

Advanced Concept Training Reinforced concrete (EN1992) – Adv. modules

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 2017 SCIA nv. All rights reserved.

Table of contents

Table of contents	3
Introduction	5
Code Dependent Deflection (CDD) calculation	6
Introduction: the difference between CDD and PNL	6
CDD in SCIA Engineer.....	6
CDD deformation for 2D elements.....	15
Theoretical background.....	20
Physical Non Linear (PNL) calculation	22
Introduction.....	22
Examples of a P(G)NL calculation	32
Punching shear check	41
1_Theoretical background – Eurocode	41
2_Punching check in SCIA Engineer	44
3_Example in SCIA Engineer.....	46
4_Extra features.....	57
Pressure only 2D members	61
Introduction & Principle of calculation	61
Comparison between isotropic and pressure only element	62
Wall with opening	63
Use of reinforcement ribs	65
Practical example.....	Error! Bookmark not defined.
Fire resistance check	71
Simplified calculation method	71
Thermal distribution curves	71
Load case: Thermal load.....	75
Load Combinations	77
Materials properties for concrete/reinforcement	77
Check	78
Advanced calculation method	84
Tabulated data	85
Load Combinations	85
Setup for tabulated data.....	85
Concrete member data	88
Check	89

Introduction

The examples in this manual can be made in a full licensed as well as in a tryout or student version of SCIA Engineer.

Here follows an overview of the required SCIA Engineer modules / editions, per subject:

- CDD calculation	
esas.18 (1D members)	Concept edition
esas.19 (2D members)	
- PNL calculation	
esas.16 (1D members)	Not part of an edition
- Punching shear check	
esacd.03	Concept edition
- Pressure only 2D members	
esas.44	Professional edition
- Fire resistance check	
esacd.07 (1D members)	Professional edition

Prerequisites are:

- Theoretical reinforcement design	
esacd.01 (1D members)	Concept edition
esacd.02 (2D members)	

and/or

- Practical reinforcement design	
esacdt.01 (1D members)	Concept edition
esacdt.03 (2D members)	

Code Dependent Deflection (CDD) calculation

Introduction: the difference between CDD and PNL

This chapter will start with an explanation of the name code dependant deflections, which is relevant for users of versions before SE 2008.1. In these previous versions, the code dependant deflections were named PNL deflections. A real non linear analysis was named a PGNL calculation. The names PNL calculation (physically non-linear calculation) and PGNL calculation (physically and geometrically non-linear calculation) were used in previous versions of the software. These terms were very similar but a bit misleading. Therefore, it is useful to start with a few words about the features and principles of the two types of analysis.

First of all, it may be reasonable to establish more explanatory names.

What was known as a PNL calculation should be called:

*calculation of deflections according to a standard: **code dependent deflections (CDD)**.*

What was known as a PGNL calculation should be called:

*physically (and geometrically) non-linear calculation: **P(G)NL calculation**.*

It is not a rule that both types of non-linearity must be engaged in the latter, there are three possibilities:

- (i) only physical non-linearity is taken into account (PNL),
- (ii) only geometrical non-linearity is taken into account (GNL),
- (iii) both physical non-linearity and geometrical non-linearity are taken into account (PGNL).

Now you can see one of the reasons why a more reasonable naming should be introduced.

The second reason is that the CDD calculation is not a real physically non-linear calculation in terms of finite element method (contrary to PNL that is a real non-linear calculation).

CDD is a two-step solution following exactly the regulations given in technical standards for design and checking of concrete structures.

Both analyses are aimed primarily at concrete structures.

The CDD calculation has been designed exclusively for concrete beams and plates, as it is based on the wording of technical standards for design and checking of concrete structures. The aim is to calculate the long term deflections according to the rules described in the national code.

The PNL calculation is a general procedure tailored for the analysis of concrete frames (it takes into account the non linear stiffness of concrete and the reinforcement). It is a real non linear calculation.

The aim is to calculate the internal forces distribution after physical non linear calculation. The PNL calculation will be treated in the next chapter.

The principle of the CDD calculation can be explained as follows.

This calculation of deflections depends (is based) on standards. Therefore, it represents a standard-related calculation that is performed in two steps. First, a normal linear calculation is carried out and the computed internal forces are used to input the reinforcement (the provided reinforcement) or at least to determine the required reinforcement areas. The procedure continues with the calculation of cracks and their effect on the stiffness of individual elements. This weakening is then input into the solver. Finally, the calculation (linear one) is run once more with these reduced stiffnesses taken into account. Which is exactly what the technical standards require.

CDD in SCIA Engineer

Creation of concrete combinations

To perform a CDD calculation in SE, it is necessary first to create what is called 'concrete combinations'.

Three types of concrete combinations have to be created:

-Immediate (CC1): containing all permanent loads: checkbox: '*used to determine permanent Code Dependant Deflections*'

-Creep (CC2): containing all permanent loads and a percentage of variable loads: checkbox: '*used to determine permanent Code Dependant Deflections caused by creep*'

-Total (CC3): containing all load cases

Take the example CDD1.esa and create these three concrete combinations:

Concrete combinations

Input

Name	Immediat
Contents of combination	
LC1 - Self weight [-]	1,00
LC2 - Permanent [-]	1,00
use to determine permanent Code De...	<input checked="" type="checkbox"/>

Concrete combinations

Input

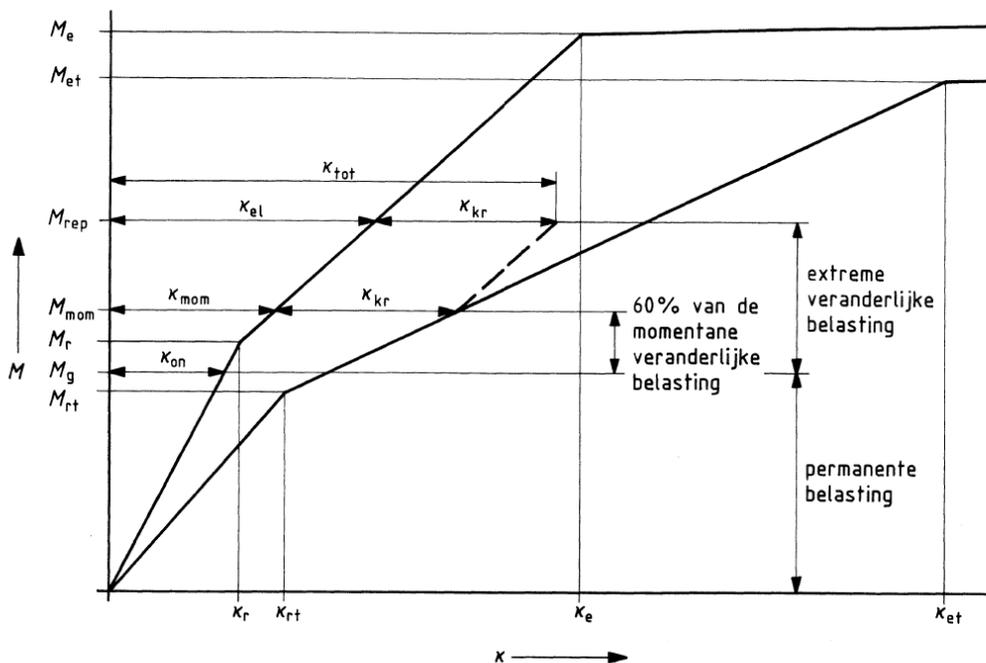
Name	Creep
Contents of combination	
LC1 - Self weight [-]	1,00
LC2 - Permanent [-]	1,00
LC3 - Variable [-]	0,30
use to determine Code Dependent D...	<input checked="" type="checkbox"/>

Concrete combinations

Input

Name	Total
Contents of combination	
LC1 - Self weight [-]	1,00
LC2 - Permanent [-]	1,00
LC3 - Variable [-]	1,00

The necessity for the creation of these tree types of combinations can be explained based on the following diagram of the code NEN6720.



Figuur 73 – M - k -diagram

With:

- M_r : Cracking moment in the case of a short-term load
- M_{rt} : Cracking moment in the case of a long-term load
- M_g : Maximal bending moment caused by permanent loads
- M_{rep} : Maximal bending moment caused by permanent loads and variable loads
- M_{mom} : Sum of the permanent loads + 60% of the variable load
- k_{kr} : Deformation caused by creep
- k_{on} : Immediate deformation at M_g
- k_{mom} : Deformation at M_{mom}
- k_{el} : Deformation at M_{rep}
- k_{tot} : Sum of k_{el} and k_{kr}

It is clear from this diagram that for the calculation of the creep deformation, not the total variable load is taken into account.

In fact the creep deformation is only calculated for CC2 . This creep deformation is kept constant and added to the elastic deformation of CC3 to obtain the long term deformation of CC3.

The percentage of variable load taken into account in CC2 for calculation can be set by user.

Refining the mesh size

With a CDD calculation it is important that the 'average number of tiles of 1D element' is at least on 5, see Figure below:

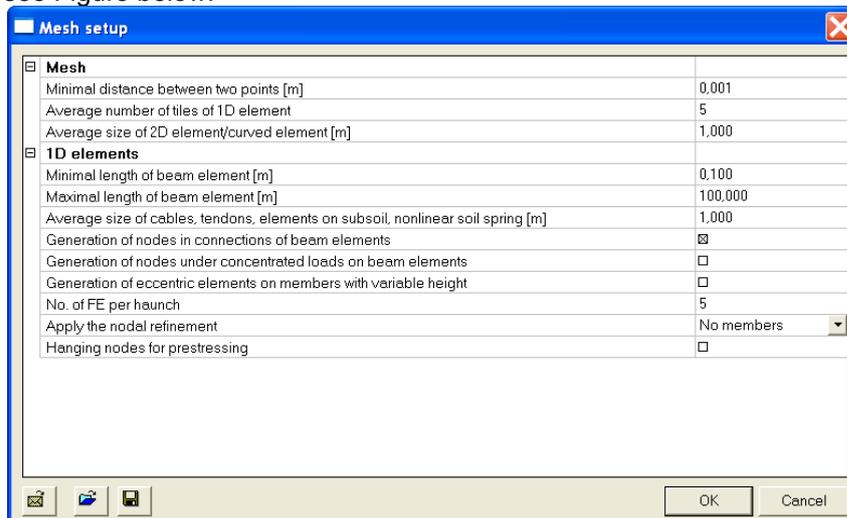


Figure: Mesh setup

This is necessary because a reduced stiffness is calculated for every mesh element.

Calculation of theoretical reinforcement

Continue with the example CDD2.esa.

As mentioned before CDD calculation is a two step calculation, starting with a linear calculation of reinforcement.

So perform a linear calculation. And calculate the theoretical needed reinforcement afterwards.

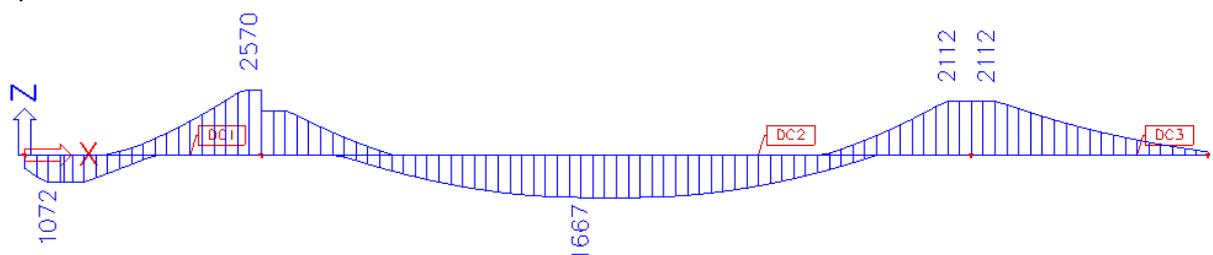


Figure: A_s ; required

Calculation of code dependent deflections (CDD)

As soon as the linear calculation and the calculation of the theoretical reinforcement have been finished successfully, the option 'Concrete – Code dependent deflections' is available.

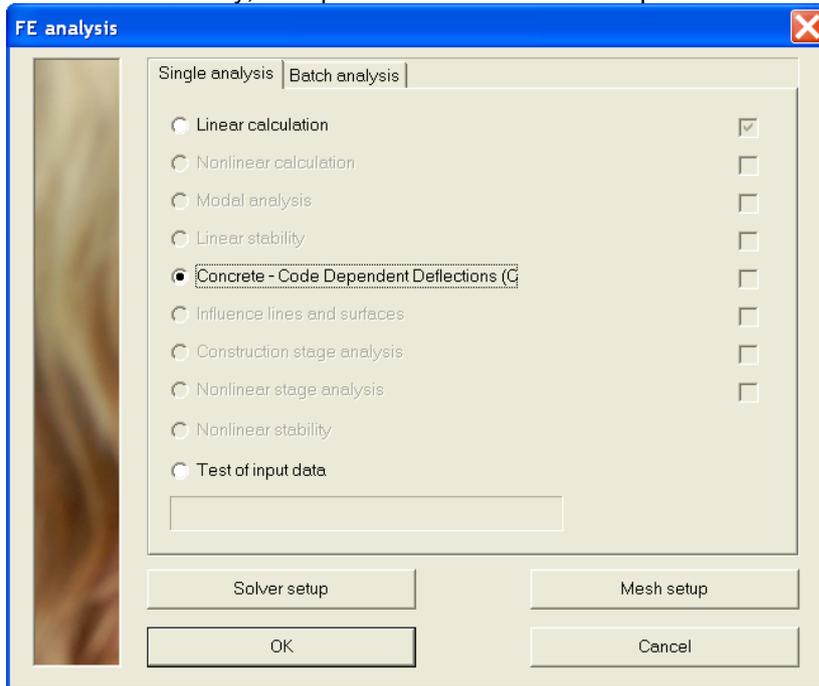


Figure: Option 'Concrete – CDD deformations'

Perform the CDD calculation now. Go to **Concrete > 1D member > Check** and notice that the extra functionality has become available in the menu, namely '**Deformation**'.

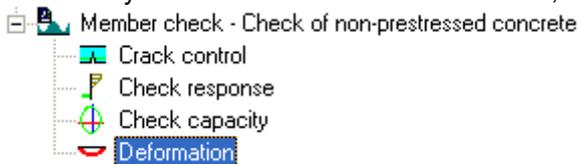


Figure: 'Deformation' in concrete-menu

The following results can be viewed for every concrete combination:

- *Linear: deformation as result of linear calculation
- *Non linear: deformation as result of CDD calculation
- *Non linear + creep: deformation as result of CDD calculation + creep deformation

The creep deformation is calculated based on CC2 with the checkbox '*used to determine permanent Code Dependant Deflections caused by creep*'.

Let us calculate the non linear + creep deflection for CC3 (total).

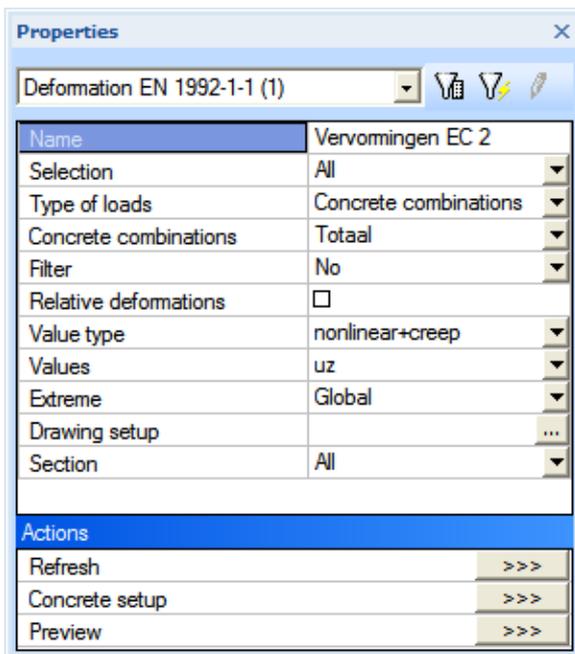


Figure: Properties

Now if we press 'Refresh', the total deformation in the long term is shown, see Figure 203

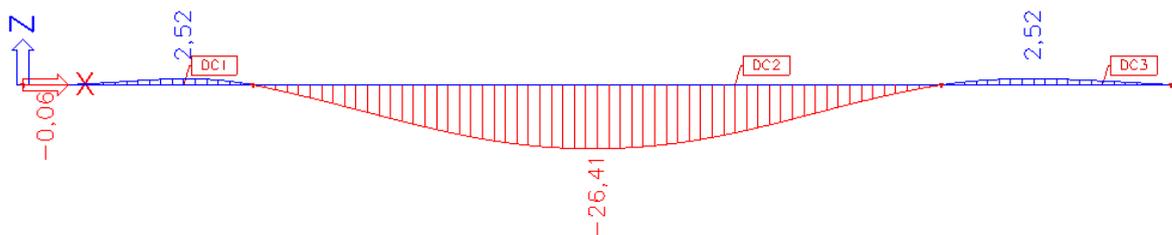


Figure: Graphical representation of the total deformation in the long term

The preview shows following values, see Figure below:

Deformation EN 1992-1-1

CDD deformations uz for selected members

Member	d_x [m]	Case	$\delta_{elastic}$ [mm]	δ_{creep} [mm]	δ_{lim} [mm]	δ_{add} [mm]	δ_{tot} [mm]	$\delta_{tot}/\delta_{lim,tot}$ [-]	Check _{calc} [-]	Check _{lim} [-]	Check
B2	5,220	Total	-17,46	-8,95	-11,14	-15,26	-26,40	0,37	0,42	1,00	OK
B3	1,440	Total	1,67	0,85	1,00	1,51	2,52	0,03	0,04	1,00	OK
B2	5,400	Total	-17,46	-8,95	-11,12	-15,30	-26,41	0,37	0,42	1,00	OK
B1	2,520	Total	1,74	0,78	1,15	1,37	2,52	0,03	0,04	1,00	OK

Figure: Preview deformation

To find out how these values can be reduced back, we take a look at the results in section $dx = 5,04$ of member B2:

- Concrete combination CC1 (Immediately) – nonlinear : uz = -11,14 mm
- Concrete combination CC2 (Creep effect) – nonlinear : uz = -12,79 mm
- Concrete combination CC2 (Creep effect) – nonlinear + creep : uz = -21,74 mm
- Concrete combination CC3 (Total) – nonlinear : uz = -17,46 mm
- Concrete combination CC3 (Total) – nonlinear + creep : uz = -26,41 mm

*elastic deformation $u_{el} = -17,46$ mm based on the concrete combination CC3 (Total) and the short-term stress-strain diagram (nonlinear).

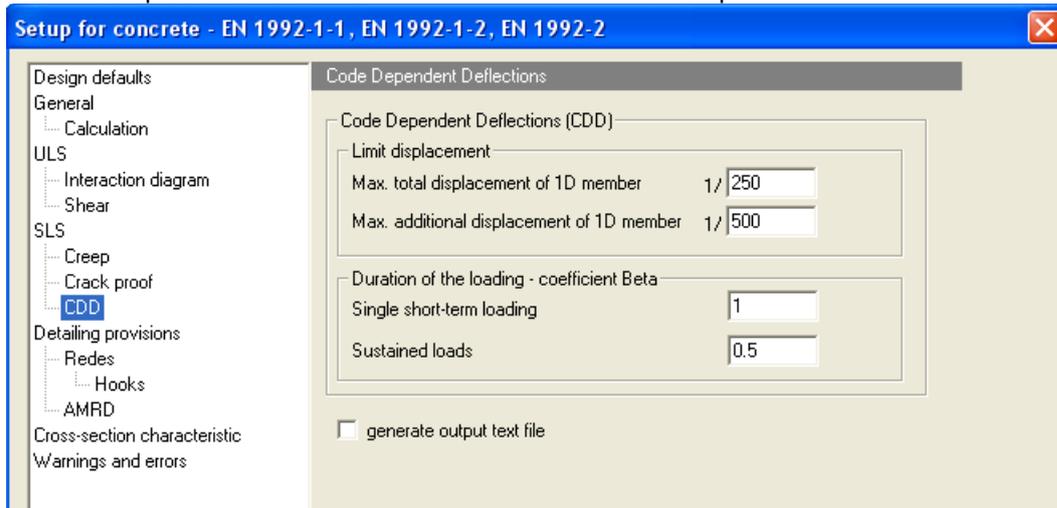
*creep deformation $U_{\text{creep}} = -21,74 - (-12,79) = -8,95 \text{ mm}$
 based on the concrete combination CC2 (Creep effect) and the long term stress-strain diagram (nonlinear + creep) minus the short term stress-strain diagram (nonlinear).

*total deformation $U_{\text{to}} = U_{\text{el}} + U_{\text{creep}}$
 $= -17,46 + (-8,95) = -26,41 \text{ mm}$

*immediate deformation $U_{\text{immediately}} = -11,14 \text{ mm}$
 based on the concrete combination CC1 (Immediately) and the short-term stress-strain diagram (nonlinear).

*additional deformation $U_{\text{add}} = U_{\text{to}} - U_{\text{immediately}}$
 $= -26,41 - (-11,14) = -15,26 \text{ mm}$

User has to check on the total deformation and on the additional deformation.
 The limit displacements for the checks can be set in the setup:



This first example was based on theoretical reinforcement. Other examples will follow with basic reinforcement or an augmentation of theoretical reinforcement through the coefficient of reinforcement.

Basic reinforcement

Also for the CDD calculation it is possible to define the basic reinforcement. This means that a number of members can be defined for the calculation. These members are taken into the arithmetical model. Subsequently an additional reinforcement can be calculated. Open file CDD3 and perform a linear calculation. From this file a basic reinforcement from above 3Ø20 and under 3Ø16 for all members is applied, see Figure 205.

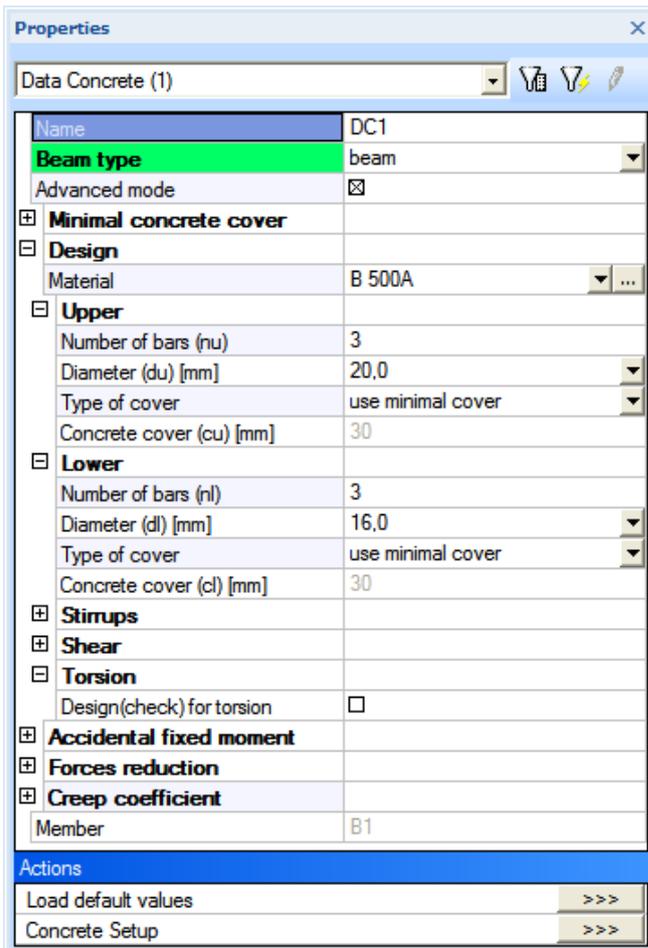


Figure : Input of basic reinforcement

Now perform a linear calculation and calculate the required reinforcement, see Figure 206.

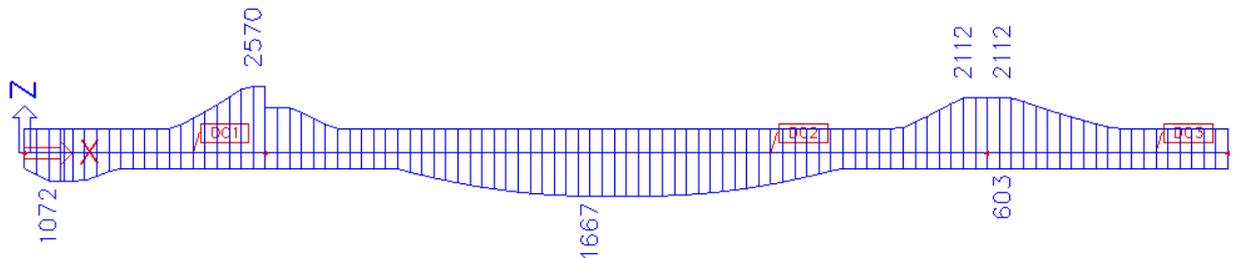
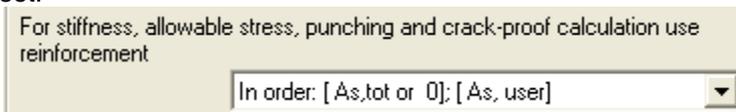


Figure : As; required

Note: to take into account this total required reinforcement (with a minimum of basic reinforcement) and not only the basic reinforcement (user reinforcement), we have to check the setup in which this can be set:



Now perform a CDD calculation and look at the results. As expected, the deformations are somewhat reduced, see Figure 207 and 208.

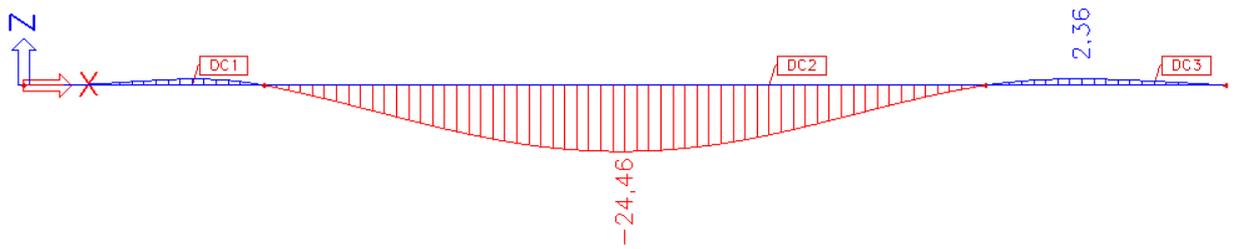


Figure : Graphic reproduction total deformation on long term

Deformation EN 1992-1-1

CDD deformations uz for selected members

Member	d_x [m]	Case	$\delta_{elastic}$ [mm]	δ_{creep} [mm]	δ_{imm} [mm]	δ_{add}	δ_{tot}	$\frac{\delta_{tot}}{\delta_{lim,tot}}$	Check _{calc}	Check
						[mm]	[mm]	[-]	[-]	
						$\delta_{lim,add}$	$\delta_{lim,tot}$	$\frac{\delta_{add}}{\delta_{lim,add}}$	Check _{lim}	
B2	5,220	Total	-16,97	-7,48	-10,80	-13,65	-24,45	0,34	0,38	OK
						36,00	72,00	0,38	1,00	
B3	1,440	Total	1,63	0,73	0,97	1,39	2,36	0,03	0,04	OK
						36,00	72,00	0,04	1,00	
B2	5,400	Total	-16,97	-7,49	-10,77	-13,68	-24,46	0,34	0,38	OK
						36,00	72,00	0,38	1,00	
B3	1,620	Total	1,58	0,73	0,92	1,39	2,31	0,03	0,04	OK
						36,00	72,00	0,04	1,00	
B1	2,520	Total	1,69	0,64	1,11	1,22	2,33	0,03	0,03	OK
						36,00	72,00	0,03	1,00	

Figure : Preview deformations

Coefficient for reinforcement

In SCIA Engineer it is possible to attribute a coefficient for the reinforcement for a CDD calculation to the theoretical required reinforcement. This coefficient is an increase factor and increases the amount of reinforcement for the CDD deformation calculation.

Open file **CDD2** again and go to **Settings > Solver > Reinforcement coefficient** and change the coefficient for the reinforcement in 2, see Figure 209.

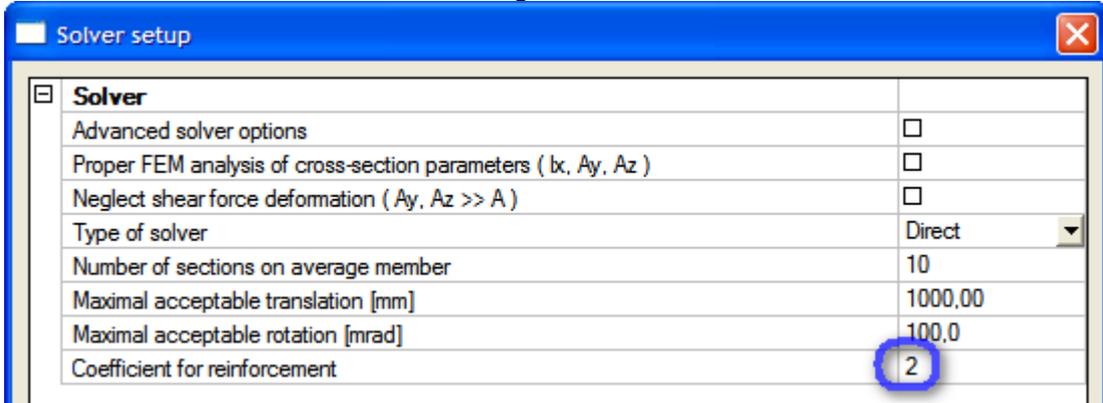


Figure : Coefficient for reinforcement in settings solver

Now perform successively a linear calculation, a calculation of the reinforcement and a CDD deformation calculation and take a look at the results. As expected, the deformations have been reduced, see Figure 210 and 211. These results are stored in file CDD4.

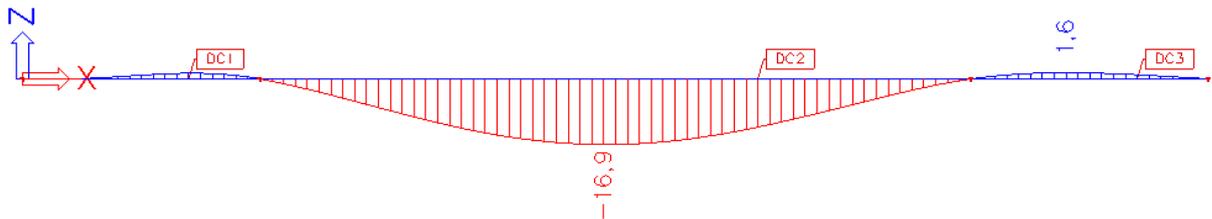


Figure : Graphic reproduction total deformation on long term

Deformation EN 1992-1-1

CDD deformations uz for selected members

Member	d _x [m]	Case	δ _{elastic} [mm]	δ _{creep} [mm]	δ _{imm} [mm]	δ _{add} [mm]	δ _{tot} [mm]	δ _{tot} /δ _{lim,tot} [-]	Check _{calc} [-]	Check
						δ _{lim,add} [mm]	δ _{lim,tot} [mm]	δ _{add} /δ _{lim,add} [-]	Check _{lim} [-]	
B2	5,220	Total	-10,4	-6,5	-7,0	-10,0	-16,9	0,24	0,28	OK
						36,0	72,0	0,28	1,00	
B3	1,440	Total	1,0	0,6	0,6	1,0	1,6	0,02	0,03	OK
						36,0	72,0	0,03	1,00	
B3	1,620	Total	1,0	0,6	0,6	1,0	1,6	0,02	0,03	OK
						36,0	72,0	0,03	1,00	
B1	2,520	Total	0,9	0,5	0,6	0,8	1,5	0,02	0,02	OK
						36,0	72,0	0,02	1,00	

Figure : Preview deformations

CDD deformation for 2D elements

Open file **CDD_plate**. Go to Load cases, combinations > concrete combinations and make following concrete combinations:

CC1	(Immediately)	1,00 x SW + 1,00 x PL
CC2	(Creep effect)	1,00 x SW + 1,00 x PL + 0,30 x VL
CC3	(Total)	1,00 x SW + 1,00 x PL + 1,00 x VL

Now perform a linear calculation. And then calculate the theoretical required reinforcement.

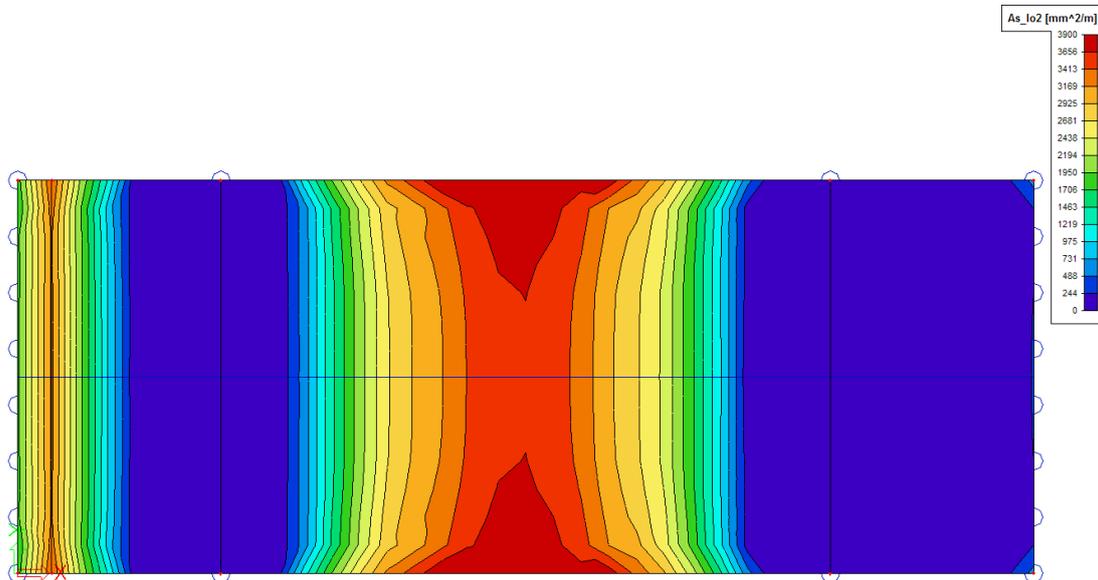


Figure : Required lower reinforcement – direction 2 (longitudinal direction)

As soon as the linear calculation and the calculation of the theoretical reinforcement have been completed successfully, the option '**Concrete – code dependent deflections**' is available. Now perform this CDD calculation. Go to **Concrete > 2D element > Check** and see that an extra functionality has become available in the menu, namely '**Deformations**'.

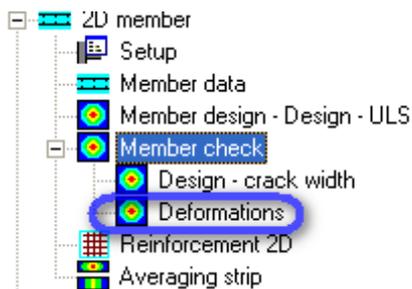


Figure : 'Deformations' in concrete- menu

The total deformation on long term will be:

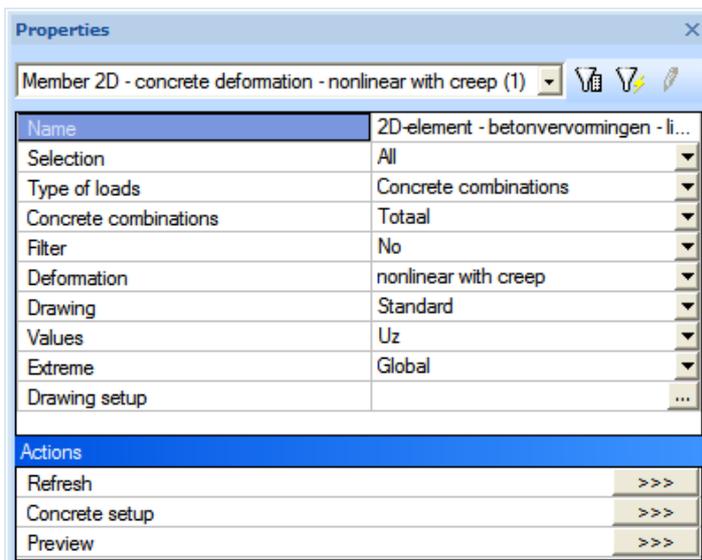


Figure : Properties

Now if we press 'Refresh', the total deformation is shown on the long term.

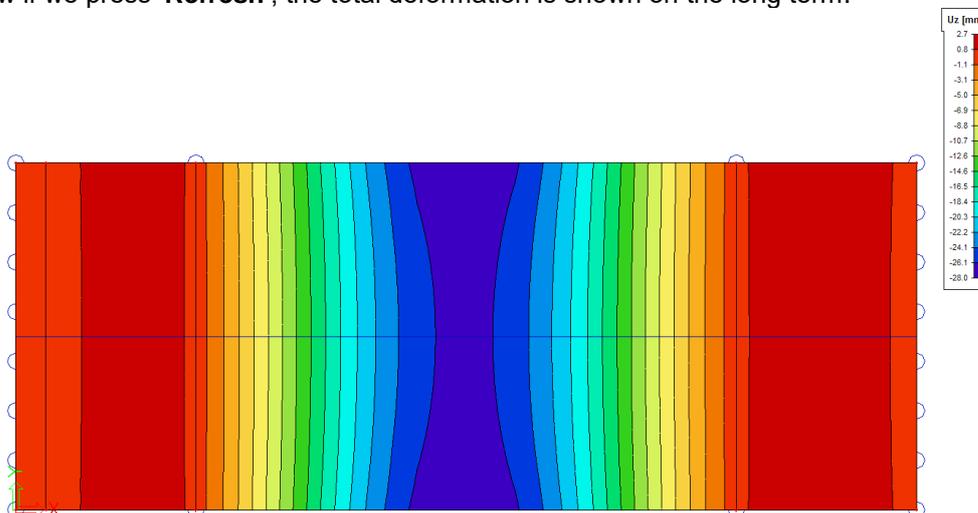


Figure : Graphic reproduction total deformation on long term

That is how the next values are obtained:

Concrete construction CC1 (Immediately) – nonlinear	: uz = -15,6 mm
Concrete construction CC2 (Creep effect) – nonlinear	: uz = -17,8 mm
Concrete construction CC2 (Creep effect) – nonlinear + creep	: uz = -23,8 mm
Concrete construction CC3 (Total) – nonlinear	: uz = -22 mm
Concrete construction CC3 (Total) – nonlinear + creep	: uz = -28 mm

$$\begin{aligned}
 U_{el} &= & &= & -22 & \text{mm} \\
 U_{creep} &= -23,8 - (-17,8) & &= & -6 & \text{mm} \\
 U_{to} &= U_{el} + U_{creep} \\
 &= -22 + (-6) & &= & -28 & \text{mm} \\
 U_{immediately} &= & &= & -15,6 & \text{mm} \\
 U_{add} &= U_{to} - U_{immediately} \\
 &= -28 - (-15,6) & &= & -12,4 & \text{mm}
 \end{aligned}$$

Basic reinforcement

It is also possible to define the basic reinforcement for the physical nonlinear deformation calculation of 2D elements.

Apply a basic reinforcement of the upper $\varnothing 20-100$ (axial) + $\varnothing 8-100$ (transversal) and lower $\varnothing 16-100$ (axial) + $\varnothing 8-100$ (transversal).

Properties	
Data slab concrete (1)	
Name	BPG2
Type	Plate
Advanced mode	<input checked="" type="checkbox"/>
Basic data	
Type of reinforcement geometry	Orthogonal
Type of cover	use minimal cover
Different layers per side	<input checked="" type="checkbox"/>
User reinforcement	<input checked="" type="checkbox"/>
User input thickness	<input type="checkbox"/>
Upper reinforcement steel	B 500A
Lower reinforcement steel	B 500A
Shear reinforcement steel	B 500A
Upper	
First direction angle [deg]	90,00
Number of reinforcement layer	2
1	
Diameter (du) [mm]	8,0
Layer angle	90.000
Concrete cover (cu) [mm]	30
Basic distance [mm]	100
2	
Diameter (du) [mm]	20,0
Layer angle	180.000
Type of cover	layer on previous layer
Concrete cover [mm]	38
Basic distance [mm]	100
Lower	
First direction angle [deg]	90,00
Number of reinforcement layer	2
1	
Diameter (dl) [mm]	8,0
Layer angle	90.000
Concrete cover (cl) [mm]	30
Basic distance [mm]	100
2	
Diameter (dl) [mm]	16,0
Layer angle	180.000
Type of cover	layer on previous layer
Concrete cover [mm]	38
Basic distance [mm]	100
Minimal concrete cover	
Creep coefficient	
2D member	S2
Actions	
Load default values	>>>
Concrete Setup	>>>

Figure : Input of basic reinforcement

Now perform a linear calculation and then calculate the required reinforcement, see Figure 221.

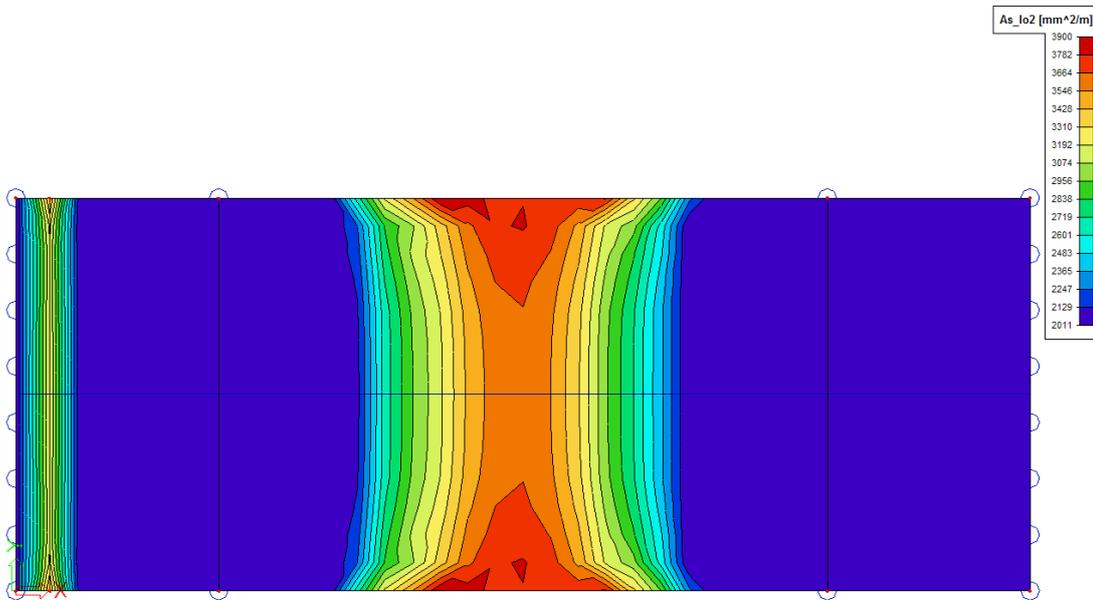


Figure : Required lower reinforcement – direction 2 (longitudinal direction)

Now perform a CDD deformation calculation and look at the results. As expected, the deformations have been slightly reduced, see Figure below.

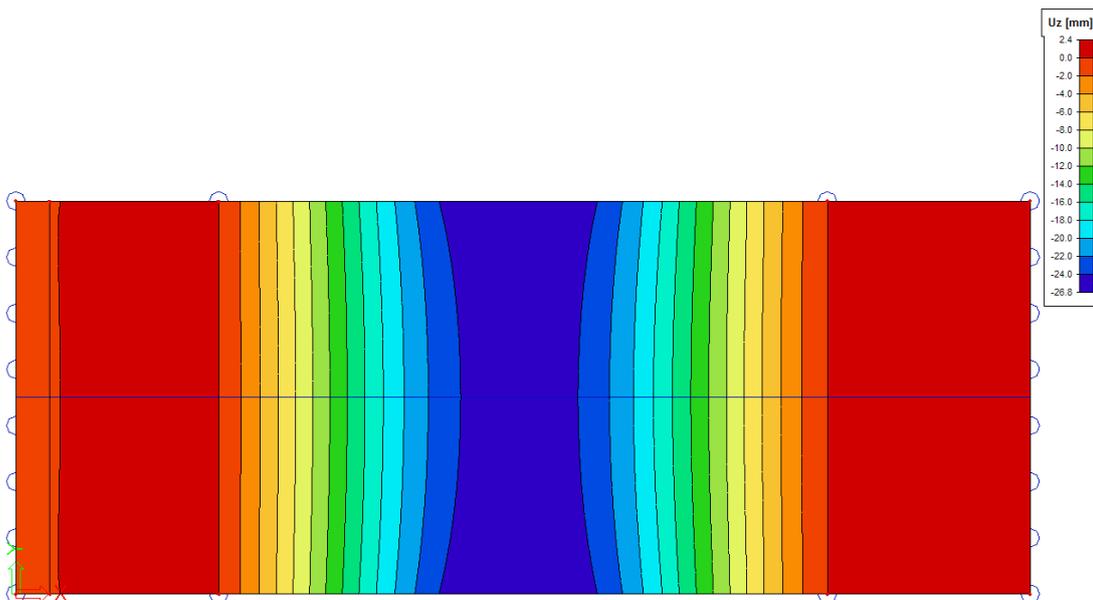


Figure : Graphic reproduction tot deformation on long term

Coefficient for reinforcement

In SCIA Engineer it is also possible to attribute a coefficient for the reinforcement for a CDD calculation of 2D elements to the theoretical required reinforcement.

Open file **WS_PNL04** and go to **solver setup** and change the coefficient for reinforcement 2, see Figure 223.

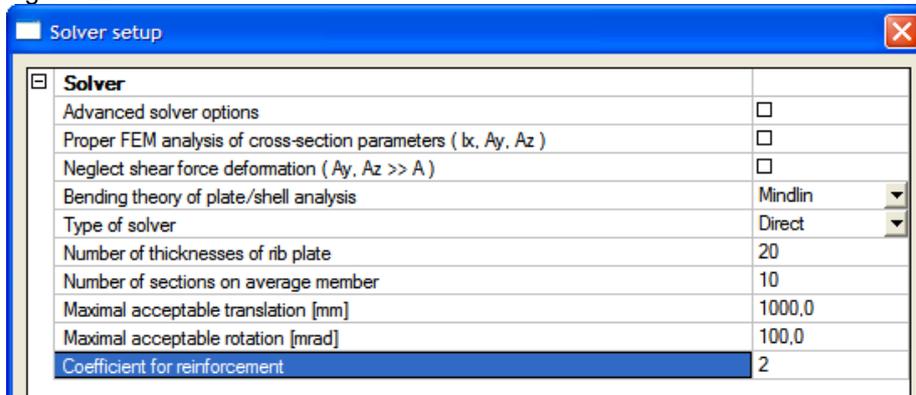


Figure : Coefficient for reinforcement in concrete settings

Now perform, successively a linear calculation, a calculation of the reinforcement and a CDD calculation and look at the results. As expected the deformations have been reduced.

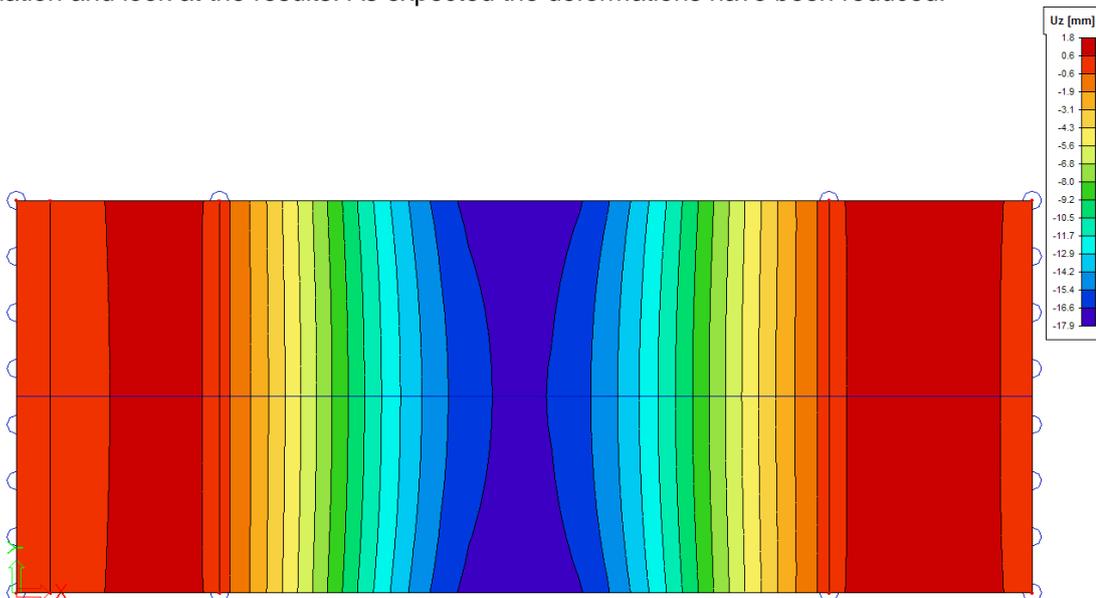


Figure : Graphic reproduction total deformation on long term

Theoretical background

SCIA Engineer is able to perform a CDD calculation for both 1D elements as well as 2D elements. The calculation of physical nonlinear deformations is based on the concept of “quasi” nonlinearity. This means that the linear calculations are used to model nonlinear behavior of the construction.

The CDD calculation is actually a two step linear calculation.

First linear calculation. Based on the linear calculated internal forces the reinforcement is determined.

This can be theoretical reinforcement or user reinforcement (basic or practical). If desired the coefficient of reinforcement can be augmented.

Based on this reinforcement the reduced stiffnesses are calculated.

The reduced stiffnesses are calculated using the following formulas (derived from EN2, formula 7.18).

$$(EI)_r = \frac{1}{\frac{\zeta}{(EI)_{II}} + \frac{1-\zeta}{(EI)_I}} \quad (7.18) \text{ and}$$

$$\zeta = 1 - \beta \left(\frac{\sigma_{sr}}{\sigma_s} \right)^2 \quad (7.19)$$

$\zeta = 0$ for uncracked sections

β is a coefficient taking account of the influence of the duration of the loading or of repeated loading on the average strain

= 1,0 for a single short-term loading

= 0,5 for sustained loads or many cycles of repeated loading

σ_s is the stress in the tension reinforcement calculated on the basis of a cracked section

σ_{sr} is the stress in the tension reinforcement calculated on the basis of a cracked section under the loading conditions causing first cracking

Based on these reduced stiffnesses a second linear calculation is done to calculate the ‘non linear’ deformations.

The creep deformation is calculated only for the concrete combination CC2 with checkbox ‘used to determine Code Dependant Deflection caused by creep’. This concrete combination contains the quasi-permanent load [1,0 x permanent load + factor Ψ x variable load]. This factor Ψ is in most cases about 30%.

The ‘creep’ deformation is obtained by subtracting the short-term deformation from the long-term deformation.

The long-term deformation is calculated using the long term stress-strain diagram of concrete based on the effective E-modulus $E_{c,ef} = E_{cm} / (1 + \Phi_{creep})$. The long term stress-strain diagram is only used to calculate creep deformation. As stated before this creep deformation is only calculated for CC2.

Through adding this ‘creep’ deformation to the elastic deformation, caused by the representative load [1,0 x permanent load + 1,0 x variable load], the total ‘non linear + creep’ deformation is obtained.

To calculate the immediate deformation, the ‘non linear’ deformation is calculated for the permanent load, by using the short-term stress-strain diagram. The program then calculates the additional deformation by deducing the immediate deformation of the total deformation.

The calculated deformations after a CDD deformation calculation are:

Elastic deformation

(based on the short-term stress-strains diagram and representative load combinations [1,0 x PB + 1,0 x VB]);

Creep deformation

(based on the short-term and long-term stress-strain diagram and the quasi-permanent load combination [1,0 x PB + Ψ x VB]);

Total deformation (Elastic deformation + creep deformation);

Immediate deformation (based on the short-term stress-strain diagram and the permanent load combination [1,0 x PB + 1,0 x VB]);

Additional deformation (Total deformation – immediate deformation).

Physical Non Linear (PNL) calculation

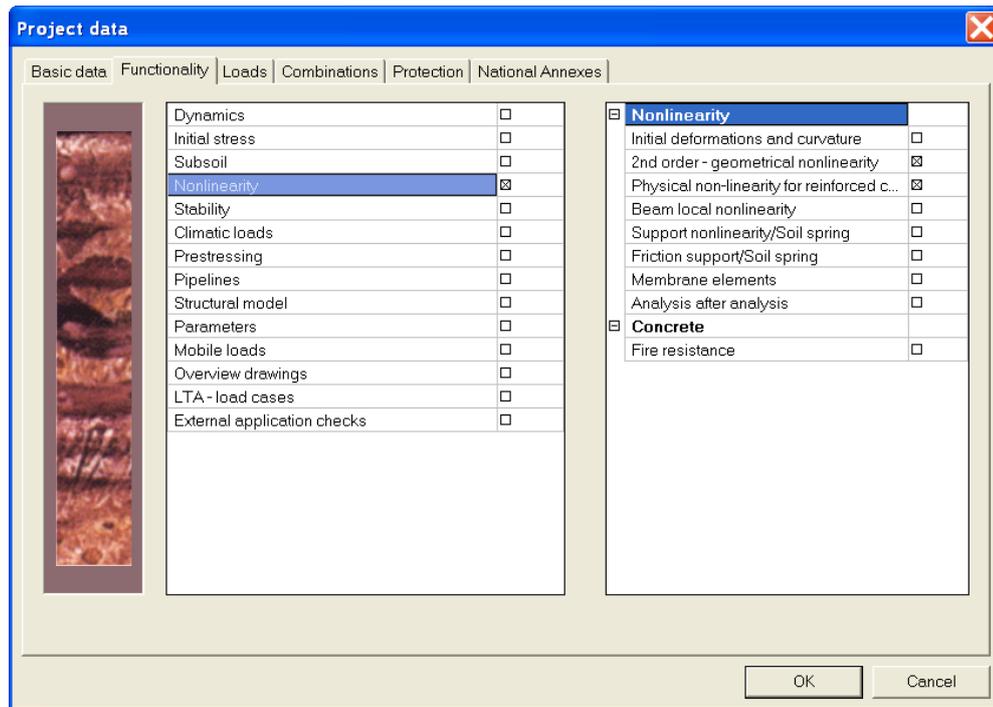
Introduction

The physically and/or geometrically non-linear calculation can be applied only to frame structures. The effect of cracks, plasticity and other factors on the stiffness are taken into account.

It is necessary to select in the Functionality settings which non-linearity is used:

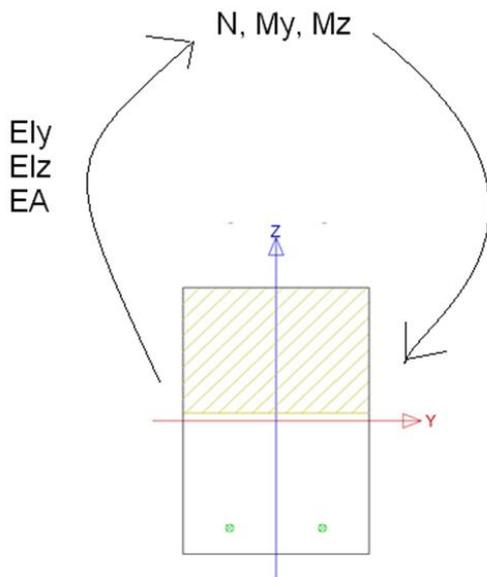
- Physical
- Geometrical
- Both

Through these settings the solver recognizes what type of calculation should be started:



Some load is applied and it is found out that cracking occurs and the weakened cross-section is determined (in fact, this is a bit more complicated, but the full description of the problem exceeds the scope of this text). The stiffness is modified in places where cracks appear and the calculation runs once more. Internal forces that are based on the modified stiffness are calculated, but the new internal forces usually still do not correspond to the stiffnesses. Therefore, the stiffnesses have to be determined once more. This procedure is repeated as many times as necessary to meet the convergence criterion. The Physically (and geometrically) non-linear calculation is a calculation performed in several, usually many iteration steps and the result is such a state of the structure when strain corresponds to internal forces (contrary to Calculation of deflections according to a standards calculation).

The following diagram shows this principle:



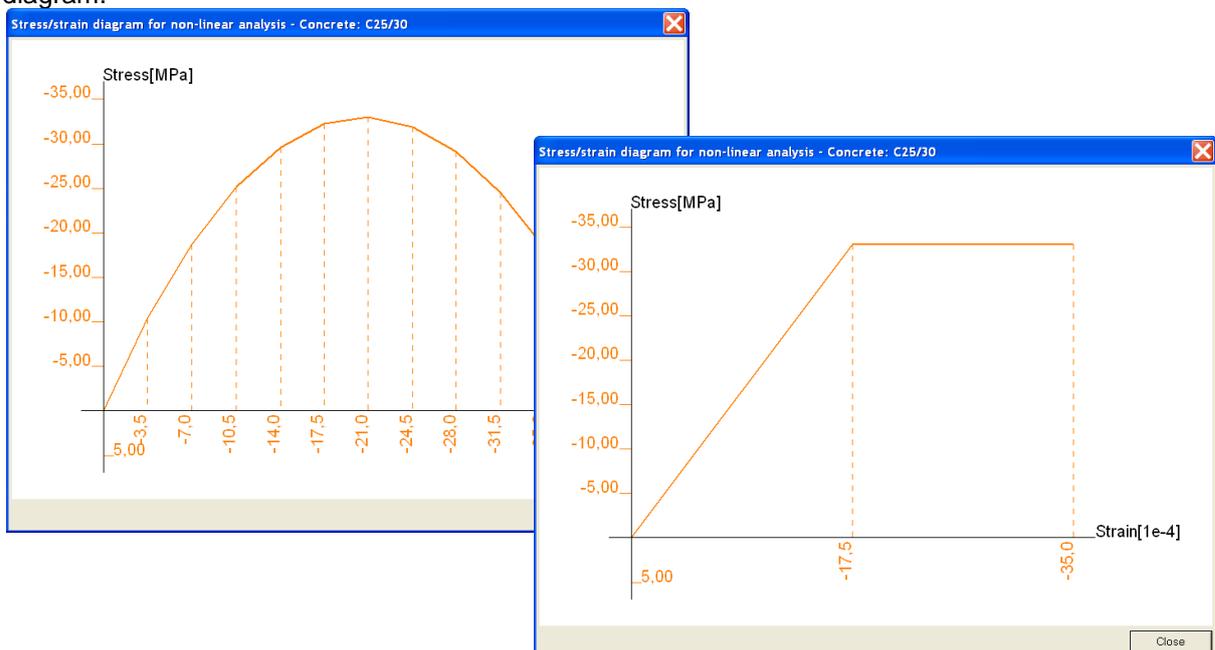
This non-linear calculation is the way to find the limit resistance not of a single section, but of the whole structure. And it is the main goal of the Physically (and geometrically) non-linear calculation analysis.

Material properties for a physically non-linear calculation

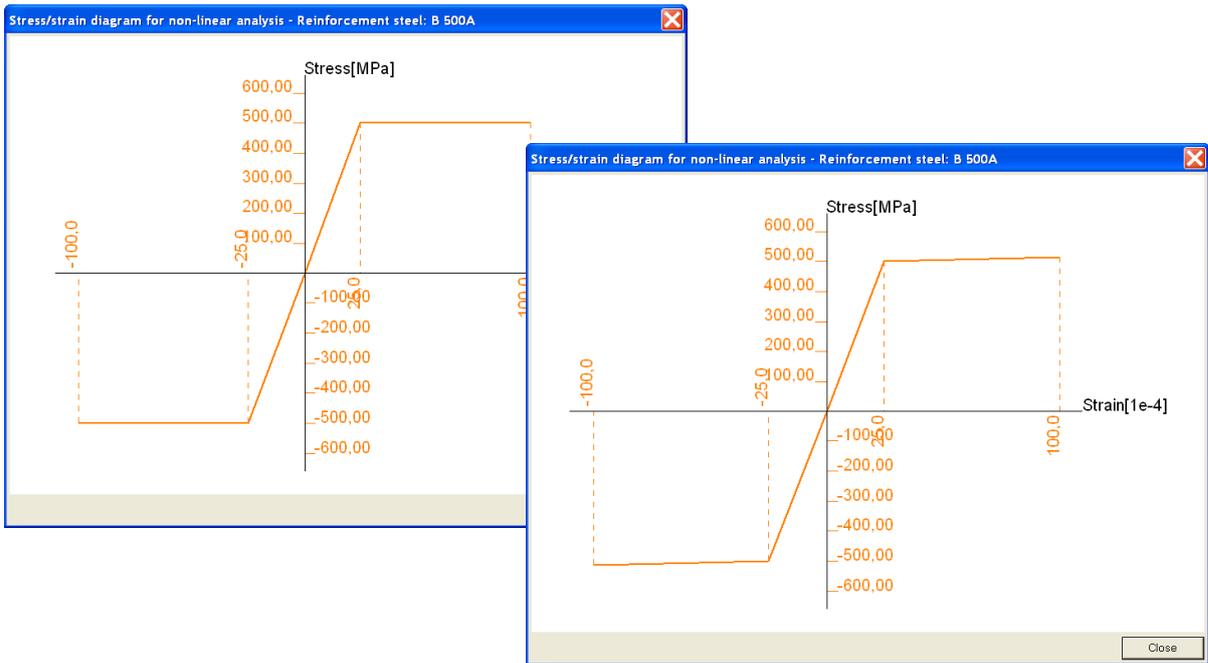
At the very beginning, we must check the settings in the material manager, verify that the option non-linear stiffness is selected, and if required, to make necessary settings. This non-linear stiffness is selected by default in a new project where Physical non-linearity is adjusted on the Functionality tab of the Project setup.

<input type="checkbox"/> Non-linear behaviour	
Non-linear behaviour	<input checked="" type="checkbox"/>
<input type="checkbox"/> Stress/strain for non-linear analysis	
Stress/strain	By code without tension part
Type of diagram	Parabolic
Diagram	...

If we have adjusted the non-linear stiffness of the concrete material, we can adjust its stress-strain diagram.

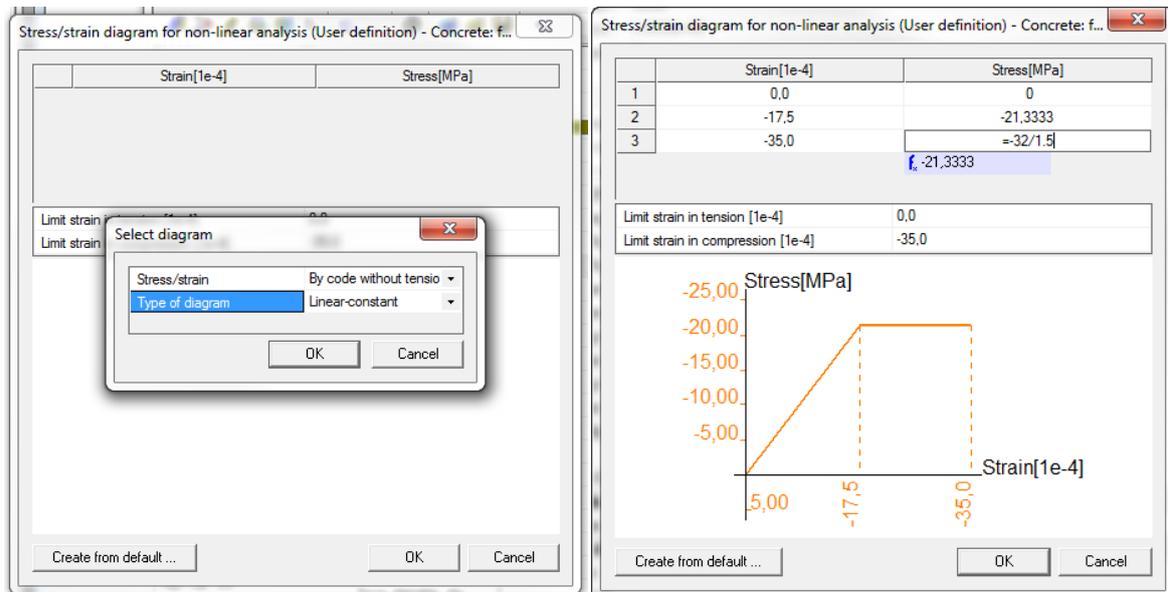


The same can be done for the reinforcement:



Remark:

1. If we open the material manager and start editing of a selected material, we can (better must) select the non-linear stiffness. Then we can select the stress-strain diagram that corresponds to our particular situation. Alternatively, we can input our own stress-strain diagram that can be input "from scratch" or derived from values stipulated in a standard.
2. In a physically non-linear calculation the stress-strain diagrams are NOT taken as defined in the appropriate standard, but AS defined by the user in the material manager. So to take into account the partial safety factors for concrete and steel (when using bi-linear or parabolic-with-linear-part), then you will have to adjust the stress-strain diagram manually



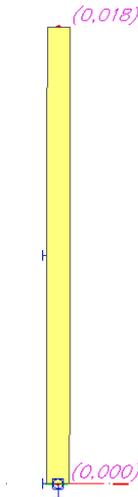
Properties for a geometrically non-linear calculation

Geometrically refers to the initial inclination of a 1D element.

Suppose a concrete column. In reality the column will not be perfectly vertical, but it will have a slight inclination.

This can be input on two ways:

- Manual introduction of the inclination. In the following picture this principle is shown for the X-coordinate:



- A second option is the input of the inclination as an option in the non-linear combinations:

Name	NLCombi1
Description	geometrical calculation
Type	Ultimate
Contents of combination	
P [-]	1,20
v [-]	1,50
Bow imperfection	None
Global imperfection	Simple inclination
dx [mm/m]	5,0
dy [mm/m]	5,0

Configuration of the solver and setup

Solver

The following properties are important for a PGNL-calculation:

Nonlinearity	
Maximum iterations	50
Geometrical nonlinearity - II. order	Newton-Raphson
Number of increments	5
Coefficient for reinforcement	1
Physical nonlinearity	
Convergence criteria [%]	5
Precision of cross-section iterati...	1
Number of cross-section iteratio...	50

First of all, the second order method has to be defined, namely Timoshenko or Newton-Raphson. For more information about these two methods, we refer to the Workshop 'Non linear and Stability'.

In the case of Newton-Raphson, the number of increments has to be defined. For both, Timoshenko and NR a number of iterations has to be input, the default value is 50.

Next, we see some parameters for the physical nonlinearity. The iteration stops if the following convergence criterion is met:

$$\left(\sqrt{\sum_{j=1}^3 u_{i,j}^2} - \sqrt{\sum_{j=1}^3 u_{i-1,j}^2} \right) / \sqrt{\sum_{j=1}^3 u_{i,j}^2} \leq 0,005,$$

where

u_j is displacement ($j=1$ for x , 2 for y , 3 for z),
 i is the i -th step of the physically nonlinear calculation.

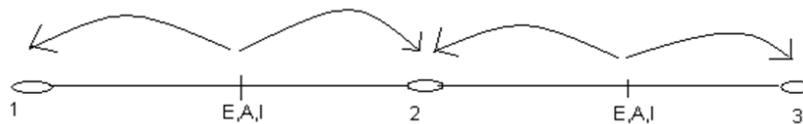
If the condition is met, the problem converges, the calculation is stopped and the results are published. If the convergence criterion is not met even after the specified maximum number of steps of iteration, the problem does not converge. In that case the calculation is terminated and no results are presented.

Mesh

For a PGNL-calculation it's very important to refine the mesh. Pay attention to the option Average number of tiles of 1D element:

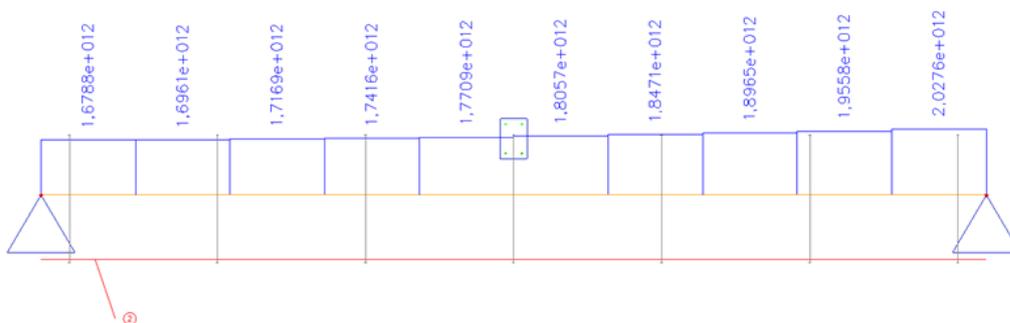
Mesh	
Minimal distance between two points [m]	0,001
Average number of tiles of 1D element	25
Average size of 2D element/curved element [m]	1,000

The new stiffnesses are calculated in the middle of every mesh element. However, since the internal forces are not shown, the stiffnesses are distributed to the nodes of the mesh elements.



So, this means that if the default value of 1 is kept, only one constant value of the stiffness is obtained. But if we divide the beam into different mesh elements (recommended > 10 elements, but this depends on the size of the beam), more precise results will be given.

You can see this very clear in the following picture:



The importance of this will be emphasized later if we have a look at the examples.

Beam related parameters for PNL-calculations

To define which concrete elements have to be calculated with a non-linear stress-strain diagram, we have to assign member data to the concerned beams.

For this, go to Structure > Model data > Beam-PNL member data

PNL Member data	
Name	FNL 2
Physical non-linear stiffness	<input checked="" type="checkbox"/> yes
Reinforcement taken from	Practical ONLY

Name	Identifies the member data.
Physical non-linear stiffness	<p>If ON, the particular beam is considered to behave physically non-linearly.</p> <p>If OFF, that beam is treated as in a standard linear calculation.</p>
Reinforcement taken from	<p>Specifies the reinforcement in the beam.</p> <p>The meaning of individual options is explained in the following text.</p>

When these data are input, two questions must be answered:

- "Do you want to have physically non-linear stiffness?"

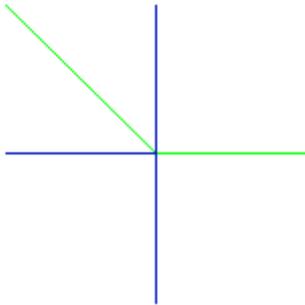
And if so,:

- "What type of reinforcement do you want:
 - (i) no reinforcement, (ii) only practical (provided), (iii) only design (or required) areas, (iv) practical and if no such reinforcement is found, then use the design A_s , or (v) the design reinforcement and if it is missing, then use the practical (provided) reinforcement?"

PNL Member data	
Name	PNL 1
Physical non-linear stiffness	<input checked="" type="checkbox"/> yes
Reinforcement taken from	1st Practical, 2nd Design A_s No reinforcement Practical ONLY Design A_s ONLY 1st Practical, 2nd Design A_s 1st Design A_s , 2nd Practical

For each choice, a different type of label is shown:

- No reinforcement
- 1st Practical, 2nd Design As (if Practical reinforcement is not available and no theoretical design was performed)
- 1st Design As, 2nd Practical (if no theoretical design was performed and no practical reinforcement is available)



- Practical only (if practical reinforcement is present)
- 1st Practical, 2nd Design As (if Practical reinforcement is available)
- 1st Design As, 2nd Practical (if no theoretical design was performed and practical reinforcement is available):



- Design only (if a theoretical design was performed)

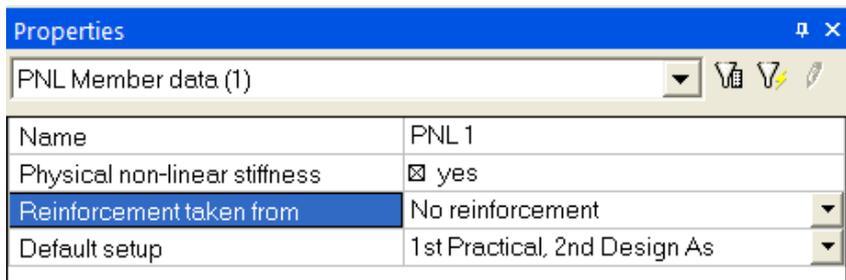


Once, the member data are input, a new options appears in the property menu: Default setup. This will be explained by means of an example:

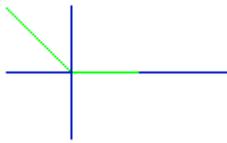
The following parameters are input in the member data:

PNL Member data	
Name	PNL 2
Physical non-linear stiffness	<input checked="" type="checkbox"/> yes
Reinforcement taken from	1st Practical, 2nd Design As

However, there was no input of practical reinforcement and above this, no theoretical reinforcement design was performed. In that case, the program recognizes that there is no reinforcement in the member and the following properties can be seen after the input of the member data:



With the following symbol:



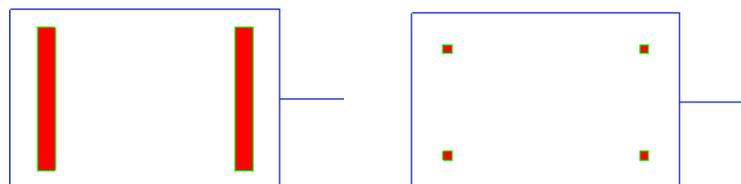
So, the program will perform a PGNL-calculation with the parameter in 'Reinforcement taken from' but it also remembers the initial setting.

Reinforcement in the physically non-linear calculation

Although it is possible to run the Physically (and geometrically) non-linear calculation without any reinforcement, it is strongly recommended to input the real reinforcement (i.e. the provided reinforcement and not just the required reinforcement areas) before running the Physically (and geometrically) non-linear calculation. On the other hand, it is true that in case of planar frames it is possible to perform the calculation also for the required reinforcement areas

Calculation for design As (required reinforcement areas)

So, for planar frames, the procedure may be as follows. We define a standard combination of load cases and use it for the design of reinforcement (after performing a linear static calculation). We compute the required reinforcement areas and run a non-linear calculation of the planar frame. In this calculation we do not have the "balls" of the reinforcement, but a "strip" of it (see the figure below).



Strip of reinforcement – idealised design (required reinforcement)

Balls of reinforcement – real position of individual reinforcement bars

This strip will behave in a certain way. It can plasticise, or even break. This is not possible in a spatial frame as an unsymmetrical bending occurs there and we would not know what to do with that strip. We need the balls in that frame, i.e. we need the real (provided) reinforcement. That is why a spatial frame cannot be calculated with required reinforcement.

Calculation without any reinforcement

We can say that there will be no reinforcement. It means that we do not have to have any reinforcement and still we can perform the non-linear calculation. It is a reasonable choice for e.g. masonry or plain concrete. If required, the FEM solver is run repeatedly for geometrical non-linearity within each step of the physical non-linearity and this approach combines the two non-linearities. To conclude, we can run the calculation without any reinforcement, but e.g. a simply supported beam subjected to bending would fail and collapse. Such a calculation can be applied to columns subjected to eccentric compression as we know that there is a compression zone there, that a part of the section plasticises, that cracks open on one side, and that it does not matter as it shifts the centre of gravity, and that there is a compressive stress causing balance in the section and everything is in order.

Procedure for a PNL-calculation

The procedure can be briefly summarised in the following steps:

1. Definition of material properties
2. Input of the structure model including load, boundary conditions, etc.
3. Input of member-related parameters (called "PNL Member data")
4. Refinement of the mesh (number of tiles of 1D elements)
5. Linear calculation – to be able to design or input the reinforcement
6. Design or input of reinforcement
7. Non-linear calculation
8. Review and assessment of results

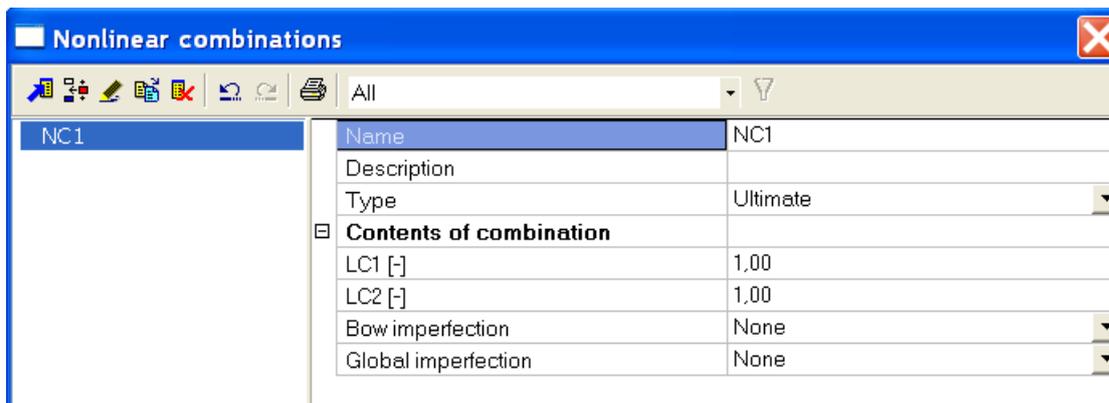
Type of load case combinations

This text is not exclusively related to the Physically (and geometrically) non-linear calculation, but it may be useful as a kind of revision of what should be clear in this context.

In SCIA Engineer there are:

- Usual combinations of load cases (let us call them common combinations)
- Non-linear combinations for a physically non-linear calculation
- Special combinations for concrete that are reserved for the earlier-mentioned Calculation of deflections according to a standard (CDD-calculation).

Unless we have defined a non-linear combination, the Physically (and geometrically) non-linear calculation cannot be performed.



For a physically non-linear calculation it is sufficient to define a non linear combination like this.

For a physically and geometrically non linear calculation the global imperfection still has to be set.

Analysis of the results

First of all, the following remarks have to be made:

- The results cannot be reviewed in the service Concrete, as they do not represent any check. It is true that we have calculated with concrete, but the results are general results of a general non-linear calculation and therefore they are in the service Results > PNL stress/strain and PNL stiffness.
- Steel beams are calculated linearly

The following results can be obtained:

PNL stress/strain

eps cc	compressive strain in concrete, meaningful only for beams made of concrete
eps ct	tensile strain in concrete, meaningful only for beams made of concrete
eps rt	tensile strain in reinforcement, meaningful only for beams made of reinforced concrete
eps rc	compressive strain in reinforcement, meaningful only for beams made of reinforced concrete
eps st	tensile strain in steel, meaningful only for beams made of steel
eps sc	compressive strain in steel, meaningful only for beams made of steel
sigma cc	compressive stress in concrete, meaningful only for beams made of concrete
sigma ct	tensile stress in concrete, meaningful only for beams made of concrete
sigma rt	tensile stress in reinforcement, meaningful only for beams made of reinforced concrete
sigma rc	compressive stress in reinforcement, meaningful only for beams made of reinforced concrete
sigma st	tensile stress in steel, meaningful only for beams made of steel
sigma sc	compressive stress in steel, meaningful only for beams made of steel

PNL stiffnesses

EAx	Non-linear axial stiffness.
Ely	Non-linear flexural stiffness, i.e. the connection presented in the last iteration step.
Elz	Non-linear flexural stiffness, i.e. the connection presented in the last iteration step.
xr	Height of compression zone, or the position of neutral axis.
As	Total reinforcement area in a given section that is considered in the calculation of non-linear stiffness. This area is multiplied by the coefficient for reinforcement from the Solver setup.

Examples of a P(G)NL calculation

- **PNL01.esa: Plasticization in supports**

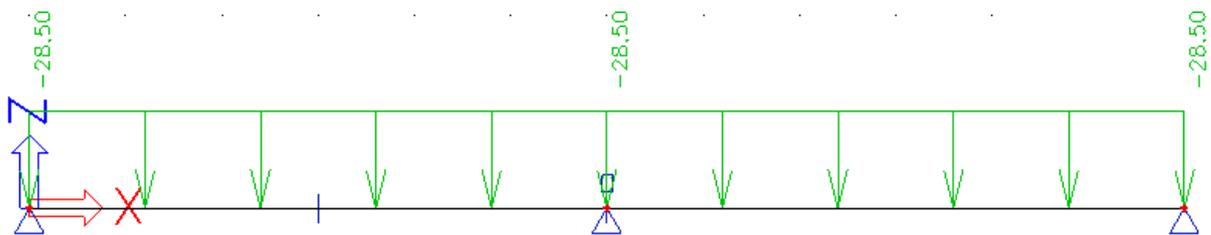
To show the effect of plasticizing that may be "discovered" in the physically non-linear calculation, let us consider the following simple structure.

We create a two-span (2 × 6.0 m) continuous beam with concrete cross-section 500 × 300 mm. We input the practical reinforcement whose details are not important for the purpose of our explanation. Only physical non-linearity is considered.

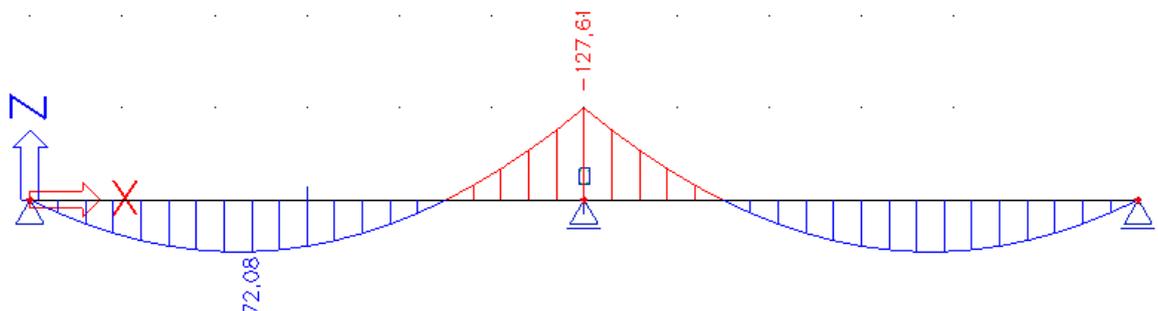
The beam is subjected to a uniformly distributed load $q = 28,5 \text{ kNm}^{-1}$.

When we apply the analytically derived formula for the hogging moment above the intermediate support of a two-span continuous beam, we obtain:

$$M = 0.125 \times q \times l^2 = 0.125 \times 28.5 \times 6^2 = \underline{128.25 \text{ kNm}}.$$

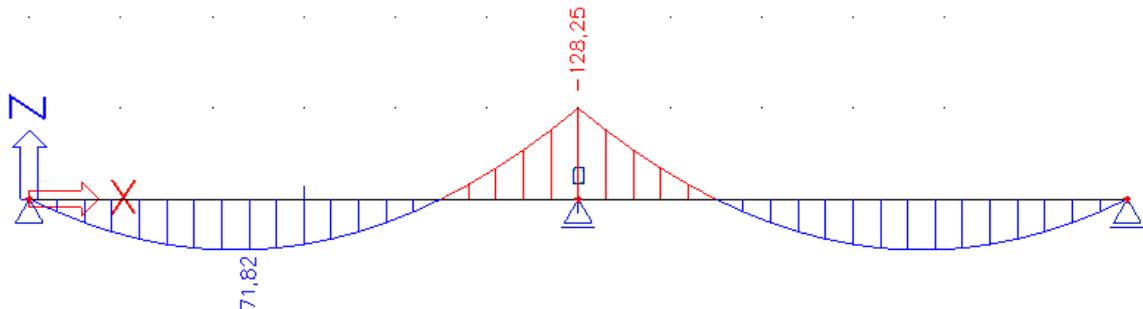
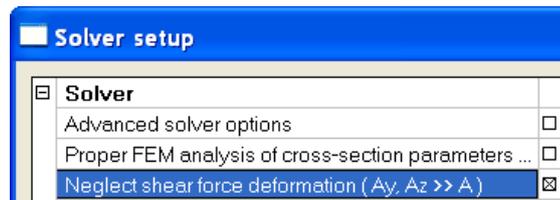


The linear calculation in SCIA Engineer gives the bending moment calculated above the intermediate support is equal to 127,6 kNm, which is very close to the direct solution.

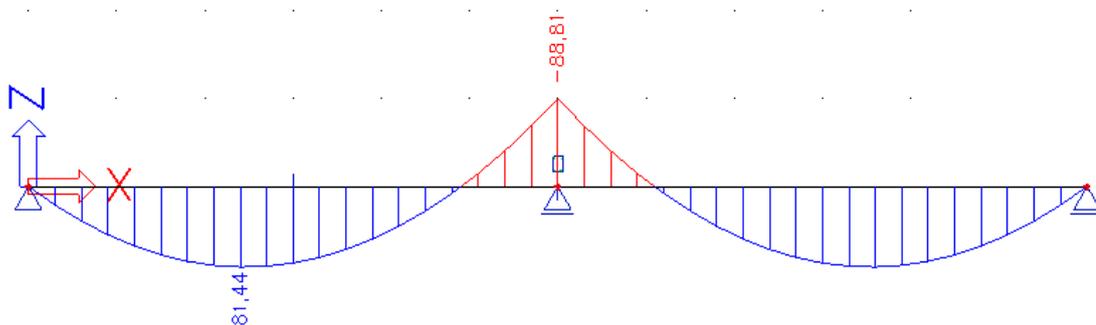


Remark:

The small difference is due to the fact that SCIA Engineer takes into account the shear force deformation. If we neglect this, the same result as in the manual calculation is obtained:



When we apply the non-linear calculation, the change of cross-section stiffness due to formed cracks results in a redistribution of internal forces.



The bending moment calculated above the intermediate support is equal to 88.8 kNm.

We may look at the correctness of the moment redistribution by taking the bending moment in the middle of the span (not shown in the picture) of 83.9 kNm and performing a simple calculation: moment in the middle of the span + half of the redistributed hogging moment = hogging moment before the redistribution = $83.9 + 88.8/2 = 128.3$, which almost precisely equals to the direct solution stated above.

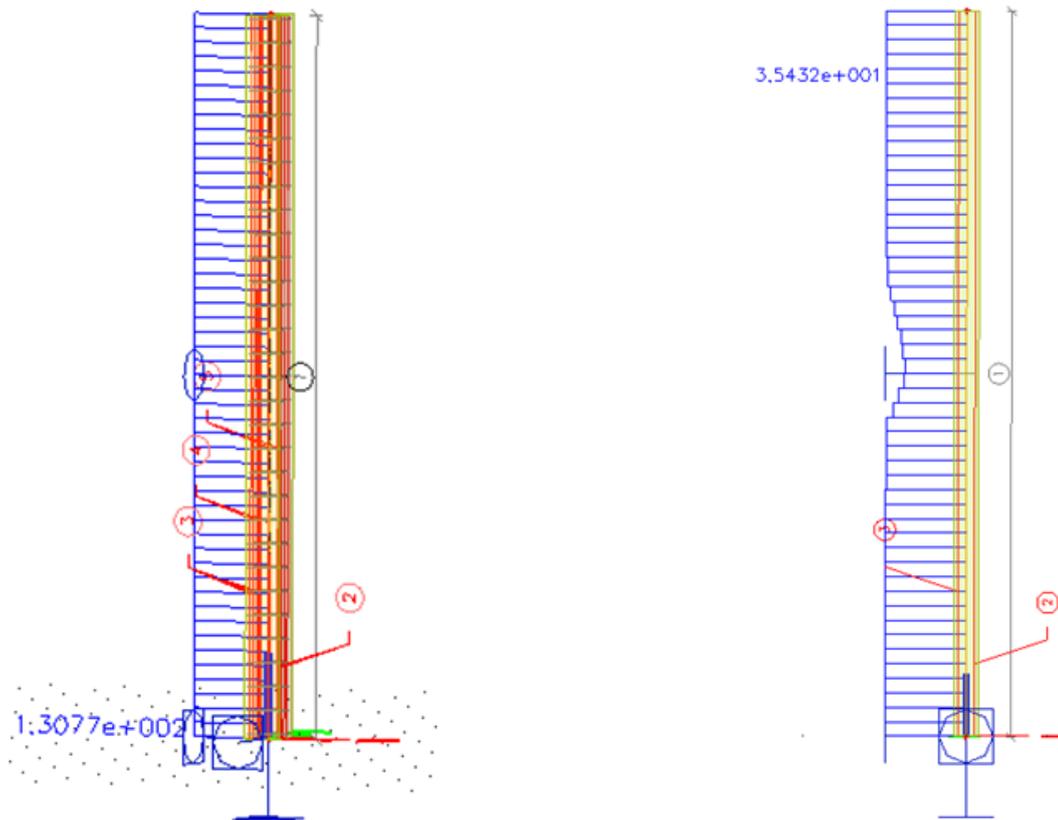
If the loading would be higher then an instability would be found, because a hinge in the middle of the beam causes a mechanism. By using the batch optimizer in ODA, the maximum loading on the beam could be determined. By this method the maximum loading on a structure can be defined.

This indicates the practical use of a PNL calculation. It is possible to define whether a structure is stable or unstable. In case of a stable structure it is possible to find the weak points of the structure and the stress/strain distribution on these points. In this way proper measures can be taken to make the structure stronger.

- **PNL02.esa and PNL03.esa: Uncracked columns versus Cracked column**

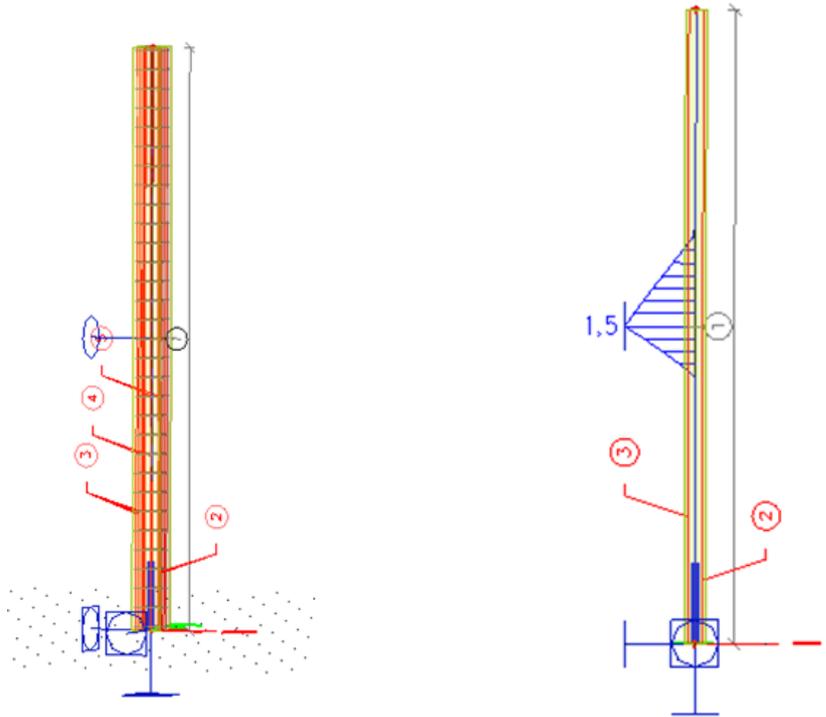
Both projects contain a concrete column with practical reinforcement. The column in PGNL03.esa however is supported in the middle.

After a non-linear calculation, the stiffnesses EI_y are regarded:



It's obvious that the right column will crack in the middle at the location of the support.

If we display the strain ϵ_{ct} (concrete in tension) we see the following:



The single check for the stresses shows the cracked section (part of the section is in tension):

Single cross-section

Calculation

Step / position: 0.5 / 4.5

automatic recalculation-redraw

Selected section

Calculation

Code setup

Extreme | My-

N	2130.2782	kN
Vy	0	kN
Vz	-33.7872	kN
Mx	0	kNm
My	-112.805	kNm
Mz	0	kNm

Calculation

Numerical output

Change of setup

Output

Cross-section | Loads | Strain | Stress | Strain 3D | Stress 3D | Forces 3D | Stress/strain diagram

Stress [MPa]

Dimensioning is drawn in the direction of in-plane gradient of the strain plane [mm]

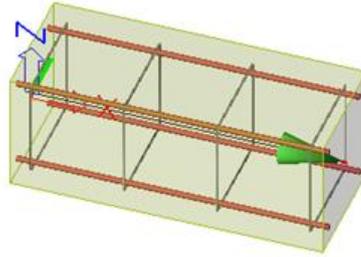
Stress distribution diagram showing values: 181.3 and -31.0

eps cc=-11.0724 *1e-4 > eps cc.lim=-35.0000 *1e-4
 eps ct=-9.0648 *1e-4 < eps ct.lim=5000.0000 *1e-4
 eps sc=-9.0648 *1e-4 > eps sc.lim=-100.0000 *1e-4
 eps st

Calculation successful. There are no warnings or no errors. (1)

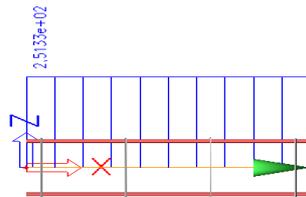
OK Cancel Apply

- **PNL04.esa: beam under tension**



Calculation of the physical non linear stiffness EA_x :

$$EA_{x,PNL} = E_b \cdot A_s \cdot n \quad (\text{with } n = \frac{E_s}{E_b})$$



Calculation of the linear stiffness EA_x :

$$EA_{x,lin} = E_b \cdot A_b + E_s \cdot A_s$$

The linear stiffness takes into account the stiffness of the concrete in tension as well.

The difference in results between linear and non linear calculation becomes clear if we look at the axial deformation of the beam.

$$U_x = \frac{N}{EA_x} * L$$

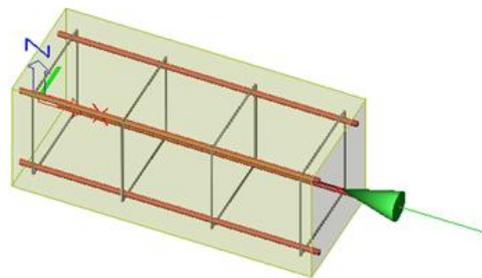
Linear calculation shows a U_x deformation of 2mm.

The PNL calculation shows a U_x deformation of 0.1mm.

So by not taking into account the stiffness of the concrete in tension, the stiffness is reduced 20 times.

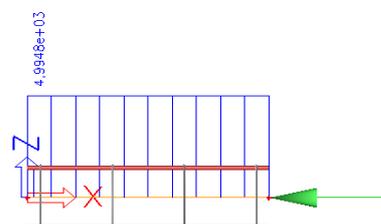
- **PNL05.esa: beam under pressure**

In this case, the part of the concrete cross-section is also taken into account:

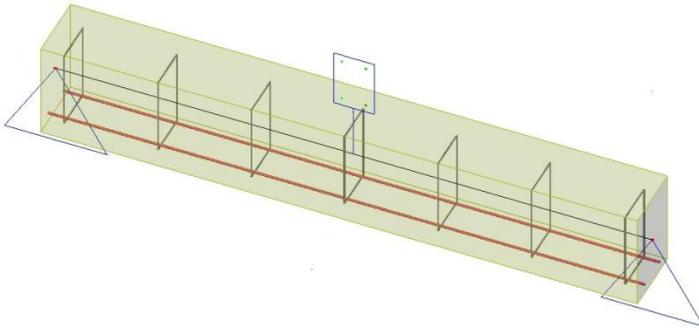


Calculation of the stiffness EA_x :

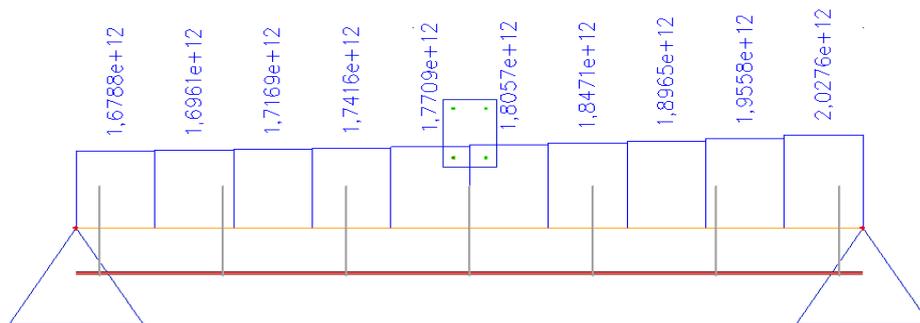
$$EA_x = E_b \cdot A_c + E_b \cdot A_s \cdot n \quad (\text{with } n = \frac{E_s}{E_b})$$



- PNL06.esa: Beam supported at two ends

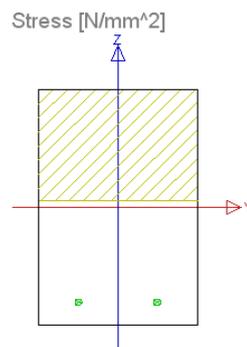


If we calculate the stiffness EI_y , we remark a slight decrease at the left support:



Pay attention to the number of mesh elements. If we had taken the default value of 1 element per beam, the stiffness was constant along the length. So, to obtain precise results the mesh has to be refined.

If we go to the section check, it's clear that the section is assumed to crack:



A new surface is calculated:

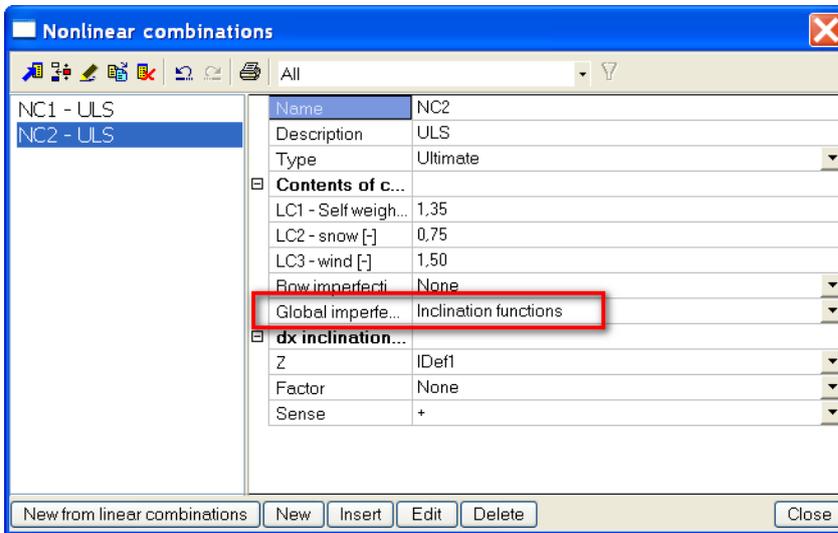
$$A_c = A_{s,tot} \cdot n + b \cdot x_u$$

And the new stiffness I is calculated by means of the following formula:

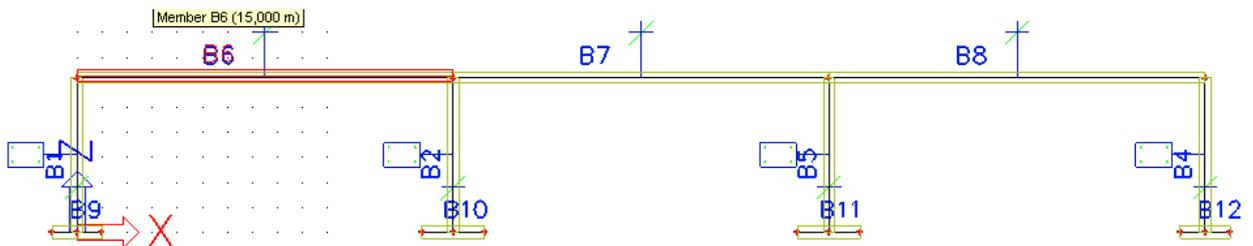
$$I = e_1^2 \cdot n \cdot A_{s,tot} + b \cdot x_u \cdot e_2^2 + \frac{b}{12} \cdot x_u^3$$

- **PGNL07.esa: Practical example**

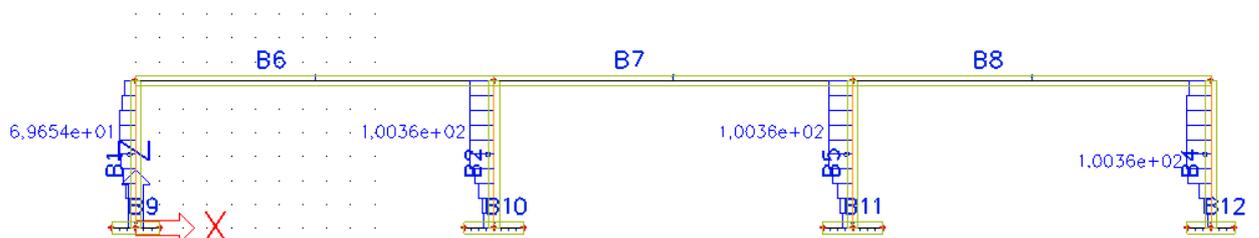
This example is about a concrete construction with columns and beams. This will be a PGNL calculation because also a geometrical non linearity is defined for the columns.



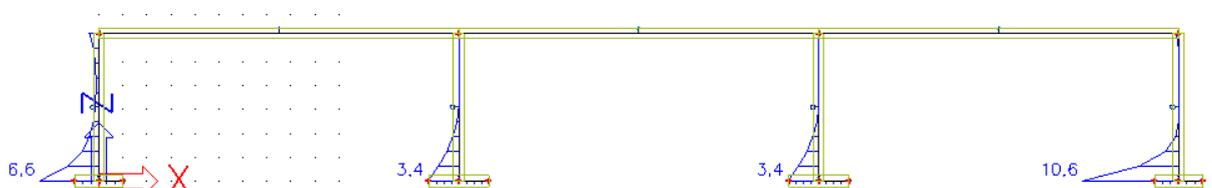
PNL data will have to be set on every beam of the construction. To simplify the calculation we are only going to take into account the Physical non linear stiffness of the columns. The columns are expected to be the critical part of the structure and are therefore calculated non linear.



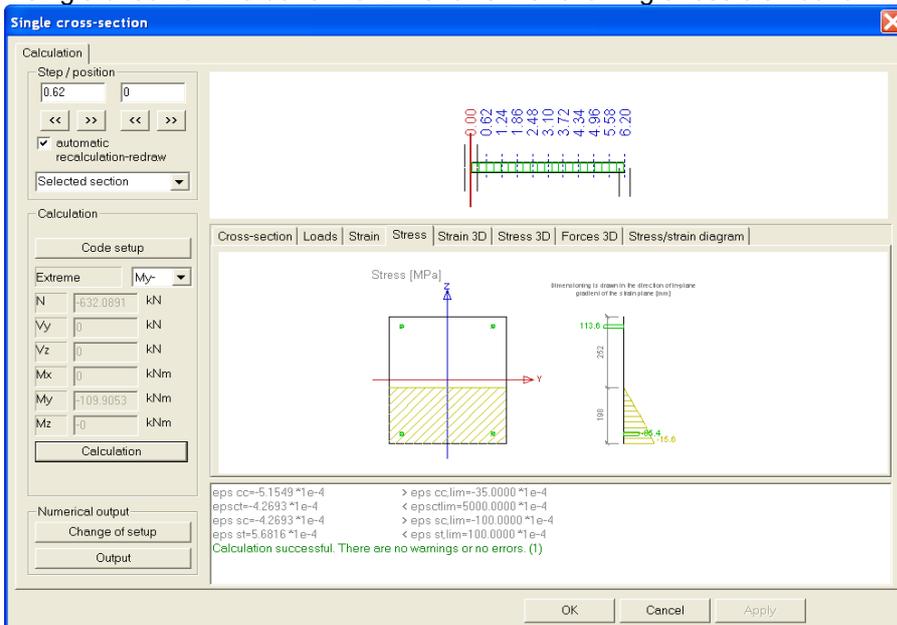
A PGNL calculation shows the adapted stiffnesses EI_y of the columns.



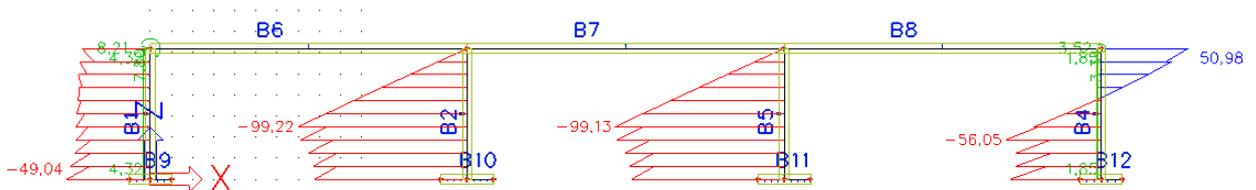
It is clear that reduced stiffnesses are found on the bottom of the columns because there tension in the concrete is expected. This can be verified if we look at the tension strains in the concrete.



A single check on the bottom of B1 shows the following stress distribution.



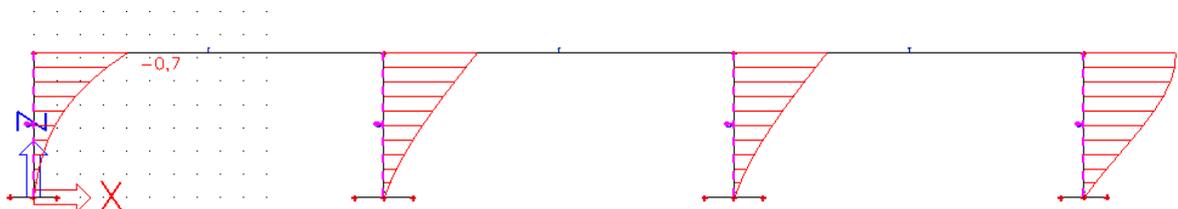
The moment distribution in the columns is as shown below. The reduced stiffnesses have led to a redistribution of internal forces during the iteration process.



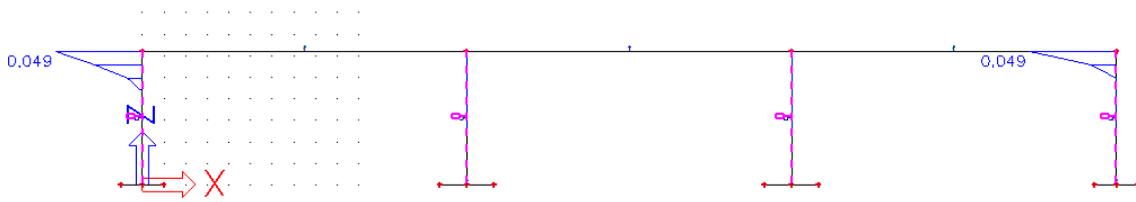
To conclude this construction is still stable but cracks will occur on the bottom of the columns.

To show more clearly the effect of the geometrical non linearity we are taking a look at the snow load only.

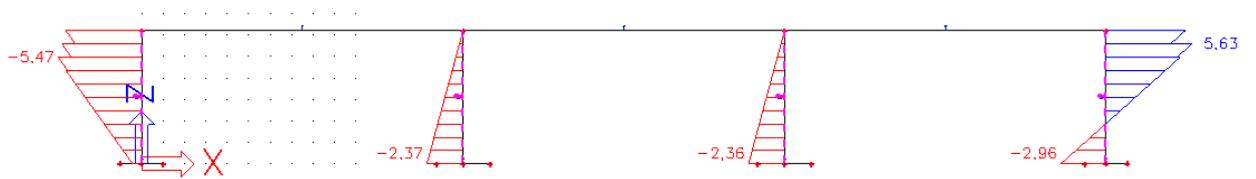
This snow load will cause the following deformation u_z of the structure, taking into account the geometric non linearity which is defined by a global imperfection.



This will cause tension strains ϵ_{ct} in the concrete on top of the side columns and thus leading to a reduced stiffness there.



The moment distribution is then as shown below.



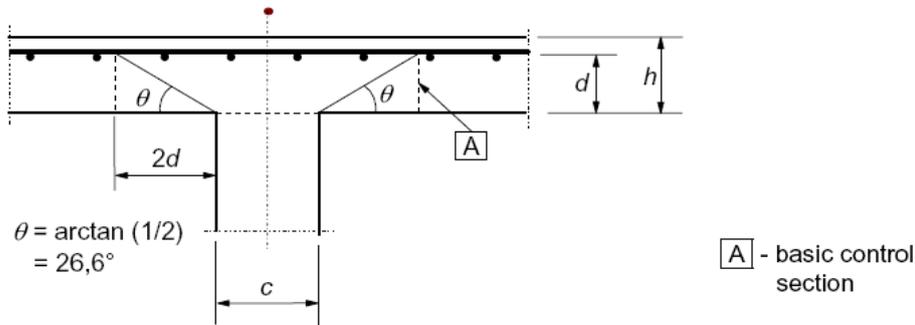
Punching shear check

1_Theoretical background – Eurocode

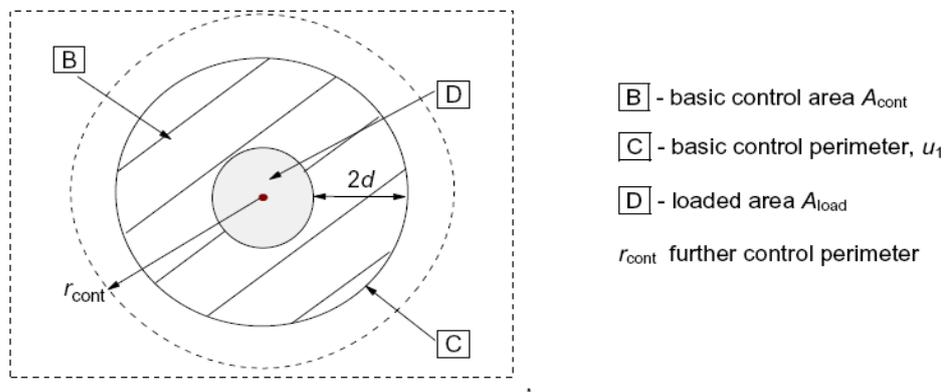
Punching shear can result from a concentrated load or reaction acting on a relatively small area, called the loaded area A_{load} of a slab or a foundation.

Following pictures show the verification model for checking punching failure at the ultimate limit state.

The design shear stress around a column-plate connection will be checked along one or more control perimeters, which are defined as follows: (EC § 6.4.1)

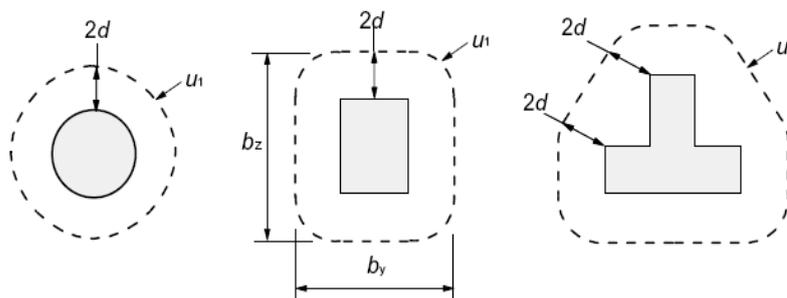


a) Section

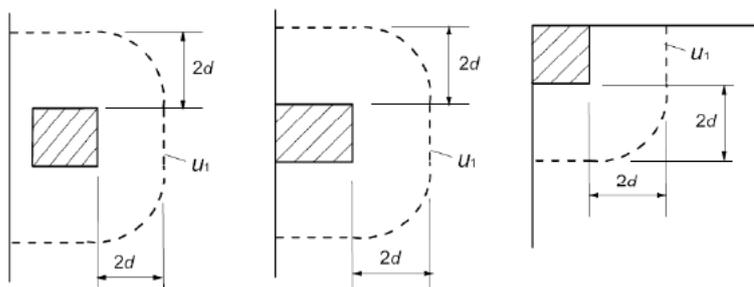


b) Plan

The basic control perimeter u_1 is normally taken at a distance $2d$ from the loaded area:



In case the loaded area is close to an edge or corner:



The value of d , the effective depth of the slab, is assumed constant and may normally be taken as:

$$d_{\text{eff}} = (d_y + d_z)/2 \quad (\text{EC formula 6.32})$$

where d_y and d_z are the effective depths of the reinforcement in two orthogonal directions.

The punching shear calculation according to EC § 6.4.3 is as follows:

The design procedure for punching shear is based on checks at the face of the column and at the basic control perimeter u_1 . If shear reinforcement is required, a further perimeter $u_{\text{out,ef}}$ should be found where shear reinforcement is no longer required.

The design shear resistances along the control sections are:

- $V_{Rd,c}$: design value of the shear resistance of a slab *without* punching shear reinforcement along the control section considered
- $V_{Rd,cs}$: design value of the punching shear resistance of a slab *with* punching shear reinforcement along the control section considered
- $V_{Rd,max}$: design value of the *maximum* punching shear resistance along the control section considered

The following checks have to be performed:

- Check at the face of the column (perimeter u_0):

$$v_{Ed0} \leq v_{Rd,max}$$

with v_{Ed0} the design shear stress at the column perimeter u_0

- Check at the basic control perimeter u_1 :

If $v_{Ed} \leq v_{Rd,c}$: Punching reinforcement is not needed

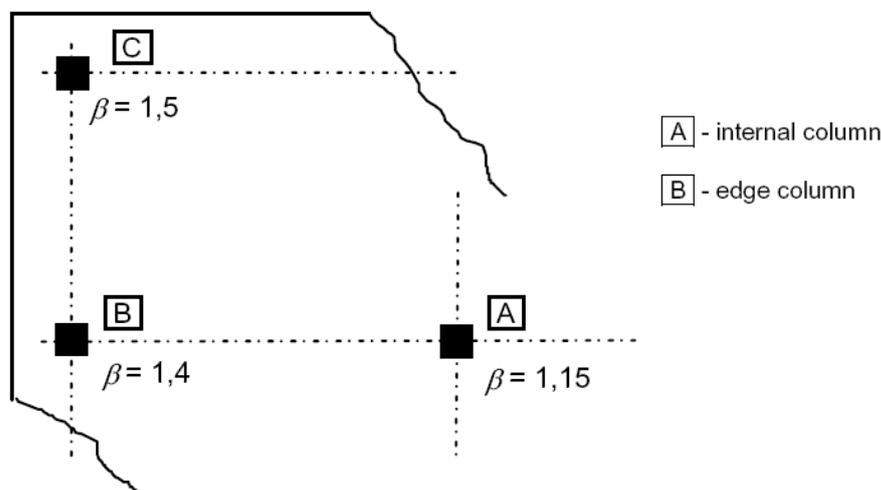
If $v_{Ed} > v_{Rd,c}$: Design of the punching reinforcement A_{sw} according to:

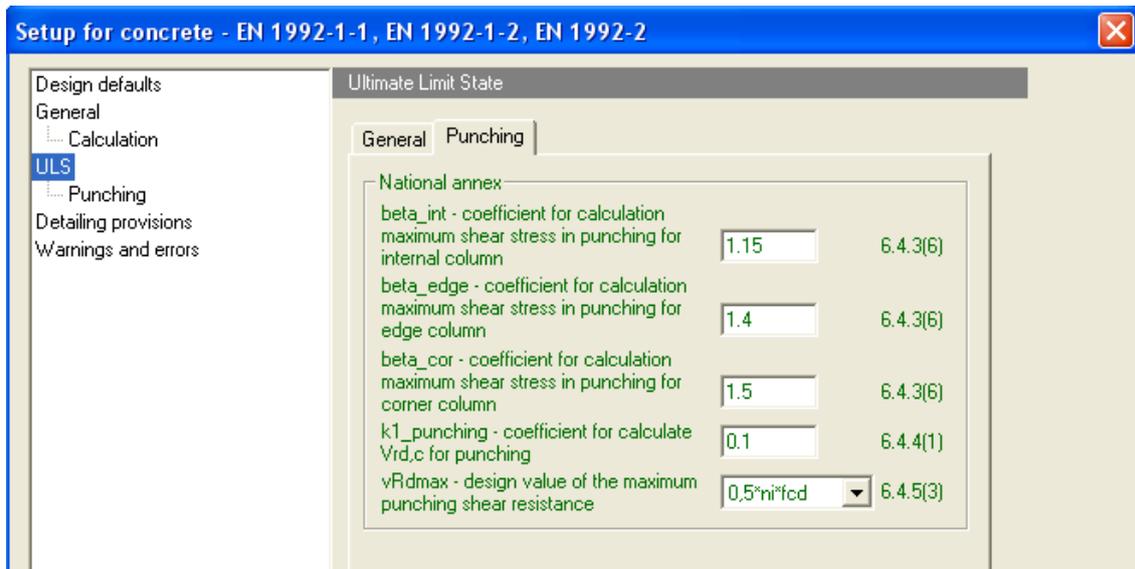
$$v_{Ed} = V_{Rd,cs} = 0,75v_{Rd,c} + 1,5\left(\frac{d}{s_r}\right)A_{sw}f_{ywd,ef}\left(\frac{1}{u_1d}\right)\sin\alpha \quad (\text{EC formula 6.52})$$

Remark: Where the support reaction is eccentric with regard to the control perimeter:

$$v_{Ed} = \beta \frac{V_{Ed}}{u_i d} \quad (\text{EC formula 6.38})$$

Values for β are given in the National Annexes of the Eurocode, but the recommended values for EN are set as default in the Concrete Setup:

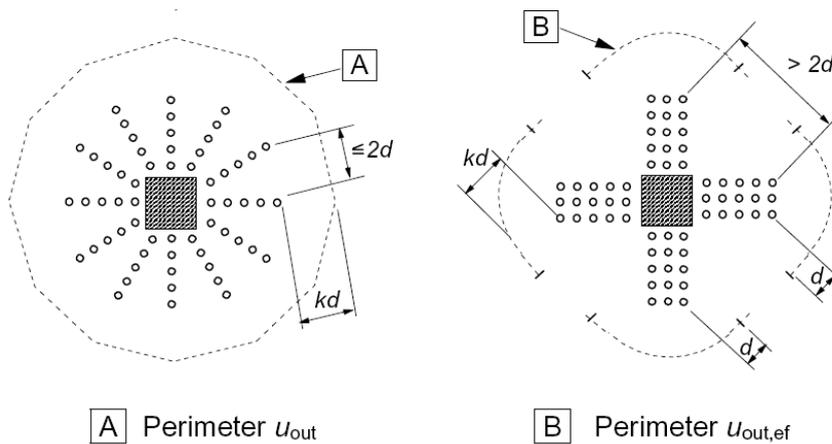




The control perimeter at which shear reinforcement is no longer required, u_{out} or $u_{out,ef}$:

$$u_{out,ef} = \frac{\beta V_{Ed}}{v_{Rd,c} d} \quad (\text{EC formula 6.54})$$

The outermost perimeter of shear reinforcement should be placed at a distance not greater than $k*d$ within u_{out} or $u_{out,ef}$ (recommended value of k is 1,5).



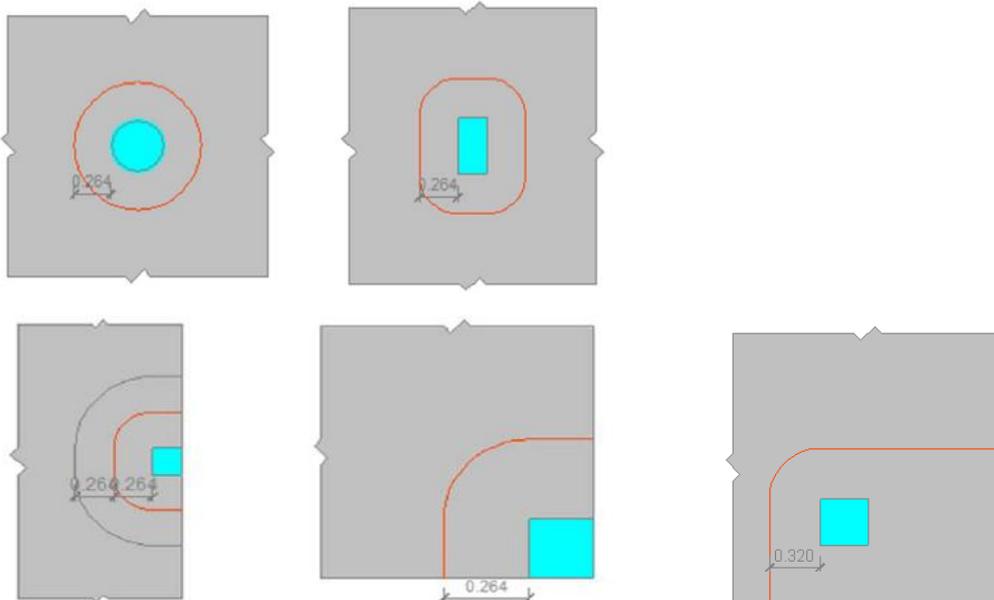
2_Punching check in SCIA Engineer

Configurations

The punching check in SCIA Engineer is only available when a real column or a nodal support have been connected to a plate. No punching check can be performed for a point load or a little surface load applied to the plate.

SCIA Engineer supports circular and rectangular cross sections for the punching check. The column position with regard to the edges of the plate is recognized; but the proximity of openings (as additional data to the plate) is not taken into account for the column position (see Extra Features). Also, for the punching check, all edges and angles of the plate are taken to be straight... so if they are not in your model, the program makes an approximation.

To sum up: the following configurations are recognized by SCIA Engineer:



Control perimeters

If required, the user can adjust the parameters for the control perimeters in the Concrete menu > Punching > Setup. By default the values given by the code are taken:

Setup for concrete - EN 1992-1-1, EN 1992-1-2, EN 1992-2

Design defaults
 General
 Calculation
 ULS
 Punching
 Detailing provisions
 Warnings and errors

Punching

Loaded area

Control perimeters for slab of ceiling at a distance: 2 d

Control perimeters for foundational slab at a distance: 0.5 d

Distance between the perimeter of the loaded area and the edge of the opening does not exceed: 6 d

Dimensions of column for use the complete perimeter does not exceed value: 3 d

Shear reinforcement

Min thickness of plate (9.3.2(1)): 200 mm

Calculation of shear resistant

Normal force to punching calculation

Column heads

Rectengular column with rect. column head $l_h \leq 2.0(0.5)h_h$ (6.4.2(8))

use rectangular loaded area

use equivalent diameter d_{crit}

Choice of A_s used for check

The punching check will check if the longitudinal reinforcement A_s in the plate is sufficient to resist to the shear force around a **column-plate** or **nodal support-plate** connection.

In SCIA Engineer the user can choose for which type of reinforcement the punching check has to be done: for A_s ,theoretical required or for A_s ,user defined.

Go to Concrete menu > 2D member > Setup > General: Calculation, to make your choice:

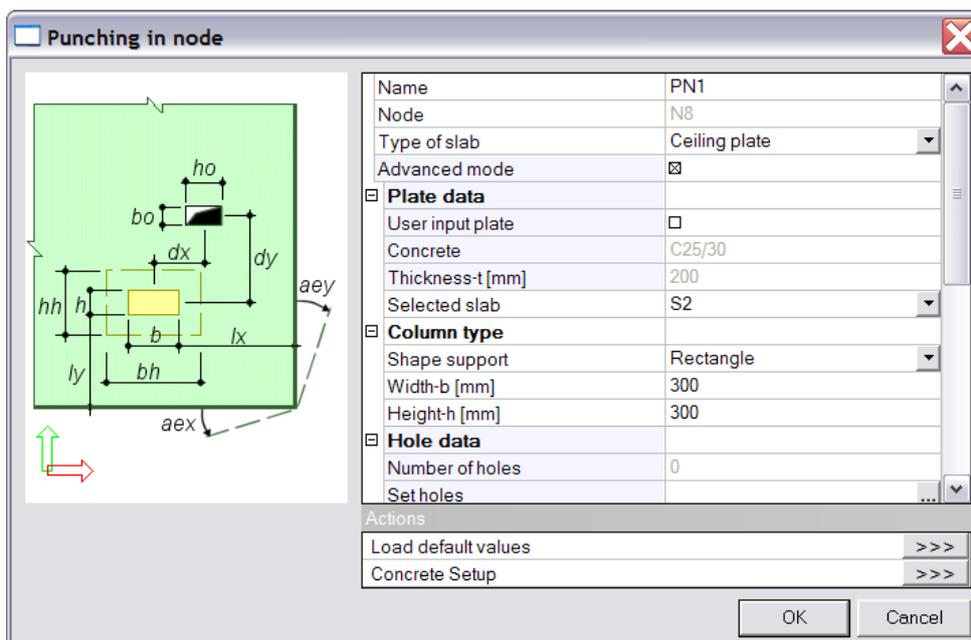
For stiffness, allowable stress, punching and crack-proof calculation use reinforcement

In order: [A_s , user]; [A_s ,tot or 0]

This default option means that when there is no A_s ,user added to a certain member, the A_s ,theoretical required will be used for the check.

Punching data

The user can add Punching data to the nodes in which there is a connection between a column or nodal support and the plate. This option can be found in the Concrete menu > Punching > Punching data:



It is not required to add Punching data to each connecting node; SCIA Engineer automatically detects the right configuration in case of a column. In case of a nodal support, and no Punching data are added, default dimensions and settings are taken into account for the check. The default support shape can be found in the Concrete menu > Setup > Design defaults. The default support dimension can be found in the properties of the support.

In case the plate is supported by a subsoil, the Type of slab is automatically put to 'Foundational plate'. Any other plate not on a subsoil is detected as a 'Ceiling plate'. This is important, because the control perimeters for a ceiling plate are taken at a distance of $2d$; for a foundational plate at a distance of $0.5d$ (by default).

For a ceiling plate, the position of each column – above or below the plate – is detected automatically by SCIA Engineer. This is important, because the position of the column determines if the upper or lower reinforcement of the plate will be taken into account for the punching check.

(If the column is above the plate, the lower reinforcement will be taken into account, because the tension is at the lower surface of the plate in this case... and vice versa.)

Punching check

Before a punching check according to the As,ULS (theoretical required) can be performed, the user has to do a design of the reinforcement in the Concrete menu > 2D member > Member design – Design ULS.

Ask for instance the results for the Class 'All ULS': the values for the theoretical reinforcement have to be loaded into the memory of SCIA Engineer, otherwise the punching check will not be available.

The results for a punching check in SCIA Engineer will be illustrated in the following example.

3_Example in SCIA Engineer

Description of used example

Open the example punching.esa

Geometry:

Concrete class C20/25

Reinforcement class B500A

Plate thickness 200 mm

Column cross-section R 300x300 mm²

Plate and columns are connected to each other by means of the action Connect members/nodes.

Loading:

*Load cases

LC 1: Self weight

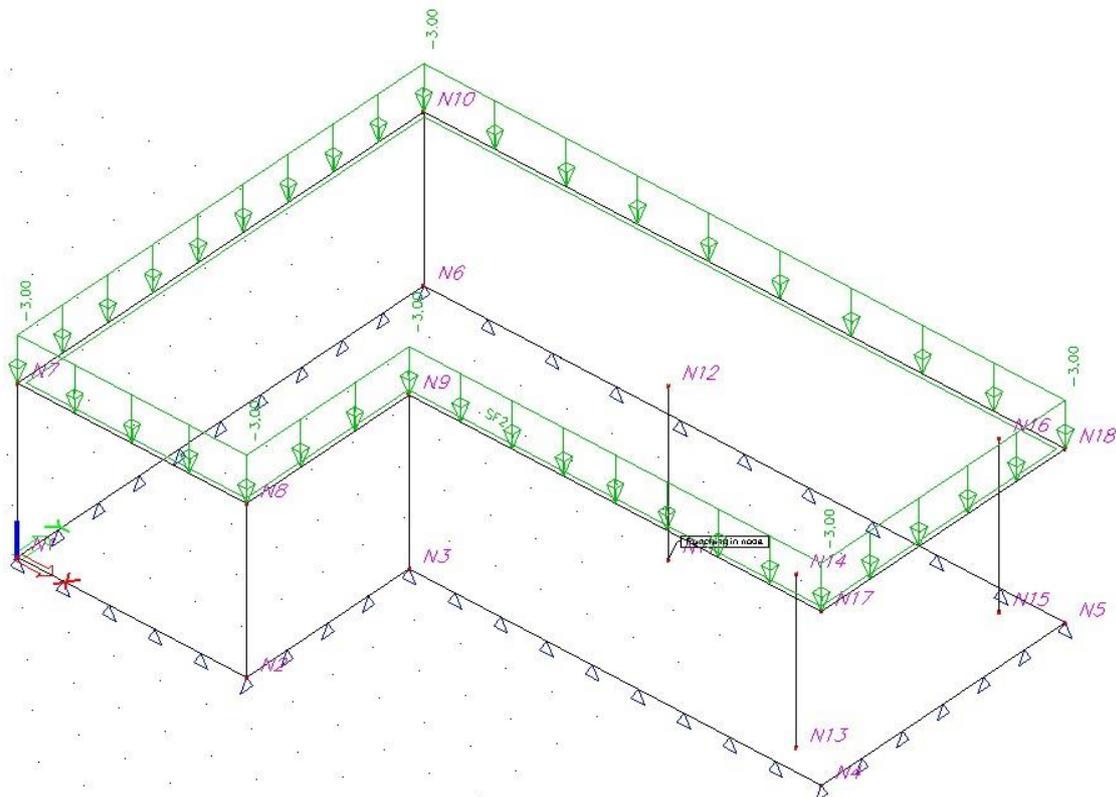
LC2: Permanent 1 kN/m²

LC 3: Service load 3 kN/m²

*Combinations

ULS (Type EN – ULS (STR/GEO Set B))

SLS (Type EN – SLS Quasi Permanent)



Work method

This model will be checked on punching, which can occur as a consequence of a point load or reaction that is applied on a small area. It's actually an addition to/refinement of the shear force check. As stated already above, SCIA Engineer can perform a punching check for a column-plate or a nodal support-plate connection.

Step 1: Mesh refinement & Calculation

*As basic rule we take the width of the mesh equal to 1 or 2 times the thickness of the 2D elements in the project. In this case we will take a mesh width of 200 mm.

Go to Main menu > Calculation, mesh > Mesh setup > Average size of 2D (mesh) element/curved element = 0,2 m.

*Start a linear calculation.

Step 2: Reinforcement results – As and Asw

*Go to Concrete menu > 2D member > Member design – Design ULS; ask the values for As and Asw. Asw is the shear force reinforcement, which seems to be required around the connecting nodes with the columns. And even Error 8 appears at some connecting nodes: 'Not dimensionable due to shear'. This shear force reinforcement is not necessarily required in reality: to calculate Asw, a check is done in the connection point where all the forces are concentrated, which results in high peaks. These peaks are singularities, and exist as a consequence of calculating with the finite element method. In reality, the connection is not a single point, but the whole area where column and plate meet.

*The peak values can both be ignored or averaged.

SCIA Engineer offers the possibility to work with averaging strips: the values of all of the finite elements which lie *entirely* in the averaging strip, are averaged. Go to Concrete menu > 2D member > Averaging strip. Set 'Type' to 'Point' and Direction to 'Both' (this means the values will be averaged in both local x and y directions). As basic rule for 'Width' and 'Length', the dimension of the column + 1 to 2 times the thickness of the plate can be chosen.

Example – Add an averaging strip to nodes N11 and N12, and set 'Width' and 'Length' to 0,8 m. Ask now again for the value for Asw – select the option 'Averaging of peak' in the Properties menu. The peak will be reduced, but there's a great chance that shear force reinforcement will still be necessary.

*To be sure if shear force reinforcement is required in reality, a punching check has to be carried out. This is a check of the shear force, not in the connection point, but at the face of the column and at some further perimeters.

So we can conclude that a high value for Asw is only an indication that shear force reinforcement might be required.

Step 3: Choice of As used for check

Go to Concrete menu > 2D member > Setup > General: Calculation.

Here you can make the choice for which longitudinal reinforcement will be taken into account for the punching check: the As,req (theoretical), or the As,user which has been added by the user.

Step 4: Punching data

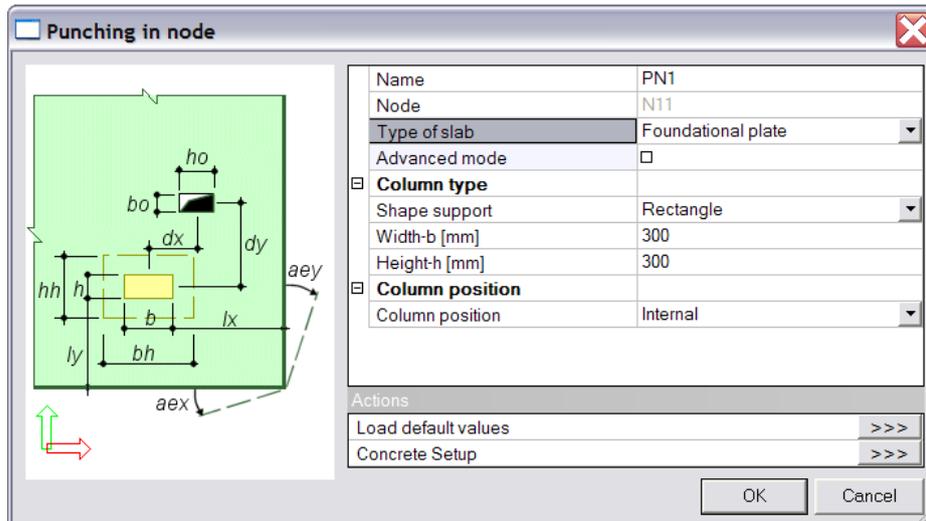
*Go to Concrete menu > Punching > Punching data.

It is not required to add Punching data to each connecting node; SCIA Engineer automatically detects the right configuration in case of a column. In case of a nodal support, and no Punching data are added, default dimensions and settings are taken into account for the check. The default values can be found in the Concrete menu > Setup > Design defaults and ULS: Punching.

By adding Punching data to a node, you can always check which configuration is proposed by SCIA Engineer:

- The specific 'Type of slab' is chosen by SCIA Engineer: Ceiling or Foundational plate.
- The dimensions are filled in: for a column, the 'Width' and 'Height' of the cross-section are correctly taken into account, in case the 'Shape of support' is a rectangle or a circle.
- The column position is filled in correctly, in case the plate consists of straight edges and 90° angles.
- Some more advanced options can be added by the user: Holes and Column heads (see paragraph 5).

Example – Add Punching data to node N13.



SCIA Engineer recognizes the following things:

- Type of slab: Foundational plate (= Plate on subsoil)
- Column type: R 300x300 mm²
- Column position: Internal column

Step 5: Punching check

*When a punching check according to $A_{s,req}$ (theoretical) will be carried out, be sure to at first go to Concrete menu > 2D member > Member design – Design ULS, and refresh the results. The values for the theoretical reinforcement are loaded to the memory in this way, so a punching check can be performed afterwards based on these results.

Be careful with the option 'Averaging of peak' when you have added averaging strips to the project: when this option is selected in the Properties menu, the averaged values are taken into account for the punching check!

*Go to Concrete menu > Punching > Punching check.

4 different results can be obtained when performing a punching check. They are discussed in the next paragraph.

Step 6: Detailed results

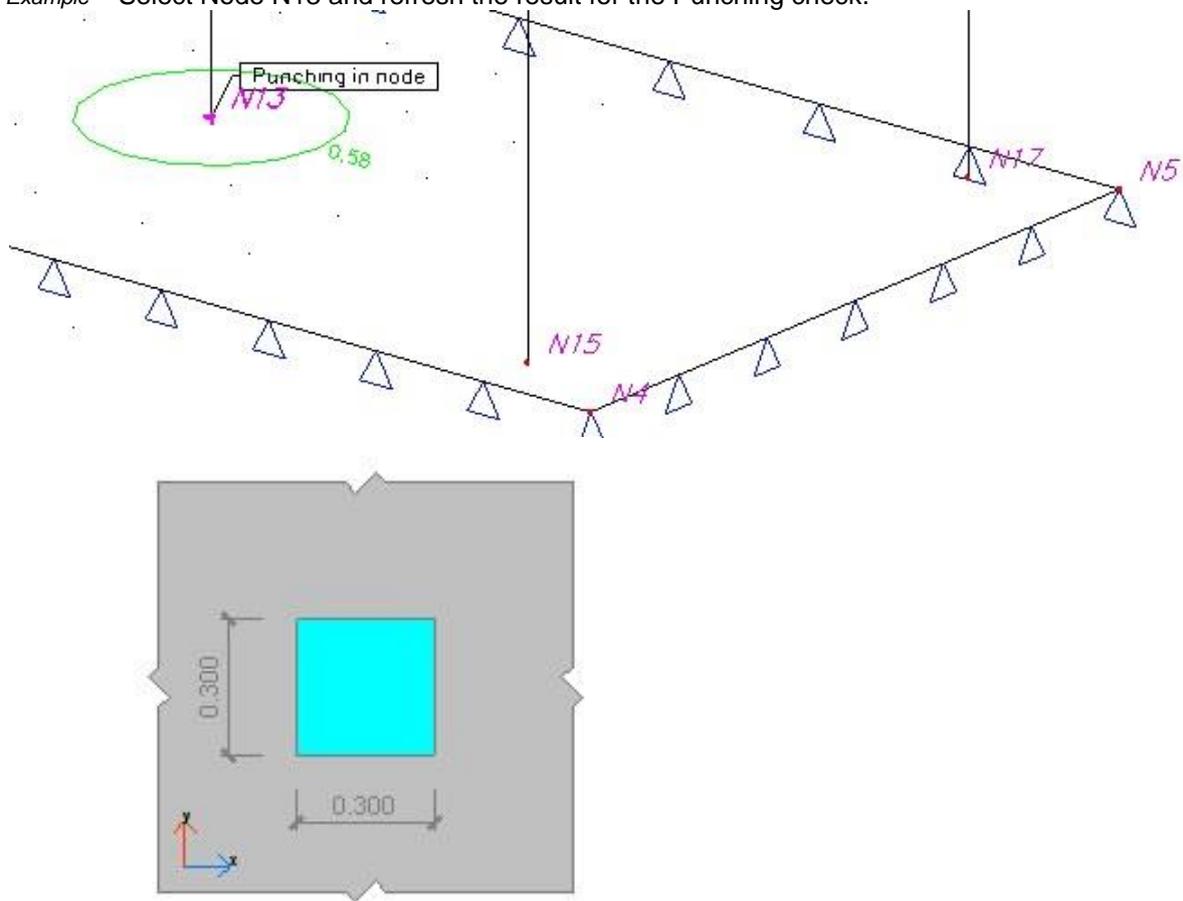
*Go to Concrete menu > Punching > Punching check. At the bottom of the Properties menu, the action Single Check can be activated.

A connecting node has to be selected, after which a new window with detailed information is opened. On the first tab 'Geometry of punching' the configuration taken into account for the punching check can be viewed. On the tab 'Design of reinforcement' the calculation parameters and detailed results for the punching check can be found.

Results for punching check

1) No punching reinforcement needed – Check value < 1.00

Example – Select Node N13 and refresh the result for the Punching check.



Activate the action Preview at the bottom of the Properties menu:

Punching check

Linear calculation, Extreme : Global

Selection : N13

Combinations : ULS

Check maximum shear resistance

Node	Case	Per.	lx_col [m]	ly_col [m]	u ₀ [m]	v [-]	V _{Ed0} [MPa]	V _{Rd,max} [MPa]
N13	ULS	1	0,080	0,080	1,200	0,55	2,09	3,68

Reinforcement in plate

Node	Type of reinforcement	A _{s1} [mm ²]	A _{s2} [mm ²]	α _{s1} [deg]	α _{s2} [deg]
N13	Required	1364	1388	0,00	90,00

Load in critical section

Node	Case	f _d [kN/m ²]	R _{Ed} [kN]	M _{Edx} [kNm]	M _{Edy} [kNm]
N13	ULS	52,11	-348,43	80,53	81,80

Check punching shear resistance and design shear reinforcement

Node	Case	Per.	d [mm]	u [m]	V _{Ed} [MPa]	V _{Rd,c} [MPa]	A _{sv} /u [mm ² /m]	V _{Rd,c} [MPa]	Check	Check value [-]	W/E
N13	ULS	1	160	1,702	1,43	2,48	0	2,48	OK	0,58	6

Conclusion:

$$V_{Ed} < V_{Rd,max}$$

$$V_{Ed} < V_{Rd,c}$$

Out of this result follows that no punching reinforcement is required $\rightarrow A_{sw}/u = 0$.
Only the first control perimeter (= critical section) has to be considered.

Activate the action Single Check at the bottom of the Properties menu. Here the calculation parameters and results can be regarded in detail:

The screenshot shows the 'Slab punching N13' dialog box with the following data:

Loads (Case/Combi)			
Case	ULS/3	fd	52.1074 kN/m ²
Red	-348.4331 kN	Medx	80.5304 kNm
		Medy	81.8014 kNm

First critical section			
Distances from column face :			
lx_column	0.08 m	ly_column	0.08 m
Beta	1.15	rho_l	0.0086
		sigma_cp	0 MPa
d	160 mm	A_crit	0.2061 m ²
		A_load	0.09 m ²

Check punching resistance and design shear reinforcement			
u	1.7025 m	u_out,eff	1.0104 m
		Angle of Asw	90 deg
sr	0.75 d	sr	120 mm
		Asw/u	0 mm ² /m
vEd	1.4257 MPa	vRd,c	2.4785 MPa
		vRd,cs	2.4785 MPa

Check maximum shear resistance			
u_0	1.2 m	vEd_0	2.087 MPa
		vRd,max	3.68 MPa

Results :

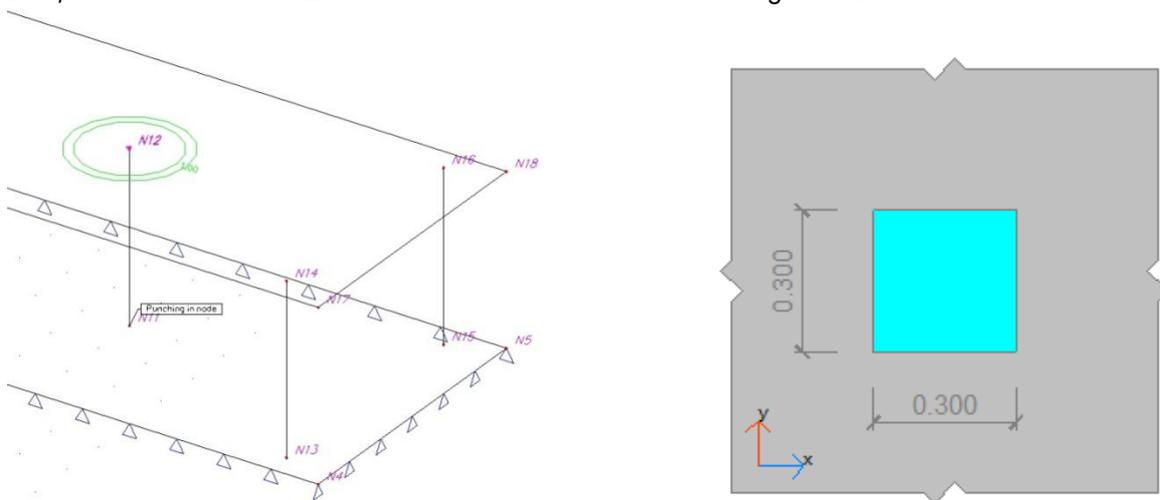
Actual critical section : Shear force carried by concrete. No shear reinforcement required.

Total : Calculation is satisfactory.

Buttons: Calculation, Document, Reinforcement, OK, Cancel

2) Punching reinforcement needed – Check value = 1.00

Example – Select Node N12 and refresh the result for the Punching check.



Activate the action Preview at the bottom of the Properties menu:

Punching check

Linear calculation, Extreme : Global
 Selection : N12
 Combinations : ULS
 Check maximum shear resistance

Node	Case	Per.	lx_col [m]	ly_col [m]	u ₀ [m]	v [-]	V _{Ed0} [MPa]	V _{Rd,max} [MPa]
N12	ULS	1	0,320	0,320	1,200	0,55	2,53	3,68

Reinforcement in plate

Node	Type of reinforcement	A _{s1+} [mm ²]	A _{s2+} [mm ²]	A _{s1-} [mm ²]	α _{s1+} [deg]	α _{s2+} [deg]	α _{s1-} [deg]
N12	Required	3199	3209	200	0,00	90,00	0,00

Load in critical section

Node	Case	f _d [kN/m ²]	R _{Ed} [kN]	M _{Edx} [kNm]	M _{Edy} [kNm]
N12	ULS	-12,15	421,82	-138,43	-99,39

Check punching shear resistance and design shear reinforcement

Node	Case	Per.	d [mm]	u [m]	V _{Ed} [MPa]	V _{Rd,c} [MPa]	A _{sw} /u [mm ² /m]	V _{Rd,s} [MPa]	Check	Check value [-]	WE
N12	ULS	1	160	3,210	0,92	0,82	85	0,92	OK	1,00	226

Punching reinforcement

Node	Row	Line	Per.	Number	s _i [m]	Height [m]	Length [m]	x _{beg} [m]	y _{beg} [m]	x _{beg} [m]	y _{beg} [m]
N12	1	1	1	2	0,240	0,120	0,396	-0,198	-0,198	-0,198	0,198
N12	1	2	1	2	0,240	0,120	0,396	-0,198	0,198	0,198	0,198
N12	1	3	1	2	0,240	0,120	0,396	0,198	0,198	0,198	-0,198
N12	1	4	1	2	0,240	0,120	0,396	0,198	-0,198	-0,198	-0,198
N12	2	1	1	3	0,240	0,120	0,636	-0,318	-0,318	-0,318	0,318
N12	2	2	1	3	0,240	0,120	0,636	-0,318	0,318	0,318	0,318
N12	2	3	1	3	0,240	0,120	0,636	0,318	0,318	0,318	-0,318
N12	2	4	1	3	0,240	0,120	0,636	0,318	-0,318	-0,318	-0,318
N12	3	1	1	4	0,240	0,120	0,876	-0,438	-0,438	-0,438	0,438
N12	3	2	1	4	0,240	0,120	0,876	-0,438	0,438	0,438	0,438
N12	3	3	1	4	0,240	0,120	0,876	0,438	0,438	0,438	-0,438
N12	3	4	1	4	0,240	0,120	0,876	0,438	-0,438	-0,438	-0,438

Conclusion:

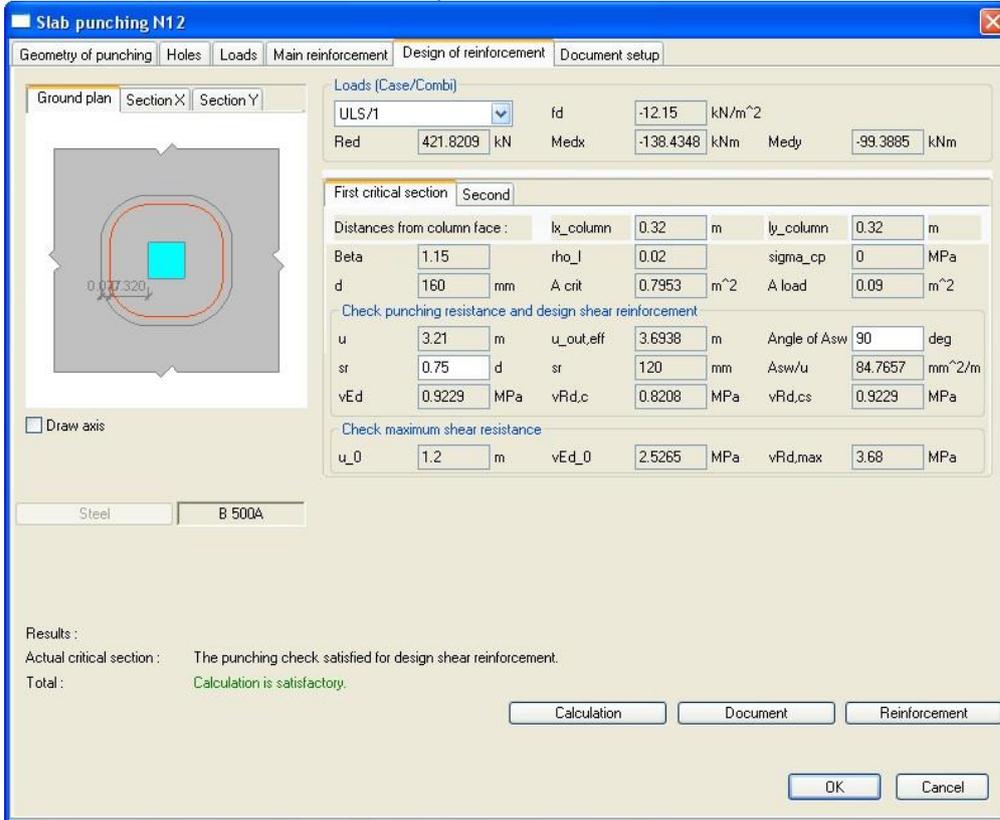
$V_{Ed} < V_{Rd,max}$

$V_{Ed} > V_{Rd,c}$

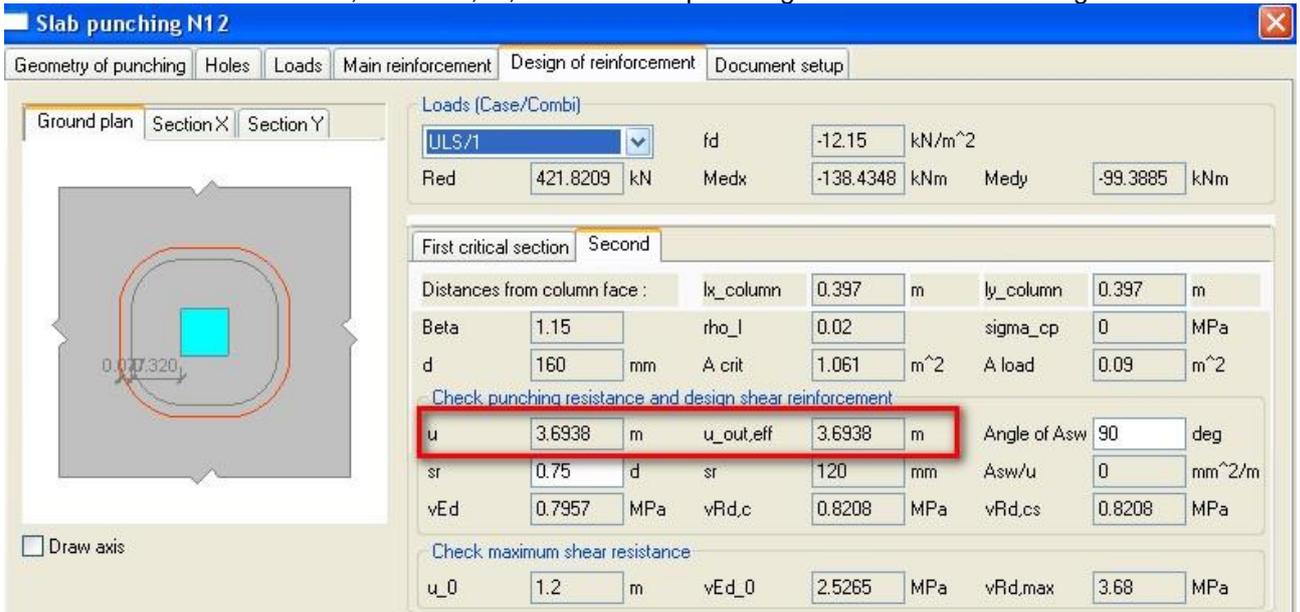
Out of this result follows that punching reinforcement is needed $\rightarrow A_{sw}/u > 0$. To calculate the value of A_{sw}/u , V_{Ed} is taken equal to $v_{Rd,s}$.

More control perimeters have to be considered; the program stops when u_{out} or $u_{out,eff}$ is reached.

Activate the action Single Check at the bottom of the Properties menu. Notice the two tabs, namely for the first critical section and the outer perimeter:

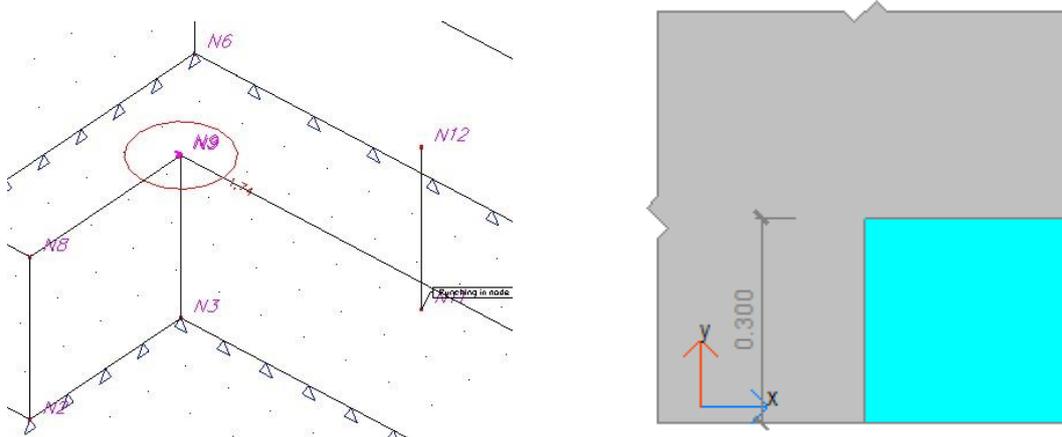


On the second critical section, $u = u_{out,eff}$, which means punching reinforcement is no longer needed:



3) Punching of column through plate – Check value > 1.00

Example – Select Node N9 and refresh the result for the Punching check.



Activate the action Preview at the bottom of the Properties menu:

Punching check

Linear calculation, Extreme : Global
 Selection : N9
 Combinations : ULS
 Check maximum shear resistance

Node	Case	Per.	lx_col [m]	ly_col [m]	u ₀ [m]	v [-]	V _{Ed0} [MPa]	V _{Rd,max} [MPa]
N9	ULS	1	0,320	0,320	0,480	0,55	6,40	3,68

Reinforcement in plate

Node	Type of reinforcement	A _{s1+} [mm ²]	A _{s2+} [mm ²]	α _{s1+} [deg]	α _{s2+} [deg]
N9	Required	4125	3435	0,00	90,00

Load in critical section

Node	Case	f _d [kN/m ²]	R _{Ed} [kN]	M _{Edx} [kNm]	M _{Edy} [kNm]
N9	ULS	-12,15	327,61	-169,09	-120,02

Check punching shear resistance and design shear reinforcement

Node	Case	Per.	d [mm]	u [m]	V _{Ed} [MPa]	V _{Rd,c} [MPa]	A _{sv} /u [mm ² /m]	V _{Rdcs} [MPa]	Check	Check value [-]	WE
N9	ULS	1	160	1,102	2,75	0,82	0	0,00	NOT OK	1,74	785

Conclusion:

V_{Ed0} > V_{Rd,max}

The maximum punching shear stress is exceeded at the face of the column.

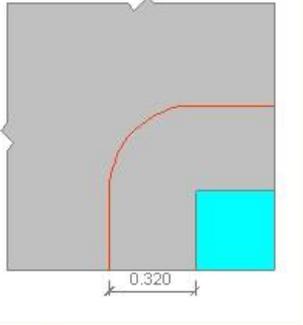
Out of this result follows that the column will punch through the plate, so the punching check doesn't satisfy. In this case, the structure has to be remodelled: use a thicker plate, higher concrete grade, more longitudinal reinforcement in the plate, ...

Activate the action Single Check at the bottom of the Properties menu. Here the calculation parameters and results can be regarded in detail:

Slab punching N9

Geometry of punching | Holes | Loads | Main reinforcement | Design of reinforcement | Document setup

Ground plan | Section X | Section Y



Draw axis

Steel: B 500A

Results:

Actual critical section: The maximum punching shear stress was exceeded

Total: Calculation is not satisfactory.

Calculation | Document | Reinforcement

OK | Cancel

Loads (Case/Combi)

ULS/1 | fd: -12.15 kN/m²

Red: 327.6074 kN | Medx: -169.0947 kNm | Medy: -120.0158 kNm

First critical section

Distances from column face: | lx_column: 0.32 m | ly_column: 0.32 m

Beta: 1.5 | rho_l: 0.02 | sigma_cp: 0 MPa

d: 160 mm | A_crit: 0.3623 m² | A_load: 0.09 m²

Check punching resistance and design shear reinforcement

u: 1.1025 m | u_out,eff: 3.7419 m | Angle of Asw: 90 deg

sr: 0.75 d | sr: 120 mm | Asw/u: 0 mm²/m

vEd: 2.7484 MPa | vRd,c: 0.8208 MPa | vRd,cs: 0 MPa

Check maximum shear resistance

u_0: 0.48 m | vEd_0: 6.3986 MPa | vRd,max: 3.68 MPa

Note that u_0 is only 0,48m.

This is because it is a corner column.

EN2 (art 6.4.5 (3)) states that:

(3) Adjacent to the column the punching shear resistance is limited to a maximum of:

$$v_{Ed} = \frac{\beta V_{Ed}}{u_0 d} \leq v_{Rd,max} \quad (6.53)$$

where

u_0 for an interior column $u_0 = \text{length of column periphery [mm]}$

for an edge column $u_0 = c_2 + 3d \leq c_2 + 2c_1$ [mm]

for a corner column $u_0 = 3d \leq c_1 + c_2$ [mm]

c_1, c_2 are the column dimensions as shown in Figure 6.20

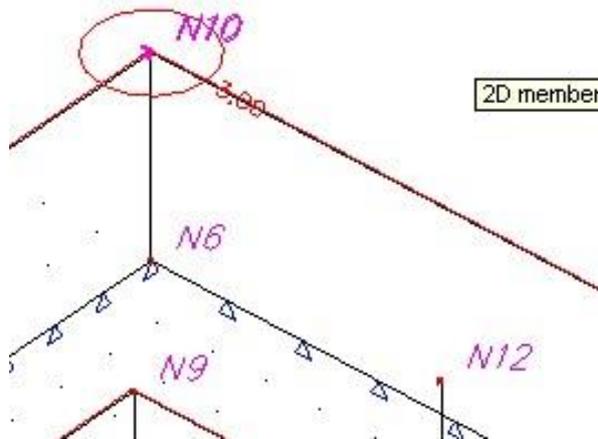
v see Expression (6.6)

β see 6.4.3 (3), (4) and (5)

Note: The value of $v_{Rd,max}$ for us in a Country may be found in its National Annex. The recommended value is $0,5v_{fd}$.

4) Value = 3.00 – No punching check possible

Example – Select Node N10 and refresh the result for the Punching check.



Activate the action Preview at the bottom of the Properties menu:

Punching check

Linear calculation, Extreme : Global

Selection : N10

Combinations : ULS

Check maximum shear resistance

Node	Case	Per.	lx_col [m]	ly_col [m]	u ₀ [m]	v [-]	V _{Ed0} [MPa]	V _{Rd,max} [MPa]
N10	ULS	1	0,000	0,000	0,000	0,15	0,00	0,00

Reinforcement in plate

Node	Type of reinforcement
N10	No reinf

Load in critical section

Node	Case	f _d [kN/m ²]	R _{Ed} [kN]	M _{Edx} [kNm]	M _{Edy} [kNm]
N10	ULS	0,00	0,00	0,00	0,00

Check punching shear resistance and design shear reinforcement

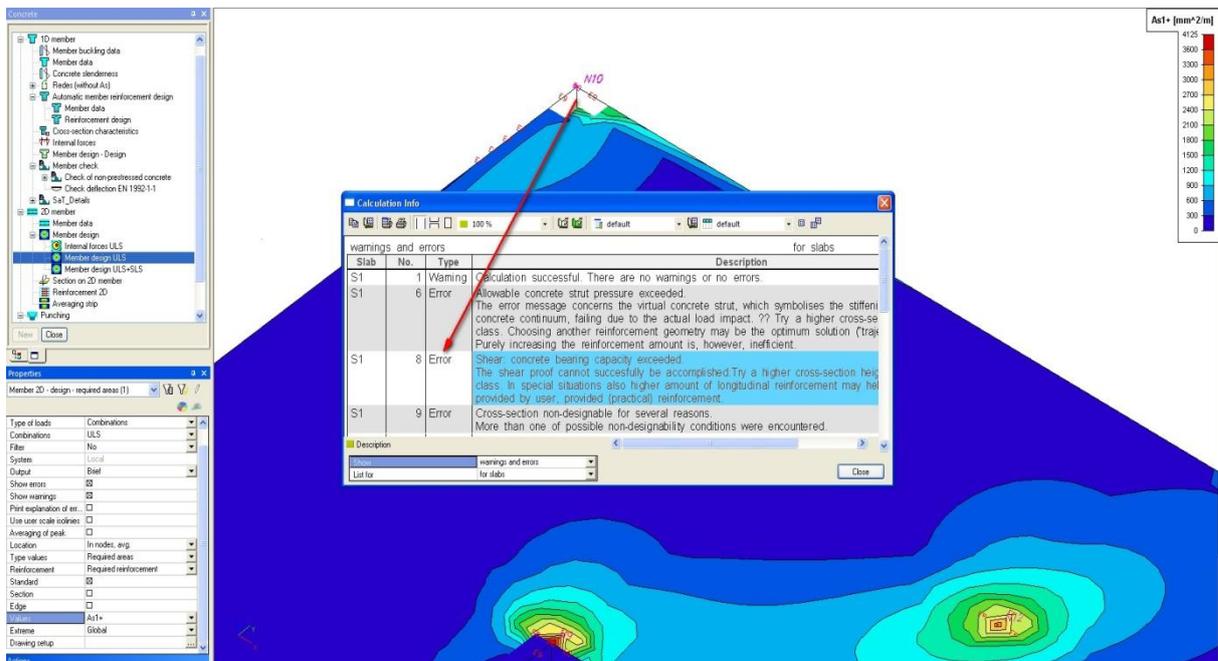
Node	Case	Per.	d [mm]	u [m]	V _{Ed} [MPa]	V _{Rd,c} [MPa]	A _{sv} /u [mm ² /m]	V _{Rdcs} [MPa]	Check	Check value [-]	W/E
N10	ULS	1	0	0,000	0,00	0,00	0	0,00	NOT OK	3,00	741

All of the values needed for the punching check are filled in as “0”, which means one or more of the calculation parameters are unknown.

The explanation of error 741 leads us in the good direction: Main reinforcement was not found in the plate.

Since we are dealing with a column at the bottom of a plate, the longitudinal reinforcement which is taken into account for the punching check is the upper reinforcement.

If we go to the Concrete menu > 2D member > Member design – Design ULS, and take a look at the results for As_{up1} and As_{up2}, we notice that Error 8 appears in the finite element next to the connecting node.



Conclusion:

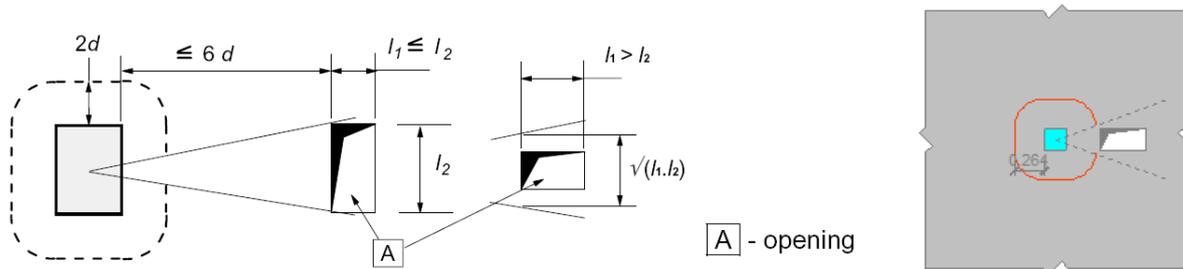
Because no upper reinforcement A_s can be calculated around the connecting node, the punching check cannot be effectuated, and the 'error' check value of 3.00 appears.

In this case, the structure has to be remodelled: use a thicker plate, higher concrete grade, other column dimensions, other column position, ...

4_Extra features

Openings near the loaded area

Openings near the loaded area have to be dealt with as follows, according to *EC § 6.4.2 (3)*:
For loaded areas situated near openings, if the shortest distance between the perimeter of the loaded area and the edge of the opening does not exceed $6d$ (see figure), the part of the control perimeter contained between two tangents drawn to the outline of the opening from the centre of the loaded area is considered to be ineffective.



This requirement is also implemented in SCIA Engineer.

It is important to know that openings inputted in a slab (via the Structure menu) are not taken into account by default for the punching check: there is no active link between the slab data and the punching check.

An opening has to be added manually in the single check menu. Only then this opening will be taken into account for the punching calculation.

Column Heads

Column heads have to be dealt with as follows, according to EC § 6.4.2 (8), (9), (11):

$l_H < 2h_H$

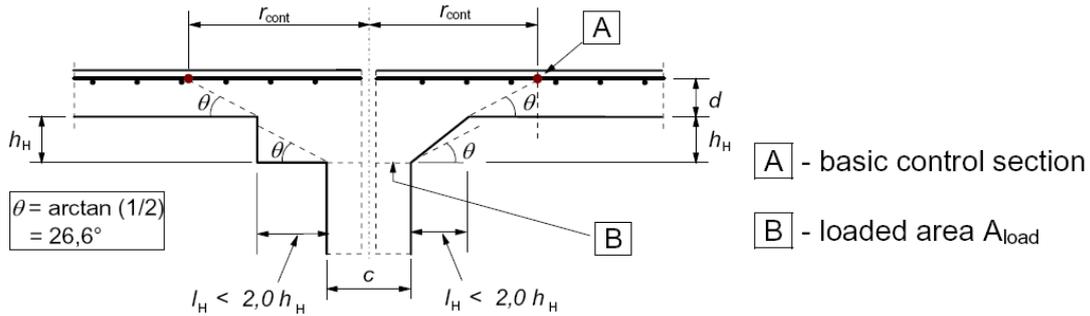
For slabs with circular column heads for which $l_H < 2h_H$ a check of the punching shear stresses according to EC § 6.4.3 is only required on the control section outside the column head. The distance of this section from the centroid of the column r_{cont} may be taken as:

$$r_{cont} = 2d + l_H + 0,5c$$

where:

l_H is the distance from the column face to the edge of the column head

c is the diameter of a circular column



For a rectangular column with a rectangular head with $l_H < 2,0h_H$ and overall dimensions l_1 and l_2 ($l_1 = c_1 + 2H_1$, $l_2 = c_2 + 2H_2$, $l_1 \leq l_2$), the value r_{cont} may be taken as the lesser of:

$$r_{cont} = 2d + 0,56 \sqrt{l_1 l_2} \quad \text{and} \quad r_{cont} = 2d + 0,69 l_1$$

$l_H > 2h_H$

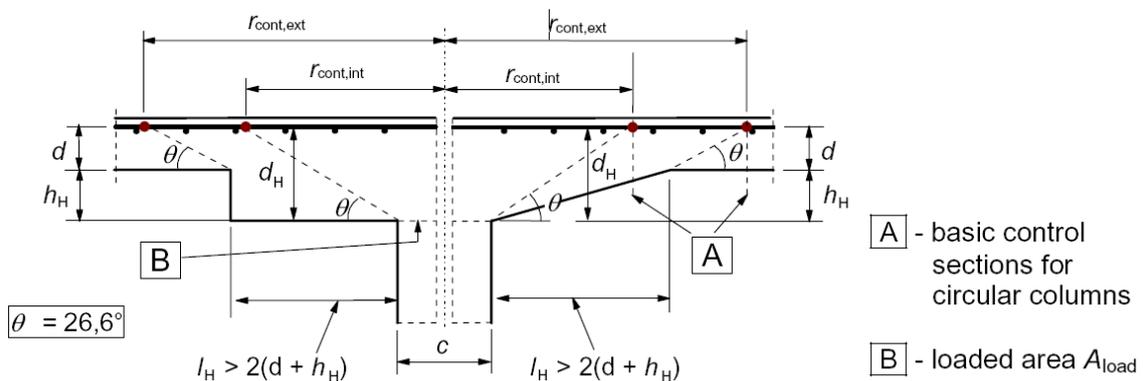
For slabs with enlarged column heads where $l_H > 2h_H$ control sections both within the head and in the slab should be checked.

$l_H > 2(d + h_H)$

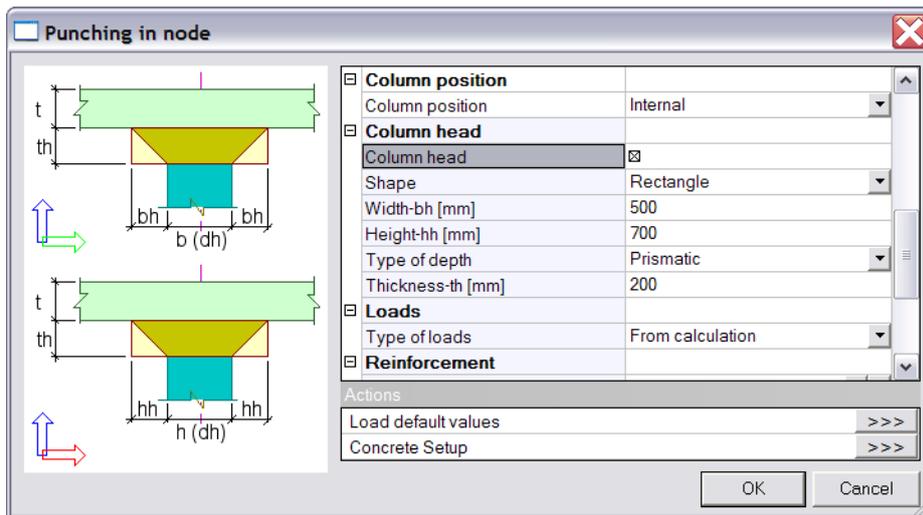
For circular columns the distances from the centroid of the column to the control sections may be taken as:

$$r_{cont,ext} = l_H + 2d + 0,5c$$

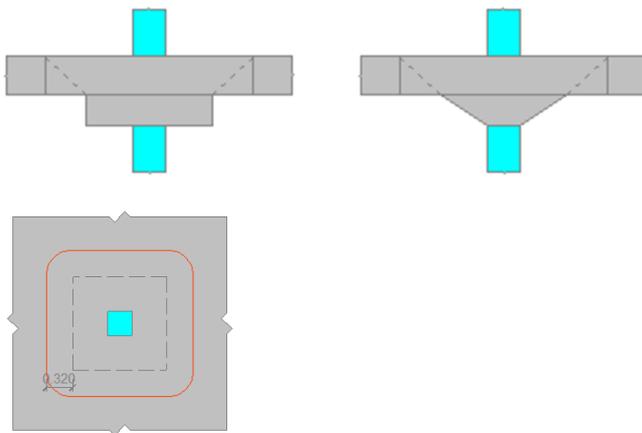
$$r_{cont,int} = 2(d + h_H) + 0,5c$$



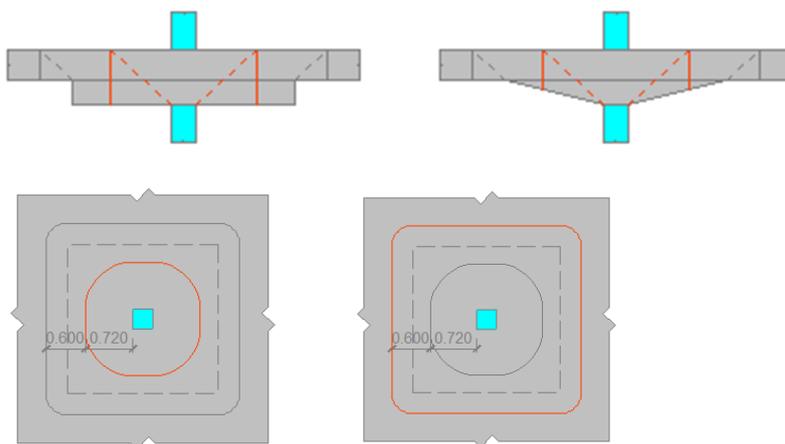
All of the cases above can be inputted and designed in SCIA Engineer, by means of Punching data.



For $l_H < 2h_H$:



For $l_H > 2(d + h_H)$:



Pressure only 2D members

Introduction & Principle of calculation

Pressure only elements were developed for the analysis of masonry walls, as this is a typical case of a pressure only element. However also for concrete wall analysis this can be a useful feature, especially in cases where the reinforcement configuration is not a standard 2 course orthogonal net. Examples of both will be treated in this chapter.

The principle of this calculation is an iterative process. In every finite element where tension is found an orthotropy will be given in such way that the stiffness in the direction of tension is lower, which causes that the force will find its way through the elements in the direction of the pressure lines in which the element is given a higher stiffness. This iterative process continues until equilibrium is found, but never a stiffness less than 5% will be given in a direction. So it is possible in some cases that still some tension will be found in some elements, this is then because else no equilibrium can be made, but the stiffness in this tension direction will be very low.

In SCIA Engineer, 2D members can be defined as 'pressure only' members after the specific functionality has been activated:

The image shows two overlapping dialog boxes in the SCIA Engineer software. The background dialog is 'Project data', which has several tabs: 'Basic data', 'Functionality', 'Loads', 'Combinations', 'Protection', and 'National Annexes'. The 'Functionality' tab is active, showing a list of features with checkboxes. The 'Nonlinearity' feature is checked. The foreground dialog is 'Properties' for a '2D member (1)'. It contains various parameters for the member, such as Name (S2), Type (wall (80)), Analysis model (Standard), Shape (Flat), Material (C30/37), Thickness (200 mm), Member system-plane at (centre), Eccentricity z (0 mm), LCS Type (Standard), Swap orientation (no), LCS Angle (0.00 deg), Layer (Layer1), Nodes (N5, N6, N7, N8), and Data (Line force on 2D member... LFS2, Line support on 2D member... Sls2, Mesh data MS2). The 'FEM model' dropdown is highlighted with a red box and set to 'Press only'.

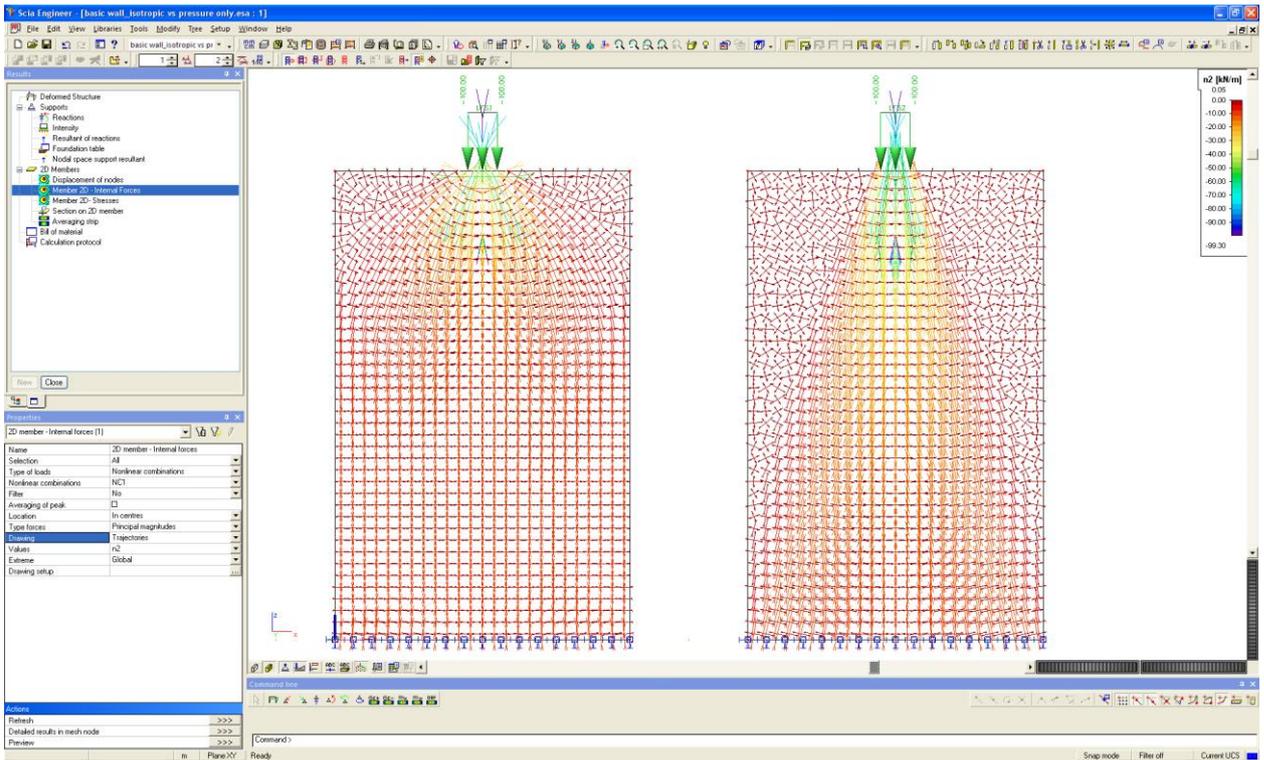
The 'pressure only' non linearity will only be taken into account by a non linear calculation. Therefore non linear combinations have to be created.

Comparison between isotropic and pressure only element

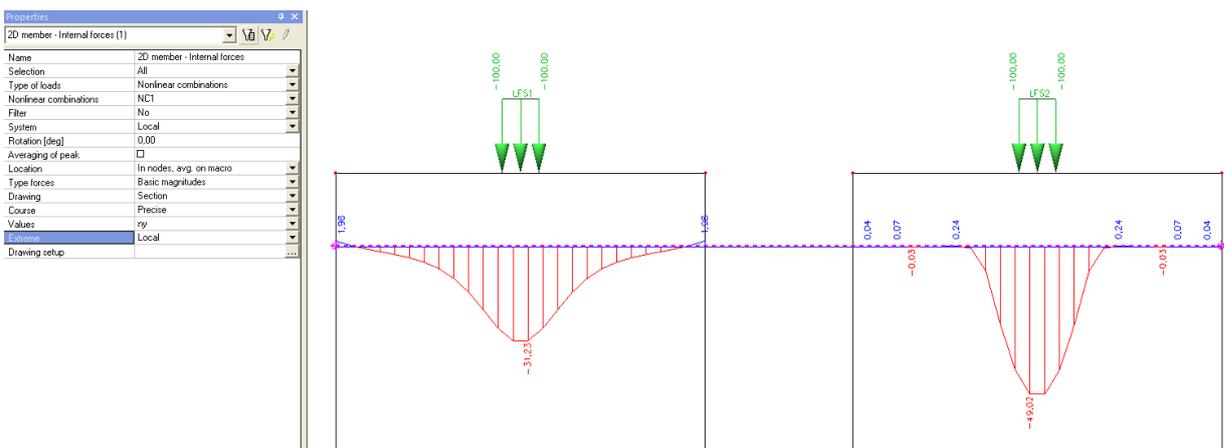
Open the example basic_wall_isotropic vs pressure_only.esa. In this project there is two time the same basic wall with line load on top and line support at the bottom. The left wall is defined isotropic. The right wall is defined pressure only.

The best way to visualize the effect of the pressure only element is by showing the trajectories of the principal pressure force n2. These trajectories show the direction of the pressure lines.

The difference between the isotropic wall and pressure only wall is clear. In the isotropic walls the pressure lines are bended through the effect of tension that can taken by the wall. The pressure only wall shows straight pressure lines, this wall will only work in compression.



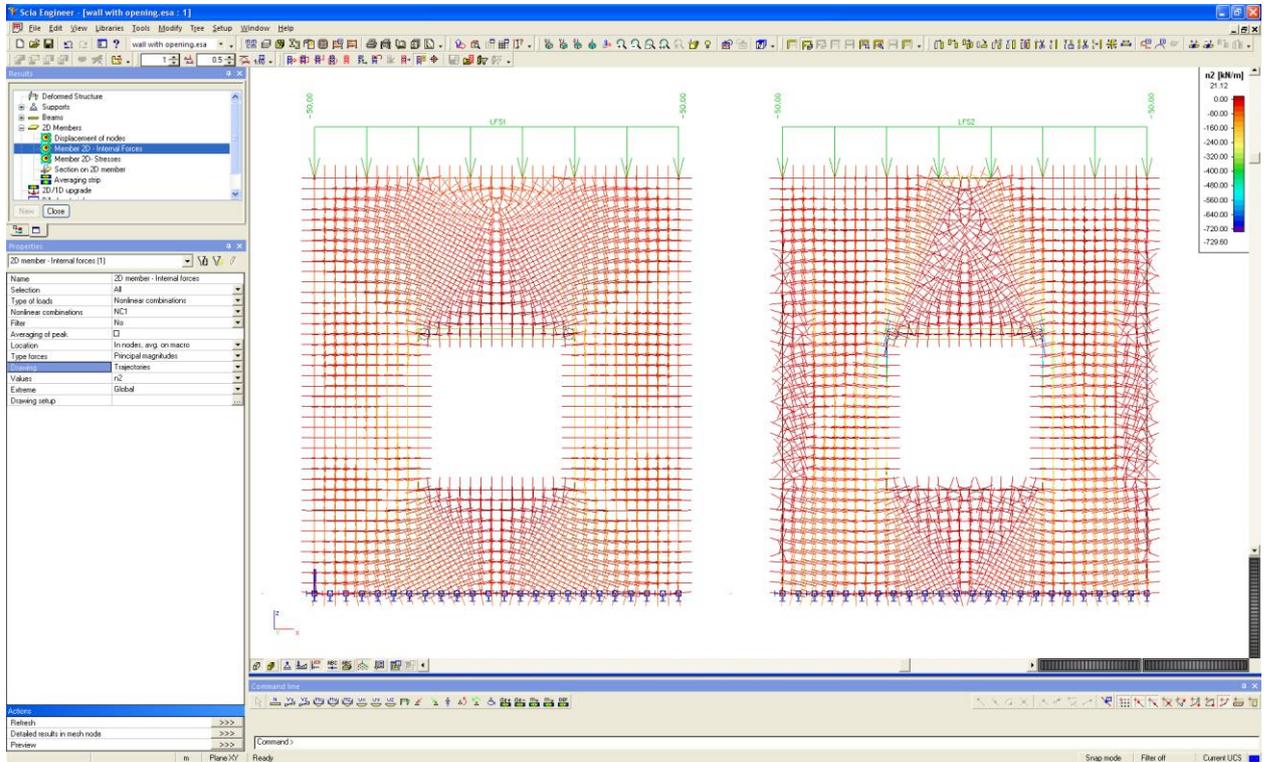
The difference is also visible in the value of the vertical pressure force n_y :



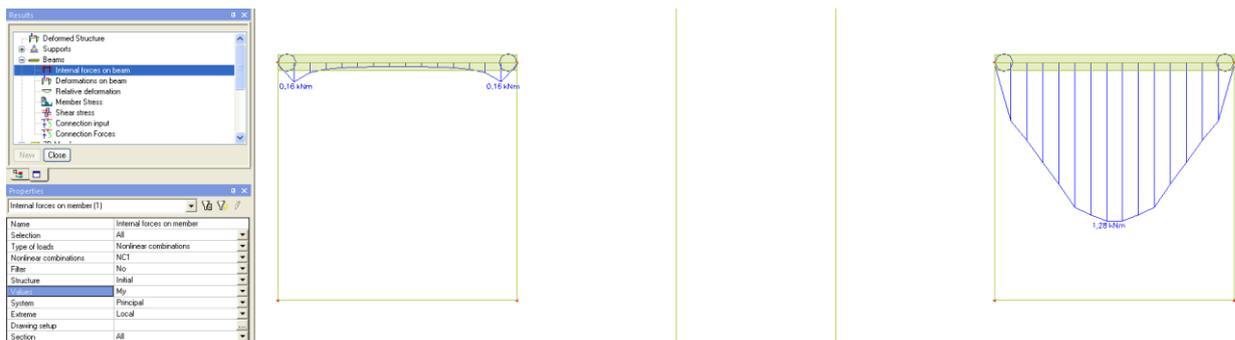
Wall with opening

Open the example wall with opening.esa. In this project there is two time the same wall with opening with line load at top and line support at bottom. The left wall is defined isotropic. The right is defined pressure only.

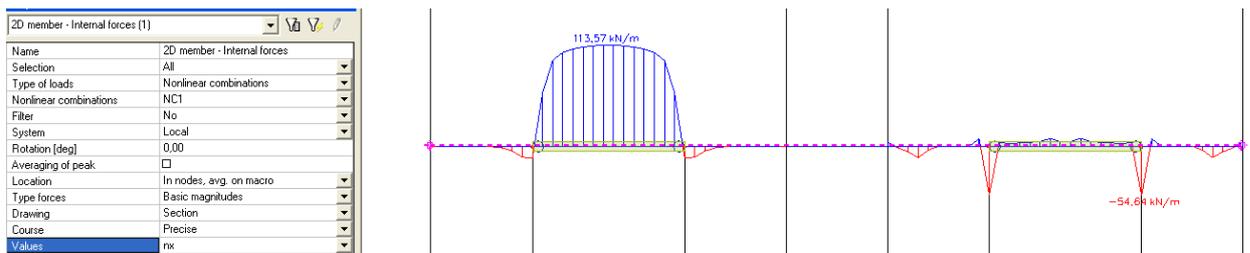
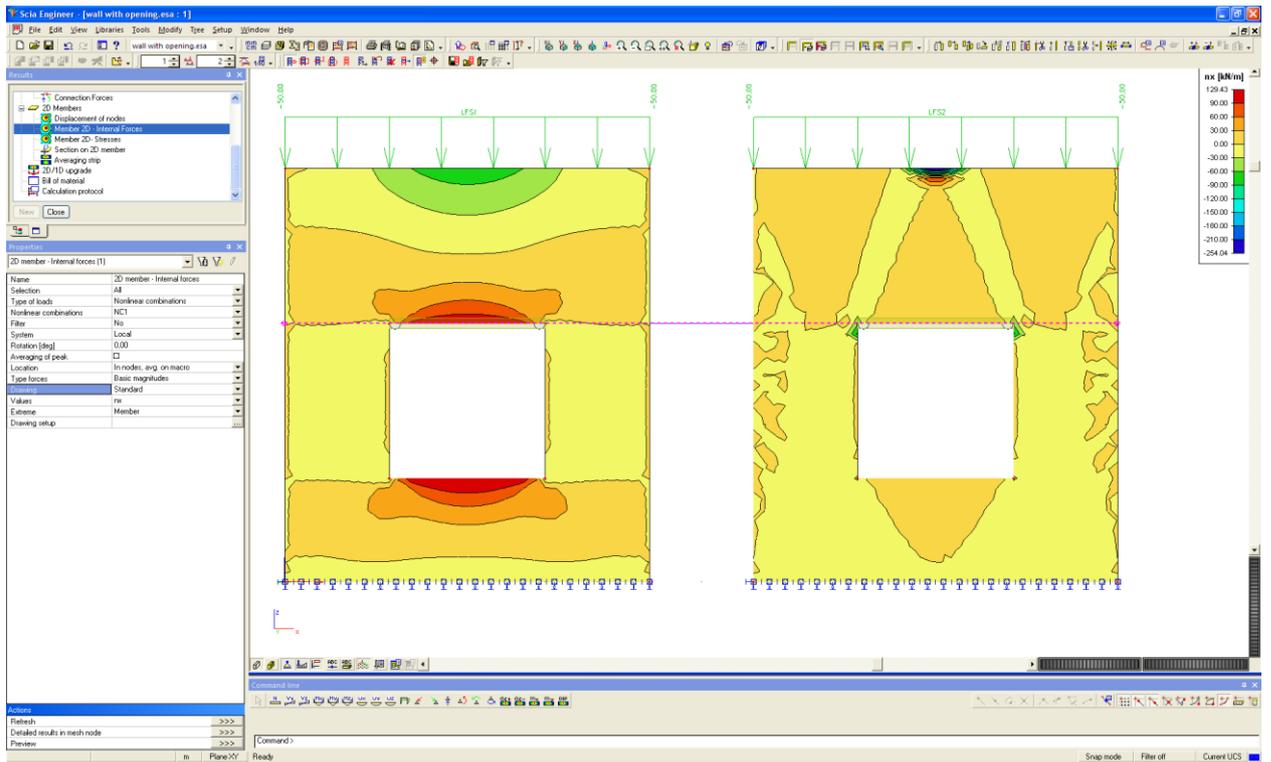
In a first image the trajectories are shown:



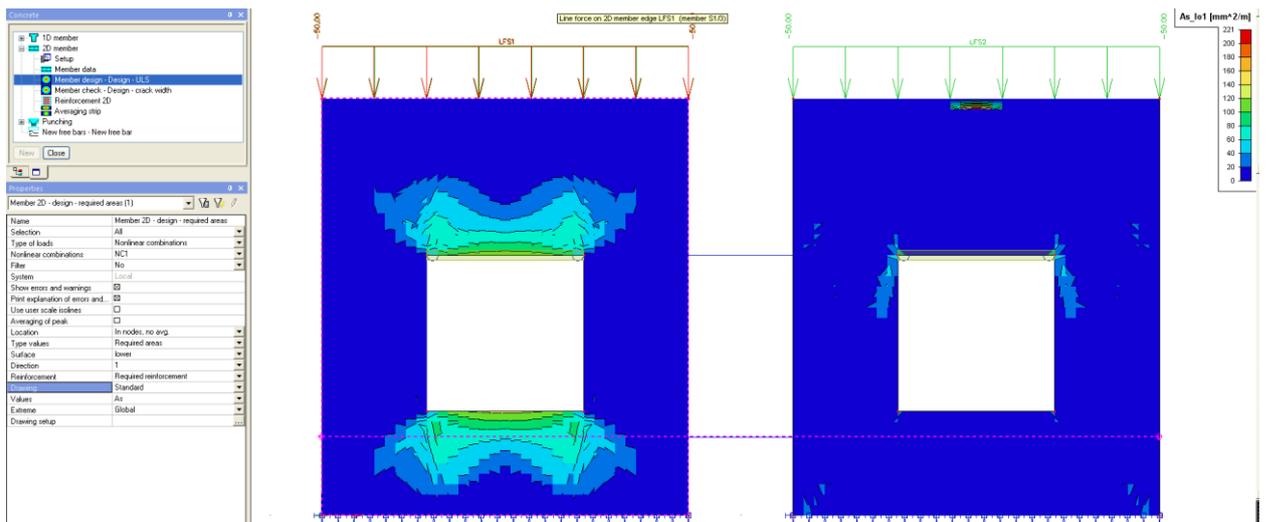
Above the opening there is a beam element. It is typical for a masonry wall that above the openings beam elements are used to take the tension. If we now take a look at the moment M_y in this beam element it becomes clear that the pressure only wall gives a more realistic result.



This can be explained by looking at the horizontal tension forces n_x in the wall. In the picture below can be seen that the isotropic element will take the tension itself, so it is not transferred to the beam element.

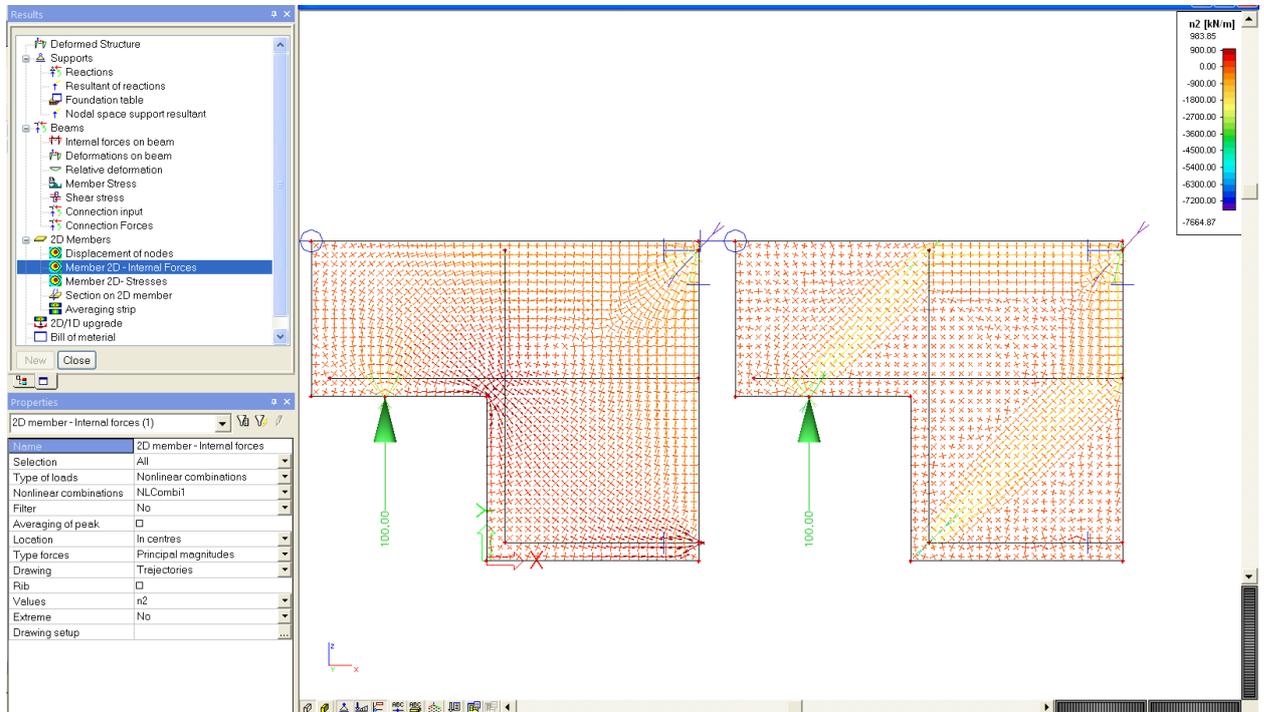


The tension in the horizontal direction also leads to tension reinforcement in this direction. It is visible that in the isotropic element the beam is not loaded but tension reinforcement is necessary.

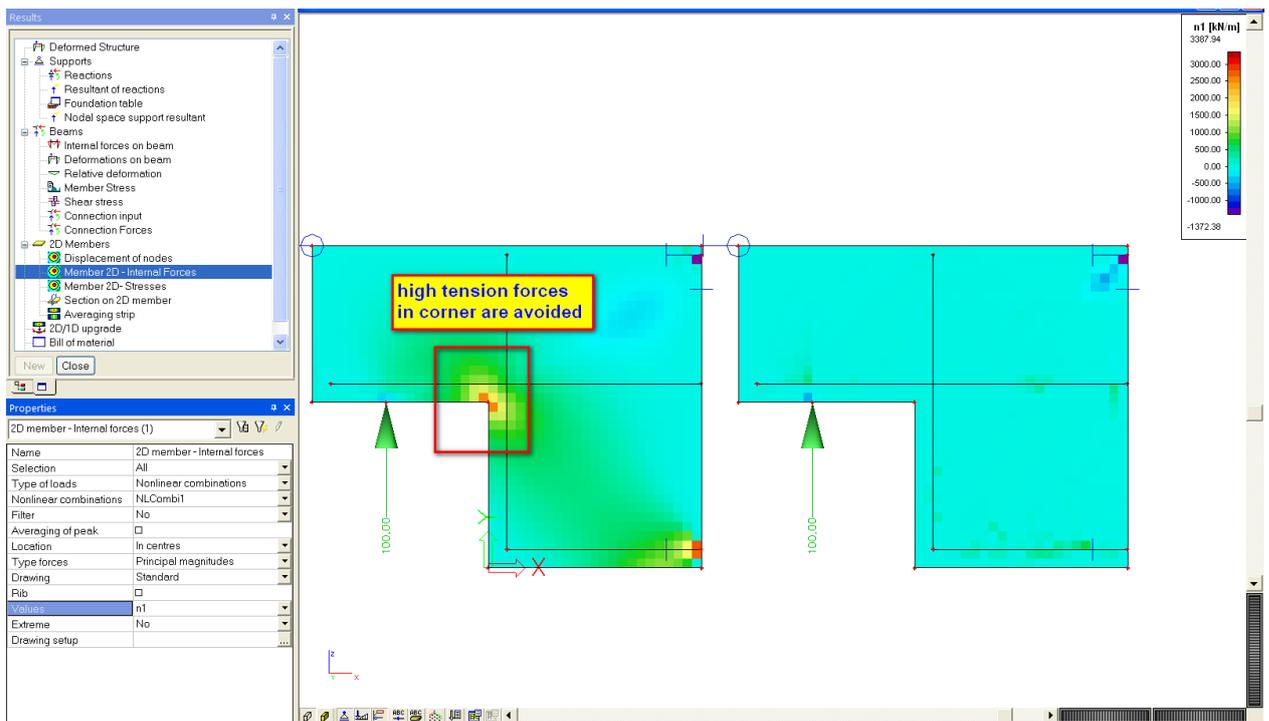


Use of reinforcement ribs

The principle of the use of reinforcement ribs is shown in the example strut model.esa. The reinforcement ribs are introduced in the model as a rib. The cross-section consists of reinforcement material and is a numerical cross-section with the properties of a reinforcement rib. The trajectories clearly show a difference between the isotropic model and the pressure only model.



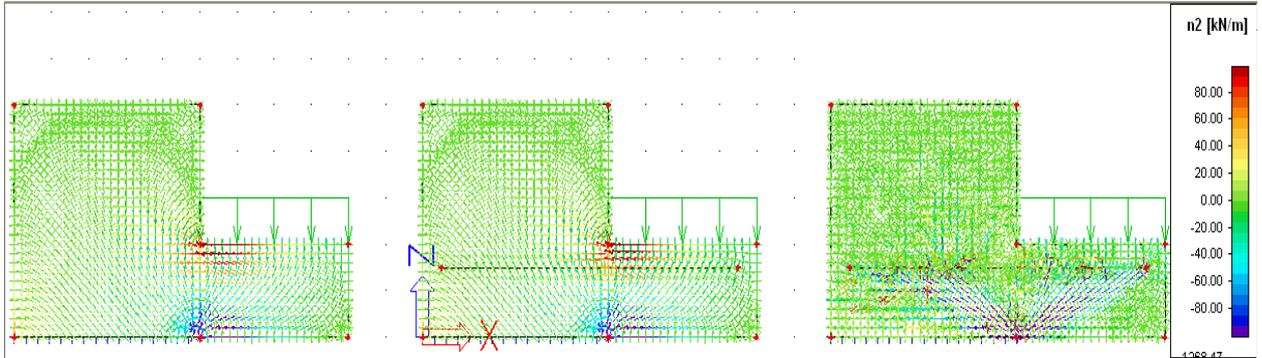
The pressure only walls show straight pressure lines that are only bended on the reinforcement ribs that will take the tension. To dimension these reinforcement ribs the normal forces working in these ribs can be asked. Based on these tension forces the user can dimension the reinforcement ribs. For the wall design the high tension forces that appear in the corner will be avoided. That is why this reinforcement configuration is a better design then a standard 2D reinforcement net.



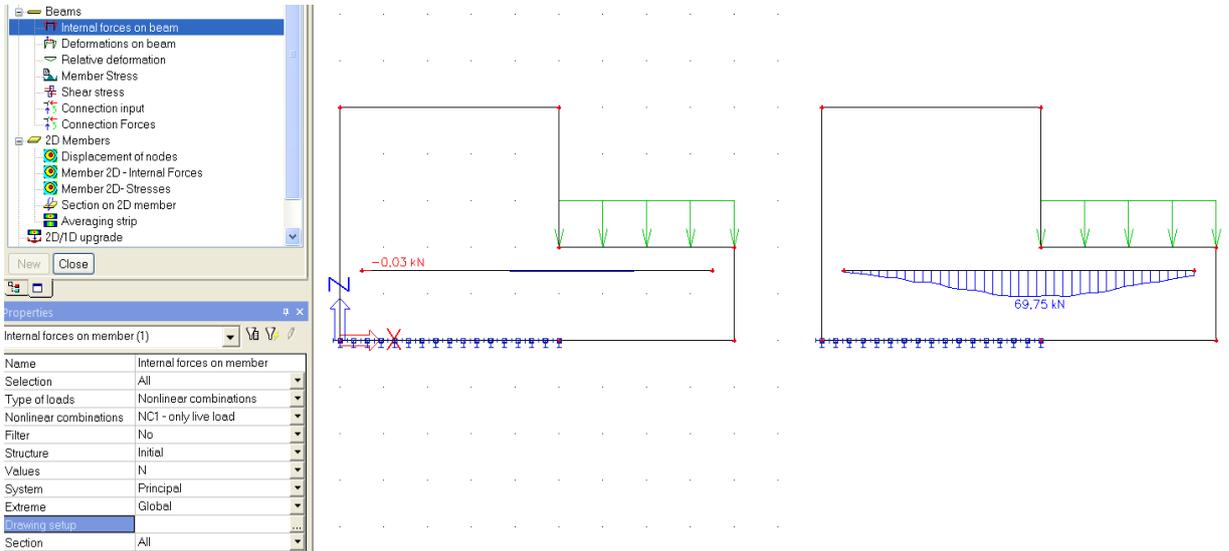
In the next example cantilever.esa the use of reinforcement ribs is illustrated.

In the project there is three times the same construction which consists of a wall with an overhang part on which some loading is set. The left wall is isotropic, the middle wall is isotropic with reinforcement ribs, the right wall is pressure only with reinforcement ribs. The reinforcement ribs are introduced in the model as a rib. The cross-section consists of reinforcement material and is a numerical cross-section with the properties of a reinforcement rib.

First the trajectories are shown.

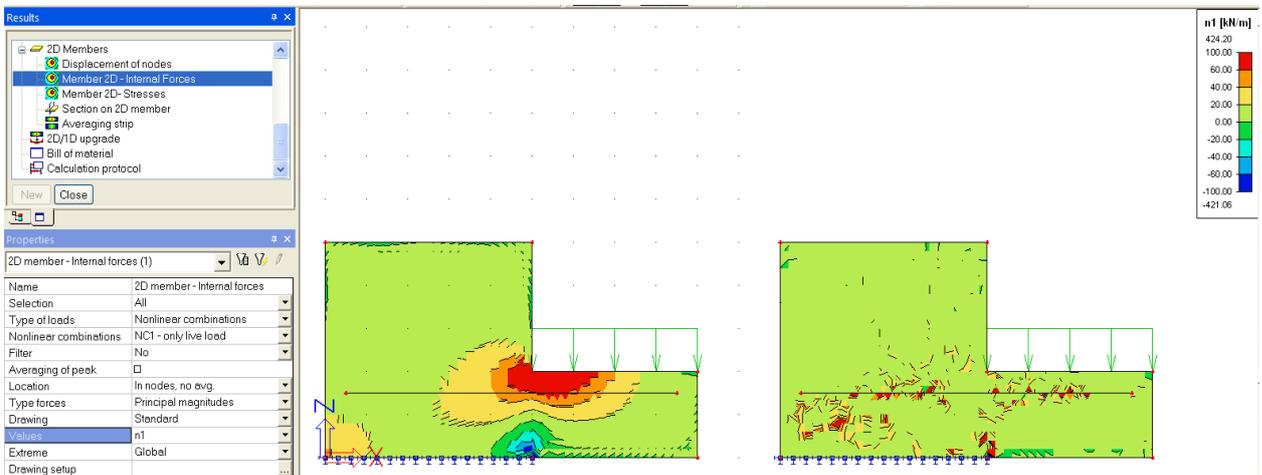


The reinforcement ribs are only effective if the concrete does not take any tension itself. To illustrate the normal force in the ribs is shown below.



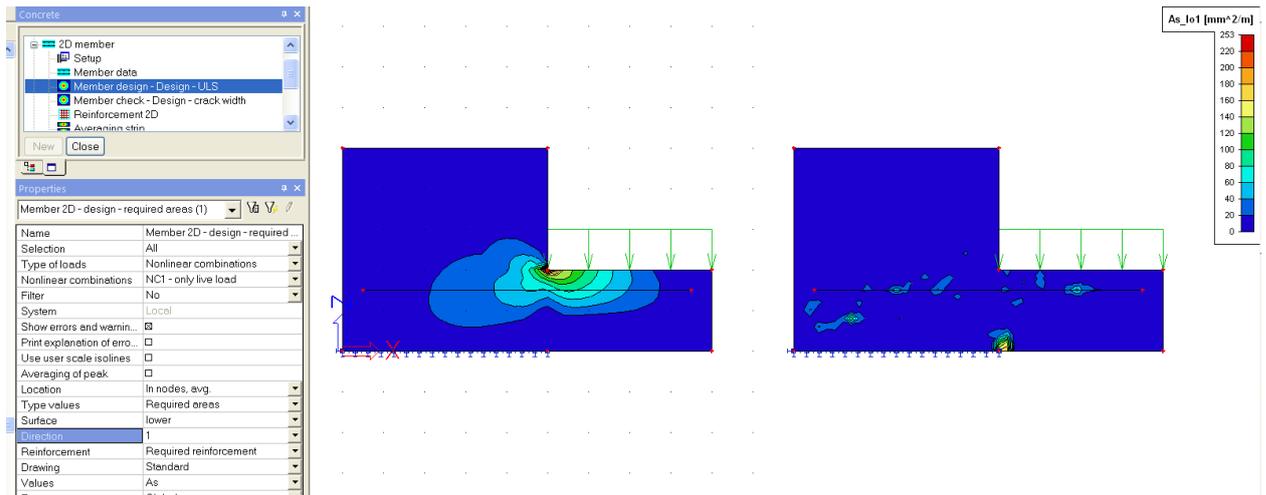
Based on this tension force, the reinforcement rib can be dimensioned.

The high tension forces in the corner of the 2D element will no longer appear because they are taken by this reinforcement rib.



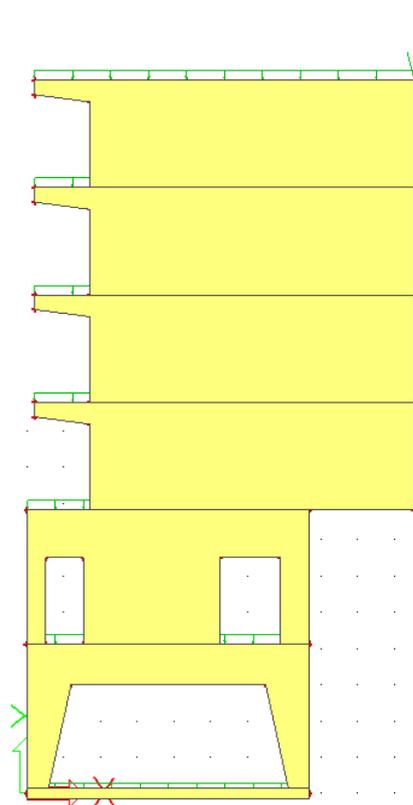
Of course this also affects the reinforcement design.

In the picture below the statically required reinforcement in horizontal direction is shown.



Practical example

A last example **practical example.esa** demonstrates a wall out of the practice:



We will consider two cases:

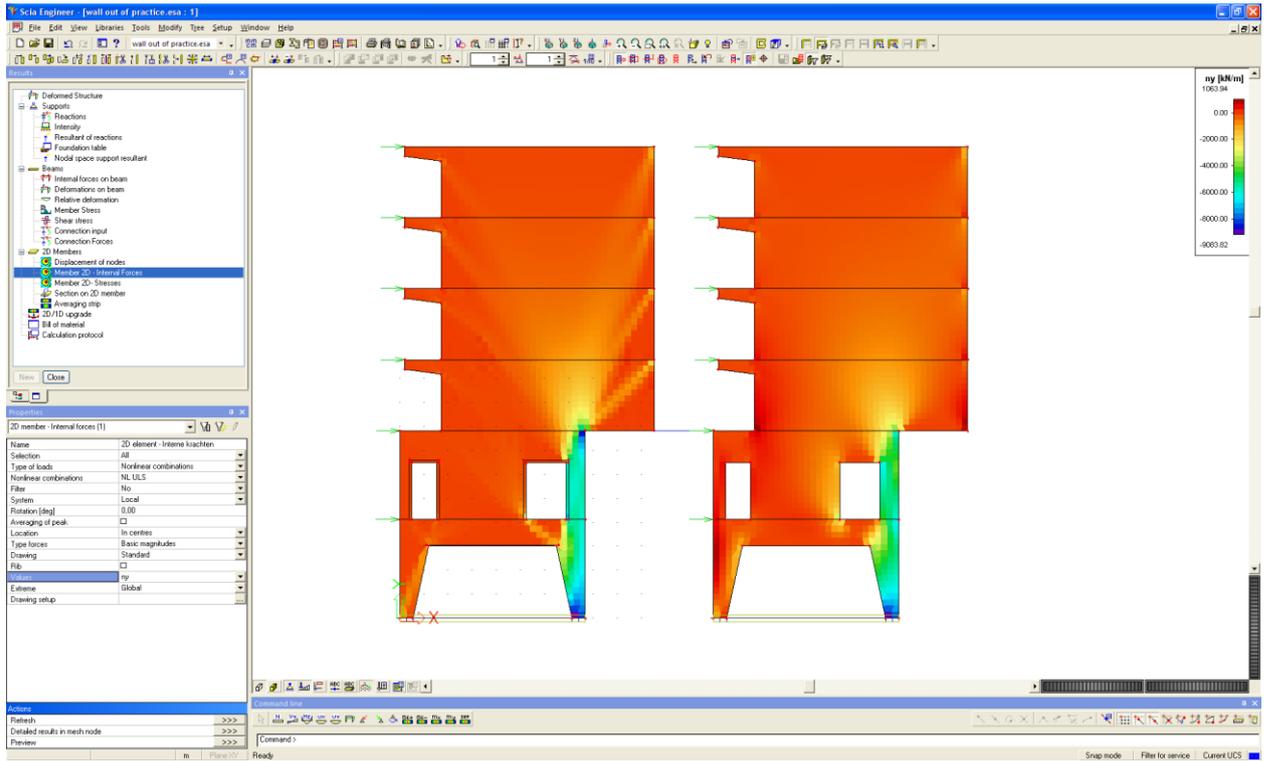
On the right is the linear elastic model.

On the left is the pressure only model.

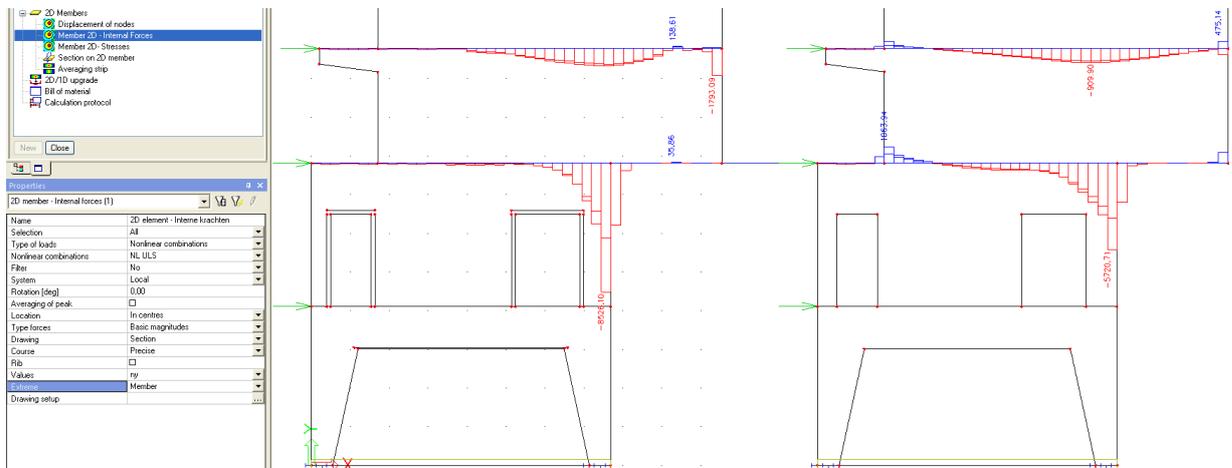
For the pressure only model, two adaptations are made:

- The FEM-type is changed into 'Press only'
- Ribs are introduced at the critical zones where we expect large tensile forces. They will simulate the reinforcement.

The linear elastic results are as follows:

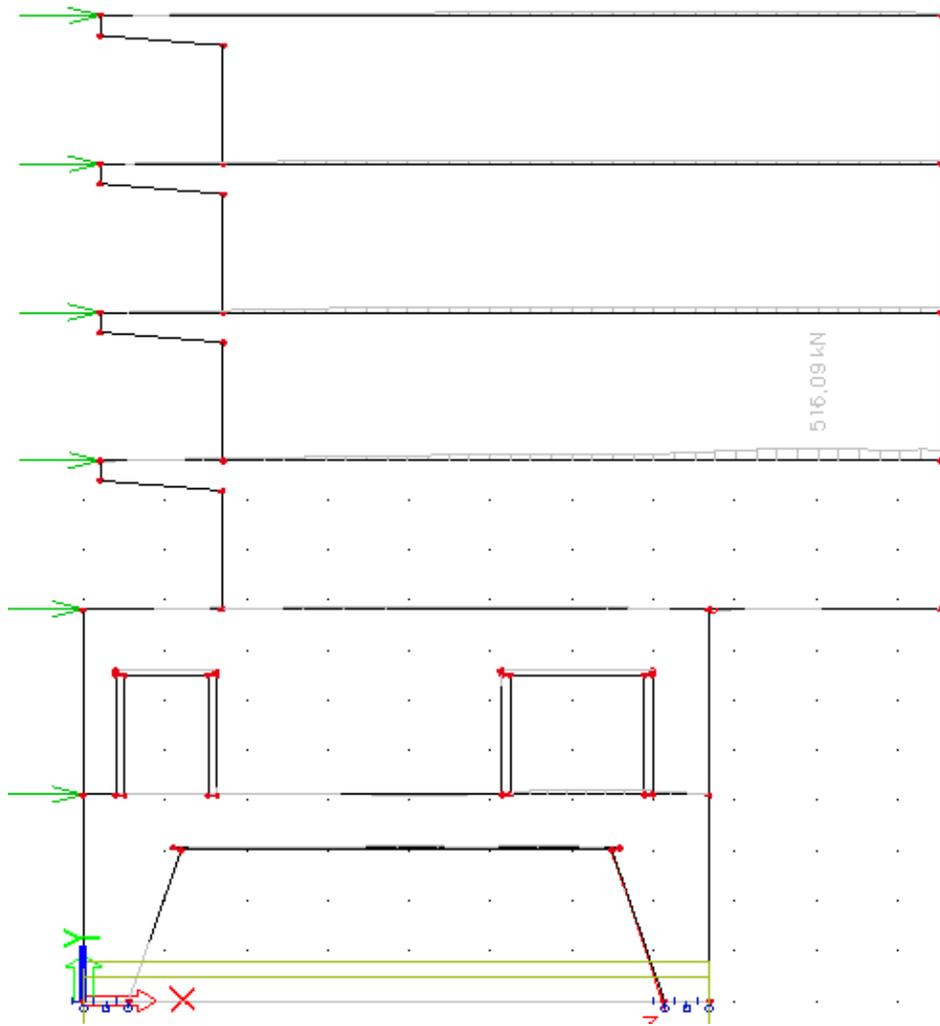


If we take a look at the corners:



We can clearly see the effect of the pressure only wall in which the tensile forces are almost completely eliminated.

The tensile forces in the reinforcement ribs show the tensile stress to be taken by the reinforcement.



Fire resistance check

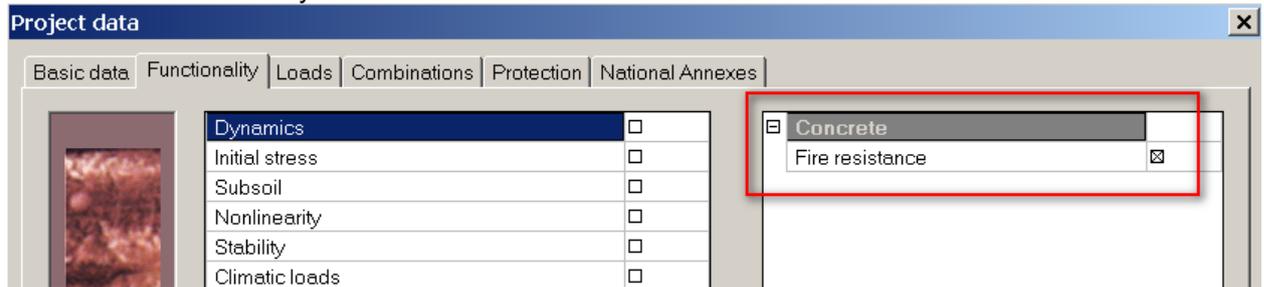
The Fire resistance module can be used to check both prestressed and non-prestressed 1D members according to EN1992-1-2.

In the EN1992-1_2, three types of fire-calculation are described:

- Simplified calculation method (Chapter 4.2 of 1992-1-2)
- Advanced calculation methods (Chapter 4.3 of 1992-1-2)
- Tabulated data (Chapter 5 of 1992-1-2)

In SCIA Engineer all three types are possible. So in this chapter all three possibilities will be explained.

First of all the functionality “Fire Resistance” has to be activated.



Simplified calculation method

See also example SimplifiedMethod.esa.

Annex B of EN 1992-1-2 provides two alternative methods for calculating the resistance to bending moments and axial forces

- B.1 “500°C isotherm method”
- B.2 “Zone method”

In SCIA Engineer the Zone method is implemented.

Thermal distribution curves

Form § 4.2.2 Temperature profiles of 1992-1-2:

“Temperatures in a concrete structure exposed to a fire may be determined from tests or by calculation

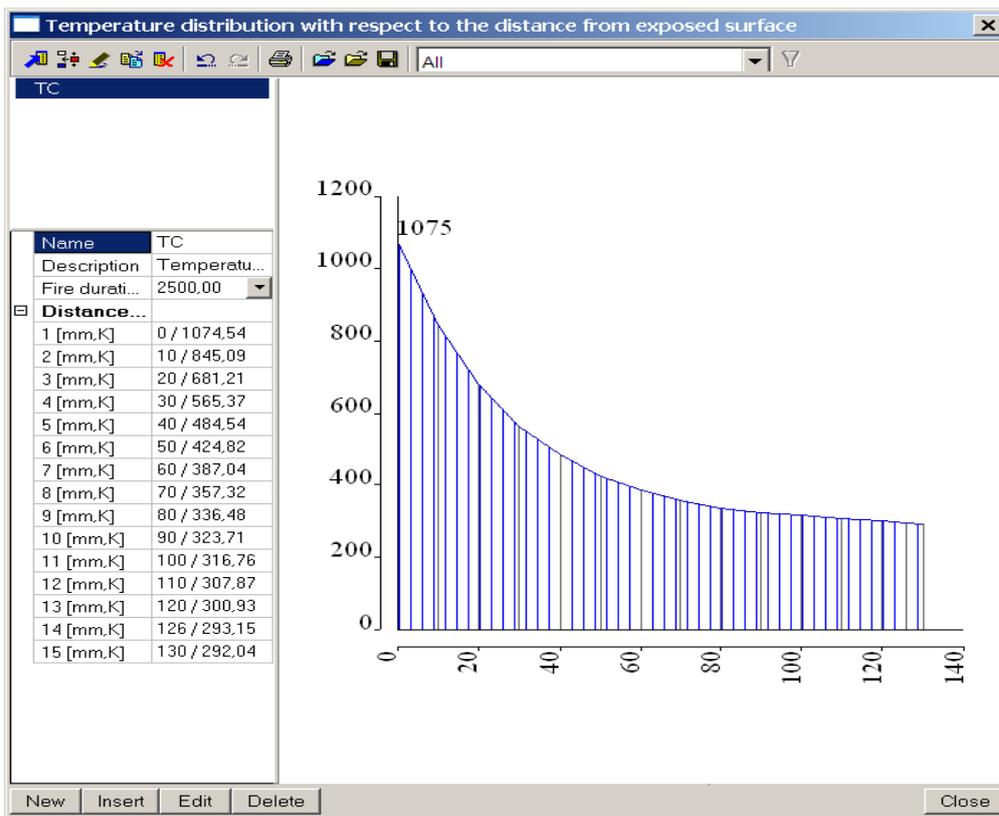
Note: The temperature profiles given in Annex A may be used to determine the temperatures in cross-sections with siliceous aggregate exposed to a standard fire up to the time of maximum gas temperature. The profiles are conservative for most aggregates.”

In below one example of a temperature profile of annex A is displayed:

One temperature distribution curve defined in the temperature curves manager may in fact consist of several individual curves. Each individual curve is defined for one time duration of fire.

The final temperature distribution curve stored in the Temperature curves manager can be used for any duration of fire that is within the interval of durations of individual curves. If the user specified a time duration which coincides with one of the individual curves, then that corresponding curve is used in the calculation. If the user specified a time duration which is somewhere in between, the curve to be used in the calculation is automatically interpolated by the program.

If a fire duration of 2500 sec is chosen in the Temperature curves manager, then the new curve will be automatically calculated by means of an interpolation:

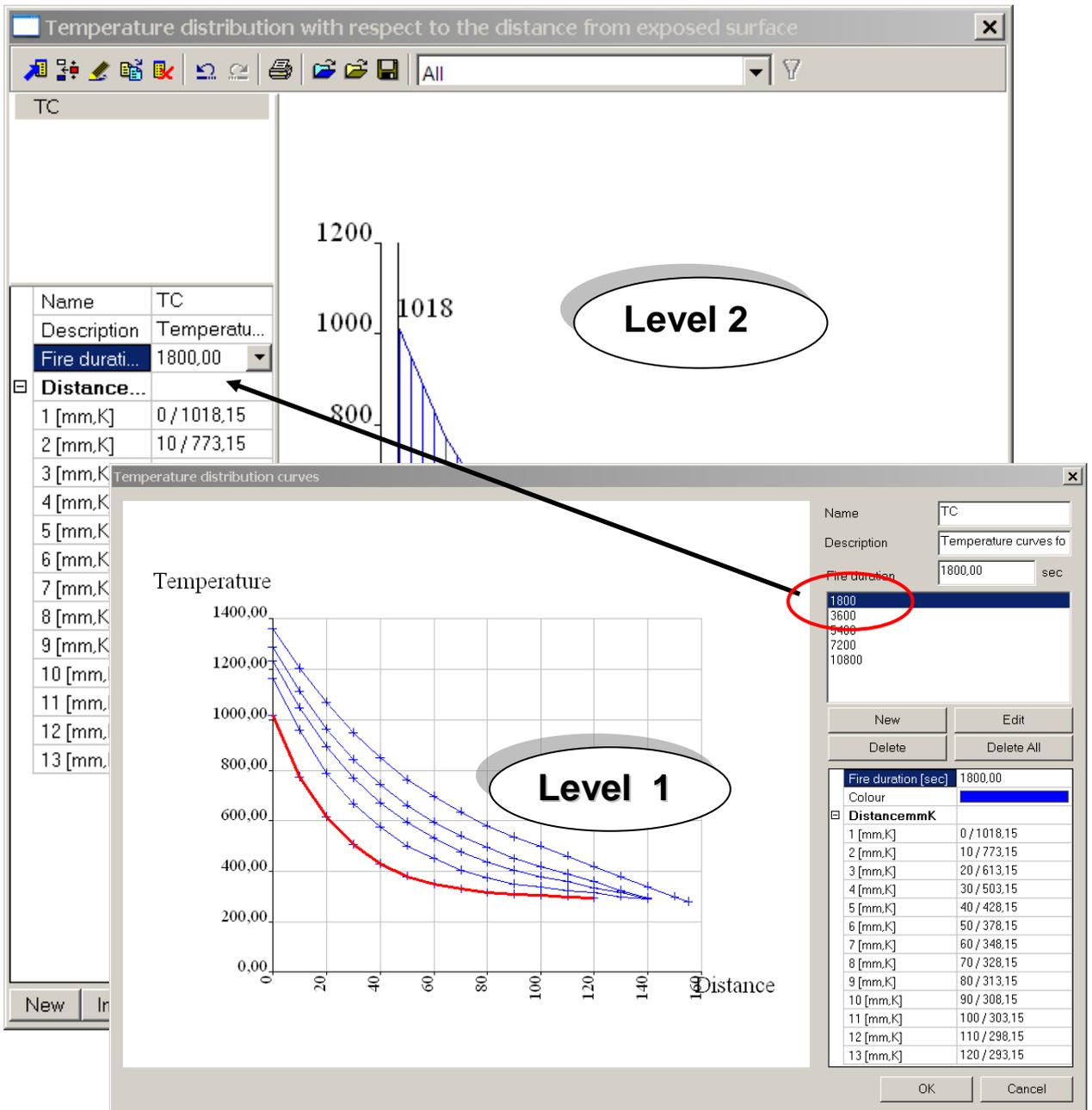


In this example a fire of 3600sec is implemented.

Consequently, the input dialogue is a two-level input dialogue.

The first level displays the whole set of defined curves. The second level enables the user to work with one particular individual curve.

It is, however, also possible that the final curve consists just of one individual curve.



Explanation of some of these parameters:

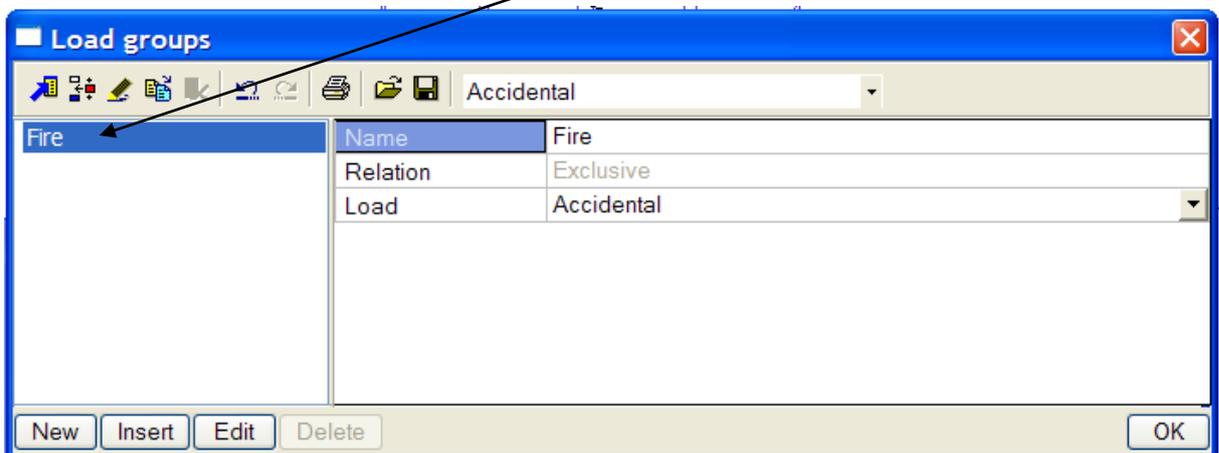
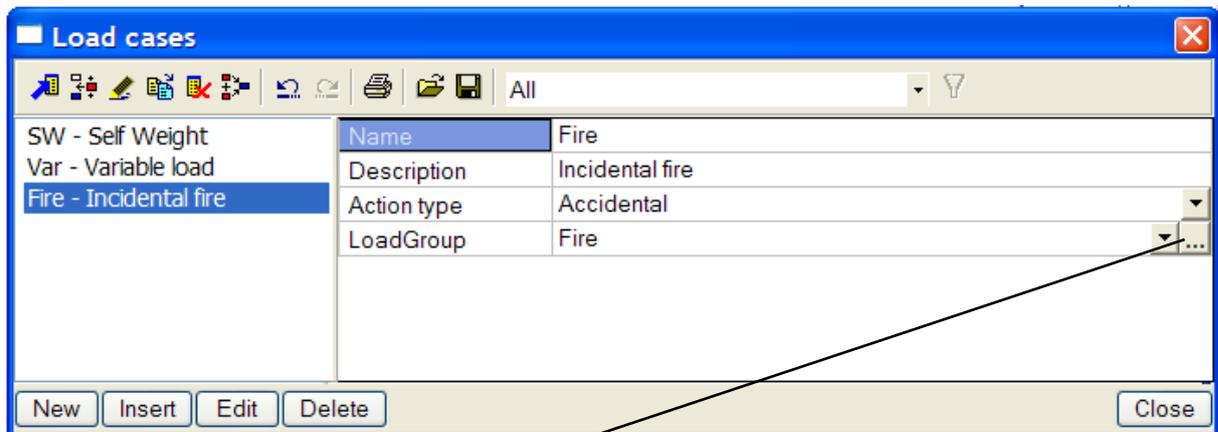
Fire duration	The final fire duration used in the calculation. If the value specified here coincides with one of the individual curves, the corresponding curve is used. Otherwise the final curve is obtained through interpolation.
Distance:	The distance from the face of the cross-section that is exposed to fire. The distance is measured from the face inwards.
Temperature	The temperature at the given distance.

Load case: Thermal load

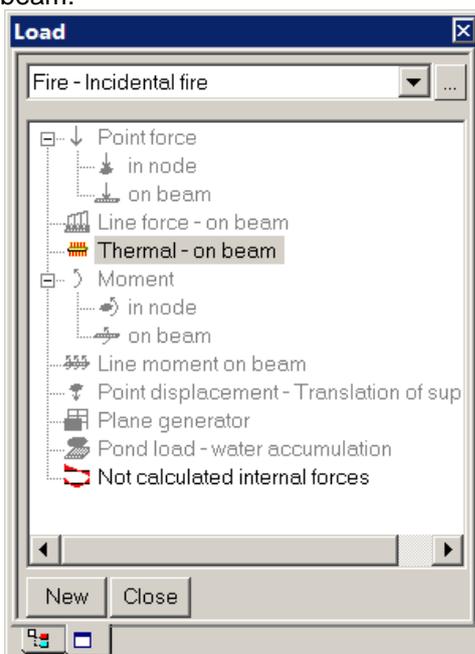
Once, the thermal curves are defined, a thermal load case can be created.

Open the load cases manager and create a load case *Fire*.

The action type and type of load group are put to **accidental**.



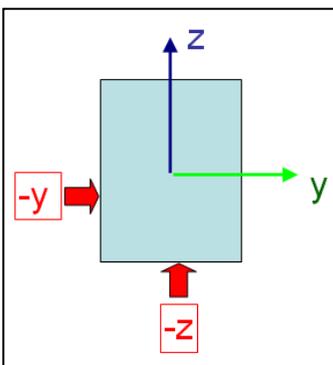
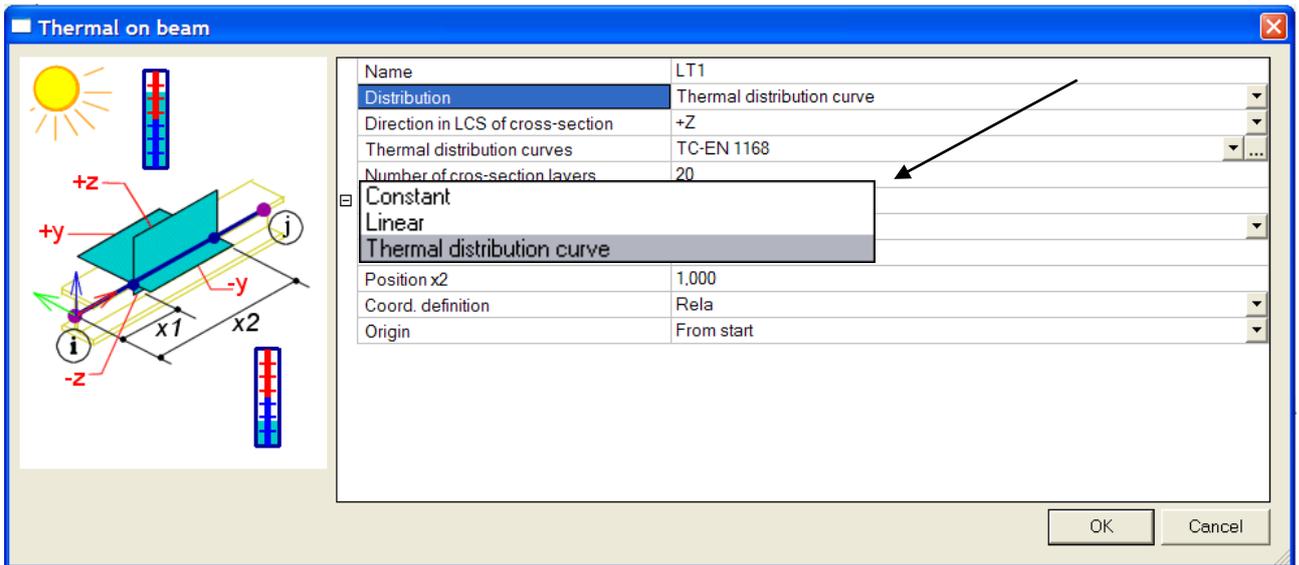
With these chosen options, only one type will be available in the load menu, namely: Thermal-on beam:



Select this type of load case.

A thermal load can be defined through a distribution curve that defines the distribution of the heat across the cross-section of a member.

When a member is exposed to fire, one side of the cross-section is directly exposed to the source of the heat. The temperature to which the interior of the cross-section is exposed decreases with the distance from the directly exposed face. The temperature distribution curve defines this drop in the temperature.



Direction in LCS of cross-section	This parameter specifies the direction from which the heat "comes", i.e. which face of the beam is exposed to the fire.
Temperature distribution curve	Specifies the temperature distribution curve that is used to determine the final load.
Number of cross-section layers	The actual distribution of the heat over the cross-section is generally non-linear (defined by the temperature distribution curve). The calculation algorithm, however, required a linear distribution. Therefore, the program must transform the curve into a trapezoid. The accuracy of this approximation can be controlled by this parameter. The higher the input number, the better accuracy and the more demanding (longer) the calculation. The parameter can be from the interval <5, 50>.

Load Combinations

With regard to the fire resistance calculations there are certain limitations and assumptions concerning load case combinations.

This type of calculation is supported only for accidental combinations:

EN-ULS Accidental – Psi1

EN-ULS Accidental – Psi2

Note :

- For detailing provisions (Type of calculation ="from temp. curves") and for simplified calculation (check response and check capacity) only accidental combinations (EN-ULS Accidental – Psi1 and EN-ULS Accidental – Psi2) can be used,
- **Temperature curves** can be defined only in a load case with **action type = "accidental"**.
- A load case with action type = "accidental" in an accidental combination is **in every linear combination** that will be generated from these accidental combinations.
- **Only one load** case with action type = "accidental" can be in one accidental combination.
- If an accidental combination **without load case with action type = "accidental"** appears in checks of fire resistance, the document will contain an error message.

Materials properties for concrete/reinforcement

For the simplified and advanced calculation new material properties of concrete must be defined. Other properties will be calculated using the simplified calculation method 'Zone method' (reduced strength, reduced Young's module, new strain-stress diagram).

Type of diagram

<input type="checkbox"/> Stress-strain diagram (fire resistance)	
Type of diagram	By code with automatic temperature
Picture of Stress-strain diagram	By code with automatic temperature
	By code with fixed temperature

There are two options:

By code with automatic temperature (default):

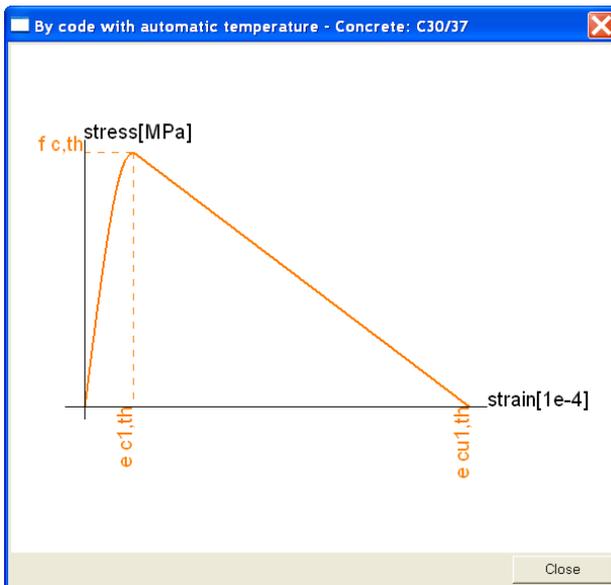
The temperature will be loaded automatically as the temperature in the centroid of a layer of a cross-section. For reinforcement this will be in position of the bar.

Property Picture of Stress-strain diagram is active, but the graphical interpretation of the stress/strain diagram is without numerical values.

By code with fixed temperature:

A new appearing item Temperature can be used to input the value of the temperature and a stress/strain diagram for this temperature will be the same as for all layers of the cross-section.

Property Picture of "Stress-strain diagram" is active and the graphical interpretation of the stress/strain diagram contains numerical values.



Type of aggregate

The material property **Type of aggregate** is also important for the fire resistance calculation and should be therefore adjusted with proper carefulness.

Siliceous aggregates are: quartzite aggregates, sandstone aggregates and basalt aggregates. Calcareous aggregates are only limestone aggregates.

Check

This type of method allows for the following checks:

- Method of limit strain
- Interaction diagram (resistance check)

Check capacity and **Check response** are performed for changed cross-section and materials properties. The following parameters are considered in the fire resistance check:

- Use of nonlinear and linear distribution temperature curves
- Changes in cross-section properties:
 - reduced height of section (damaged zone)
 - new eccentricities.
- Changes in material properties of concrete and non-prestressed reinforcement:
 - new stress/strain diagram
 - thermal expansion coefficient
 - reduced ultimate limit strain
 - reduced strength
 - reduced module of elasticity.

The calculation of the **internal forces** assumes or does not take into account the following:

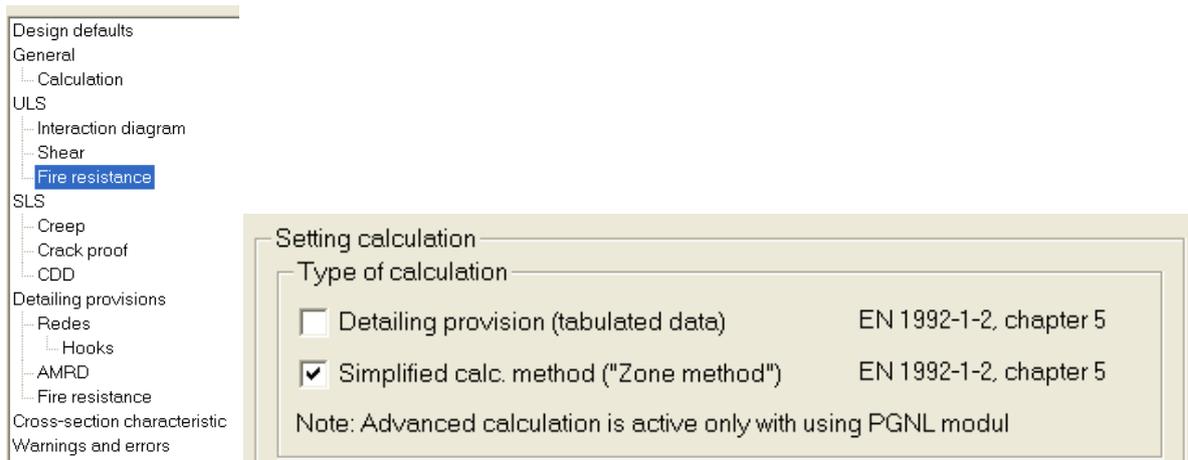
- Thermal expansion coefficient of concrete changes depending on the temperature
- Height of the cross-section is not reduced according to the temperature
- Material characteristics of reinforcement and concrete do not change depending on the temperature.
- Stress-strains diagrams of concrete and reinforcement depend on the temperature.
- Only a linear distribution temperature curve for temperature load.

Shear capacity is calculated according to EN 1992-1-1, but with following changes:

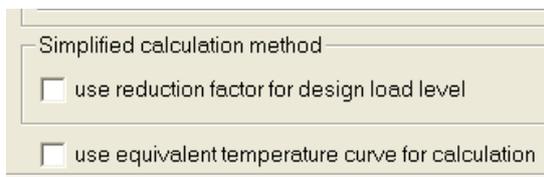
- reduced strength f_{cd} (temperature in centroid of section)
- reduced strength f_{ctd} (temperature in centroid of section)
- reduced geometry of cross-section
- reduced strength of stirrups f_{sd} (the reference temperature is the temperature in the point P (intersection of the branch of a stirrup with the section a–a, where the section a–a is the boundary of the effective tension area calculated according to EN 1992-1-1, chapter 7.3.2 (3))

How can this method be chosen in SCIA Engineer?

Go to **Concrete > Setup > ULS > Fire resistance** and check the option Simplified Calc. Method ("Zone method"):



Above this, the following options can be checked:



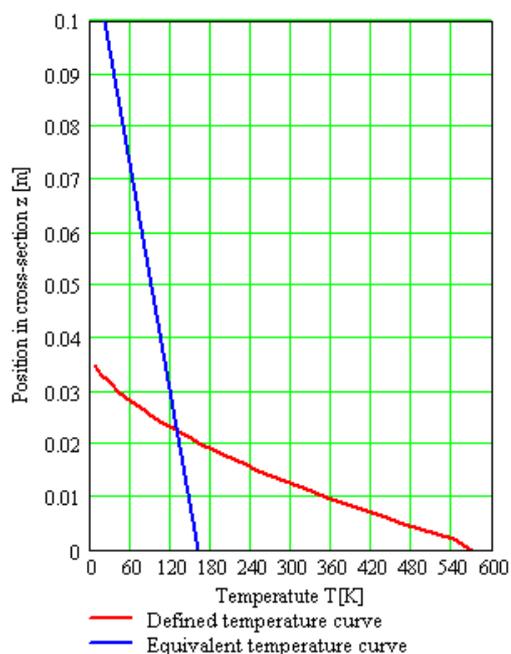
- **Use reduction factor for design load level:**

If ON, the internal forces for the check of fire resistance with simplified method will be multiplied by factor η_{fi} (value η_{fi} is set in **Concrete > Setup > ULS > Fire resistance**)

- **Use equivalent temperature curve for calculation:**

If item is turned OFF the calculation values, which depend on temperature, will use the defined temperature distribution curve

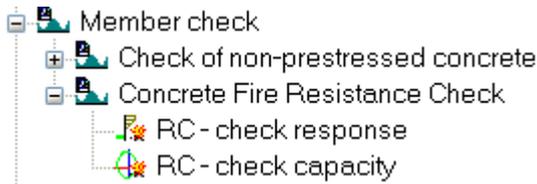
If item is turned ON the calculation values, which depend on temperature, will use an equivalent temperature distribution curve, see the picture below.



Note:

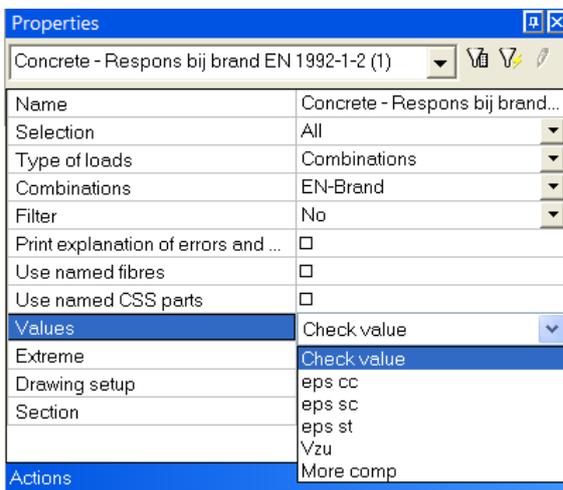
- Only **Check capacity** and **Check response** functions are supported.
- This method can be used only in the case of practical reinforcement and rectangular cross-sections.
- Simplified method is supported only for:
 - EN-ULS Accidental – Psi1
 - EN-ULS Accidental - Psi2

To see those results go to Concrete > 1D member > Member check > Concrete fire resistance check:



Two possible checks are available:

- **RC/PC-check response:**
Checks the limit strain in reinforced/prestressed concrete.



Check fire resistance response EN 1992-1-2

Linear calculation, Extreme : Member

Selection : All

Combinations : EN-Brand

Method of limit strain for selected members

Member	d _x [m]	Case	R [sec] Fibre	N [kN]	V _z [kN]	M _y [kNm]	Check ^{calc} [-]	Check ^{lim} [-]	Check W/E
				N ^(r) [kN]	V _{z,rec} [kN]	M _{y(r)} [kNm]			
S1	0,000	EN-Brand/1	7200,00	-158,83	12,96	-23,33	0,23	1,00	OK
				1 -158,83	12,96	-23,33			

- **RC/PC-check capacity:**
Checks the limit internal forces that the concrete cross-section can transfer.

Properties	
Concrete - Brandcapaciteit EN 1992-1-2 (1)	
Name	Concrete - Brandcapaciteit E...
Selection	All
Type of loads	Combinations
Combinations	EN-Brand
Filter	No
Print explanation of errors and ...	<input type="checkbox"/>
Values	Check value
Extreme	Check value
Drawing setup	Vzu
Section	Nu Myu Mzu More comp

Check fire resistance capacity EN 1992-1-2

Linear calculation, Extreme : Member

Selection : All

Combinations : EN-Brand

Method of interactive diagram for selected members

Member	d _x [m]	Case	R [sec]	N [kN]	V _z [kN]	M _y [kNm]	Check _{calc} [-]	Check W/E
			Check type	N _(t) [kN]	V _{z,rec} [kN]	M _{y(t)} [kNm]	Check _{lim} [-]	
S1	0,000	EN-Brand/1	7200,00	-158,83	12,96	-23,33	0,83	OK
			Mu	-158,83	12,96	-23,33	1,00	

In both cases, it's also possible to perform a single check:

The tab-sheet Cross-section shows the reduced height of the cross-section.
There is also a new tab-sheet Temperature.

- The following can be displayed:
 - temperature in each layer of the cross-section
 - temperature at the edges of the cross-section
 - fire duration
 - direction of temperature

Single cross-section

Calculation

Step / position: 0.5 | 1.8

<< >> << >>

automatic recalculation-redraw

Selected section

Code setup

Extreme: My+

N: -4.4149 kN
 Vy: 0 kN
 Vz: 0 kN
 Mx: 0 kNm
 My: -0 kNm
 Mz: -0 kNm

Calculation

Numerical output

Change of setup

Output

Cross-section | Loads | Vertical section | Horizontal section | Interaction diagram | Temperature

6x B 600C (12)
 Concrete: C30/37
 6x B 600C (12)
 Shear width bw= 500 mm
 Stirrups: 2x B 600C (8) Dist= 291 mm

Nu,min=-4.4149 kN = Nd=-4.4149 kN = Nu,max=-4.4149 kN
 Myu,min=-0.3135 kNm < Myd=-0.0000 kNm < Myu,max=33.2106 kNm
 Mzu,min=0.0000 kNm = Mzd=-0.0000 kNm = Mzu,max=0.0000 kNm

Calculation successful. There are no warnings or no errors. (1)
 Vz=0.0000 kN = Vz=0.0000 kN
 Stirrups distance not satisfy to detailing provision for column. (194)
 The calculation of cross-section satisfied. (161)

OK Cancel Apply

Single cross-section

Calculation

Step / position: 0.5 | 1.8

<< >> << >>

automatic recalculation-redraw

Selected section

Code setup

Extreme: My+

N: -4.4149 kN
 Vy: 0 kN
 Vz: 0 kN
 Mx: 0 kNm
 My: -0 kNm
 Mz: -0 kNm

Calculation

Numerical output

Change of setup

Output

Cross-section | Loads | Vertical section | Horizontal section | Interaction diagram | Temperature

Nu,min=-4.4149 kN = Nd=-4.4149 kN = Nu,max=-4.4149 kN
 Myu,min=-0.3135 kNm < Myd=-0.0000 kNm < Myu,max=33.2106 kNm
 Mzu,min=0.0000 kNm = Mzd=-0.0000 kNm = Mzu,max=0.0000 kNm

Calculation successful. There are no warnings or no errors. (1)
 Vz=0.0000 kN = Vz=0.0000 kN
 Stirrups distance not satisfy to detailing provision for column. (194)
 The calculation of cross-section satisfied. (161)

OK Cancel Apply

Advanced calculation method

This type of method can be used only in combination with the PGNL module. With other words, only nonlinear combinations are supported.

For the definition of the nonlinear combination a new combo box for the Fire type appears:

- Prefire (internal forces without influence of thermal load, material characteristic and height of cross-section are not reduced)
- during fire (internal forces with influence of thermal load, material characteristic and height of cross-section are reduced)
- post fire (internal forces without influence of thermal load, material characteristic and height of cross-section are reduced)

These nonlinear combinations are available only if the functionalities Fire resistance for concrete and Physical non-linearity for reinforced concrete are checked.

The following assumptions are taken into account for the check:

- Reduced height of section
- Stress-strain diagram with influence of temperature
- Coefficient of thermal expansion with influence of temperature
- Module of elasticity of concrete with influence of temperature
- Nonlinear or linear distribution temperature curves
- Load by temperature is calculated for linear distribution temperature

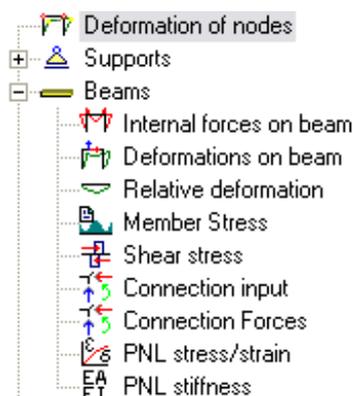
Note:

Only for practical reinforcement and rectangular cross-sections

Results for the check of the advanced method

In the property menu, there appears a new combobox in which the option pre-, during- and post-fire can be chosen.

In this method the PNL stresses and strains can be asked.
Also the PNL stiffnesses are showed.



Tabulated data

See also example TabulatedData.esa.

Load Combinations

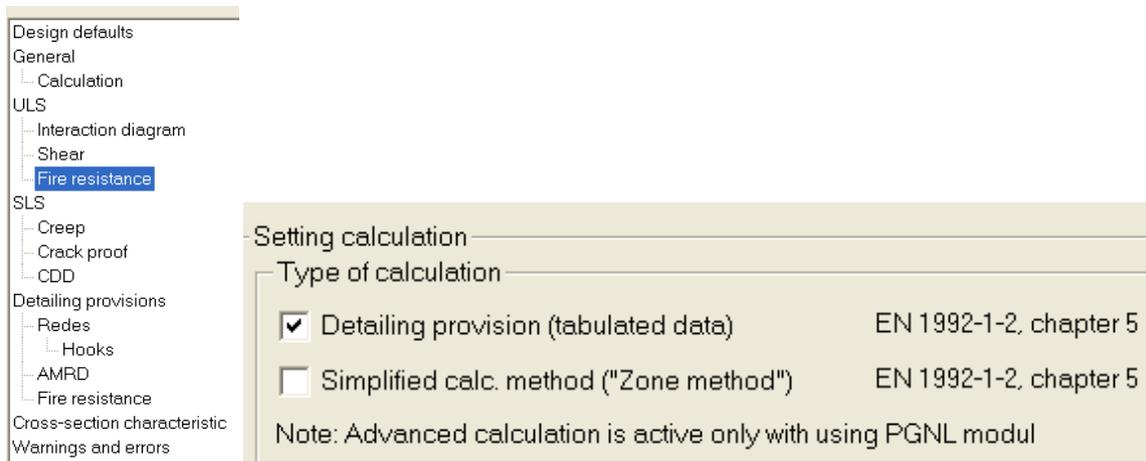
For type of calculation = "Input by user" and "Automatic" all types of load cases/combinations/ classes can be used.

In this example an ULS combinations is inserted (to calculate the theoretical reinforcement) and an accidental combination.

Setup for tabulated data

So activate the calculation following this tabulated data, activate this option

Go to **Concrete > Setup > ULS > Fire resistance** and check the option Simplified Calc. Method ("Zone method"):



This check is performed following

- Table 5.2a (for columns)
- Table 5.5 or Table 5.6 (for beams).

Columns | **Beams** | Slabs

Reduced minimum axis distance a_{min} according to 5.2(7c)

Reduced cross-sectional dimensions b_{min} according to 5.2(10)

Table for min. dim. b_{min} and min. axis dist. a_{min} (Table 5.5 and 5.6)

SFR	$b_{min}(S)[mm]$	$a_{min}(S)[mm]$	$b_{min}(C)[mm]$	$a_{min}(C)[mm]$
R30	120	20	160	12
R60	160	35	200	12
R90	200	45	250	25
R1...	240	60	300	35
R1...	300	70	400	50
R2...	350	80	500	60

Type of beam: (S): simply supported beam (C): continuous beam

SFR: Standard fire resistance

Load default values

Columns | Beams | **Slabs**

Reduced minimum axis distance a_{min} according to 5.2(7c)

Table for $h_{s,min}$ and a_{min} for slabs (Table 5.8)

SFR	$h_{s,min}[mm]$	$a_{min}[mm]$
R30	60	10
R60	80	20
R90	100	30
R120	120	40
R180	150	55
R240	175	65

Table for $h_{s,min}$ and a_{min} for hollow core slabs (EN 1168, Table G1)

SFR	$h_{s,min}[mm]$	$a_{min}[mm]$
R15	100	10
R20	100	10
R30	100	10
R40	100	15
R60	120	20
R90	140	30
R120	160	40
R180	200	55

SFR: Standard fire resistance

Load default values

In SCIA Engineer, the tables can be verified and adapted under '**Setup > Detailing provisions > Fire resistance**'. Also reductions of the tabulated data can be input here:

In this table 2 options are checked:

- b_{min} : minimum width of the beam
- a_{min} : minimum for the average axis distance of the longitudinal reinforcement

As shown in those tables, the minimum values for only depends on the time of the fire resistance. E.g. a R60 corresponds with 60 minutes of fire resistance.

The calculation of b_{min} and a_{min} is discussed in the next paragraph.

To change the fire (time) resistance R, go to **Concrete > Setup > ULS > Fire resistance** and check the option 'Detailing provision'.

Detailing provisions

Fire(time) resistance R and critical temperature Theta_cr

automatic by code Fire(time) resistance R 1800 sec
 from temp. curve Theta_cr 773.2 K
 user input

Automatic by code = calculation according chapter 5.2(7) [2]

This can be adjusted in the setup for the entire structure but can also be overwritten for each beam by means of the Member Data (see later).

It is possible to activate the changes in the tabulated data in SCIA Engineer. Those options can also be found at “Concrete -> 1D Member -> Setup” when selecting “Detailing provisions -> Fire resistance”:

Changes in tabulated data for beams and slabs

Reduce values bmin (hs,min) for beams and slabs for calcareous aggregate by 10 % 5.1.(2) [2]

Increase values amin for beams and slabs for prestressing bars 10 mm 5.2.(5) [2]

Increase values amin for beams and slabs for prestressing wires and strands 15 mm 5.2.(5) [2]

Note:

- Only the check of practical reinforcement is supported.
- In this method, there is no reduction of the height of section. There are also no changes in the material characteristics.
- Detailing provisions are supported only for:
 - any load case, combination and class in case of "Automatic" and "Input by user" calculation
 - only for a load case, combination and class with defined Temperature distribution curve in case of "From temp. curve"

Concrete member data

The Member data dialogue contains group **Fire resistance** for all types of beam if the functionality **Concrete >Fire resistance** is turned on in the Project data dialogue.

Detailing provisions

For beam type ‘beam’:

Fire resistance	
Detailing provisions	
Fire (time) resistance R [sec]	8200,00
Column exposed	more than one side
Member	S1

For beam type ‘column’:

Fire resistance	
Detailing provisions	
Simply supported member	<input type="checkbox"/>
Type of calculation	automatic by code
Fire (time) resistance R [sec]	1800,00
Member	S1

Simply supported member:

This option is active only for **Beam type** = "beam". If ON, the calculation of values a_{min} and b_{min} in table **Table for minimum dimension b_{min} and minimum axis distance a_{min} (Table 5.5 and 5.6)** uses the column for simply supported member.

Type of calculation:

The option is active only for **Beam type** = "Beam" and "Beam slab".

The options are:

- from. temp.curve (default)
- automatic
- input by user

Theta_cr:

Default value is 773.15C.

This item is active only for **Beam type** = "Beam" and "Beam slab" and if the **Type of calculation** = "input by user".

Fire(time) resistance R [sec]:

Default value is 7200s.

This item is active only if the **Type of calculation** = "input by user" or "automatic".

Column exposed:

The item offers two items:

- one side
- more than one side (default)

This item is active only for **Type of beam** = "column".

Note:

If **no member data** are assigned to a particular member, the following options are considered:
For columns without member data, **Column exposed** = **More than one side**

For beams without member data, **Type of beams** = **continuous beam**.

Check

Default, the tabulated data are given at a critical Temperature equal to 500°C. The tabulated data has to be reduced for a critical temperature different from this one with

$$\Delta a = 0,1 * (500 - \theta_{cr}) \text{ (mm)} \quad \text{(formula (5.3) EC1992-1-2)}$$

With $350^{\circ}\text{C} < \theta_{cr} < 700^{\circ}\text{C}$

$$b_{mod} \geq b_{min} + 0,8 * (400 - \theta_{cr}) \text{ (mm)} \text{ (formula (5.4) EC1992-1-2)}$$

So $\theta_{cr} < 400^{\circ}\text{C}$

First the calculation of this critical temperature is explained. Afterwards the checks will be discussed.

Calculation of θ_{cr}

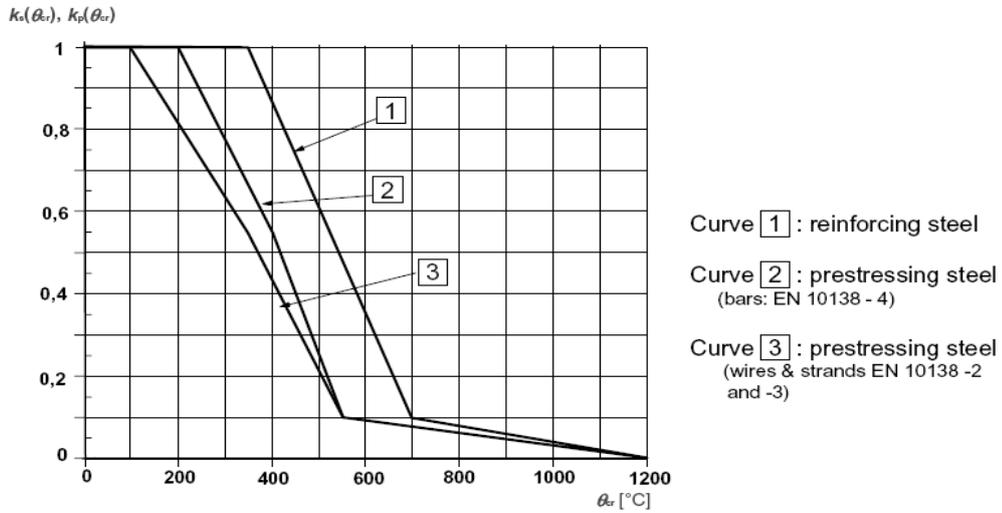
The value **Critical temperature θ_{cr}** is used for modification of a_{min} and b_{min}

This value can be calculated with 3 options in SCIA Engineer and those options can be found at **Concrete > Setup > ULS > Fire resistance** and choose the option 'Detailing provision'.



- loaded from **Concrete setup** or **Member data** for type of calculation = "Input by user",

- loaded from Temperature distribution curve for type of calculation ="from temp. curve",
- calculated according to chapter 5.2(6) for type of calculation ="Automatic" . Figure 5.1 gives the reduction k_s of the characteristic strength of reinforcing and prestressing steel as a function of the temperature θ_{cr}



Note:

- The option automatic and input by user are available for all type of cross-sections
- The option Temperature curves are only available for rectangular cross-sections

This last option is used in the example.

So θ_{cr} can be calculated with k_s .

And :

$$k_s(\theta_{cr}) = \frac{\sigma_{s,fi}}{f_{yk}}$$

And

$$\sigma_{s,fi} = \eta_{s,fi} \frac{f_{yk}(20C)}{\gamma_s} \frac{A_{s,req}}{A_{s,prov}}$$

Conservative $\eta_{fi} = 0,7$

And in this example: $A_{s,req}$ can be calculated using SCIA Engineer (theoretical reinforcement):

Design As EN 1992-1-1

Linear calculation, Extreme : Member

Selection : All

Combinations : CO2

Main lower reinforcement for selected beams

Member	d_x [m]	Case	N [kN]	$M_{y,d}$ [kNm]	x_{rel} [mm]	d [mm]	$A_{s,req}$ [mm ²]	Reinf.[no.]	W/E
B1	2,500	CO2/2	0,00	60,67	42	554	225	2x16,0(402)	147

Afterwards a user reinforcement is inputted and this is a reinforcement of:

Design As EN 1992-1-1

Linear calculation, Extreme : Member

Selection : All

Combinations : CO2

Main upper reinforcement for selected beams

Member	d _x [m]	Case	N _d [kN]	M _{yd} [kNm]	x _u [mm]	d [mm]	A _{s,req} [mm ²]	A _{s,user} [mm ²]	Reinf.[no.]
B1	0,000	CO2/1	0,00	2,95	0	0	0	603	3d16(B 500A)(603)

Main lower reinforcement for selected beams

Member	d _x [m]	Case	N _d [kN]	M _{yd} [kNm]	x _u [mm]	d [mm]	A _{s,req} [mm ²]	A _{s,user} [mm ²]	Reinf.[no.]
B1	0,000	CO2/2	0,00	12,98	0	0	0	603	3d16(B 500A)(603)

This can be inputted in the formula above:

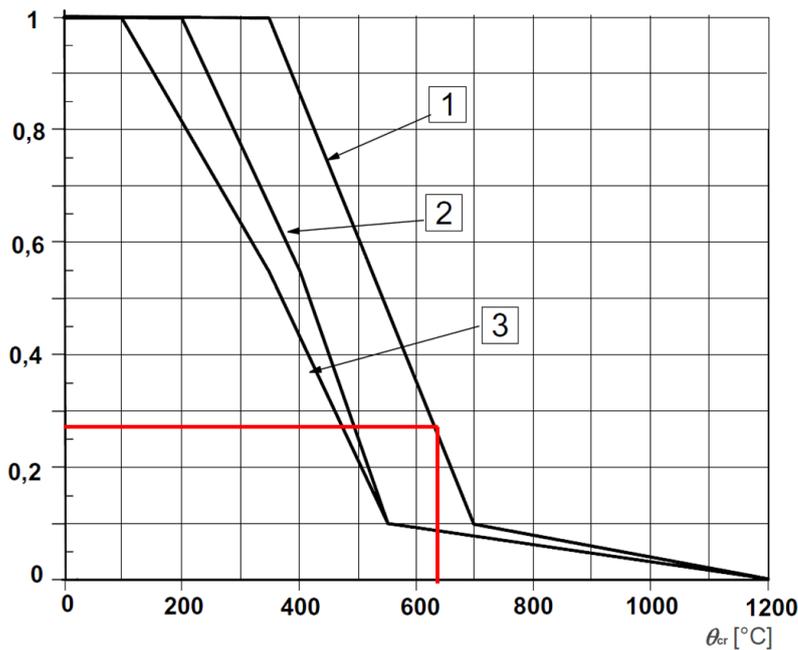
$$\sigma_{s,fi} = \eta_{s,fi} \frac{f_{yk}(20C)}{\gamma_s} \frac{A_{s,req}}{A_{s,prov}}$$

$$\sigma_{s,fi} = 0,7 \frac{500 \cdot 225}{1 \cdot 603} = 130,6 \text{ MPa}$$

And

$$k_s(\theta_{cr}) = \frac{\sigma_{s,fi}}{f_{yk}} = \frac{130,6 \text{ MPa}}{500 \text{ MPa}} = 0,26$$

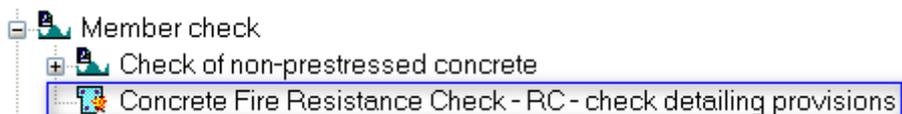
$k_s(\theta_{cr}), k_p(\theta_{cr})$



So $\theta_{cr} = 680^\circ\text{C}$.

Check results

Go to Concrete > 1D member > Member Check.



The following values can be asked now:

Concrete - Detaillierungsregeln EN 1992-1-2 (1)	
Name	Concrete - Detaillierungsregeln EN 1992-...
Selection	All
Type of loads	Combinations
Combinations	EN-Brand
Filter	No
Print explanation of errors and warnings	<input type="checkbox"/>
Values	b_min / b_cal
Extreme	a_min / a_cal
Drawing setup	b_min / b_cal
Section	R_cal
	More comp

With:

amin/acal	Check value for axial distance of reinforcement from the edge of cross-section.
bmin/bcal	Check value for the minimum dimension of the cross-section.
Rcal	Calculated value of fire resistance of section calculated from acal and bcal.

In this example following results are displayed:

Check detailing provisions EN 1992-1-2

Linear calculation, Extreme : Member

Selection : All

Combinations : CO2

Concrete fire resistance - detailing rules

Member	d _{cr} [m]	Case	a _{min} /a _{cal} [-]	b _{min} /b _{cal} [-]	R [sec]	R _{cal} [sec]	Check _{calc} [-]	Check _{lim} [-]	Check	W/E
B1	0,000	CO2/1	0,35	1,00	7200,00	7200,00	1,00	1,00	OK	213,212

Explanation of warnings and errors

- 212 | Minimum cross-sectional dimension for beams is not reduced, because critical temperature is higher than 400C(673.15K).
- 213 | Minimum dimension of section and minimum axis distance are satisfied.

b_{min} can be recalculated with following formula (as given before):

$b_{mod} \geq b_{min} + 0,8 * (400 - \theta_{cr})$ (mm) (formula (5.4) EC1992-1-2)

So $\theta_{cr} < 400^\circ\text{C}$ and $\theta_{cr} = 680^\circ\text{C}$, so b_{min} will not be adapted

So this check can be done just using the table below:

Columns **Beams** Slabs

Reduced minimum axis distance a_{min} according to 5.2(7c)
 Reduced cross-sectional dimensions b_{min} according to 5.2(10)

Table for min. dim. b_{min} and min. axis dist. a_{min} (Table 5.5 and 5.6)

SFR	$b_{min}(S)[mm]$	$a_{min}(S)[mm]$	$b_{min}(C)[mm]$	$a_{min}(C)[mm]$
R30	120	20	160	12
R60	160	35	200	12
R90	200	45	250	25
R1...	240	60	300	35
R1...	300	70	400	50
R2...	350	80	500	60

Type of beam: (S): simply supported beam (C): continuous beam

SFR: Standard fire resistance

Load default values

After 7200 sec (= 120 minutes), the minimum for $b_{min} = 300mm$
 A cross section of 600mm x 300mm has been inputted, so this check is 1.

a_{min} can be adapted using the following formula:

$$\Delta a = 0,1 * (500 - \theta_{cr}) \text{ (mm)} \quad \text{(formula (5.3) EC1992-1-2)}$$

With $350^\circ C < \theta_{cr} < 700^\circ C$ and $\theta_{cr} = 680^\circ C$

$$\text{so } a_{min} = 35mm + 0,1 * (500 - 680) = 17mm$$

In this example $a = c + \text{diameter_stirrups} + (\text{diameter_logitunal_reinforcement})/2$

$$a = 30 + 8 + 16/2 = 46mm$$

This check = $17mm / 46mm = 0,36$

The last check is R_{cal} . For a column, this is calculated with the formula given in (5.7) of EN1992-1-2:

$$R = 120 ((R_{\eta,fi} + R_a + R_l + R_b + R_n) / 120)^{1,8}$$

where

$$R_{\eta,fi} = 83 \left[1,00 - \mu_{fi} \frac{(1+\omega)}{(0,85/\alpha_{cc}) + \omega} \right]$$

$$R_a = 1,60 (a - 30)$$

$$R_l = 9,60 (5 - l_{0,fi})$$

$$R_b = 0,09 b'$$

$$R_n = 0 \quad \text{for } n = 4 \text{ (corner bars only)}$$

$$= 12 \quad \text{for } n > 4$$

a is the axis distance to the longitudinal steel bars (mm); $25 \text{ mm} \leq a \leq 80 \text{ mm}$

$l_{0,fi}$ is the effective length of the column under fire conditions; $2 \text{ m} \leq l_{0,fi} \leq 6 \text{ m}$;
 values corresponding to $l_{0,fi} = 2 \text{ m}$ give safe results for columns with $l_{0,fi} < 2 \text{ m}$

$b' = 2A_c / (b+h)$ for rectangular cross-sections or the diameter of circular cross sections

$200 \text{ mm} \leq b' \leq 450 \text{ mm}$; $h \leq 1,5 b$.

ω denotes the mechanical reinforcement ratio at normal temperature conditions :

$$= \frac{A_s f_{yd}}{A_c f_{cd}}$$

α_{cc} is coefficient for compressive strength (see EN 1992-1-1)

For beams this is calculated by interpolation of the tabulated data.