



# Advanced Expert Trainig Pre-tensioning (Precast)

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2021 SCIA nv. All rights reserved.

## Table of Contents

Table of	Conte	ents	3	
Introduct	ion		4	
Chapter 1	1:	Pre-te	ensioned precast beam5	
1	1.1.	Input	geometry and prestress	5
	1.1	1.1.	Project settings	5
	1.1	1.2.	Input geometry	6
	1.1	1.3.	Load cases + loading	7
	1.1	1.4.	Bore hole pattern	7
	1.1	1.5.	Pre-tensioned tendons	10
1	.2.	TDA C	Construction stages (and loading)	15
	1.2	2.1.	Construction stages setup	15
	1.2	2.2.	Adding construction stages	16
	1.2	2.3.	Defining boundary conditions	18
1	I. <b>3</b> .	Calcu	lation	20
1	I. <b>4</b> .	Resul	ts	21
	1.4	4.1.	Tendon stresses	21
	1.4	1.2.	Deformations	22
	1.4	4.3.	Internal forces	23
	1.4	4.4.	Stresses	23
1	l.5.	Check	ks in the concrete menu	27
	1.5	5.1.	Deflection check	27
	1.5	5.2.	Check of allowable concrete stress in compression	28
	1.5	5.3.	Capacity check	29
	1.5	5.4.	Design of passive reinforcement As	29
	1.5	5.5.	Design of vertical shear reinforcement Ass	30
	1.5	5.6.	Design of shear reinforcement in construction joint Asj	30
	1.5	5.7.	Decompression check	32
Chapter 2	2:	2D/1D	upgrade35	
2	2.1.	Model	lling of a prefab slab model	35
2	2.2.	Applic	cation of the loading	39
2	2.3.	Calcu	lation	40
2	2.4.	Resul	ts	41
2	2.5.	2D →	1D upgrade	44

## 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.

Project data						×
Basic data Fu	nctionality Actions	Unit Set Protection				
	Data			Material		
111	Name:	-		Concrete	$\checkmark$	
				Material	C50/60	·
	Part:	-		Reinforcement m	B 500B	·
				Steel		
	Description:	-		Masonry		
	Author			Aluminium		
				Timber		
	Date:	-		Steel fibre concrete		
				Other		
	Structure:	In Frame XZ		Code		
(Final Street		+		National Code:		
	Post processing	No. 16 and older	-	EC - EN	-	
	environmeni					
	Model:	😝 Construction stages	•	National annex:		
an over the		-		Standard EN		
				Ok		Cancel

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

	General	_		Detailed	
11	Property modifiers		4	Subsoil	
1	Parametric input			Pad foundation check	
	Climatic loads		4	Concrete	
	Mobile loads			Fire resistance checks	
	Dynamics		Ho	Hollow core slab checks	
	Stability				
F	Nonlinearity				
	Structural model				
	IFC properties				
	Advanced concrete checks	✓			
	Prestressing	$\checkmark$			
	Bridge design				
	Excel checks				
	Document				
and the second second					
-					

## 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.

New cross-section		×
Available groups	Available items of this group	
Available groups Concrete Geometric shapes Numerical General Precast Bridge	Available items of this group       Items in project         Image: Control of the second	
Precast3	Profile Library filter Add Close	

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.

Advanced Expert Training – Pre-tensioning (Precast)



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:



🔳 Edit	database													>	×
<b>,1</b> 👫	🥖 💕 🛛	k 🖸	a 🕘	🎬 🖻	; 🔒										
BH		Ν	lame					BH							
		C	ross-sectio	n nam	e			CS1							
								-							Ξ
			•			•		•		•	•	•	•		
			•	•••	•••	•   •	• •	·	••	•	•	•	•		
			:	::	::	: I:	: :	:1:	::	:	:	:	:		
			•	• •	• •	• •	• •	• •	• •	٠	•	•	•		
			•	•••	•••	•   ·	•••		••	•	•	•	•		
			:	::	::					:	:	:			
			•			.).	• •	.ا	• •	•	•	•	•		
			م		_	-	• •	• )	-	-	-	-	•		
			1:	::	::	•				:	:	:	:		
			Ľ				-8	-		-	-	-			
New	Insert	Edit	Delete											OK	

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.

10-	O4 ↑ O7
20	
_0£ !	50 <b>@</b> 6
Position	5 • 6 - Bottom centre +
Position Dy [mm]	5 • 6 6 - Bottom centre • 0
Position Dy [mm] Dz [mm]	6 - Bottom centre → 0 0
Position Dy [mm] Dz [mm] Cover	6 - Bottom centre • 0 0

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.

B1-1(0,000 m) Add.			
Name	B1-1		
Position (x) [m]	0,000	Strand pattern - x	×
		Name B1-1	^
		Asymmetrical s 🗌 no	
		Coord. definition Abso -	
		Position (x) [m] 3,000	۷.
		OK Cancel	

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.

trand property	Geometry
	V
Group	1
Material	Y1860S7 +
Position in hole	e Centre 🔹
Fixed	
Draped	
Asymmetrical	
Debonding le	No
Stressing seq	1 +
Type of stressi	. Type 3 🔹
Stress during	1440,00
Duration of k	300,00
Initial stress [	1440,00
Anchorage se	6,00
Determine tra	By user inpu 👻
Transmission I	. 1,00
Distance betw	. 0,500
Position	
X [m]	3,000
Drape	
Dy [mm]	0
Dz [mm]	-800
Remove	>>>

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.

Pre-tensioned tendon (1)   Pre-tensioned tendon (1)  Pre-tensioned ten
👟 🍣 🗞
DCD.
Name BSP
Edit strand patterns
Sectional strand patt SSP3 +
LC prestress LC1 - Prestress +
Stressing beds SB +
Strand patterns
B1-1 SYM. [m] 3,000
4 Debonding leng
L1 [m] 3,00
Member B1

## 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".

Tree 🔻 🕂 🗙			
B Main Construction stages X	Construction stages setup		$\times$
ST1 *	News		
	Trac	Time dependent analysis	
I Setup	Structure	Building	
Time axis	structure	building	
Members      Supports	Load factors(Code independent combinations only)		
Delete input data of stage	Common min [ ]	0.000	
III III	Gamma min [-]	1,000	
	Gamma max [-]	1,000	
	Prestressed load cases	0.000	
	Gamma min [-]	1,000	
	Gamma max [-]	1,000	
	Long-term part of variable loads	0.200	
	Factor Psi [-]	0,300	
	<sup>∡</sup> IDA		
	4 Load factors for generated load cases	1.000	
	gamma-creep min [-]	1,000	
	gamma-creep max [-]	1,000	
	A Time - History		
	Number of subintervals	1,0	
	Ambient humidity [%]	70,00	
	Automatic calculation of subintervals	L] no	
	4 Local time axis		
	Time of casting [day]	-1,00	
	Time of curing [day]	3,00	
	Duration of curing of composite parts of cross-section [day]	3,00	
	Line support (formwork)		
	Time of releasing of displacements in X direction [day]	14,00	
	Time of releasing of displacements in Z direction [day]	14,00	
	Generate output text file		
	4 Results		
	Name of gener. ultimate combination (max)	F{O}-MAX	
	Name of gener. ultimate combination (min)	F{O}-MIN	
	Name of gener. creep load case	F{O}-Creep	
	Name of gener. serviceability combination	F{O}-SLS	
	Name of gener. code combination	F{O}-{CODE}	
		OK	Cancel
		- OK	

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.

Construction stages			×
🏓 🤮 🇶 🖳 🔉	2 😂 🗛	• 7	
ST1 - placing of prefa	Name	ST1	
ST2 - casting in situ c	Order of stage	1	
ST3 - finishes	Description	placing of prefab beam	
ST4 - start of service st	Global time [day]	0,00	
ST5 - end of service st	Number of subintervals	10	
	Ambient humidity [%]	70,00	
	Last construction stage		
	Load case permanent or long-term		
	Load case	LC2 - Placing of concrete	beam 👻
	Load case prestress		
	Load case	LC1 - Prestress	·
	Type of generated combinations	All code dependent	*
	Structure	Building	
	Actions		
	Variable load cases and type of generated combinat	tions	>>>
New Insert Edi	it Delete		Close

Construction stages					×
🎵 🤮 🗶 💽 🗠	AI -	7			
ST1 - placing of prefa	Name	ST2			
ST2 - casting in situ c	Order of stage	2			
ST3 - finishes	Description	casting in situ concrete			
ST4 - start of service st	Global time [day]	20,00			
ST5 - end of service st	Number of subintervals	10			
	Ambient humidity [%]	70,00			
	Last construction stage				
	Load case permanent or long-term				
	Load case	LC3 - Weight of in situ to	pping	*	
	Load case prestress				
	Load case	None		*	
	Type of generated combinations	All code dependent			*
	Structure	Building			
	Actions				
	Variable load cases and type of generated combination	15		>>>	>
New Insert Edit	Delete			Clos	e

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.



Construction stages			×
🎜 🤮 🇶 😰 🔅	:   🚑   [Ali	• 7	
ST1 - placing of prefa	Name	ST4	
ST2 - casting in situ c	Order of stage	4	
ST3 - finishes	Description	start of service stage	
ST4 - start of service st	Global time [day]	50,00	
ST5 - end of service st	Number of subintervals	1	
	Ambient humidity [%]	70,00	
	Last construction stage		
	4 Load case permanent or long-term		
	Load case	LC5 - Service load	·
	Psi [-]	0,300	
	Type of generated combinations	All code dependent	*
	Structure	Building	
	Actions		
	Variable load cases and type of generated combination	ons	>>>
New Insert Edit	Delete		Close

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.

Construction stages				$\times$
🏓 🦆 🍠 💽 🖸	: 🞒 🛯 Ali 🗸 🗸	V		
ST1 - placing of prefa	Name	ST5		
ST2 - casting in situ c	Order of stage	5		
ST3 - finishes	Description	end of service stage		
ST4 - start of service st	Global time [day]	18000,00		
ST5 - end of service st	Number of subintervals	1		
	Ambient humidity [%]	70,00		
	Last construction stage			
	Load case permanent or long-term			
	Load case	LC6 - empty (check a	fter 50 years)	·
	Type of generated combinations	All code dependent		*
	Structure	Building		
	Actions			
	Variable load cases and type of generated combination	15		>>>
New Insert Edit	Delete			Close

## 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.

Properties	<b>▼</b> 4	ЧX
Support in node (2)	- 19 13/	Ø
	S 📀	8
Туре	Standard	-
Angle [deg]		
Constraint		*
х		*
Z	Rigid	*
Ry	Free	*
Default size [m]	0,200	
Construction sta		
Add	ST1 - placing of	
Remove	No	
Remove Geometry	No	
Remove Geometry System	No -	• •••
Remove Geometry System	No -	
Remove Geometry System	No -	·
Remove Geometry System	No -	•
Remove Geometry System	No •	•
Remove Geometry System	No •	-
Remove Geometry System	No •	•



Properties	▼ ₽ ×
Member (1)	- Va V/ /
	of 📀 🗯
Name	B1
Туре	beam (80) -
Analysis model	Standard +
Cross-section	CS1 - Precast3 (: *
Alpha	0 -
Member system-line at	Centre -
ez [mm]	0
LCS	standard -
FEM type	standard -
Layer	Layer1
4 Buckling	
System lengths and	Default
Material and no. of	Concrete - 1
Secondary member	
Construction sta	
Add	ST1 - placing of 🔹
Add Phase 2	ST2 - casting in : *
Remove	No *
Geometry	
Length [m]	30,000
Shape	Line
Beg. node	N1
End node	N2
Modes	
N1	abso
N2	abso
112	
4 Data	



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.

Tree V 4 X	Beam construction stages settings	×
ST1 - placing of prefab beam	Name	LBH
💷 Setup	Local time axis	28.00
Time axis	Time of end of curing [day]	0,00
Add member	Line support (formwork)	
Beam settings	Time of instalation of formwork [day]	-28,00
	Time of releasing of displacements in X direction [day]	0,00
💢 Delete input data of stage	Time of releasing of displacements in 2 direction [day]	0,00

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.

Tree		
🔁 Main 🗖 Construction stages	III Mesh setup	×
Project	Name	MeshSetup1
PlbAte all and storeys	Average number of 1D mesh elements on straight 1D members	30
Structure	Average size of 1D mesh element on curved 1D members [m]	0,200
and Load	Average size of 2D mesh element [m]	1,000
□ J≧ Load cases, Combinations	Connect members/nodes	$\checkmark$
Load Cases	Setup for connection of structural entities	
J++ Load Groups	4 Advanced mesh settings	
👫 Combinations	4 General mesh settings	
Result classes	Minimal distance between definition point and line [m]	0,001
Construction stades	Definition of mesh element size for panels	Manual +
Check structure data	Average size of panel element [m]	1,000
Connect members/nodes	Elastic mesh	<b>V</b>
J++ Mesh setup	Hanging nodes for prestressing	<b>V</b>
J++ Solver setup	4 1D elements	
Local mesh refinement	Minimal length of beam element [m]	0,100
Calculation	Maximal length of beam element [m]	1000,000
Hidden calculation	Average size of tendons, elements on subsoil, nonlinear soil spring [m]	1,000
Autodesign	Generation of nodes in connections of beam elements	
E Integrated Design Forms	Generation of nodes under concentrated loads on beam elements	<b>V</b>
Concrete	Generation of eccentric elements on members with variable height	
Engineering report	Division on haunches and arbitrary members	5
Drawing Tools	Division for integration strip and 2D-1D upgrade	50
	Mesh refinement following the beam type	None -
		OK Cancel
		OK Cancel

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.)

FE analysis	×
Single analysis Batch analysis	
Nonlinear calculation	
Modal analysis	
C Linear stability	
Concrete - Code Dependent Deflections (CDD)	
Construction stage analysis	
Nonlinear stage analysis	
Nonlinear stability	
Number of stages: 5, TDA	
Colver return Mark	catup
	rsetup
ОК Са	ncel
	FE analysis         Single analysis       Batch analysis         O Linear calculation         Nonlinear calculation         Modal analysis         Utinear stability         Concrete - Code Dependent Deflections (CDD)         O Construction stage analysis         Nonlinear stage analysis         Nonlinear stability         Test of input data         Number of stages: 5, TDA         OK       Ca

## 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:



Properties Va V/ / Deformations on member (1) R Name Selection Deform All Type of load Class ST1 (SLS) - p Class lacing of pr Filter Structure Values Extreme No Initial uz Global Drawing setup 1D Section All

Properties	
Deformations on member (1)	- Va V/ /
	💰 🌮 🧯
Name	Deformations on member
Selection	All -
Type of loads	Class -
Class	ST2 (SLS) - casting in situ ( +
Filter	No -
Structure	Initial +
Values	uz -
Extreme	Global -
Drawing setup 1D	
Section	All -

Stage 3:



Stage 4:



Stage 5:



Deformations on member (1)	- Va V/ /
Name	Deformations on member
Selection	All
Type of loads	Class
Class	ST3 (SLS) - finishes 🔹
Filter	No
Structure	Initial
Values	uz
Extreme	Global
Drawing setup 1D	
Section	All

Properties	▲ <sup>†</sup>
Deformations on member (1)	- 🛛 🎶 🖉
	S 🌮 🌶
Name	Deformations on member
Selection	All
Type of loads	Class
Class	ST4 (SLS) - start of service 👻
Filter	No
Structure	Initial
Values	uz
Extreme	Global
Drawing setup 1D	
Section	All

Properties	* Ú
Deformations on member (1)	- Va V/ /
	of 😴 🈹
Name	Deformations on member
Selection	All
Type of loads	Class
Class	ST5 (SLS) - end of service 5 -
Filter	No
Structure	Initial
Values	uz
Extreme	Global
Drawing setup 1D	
Section	All

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:



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:



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.



## 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.

Tree	Project data				X
ta Main					
Project	Basic data Fund	ctionality Actions Unit Set Pro	otection		
:##: Line grid and storeys	11	General		Detailed	
🖶 BIM toolbox		Property modifiers		Subsoil	
Structure	1200	Parametric input		Pad foundation check	1
and Load		Climatic loads		Concrete	
		Mobile loads		Fire resistance checks	1
Construction stages     Colculation math		Dynamics		Hollow core slab checks	]
Results		Stability			
F Integrated Design Forms		Nonlinearity			
Concrete		Structural model			
Concrete Advanced		IFC properties			
Engineering report		Advanced concrete checks			
W Drawing Tools		Prestressing	~		
Tools		Bridge design			
		Excel checks			
		Document			
		Substitution beam			
	1			ок с	ancel

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.

Tree 👻 🔻 🛪							
Hain - Concrete Advanced X	Concrete setup						~
Tree	Concrete setup  Vype of values Design default Drawing settings  Vype of functionality Prestressing		Standard EN Concrete	<ul> <li>Concrete</li> <li>Design defaults</li> <li>General</li> <li>ULS</li> <li>SLS</li> <li>Allowable stress</li> <li>Stress limitation during tensioning</li> <li>SLS stress limitation</li> <li>k - factor for maximum tensile stress. Value [-]</li> <li>National annex</li> <li>k<sub>1</sub> - factor for maximum compressiv Value [-]</li> <li>k<sub>2</sub> - factor for maximum compressiv Value [-]</li> <li>k<sub>2</sub> - factor for maximum stress in r Value [-]</li> <li>k<sub>3</sub> - factor for maximum stress in r Value [-]</li> <li>k<sub>4</sub> - factor for maximum stress in r Value [-]</li> <li>k<sub>5</sub> - factor for maximum stress in privalue [-]</li> <li>k<sub>5</sub> - factor for maximum stress in privalue [-]</li> <li>k<sub>5</sub> - factor for maximum stress in privalue [-]</li> <li>Detailing norvisions</li> <li>Reference: EN 1992-1-1, Claue 72 (3)</li> </ul>	<ul> <li>1,00</li> <li>0,60</li> <li>045</li> <li>0,80</li> <li>1,00</li> <li>1,00</li> <li>0,75</li> </ul>	an fistarrin	× ^
			< >>	concrete exceeds k2.fck, nonlinear creep should be Application: Prestressed concrete checks with SLS	considered quasi-permanent combination	rep. in scress in	
	Select all Unsele	ect all	Refresh	Load default non-NA parameters	Load default NA parameters	OK Car	ncel

## 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.

Tree • 3 × Main • Concrete Advanced × IBD Design defaults System lengths and buckling groups	Properties Concrete prestress design EN 1992-1- Name	1 (1) • Va V. Concrete prestress design EN 1992-1-1	₽ × //
Windowski Member data     Soncrete slenderness     G Redes (without As)     Automatic member reinforcement der     Cores section einerberteitige	Selection Type of loads Combinations Filter	All Combinations FS-EN-ULS (STR/GEO) Set B - end of service sta No	- ige - -
Cross-section characteristics  Cross-section characteristics  Cross-section characteristics  Cross-section  Cr	Print explanation of errors and war Use named joints Use named cuts Values Extreme	Ass	•
Ar Memoer Check     Ar St. Details     Book St. Details     Bill of reinforcement     St. Pretensioning - Pre-tensioned tendon     Free bars - New free bar	Drawing setup 1D Section	All	•

## 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

Mem	ber d <sub>x</sub>	Case		Joint	VEd	VEdi	VRdi	As	As,req	Reinf.[no.]	
	[m]				[kN]	[MPa]	[MPa]	[mm <sup>2</sup> /m]	[mm <sup>2</sup> /m]		
B1	3,000	F5-EN-ULS (STR/GEO) S	et B/1	1	1323,87	1,87	0,06	0	2499	2x8,0-40	]
Explanations of symbols Explanations of symbols						Explana	ations of syn	nbols			
V <sub>Ed</sub>	V <sub>Ed</sub> Shear force v <sub>Rdi</sub> Design			in shear re	sistance a	t the	A <sub>s,req</sub>	Required area	of reinforceme	ent	
VEdi	vEdi Design value of the shear stress in			inter	face				crossing the i	nterface	
	the interface As			Area	of reinforc	ement cro	ssing the				
				inter	face						

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  $\mu$ .

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

$$v_{\text{Rdi}} = c f_{\text{ctd}} + \mu \sigma_{\text{n}} + \rho f_{\text{yd}} (\mu \sin \alpha + \cos \alpha) \le 0.5 \text{ v} f_{\text{cd}}$$
(6.25)

#### where:

ρ

- $c, \mu$  are factors depend on roughness of joint
- $f_{ctd}$  is design tensile concrete strength
- $\sigma_N$  is normal stress preloaded construction joint
  - $\sigma_N > 0$  for compression
  - $\sigma_N < 0$  for tension
  - $\sigma_N$  is limited by  $0.6*f_{ctd}$

is reinforcement ratio

 $\rho = A_{sj}/A_i$  where:

 $\mathbf{A}_{sj}$  area of reinforcement used for shear in construction joint

A<sub>i</sub> 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:

$$v_{\rm Edi} = \beta V_{\rm Ed} / (z b_{\rm i})$$
 (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)
- $\mathbf{V}_{\mathsf{Ed}}$  is the transverse shear force
- z is the lever arm of composite section
- $\mathbf{b}_{i}$  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:

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

## 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.

Teas - 0 V	E Concrete setue	Presentiar
	Concrete setup	Properties
18 Main - Concrete Advanced ×	Type of values     Standard EN     Standard EN	Project data (1)
Design defaults	Design default Design default	
System lengths and buckling groups	Drawing settings	Licence name SCIA
🖃 📅 1D member	Concrete cover	Licence number 660341
Member data	- type of functionality - Columns - Cracking forces	National code EC - EN
Concrete slenderness	Prestressing Beam slabs	Structure Frame X7
Redes (without As)	Default sway type (for columns     National annex	No of poder 1 2
Automatic member reinforcement de:	General Kageraek - coefficient for calculation m	No. of houses.
the internal forces	Concerts     Value [-]     3,40	No. of Deans .
H Member design	Prestressed reinforcement     * k <sub>4.crack</sub> - coefficient for calculation m	No. of stabs : 0
B Member check	- Durability and concrete cover Value [-] 0,42	No. of solids :
Check of non-prestressed concrete	Calculation     w <sub>max</sub> - maximal crack width 7.3.1(5)	No. of used profiles : 1
Check of prestressed concrete	- General Values	No. of load cases : 11
Crack control	Beams / EN_1992_2	No. of used materials : 3
Check response	ULS <sup>4</sup> w <sub>max</sub> - maximal crack width 7.3.1(10)	Linear calculation Finished
Check capacity	General Values	National annex Standard EN
Allowable stress of concrete	Interestance - z <sub>dec</sub> - decompression distance	
Check of prestressing reinforce	Shear Value [m] 0,100	
Check deflection (linear)	1D structures D Prestressing	
🗑 🖫 SaT Details	Creep Creep	×
Bill of reinforcement	Details     Details     Anthorage check     Crack proof     Maximal crack widt	th
St Pretensioning - Pre-tensioned tendon	Bearing checks Direct Deflections	un clare PM (Quari) [mm] PM (Erec) [mm] PM (Quari) [mm]
Free bars - New free bar	Fire resistance D Allowable storer 1 V9C1	0.200 0.200 0.000
	B-SIS Condition providing a service of the service	0,300 0,200 0,000
	General Reference EN 1992-2 Clause 7.3.1(105)	0,500 0,200 0,000
	Prestressing Description : Maximum calculated crack width f	X31,X32,X33 0,300 0,000 0,000 •
	T Creep v and prestressed member with bonded tendons Not editable value	e 0.0 = not checked in this case
	Application : Por 10 member Check Clacks, for 2	ock
	Select all Unselect all Refresh Load default non-NA parameters	reichere PM (Quari) PM (Eren) PM (Quari)
		No No
	Structure 2 N2 30.000 0.000 B1 2 2 2 2 2 2 2 2 2 2 2	No No Ver
	+	IST XS2 XS3 No. Vec No.
	Load Explanation :	
	RM = reinforced	d member Quasi = under quasi-permanent load Set default values
	Libraries PM = prestresses	d member Char = under characteristic load combination
		Freq = under frequent load 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:

Tree		Properties	- 3 - 4
Setting County Strength M			CL 57 #
18 Main - Concrete Advanced X		Concrete prestress check cracks EN 1	992-1-1 (1) 💌 VI V/ V
Design defaults	N		💰 🧒 🛎
System lengths and buckling groups	Histor	Name	Concrete prestress check cracks EN 1992-1-1
😑 👕 1D member	BSP /	Selection	All -
Member data		Type of loads	Combinations -
Concrete slenderness		Combinations	F5-EN-SLS Quasi-permanent - end of service stag -
Integes (without As)		Filter	Ne
Cross-section characteristics		Print explanation of errors and war	
to Internal forces		Time of used scieforement	Artot
🗉 🏝 Member design		Voluee	Decomprarring
Member check		Values Colores	Mancher
Check of non-prestressed concrete		Extreme	Member
Check of prestressed concrete		Drawing setup 1D	
Crack control		Section Section	All ·
Check response			
Allevable stress of conserts			
Allowable principal stresses	Report preview ·	×	
Check of prestressing reinforce	🔅 🍂 🕼 🔍 Default - 🖪 🗍 🕅 🐻		
Check deflection (linear)		^	
B SaT_Details			
Bill of reinforcement	Concrete prestress check cracks EN 1992-1-1		
Pretensioning - Pre-tensioned tendon	Concrete presidess check tracks En 1992-1-1		
E Free bars - New free bar	Linear calculation, Extreme : Member Selection : 41		
	Combinations : F5-EH-SLS Quasi-permanent		
	Decompression check of concrete for selected members		
	Member 0 <sub>x</sub> Case Name 0 <sub>ktrand</sub> /uct ystrand ztranad zt Zdae Check <sub>tale</sub> Check <sub>bale</sub> Check W/E		
	B1 12,000 F5-EN-SLS Quasi-permanent/1 BSP-Strand1 13,82 -700,00 -649,34 54,26 25,00 3,00 I 1,00 NOT OK 616		

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







## 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.

New cross-section		×
Available groups Concrete Geometric shapes Numerical General Precast W Bridge	Available items of this group	ltems in project CS1 - Rectangle (500; 300)
VSTI - Precast6		->
	Profile Library filter	Add Close





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.

	Properties		▲ 廿 ×
	Support in node (11)		- 🖓 🎶 /
			S 😴 🛎
BSP10	Туре	Standard	*
BSPB	Angle [deg]		
BSP //	Constraint	Custom	-
BCDFC	Х	Rigid	-
BSPt 7	Y	Rigid	•
Rep-3	Z	Rigid	*
BSP	Rx	Rigid	*
BSP	Ry	Free	*
	Rz	Rigid	*
	Default size [m]	0,200	
	4 Geometry		
	System	GCS	*

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.

Tree		- + ×	Mesh setup			×
e Mair	n 💶 Load		Name	MechSetun1		^
	Project		Average number of 1D much elements on s	1		
	Line grid and storeys		Average number of 1D mesh elements on s	0.200		
-	BIM toolbox		Average size of 1D mesh element on curve	0,200		
	Structure		Average size of 2D mesh element [m]	0,230		
24	Load		Connect members/nodes			
- <u>*</u>	Load Cases		Setup for connection of structural entities		•••	
	1++ Load Groups	4	Advanced mesh settings			
	Nt Combinations	· · · · · · · · · · · · · · · · · · ·	General mesh settings			
	👫 Concrete combinations		Minimal distance between definition poi	0,001		
	💭 Result classes		Definition of mesh element size for panels	Manual	-	
= 🔳	Calculation, mesh		Average size of panel element [m]	1,000		
	Check structure data		Elastic mesh			
	Connect members/nodes		Use automatic mesh refinement			
	Ut Solver setup		Hanging nodes for prestressing	$\checkmark$		
	V Local mesh refinement		1D elements			
	Mesh generation		Minimal length of beam element [m]	0,100		
	E Calculation		Maximal length of beam element [m]	1000,000		
	Hidden calculation		Average size of tendons, elements on su	1,000		
	Autodesign		Generation of nodes in connections of b			
l <b>†</b> ⊧	Integrated Design Forms		Generation of nodes under concentrate	$\checkmark$		
1	Concrete		Generation of eccentric elements on me			
	Concrete Advanced		Division on haunches and arbitrary me	5		
	Drawing Tools		Division for integration strip and 2D-1D	50		
	Libraries		Mesh refinement following the beam type	None	-	
. 🗟	Tools		4 2D elements			
						~
		A	verage size of 2D mesh element			
				г		
			) <sup>1</sup> A <sup>2</sup> A <sup>2</sup>		OK Cancel	L .

## 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.

Advanced Expert Training - Pre-tensioning (Precast)



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.





Properties		▼ ₽ ×
2D member - Internal forces (1)		- Va V/ /
		s 🌮 🔌
Name	2D member - Internal forces	
Selection	All	*
Type of loads	Load cases	*
Load cases	LC4 - Service Load	*
Filter	No	*
System	Local	
Rotation [deg]	0,00	
Averaging of peak		
Location	In nodes, avg. on macro	
Type forces	Basic magnitudes	*
Standard		
Section		
Edge		
Trajectories		
Values	mx	
Extreme	Global	*
Drawing setup 2D		

Va V/ /

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:

				Properties		<b>▼</b> ₽ ×
				Internal forces on member (1)		- Va V/ /
						I I I I I I I I I I I I I I I I I I I
				Name	Internal forces on member	
				Selection	All	-
				Type of loads	Load cases	-
				Load cases	LC4 - Service Load	-
	AMP A			Filter	No	-
				Structure	Initial	-
				Prefab slab beam		
				Values	My	-
				System	Principal	-
				Extreme	Global	-
				Drawing setup 1D		
			ų ų į	Section	All	-
$\rightarrow$						
	Real and the second					
$\sim \sim \sim$	IN M					
		ANN NY				

## 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.

Upgrade 2D->1D export			×				
	Type of export	New project					
xp	Effective width slab	$\checkmark$					
	Export into frame XZ	$\checkmark$					
	Export to UCS from member1D	$\checkmark$					
	Change self weight to standard LC	$\checkmark$					
yb yb	Upgraded internal forces	$\checkmark$					
xb zb _My [kNm] = my[kN].a	Load cases, Combinations						
FTT	<ul> <li>Upgraded internal forces</li> </ul>						
	N	$\checkmark$					
↓ niv	Vy	$\checkmark$					
	Vz	$\checkmark$					
<sub>b</sub> <sub>p</sub> <sub>ρ</sub> Θ	Mx	$\checkmark$					
My + My	My	$\checkmark$					
- 40 <sup>11</sup>	Mz	$\checkmark$					
Ð							
D' P' P		OK Can	el				

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.