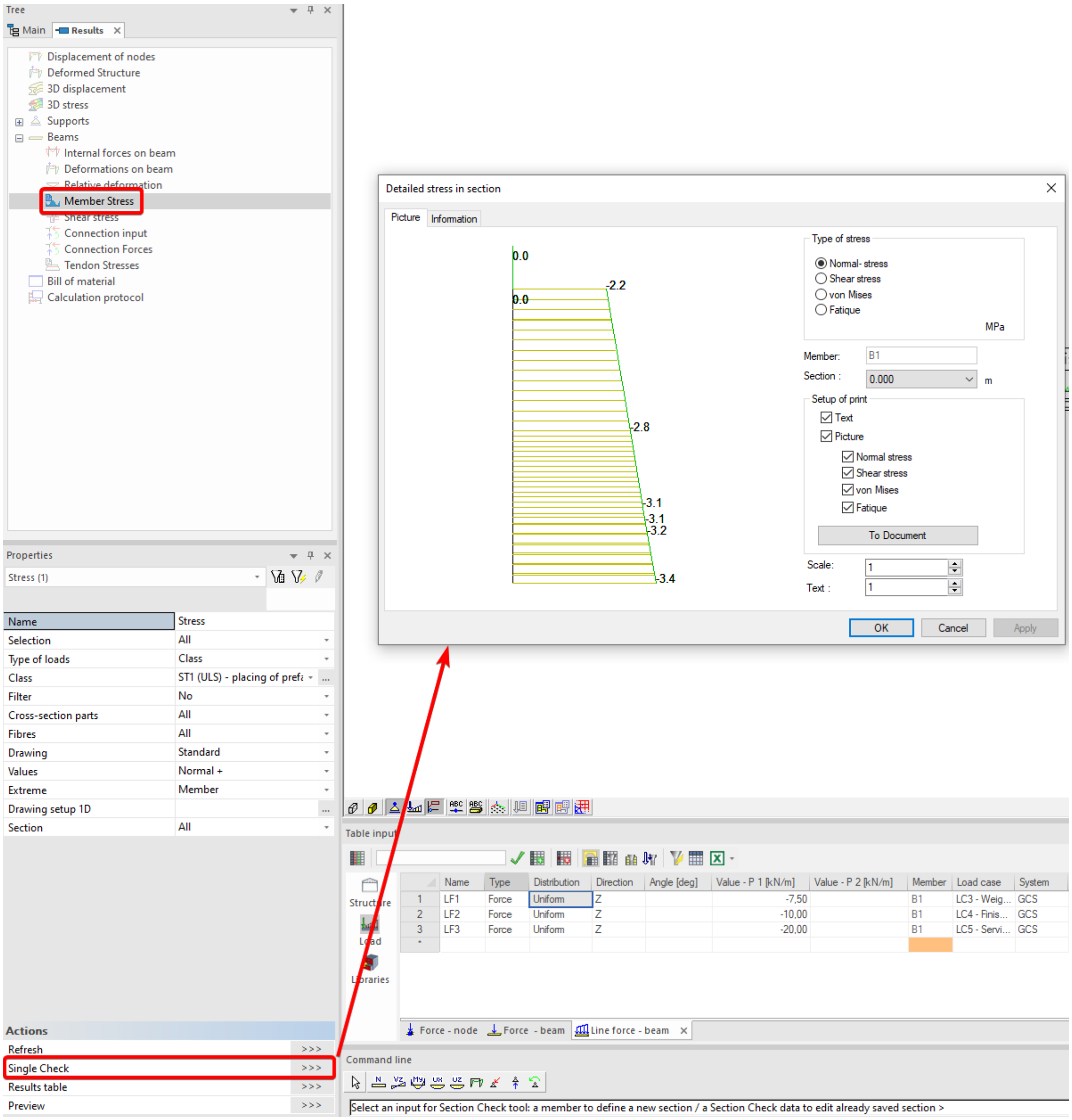


The maximal resistance moment can be viewed in the concrete menu. This means a capacity check of the internal forces will be possible in the concrete menu.

1.4.4. Stresses

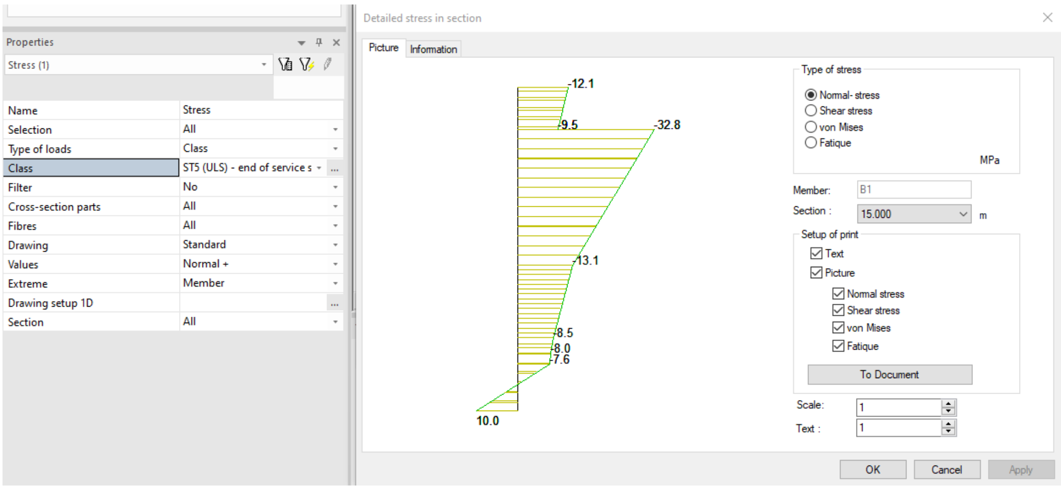
The stresses can be viewed over the height of the cross-section. This is interesting to check if there would be tension on the top fibres in the end sections of the beam.

The stress in the starting section is shown below for stage 1.



There seems to be no tension on the top fibre. This is prevented by the debonding and the bending of some strands. If there would be tension on the top fibre, then more strands should be bend or debonded.

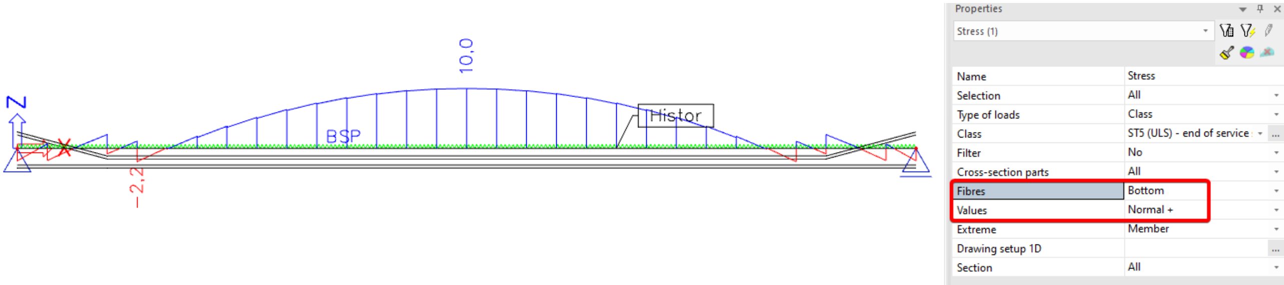
The stress in the midsection in stage 5 is shown below. Notice the jump in stress between the two phases of the cross-section.



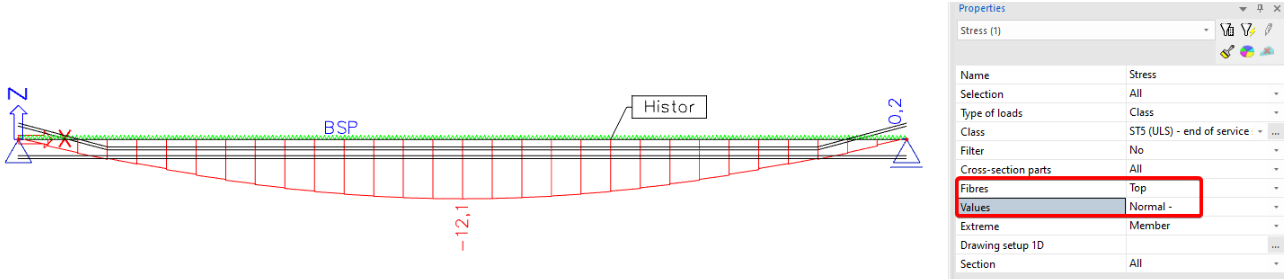
It is also possible to show the stress over the length of the beam. Hereby can be chosen to show the stress on the bottom or the top fibre.

The stress in stage 5 on respectively the bottom and top fibre are shown below.

Tension stress at the bottom fibre in stage 5:

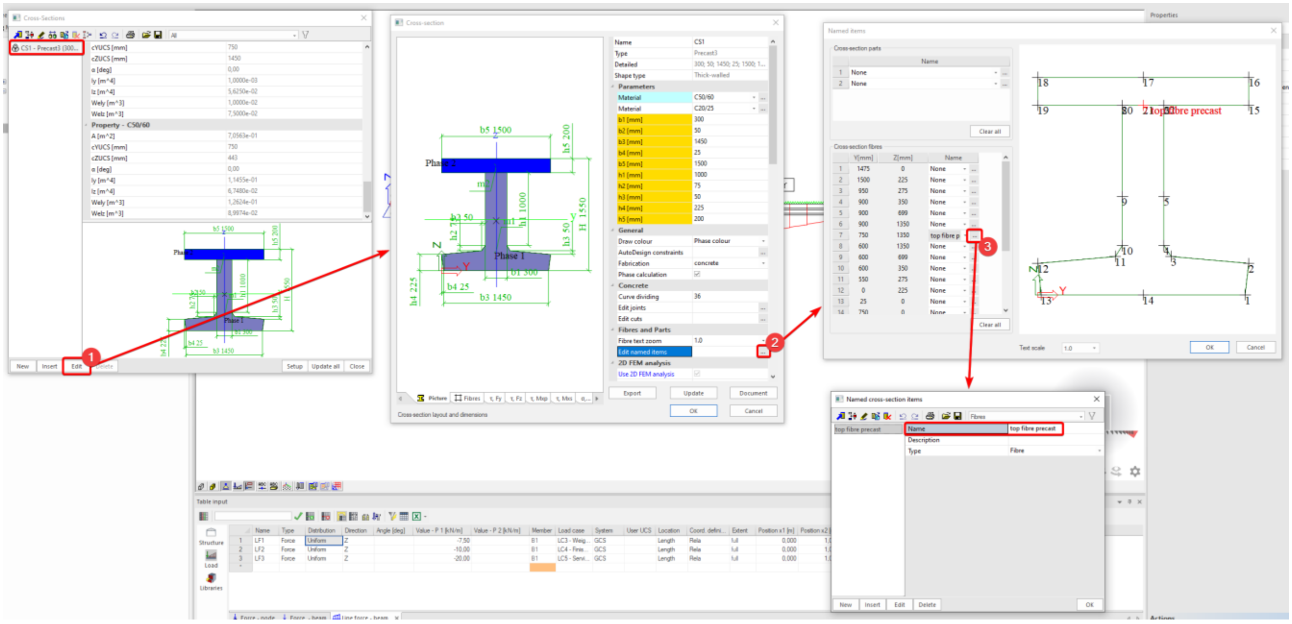


Compression stress at the top fibre in stage 5:

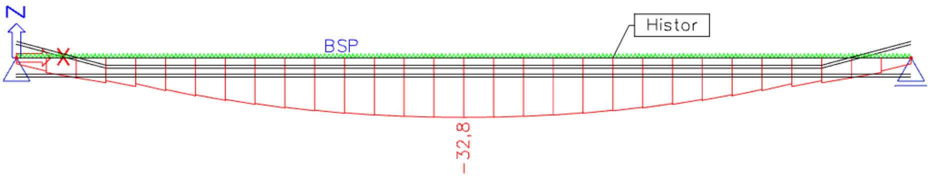


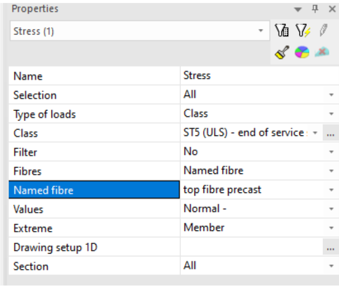
Note that the top fibre corresponds to the top of the second phase of the cross-section.

To show the stress on the top fibre of the precast beam (first phase of cross-section), you could manually predefine a named fibre. This is done in the properties of the cross-section.



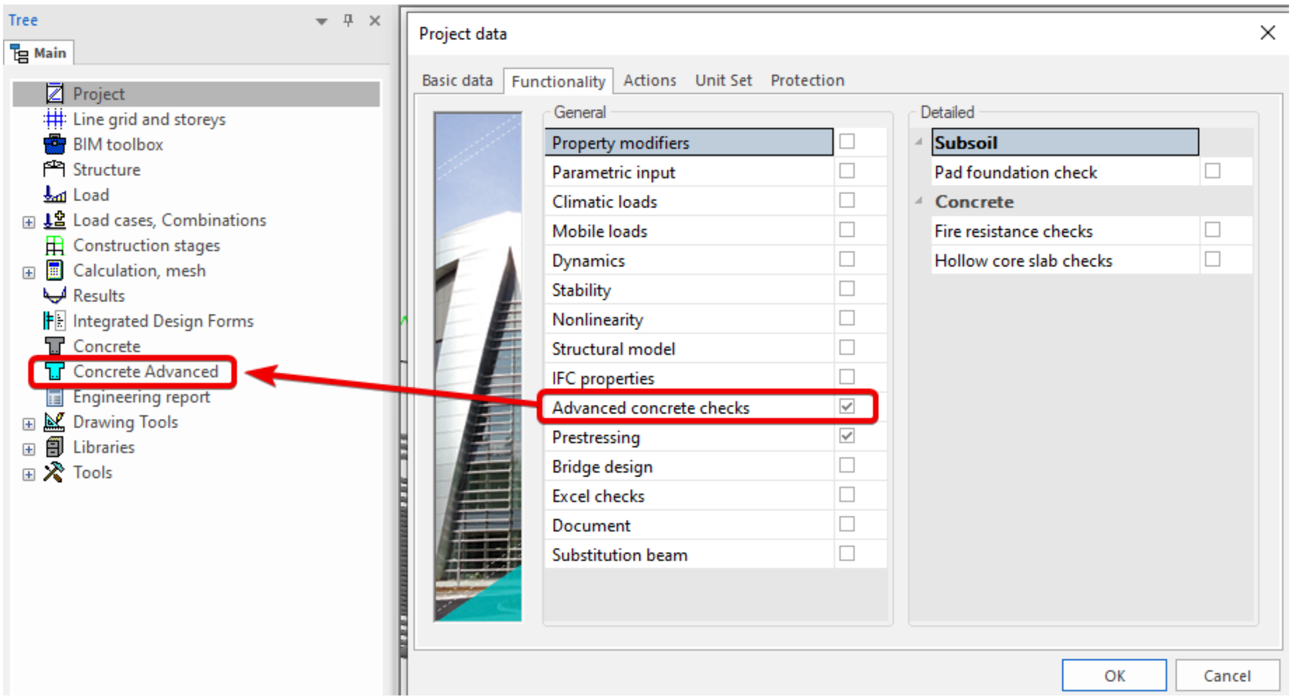
Afterwards stresses can be displayed over this named fibre. The stress in stage 5 on this fibre of the precast beam is displayed below.


Z
X
BSP
Histor
-32,8


Properties
Stress (1)
Name | Stress
Selection | All
Type of loads | Class
Class | ST5 (ULS) - end of service
Filter | No
Fibres | Named fibre
Named fibre | top fibre precast
Values | Normal -
Extreme | Member
Drawing setup 1D
Section | All

1.5. Checks in the concrete menu

All of the following checks were executed out in the old concrete menu. To be able to consult this menu, the “Advanced concrete check” must be ticked in the functionalities.



A lot of checks are possible in the concrete menu. The user should be aware of which checks he wants to run before going to the concrete menu. If the user would miss a check, for instance check of fatigue, embedded excel checks could be a work-around.

In chapter 1.4 Results a check for tension stress at the top fibre right after installation was already discussed. A check for the maximum tendon stress is usually done manually at the input of the initial stress in the strands.

In this chapter a deflection check, a check for the maximum concrete compression stress, a capacity check on the resistance moment, and a design of additional passive reinforcement (if required) will be illustrated.

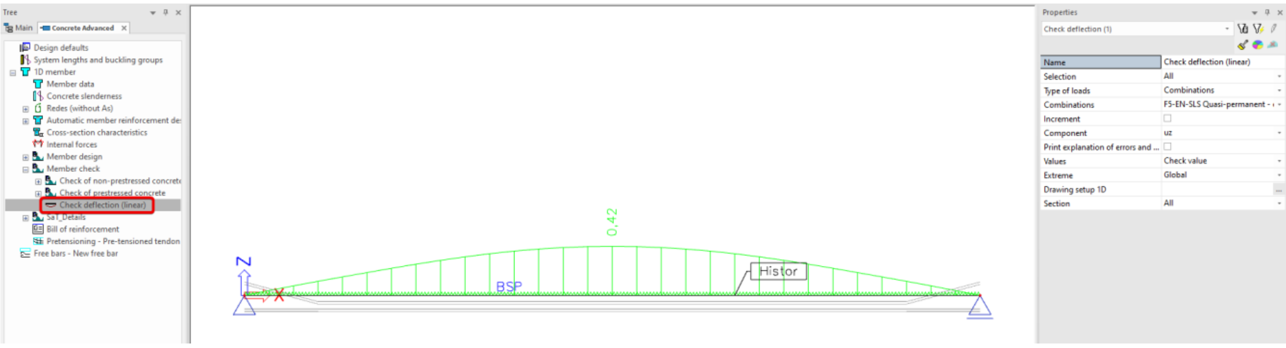
The design of vertical shear reinforcement Ass and the design of shear reinforcement As in the construction joint will also be treated. The check for decompression will be illustrated as well.

Note: it is advised to create a document template in which all the checks required for the user are assembled.

1.5.1. Deflection check

This check is comparable with the results for deformation that can be checked manually in the results menu. However in the concrete menu a difference is made between the total deflection and the harmful deflection (which is the deflection caused by creep).

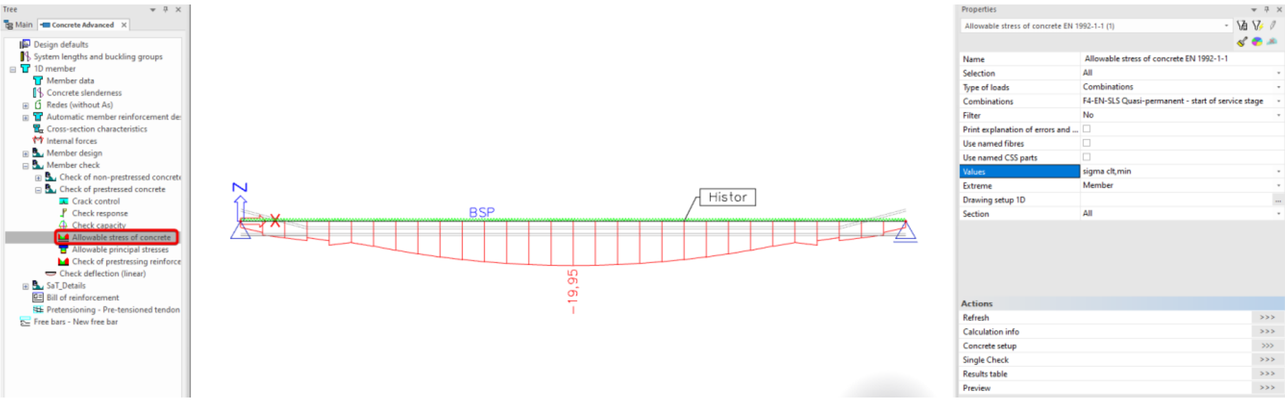
The results for the deflection check in the final stage are shown below.



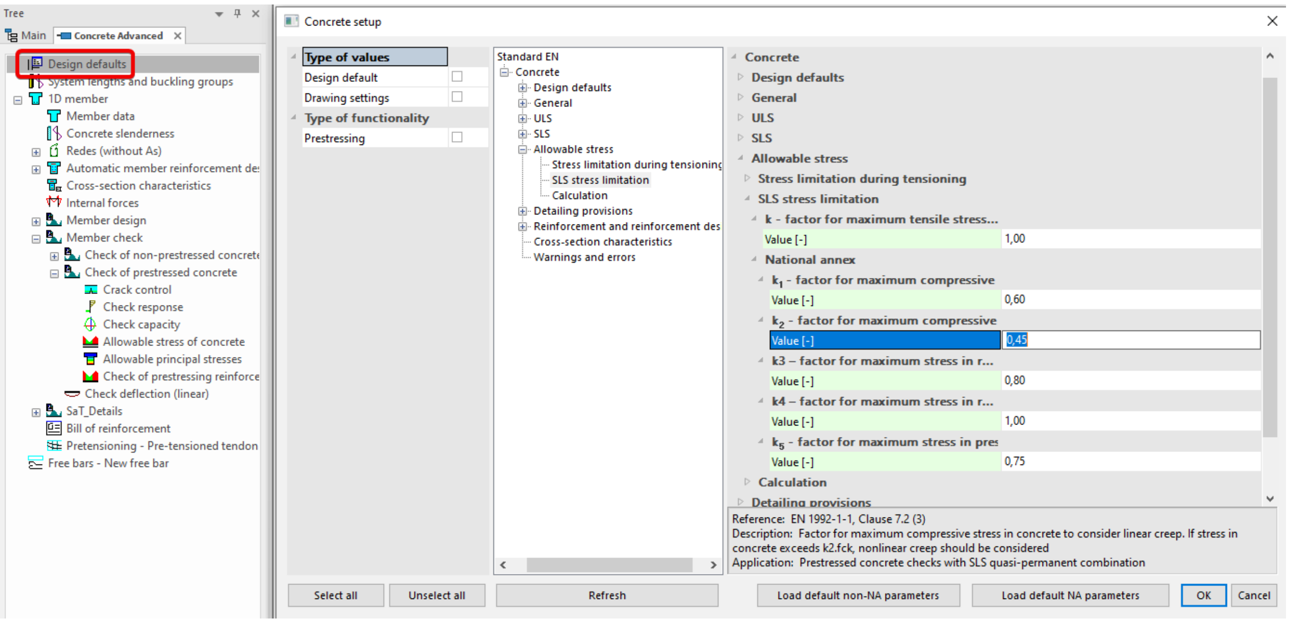
1.5.2. Check of allowable concrete stress in compression

The maximum concrete compression stress is to be expected on the top fibre of the precast beam in ST4, where all loading is applied and the LCS losses of the prestresses are not yet complete.

The results for the SLS quasi perm combination in stage 4 are shown below.

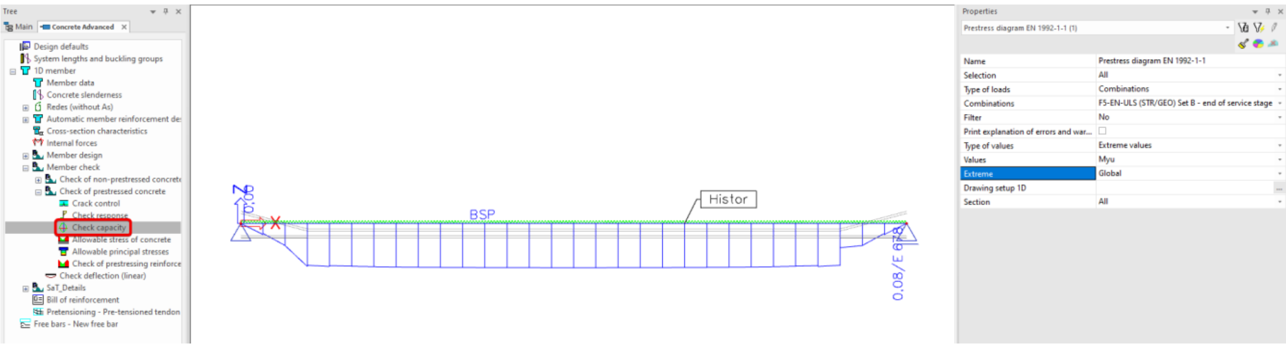


The maximum allowed concrete compression stress (sigma clt,min in Scia Engineer) is calculated according to art 7.2 (3) of EN 1992-1-1. The k2 factor used in that formula can be found in the concrete setup.



1.5.3. Capacity check

The moment of resistance is shown below.



We will not consider the end zones here. It is due to numerical effects, check of tension stress at top fibre in first stage was already performed and this was ok.

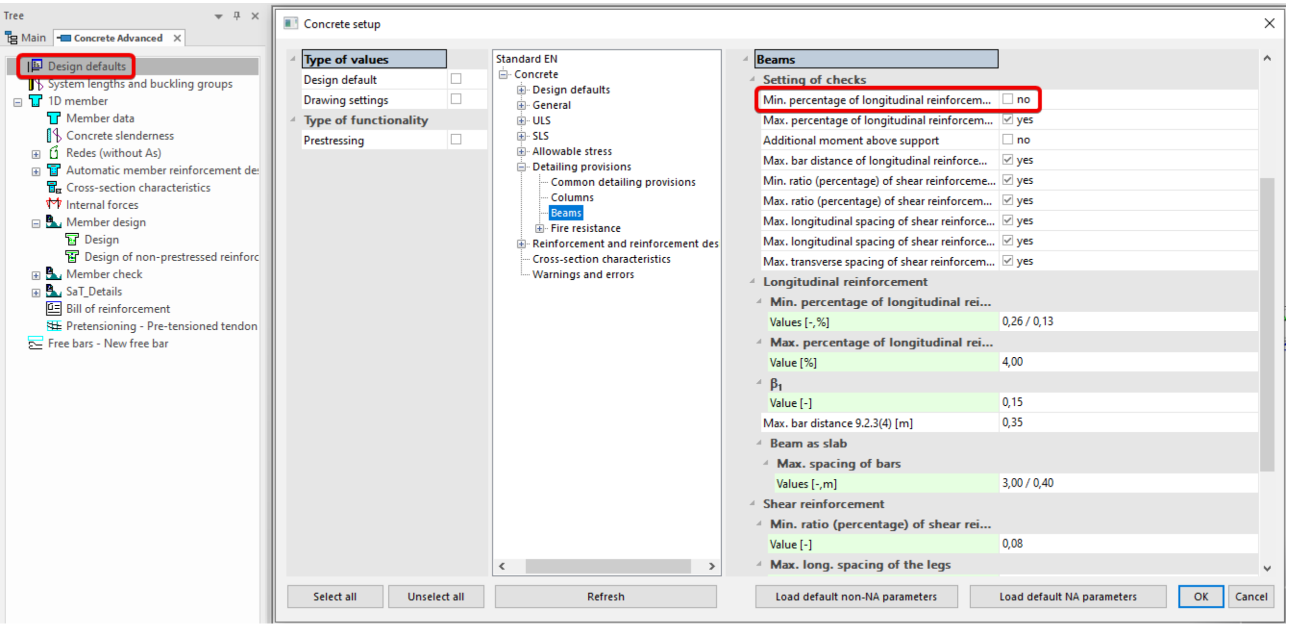
The unity check in the midsection of the beam indicates that the moment My is greater than the moment of resistance.

There are two possible solutions. The user could:

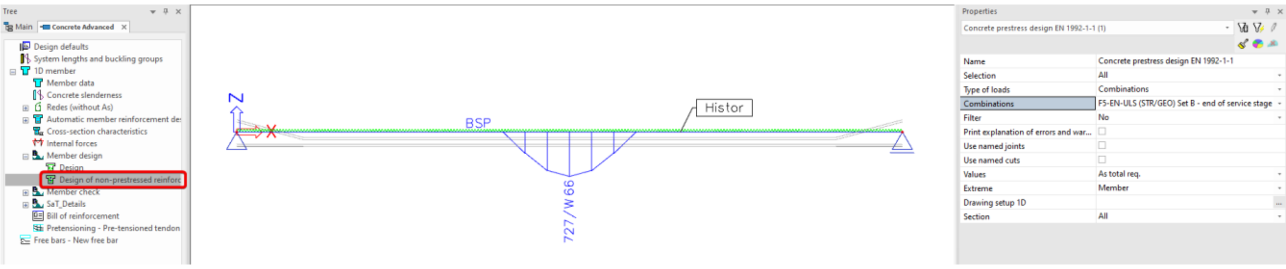
1. increase the prestress reinforcement by adding more strands.
2. design the additional passive reinforcement that is required in this case.

1.5.4. Design of passive reinforcement As

Note: to see how much passive reinforcement As is really required to increase the capacity, it is advised first to tick off the detailing provision of minimum required reinforcement in the concrete setup.

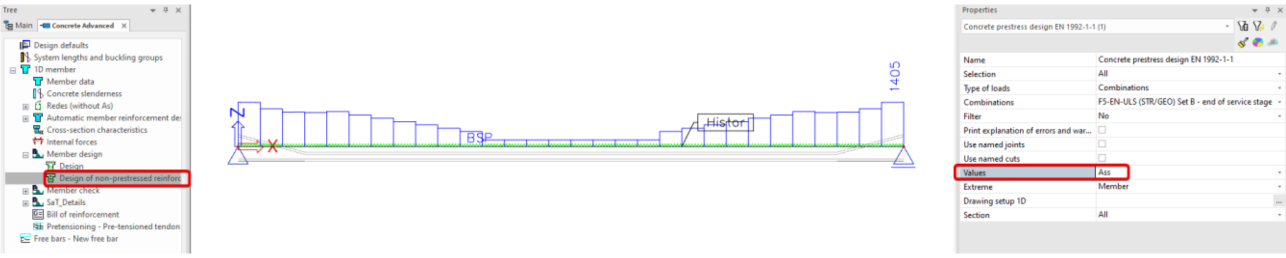


The additional required passive reinforcement for the ULS combination in stage 5 in shown below.



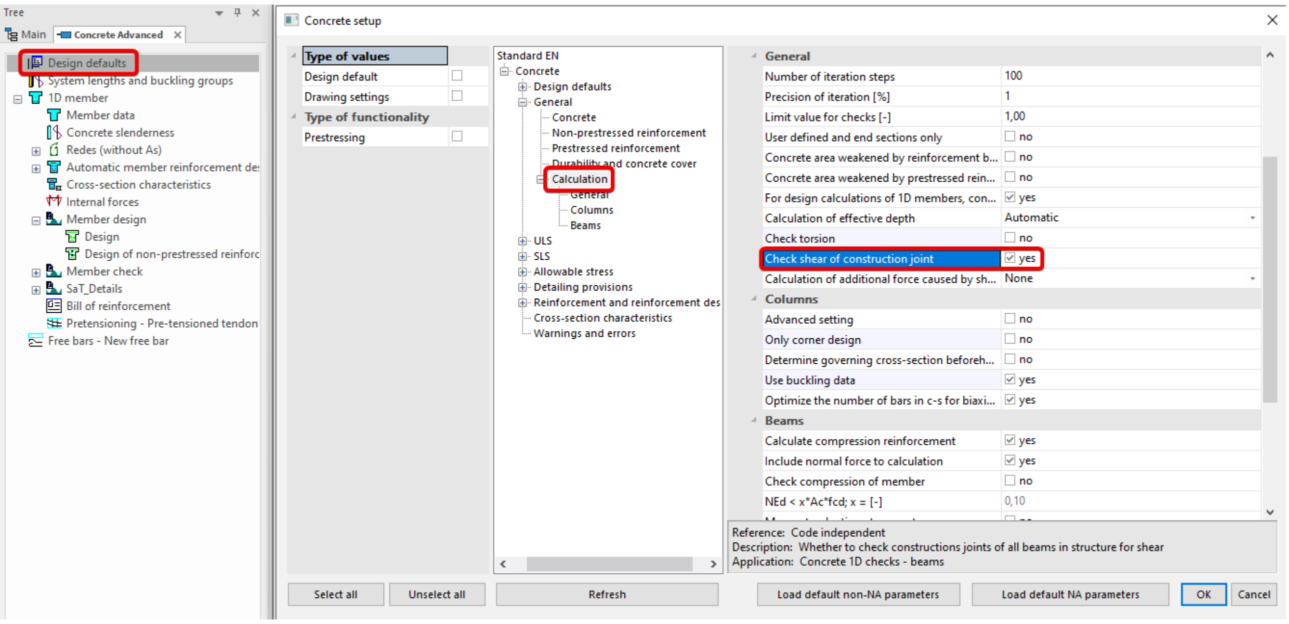
1.5.5. Design of vertical shear reinforcement Ass

It is also possible to design the required shear reinforcement Ass. This is shown below.

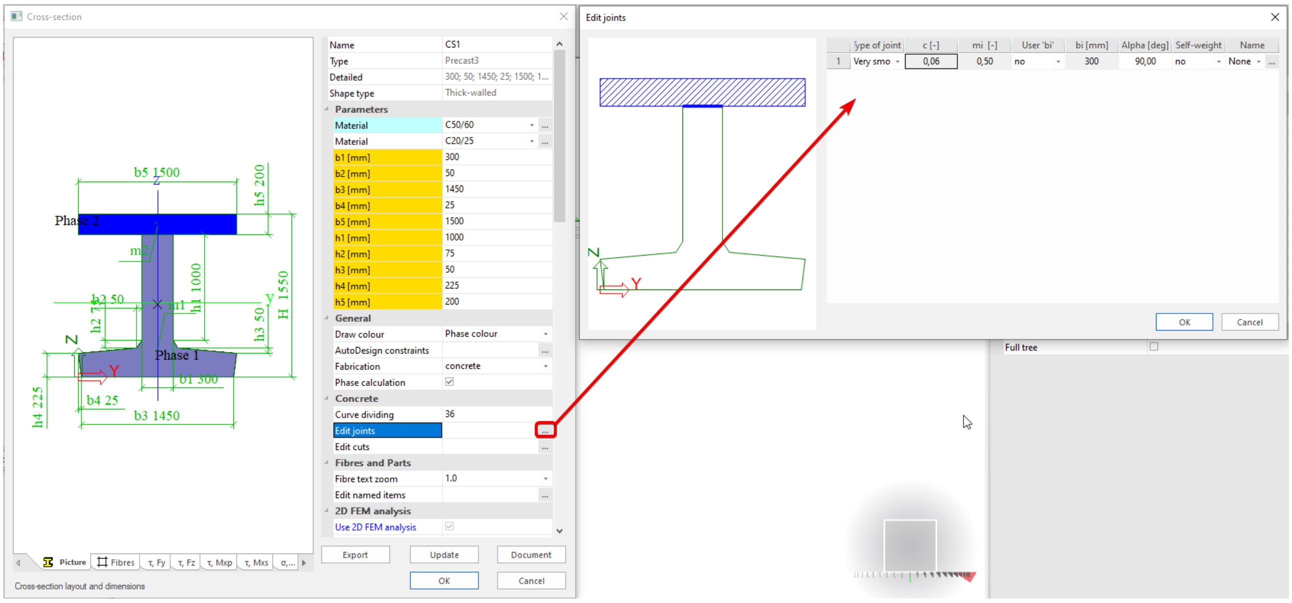


1.5.6. Design of shear reinforcement in construction joint Asj

If the user wants to preform a check of shear in construction joint, this first needs to be ticked on in the concrete setup.

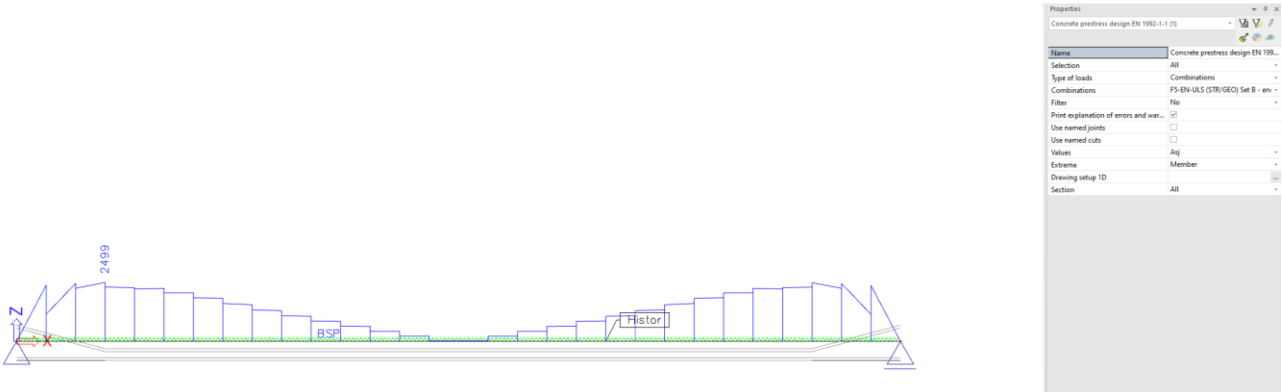


The roughness of the joint can be defined in the section properties.



By default the roughness of the joint will be defined as very smooth, which is the most conservative.

The required shear reinforcement Asj is shown below.



1. Concrete prestress design EN 1992-1-1

Linear calculation, Extreme : Member
Selection : All
Combinations : F5-EN-ULS (STR/GEO) Set B
The shifted moments (tensile forces caused by shear and torsion) are not taken into account

Shear reinforcement in joints for selected members

Member	d_x [m]	Case	Joint	V_{Ed} [kN]	v_{Edi} [MPa]	v_{Rdi} [MPa]	A_s [mm²/m]	$A_{s,req}$ [mm²/m]	Reinf.[no.]
B1	3,000	F5-EN-ULS (STR/GEO) Set B/1	1	1323,87	1,87	0,06	0	2499	2x8,0-40

Explanations of symbols	
V_{Ed}	Shear force
v_{Edi}	Design value of the shear stress in the interface

Explanations of symbols	
v_{Rdi}	Design shear resistance at the interface
A_s	Area of reinforcement crossing the interface

Explanations of symbols	
$A_{s,req}$	Required area of reinforcement crossing the interface

The theoretical background behind the calculation of Asj is added below:

In EN 4 types of joints are defined (very smooth, smooth, rough and indented). These types define the parameters c and μ.

The formula for the resistance of the concrete without shear reinforcement is shown below:

$$v_{Rdi} = c\, f_{ctd} + \mu\, \sigma_n + \rho\, f_{yd}\, (\mu \sin \alpha + \cos \alpha) \leq 0{,}5\, \nu\, f_{cd} \quad (6.25)$$

where:

c, μ	are factors depend on roughness of joint
f_{ctd}	is design tensile concrete strength
σ_N	is normal stress preloaded construction joint
	$\sigma_N > 0$ for compression
	$\sigma_N < 0$ for tension
	σ_N is limited by $0,6 \cdot f_{ctd}$
ρ	is reinforcement ratio
$\rho = A_{sj}/A_i$	where:
	A_{sj} area of reinforcement used for shear in construction joint
	A_i area of concrete joint Ai=bi * 1current meter
α	is angle of reinforcement efficient for shear in construction joint, defined by user in CSS dialog, limited by values 45°-90°

The formula for the shear force in the construction joint is shown below:

$$v_{Edi} = \beta V_{Ed} / (z\, b_i) \quad (6.24)$$

Where:

β	is the ratio of the longitudinal force in the new concrete area and the total longitudinal force either in the compression or tension zone, both calculated for the section considered. β is calculated from response with ULS precondition (6.1(2) EN1992-1-1)
V_{Ed}	is the transverse shear force
z	is the lever arm of composite section
b_i	is the width of construction joint

These formulas are taken from the EN 1992-1-1 chapter 6.2.5.

Shear reinforcement Asj will be required if $v_{Edi} > v_{Rdi}$.

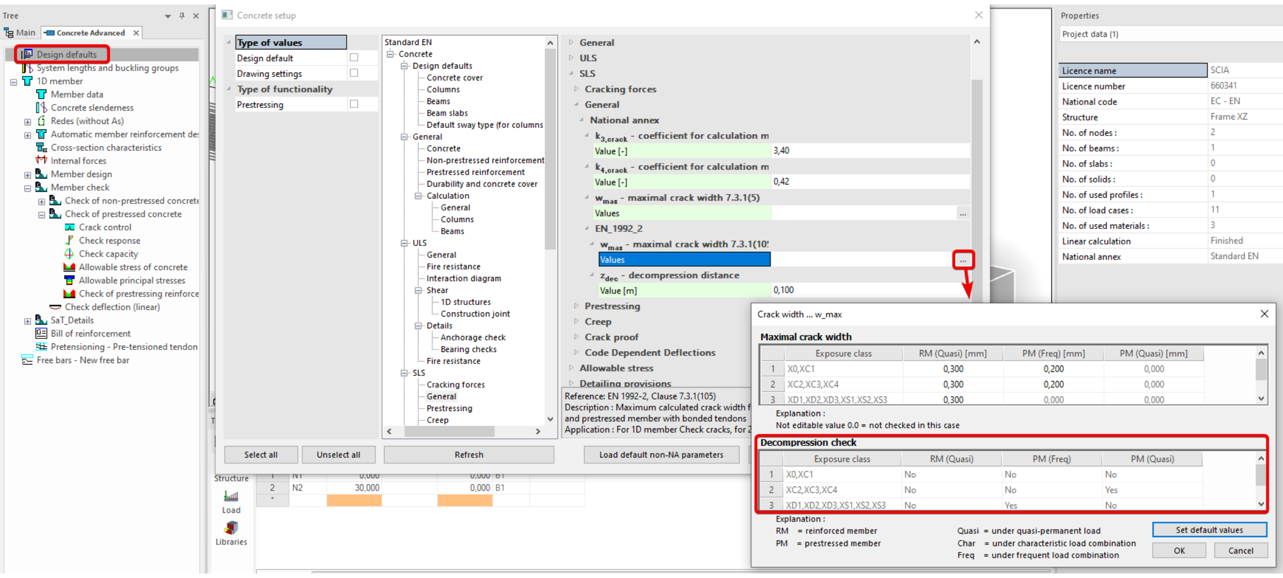
The formula for the required shear reinforcement Asj is shown below:

$$A_{sj,req} = \frac{A_i \cdot (v_{Edi} - c \cdot f_{ctd} - \mu \cdot \sigma_N)}{f_{yd} \cdot (\mu \cdot \sin\alpha + \cos\alpha)}$$

1.5.7. Decompression check

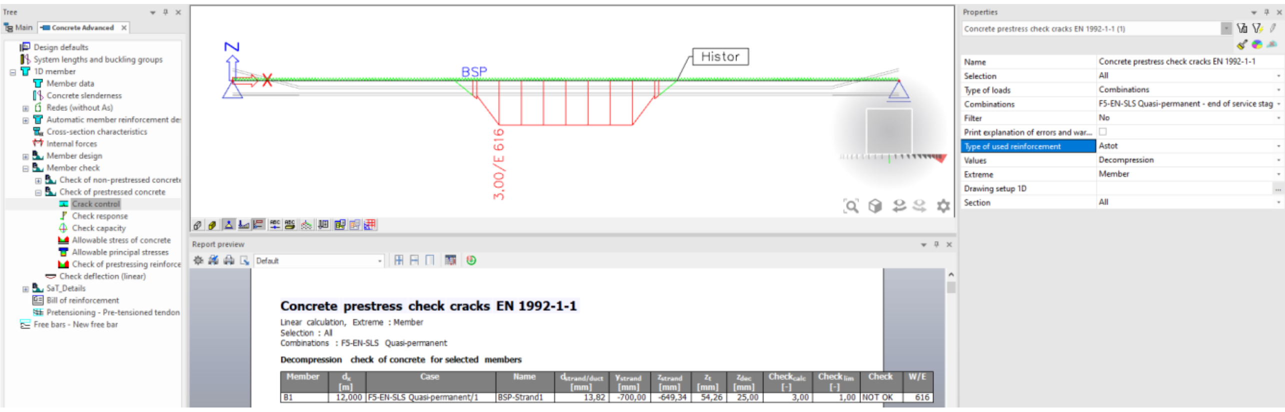
This SLS check is described in art 7.3.1(5) from EN 1992-1-1.

Decompression check only needs to be done if part of the pretensioned cross-section is in tension for the considered type of SLS combination.

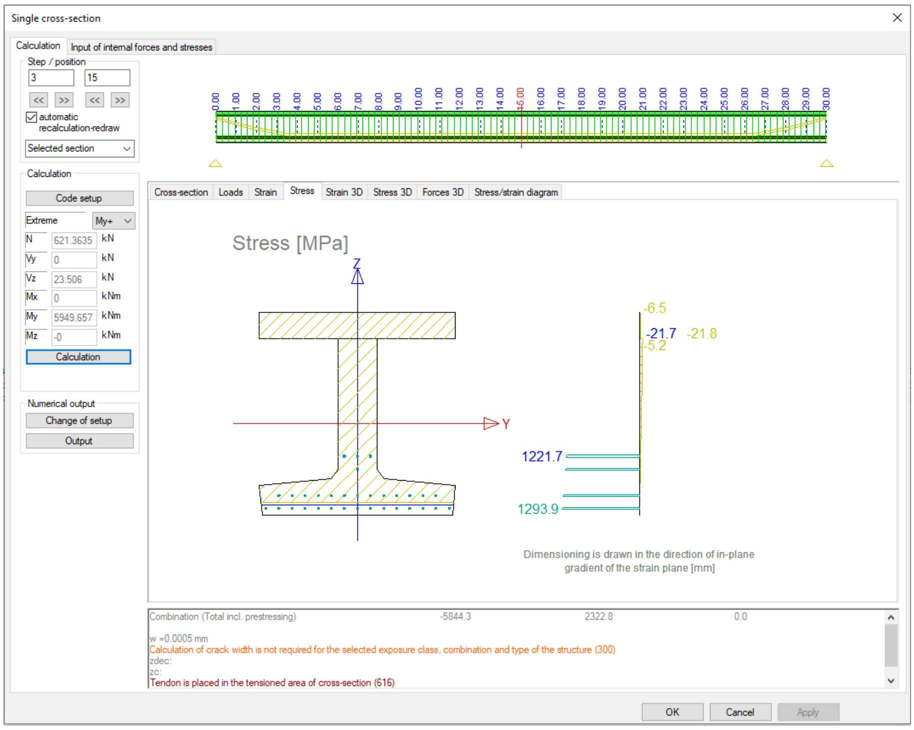


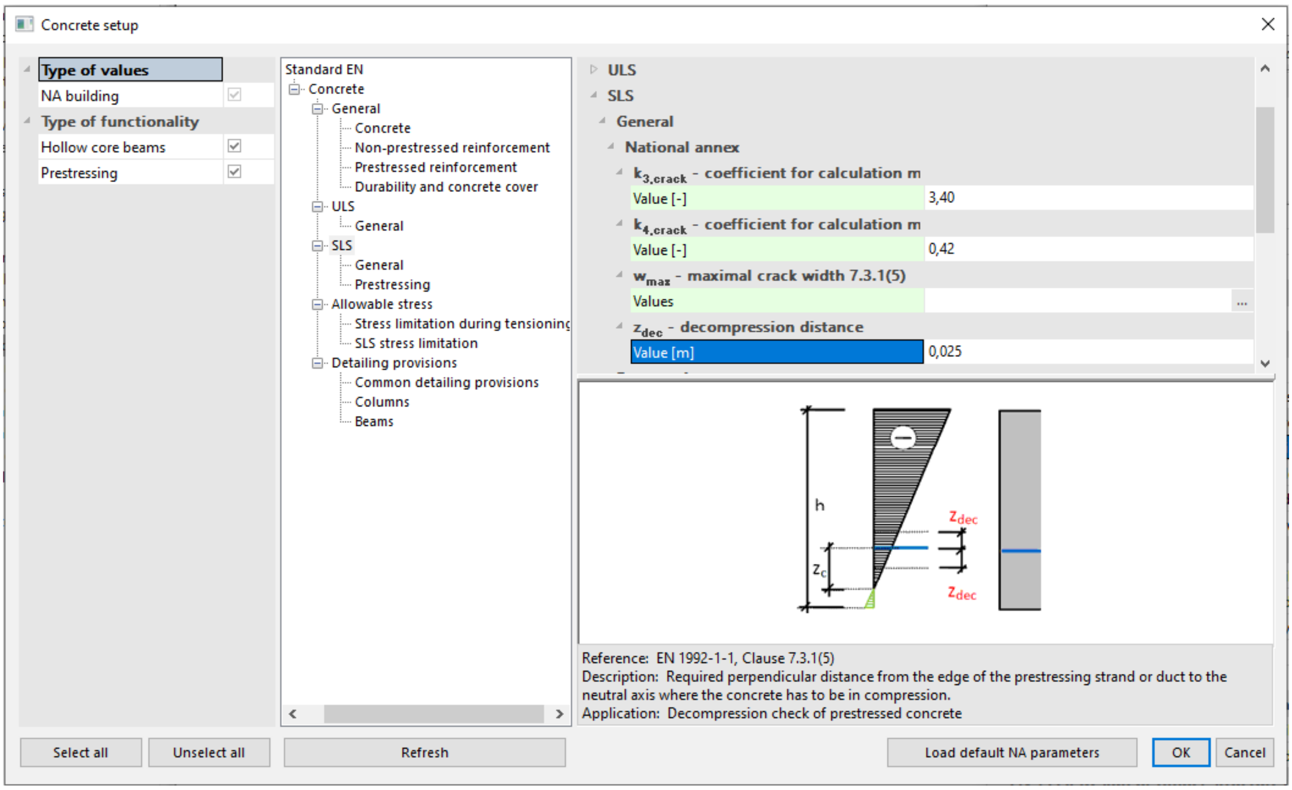
In this example we are in default environmental class XC3.
This means we will need to check for the SLS quasi-permanent combination.

Note that if we would make the check for the combination F5-EN-SLS Quasi, no tension would be found on the bottom fibre, so no decompression check is necessary (->warning 298).
Therefore the service load is increased from 20kN/m to 50kN/m to achieve results shown below:



The check is not okay because the bottom line of the strands is placed in the tensile zone of the concrete.


Single cross-section
Calculation
Input of internal forces and stresses
Stress [MPa]
Dimensioning is drawn in the direction of in-plane gradient of the strain plane [mm]
Calculation of crack width is not required for the selected exposure class, combination and type of the structure (300)
Tendon is placed in the tensioned area of cross-section (616)
OK
Cancel
Apply


Concrete setup
Type of values
NA building
Type of functionality
Hollow core beams
Prestressing
Standard EN
Concrete
General
Non-prestressed reinforcement
Prestressed reinforcement
Durability and concrete cover
ULS
SLS
Allowable stress
Stress limitation during tensioning
SLS stress limitation
Detailing provisions
Common detailing provisions
Columns
Beams
National annex
k3,crack - coefficient for calculation m
Value [-] 3,40
k4,crack - coefficient for calculation m
Value [-] 0,42
wmax - maximal crack width 7.3.1(5)
zdec - decompression distance
Value [m] 0,025
Reference: EN 1992-1-1, Clause 7.3.1(5)
Description: Required perpendicular distance from the edge of the prestressing strand or duct to the neutral axis where the concrete has to be in compression.
Application: Decompression check of prestressed concrete
Select all
Unselect all
Refresh
Load default NA parameters
OK
Cancel

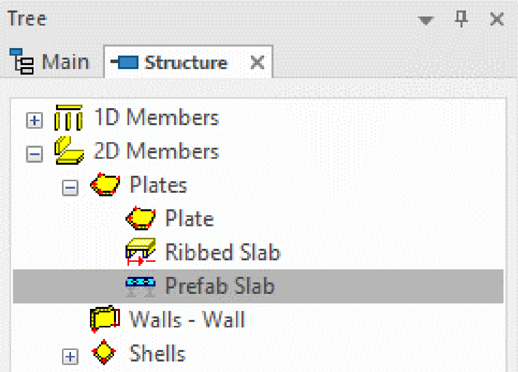
Chapter 2: 2D/1D upgrade

The purpose of this example will be to illustrate the 2D->1D upgrade functionality. This functionality allows to build up the model with a 2D element that can be used for analysis of load distribution. After the calculation of the internal forces a 1D element model can be exported out of this 2D element model. This 1D element model will contain calculated internal forces from the 2D element model and can be used to make the design checks.
For the design calculation of a 1D element model, reference is made to the previous chapter.

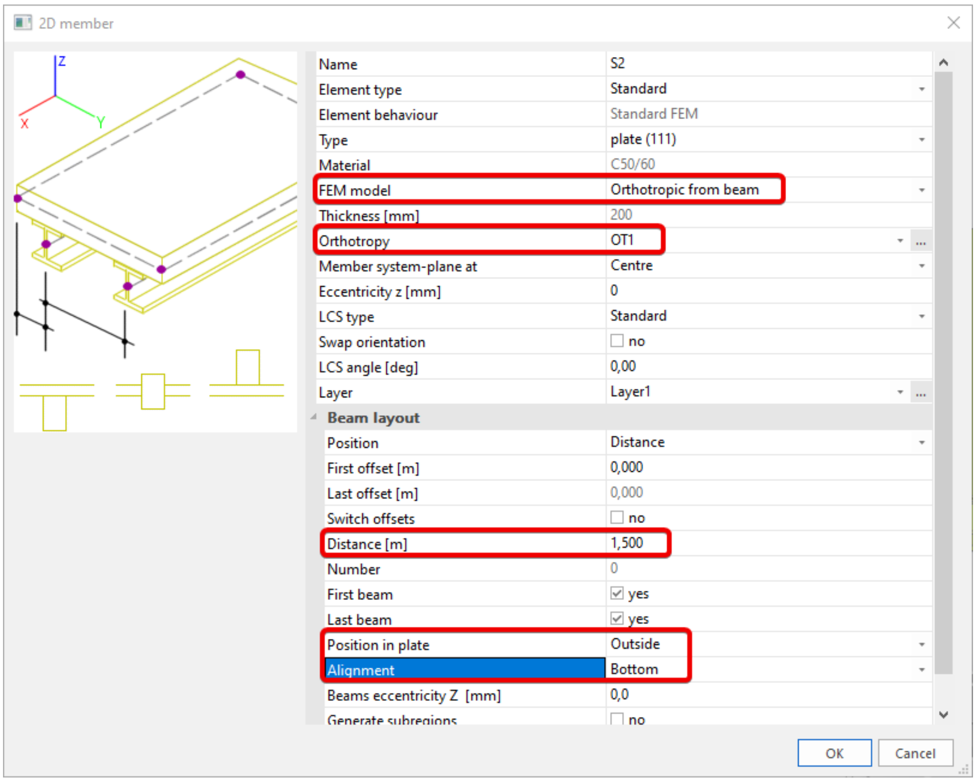
2.1. Modelling of a prefab slab model

Open a new project with a general XYZ structure (or plate XY for precast edition).

In the structure menu the Prefab Slab can be found under 2D member.

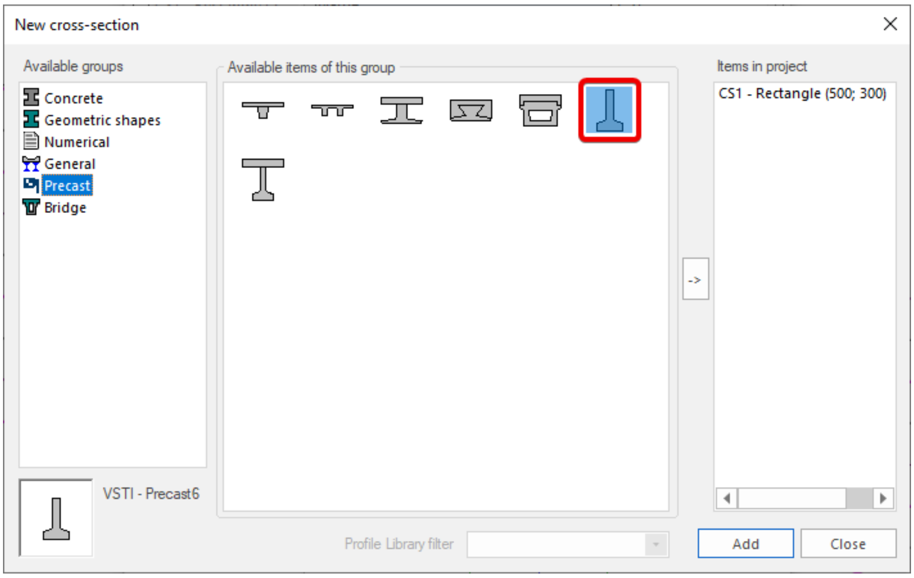


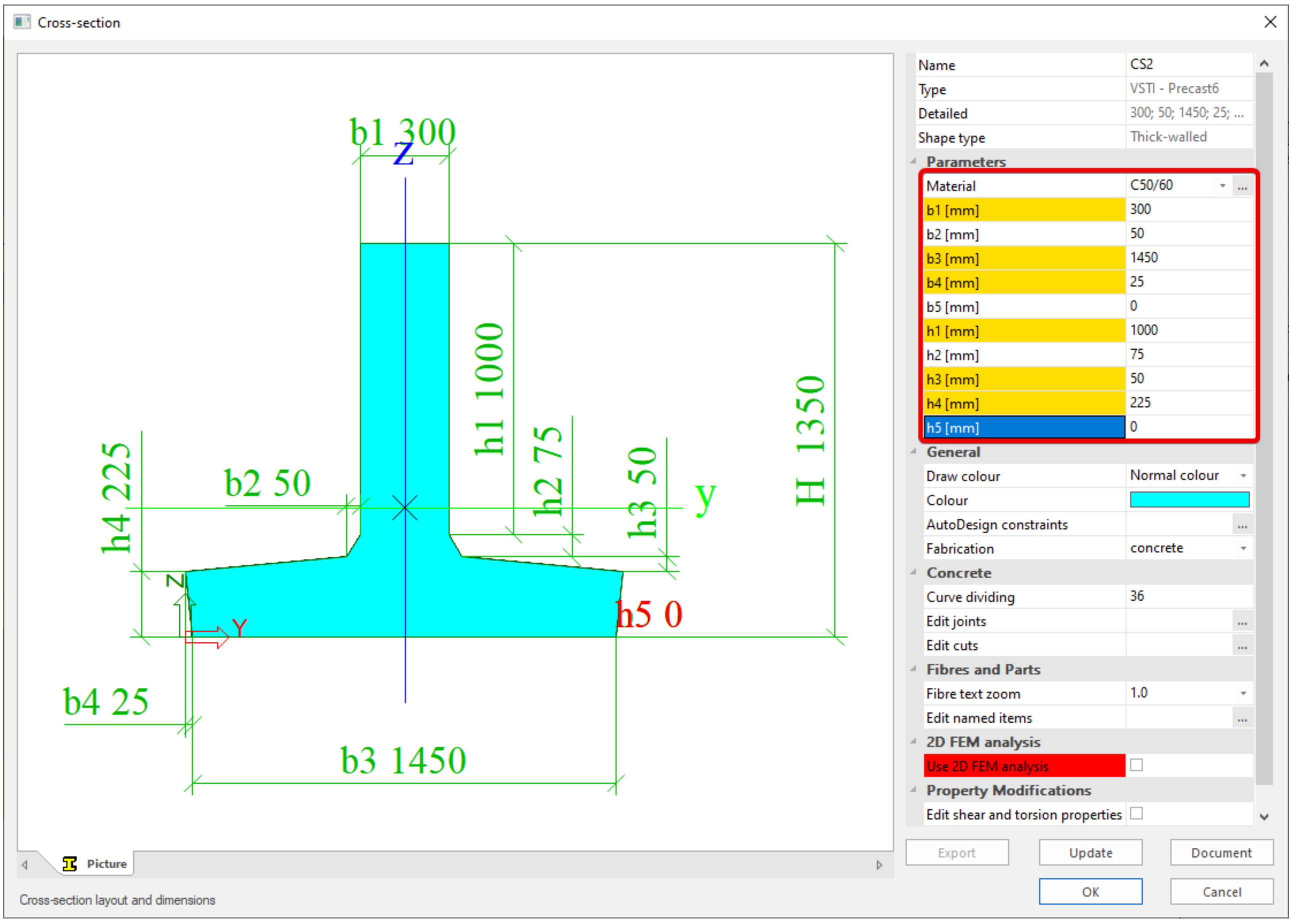
The prefab slab will be considered as a 'normal' plate in this 2D element model.
The beam layout that can be defined in the prefab slab properties will only be important in a later stage for the 1D export. This means that the beams that are added in this way will have no influence at all on the stiffness or the self-weight of the plate.
In this example beams are added every 1,5m.

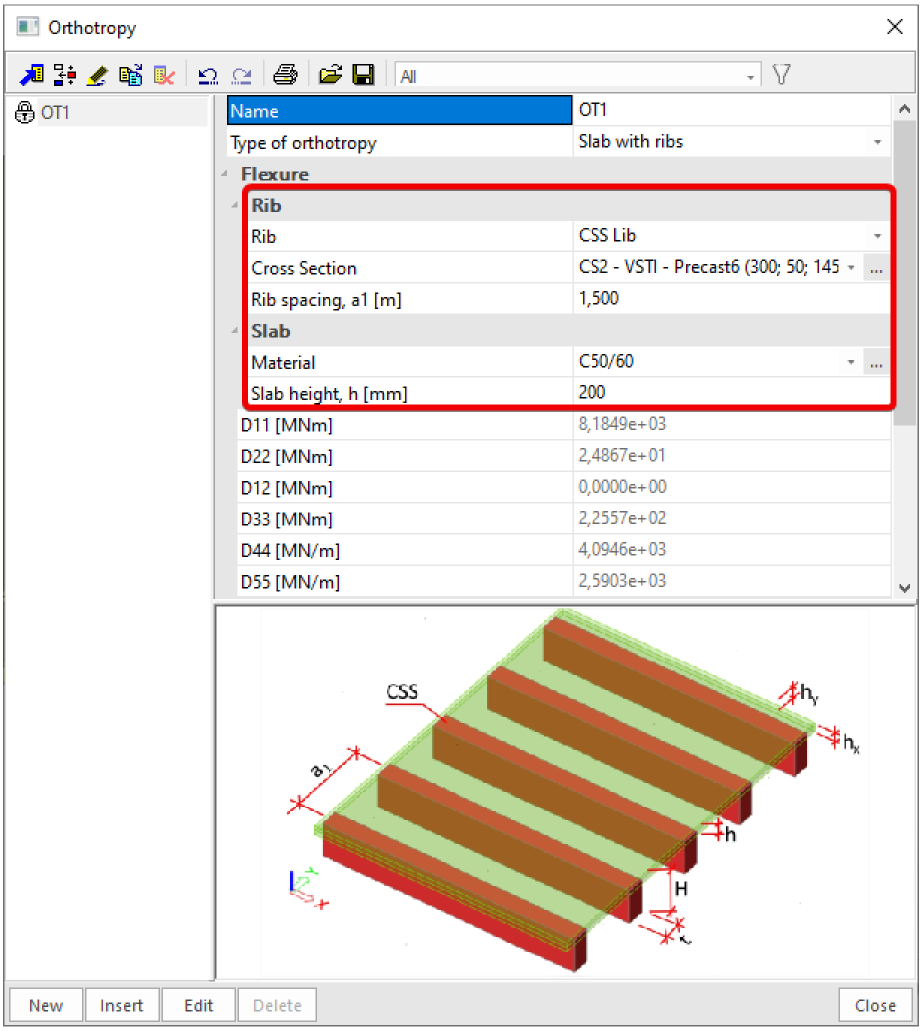


To assign a correct stiffness to the plate, which is of course important for the load distribution, the FEM model of the plate will be set to orthotropic. As type of orthotropy the OT type 'slab with ribs' could be used. The stiffness will then be calculated based upon the orthotropic parameters.
Only the self-weight is not yet taken into account correctly then, but this is no problem, since it can be recalculated in the 1D element model.

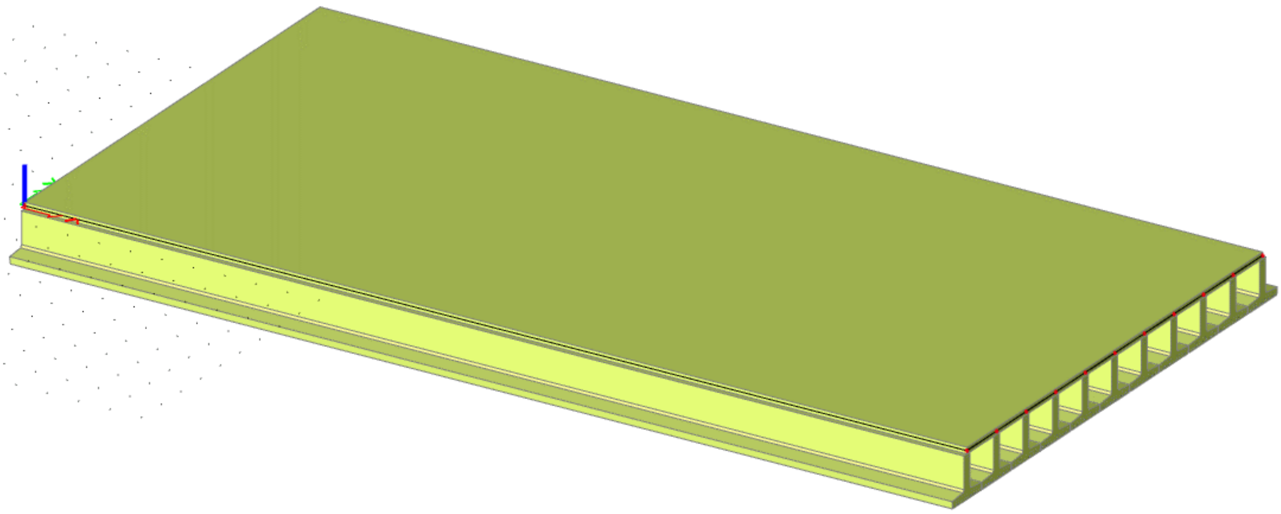
The cross-section that will be used for the ribs is the same as in the example above, with as only difference that the second phase (formed by the in situ bridge deck) is not part of the cross-section. The bridge deck will be modeled by the plate height of the 2D member because it also has to contribute for lateral stiffness.



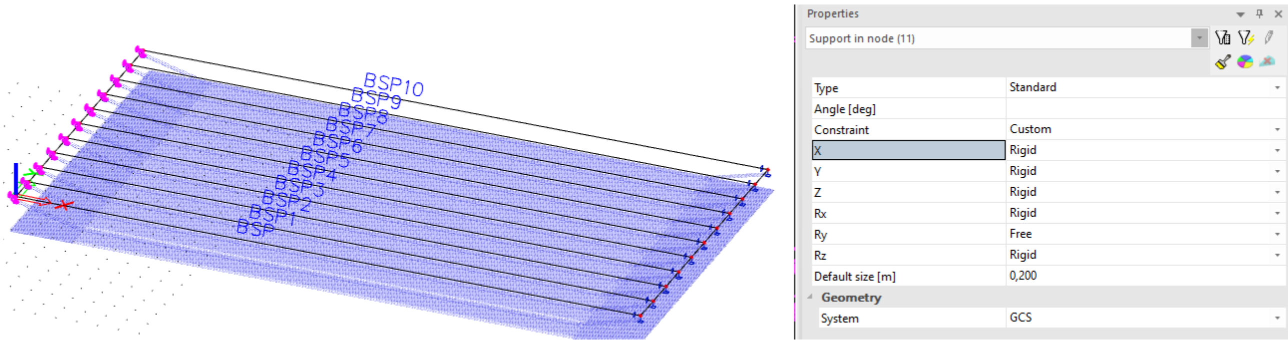




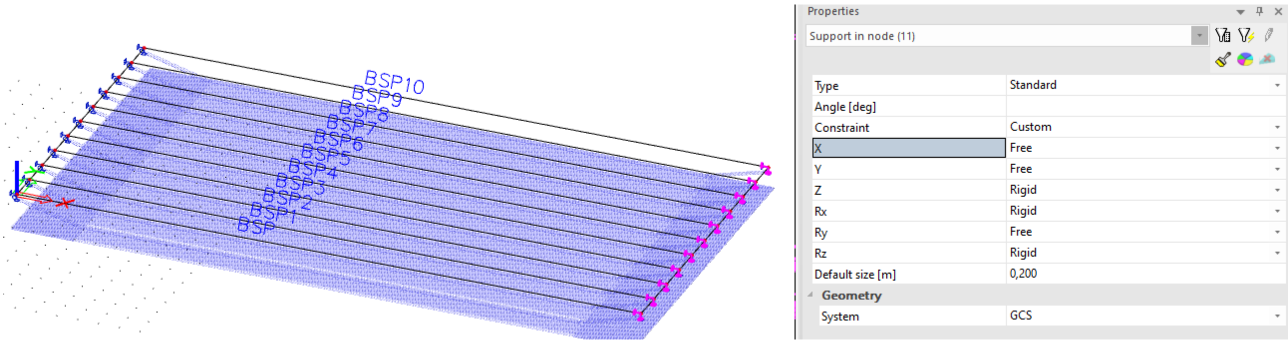
The prefab plate will be modelled with a total width of 15m and a span of 30m. The supports have to be applied as nodal supports on the beam edges.



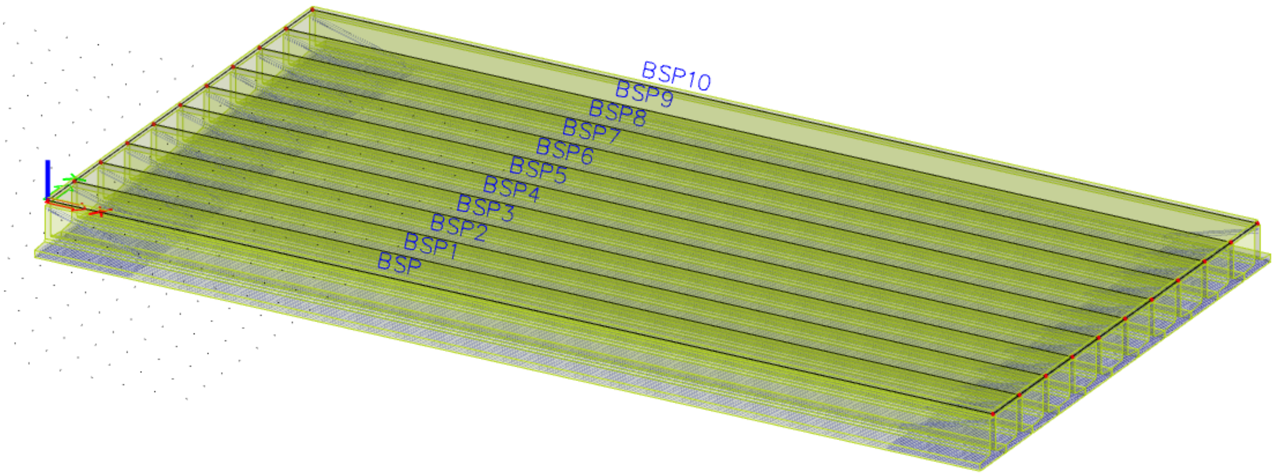
The supports on the left (X=0m) are completely fixed with exception of Ry that is free.



The supports on the right (X=30m) are the same as the ones on the left but without support in the horizontal X- and Y-directions as shown below.



It is also possible to define the prestress reinforcement in this model, but it will not be taken into account for the calculation. It will however be exported to the 1D element model where it will be taken into account in the analysis.
The same beam strand patterns as the ones used in Chapter 1 will be used. See paragraph 1.1.4 and 1.1.5.

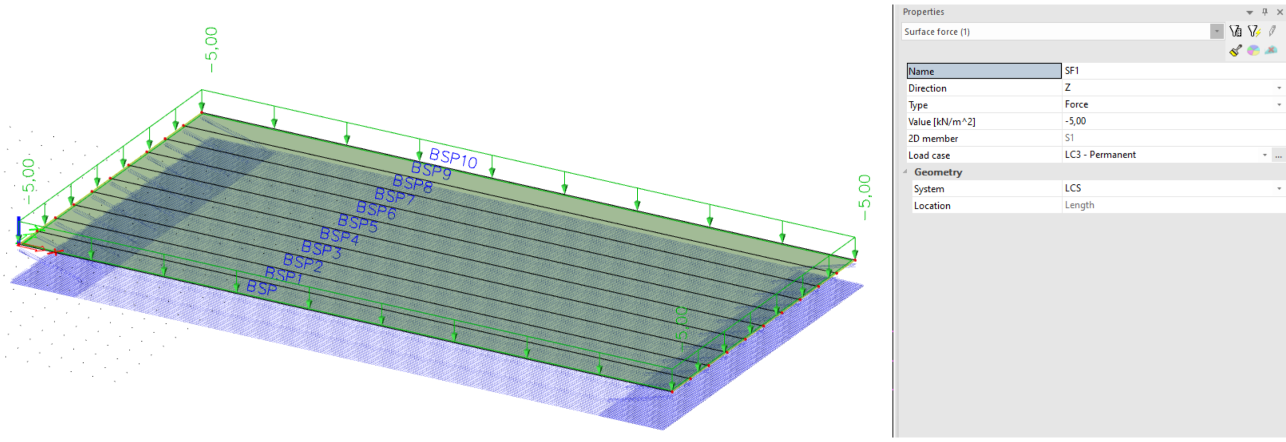


2.2. Application of the loading

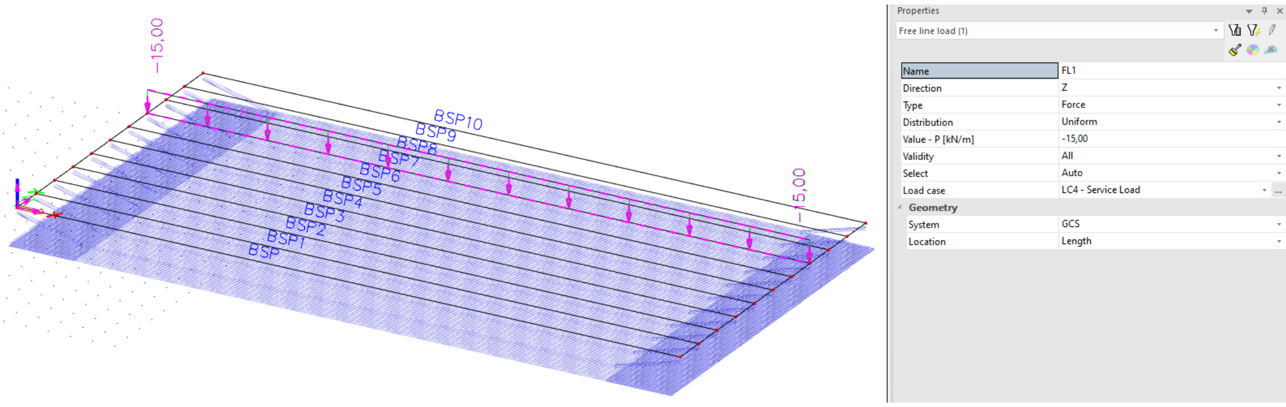
The first load case that was created automatically is the load case LC1 – Self weight.
This load case will be analyzed later on in the 1D element model, so we will not consider the internal forces for self-weight in this model.

The second load case is LC2 - Prestresses.

The third load (LC3 - Permanent) will be a surface load of 5kN/ m2.



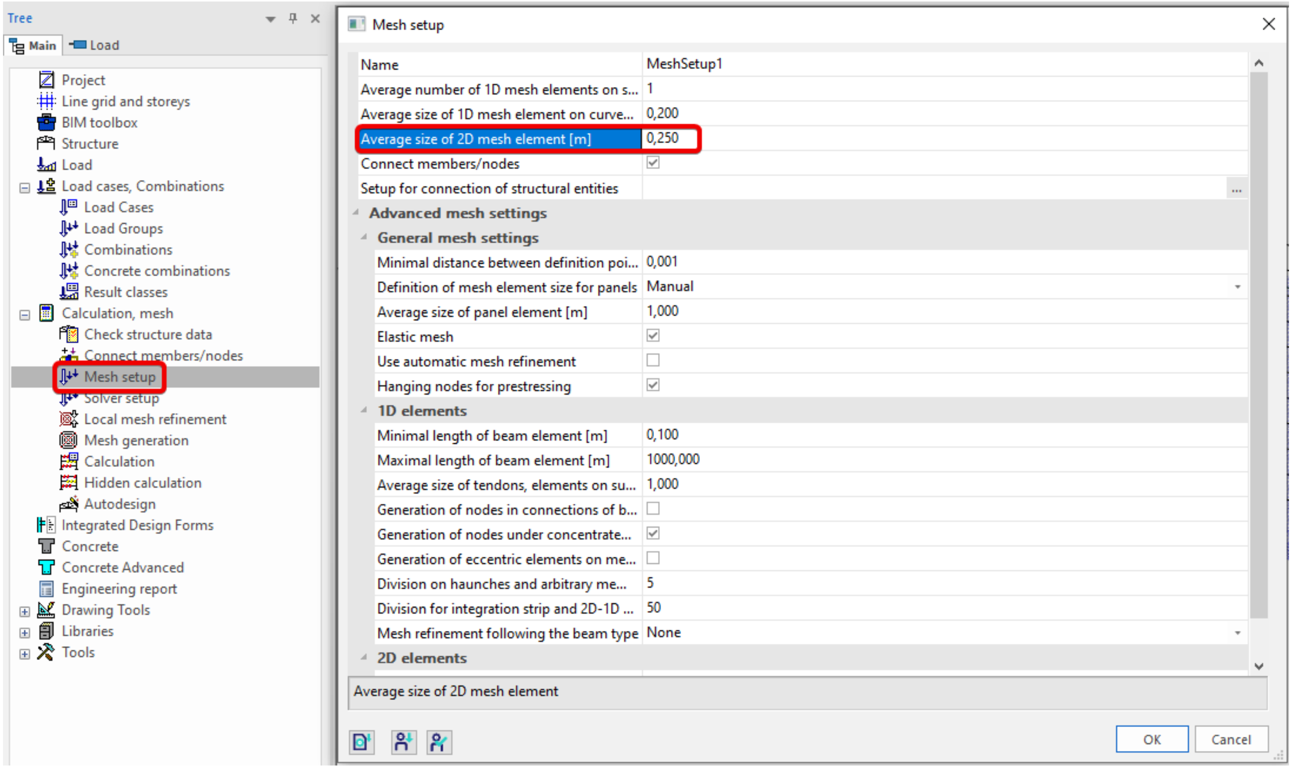
As fourth load case (LC4 –Service load), a free line load will be applied on one side of the deck.
This line load is only applied above the beam with 10.5 m set as y-coordinate, but because of the transversal stiffness there will be a transfer of load to multiple beams. The calculation of the spreading of this load is the whole purpose of this 2D prefab plate model.



2.3. Calculation

A linear calculation of these load cases will be made.

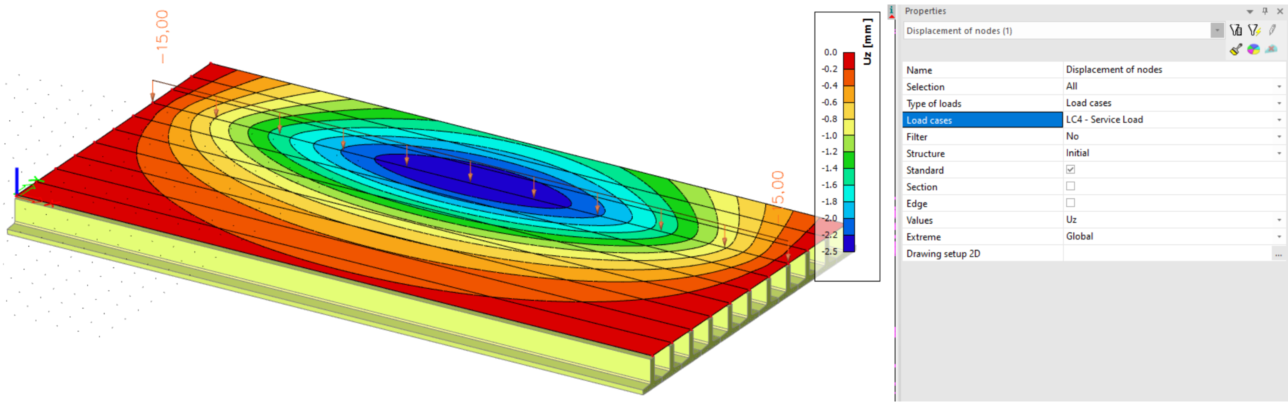
It is advised to have a look at the mesh setup first, since the default mesh size of 1m may be thorough.
A mesh size of 0,25m is set in this example.



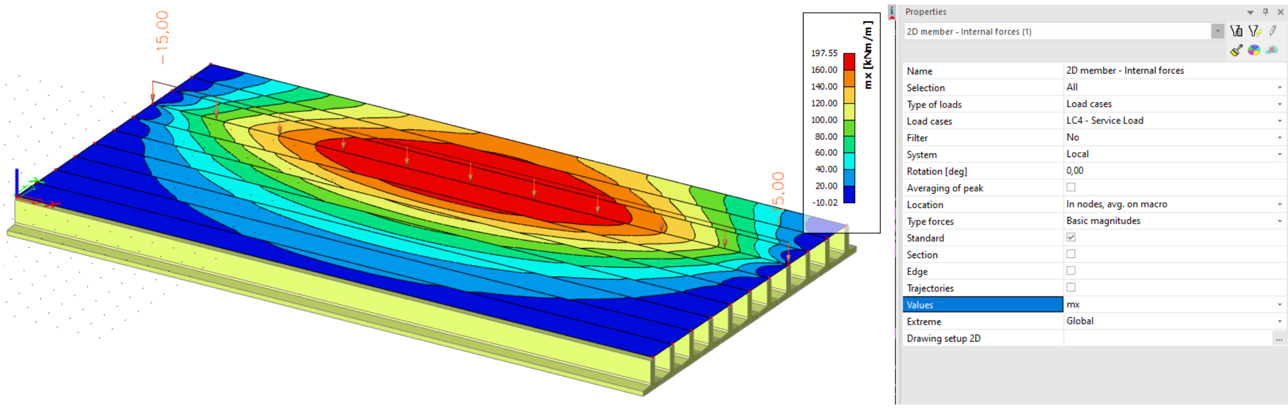
2.4. Results

The available results in this model are only results on 2D element, since in this calculation model no 1D elements are present. It is only the orthotropic 2D element that is in the calculation model.
The effect of the longitudinal ribs is taken into account in the orthotropic stiffness, which will lead to a correct spreading of the loads, which is of course the purpose of this prefab plate model.

The displacement for load case 4 are shown below.

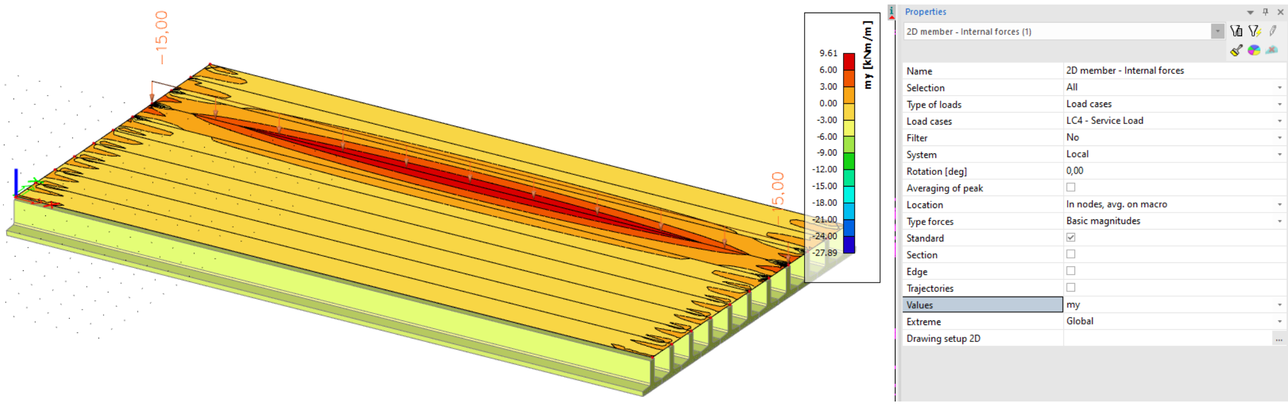


The moments for load case 4 are shown below.



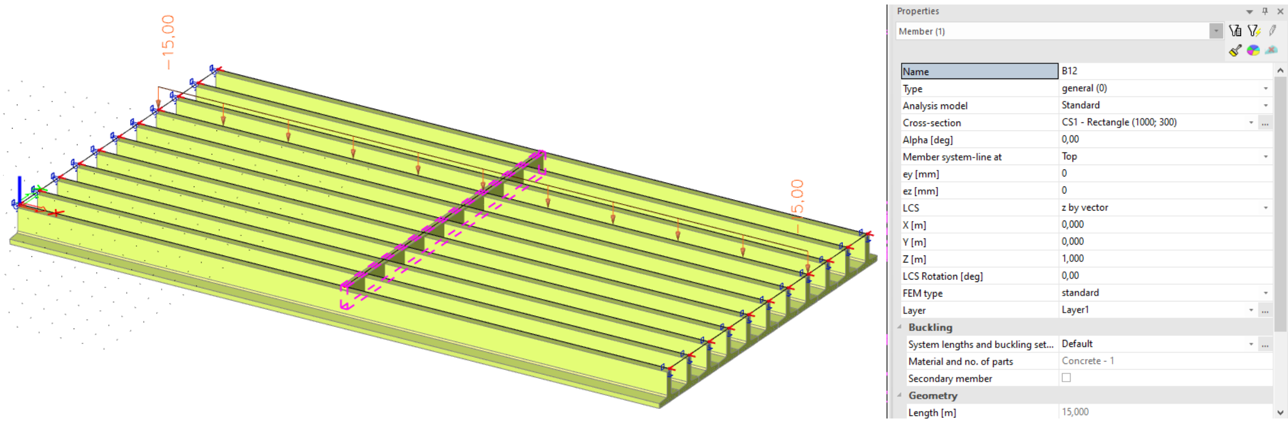
There is a clear spreading noticeable. The local line load of 15kN/m will be spread over multiple beams. This spreading is caused by the transversal stiffness of the prefab plate model. This transversal stiffness will be delivered by the in situ topping of the bridge deck (and not by the prefab beam elements themselves who will serve of course for the longitudinal stiffness).

There is also a moment my in transverse direction noticeable.

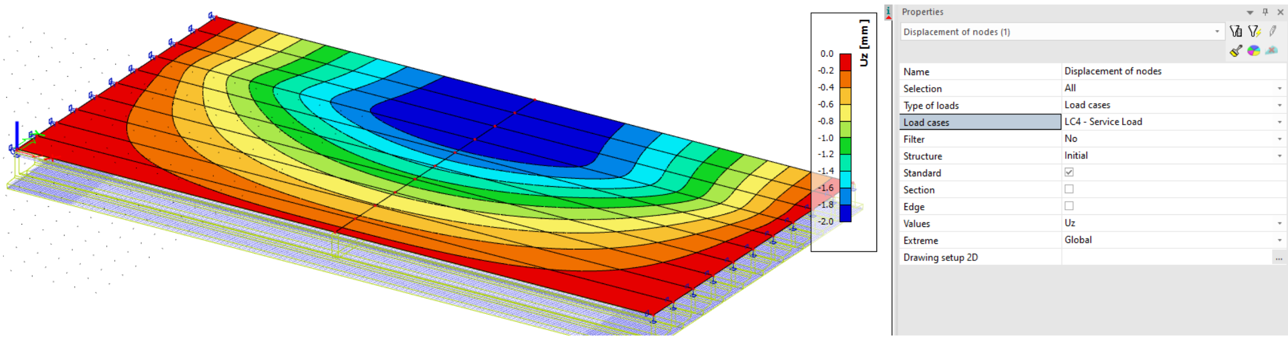


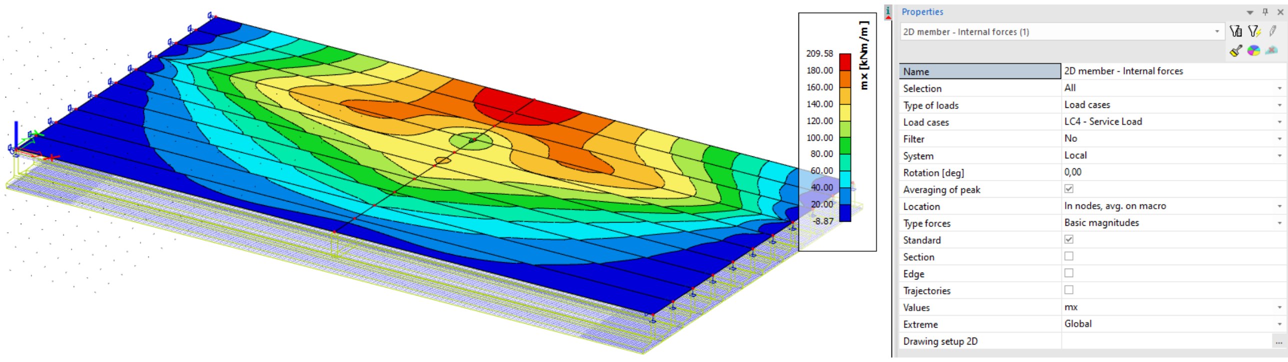
To increase the spreading of the loads, a transverse beam could be added to the model.

In this example a transverse beam (R[1000,300])is added in the middle of the span.
(please note that for correct modelling an internal edge should be created in the plate element to connect the transverse beam to the plate element).



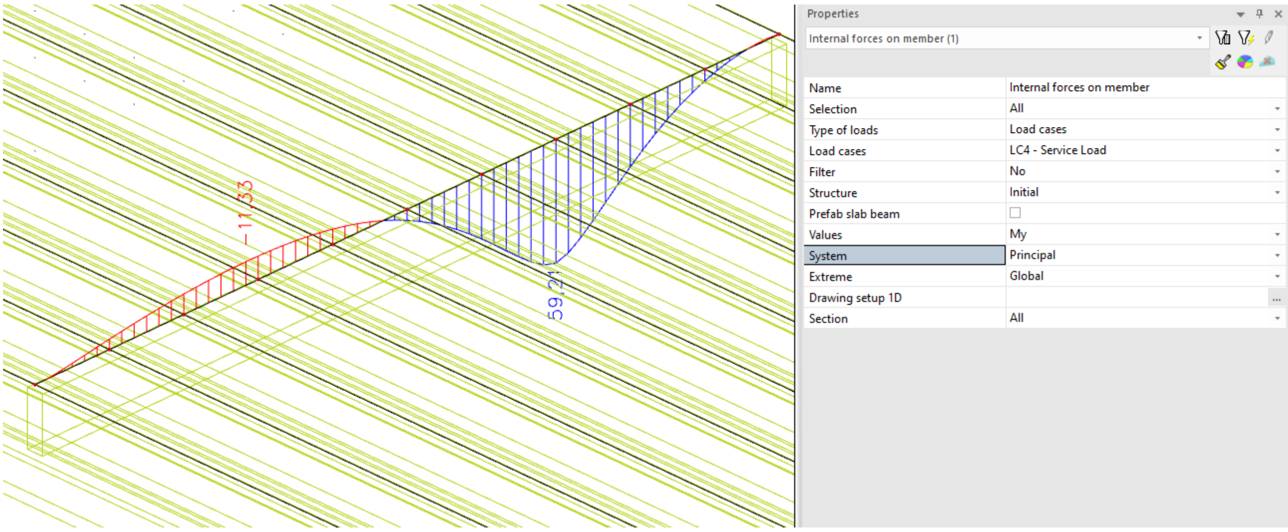
The effect on the displacements and moments are shown below.





The transverse beam is clearly providing a better spreading of the loads.

It is also possible to design the transverse beam itself, since this 1D element is part of the calculation model. The moment on this transverse beam is shown below:

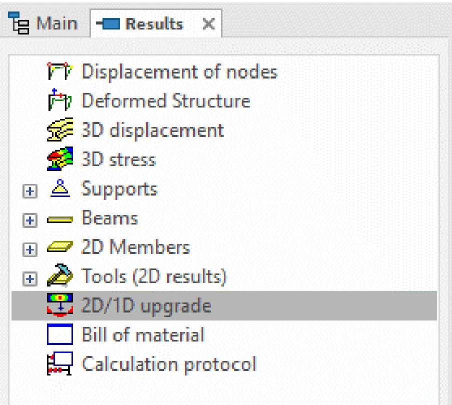


2.5. 2D → 1D upgrade

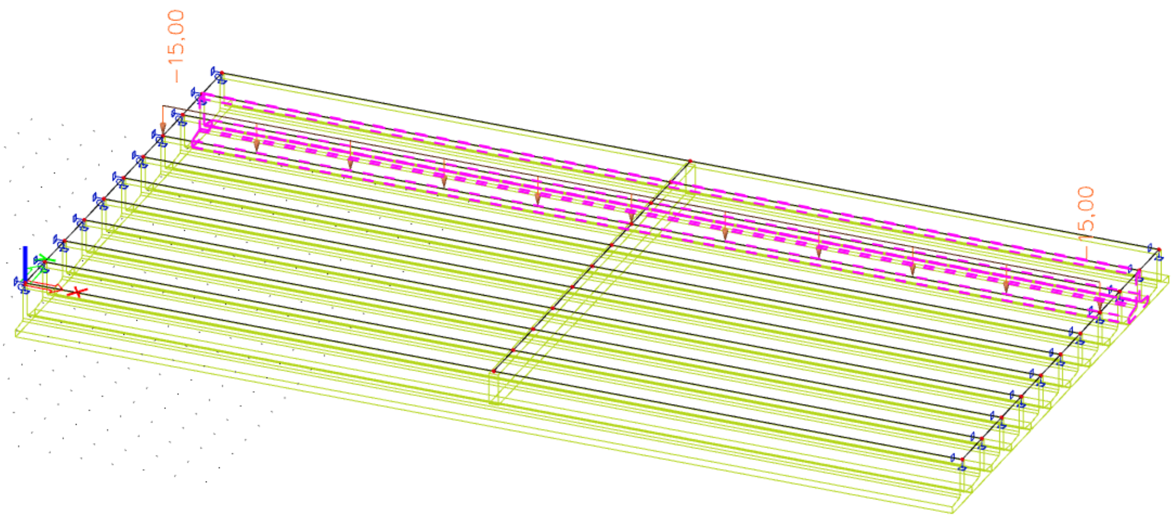
As it was mentioned before, the purpose of this 2D prefab plate model was to make an analysis of the spreading of the loads. The dimensioning of the prefab beam elements should be done in the 1D element model.

In the prefab plate model the prefab beam element(s) that take(s) most loading will be exported to a 1D element model for dimensioning. This can be done through the 2D/1D upgrade functionality in the results menu.

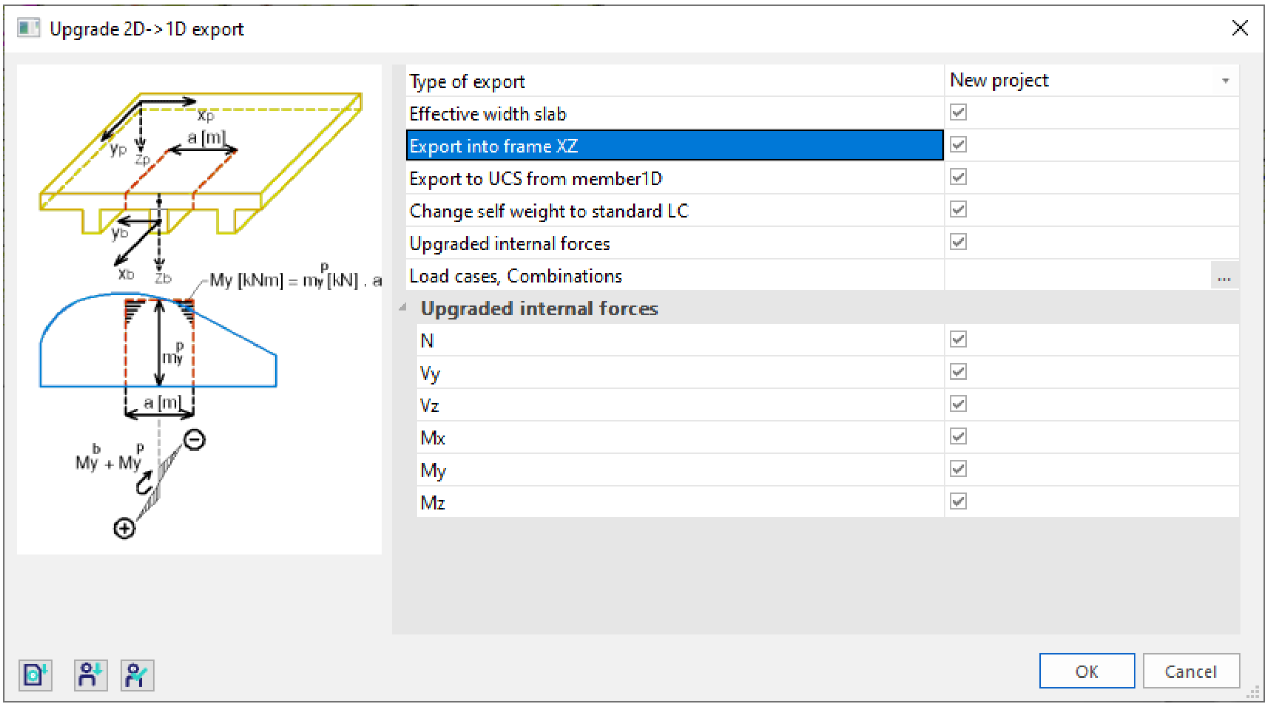
2D/1D upgrade will automatically create a 1D element model with geometry and internal forces taken from this 2D element model.



Select the beam that takes the most loading. In this case, it is the second to last beam called B10.



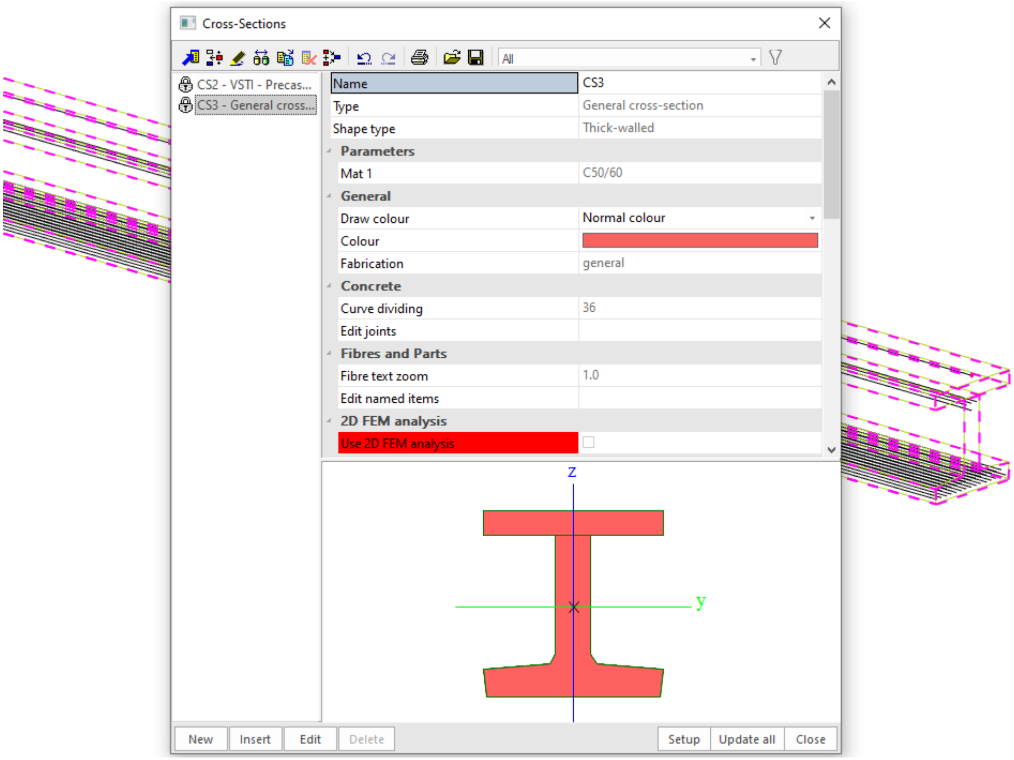
For the 2D->1D export the following setting can be used.



The width of bridge deck that is taken into account in the export is defined by the effective width (which could be entered in the properties of the prefab slab). This effective width is also used as integration width to define the equivalent internal forces on the 1D element model.
It is advised to export to frame XZ environment, since in this environment TDA calculation is possible. This is only possible if you only export only one beam at the time.
The user could also define which internal forces and for which load cases they should be exported.

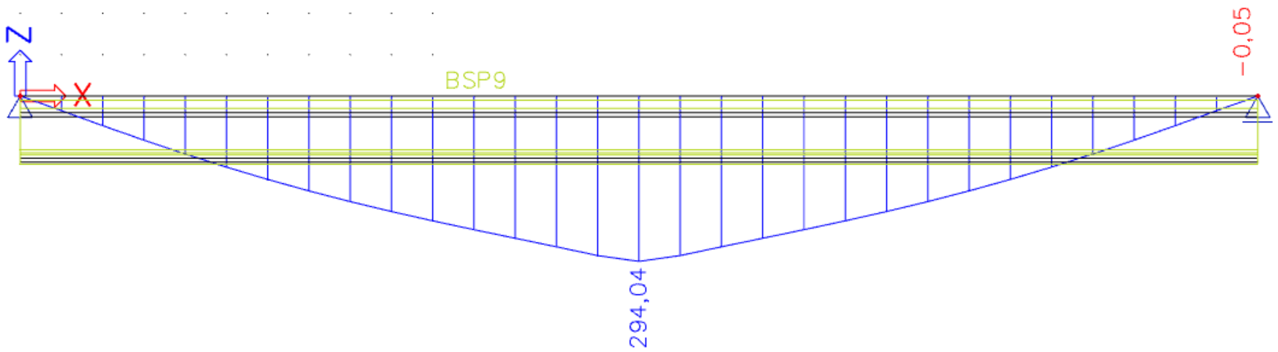
It is also possible to export into a template instead of a new project.
Exporting to a template will open a user template (with parameters) instead of just a new project.

The exported 1D element model is shown below.



The geometry of the model is recuperated from the prefab slab model.

The internal forces are also exported to this 1D element model.
As an example the moment line My is shown below



This moment line is a result of integration from the internal forces on the 2D prefab slab model.

In the 1D element model these internal forces could be used in the EN checks to make the dimensioning of the prefab beam elements. Therefore, reference is made to the 1D element model treated in chapter 1.