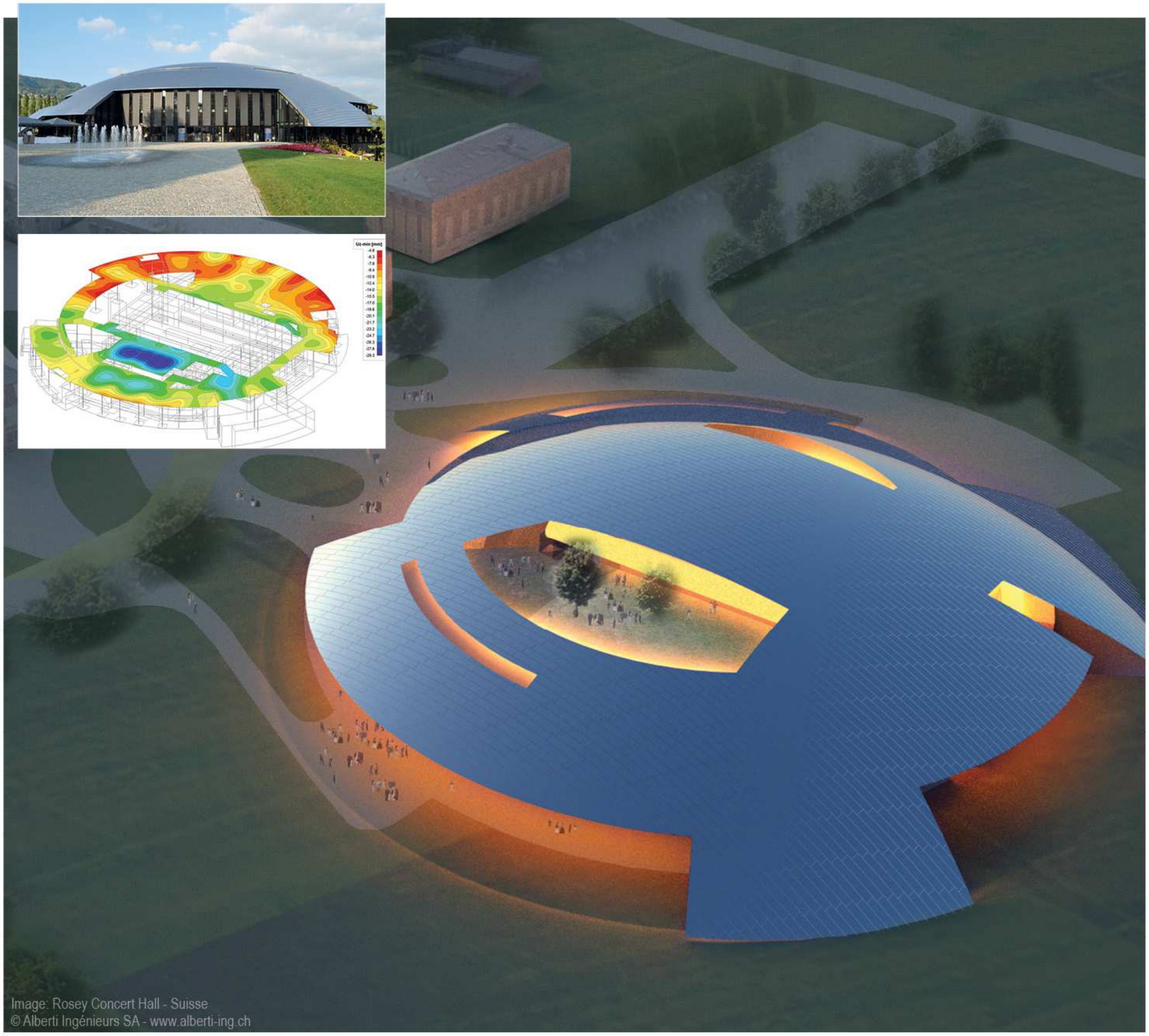
SCIA ENGINEER



Advanced Expert Training

Pre-tensioning (Precast)

Table of contents

Introduction

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

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 dependant 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.

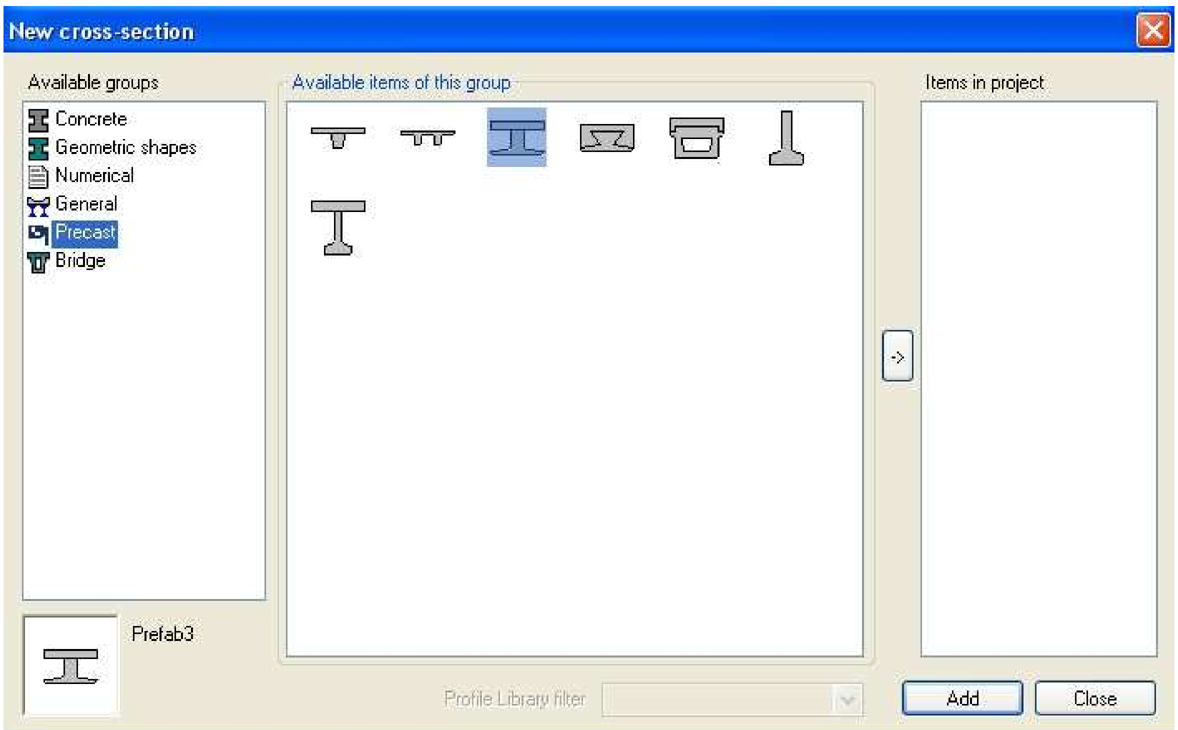
Pre-tensioned precast beam

1_Input geometry and prestress

In the project data a frame XZ environment and a construction stages model will be chosen.
The functionality prestressing will be ticked on.

The model will consist of a simply supported precast beam with a length of 30m.

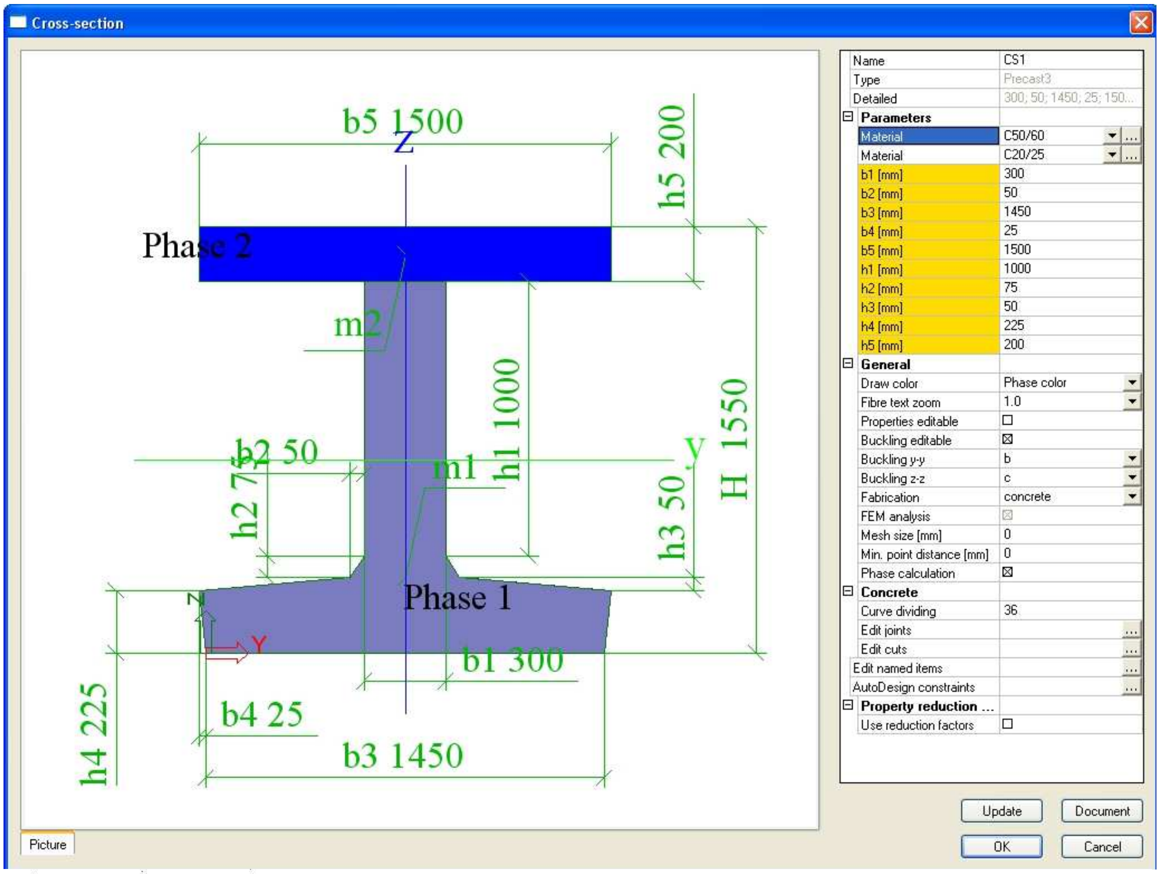
In the SCIA Engineer profile library we will find some preparametrised prefab profiles.

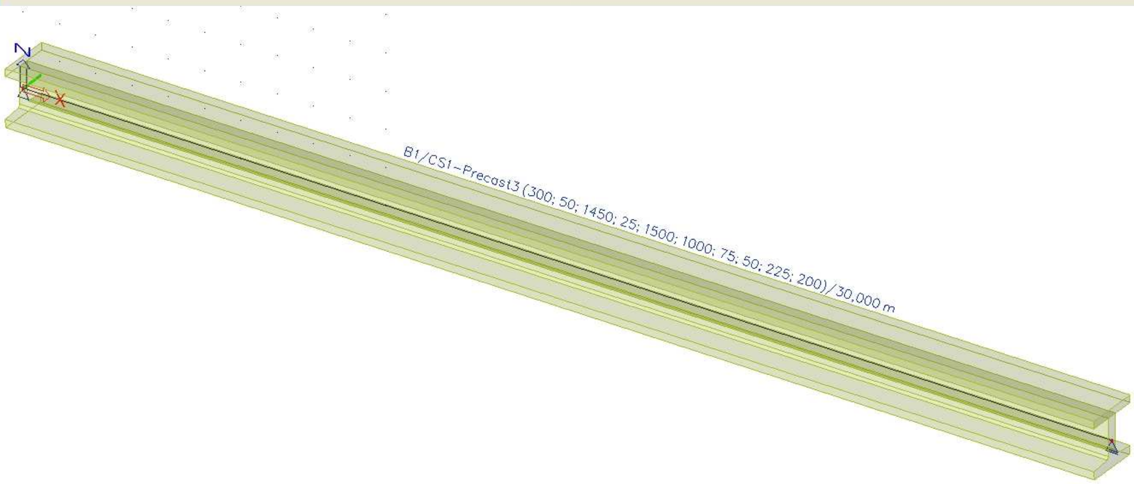


In this example we will choose for the Prefab3 profile type.

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.

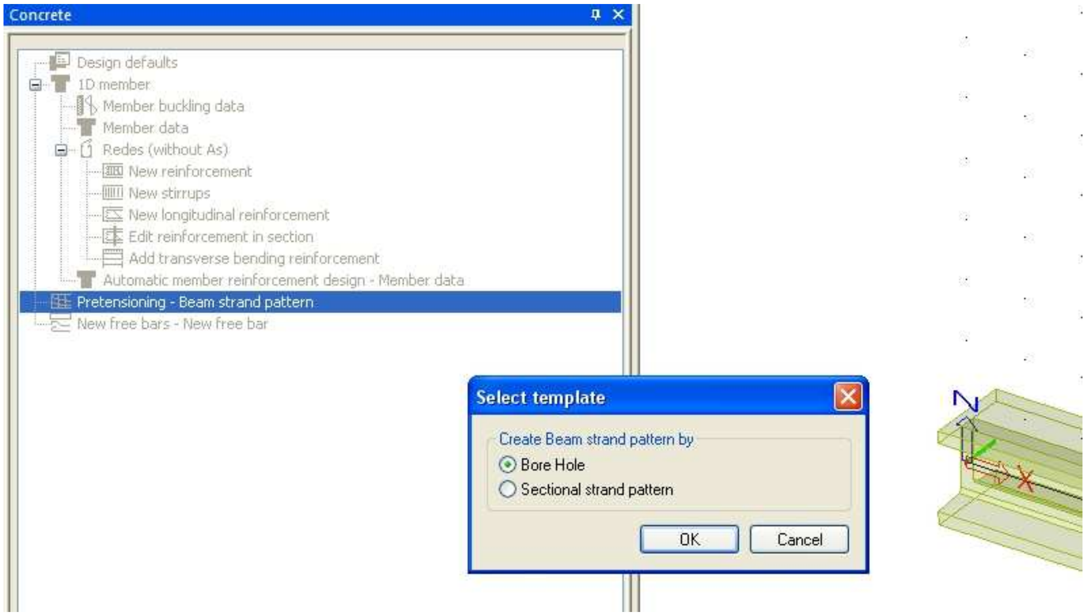
In this example, we will use the prefab I-beam with an in situ cast topping.
The profile has the following dimensions:





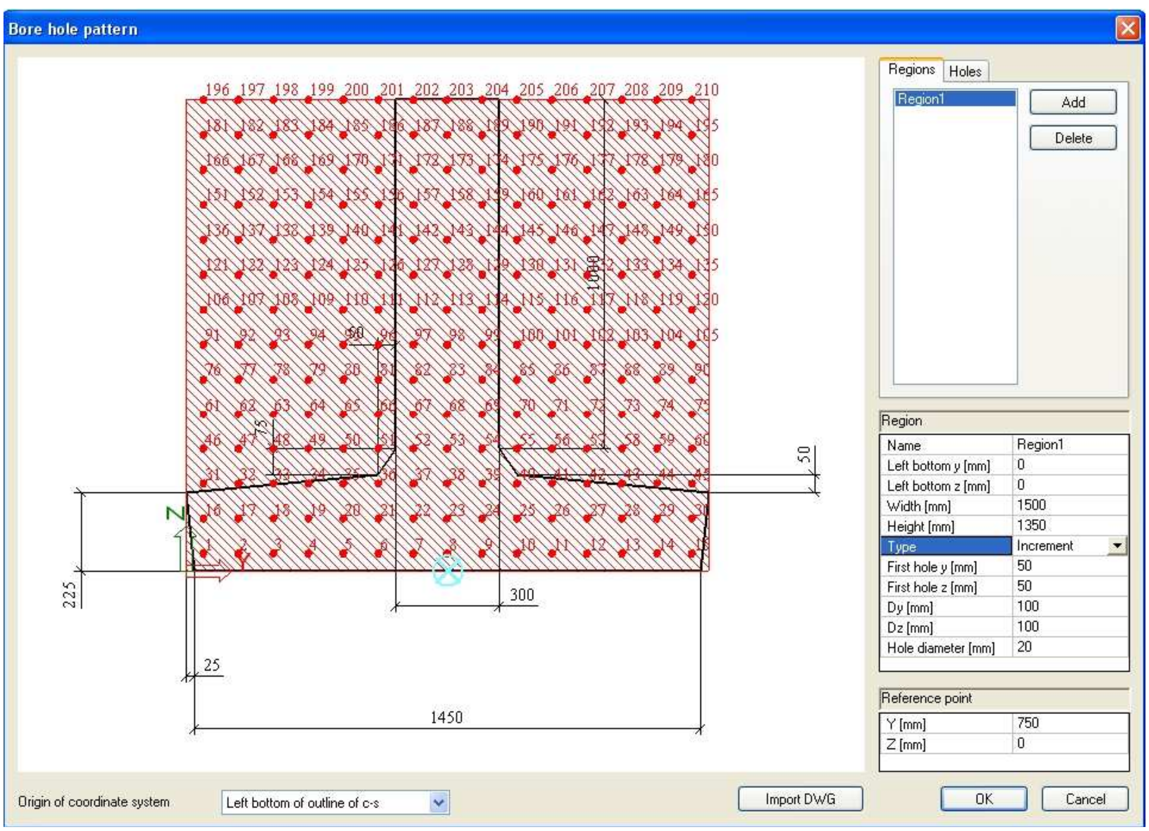
The input of the pretensioning is done in the concrete menu.

We will start with the creation of a bore hole pattern:

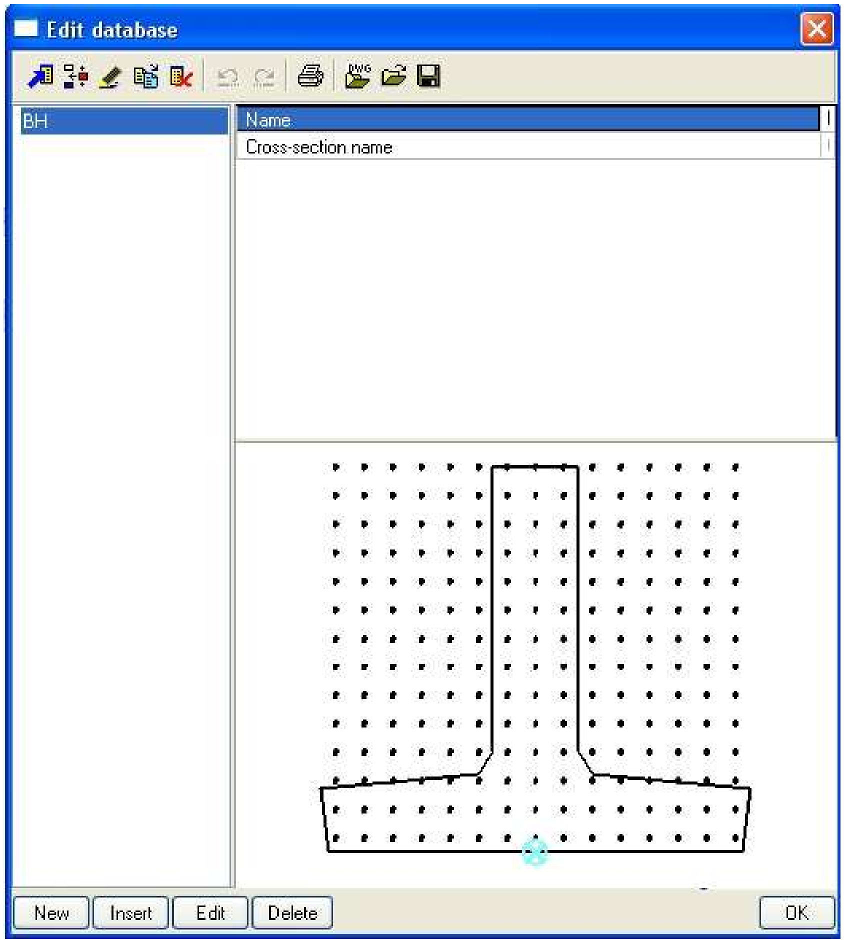


The bore hole pattern will be defined on a profile. We choose of course the above defined cross-section. The definition of the bore hole pattern can be done by means of the input of individual holes or a whole region.

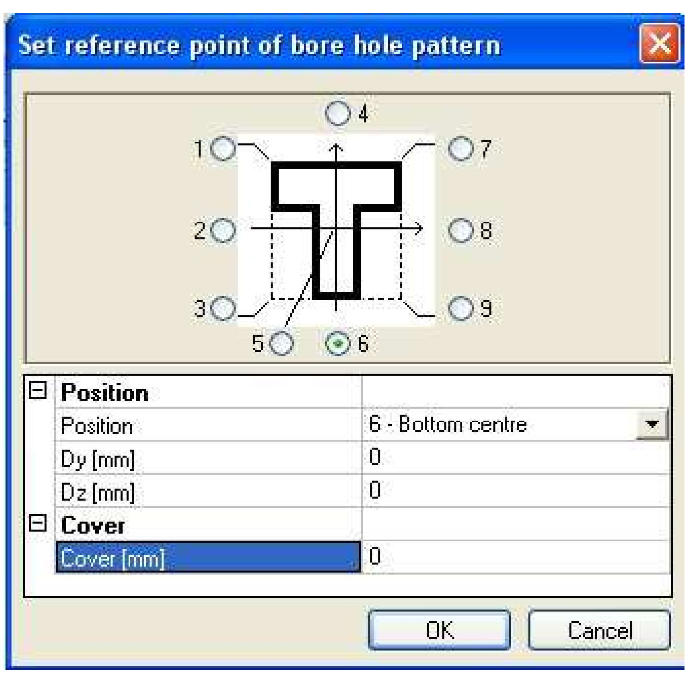
In this example a whole region over the prefabbeam will be created.
(Notice that the program automatically detects that no bore hole pattern can be defined in the in situ concrete pressure zone.)
The geometry of the bore hole pattern is shown below:



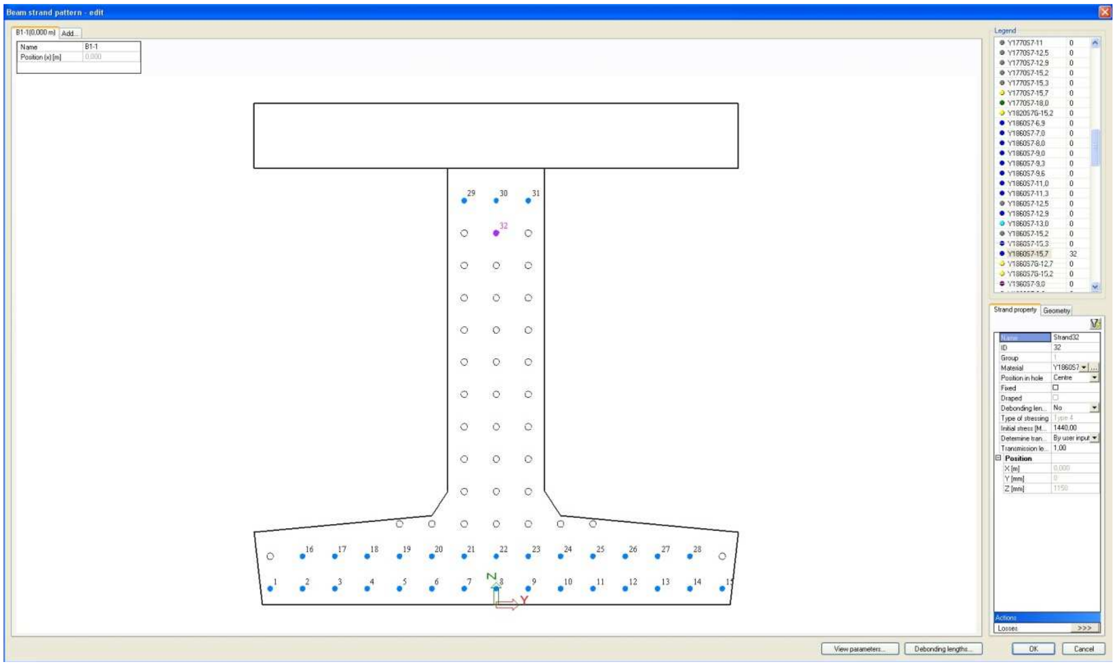
This borehole profile could be stored to use in other projects
.



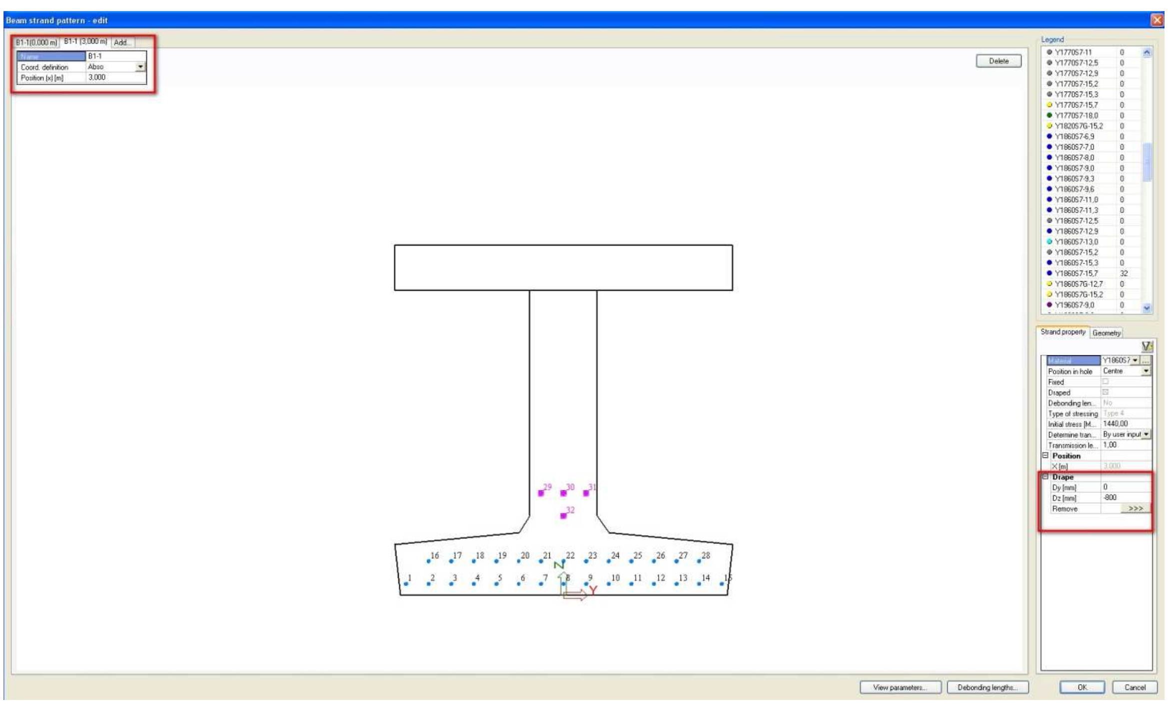
Now we will use this bore hole pattern to define a strand pattern.
As reference point for the input of the bore hole pattern, a cover of 0mm will be applied because the cover is already taken into account in the definition of the bore holes.



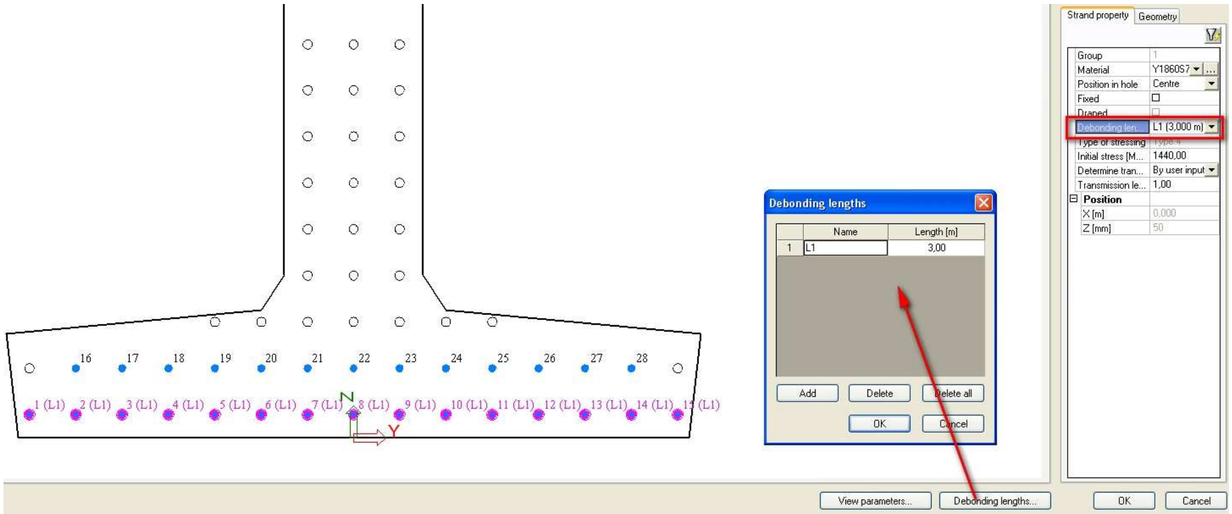
Below is shown the beam strand pattern.
28 strands will be inputted on the bottom fibre and another 4 strands at the top fibre. The strand quality is Y1860S7-15,7.



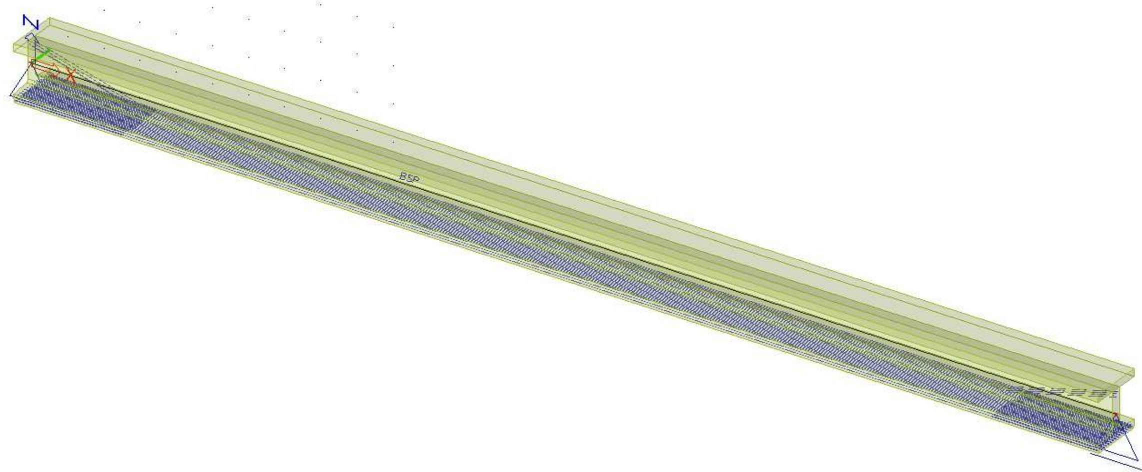
The top strands are bent so they will join the bottom strands at a distance of 3m from the support. To achieve this an extra section at 3m is created and on this section a drape of 800mm is given to the top strands.



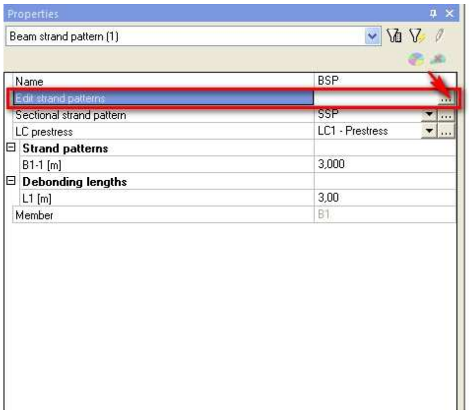
Another measure that could be taken to avoid that in the begin section a tension stress at the top fibre would occur is the debonding of some strands over the first meters. In this example the strands on the bottom row are all given a debonding length of 3m.

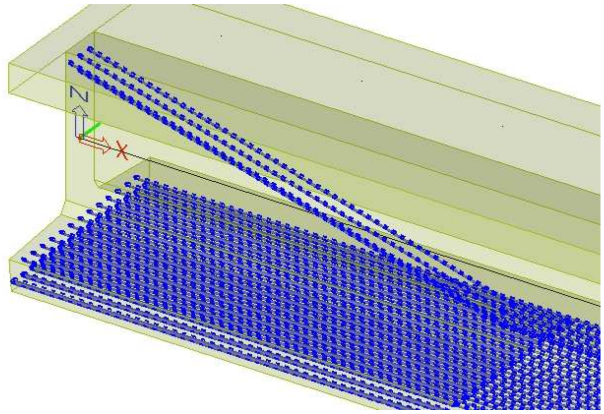


The final strand pattern will look as shown below:

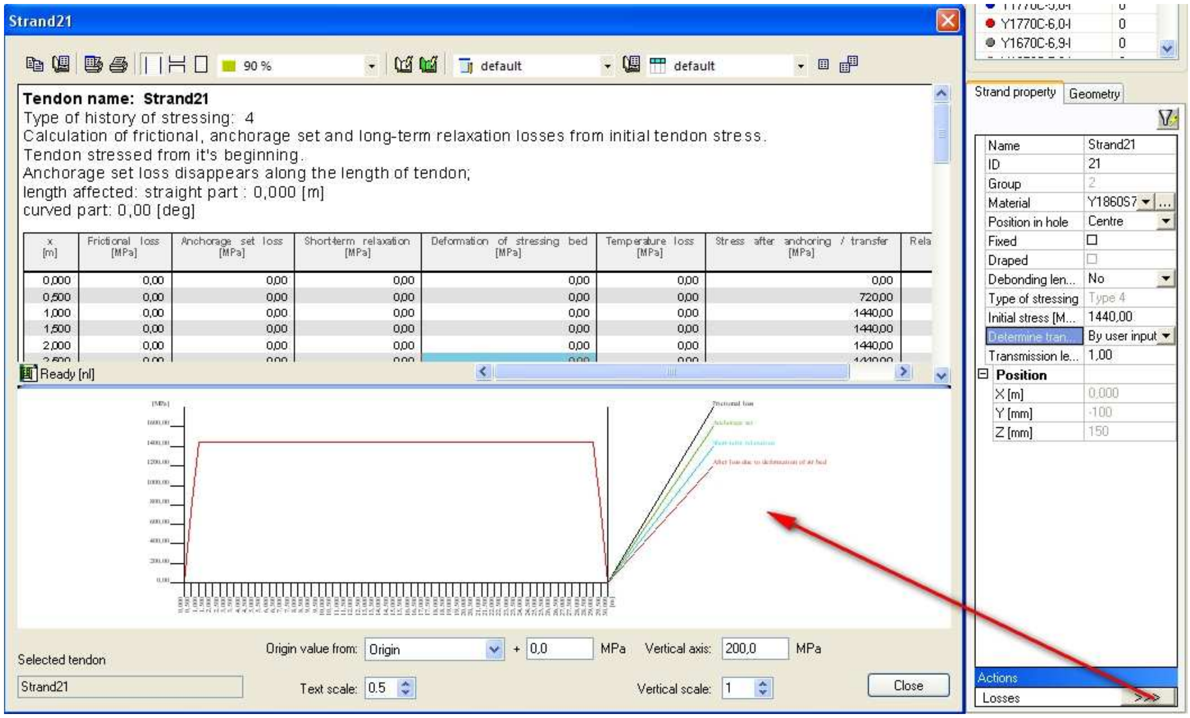


To edit the properties afterwards, the beam strand pattern should be selected. Trough the property menu the user can click on edit strand pattern to edit the strand pattern at any time.





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 through 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. (To have access to these parameters the functionality prestressing>advanced needs to be ticked on.)

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.

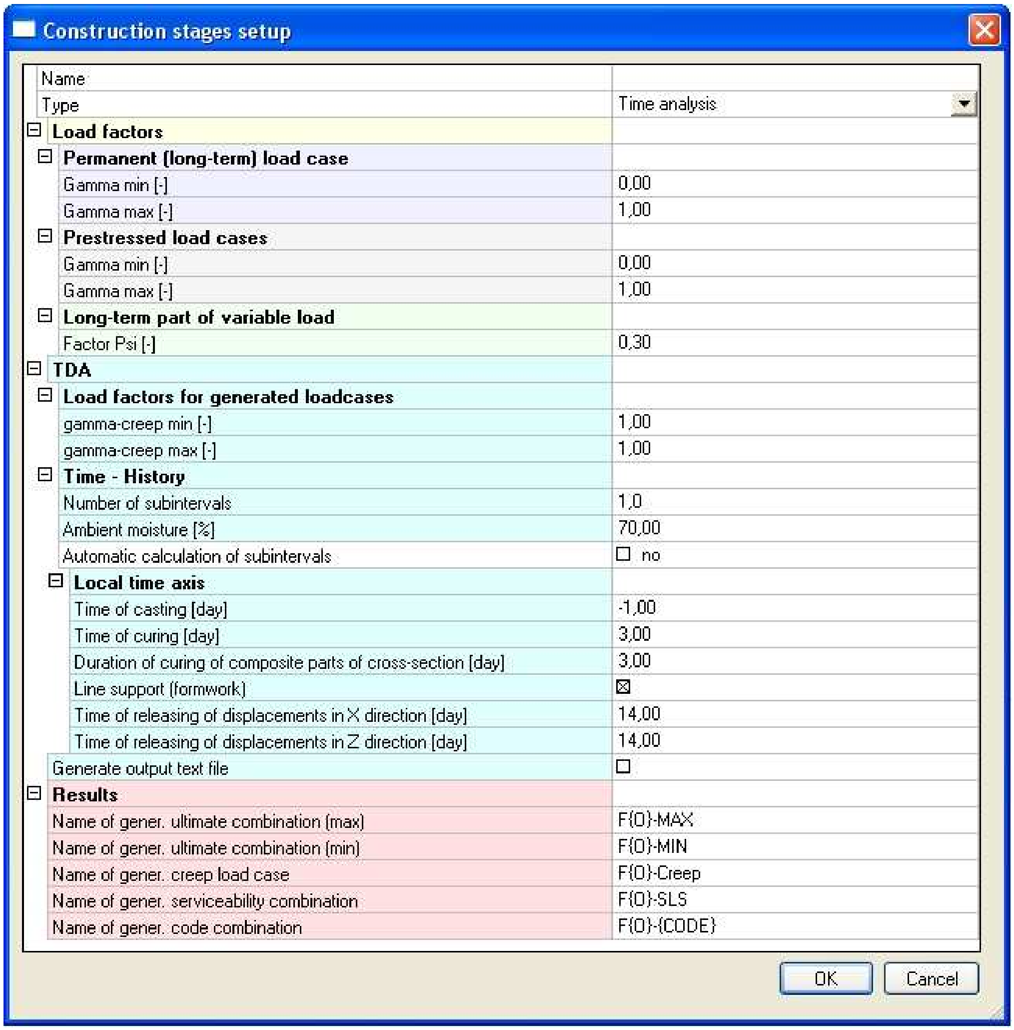
We will start by creating the following load cases:
LC2: self weight prefab beam
LC3: self weight in situ concrete (7,5kN/m) : attention: manual input required !
LC4: permanent load for finishes (10kN/m)
LC5: service load (20kN/m) : to be defined as a variable load of long term duration
LC6: empty load case (for check after 50 years)

It is easier to create the load cases before the stages, because for every stage at least one permanent load case is required.

Afterwards the stages can be defined:

Stage 1: Placing of precast beam: day 0
Stage 2: Casting of in situ concrete: dag 20
Stage 3: Application of permanent load for finishes:day 40
Stage 4: Start of the service stage: day 50
Stage 5: Check on long term: day 18000

The configuration for a time dependant analysis looks as shown below:



The most important properties are:

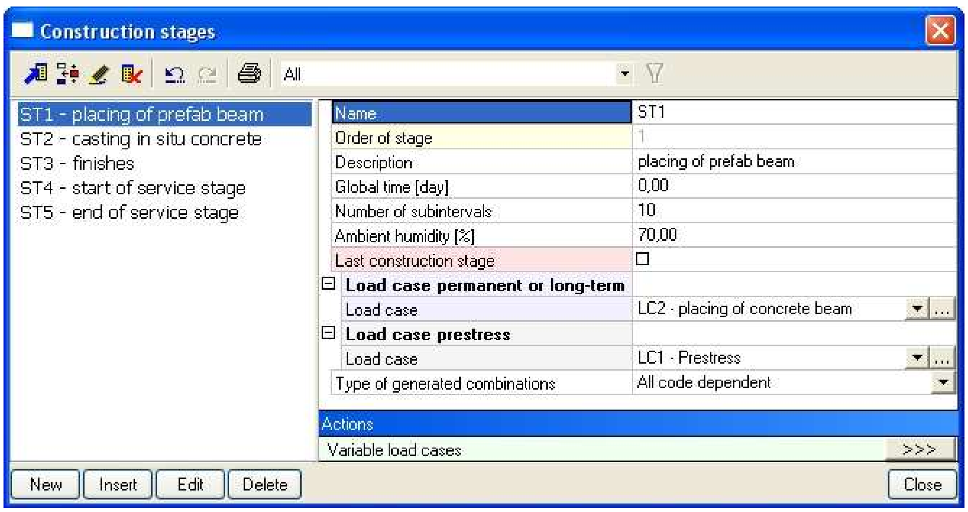
- Long term part of variable load: Factor Psi
  This implies that variable loads of long term are also taken into account for the time dependant analysis of the long term losses, but then only for a certain percentage.
  This is only valid for variable load cases of long term duration.
  Variable load 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 dependant 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.

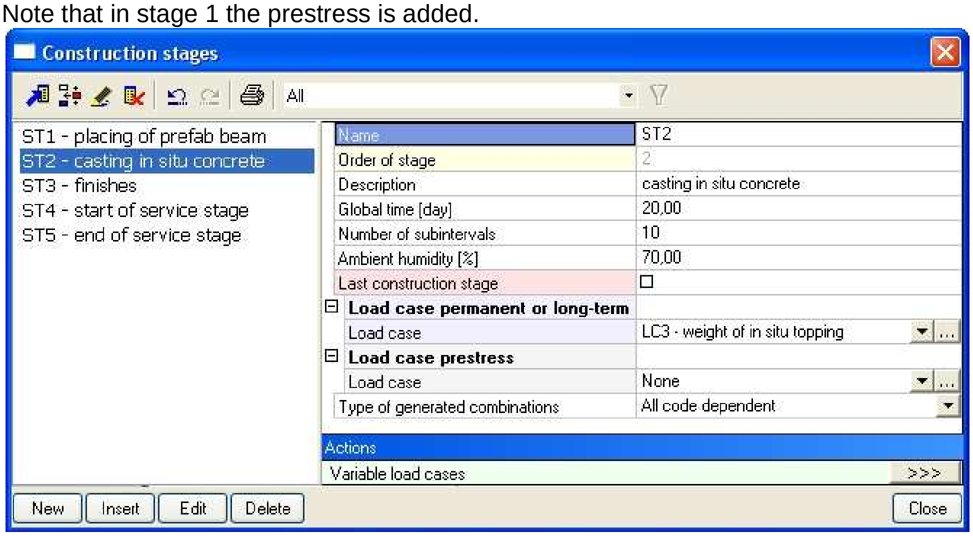
- Ambient moisture [%]
  This property has an influence on the shrinkage. The higher the ambient moisture, the lower the shrinkage. The default value of 70% is kept in this example.

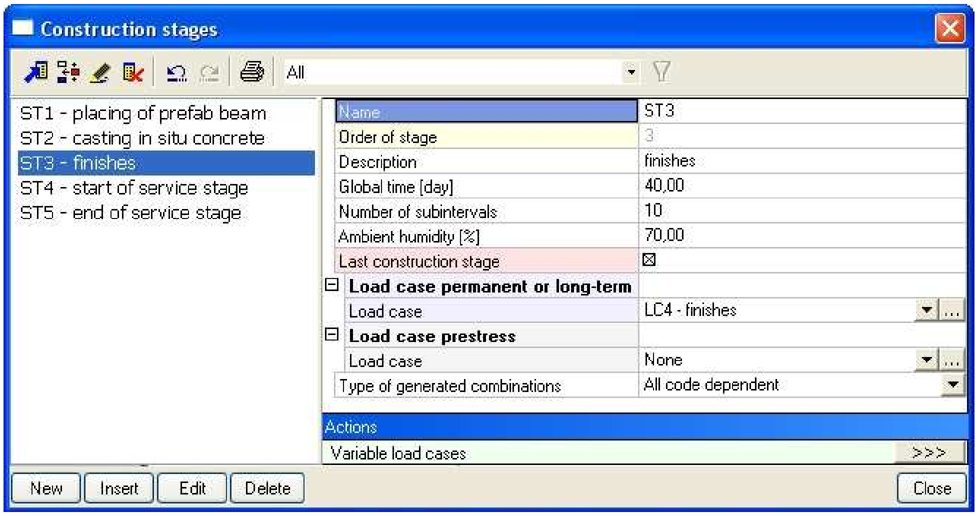
- o The local time line introduces a few extra time nodes on the timeline for which no construction stage was created.
    - Time of casting: the default here is taken is -1 day, because the introduction of an element that has the stiffness of day 0 will cause problems in the solver, because the solver 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.

De definition of each construction stages for this example is shown below:

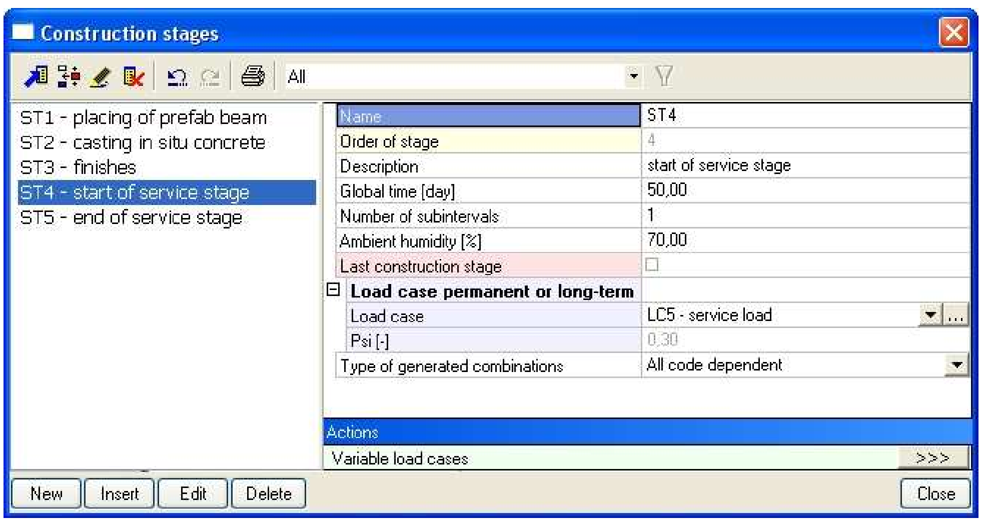


Note that in stage 1 the prestress is added.

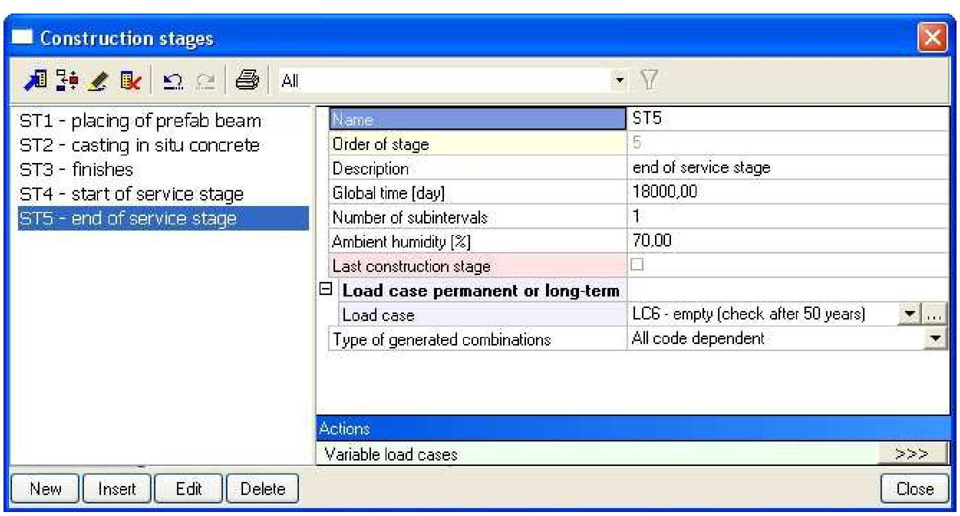




Note that stage 3 is the last construction stage.
This means that after this stage, it is allowed to add service loads of type long term variable.



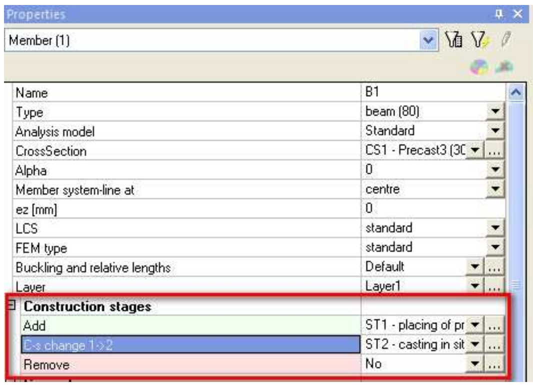
The Psi factor of 0,3 will only influence the participation of LC5 to the long term losses calculated by TDA solver. In the generated combinations, this load case will be considered without this reduction factor, so this reduction factor is only for the calculation of losses, not for the concrete checks.

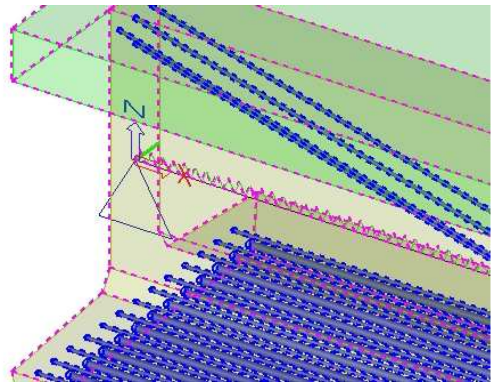


A control stage at 50 years (18000 days) is created to take into account the long term losses.

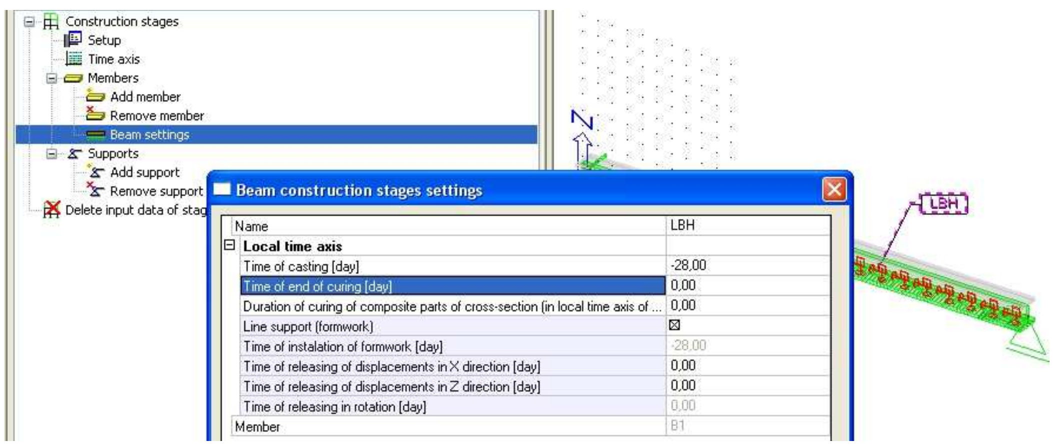
For the actual definition of the stages, we will add the precast beam and its supports in stage 1.
The prestress is already added as a load case assigned in the stage properties.

The in situ concrete is added in stage 2 trough the properties of the beam as shown below.



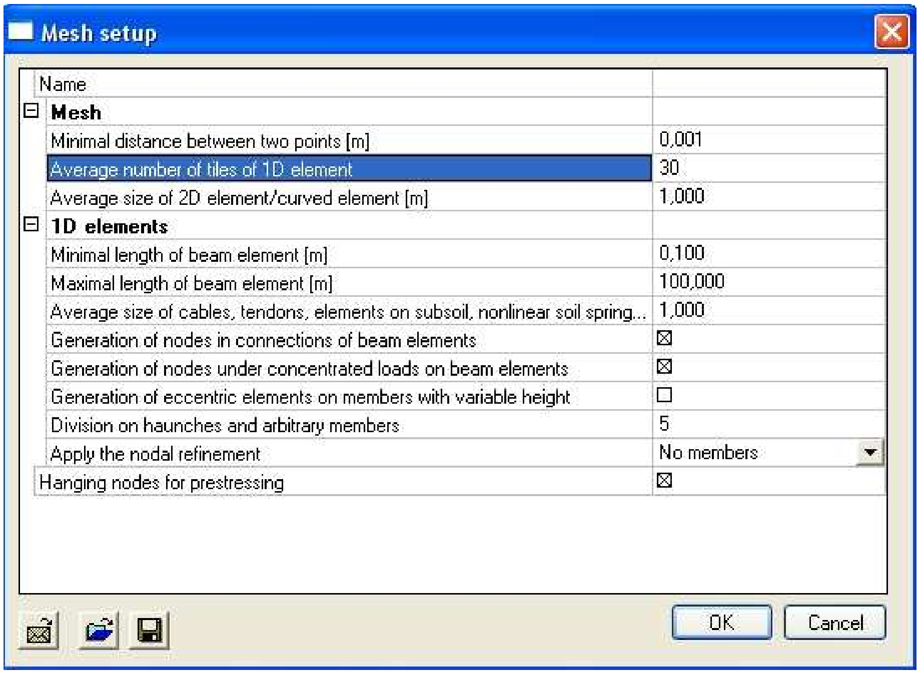


The last point to take care off is the age of the concrete.
Since the precast beam which is added in stage 1, is assumed to have full stiffness at the moment of placing, we will define an age of this precast beam of 28 days trough local beam settings.
Note that this will not influence the default age of the in situ concrete which is added fresh (age 0) in stage 2.

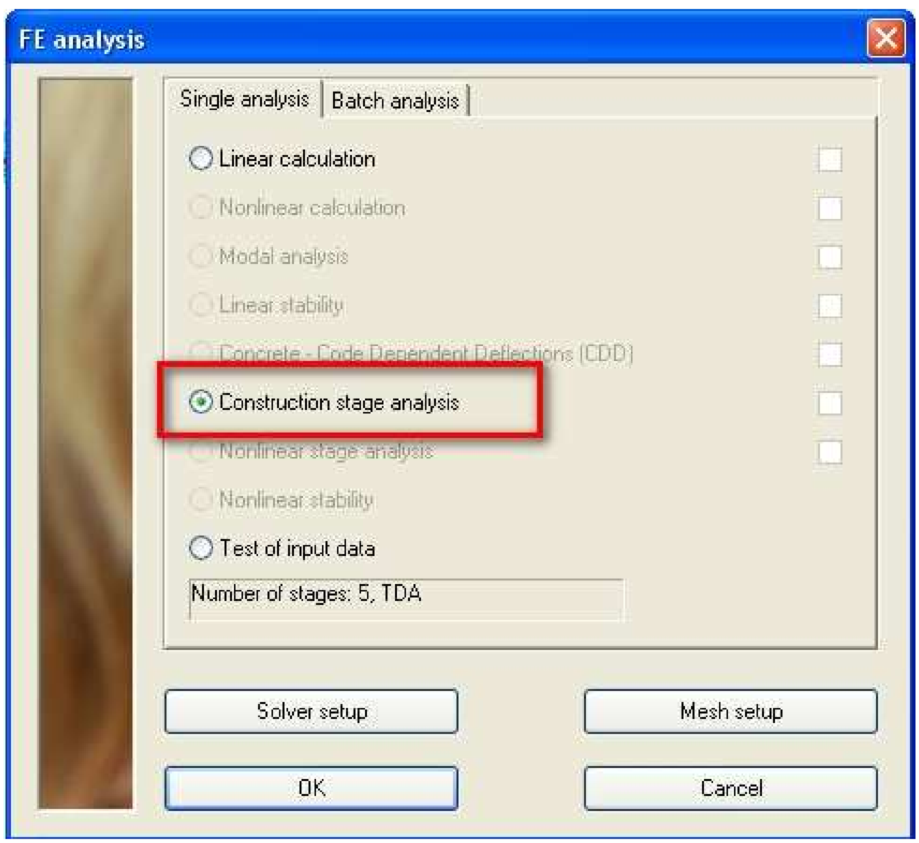


3_Calculation

Before running the TDA analysis, the beam and the patterns are meshed. The beam is divided into 30 mesh elements 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 by mistake the linear calculation is run, everything defined in construction stages will be ignored.)



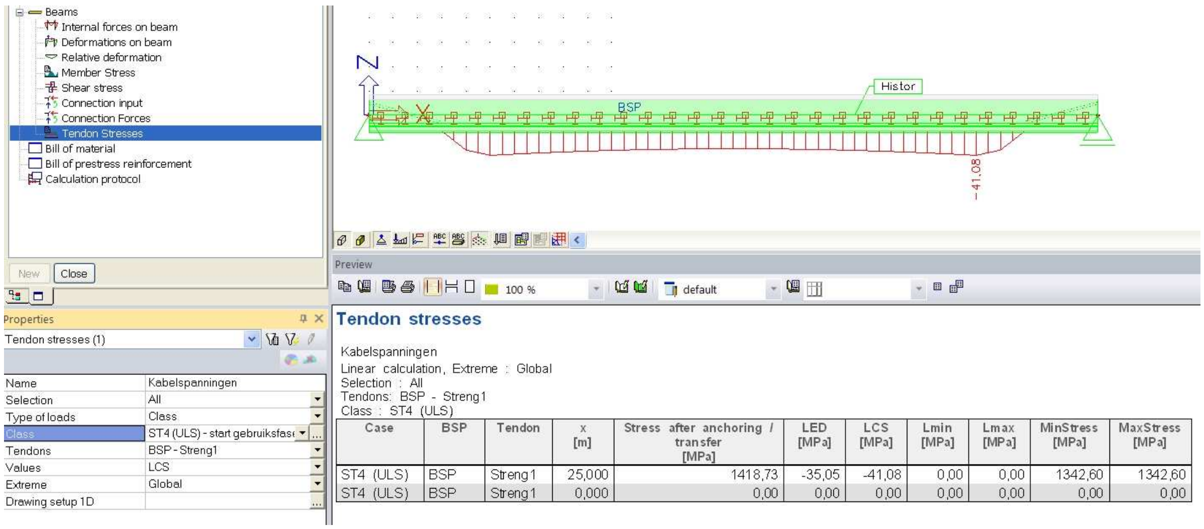
4_Results

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

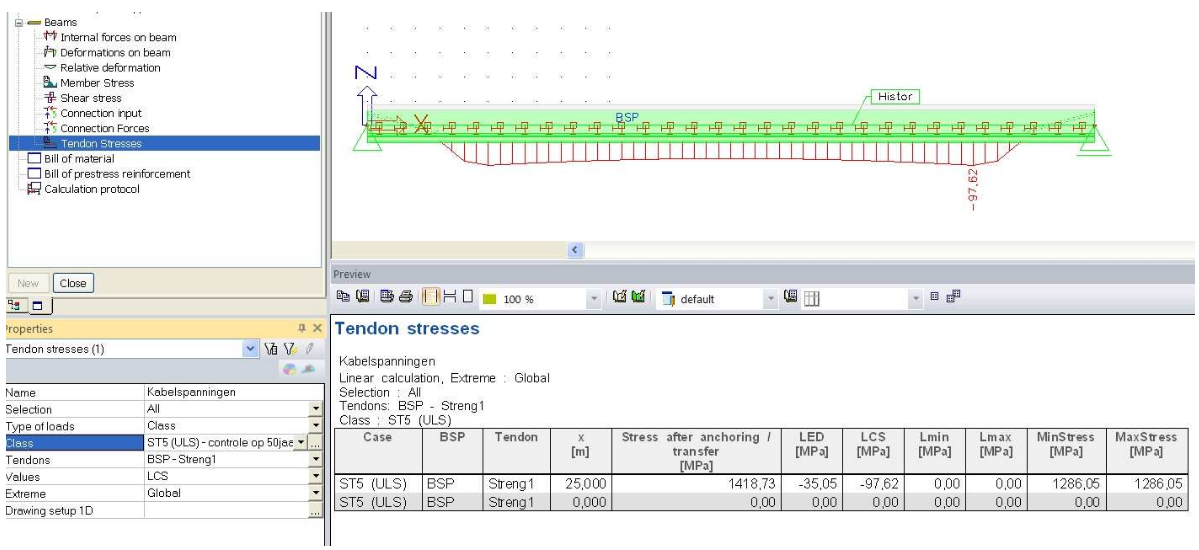
Tendon stresses

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 dependant 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:



Case	BSP	Tendon	x [m]	Stress after anchoring / transfer [MPa]	LED [MPa]	LCS [MPa]	Lmin [MPa]	Lmax [MPa]	MinStress [MPa]	MaxStress [MPa]
ST4 (ULS)	BSP	Streng1	25,000	1418,73	-35,05	-41,08	0,00	0,00	1342,60	1342,60
ST4 (ULS)	BSP	Streng1	0,000	0,00	0,00	0,00	0,00	0,00	0,00	0,00



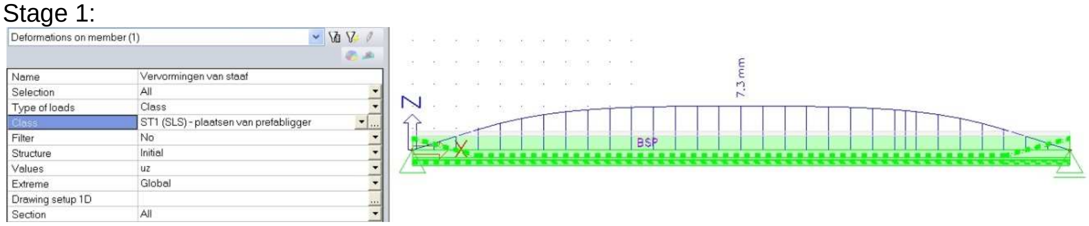
Case	BSP	Tendon	x [m]	Stress after anchoring / transfer [MPa]	LED [MPa]	LCS [MPa]	Lmin [MPa]	Lmax [MPa]	MinStress [MPa]	MaxStress [MPa]
ST5 (ULS)	BSP	Streng1	25,000	1418,73	-35,05	-97,62	0,00	0,00	1286,05	1286,05
ST5 (ULS)	BSP	Streng1	0,000	0,00	0,00	0,00	0,00	0,00	0,00	0,00

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

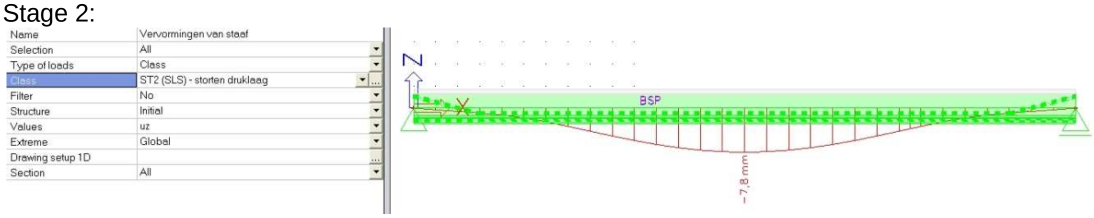
Deformations

The deformation of the pretensioned prefab beam in the different Construction stage is also given in the results menu. Below the results for the deformation in the 5 construction stages is shown:

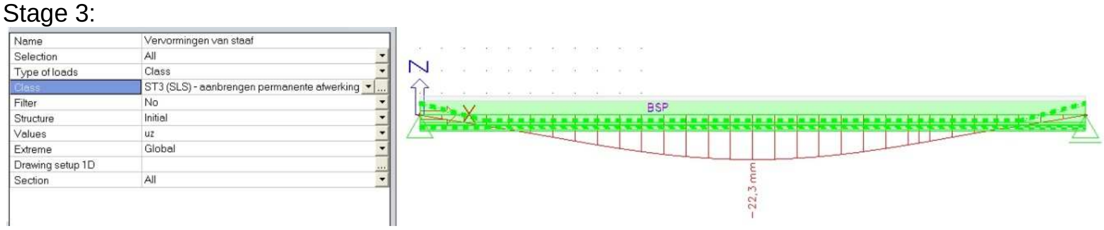
Stage 1:



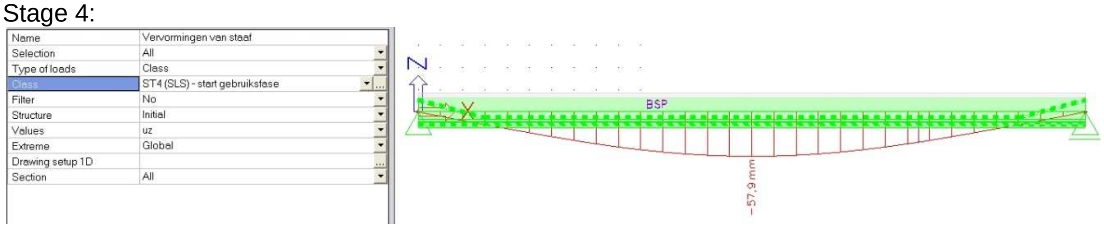
Stage 2:



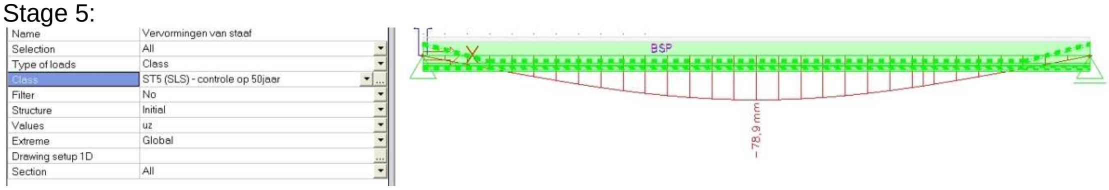
Stage 3:



Stage 4:

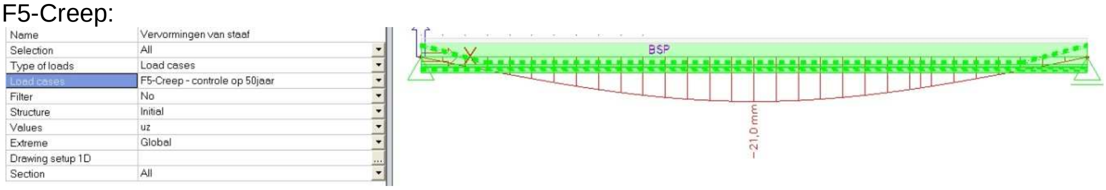


Stage 5:



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:



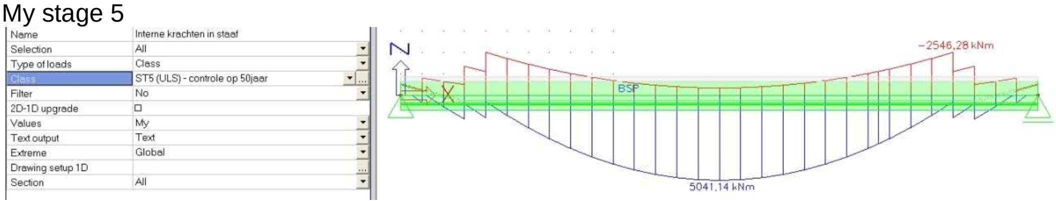
Note that also a relative deformation could be given for checking.
This relative deformation could be checked manually or by means of an imbedded excel check.

Internal forces

The (enveloppe) moment lines for stage 1 and stage 5 are shown below:

My stage 1

My stage 5

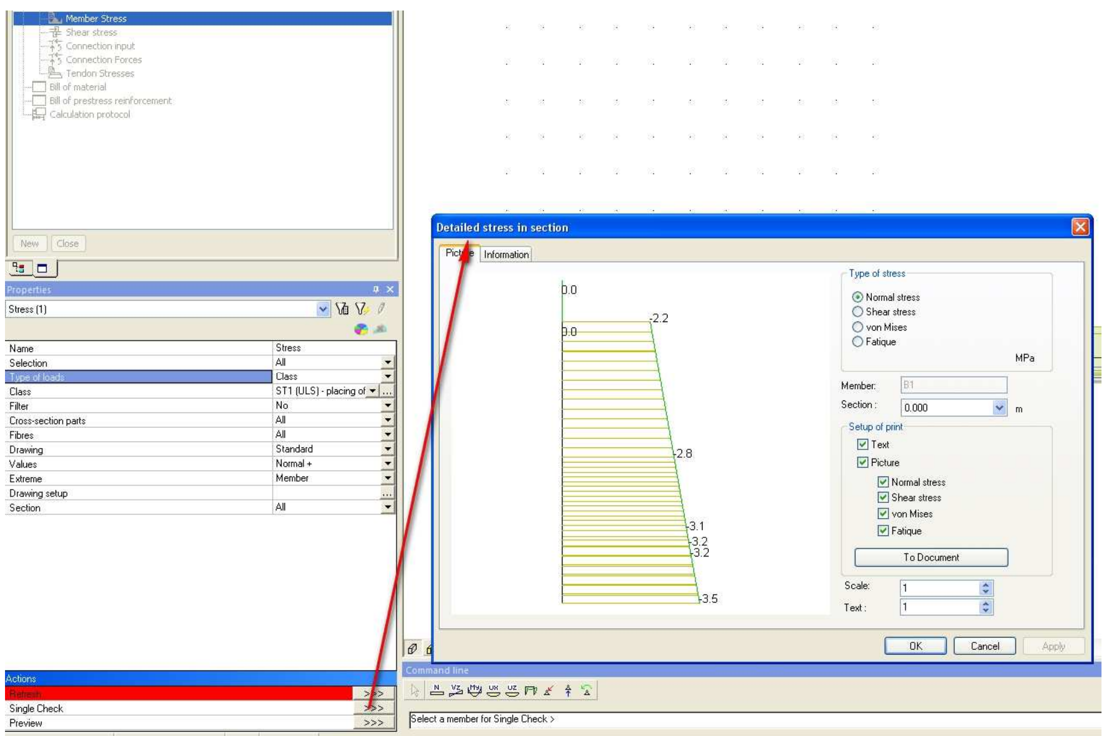


The maximal resistance moment can be viewed in the concrete menu.
This means a capacity check of the internal forces will be possible in the concrete menu.

Stresses

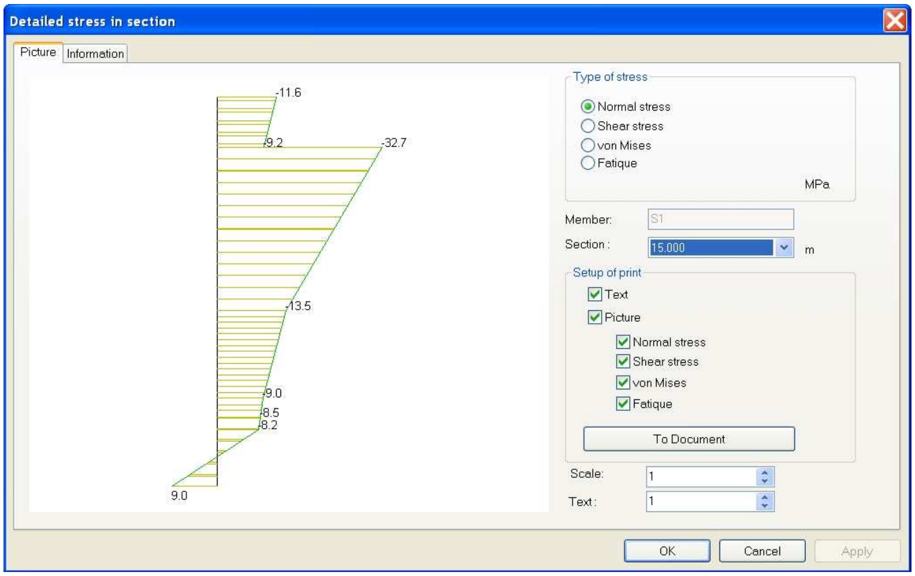
The stresses can be viewed over the height of the cross-section.
This is interessant to check if there would be tension on the top fibres in the end sections of the beam.

The stress in the begin 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:

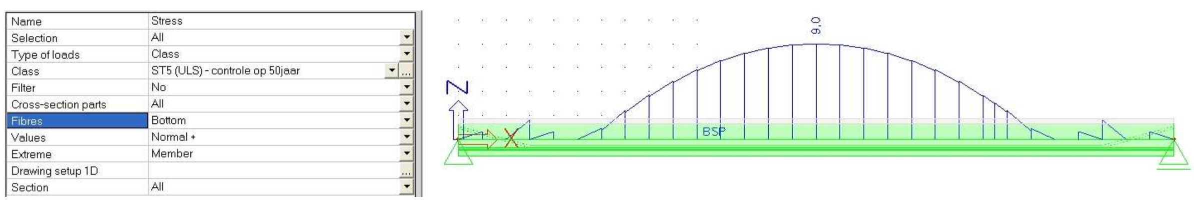


Notice the jump in stress between the two phases of the cross-section.

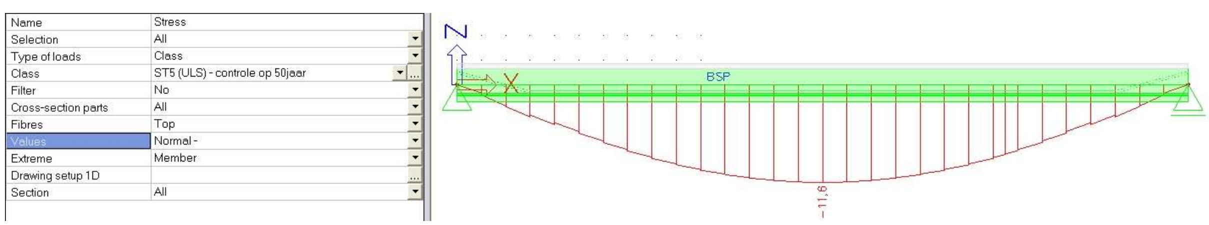
It is also possible to show the stress over the length of the beam.
Hereby can be chosen to show the stress on the bottom or the top fibre.

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

Tension stress at the bottom fibre in stage 5:



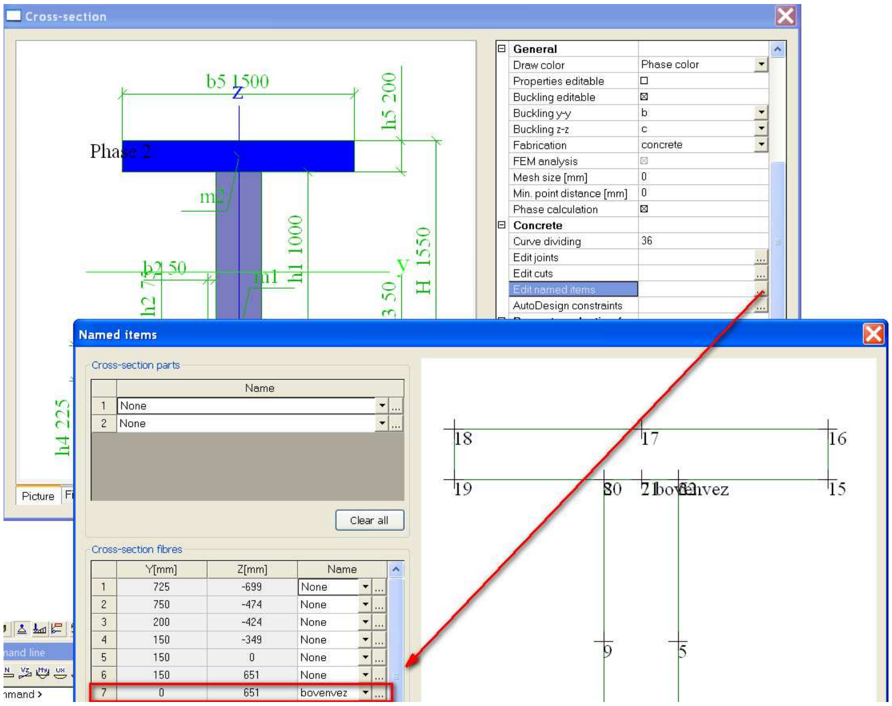
Compression stress at the top fibre in stage 5:



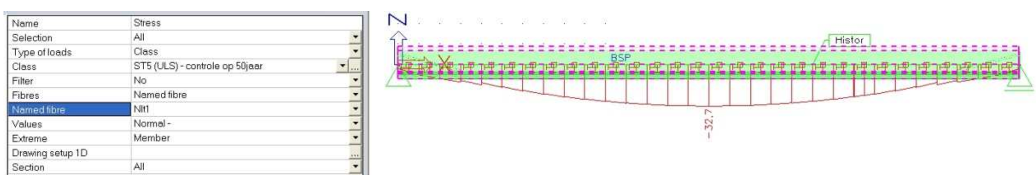
Note that the top fibre corresponds to the top of the second phase of the css.

To show the stress on the top fibre of the precast beam (first phase of css), the user could manually predefine a named fibre.

This is done in the properties of the cross-section:



Afterwards stresses can be shown over this named fibre.
In the picture below the stress in stage 5 on the top fibre of the precast beam is shown:



5_Checks in the concrete menu

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 4 (results menu) 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.

Also the design of vertical shear reinforcement Ass and the design of shear reinforcement Asj in the construction joint will be treated.

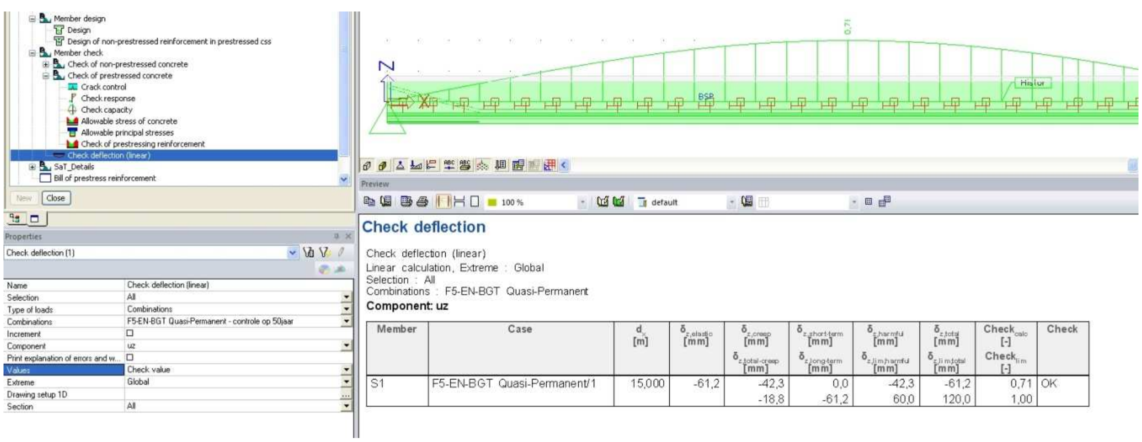
At the end also the check for decompression will be illustrated. This is a new feature since version SCIA Engineer 2011.

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

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 harmfull deflection (which is the deflection caused by creep).

Below the results are shown for the deflection check in the final stage:

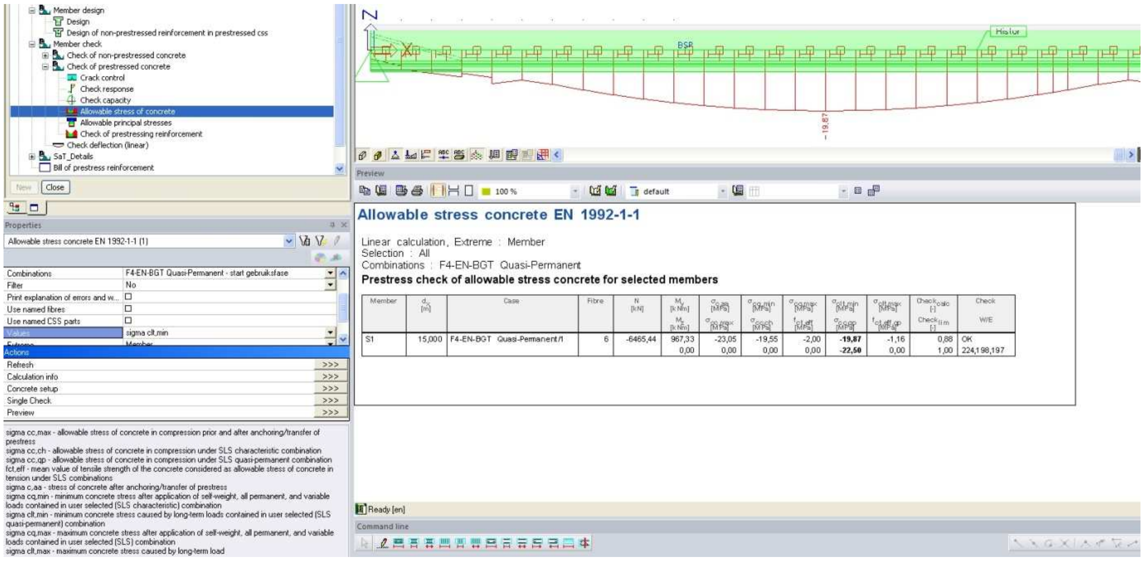


Member	Case	d_x [m]	$\delta_{z,elastic}$ [mm]	$\delta_{z,creep}$ [mm] / $\delta_{z,total-creep}$ [mm]	$\delta_{z,short-term}$ [mm] / $\delta_{z,long-term}$ [mm]	$\delta_{z,harmful}$ [mm] / $\delta_{z,lim,harmful}$ [mm]	$\delta_{z,total}$ [mm] / $\delta_{z,lim,total}$ [mm]	$Check_{calc}$ [-] / $Check_{lim}$ [-]	Check
S1	F5-EN-BGT Quasi-Permanent/1	15,000	-61,2	-42,3 -18,8	0,0 -61,2	-42,3 60,0	-61,2 120,0	0,71 1,00	OK

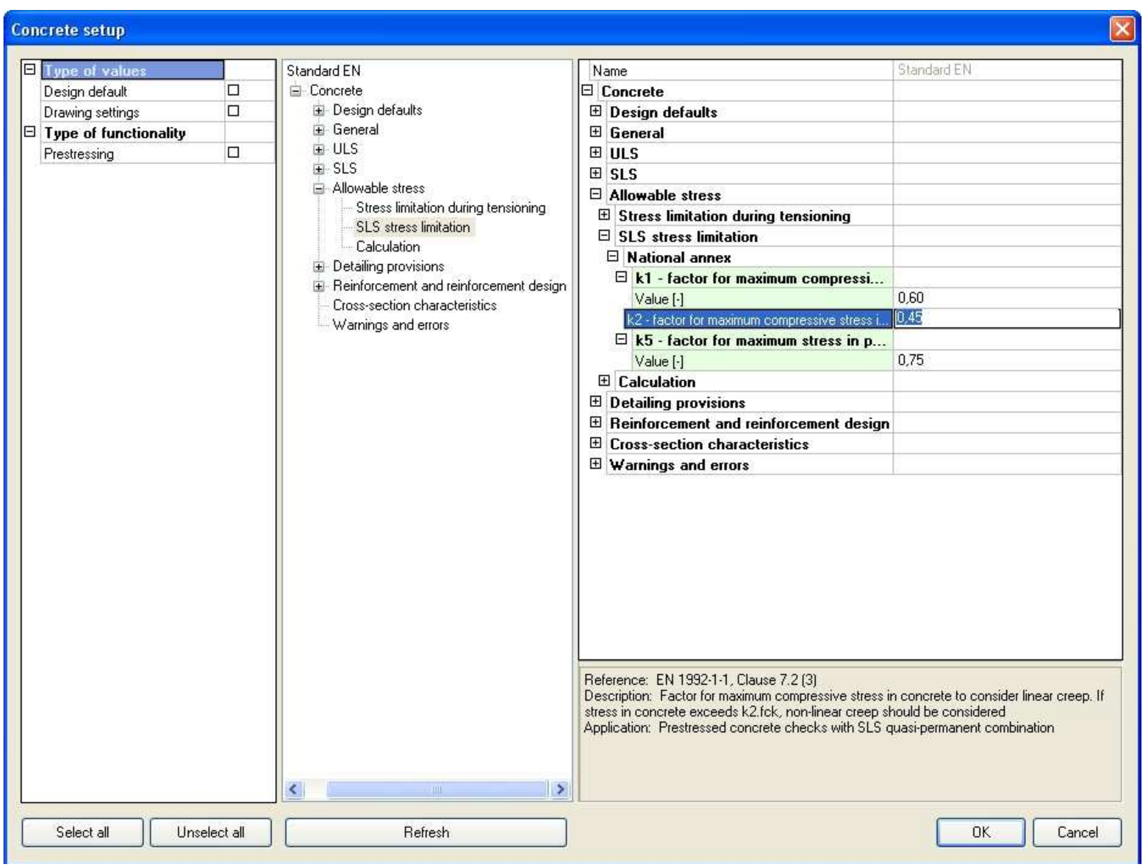
Check of allowable concrete stress in compression

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 prestress are not yet complete.

The result for the SLS quasi perm combination in stage 4 is shown below:

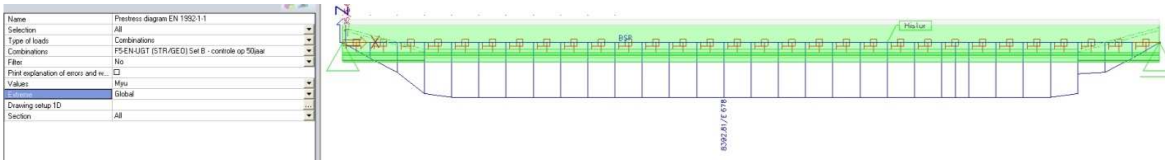


The maximum allowed concrete compression stress (sigma,lt,min in SCIA Engineer) is calculated according to art 7.2 (3) of EN 1992-1-1. The k2 factor used in that formula can be found in the concrete setup:

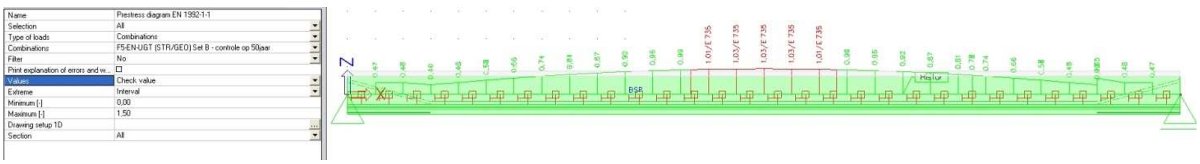


Capacity check

The resistance moment My is shown below:



This results in the following capacity check:

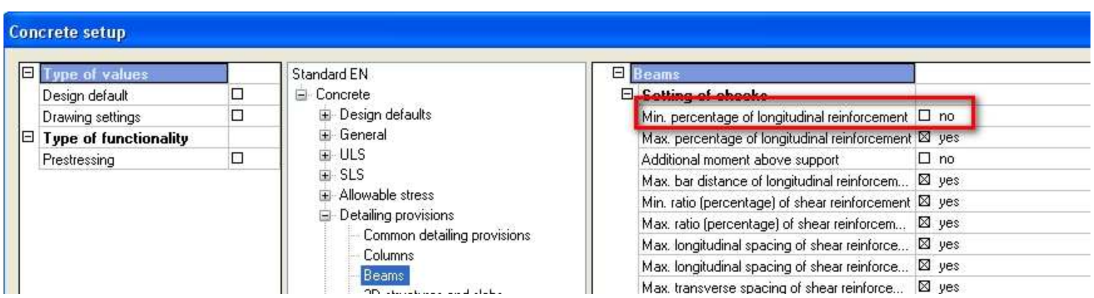


We will not consider the end zones here. It is due to numerical effects, check of tension stress at top fibre in first stage was already performed and this was ok.

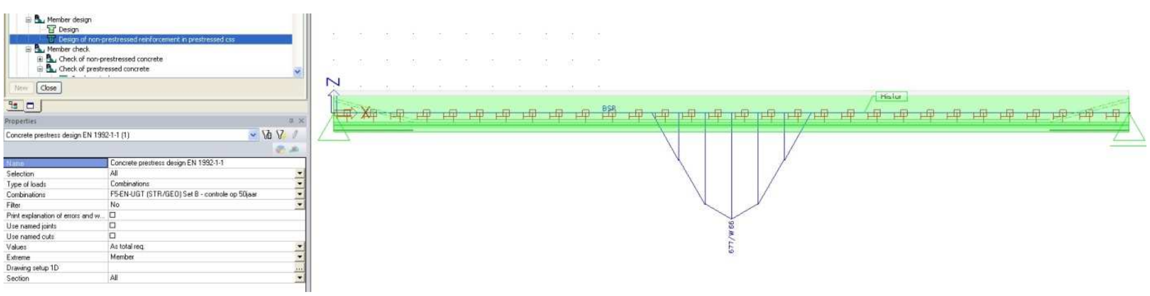
The unity check in the midsection of the beam indicates that the moment My passes the resistance moment My.
Two solutions could be possible.
The user could increase the prestress reinforcement by adding more strands.
Or the user could design the additonal passive reinforcement that is required in this case.

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 is shown below:

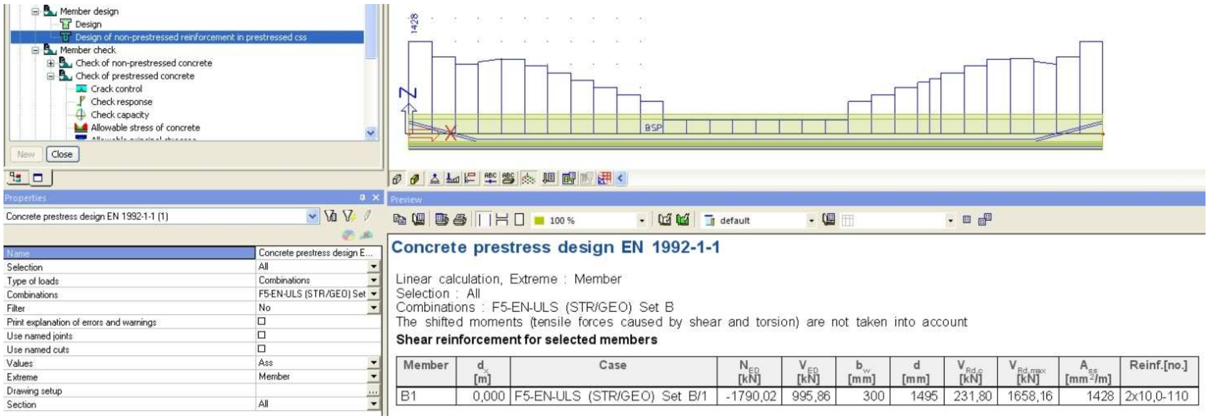


Design of vertical shear reinforcement Ass

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

The required shear reinforcement is shown below:

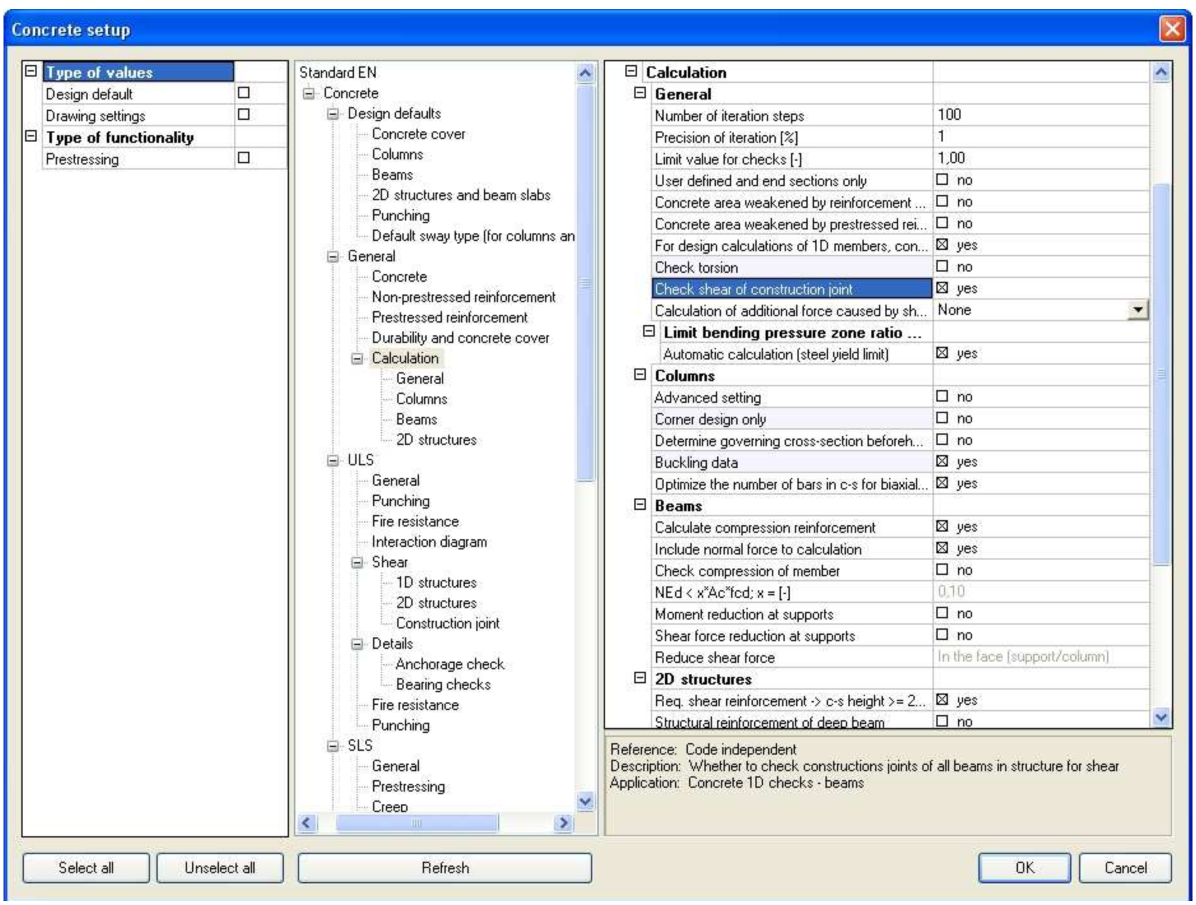
:



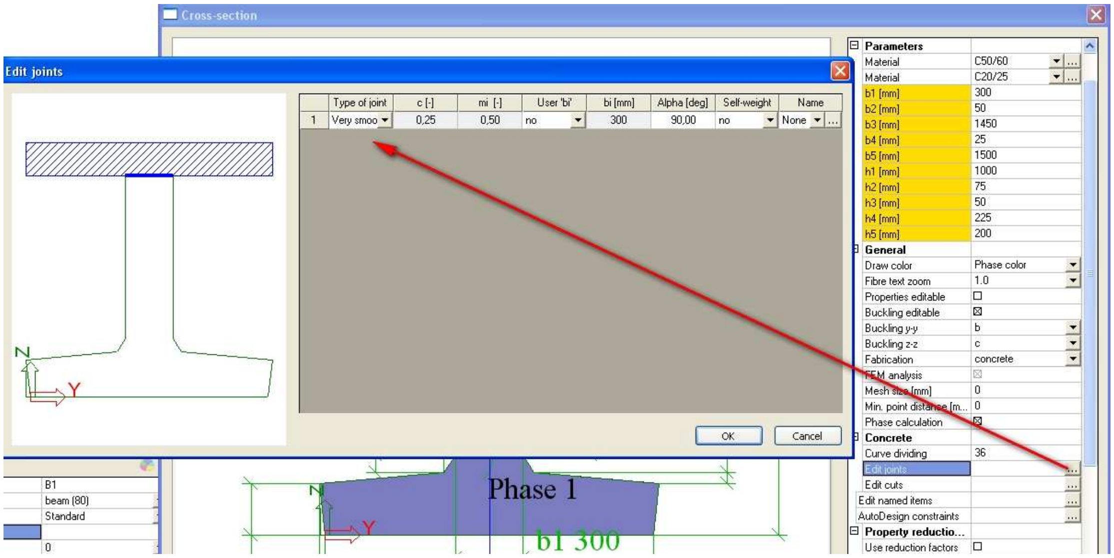
Member	d_x [m]	Case	N_{ED} [kN]	V_{ED} [kN]	b_w [mm]	d [mm]	$V_{Rd,c}$ [kN]	$V_{Rd,max}$ [kN]	A_{ss} [mm²/m]	Reinf.[no.]
B1	0,000	F5-EN-ULS (STR/GEO) Set B/1	-1790,02	995,86	300	1495	231,80	1658,16	1428	2x10,0-110

Design of shear reinforcement in construction joint Asj

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

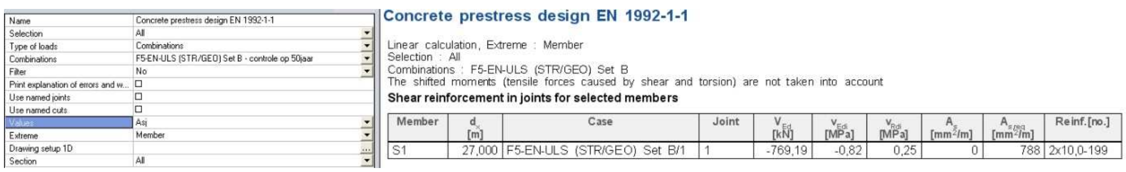


The roughness of the joint can be defined in the section properties:



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:



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

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

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

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

where:

c, μ are factors depend on roughness of joint

$\mathbf{f_{ctd}}$ 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 $\mathbf{0,6 * f_{ctd}}$

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 $A_i = b_i$ * 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 :

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

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

Value Beta 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 $v_{Edi} > v_{Rdi.}$

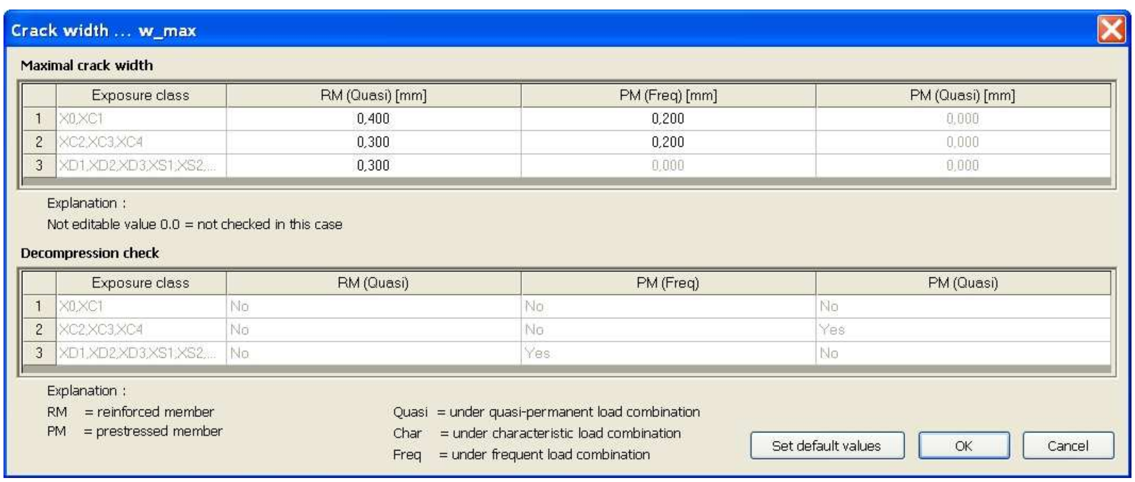
The formula for the required shear reinforcement Asj is shown below:

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

Decompression check

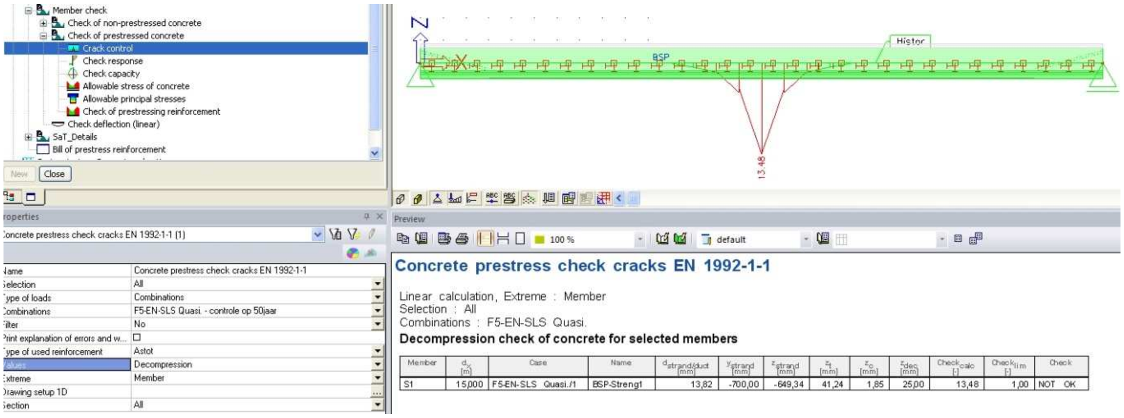
This SLS check is described in art 7.3.1(5) from EN 1992-1-1.

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

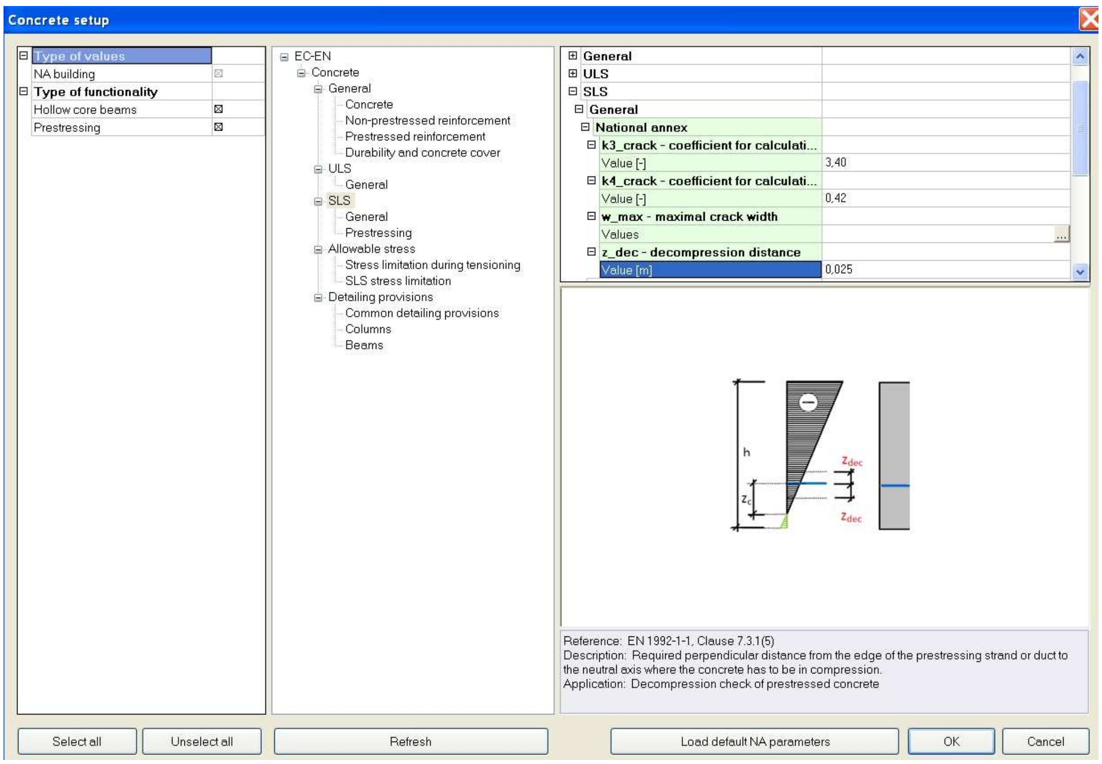


In this example we are in default environmental class XC3.
This means we will need to check for the SLS quasi-permanent combination.

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:



As z_c is lower then z_{dec}, the decompression check will not pass.



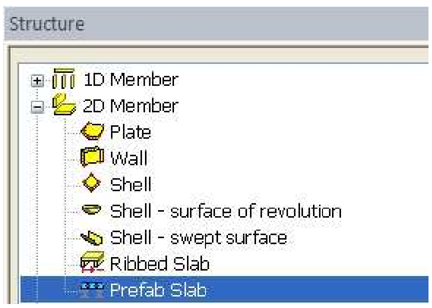
2D/1D upgrade

The purpose of this example will be to illustrate the 2D->1D upgrade functionality.
This functionality allows to built 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.

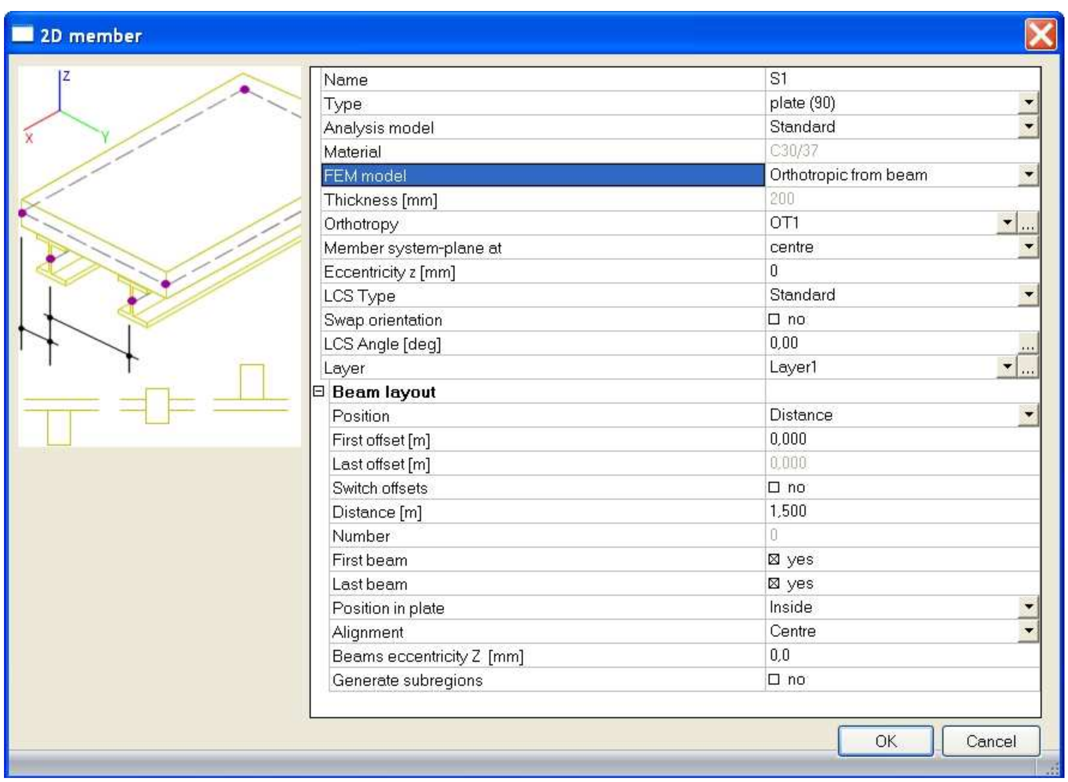
1_ Modeling of a prefab slab model

Open a new project in a general xyz model (or plate xy environment for precast edition).

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

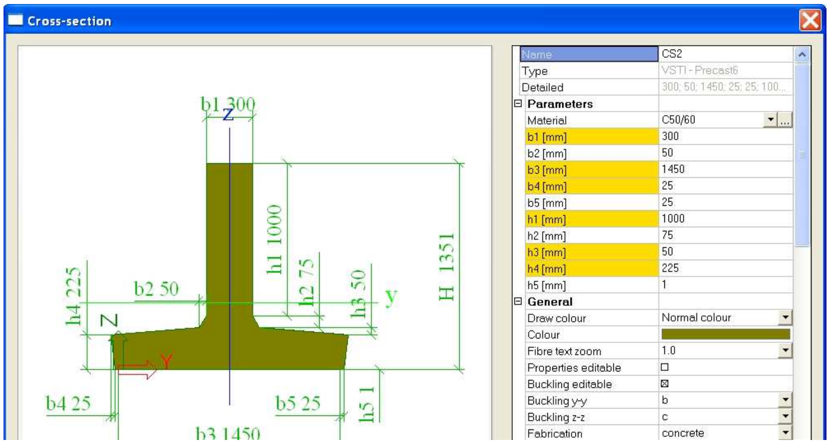


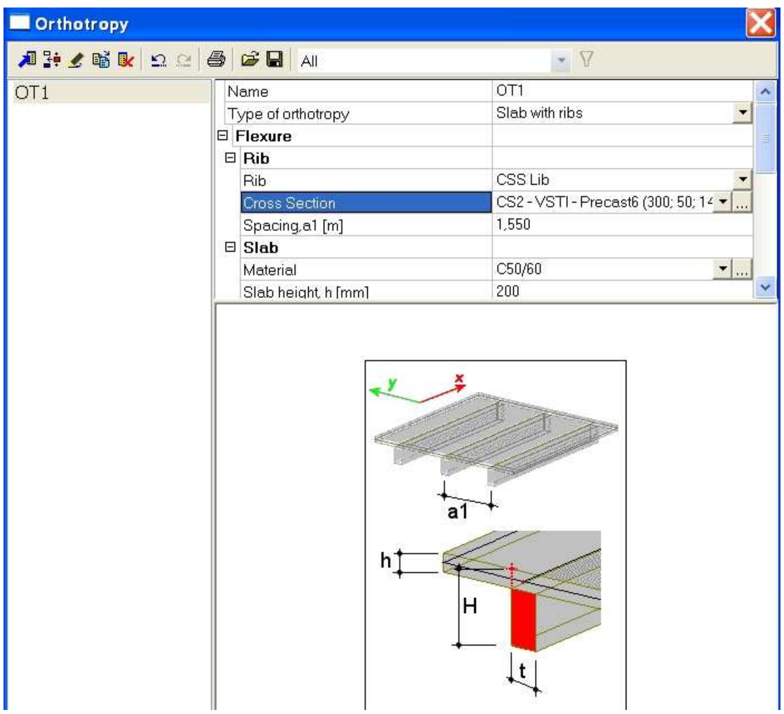
The prefab slab will be considered as a 'normal' plate in this 2D element model.
The beam layout that can be defined in the prefab slab properties will only be important in a later stage for the 1D export. This means that the beams that are added in this way will have no influence at all on the stiffness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



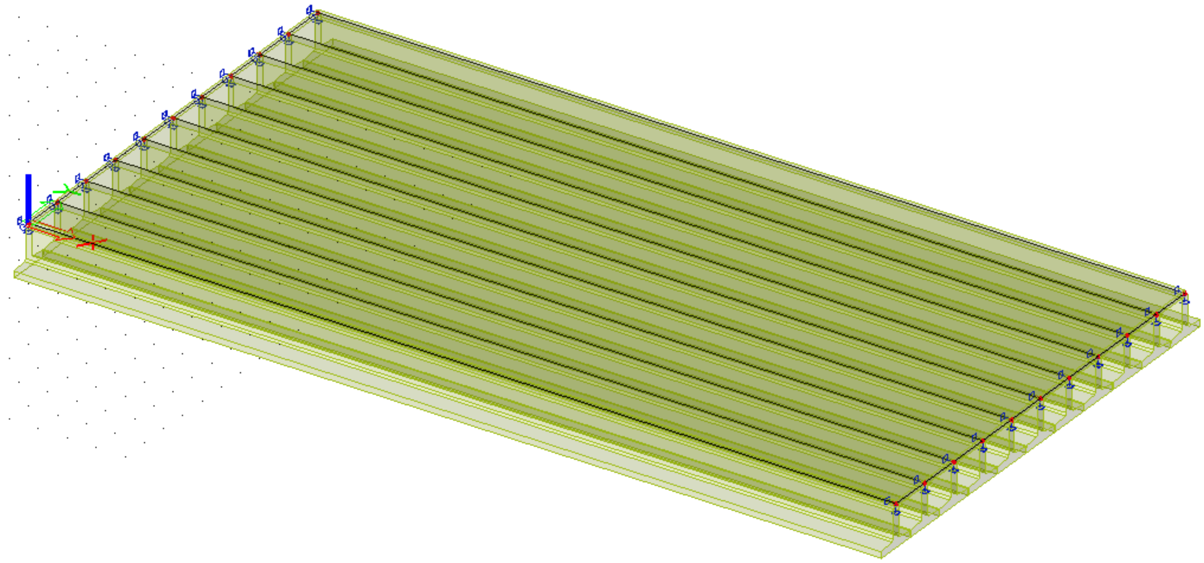
To assign a correct stiffness to the plate, which is of course important for the load distribution, the FEM model of the plate will be set to orthotropic. As type of orthotropy the OT type ‘slab with ribs’ could be used. The stiffness will then be calculated based upon the orthotropic parameters.
Only 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.

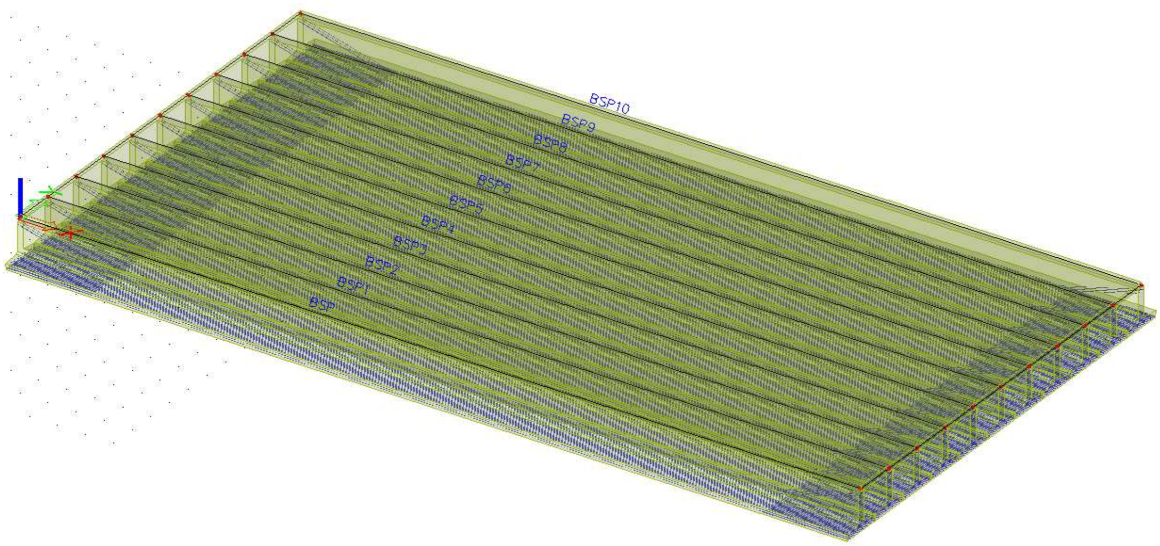




The prefabplate will be modelled with a total width of 15m and a span of 30m. The supports have to be applied as nodal supports on the beam edges.



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 elementmodel where it will be taken into account in the analysis.
The same beam strand patterns as the ones used in the example above will be used.

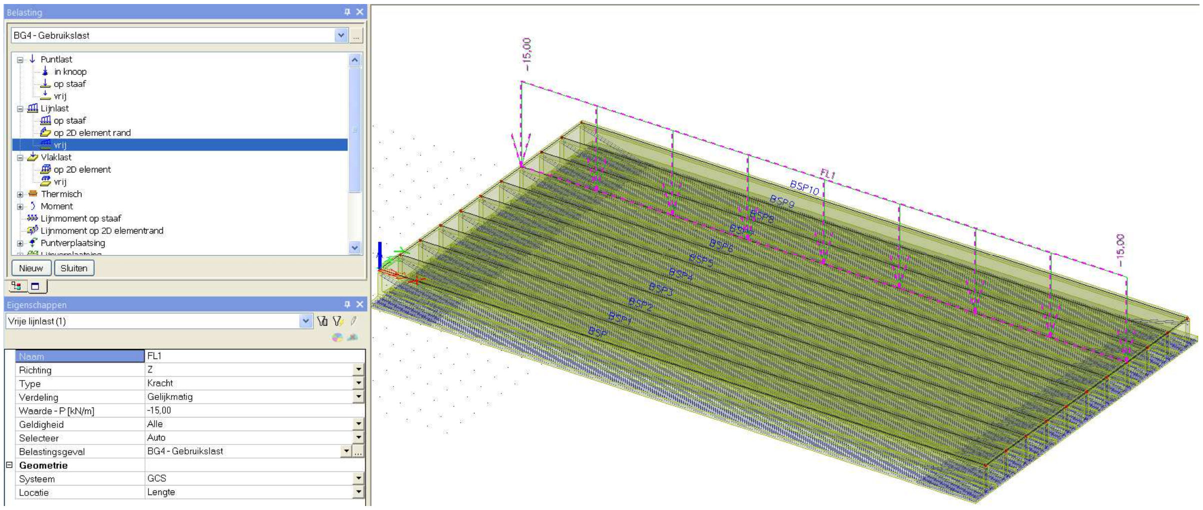


2_ Application of the loading

The first load case that was created is the load case LC1-Prestress.
The second load case will be LC2-Self weight.
This second load case will be analysed later on in the 1D element model, so we will not consider the internal forces for self weight in this model.

The third load (LC3-Permanent) will be a surface load of 5kN/ m^2.

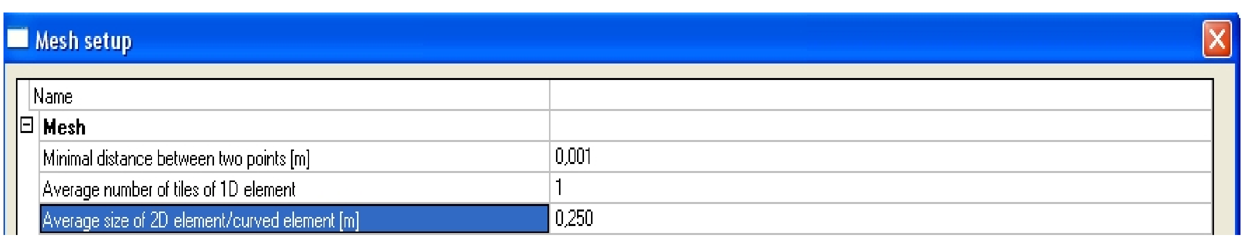
As fourth load case (LC4 – Service load), a line load will be applied on one side of the deck.
This line load is only applied above 1 beam, but because of the transversal stiffness there will be a transfer of load to multiple beams. The calculation of the spreading of this load is the whole purpose of this 2D prefab plate model.



3_ Calculation

A linear calculation of these load cases will be made.

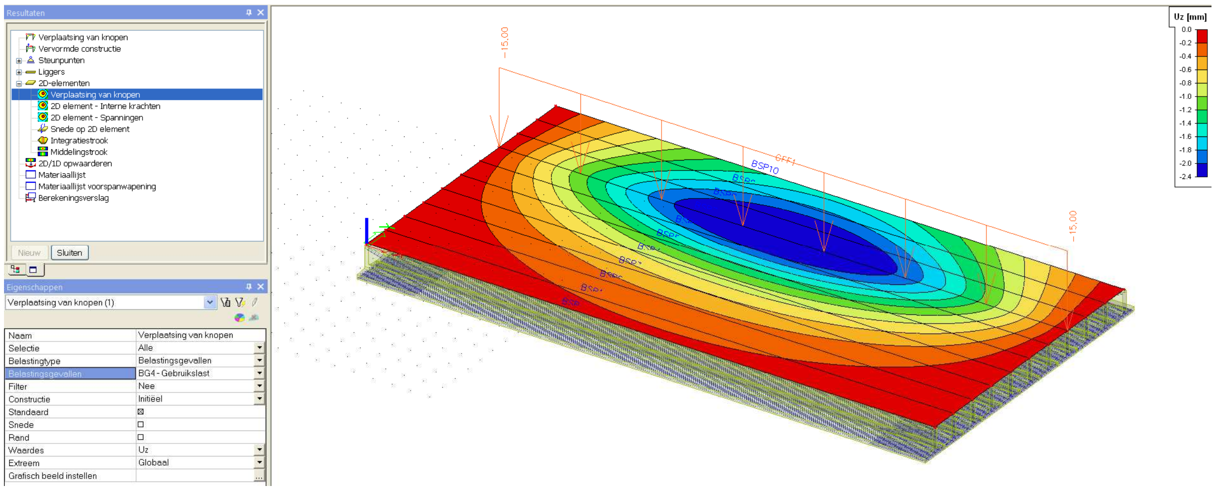
It is advised to have a look at the mesh setup first, since the default mesh size of 1m may be to rough.
A mesh size of 0,25m is set in this example.



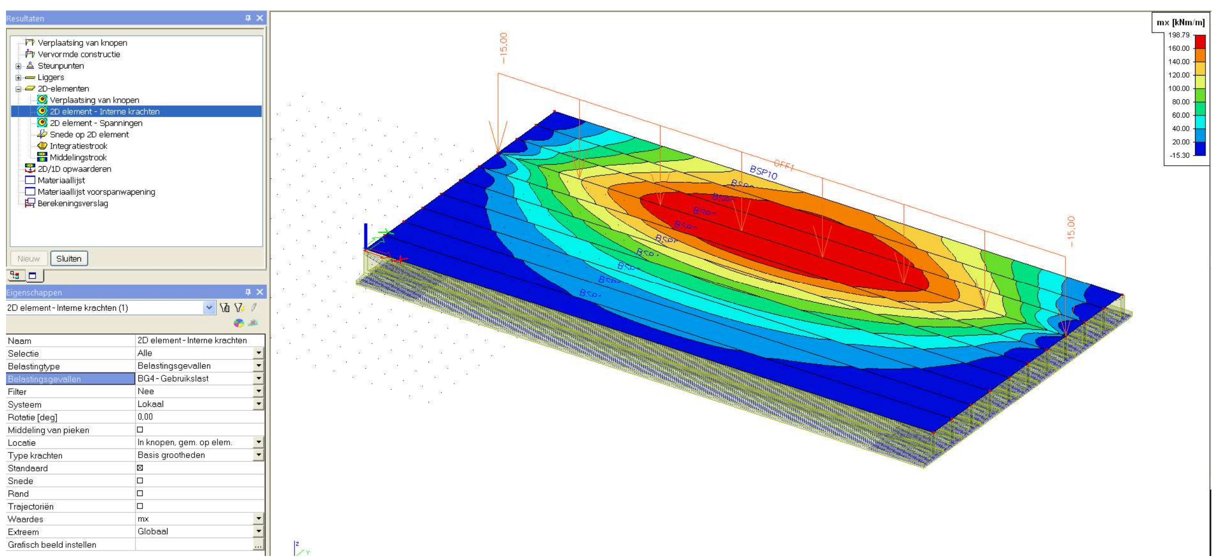
4_Results

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:

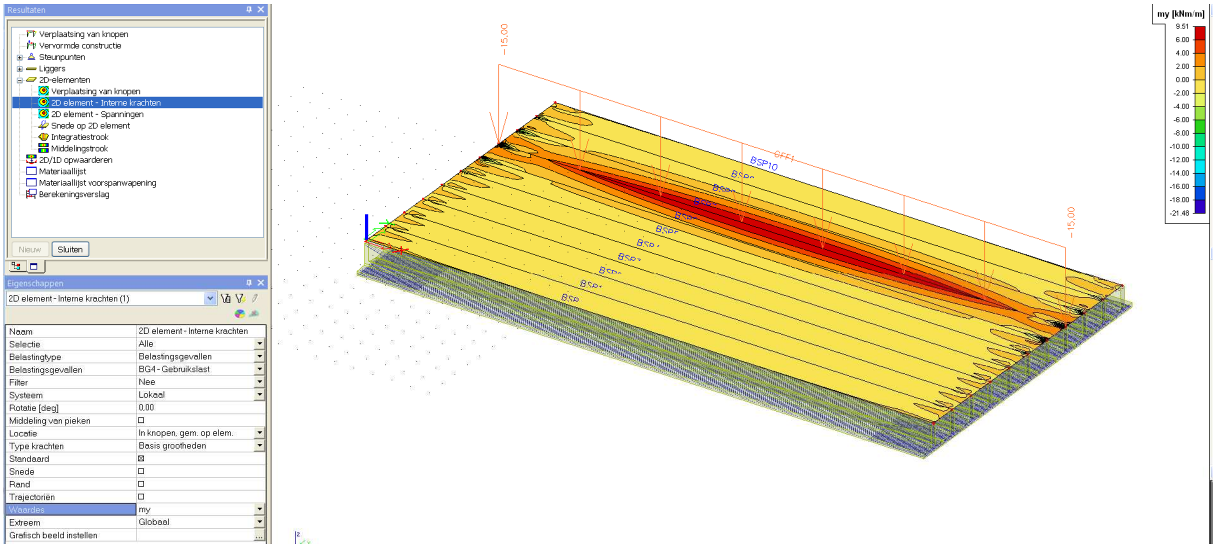


The moments for load case 4 are shown below:



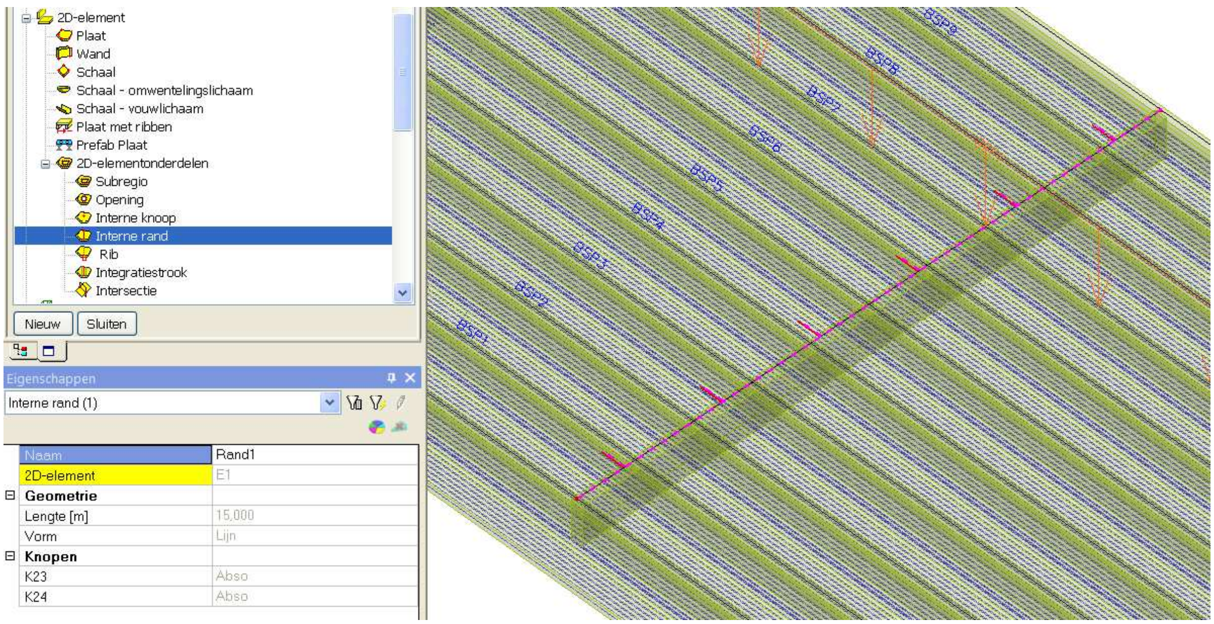
There is a clear spreading noticeable. The local line load of 15kN/m will be spread over multiple beams. This spreading is caused by the transversal stiffness of the prefab plate model. This transversal stiffness will be delivered by the in situ topping of the bridge deck (and not by the prefab beam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

There is also a moment my in transvere direction noticeable:

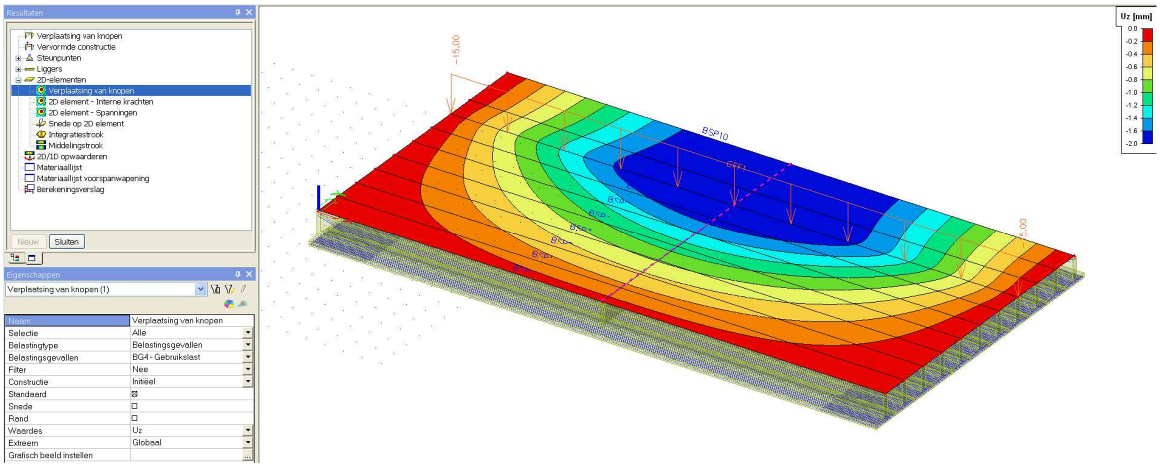


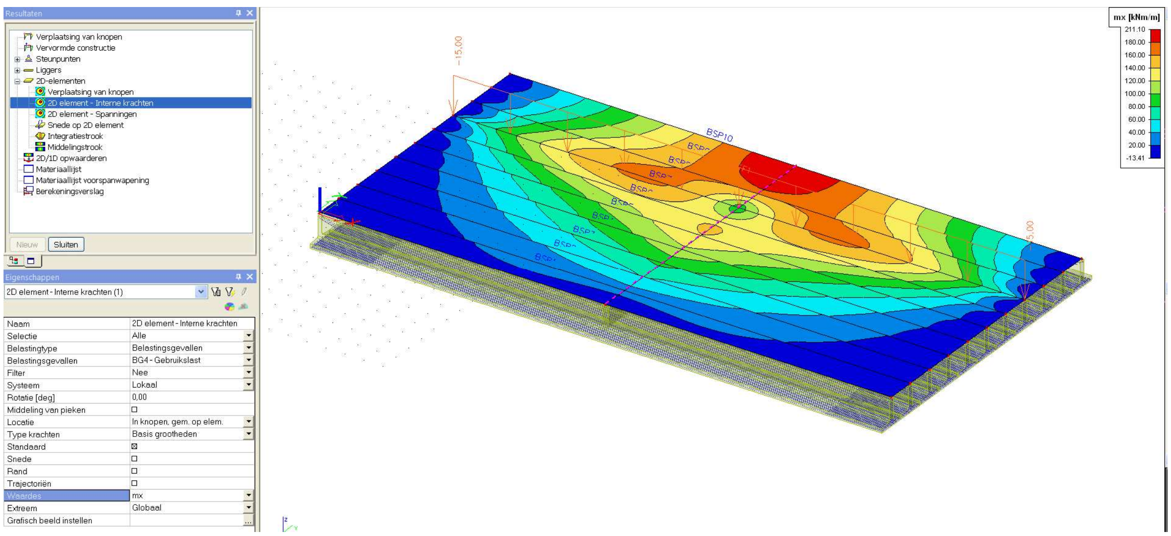
To increase the spreading of the loads, a transverse beam could be added to the model.

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



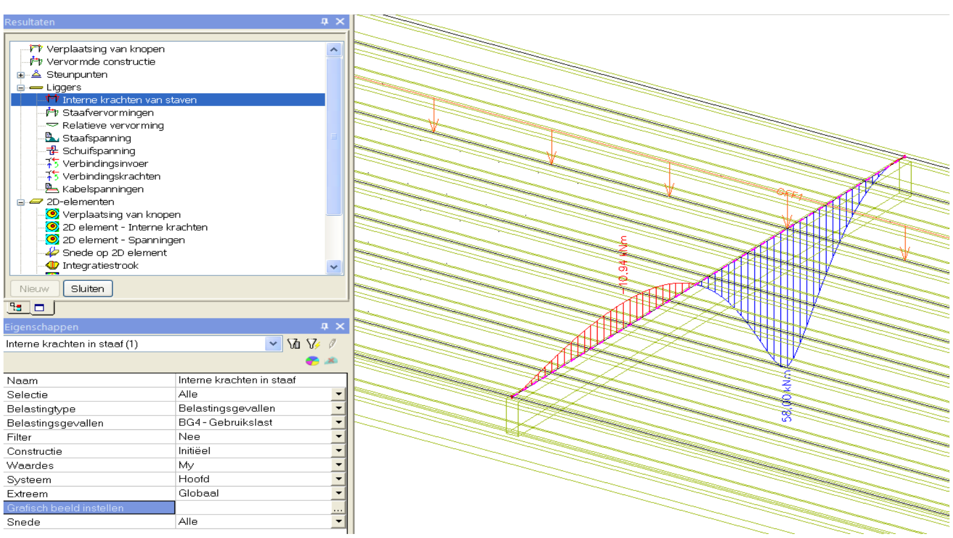
The effect on the displacements and moments is shown below:





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:

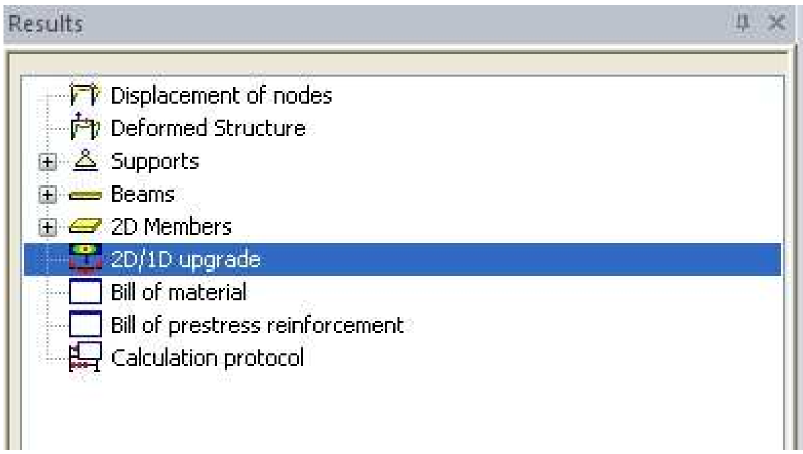


5_2D->1D upgrade

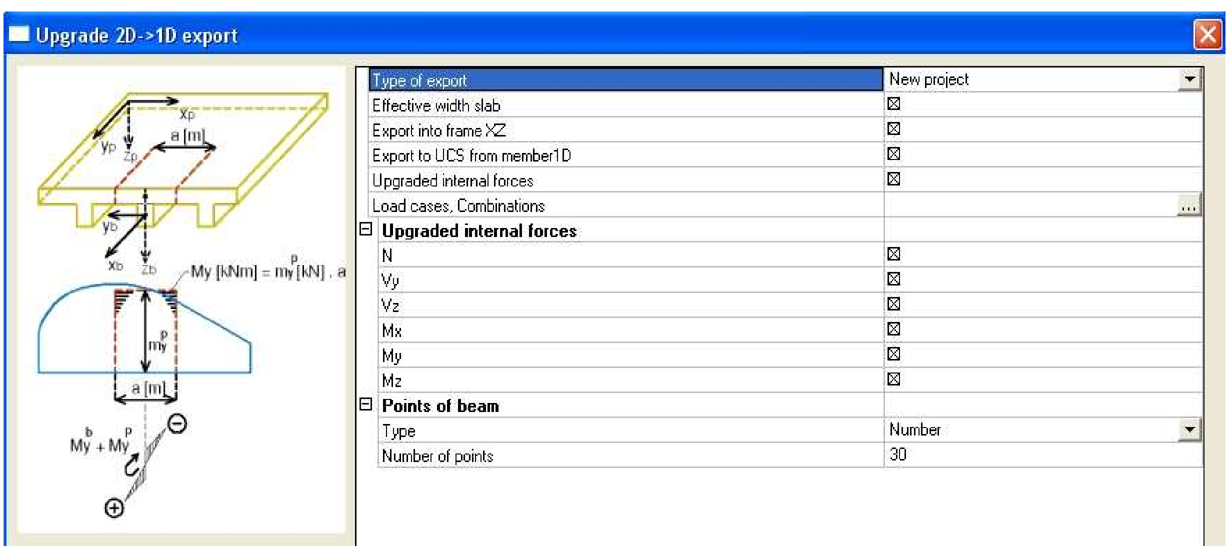
As it was mentioned before, the purpose of this 2D prefab plate model was to make an analysis of the spreading of the loads. The dimensioning of the prefab beam elements should be done in the 1D element model.

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.



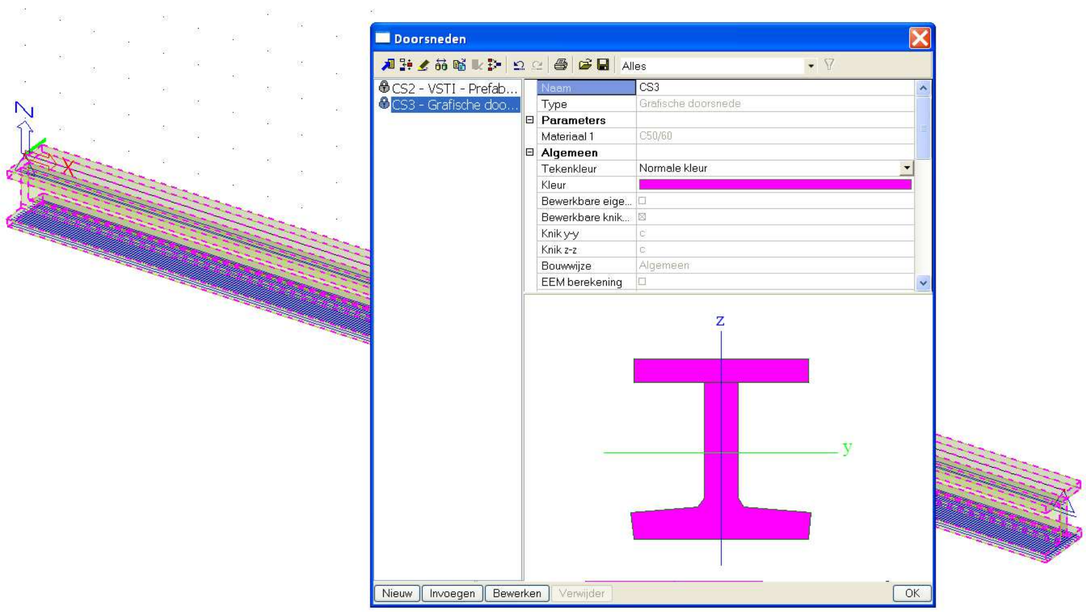
For the 2D->1D export the following setting can be defined:



The width of bridge deck that is taken into account in the export is defined by the effective width (which could be enterred 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.
The user could define also which internal forces and for which load cases they should be exported.
The number of points on beam defines the precision of these internal forces.

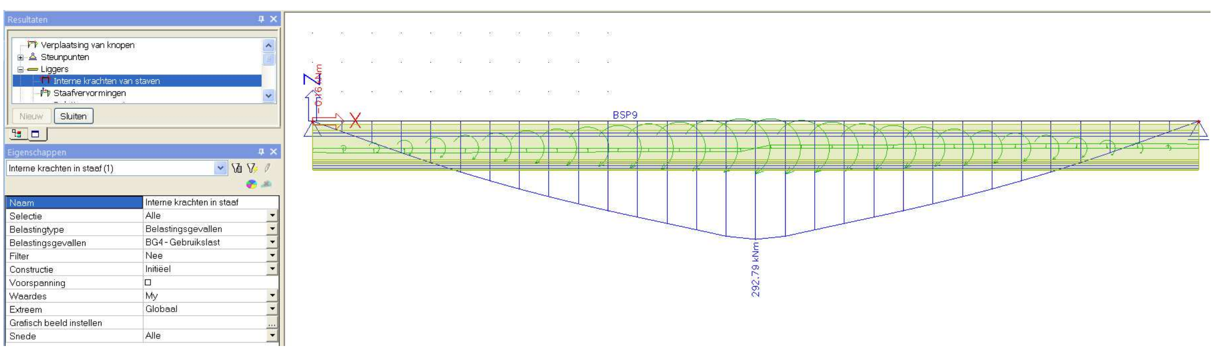
A new option since SCIA Engineer 2011 is to export to a template instead of a new project.
Exporting to a template will open a user template (with parameters) instead of just a new project.

The exported 1D element model is shown below:



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

Also the internal forces are 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. Herefore reference is made to the 1D element model treated in the first chapter.