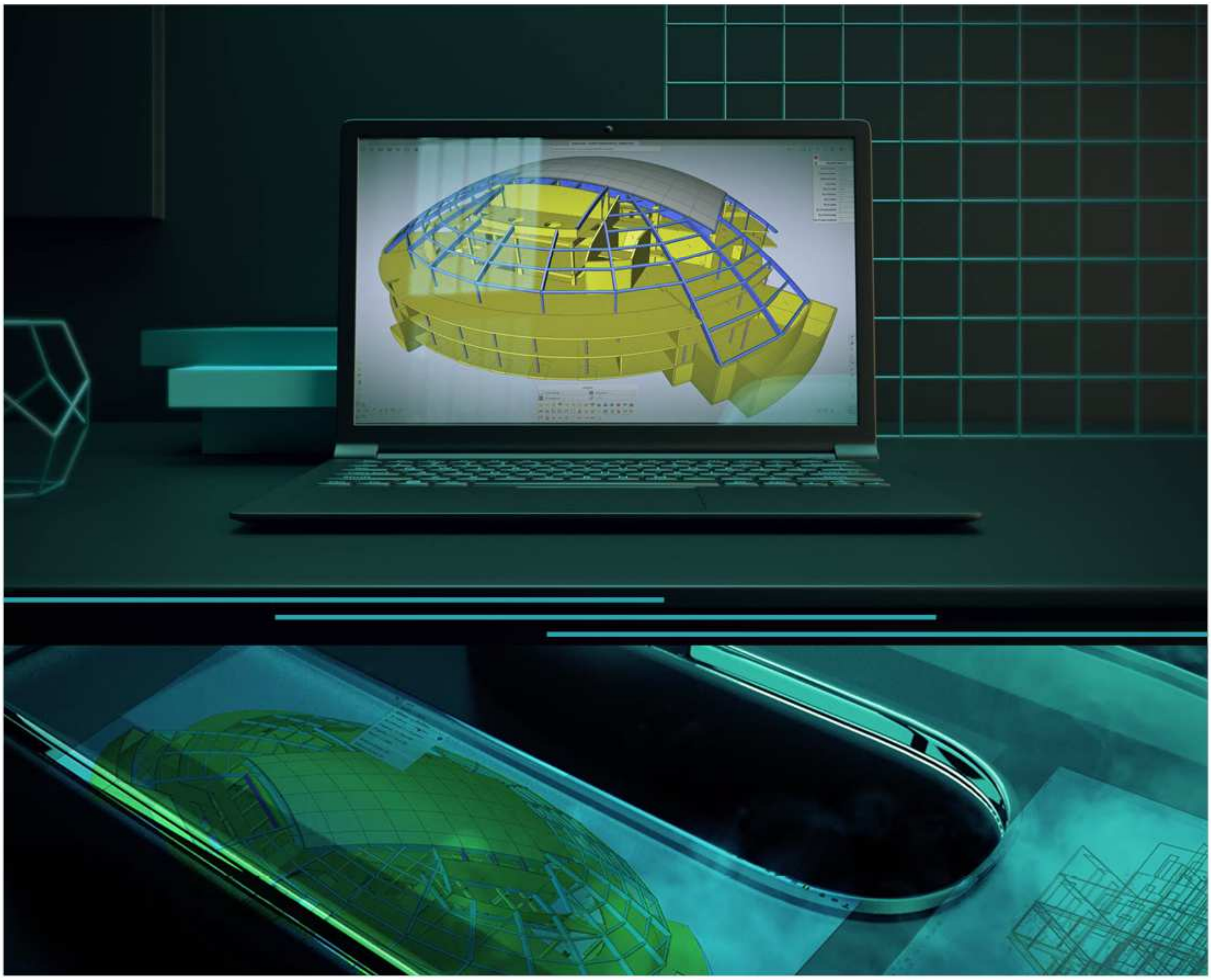


ADVANCED CONCEPT TRAINING

Concrete

Table of Contents

Chapter 1: Materials

1.1. Verification by the partial factor method

Cf art 2.4.2.4.

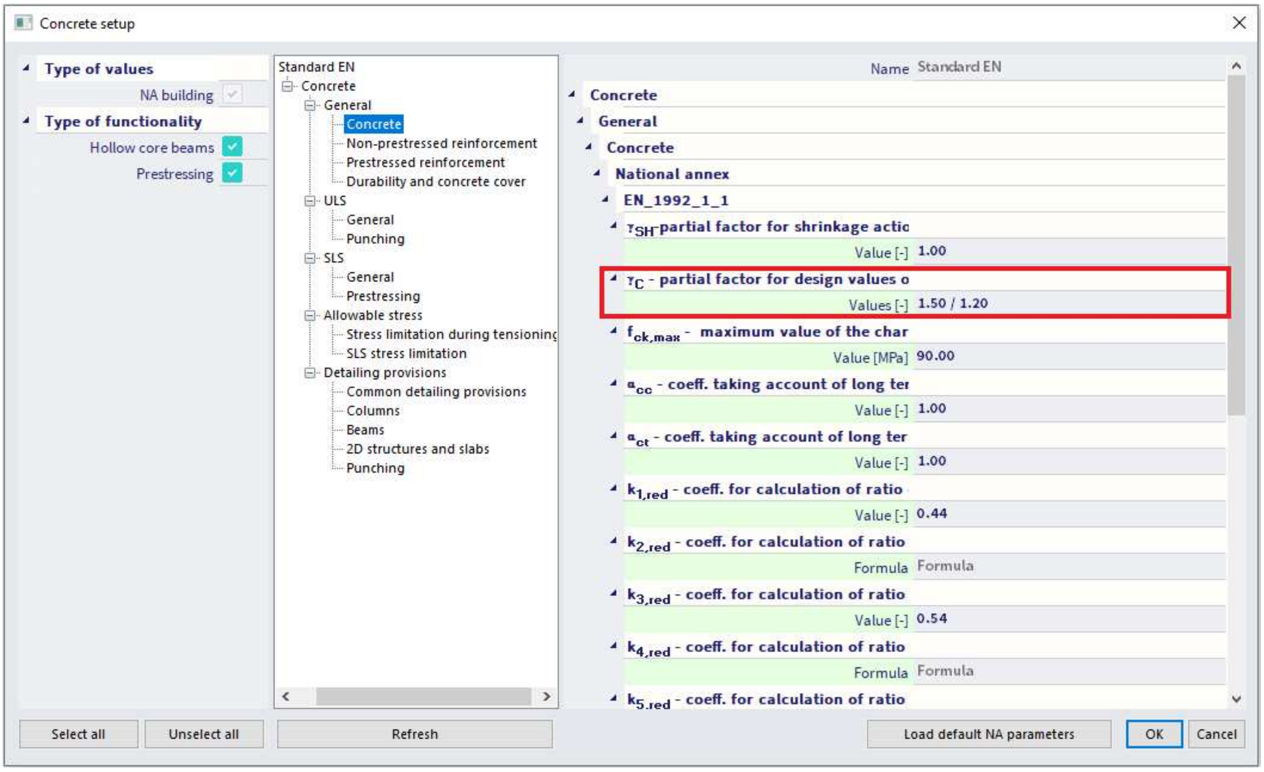
Partial factors for materials for ultimate limit states, γ_c and γ_s should be used.

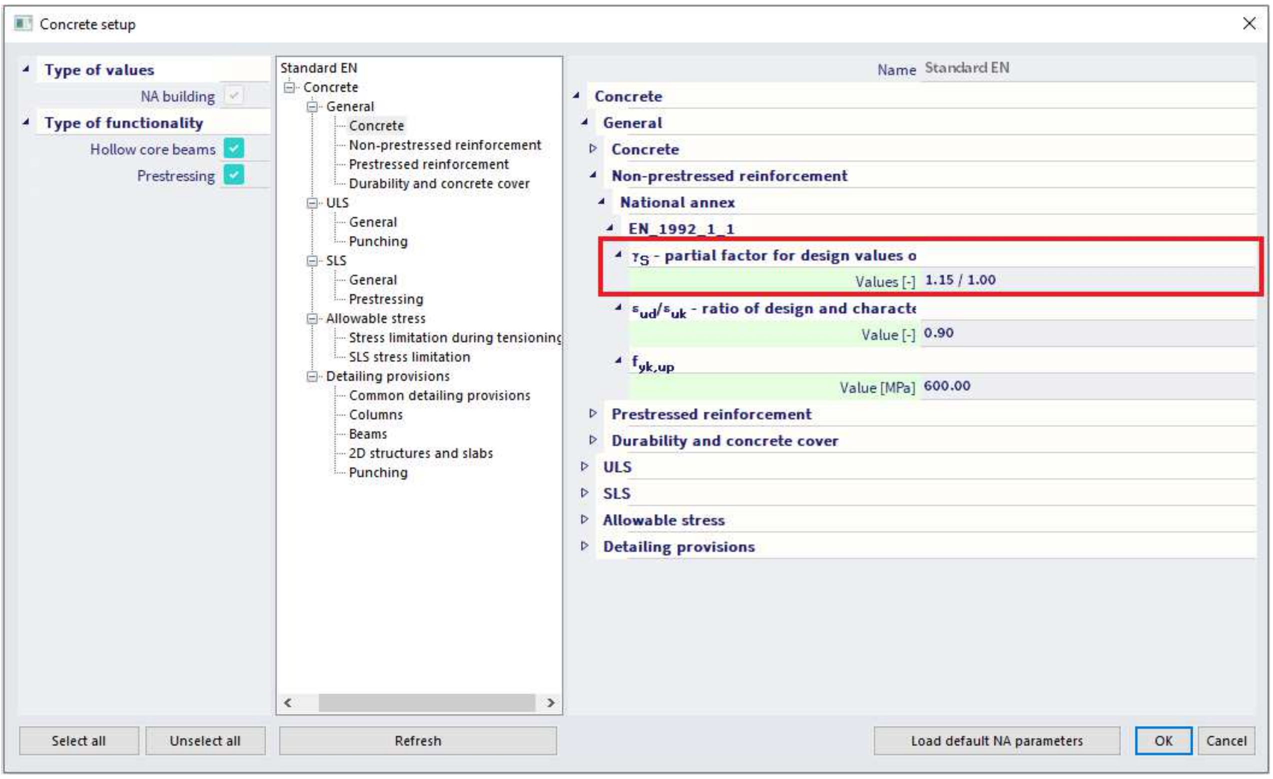
The recommended values of γ_c and γ_s for 'persistent & transient' and 'accidental, design situations are given in the following table. These are not valid for fire design for which reference should be made to EN 1992-1-2.

For fatigue verification the partial factors for persistent design situations given in this table are recommended for the values of $\gamma_{c,fat}$ and $\gamma_{s,fat}$.

Design situations	γ_C for concrete	γ_S for reinforcing steel	γ_S for prestressing steel
Persistent & Transient	1,5	1,15	1,15
Accidental	1,2	1,0	1,0

These values can also be found in the Concrete setup of the National Annex:





All factors related to the code are shown in green on the screen. By default, the values of the chosen code are taken.

The values for partial factors for materials for serviceability limit state verification should be taken as those given in the particular clauses of this Eurocode.

The recommended values of γ_c and γ_s in the serviceability limit state for situations not covered by particular clauses of this Eurocode is 1,0.

Lower values of γ_c and γ_s may be used if justified by measures reducing the uncertainty in the calculated resistance.

1.2. Concrete

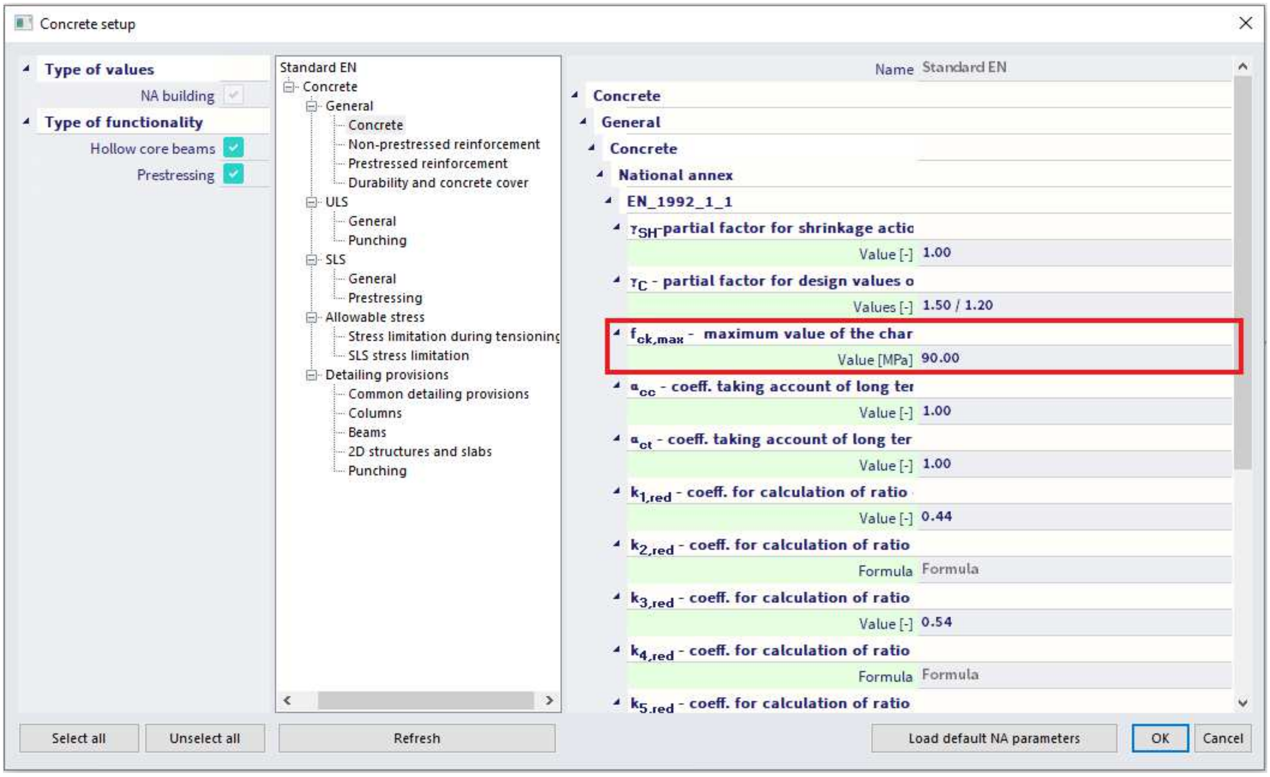
The following clauses give principles and rules for normal and high strength concrete.

1.2.1. Strength (art 3.1.2)

The compressive strength of concrete is denoted by concrete strength classes which relate to the characteristic (5%) cylinder strength f_{ck}, or the cube strength $f_{ck,cube}$.

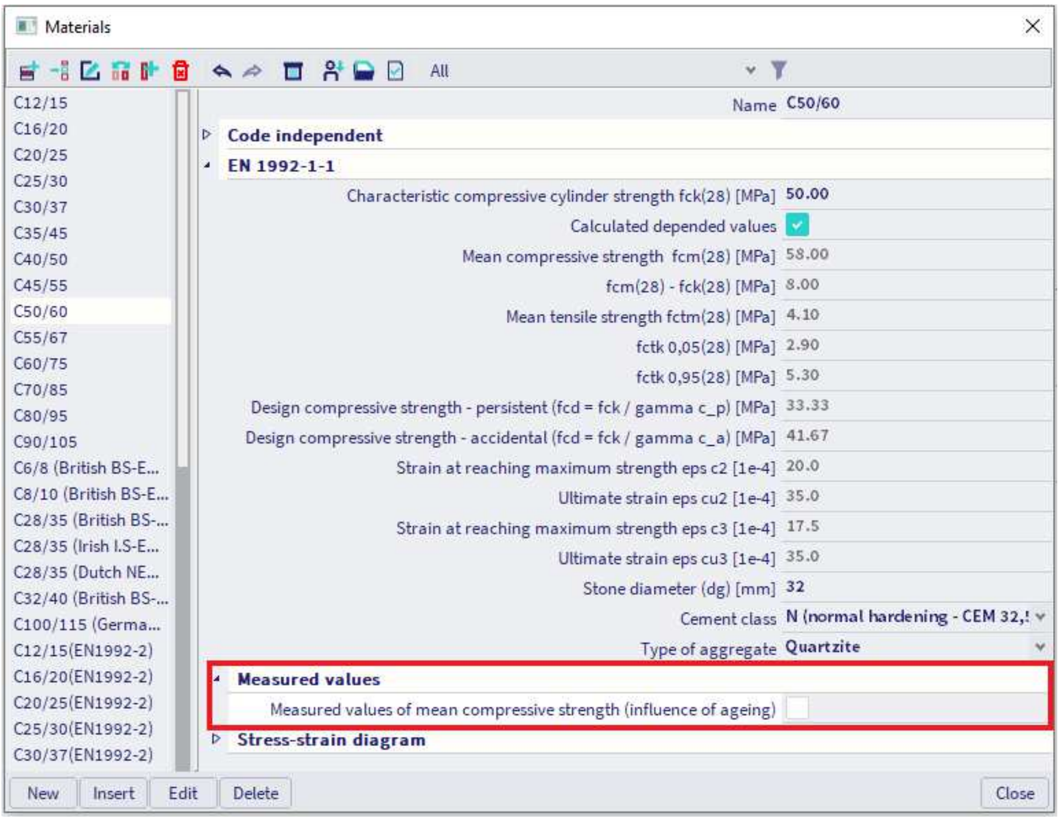
The strength classes in this code are based on the characteristic cylinder strength f_{ck} determined at 28 days with a maximum value of C_{max}.

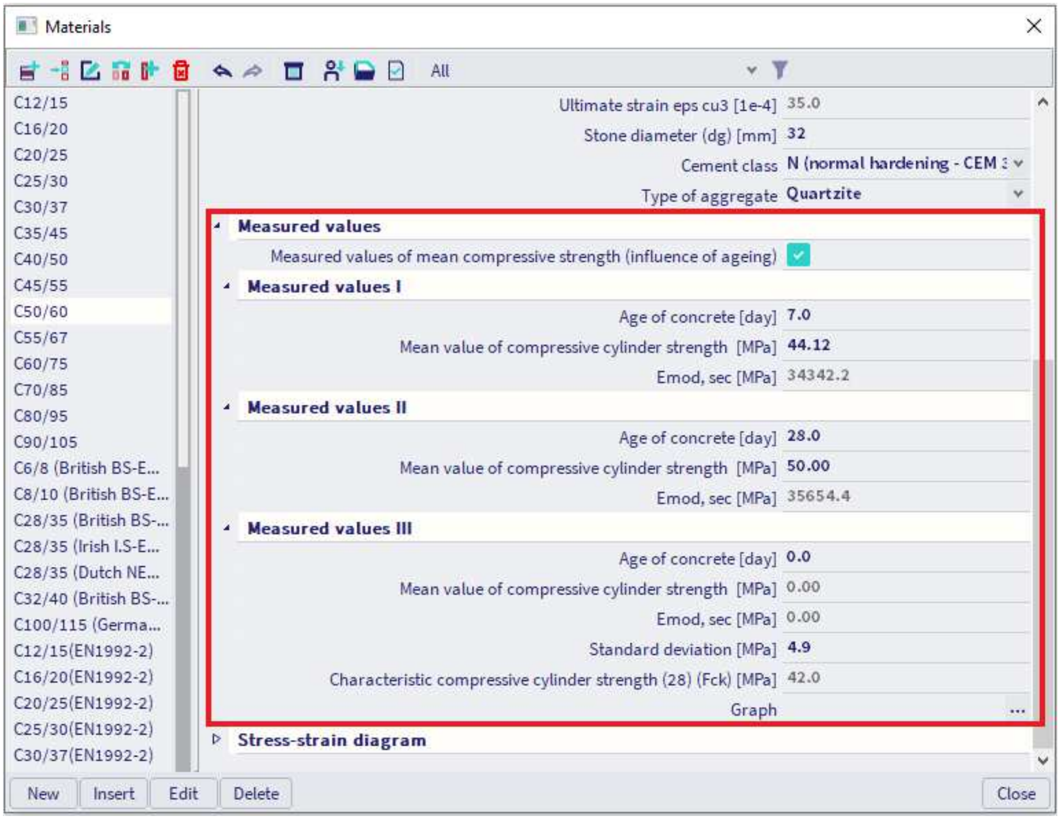
The recommended value of C_{max} is C90/105.



In certain situations (e.g. prestressing) it may be appropriate to assess the compressive strength for concrete before or after 28 days, on the basis of test specimens stored under other conditions than prescribed in EN 12390.

All values can also be found in the material library of SCIA Engineer:





It may be required to specify the concrete compressive strength, $f_{ck}(t)$, at time t for a number of stages (e.g. demoulding, transfer of prestress), where:

$f_{ck}(t) = f_{cm}(t) - 8$ (MPa) for $3 < t < 28$ days

$f_{ck}(t) = f_{ck}$ for $t \geq 28$ days

The compressive strength of concrete at an age t depends on the type of cement, temperature and curing conditions. For a mean temperature of 20°C and curing in accordance with EN 12390 the compressive strength of concrete at various ages $f_{cm}(t)$ may be estimated from:

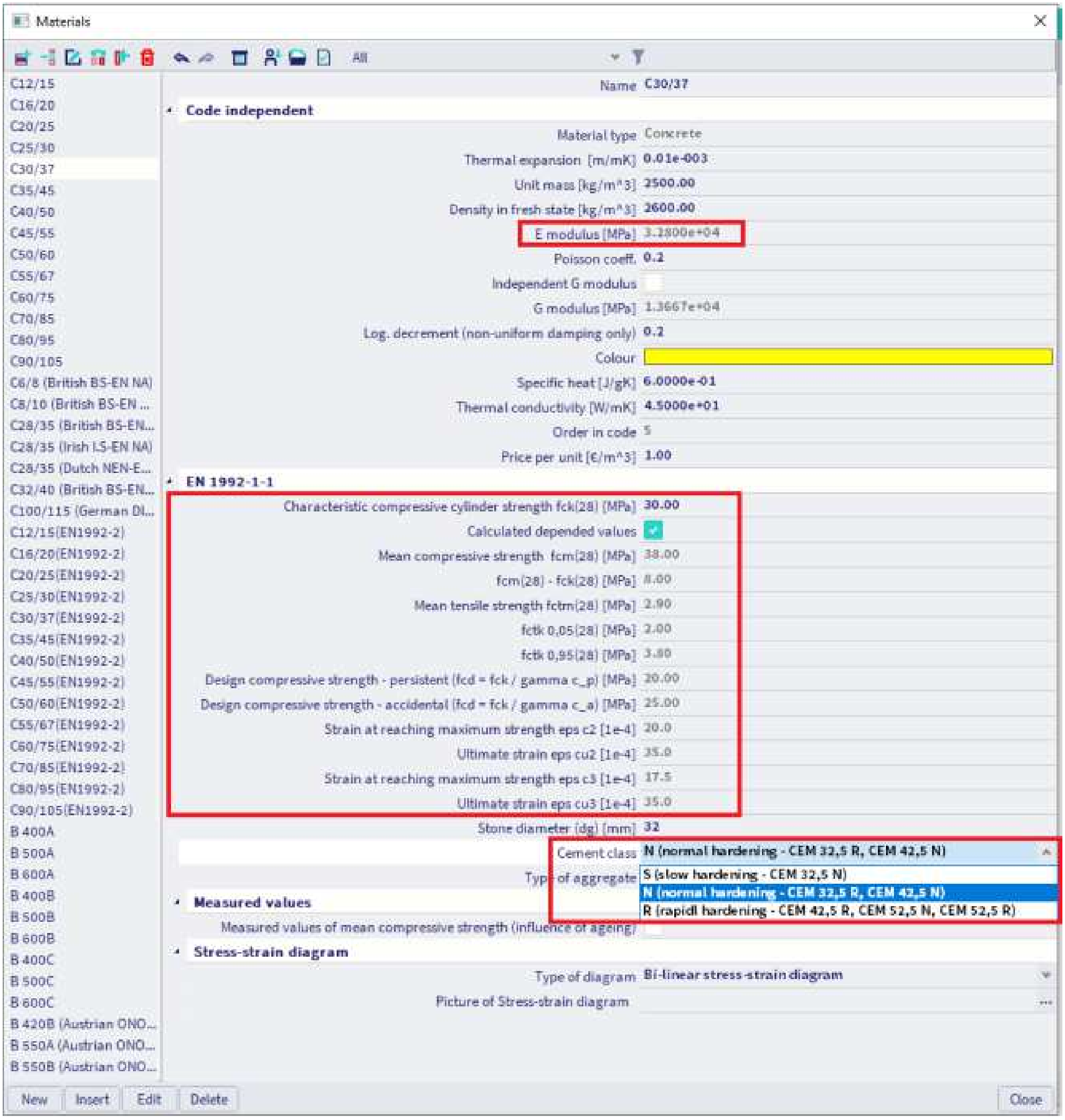
$$f_{cm}(t) = \beta_{cc}(t)\, f_{cm} \tag{3.1}$$

$$\text{with } \beta_{cc}(t) = e^{\left\{s\left[1-\left(\frac{28}{t}\right)^{\frac{1}{2}}\right]\right\}} \tag{3.2}$$

where:

- $f_{cm}(t)$ is the mean concrete compressive strength at an age of t days
- f_{cm} is the mean compressive strength at 28 days according to Table 3.1
- $\beta_{cc}(t)$ is a coefficient which depends on the age of the concrete t
- t is the age of the concrete in days
- s is a coefficient which depends on the type of cement:
  = 0,20 for cement of strength Classes CEM 42,5 R, CEM 52,5 N and CEM 52,5 R (Class R)
  = 0,25 for cement of strength Classes CEM 32,5 R, CEM 42,5 N (Class N)
  = 0,38 for cement of strength Classes CEM 32,5 N (Class S)

The type of cement can be chosen in the material library:



The tensile strength refers to the highest stress reached under concentric tensile loading.

The characteristic strengths for f_{ck} and the corresponding mechanical characteristics necessary for design, are given in Table 3.1:

Table 3.1 Strength and deformation characteristics for concrete

	Strength classes for concrete														Analytical relation / Explanation
f_{ck} (MPa)	12	16	20	25	30	35	40	45	50	55	60	70	80	90	
$f_{ck,cube}$ (MPa)	15	20	25	30	37	45	50	55	60	67	75	85	95	105	
f_{cm} (MPa)	20	24	28	33	38	43	48	53	58	63	68	78	88	98	$f_{cm} = f_{ck}+8$(MPa)
f_{ctm} (MPa)	1,6	1,9	2,2	2,6	2,9	3,2	3,5	3,8	4,1	4,2	4,4	4,6	4,8	5,0	$f_{ctm}=0{,}30\times f_{ck}^{(2/3)} \leq C50/60$ $f_{ctm}=2{,}12\cdot\ln(1+(f_{cm}/10)) > C50/60$
$f_{ctk,0,05}$ (MPa)	1,1	1,3	1,5	1,8	2,0	2,2	2,5	2,7	2,9	3,0	3,1	3,2	3,4	3,5	$f_{ctk,0,05} = 0{,}7\times f_{ctm}$ 5% fractile
$f_{ctk,0,95}$ (MPa)	2,0	2,5	2,9	3,3	3,8	4,2	4,6	4,9	5,3	5,5	5,7	6,0	6,3	6,6	$f_{ctk,0,95} = 1{,}3\times f_{ctm}$ 95% fractile
E_{cm} (GPa)	27	29	30	31	33	34	35	36	37	38	39	41	42	44	$E_{cm} = 22[(f_{cm})/10]^{0,3}$ (f_{cm} in MPa)
ε_{c1} (‰)	1,8	1,9	2,0	2,1	2,2	2,25	2,3	2,4	2,45	2,5	2,6	2,7	2,8	2,8	see Figure 3.2 $\varepsilon_{c1}(‰) = 0{,}7\, f_{cm}^{0,31} < 2{,}8$
ε_{cu1} (‰)	3,5									3,2	3,0	2,8	2,8	2,8	see Figure 3.2 for $f_{ck} \geq 50$ Mpa $\varepsilon_{cu1}(‰)=2{,}8+27[(98-f_{cm})/100]^4$
ε_{c2} (‰)	2,0									2,2	2,3	2,4	2,5	2,6	see Figure 3.3 for $f_{ck} \geq 50$ Mpa $\varepsilon_{c2}(‰)=2{,}0+0{,}085(f_{ck}-50)^{0,53}$
ε_{cu2} (‰)	3,5									3,1	2,9	2,7	2,6	2,6	see Figure 3.3 for $f_{ck} \geq 50$ Mpa $\varepsilon_{cu2}(‰)=2{,}6+35[(90-f_{ck})/100]^4$
n	2,0									1,75	1,6	1,45	1,4	1,4	for $f_{ck} \geq 50$ Mpa $n=1{,}4+23{,}4[(90-f_{ck})/100]^4$
ε_{c3} (‰)	1,75									1,8	1,9	2,0	2,2	2,3	see Figure 3.4 for $f_{ck} \geq 50$ Mpa $\varepsilon_{c3}(‰)=1{,}75+0{,}55[(f_{ck}-50)/40]$
ε_{cu3} (‰)	3,5									3,1	2,9	2,7	2,6	2,6	see Figure 3.4 for $f_{ck} \geq 50$ Mpa $\varepsilon_{cu3}(‰)=2{,}6+35[(90-f_{ck})/100]^4$

1.2.2. Design compressive and tensile strengths (art 3.1.6)

The value of the design compressive strength is defined as

$$f_{cd} = \alpha_{cc} \, f_{ck} / \gamma_C \tag{3.15}$$

where:

- γ_C is the partial safety factor for concrete.
- α_{cc} is the coefficient taking account of long term effects on the compressive strength and of unfavourable effects resulting from the way the load is applied.

The value of α_{cc} should lie between 0,8 and 1,0. The recommended value is 1,0.

Remark: the Belgian National Annex recommends the use of the value 0,85.

The value of the design tensile strength, f_{ctd}, is defined as

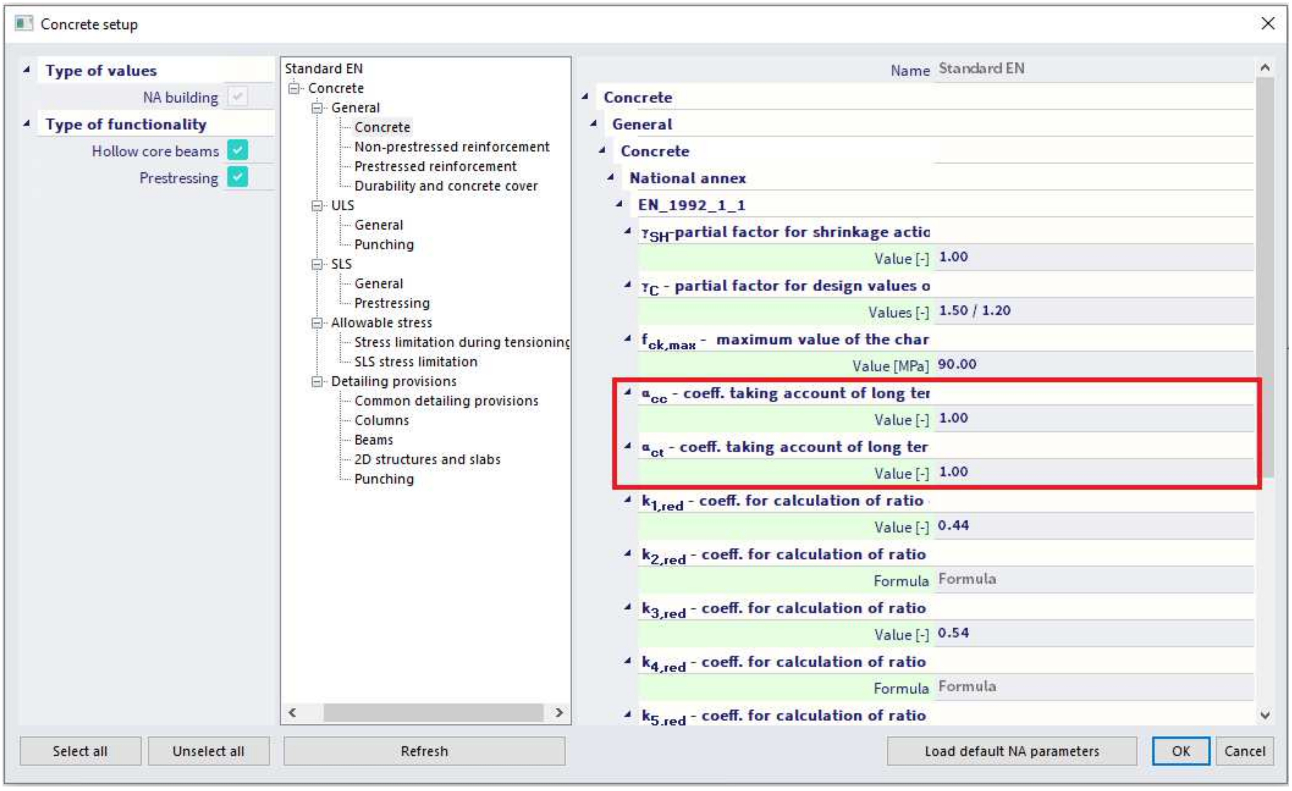
$$f_{ctd} = \alpha_{ct} \, f_{ctk,0,05} / \gamma_C \tag{3.16}$$

where:

- γ_C is the partial safety factor for concrete.
- α_{ct} is a coefficient taking account of long term effects on the tensile strength and of unfavourable effects, resulting from the way the load is applied.

The recommended value of α_{ct} is 1,0.

The values of the coefficients taking account of long term effects can be found in the Concrete setup of the National Annex:



If the concrete strength is determined at an age t > 28 days the values α_{cc} and α_{ct} should be reduced by a factor k_t.

The recommended value of k_t is 0,85.

1.2.3. Elastic deformation (art 3.1.3)

The elastic deformations of concrete largely depend on its composition (especially the aggregates). The values given in this Standard should be regarded as indicative for general applications. However, they should be specifically assessed if the structure is likely to be sensitive to deviations from these general values.

The modulus of elasticity of a concrete is controlled by the moduli of elasticity of its components. Approximate values for the modulus of elasticity E_{cm}, secant value between $\sigma_c = 0$ and $0{,}4f_{cm}$, for concretes with quartzite aggregates, are given in Table 3.1.

For limestone and sandstone aggregates the value should be reduced by 10% and 30% respectively. For basalt aggregates the value should be increased by 20%.

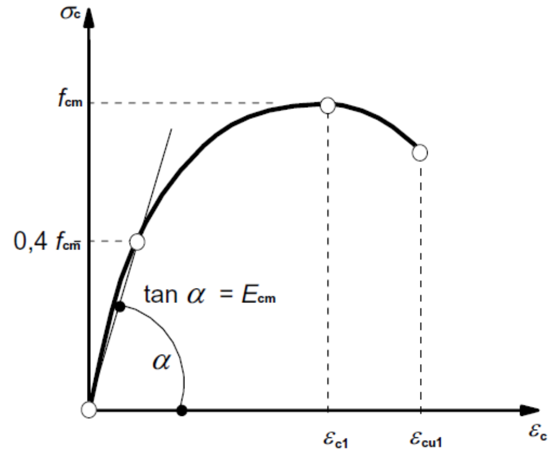


Figure 3.2: Schematic representation of the stress-strain relation for structural analysis (the use $0{,}4f_{cm}$ for the definition of E_{cm} is approximate).

Variation of the modulus of elasticity with time can be estimated by:

$$E_{cm}(t) = (f_{cm}(t) / f_{cm})^{0,3}\, E_{cm} \tag{3.5}$$

where $E_{cm}(t)$ and $f_{cm}(t)$ are the values at an age of *t* days and E_{cm} and f_{cm} are the values determined at an age of 28 days. The relation between $f_{cm}(t)$ and f_{cm} follows from Expression (3.1).

Poisson's ratio may be taken equal to 0,2 for uncracked concrete and 0 for cracked concrete.

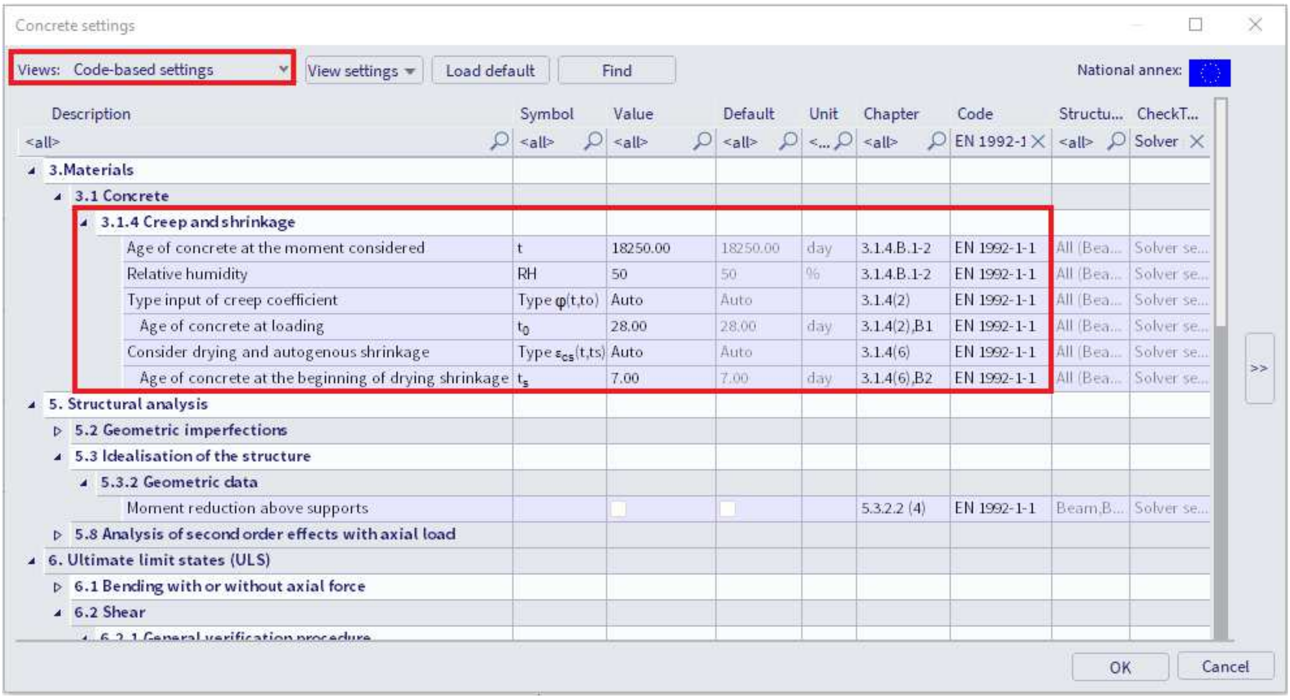
Another option to take into account cracked concrete, is to deactivate the option 'Calculate dependent values'. This allows you to set any user value for the E modulus.

1.2.4. Creep and shrinkage (art 3.1.4)

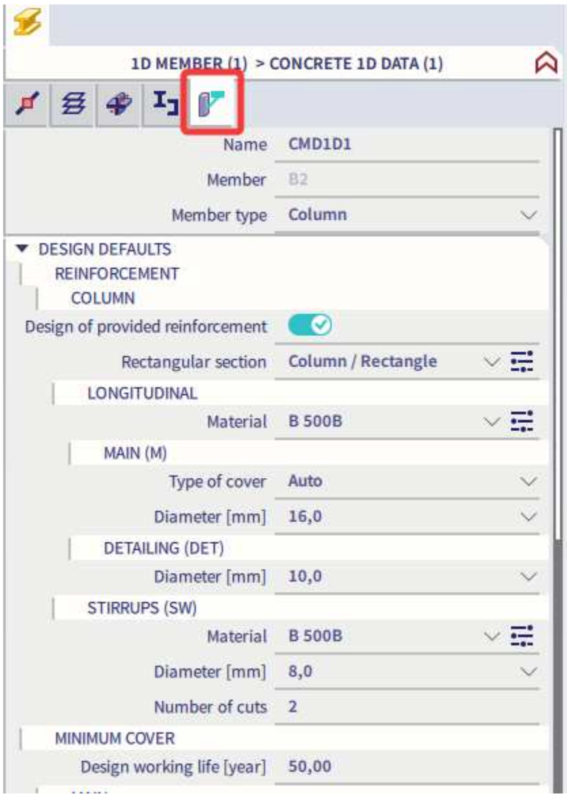
Creep and shrinkage of the concrete depend on the ambient humidity, the dimensions of the element and the composition of the concrete. Creep is also influenced by the maturity of the concrete when the load is first applied and depends on the duration and magnitude of the loading.

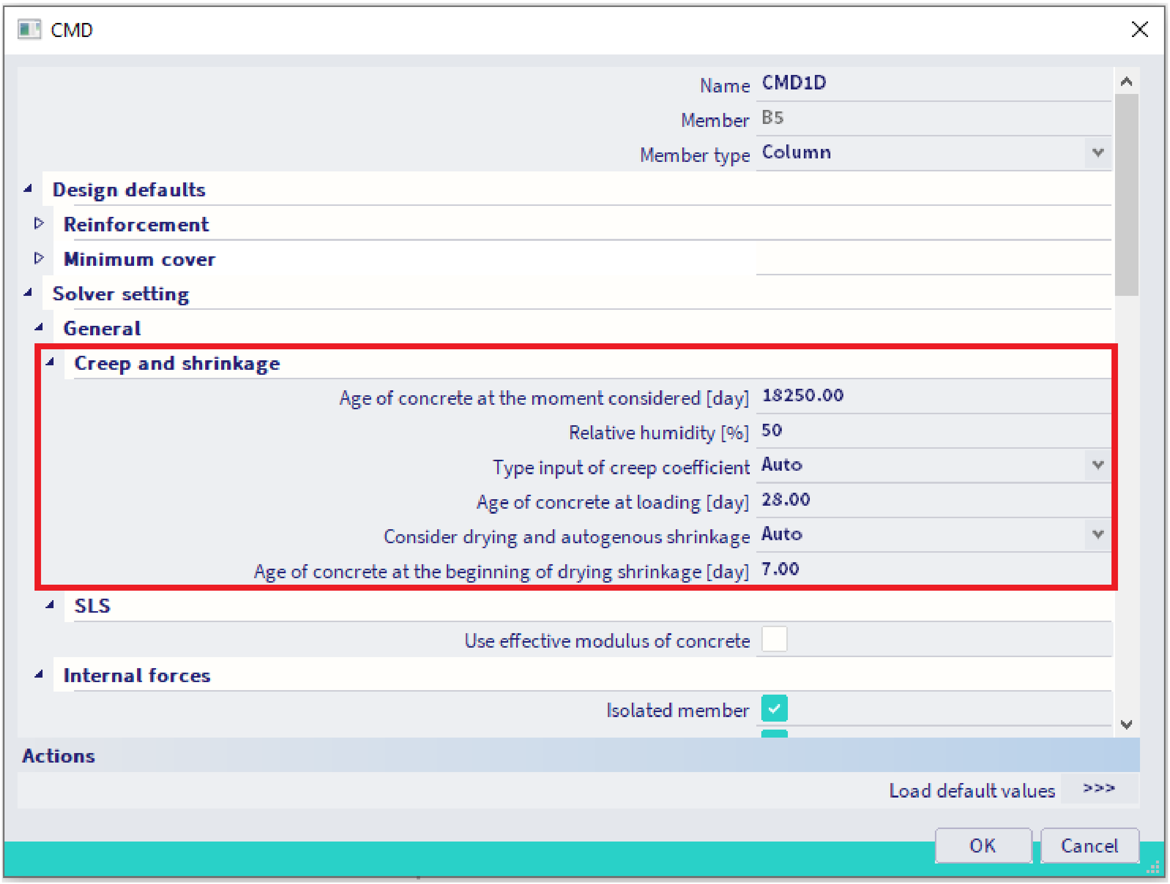
The value of the creep coefficient can be set in the concrete settings by using the “Code-based settings” view or in the 1D member data. If the type input of the creep coefficient is **“Auto”**, the creep coefficient can be calculated automatically by inputting the age of concrete and the relative humidity (see annex B.1. in EN 1992-1-1).

If the type input of the creep coefficient is **“User value”**, you can directly input the creep coefficient.



Note that the concrete member data is automatically added on each 1D and 2D element. It can be accessed by selecting the entity and going to the extra options below:





(1) The creep coefficient $\varphi(t,t_0)$ may be calculated from:

$$\varphi(t,t_0) = \varphi_0 \cdot \beta_c(t,t_0) \qquad \text{(B.1)}$$

where:

φ_0 is the notional creep coefficient and may be estimated from:

$$\varphi_0 = \varphi_{RH} \cdot \beta(f_{cm}) \cdot \beta(t_0) \qquad \text{(B.2)}$$

φ_{RH} is a factor to allow for the effect of relative humidity on the notional creep coefficient:

$$\varphi_{RH} = 1 + \frac{1 - RH/100}{0{,}1 \cdot \sqrt[3]{h_0}} \qquad \text{for } f_{cm} \le 35 \text{ MPa} \qquad \text{(B.3a)}$$

$$\varphi_{RH} = \left[1 + \frac{1 - RH/100}{0{,}1 \cdot \sqrt[3]{h_0}} \cdot \alpha_1\right] \cdot \alpha_2 \qquad \text{for } f_{cm} > 35 \text{ MPa} \qquad \text{(B.3b)}$$

RH is the relative humidity of the ambient environment in %

$\beta(f_{cm})$ is a factor to allow for the effect of concrete strength on the notional creep coefficient:

$$\beta(f_{cm}) = \frac{16{,}8}{\sqrt{f_{cm}}} \qquad \text{(B.4)}$$

f_{cm} is the mean compressive strength of concrete in MPa at the age of 28 days

$\beta(t_0)$ is a factor to allow for the effect of concrete age at loading on the notional creep coefficient:

$$\beta(t_0) = \frac{1}{(0{,}1 + t_0^{0{,}20})} \qquad \text{(B.5)}$$

h_0 is the notional size of the member in mm where:

$$h_0 = \frac{2A_c}{u} \qquad \text{(B.6)}$$

A_c is the cross-sectional area

u is the perimeter of the member in contact with the atmosphere

$\beta_c(t,t_0)$ is a coefficient to describe the development of creep with time after loading, and may be estimated using the following Expression:

$$\beta_c(t,t_0) = \left[\frac{(t - t_0)}{(\beta_H + t - t_0)}\right]^{0{,}3} \qquad \text{(B.7)}$$

t is the age of concrete in days at the moment considered

t_0 is the age of concrete at loading in days

$t - t_0$ is the non-adjusted duration of loading in days

β_H is a coefficient depending on the relative humidity (RH in %) and the notional member size (h_0 in mm). It may be estimated from:

$$\beta_H = 1{,}5\,[1 + (0{,}012\,RH)^{18}]\,h_0 + 250 \le 1500 \qquad \text{for } f_{cm} \le 35 \qquad \text{(B.8a)}$$

$$\beta_H = 1{,}5\,[1 + (0{,}012\,RH)^{18}]\,h_0 + 250\,\alpha_3 \le 1500\,\alpha_3 \qquad \text{for } f_{cm} \ge 35 \qquad \text{(B.8b)}$$

$\alpha_{1/2/3}$ are coefficients to consider the influence of the concrete strength:

$$\alpha_1 = \left[\frac{35}{f_{cm}}\right]^{0{,}7} \quad \alpha_2 = \left[\frac{35}{f_{cm}}\right]^{0{,}2} \quad \alpha_3 = \left[\frac{35}{f_{cm}}\right]^{0{,}5} \qquad \text{(B.8c)}$$

Where great accuracy is not required, a value found from a figure (Figure 3.1) may be considered as the creep coefficient, provided that the concrete is not subjected to a compressive stress greater than 0,45 f_{ck} (t_0) at an age t_0, the age of concrete at the time of loading.

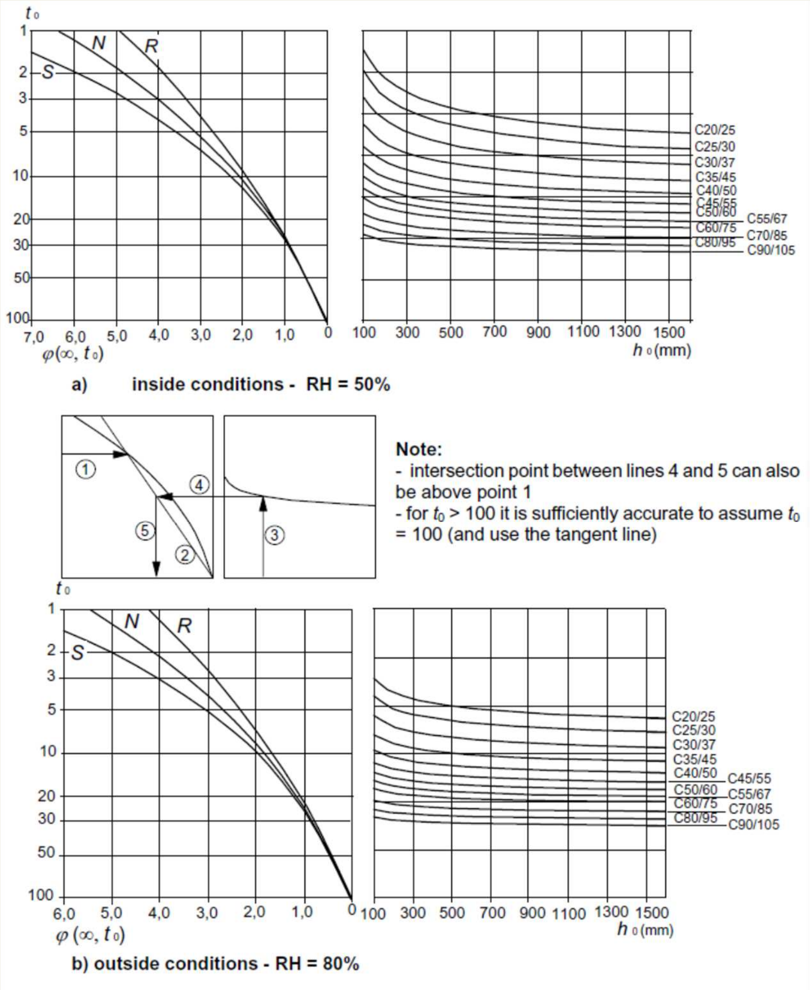


Figure 3.1: Method for determining the creep coefficient $\varphi(\infty, t_0)$ for concrete under normal environmental conditions

1.2.5. Stress-strain relations for the design of cross-sections (art 3.1.7)

For the design of cross-sections, the following stress-strain relationship may be used:

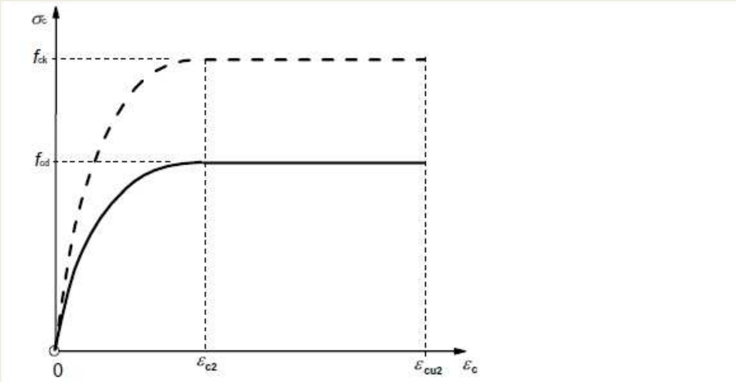


Figure 3.3: Parabola-rectangle diagram for concrete under compression.

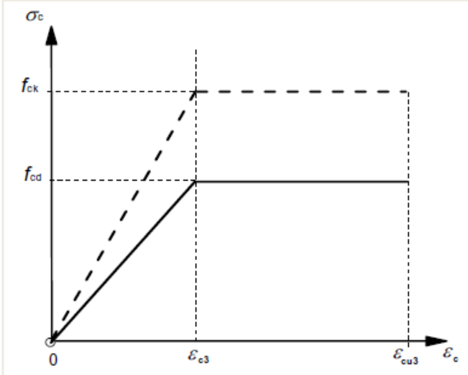


Figure 3.4: Bi-linear stress-strain relation.

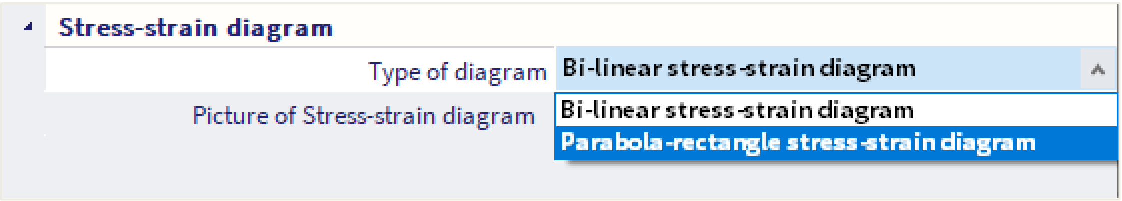
ε_{c2} is the strain at reaching the maximum strength in the parabola-rectangle diagram

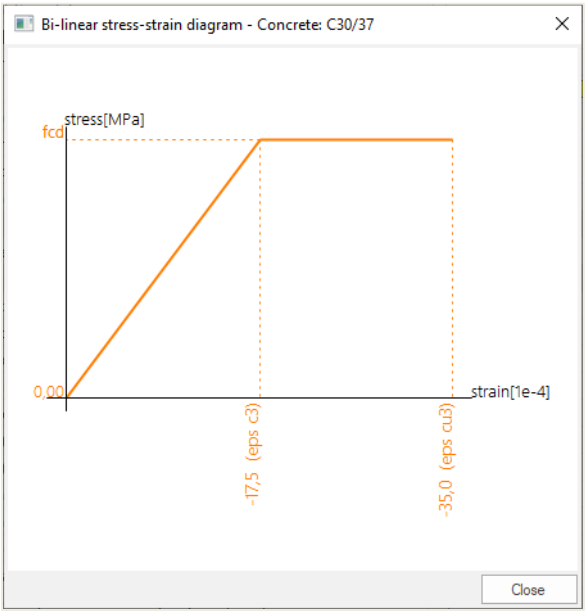
ε_{cu2} is the ultimate strain in the parabola-rectangle diagram

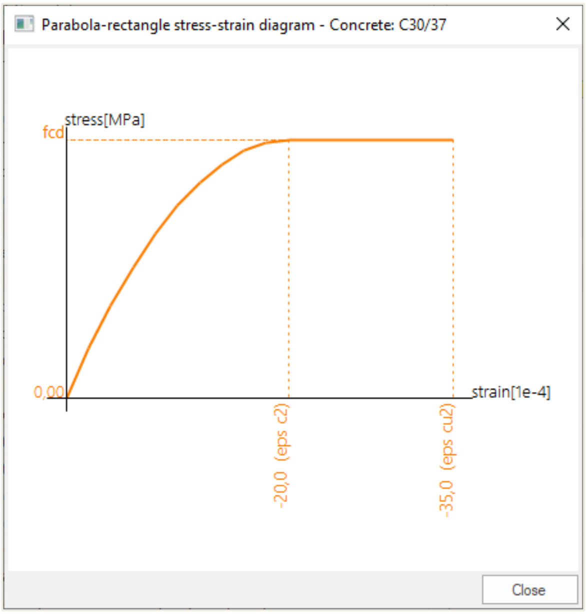
ε_{c3} is the strain at reaching the maximum strength in the bi-linear diagram

ε_{cu3} is the ultimate strain in the bi-linear diagram

You can choose in the material library which one of the diagrams should be used for the calculation:







1.3. Reinforcing Steel

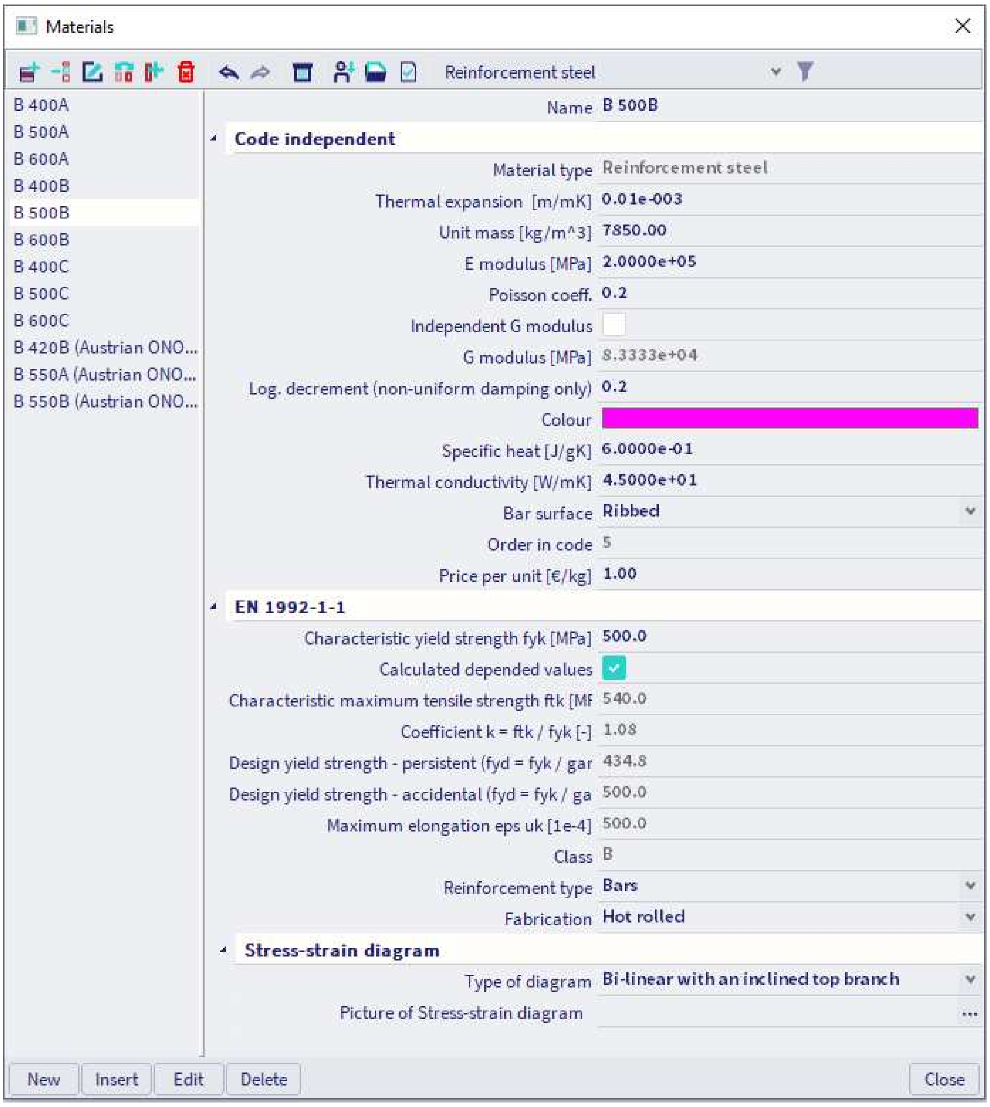
The following clauses give principles and rules for reinforcement which is in the form of bars, de-coiled rods, welded fabric and lattice girders. They do not apply to specially coated bars.

1.3.1. Properties (art 3.2.2)

The behaviour of reinforcing steel is specified by the following properties:

- yield strength (f_{yk} or $f_{0,2k}$)
- maximum actual yield strength ($f_{y,max}$)
- tensile strength (f_t)
- ductility (ε_{uk} and f_t/f_{yk})
- bendability
- bond characteristics (f_R)
- section sizes and tolerances
- fatigue strength
- weldability
- shear and weld strength for welded fabric and lattice girders

The steel properties can be found in the material library:



The mean value of density may be assumed to be 7850 kg/m^3.

The design value of the modulus of elasticity E_s may be assumed to be 200GPa.

This Eurocode applies to ribbed and weldable reinforcement, including fabric.

The application rules for design and detailing in this Eurocode are valid for a specified yield strength range, f_{yk} = 400 to 600 MPa.

Table C.1 gives the properties of reinforcement suitable for use with this Eurocode:

Table C.1: Properties of reinforcement

Product form	Bars and de-coiled rods			Wire Fabrics			Requirement or quantile value (%)
Class	A	B	C	A	B	C	-
Characteristic yield strength f_{yk} or $f_{0,2k}$ (MPa)	400 to 600						5,0
Minimum value of $k = (f_t/f_y)_k$	≥1,05	≥1,08	≥1,15 <1,35	≥1,05	≥1,08	≥1,15 <1,35	10,0
Characteristic strain at maximum force, ε_{uk} (%)	≥2,5	≥5,0	≥7,5	≥2,5	≥5,0	≥7,5	10,0
Bendability	Bend/Rebend test			-			
Shear strength	-			0,3 A f_{yk} (A is area of wire)			Minimum
Maximum deviation from nominal mass (individual bar or wire) (%) — Nominal bar size (mm): ≤ 8 / > 8	± 6,0 ± 4,5						5,0

1.3.2. Design assumptions (art 3.2.7)

For normal design, either of the following assumptions may be made:

B1) an inclined top branch with a strain limit of ε_{ud} and a maximum stress of kf_{yk} / γ_s at ε_{uk}, where $k = (f_t/f_y)_k$.

B2) a horizontal top branch without the need to check the strain limit.

The recommended value of ε_{ud} is 0,9 ε_{uk}. The value of $(f_t/f_y)_k$ is given in Table C.1.

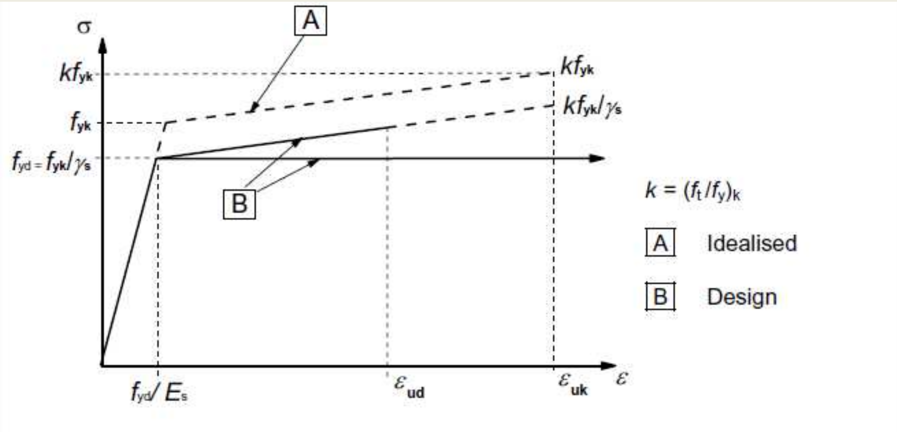
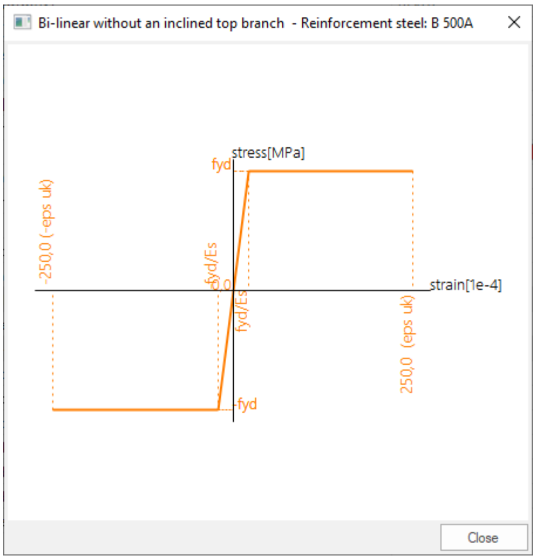
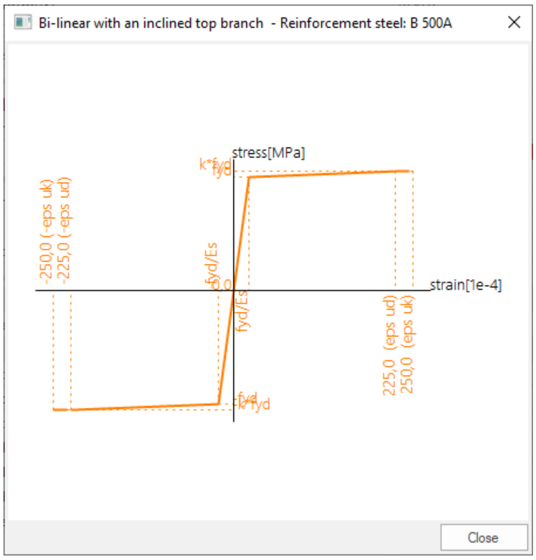


Figure 3.8: Idealised and design stress-strain diagrams for reinforcing steel (for tension and compression)

In the material library you can choose between the two assumptions:





1.4. Durability and cover to reinforcement

1.4.1. Environmental conditions (art 4.2)

Exposure conditions are chemical and physical conditions to which the structure is exposed in addition to the mechanical actions.

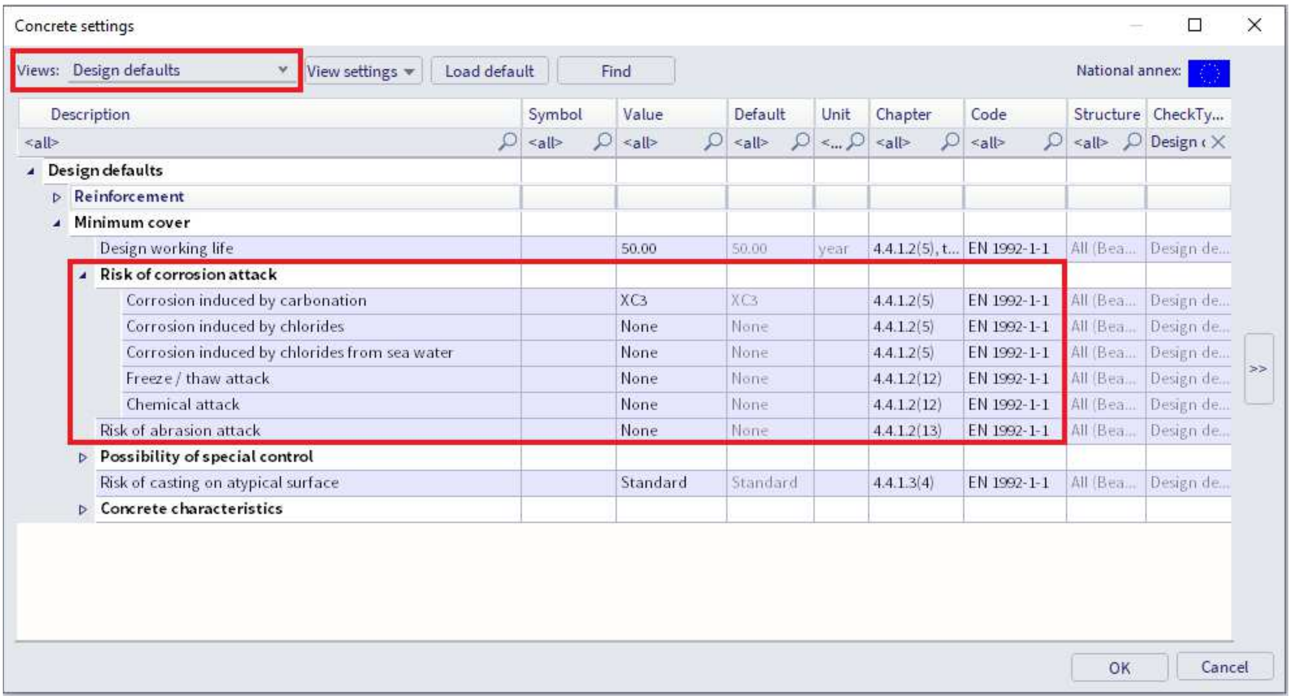
Environmental conditions are classified according to Table 4.1:

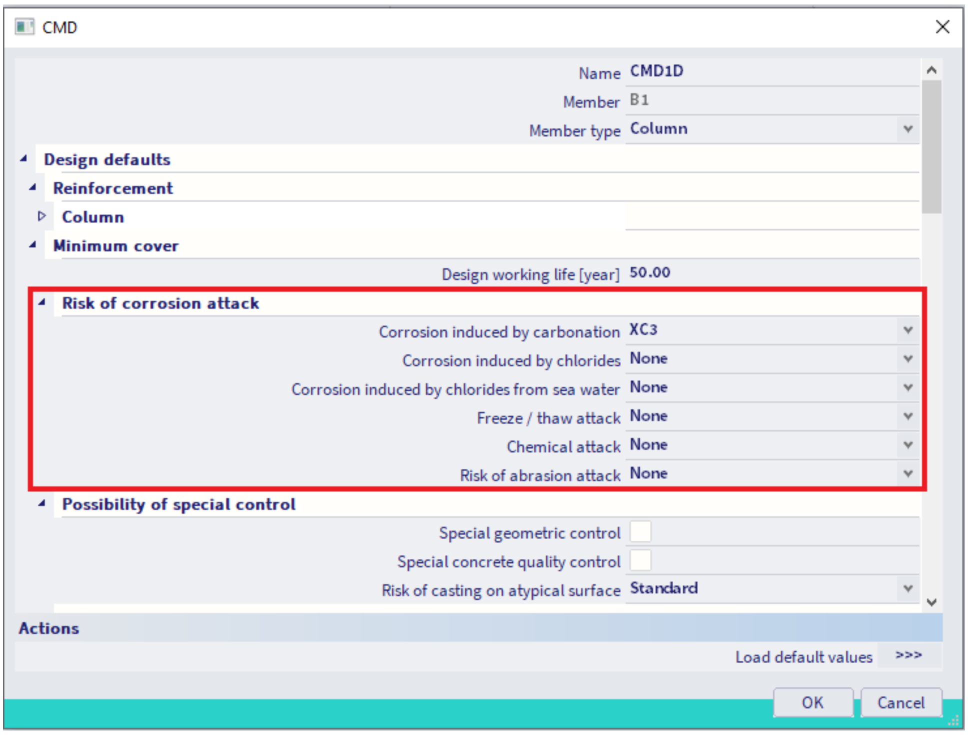
Table 4.1: Exposure classes related to environmental conditions in accordance with EN 206-1

Class designation	Description of the environment	Informative examples where exposure classes may occur
1 No risk of corrosion or attack		
X0	For concrete without reinforcement or embedded metal: all exposures except where there is freeze/thaw, abrasion or chemical attack For concrete with reinforcement or embedded metal: very dry	Concrete inside buildings with very low air humidity
2 Corrosion induced by carbonation		
XC1	Dry or permanently wet	Concrete inside buildings with low air humidity Concrete permanently submerged in water
XC2	Wet, rarely dry	Concrete surfaces subject to long-term water contact Many foundations
XC3	Moderate humidity	Concrete inside buildings with moderate or high air humidity External concrete sheltered from rain
XC4	Cyclic wet and dry	Concrete surfaces subject to water contact, not within exposure class XC2

3 Corrosion induced by chlorides		
XD1	Moderate humidity	Concrete surfaces exposed to airborne chlorides
XD2	Wet, rarely dry	Swimming pools Concrete components exposed to industrial waters containing chlorides
XD3	Cyclic wet and dry	Parts of bridges exposed to spray containing chlorides Pavements Car park slabs
4 Corrosion induced by chlorides from sea water		
XS1	Exposed to airborne salt but not in direct contact with sea water	Structures near to or on the coast
XS2	Permanently submerged	Parts of marine structures
XS3	Tidal, splash and spray zones	Parts of marine structures
5. Freeze/Thaw Attack		
XF1	Moderate water saturation, without de-icing agent	Vertical concrete surfaces exposed to rain and freezing
XF2	Moderate water saturation, with de-icing agent	Vertical concrete surfaces of road structures exposed to freezing and airborne de-icing agents
XF3	High water saturation, without de-icing agents	Horizontal concrete surfaces exposed to rain and freezing
XF4	High water saturation with de-icing agents or sea water	Road and bridge decks exposed to de-icing agents Concrete surfaces exposed to direct spray containing de-icing agents and freezing Splash zone of marine structures exposed to freezing
6. Chemical attack		
XA1	Slightly aggressive chemical environment according to EN 206-1, Table 2	Natural soils and ground water
XA2	Moderately aggressive chemical environment according to EN 206-1, Table 2	Natural soils and ground water
XA3	Highly aggressive chemical environment according to EN 206-1, Table 2	Natural soils and ground water

In the Concrete settings, in the "Design defaults" view, you can choose the desired exposure class. All items with a blue background colour can be overwritten in the 1D member data.





1.4.2. Methods of verification (art 4.4)

Concrete Cover: art 4.4.1

General (art 4.4.1.1)

The concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and stirrups and surface reinforcement where relevant) and the nearest concrete surface.

The nominal cover shall be specified on the drawings. It is defined as a minimum cover, c_{min}, plus an allowance in design for deviation, Δc_{dev} :

$$c_{nom} = c_{min} + \Delta c_{dev}$$

Minimum cover, c_{min} (art 4.4.1.2)

Minimum concrete cover, c_{min}, shall be provided in order to ensure:

- the safe transmission of bond forces
- the protection of the steel against corrosion (durability)
- an adequate fire resistance

The greater value for c_{min} satisfying the requirements for both bond and environmental conditions shall be used:

$$c_{min} = \max \{c_{min,b};\ c_{min,dur} + \Delta c_{dur,\gamma} - \Delta c_{dur,st} - \Delta c_{dur,add};\ 10\text{ mm}\} \qquad (4.2)$$

where:

- $c_{min,b}$ minimum cover due to bond requirement
- $c_{min,dur}$ minimum cover due to environmental conditions
- $\Delta c_{dur,\gamma}$ additive safety element
- $\Delta c_{dur,st}$ reduction of minimum cover for use of stainless steel
- $\Delta c_{dur,add}$ reduction of minimum cover for use of additional protection

The recommended value of $\Delta c_{dur,\gamma}$, $\Delta c_{dur,st}$ and $\Delta c_{dur,add}$, without further specification, is 0 mm.

- In order to transmit bond forces safely and to ensure adequate compaction of the concrete, the minimum cover should not be less than $c_{min,b}$ given in table 4.2.

Table 4.2: Minimum cover, $c_{min,b}$, requirements with regard to bond

Bond Requirement	
Arrangement of bars	Minimum cover $c_{min,b}$*
Separated	Diameter of bar
Bundled	Equivalent diameter (ϕ_n)(see 8.9.1)
*: If the nominal maximum aggregate size is greater than 32 mm, $c_{min,b}$ should be increased by 5 mm.	

- The minimum cover values for reinforcement and prestressing tendons in normal weight concrete taking account of the exposure classes and the structural classes is given by $c_{min,dur}$.

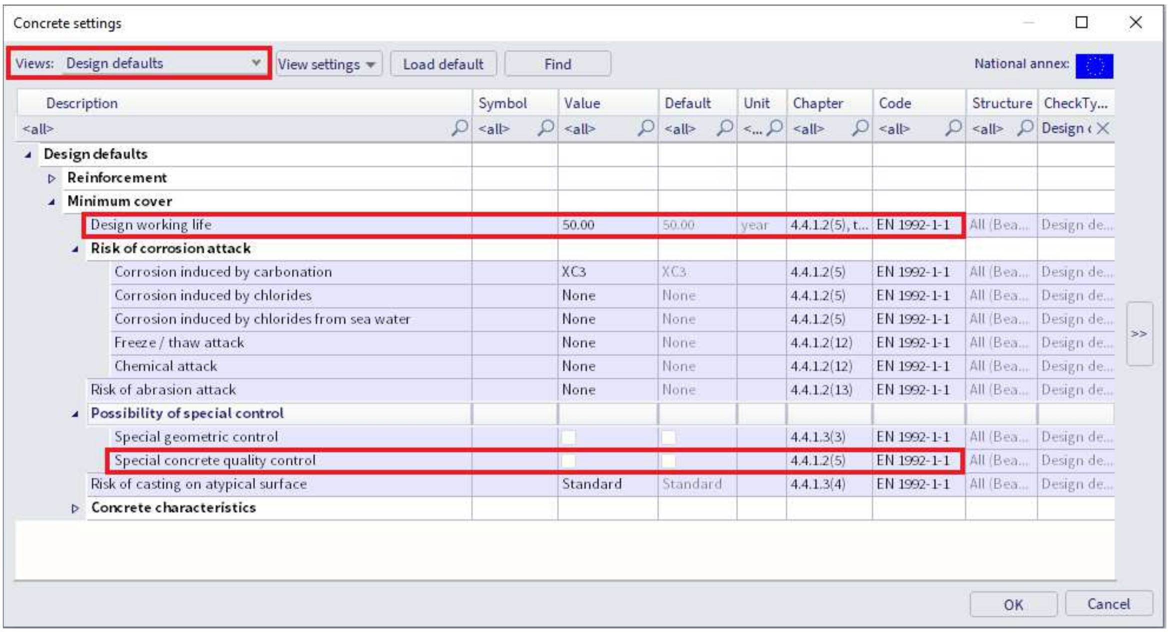
The recommended Structural Class (design working life of 50 years) is S4 for the indicative concrete strengths (given in Annex E of EN 1992-1-1). The recommended minimum Structural Class is S1.

The recommended modifications to the structural class is given in Table 4.3N:

Table 4.3N: Recommended structural classification

Structural Class							
Criterion	Exposure Class according to Table 4.1						
	X0	XC1	XC2 / XC3	XC4	XD1	XD2 / XS1	XD3 / XS2 / XS3
Design Working Life of 100 years	increase class by 2	increase class by 2	increase class by 2	increase class by 2	increase class by 2	increase class by 2	increase class by 2
Strength Class [1) 2)]	≥ C30/37 reduce class by 1	≥ C30/37 reduce class by 1	≥ C35/45 reduce class by 1	≥ C40/50 reduce class by 1	≥ C40/50 reduce class by 1	≥ C40/50 reduce class by 1	≥ C45/55 reduce class by 1
Member with slab geometry (position of reinforcement not affected by construction process)	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1
Special Quality Control of the concrete production ensured	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1	reduce class by 1

The design working life and the special quality control can be defined in the concrete settings or in the 1D member data:



The recommended values of $c_{min,dur}$ are given in Table 4.4N (reinforcing steel):

Table 4.4N: Values of minimum cover, $c_{min,dur}$, requirements with regard to durability for reinforcement steel in accordance with EN 10080.

Environmental Requirement for $c_{min,dur}$ (mm)							
Structural Class	Exposure Class according to Table 4.1						
	X0	XC1	XC2 / XC3	XC4	XD1 / XS1	XD2 / XS2	XD3 / XS3
S1	10	10	10	15	20	25	30
S2	10	10	15	20	25	30	35
S3	10	10	20	25	30	35	40
S4	10	15	25	30	35	40	45
S5	15	20	30	35	40	45	50
S6	20	25	35	40	45	50	55

- The concrete cover should be increased by the additive safety element $\Delta c_{dur,\gamma}$.

  Where stainless steel is used or where other special measures have been taken, the minimum cover may be reduced by $\Delta c_{dur,st}$. For such situations the effects on all relevant material properties should be considered, including bond.

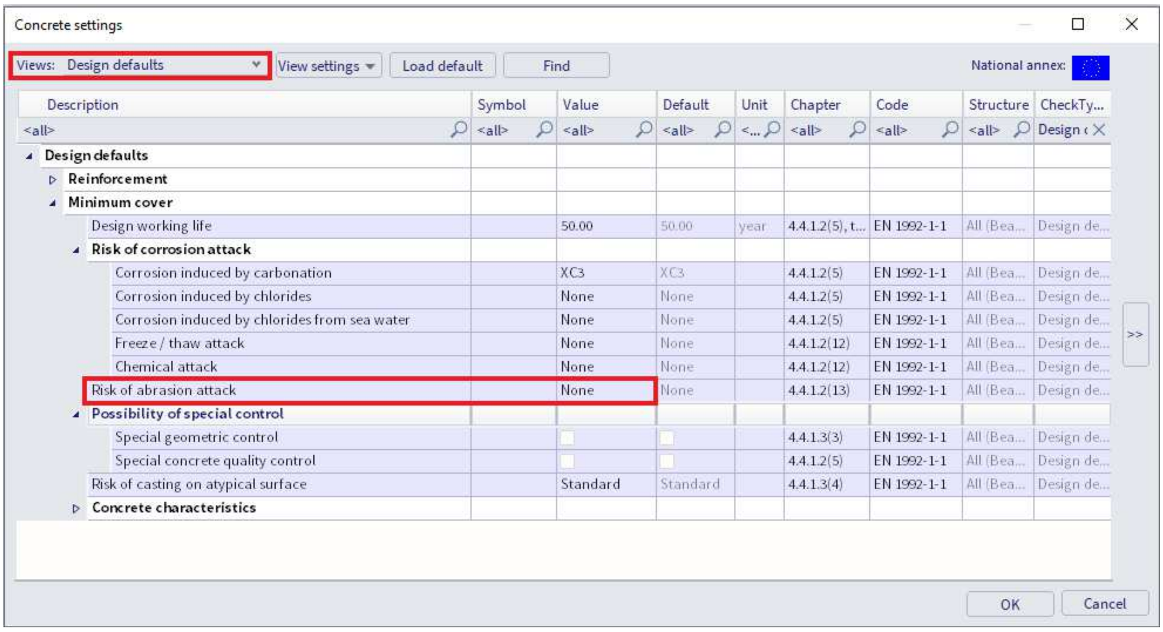
  For concrete with additional protection (e.g. coating) the minimum cover may be reduced by $\Delta c_{dur,add}$.

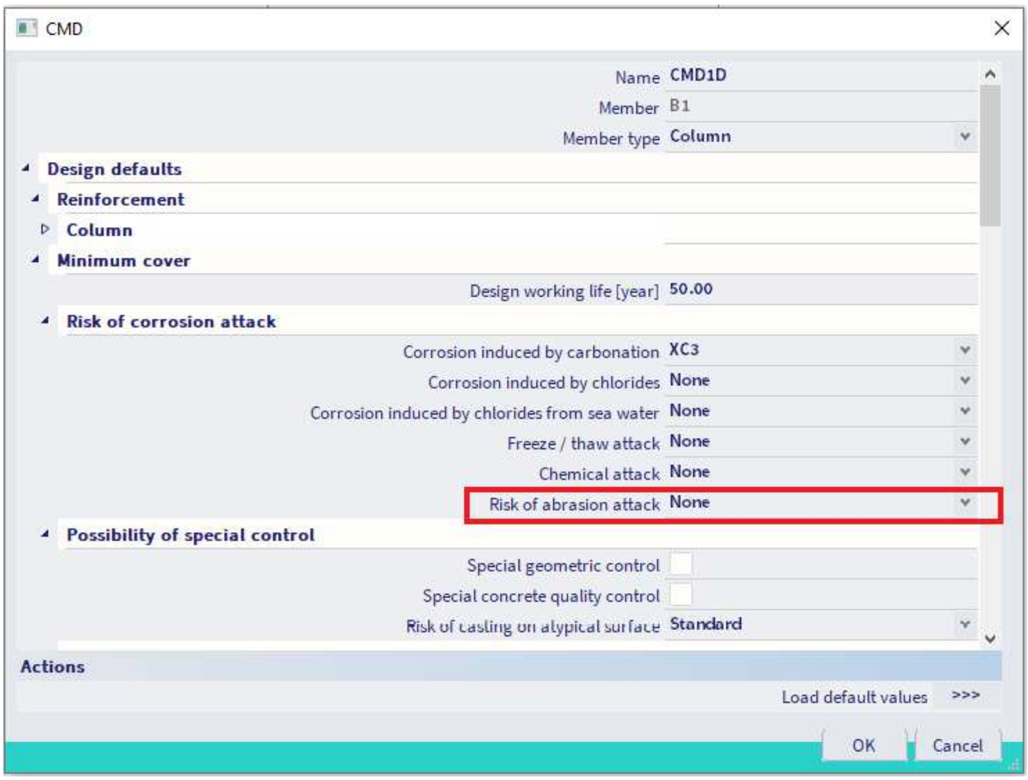
  For concrete abrasion special attention should be given on the aggregate. Optionally concrete abrasion may be allowed for by increasing the concrete cover (sacrificial layer). In that case, the minimum cover c_{min} should be increased by k_1 for Abrasion Class XM1, by k_2 for XM2 and by k_3 for XM3.

  Abrasion Class XM1 means a moderate abrasion like for members of industrial sites frequented by vehicles with air tyres. Abrasion Class XM2 means a heavy abrasion like for members of industrial sites frequented by fork lifts with air or solid rubber tyres. Abrasion Class XM3 means an extreme abrasion like for members industrial sites frequented by fork lifts with elastomer or steel tyres or track vehicles.

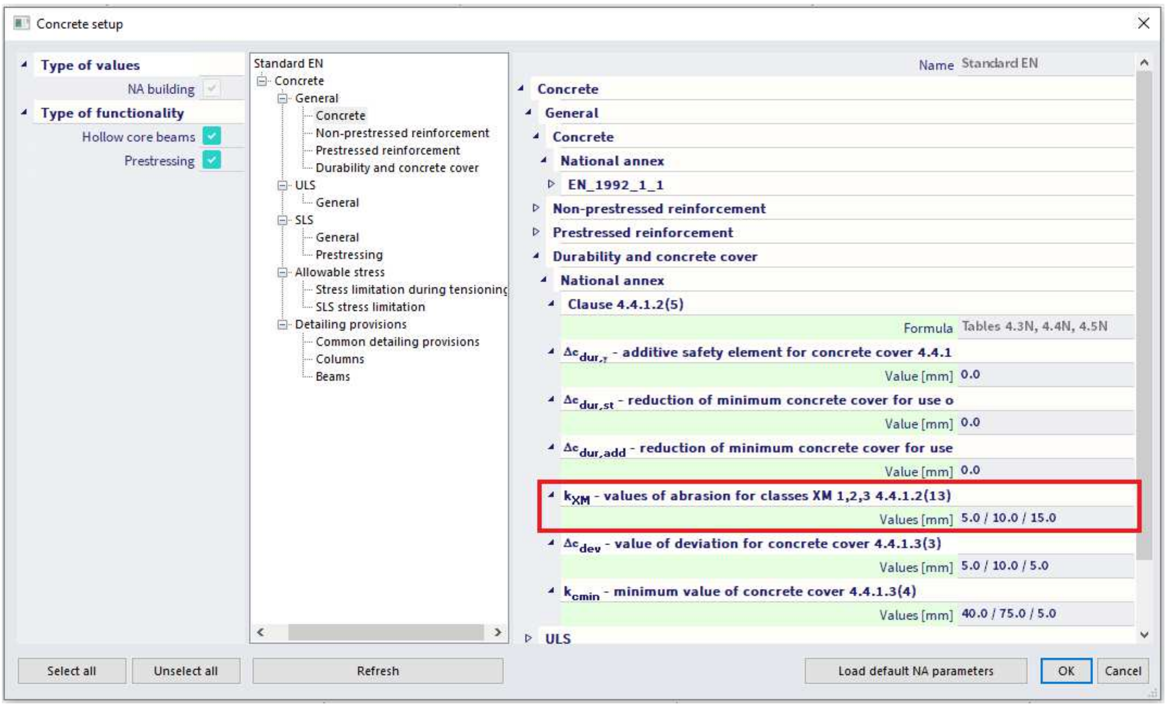
  The recommended values of k_1, k_2 and k_3 are respectively: 5 mm, 10 mm and 15 mm.

The abrasion class can be inputted in the concrete settings or the 1D member data:





The values of k_1, k_2 and k_3 are available in the National Annex:



Allowance in design for deviation (art 4.4.1.3)

To calculate the nominal cover, c_{nom}, an addition to the minimum cover shall be made in design to allow for the deviation (Δc_{dev}). The required minimum cover shall be increased by the absolute value of the accepted negative deviation.

The recommended value of Δc_{dev} is 10 mm.

In certain situations, the accepted deviation and hence allowance, Δc_{dev}, may be reduced.

The recommended values are:

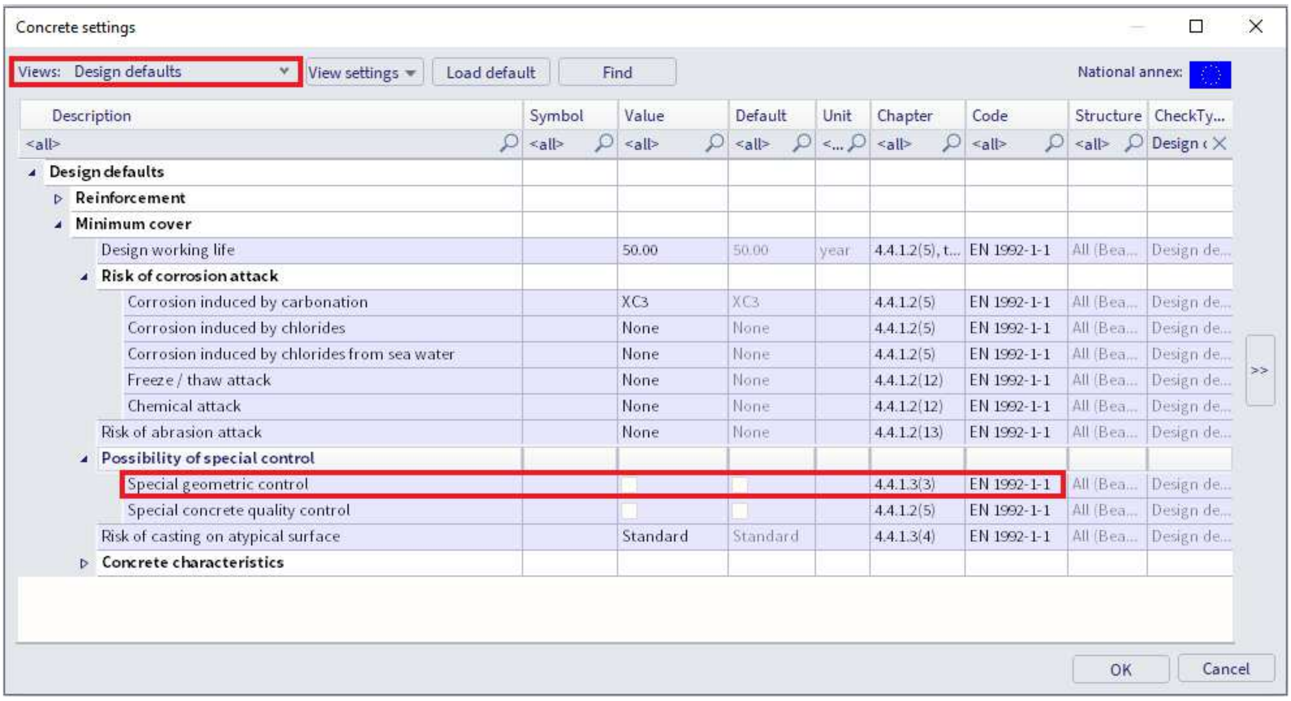
- *where fabrication is subjected to a quality assurance system, in which the monitoring includes measurements of the concrete cover, the allowance in design for deviation Δc_{dev} may be reduced:*

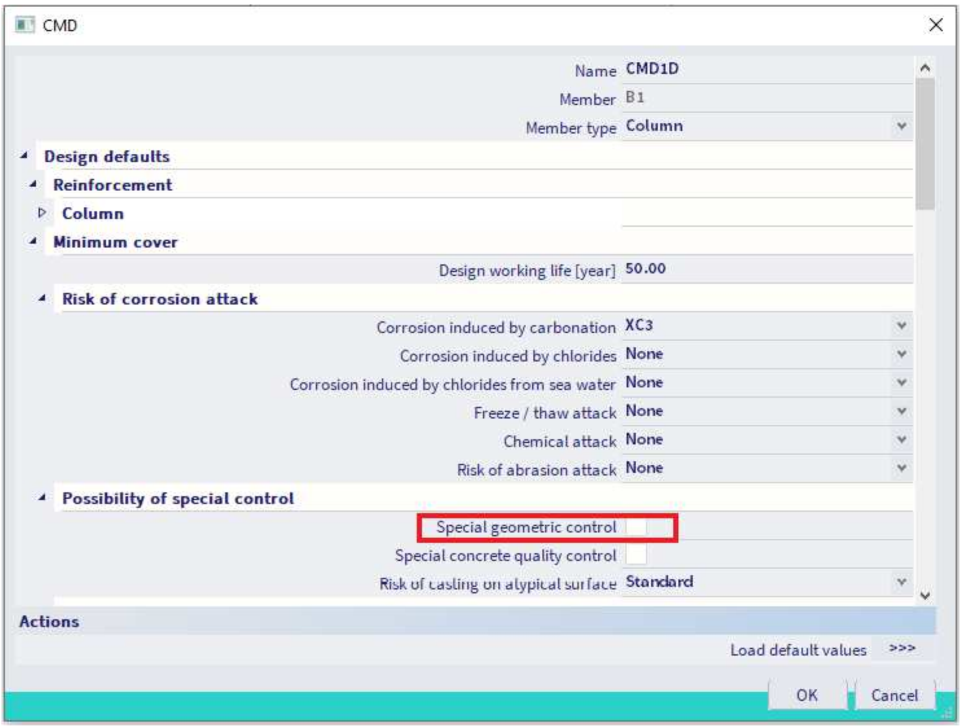
$$10 \text{ mm} \geq \Delta c_{dev} \geq 5 \text{ mm}$$

- *where it can be assured that a very accurate measurement device is used for monitoring and non-conforming members are rejected (e.g. precast elements), the allowance in design for deviation Δc_{dev} may be reduced:*

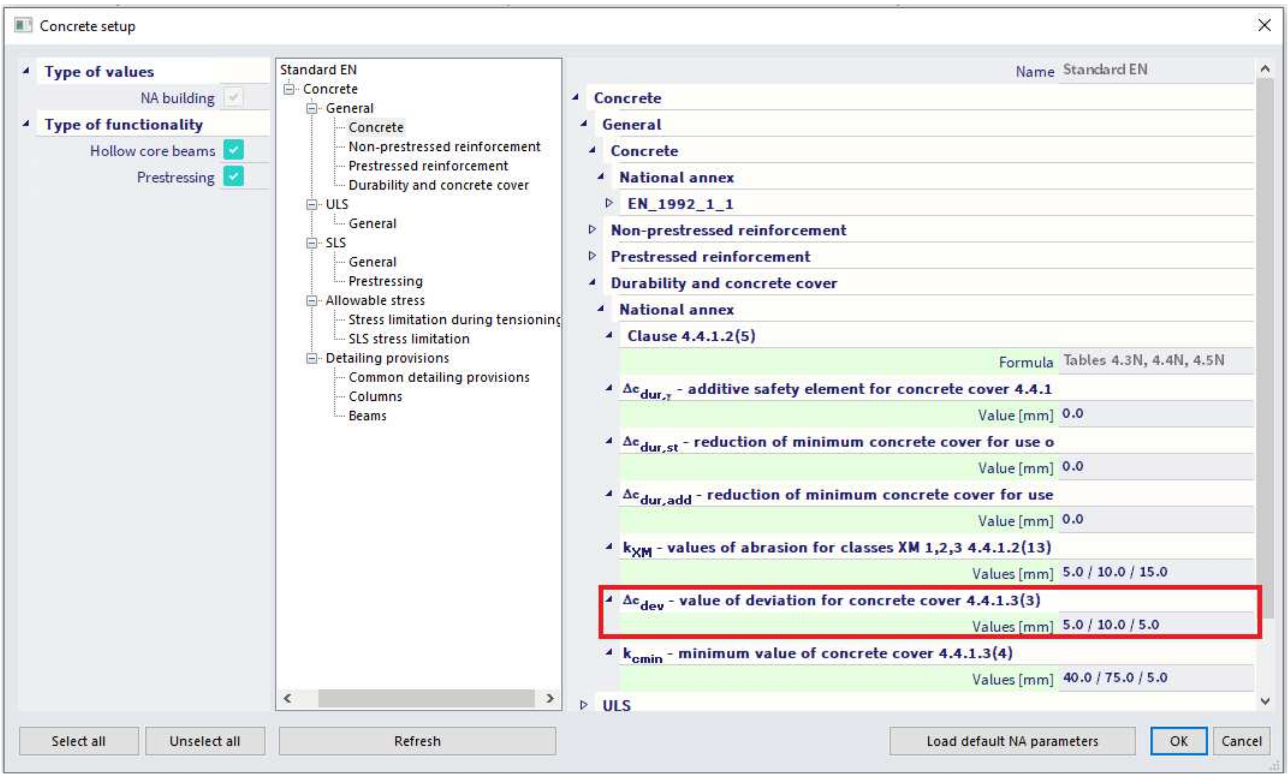
$$10 \text{ mm} \geq \Delta c_{dev} \geq 0 \text{ mm}$$

The special geometric control can be checked in the concrete settings or the 1D member data:





The values of Δc_{dev} can be found in the National Annex:



Chapter 2: Design and Check

2.1 Analysis models

2.1.1 Eurocode

Structural models for overall analysis (art 5.3.1)

The elements of a structure are classified, by consideration of their nature and function, as beams, columns, slabs, walls, plates, arches, shells etc. Rules are provided for the analysis of the commoner of these elements and of structures consisting of combinations of these elements.

For buildings the following provisions are applicable:

1) A beam is a member for which the span is not less than 3 times the overall section depth. Otherwise it should be considered as a deep beam.

2) A slab is a member for which the minimum panel dimension is not less than 5 times the overall slab thickness.

3) A slab subjected to dominantly uniformly distributed loads may be considered to be one way spanning if either:
   - it possesses two free (unsupported) and sensibly parallel edges.
   - it is the central part of a sensibly rectangular slab supported on four edges with a ratio of the longer to shorter span greater than 2.

4) Ribbed or waffle slabs need not be treated as discrete elements for the purposes of analysis, provided that the flange or structural topping and transverse ribs have sufficient torsional stiffness. This may be assumed provided that:
   - the rib spacing does not exceed 1500 mm
   - the depth of the rib below the flange does not exceed 4 times its width.
   - the depth of the flange is at least 1/10 of the clear distance between ribs or 50 mm, whichever is the greater.
   - transverse ribs are provided at a clear spacing not exceeding 10 times the overall depth of the slab.

The minimum flange thickness of 50 mm may be reduced to 40 mm where permanent blocks are incorporated between the ribs.

A column is a member for which the section depth does not exceed 4 times its width and the height is at least 3 times the section depth. Otherwise it should be considered as a wall.

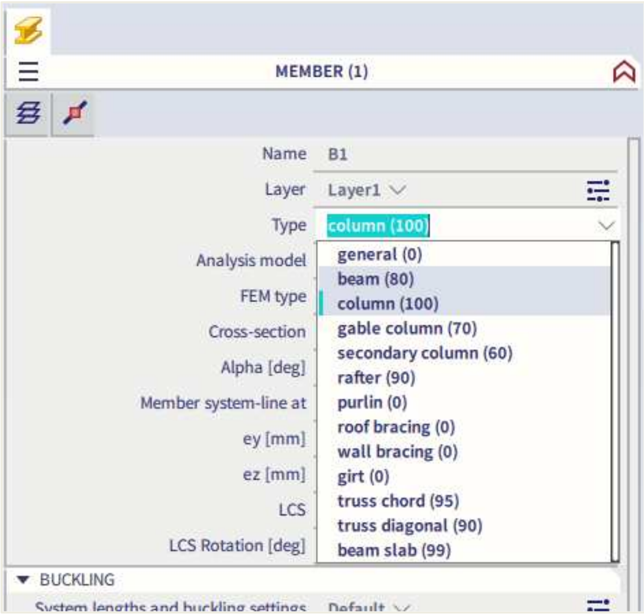
2.1.2 SCIA Engineer

ASSIGNMENT OF ANALYSIS MODEL

In SCIA Engineer, several types of analysis models are available. It is up to the user to decide which model should be used for which element.

For 1D members, there is the choice between Beam, Beam slab and Column calculation.

Each element has a property ‘Type’ assigned to it, to determine which type of calculation will be used:

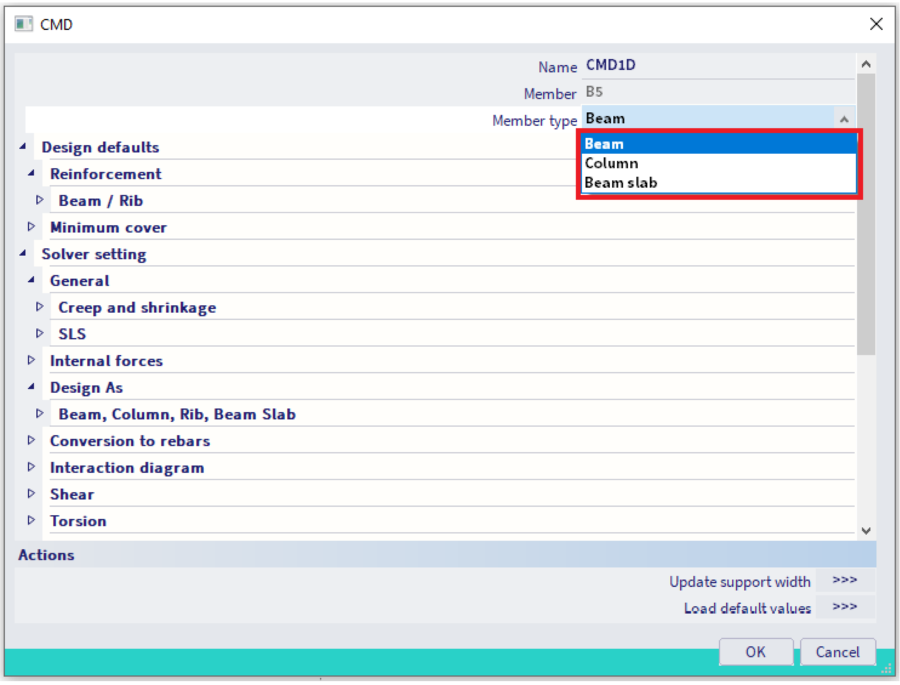


The Beam calculation is used for the Types ‘General’, ‘Beam, ‘Rafter’, ‘Purlin’, ‘Roof bracing’, ‘Wall bracing’, ‘Girt’, ‘Truss chord’ and ‘Truss diagonal’.

The Beam slab calculation is used only for the Type ‘Beam slab’. For this type, by default no shear reinforcement is added (unless necessary in case of a slab thickness of 200 mm or more, as defined in the Concrete Settings for slabs). As diameter for the longitudinal reinforcement, the default diameter for 2D structures – and not for beams! – is taken from the Concrete Settings.

The Column calculation is used for the Types ‘Column’, ‘Gable column’ and ‘Secondary column’.

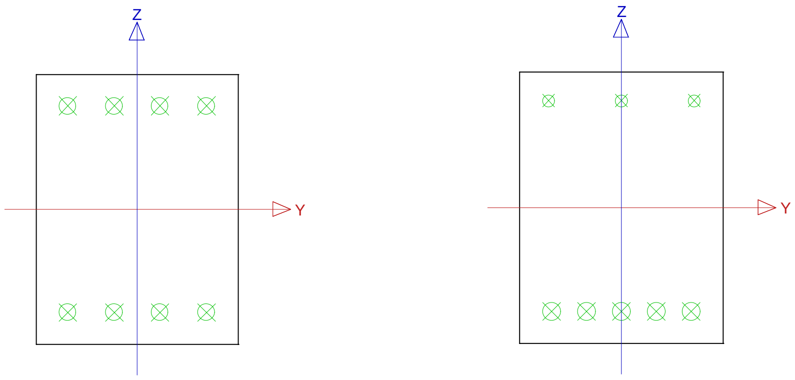
Also there, you have the choice for the 3 different analysis models, by means of the option “Member type”:



The 1D member data *overwrite* both the element properties and the default settings in the Concrete settings.

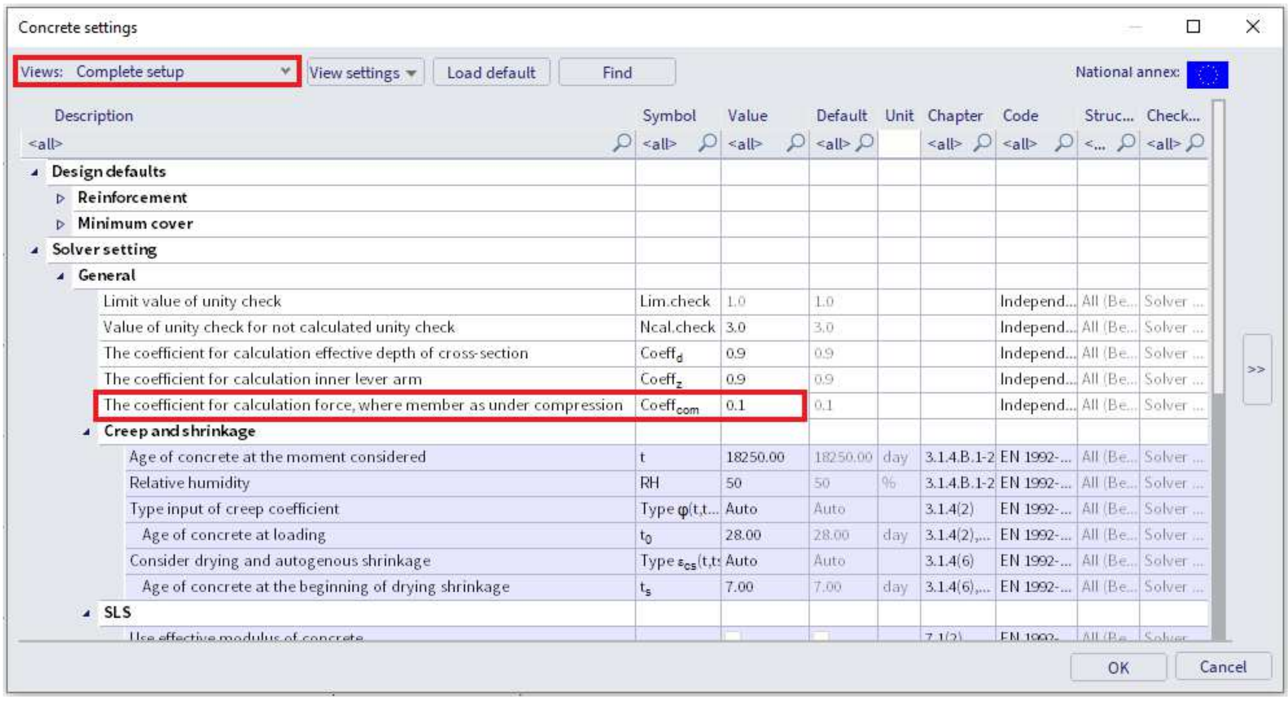
DIFFERENCE BETWEEN BEAM AND COLUMN ANALYSIS MODEL

The most important difference between Beam and Column calculation is the difference in reinforcement area per direction. A beam has an upper reinforcement area that differs from the lower reinforcement area. A column always has the same reinforcement configuration for the parallel sides, per direction.



These configurations are obvious, and caused by the difference in dominant internal forces per calculation type. For a beam calculation the bending moment is dominant, while for a column calculation the axial compression force + bending moments (if present).

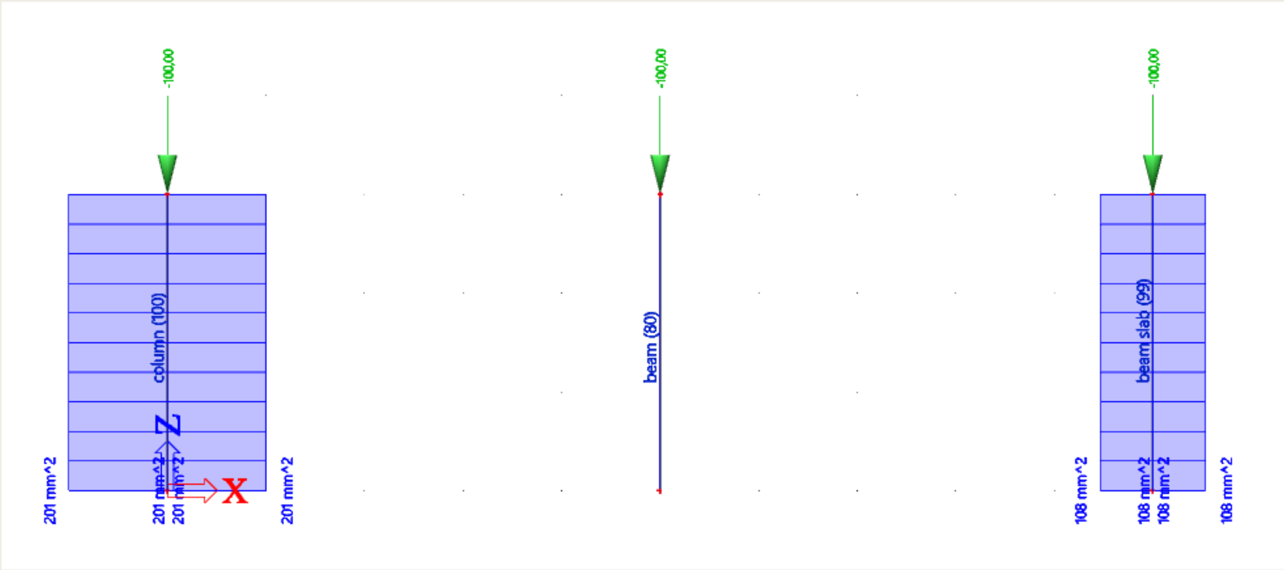
So in fact, when the axial pressure on a beam is too high, you should choose to calculate the element as a column. In the concrete settings an option is available to consider if the member is in compression or not. If the member is compressed, the second order effect is taken into account. Go to the Concrete workstation and “Concrete settings”, on the “Complete setup” view :



This option ‘The coefficient for calculation force, where member as under compression’ will check how important the contribution of the axial compression force is:

- If the axial compression load $N_{Ed} < 0{,}1{*}A_c{*}f_{cd}$, the member is not considered to be in compression, which means the type ‘Beam’ is the right choice.
- If the axial compression load $N_{Ed} > 0{,}1{*}Ac{*}f_{cd}$, the member is considered to be in compression, which means the beam has to be modelled as type ‘Column’ and the second order effect will be taken into account.

2.1.3 Example



Overall Design (ULS)

Linear calculation
Load case: LC2
Coordinate system: Member
Extreme 1D: Member
Selection: All

Longitudinal required reinforcement

Name	dx [m]	Case	Member	A_{sz} req+ [mm^2] A_{sz} req bar+ [mm^2]	A_{sz} req- [mm^2] A_{sz} req bar- [mm^2]	A_{sy} req+ [mm^2] A_{sy} req bar+ [mm^2]	A_{sy} req- [mm^2] A_{sy} req bar- [mm^2]	A_{sz} req [mm^2] A_{sz} req bar [mm^2]	A_{sy} req [mm^2] A_{sy} req bar [mm^2]	A_s req [mm^2] A_s req bar [mm^2]	ReinfReq
B1	0,000	LC2	Column	**201** **201**	**201** **201**	**201** **201**	**201** **201**	**402** **402**	**402** **402**	**804** **804**	**[z]4ϕ16*,** **[y]4ϕ16***
B2	0,000	LC2	Beam	**0** **0**	**0** **0**	**0** **0**	**0** **0**	**0** **0**	**0** **0**	**0** **0**	
B3	0,000	LC2	Beam slab	**108** **201**	**108** **201**	**108** **201**	**108** **201**	**215** **402**	**215** **402**	**430** **804**	**[z+]2ϕ16*,** **[z-]2ϕ16*,** **[y+]2ϕ16*,** **[y-]2ϕ16***

Shear reinforcement

Name	dx [m]	Case	Member	A_{swm} req [mm^2/m]	A_{swm} prov [mm^2/m]	ShearReinf
B1	0,000	LC2	Column	**0**	**0**	
B2	0,000	LC2	Beam	**0**	**0**	
B3	0,000	LC2	Beam slab	**0**	**0**	**Not required**

Under internal forces, a warning will be displayed in the detailed output whether it is necessary to calculate an element as column, to take into account the compression forces. If needed, the type has to be changed manually to column in the member properties or via 1D member data.

Compression member

Limit axial force to consider member as compression:

$$N_{com} = -\,Coeff_{com} \cdot \left(f_{cd} \cdot A_c \right) = -\,0.1 \cdot \left(6.4 \cdot 10^{6} \cdot 0.09 \right) = -57.6 \text{ kN}$$

Check condition:

$N_{Ed} < N_{com}$ = -100 kN < -58 kN ... compression member

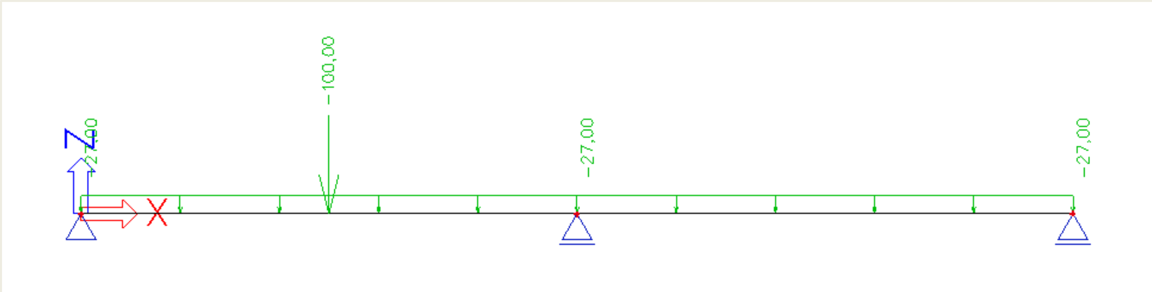
Warning: First and second order eccentricities should be taken to account, member should be evaluated as column (significant compressive normal force). Change type of member to Column.

2.2 Beam design

2.2.1 Description of used example

The example that will be used to explain reinforcement calculation in a beam is called 'beam.esa'.

The beam reinforcement calculation is explained by means of the following two span beam:



The length of the total beam is 10m and it has a dimension of 500x300mm.

The inputted loads are:

- BG1: self-weight
- BG2: permanent load
  - Line load: -27 kN/m
  - Point load: -100 kN at position x = 0,25
- BG3: variable load
  - Line load: -15 kN/m
  - Point load: -150 kN at position x = 0

2.2.2 Recalculated internal forces

Reinforcement calculation in SCIA Engineer is based on recalculated internal forces. The pure internal forces calculated by the mechanical FEM calculation are transformed according to code regulation into 'recalculated internal forces' to design the reinforcement.

These recalculated internal forces can be viewed in the Concrete settings of SCIA Engineer.

Shifting of bending moments (art 9.2.1.3)

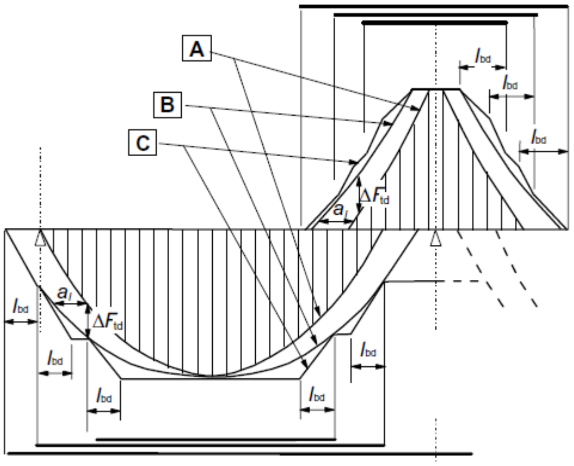
Sufficient reinforcement should be provided at all sections to resist the envelope of the acting tensile force, including the effect of inclined cracks in webs and flanges.

Additional tensile forces caused by shear and torsion are taken into account in SCIA Engineer by using the simplified calculation based on shifting of bending moments according to clause 9.2.1.3(2). Shifting of bending moments is calculated only for beams and beams as slab.

For members with shear reinforcement the additional tensile force, ΔF_{td}, should be calculated. For members without shear reinforcement, ΔF_{td} may be estimated by shifting the moment curve a distance a_l = d (for beams as slab). This "shift rule" may also be used as an alternative for members with shear reinforcement, where:

$$a_l = z\,(\cot\theta - \cot\alpha)/2 \qquad \text{(for beams)} \qquad (9.2)$$

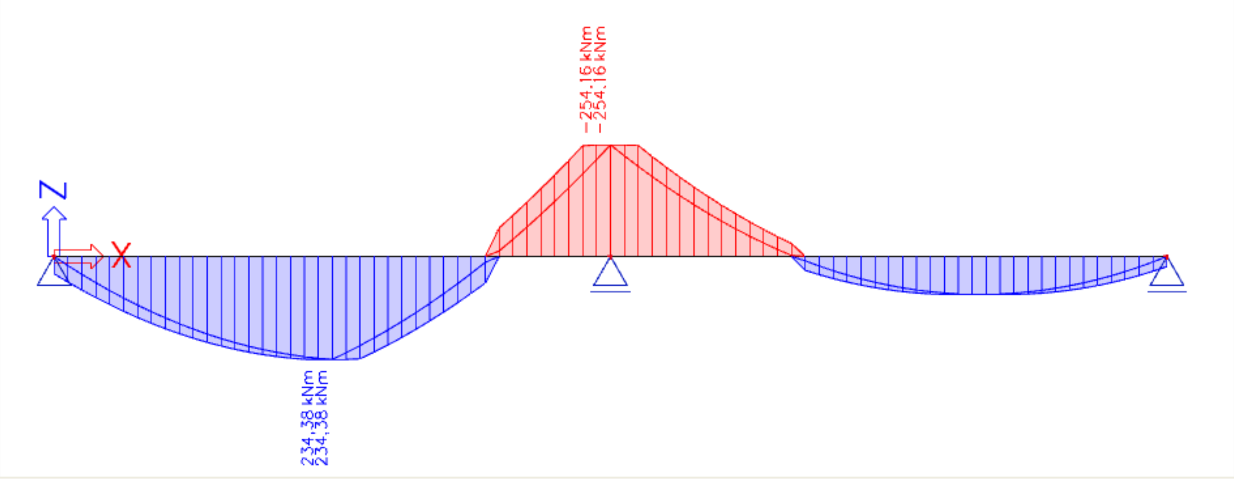
The additional tensile force is illustrated in Figure 9.2:



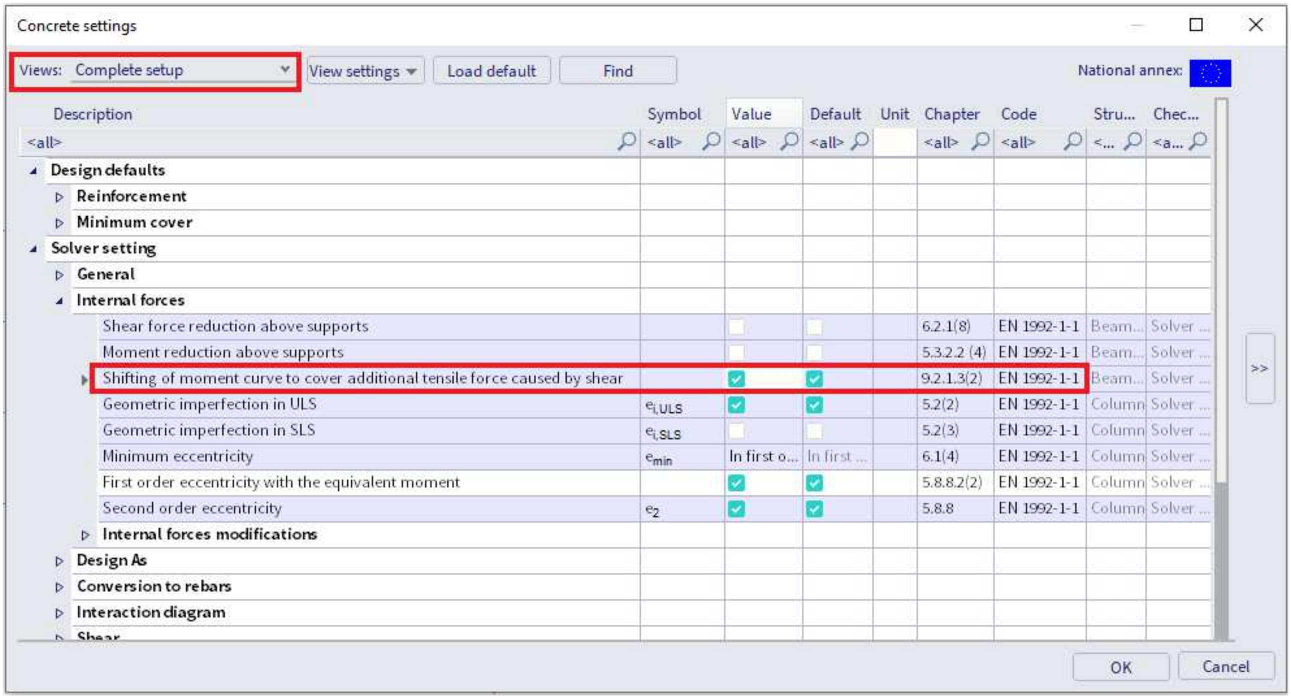
A - Envelope of $M_{Ed}/z + N_{Ed}$ B - acting tensile force F_s C - resisting tensile force F_{Rs}

Figure 9.2: Illustration of the curtailment of longitudinal reinforcement, taking into account the effect of inclined cracks and the resistance of reinforcement within anchorage lengths

In SCIA Engineer, you can review the recalculated internal forces. In the Concrete menu it is possible to view the internal forces and recalculated internal forces. In the figure below the difference is clearly visible:



The shifted moment line is taken into account for recalculated internal forces and by this also for the calculation of longitudinal reinforcement, if activated in the concrete settings (for the global structure) or in the 1D member data (individually per member):



REDUCTION OF BENDING MOMENT (art 5.3.5.5 (3) & 5.3.2.2 (4))

Another typical case of recalculated internal forces is the moment capping at supports.

Where a beam or slab is monolithic with its supports, the critical design moment at the support should be taken as that at the face of the support. The design moment and reaction transferred to the supporting element (e.g. column, wall, etc.) should be generally taken as the greater of the elastic or redistributed values.

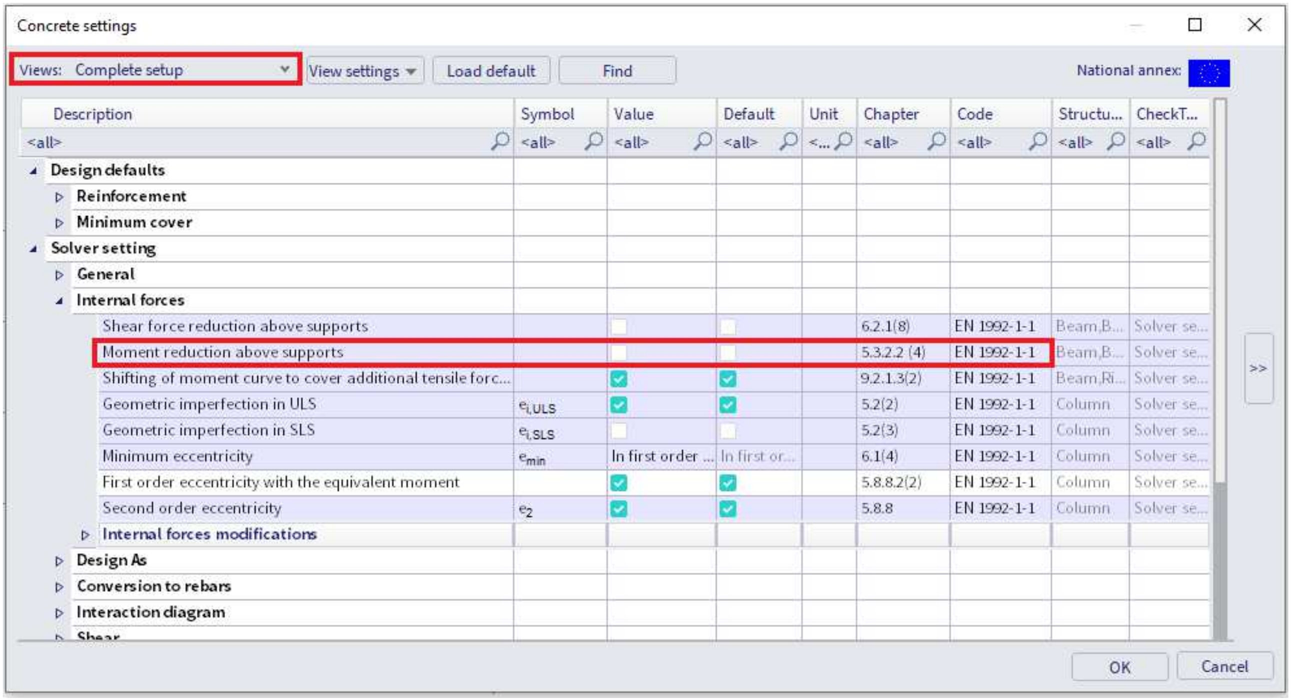
Regardless of the method of analysis used, where a beam or slab is continuous over a support which may be considered to provide no restraint to rotation (e.g. over walls), the design support moment, calculated on the basis of a span equal to the center-to-center distance between supports, may be reduced by an amount ΔM_{Ed} as follows:

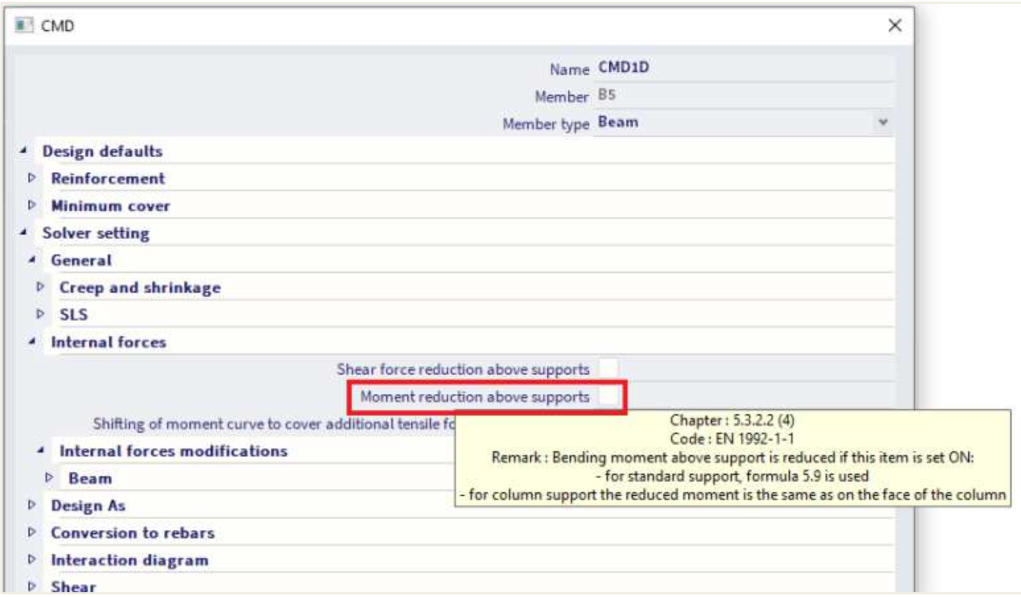
$$\Delta M_{Ed} = F_{Ed,sup}\, t / 8 \qquad (5.9)$$

where:

- $F_{Ed,sup}$ is the design support reaction
- t is the width of the support

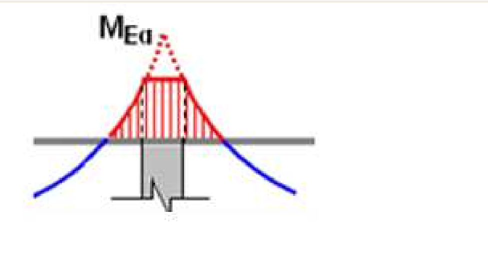
In SCIA Engineer this reduction of bending moment is only taken into account if it is activated in the concrete settings (for the global structure) or in the 1D member data (individually per member):



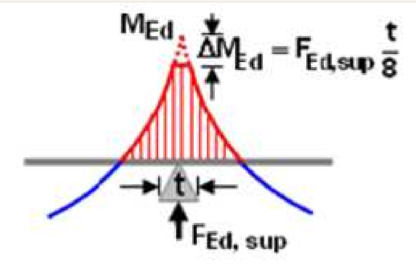


The way in which the moment reduction is performed, is based on the type of support. If a standard support is defined, the reduction will be done following formula 5.9. If a column is defined, the reduction at the face of the column is used.

At the face of the column (5.3.2.2 (3))

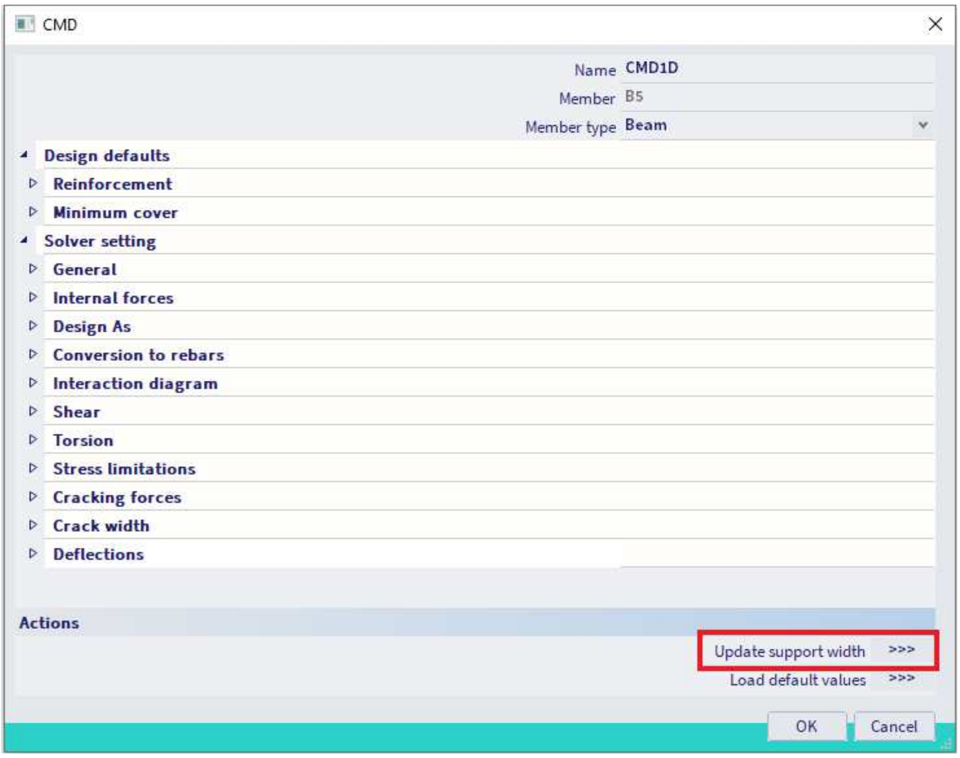


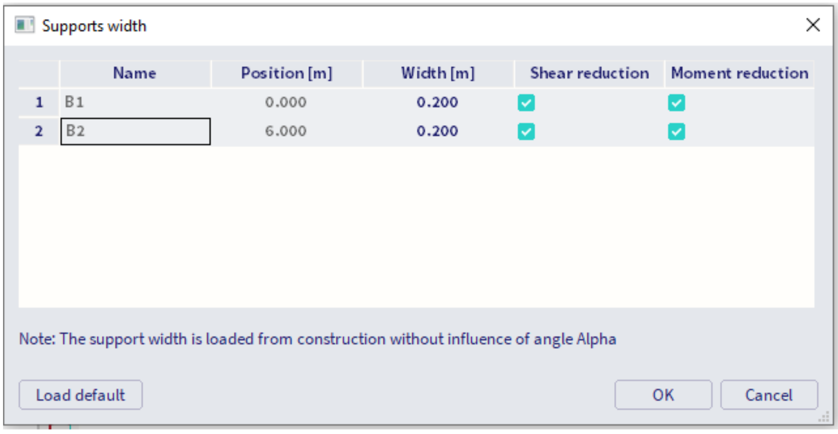
Using formula 5.9 (5.3.2.2 (4))



In SCIA Engineer, the width “t”, used for the moment reduction at supports, can be set in the properties of that support:

In the bottom of the 1D member data, there is an action button “Update support width”. This button collects all linked members or supports of the selected member and reads their support widths.

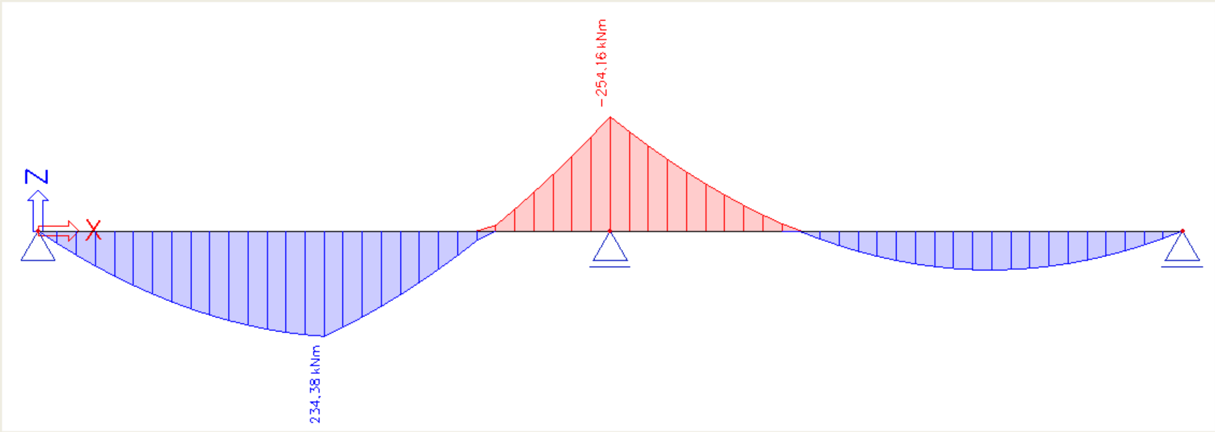




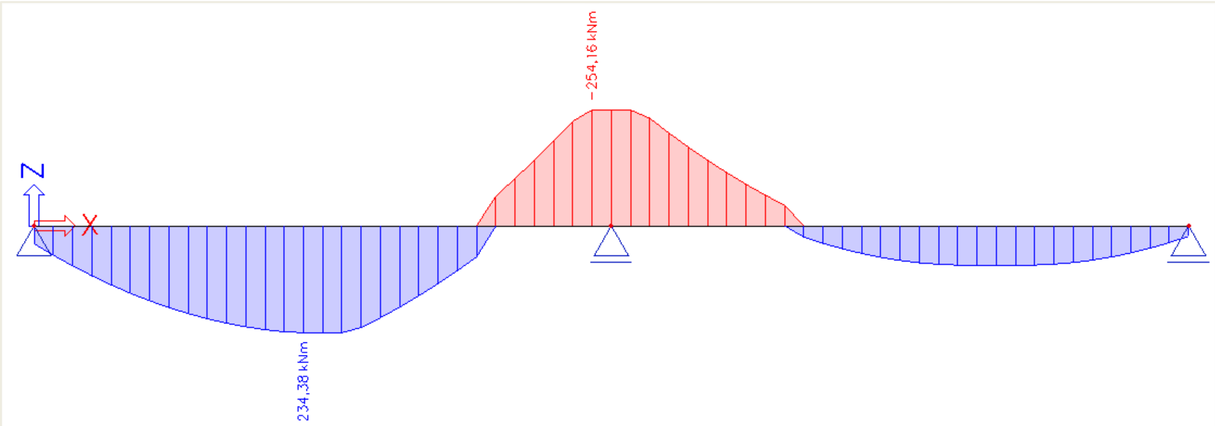
The reduction of moment by moment capping at supports is illustrated for our example below:

- t = 0,2m
- $F_{Ed,sup}$ = 477,5kN
- ΔM_{Ed} = 477,5*0,2 / 8 = 11,94kNm

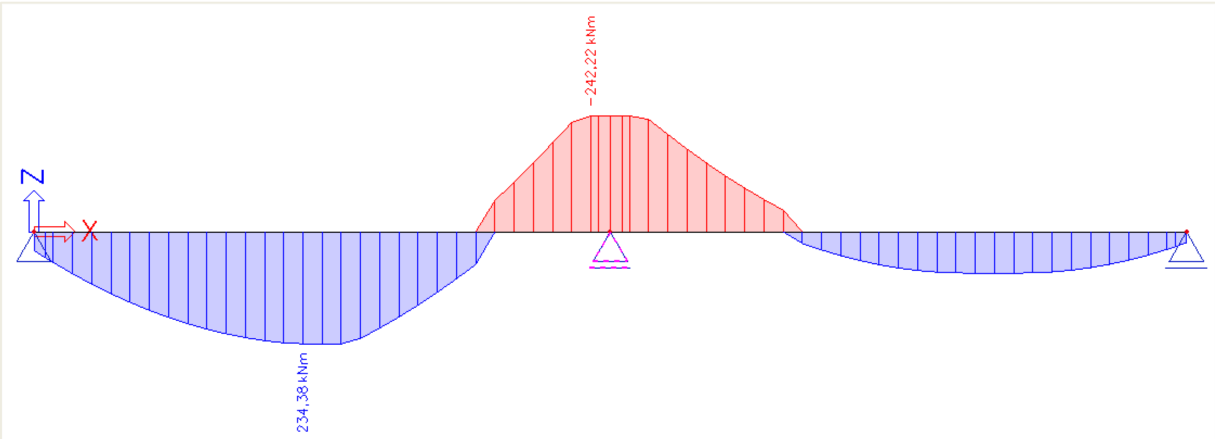
The original moment M_y at the support was 254,16 kNm:



The recalculated moment clearly shows the shifting of the moment line.



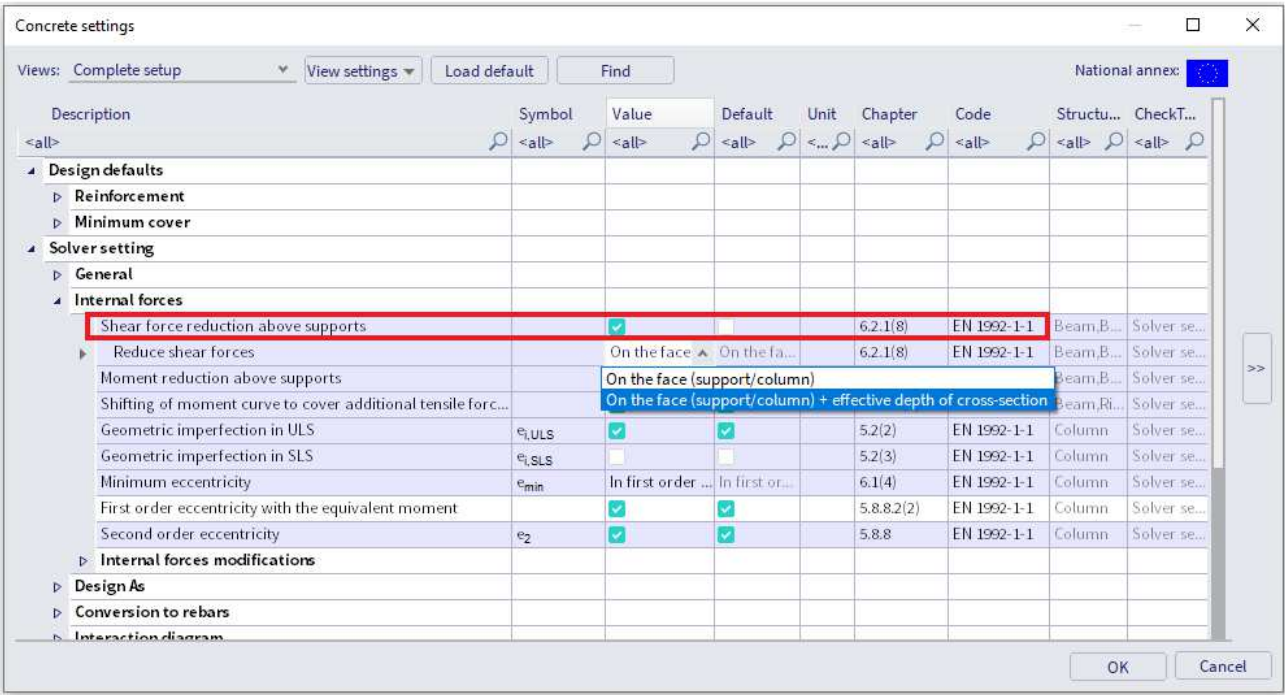
With moment capping at support taken into account the recalculated moment is 242,22 kNm.

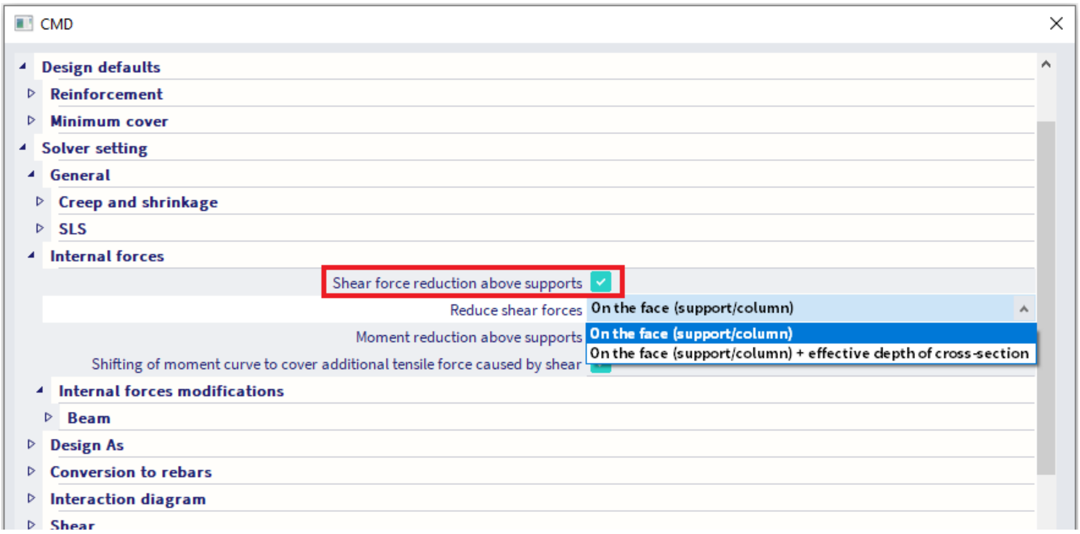


REDUCTION OF SHEAR FORCES (art 6.2.1 (8))

For members subject to predominantly uniformly distributed loading, the design shear force does not need to be checked at a distance less than d from the face of the support. Any shear reinforcement required should continue to the support. In addition it should be verified that the shear at the support does not exceed $V_{Rd,max}$.

In SCIA Engineer, this reduction of shear forces is only taken into account if it is activated in the concrete settings (for the global structure) or in the 1D member data (individually per member):



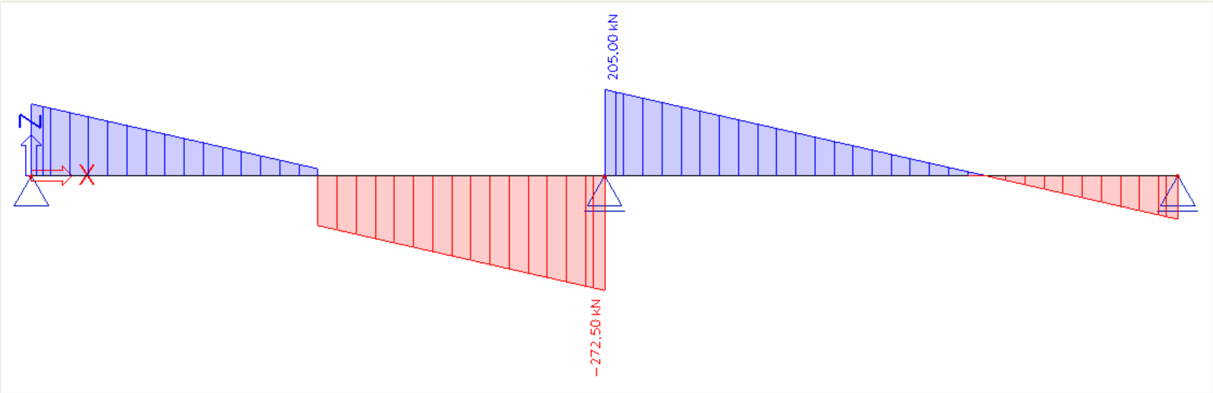


It is possible to choose the type of reduction of shear forces at the face of the support or at a distance d from the face of the support:

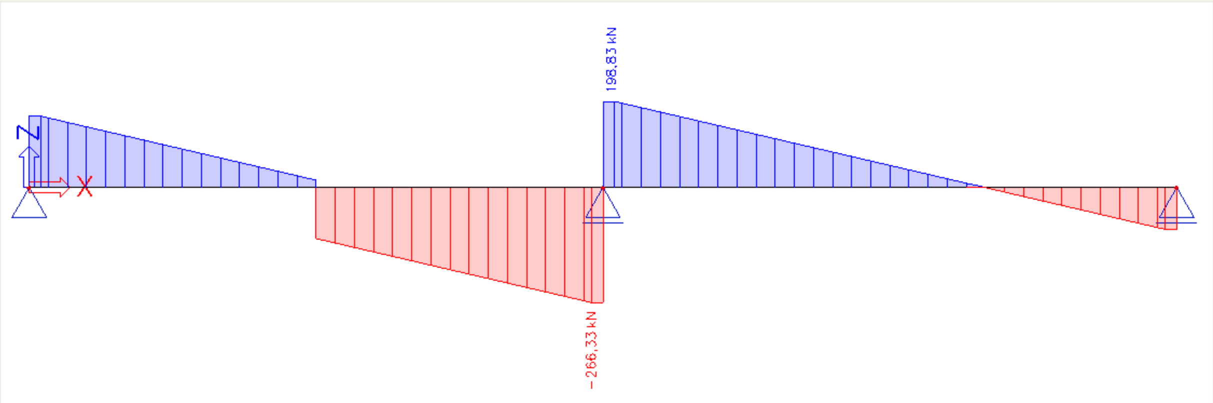
At the face of the column	At the face of the column + effective depth of the cross section (based on

Also for the reduction of shear forces, the support width "t" is taken into account, which is taken from the properties of the support or the 1D member data. The reduction of shear forces at supports is illustrated for our example below with t = 0,2 m.

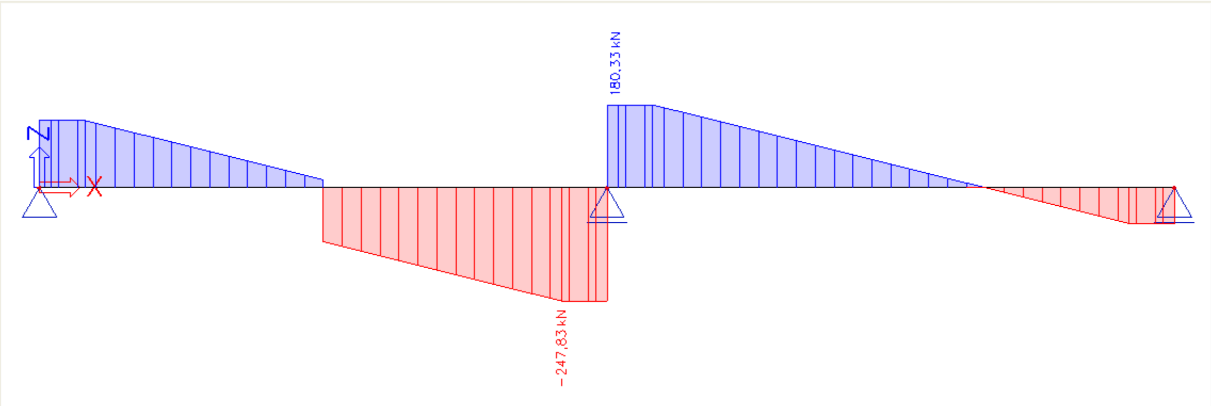
The first image displays the original V_z:



The second image shows the reduction at the face of the support:



The last image shows the reduction at the effective depth from the face:



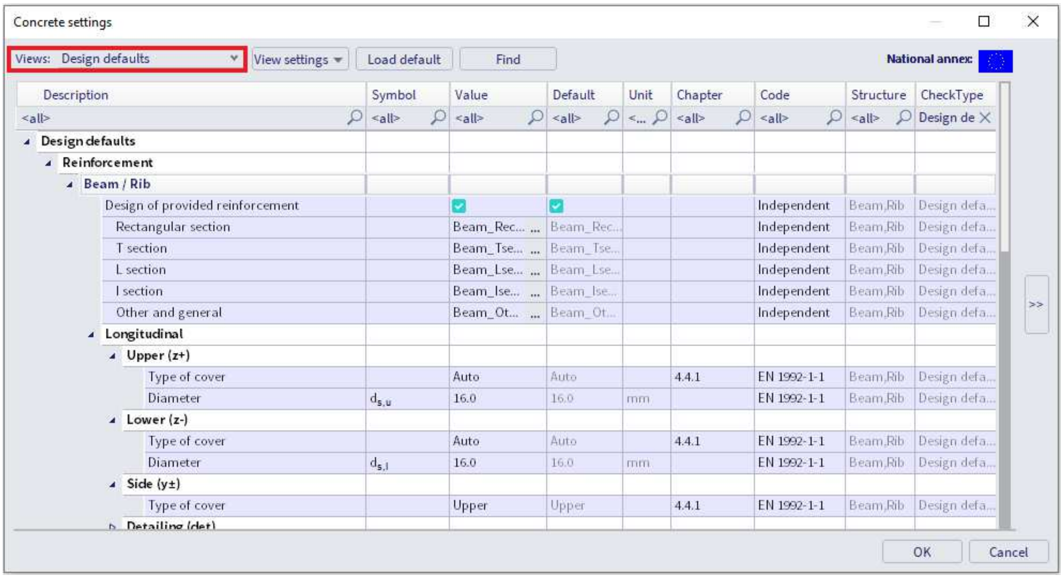
2.2.3 Theoretical reinforcement

CONFIGURATION

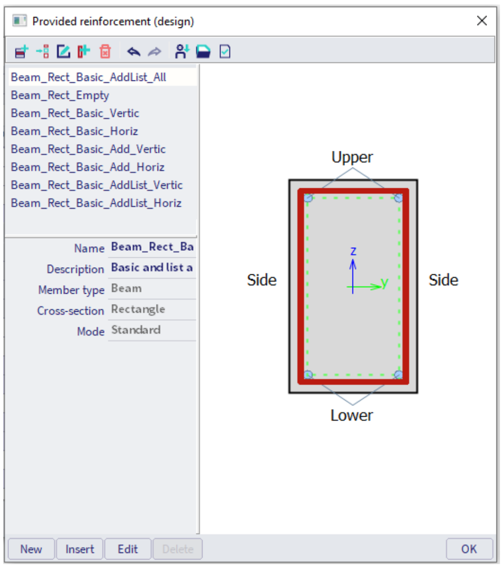
The theoretical reinforcement is calculated out of the recalculated internal forces. It gives the amount of reinforcement needed to resist the internal forces induced by ULS loads. Since there are several workflows possible to design concrete beam elements, the theoretical reinforcement design is not mandatory to perform. Experienced users can directly jump to practical reinforcement to perform the checks on, but this theoretical approach gives a good idea of how this practical reinforcement should look like. There are two types of theoretical reinforcement:

- **Required reinforcement:** The required reinforcement is a numerical value (mm²) of the reinforcement that is necessary in every section of the beam.
- **Provided reinforcement:** The provided reinforcement is a template added to each beam/column consisting of basic and additional reinforcement.

The configuration of theoretical reinforcement can be found in the Concrete settings, in the “Design defaults” view. Templates of longitudinal reinforcement and stirrups for different shapes of beam are available. The concrete cover can be set for upper, lower and side faces.



Several default templates for longitudinal reinforcement and stirrups are available for the different section types (provided reinforcement). These can be adapted or new ones can be made.

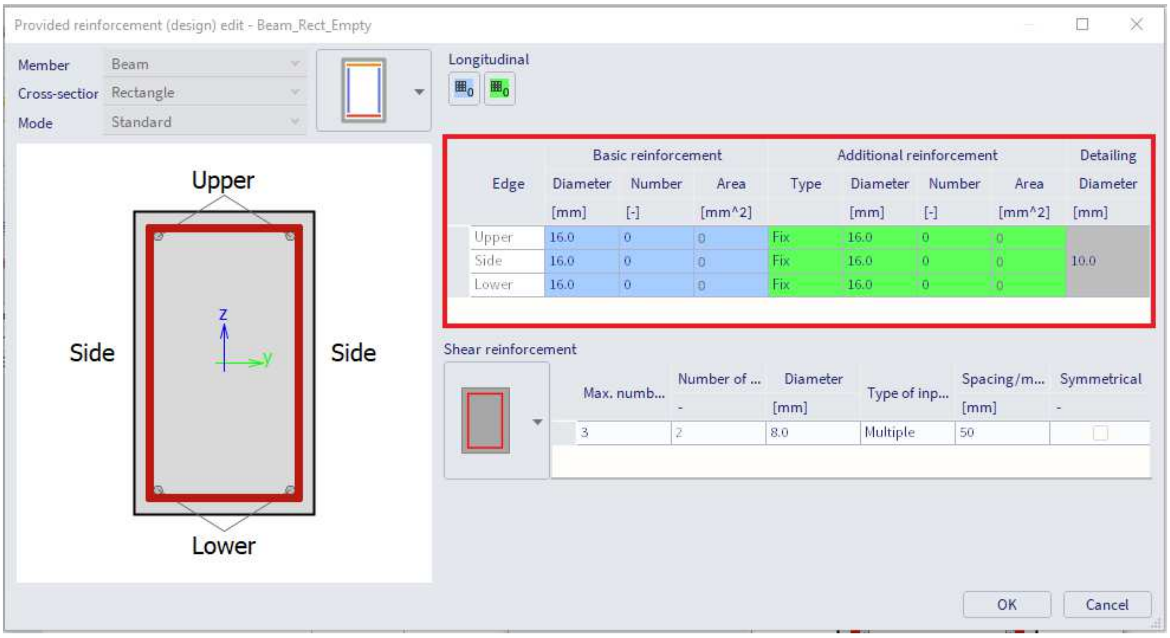


This template exists of basic, additional and shear reinforcement. The purpose is to compare these templates with the required reinforcement, to model the user reinforcement that is introduced later on or to convert it automatically to user reinforcement.

⇨ ***Longitudinal reinforcement***

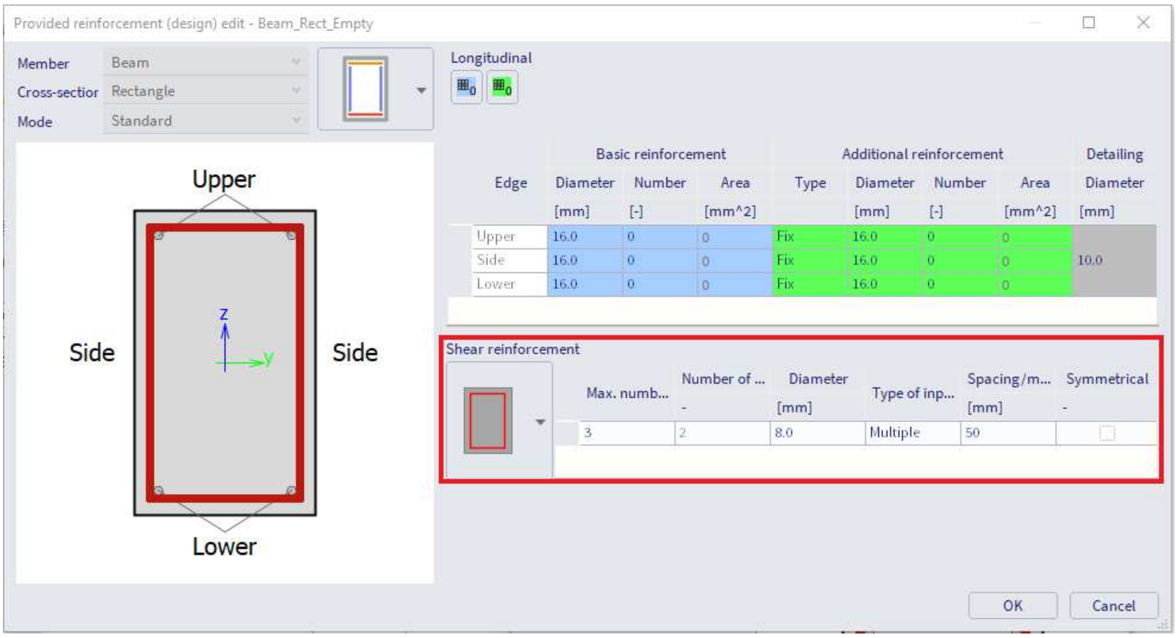
The basic reinforcement is present along the whole length of the beam; the additional reinforcement is present only at the zones where basic reinforcement is not sufficient to withstand (recalculated) internal forces.

A choice can be made between fixed additional bars (diameter and number) or a list with different numbers of bars with a fixed diameter. SCIA Engineer uses the least amount of necessary additional bars or places the maximum if this template is still not sufficient to resist the (recalculated) internal forces. Next to the basic and additional reinforcement you can also set a diameter for the detailing reinforcement. The detailing reinforcement is reinforcement that statically is not required but that needs to be added to the cross-section to fulfil the detailing provisions.



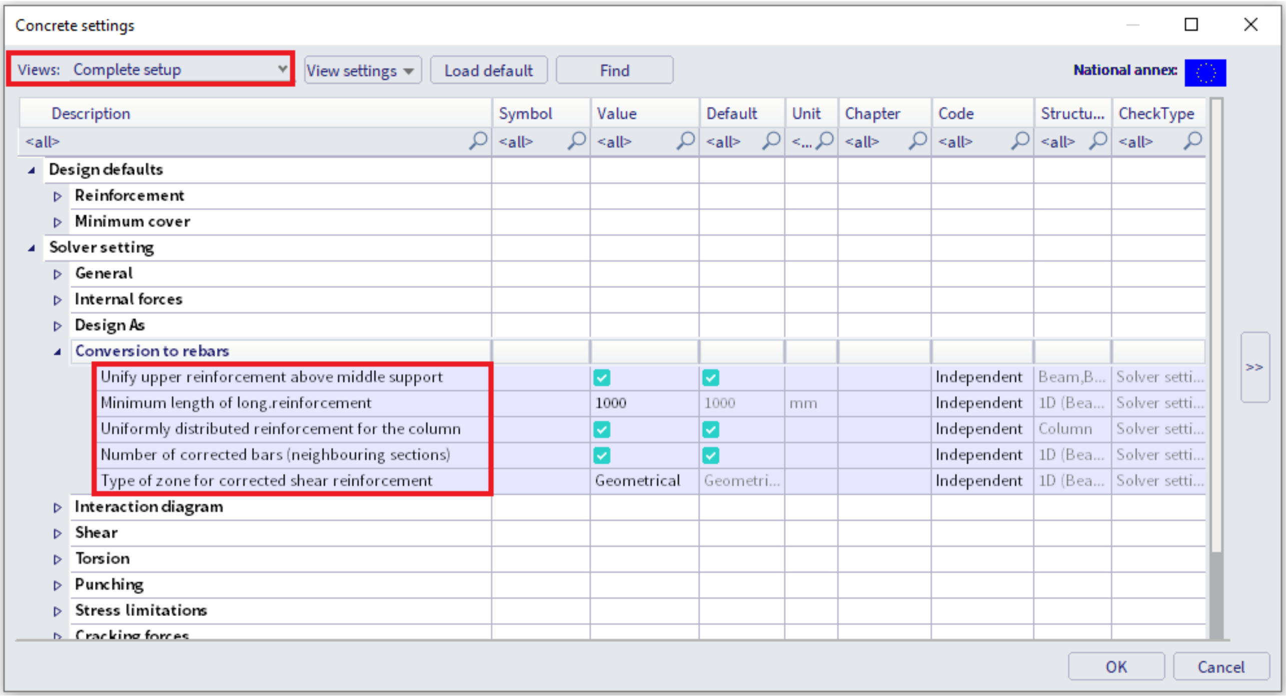
⇨ ***Shear reinforcement***

For the shear reinforcement the number of cuts, the maximum number of stirrup zones, the diameter and the spacing can be set. For the spacing different types of input can be used: **Multiple** and **User defined**. Multiple means that the spacing between the stirrups will be the multiple of a set value. With User defined reinforcement you can set the spacings that can be used. SCIA Engineer will automatically select the spacing depending on this template and the general settings in the design defaults. The option **Symmetrical** allows you to define whether the zones in each span will be symmetrical or not.



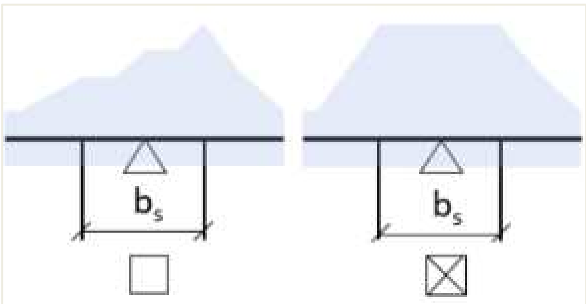
CONFIGURATION FOR CONVERSION TO REBARS

The configuration for conversion to rebars can be found in the Concrete settings, in the “Complete setup” view. Different options are available:



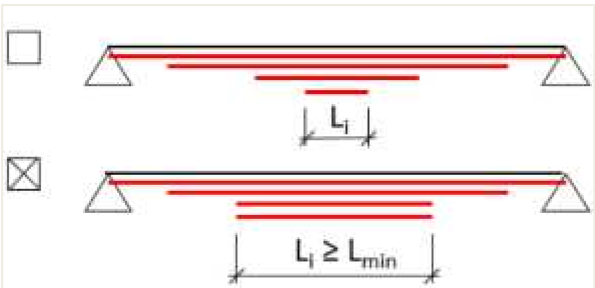
⇨ ***Unify upper reinforcement above middle support***

Unifies the number of bars of upper reinforcement at the middle support. The maximum number of bars from the left and right side of the support are taken into account.



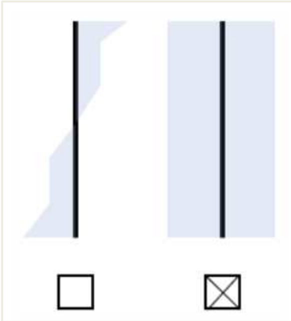
⇨ ***Minimum length of longitudinal reinforcement***

Sets a minimum length for the longitudinal reinforcement.



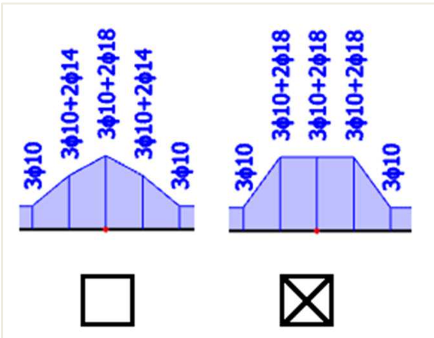
⇨ ***Uniformly distributed reinforcement for the column***

Uniform distribution of reinforcement along the whole length of the column, with maximum area from y and z edges in all sections taken into account.



⇨ ***Number of corrected bars (neighbouring sections)***

Additional reinforcement is tested in each section for number of bars and diameter in neighbouring sections. If the additional reinforcement can be distributed to the stirrup links between basic reinforcement bars, the number of bars and diameter of additional reinforcement is increased to fulfil conditions. The reason for the correction of the number of bars of additional provided reinforcement is to have logic and symmetrical reinforcement in the cross-section along the beam.

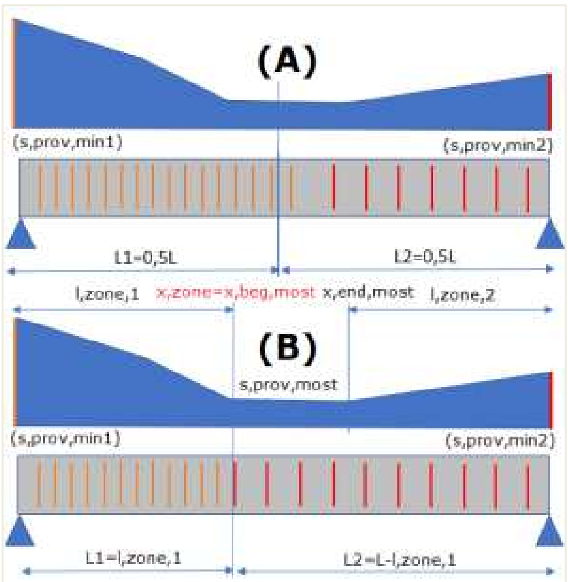


⇨ ***Type of zone for corrected shear reinforcement***

None - Zones for shear reinforcement are not created. Conversion of provided reinforcement to real bars is not possible.

(A) Geometrical – Member is in every span divided geometrically in zones with the same length.

(B) Spacing – Member is in every span divided in zones according to the most occurrent spacing.



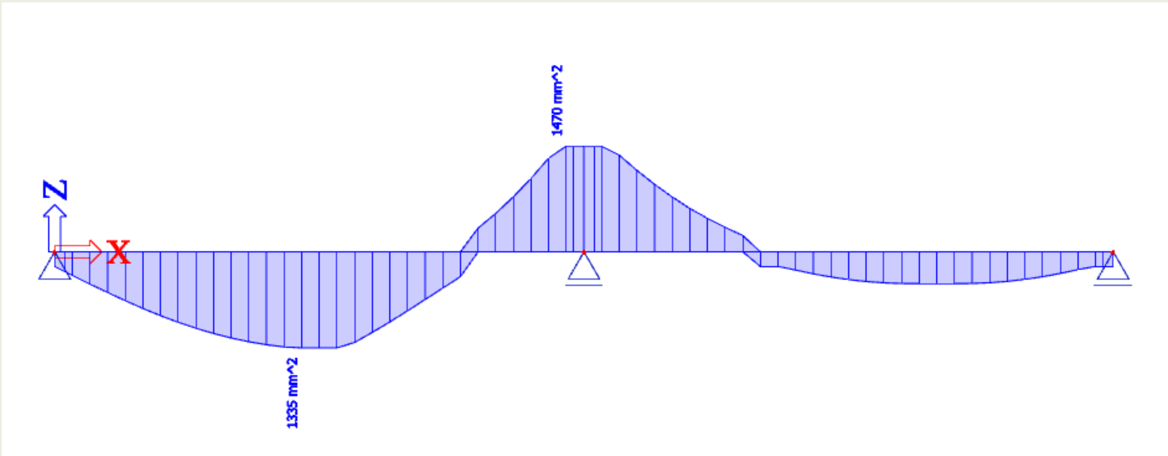
CALCULATION OF LONGITUDINAL REINFORCEMENT As

The longitudinal reinforcement calculation is based on $M_{y,recalc}$ represented in the previous chapter.

The only thing left to be set in the concrete setup is the material quality and default diameter:

- Material quality is set to B 500A. This can be changed in the project data or concrete 1D member data.
- The default diameter is set to 16 mm. This parameter is taken from the additional reinforcement diameter of the reinforcement template under Design defaults, or from 1D member data.

The following results are obtained with these settings:



In the following image you can see the brief output in the preview:

Longitudinal required reinforcement

Name	dx [m]	Case	Member	A_{sz_req+} [mm²] $A_{sz\_req\_bar+}$ [mm²]	A_{sz_req-} [mm²] $A_{sz\_req\_bar-}$ [mm²]	A_{sy_req+} [mm²] $A_{sy\_req\_bar+}$ [mm²]	A_{sy_req-} [mm²] $A_{sy\_req\_bar-}$ [mm²]	A_{sz_req} [mm²] $A_{sz\_req\_bar}$ [mm²]	A_{sy_req} [mm²] $A_{sy\_req\_bar}$ [mm²]	A_{s_req} [mm²] $A_{s\_req\_bar}$ [mm²]	ReinfReq
S1	2,333-	ULS	Beam	0 0	**1335** **1407**	**0** **0**	**0** **0**	1335 1407	**0** **0**	1335 1407	**[z-]7ϕ16**
S1	4,833-	ULS	Beam	**1470** **1608**	0 0	0 0	0 0	**1470** **1608**	0 0	**1470** **1608**	[z+]8ϕ16

You can also ask a standard or a more detailed output where you can find more information about certain parameters used in the calculation, for example about d = lever arm of reinforcement:

$$d = h - cover - \Phi_{stirrup} - \Phi_{longitudinal\ beam}/2 = 500 - 35 - 8 - \mathbf{16}/2 = 449\text{ mm}$$

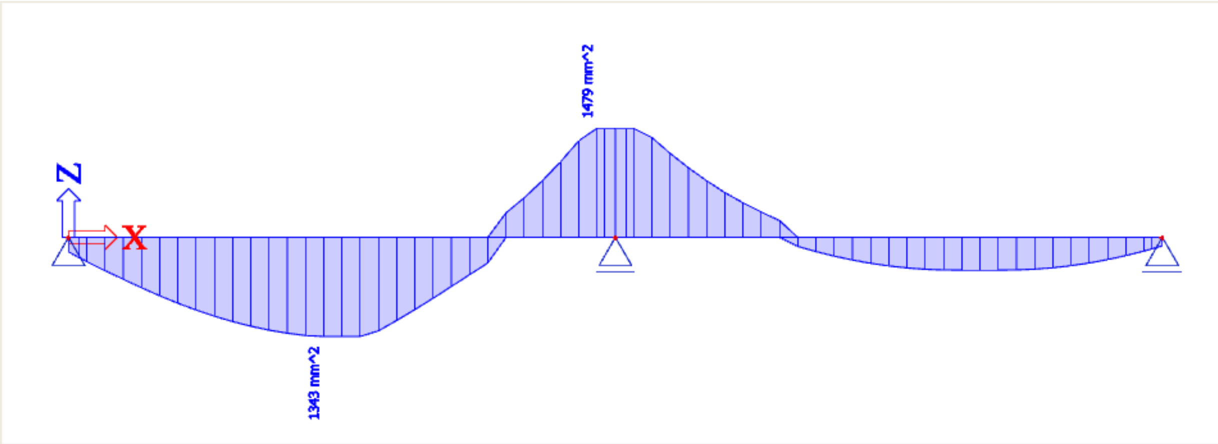
(the cover is defined by the environmental class and is 35 mm for XC3)

The only internal force working on this beam is M_{yd}. N_d and T_d are zero.

A_{sy_req} = 0 because there is no torsion on this beam.

Note that the detailing provisions are deactivated. Otherwise, no reinforcement of ϕ = 16 mm could be proposed, since the detailing provisions are not met (bar distance too small).

If the default diameter is set to 20 mm, the following results are obtained:



Longitudinal required reinforcement

Name	dx [m]	Case	Member	A_{sz_req+} [mm²] $A_{sz\_req\_bar+}$ [mm²]	A_{sz_req-} [mm²] $A_{sz\_req\_bar-}$ [mm²]	A_{sy_req+} [mm²] $A_{sy\_req\_bar+}$ [mm²]	A_{sy_req-} [mm²] $A_{sy\_req\_bar-}$ [mm²]	A_{sz_req} [mm²] $A_{sz\_req\_bar}$ [mm²]	A_{sy_req} [mm²] $A_{sy\_req\_bar}$ [mm²]	A_{s_req} [mm²] $A_{s\_req\_bar}$ [mm²]	ReinfReq
S1	2,333-	ULS	Beam	0 0	**1343** **1571**	**0** **0**	**0** **0**	1343 **1571**	**0** **0**	1343 **1571**	**[z-]5ϕ20**
S1	4,833-	ULS	Beam	**1479** **1571**	0 0	0 0	0 0	**1479** 1571	0 0	**1479** 1571	[z+]5ϕ20

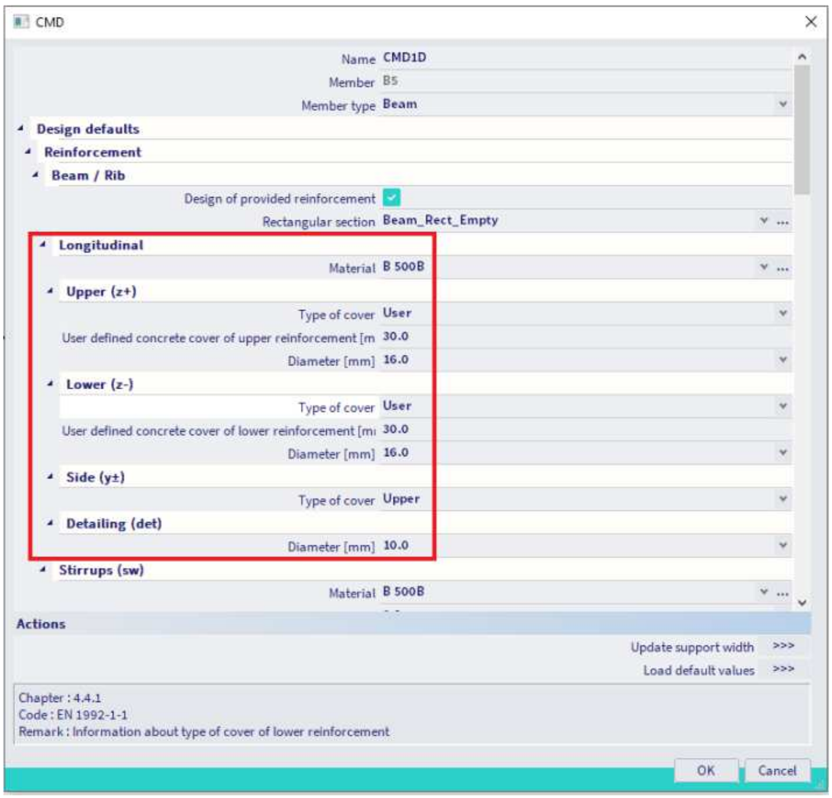
If you take a close look at these results, you can see that also the value for $A_{s,req}$ has changed.

This is because the lever arm d has decreased:

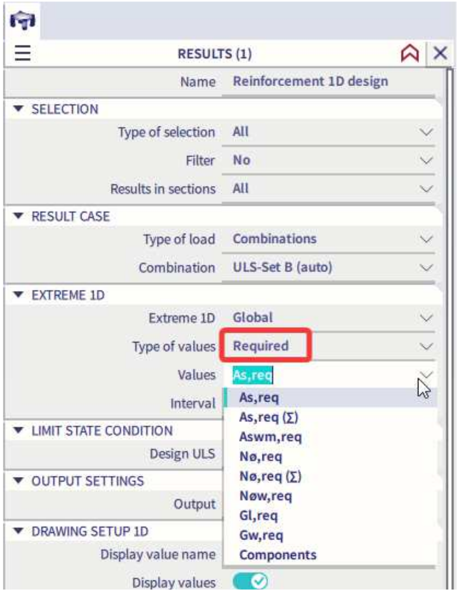
$$d = h - cover - \Phi_{stirrup} - \Phi_{longitudinal\ beam}/2 = 500 - 35 - 8 - \mathbf{20}/2 = 447 \text{ mm}$$

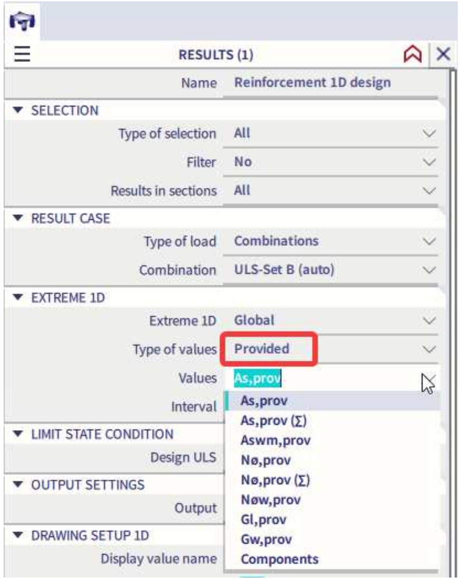
As you can see, the default diameter has also a slight effect on the amount of reinforcement that is required, because of the changed lever arm.

Note: 1D member data can be used to change the default diameter for the bar to which these data are assigned. It is obvious that the 1D member data have higher priority than the Concrete settings.

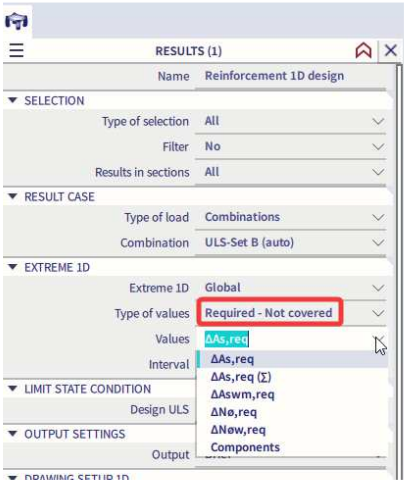


Next to the required reinforcement area, also the provided reinforcement can be viewed. Both can be displayed as a value in reinforcement area (value As or Aswm) or as the amount of bars (value Nφ or Nφw). Also the weight of the reinforcement can be shown (value Gl or Gw).

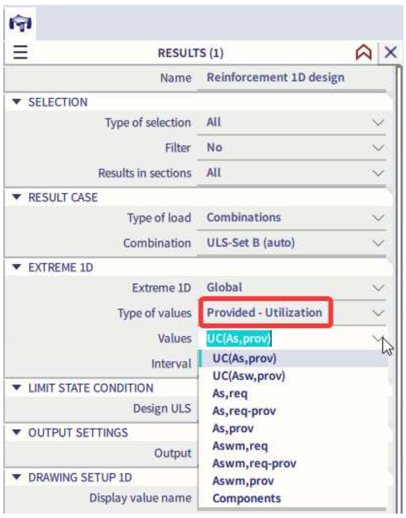




The option Required – not covered will show the amount of provided reinforcement that is missing. For example: $\Delta A_{s,req} = A_{s,req} - A_{s,prov}$, thus the amount of reinforcement which still has to be added to the template to resist the (recalculated) internal forces. If $A_{s,prov} > A_{s,req}$, $\Delta A_{s,req} = 0$



Unity checks can be performed on the provided reinforcement compared to the required reinforcement. This will give you an idea of the efficiency of the reinforcement.



CALCULATION OF SHEAR REINFORCEMENT Aswm

Shear reinforcement

Name	dx [m]	Case	Member	A_{swm_req} [mm²/m]	A_{swm_prov} [mm²/m]	ShearReinf
S1	7,333-	ULS	Beam	**298**	**309**	**ϕ8/325mm, (ns=2)**
S1	4,900	ULS	Beam	**1315**	**1340**	**ϕ8/75mm, (ns=2)**

V_{Ed} = design shear force resulting from external loading

$V_{Rd,c}$ = design shear resistance of the member without shear reinforcement

$V_{Rd,s}$ = design value of the shear force which can be sustained by the yielding shear reinforcement

$V_{Rd,max}$ = design value of the maximum shear force which can be sustained by the member, limited by crushing of the compression struts

In general we can have three cases:

- $V_{Ed} > V_{Rd,max}$ Concrete strut failure
- $V_{Ed} \leq V_{Rd,c}$ Shear force carried by concrete. No shear reinforcement necessary (minimum shear reinforcement according to detailing provisions)
- $V_{Ed} > V_{Rd,c}$ and $V_{Ed} < V_{Rd,max}$ Shear reinforcement necessary in order that: $V_{Ed} \leq V_{Rd}$

⇨ ***Members NOT requiring design shear reinforcement: VEd < VRd,c (art 6.2.2)***

$$V_{Rd,c} = [C_{Rd,c}\, k(100\, \rho_l\, f_{ck})^{1/3} + k_1\, \sigma_{cp}]\, b_w\, d \qquad (6.2.a)$$

with a minimum of:

$$V_{Rd,c} = (v_{min} + k_1\, \sigma_{cp})\, b_w\, d \qquad (6.2.b)$$

where:

- f_{ck} = characteristic concrete compressive strength [MPa]
- k = size factor: $k = 1 + \sqrt{(200/d)} \leq 2{,}0$ (with d in mm)
- ρ_l = longitudinal reinforcement ratio: $\rho_l = A_{sl}/b_w d \leq 0{,}02$
- b_w = smallest web width of the cross-section in the tensile area [mm]
- σ_{cp} = concrete compressive stress due to loading: $\sigma_{cp} = N_{Ed}/A_c < 0{,}2\, f_{cd}$ [MPa]
- d = effective height of cross section

The recommended value for $C_{Rd,c}$ is $0{,}18/\gamma_c$, that for k_1 is 0,15 and that for v_{min} is given by expression:

$$v_{min} = 0{,}035\, k^{3/2}.\, f_{ck}^{1/2} \qquad (6.3N)$$

The shear force V_{Ed}, calculated without reduction by β, should always satisfy the condition:

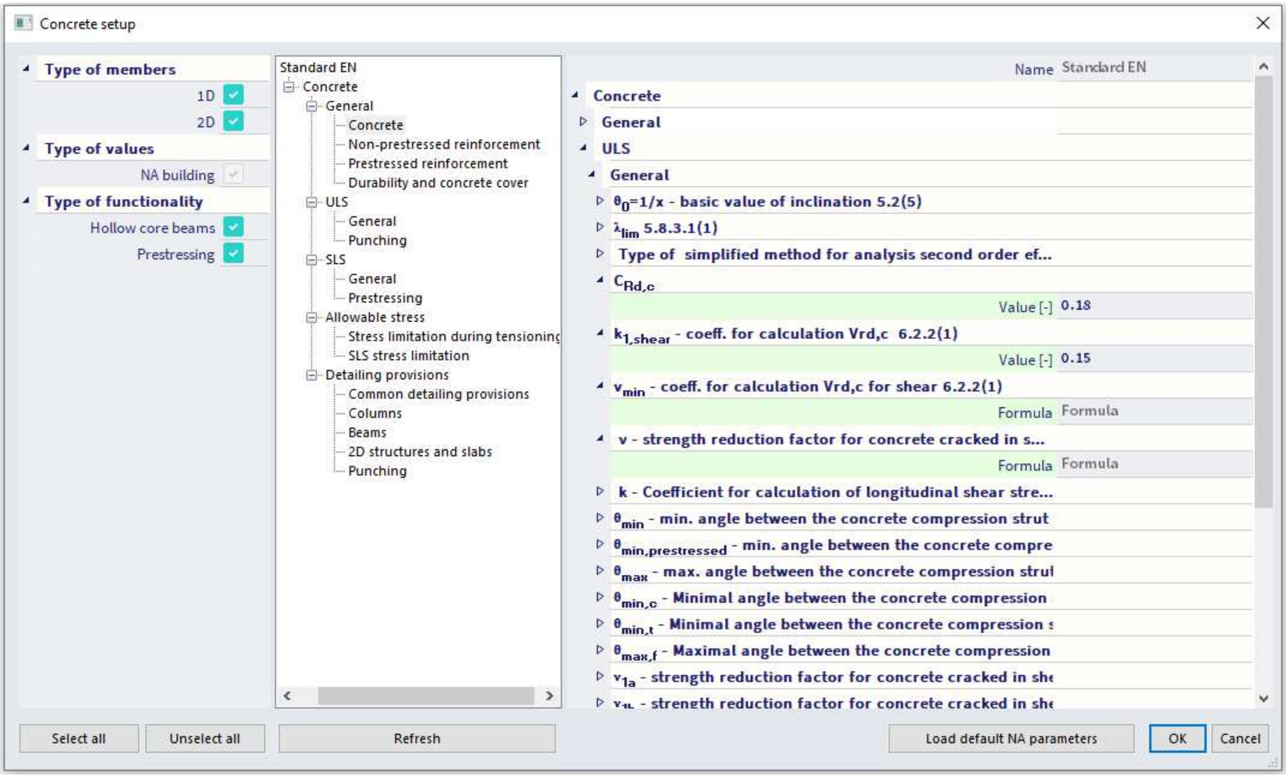
$$V_{Ed} \leq 0{,}5\, b_w\, d\, v\, f_{cd} \qquad (6.5)$$

where v is a strength reduction factor for concrete cracked in shear.

The recommended value for v follows from:

$$v = 0{,}6\left[1-\frac{f_{ck}}{250}\right] \qquad (6.6N)$$

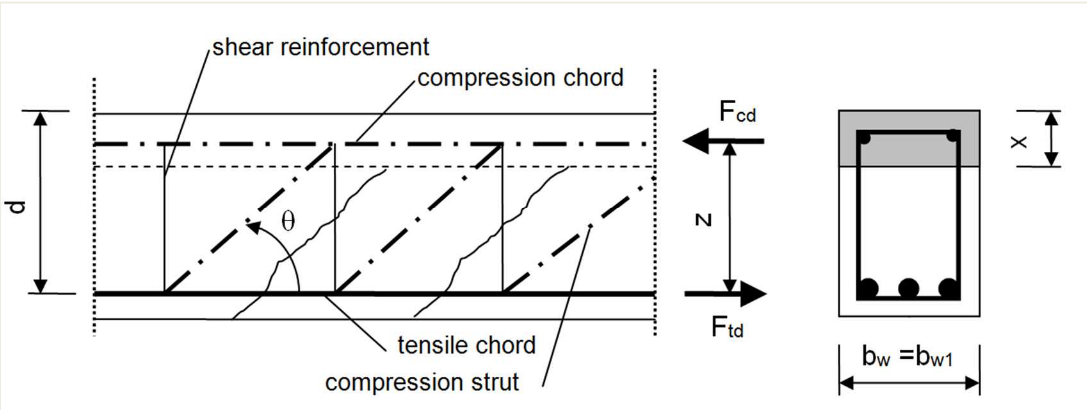
In SCIA Engineer, it is possible to input following parameters:



Note: the green values are according to EN code.

⇨ ***Members requiring design shear reinforcement VEd > VRd,c (art 6.2.3)***

The design of members with shear reinforcement is based on the theory of the concrete truss-model. In this theory, a virtual truss-model is imagined in a concrete beam. This truss-model has a set of vertical (or slightly diagonal), horizontal and diagonal members. The vertical bars are considered to be the stirrups, the horizontal bars are the longitudinal reinforcement bars and the diagonal bars are the concrete struts.

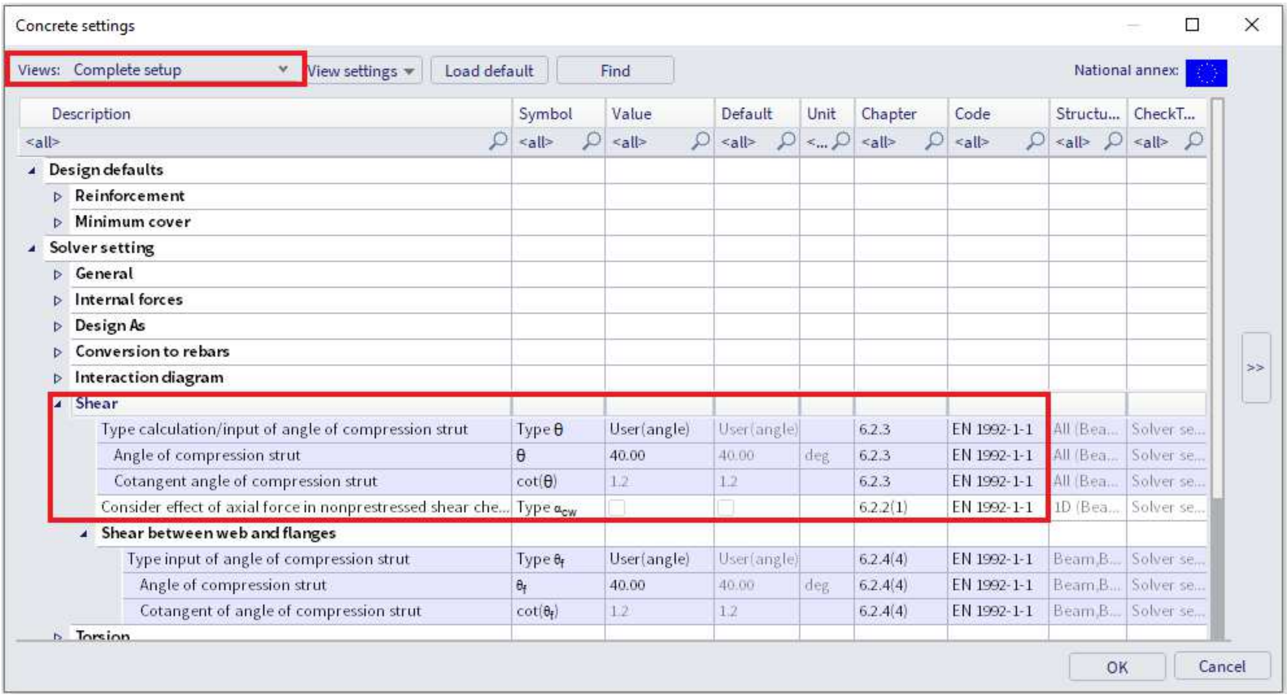


The angle θ should be limited.

The recommended limits of cot θ are given:

$$1 \leq \cot\theta \leq 2{,}5 \qquad (6.7N)$$

The angle θ can be inserted in SCIA Engineer:



For members with vertical shear reinforcement, the shear resistance V_{Rd} is the smaller value of:

$$V_{Rd,s} = \frac{A_{sw}}{s} z\, f_{ywd} \cot\theta \qquad (6.8)$$

and

$$V_{Rd,max} = \alpha_{cw}\, b_w\, z\, v_1\, f_{cd}/(\cot\theta + \tan\theta) \qquad (6.9)$$

where:

- A_{sw} = cross-sectional area of the shear reinforcement
- s = spacing of the stirrups
- f_{ywd} = design yield strength of the shear reinforcement
- v_1 = strength reduction factor for concrete cracked in shear
- α_{cw} = coefficient taking account of the state of the stress in the compression chord

The recommended value of v_1 is v (see Expression 6.6N)

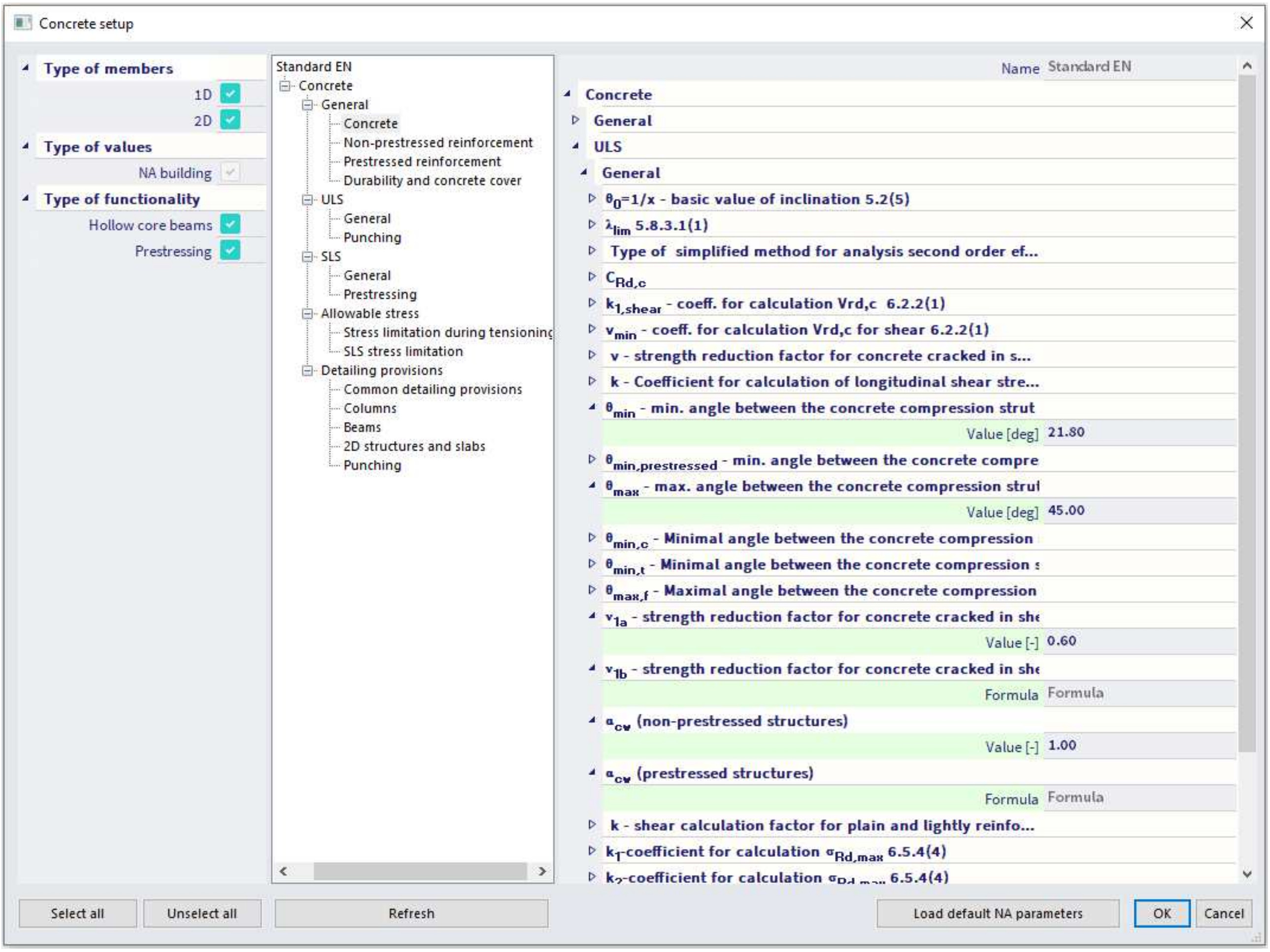
If the design stress of the shear reinforcement is below 80% of the characteristic yield stress f_{yk}, v_1 may be taken as:

$$v_1 = 0{,}6 \qquad \text{for } f_{ck} \leq 60 \text{ MPa} \qquad (6.10.aN)$$

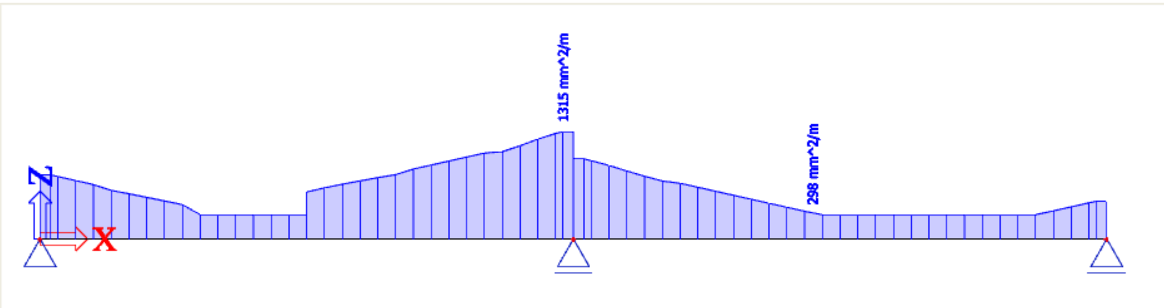
$$v_1 = 0{,}9 - f_{ck}/200 > 0{,}5 \qquad \text{for } f_{ck} \geq 60 \text{ MPa} \qquad (6.10.bN)$$

The recommended value of α_{cw} is 1 for non-prestressed structures.

These code related parameters can be found in the Concrete setup:



If we go back to our example in SCIA Engineer, we find the following $A_{swm,req}$ for the whole beam:



Shear reinforcement

Name	dx [m]	Case	Member	A_{swm_req} [mm²/m]	A_{swm_prov} [mm²/m]	ShearReinf
S1	7,333-	ULS	Beam	298	309	ϕ8/325mm, (ns=2)
S1	4,900	ULS	Beam	1315	1340	ϕ8/75mm, (ns=2)

The maximum value of 1315 mm² corresponds to a two section stirrup of ϕ = 8mm every 75 mm.

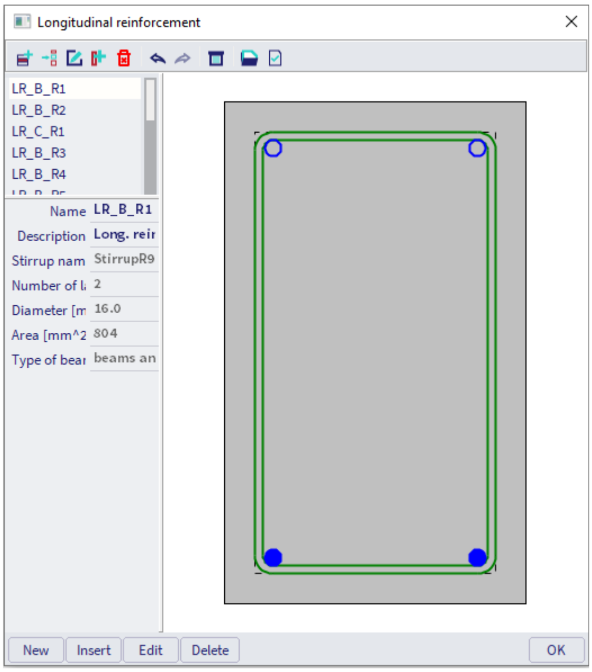
2.2.4 Practical reinforcement

We will now pass on to the level of practical reinforcement. This will allow us to specify the reinforcement locally over the beam.

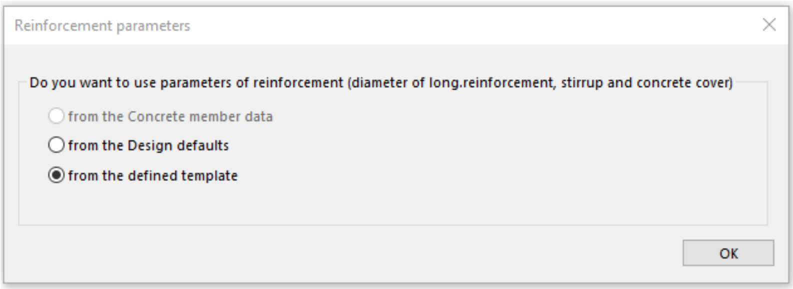
In the theoretical reinforcement design, we have calculated where reinforcement is needed.

This allows us to input manually the practical reinforcement by adding New reinforcement for the whole length of the beam.

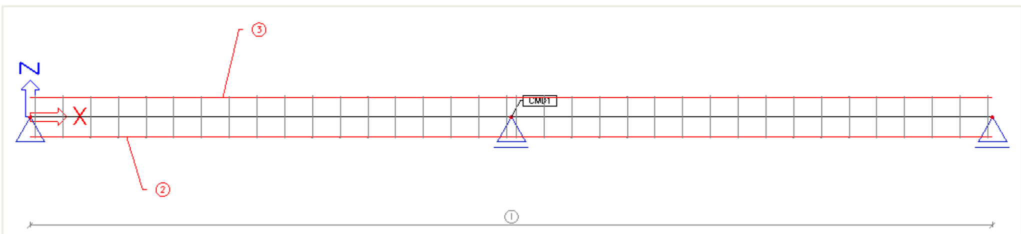
We can first select a template for the longitudinal reinforcement:



Next, we have to decide where the parameters of reinforcement are coming from:

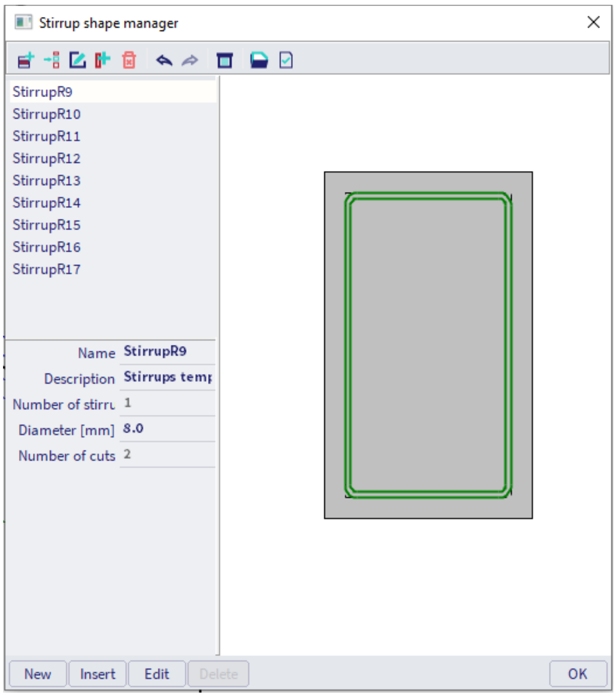


The practical reinforcement is shown graphically on the screen:

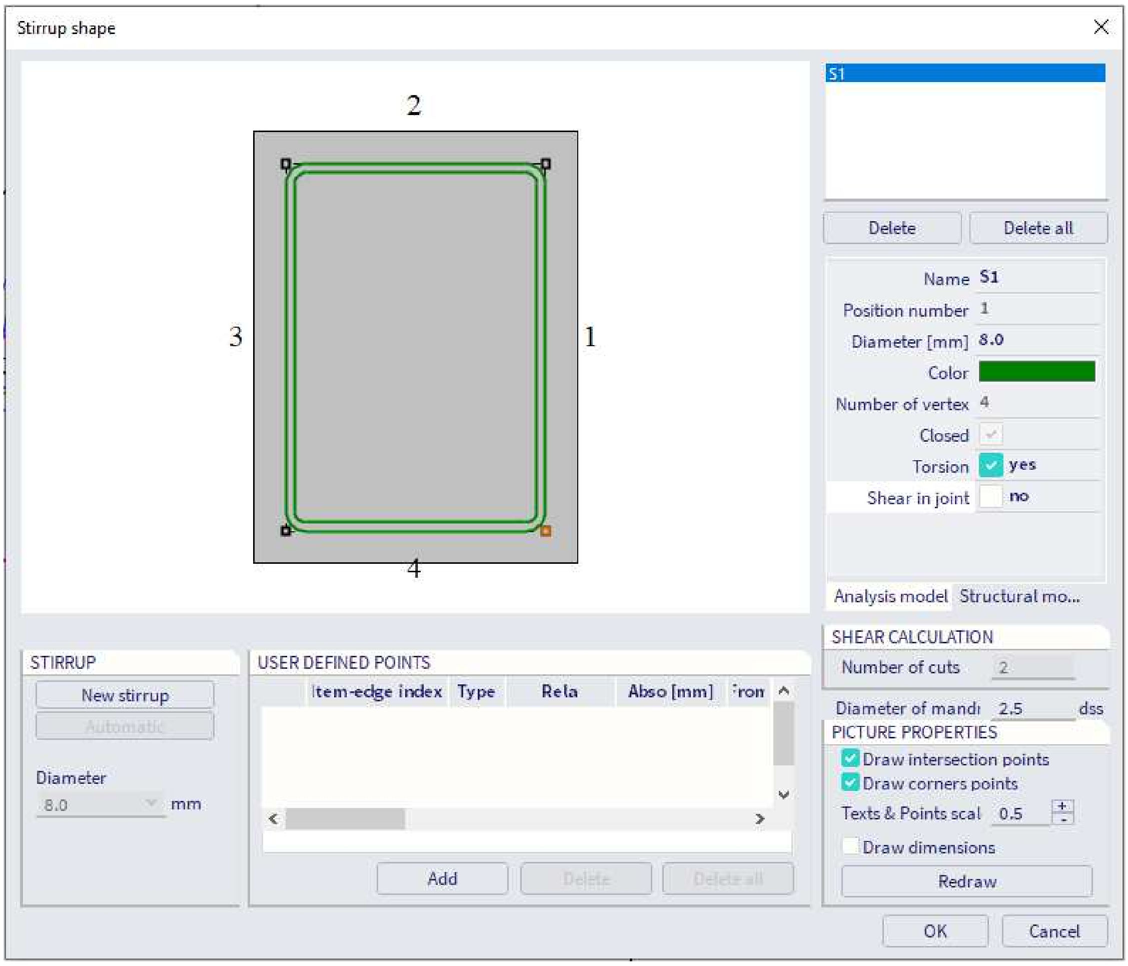


As a user, you can add locally New stirrups or New longitudinal bars.

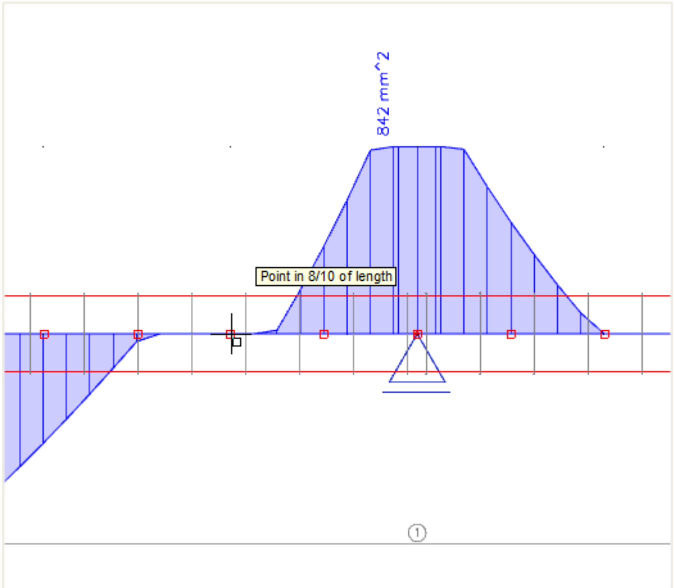
For the stirrups, you can select a certain stirrup shape:



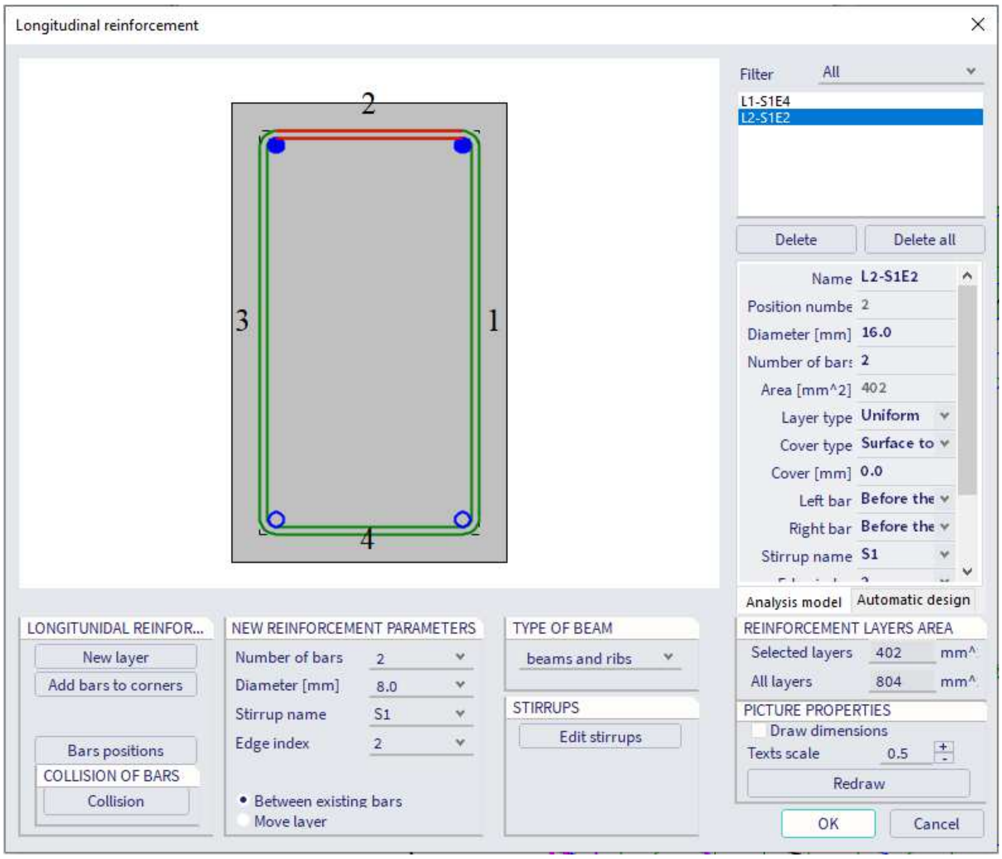
The stirrup shape can be edited or a new one can be made. Therefore user points may be added.



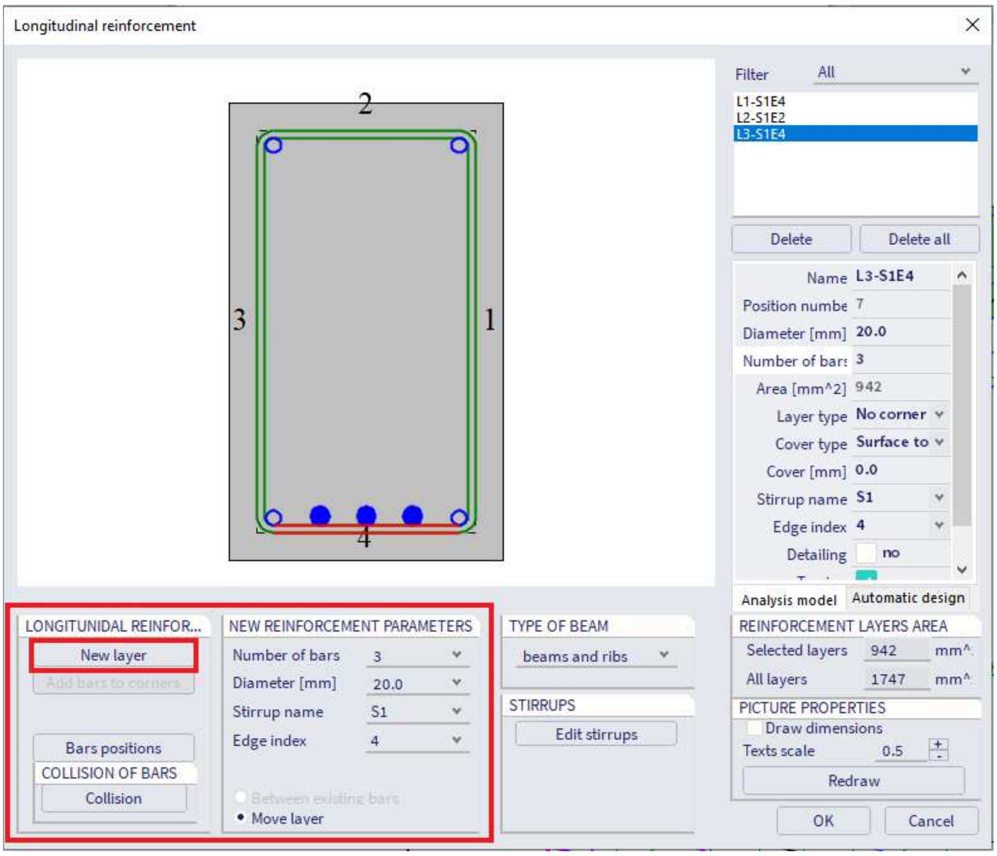
For the longitudinal reinforcement, we can define precisely where the extra practical reinforcement needs to be putted:



The selected zone of the member can be modified by the properties panel or by the menu Library / Concrete, Reinforcement / Longitudinal Reinforcement Library:



Here can be set on which face extra reinforcement needs to be added:

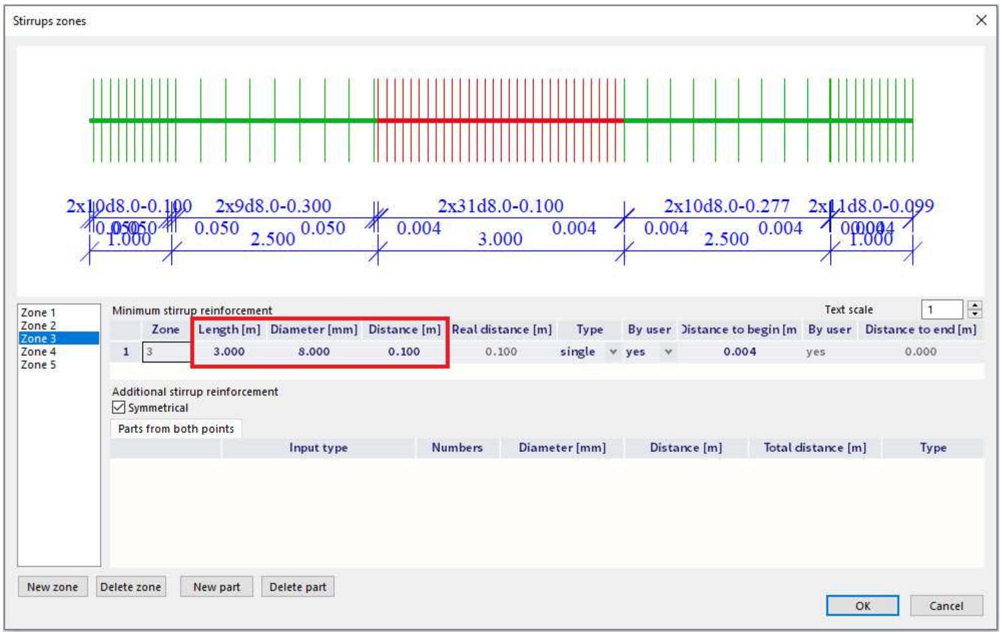


For reasons of simplicity, we will add 3 bars of 20 mm that are still needed over the whole area where extra reinforcement is required. This can of course be done more detailed.

The same procedure will be repeated for the upper reinforcement over the support.

Also, the shear reinforcement needs to be increased in the zones over the support. This can be done by increasing the diameter of the stirrups or by decreasing the distance between the stirrups.

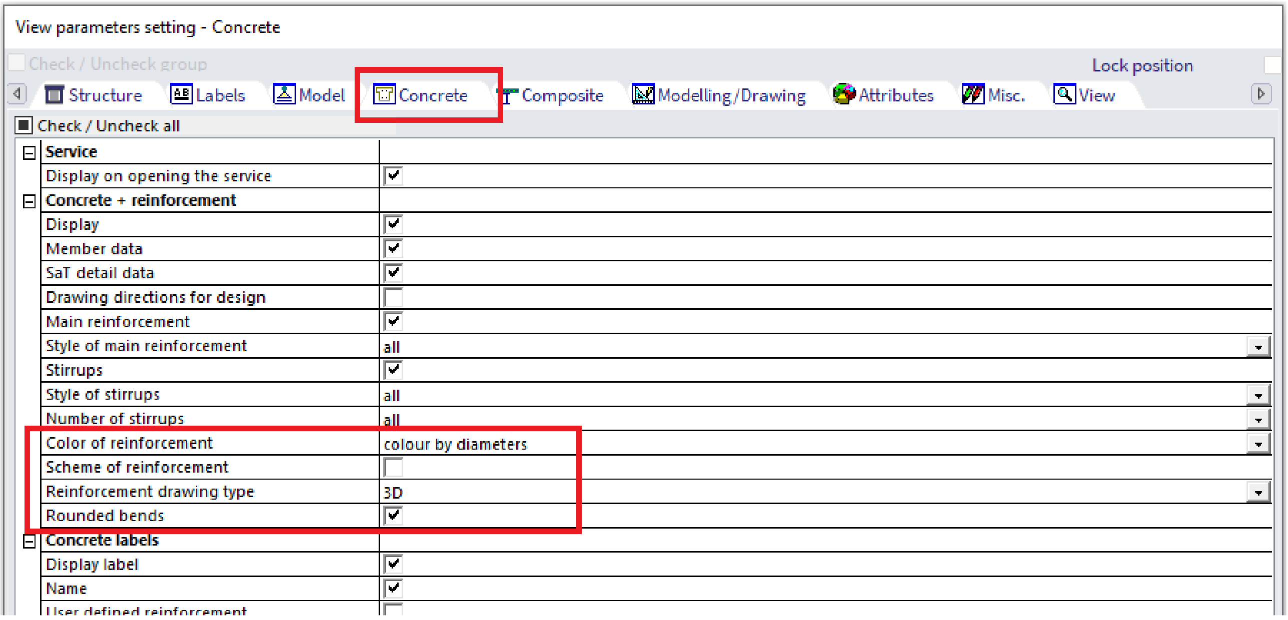
Different stirrup zones can be created:



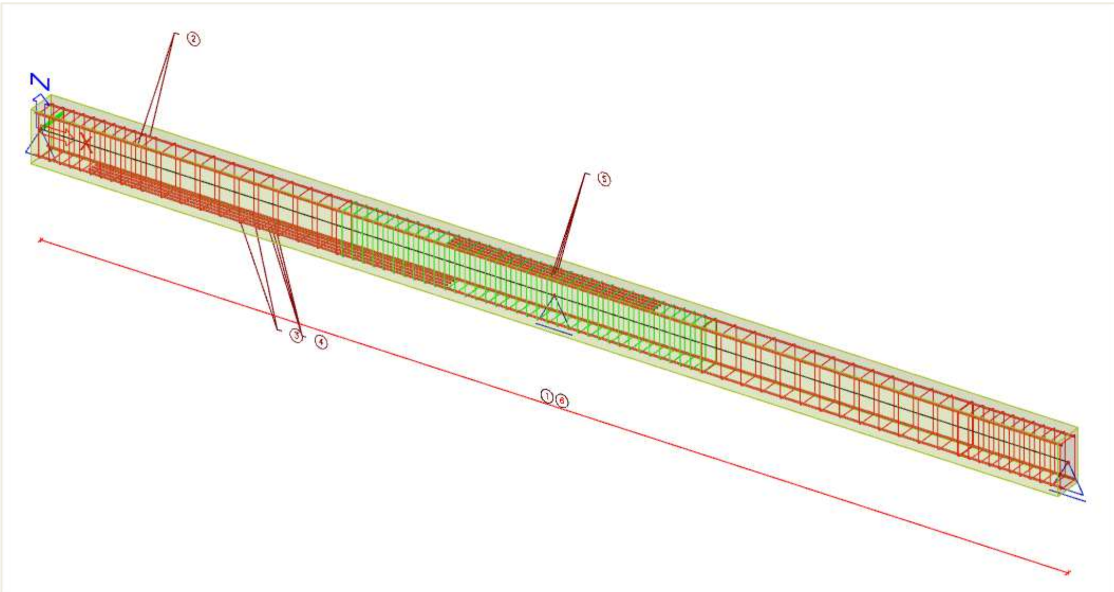
To check if there is enough shear reinforcement, a capacity check needs to be performed. This will be explained in the next chapter.

By selecting the reinforcement it is always possible to change the parameters afterwards through the property window.

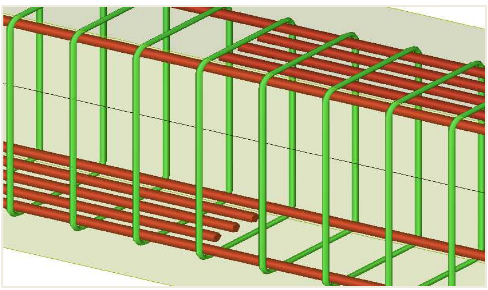
Through view parameter settings a 3D representation of the reinforcement can be obtained:



The total practical reinforcement of the beam is shown below:



A zoomed view shows the 3D representation:

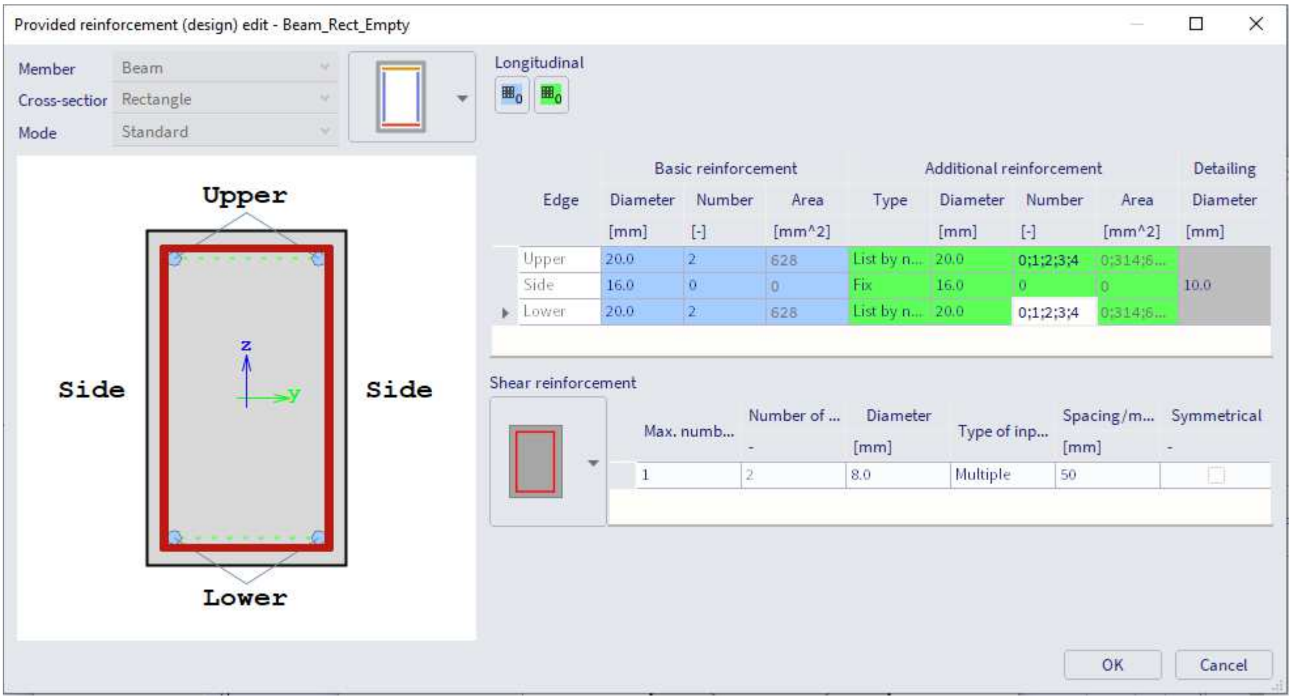


2.2.5 Conversion of theoretical reinforcement into practical reinforcement

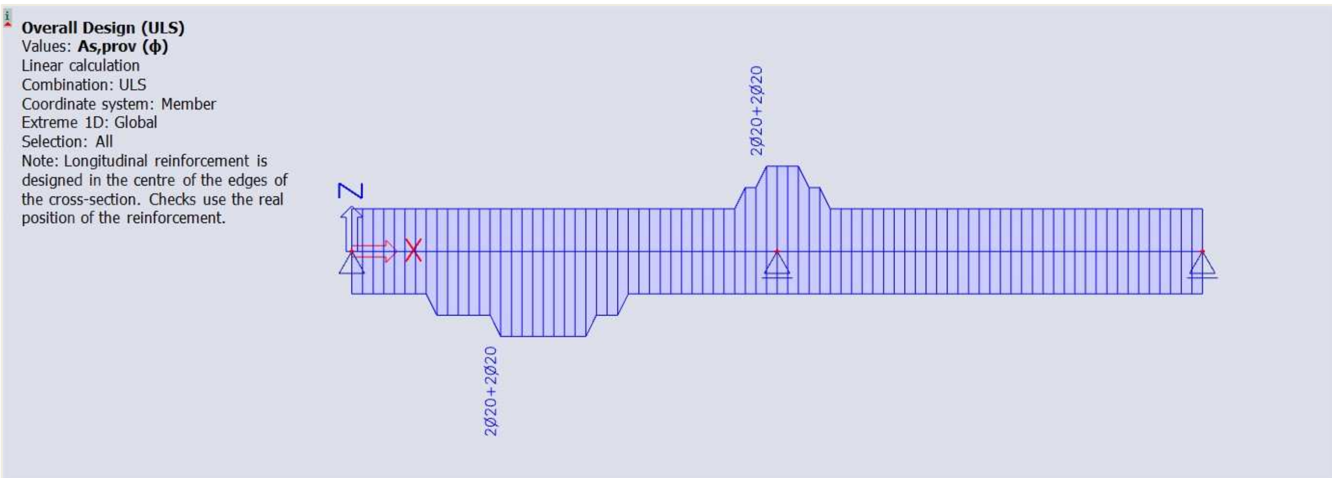
Since SCIA Engineer 19 it is also possible to convert theoretical reinforcement into practical reinforcement. As mentioned in previous chapter there are two types of theoretical reinforcmeent: **Required reinfocement** (= mm² necessary in each section) and **Provided reinforcement** (= template of reinforcement with various ammounts of additional reinforcement possible). It is only possible to convert **Provided reinforcement** into practical (=user) reinforcement.

Let's have a look at this example : open beam.esa

Set the template of provided reinforcement.

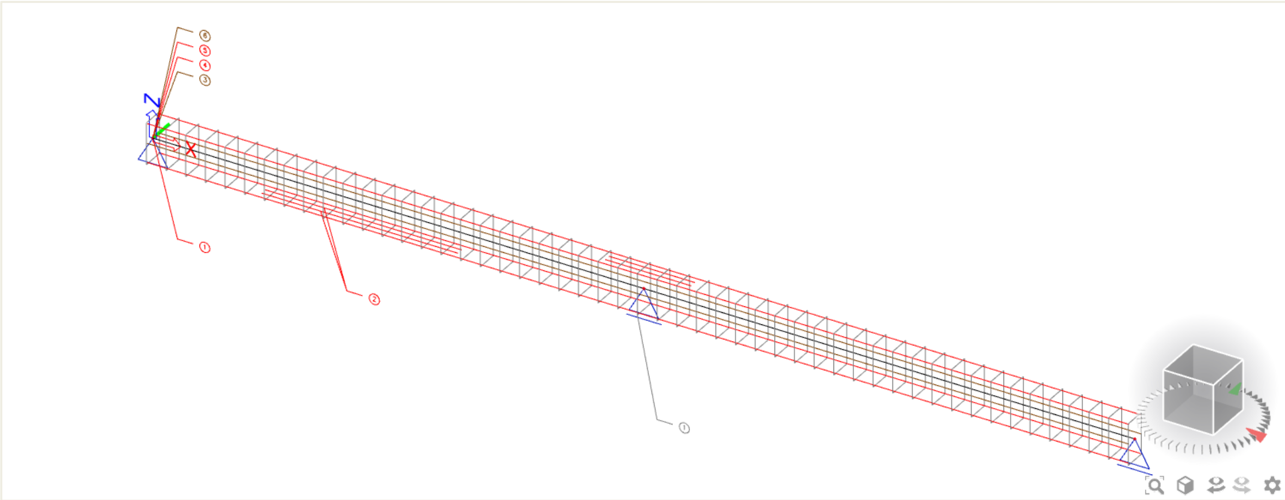


Go to Reinforcement design and look at the value As,prov(ϕ). This is the provided reinforcement that will be converted into practical reinforcement.



Press 'Conversion for real bars'

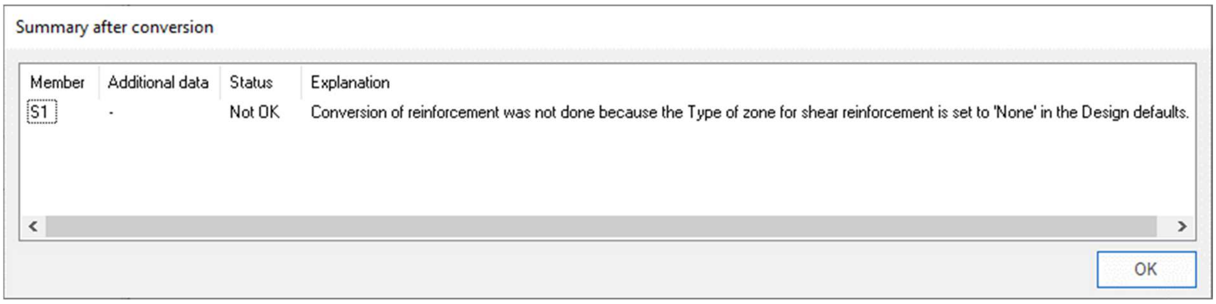
The following reinforcement is generated.



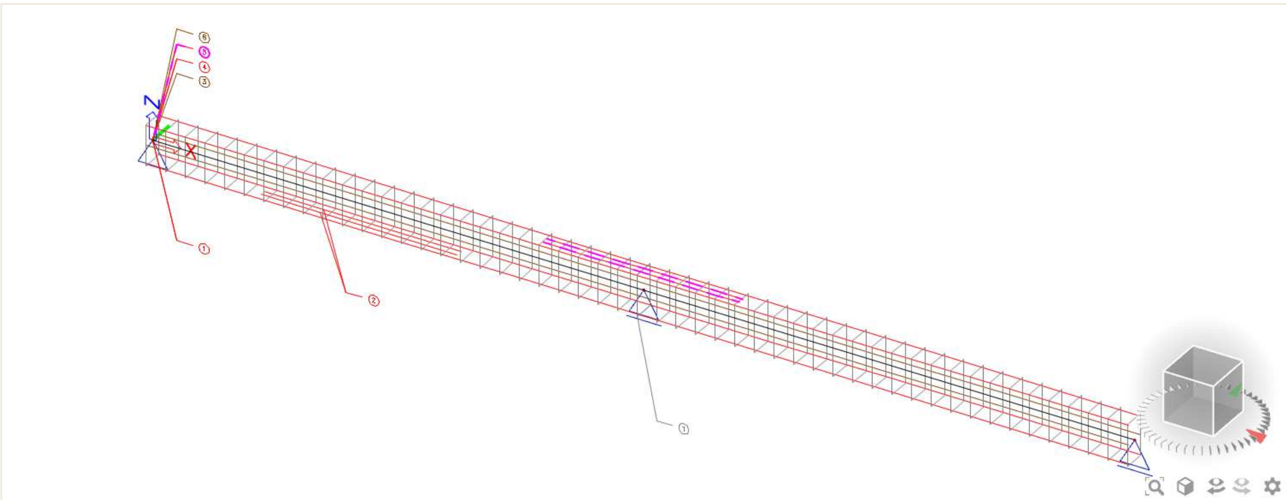
The practical reinforcement is added as reinforcement data. You can edit the reinforcement by selecting it and then click on ‘Edit reinforcement’.

Now the parts of the reinforcement that needs edditing can be slected. The diameter, number of bars, length, spacing , … can be changed in the properties window.

<u>Remark:</u>
It might occur the error message ‘**Conversion of reinforcement was not done because the Type of zone of shear reinforcement is set to ‘None’ in the Design defaults’** appears within the summary after conversion when converting the provided reinforcement into real bars. This behaviour is caused due to the option ‘**None**’ is selected for the setting ‘**Type of zone for corrected shear reinforcement**’ within the design defaults.



In the example, we will increase the length and diameter of reinforcement area 5.



Note that the converted reinforcement cannot be put together with the practical user reinforcement on the same element. You can either input your own reinforcement template using the first option or convert the reinforcement template using the second option. In general, it is advisable to use the first option for elements with a more difficult layout of the reinforcement (multiple reinforcement layers) as this is easier to adjust. For elements that have a very similar reinforcement layout, the converted reinforcement can be a useful tool.

2.2.6 Checks

In SCIA Engineer, checks can be performed in three different ways:

1. With practical reinforcement inputted on the member, checks can be done one by one for all sections of the member
2. With practical reinforcement inputted on the member, overall ULS or SLS checks can be done for a specific section of the member with the tool "Section check"
3. Without practical reinforcement, overall ULS or SLS checks can be done for a specific section of the member with the tool "Section Check". Reinforcement will then be added locally in the Section check tool to be able to perform the various available checks.

First you get an overview of the input data for the checks:

- **Internal forces:** displaying the characteristic and design values
- **Slenderness:** determining if 2^{nd} order effects need be considered (for member type 'column')
- **Stiffnesses:** displaying the values EA, EI_y and EI_z

Available checks at the Ultimate Limit State are:

- **Capacity check:** for N-My-Mz interaction - based on resistance calculated from interaction diagram
- **Response check:** based on check of ultimate stresses and strains for N-My-Mz interaction
- **Check of shear and torsion**
- **Check of interaction of shear, torsion, bending and normal force**

Available checks at the Serviceability Limit State are:

- **Stress limitation** (for concrete as well as reinforcing steel)
- **Crack width limitation**
- **Simple check for deflection:** based on calculation of stiffness ratio, without necessity to calculate Code Dependent Deflection (CDD)

There is also an Overall check available. This will simply check all the checks you have activated, but for a more detailed report, you will need to go into the check itself.

The capacity, response and shear + torsion check should be okay if no additional reinforcement is required.

However, these checks give interesting information on the efficiency of reinforcement. For instance, if in a section only 50% of reinforcement is used, then we can conclude that here less reinforcement would have been sufficient.

The detailing provisions and the crack limitation are extra checks that are not accounted in the reinforcement design. If these checks are not okay, then the practical reinforcement needs to be changed.

In the following chapters, we will explain the checks one by one when practical reinforcement is inputted. It corresponds to the 1^{st} method to perform a check (see above).

Example 1: beam_practical reinforcement.esa

The last chapter will be focused on the Section check tool, corresponding to 2^{nd} and 3^{rd} methods to perform a check (see above).

Example 2: beam_without practical reinforcement.esa

CAPACITY RESPONSE

The Capacity - response is based on the calculation of strain and stress in a particular component (concrete fibre or reinforcement bar).

The check consists of the comparison of those strains and stresses with the limited values according to EN 1992-1-1 requirements.

However, this method does not calculate extremes (capacities of the cross-section) like the interaction diagram but calculates the state of equilibrium for that section (response).

For capacities of the member, please refer to the "Capacity – diagram" check.

The following checks are performed:

- Check of compressive concrete (cc)
- Check of compressive reinforcement (sc)
- Check of tensile reinforcement (st)

The Unity Check, UC, displayed on the screen will be the maximum value of those 3 checks.

Example: beam_practical reinforcement.esa

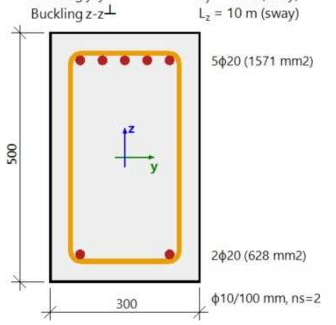
Run the Capacity – Response check in Design > Concrete 1D > ULS response check.

The maximum value of the check is given on the middle support. The Standard output gives:

Beam S1 — **RECT (500; 300)**

EC EN 1992-1-1:2004/AC:2008 — Section 26 [dx = 5 m]

Member length: L = 10 m
Buckling y-y⊥ L_y = 10 m (sway)
Buckling z-z⊥ L_z = 10 m (sway)



Concrete: C30/37
Bi-linear stress-strain diagram
Exposure class: XC3
Longitudinal reinforcement: B 500A
Bi-linear with an inclined top branch
7ϕ20 mm (A_s = 2199 mm^2)
ρ_l = 1.466 % (17.3 kg/m)
Shear reinforcement: B 500A
Bi-linear with an inclined top branch
ϕ10/99.7 mm (n_s = 2) (A_{sw} = 157 mm^2)
ρ_w = 1.051 % (12.4 kg/m) (A_{swm} = 1576 mm^2/m)
Cover (stirrup)
Top: 36 mm
Bottom: 36 mm
Left: 36 mm
Right: 36 mm

Summary of check

Type of component	Fibre / Bar	ε_{extr} [‰]	σ_{extr} [MPa]	Check strain [-]	Check stress [-]	UC [-]	Limit [-]	Status
Concrete	1	-1.63	-18.7	0.47	0.93	0.95	1	OK
Reinf.	1	2.17	434	0.10	0.95			

In the Standard output you can read the UC, and the extreme strain and stress in the studied section.

In the Detailed output you will get all the strains and stresses and the limit strains and stresses:

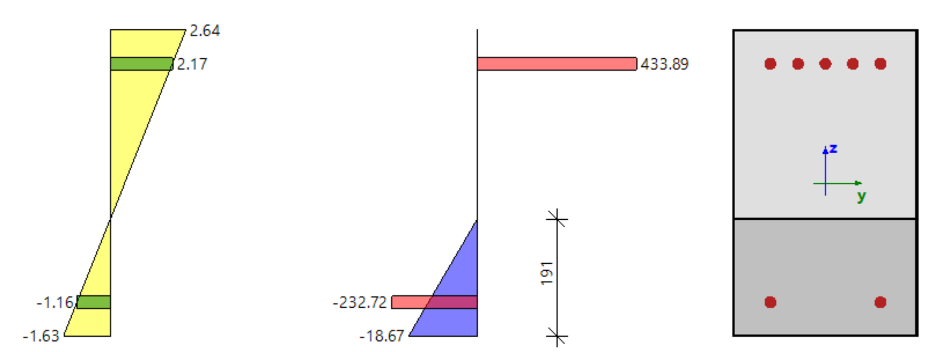
Extreme values of stress/strain in component

Type of component	Fibre / Bar	ε [‰]	ε_{lim} [‰]	σ [MPa]	σ_{lim} [MPa]	UC [-]	Status
Concrete - compression	1	-1.63	-3.5	-18.7	-20	0.93	OK
Concrete - tension	3	2.64	0	0	0	0.00	OK
Reinforcement - compression	3	-1.16	-22.5	-233	-454	0.51	OK
Reinforcement - tension	1	2.17	22.5	434	454	0.95	OK

Note that the tensile stress in concrete is not considered, therefore the corresponding UC is 0.

Stress and strain diagrams are also available in the Detailed output:

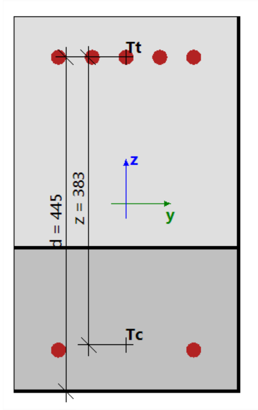
Stress and strain distribution



Settings that might influence the check:

- **Effective depth of cross-section - d**

It is usually defined as distance of the most compressive fibre of concrete to center of gravity of tensile reinforcement. In SCIA Engineer, the effective depth of cross-section is defined as distance of the most compressive fibre of concrete to position resultant of forces in tensile reinforcement.



The effective depth d cannot be calculated in the following cases:

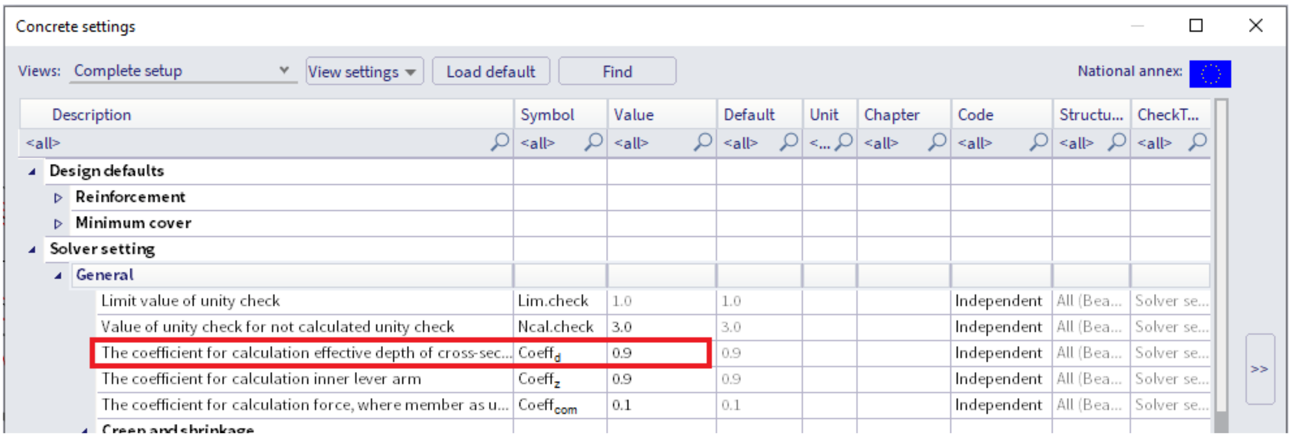
- The most compressive fibre cannot be determined (the whole cross-section is in tension)
- Resultant of forces in tensile reinforcement cannot be determined (whole section is in compression)
- Equilibrium is not found
- Distance of the most compressive fibre and Resultant of forces in tensile reinforcement is less than 0,5*h

In those cases, the effective depth is calculated according to formula :

$$d = Coeff_d * h_l$$

With:

- $Coeff_d$ by default 0,9 in Concrete settings, in “Complete Setup” view, and in “Solver settings” / “General”
- h_l height of cross-section perpendicular to neutral axis



- **Inner lever arm**

z is defined in EN 1992-1-1, clause 6.2.3 (3) as the distance between position resultant of tensile force (tensile reinforcement) and position of resultant of compressive force (compressive reinforcement and compressive concrete).

The inner lever arm cannot be calculated in the following cases:

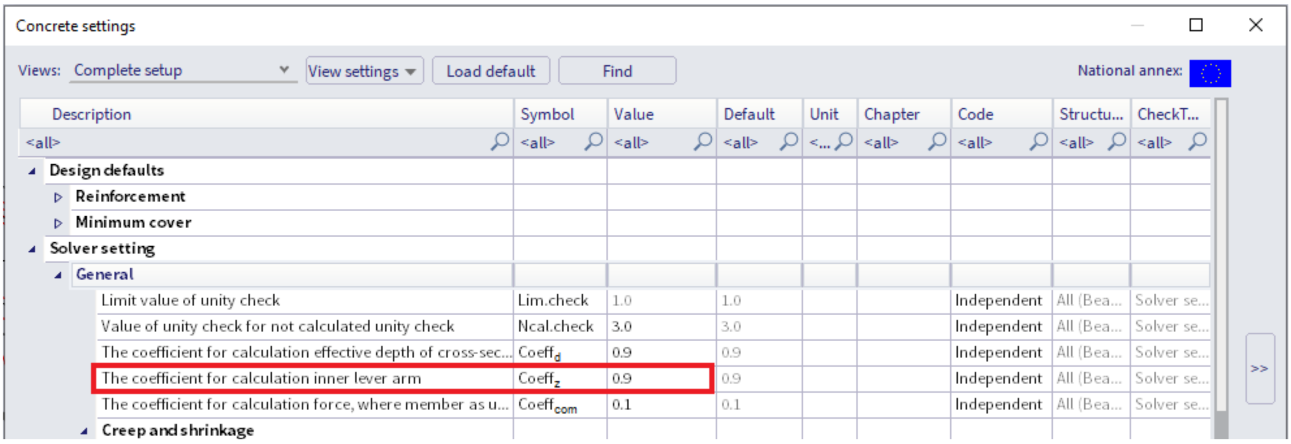
- The most compressive fibre cannot be determined (the whole cross-section is in tension)
- Resultant of forces in tensile reinforcement cannot be determined (whole section is in compression)
- Equilibrium is not found

In those cases, it is calculated according to formula:

$$z = Coeff_z * d$$

With:

- $Coeff_z$ by default 0,9 in Concrete settings, in “Complete Setup” view, and in “Solver settings” / “General”



For additional information about this check and the theoretical background, please refer to our web help.

CAPACITY DIAGRAM

Capacity - diagram services uses the creation of interaction diagram (graph presenting the capacity of a concrete member to resist a set of N + My + Mz).

This check calculates the extreme allowable interaction between the normal force N and bending moments My and Mz.

Example: beam_practical reinforcement.esa

Run the Capacity – Diagram check in Design menu > Concrete 1D > ULS capacity diagram check

The standard output gives the summary result of the check:

Summary of check

N	N_{Ed}	N_{Rd+}	M_y	M_{Edy}	M_{Rdy+}	M_{Rdy-}	UC	Status
		N_{Rd-}	M_z	M_{Edz}	M_{Rdz+}	M_{Rdz-}		
[kN]	[kN]	[kN]	[kNm]	[kNm]	[kNm]	[kNm]	[-]	
0	0	0	-261	-261	119	-278	0.939	OK
		0	0	0	0	0		M_{Edz}/M_{Rdz}

The Detailed output gives additional info about how the check is performed:

Summary of check

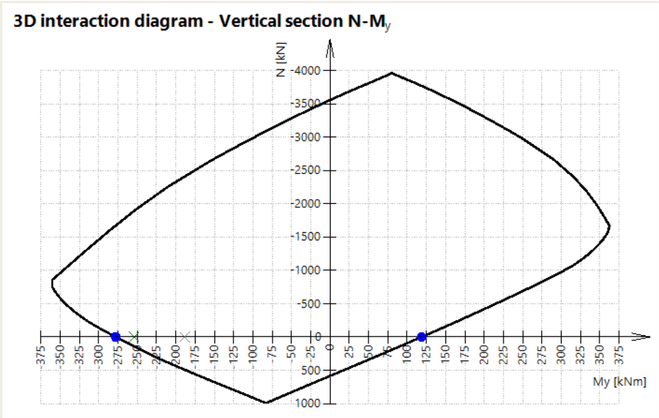
Forces: N_{Ed} = 0 kN M_{Edy} = -261 kNm M_{Edz} = 0 kNm

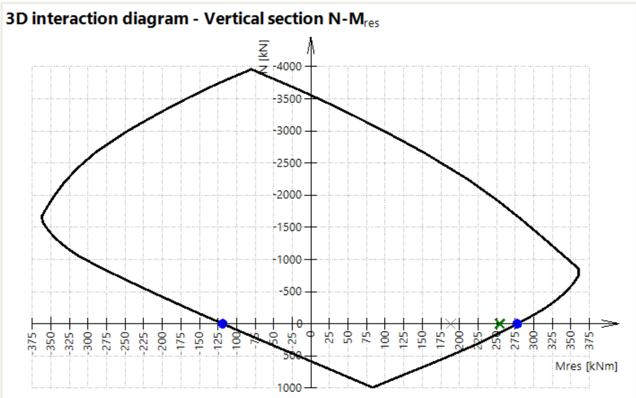
Resistance: N_{Rd} = 0 kN M_{Rdy} = -278 kNm M_{Rdz} = 0 kNm

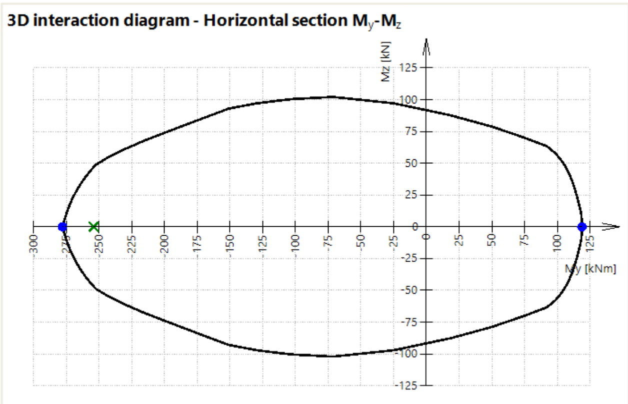
Calculation of unity check

$$UC = \frac{\sqrt{N_{Ed}^2 + M_{Edy}^2 + M_{Edz}^2}}{\sqrt{N_{Rd}^2 + M_{Rdy}^2 + M_{Rdz}^2}} = \frac{\sqrt{0^2 + -261^2 + 0^2}}{\sqrt{0^2 + -278^2 + 0^2}} = 0.939 \quad <= 1 \quad \text{OK}$$

Interaction diagrams are also drawn in the Detailed output:







Settings that might influence the check:

- **Interaction diagram method**
- **Division of strain**
- **Number of points in vertical cuts**

For additional information about this check and the theoretical background, please refer to our web help.

SHEAR + TORSION

Check of Interaction shear and torsion consists of three checks according to clause 6.1 - 6.3 in EN 1992-1-1:

- check of shear
- check of torsion
- check of interaction of shear and torsion

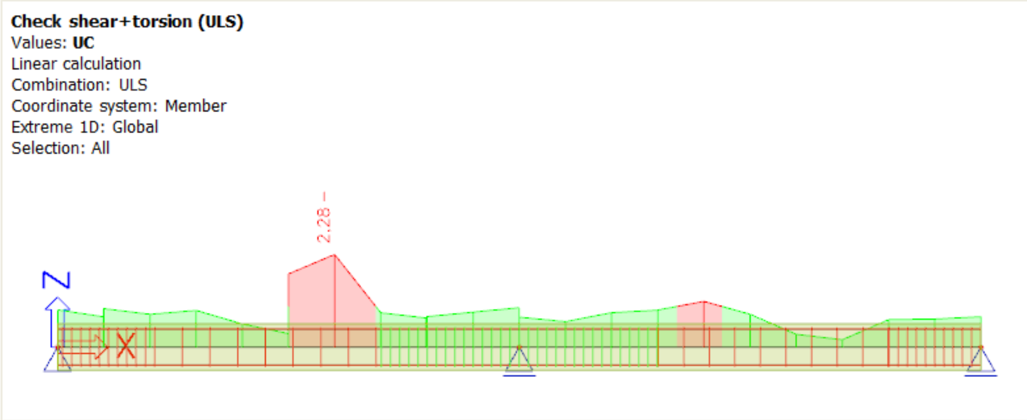
This check can be performed if the following conditions are met:

- The material of all reinforcement bars and stirrups are the same
- The angle between gradient of the strain plane and the resultant of shear forces is not greater than 15°
- Cross-section with one polygon and one material

Example: beam_practical reinforcement.esa

Run the Shear + Torsion check in Design > Concrete 1D > ULS Shear and Torsion check

Some parts of the beam do not satisfy:



The Standard output allows us to identify which specific check is not satisfied:

Forces

Content of combination: 1.35*LC1+1.35*LC2+1.50*LC3

N_{Ed} = 0 kN M_{Edy} = 203 kNm M_{Edz} = 0 kNm V_{Edy} = 0 kN V_{Edz} = -152 kN T_{Ed} = 0 kNm

Resultant of shear force

$$V_{Ed} = \sqrt{V_{Edy}^2 + V_{Edz}^2} = \sqrt{0^2 + -152^2} = 152 \text{ kN}$$

Difference between angles α_M and α_V

$$\alpha_{MV} = \text{abs}(\alpha_M - \alpha_V) = \text{abs}(90 - 90) = 0\ °$$

Summary of check

d = 445 mm z = 383 mm b_w = 300 mm b_{w1} = 300 mm V_{Rdc} = 87.8 kN V_{Rds} = 66.5 kN V_{Edmax} = 705 kN V_{Rdmax} = 598 kN

Type of check	Forces	Resistances	UC [-]	Status
Check shear Vy+Vz	151.7 kN	66.5 kN	2.28	Not OK
Check torsion	0.0 kNm	0.0 kNm	0.00	OK
Interaction check Vy+Vz+T (concrete)			0.00	OK
Interaction check Vy+Vz+T (shear)	0.0 kN	0.0 kN	0.00	OK
Interaction check Vy+Vz+T (long. reinf.)	0.0 kN	0.0 kN	0.00	OK
Summary of check			2.28	Not OK

Here the shear forces cause a unity check >1.

In the Detailed output we can read notes, warning and errors about the design. For example, for the shear forces check not satisfied, the report clearly explains that the shear reinforcement is not sufficient and that we have to increase it.

Shear check

Check V_{Rdmax}

$$V_{Ed} = 152 \text{ kN} \le V_{Rdmax} + V_{ccd} + V_{td} = 598 \text{ kN}$$

Note: The check satisfies for crushing of the compression strut ($V_{Ed} \le V_{Rd,max} + V_{td} + V_{ccd}$).

Check V_{Edmax}

$$V_{Ed} = 152 \text{ kN} \le V_{Edmax} + V_{ccd} + V_{td} = 705 \text{ kN}$$

Note: The check satisfies for shear force near the support ($V_{Ed} \le V_{Ed,max} + V_{td} + V_{ccd}$).

Check V_{Rdc} and V_{Rds}

$$V_{Ed} = 152 \text{ kN} > V_{Rdc} = 87.8 \text{kN and } V_{Ed} = 152 \text{ kN} > V_{Rds} + V_{ccd} + V_{td} = 66.5 \text{ kN}$$

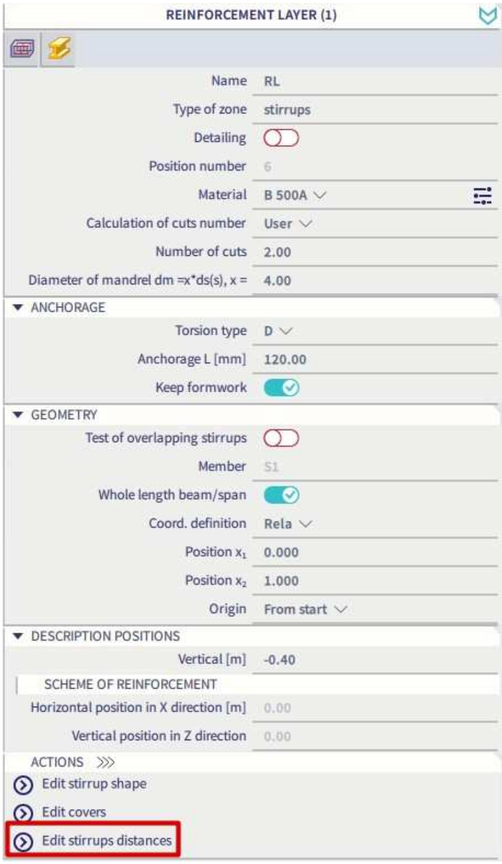
Error: The check does not satisfy, because of shear reinforcement ($V_{Ed} > V_{Rds} + V_{ccd} + V_{td}$). It is necessary to increase area of shear reinforcement or to increase dimensions of the cross-section or quality of shear reinforcement.

Unity check

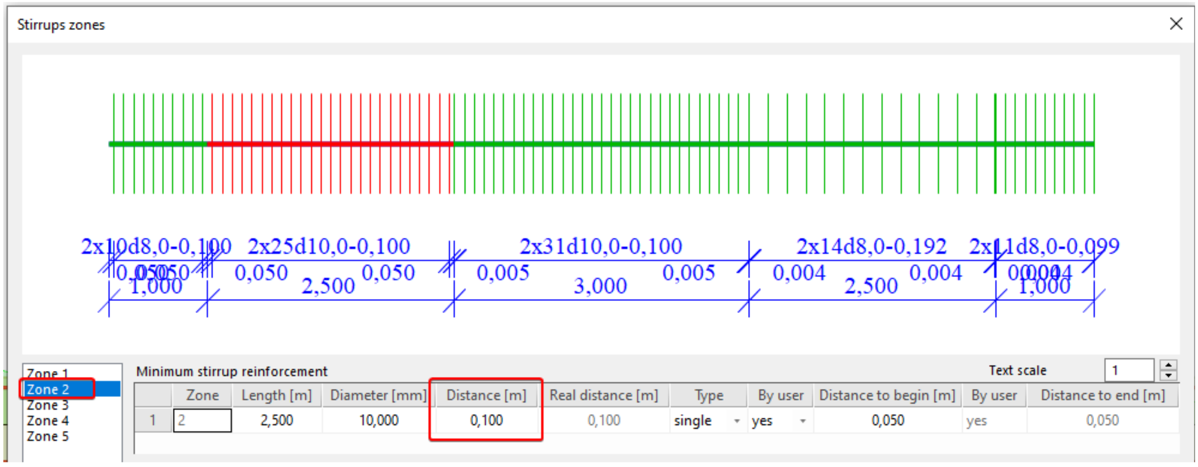
$$UC = \frac{\text{abs}(V_{Ed})}{V_{Rd}} = \frac{\text{abs}(152 \text{ kN})}{66.5 \text{ kN}} = 2.28$$

Various actions can be done to fix this issue. In this example, we choose to decrease the spacing of the stirrups in the section where there is an issue.

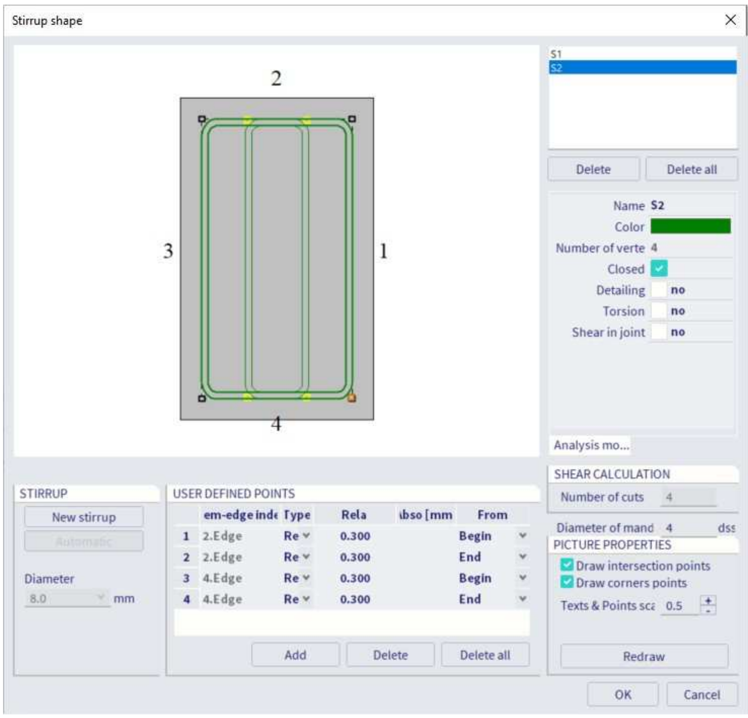
Select stirrups and click on “Edit stirrups distances” at the bottom of the Properties of the stirrup layers:



Select “Zone 2” and change the distance between stirrups from 0.3 m to 0.1 m. Apply the same procedure for “Zone 4” and modify the spacing to 0,2 m:

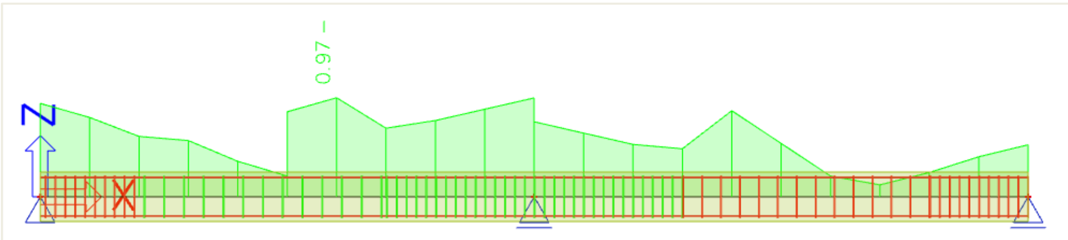


We could also have added more stirrups like below:



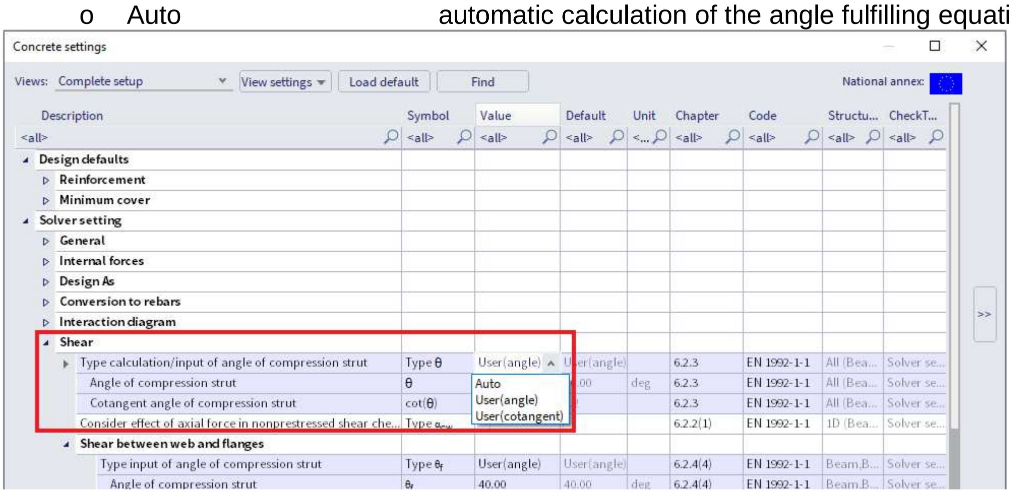
Changing the stirrup shape allows us to keep a bigger distance of 0.2m between stirrups in “Zone 2”.

After modification, the shear + torsion check is satisfied:

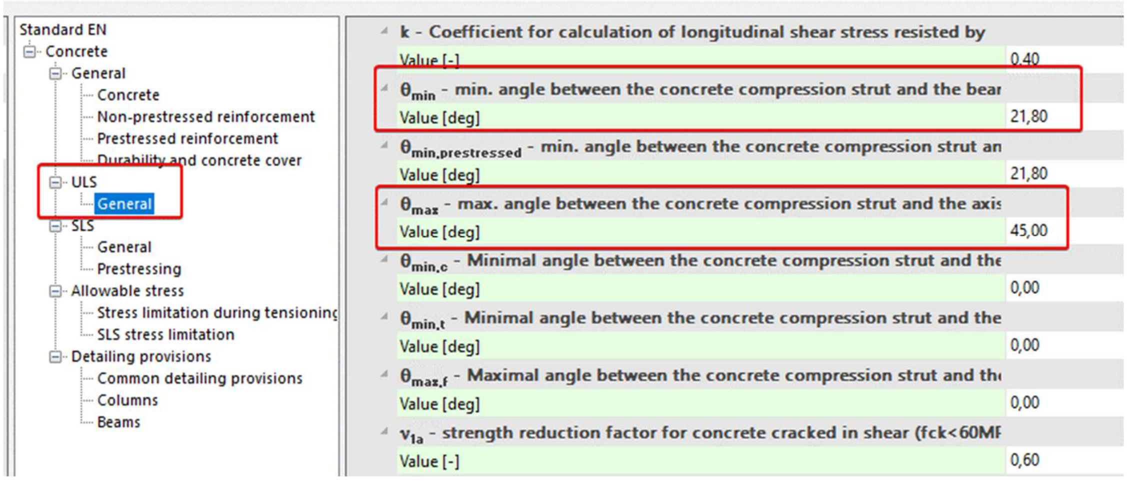


Settings that might influence the check:

- **Coefficient for calculation of effective depth of cross-section**
  Default value 0,9 in Concrete settings > Complete Setup view > Solver settings > General
- **Coefficient for calculation of inner lever arm**
  Default value 0,9 in Concrete settings > Complete Setup view > Solver settings > General
- **Angle of concrete compression strut**
  3 types of input in Concrete settings > Solver settings > Shear:
  - User (angle) user input of the angle – by default
  - User (cotangent) user input of the cotangent
  - Auto automatic calculation of the angle fulfilling equation 6.29



The angle should be between θmin and θmax defined in the NA for EN1992-1-1.



- **Angle of shear reinforcement**
  Practical reinforcement can only be introduced at 90°.
- **Type for determination equivalent thin-walled cross-section**

For additional information about this check and the theoretical background, please refer to our web help.

STRESS LIMITATION

Stress limitation is based on the verification of:

- **compressive stress in concrete** - the high value of compressive stress in concrete could lead to appearance of longitudinal cracks, spreading of micro-cracks in concrete and higher values of creep (mainly nonlinear). This effect can lead to a state where the structure is unusable.
- **tensile stress in reinforcement** - stress in reinforcement is verified due to limitation of unacceptable strain existence and thus appearance of cracks in concrete.

Example: beam_practical reinforcement.esa

The stress limitation check is done according to the following steps:

- Verification of crack appearance
- Verification of the stresses

The Standard output shows those 2 steps:

Verification of cracks in cross-section

Load	Type of module	E_c [MPa]	Combi.	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	σ_{ct} [MPa]	h [mm]	$f_{ct,eff}$ [MPa]	Cracks appear
Short	E_c	0	Char.	0	-188	0	12.6	500	2.9	YES

Stress limitation in concrete

Check type	Load	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	y_i [mm]	z_i [mm]	σ_c [MPa]	$\sigma_{c,lim}$ [MPa]	$\sigma_c/\sigma_{c,lim}$ [-]	Status
§7.2(2) Char.	Short	0	-188	0						OFF
§7.2(3) Q.-P.	Short	0	-188	0	0.15	-0.25	-21.2	-13.5	1.57	Not OK

Stress limitation in non-prestressed reinforcement

Check type	Load	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	y_i [mm]	z_i [mm]	σ_s [MPa]	$\sigma_{s,lim}$ [MPa]	$\sigma_s/\sigma_{s,lim}$ [-]	Status
§7.2(5) Char.	Short	0	-188	0	0.09	0.2	300	400	0.75	OK

Verification of crack appearance

Crack appearance is verified for characteristic load combination in accordance to chapter 7.1(2) in EN1992-1-1:

- **$\sigma_{ct} \leq f_{ct,eff}$ no crack appears**
- **$\sigma_{ct} > f_{ct,eff}$ crack appears**

With:

- σ_{ct} maximal tensile stress in concrete fibre
- $f_{ct,eff}$ effective concrete tensile strength

Verification of stresses

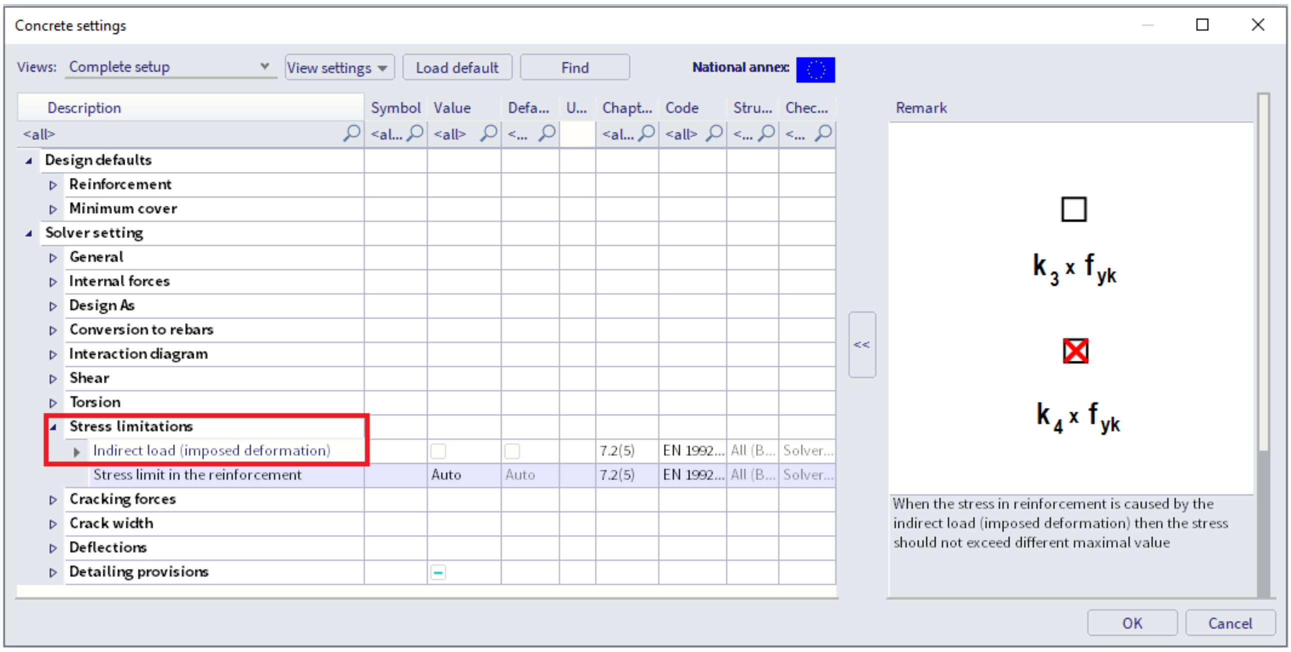
There are 3 stress limitations checked:

- $\sigma_{c,char,lim} \leq k_1 * f_{ck}$ concrete stress under Char. load – 7.2(2) - exposure classes XD, XF, XS
- $\sigma_{c,qp,lim} \leq k_2 * f_{ck}$ concrete stress under Quasi Perm. load – chapter 7.2(3)
- $\sigma_{s,char,lim} \leq k_3 * f_{yk}$ reinforcement stress under Char. Load – chapter 7.2(5)

Values of k_1, k_2, k_3, are defined in the NA, standard values are respectively 0.6, 0.45, 0.8

Additionally, when the stress in the reinforcement is caused by an imposed deformation, then the maximal strength is increased to $k_4 * f_{yk}$, where k_4 is NA parameter with standard value $k_4 = 1,0$.

This option can be activated in Concrete settings > Stress limitations:



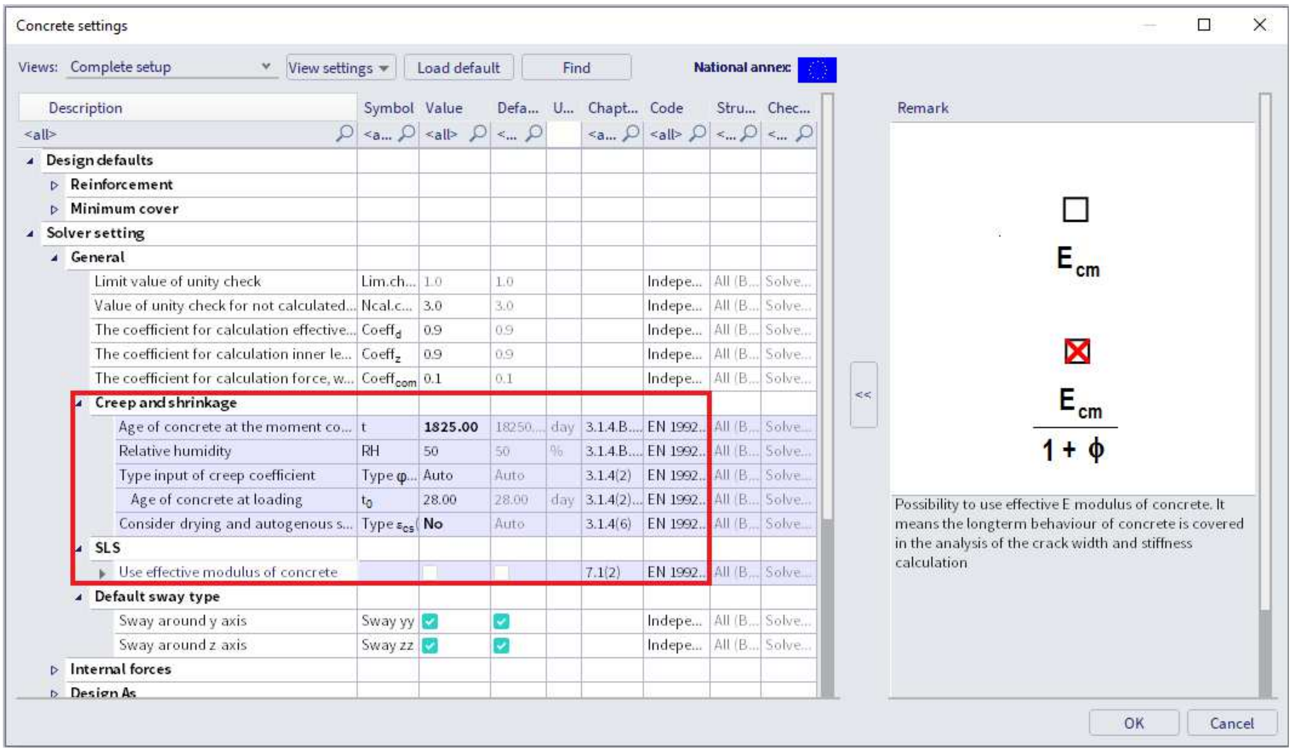
By default, stress limitation check is done for short-term state.

It is possible to perform a long-term state. Effective E modulus of elasticity is calculated as follows, using the creep coefficient:

$$E_{c,eff} = E_{cm} / (1+\varphi)$$

Long-term behaviour can be activated in Concrete Setting > Complete Setup view > Solver settings > General > SLS > Use effective modulus of elasticity.

The creep coefficient can whether be calculated by the software or inputted manually in the Concrete settings.



<u>Note:</u> SCIA Engineer is not able to use characteristic or quasi-permanent combinations together in one step. Therefore, the same forces (load combination) are used for crack appearance and final stress values.

CRACK WIDTH

The crack width is calculated according to clause 7.3.4 in EN 1992-1-1.

The following preconditions are used for calculation:

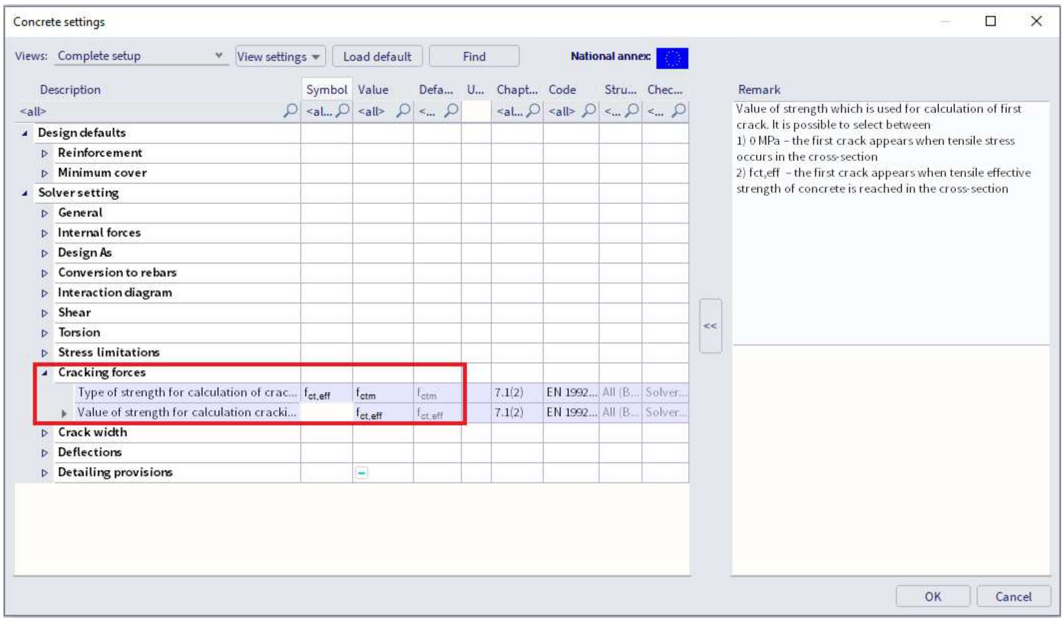
- The crack width is calculated for beams and columns and for general loads (N + My + Mz)
- Cross-section with one polygon and one material is considered in version SEN 17
- The material of all reinforcement bars must be the same in SEN 17
- Appearance of cracks should be calculated for a characteristic combination according to EN 1992-1-1, clause 7.2(2). A simplification is made in SEN 17 that the normal stress is calculated for the same type of combination as used for the calculation of crack width, inputted in service Crack control.

Example: <u>beam_practical reinforcement.esa</u>

First a determination whether the section is cracked or un-cracked is performed by comparing:

- $\sigma_{ct} \leq \sigma_{cr}$ Uncracked
- $\sigma_{ct} > \sigma_{cr}$ Cracked

Value for σ_{cr} can be set in the Concrete settings > Cracking forces. Two options can influence this value:



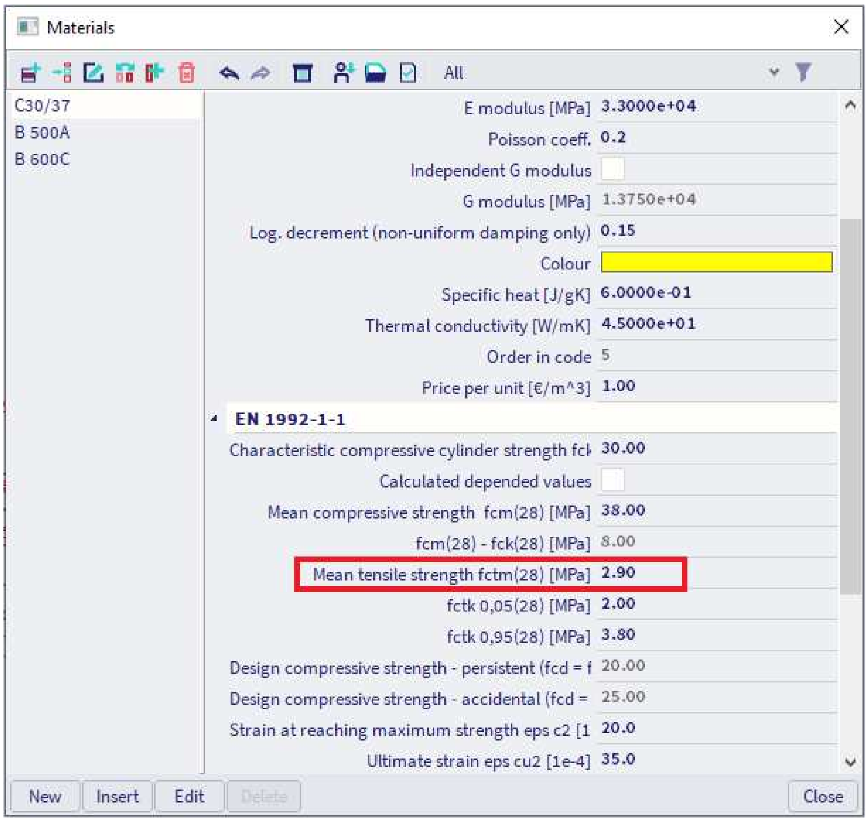
Value of strength for calculation of cracking forces:

- σ_{cr} = 0 MPa cracks appear when tensile stress occurs in the section
- $\sigma_{cr} = f_{ct,eff}$ cracks appear when tensile effective strength of concrete is reached in the section

Type of strength for calculation of cracking forces:

If previous option is set on $\sigma_{cr} = f_{ct,eff}$, which is the default value then:

- $f_{ct,eff} = f_{ctm}$ mean tensile strength of concrete at 28 days set in the material properties.
- $f_{ct,eff} = f_{ctm,fl}$ mean flexural tensile strength (EN 1992-1-1,clause 3.1.8(1)). This value should be used if restrained deformations such as shrinkage or temperature movements are considering for calculation crack width.



Note: The value presented in material properties (picture above) is the mean tensile strength at 28 days. If cracking is expected earlier than 28 days, it is necessary to input this value $f_{ctm}(t)$ into the material properties (EN 1992-1-1,clause 3.1.2(9)).

The check of crack appearance, with values of cracking forces (N_{cr}, M_{cry}, M_{crz}) can be read in the Detailed output:

Material characteristics

Effective strength of concrete:

$f_{ct.eff} = f_{ctm} = 2.9$ MPa

Strength in concrete , when crack is appeared:

$\sigma_{cr} = 2.9$ MPa

Modulus of elasticity of concrete:

$E_c = E_{cm} = 33$ GPa

Forces

Content of combination:

LC1+LC2+LC3

Characteristic values

$N_{char} = 0$ kN $M_{y.char} = -188$ kNm $M_{z.char} = 0$ kNm

Quasi-permanent values

$N_{qp} = 0$ kN $M_{y.qp} = -188$ kNm $M_{z.qp} = 0$ kNm

Angle of bending moment resultant

$\alpha_M = -90°$

Cross-section characteristics

Type	Css-uncracked	Css cracked
t_{iy} [m]	0	0
t_{iz} [m]	$6.82 \cdot 10^{-3}$	-0.117
A_i [m^2]	0.163	0.0533
I_{iy} [m^4]	$3.63 \cdot 10^{-3}$	$1.91 \cdot 10^{-3}$
I_{iz} [m^4]	$1.19 \cdot 10^{-3}$	$370 \cdot 10^{-6}$

Calculation of cracking forces (uncracked section)

Maximal stress in concrete

$\sigma_{ct} = 12.6$ MPa

Cracking forces

$N_{cr} = 0$ kN $M_{cry} = -43.3$ kNm $M_{crz} = 0$ kNm

$\sigma_{ct} = 12.6 \text{ MPa} > \sigma_{cr} = 2.9 \text{ MPa} \Rightarrow$ Cracks appear

Note: The crack is appeared, because maximal tensile stress is greater than cracking strength.

Here, modulus E is taken for short-term state. As mentioned previously, long-term state with an effective modulus E_{eff} can be chosen in Concrete settings > Complete Setup view > General > SLS > Use effective modulus E.

In this example, cracks appear.

Crack width is then calculated according to EN 1992-1-1, formula 7.8:

$$w = s_{r,max} \bullet (\varepsilon_{sm} - \varepsilon_{cm})$$

For further details about the calculation, the Detailed output can be analysed. The following picture shows only a part of the report:

Maximum crack spacing

$s_{max} = 45 \text{ mm} \le 5^*(c+0.5^*\phi_{eq}) = 275 \text{ mm or } \rho_{p.eff} = 0,$ therefore:

$$s_{r.max} = k_3 \cdot c + \frac{k_1 \cdot k_2 \cdot k_4 \cdot \phi_{eq}}{\rho_{p.eff}} = 3.4 \cdot 0.045 + \frac{0.8 \cdot 0.5 \cdot 0.425 \cdot 0.02}{0.0428} = 232 \text{ mm} \quad (7.11)$$

Mean strain in the reinforcement

$$\varepsilon_{sm\_cm} = \max\left(\frac{\sigma_s - k_t \cdot \left(\frac{f_{ct.eff}}{\rho_{p.eff}}\right) \cdot \left(1 + \alpha_E \cdot \rho_{p.eff}\right)}{E_s}; \frac{0.6 \cdot \sigma_s}{E_s}\right) \quad (7.9)$$

$$= \max\left(\frac{300 \cdot 10^6 - 0.46 \cdot \left(\frac{2.9 \cdot 10^6}{0.0428}\right) \cdot \left(1 + 6.06 \cdot 0.0428\right)}{200 \cdot 10^9}; \frac{0.6 \cdot 300 \cdot 10^6}{200 \cdot 10^9}\right) = 1.3 ‰$$

Calculated crack width

$$w = \varepsilon_{sm\_cm} \cdot s_{r.max} = 1.3 \cdot 232 = 0.303 \text{ mm} \quad (7.8)$$

Limit value of crack width

$w_{max} = 0.4$ mm

Unity check

Calculation unity check

$$UC = \frac{w}{w_{max}} = \frac{0.303\ \text{mm}}{0.4\ \text{mm}} = \mathbf{0.757}$$

Check crack width

$\mathbf{w = 0.303\ mm = < w_{max} = 0.4\ mm}$

Note: Check crack width satisfies, because the crack width is lesser than limit value.

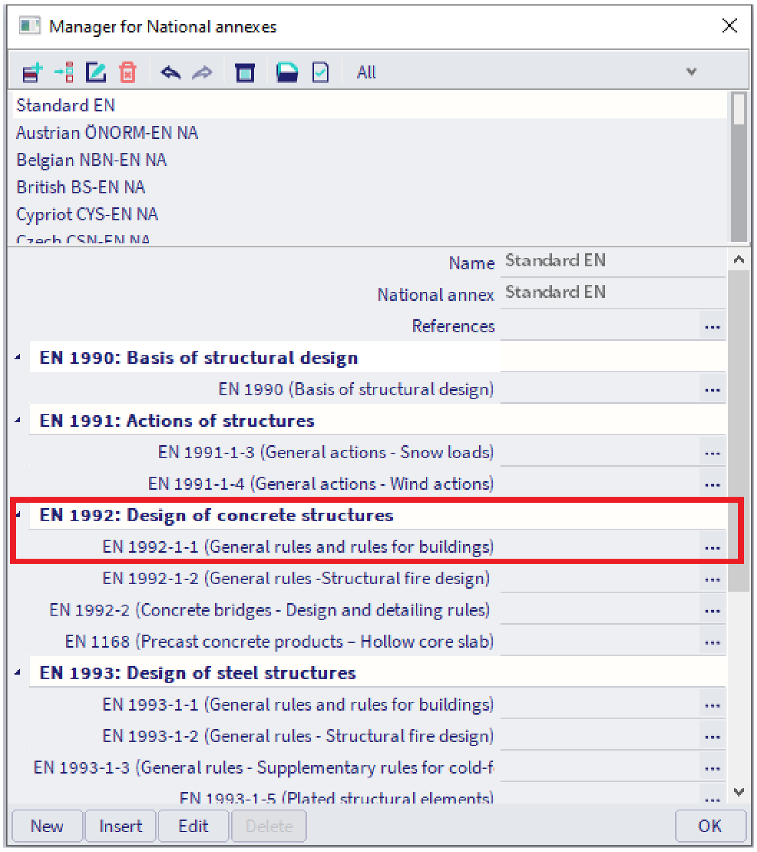
Standard output will give the summary values:

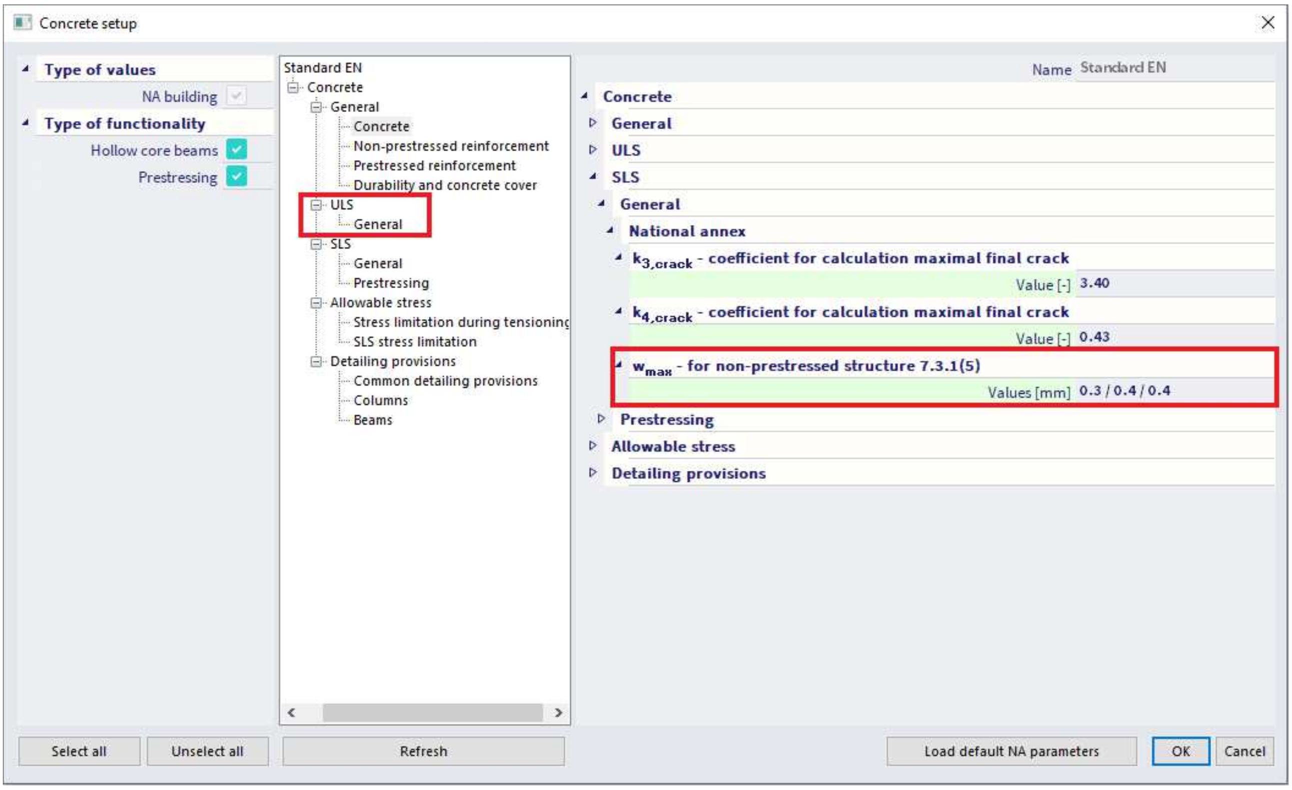
Summary of check

N_{cr} = 0 kN M_{cry} = -43.3 kN M_{crz} = 0 kN σ_s = 300 MPa $s_{r,max}$ = 232 mm ε_{sm_cm} = 1.3 ‰

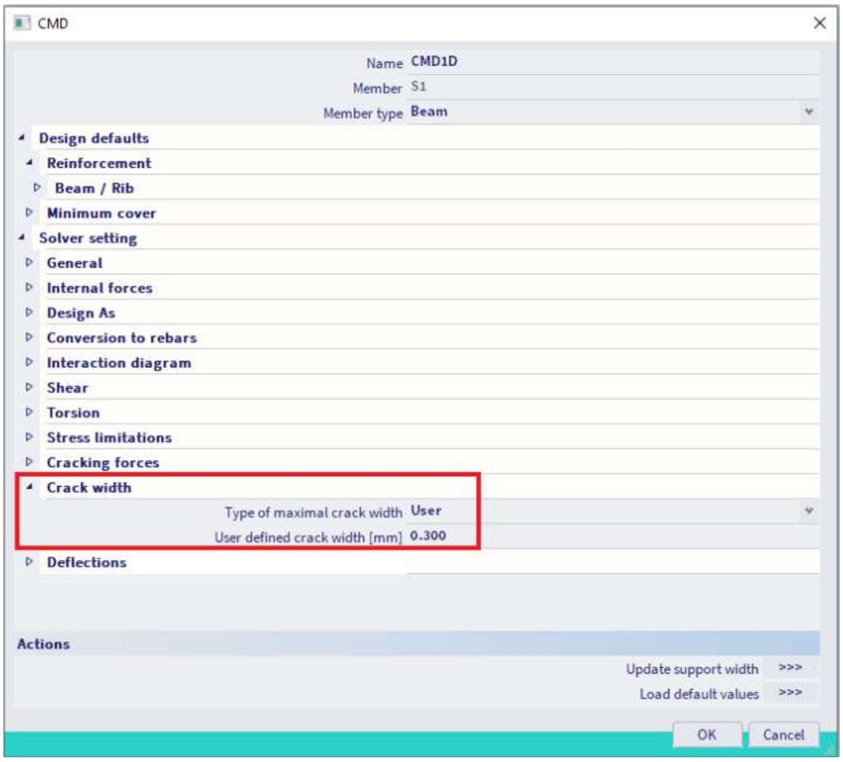
σ_{ct} [MPa]	σ_{cr} [MPa]	Cracked	w [mm]	w_{lim} [mm]	UC [-]	Limit check [-]	Status
12.6	2.9	YES	0.303	0.4	0.76	1	OK

The limit value of the crack width w_{max} is by default automatically calculated according to EN 1992-1-1 (Table 7.1N). The allowable crack width can be seen in the NA setup:





You can manually input the limiting crack width in the 1D member data:



DEFLECTION

The calculation of deflection is done according to chapter 7.4.3 from EN 1992-1-1.

Two kinds of deflection calculations are possible in the software:

- Simplified method where the calculation is done twice, assuming the whole member to be uncracked and fully cracked, and then interpolating formula 7.18 according to clause 7.4.3(7). This is the default used method.
- Code dependent deflection. This is the most rigorous method to calculate deflection by computing the calculation of curvatures at frequent sections along the member and then calculate the deflection by numerical integration. More information about this method can be found in the chapter **Code dependent deflections.**

The calculation procedure for the simplified method can be described in the following steps:

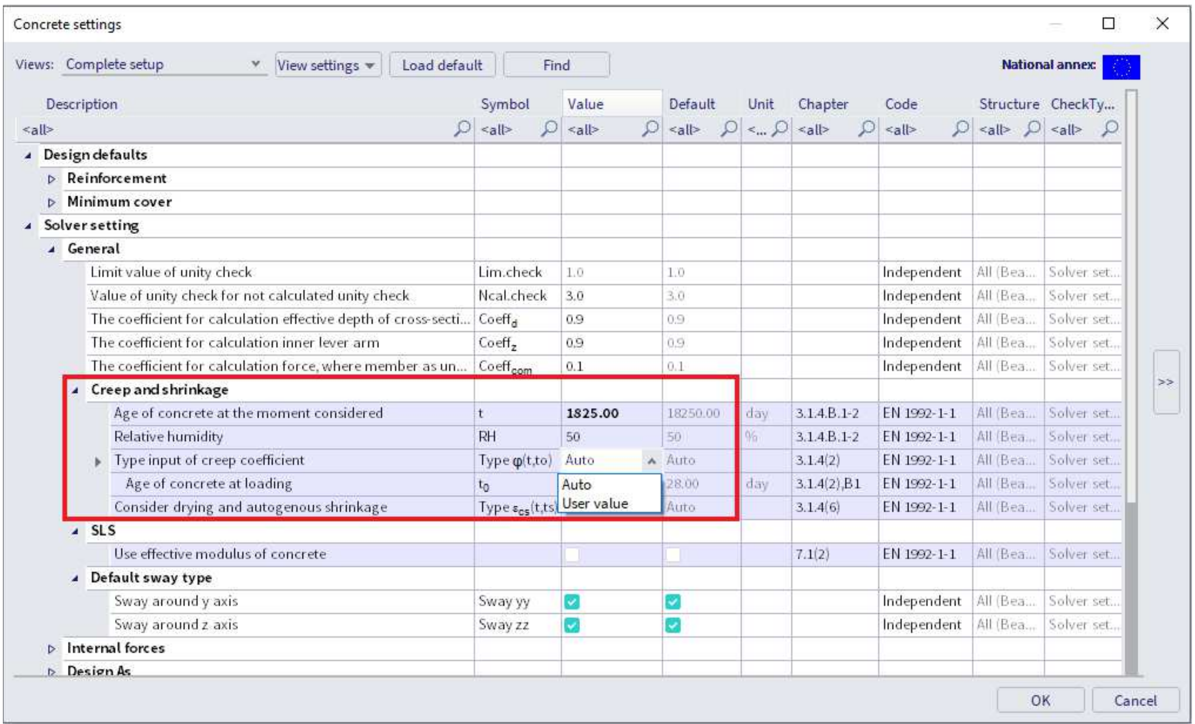
1. **Calculation of short-term stiffness** using E modulus at 28 days.

2. **Calculation of long-term stiffness** using effective E modulus based on creep coefficient.

   In the current version of the software, it is not possible to distinguish between the short-term and long-term part of the load in a combination. Therefore, some preconditions have been established for determination of the long-term part of the load. The long-term part of the load (LongTermPercentage) is estimated based on the type of combination. There are three main SLS combinations:

   - SLS characteristics - LongTermPercentage = 70 %
   - SLS frequent - LongTermPercentage = 85 %
   - SLS quasi-permanent- LongTermPercentage = 100 %

   The creep-factor is calculated by the software depending on the relative humidity, outline of the cross-section, reinforcement percentage, concrete class, etc. It can also be manually inputted in the Concrete setup > Complete setup view > General > Creep:



3. **Calculation of stiffness ratios** between each state, short and long term.

   It is the ratio of linear stiffness of the concrete component divided by the resultant stiffness taking cracks into account. The calculation of resultant stiffness is based on clause 7.4.3 (3), formula 7.18.

   bending stiffness around y-axis (EIy) = $1 / [\zeta/(EIy)_{II} + (1-\zeta) / (EIy)_{I}]$
   bending stiffness around z-axis (EIz) = $1 / [\zeta/(EIz)_{II} + (1-\zeta) / (EIz)_{I}]$
   axial stiffness (EA) = $1 / [(\zeta/(EA)_{II} + (1-\zeta) / (EA)_{I}]$

In this formula $(EI)_I$ is the linear stiffness, $(EI)_{II}$ is the stiffness of the cracked element (= long term stiffness = E_{lin} / 1 + φ) and ζ is the distribution coefficient.

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

ratio = Stiffness_{lin} / Stiffness_{res}, for example $\text{ratio}_{uz} = EI_{z,lin} / EI_{z,res}$

4. **Calculation of deflection components**

Several components are needed to calculate the total and additional deflection.
In the following part we will note "s" for short term and "l" for long term.
The components are:

- δ_{lin} linear (elastic) deflection, $\delta_{lin} = \delta_{lin,s} + \delta_{lin,l}$
- δ_{imm} immediate deflection, $\delta_{imm} = \delta_{lin,l} \cdot \text{ratio}_s$
- δ_s short-term deflection, $\delta_s = \delta_{lin,s} \cdot \text{ratio}_s$
- $\delta_{l,creep}$ long-term deflection + creep, $\delta_{l,creep} = \delta_{lin,l} \cdot \text{ratio}_l$
- δ_{creep} creep deflection, $\delta_{creep} = \delta_{lin,l} \cdot (\text{ratio}_l - \text{ratio}_s)$
- δ_l long-term deflection, $\delta_l = \delta_{l,creep} - \delta_{creep}$
- δ_{add} additional deflection, $\delta_{add} = \delta_s + \delta_{l,creep} - \delta_{imm}$
- δ_{tot} total deflection, $\delta_{tot} = \delta_s + \delta_{l,creep}$

5. **Check of deflections**

Two deflections are checked:
Total deflection: The appearance and general utility of the structure could be impaired when the calculated sag of a beam, slab or cantilever subjected to quasi-permanent loads exceeds span/250.

$$\delta_{tot,lim} = L / 250$$

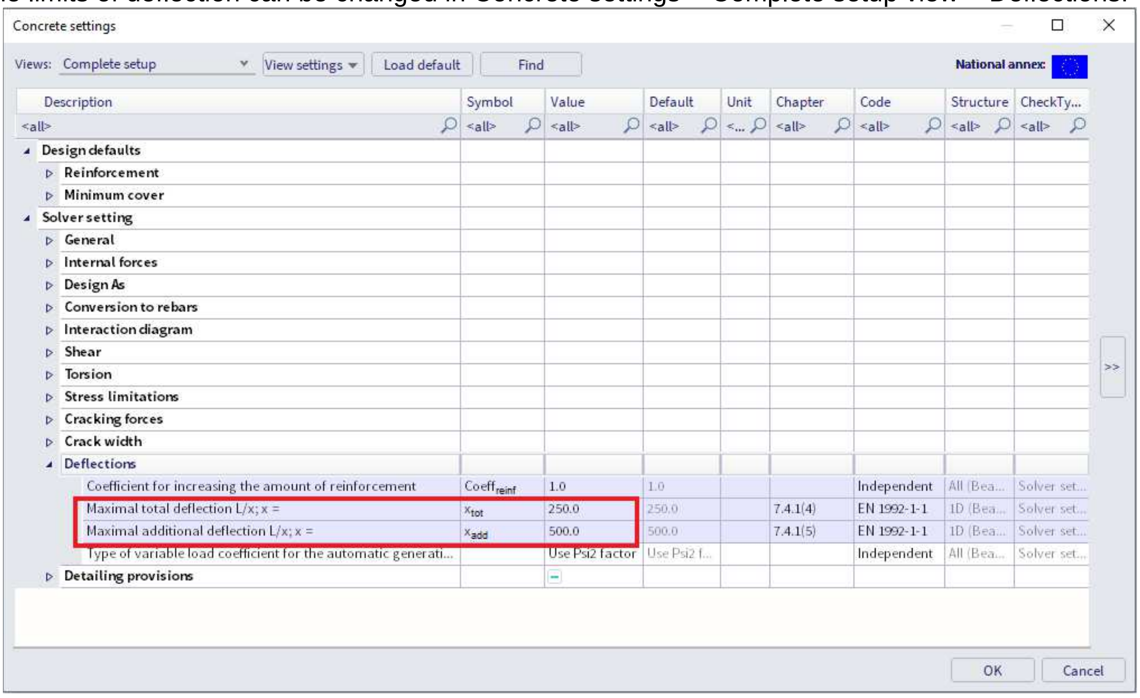
Additional deflection: Deflections that could damage adjacent parts of the structure should be limited.

$$\delta_{add,lim} = L / 500$$

L is the buckling length multiplied by a β factor of the member in the corresponding direction.
Final unity check is:

$$\text{Unity check} = \max \left\{ \frac{\delta tot}{\delta tot, lim}; \frac{\delta add}{\delta add, lim} \right\}$$

The limits of deflection can be changed in Concrete settings > Complete setup view > Deflections:



Example: beam_practical reinforcement.esa

Look at deflection check for the “SLS qp” combination.

Various results can be displayed on the screen: UC, total and additional deflection or limits for total and additional deflection.

Open the Standard output for the UC. At position dx = 2,5 m we have the following result:

Basic values of deflections

Type of deflection	Ratio short [-]	Ratio long [-]	δ_{lin} [mm]	δ_{imm} [mm]	δ_{add} [mm]	δ_{short} [mm]	δ_{long} [mm]	$\delta_{long+creep}$ [mm]	δ_{creep} [mm]
u_y	2.88	5.22	0	0	0	0	0	0	0
u_z	2.5	3.38	-3.08	-7.7	-2.69	0	-7.7	-10.4	-2.69

Check of additional and total deflections

Type of deflection	L [m]	δ_{add} [mm]	$\delta_{add.lim}$ [mm]	UC_{add} [-]	δ_{tot} [mm]	$\delta_{tot.lim}$ [mm]	UC_{tot} [-]	UC [-]	Limit [-]	Status
u_y	10	0	0	0	0	0	0	0	1	OK
u_z	10	-2.69	-20	0.13	-10.4	-40.1	0.26	0.26	1	OK

List of errors/warnings/notes: NO

All ratio of stiffnesses and deflection components are resumed in a table.

Open the Detailed output, for the same position dx = 2,5 m.

All previously mentioned steps for the calculation of the deflections can be found here.

For example for the long-term stiffness, we can obtain the long-term part of the loads and the calculated creep coefficient:

Long-term stiffnesses and curvatures under total load

Settings

Long-term part of applied load = 100%

Creep coefficient $\varphi = 2.21$

Uncracked (state I) and cracked (state II) cross section properties are also shown in a table:

Cross-section characteristics

Type of component	t_y [m]	t_z [m]	A [m^2]	I_y [m^4]	I_z [m^4]	x_i [m]	A_{st} [m^2]	A_{sc} [m^2]	A_s [m^2]
Linear	0	0	0.15	$3.13 \cdot 10^{-3}$	$1.13 \cdot 10^{-3}$	0.25	-	-	-
Uncracked	0	-0.019	0.193	$4.69 \cdot 10^{-3}$	$1.35 \cdot 10^{-3}$	0.27	$1.57 \cdot 10^{-3}$	$628 \cdot 10^{-6}$	$2.2 \cdot 10^{-3}$
Cracked	0	0.053	0.102	$2.89 \cdot 10^{-3}$	$667 \cdot 10^{-6}$	0.197	$1.57 \cdot 10^{-3}$	$628 \cdot 10^{-6}$	$2.2 \cdot 10^{-3}$

Check of concrete stresses and calculation of cracking forces

Maximal tensile stress in concrete fibre

$\sigma_{ct} = 7.76$ MPa

Cracking status

$\sigma_{ct} > f_{ct,eff}$ = 7.76 MPa > 2.9 MPa => Cracks appear.

Stress in reinforcement for cracking load

$\sigma_{sr} = 99.3$ MPa

Stress in reinforcement for acting load

$\sigma_s = 262$ MPa

Distribution coefficient

$$\zeta = \max\left(0; 1 - \beta \cdot \left(\frac{\sigma_{sr}}{\sigma_s}\right)^2\right) = \max\left(0; 1 - 0.5 \cdot \left(\frac{99.3}{262}\right)^2\right) = 0.928 \quad (7.19)$$

N_{cr} [kN]	$M_{y,cr}$ [kNm]	$M_{z,cr}$ [kNm]	σ_{ct} [MPa]	$f_{ct,eff}$ [MPa]	Cracked section	σ_{sr} [MPa]	σ_s [MPa]	β [-]	ζ [-]	E_c [GPa]
0	59.6	0	7.76	2.9	YES	99.3	262	0.5	0.928	33

Which allows to calculate the stiffness's ratio, for example the bending stiffness's ratio:

Bending stiffness EIy

$$EI_{y,lin} = E_c \cdot I_y = 33 \cdot 3.13 \cdot 10^9 = 103\ MNm^2$$

$$EI_{y,I} = E_{c,eff} \cdot I_{y,I} = 10.3 \cdot 4.69 \cdot 10^9 = 48.1\ MNm^2$$

$$EI_{y,II} = E_{c,eff} \cdot I_{y,II} = 10.3 \cdot 2.89 \cdot 10^9 = 29.7\ MNm^2$$

$$EI_y = \frac{1}{\frac{\zeta}{EI_{y,II}} + \frac{1-\zeta}{EI_{y,I}}} = \frac{1}{\frac{0.928}{29.7} + \frac{1-0.928}{48.1}} = 30.5\ MN{\bullet}m^2 \qquad (7.18)$$

$$RatioEIy = \frac{EI_y}{EI_{y,lin}} = \frac{30.5}{103} = 0.296$$

Bending stiffness EIz

$$EI_{z,lin} = E_c \cdot I_z = 33 \cdot 1.13 \cdot 10^9 = 37.1\ MNm^2$$

$$EI_{z,I} = E_{c,eff} \cdot I_{z,I} = 10.3 \cdot 1.35 \cdot 10^9 = 13.8\ MNm^2$$

$$EI_{z,II} = E_{c,eff} \cdot I_{z,II} = 10.3 \cdot 667 \cdot 10^6 = 6.85\ MNm^2$$

$$EI_z = \frac{1}{\frac{\zeta}{EI_{z,II}} + \frac{1-\zeta}{EI_{z,I}}} = \frac{1}{\frac{0.928}{6.85} + \frac{1-0.928}{13.8}} = 7.11\ MN{\bullet}m^2 \qquad (7.18)$$

$$RatioEIz = \frac{EI_z}{EI_{z,lin}} = \frac{7.11}{37.1} = 0.191$$

And final the short and long-term ratios:

Short-term ratios

Bending stiffness EIy

$$RatioEIys = \frac{EI_{y,s}}{EI_{y,lin}} = \frac{41.2 \cdot 10^6}{103 \cdot 10^6} = 0.4$$

Bending stiffness EIz

$$RatioEIzs = \frac{EI_{z,s}}{EI_{z,lin}} = \frac{12.9 \cdot 10^6}{37.1 \cdot 10^6} = 0.347$$

Ratios

$$ratio_{uys} = \frac{1}{RatioEIzs} = \frac{1}{0.347} = 2.88$$

$$ratio_{uzs} = \frac{1}{RatioEIys} = \frac{1}{0.4} = 2.5$$

Long-term ratios

Bending stiffness EIy

$$RatioEIyl = \frac{EI_{y,l}}{EI_{y,lin}} = \frac{30.5 \cdot 10^6}{103 \cdot 10^6} = 0.296$$

Bending stiffness EIz

$$RatioEIzl = \frac{EI_{z,l}}{EI_{z,lin}} = \frac{7.11 \cdot 10^6}{37.1 \cdot 10^6} = 0.191$$

Ratios

$$ratio_{uyl} = \frac{1}{RatioEIzl} = \frac{1}{0.191} = 5.22$$

$$ratio_{uzl} = \frac{1}{RatioEIyl} = \frac{1}{0.296} = 3.38$$

Then all deflection components are calculated together with the limit deflections:

Deflections

Linear deflection

$$\delta_{lin,y} = u_{ys} + u_{yl} = 0 + 0 = 0\ mm$$

$$\delta_{lin,z} = u_{zs} + u_{zl} = 0 + -3.08 = -3.08\ mm$$

Immediate deflection

$$\delta_{imm,y} = u_{yl} \cdot ratio_{uys} = 0 \cdot 2.88 = 0\ mm$$

$$\delta_{imm,z} = u_{zl} \cdot ratio_{uzs} = -3.08 \cdot 2.5 = -7.7\ mm$$

Short-term deflection

$$\delta_{short,y} = u_{ys} \cdot ratio_{uys} = 0 \cdot 2.88 = 0\ mm$$

$$\delta_{short,z} = u_{zs} \cdot ratio_{uzs} = 0 \cdot 2.5 = 0\ mm$$

Long-term + creep deflection

$$\delta_{long,creep,y} = u_{yl} \cdot ratio_{uyl} = 0 \cdot 5.22 = 0\ mm$$

$$\delta_{long,creep,z} = u_{zl} \cdot ratio_{uzl} = -3.08 \cdot 3.38 = -10.4\ mm$$

Creep deflection

$$\delta_{creep,y} = u_{yl} \cdot \left(ratio_{uyl} - ratio_{uys}\right) = 0 \cdot \left(5.22 - 2.88\right) = 0\ mm$$

$$\delta_{creep,z} = u_{zl} \cdot \left(ratio_{uzl} - ratio_{uzs}\right) = -3.08 \cdot \left(3.38 - 2.5\right) = -2.69\ mm$$

Long-term deflection

$$\delta_{long,y} = \delta_{long,creep,y} - \delta_{creep,y} = 0 - 0 = 0\ mm$$

$$\delta_{long,z} = \delta_{long,creep,z} - \delta_{creep,z} = -10.4 - -2.69 = -7.7\ mm$$

Additional deflection

$$\delta_{add,y} = \delta_{short,y} + \delta_{long,creep,y} - \delta_{imm,y} = 0 + 0 - 0 = 0\ mm$$

$$\delta_{add,z} = \delta_{short,z} + \delta_{long,creep,z} - \delta_{imm,z} = 0 + -10.4 - -7.7 = -2.69\ mm$$

Limit additional deflection

$$\delta_{add,lim,y} = 0\ mm$$

$$\delta_{add,lim,z} = \frac{-l_{0z}}{Lim_{add}} = \frac{-10}{500} = -20\ mm$$

Total deflection

$$\delta_{tot,y} = \delta_{short,y} + \delta_{long,creep,y} = 0 + 0 = 0\ mm$$

$$\delta_{tot,z} = \delta_{short,z} + \delta_{long,creep,z} = 0 + -10.4 = -10.4\ mm$$

Limit total deflection

$$\delta_{tot,lim,y} = 0\ mm$$

$$\delta_{tot,lim,z} = \frac{-l_{0z}}{Lim_{tot}} = \frac{-10}{250} = -40\ mm$$

Limitations of the deflection check:

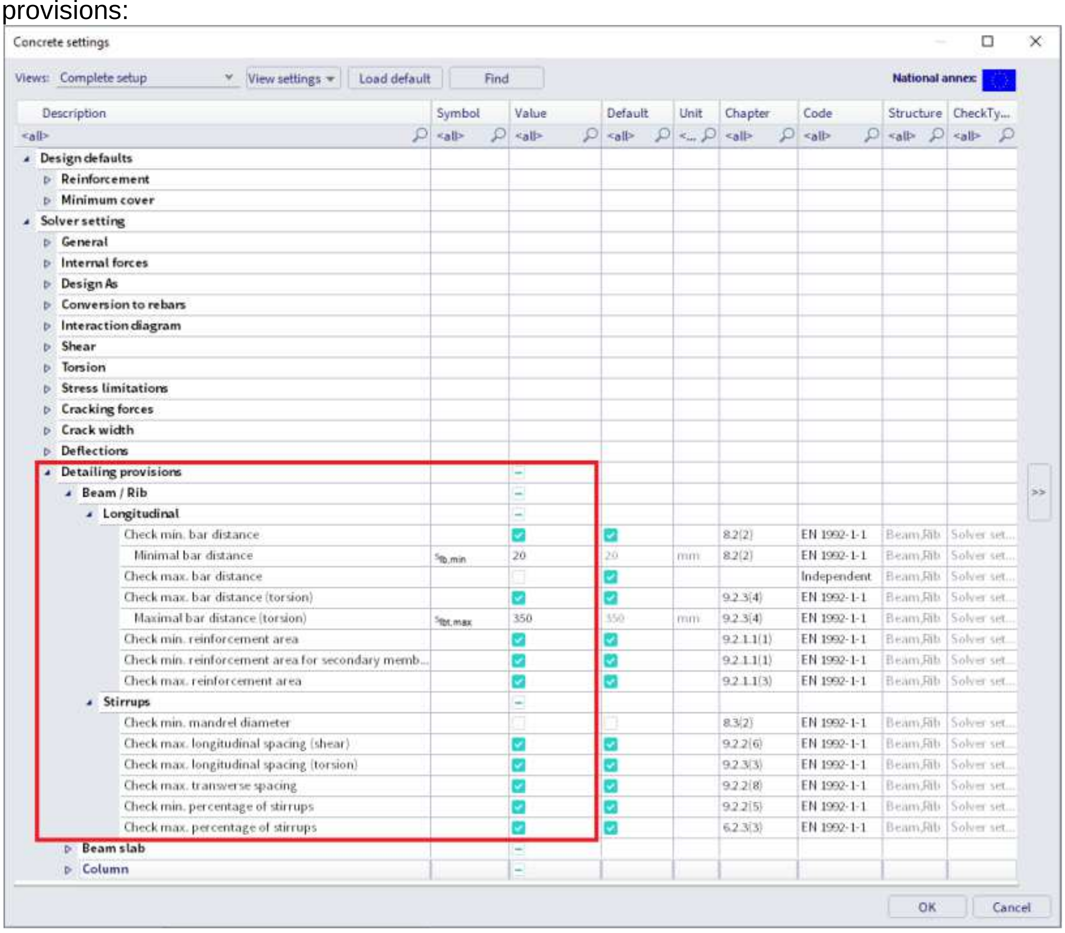
- Deformation caused by shrinkage is not automatically considered.
- Verification based on limiting span / depth ratio according to 7.4.2 is not implemented.
- Calculation of deflection depends on the internal forces used for the reduced stiffness. Therefore, the check of deflection doesn't work for cases where the internal forces are equal to zero but deflections are not zero. Typically, this is the case for a cantilever structure with free overhang.

DETAILING PROVISIONS

SCIA Engineer distinguishes three types of member with their detailing provisions:

- Beam - verification of longitudinal and shear reinforcement
- Column - verification of main and transverse reinforcement
- Beam slab - verification of longitudinal reinforcement only

All detailing provisions are taken into account automatically in Concrete settings > Complete setup view > Detailing provisions:



Following table shows which checks of detailing provisions are performed:

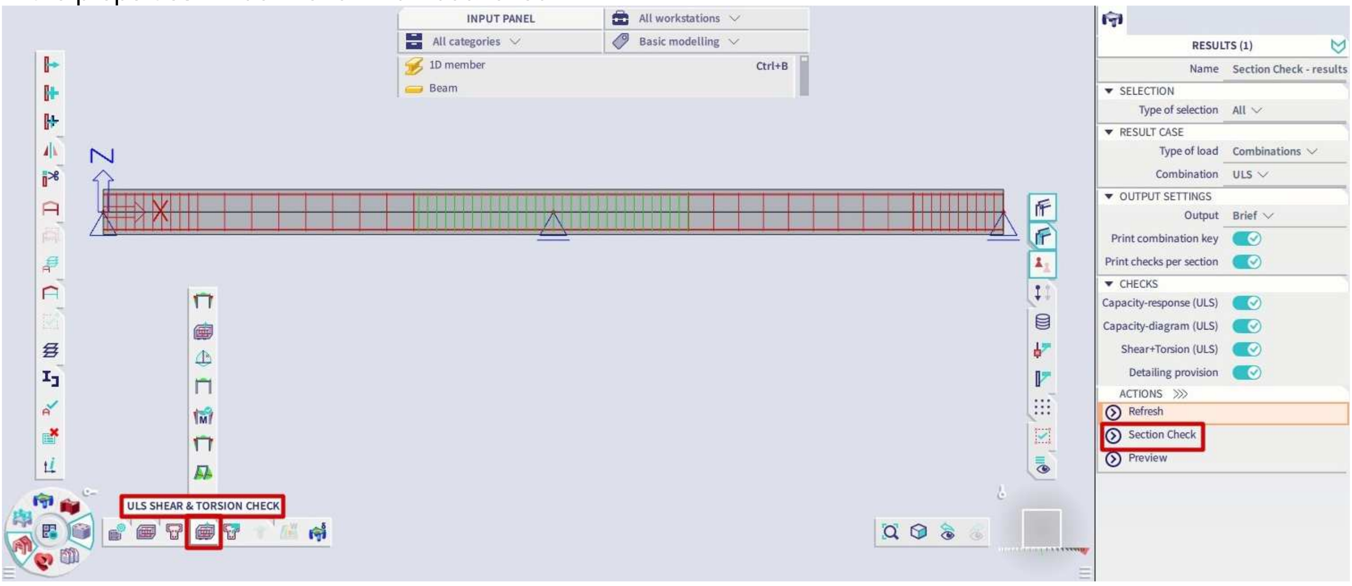
Member type	Longitudinal (main)	Shear (transverse)
Beam	8.2(2) - Minimal clear spacing of bars 9.2.1.1(1) - Minimal area of longitudinal reinforcement 9.2.1.1(3) - Maximal area of longitudinal reinforcement 9.2.3(4) - Maximal center-to-center bar distance based on torsion Code-Independent - Maximal clear spacing	6.2.3(3) - Maximal percentage of shear reinforcement 9.2.2(5) - Minimal percentage of shear reinforcement 9.2.2(6) - Maximal longitudinal spacing of stirrups (shear) 9.2.2(8) - Maximal transverse spacing of stirrups (shear) 9.2.3(3) - Maximal longitudinal spacing of stirrups (torsion)
Column	8.2(2) - Minimal clear spacing of bars 9.5.2(1) - Minimal bar diameter of longitudinal reinforcement 9.5.2(2) - Minimal area of longitudinal reinforcement 9.5.2(3) - Maximal area of longitudinal reinforcement 9.5.2(4) - Minimal number of longitudinal reinforcement bars	9.2.3(3) - Maximal longitudinal spacing of stirrups (torsion) 9.5.3(1) - Minimal diameter of transverse reinforcement 9.5.3(3) - Maximal longitudinal spacing of transverse reinforcement
Beam Slab	8.2(2) - Minimal clear spacing of bars 9.3.1.1(3) - Maximal bar distance of longitudinal reinforcement	-

SECTION CHECK

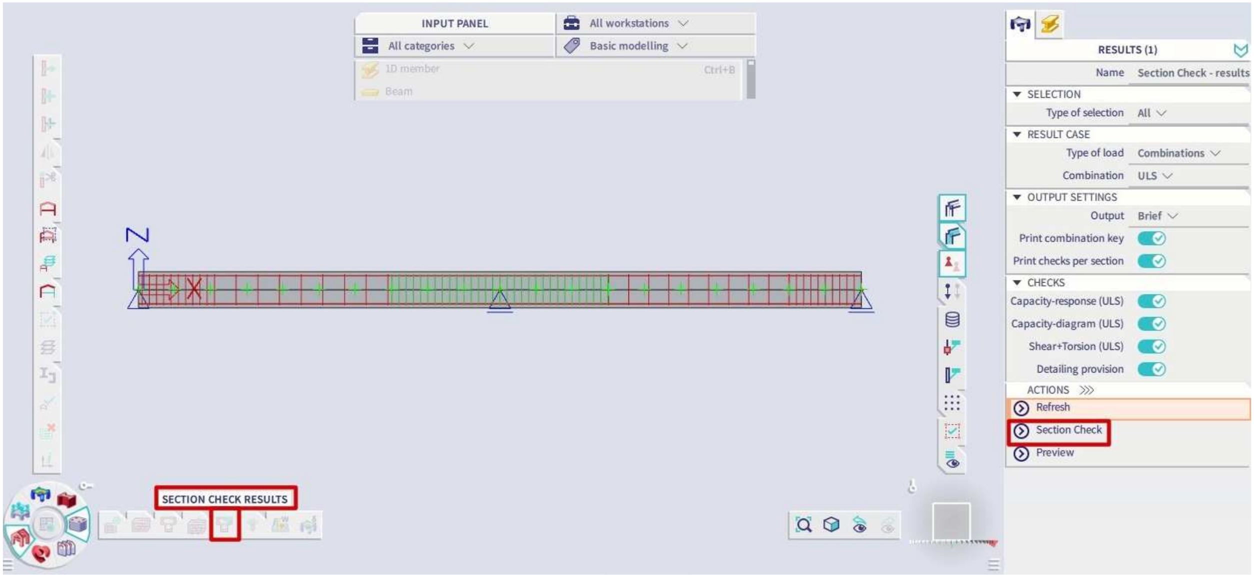
The Section check tools can be used in two different ways: with or without practical reinforcement inputted beforehand.

Section check can be launched:

- In the properties window for an individual check



- In the properties window for the Section Check – results service



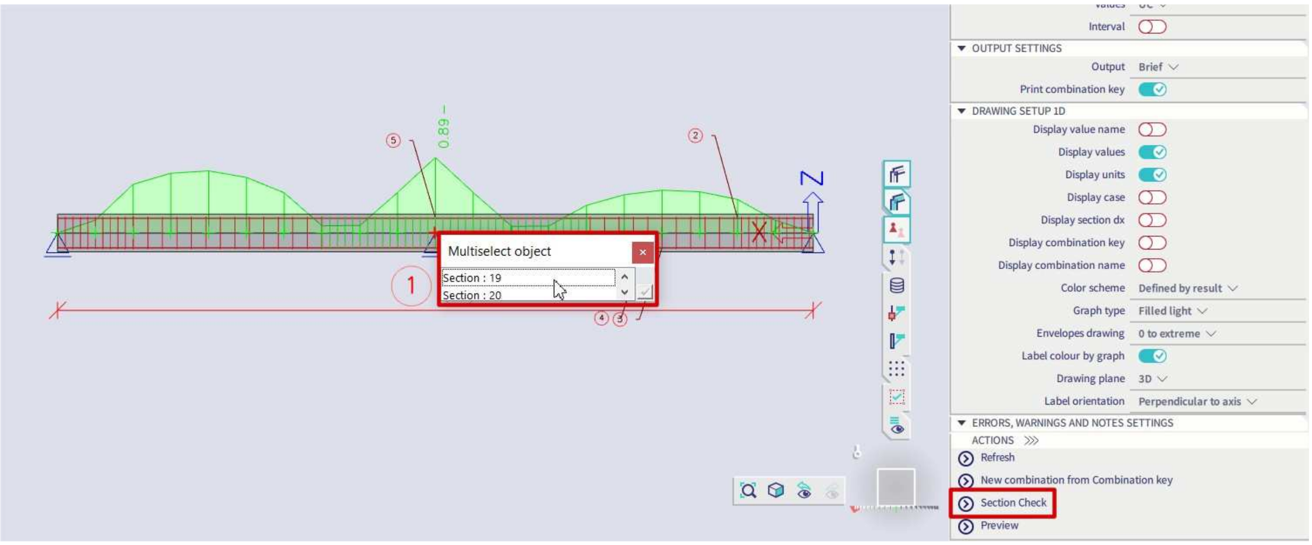
⇨ *With practical reinforcement*

Example 1: beam_practical reinforcement SC.esa

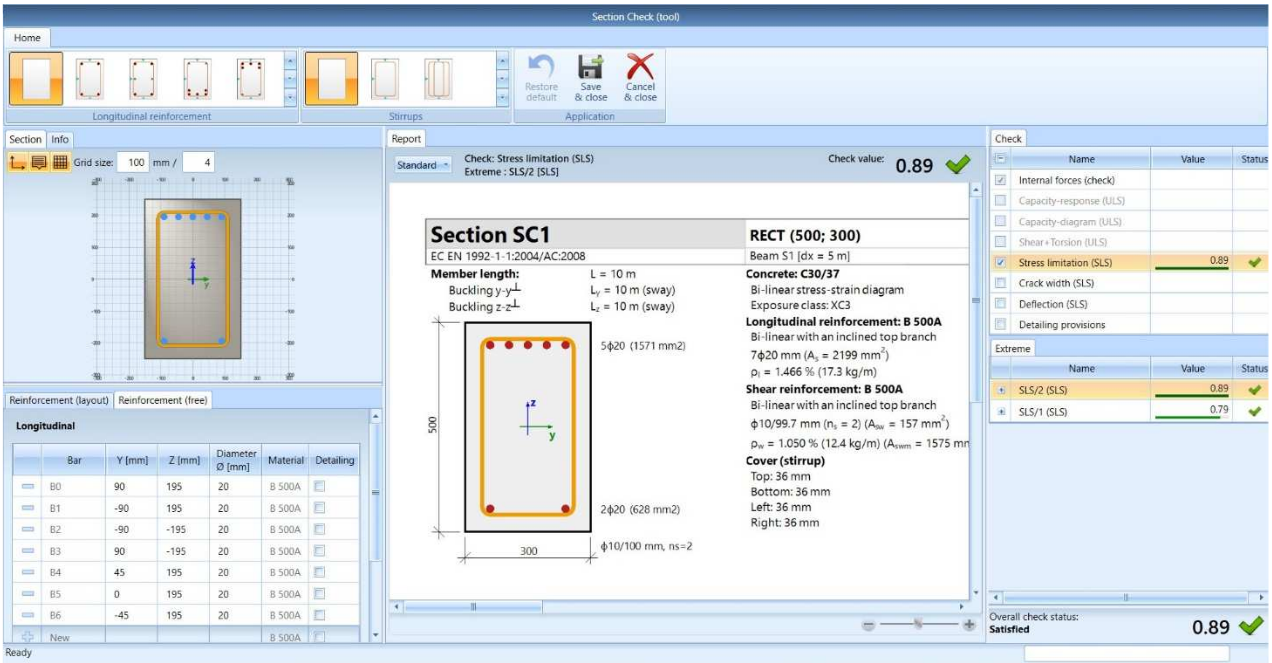
Section check can be opened from all individual checks.

In this example, select Design > Concrete 1D > SLS reinforcement stress limitation check (SLS) and click on "Section check" in the Properties window:

Select the beam and then click on the position for which the check should be done. Choose section 20 at the middle of the beam:



The Section check tool opens:



This window is composed of 3 mains parts:

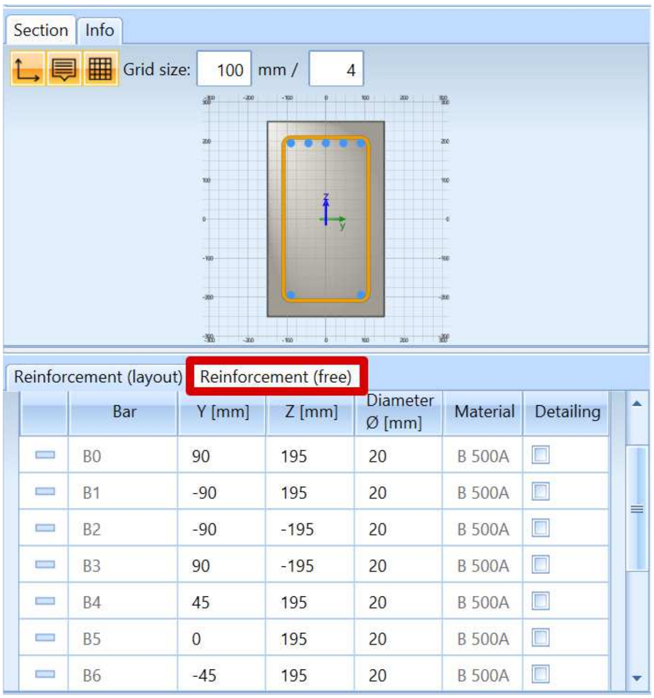
- Definition / modification of the reinforcement
- Preview of the report
- Checks to be performed according to the previous selected combinations or load cases. By default, only the individual selected check will be performed. You can activate more checks if wanted.

When selecting a SLS combination in the Properties windows, only SLS checks will be available.

When selecting a ULS combination in the Properties windows, only ULS checks will be available.

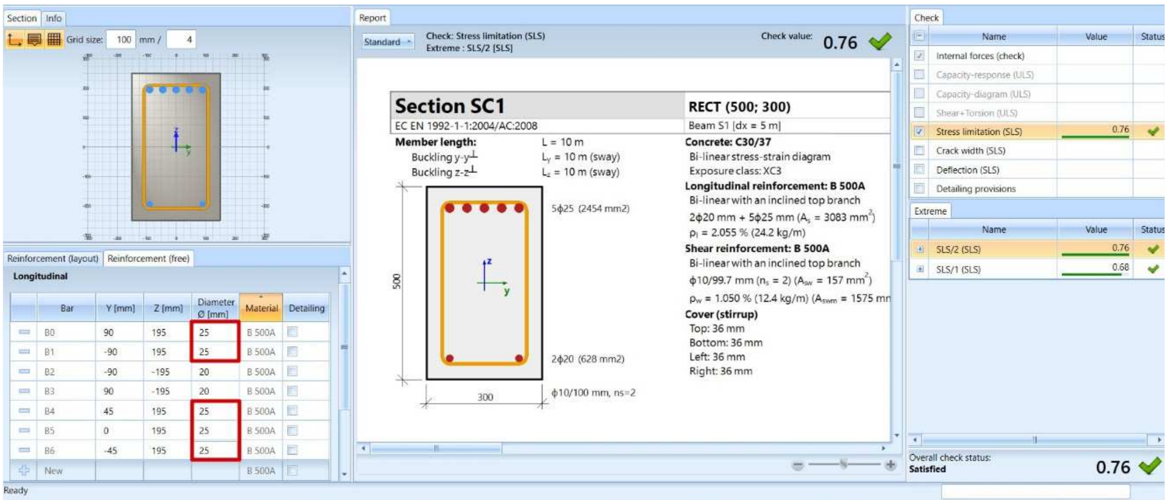
In this example, stress limitation in the concrete is not OK. One solution is to redesign the longitudinal reinforcement to satisfy the SLS stress limitations. We could then close the Section check tool and change the practical reinforcement for this beam or we can adapt locally the reinforcement in the studied section (Section 19). We will choose to adapt the reinforcement in the Section check tool itself.

When practical reinforcement was already inputted, it can be edited in the tab “Reinforcement (free)”:



Each present bar, position and diameter, is listed in the table. They can be modified, deleted or new bars can be added.

Increase the diameter of top layer bars B0, B1, B4 and B6 from 20 mm to 25 mm:



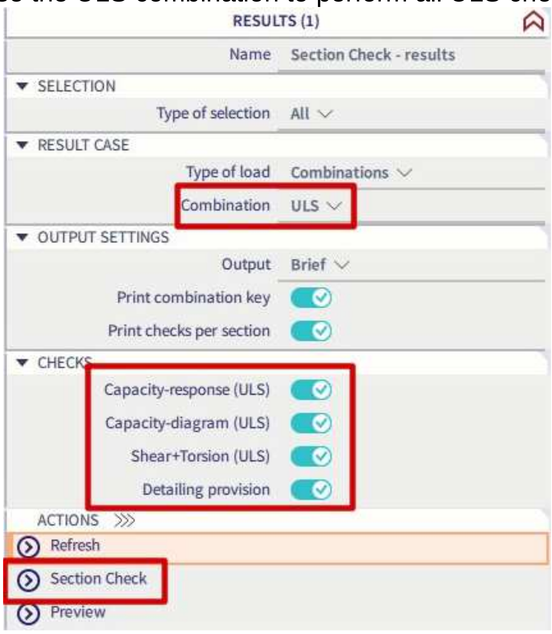
⇨ *Without practical reinforcement*

Example 2: <u>beam_without practical reinforcement SC.esa</u>

When no practical reinforcement was inputted beforehand, it is possible to run the section check tool in order to check a specific section of a member with a local reinforcement on this specific section.

In the Concrete menu, select “Section check results”.

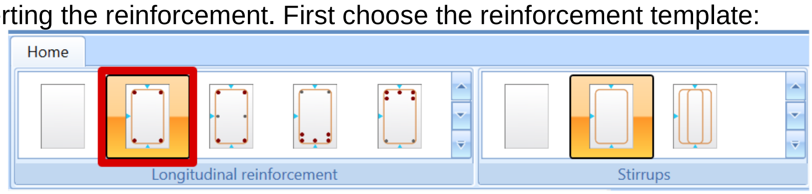
In the properties window, choose the ULS combination to perform all ULS checks:



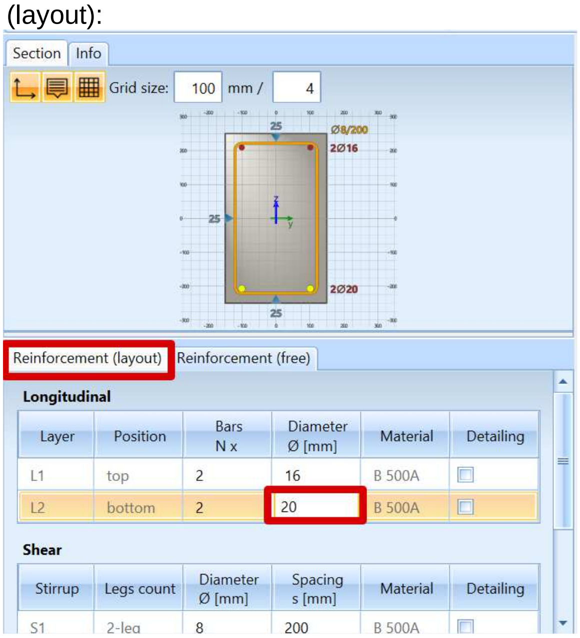
Select Section 9, in the middle of the first span.

All checks are not satisfied, and the overall UC is 3. The value 3 means that the check could not be performed due to an error in the calculation. In this case, it is because there is no reinforcement yet.

We will start by inserting the reinforcement. First choose the reinforcement template:

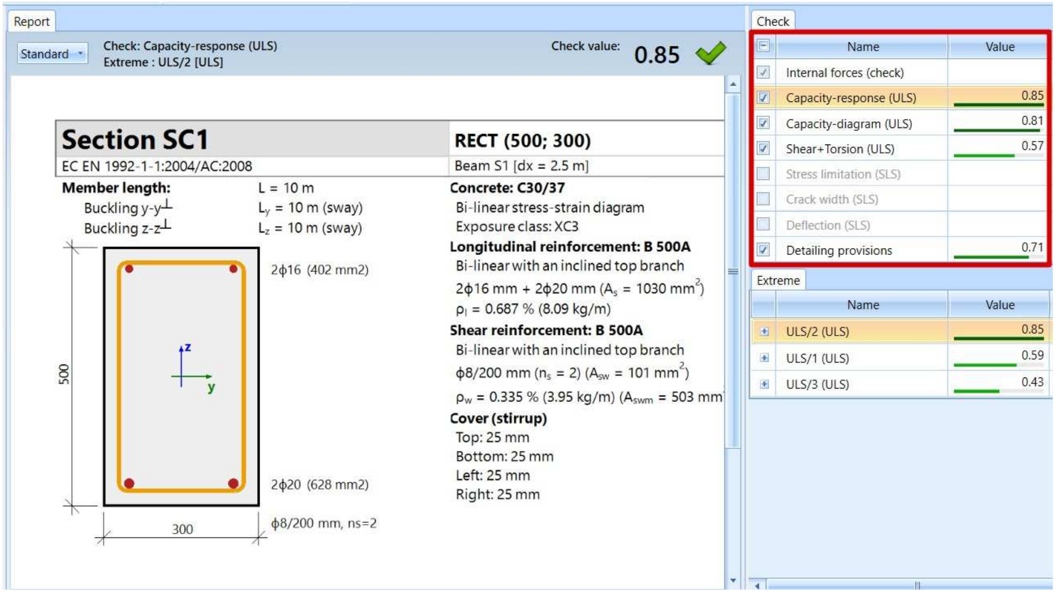


Then change the diameter of the reinforcement template. For bottom longitudinal bars, change diameter to 20 mm in the tab “Reinforcement (layout):



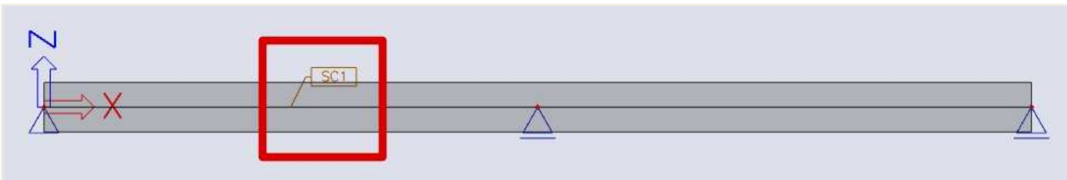
Note that it is also possible to define the shear reinforcement in this window.

The results for all ULS checks are now:

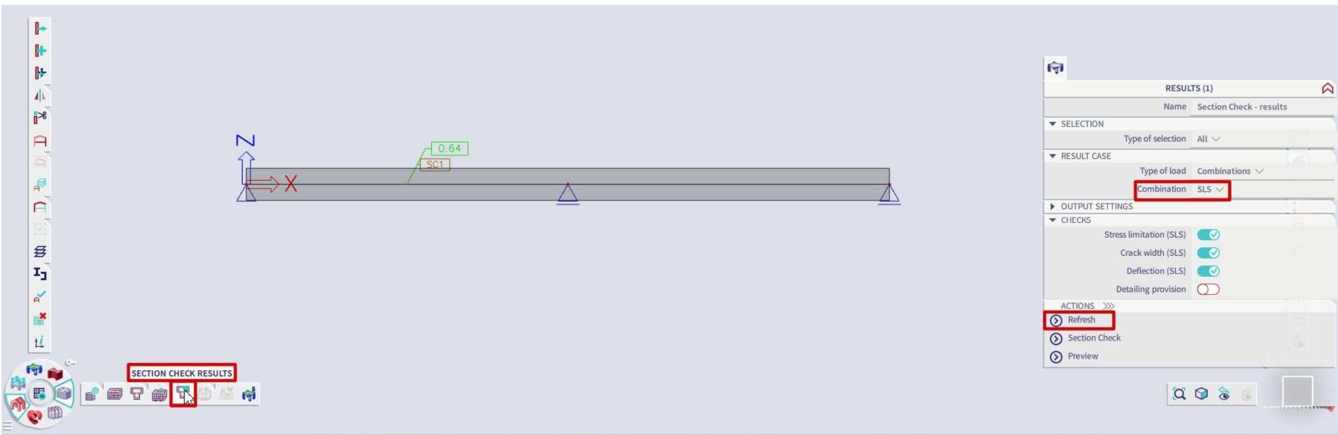


Once the section is reinforced and checks are satisfied, you can save the design of this section with the option "Save and close":

A label will then be added on the beam:



It is possible to run the Section check for SLS combination as below:



If required, Section check tool can still be opened to redesign the section to satisfy the SLS checks by clicking on Section check in the Properties window.

2.3 Column design

2.3.1 Reinforcement design methods

For column design, there are 3 types of calculation:

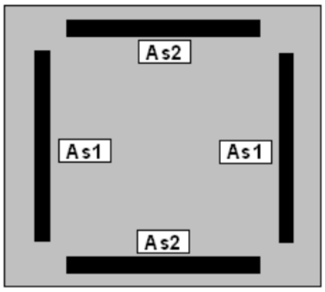
- Axial compression only
- Uniaxial bending
- Biaxial bending

When taking a closer look at the column calculation, 2 different approaches can be distinguished:

- For the ‘Axial compression only’ and ‘Uniaxial bending’ calculation, SCIA Engineer uses the same computing heart as for beams.
- For ‘Biaxial bending’ calculations, SCIA Engineer uses a combination of the computing heart for beams and the so-called interaction formulas.

Furthermore, the uniaxial bending calculation always has as result a 1-directional reinforcement configuration, with the same number of reinforcement bars at parallel sides.

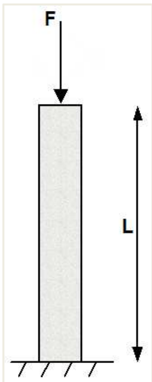
The biaxial bending calculation has as result a 2-directional reinforcement configuration. The number of bars may differ per direction, but is always the same for parallel sides:



The uniaxial bending calculation is a relatively simple calculation type, while the biaxial bending calculation requires an iterative process.

Keep this in mind as the reason why the uniaxial bending calculation will go a lot faster.

DESIGN WITH AXIAL COMPRESSION ONLY



⇨ ***No reinforcement required: $N_{Ed} < N_{Rd}$***

Example: Axial compression only.esa

Studied column: B1

Geometry

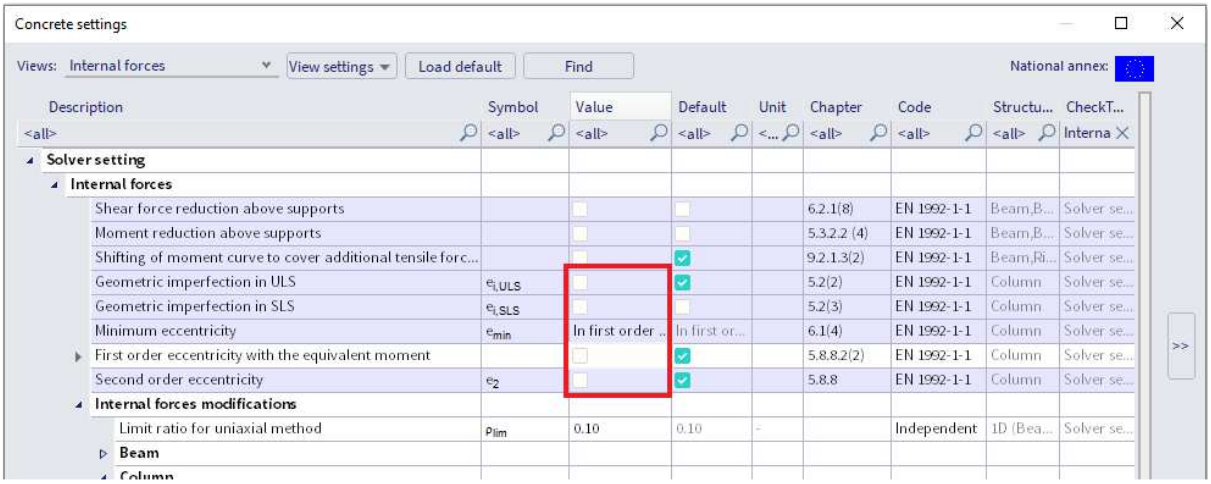
Column cross-section: RECT 350x350 mm²

Height: 4,5 m

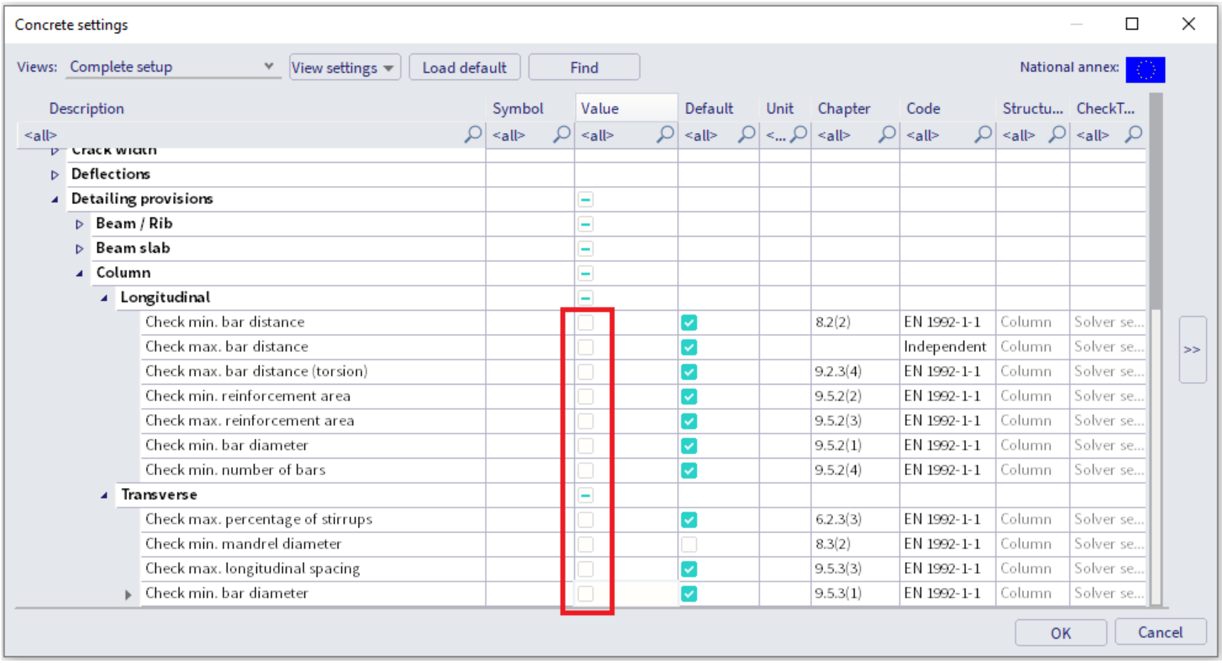
Concrete grade: C45/55

Concrete Setup

Item Concrete settings > Internal forces ULS: ‘eccentricities’ are not taken in account.



The Detailing provisions are not taken in account, in order to view the pure results (according to the Eurocode, always a minimum reinforcement percentage must be added).



Loads

- LC1: Permanent load > F = 1100 kN
- LC2: Variable load > F = 1000 kN

This means the column is loaded with a single compression force.

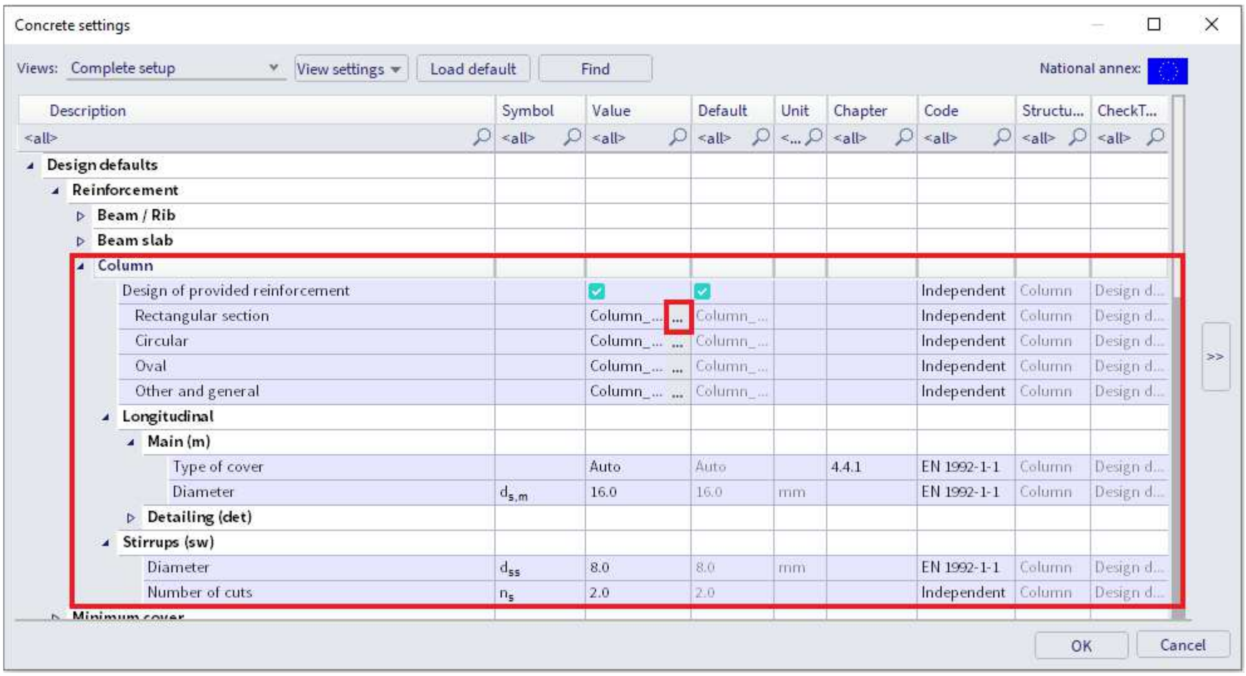
Combination according to the Eurocode:

$$\text{ULS Combination} = 1{,}35 * LC1 + 1{,}50 * LC2$$

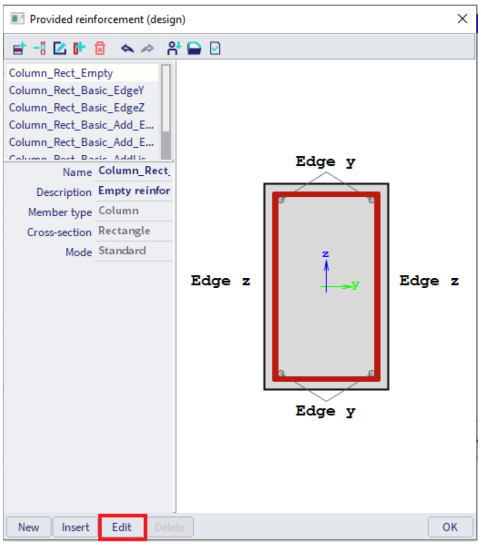
Design normal force N_{Ed} = 1,35 * 1100 + 1,50 * 1000 = 2985kN

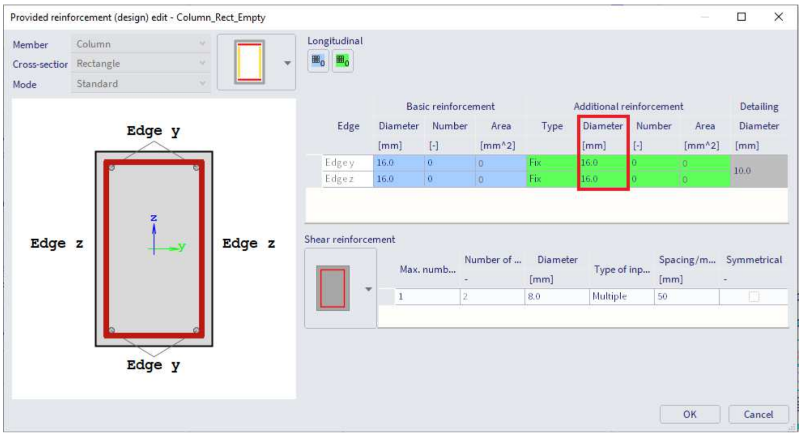
Bar diameter

The bar diameter is taken from the Concrete Settings > Complete setup View, or from 1D member data if applied (1D member data always overwrite the Concrete Settings data, for the specific member they are assigned to).



By default, the diameter for the main column reinforcement is put to ϕ16mm. Based on this diameter and the exposure class (by default XC3), the concrete cover is calculated. This information is necessary to be able to calculate the lever arm of the reinforcement bars.





<u>Note:</u> To change the default diameter from ϕ16mm to ϕ20mm for example, edit the template “Column_Rect_Empty” (or the corresponding empty template for the specific columns shape), and change the value of the diameter to be taken into account (additional provided reinforcement).

Results

Go to Steel workstation > 1D Reinforcement design:

Ask the value of $A_{s\ reg}$ for member B1, and click the action button [Refresh].

The graph appears to be null on the screen. The Brief output (Preview button), gives As,req = 0.

Overall Design (ULS)

Linear calculation
Combination: CO1
Coordinate system: Principal
Extreme 1D: Global
Selection: All

Longitudinal required reinforcement

Name	dx [m]	Case	Member	A_{sz_req+} [mm²] $A_{sz\_req\_bar+}$ [mm²]	A_{sz_req-} [mm²] $A_{sz\_req\_bar-}$ [mm²]	A_{sy_req+} [mm²] $A_{sy\_req\_bar+}$ [mm²]	A_{sy_req-} [mm²] $A_{sy\_req\_bar-}$ [mm²]	A_{sz_req} [mm²] $A_{sz\_req\_bar}$ [mm²]	A_{sy_req} [mm²] $A_{sy\_req\_bar}$ [mm²]	A_{s_req} [mm²] $A_{s\_req\_bar}$ [mm²]	ReinfReq
B1	0,000	CO1	Column	0 0	0 0	0 0	0 0	0 0	0 0	0 0	

Shear reinforcement

Name	dx [m]	Case	Member	A_{swm_req} [mm²/m]	A_{swm_prov} [mm²/m]	ShearReinf
B1	0,000	CO1	Column	0	0	

If you set output settings on Detailed, you can see the explanation that reinforcement is not necessary.

Explanation errors/warnings and notes

Index	Type	Description	Solution
N1/1	Note	Statically required reinforcement: The reinforcement is not neccessary.	
		Shear design: Design is not done, because	

Remark: this result is obtained only because **all detailing provisions are deactivated** in the Concrete Settings!

Check of reinforcement

$N_{Rd} = f_{cd} \cdot \alpha \cdot A_c = 30 * 1 * 350^2 / 1000 = 3675kN$

Since $N_{Rd} = 3675kN > N_{Ed} = 2985kN$, indeed no theoretical reinforcement is required.

⇨ ***Reinforcement required: $N_{Ed} > N_{Rd}$***

Example: Axial compression only.esa

Studied column: B2

For this example, the same configuration as above is used, only the permanent point load is increased to 2000kN.

Loads

- LC1: Permanent load > F = 2000kN
- LC2: Variable load > F = 1000kN

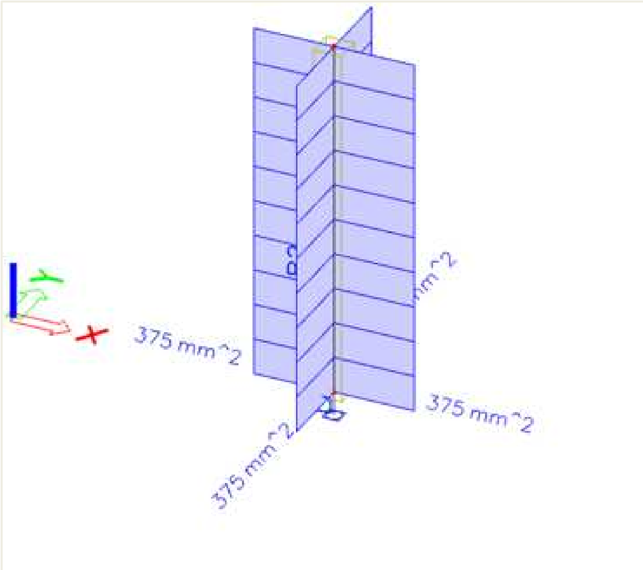
Combination according to the Eurocode:

ULS Combination = 1,35 * LC1 + 1,50 * LC2

Design normal force $N_{Ed} = 1,35 * 2000 + 1,50 * 1000 = 4200kN$

Results

Remark that SCIA Engineer shows on the screen the reinforcement per direction. The total reinforcement area is in fact 750 + 750 = 1500mm².



Overall Design (ULS)

Linear calculation
Combination: ULS
Coordinate system: Principal
Extreme 1D: Global
Selection: B2

Longitudinal required reinforcement

Name	dx [m]	Case	Member	A_{sz_req+} [mm²] $A_{sz\_req\_bar+}$ [mm²]	A_{sz_req-} [mm²] $A_{sz\_req\_bar-}$ [mm²]	A_{sy_req+} [mm²] $A_{sy\_req\_bar+}$ [mm²]	A_{sy_req-} [mm²] $A_{sy\_req\_bar-}$ [mm²]	A_{sz_req} [mm²] $A_{sz\_req\_bar}$ [mm²]	A_{sy_req} [mm²] $A_{sy\_req\_bar}$ [mm²]	A_{s_req} [mm²] $A_{s\_req\_bar}$ [mm²]	ReinfReq
B2	0.000	ULS	Column	**375** **402**	**375** **402**	**375** **402**	**375** **402**	**750** **804**	**750** **804**	**1500** **1608**	**[z]6ϕ16,** **[y]6ϕ16**

Shear reinforcement

Name	dx [m]	Case	Member	A_{swm_req} [mm²/m]	A_{swm_prov} [mm²/m]	ShearReinf
B2	0.000	ULS	Column	**0**	**0**	

When asking for the Standard output for Reinforcement design, the proposed configuration can be found:

Overall Design (ULS)

Linear calculation
Combination: CO1
Coordinate system: Principal
Extreme 1D: Global
Selection: All

Column B1

Column B1	Rectangle (350; 350)
EC EN 1992-1-1:2004/AC:2008	Section 0 [dx = 0 m]

		Materials	
Member length	Ld = 4.5 m	Concrete	C45/55
Buckling length y	Ly = 9.01 m	Reinforcement	B 500B
Buckling length z	Lz = 9.01 m		

Longitudinal reinforcement

ϕ = 16 mm, c = 30 mm,

Shear reinforcement

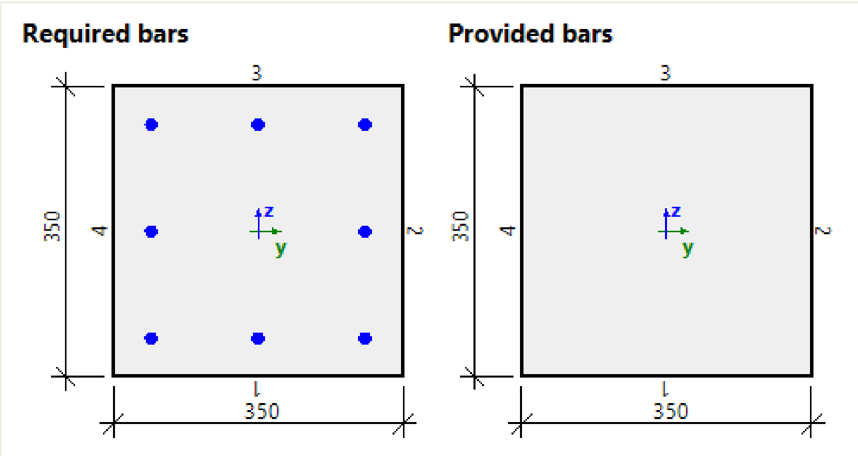
$n_{s,req}$ = 2, $\phi_{s,req}$ = 8 mm, $\alpha_{s,req}$ = 90 °

Design of longitudinal reinforcement

A_s: 1.35*LC1+1.50*LC2 : N_{Ed} = -4200 kN, M_{Edy} = 0 kNm, M_{Edz} = 0 kNm

Required

Edge	Layer	y [m]	z [m]	$A_{s,stat}$ [mm^2]	$A_{s,det,min}$ [mm^2]	$A_{s,det,max}$ [mm^2]	$\Delta A_{s,tor}$ [mm^2]	$A_{s,req}$ [mm^2]	$A_{s,req,bar}$ [mm^2]	Reinf
1	1	0	-0.129	375	0	0	0	375	402	3ϕ16
2	1	0.129	0	375	0	0	0	375	402	3ϕ16
3	1	0	0.129	375	0	0	0	375	402	3ϕ16
4	1	-0.129	0	375	0	0	0	375	402	3ϕ16



Explanation of the number of reinforcement bars

Default bar diameter has been set to ϕ16 in Design default.

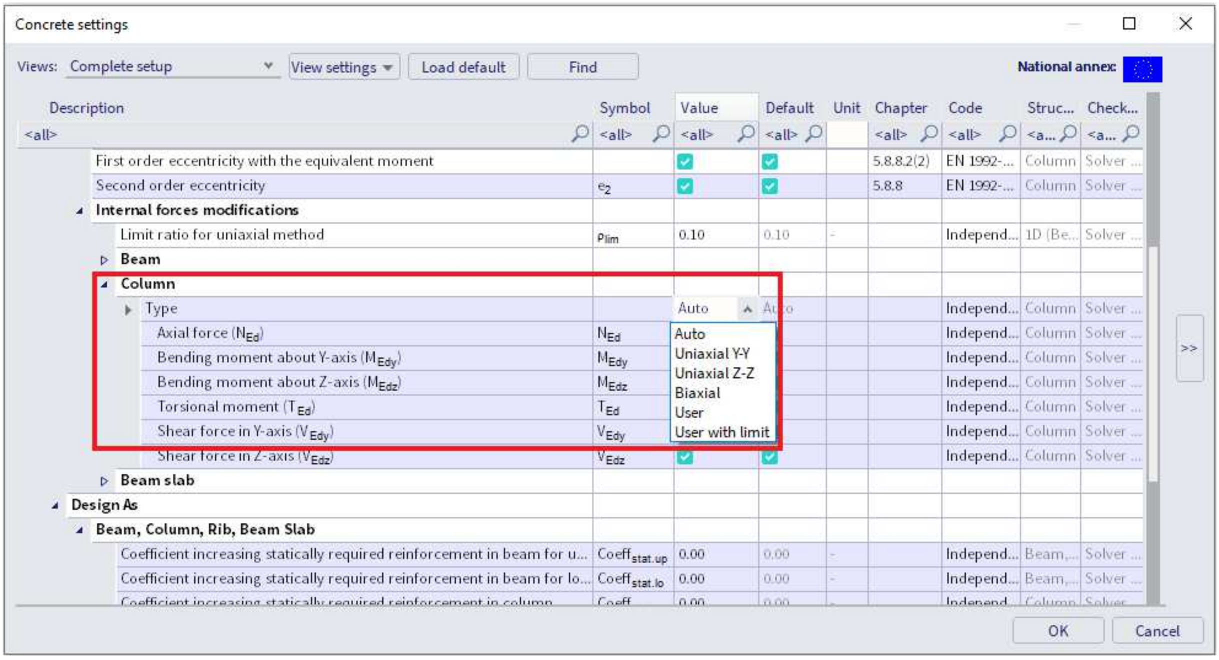
The table indicates that each edge needs 3ϕ16.

On the final picture, this leads to a total of 8ϕ16 in the section of the column.

DESIGN WITH BENDING MOMENT AND AXIAL FORCE

Four calculation methods are available in SCIA Engineer in concrete settings > Design As > Beam, Column, Rib, … > Design method:

- Auto (by default)
- Uniaxial around y axis
- Uniaxial around z axis
- Biaxial (always used for circular and oval columns)



The “Auto” selection of the design method is based on the limit ratio of bending moment for the uniaxial method. The program will automatically select the uniaxial or biaxial method depending on the values of bending moments around y and z axis.

Rule for automatic selection of the design method:

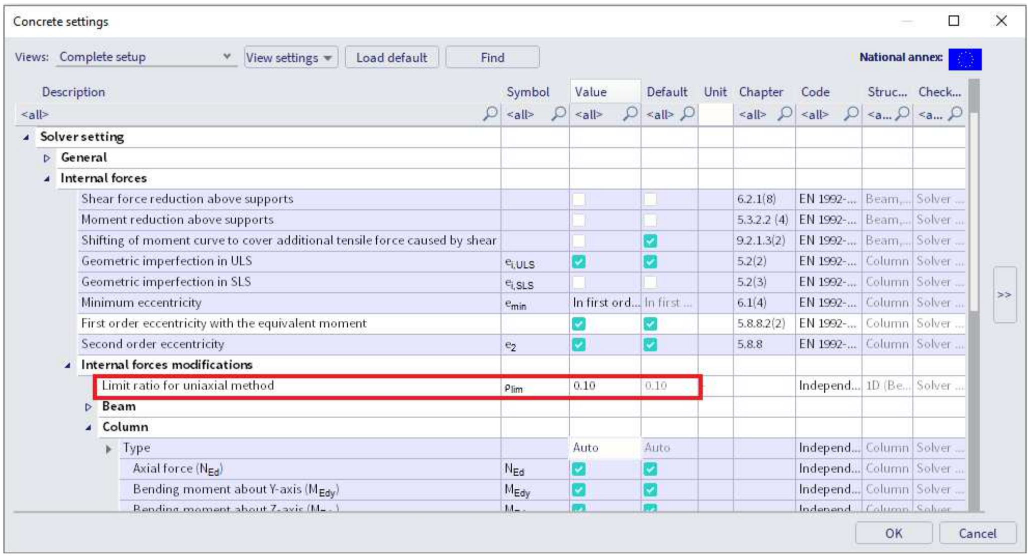
- If $\rho_M \leq \rho_{M,lim}$ Uniaxial method
- If $\rho_M \geq \rho_{M,lim}$ Biaxial method

$$\rho_M = \frac{\mathrm{Min}\{|MEd_{y,max}|,|MEd_{z,max}|\}}{\mathrm{Max}\{|MEd_{y,max}|,|MEd_{z,max}|\}}$$

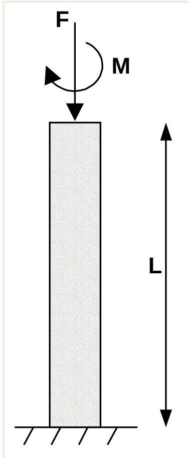
With:

- $M_{Edy.max}$ maximal design moment around y axis from all combinations in current section
- $M_{Edz.max}$ maximal design moment around z axis from all combinations in current section
- $\rho_{M,lim}$ limit ratio of bending moments for uniaxial method loaded from Concrete settings

Settings for limit ratio:



⇨ *Uniaxial bending calculation*



Principle

The reinforcement is designed for N_{Ed} and one bending moment $M_{Ed,y}$ or $M_{Ed,z}$:

- Uniaxial around y: M_{Edz} is ignored, the reinforcement is designed only for N_{Ed} and $M_{Ed,y}$
- Uniaxial around z: M_{Edy} is ignored, the reinforcement is designed only for N_{Ed} and $M_{Ed,z}$

If Auto selection of design method is selected and $\rho_M \leq \rho_{M,lim}$, the rule to choose between uniaxial method around y or z is:

- If $M_{Ed,y} > M_{Ed,z} \rightarrow A_s = A_{sy}$ is designed for forces N_{Ed} and $M_{Ed,y}$
- If $M_{Ed,z} > M_{Ed,y} \rightarrow A_s = A_{sz}$ is designed for forces N_{Ed} and $M_{Ed,z}$

Example: Uniaxial bending.esa

Geometry

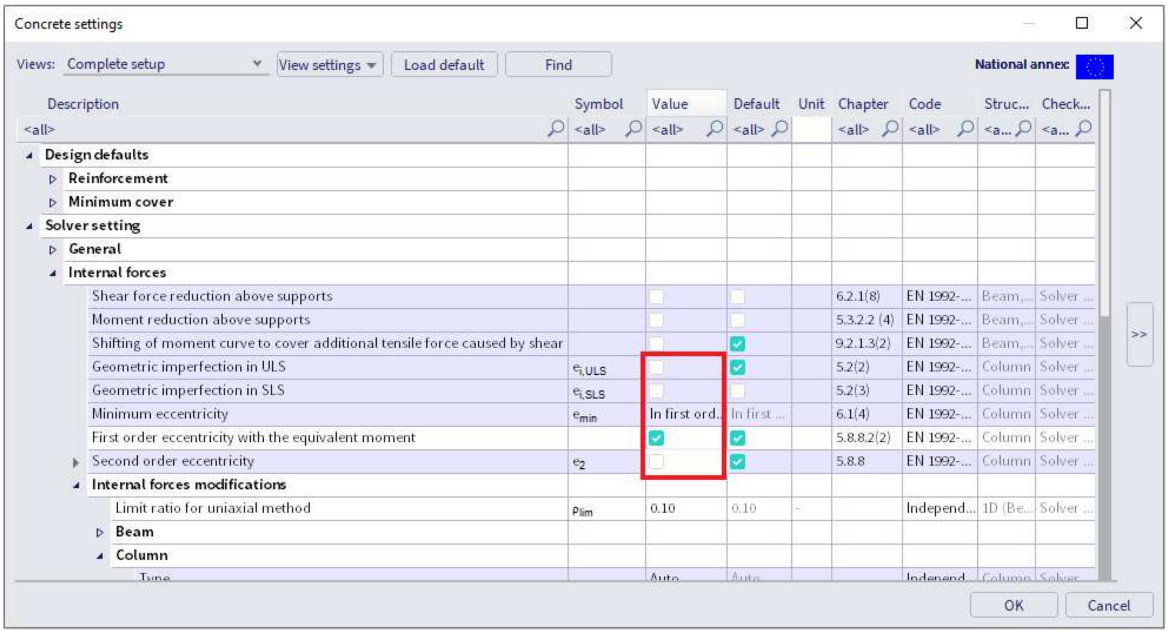
Column cross-section: RECT 350x350mm²

Height: 4,5 m

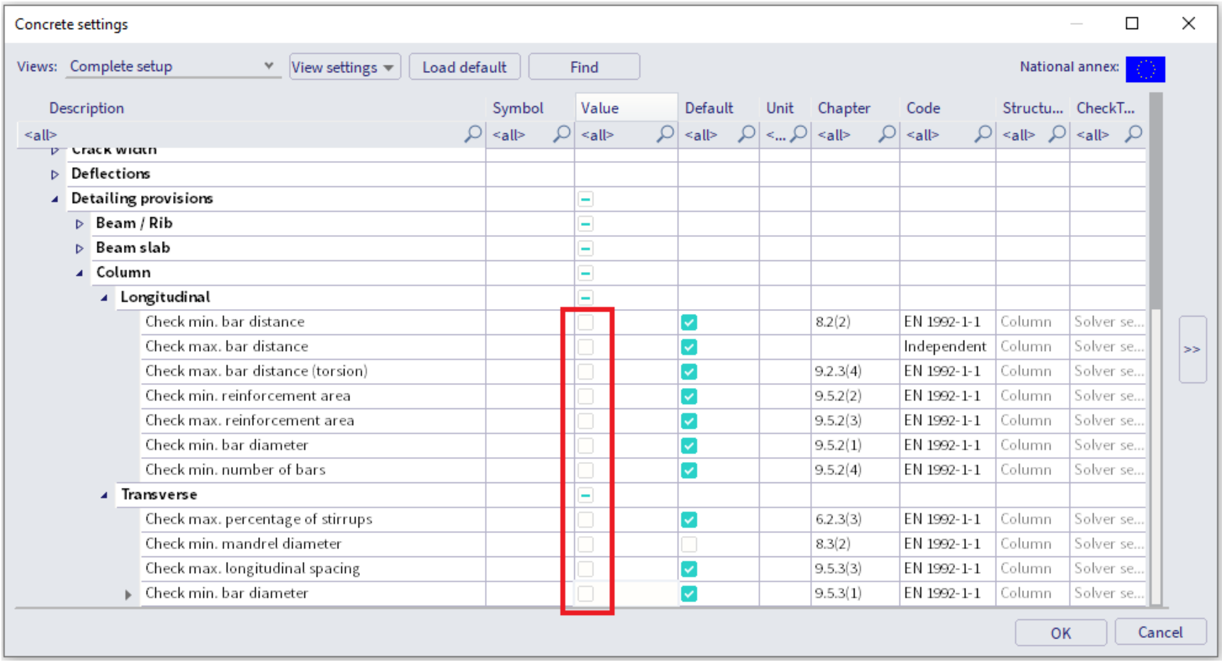
Concrete grade: C45/55

Concrete Setup

Item Concrete settings > Internal forces ULS: 'eccentricities' are not taken in account (only 1st order moments are considered).



Item Detailing provisions are not taken in account, to view the pure results (according to the Eurocode, always a minimum reinforcement percentage must be added).



Loads

Column B1:

- LC1: Permanent load > F = 500 kN; M_y = 100 kNm
- LC2: Variable load > F = 1000 kN; M_y = 100 kNm

Column B2:

- LC1: Permanent load > F = 500 kN; M_y = 100 kNm
- LC2: Variable load > F = 1000 kN; M_y = 100 kNm; M_z = 10 kNm

Combination according to the Eurocode:

ULS Combination = 1,35 * LC1 + 1,50 * LC2

Design normal force N_{Ed} = 1,35 * 500 + 1,50 * 1000 = 2175 kN

Design moment M_{yd} = 1,35 * 100 + 1,50 * 100 = 285 kNm

Additional design moment in column B2 M_{zd} = 22,5 kNm

Results

Go to Reinforcement design > 1D members > Reinforcement design, ask the value for $A_{s,req}$, and click the action buttons [Refresh] and [Preview].

Looking at the Detailed output for column B1:

Determination type of calculation

Calculation maximum bending moments around y and z axis

$M_{y.max}$ = -285 kNm $M_{z.max}$ = 0 kNm

Calculation maximum ratio of bending moments

ρ_M = 0

Determination type of calculation

$\rho_M = 0 < \rho_{M,lim} = 0.1$ and $|M_{y.max}|$ = 285 kNm > $|M_{z.max}|$ = 0 kNm =>

=> Uniaxial method around y axis. Moment M_z will not take into account (M_z = 0 kNm).

The numerical results of the calculation are as follows (standard output):

Column B1	**RECT (350; 350)**
EC EN 1992-1-1:2004/AC:2008	Section 0 [dx = 0 m]

Member length Ld = 4.5 m

Buckling length y Ly = 9.01 m

Buckling length z Lz = 9.01 m

Materials

Concrete C45/55

Reinforcement B 500A

Longitudinal reinforcement

ϕ = 16 mm, c = 30 mm,

Shear reinforcement

$n_{s.req}$ = 2, $\phi_{s.req}$ = 8 mm, $\alpha_{s.req}$ = 90 °

Design of longitudinal reinforcement

$A_{s.z+}$: 1.35*LC1+1.50*LC2 : N_{Ed} = -2175 kN, M_{Edy} = -285 kNm, M_{Edz} = 0 kNm

$A_{s.z-}$: 1.35*LC1+1.50*LC2 : N_{Ed} = -2175 kN, M_{Edy} = -285 kNm, M_{Edz} = 0 kNm

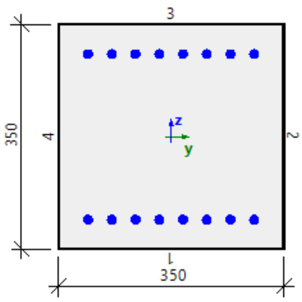
Required

Edge	Layer	y [m]	z [m]	$A_{s.stat}$ [mm^2]	$A_{s.det.min}$ [mm^2]	$A_{s.det.max}$ [mm^2]	$\Delta A_{s.tor}$ [mm^2]	$A_{s.req}$ [mm^2]	$A_{s.req.bar}$ [mm^2]	Reinf
1	1	0	-0.129	1552	0	0	0	1552	1608	8ϕ16
3	1	0	0.129	1552	0	0	0	1552	1608	8ϕ16

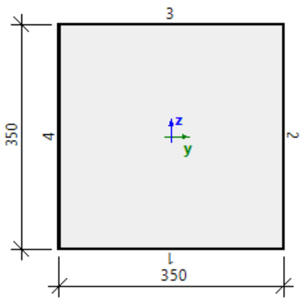
Summary of reinforcement

Top :	$A_{sz.req+}$ = 1552 mm^2	$A_{sz.prov+}$ = 0 mm^2
Bottom :	$A_{sz.req-}$ = 1552 mm^2	$A_{sz.prov-}$ = 0 mm^2
Right :	$A_{sy.req+}$ = 0 mm^2	$A_{sy.prov+}$ = 0 mm^2
Left :	$A_{sy.req-}$ = 0 mm^2	$A_{sy.prov-}$ = 0 mm^2
Total vertical :	$A_{sz.req}$ = 3104 mm^2	$A_{sz.prov}$ = 0 mm^2
Total horizontal :	$A_{sy.req}$ = 0 mm^2	$A_{sy.prov}$ = 0 mm^2
Total :	$A_{s.req}$ = 3104 mm^2	$A_{s.prov}$ = 0 mm^2

Required bars



Provided bars



Looking at the Detailed output for column B2:

Determination type of calculation

Calculation maximum bending moments around y and z axis

$M_{y.max}$ = -285 kNm $M_{z.max}$ = -22.5 kNm

Calculation maximum ratio of bending moments

ρ_M = 0.0789

Determination type of calculation

ρ_M = 0.0789 < $\rho_{M,lim}$ = 0.1 and $|M_{y.max}|$ = 285 kNm > $|M_{z.max}|$ = 22.5 kNm =>

= > Uniaxial method around y axis. Moment M_z will not take into account (M_z = 0 kNm).

And the Standard output:

Column B2

EC EN 1992-1-1:2004/AC:2008

RECT (350; 350)

Section 0 [dx = 0 m]

Member length	Ld = 4.5 m
Buckling length y	Ly = 9.01 m
Buckling length z	Lz = 9.01 m

Materials

Concrete	C45/55
Reinforcement	B 500A

Longitudinal reinforcement

ϕ = 16 mm, c = 30 mm,

Shear reinforcement

$n_{s,req}$ = 2, $\phi_{s,req}$ = 8 mm, $\alpha_{s,req}$ = 90 °

Design of longitudinal reinforcement

$A_{s,z+}$: 1.35*LC1+1.50*LC2 : N_{Ed} = -2175 kN, M_{Edy} = -285 kNm, M_{Edz} = 0 kNm

$A_{s,z-}$: 1.35*LC1+1.50*LC2 : N_{Ed} = -2175 kN, M_{Edy} = -285 kNm, M_{Edz} = 0 kNm

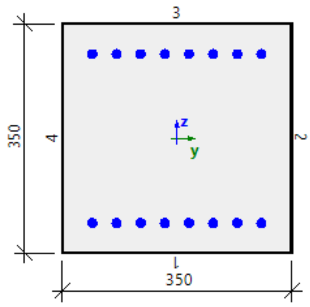
Required

Edge	Layer	y [m]	z [m]	$A_{s,stat}$ [mm²]	$A_{s,det,min}$ [mm²]	$A_{s,det,max}$ [mm²]	$\Delta A_{s,tor}$ [mm²]	$A_{s,req}$ [mm²]	$A_{s,req,bar}$ [mm²]	Reinf
1	1	0	-0.129	1552	0	0	0	1552	1608	8ϕ16
3	1	0	0.129	1552	0	0	0	1552	1608	8ϕ16

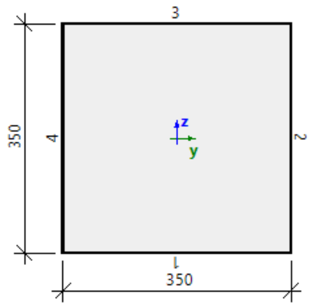
Summary of reinforcement

Top :	$A_{sz,req+}$ = 1552 mm²	$A_{sz,prov+}$ = 0 mm²
Bottom :	$A_{sz,req-}$ = 1552 mm²	$A_{sz,prov-}$ = 0 mm²
Right :	$A_{sy,req+}$ = 0 mm²	$A_{sy,prov+}$ = 0 mm²
Left :	$A_{sy,req-}$ = 0 mm²	$A_{sy,prov-}$ = 0 mm²
Total vertical :	$A_{sz,req}$ = 3104 mm²	$A_{sz,prov}$ = 0 mm²
Total horizontal :	$A_{sy,req}$ = 0 mm²	$A_{sy,prov}$ = 0 mm²
Total :	$A_{s,req}$ = 3104 mm²	$A_{s,prov}$ = 0 mm²

Required bars

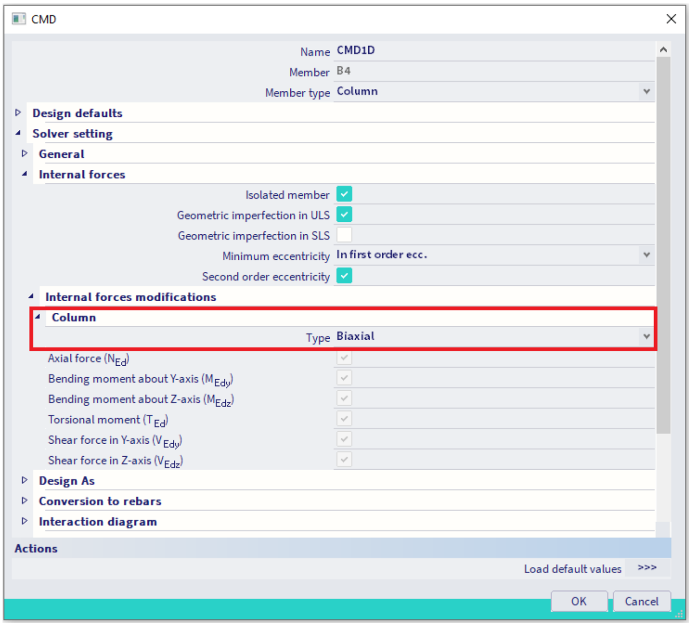


Provided bars



Even if an additional bending moment in the z direction is present in column B2, according to the limit ratio the uniaxial method was used, and the same amount of reinforcement is required for columns B1 and B2.

You have the possibility to force the biaxial method design on column B2 using 1D member data:



Amount of required reinforcement will be slightly higher in this case since M_{Edz} is also considered.

Column B2	**RECT (350; 350)**
EC EN 1992-1-1:2004/AC:2008	Section 0 [dx = 0 m]

		Materials	
Member length	Ld = 4.5 m		
Buckling length y	Ly = 9.01 m	Concrete	C45/55
Buckling length z	Lz = 9.01 m	Reinforcement	B 500A

Longitudinal reinforcement

ϕ = 16 mm, c = 30 mm,

Shear reinforcement

$n_{s.req}$ = 2, $\phi_{s.req}$ = 8 mm, $\alpha_{s.req}$ = 90 °

Design of longitudinal reinforcement

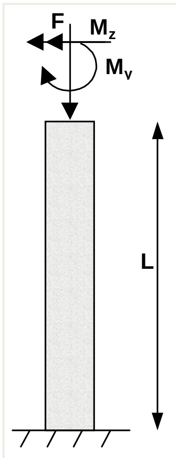
$A_{s.z+}$: 1.35*LC1+1.50*LC2 : N_{Ed} = -2175 kN, M_{Edy} = -285 kNm, M_{Edz} = -23 kNm

$A_{s.z-}$: 1.35*LC1+1.50*LC2 : N_{Ed} = -2175 kN, M_{Edy} = -285 kNm, M_{Edz} = -23 kNm

Required

Edge	Layer	y [m]	z [m]	$A_{s.stat}$ [mm²]	$A_{s.det.min}$ [mm²]	$A_{s.det.max}$ [mm²]	$\Delta A_{s.tor}$ [mm²]	$A_{s.req}$ [mm²]	$A_{s.req.bar}$ [mm²]	Reinf
1	1	0	-0.129	2046	0	0	0	2046	2212	11ϕ16
3	1	0	0.129	2046	0	0	0	2046	2212	11ϕ16

⇨ *Biaxial bending calculation*



This method allows to design reinforcement for a normal force (N_{Ed}) and biaxial bending moments. This method is based on an interaction formula, equation 5.39 in EN 1992-1-1.

$$\left(\frac{M_{Edz}}{M_{Rdz}}\right)^a + \left(\frac{M_{Edy}}{M_{Rdy}}\right)^a \leq 1{,}0 \qquad (5.39)$$

where:

- $M_{Edz/y}$ design moment, including a 2nd order moment (if required)
- $M_{Rdz/y}$ moment resistance
- a exponent:
  for circular and elliptical cross sections: a = 2
  for rectangular cross sections:

*N*Ed/*N*Rd	0,1	0,7	1,0
a =	1,0	1,5	2,0

  with linear interpolation for intermediate values
  - N_{Ed} design value of axial force
  - N_{Rd} = $A_c \cdot (f_{cd} + \mu_s \cdot f_{yd})$, design axial resistance of the section, where:
    - A_c gross area of the concrete section
    - f_{cd} design value of concrete compressive strength
    - f_{yd} design yield strength of reinforcement
    - μ_s mechanical reinforcement ratio in the calculation of limit slenderness obtained with an iterative calculation

CIRCULAR COLUMN

For circular and oval columns, the design method is always the biaxial calculation, regardless of the design method set in the Concrete settings.

For circular and oval columns, the required number of reinforcement bars is spread equally along the face of the column.

Example: Circular column.esa

Geometry

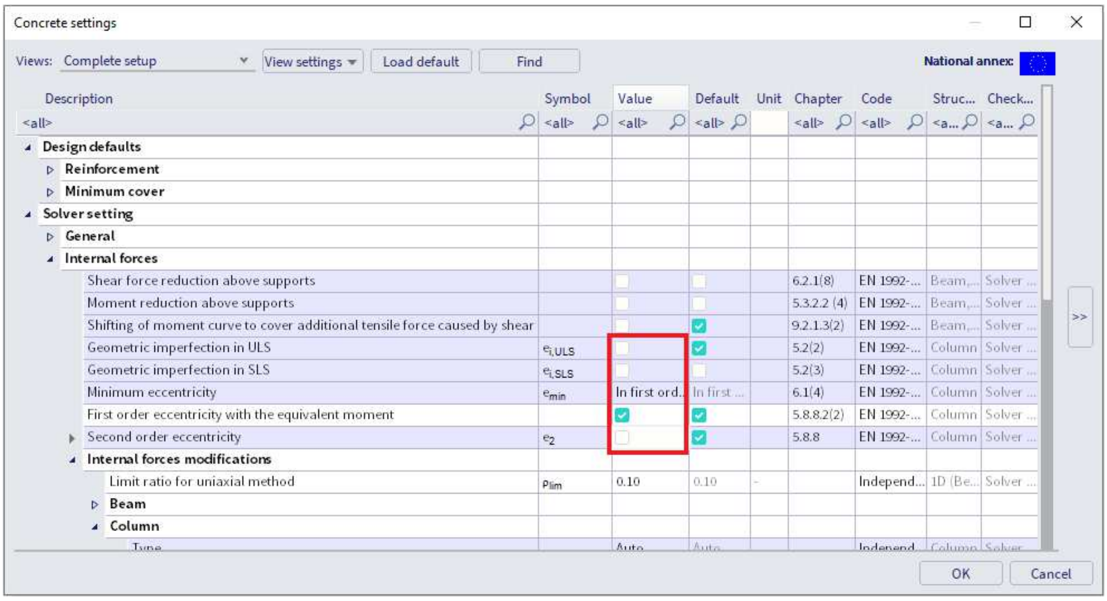
Column cross-section: CIRC diameter 400mm
Height: 4,5 m
Concrete grade: C45/55

Loads

Load configuration: N_{Ed} = 2175,00kN
M_{yd} = 142,50kNm
M_{zd} = 0kNm

Concrete Setup

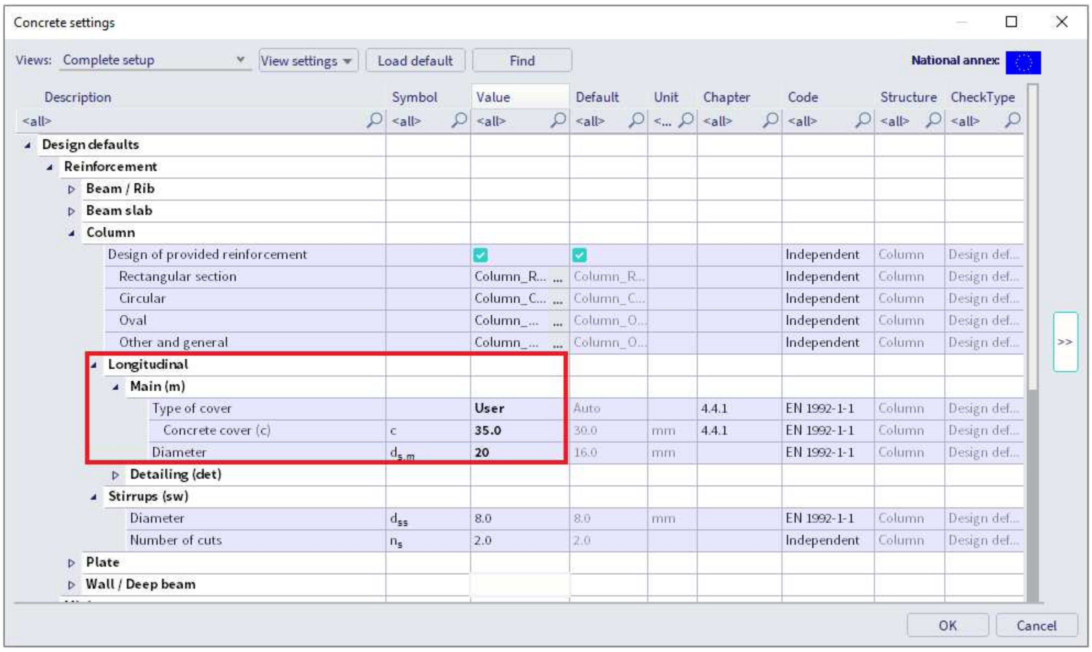
Geometrical imperfection and 2nd order moments are deactivated: Concrete settings > Complete Setup view :



All detailing provisions are considered.

Design defaults

The bar diameter is set to ϕ20 mm in Reinforcement design > Design defaults > Tab Columns, or from 1D Member data if applied.



Results

Go to Reinforcement design > 1D members > Reinforcement design.

Choose Standard output in the Properties window and open the Preview at the bottom of the Properties window:

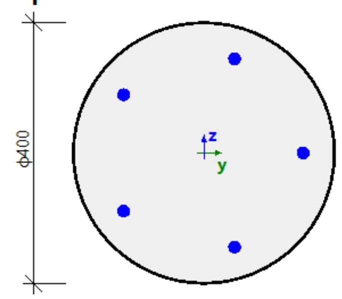
Required

Edge	Layer	y [m]	z [m]	$A_{s,stat}$ [mm²]	$A_{s,det,min}$ [mm²]	$A_{s,det,max}$ [mm²]	$\Delta A_{s,tor}$ [mm²]	$A_{s,req}$ [mm²]	$A_{s,req,bar}$ [mm²]	Reinf
-	-	-	-	1142	1257	5001	0	1257	1571	5ϕ20*

Summary of reinforcement

Total : $A_{s,req}$ = 1257 mm²

Required bars



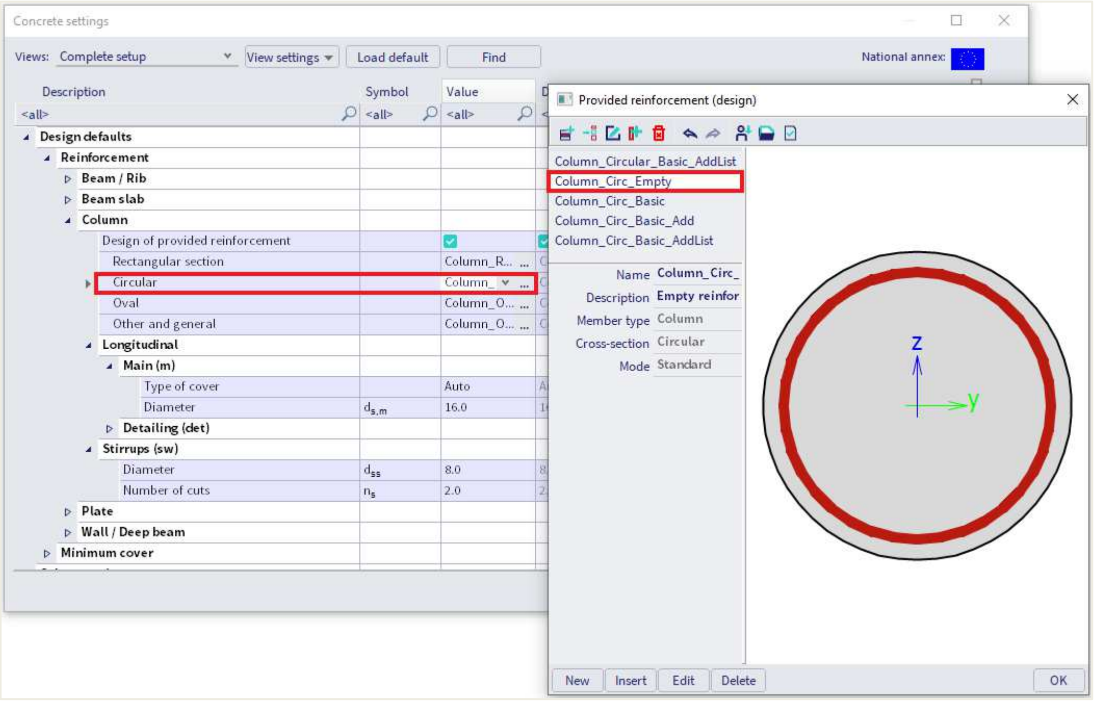
In this example $A_{s,req}$ is determined by the minimum amount of reinforcement according to the detailing provision, $A_{s,det,min}$.

Since $A_{s,req}$ = 1257mm², the software will propose 5 bars of ϕ20mm (5*314mm² = 1571mm² = $A_{s,req,bar}$) which is the closest amount of bar with $A_{s,req,bar} > A_{s,req}$.

Note that SCIA Engineer uses the real area of the bars to calculate the required reinforcement area.

So, the final required reinforcement displayed on the screen is $A_{s,req,bar}$.

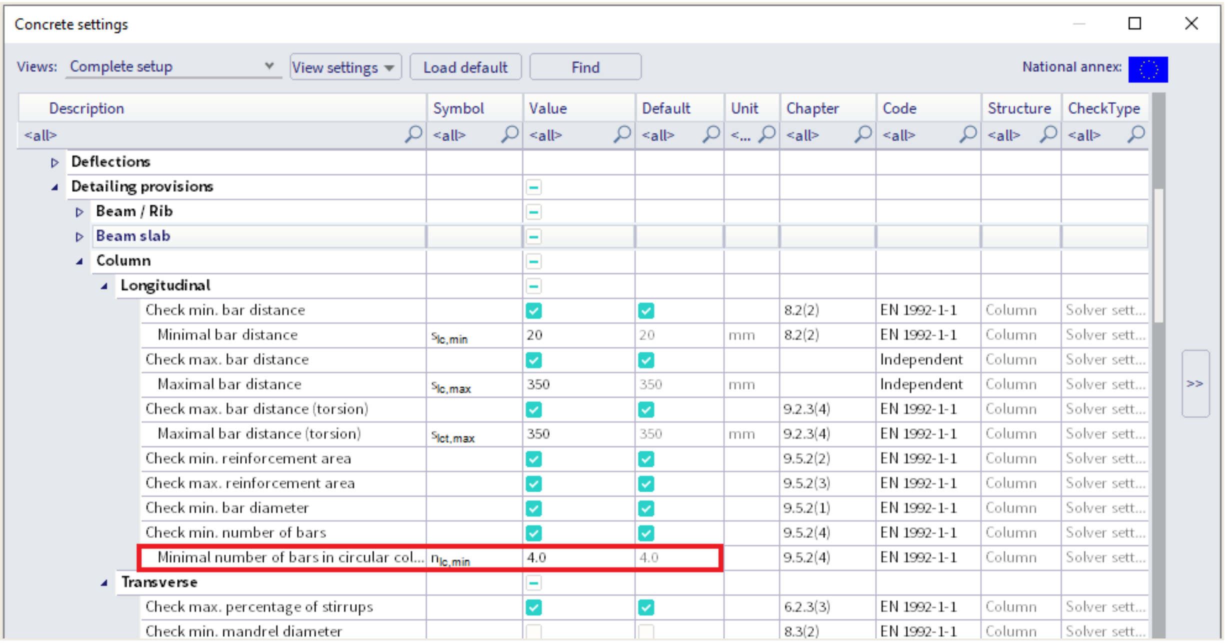
Remark 1: If you choose a template without bars predefined in Design Default, for example "Column_Circ-Empty", the software will display only the $A_{s,req}$ and not $A_{s,req,bar}$ as mentioned above.



Remark 2:

According to *EN1992-1-1 art 9.5.2(4)*, there is a minimum number of bars in a circular column.

This parameter is set by default to "4" in Concrete Settings > Complete setup view.



If we increase the loads:

F_z = -1250kN

M = 50kNm

The results are as follows.

Example: Circular column_increase.esa

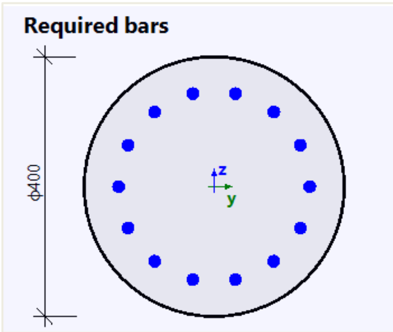
Overall Design (ULS)

Linear calculation
Combination: CO1
Coordinate system: Principal
Extreme 1D: Global
Selection: All

Longitudinal required reinforcement

Name	dx [m]	Case	Member	A_{sz} req+ [mm²] A_{sz} req bar+ [mm²]	A_{sz} req- [mm²] A_{sz} req bar- [mm²]	A_{sy} req+ [mm²] A_{sy} req bar+ [mm²]	A_{sy} req- [mm²] A_{sy} req bar- [mm²]	A_{sz} req [mm²] A_{sz} req bar [mm²]	A_{sy} req [mm²] A_{sy} req bar [mm²]	A_s req [mm²] A_s req bar [mm²]	ReinfReq
B1	0,000	CO1	Column	1041 1100	1041 1100	1041 1100	1041 1100	2082 2199	2082 2199	4164 4398	14ϕ20

The corresponding bar configuration is:



2.3.2 Calculation of internal forces

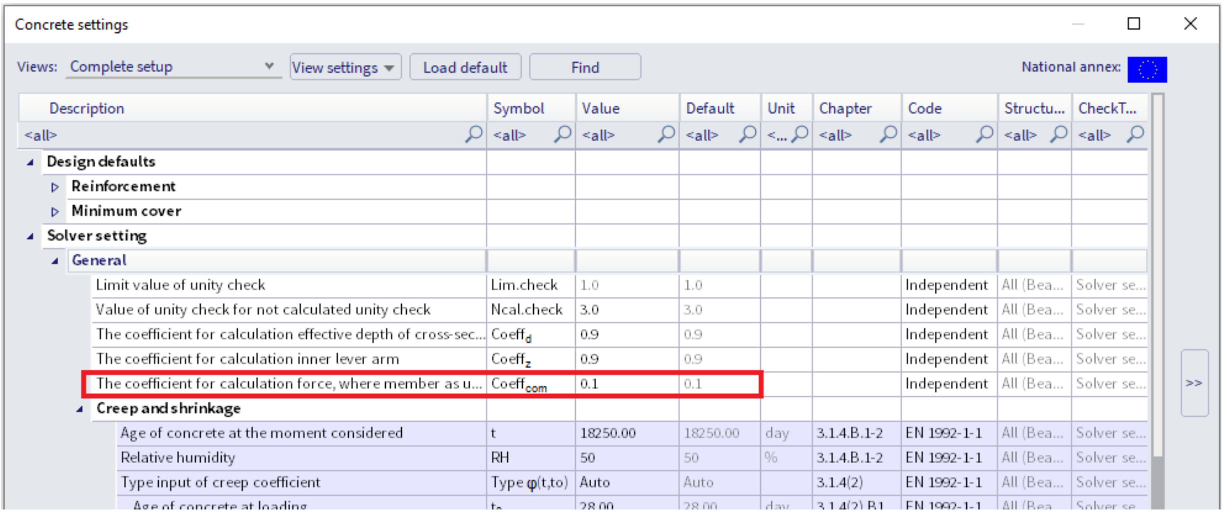
DETERMINING IF MEMBER IS IN COMPRESSION

2nd order effects, geometrical imperfection and minimal eccentricity are considered only if:

- Member type = Column
- Compression in the column is relatively high

In SCIA Engineer, there is a parameter which allows to decide whether a member is in compression or if the compression is too small to be considered.

In Concrete settings > Complete setup view:



Condition is:

- If $N_{Ed} \leq -\ Coeff_{com} * f_{cd} * A_c$ Member is in compression
- If $N_{Ed} > -\ Coeff_{com} * f_{cd} * A_c$ Compression is not sufficient (zero or relatively small)

This result can be viewed in Reinforcement design > 1D member > Internal forces.

The Detailed output gives:

Compression member

Limit axial force to consider member as compression:

$$N_{com} = -\,Coeff_{com} \cdot \left(f_{cd} \cdot A_c\right) = -0.1 \cdot \left(30 \cdot 10^6 \cdot 0.123\right) = -368\ \text{kN}$$

Check condition:

$$N_{Ed} < N_{com} = -1100\ \text{kN} < -368\ \text{kN} \ldots \quad \text{compression member}$$

> Note: First and second order eccentricity shall be taken into account, because the member is considered as a compression member (significant normal force is presented).

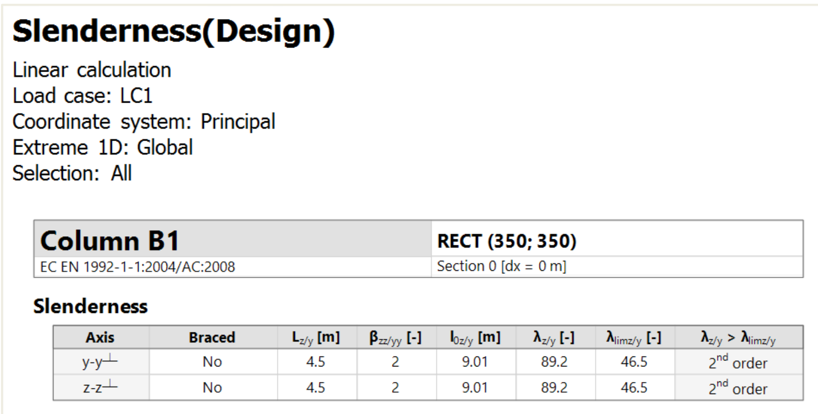
CHOICE BETWEEN 1st and 2nd ORDER CALCULATION

Slenderness – Check of the criteria $\lambda < \lambda_{lim}$

- If $\lambda < \lambda_{lim}$, 1st order effects have to be taken into account with geometric imperfection (art 5.2)
- If $\lambda > \lambda_{lim}$, 2nd order effects have to be taken into account with geometric imperfection (art 5.2)

The values for λ and λ_{lim}, and the corresponding check, can be found in the main menu Deign > Concrete 1D > Slenderness for design :

The Standard output shows the check of $\lambda > \lambda_{lim}$ and indicates whether a 1st or 2nd order calculation should be done.



Slenderness(Design)

Linear calculation
Load case: LC1
Coordinate system: Principal
Extreme 1D: Global
Selection: All

Column B1	**RECT (350; 350)**
EC EN 1992-1-1:2004/AC:2008	Section 0 [dx = 0 m]

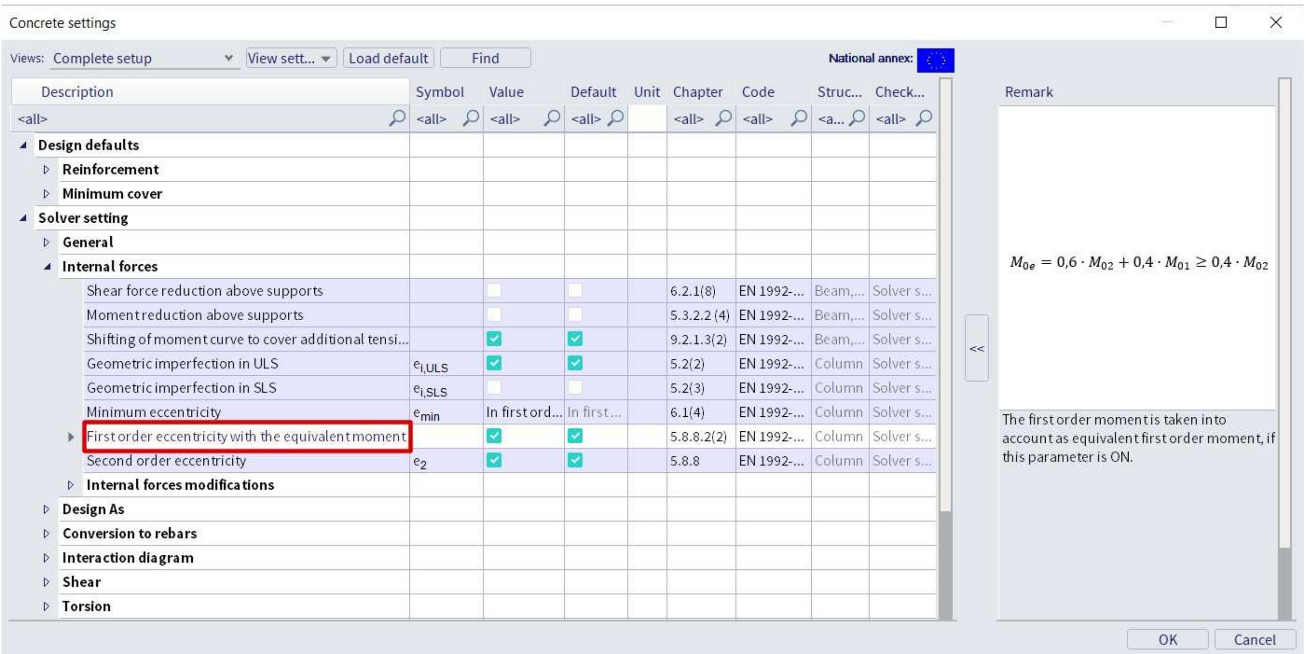
Slenderness

Axis	Braced	$L_{z/y}$ [m]	$\beta_{zz/yy}$ [-]	$l_{0z/y}$ [m]	$\lambda_{z/y}$ [-]	$\lambda_{limz/y}$ [-]	$\lambda_{z/y} > \lambda_{limz/y}$
y-y	No	4.5	2	9.01	89.2	46.5	2nd order
z-z	No	4.5	2	9.01	89.2	46.5	2nd order

1^{st} ORDER EFFECTS

1st order effects (eccentricity) are always considered.

There are 2 ways to calculate the 1^{st} order moments and eccentricity in SCIA Engineer depending on check box **First order eccentricity with the equivalent moment** in Concrete Setup > Solver setting > Internal forces.



The 2 options are:

- **First order eccentricity with the equivalent moment = YES**, bending moments at the ends of the column will be taken to calculate an equivalent 1^{st} order bending moment. This leads to the same 1^{st} order bending moment along the whole length of the member.

$$e_{0y} = M_{0ez}/N_{Ed} \text{ and } e_{0z} = M_{0ey}/N_{Ed}$$

With

$$M_{0e} = (0{,}6 * M_{02}) + (0{,}6 * M_{01}) \geq 0{,}4 * M_{02}$$

- **First order eccentricity with the equivalent moment = NO**, 1^{st} order eccentricity is calculated from bending moments in current section. As a result, bending moments in each section can be different.

$$e_{0y} = M_z/N_{Ed} \text{ and } e_{0z} = M_y/N_{Ed}$$

Values of the 1st order eccentricities and moments can be viewed in Design > Concrete 1D > Internal forces for design.

Standard output gives:

Internal forces (Design)

Linear calculation
Combination: ULS
Coordinate system: Principal
Extreme 1D: Global
Selection: All

Column B1	**RECT (350; 350)**
EC EN 1992-1-1:2004/AC:2008	Section 0 [dx = 0 m]

Internal forces (FEM-based)

Extreme: ULS/1 (ULS)
Type: Combination (linear)
Design situation: EN-ULS (STR/GEO) Set B

Type of load	N [kN]	M_y [kNm]	M_z [kNm]	V_y [kN]	V_z [kN]	M_x [kNm]
Internal forces (FEM-based)	-300.0	-30.0	0.0	0.0	0.0	0.0

Content: LC1

Slenderness

Axis	Braced	$L_{z/y}$ [m]	$\beta_{zz/yy}$ [-]	$l_{0z/y}$ [m]	$\lambda_{z/y}$ [-]	$\lambda_{limz/y}$ [-]	$\lambda_{z/y} > \lambda_{limz/y}$
y-y⊥	No	4.5	2	9.01	89.2	46.5	2nd order
z-z⊥	No	4.5	2	9.01	89.2	46.5	2nd order

Unfavourable direction

Second order effect and imperfections

Axis	N_{Ed} [kN]	$M_{0Edy/z}$ [kNm]	$M_{2y/z}$ [kNm]	$M_{Edy/z}$ [kNm]	$e_{0z/y}$ [mm]	$e_{iz/y}$ [mm]	$e_{0min,z/y}$ [mm]	$e_{0Edz/y}$ [mm]	$e_{2z/y}$ [mm]	$e_{Edz/y}$ [mm]
y-y⊥	-300	-30	0	-30	100	0	0	100	0	100
z-z⊥	-300	0	0	0	0	0	0	0	0	0

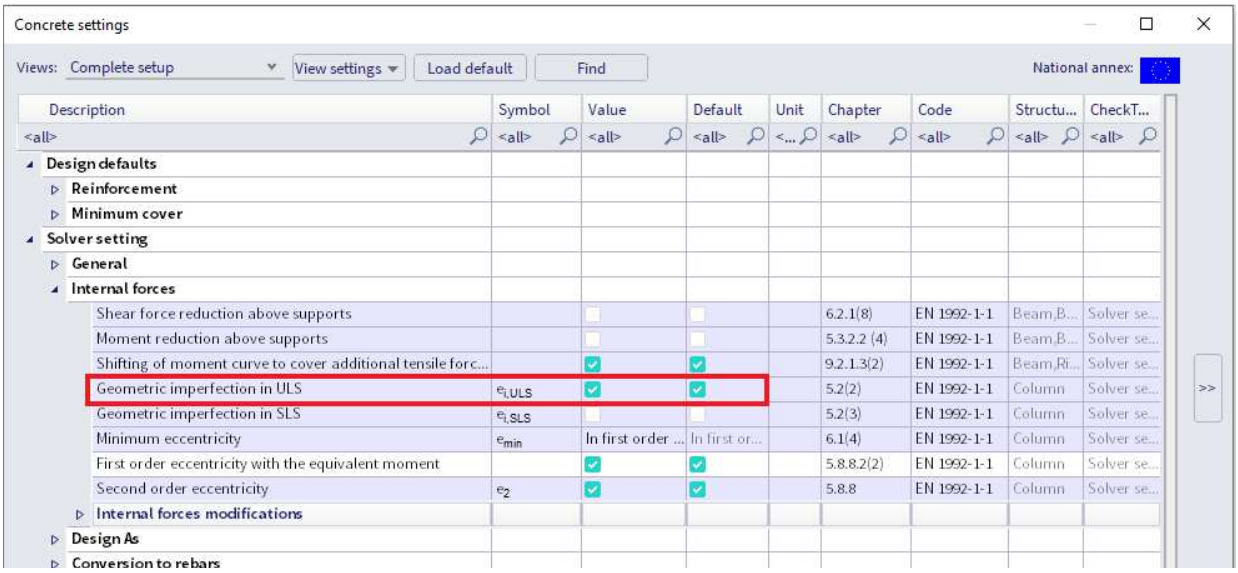
Design forces (recalculated)

Type of load	N_{Ed} [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,z}$ [kNm]	$V_{Ed,y}$ [kN]	$V_{Ed,z}$ [kN]	$M_{Ed,x}$ [kNm]
Design forces (recalculated)	-300.0	-30.0	0.0	0.0	0.0	0.0

GEOMETRICAL IMPERFECTION (art 5.2)

The effect of geometric imperfections always have to be taken into account: both in a 1st and 2nd order calculation.

Geometrical imperfection is by default activated in Concrete settings > Internal forces



In SCIA Engineer, the geometrical imperfection is represented by an inclination according to clause 5.2(5) in EN 1992-1-1.

For both axis (y and z of LCS), the inclination is calculated as followed:

$$\theta_{i,y(z)} = \theta_0 \cdot \alpha_h \cdot \alpha_{m,y(z)} \tag{5.1}$$

- θ_0 basic value of inclination
- α_h reduction factor for length of column or height of structure: $\alpha_h = 2/\sqrt{l}\,; 2/3 \leq \alpha_h \leq 1$
- $\alpha_{m,y(z)}$ reduction factor for numbers of members: $\alpha_{m,y(z)} = \sqrt{\left(0{,}5 \cdot \left(1 + 1/m_{y(z)}\right)\right)}$
- l length of column or height of structure depending on:
  - o isolated member l = L, where L is the length of the member
  - o not isolated member l = H, where H is the total height of building (buckling system)
- $m_{y(z)}$ number of vertical members contributing to the total effect of the imperfection perpendicular to y(z)

Values of l and $m_{y(z)}$ will be defined in the buckling data.

The effect of imperfection for isolated column and for structure is always taken into account as an eccentricity according to clause 5.2(7a) in EN 1992-1-1:

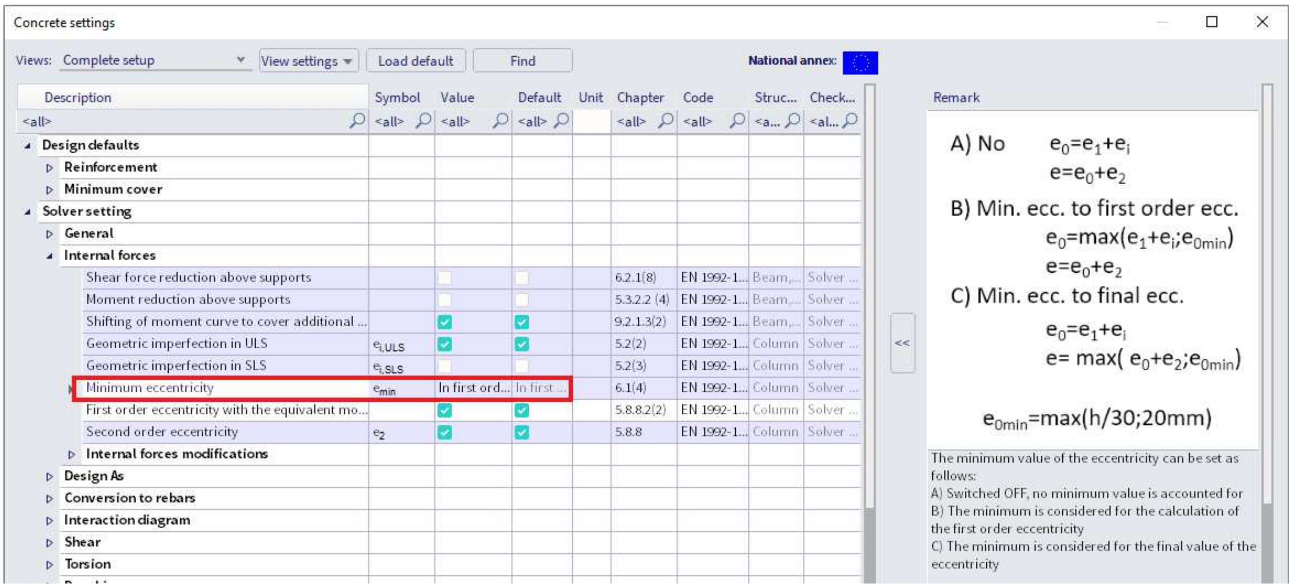
$$e_{i,y} = \theta_{i,z} \cdot l_{0,z}/2, e_{i,z} = \theta_{i,y} \cdot l_{0,y}/2$$

The imperfection shall be taken into account in ultimate limit states and does not need to be considered for serviceability limit states, see clause 5.2(2P) and 5.2(3) in EN 1992-1-1.

You can set independently if the imperfection will be taken into account for ULS or SLS in the Concrete settings.

A minimum 1st order eccentricity is also calculated according to clause 6.1(4) in EN 1992-1-1.

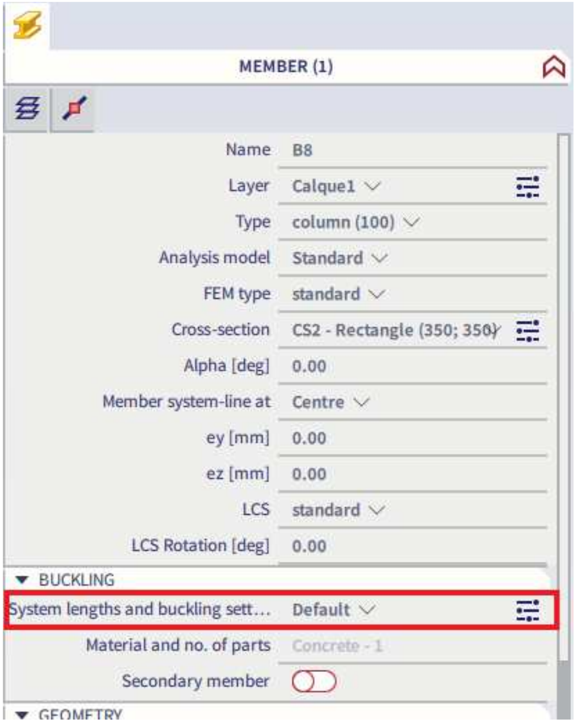
This can be viewed in Concrete settings > Internal forces > Use minimum value of eccentricity

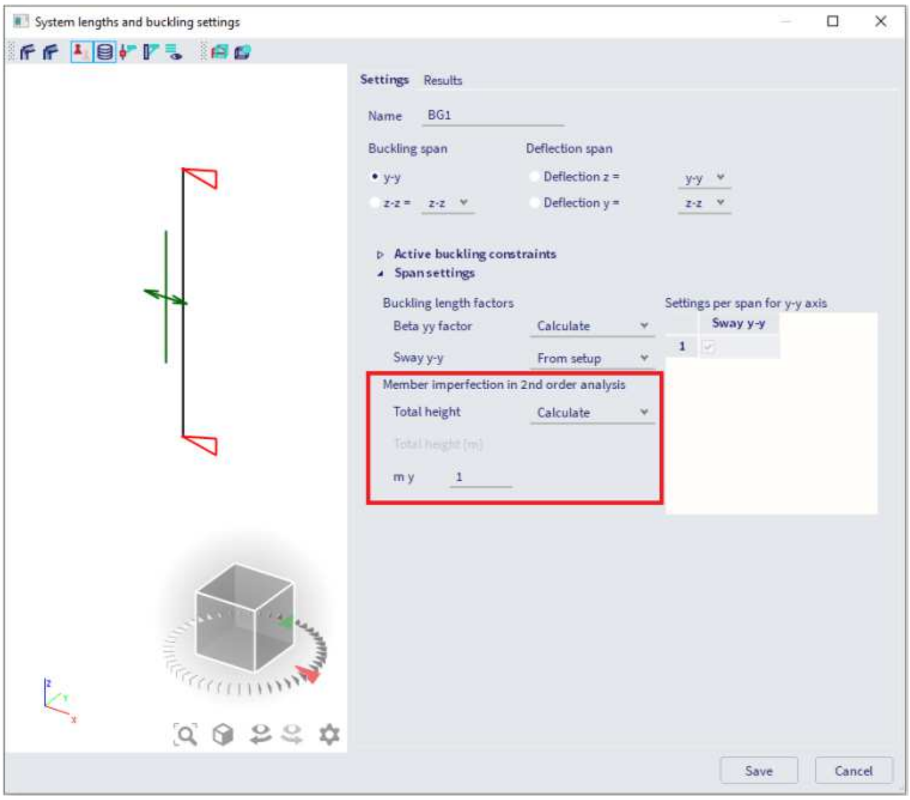


Buckling data for l and $m_y(z)$

Settings for l and $m_y(z)$ for the calculation of the geometrical imperfection can be set in the properties of the columns.

Properties > System lengths and buckling settings





When opening the buckling menu, you need to define both the '**Active buckling constraints**' and '**Span settings**' for buckling around the local y-axis (buckling span y-y) and local z-axis (buckling span z-z).

- **Total height determination**: set type of calculation of total height of building or length of the isolated columns.
    - o *Calculate*: H tot will be calculated automatically as sum of lengths of all the members in the buckling system
    - o *Input*: manual input value for H_{tot} in edit box Tot. height
- **my/z**: number of vertical members contributing to the total effect of the imperfection perpendicular to y/z axis of LCS.

Eccentricities due to geometrical imperfections can be viewed in Reinforcement design > 1D member > Internal Forces:

Second order effect and imperfections

Axis	N_{Ed} [kN]	$M_{0Edy/z}$ [kNm]	$M_{2y/z}$ [kNm]	$M_{Edy/z}$ [kNm]	$e_{0z/y}$ [mm]	$e_{iz/y}$ [mm]	$e_{0min,z/y}$ [mm]	$e_{0Edz/y}$ [mm]	$e_{2z/y}$ [mm]	$e_{Edz/y}$ [mm]
y-y⊥	-405	-49.1	0	-49.1	100	21.2	20	121	0	121
z-z⊥	-405	8.1	0	8.1	0	0	-20	-20	0	-20

After calculation of 1st order eccentricity including effect of imperfection, the 1st order moment, including the effect of imperfections around y (z) axis of LCS is calculated:

$$M_{0Ed,y(z)} = N_{Ed} \cdot e_{0Ed,z(y)} \qquad e_{0Ed,z(y)} = e_{0,y(z)} + e_{i,y(z)} > e_{0,min,y(z)}$$

- $e_{0,y(z)}$ 1st order eccentricity
- $e_{i,y(z)}$ eccentricity caused by geometrical imperfection
- $e_{0,min}$ minimum first order eccentricity

2nd ORDER EFFECTS

The EN 1992-1-1 defines several methods for 2nd order effects with axial loads(general method, simplified method based on nominal stiffness, simplified method based on nominal curvature...).

In SCIA Engineer the following methods are available:

- General method according to clause 5.8.2(2) – based on a nonlinear calculation
- Simplified method based on nominal curvature according to clause 5.8.8
- Simplified method based on nominal stiffness according to clause 5.8.7

The simplified method is taken into account:

- For ultimate limit state
- For Member type = Column with compression according to "Determination if member is in compression"
- If option "Use second order effect" in switched ON, see Concrete settings > Internal forces.
  This option is activated by default.
- If slenderness $\lambda > \lambda_{lim}$, see chapter "Slenderness criteria"

<u>Simplified method based on nominal stiffness</u>

The total design moment, including second order moment, may be expressed as a magnification of the bending moments resulting from a first analysis, namely:

$$M_{Ed} = M_{0Ed} \cdot \left[1 + \frac{\beta}{\left(\frac{N_B}{N_{Ed}}\right) - 1}\right]$$

Where:

- M_{0Ed} is the first order moment
- β is a factor which depends on distribution of 1st and 2nd order moments:
  $\beta = \pi^2 / c_0$
- c_0 is a coefficient which depends on the distribution of first order moment
- N_{Ed} is the design value of axial load
- N_B is the buckling load based on nominal stiffness

<u>Simplified method based on nominal curvature</u>

The nominal 2nd order moment is calculated according to clause 5.8.8.2(3) in EN 1992-1-1:

$$M_{2,y(z)} = N_{Ed} \cdot e_{2,z(y)}$$

With:

- N_{Ed} design axial force
- $e_{2,z(y)}$ 2nd order eccentricity

When all mentioned criteria above are met for the simplified method, the 2nd order eccentricity is calculated according to formula:

$$e_{2y(z)} = (1/r)_{z(y)} \cdot {l_{0z(y)}}^2 / c_{z(y)}$$

Otherwise:

$$e_{2,y(z)=0}$$

With:

- $(1/r)_{z(y)}$curvature around z(y), calculated according to clause 5.8.8.3
  $c_{z(y)}$ factor depending on the curvature distribution around z(y) axis according to clause 5.8.8.2(4)
    - o = 8, for constant 1st order bending moment (non zero) along the column and in case that equivalent bending moment is taken into account ("Use equivalent first order value" ON).
    - o = 10 otherwise.
- $\lambda_{z(y)}$ slenderness
- $\lambda_{z(y),lim}$ limit slenderness
- $l_{0,z(y)}$ effective length of the column around z(y) – buckling length

Effective length

The effective length, or buckling length, is by default calculated by SCIA Engineer. Be aware that formulas for automatic calculation are only valid for simple structures!

Otherwise, it is also possible to input the value of the effective length manually.

Automatic calculation of effective length

Calculation of effective length depends on the type of structure, sway or non-sway.

Two approximate formulas are used: one formula for a non-sway structure (resulting in a buckling factor β ≤ 1) and one formula for a sway structure (resulting in a buckling factor β ≥ 1):

- For a non-sway structure:

$$\beta = \frac{(\rho_1\rho_2 + 5\rho_1 + 5\rho_2 + 24)(\rho_1\rho_2 + 4\rho_1 + 4\rho_2 + 12)2}{(2\rho_1\rho_2 + 11\rho_1 + 5\rho_2 + 24)(2\rho_1\rho_2 + 5\rho_1 + 11\rho_2 + 24)}$$

- For a sway structure:

$$\beta = x\sqrt{\frac{\pi^2}{\rho_1 x} + 4}$$

With:

- B buckling factor
- L system length
- E modulus of Young
- I moment of inertia
- C_i stiffness in node i
- M_i moment in node i
- Φ_i rotation in node i

$$x = \frac{4\rho_1\rho_2 + \pi^2\rho_1}{\pi^2(\rho_1 + \rho_2) + 8\rho_1\rho_2}$$

$$\rho_i = \frac{C_i L}{EI} \qquad C_i = \frac{M_i}{\phi_i}$$

The values for M_i and ϕ_i are approximately determined by the internal forces and the deformations, calculated by load cases which generate deformation forms, having an affinity with the buckling form.

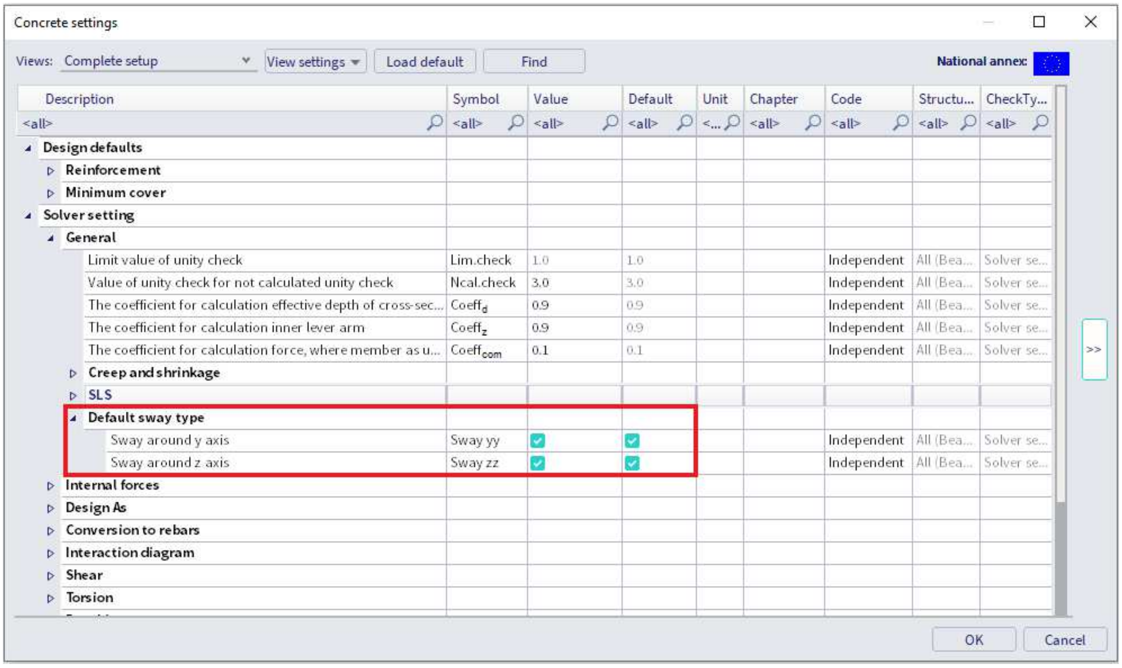
The calculation of the β ratios is automatically done when calculating the structure linearly. For this, two additional load cases are calculated in the background:

- Load case 1:
  - o on the beams, the local distributed loads qy=1 N/m and qz=-100 N/m are used
  - o on the columns the global distributed loads Qx =10000 N/m and Qy =10000 N/m are used
- Load case 2:
  - o on the beams, the local distributed loads qy=-1 N/m and qz=-100 N/m are used
  - o on the columns the global distributed loads Qx =-10000 N/m and Qy=-10000 N/m are used

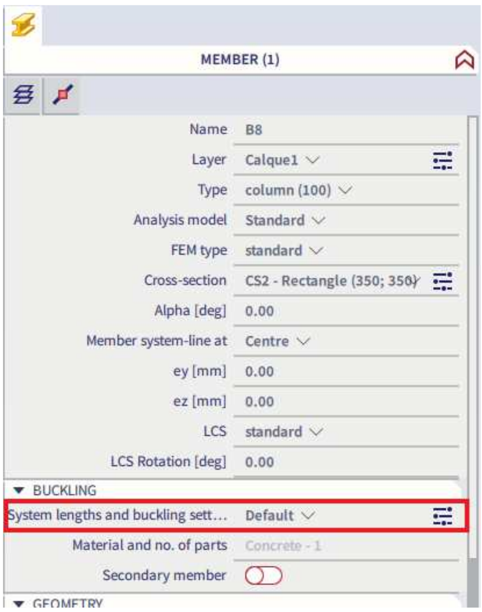
Since these load cases, and thus the buckling ratios, are calculated during the linear calculation, it is necessary to always perform a linear calculation of the structure.

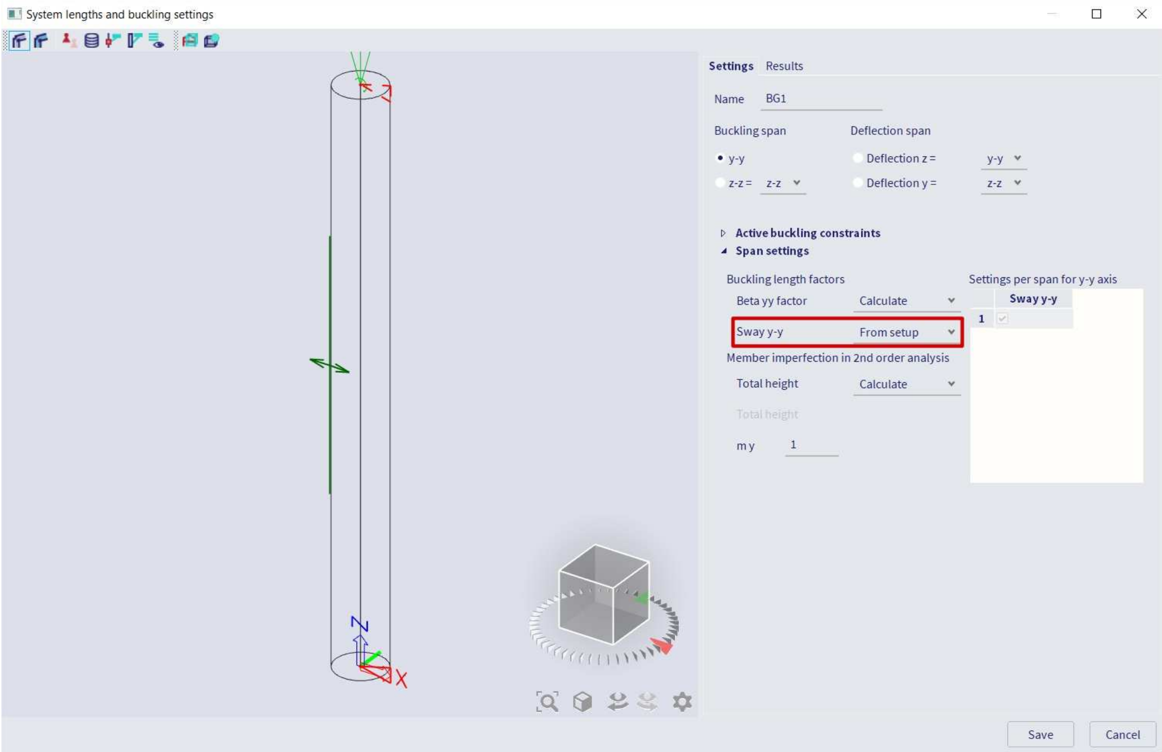
<u>Note:</u> The used approach gives good results for frame structures with perpendicular rigid or semi-rigid beam connections. For other cases, you must evaluate the presented bucking ratios.

By default, the structure is considered as sway in y and z direction. It can be modified for the whole project in Concrete settings > General > Default sway type.

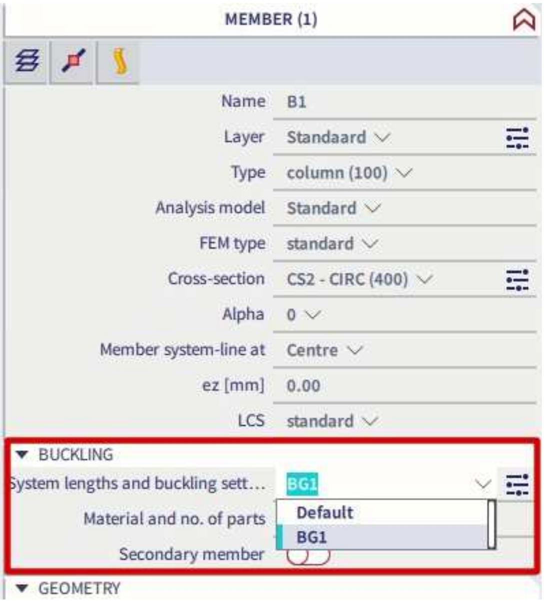


You can easily modify these default settings for a specific column in the project within the buckling menu. This menu can be accessed – as explained in the previous section – by navigating to the option '**System lengths and buckling settings**' within the properties of the member.

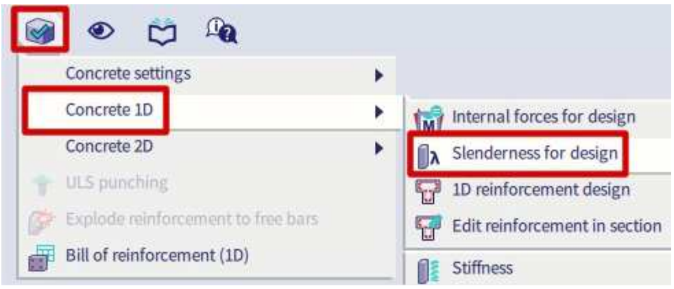


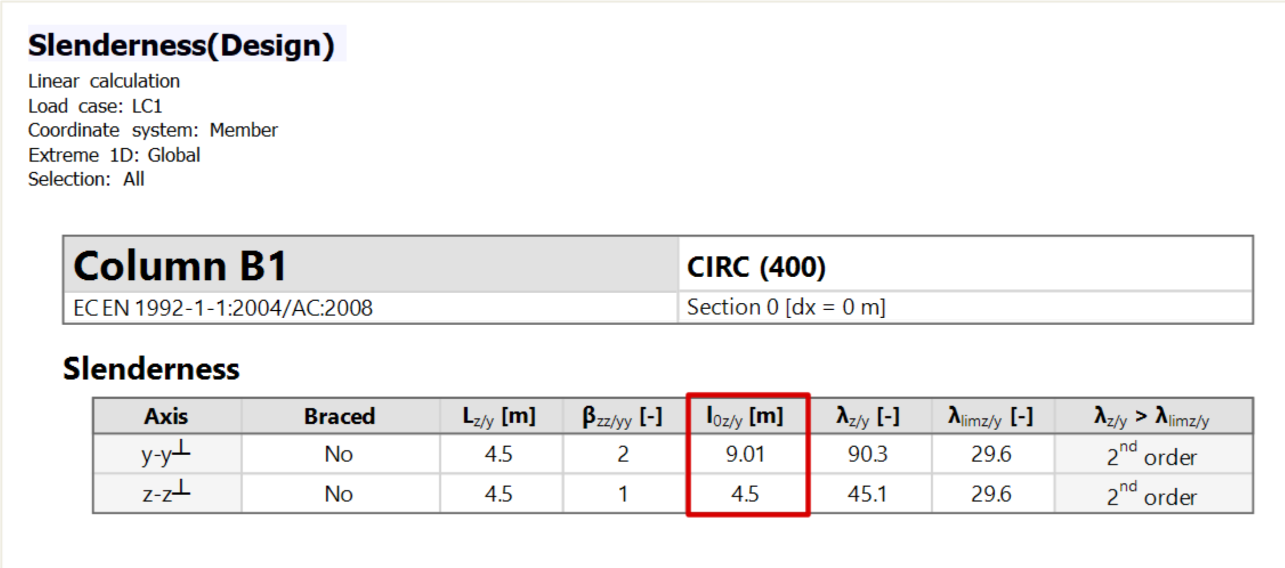


This new setting has the name, here **BG1**, which you can attribute to others similar columns in their properties window:



The calculated effective length can be viewed in Design menu > Concrete 1D > Slenderness for design:





Slenderness(Design)

Linear calculation
Load case: LC1
Coordinate system: Member
Extreme 1D: Global
Selection: All

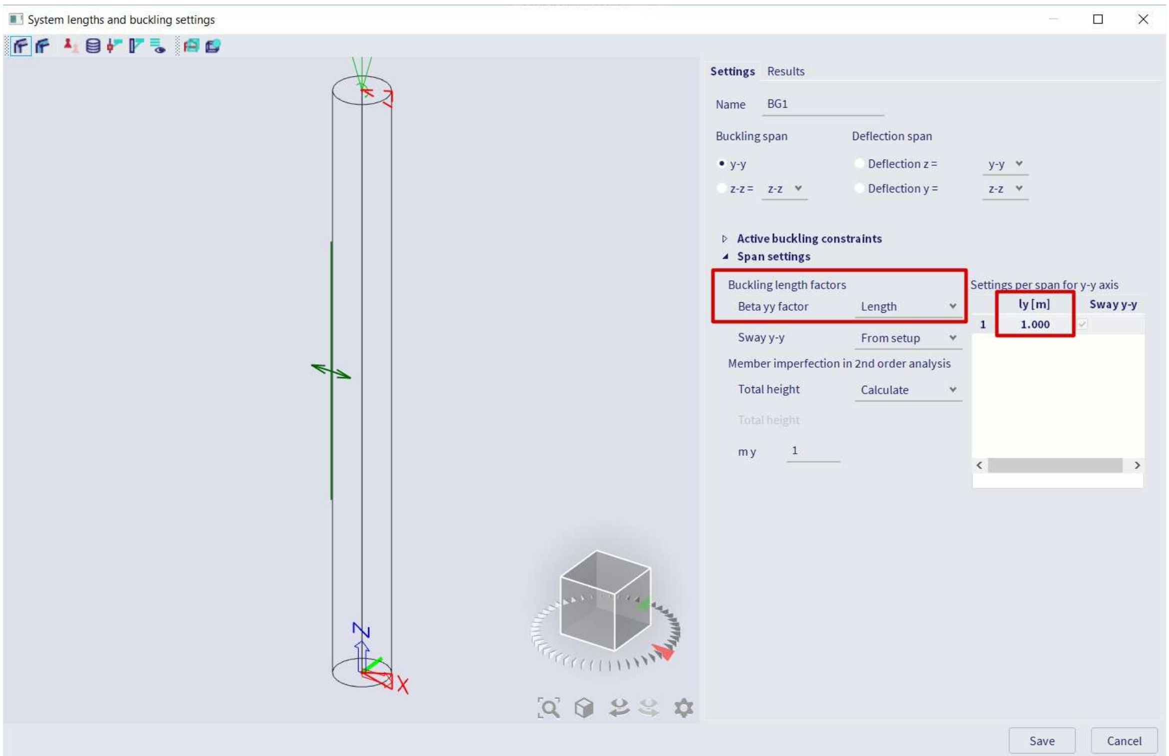
Column B1	**CIRC (400)**
EC EN 1992-1-1:2004/AC:2008	Section 0 [dx = 0 m]

Slenderness

Axis	Braced	$L_{z/y}$ [m]	$\beta_{zz/yy}$ [-]	$l_{0z/y}$ [m]	$\lambda_{z/y}$ [-]	$\lambda_{limz/y}$ [-]	$\lambda_{z/y} > \lambda_{limz/y}$
y-y⊥	No	4.5	2	9.01	90.3	29.6	2nd order
z-z⊥	No	4.5	1	4.5	45.1	29.6	2nd order

Manual input of effective length

The same option – as seen for the automatic calculation – allows you to manually define the buckling length of the system. The option '**Buckling length factors**' can be accessed within the section '**Span settings**'. In the table '**Settings per span for y-y/z-z axis**' you can insert the buckling length which needs to be taken into account.



Comparison of the two simplified methods

Whatever the reinforcement in the column, the method based on nominal curvature gives more or less the same result, while the method based on the nominal stiffness is very affected by the reinforcement of the column.

And on the other hand, the method based on the nominal stiffness is not usable anymore if the applied normal effort N_{Ed} is too close to the buckling effort N_B.

RECALCULATED INTERNAL FORCES

In Concrete Menu > Reinforcement Design > 1D member > Internal forces.

The design moment, M_{Ed}, is equal to $M_{Ed} = M_{0Ed} + M_2$.

With:

- M_2 2nd order bending moment
- M_{0Ed} bending moment taking into account 1st order and geometrical imperfections

Example: 2nd order.esa

Geometry

Column cross-section: RECT 350x350mm²
Height: 4,5 m
Concrete grade: C45/55

Concrete Setup

All of the default values are kept.

This means that geometrical imperfection and 2nd order effects are taken into account.

Loads

Load configuration: N_d = 405,00 kN
M_{yd} = 40,50 kNm
M_{zd} = 0 kNm

Buckling data

Sway type is set by default.

Calculation of the effective length is done automatically by the software.

Slenderness criterion

Check if 2nd order calculation is required following art 5.8.3.1:

Since $\lambda > \lambda_{lim}$, a 2nd order calculation will be required.

Note: the program automatically takes into account a second order moment if required. So, this check is just extra information for the user.

Internal forces

Ask for M_{Ed} in Design > Concrete 1D > Internal forces for design.

The Standard output is chosen:

Internal forces (FEM-based)

Extreme: ULS/2 (ULS)
Type: Combination (linear)
Design situation: EN-ULS (STR/GEO) Set B

Type of load	N [kN]	M_y [kNm]	M_z [kNm]	V_y [kN]	V_z [kN]	M_x [kNm]
Internal forces (FEM-based)	-405.0	-40.5	0.0	0.0	0.0	0.0

Content: 1.35*LC1

Second order effect and imperfections

Axis	N_{Ed} [kN]	$M_{0Edy/z}$ [kNm]	$M_{2y/z}$ [kNm]	$M_{Edy/z}$ [kNm]	$e_{0z/y}$ [mm]	$e_{iz/y}$ [mm]	$e_{0min,z/y}$ [mm]	$e_{0Edz/y}$ [mm]	$e_{2z/y}$ [mm]	$e_{Edz/y}$ [mm]
y-y⊥	-405	-49.1	-73.2	-122	100	21.2	20	121	181	302
z-z⊥	-405	8.6	63	71.6	0	-21.2	-20	-21.2	-156	-177

Design forces (recalculated)

Type of load	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	V_{Edy} [kN]	V_{Edz} [kN]	M_{Edx} [kNm]
Design forces (recalculated)	-405.0	-122.3	71.6	0.0	0.0	0.0

Results

The results for the reinforcement design are shown below:

Design forces

Case	N_{Ed} [kN]	V_{Edy} [kN]	V_{Edz} [kN]	T_{Ed} [kNm]	M_{Edy} [kNm]	M_{Edz} [kNm]	λ/λ_{lim} y-y⊥		λ/λ_{lim} z-z⊥	
ULS/1	-300.0	0.0	0.0	0.0	-30.0	0.0	-	-	-	-
ULS/2	-405.0	0.0	0.0	0.0	-122.3	71.6	2.38	2nd	2.38	2nd

ULS/1	LC1
ULS/2	1.35*LC1

Longitudinal reinforcement

	Basic	Additional	Detailing	$A_{s,ult}$ [mm²]	$A_{s,min}$ [mm²]	$A_{s,req}$ [mm²]	$A_{s,prov}$ [mm²]	$A_{s,max}$ [mm²]	s_{min} [mm]	s_{max} [mm]	Status
[1]	**2ϕ16**	---	---	500	61	**500**	**402**	1225	242	258	**Not OK**
									≥37	-	
[2]	**2ϕ16**	---	---	385	61	**385**	**402**	1225	70	86	**OK**
									≥37	-	
[3]	**2ϕ16**	---	---	500	61	**500**	**402**	1225	242	258	**Not OK**
									≥37	-	
[4]	**2ϕ16**	---	---	385	61	**385**	**402**	1225	70	86	**OK**
									≥37	-	
ΣY	**4ϕ16**	---	---			**1000**	**804**				
ΣZ	**4ϕ16**	---	---			**770**	**804**				
Σ	**8ϕ16**	---	---			**1770**	**1608**				

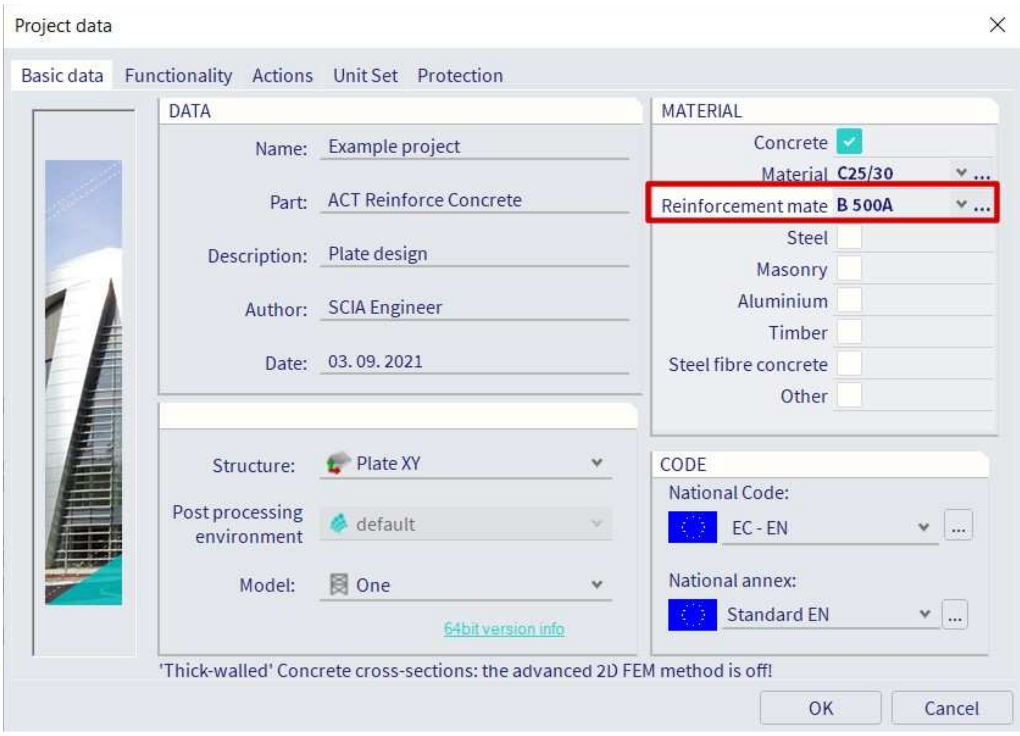
Note that biaxial bending method was used for reinforcement calculation.

2.4 Plate design

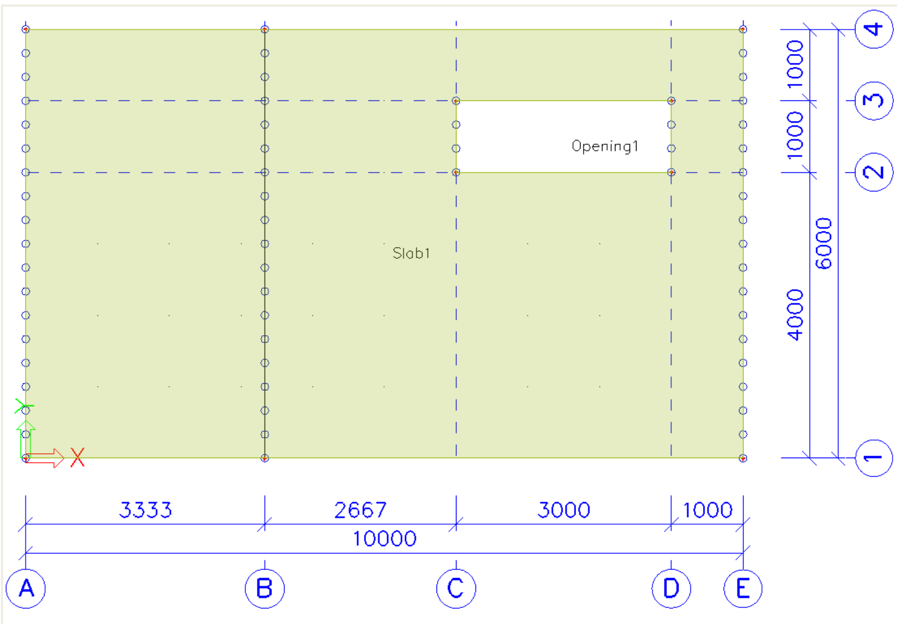
2.4.1 Used example

INPUT OF GEOMETRY

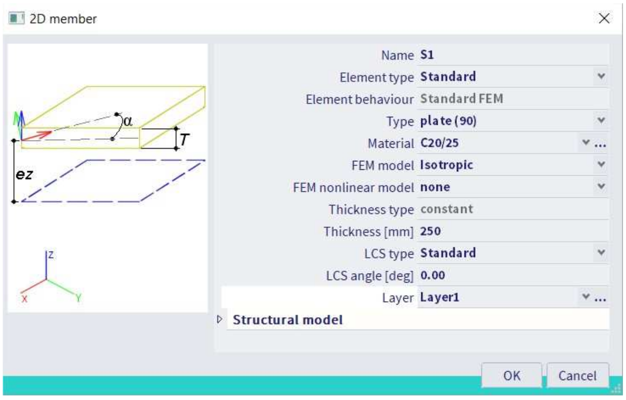
Project data: 2D environment = Plate XY

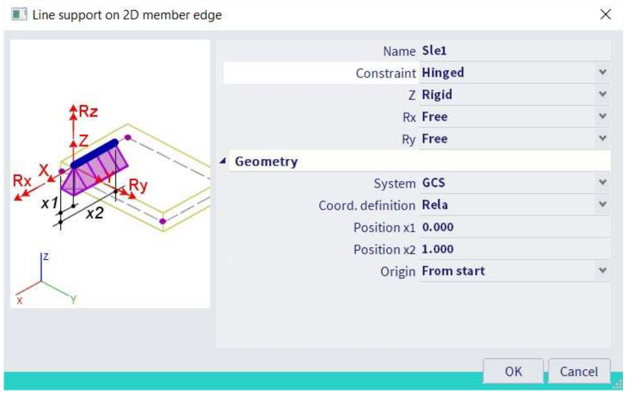


The Reinforcement material (e.g. B500A) chosen in the Project data window, will define the steel quality used for the theoretical reinforcement design.



Properties of the slab and the line supports:

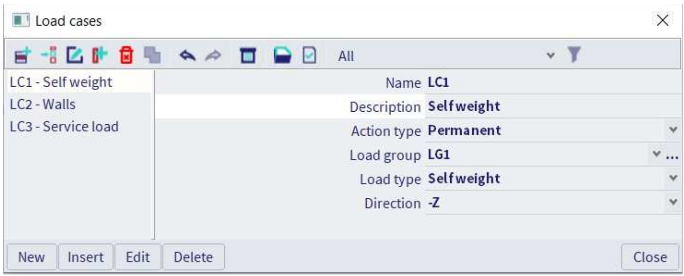




LOADS

⇨ Load cases & Load groups

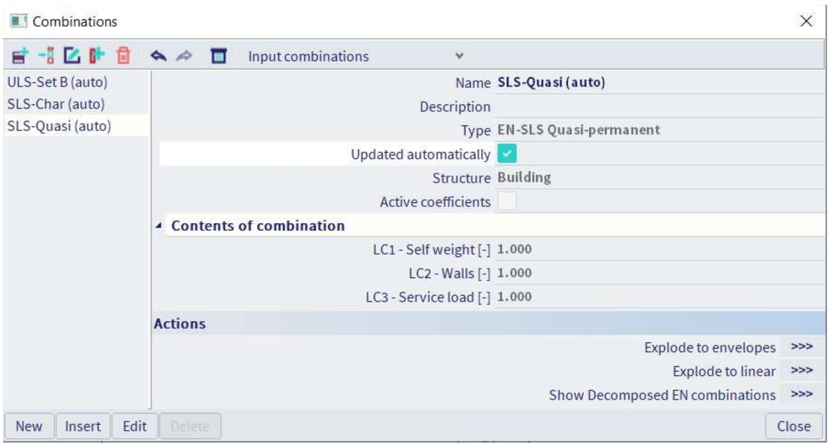
Load Case	Action type	Load Group	Relation	EC1-Load type
Self-weight	Permanent	LG1	/	/
Walls	Permanent	LG1	/	/
Service load	Variable	LG2	Standard	Cat B: Offices



⇨ Load combinations

Type EN-ULS (STR/GEO) Set B

Type EN-SLS Quasi Permanent



⇨ *Result classes*

All ULS+SLS

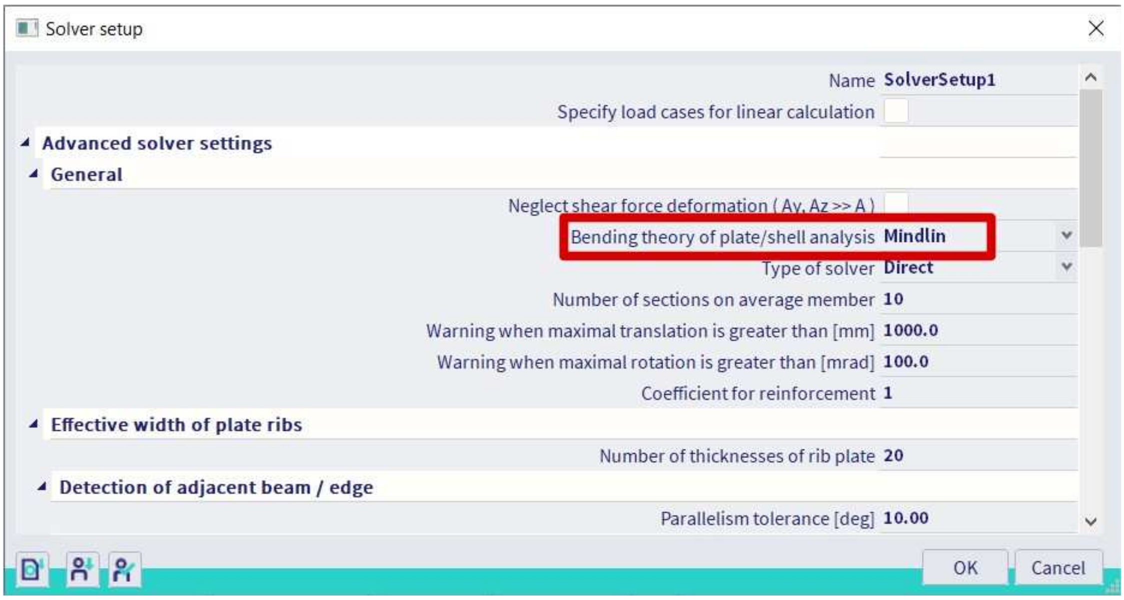
FINITE ELEMENT MESH

⇨ *Introduction*

2 types of finite elements are implemented in SCIA Engineer:

- The **Mindlin element** including shear force deformation, which is the standard in SCIA Engineer. The Mindlin theory is valid for the calculation of both thin and thick plates.
- The **Kirchhoff element** without shear force deformation, which can be used to calculate and design only thin plates.

The element type used for the current calculation is defined in the Tools menu > Calculation & Mesh > Solver Settings:



⇨ *Mesh generation*

Via the tools menu → Calculation & mesh → Generate mesh

⇨ ***Graphical display of the mesh***

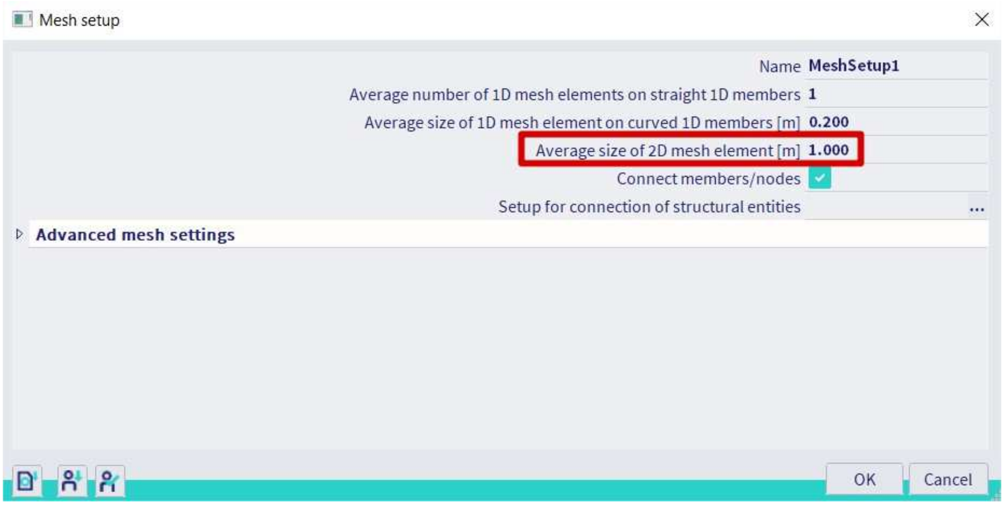
Set view settings for all entities, via right mouse click in screen or more options > View settings for all entities

- Structure tab → Mesh → Draw mesh
- Labels tab → Mesh → Display label

⇨ ***Mesh refinement***

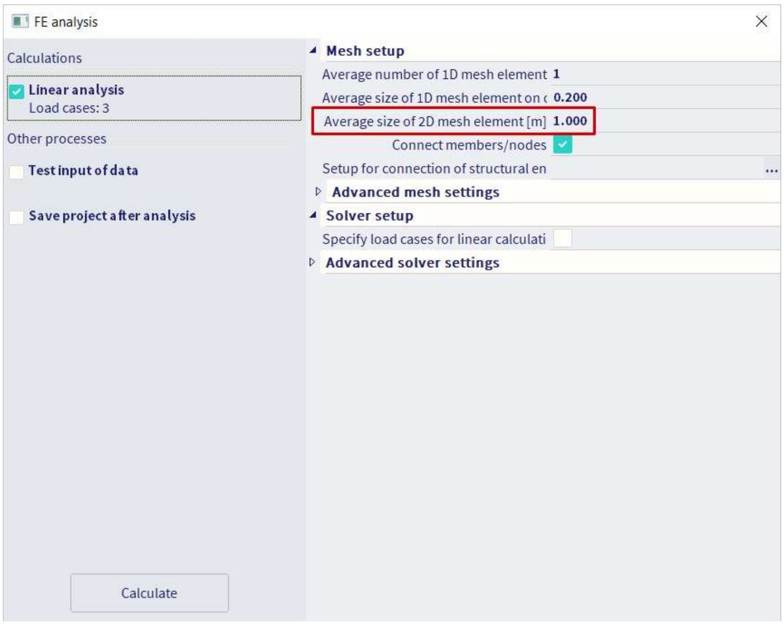
Via the tools menu → Calculation & mesh → Mesh settings

Average size of 2D (mesh) elements is by default = 1 m

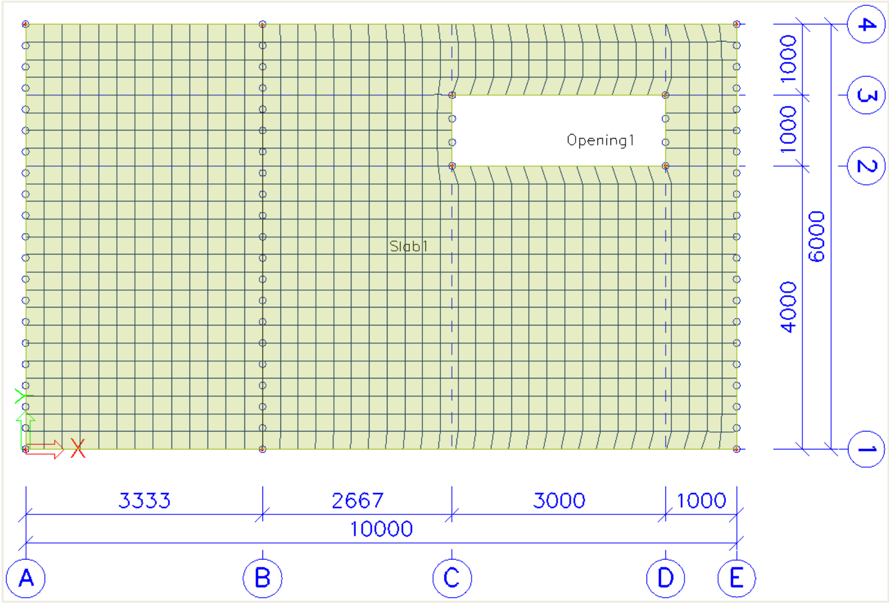


OR

The mesh size can be changed in the FE analysis window before running the calculation.



'Basic rule' for the size of 2D mesh elements: take 1 to 2 times the thickness of the plates in the project. For this example, take a mesh size of 0,25 m.



2.4.2 Results for the linear calculation

SPECIFICATION OF RESULTS

After running the linear calculation, go to the Results menu → 2D members → 2D Internal Forces.

Specify the desired result in the Properties panel:

System:

- LCS mesh element: according to the local axes of the *individual* mesh elements
- LCS - Member 2D: according to the LCS of the 2D member (Pay attention when working with shell elements!)

Location: 4 different ways to ask for the results, see chapter Results

Type forces: Basic, Principal or Design magnitudes, see Annex 1

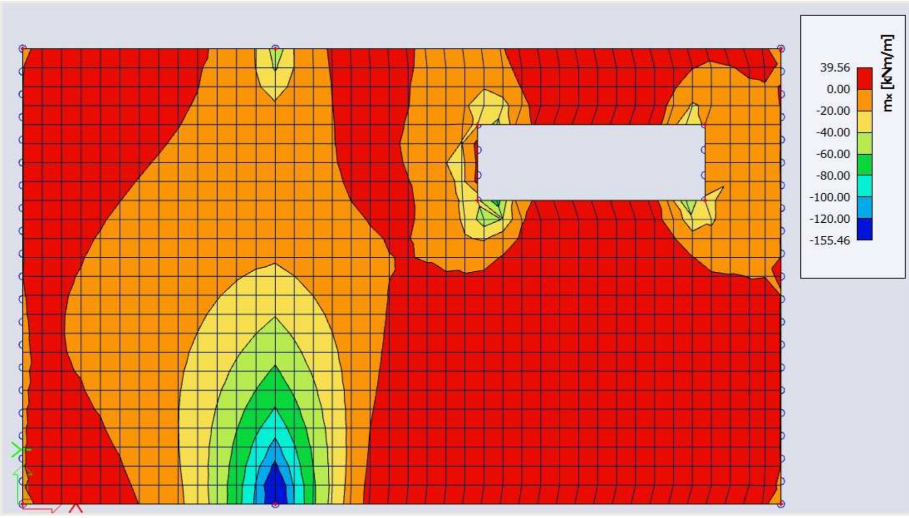
Drawing setup 2D: Click on the button [...]. Here you can modify the display of 2D results (Isobands / Isolines / Numerical results / ...), modify the minimum and maximum settings, ...

After making changes in the Properties panel, you always have to execute the 'Refresh' action.

TYPES OF RESULTS

Basic magnitudes

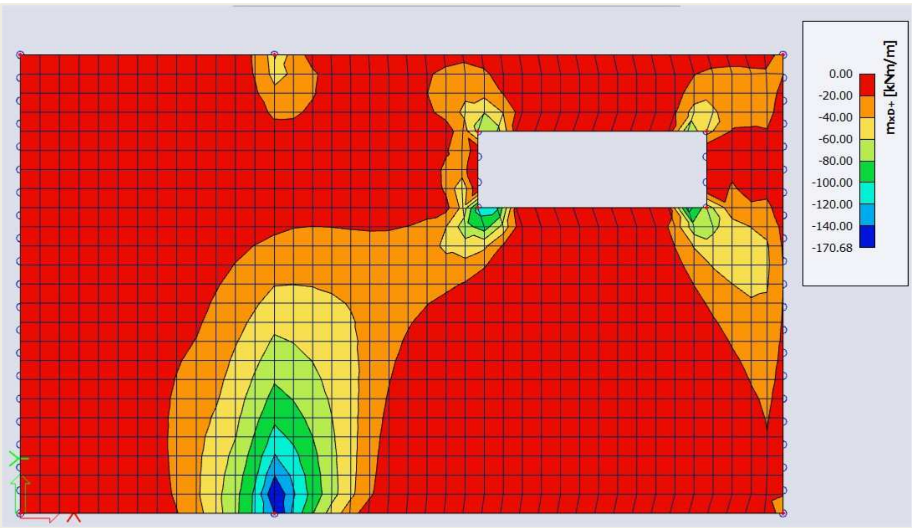
Combination = ULS; Type forces = Basic magnitudes; Envelope = Minimum; Values = **m_x**



These are the characteristic values coming from de FE-analysis in the center of the plate.

Elementary design magnitudes

Combination = ULS; Type forces = Design magnitudes; Envelope = Maximum; Values = **m_xD+**



The convention for the sign of the design moments has been changed since the v17 post-processor. Now a moment is positive when it causes a tensile force on the bottom of the plate and negative when it causes tensile force at the top of the plate.

In the v16 post-processor a design moment is positive when you should reinforce for this moment. This means that for a positive value for m_xD+ there is a tensile force at the top of the plate and that for a positive value for m_xD- there is a tensile force at the bottom of the plate.

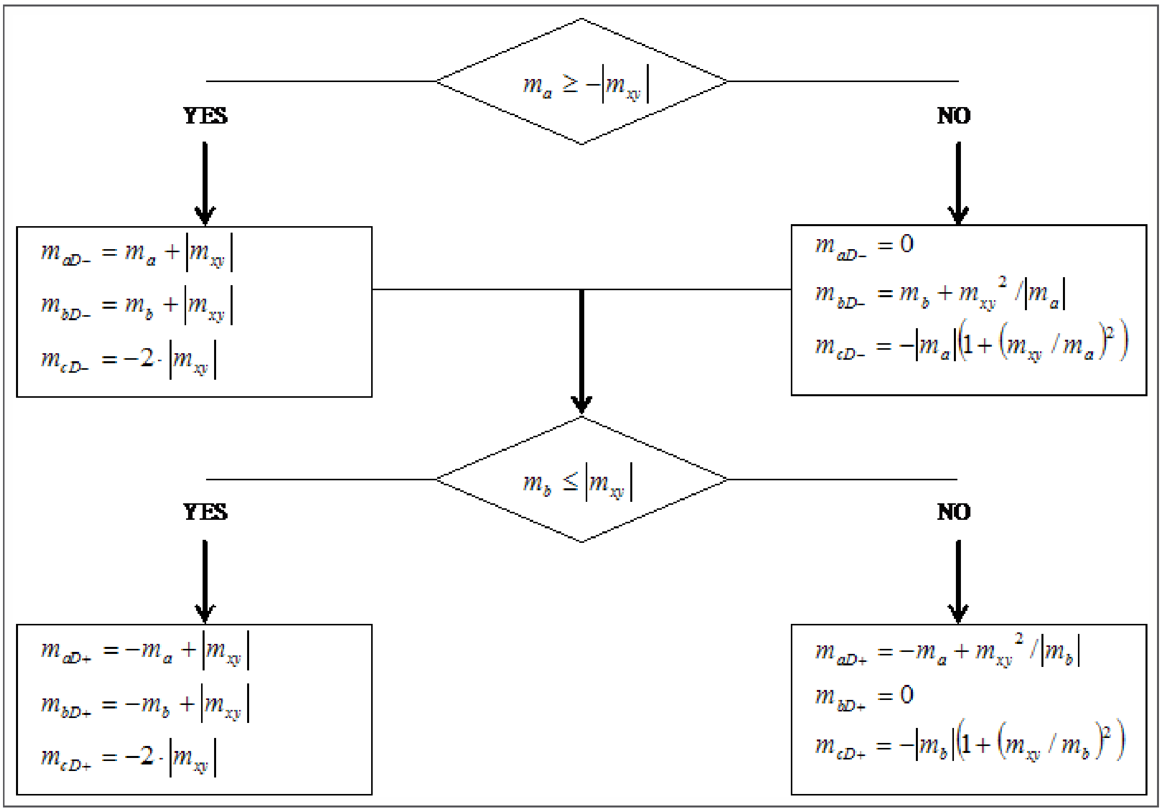
The available values are mxD, myD and mcD, where ‘D’ stands for design. The ‘+’ and ‘-‘ respectively stand for the values at the positive and negative side of the local z axis of the 2D member.

So for instance the value mxD+ is the moment that will be used for the design of the upper reinforcement in the local x-direction of the 2D member.

The calculation of design moments for *plates* and *shells* according to the EC2 algorithm follows the chart from CSN P ENV 1992-1-1, Annex 2, paragraph A2.8.

$m_y \geq m_x \quad : \quad a = x, b = y$

$m_y < m_x \quad : \quad a = y, b = x$



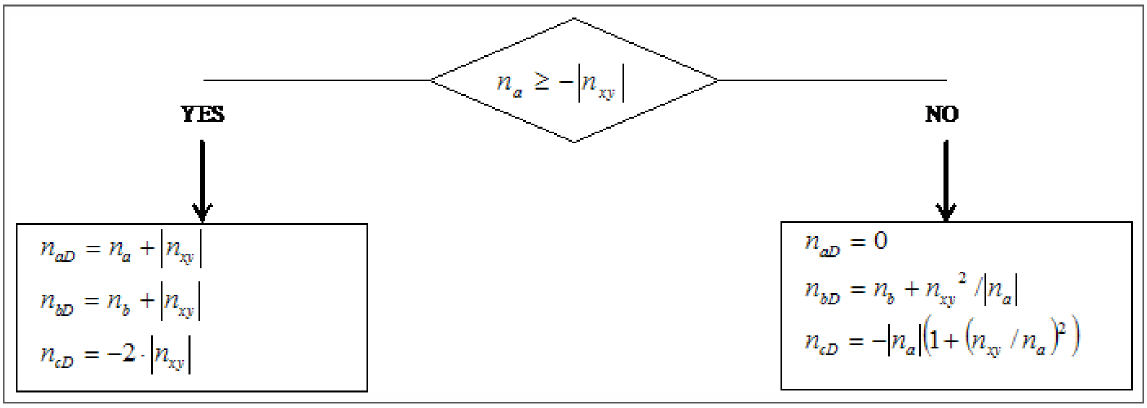
What happens, is that for the 3 characteristic (bending and torsion) moments an equivalent set of 3 design moments is calculated:

mx		mxD
my	≈	myD
mxy		mcD

It is clear that mxD and myD are the moments to be used for the reinforcement design in the respective direction. The quantity mcD is the design moment that has to be taken by the concrete. The Eurocode does not mention any check for this value, but it is however available in SCIA Engineer for the reason of completeness.

The calculation of design forces for *walls* according to the EC2 algorithm follows the chart from CSN P ENV 1992-1-1, Annex 2, paragraph A2.9.

$$n_y \geq n_x \quad : \quad a = x, b = y$$
$$n_y < n_x \quad : \quad a = y, b = x$$



Analogously, if membrane effects are present, for the 3 characteristic membrane forces an equivalent set of 3 design forces is calculated:

nx		nxD
ny	≈	nyD
nxy		ncD

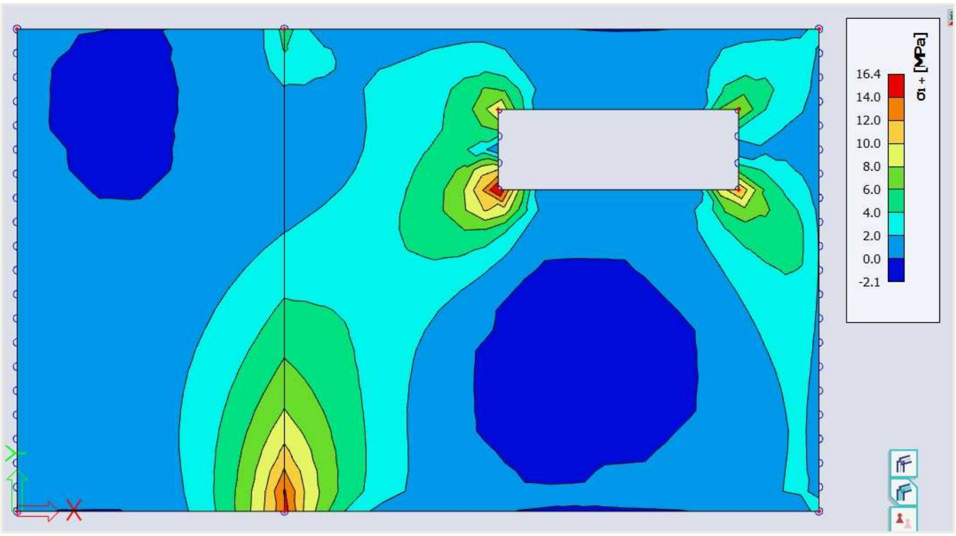
Here, the quantity ncD does have a clear meaning: it is the compression force that has to be taken by the concrete compression struts. Therefore, to make sure that concrete crushing will not occur, the value ncD should be checked to be ≤ fcd.

<u>Attention</u>: These design magnitudes are not the ones used by SCIA Engineer for the reinforcement design in the Concrete menu. A much more refined transformation procedure is implemented there to calculate the design magnitudes from the basic magnitudes.

⇨ Principal magnitudes

Results menu → 2D members → 2D stresses/strain

Combination = ULS; Type forces = Principal stress; Envelope = Maximum; Values = σ**1+**



‘1’ and ‘2’ refer to the principal directions, calculated based on Mohr’s circle.

The first direction is the direction of maximum tension (or minimum compression). The second direction is the direction of maximum compression (or minimum tension).

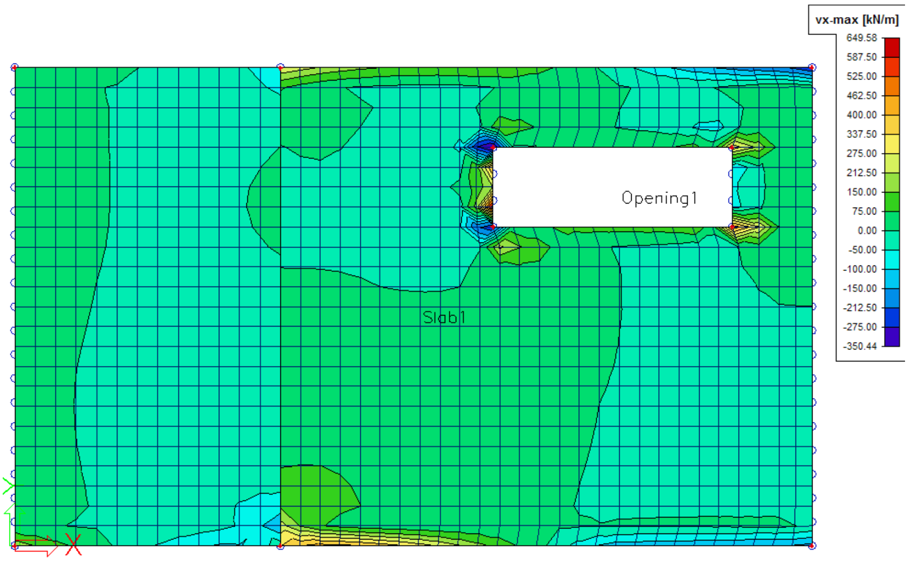
Keep in mind that the most economic reinforcement paths are the ones that follow the trajectories of the principal directions!

COMPARISON MINDLIN ⬄ KIRCHHOFF

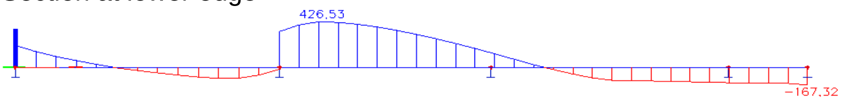
⇨ Shear force vx

Combination = ULS; Type forces = Basic magnitudes; Envelope = Maximum; Values = **v_x**

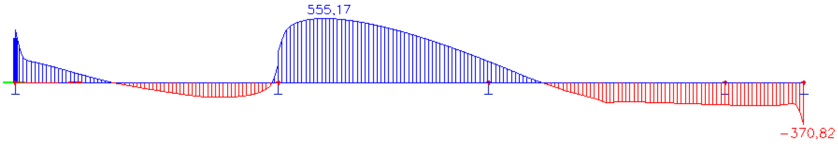
Mindlin



Section at lower edge

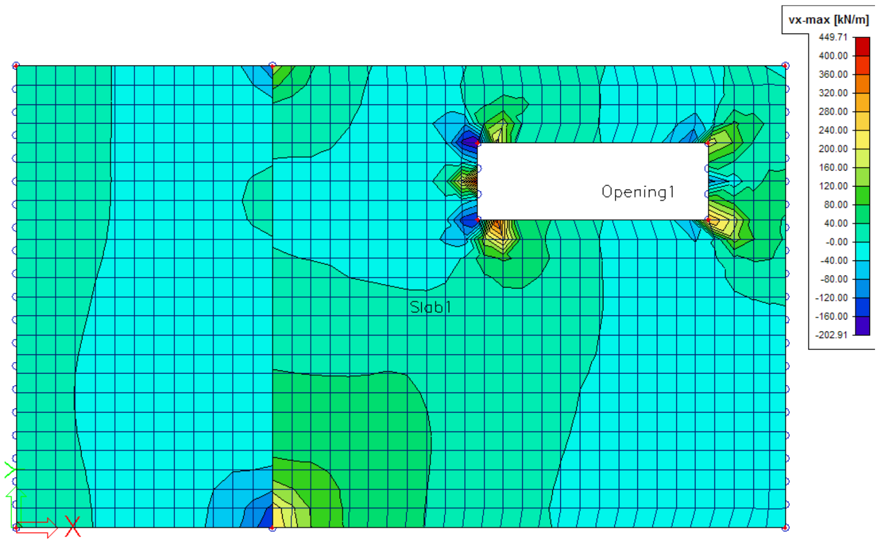


Mesh size = 0,25 m

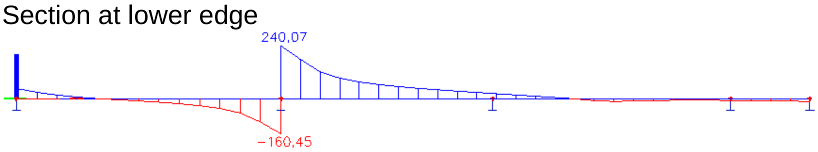


Mesh size = 0,05 m

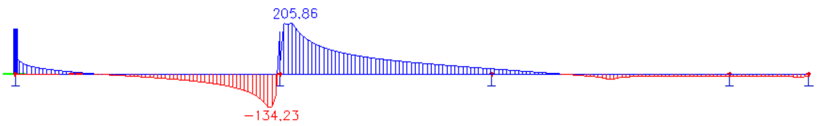
Kirchhoff



Section at lower edge



Mesh size = 0,25 m

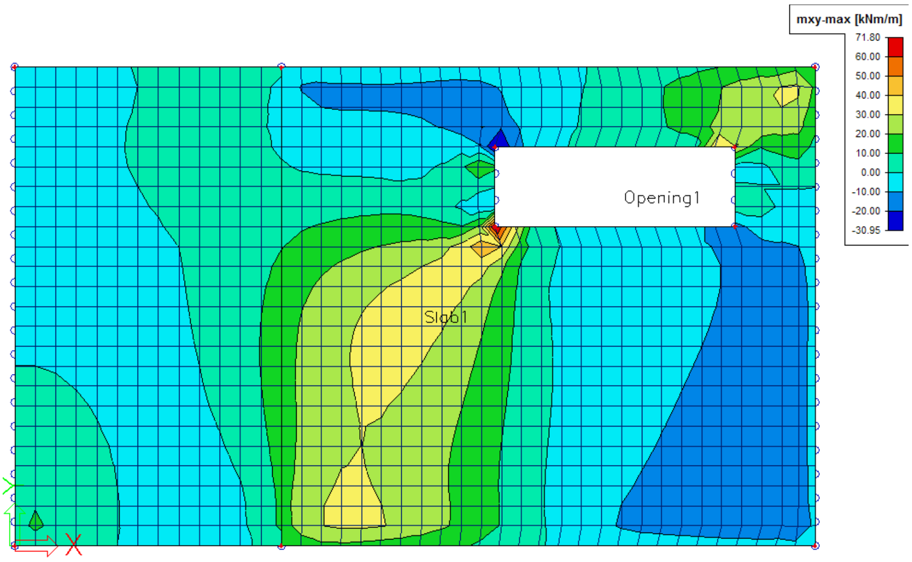


Mesh size = 0,05 m

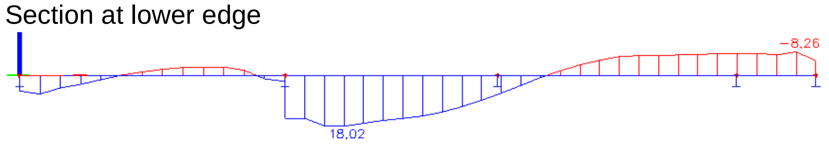
⇨ Torsion moment mxy

Combination = ULS; Type forces = Basic magnitudes; Envelope = Maximum; Values = **m_xy**

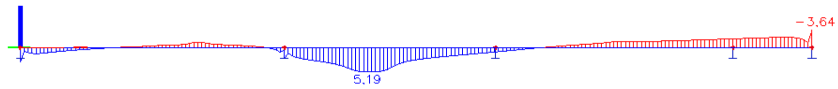
Mindlin



Section at lower edge

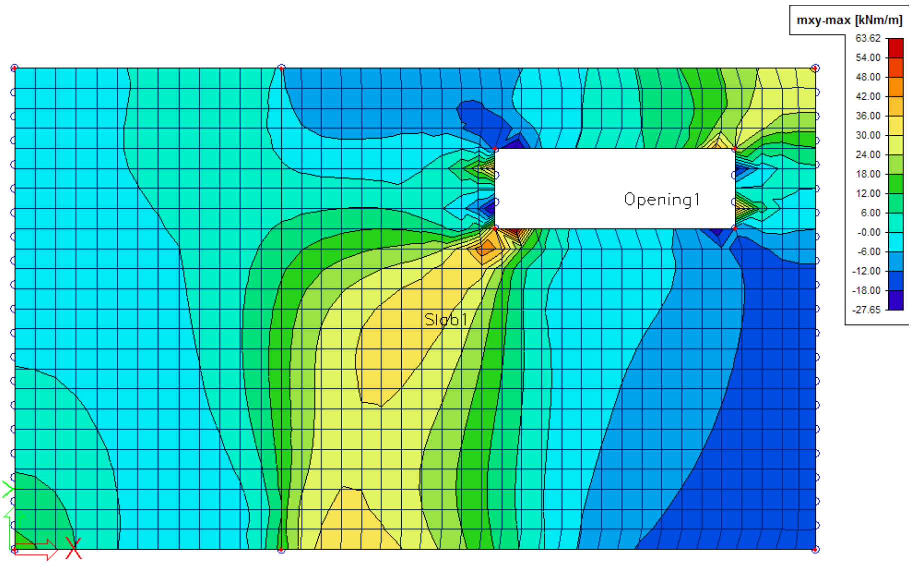


Mesh size = 0,25 m

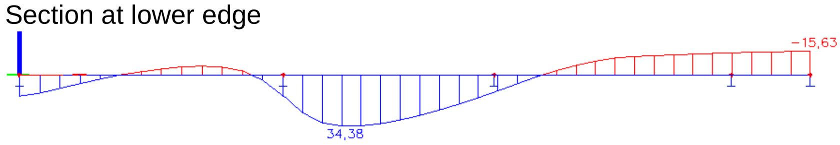


Mesh size = 0,05 m

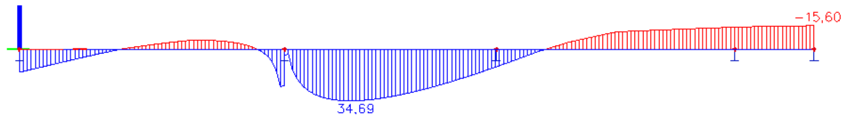
Kirchhoff



Section at lower edge



Mesh size = 0,25 m



Mesh size = 0,05 m

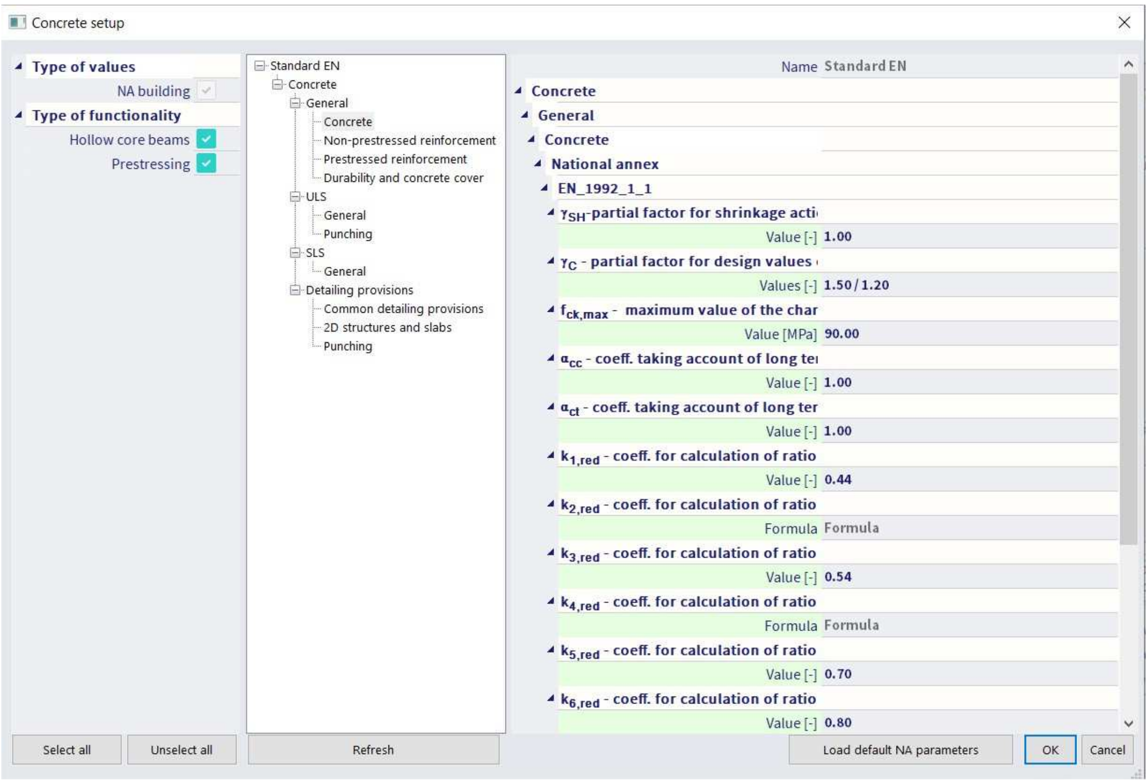
<u>**Conclusion**</u>: Kirchhoff gives the expected shear force values, Mindlin gives the expected torsion moments.

2.4.3 Concrete setups

GENERAL SETUPS

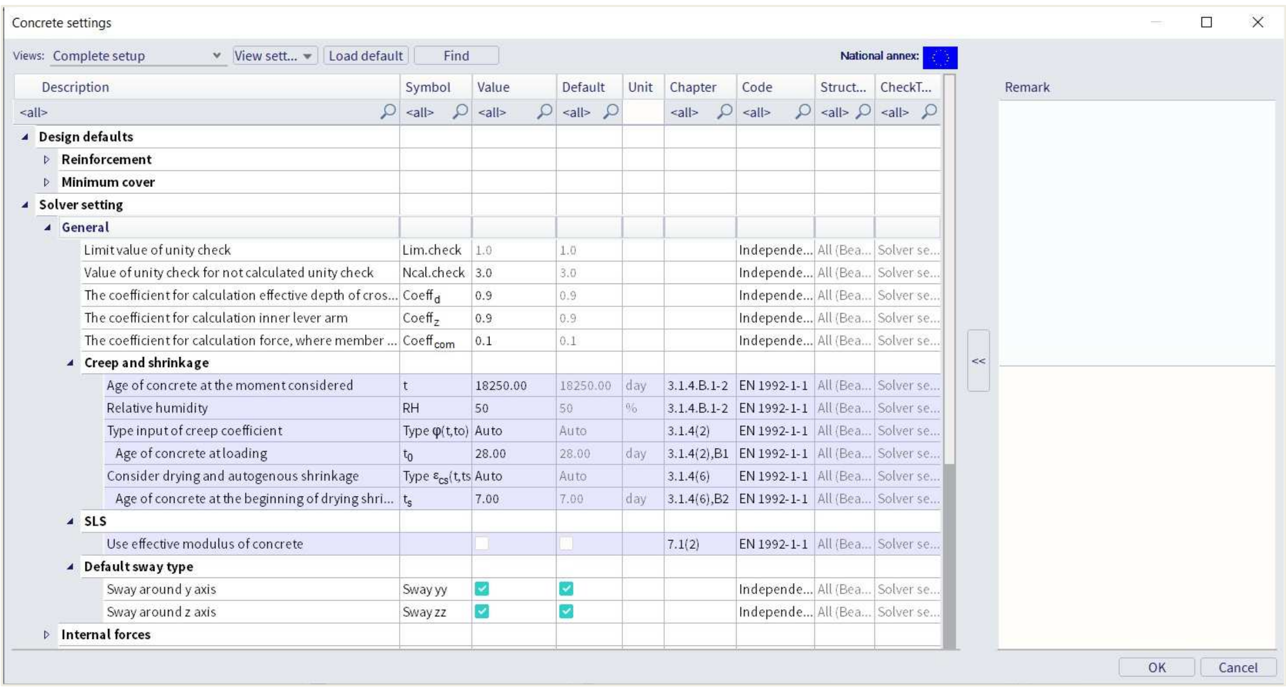
⇨ ***Setup 1: National Determined parameters***

File → Project settings → National annex […] → EN 1992-1-1 […]
OR
Click on the flag at the top right of SCIA Engineer → Manage annexes → EN 1992-1-1 […]



⇨ ***Setup 2: Concrete settings***

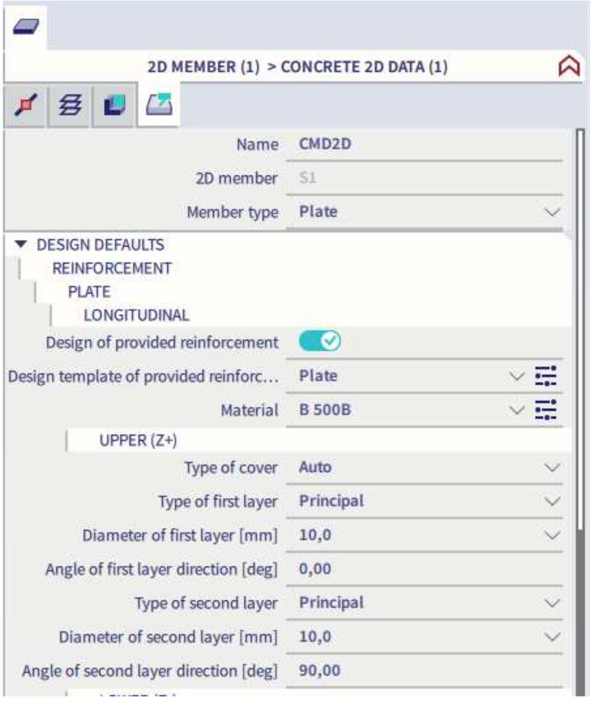
Concrete menu → Concrete settings



All of the adjustments made in one of the two general setups are valid for the **whole project**.

MEMBER DATA

It is possible to **overwrite** the data from the general setups per 2D member, namely by means of Member data. This is automatically defined on any 2D member (same is true for 1D members).



2.4.4 ULS design

REINFORCEMENT DESIGN

⇨ Internal forces

Design menu → Concrete 2D → Internal forces

Basic (centroid) : the values shown here are exactly the same as in the Results menu; they are calculated by the FEM solver.

Design (centroid) : the values shown here are different from those in the Results menu.

- The design magnitudes in the **Results** menu are calculated by the **FEM** solver according to some simple formulas specified in EC-ENV.
- The design magnitudes in the **Concrete** menu are calculated by the **NEDIM** solver, where a much finer transformation procedure is implemented, based on the theory of Baumann.
  <u>These are the values that will be used for the SCIA Engineer reinforcement design.</u>

Theory of Baumann:

1) Calculation of the lever arm.

The lever arm is necessary for the calculation of surface forces. Value z will be calculated in the direction of the angle of the first principal moment. The forces will be recalculated and a cross-section set will be created in this direction. The reinforcement will be designed for these recalculated forces and from the designed reinforcement the inner lever arm will be calculated.

Principal stresses and directions at both surfaces

$\sigma_{I-} = 0.48$ MPa $\sigma_{II-} = 0.11$ MPa -> $\alpha_{z-} = -5.86 = -5.86\ °$

$\sigma_{I+} = -0.11$ MPa $\sigma_{II+} = -0.48$ MPa -> $\alpha_{z+} = -5.86\ °$

-> direction for calculation inner lever arm

$\alpha_z = -5.86$

Recalculated forces to direction of inner lever arm

$n_z = 0.0 \quad m_z = 4970.4$

$$f_{cd} = \frac{\alpha_{cc} \cdot f_{ck}}{\gamma_C} = \frac{1 \cdot 20 \cdot 10^6}{1.5} = 13.33\ \text{MPa}$$

$d = 210$ mm

$$\eta = 1 - 0.5 \cdot \frac{\varepsilon_{c2}}{\varepsilon_{cu2}} = 1 - 0.5 \cdot \frac{0.0018}{0.0035} = 0.75$$

$$\beta = 1 - \frac{\frac{\varepsilon_{cu2}^2}{2} - \frac{\varepsilon_{c2}^2}{6}}{\varepsilon_{cu2}^2 - \frac{\varepsilon_{cu2} \cdot \varepsilon_{c2}}{2}} = 1 - \frac{\frac{0.0035^2}{2} - \frac{0.0018^2}{6}}{0.0035^2 - \frac{0.0035 \cdot 0.0018}{2}} = 0.389$$

$$\xi_{bal} = \frac{\varepsilon_{cu2}}{\varepsilon_{cu2} + \frac{f_{yk}}{\gamma_S \cdot E_s}} = \frac{0.0035}{0.0035 + \frac{500}{1.15 \cdot 200000}} = 0.617$$

$x_{bal} = \xi_{bal} \cdot d = 0.617 \cdot 210 = 0.13$

$n_{c,bal} = -\xi_{bal} \cdot d \cdot b \cdot \eta \cdot f_{cd} = -0.617 \cdot 210 \cdot 1000 \cdot 0.75 \cdot 13.33 = -1295$ kN/m

$n_z = 0$ kN/m $> n_{c,bal} = -1295$ kN/m => predominant tension

$$x = \frac{d_-}{2 \cdot \beta} \cdot \left(1 - \sqrt{1 - 4 \cdot \beta \cdot \frac{abs(m_z) - n_z \cdot (d_- - 0.5 \cdot h)}{b \cdot d_-^2 \cdot \eta \cdot f_{cd}}} \right)$$

$$= \frac{0.21}{2 \cdot 0.389} \cdot \left(1 - \sqrt{1 - 4 \cdot 0.389 \cdot \frac{abs(4970) - 0 \cdot (0.21 - 0.5 \cdot 250)}{1000 \cdot 0.21^2 \cdot 0.75 \cdot 13.33}} \right) = 2\ \text{mm}$$

$z = d - \beta \cdot x = 210 - 0.389 \cdot 2 = 209$ mm

$z_+ = 124$ mm

$z_- = 85$ mm

If value z cannot be calculated it will be calculated according to formula: z = 0,9 * d

2) Calculation of normal forces at the surfaces of 2D element.

The inputted internal forces will be recalculated to both surfaces according the following formulas:

Lower surface

$$n_{x-} = \frac{n_x}{2} + \frac{m_x}{z} = \frac{0}{2} + \frac{4.93}{0.209} = 23.6 \text{ kN/m}$$

$$n_{y-} = \frac{n_y}{2} + \frac{m_y}{z} = \frac{0}{2} + \frac{1.22}{0.209} = 5.8 \text{ kN/m}$$

$$n_{xy-} = \frac{n_{xy}}{2} + \frac{m_{xy}}{z} = \frac{0}{2} + \frac{-0.385}{0.209} = -1.8 \text{ kN/m}$$

Upper surface

$$n_{x+} = \frac{n_x}{2} - \frac{m_x}{z} = \frac{0}{2} - \frac{4.93}{0.209} = -23.6 \text{ kN/m}$$

$$n_{y+} = \frac{n_y}{2} - \frac{m_y}{z} = \frac{0}{2} - \frac{1.22}{0.209} = -5.8 \text{ kN/m}$$

$$n_{xy+} = \frac{n_{xy}}{2} - \frac{m_{xy}}{z} = \frac{0}{2} - \frac{-0.385}{0.209} = 1.8 \text{ kN/m}$$

3) Calculation of principal forces at surfaces of 2D element.

The principal forces at both surfaces and the direction of the first principal force will be calculated according to the following formulas:

Lower surface

Principal forces at lower surface:

$$n_{I-} = \frac{n_{x-} + n_{y-}}{2} + \frac{1}{2} \cdot \sqrt{\left(n_{x-} - n_{y-}\right)^2 + 4 \cdot n_{xy-}^{\;2}}$$

$$= \frac{23.6 + 5.8}{2} + \frac{1}{2} \cdot \sqrt{\left(23.6 - 5.8\right)^2 + 4 \cdot -1.8^2} = 23.8 \text{ kN/m}$$

$$n_{II-} = \frac{n_{x-} + n_{y-}}{2} - \frac{1}{2} \cdot \sqrt{\left(n_{x-} - n_{y-}\right)^2 + 4 \cdot n_{xy-}^{\;2}}$$

$$= \frac{23.6 + 5.8}{2} - \frac{1}{2} \cdot \sqrt{\left(23.6 - 5.8\right)^2 + 4 \cdot -1.8^2} = 5.7 \text{ kN/m}$$

Direction of principal forces:

$$\alpha_{I-} = 0.5 \cdot \text{ArcTg}\left(\frac{2 \cdot n_{xy-}}{n_{x-} - n_{y-}}\right) = 0.5 \cdot \text{ArcTg}\left(\frac{2 \cdot -1.8}{23.6 - 5.8}\right) = -6\,°$$

Upper surface

Principal forces at upper surface:

$$n_{I+} = \frac{n_{x+} + n_{y+}}{2} + \frac{1}{2} \cdot \sqrt{\left(n_{x+} - n_{y+}\right)^2 + 4 \cdot n_{xy+}^{\;2}}$$

$$= \frac{-23.6 + -5.8}{2} + \frac{1}{2} \cdot \sqrt{\left(-23.6 - -5.8\right)^2 + 4 \cdot 1.8^2} = -5.7 \text{ kN/m}$$

$$n_{II+} = \frac{n_{x+} + n_{y+}}{2} - \frac{1}{2} \cdot \sqrt{\left(n_{x+} - n_{y+}\right)^2 + 4 \cdot n_{xy+}^{\;2}}$$

$$= \frac{-23.6 + -5.8}{2} - \frac{1}{2} \cdot \sqrt{\left(-23.6 - -5.8\right)^2 + 4 \cdot 1.8^2} = -23.8 \text{ kN/m}$$

Direction of principal forces:

$$\alpha_{I+} = 0.5 \cdot \text{ArcTg}\left(\frac{2 \cdot n_{xy+}}{n_{x+} - n_{y+}}\right) - 90 = 0.5 \cdot \text{ArcTg}\left(\frac{2 \cdot 1.8}{-23.6 - -5.8}\right) - 90 = -96\,°$$

4) Recalculation of principal forces at both surfaces to inputted directions.

The recalculation of the principal forces to the inputted direction will be done separately for both surfaces by using Baumann's transformation formula.

Lower surface

Angles for Baumann's transformation formula

$$\alpha_{1-} = \alpha_{inp,1-} - \alpha_{I-} = 0 - -6 = 6\ °$$
$$\alpha_{2-} = \alpha_{inp,2-} - \alpha_{I-} = 90 - -6 = 96\ °$$
$$\alpha_{3-} = \alpha_{con-} - \alpha_{I-} = 45 - -6 = 51\ °$$

Recalculated dimensional forces at lower surface (acc. to Baumann)

$$n_{Eds1-} = \frac{n_{I-} \cdot \sin(\alpha_{2-}) \cdot \sin(\alpha_{3-}) + n_{II-} \cdot \cos(\alpha_{2-}) \cdot \cos(\alpha_{3-})}{\sin(\alpha_{2-} - \alpha_{1-}) \cdot \sin(\alpha_{3-} - \alpha_{1-})}$$
$$= \frac{23.8 \cdot \sin(96) \cdot \sin(51) + 5.7 \cdot \cos(96) \cdot \cos(51)}{\sin(96 - 6) \cdot \sin(51 - 6)} = 25.4 \text{ kN/m}$$

$$n_{Eds2-} = \frac{n_{I-} \cdot \sin(\alpha_{3-}) \cdot \sin(\alpha_{1-}) + n_{II-} \cdot \cos(\alpha_{3-}) \cdot \cos(\alpha_{1-})}{\sin(\alpha_{3-} - \alpha_{2-}) \cdot \sin(\alpha_{1-} - \alpha_{2-})}$$
$$= \frac{23.8 \cdot \sin(51) \cdot \sin(6) + 5.7 \cdot \cos(51) \cdot \cos(6)}{\sin(51 - 96) \cdot \sin(6 - 96)} = 7.7 \text{ kN/m}$$

$$n_{Eds3-} = \frac{n_{I-} \cdot \sin(\alpha_{1-}) \cdot \sin(\alpha_{2-}) + n_{II-} \cdot \cos(\alpha_{1-}) \cdot \cos(\alpha_{2-})}{\sin(\alpha_{1-} - \alpha_{3-}) \cdot \sin(\alpha_{2-} - \alpha_{3-})}$$
$$= \frac{23.8 \cdot \sin(6) \cdot \sin(96) + 5.7 \cdot \cos(6) \cdot \cos(96)}{\sin(6 - 51) \cdot \sin(96 - 51)} = -3.7 \text{ kN/m}$$

Upper surface

Angles for Baumann's transformation formula

$$\alpha_{1+} = \alpha_{inp,1+} - \alpha_{I+} = 0 - -96 = 96\ °$$
$$\alpha_{2+} = \alpha_{inp,2+} - \alpha_{I+} = 90 - -96 = 186\ °$$
$$\alpha_{3+} = \alpha_{con+} - \alpha_{I+} = 135 - -96 = 231\ °$$

Recalculated dimensional forces at upper surface (acc. to Baumann)

$$n_{Eds1+} = \frac{n_{I+} \cdot \sin(\alpha_{2+}) \cdot \sin(\alpha_{3+}) + n_{II+} \cdot \cos(\alpha_{2+}) \cdot \cos(\alpha_{3+})}{\sin(\alpha_{2+} - \alpha_{1+}) \cdot \sin(\alpha_{3+} - \alpha_{1+})}$$
$$= \frac{-5.7 \cdot \sin(186) \cdot \sin(231) + -23.8 \cdot \cos(186) \cdot \cos(231)}{\sin(186 - 96) \cdot \sin(231 - 96)} = -21.7 \text{ kN/m}$$

$$n_{Eds2+} = \frac{n_{I+} \cdot \sin(\alpha_{3+}) \cdot \sin(\alpha_{1+}) + n_{II+} \cdot \cos(\alpha_{3+}) \cdot \cos(\alpha_{1+})}{\sin(\alpha_{3+} - \alpha_{2+}) \cdot \sin(\alpha_{1+} - \alpha_{2+})}$$
$$= \frac{-5.7 \cdot \sin(231) \cdot \sin(96) + -23.8 \cdot \cos(231) \cdot \cos(96)}{\sin(231 - 186) \cdot \sin(96 - 186)} = -4.0 \text{ kN/m}$$

$$n_{Eds3+} = \frac{n_{I+} \cdot \sin(\alpha_{1+}) \cdot \sin(\alpha_{2+}) + n_{II+} \cdot \cos(\alpha_{1+}) \cdot \cos(\alpha_{2+})}{\sin(\alpha_{1+} - \alpha_{3+}) \cdot \sin(\alpha_{2+} - \alpha_{3+})}$$
$$= \frac{-5.7 \cdot \sin(96) \cdot \sin(186) + -23.8 \cdot \cos(96) \cdot \cos(186)}{\sin(96 - 231) \cdot \sin(186 - 231)} = -3.7 \text{ kN/m}$$

5) Calculation of virtual forces at both surfaces to inputted directions.

The virtual forces are necessary to convert the pressure/tensile forces at the surface back to the center of the plate. The virtual force represents the equivalent force at the other side of the plate.

Virtual forces at both surfaces

Lower surface

Angles for Baumann's transformation formula

$$\alpha_{1+} = \alpha_{inp,1+} - \alpha_{I-} = 0 - -6 = 6\,°$$

$$\alpha_{2+} = \alpha_{inp,2+} - \alpha_{I-} = 90 - -6 = 96\,°$$

$$\alpha_{3+} = \alpha_{con-} - \alpha_{I-} = 45 - -6 = 51\,°$$

Recalculated virtual forces at lower surface (acc. to Baumann)

$$n_{Edsvirt1-} = \frac{n_{I-} \cdot \sin(\alpha_{2+}) \cdot \sin(\alpha_{3+}) + n_{II-} \cdot \cos(\alpha_{2+}) \cdot \cos(\alpha_{3+})}{\sin(\alpha_{2+} - \alpha_{1+}) \cdot \sin(\alpha_{3+} - \alpha_{1+})}$$

$$= \frac{23.8 \cdot \sin(96) \cdot \sin(51) + 5.7 \cdot \cos(96) \cdot \cos(51)}{\sin(96 - 6) \cdot \sin(51 - 6)} = 25.4 \text{ kN/m}$$

$$n_{Edsvirt2-} = \frac{n_{I-} \cdot \sin(\alpha_{3+}) \cdot \sin(\alpha_{1+}) + n_{II-} \cdot \cos(\alpha_{3+}) \cdot \cos(\alpha_{1+})}{\sin(\alpha_{3+} - \alpha_{2+}) \cdot \sin(\alpha_{1+} - \alpha_{2+})}$$

$$= \frac{23.8 \cdot \sin(51) \cdot \sin(6) + 5.7 \cdot \cos(51) \cdot \cos(6)}{\sin(51 - 96) \cdot \sin(6 - 96)} = 7.7 \text{ kN/m}$$

$$n_{Edsvirt3-} = \frac{n_{I-} \cdot \sin(\alpha_{1+}) \cdot \sin(\alpha_{2+}) + n_{II-} \cdot \cos(\alpha_{1+}) \cdot \cos(\alpha_{2+})}{\sin(\alpha_{1+} - \alpha_{3+}) \cdot \sin(\alpha_{2+} - \alpha_{3+})}$$

$$= \frac{23.8 \cdot \sin(6) \cdot \sin(96) + 5.7 \cdot \cos(6) \cdot \cos(96)}{\sin(6 - 51) \cdot \sin(96 - 51)} = -3.7 \text{ kN/m}$$

Upper surface

Angles for Baumann's transformation formula

$$\alpha_{1-} = \alpha_{inp,1-} - \alpha_{I+} = 0 - -96 = 96\,°$$

$$\alpha_{2-} = \alpha_{inp,2-} - \alpha_{I+} = 90 - -96 = 186\,°$$

$$\alpha_{3-} = \alpha_{con+} - \alpha_{I+} = 135 - -96 = 231\,°$$

Recalculated virtual forces at upper surface (acc. to Baumann)

$$n_{Edsvirt1+} = \frac{n_{I+} \cdot \sin(\alpha_{2-}) \cdot \sin(\alpha_{3-}) + n_{II+} \cdot \cos(\alpha_{2-}) \cdot \cos(\alpha_{3-})}{\sin(\alpha_{2-} - \alpha_{1-}) \cdot \sin(\alpha_{3-} - \alpha_{1-})}$$

$$= \frac{-5.7 \cdot \sin(186) \cdot \sin(231) + -23.8 \cdot \cos(186) \cdot \cos(231)}{\sin(186 - 96) \cdot \sin(231 - 96)} = -21.7 \text{ kN/m}$$

$$n_{Edsvirt2+} = \frac{n_{I+} \cdot \sin(\alpha_{3-}) \cdot \sin(\alpha_{1-}) + n_{II+} \cdot \cos(\alpha_{3-}) \cdot \cos(\alpha_{1-})}{\sin(\alpha_{3-} - \alpha_{2-}) \cdot \sin(\alpha_{1-} - \alpha_{2-})}$$

$$= \frac{-5.7 \cdot \sin(231) \cdot \sin(96) + -23.8 \cdot \cos(231) \cdot \cos(96)}{\sin(231 - 186) \cdot \sin(96 - 186)} = -4.0 \text{ kN/m}$$

$$n_{Edsvirt3+} = \frac{n_{I+} \cdot \sin(\alpha_{1-}) \cdot \sin(\alpha_{2-}) + n_{II+} \cdot \cos(\alpha_{1-}) \cdot \cos(\alpha_{2-})}{\sin(\alpha_{1-} - \alpha_{3-}) \cdot \sin(\alpha_{2-} - \alpha_{3-})}$$

$$= \frac{-5.7 \cdot \sin(96) \cdot \sin(186) + -23.8 \cdot \cos(96) \cdot \cos(186)}{\sin(96 - 231) \cdot \sin(186 - 231)} = -3.7 \text{ kN/m}$$

6) Recalculation of forces at surfaces to center of gravity of cross-section.

Using the transformed dimensional forces and virtual forces the internal forces at the center of the plate can be calculated.

Lower surface

Dimensional forces of lower surface transformed to centroid

$n_{Ed1-} = n_{Eds1-} + n_{Edsvirt1+} = 25.4 + -21.7 = 3.7$ kN/m

$m_{Ed1-} = n_{Eds1-} \cdot z_- - n_{Edsvirt1+} \cdot z_+ = 25.4 \cdot 85 - -21.7 \cdot 124 = 4.9$ kNm/m

$n_{Ed2-} = n_{Eds2-} + n_{Edsvirt2+} = 7.7 + -4.0 = 3.7$ kN/m

$m_{Ed2-} = n_{Eds2-} \cdot z_- - n_{Edsvirt2+} \cdot z_+ = 7.7 \cdot 85 - -4.0 \cdot 124 = 1.2$ kNm/m

$n_{Ed3-} = n_{Eds3-} + n_{Edsvirt3+} = -3.7 + -3.7 = -7.4$ kN/m

$m_{Ed3-} = n_{Eds3-} \cdot z_- - n_{Edsvirt3+} \cdot z_+ = -3.7 \cdot 85 - -3.7 \cdot 124 = 0.1$ kNm/m

Upper surface

Dimensional forces of upper surface transformed to centroid

$n_{Ed1+} = n_{Eds1+} + n_{Edsvirt1-} = -21.7 + 25.4 = 3.7$ kN/m

$m_{Ed1+} = - n_{Eds1+} \cdot z_+ + n_{Edsvirt1-} \cdot z_- = - -21.7 \cdot 124 + 25.4 \cdot 85 = 4.9$ kNm/m

$n_{Ed2+} = n_{Eds2+} + n_{Edsvirt2-} = -4.0 + 7.7 = 3.7$ kN/m

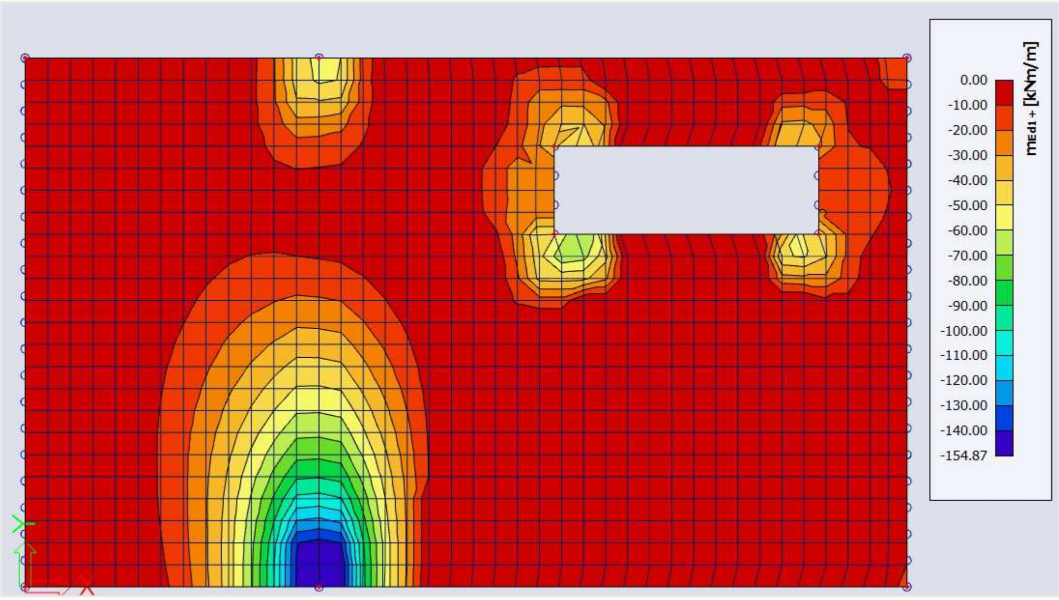
$m_{Ed2+} = - n_{Eds2+} \cdot z_+ + n_{Edsvirt2-} \cdot z_- = - -4.0 \cdot 124 + 7.7 \cdot 85 = 1.2$ kNm/m

$n_{Ed3+} = n_{Eds3+} + n_{Edsvirt3-} = -3.7 + -3.7 = -7.4$ kN/m

$m_{Ed3+} = - n_{Eds3+} \cdot z_+ + n_{Edsvirt3-} \cdot z_- = - -3.7 \cdot 124 + -3.7 \cdot 85 = 0.1$ kNm/m

The available values are: mEd,1+, mEd,2+, mEd,c+, mEd,1-, mEd,2-, mEd,c-, nEd,1+, nEd,2+, nEd,c+, nEd,1-, nEd,2-, nEd,c- and vEd. “+” and “-“ stand for the design values at respectively the positive and the negative side of the local z-axis of the 2D member. “1” and “2” stand for the reinforcement directions, which are by default respectively the local x- and y- direction of the 2D member. (mEd,c+ and mEd,c- are the design moments that would have to be taken by the concrete, but they have no real significance for the reinforcement design.)

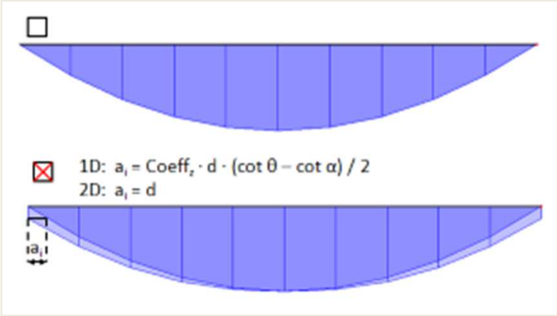
Combination = ULS; Type values = Design internal forces; Value = **mEd,1+**



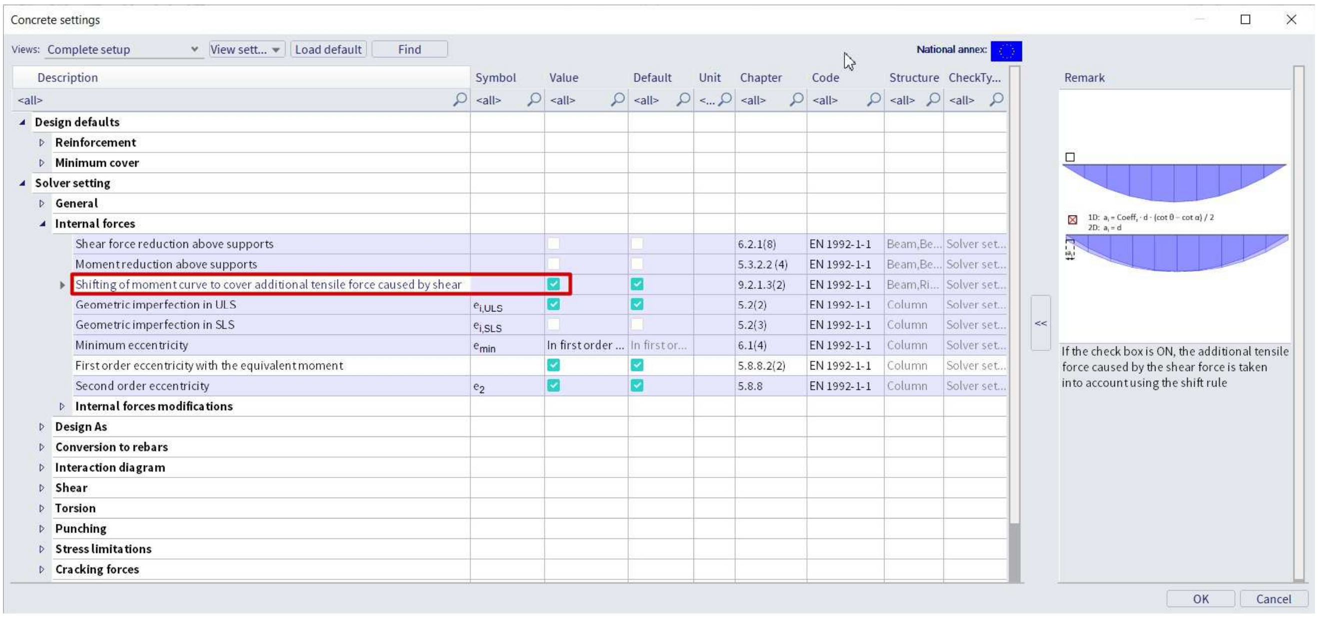
Compare the result for this value mEd,1+ (Concrete menu) with the result for the equivalent value mxD+ (Result menu) shown on page 120.

Despite the different transformation procedures, the general image of the results will be similar for *orthogonal* reinforcement directions (acc. to the local x and y axes). The largest difference is caused by the shift rule that is only taken into account in the design magnitudes calculated by the NEDIM solver (values mEd,1 and mEd,2).

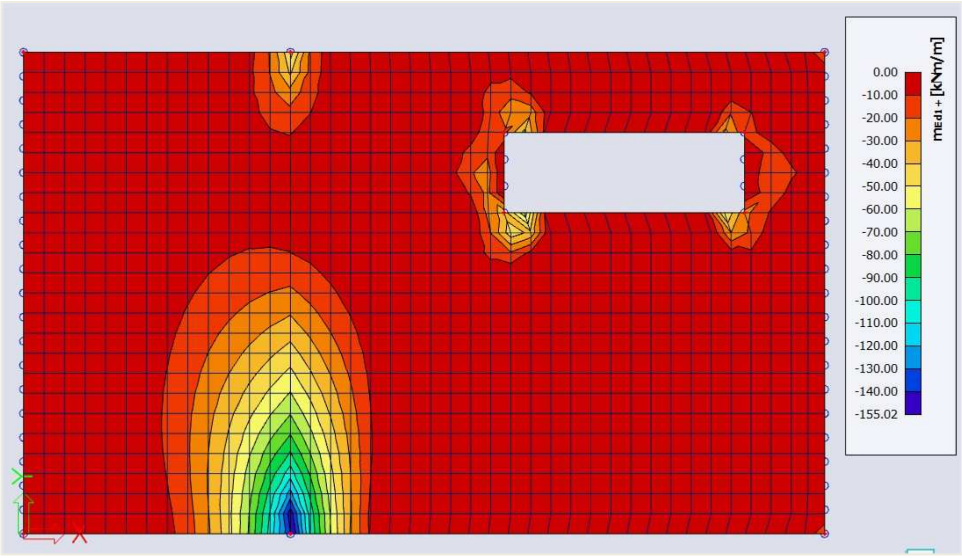
The shift rule takes into account the additional tensile force caused by the shear force by shifting the moment line by a distance a_i. a_i is determined as in the image below.



The shift rule is taken into account in the default concrete settings. You can deactivate this option in the concrete settings.



If we uncheck this option the general image of mEd,1+ is closer to the one obtained for mxD+ (page 120).



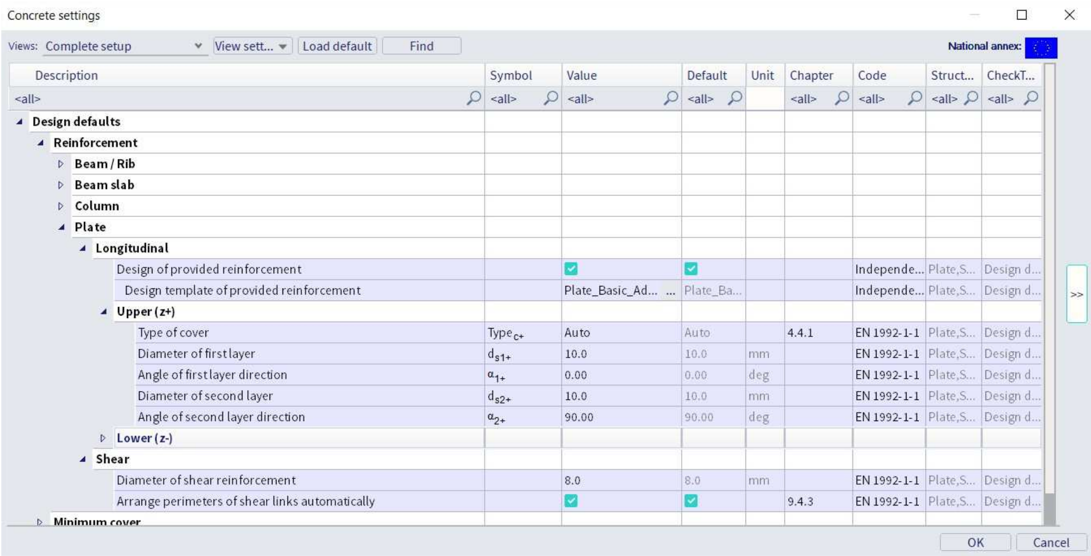
⇨ Provided reinforcement

Before calculating the theoretical reinforcement it is possible to add a template of reinforcement to your plate(s). This template can be used to:

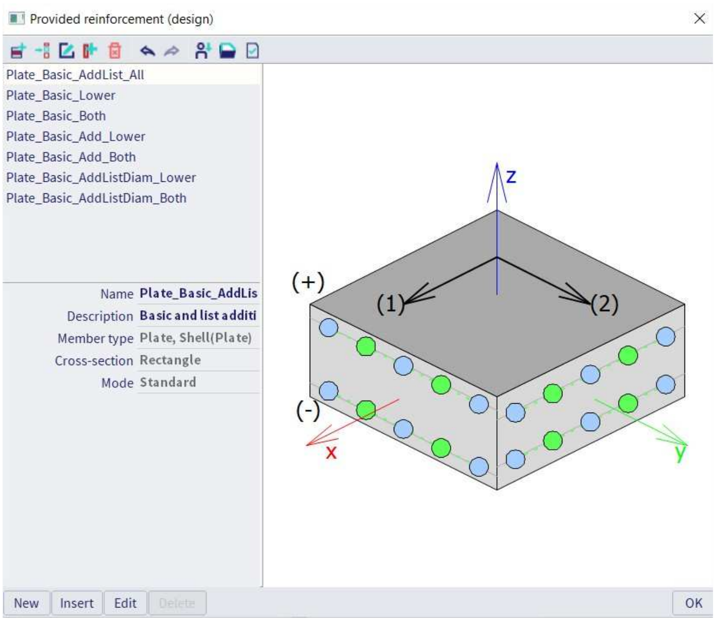
- Compare the template with the calculated theoretical reinforcement. By doing this it is easy to see where this basic template is not sufficient.
- Perform the punching design, Crack width check and the code dependent deflections.

The reinforcement added by the template is called **Provided reinforcement**.

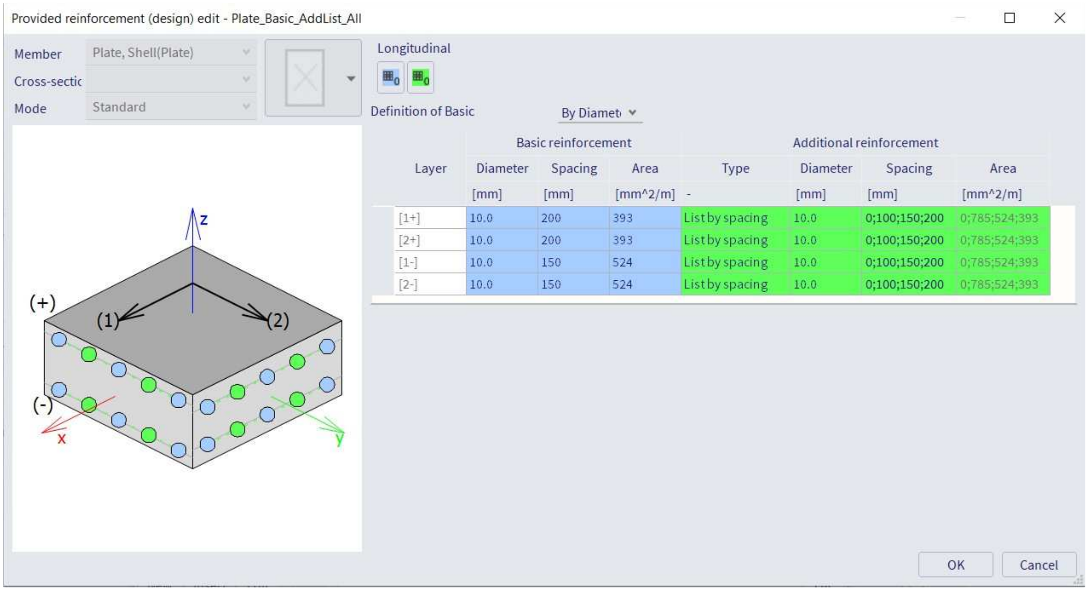
To add **Provided reinforcement** go to Concrete menu → Concrete settings → Design defaults



Click on the 3 dots next to the ‘Design template of provided reinforcement’. This opens a window with all the default templates.



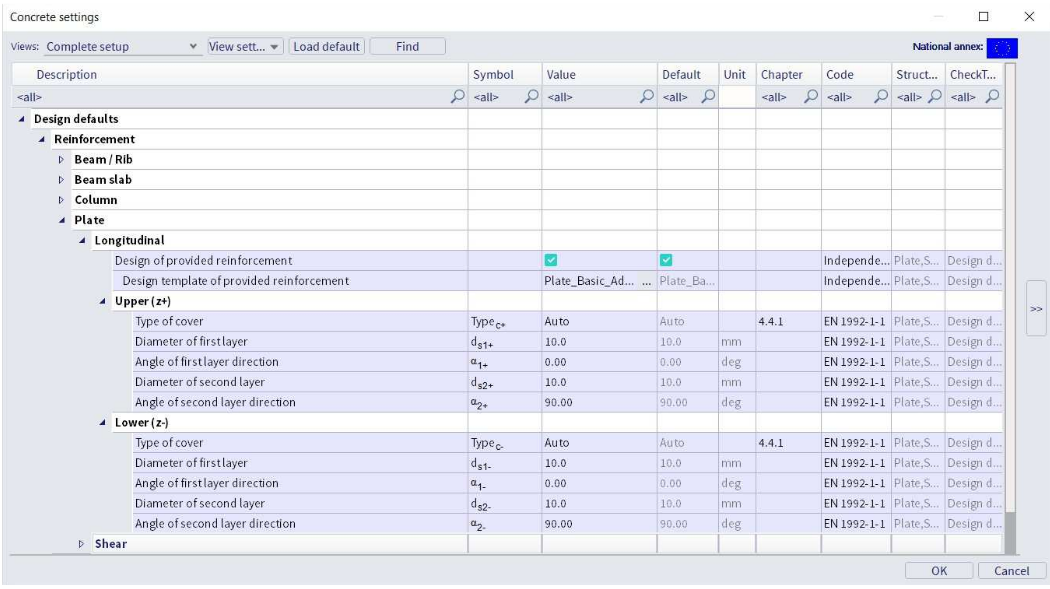
You can select one of these templates, make a new one or edit one of the existing templates. Select the first template and click 'Edit'.



In this window the reinforcement can be defined. There are 2 types of reinforcement in templates:

- **Basic reinforcement:** This type of reinforcement is added over the entire plate.
- **Additional reinforcement:** This type of reinforcement is only added in zones where, according to the calculated theoretical reinforcement , extra reinforcement is needed. You can define a single diameter and spacing as extra reinforcement. Or a list of reinforcement with either various diameters or various spacings.

<u>Note:</u> In the design defaults you can change the reinforcement directions. These directions are respected by as well the provided as the theoretical required reinforcement.



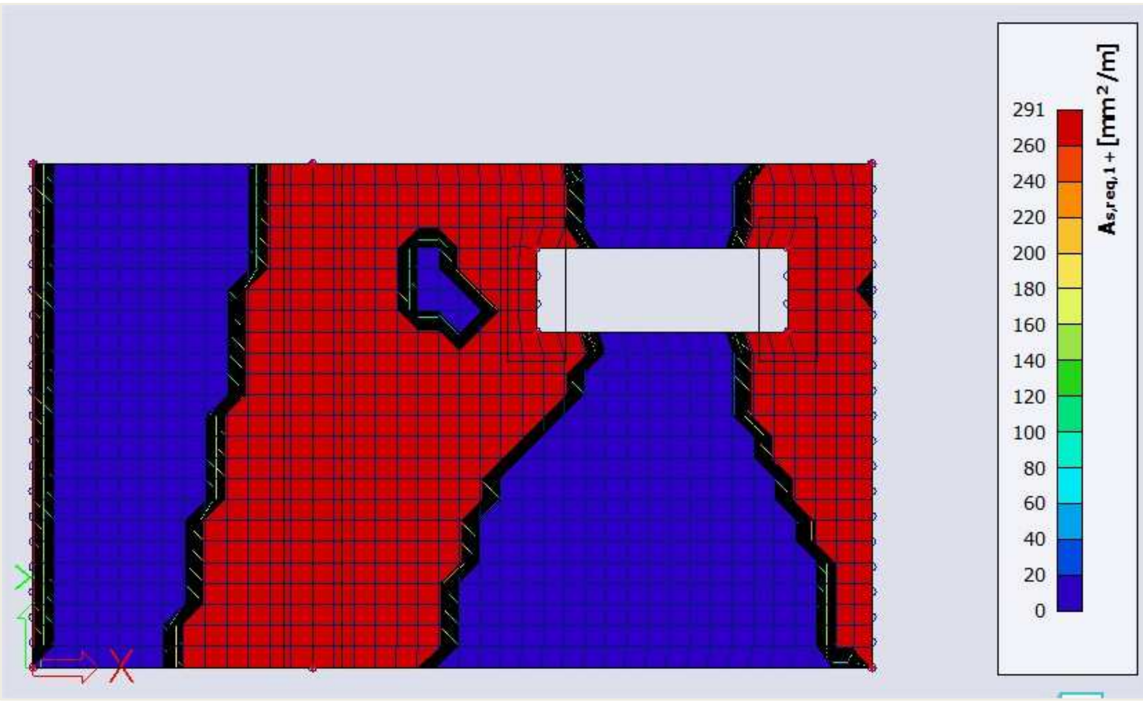
⇨ Theoretical reinforcement

Concrete Menu→ ULS & SLS 2D Reinforcement design

In the menu Reinforcement design (ULS) you have 5 types of values:

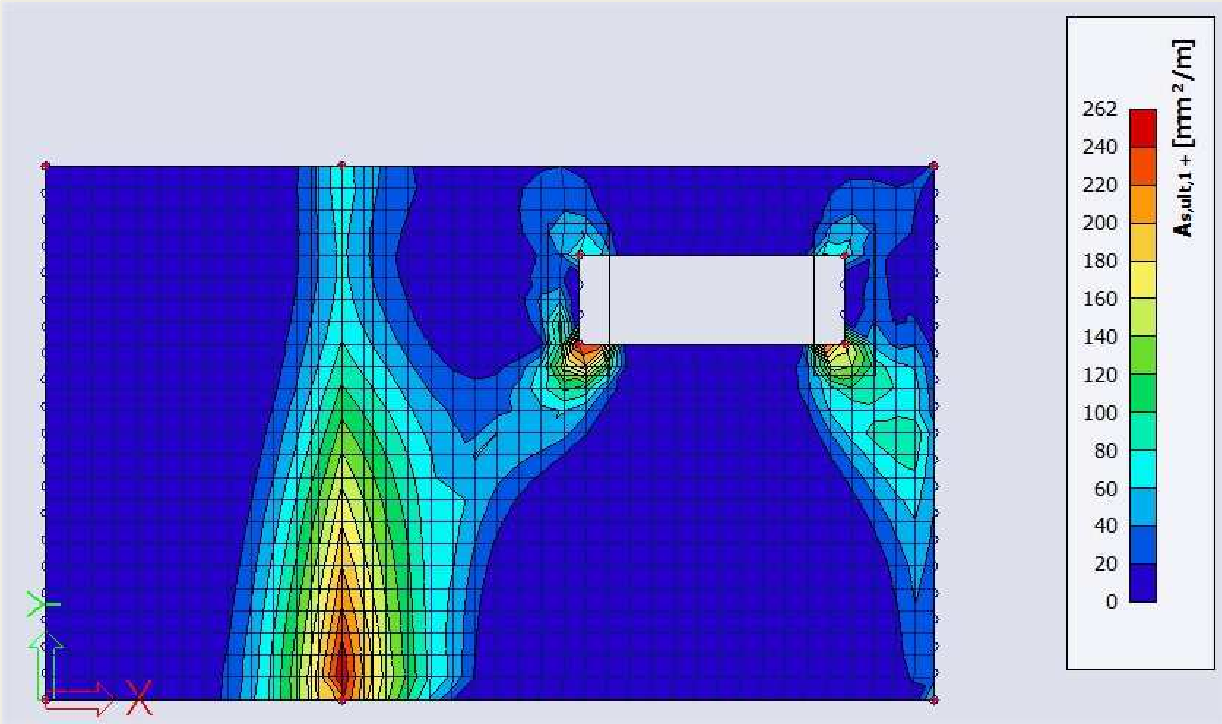
- **Required:** These values represent the theoretical reinforcement calculated by SCIA Engineer. This takes into account the detailing provisions.

Plate, Shell(Plate)		−
Longitudinal		−
Check min. ratio of principal reinforcement		☑
Type of the minimum tension principal reinforcement f...		Auto
Type of the minimum tension principal reinforcement f...		Auto
Check max. ratio of principal reinforcement		☑
Check min. transverse ratio of secondary reinforcement		☐
Check min. bar distance		☑
Minimal bar distance	slp.min	20
Check max.spacing of principal longitudinal reinforcement		☑
Check max.spacing of secondary longitudinal reinforcem...		☑
Shear		−
Check min. ratio of shear reinforcement		☑
Check min. thickness of member with shear reinforcement		☑
Min. thickness of member with shear reinforcement	h_{min}	200
Check max. spacing of shear links		☑
Max. spacing of shear links	$Coeff_{smax.p.s}$	0.8



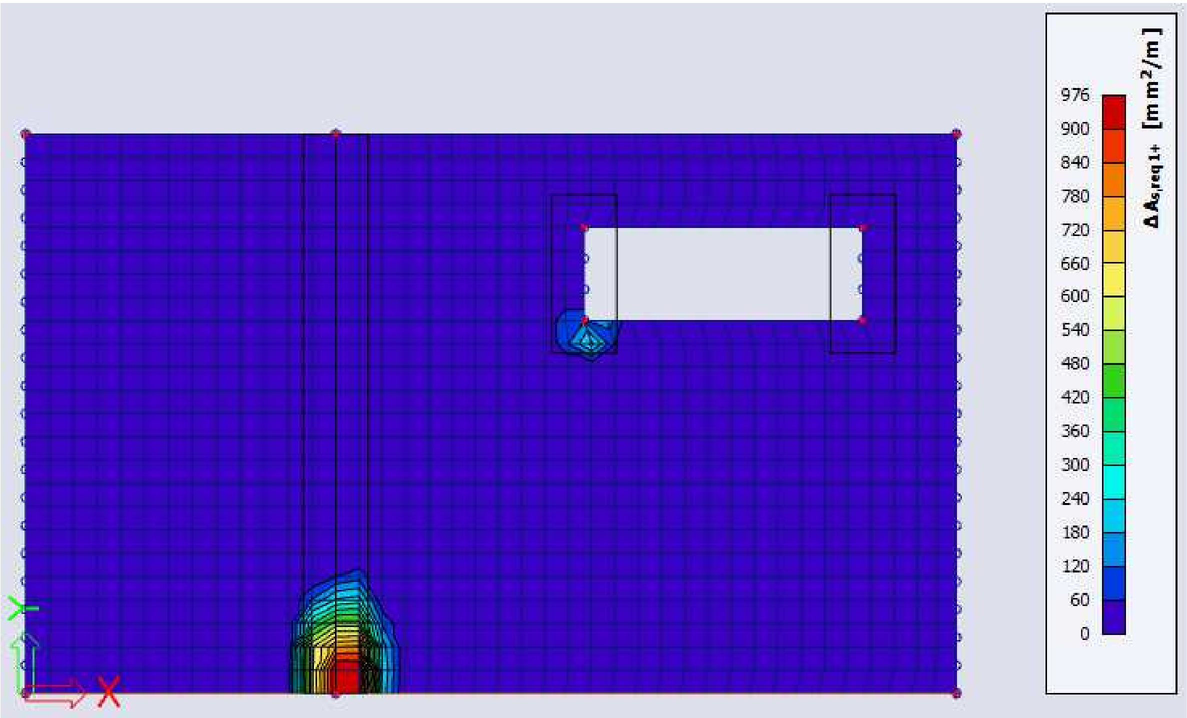
As,req1+: Theoretical required reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction. Taking into account the detailing provisions.

- **Required – Statically:** These values represent the theoretical reinforcement calculated by SCIA Engineer **without** the detailing provisions taken into account.



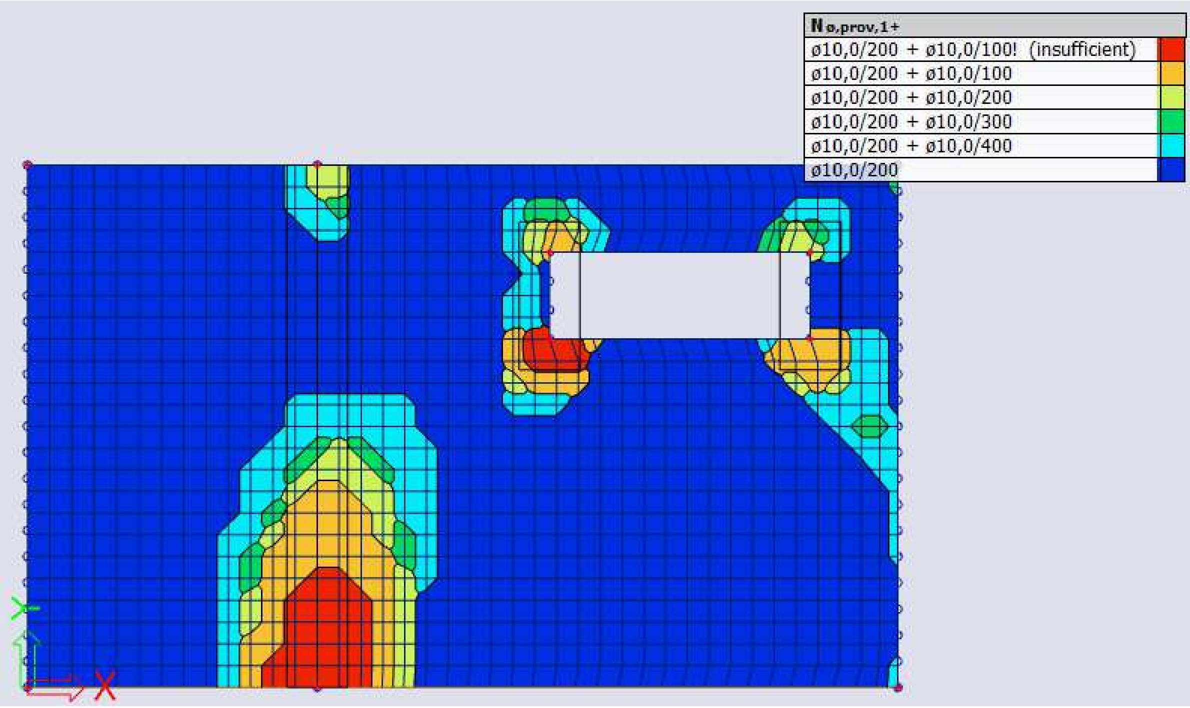
As,ult1+: Theoretical required reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction. **Without** taking into account the detailing provisions.

- **Required – Not covered:** These values show if there is extra reinforcement needed on top of the provided reinforcement. Areas where this value is 0 are areas where no extra reinforcement is needed (compared to the provided reinforcement). Areas where these values are not 0 are areas where the provided reinforcement is not sufficient.



ΔAs,req1+: Theoretical additional required reinforcement on top of the provided reinforcement on the top side of the plate (positive z direction) in the first reinforcement direction.

- **Provided:** These values show you the provided reinforcement defined in the templates.

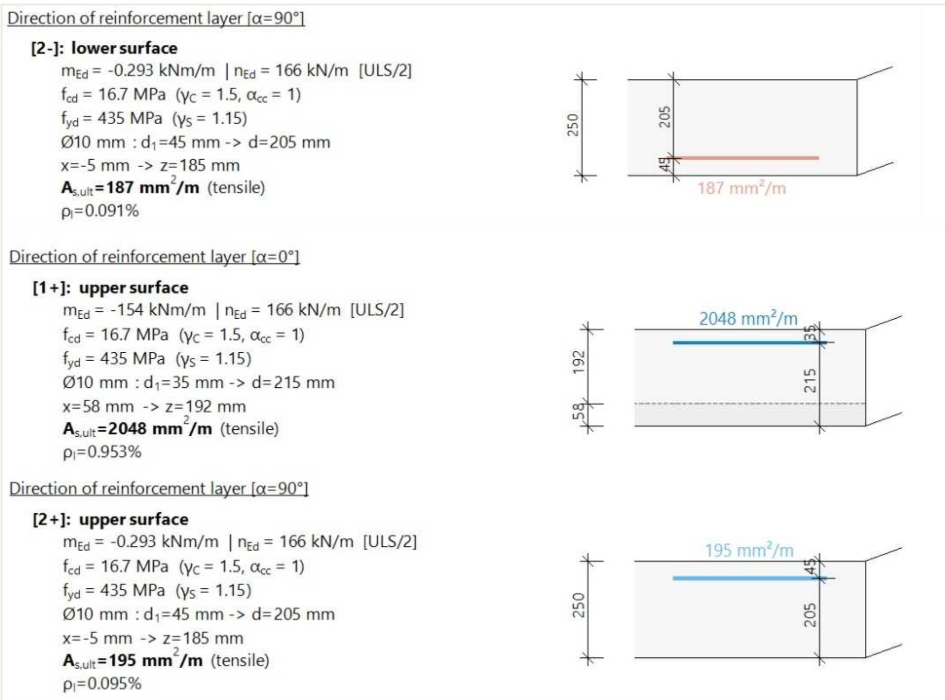


As,prov,1+ or Nφ,prov,1+: Provided reinforcement on the plate in mm²/m or as the amount of reinforcement respectively. If elements are red the additional reinforcement in the template is not sufficient.

- **Provided – Utilization:** Unity checks where provided reinforcement is compared to the required reinforcement. This will give you an idea of the efficiency of the reinforcement.

⇨ Calculation of longitudinal reinforcement

The theoretical longitudinal reinforcement is calculated out of the design internal forces.



⇨ Calculation of shear reinforcement

Before calculating the shear reinforcement two checks are done:

- $V_{Ed} \leq V_{Rd,max}$: The design internal forces on the plate should be lower or equal to the maximum shear resistance of the plate.

$$V_{Rd,max} = \frac{\alpha_{cw} \cdot b_w \cdot z \cdot v_1 \cdot f_{cd}}{\left(\text{cotg}(\theta) + \text{tg}(\theta)\right)}$$

- $V_{Ed} < V_{Rdc}$: If V_{Ed} is smaller than V_{Rdc} no shear reinforcement is required. If this is not the case punching shear reinforcement will be automatically calculated by SCIA Engineer.

$$v_{Rdc} = \max\left(10^6 \cdot \left(C_{Rdc} \cdot k \cdot \left(100 \cdot \rho_l \cdot f_{ck}\right)^{\frac{1}{3}} + k_1 \cdot \sigma_{cp}\right) \cdot d;0\right)$$
$$= \max\left(10^6 \cdot \left(0.12 \cdot 1.98 \cdot \left(100 \cdot 4.58 \cdot 10^{-3} \cdot 25\right)^{\frac{1}{3}} + 0.15 \cdot 0\right) \cdot 0.21;0\right) = 112 \text{ kN/m}$$
$$v_{Rdcmin} = \max\left(10^6 \cdot \left(v_{min} + k_1 \cdot \sigma_{cp}\right) \cdot d;0\right) = \max\left(10^6 \cdot \left(0.486 + 0.15 \cdot 0\right) \cdot 0.21;0\right) = 102 \text{ kN/m}$$
$$v_{Rdc} = \max\left(v_{Rdc}; v_{Rdcmin}\right) = \max\left(112 \text{ kN/m}; 102 \text{ kN/m}\right) = 112 \text{ kN/m}$$

Check shear capacity (without shear reinforcement)

Check $v_{Rd,max}$

$v_{Ed} = 82.3$ kN/m $\leq v_{Rd,max} = 878$ kN/m **(OK)**

Check v_{Rdc}

$v_{Ed} = 82.3$ kN/m $<= v_{Rdc} = 112$ kN/m **(OK, no shear reinforcement is required)**

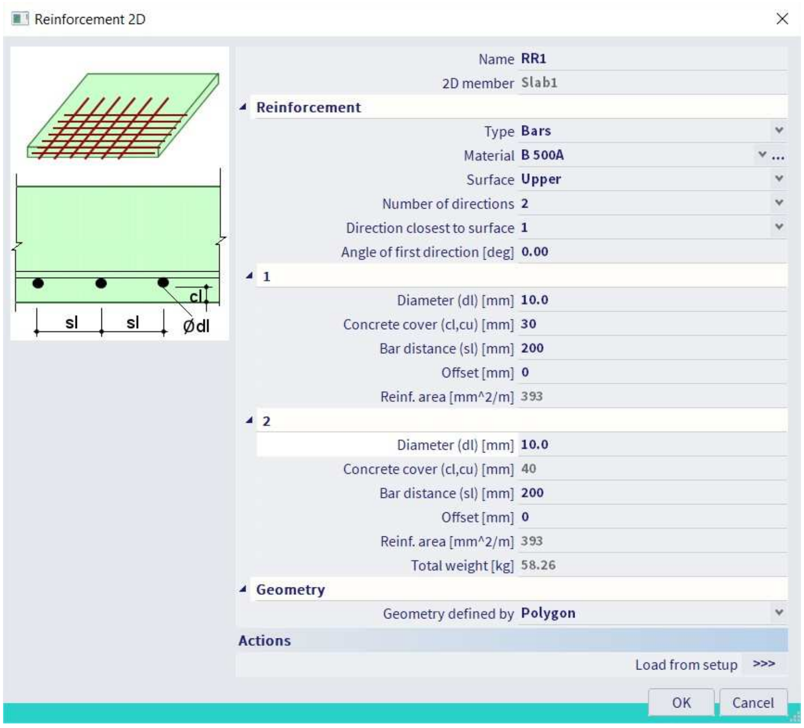
When $V_{Ed} > V_{Rd,max}$ the following error appears in the output of the reinforcement design.

Warning	Punching shear resistance at the column perimeter (vRd,max) is not sufficient acc. to §6.4.3(2).	Increase the column size or change plate properties (use higher grade of concrete material or increase the thickness).

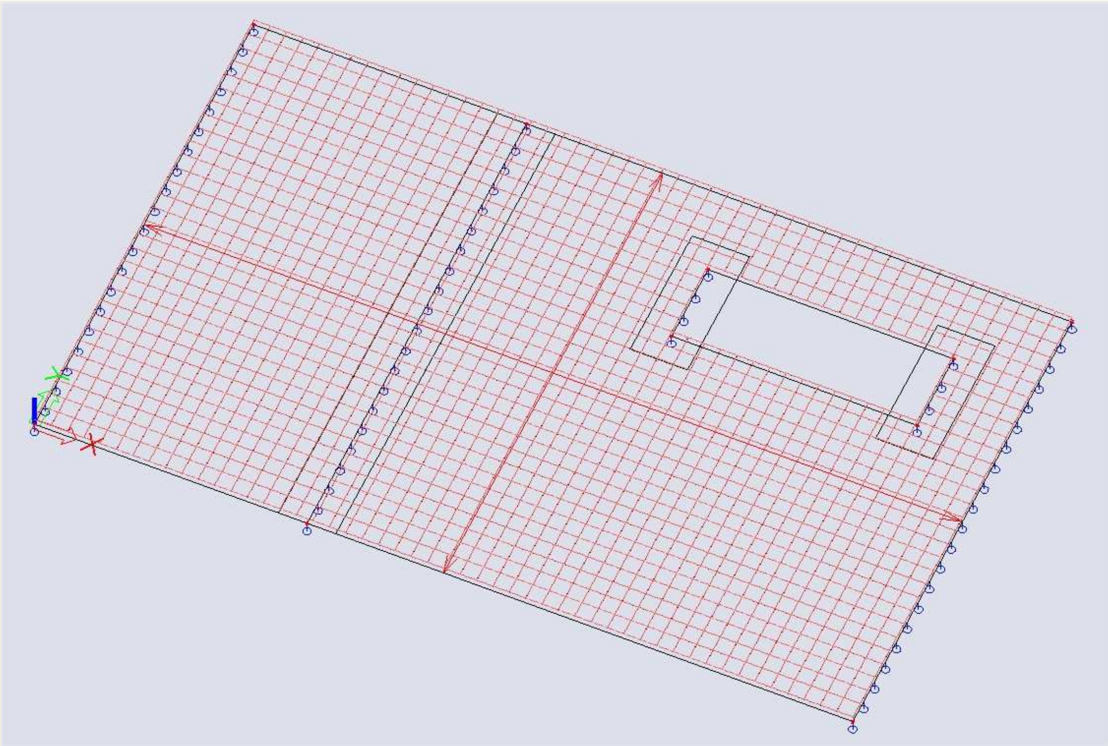
This error message is found at locations with high peak values for the shear stress. Most of the time these peak values are singularities, and do not occur in reality. You have roughly 2 options: you can just ignore the peaks or average them, for example by means of Averaging strips.

Practical reinforcement design

Next to theoretical required and provided reinforcement you have also practical or **User** reinforcement. This type of reinforcement can be added to the plate via the Concrete menu→ 2D Reinforcement.



This reinforcement is to be added separately at the upper and lower side, and in the different reinforcement directions.



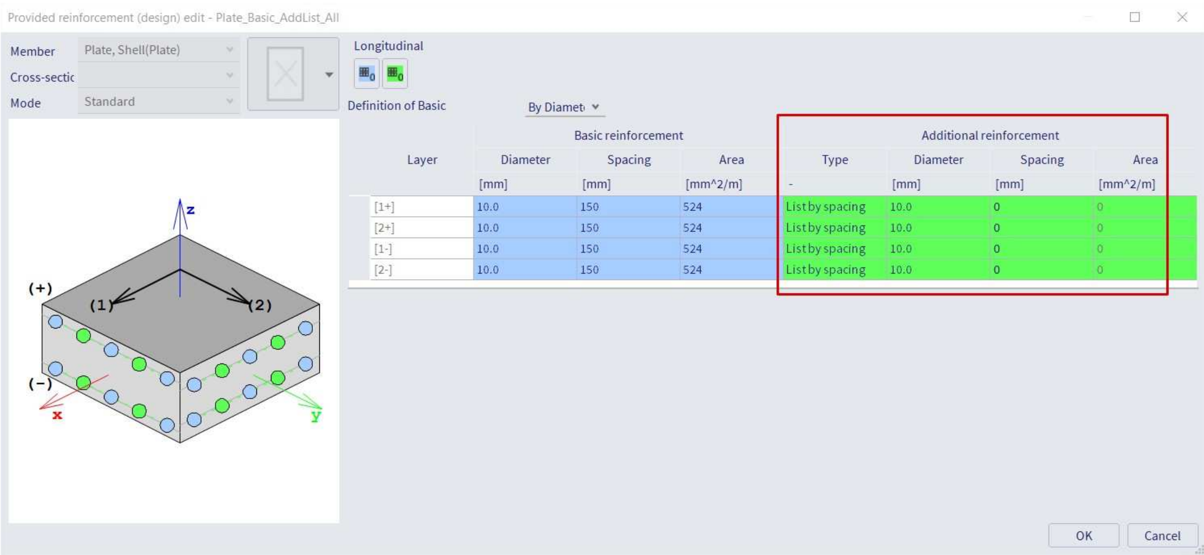
Note: You can add multiple layers of practical reinforcement on the same area. The reinforcement added to this area is the sum of all these layers.

Combination Provided reinforcement and user reinforcement

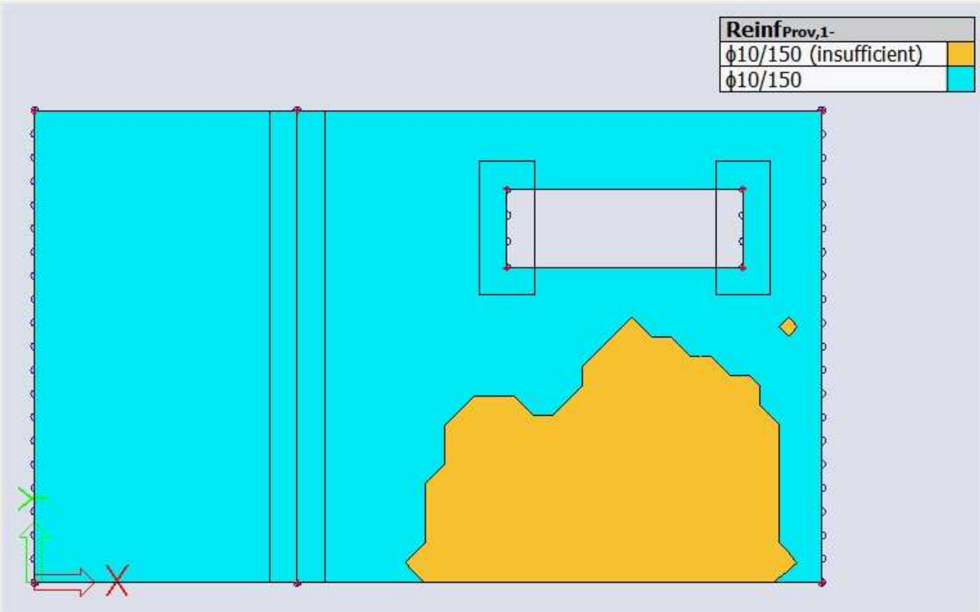
After running the reinforcement design, it might be possible the provided reinforcement is insufficient in certain areas. This means you should introduce some additional reinforcement. In this case you can apply two different workflows:

(a) Define all the reinforcement as practical reinforcement;
(b) Combine the provided reinforcement and the practical reinforcement which will only be defined in the areas where it is necessary to define additional reinforcement.

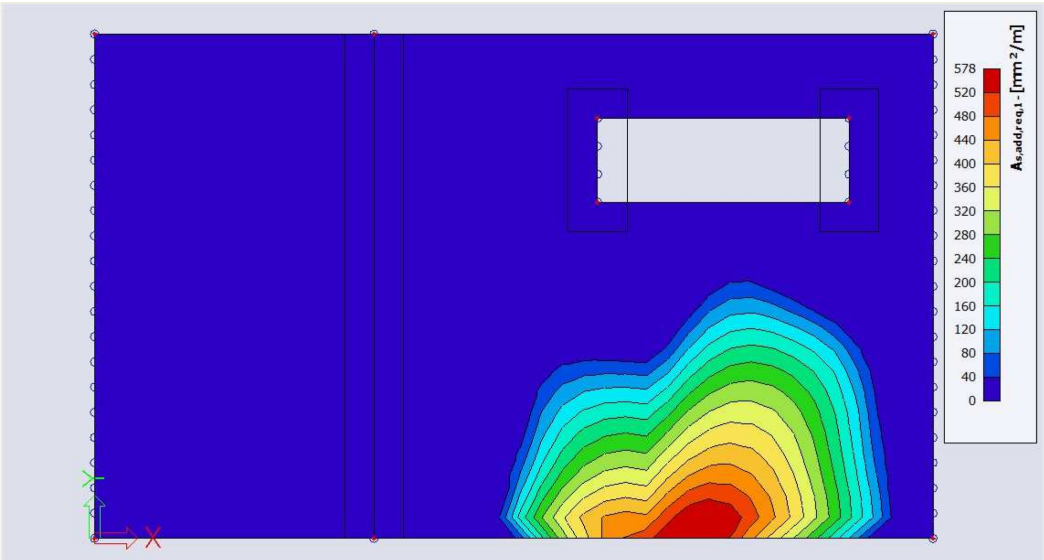
This principle will be explained by using the following example for the ULS reinforcement design in **direction 1 or the local x-direction**. Within the design defaults, you can define a template for the provided reinforcement which can be used within the actual design. In this case the basic reinforcement will be set to **Ø10 à 150** and the addition reinforcement will be set to zero.



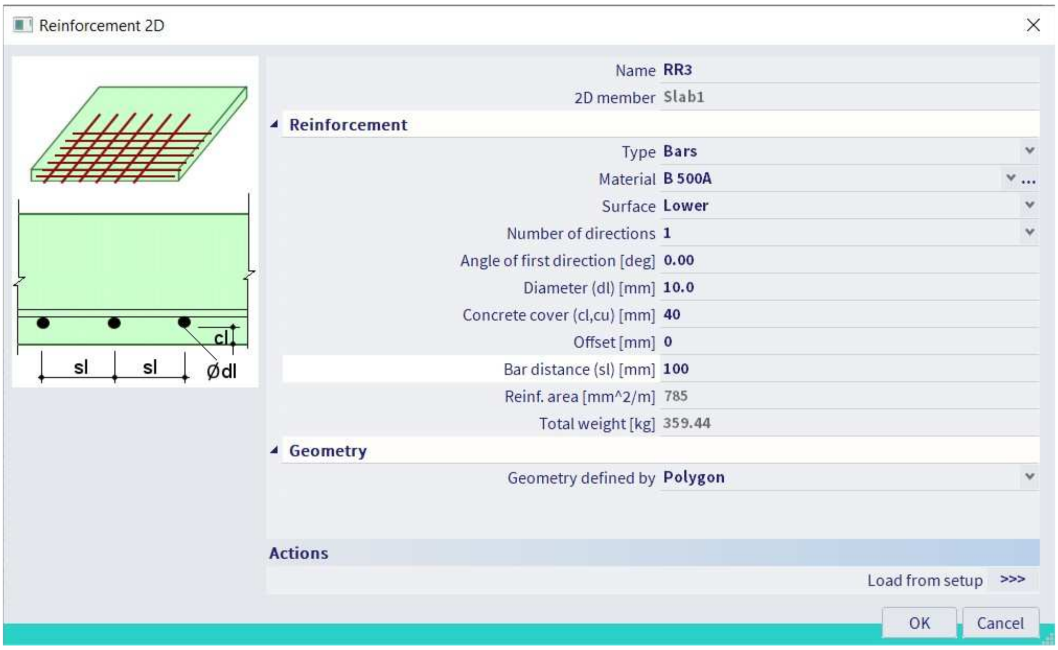
When running the ULS design for the value **As_prov,1-**, it can be seen the provided reinforcement of **Ø10 à 150** will be insufficient to withstand the acting loads. This indicates the application of additional reinforcement will be necessary.

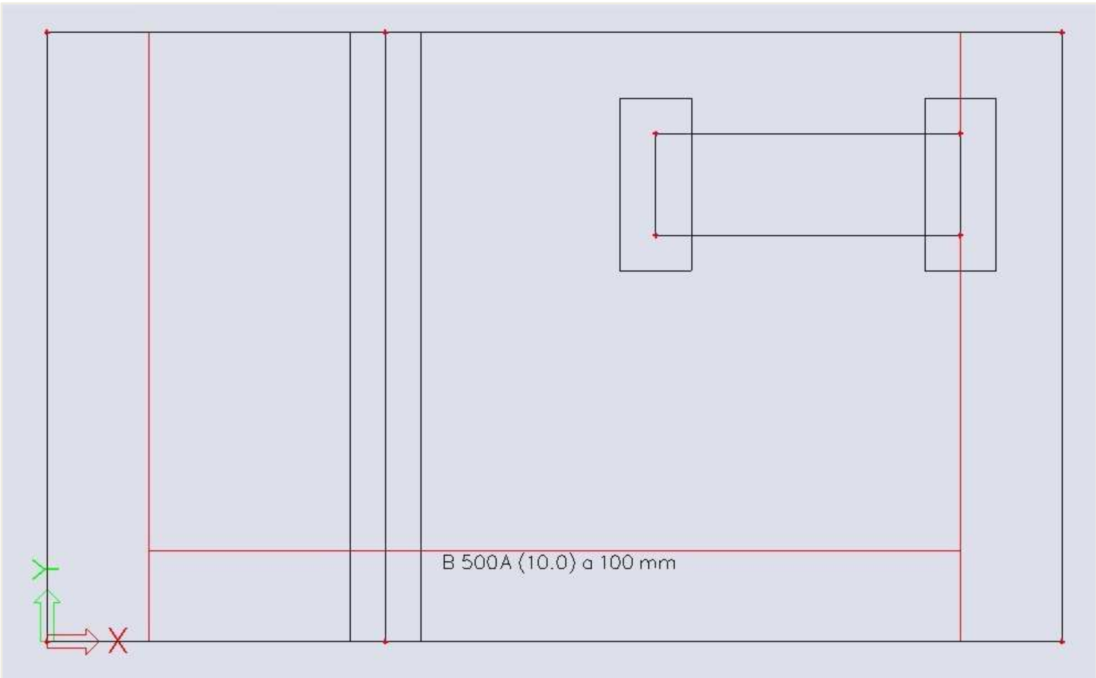


When generating the value **As_add,req,1-**, you can see the exact amount of reinforcement in mm2/m which needs to be added on top of the provided reinforcement. In this case an additional reinforcement of **578 mm2/m** will be necessary. This value can be translated into the configuration of **Ø10 à 100** as practical reinforcement.

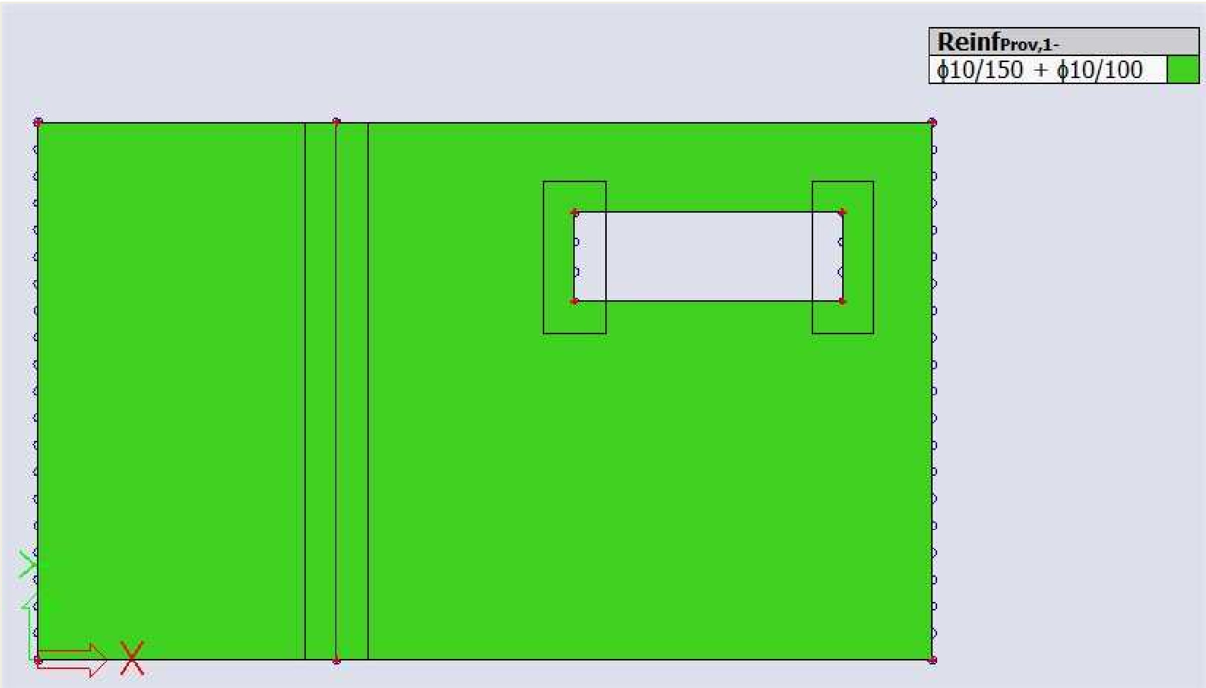


This value can be translated into the configuration of **Ø10 à 100** as practical reinforcement. Since there is no required additional reinforcement in direction 2, only one direction of reinforcement will be added to the 2D member by using practical reinforcement as defined within the previous section.





When generating the results once more for the value **As_prov,1-** and activating the option '**Consider user reinforcement**', it can be seen the user defined reinforcement of **Ø10 à 100** is added on top of the basic reinforcement of **Ø10 à 150** which is defined within the design defaults.



The applied values are visible within the preview of the reinforcement design.

Longitudinal reinforcement - Summary

Designed reinforcement layers (in direction from the member local x axis):

	Basic	Additional User	Additional Auto	α [°]	$A_{s,min}$ [mm^2]	$A_{s,ult}$ [mm^2]	$\Delta A_{s,serv}$ [mm^2]	$A_{s,req}$ [mm^2]	$A_{s,prov}$ [mm^2]	$A_{s,max}$ [mm^2]	$s_{min(cl)}$ [mm]	s_{max} [mm]	Status
[2+]	ϕ10/150	---	---	90.0	277	53	---	**277**	**524**	10000	58	60	**OK**
								0.11%	0.21%		≥37	≤400	
[1-]	ϕ10/150	ϕ10/100	---	0.0	291	1102	---	**1102**	**1309**	10000	55	60	**OK**
								0.44%	0.52%		≥37	≤400	
[2-]	ϕ10/150	---	---	90.0	277	70	---	**277**	**524**	10000	58	60	**OK**
								0.11%	0.21%		≥37	≤400	

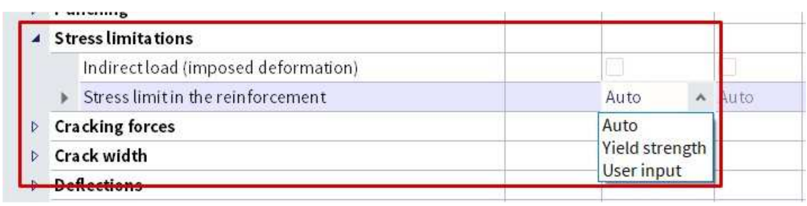
$A_{s,req}$ - required reinforcement area as max($A_{s,ult}$, $A_{s,min}$)+$\Delta A_{s,serv}$; $A_{s,prov}$ - provided reinforcement area; $A_{s,min/max}$ - min/max reinforcement area; $s_{max(min)}$ - maximum spacing of bars (minimum clear distance between bars

The option '**Consider user reinforcement**' is also accessible within all the reinforcement checks – crack width, punching and CDD. This allows you to easily check the reinforcement introduced by both the template and the practical bars.

2.4.5 SLS Design of 2D members – Crack width and stress limitation

Next to the ULS design of 2D members the Eurocode defines some restrictions related to SLS design as well, more specifically the crack width and the limitation of the tensile stress in the reinforcement. Due to these SLS conditions you might need to increase the amount of reinforcement which should be sufficient to withstand the acting ULS forces. The total amount of reinforcement to fulfil the conditions for both the ULS and SLS design can be calculated within SCIA as well as the increment of statically required reinforcement.

The principle of this design method will be explained by the following example of a 2D plate. On this member CMD will be applied in which the crack width in the first direction at the bottom surface will be limited to **0,100 mm**. The tensile stress in the reinforcement can be limited both within the design defaults and the CMD. In this example the limit will be set to **150 MPa**.



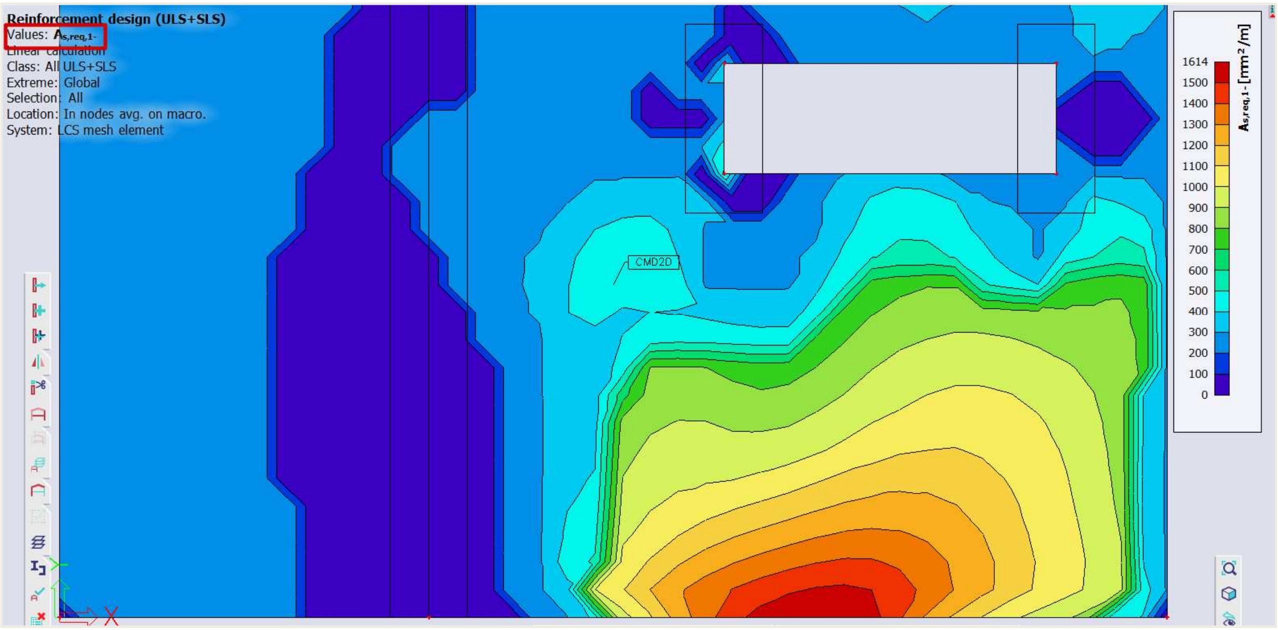
Since this design method is applicable for the ULS and SLS, it is important to select a result class which contains both ULS and SLS combination.

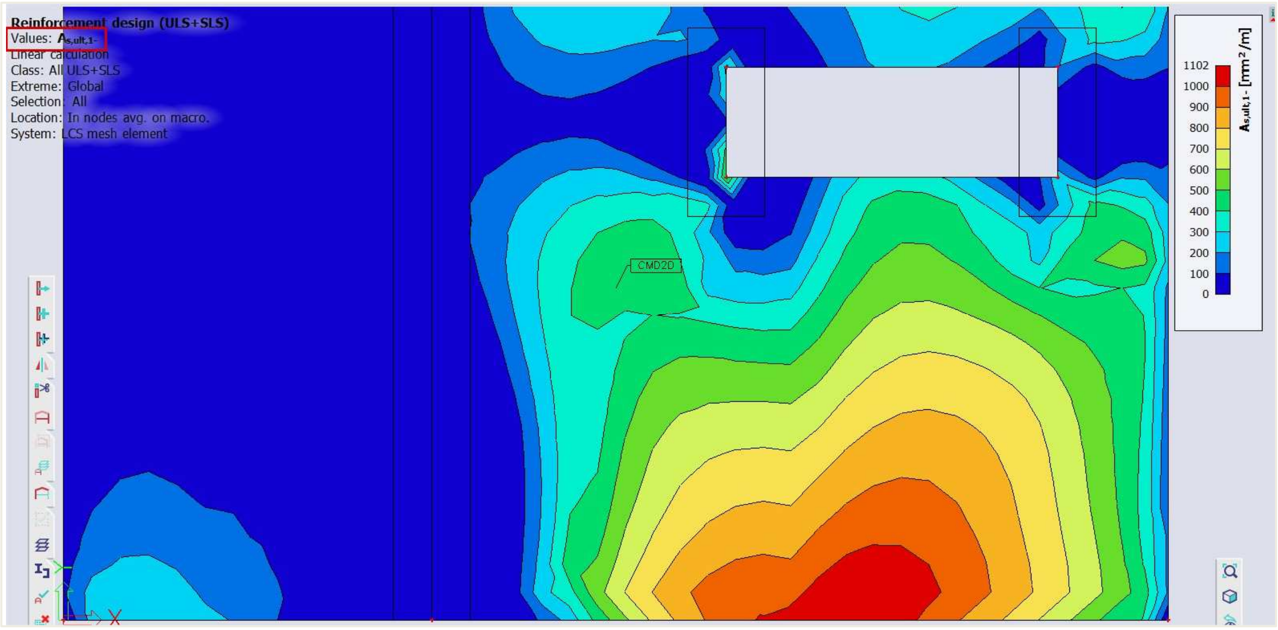
The first step of the design procedure consists of the determination of **As_req** for the ULS state for each direction and each surface. During this step SCIA will determine two values, more specifically:

(a) **As_ult:** the statically required reinforcement to withstand the ULS acting forces;

(b) **As_req:** the required reinforcement including the detailing provisions from the EN.

When looking at the given example, it can be seen the required reinforcement **As_req,1-** is equal to **1614 mm2/m**. The statically required reinforcement **As_ult,1-** is equal to **1102 mm2/m**. This value is a bit lower since it does not contain the increment of longitudinal reinforcement due to the SLS design.

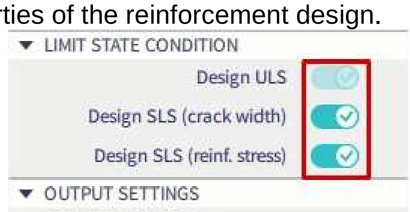




After the calculation of **As_ult** you can choose to integrate the SLS restriction and you have three possibilities:

- Combination of the ULS and SLS design based on cracks.
- Combination of the ULS and SLS design based on stress limitation.
- Combination of the ULS and SLS design based on cracks and stress limitation.

This can be defined within the properties of the reinforcement design.



After activating these settings the increment of longitudinal reinforcement can be generated, in this case the value **ΔAs_serv,1-**. SCIA will determine the principal forces **mEd,ch** and **mEd,QP** in order to calculate the appearance of cracks based on the designed ULS reinforcement **As_ult**. Next to the principal forces it is also necessary to calculate the amount of reinforcement in the direction of the principal forces.

Within the following step, SCIA will determine the maximum allowable crack width based on chapter 7.3.4 from EN 1992-1-1:2004 and compare it to the defined limit as shown below.

Principal stress σ_I[-]=-4.58°

$m_{Ed,char} = 65$ kNm/m | $n_{Ed,char} = 0$ kN/m

$m_{Ed,qp} = 47$ kNm/m | $n_{Ed,qp} = 0$ kN/m

Recalculation of required areas to direction of principal stress

$$A_{s,ult,\sigma} = A_{s,ult,1-} \cdot \cos(\Delta\alpha_{1-})^2 + A_{s,ult,2-} \cdot \cos(\Delta\alpha_{2-})^2$$
$$= 1102 \cdot \cos(-5)^2 + 277 \cdot \cos(-95)^2 = 1097 \text{ mm}^2$$
$$A_{s,serv,\sigma} = A_{s,ult,\sigma} + \Delta A_{s,serv,1-} \cdot \cos(\Delta\alpha_{1-})^2 + \Delta A_{s,serv,2-} \cdot \cos(\Delta\alpha_{2-})^2$$
$$= 1097 + 511 \cdot \cos(-5)^2 + 0 \cdot \cos(-95)^2 = 1605 \text{ mm}^2$$

Check of cracks occuring (§7.1(2))

$f_{ct,eff} = 2.6$ MPa

σ_{ct} = 5.716 MPa > σ_{cr} = 2.6 MPa => cracks appear

Check of reinforcement stress limitation (§7.2(5))

σ_s = 149.3 MPa ≤ $\sigma_{s,lim}$ = 150 MPa

Effective tension area (§7.3.2(3))

$h_{c,eff} = 64.4$ mm => $A_{s,eff} = 1605$ mm^2 ($\rho_{p,eff}$ =2.49 %)

Calculation of crack width (§7.3.4)

$$s_{r,max} = k_3 \cdot c + \frac{k_1 \cdot k_2 \cdot k_4 \cdot \phi_{eq}}{\rho_{p,eff}} = 3.4 \cdot 0.03 + \frac{0.8 \cdot 0.5 \cdot 0.425 \cdot 0.01}{0.0249} = 170 \text{ mm} \quad (7.11)$$

$$\varepsilon_{sm}\_\varepsilon_{cm} = \max\left(\frac{\sigma_s - k_t \cdot \left(\frac{f_{ct,eff}}{\rho_{p,eff}}\right) \cdot (1 + \alpha_e \cdot \rho_{p,eff})}{E_s}; \frac{0.6 \cdot \sigma_s}{E_s}\right)$$
$$= \max\left(\frac{149.3 - 0.46 \cdot \left(\frac{2.6}{0.0249}\right) \cdot (1 + 6.35 \cdot 0.0249)}{200000}; \frac{0.6 \cdot 149.3}{200000}\right) = 0.468\ ‰$$

$w_k = s_{r,max} \cdot \varepsilon_{sm}\_\varepsilon_{cm} = 170 \text{ mm} \cdot 0.468\ ‰ = 0.0797$ mm

Check of crack width

w_k = 0.0797 mm ≤ w_{max} = 0.1 mm

If the cracks are within the limit, then **As_ult** is sufficient to fulfill the restrictions for both ULS and SLS. If not, then SCIA will start the iteration process to increase the **As,ult** by an extra amount of reinforcement to ensure the crack width is within the allowable limits. When looking at the table below it can be seen an additional amount of **1166 mm2/m** for the first direction at the bottom of the member should be added to the reinforcement **As_ult,1-**.

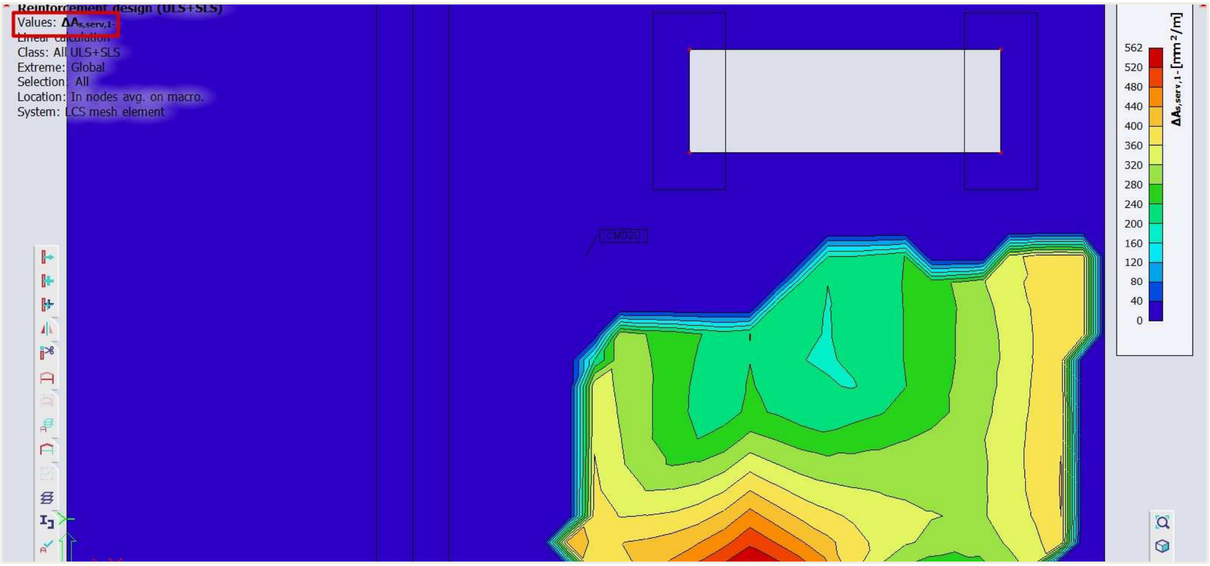
Longitudinal reinforcement - Summary

Designed reinforcement layers (in direction from the member local x axis):

	Basic	Additional User	Additional Auto	α [°]	$A_{s,min}$ [mm^2]	$A_{s,ult}$ [mm^2]	$\Delta A_{s,serv}$ [mm^2]	$A_{s,req}$ [mm^2]	$A_{s,prov}$ [mm^2]	$A_{s,max}$ [mm^2]	$s_{min(cl)}$ [mm]	s_{max} [mm]	Status
[2+]	ϕ10/150	---	---	90.0	277	53	0	**277**	**524**	10000	58	60	**OK**
								0.11%	0.21%		≥37	≤400	
[1-]	ϕ10/150	ϕ10/100	---	0.0	291	1102	511	**1613**	**1310**	10000	55	60	**Not OK**
								0.65%	0.52%		≥37	≤400	
[2-]	ϕ10/150	---	---	90.0	277	70	0	**277**	**524**	10000	58	60	**OK**
								0.11%	0.21%		≥37	≤400	

$A_{s,req}$ - required reinforcement area as max($A_{s,ult}$; $A_{s,min}$)+$\Delta A_{s,serv}$; $A_{s,prov}$ - provided reinforcement area; $A_{s,min/max}$ - min/max reinforcement area; $s_{max(min)}$ - maximum spacing of bars (minimum clear distance between bars

When looking at the output for Δ**As_serv,1-** a value of **562 mm²/m** can be generated.



If this value of Δ**As_serv,1** will be added to the value of **As_ult,1-**, it will result in the value **As_req,1-**. In short the following summary can be created:

- **As_req,i,+/-:** Required reinforcement area for ULS and SLS including detailing provisions for the particular direction (1,2) and surface (+,-).
- **As_ult,i,+/-:** Statically required reinforcement based on ULS for particular direction (1,2) and surface (+,-).
- Δ**As_serv,i,+/-:** Increment of statically required reinforcement based on SLS for particular direction (1,2) and surface (+,-).

The same procedure can be applied for the limitation of tensile stress within the reinforcement. In this case SCIA will determine the amount of reinforcement for the ULS and use this reinforcement to calculate the actual stresses in the reinforcement. This value will then be compared to the defined allowable limit. The limit can be defined in both the design defaults and CMD. You have three possibilities to define the limit of the stresses:

- **Auto**: based on definition in the national annexes 7.2(5).
- **Yield Strength**: the limit is determined based on f_{yk} (characteristic yield strength of reinforcement)
- **User input**: the limit must be decided by the user.

This can be checked within the output, in this case the user defined value of **150 MPa** can be seen.

Principal stress $\sigma_I[-] = -0.0672°$

$m_{Ed,char} = 63.2$ kNm/m | $n_{Ed,char} = 0$ kN/m

$m_{Ed,qp} = 46.5$ kNm/m | $n_{Ed,qp} = 0$ kN/m

Recalculation of required areas to direction of principal stress

$$A_{s,ult,\sigma} = A_{s,ult,1-} \cdot \cos\left(\Delta\alpha_{1-}\right)^2 + A_{s,ult,2-} \cdot \cos\left(\Delta\alpha_{2-}\right)^2$$
$$= 1024 \cdot \cos\left(-0.07\right)^2 + 277 \cdot \cos\left(-90\right)^2 = 1024 \text{ mm}^2$$
$$A_{s,serv,\sigma} = A_{s,ult,\sigma} + \Delta A_{s,serv,1-} \cdot \cos\left(\Delta\alpha_{1-}\right)^2 + \Delta A_{s,serv,2-} \cdot \cos\left(\Delta\alpha_{2-}\right)^2$$
$$= 1024 + 562 \cdot \cos\left(-0.07\right)^2 + 0 \cdot \cos\left(-90\right)^2 = 1586 \text{ mm}^2$$

Check of cracks occuring (§7.1(2))

$f_{ct,eff} = 2.6$ MPa

σ_{ct} = 5.565 MPa > σ_{cr} = 2.6 MPa => cracks appear

Check of reinforcement stress limitation (§7.2(5))

σ_s = 149.4 MPa $\leq$ $\sigma_{s,lim}$ = 150 MPa

As previously mentioned when the SLS restrictions are not fulfilled an increment must be calculated **serv_coeff** will be calculated depending on the following conditions:

- In case of crack width only: $\mathbf{serv_{coeff}=w_{k,coeff}= (w_k / w_{k,max})^{0,5}+0,01}$
- In case of reinforcement stress only: $\mathbf{serv_{coeff}=s_{s,coeff}= (s_s / s_{s,lim})+0,005}$
- In case of reinforcement stress only: $\mathbf{serv_{coeff}= max(s_{s,coeff};w_{k,coeff})}$

When the statically reinforcement is designed based on ULS +SLS, the verification of the detailing provisions must be done. The same procedure and warnings as used for ULS design will be applied for ULS+SLS design, only one step further. The final reinforcement area As_req for direction (1,2) and surface (+,-) will be determined by the following formula, taking into account the minimal and maximal areas from detailing provisions:

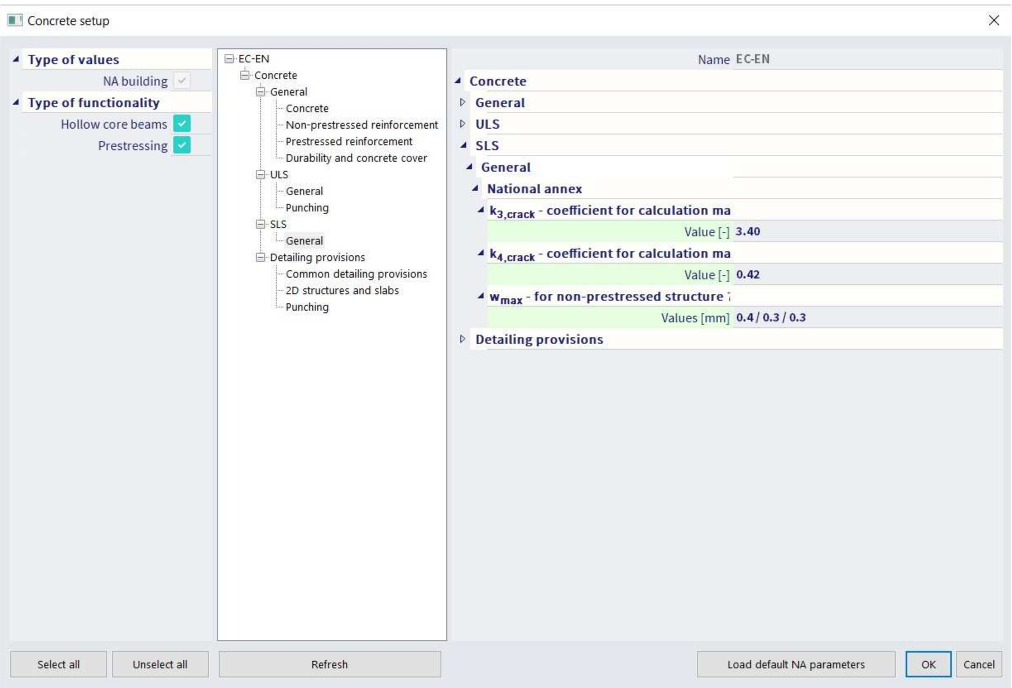
$$\mathbf{A_{s,req,1,2,\pm} = \min(\max(A_{s,ult,1,2,\pm}; A_{s,serv,1,2,\pm}; A_{s,min}); A_{s,max})}$$

2.4.6 Crack control

INPUT DATA FOR CRACK CONTROL

⇨ *Maximum crack width*

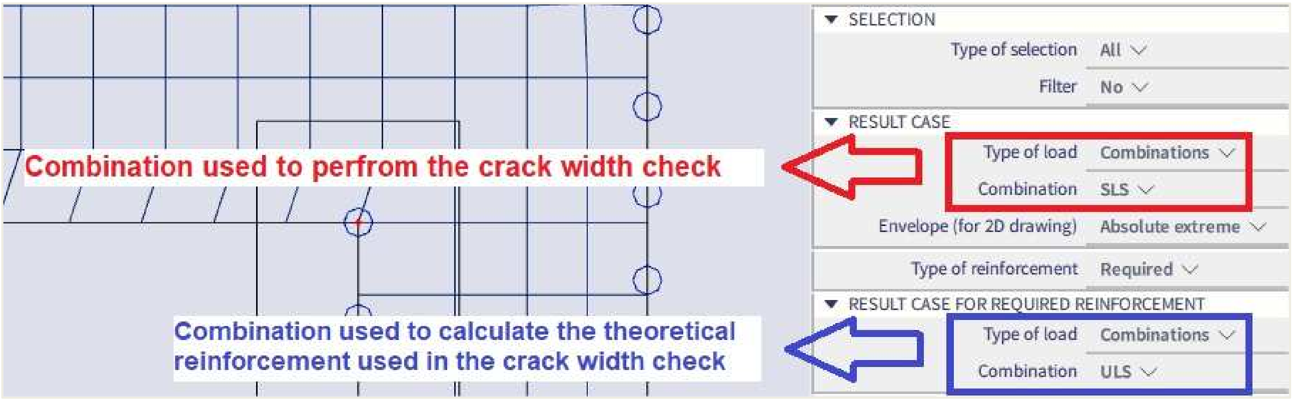
The values of the maximum crack width (w_{max}) are national determined parameters, dependent on the chosen exposure class. Therefore, this value can be found in the setup for National Determined Parameters, via the File menu → Project settings → National annex […] → EN 1992-1-1 […].



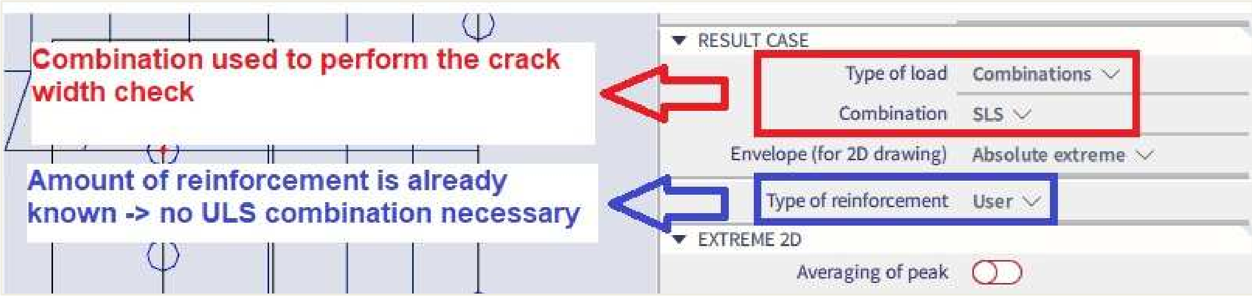
⇨ *Type of used reinforcement*

You can perform the Crack width check for all three types of reinforcement (Required, provided and user reinforcement). The crack width check is performed on a Quasi-permanent SLS combination.

If the type of reinforcement used for the crack width check is either the provided or required reinforcement an ULS combination should be chosen as well. This is necessary because the required/provided reinforcement is calculated based on an ULS combination. After this reinforcement is calculated it can be used to perform the crack width check. All this is done automatically and can be set in the properties window of the crack width check.



Required/provided reinforcement



User reinforcement

⇨ ***Theoretical background***

Crack appearance

If condition below is satisfied no cracks will appear in the concrete.

$$\sigma_{ct,max\pm} \leq f_{ct,eff}$$

With:

- $\sigma_{ct,max\pm} = \frac{n_{i\pm}}{A_{i,i\pm}} + \frac{m_{i\pm}}{I_{i,i\pm}} \cdot z_{t,max,i\pm}$ = normal concrete stress on un-cracked section at the most tensioned fiber of concrete cross-section
- $f_{ct,eff}$ = mean value of the tensile strength of the concrete effective at the time

Calculation of crack width

$$w = \varepsilon_{sm\_cm} \cdot s_{r,max}$$

With:

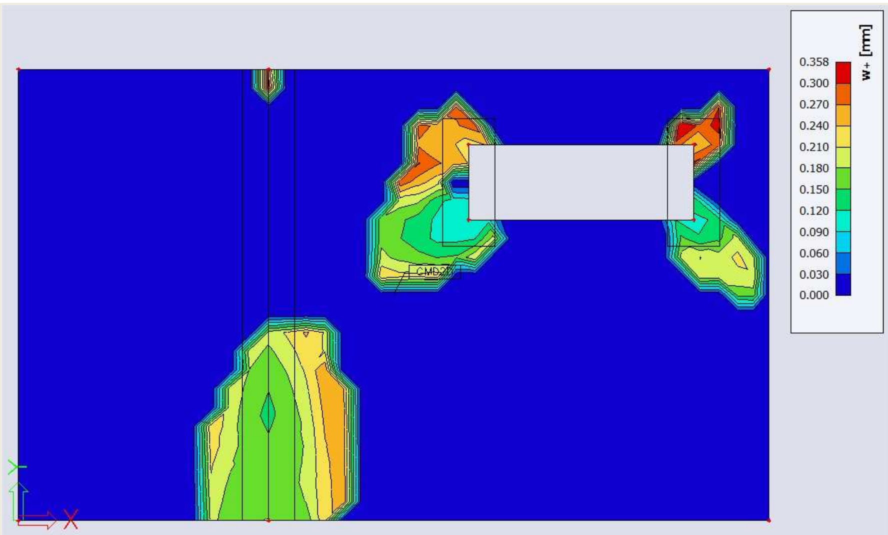
- $(\varepsilon_{sm} - \varepsilon_{cm})_{i\pm} = \max\left[\frac{\sigma_{s,i\pm} - k_t \cdot \frac{f_{ct,eff}}{\rho_{p,eff,i\pm}} \cdot (1+\alpha_{e,i\pm} \cdot \rho_{p,eff,i\pm})}{E_{s,i\pm}} ; 0,6 \cdot \frac{\sigma_{s,i\pm}}{E_{s,i\pm}}\right]$
- $s_{r,max,i\pm} = \begin{cases} \min\left(k_3 c_{i\pm} + \frac{k_{1,i\pm} k_{2,i\pm} k_4 d_{s,i\pm}}{\rho_{p,eff,i\pm}} ; 1,3 \cdot (h - x_{i\pm})\right) & \text{if } s_{s,i\pm} < 5\left(c_{i\pm} + 0,5 d_{s,i\pm}\right) \\ 1,3 \cdot (h - x_{i\pm}) & \text{f } s_{s,i\pm} \geq 5\left(c_{i\pm} + 0,5 d_{s,i\pm}\right) \end{cases}$

RESULTS FOR REQUIRED THEORITICAL REINFORCEMENT

Desing menu → Concrete 2D → SLS crack width

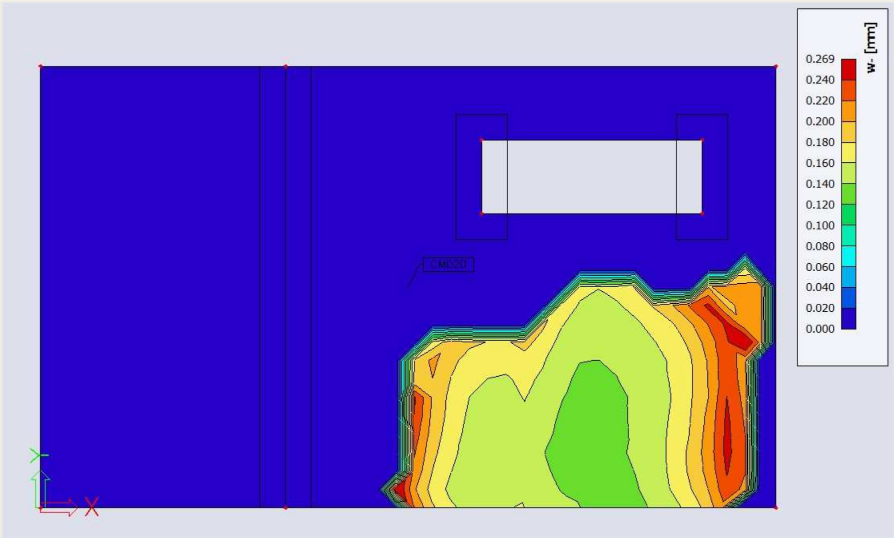
Crack width w+

Combination = SLS; Type of reinforcement = Required; Value = **w+**



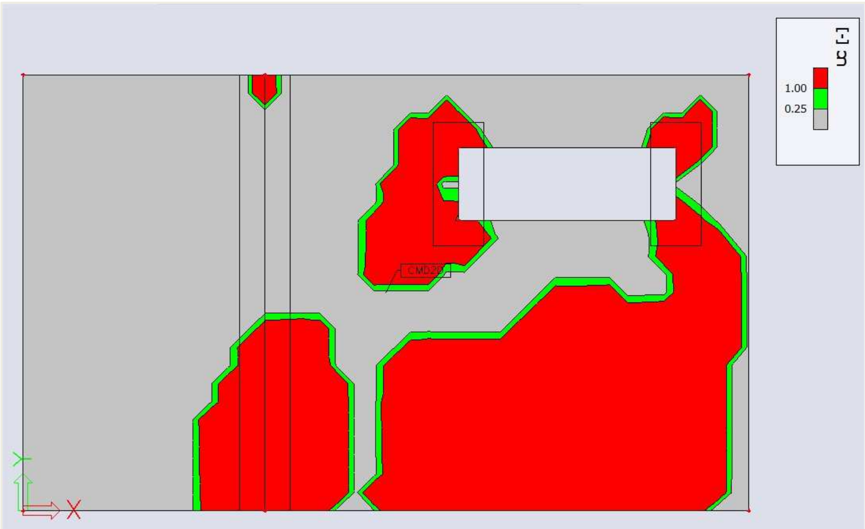
Crack width w-

Combination = SLS; Type of reinforcement = Required; Value = **w-**



Unity check

Combination = SLS; Type of reinforcement = Required; Value = **UC**



A green value stands for a unity check ≤ 1 ($w_{calc} \leq w_{max}$), a grey value stands for unity check ≤ 0,25 and a red value means that w_{max} is exceeded.

2.5 Punching

2.5.1 Theoretical background

General

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.

The most common situations where punching shear has to be considered is the region immediately surrounding a column in a flat ceiling plate or where column is supported on foundation plate.

The following problem types can be distinguished: interior, edge and corner columns.

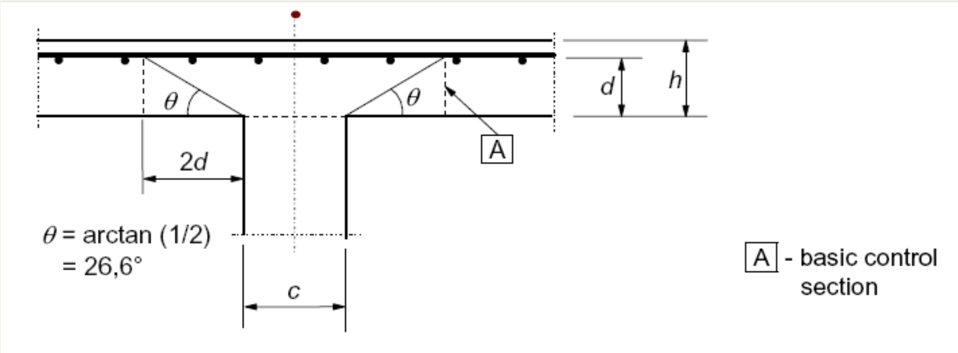
Design of punching shear reinforcement is based on clause 6.4 of EN 1992-1-1: 2004 / A1:2014 + National Annexes.

The verification reveals either that the load-bearing capacity of the reinforced concrete is sufficiently high, or that punching shear reinforcement must be designed and installed. If the verification limits are exceeded, the verification result is marked as not permissible. In this case, you must change the model parameters or select a suitable design alternative.

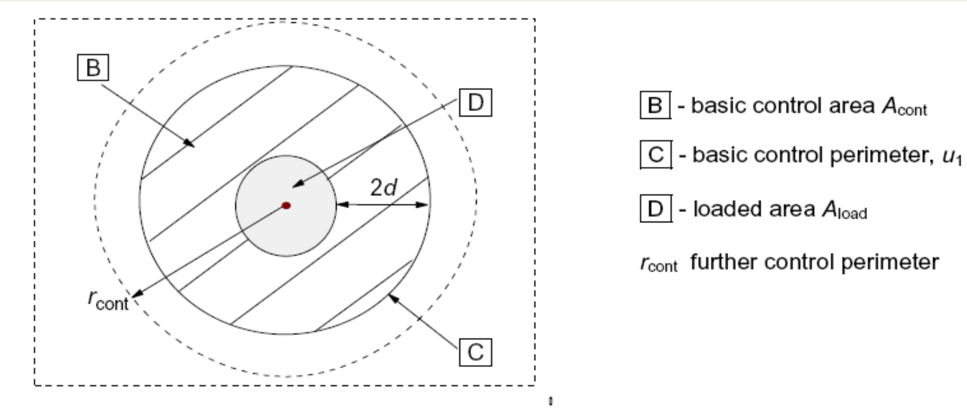
The verification of punching failure at the ultimate limit state can be resumed as follows:

- Check of the shear resistance at the face of the column noted u_0, and at the basic control perimeter named u_1.
- If shear reinforcement is required, a further perimeter $u_{out,ef}$ should be found where shear reinforcement is no longer required.

Those control perimeters are shown in the following pictures:



a) Section

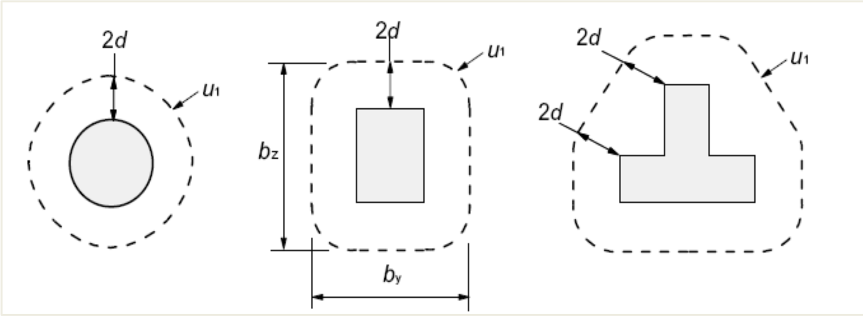


b) Plan

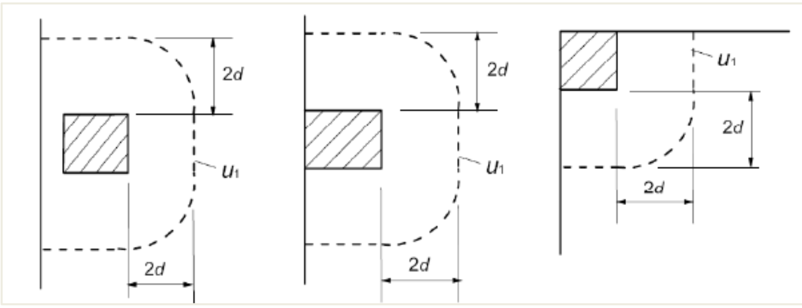
Load distribution and basic control perimeter

⇨ *Basic control perimeter u1*

The basic control perimeter u_1 is taken at a distance 2d from the loaded area, where d is the effective depth.

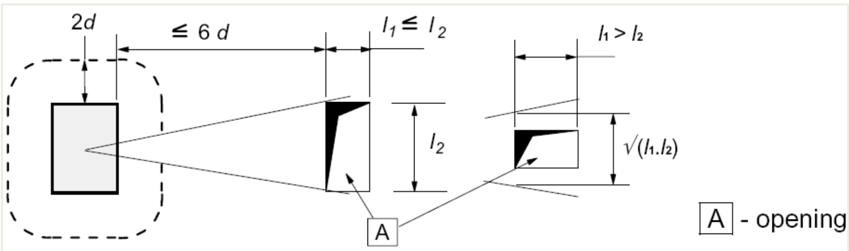


In case the loaded area is close to an edge or a corner:



In case there is openings near the loaded area, they are dealt with according to clause 6.4.2(3).

If the shortest distance between the perimeter of the loaded area and the edge of the opening does not exceed 6d (see figure), part of the control perimeter contained between two tangents drawn to the outline of the opening from the center of the loaded area is ineffective.



In SCIA Engineer, openings on 2D members are automatically considered according to the previous criteria.

⇨ *Effective depth d_{eff}*

The effective depth of the slab is assumed constant and is calculated acc. to formula 6.32 of EN1992-1-1:

$$d_{eff} = \frac{(d_y + d_z)}{2}$$

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

Punching shear calculation

The punching shear calculation is done according to EN1992-1-1 art.6.4.3.

First the design shear resistances along the control sections are calculated:

- $v_{Rd,c}$ 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

Then the following checks should be performed.

⇨ ***Check at the column perimeter u_0***

At the column perimeter u_o, or at the perimeter of the loaded area, the maximum punching shear stress should not be exceeded.

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

With :

- v_{Ed0} design shear stress at the column perimeter u_0
- $v_{Rd,max} = 0.5 * v * f_{cd}$
- $v = 0.6*(1 - f_{ck}/250)$

⇨ ***Check at the basic perimeter u_1***

At the basic control perimeter u_1:

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

The punching shear resistance of a plate $V_{Rd,c}$ is calculated according to formula 6.47, EN1992-1-1:

$$v_{Rd,c} = C_{Rd,c}.k.(100.\rho_l.f_{ck})^{1/3} + k_1\sigma_{cp} \geq (v_{min} + k_1\sigma_{cp})$$

With:

- ρ_l average reinforcement ratio in specific distance around column
- f_{ck} characteristic concrete compressive strength in MPa
- $v_{min} = 0{,}035 \cdot k^{3/2} \cdot f_{ck}^{1/2}$
- $C_{Rd,c} = \frac{0{,}18}{\gamma_c}$
- $k = 1 + \sqrt{\frac{200}{d}} \leq 2{,}0$
- d in mm

The maximum shear stress v_{Ed} is calculated for considered control perimeter u_i according to clause 6.4.3(1) as follows:

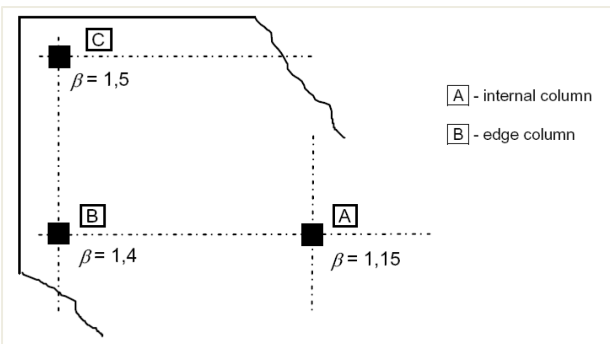
$$v_{Ed} = \beta.\frac{V_{Ed}}{u_i d}$$

The β-factor is to consider the non-uniform load transfer (due to unbalanced bending moment). If the load transfer is non-uniform, local peak loading should be compensated by help of this β-factor.

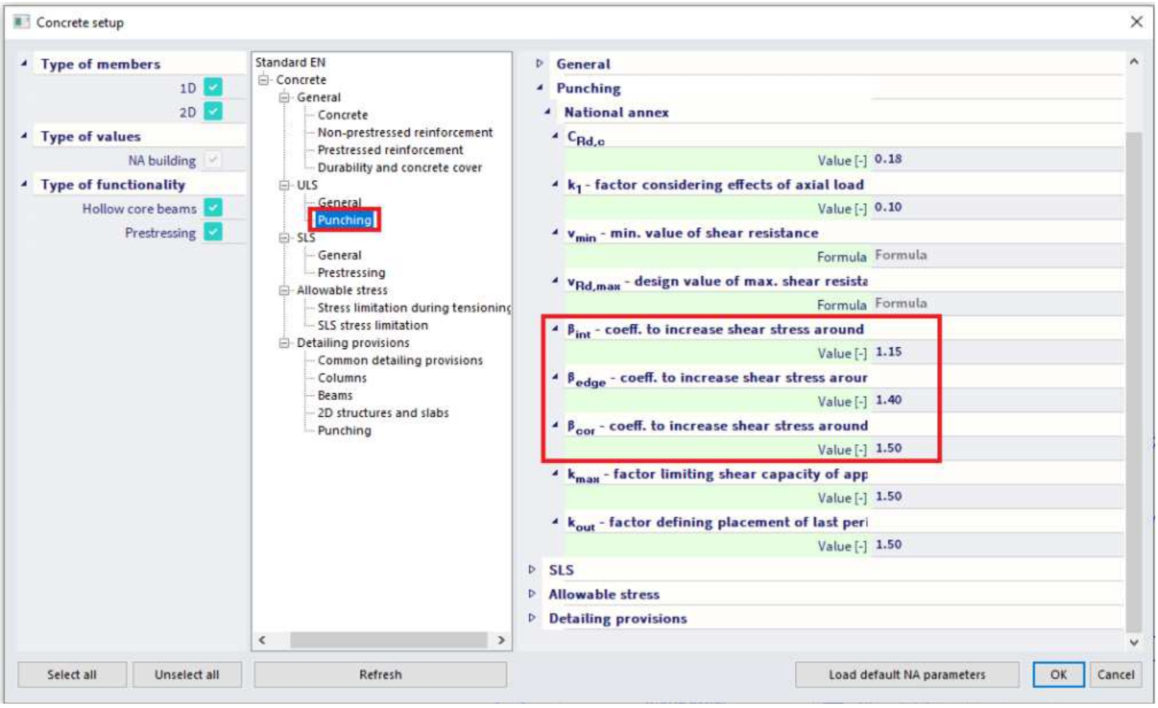
In case that lateral stability of the structure does not depend on frame action between the slabs and the columns, and where the adjacent spans do not differ in length by more than 25%, approximate values for β may be used according to clause 6.4.3(6).

In SCIA Engineer, you must decide whether these approximate values can be used, because the program cannot check the preconditions described above.

By default, the recommended approximated values are:



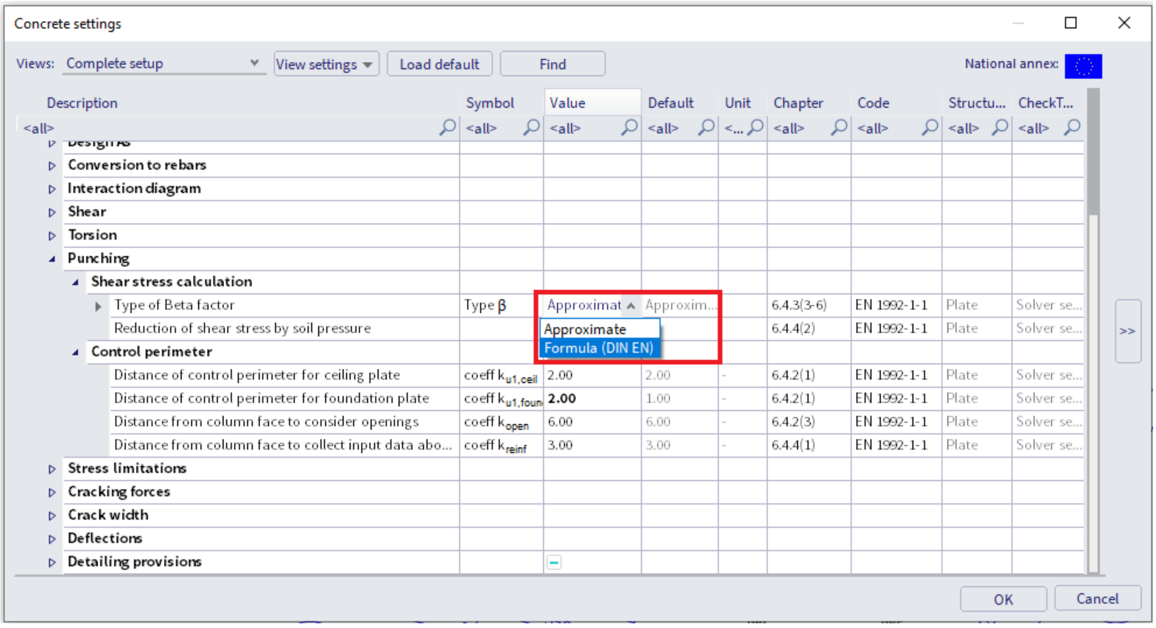
Those values might be different according to the National Annexes and can be viewed in the National Annexes setup:



Otherwise, as described in art 6.4.3, the β-factor can be calculated by the following general formula:

$$\beta = 1 + \sqrt{\left(k_y.\frac{M_{Ed,y}}{V_{Ed}}.\frac{u_1}{W_{1y}}\right)^2 + \left(k_z.\frac{M_{Ed,z}}{V_{Ed}}.\frac{u_1}{W_{1z}}\right)^2}$$

Calculation of β-factor with general formula can be set in Concrete setup > Punching:



⇨ *Design of punching reinforcement if required*

In case that $v_{Ed} > v_{Rd,c}$, punching reinforcement should be designed.

If punching reinforcement is required, the outer control perimeter u_{out} beyond which the reinforcement is no longer needed is calculated acc. to clause 6.4.5(4):

$$u_{out,ef} = \frac{\beta . V_{Ed}}{v_{Rd,c} . d}$$

Calculation of the required punching reinforcement

In SCIA Engineer, the shear reinforcement is designed using the following assumptions:

- the distribution of the shear links is considered as radial only
- only vertical shear links are supported
- the shape of reinforcement perimeters around the column is the same as for the shape of the basic control perimeter

The required area $A_{sw,req}$ of one perimeter of shear reinforcement around the column assumed as radially distributed vertical shear links is calculated as:

$$A_{sw,req} = \frac{(v_{Ed,u1} - 0.75 \cdot v_{Rd,c}) \cdot u_1 \cdot s_r}{1.5 \cdot f_{ywd,ef}}$$

$f_{ywd,ef}$ is the effective design strength of the punching reinforcement according to formula:

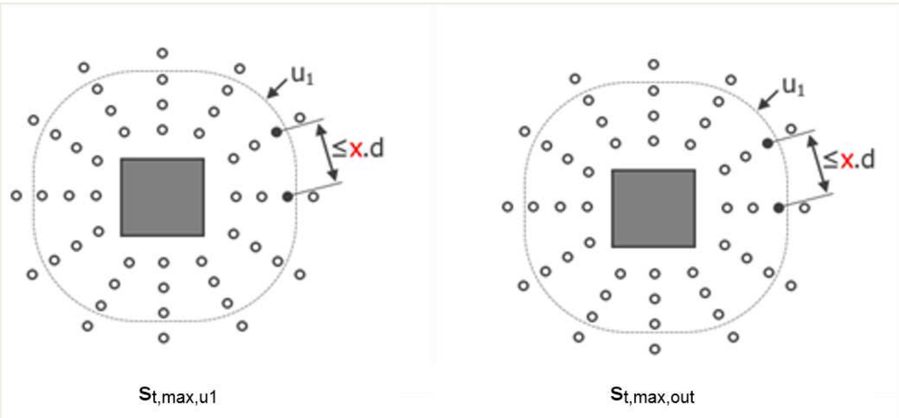
$f_{ywd,ef} = 200 + 0.25 \cdot d_{eff} \leq f_{ywd}$

Detailing provisions for the punching reinforcement

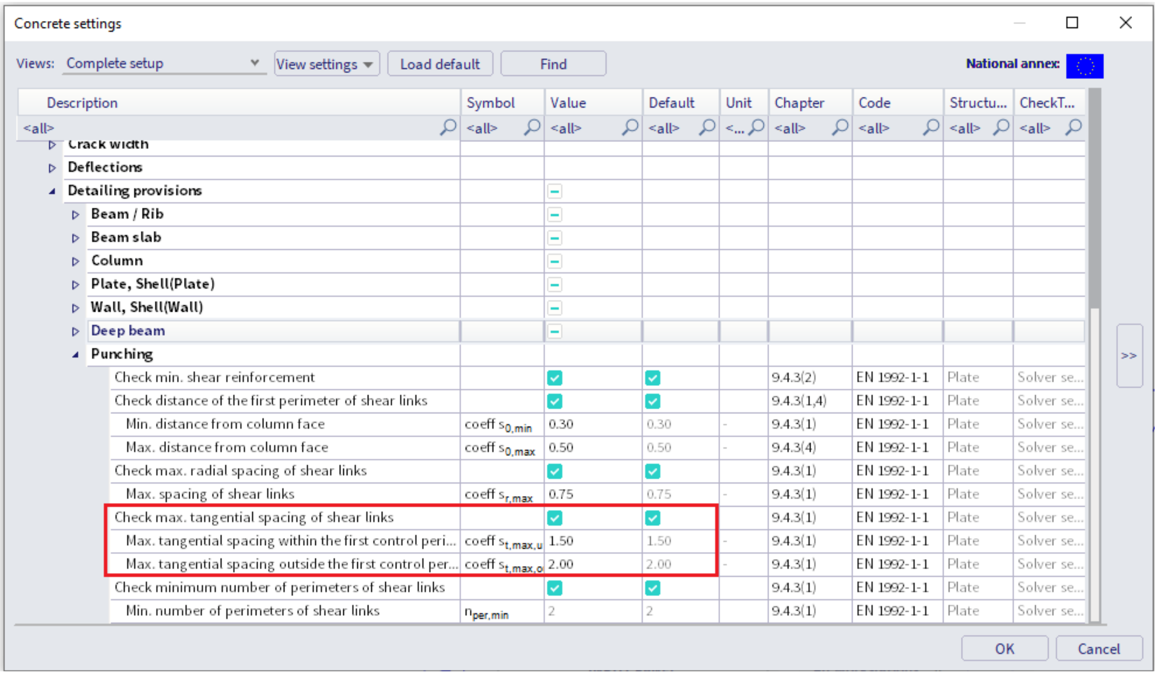
The required area might be adjusted to fulfil detailing provision rules acc. to clause 9.4.3(1), so that number of shear links n_s per each reinforcement perimeter is:

$$n_s = \max\left\{\frac{4 \cdot A_{sw,req}}{\pi \cdot d_s^{\,2}}; \frac{u_{1,last}}{s_{t,max,u1}}; \frac{u_{s,last}}{s_{t,max,out}}\right\}$$

- d_s diameter of shear link
- $\frac{u_{1,last}}{s_{t,max,u1}}$ condition of maximum allowed tangential spacing of links of reinforcement perimeters placed within the basic control perimeter ($u_{1,last}$ is length of last perimeter of shear reinforcement there)
- $\frac{u_{s,last}}{s_{t,max,out}}$ condition of maximum allowed tangential spacing of links of reinforcement perimeters placed outside the basic control perimeter ($u_{s,last}$ is length of last perimeter of shear reinforcement there)



In SCIA Engineer, limitation of spacing $s_{t,max,u1}$ and $s_{t,max,out}$ are set in Concrete setup > Detailing provisions > Punching:



The last condition, which must be fulfilled acc. to clause 9.4.3(2) is minimum reinforcement area of single shear link $A_{sw1,min}$ according to formula (9.11):

$$A_{sw1,min} = \frac{0{,}08 \cdot \sqrt{f_{ck} / f_{ywk}} \cdot s_r \cdot s_t}{1{,}5}$$

With:

- s_r spacing of shear links in the radial direction
- s_t spacing of shear links in the tangential direction

The final designed area of each perimeter of shear reinforcement around the column is:

$$A_{sw} = \frac{n_s * \pi * d_s^2}{4} \geq n_s * A_{sw1,min}$$

The required number of shear reinforcement perimeters around columns, n_{per}, is determined based on clause 6.4.5(4), which specifies that the outermost perimeter of shear reinforcement, $a_{s,\,last} = s_0 + s_r * n_{per}$, should be placed at a distance not greater than $k_{out} * d_{eff}$ within u_{out}. The following formula for n_{per} is derived:

$$n_{per} = \left\lceil \frac{a_{out} - s_0 - k_{out} * d_{eff}}{s_r} + 1 \right\rceil \geq n_{per,min}$$

With:

- K_{out} Coefficient to determine the maximum distance of last perimeter from u_{out}. Default value is 1,5. This is a National Annexes parameter.
- $N_{per,min}$ Minimum number of reinforcement perimeters around column required acc. to clause 9.4.3(1). Default value is 2 in Concrete settings > Complete setup view > Detailing provisions > Punching.
- A_{out} Distance of the outer perimeter u_{out}.

The total amount of shear reinforcement $A_{sw,tot}$ around the column is then calculated as :

$$A_{sw,tot} = n_{per} * A_{sw}$$

2.5.2 Punching design

Configuration

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 only for the punching check.

The column position with regard to the edges of the plate and the openings is recognize. Also, for the punching check, all edges and angles of the plate are taken as straight... so if they are not in your model, the program makes an approximation.

SCIA Engineer doesn't support all punching cases of column-plate connection. The list of all current limitations can be found in our webhelp. Each unsupported configuration is mentioned in the list of Errors/warning/notes of the report in the punching check report.

Summary

Name	Case	Punching case	Punching shape	$UC_{vRd,max}$ [-]	$UC_{vRd,c}$ [-]	Shear reinforcement perimeters	$UC_{vRd,cs}$ [-]	$UC_{Asw,det}$ [-]	UC [-] Check	Errors, warnings, notes
N61	ULS/1	N/A	N/A	**3.00**	**3.00**	N/A	-	-	3.00 NOT OK	W6/131
N63	ULS/1	N/A	N/A	**3.00**	**3.00**	N/A	-	-	3.00 NOT OK	W6/124

Concrete

Name	Case	Punching case	Punching shape	V_{Ed} [kN] β [-]	$M_{Ed,y}$ [kNm] $M_{Ed,z}$ [kNm]	Plate h [mm]	Material f_{cd} [MPa]	d_{eff} [mm] ρ_l [%]	u_0 [m] u_1 [m]	$v_{Ed,u0}$ [MPa] $v_{Ed,u1}$ [MPa]	$v_{Rd,max}$ [MPa] $v_{Rd,c}$ [MPa]	$UC_{vRd,max}$ [-] $UC_{vRd,c}$ [-]
N61	ULS/1	N/A	N/A	- -	- -	N/A -	N/A -	- -	- -	- -	- -	**3.00** **3.00**
N63	ULS/1	N/A	N/A	- -	- -	N/A -	N/A -	- -	- -	- -	- -	**3.00** **3.00**

E/W/N	Present on members
W6/131	N61
W6/124	N63

E/W/N	Description	Solution
W6/131	Node cannot be calculated for punching. The connected column has not supported type of cross-section.	
W6/124	Node cannot be calculated for punching. The connected column goes through the plate.	Split the column in the node to get a separate column above and below the plate.

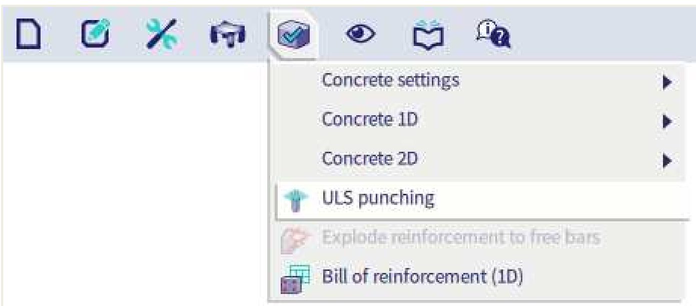
Choice of reinforcement

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

In SCIA Engineer you can choose between 3 types of reinforcement for the punching check/design:

- As, required – calculated by the software for a specific load combination
- As,provided – user set in Reinforcement design > Design defaults
- As,user - practical reinforcement inputted by user manually in 2D Reinforcement

The choice between As,required, As,provided or As,user is done in the Properties panel for Punching design:



Punching check

Studied example: punching.esa

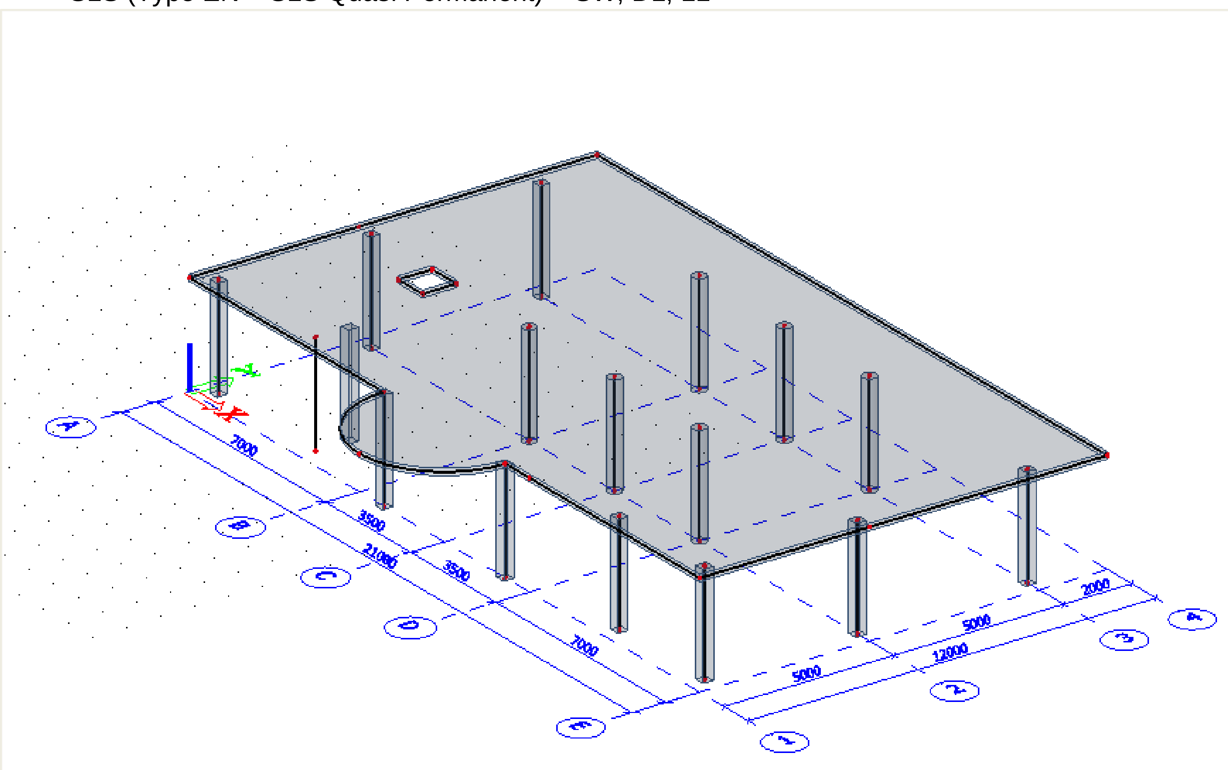
Geometry

Concrete class C30/37
Reinforcement class B500B
Plate thickness 200 mm
Column cross-section 10 x R 300x300 mm² and 6 x circular C400 mm²
Plate and columns are connected to each other by means of the action Connect members/nodes.

Loads

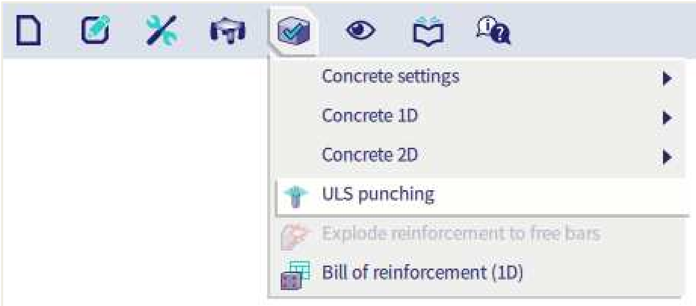
*Load cases
- SW: Self weight
- DL: Dead Load = Surface load -1 kN/m² + Line force on edges -1 kN/m
- LL: Live Load = Surface load -1 kN/m²
- LL1: Additional case for further study= -25 kN/m², to be explained later

*Combinations
- ULS (Type EN – ULS (STR/GEO Set B)) = SW, DL, LL
- SLS (Type EN – SLS Quasi Permanent) = SW, DL, LL

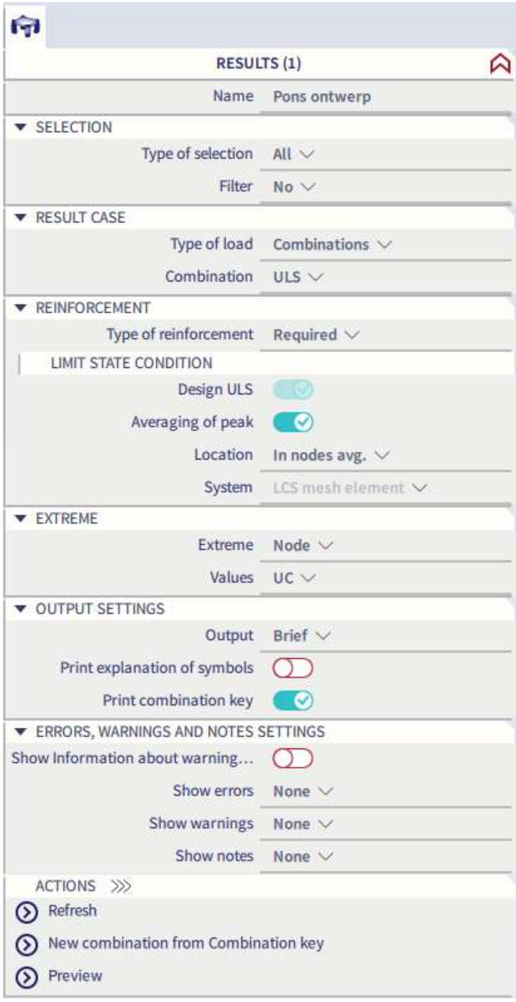


Work method

The **Punching Design** command can be selected in the main menu “Design”:



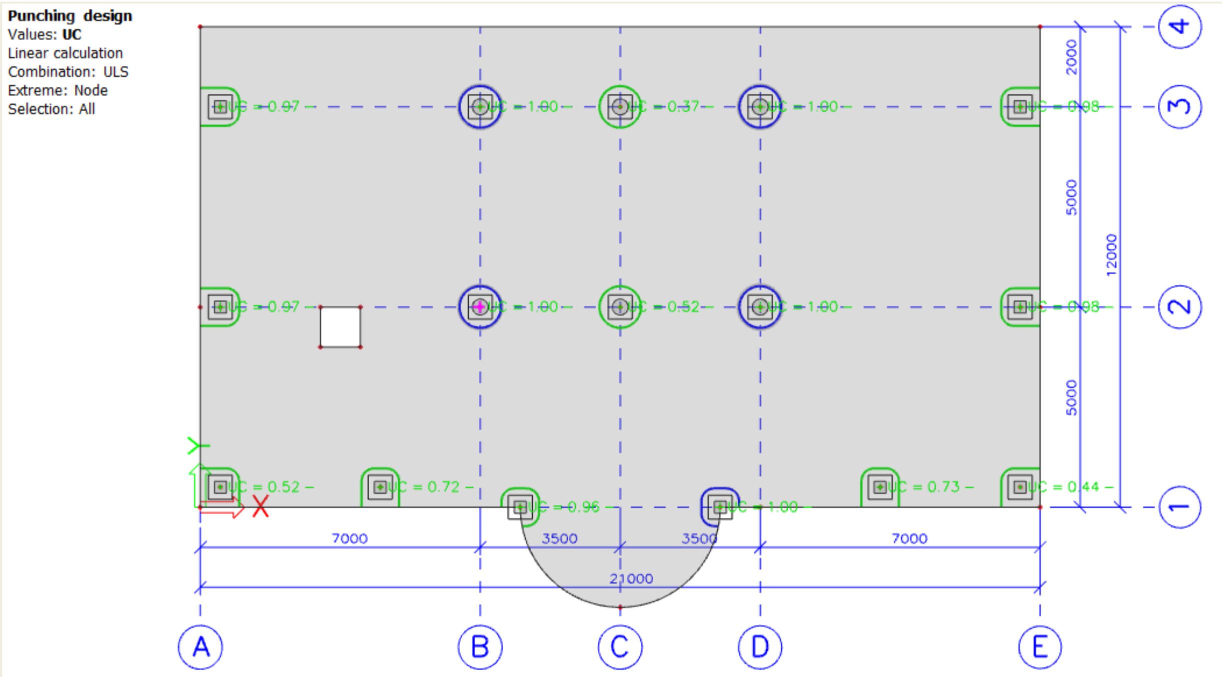
The command is available, when EC – EN national code is selected in Project data and the linear or non-linear static analysis is done for the model containing 2D members from concrete material. Once the command is selected, appropriate parameters are listed and can be adjusted in property window with following options.



Set the type of Selection to ALL, the Type of load to Combination ULS and the type of Reinforcement to Required then click “Refresh”

You will notice that the UC for every node will be displayed along with the control parameter in colour. In total there are 3 colours (Green, blue and red).

- Green: Shear capacity without reinforcement is sufficient ($UC_{vRd,c} \leq 1.0$ and $UC_{vRd,max} \leq 1.0$)
- Blue: Shear capacity with shear reinforcement is sufficient ($UC_{vRd,c} > 1.0$ but $UC_{vRd,cs} \leq 1.0$)
- Red: Plate is not designable by application of reinforcement or maximum shear capacity of concrete adjacent to the column is not sufficient ($UC_{vRd,cs} > 1.0$ or $UC_{vRd,max} > 1.0$)



Presentation of results as a numerical output is possible via Preview and / or Table results. For the Punching Design, there is available two types of output:

- Brief - contains just a summary table with basic results

Punching design

Linear calculation
Combination: ULS
Extreme: Node
Selection: All
Summary

Name	Case	Punching case	Punching shape	$UC_{vRd,max}$ [-]	$UC_{vRd,c}$ [-]	Shear reinforcement perimeters	$UC_{vRd,cs}$ [-]	$UC_{Asw,det}$ [-]	UC [-] Check
N15	ULS/1	Corner column	Rectangle (300;300)	**0.82**	**0.96**	not required	-	-	0.96 OK
N20	ULS/1	Corner column	Rectangle (300;300)	**0.86**	**1.01**	3x 9Ø8(radial) 80+2x80=240	**0.68**	**1.00**	1.00 OK, BUT
N53	ULS/1	Internal column	Circle (400)	**0.37**	**1.07**	3x 12Ø8(radial) 80+2x80=240	**0.72**	**1.00**	1.00 OK, BUT
N55	ULS/1	Internal column	Circle (400)	**0.12**	**0.37**	not required	-	-	0.37 OK
N57	ULS/1	Internal column	Circle (400)	**0.37**	**1.07**	3x 12Ø8(radial) 80+2x80=240	**0.72**	**1.00**	1.00 OK, BUT
N59	ULS/1	Internal column	Circle (400)	**0.36**	**1.06**	3x 12Ø8(radial) 80+2x80=240	**0.71**	**1.00**	1.00 OK, BUT
N61	ULS/1	Internal column	Circle (400)	**0.17**	**0.52**	not required	-	-	0.52 OK
N63	ULS/1	Internal column	Circle (400)	**0.37**	**1.08**	3x 12Ø8(radial) 80+2x80=240	**0.72**	**1.00**	1.00 OK, BUT
N88	ULS/1	Edge column	Rectangle (300;300)	**0.43**	**0.98**	not required	-	-	0.98 OK
N90	ULS/1	Edge column	Rectangle (300;300)	**0.43**	**0.98**	not required	-	-	0.98 OK
N95	ULS/1	Corner column	Rectangle (300;300)	**0.21**	**0.44**	not required	-	-	0.44 OK, BUT

- Standard - contains the same summary table as in Brief output supplemented by additional tables providing further semi-results

⇨ *Shear capacity without reinforcement is sufficient*

Select Node N61 and change the type of selection to current.

A brief output will show:

Punching design

Linear calculation
Combination: ULS
Extreme: Node
Selection: N61
Summary

Name	Case	Punching case	Punching shape	UC $_{vRd,max}$ [-]	UC$_{vRd,c}$ [-]	Shear reinforcement perimeters	UC $_{vRd,cs}$ [-]	UC $_{Asw,det}$ [-]	UC [-] Check
N61	ULS/1	Internal column	Circle (400)	**0.17**	**0.52**	not required	-	-	0.52 OK

Name	Combination key
ULS/1	1.35*SW + 1.35*DL + 1.50*LL

We can see that the UC < 1, let's look at the standard output for this node:

Punching design

Linear calculation
Combination: ULS
Extreme: Node
Selection: N61
Summary

Name	Case	Punching case	Punching shape	UC $_{vRd,max}$ [-]	UC$_{vRd,c}$ [-]	Shear reinforcement perimeters	UC $_{vRd,cs}$ [-]	UC $_{Asw,det}$ [-]	UC [-] Check
N61	ULS/1	Internal column	Circle (400)	**0.17**	**0.52**	not required	-	-	0.52 OK

Concrete

Name	Case	Punching case β [-]	Punching shape	V_{Ed} [kN] ΔV_{Ed} [kN]	$M_{Ed,y}$ [kNm] $M_{Ed,z}$ [kNm]	Plate h [mm]	Material f_{cd} [MPa]	d_{eff} [mm] ρ_l [%]	u_0 [m] u_1 [m]	$v_{Ed,u0}$ [MPa] $v_{Ed,u1}$ [MPa]	$v_{Rd,max}$ [MPa] $v_{Rd,c}$ [MPa]	UC $_{vRd,max}$ [-] UC$_{vRd,c}$ [-]
N61	ULS/1	Internal column 1.15	Circle (400)	**128.46** 0.00	0.09 13.98	Ceiling 200.00	C30/37 20.00	160.00 0.17	1.257 3.267	0.73 0.28	4.22 0.55	**0.17** **0.52**

We can see that $V_{Ed,u1}$ = 0,28MPa < $V_{Rd,c}$ = 0,55MPa so the shear capacity without reinforcement is sufficient. The control parameter is displayed in Green colour.

⇨ *Shear capacity with reinforcement is sufficient*

Let us look now at the standard output for node N59:

Punching design

Linear calculation
Combination: ULS
Extreme: Node
Selection: N59
Summary

Name	Case	Punching case	Punching shape	UC $_{vRd,max}$ [-]	UC$_{vRd,c}$ [-]	Shear reinforcement perimeters	UC $_{vRd,cs}$ [-]	UC $_{Asw,det}$ [-]	UC [-] Check
N59	ULS/1	Internal column	Circle (400)	**0.36**	**1.06**	3x 12Ø8(radial) 80+2x80=240	**0.71**	**1.00**	1.00 OK, BUT

Concrete

Name	Case	Punching case β [-]	Punching shape	V_{Ed} [kN] ΔV_{Ed} [kN]	$M_{Ed,y}$ [kNm] $M_{Ed,z}$ [kNm]	Plate h [mm]	Material f_{cd} [MPa]	d_{eff} [mm] ρ_l [%]	u_0 [m] u_1 [m]	$v_{Ed,u0}$ [MPa] $v_{Ed,u1}$ [MPa]	$v_{Rd,max}$ [MPa] $v_{Rd,c}$ [MPa]	UC $_{vRd,max}$ [-] UC$_{vRd,c}$ [-]
N59	ULS/1	Internal column 1.15	Circle (400)	**265.21** 0.00	26.85 6.10	Ceiling 200.00	C30/37 20.00	160.00 0.37	1.257 3.267	1.52 0.58	4.22 0.55	**0.36** **1.06**

Reinforcement

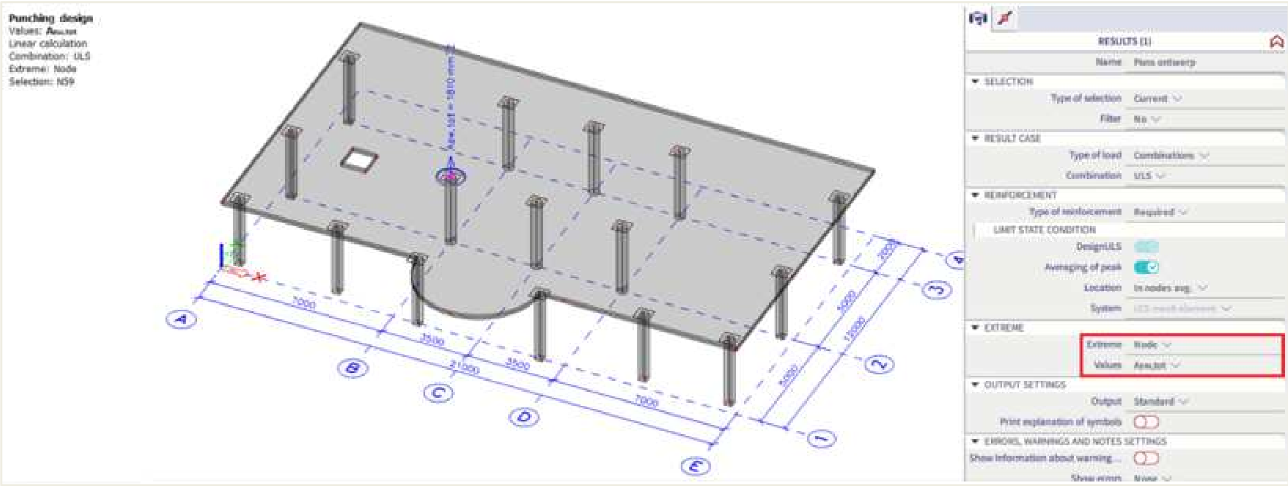
Name	Case	Shear reinforcement perimeters	u_{out} [m] a_{out} [mm]	$s_{t,u1}$ [mm] $s_{t,out}$ [mm]	Control perimeters (distance/capacity)	Material $f_{ywd,ef}$ [MPa]	$A_{sw,req}$ [mm²] $A_{sw1,min}$ [mm²]	A_{sw} [mm²] $A_{sw,tot}$ [mm²]	$v_{Rd,cs}$ [MPa] $k_{max} v_{Rd,c}$ [MPa]	UC$_{vRd,cs}$ [-] UC$_{Asw,det}$ [-]
N59	ULS/1	3x 12Ø8(radial) 80+2x80=240	3.472 354	230 230	320/71%	B 500B 290.0	103 11	603 **1810**	1.42 0.82	**0.71** **1.00**

We can see here that $V_{Ed,u1}$ = 0,58MPa < $V_{Rd,c}$= 0,55MPa and the $UC_{vRd,c}$ = 1,06 > 1.

So shear reinforcement needs to be designed. The final value is $A_{sw,tot}$ = 1810mm² which take into account detailing provisions.

The control parameter is displayed in blue colour.

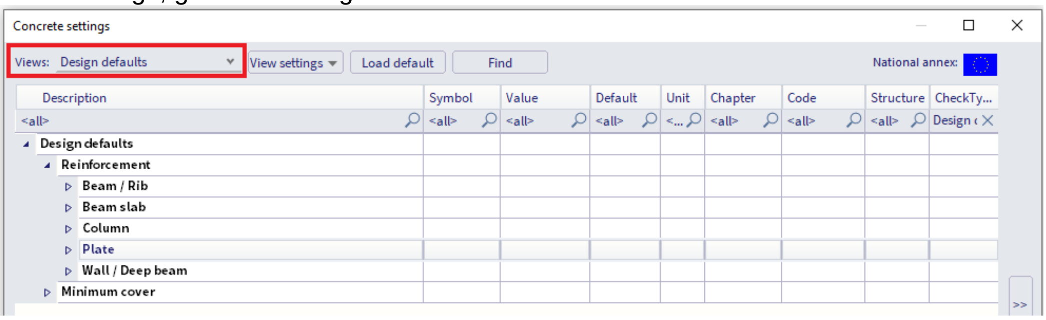
You can also show the $A_{sw,tot}$ graphically:



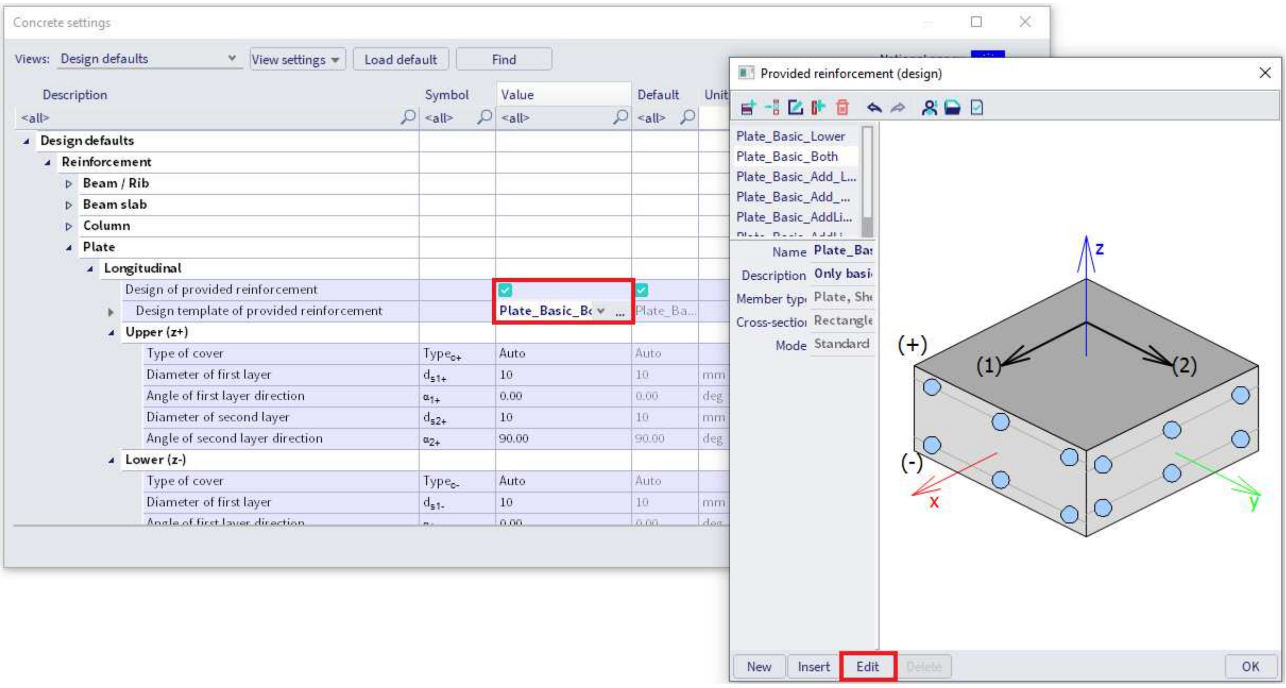
⇨ ***Use of provided reinforcement***

Let's add some provided reinforcement to the plate.

In the Concrete settings, go to the Design Defaults view:



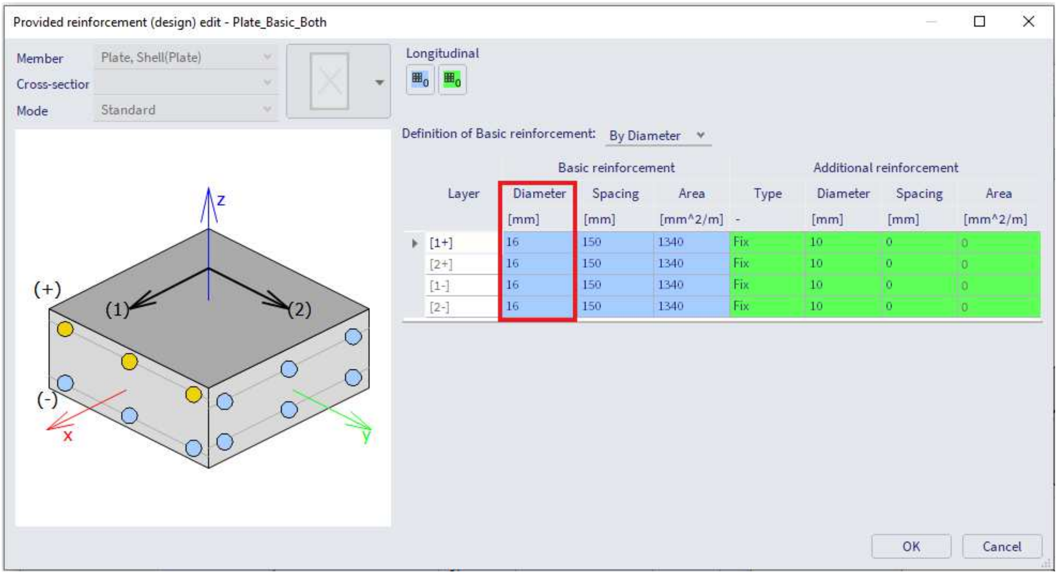
Activate the provided template for the plates:



Here you can choose between the different templates.

You can give a basic provided reinforcement without any additional reinforcement or allow SCIA Engineer to calculate additional reinforcement when needed.

For this example, we will define the basic reinforcement without additional reinforcement and we will use diameter 16 mm with a spacing of 150 mm.



Now look at the standard output for node N59. With the required reinforced we needed additional shear reinforcement but with the provided reinforcement set above no need for shear reinforcement:

Punching design

Linear calculation
Combination: ULS
Extreme: Node
Selection: N59

Summary

Name	Case	Punching case	Punching shape	UC $_{vRd,max}$ [-]	UC $_{vRd,c}$ [-]	Shear reinforcement perimeters	UC $_{vRd,cs}$ [-]	UC $_{Asw,det}$ [-]	UC [-] Check
N59	ULS/1	Internal column	Circle (400)	**0.36**	**0.82**	not required	-	-	0.82 OK

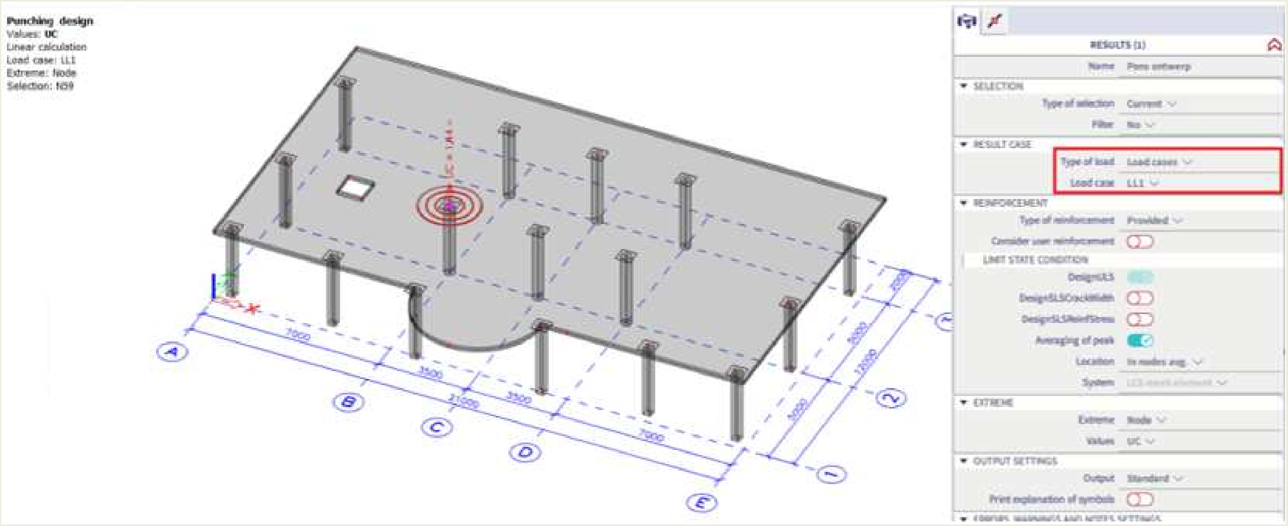
Concrete

Name	Case	Punching case β [-]	Punching shape	V_{Ed} [kN] ΔV_{Ed} [kN]	$M_{Ed,y}$ [kNm] $M_{Ed,z}$ [kNm]	Plate h [mm]	Material f_{cd} [MPa]	d_{eff} [mm] ρ_l [%]	u_0 [m] u_1 [m]	$v_{Ed,u0}$ [MPa] $v_{Ed,u1}$ [MPa]	$v_{Rd,max}$ [MPa] $v_{Rd,c}$ [MPa]	UC $_{vRd,max}$ [-] UC $_{vRd,c}$ [-]
N59	ULS/1	Internal column 1.15	Circle (400)	**265.21** 0.00	26.85 6.10	Ceiling 200.00	C30/37 20.00	160.00 0.84	1.257 3.267	1.52 0.58	4.22 0.71	**0.36** **0.82**

We can see that $V_{Ed,u1}$ = 0,58 MPa < $V_{Rd,c}$ = 0,71 MPa so the shear capacity without reinforcement is sufficient. The control parameter is now displayed in Green colour instead of blue.

⇨ ***Unity check is not ok: control perimeter is red***

Change the "Type of Result" to Load Case LL1 and display the result for node N59:



Control perimeter is now displayed in red and the UC = 1,44 > 1.

Take a look at the Standard Output:

Punching design

Linear calculation
Load case: LL1
Extreme: Node
Selection: N59

Summary

Name	Case	Punching case	Punching shape	$UC_{vRd,max}$ [-]	$UC_{vRd,c}$ [-]	Shear reinforcement perimeters	$UC_{vRd,cs}$ [-]	$UC_{Asw,det}$ [-]	UC [-] Check
N59	LL1	Internal column	Circle (400)	**0.96**	**2.17**	7x 19Ø8(radial) 80+6x110=740	**1.44**	**1.00**	1.44 NOT OK

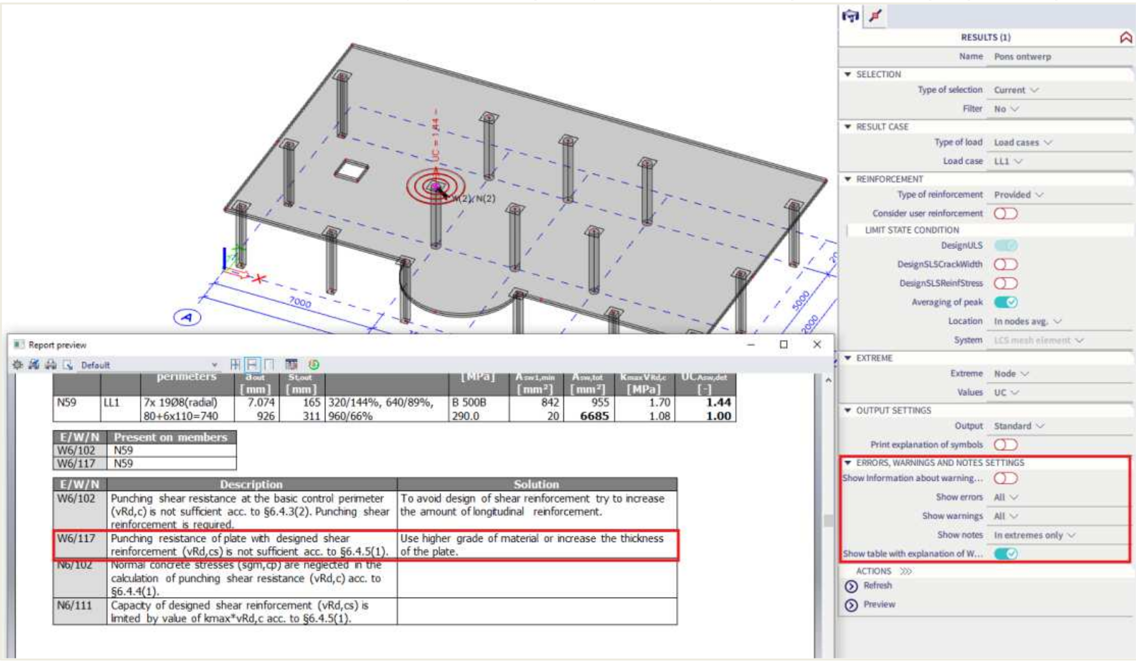
Concrete

Name	Case	Punching case β [-]	Punching shape	V_{Ed} [kN] ΔV_{Ed} [kN]	$M_{Ed,y}$ [kNm] $M_{Ed,z}$ [kNm]	Plate h [mm]	Material f_{cd} [MPa]	d_{eff} [mm] ρ_l [%]	u_0 [m] u_1 [m]	$v_{Ed,u0}$ [MPa] $v_{Ed,u1}$ [MPa]	$v_{Rd,max}$ [MPa] $v_{Rd,c}$ [MPa]	$UC_{vRd,max}$ [-] $UC_{vRd,c}$ [-]
N59	LL1	Internal column 1.15	Circle (400)	**708.92** 0.00	71.59 11.71	Ceiling 200.00	C30/37 20.00	160.00 0.84	1.257 3.267	4.05 1.56	4.22 0.72	**0.96** **2.17**

Reinforcement

Name	Case	Shear reinforcement perimeters	u_{out} [m] a_{out} [mm]	$s_{t,u1}$ [mm] $s_{t,out}$ [mm]	Control perimeters (distance/capacity)	Material $f_{ywd,ef}$ [MPa]	$A_{sw,req}$ [mm²] $A_{sw1,min}$ [mm²]	A_{sw} [mm²] $A_{sw,tot}$ [mm²]	$v_{Rd,cs}$ [MPa] $k_{max} v_{Rd,c}$ [MPa]	$UC_{vRd,cs}$ [-] $UC_{Asw,det}$ [-]
N59	LL1	7x 19Ø8(radial) 80+6x110=740	7.074 926	165 311	320/144%, 640/89%, 960/66%	B 500B 290.0	842 20	955 **6685**	1.70 1.08	**1.44** **1.00**

We can also show the errors and warning in the output by checking this option in the properties panel:



2.6 Code dependent deflection (CDD)

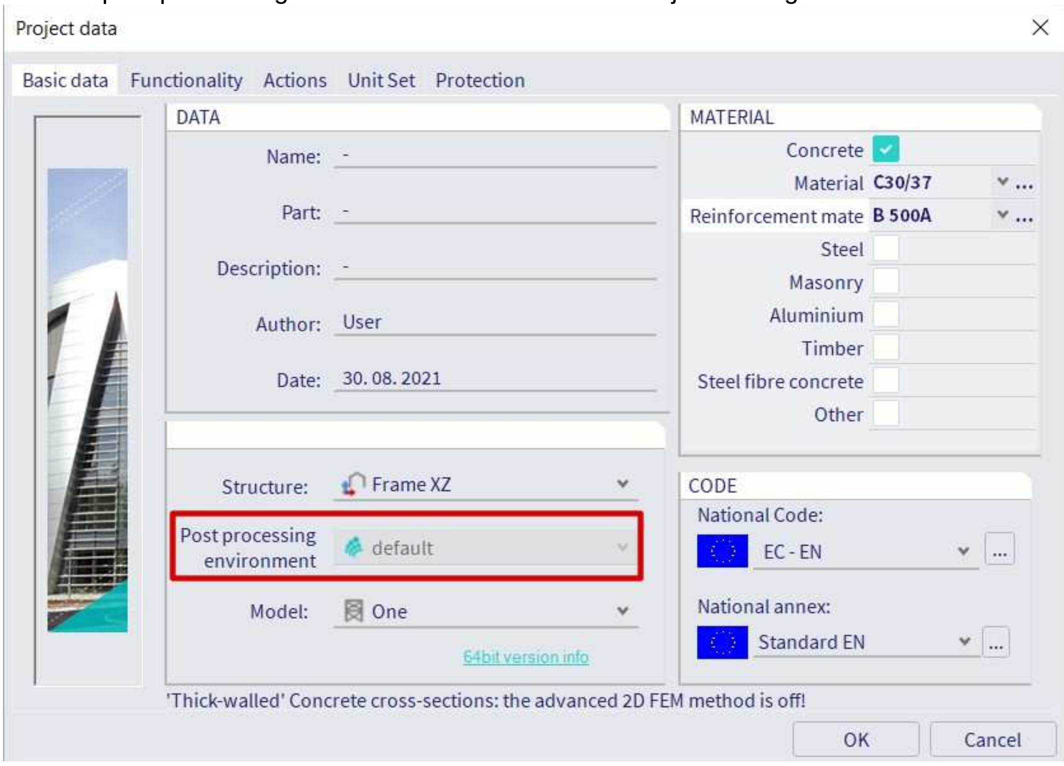
2.6.1 Intro

The CDD calculation is a more rigorous calculation of the deflection. The calculation procedure is the same as for the simplified method, but with following differences:

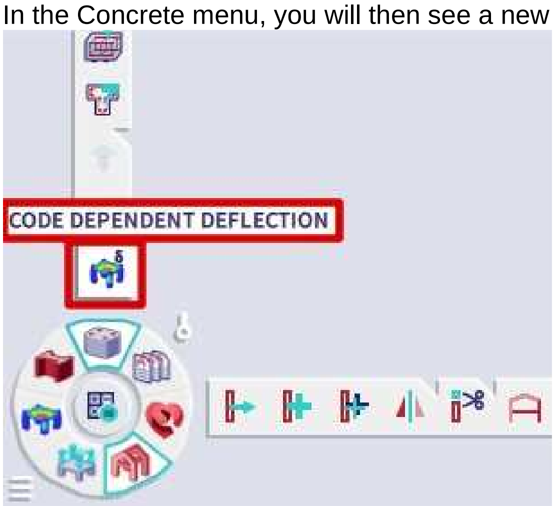
- 3 types of combinations are used to calculate the deflections
- Calculation of stiffness is more precise

To be able to use this method in SCIA Engineer, the following settings should be set beforehand:

1. Use the post processing environment '**default**' in the Project settings:



2. In the Concrete menu, you will then see a new check named Code dependent deflection:



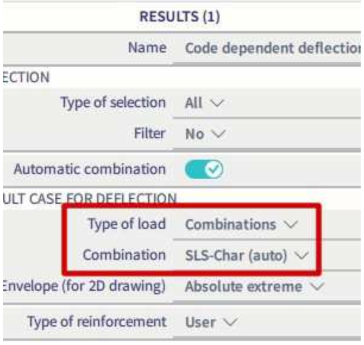
2.6.2 Types of combination for CDD

The combinations used for the CDD calculation can either be automatically generated or inserted by the user.

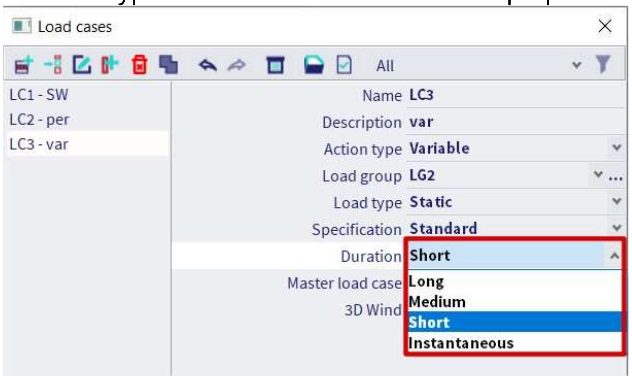
Automatic Creation of combinations for CDD

Three different combinations are automatically created by the software in the background to calculate the deflection:

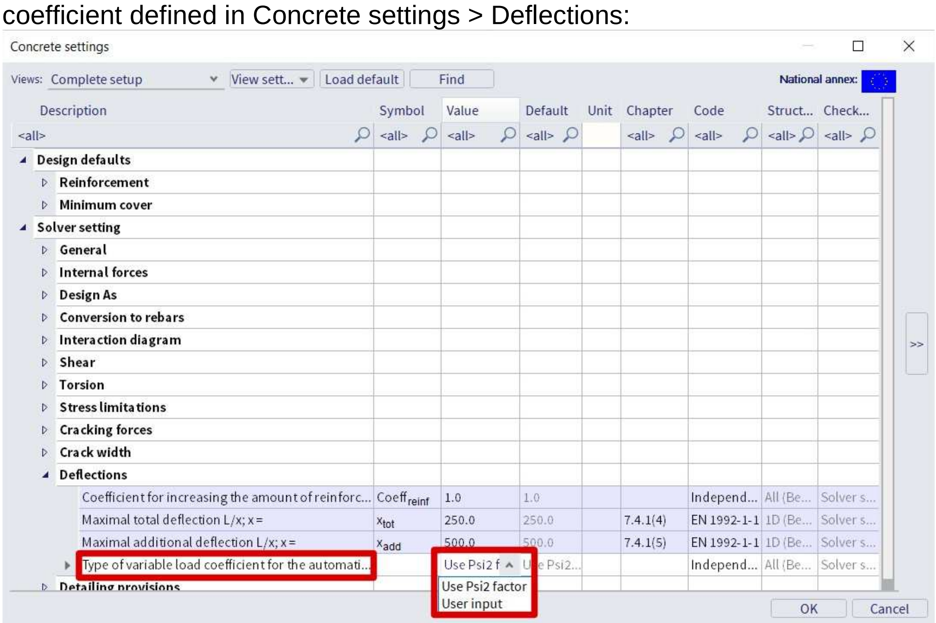
1. Combination for calculation of total deflection
   Generated directly from the user choice of combination in the CDD check, properties panel:



2. Combination for calculation of immediate deflection
   Uses the generated combination for total deflection and removes variable load cases with duration type Medium, Short or Instantaneous.
   Duration type is defined in the Load cases properties:



3. Combination for calculation of deflection due to creep
   Uses the generated combination for total deflection and multiplies variable load cases by a coefficient defined in Concrete settings > Deflections:



Additional characteristic combinations are generated for each previously mentioned combination to determine if the section is cracked or uncracked.

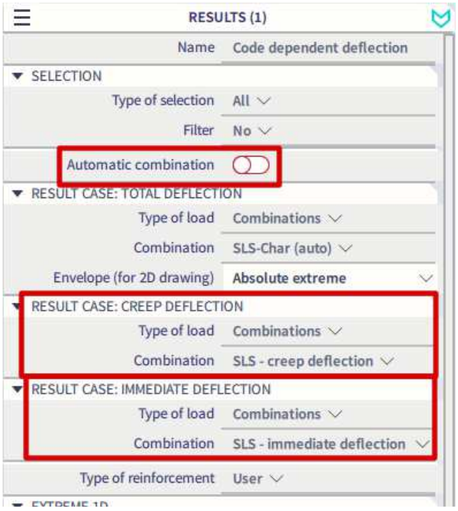
Manual input of combinations for CDD

It is possible for you to introduce your own combinations for calculation of immediate deflection and deflection due to creep.

In order to introduce these manual combinations, the option “Automatic combination” must be unchecked in the CDD check, properties panel.

Two new sections (“Result case: Creep deflection” and “Result case: Immediate deflection”) appear in the properties window where you can choose the combinations for creep and immediate deflections.

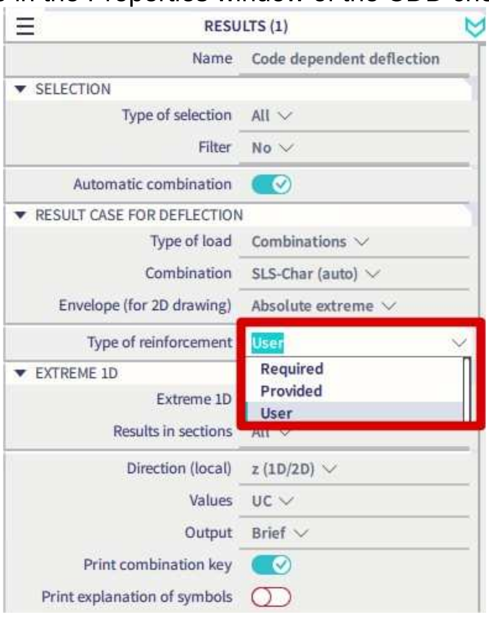
These combinations have to be linear combinations, it means that creep and immediate deflection will be the same for all sub-combinations generated from combination for total deflection.



The combination for the calculation of the total deflection remains generated directly from your choice of combination in the CDD check in the properties panel.

2.6.3 Type of reinforcement

For the CDD method, it is possible to calculate the deflection with required, provided or user inputted reinforcement. This choice is done in the Properties window of the CDD check:



2.6.4 Calculation of stiffness for 1D elements

Members which are not expected to be loaded above the level which would cause the tensile strength of the concrete to be exceeded anywhere within the member should be considered to be uncracked. Members which are expected to crack, but may not be fully cracked, will behave in a manner intermediate between the uncracked and fully cracked conditions. New stiffness (stiffness with taking into account cracking) is calculated in center of each 1D element.

Two types of stiffness are calculated:

- **Short-term stiffness** - is calculated using 28 days modulus of elasticity $E_c = E_{cm}$, it follows that value of stiffness is loaded directly from properties of the concrete material
- **Long-term stiffness** - is calculated using effective E modulus based on creep coefficient for acting load, it follows $E_c = E_{c,eff} = E_{cm}/(1+\varphi)$.

Calculation effective modulus of elasticity is based on equation 5.27 in EN 1992-1-1, but instead of effective creep coefficient φ_{ef}, only creep coefficient φ is used

The following procedure is used for the calculation of stiffnesses:

1) The transformed cross-section characteristics of uncracked section (Ai, Ii, ti...) are calculated
2) The stiffnesses of the uncracked cross-section ($(Eiy)_I$,$(Eiz)_I$, $(EA)_I$) to the center of the uncracked transformed cross-section are calculated.
3) The maximum value of tensile stress of the uncracked cross-section (value $\sigma_{ct,res}$) for respective characteristic combination ($N_{char,res}$,$M_{char,res,y}$, $M_{char,res,z}$) is calculated
4) The maximum value of tensile stress of uncracked cross-section (value $\sigma_{ct,imm}$) for immediate characteristic combination ($N_{char,im}$,$M_{char,im,y}$, $M_{char,im,z}$) is calculated
5) Compare σ_{ct} with $\sigma_{ct,imm}$
   - If $\sigma_{ct} \geq \sigma_{ct,imm}$, the respective characteristic combination will be used for calculation, $N_{char}=N_{char,res}$,$M_{char,y}=M_{char,res,y}$, $M_{char,z}=M_{char,res,z}$,$\sigma_{ct}=\sigma_{ct,res}$
   - If $\sigma_{ct} \leq \sigma_{ct,imm}$, the immediate characteristic combination will be used $N_{char}=N_{char,im}$,$M_{char,y}=M_{char,im,y}$, $M_{char,z}=M_{charim,z}$,$\sigma_{ct}=\sigma_{ct,im}$
6) Compare σ_{ct} with σ_{cr}
   - If $\sigma_{ct} \leq \sigma_{cr}$, the cross-section is uncracked:
     - o bending stiffness around y-axis $EI_y = (Eiy)_I$
     - o bending stiffness around z axis $EI_z = (Eiy)_I$
     - o axial stiffness $EA = EA_I$,
   - If $\sigma_{ct} \geq \sigma_{cr}$, the cross-section is cracked and average stiffness will be calculated.
     1. The transformed Css characteristics of the cracked section (Air, Iir, tir...) is calculated.
     2. The stiffnesses of the fully cracked cross-section ($(Eiy)_{II}$,$(Eiz)_{II}$, $(EA)_{II}$) to center of cracked transformed cross-section is calculated
     3. The stress in the tensile reinforcement of the fully cracked cross-section (value σ_{sr}) for characteristic combination (N_{char},M_{char}, $M_{char,z}$) is calculated.
     4. The stress in the tensile reinforcement of the fully cracked cross-section (value σ_s)for respective combination (N,M_y,M_z) is calculated.
     5. The distribution coefficient ζ according equation 7.19 in EN 1992-1-1 is calculated
        $$\zeta = 1 - \beta\left(\frac{\sigma_{sr}}{\sigma_s}\right)^2$$
        where β is a coefficient taking account the influence of the duration of the loading or of repeated loading on the average strain (β=1 for calculation of short-term stiffness, β=0,5 for calculation of long-term stiffness)
     6. The average value of the stiffnesses based on equation 7.18 in EN 1992-1-1 is calculated
        - o bending stiffness around y-axis $(Eiy) = 1/[\zeta/(Eiy)_{II} + (1-\zeta)/(Eiy)_I]$
        - o bending stiffness around z-axis $(Eiz) = 1/[\zeta/(Eiz)_{II} + (1-\zeta)/(Eiz)_I]$
        - o axial stiffness $(EA) = 1/[(\zeta/(EA)_{II} + (1-\zeta)/(EA)_I]$,

Stiffness is recalculated to principal axis for unsymmetrical cross-section

7) The five types of stiffnesses are calculated for each 1D element and each dangerous combination:

Type of stiffness	Respective combination
Short-term stiffness for immediate deflection	Immediate
Short-term stiffness for short-term deflection	Total
Short-term stiffness for creep deflection	Creep
Long-term stiffness for creep deflection	Creep
Long-term stiffness for shrinkage deflection	Total

8) The following stiffnesses are changes in stiffness matrix for 1D elements:

- $EA_x = EA$
- $GA_y = GA_z = G \cdot EA_x/(1,2 \cdot E_c)$
- $EI_y = Eiy$
- $EI_z = Eiz$
- $GI_x = 0,5 \cdot (1-\mu) \cdot (EI_y)EI_z)^{0.5}$

Where:

- G is shear modulus of the concrete calculated according to formula G = 0.5×Ec/(1+m)
- m is Poisson coefficient of the concrete loaded from material properties of the concrete

Eccentricity of stiffnesses (distance between center of gravity of concrete cross-section and center of gravity of cracked transformed cross-section) is not taken into account in current version

Calculation of curvature, strain and stiffness caused by shrinkage of a 1D element

Calculation of shrinkage forces

The forces caused by shrinkage are calculated according to formulas below. The forces are calculated for both states: uncracked and cracked cross-section.

- $N_{shr} = -\varepsilon_{cs}(t,t_s) \cdot Coef_{Reinf} \sum(E_{si} \cdot A_{si})$
- $M_{shr,y} = N_{shr} \cdot e_{shr,z}$
- $M_{shr,z} = N_{shr} \cdot e_{shr,y}$

Where :

- $e_{shr,y} = \sum(E_{si} \cdot A_{si}) / \sum(E_{si} \cdot A_{si} \cdot y_{si}) - t_{iy}$
- $e_{shr,z} = \sum(E_{si} \cdot A_{si}) / \sum(E_{si} \cdot A_{si} \cdot z_{si}) - t_{iz}$
- $\varepsilon_{cs}(t,t_s) \cdot$ - total shrinkage strain
- $Coef_{reinf}$ - coefficient increasing amount of reinforcement
- E_{si} - modulus of elasticity of i-th bar of reinforcement
- A_{si} - area of reinforcement of i-th bar of reinforcement
- y_{si} - position of i-th bar of reinforcement from center of gravity of cross-section in y-direction
- z_{si} - position of i-th bar of reinforcement from center of gravity of cross-section in z-direction
- t_{iy} - distance between center of gravity of transformed uncracked/cracked cross-section and center of gravity of concrete cross-section in y-direction
- t_{iz} - distance between center of gravity of transformed uncracked/cracked cross-section and center of gravity of concrete cross-section in z-direction

Shrinkage deflection (long-term stiffness)

	N [kN]	M_y [kNm]	M_z [kNm]
Combination: CO2/1_tot	0.00	159.75	0.00
Characteristic combination (char): CO2/1_tot_char	0.00	159.75	0.00

Forces caused by shrinkage: N_{shr} = 140.74 kN, $M_{shr,y}$ = 24.56 kNm, $M_{shr,z}$ = 0.00 kNm

Cross-section characteristics

Type of component	t_y [mm]	t_z [mm]	A [mm^2]	I_y [mm^4]	I_z [mm^4]	x_i [mm]	A_s [mm^2]
Linear	0.0	0.0	320000	$9.02 \cdot 10^9$	$17.6 \cdot 10^9$	218.0	-
Uncracked	0.0	-19.7	356135	$12.1 \cdot 10^9$	$19.7 \cdot 10^9$	232.2	1407
Cracked	0.0	73.4	175211	$5.31 \cdot 10^9$	$13.7 \cdot 10^9$	139.1	1407

Cracking forces

N_{cr} [kN]	$M_{y,cr}$ [kNm]	$M_{z,cr}$ [kNm]	σ_{ct} [MPa]	σ_{cr} [MPa]	Cracked section	σ_{sr} [MPa]	σ_s [MPa]	β [-]	ζ [-]	E_c [GPa]
0.00	73.03	0.00	4.87	2.20	YES	144.6	316.4	0.5	0.896	30.0

Stiffness calculation

Axial stiffness EA: EA_I = 9600.00 MN EA_{II} = 9600.00 MN

$$EA = \frac{1}{\frac{\zeta}{EA_{II}} + \frac{1-\zeta}{EA_I}} = \frac{1}{\frac{0.896}{9600.00} + \frac{1-0.896}{9600.00}} = 9600.00\ \text{MN} \tag{7.18}$$

Bending stiffness EI_y: $EI_{y,I}$ = 672.92 MN·m ^{2} $EI_{y,II}$ = 193.17 MN·m ^{2}

$$EI_y = \frac{1}{\frac{\zeta}{EI_{y,II}} + \frac{1-\zeta}{EI_{y,I}}} = \frac{1}{\frac{0.896}{193.17} + \frac{1-0.896}{672.92}} = 208.71\ \text{MN·m}^2 \tag{7.18}$$

Bending stiffness EI_y: $EI_{z,I}$ = 527.00 MN·m ^{2} $EI_{z,II}$ = 527.00 MN·m ^{2}

$$EI_z = \frac{1}{\ldots} = \frac{1}{\ldots} = 527.00\ \text{MN·m}^2 \tag{7.18}$$

Calculation of strain and curvature caused by shrinkage

Strain and curvature caused by shrinkage are calculated for each 1D element and these values are calculated for both states (uncracked and cracked cross-section).

Calculation of strain caused by shrinkage:

- $\varepsilon_x = -\varepsilon_{cs}(t,t_s) \cdot Coef_{Reinf} \cdot \sum(E_{si} \cdot A_{si})/(E_{ceff} \cdot A_i)$

Calculation of curvature around y and z axis caused by shrinkage

- $(1/r_y) = -\varepsilon_{cs}(t,t_s) \cdot Coef_{Reinf} \cdot \sum(E_{si} \cdot A_{si} \cdot (t_{iz}-z_{si}))/(E_{ceff} \cdot I_{iy})$
- $(1/r_z) = -\varepsilon_{cs}(t,t_s) \cdot Coef_{Reinf} \cdot \sum(E_{si} \cdot A_{si} \cdot (t_{iy}-y_{si}))/(E_{ceff} \cdot I_{iz})$

Where:

- $\varepsilon_{cs}(t,t_s) \cdot$ - total shrinkage strain
- $Coef_{reinf}$ - coefficient increasing amount of reinforcement
- E_{si} - is modulus of elasticity of i-th bar of reinforcement
- A_{si} - is area of reinforcement of i-th bar of reinforcement
- y_{si} - position of i-th bar of reinforcement from center of gravity of cross-section in y-direction
- z_{si} - position of i-th bar of reinforcement from center of gravity of cross-section in z-direction
- t_{iy} - distance between the center of gravity of transformed uncracked/cracked cross-section and center of gravity of concrete cross-section in y-direction
- t_{iz} - distance between the center of gravity of transformed uncracked/cracked cross-section and center of gravity of concrete cross-section in z-direction
- E_{ceff} - effective modulus of elasticity of the concrete calculated according to formula $E_c = E_{c,eff} = E_{cm}/(1+\varphi)$.
- E_{cm} - secant modulus of elasticity of concrete
- φ - creep coefficient
- A_i - transformed area of uncracked/cracked cross-section
- I_{iy} - transformed second moment of area around y-axis of uncracked/cracked cross-section calculated to center of gravity of transformed uncracked/cracked cross-section
- I_{iz} - transformed second moment of area around z axis of uncracked/cracked cross-section calculated to center of gravity of transformed uncracked/cracked cross-section

Calculation of stiffnesses for shrinkage

The stiffness of uncracked/cracked cross-section for shrinkage is calculated from strain and curvatures caused by shrinkage by using total level of load (total load combination)

- bending stiffness around y-axis $EI_y = M_{tot,y}/(1/r_y)$
- bending stiffness around z axis $EI_z = M_{tot,z}/(1/r_z)$
- axial stiffness $EA = N_{tot}/\varepsilon_x$

2.6.5 Calculation of stiffness for 2D elements

The following procedure is used for the calculation of stiffness of 2D elements:

1) The principal stresses of 2D element for both surfaces is calculated

$$\sigma_{1\mp} = \frac{\sigma_{x\mp} + \sigma_{y\mp}}{2} + \frac{1}{2}\sqrt{\left(\sigma_{x\mp} - \sigma_{y\mp}\right)^2 + 4 \cdot \sigma_{xy,\mp}}$$

$$\sigma_{2\mp} = \frac{\sigma_{x\mp} + \sigma_{y\mp}}{2} - \frac{1}{2}\sqrt{\left(\sigma_{x\mp} - \sigma_{y\mp}\right)^2 + 4 \cdot \sigma_{xy,\mp}}$$

2) The angle of principal stresses at both surfaces is calculated

$$\alpha_{\sigma 1\mp} = 0{,}5 \cdot \tan^{-1}\left(\frac{2 \cdot \sigma_{xy\mp}}{\sigma_{x\mp} - \sigma_{y\mp}}\right)$$

3) The final value of the principal stress is calculated

   $\alpha=\alpha_{\sigma 1+}$ if $\sigma_{1+} \geq \sigma_{1-}$

   $\alpha=\alpha_{\sigma 1-}$ otherwise

4) The internal forces are recalculated to the direction of the principal stresses α

$$m(\alpha) = m_x \cdot \cos^2(\alpha) + m_y \cdot \sin^2(\alpha) + m_{xy} \cdot \sin(2 \cdot \alpha)$$

$$n(\alpha) = n_x \cdot \cos^2(\alpha) + n_y \cdot \sin^2(\alpha) + n_{xy} \cdot \sin(2 \cdot \alpha)$$

   where n_x,n_y,n_{xy},m_x,m_y,m_{xy} are 2D forces in center of 2D element

5) The area of reinforcement is recalculated to the direction of of the principal stress α

   $A_s(\alpha) = A_s \cdot \cos^2(\alpha-\alpha_s)$

   where A_s,α_s is area and angle of longitudinal reinforcement

6) The non-linear stiffness in the first principal direction is calculated according to the procedure as for 1D element
   - for rectangular cross-section (b =1m, h = thickness of 2D element in center of gravity)
   - for internal forces N = $n(\alpha)$, M_y= $m(\alpha)$ and M_z=0 according procedure as for 1D element

7) The non-linear stiffness in the second principal direction is calculated according to the procedure as for 1D element
   - for rectangular cross-section (b =1m, h = thickness of 2D element in center of gravity)
   - for internal forces N = $n(\alpha+90)$, M_y= $m(\alpha+90)$ and M_z=0 according procedure as for 1D element

8) The stiffness for shrinkage deflection is calculated in both directions of principal axes as explained in the next section.

9) The five types of stiffnesses are calculated for each 2D element and each dangerous combination:

Type of stiffness	Respective combination	Direction of principal stress
Short-term stiffness for immediate deflection	Immediate	First (EA_1,EIy_1,EIz_1)
		Second (EA_2,EIy_2,EIz_2)
Short-term stiffness for short-term deflection	Total	First (EA_1,EIy_1,EIz_1)
		Second (EA_2,EIy_2,EIz_2)
Short-term stiffness for creep deflection	Creep	First (EA_1,EIy_1,EIz_1)
		Second (EA_2,EIy_2,EIz_2)
Long-term stiffness for creep deflection	Creep	First (EA_1,EIy_1,EIz_1)
		Second (EA_2,EIy_2,EIz_2)
Long-term stiffness for shrinkage deflection	Total	First (EA_1,EIy_1,EIz_1)
		Second (EA_2,EIy_2,EIz_2)

10) The following stiffnesses are changes in stiffness matrix for 2D elements:

- $D11 = EIy_1$
- $D22 = EIy_2$
- $D33 = 0.5 \cdot (1-\mu) \cdot (D11 \cdot D22)^{0.5}$
- $D44 = G \cdot h/1.2$
- $D55 = G \cdot h/1.2$
- $D12 = \mu \cdot (D11 \cdot D22)^{0.5}$
- $d11 = EA_1$
- $d22 = EA_2$
- $d33 = G \cdot h$
- $d12 = \mu \cdot (d11 \cdot d22)^{0.5}$

Where:

- G is shear modulus of the concrete calculated according to formula $G = 0.5 \cdot E_c/(1+\mu)$
- μ is Poisson coefficient of the concrete loaded from material properties of the concrete

Eccentricity of stiffnesses (distance between center of gravity of concrete cross-section and center of gravity of cracked transformed cross-section) is not taken into account in current version

Calculation of curvature, strain and stiffness caused by shrinkage of a 2D element

Calculation of shrinkage forces

The forces are calculated in the center of gravity of each element and they are calculated in two directions:

- The first one is the direction of principal stress α
- The second one is the direction of principal stress α+90°

The forces caused by shrinkage for first/second direction are calculated according to formulas below. The forces are calculated for both states: uncracked and cracked cross-section.

- $n_{shr} = -\varepsilon_{cs}(t,t_s) \cdot Coef_{Reinf} \sum(E_{si} \cdot A_{si(\alpha)})$
- $m_{shr} = n_{shr} \cdot e_{shr,z}$

Where:

- $e_{shr,z} = \sum(E_{si} \cdot A_{si(\alpha)}) / \sum(E_{si} \cdot A_{si(\alpha)} \cdot z_{si}) - t_{iz(\alpha)}$
- $\varepsilon_{cs}(t,t_s)$ - total shrinkage strain
- $Coef_{reinf}$ - coefficient increasing amount of reinforcement
- E_{si} - is modulus of elasticity of i-th bar of reinforcement
- $A_{si(\alpha)}$ - is area of reinforcement of i-th bar of reinforcement in first (angle α)/second direction (angle α+90°) of principal stress
- z_{si} - position of i-th bar of reinforcement from center of gravity of cross-section in z-direction
- $t_{iz(\alpha)}$ - distance between center of gravity of transformed uncracked/cracked cross-section and centre of gravity of concrete cross-section in z-direction and in first (angle α)/second direction (angle α+90°) of principal stress

Calculation of strain and curvature caused by shrinkage

Strain and curvature caused by shrinkage are calculated for each 2D elements and these values are calculated for both states (uncracked and cracked cross-section). The values are calculated in both directions of principal stresses.

Calculation of strain caused by shrinkage:

- $\varepsilon_x = -\varepsilon_{cs(t,ts)} \cdot CoefReinf \cdot \sum(E_{si} \cdot A_{si(\alpha)})/(E_{ceff} \cdot A_{i(\alpha)})$

Calculation of curvature around y and z axis caused by shrinkage:

- $(1/r) = -\varepsilon_{cs}(t,t_s) \cdot Coef_{Reinf} \cdot \sum(E_{si} \cdot A_{si(\alpha)} \cdot (t_{iz(\alpha)} - z_{si}))/(E_{ceff} \cdot I_{iy(\alpha)})$

Where:

- $\varepsilon_{cs}(t,t_s)$ - total shrinkage strain
- $Coef_{reinf}$ - coefficient increasing amount of reinforcement
- E_{si} - is modulus of elasticity of i-th bar of reinforcement
- $A_{si(\alpha)}$ - is area of reinforcement of i-th bar of reinforcement in first (angle α)/second direction (angle α+90°) of principal stress
- z_{si} - position of i-th bar of reinforcement from center of gravity of cross-section in z-direction
- $t_{iz(\alpha)}$ - distance between centre of gravity of transformed uncracked/cracked cross-section and centre of gravity of concrete cross-section in z-direction and in first (angle α)/second direction (angle α+90°)of principal stress
- E_{ceff} - effective modulus of elasticity of the concrete calculated according formula $E_c = E_{c,eff} = E_{cm}/(1+\varphi)$.
- E_{cm} - secant modulus of elasticity of concrete
- φ - creep coefficient
- $A_{i(\alpha)}$ - transformed area of uncracked/cracked cross-section in the first (angle α)/second direction (angle α+90°) of principal stress
- $I_{iy(\alpha)}$ - transformed second moment of area around y axis of uncracked/cracked cross-section calculated to centre of gravity transformed uncracked/cracked cross-section in the first (angle α)/second direction (angle α+90°)of principal stress

Calculation of stiffnesses for shrinkage

The stiffness of uncracked/cracked cross-section for shrinkage is calculated from strain and curvatures caused by shrinkage by using total level of load (total load combination)

- bending stiffness in direction of first principal axis $EIy_1 = m_{tot(\alpha)}/(1/r)_1$
- bending stiffness in direction of second principal axsi $EIy_2 = m_{tot(\alpha+90)}/(1/r)_2$
- axial stiffness in direction of first principal axis $EA_1 = n_{tot(\alpha)}/\varepsilon_{x,1}$
- axial stiffness in direction of second principal axis $EA_2 = n_{tot(\alpha+90)}/\varepsilon_{x,2}$

Where:

- $n_{tot(\alpha)}$, $n_{tot(\alpha+90}$ - are axial forces from total combination in 2D element recalculated to direction of first and second principal axis
- $m_{tot(\alpha)}$, $m_{tot(\alpha+90}$ - are bending moments from total combination in 2D element recalculated to direction of first and second principal axis
- $\varepsilon_{x,1(2)}$ - is strain caused by shrinkage calculated in direction of first (second) principal axis
- $(1/r)_{1(2)}$ - is curvature caused by shrinkage calculated in direction of first (second) principal axis

Deflection for shrinkage is calculated in FEM analysis for total combination, therefore the stiffness are calculated with using internal forces for total combination

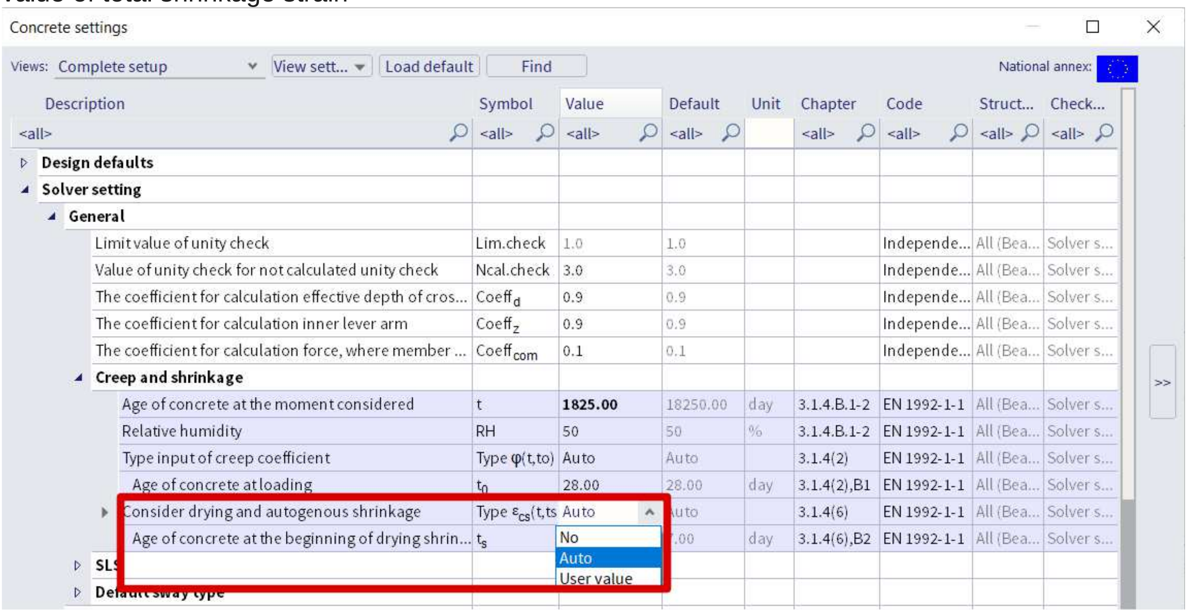
2.6.6 Parameters for the calculation of shrinkage strain

The total shrinkage strain is composed of two components, the drying shrinkage strain and the autogenous shrinkage strain. The drying shrinkage strain develops slowly, since it is a function of the migration of the water through the hardened concrete. The autogenous shrinkage strain develops during hardening of the concrete.

There are three options for calculation/input of total shrinkage strain that can be selected in the concrete settings menu:

- No **(Consider drying and autogenous shrinkage= No):** shrinkage will not be taken into account in CDD calculation
- Automatic calculation **(Consider drying and autogenous shrinkage = Auto)**, where shrinkage strain is calculated according to EN 1992-1-1, chapter 3.1.4(6) for following input parameters:
  - o Relative humidity
  - o Age of concrete at beginning of drying shrinkage
  - o Age of concrete at moment considered

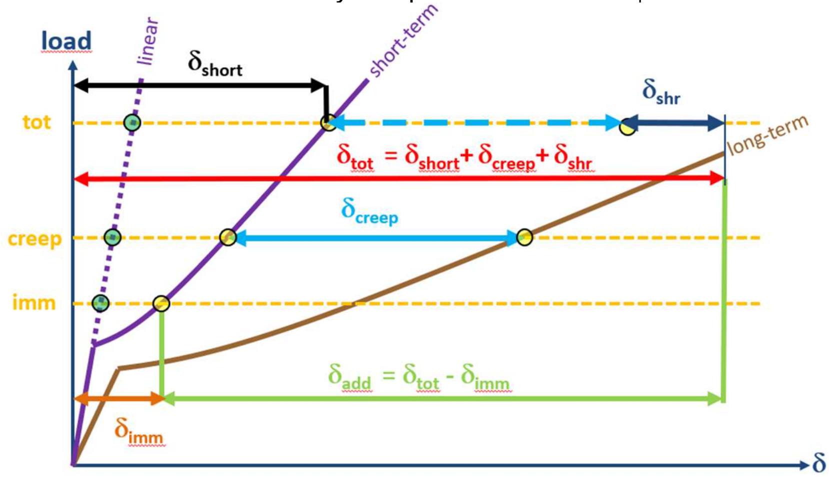
  Except of these input parameters, automatic calculation of shrinkage strain depends on material properties (mean compressive strength of concrete f_{cm}, characteristic compressive cylinder strength f_{ck}, type of cement), cross-section parameters (cross-sectional area A_c and the perimeter of the member in contact with the atmosphere u)
- User input **(Consider drying and autogenous shrinkage = User value)** and you can input directly value of total shrinkage strain



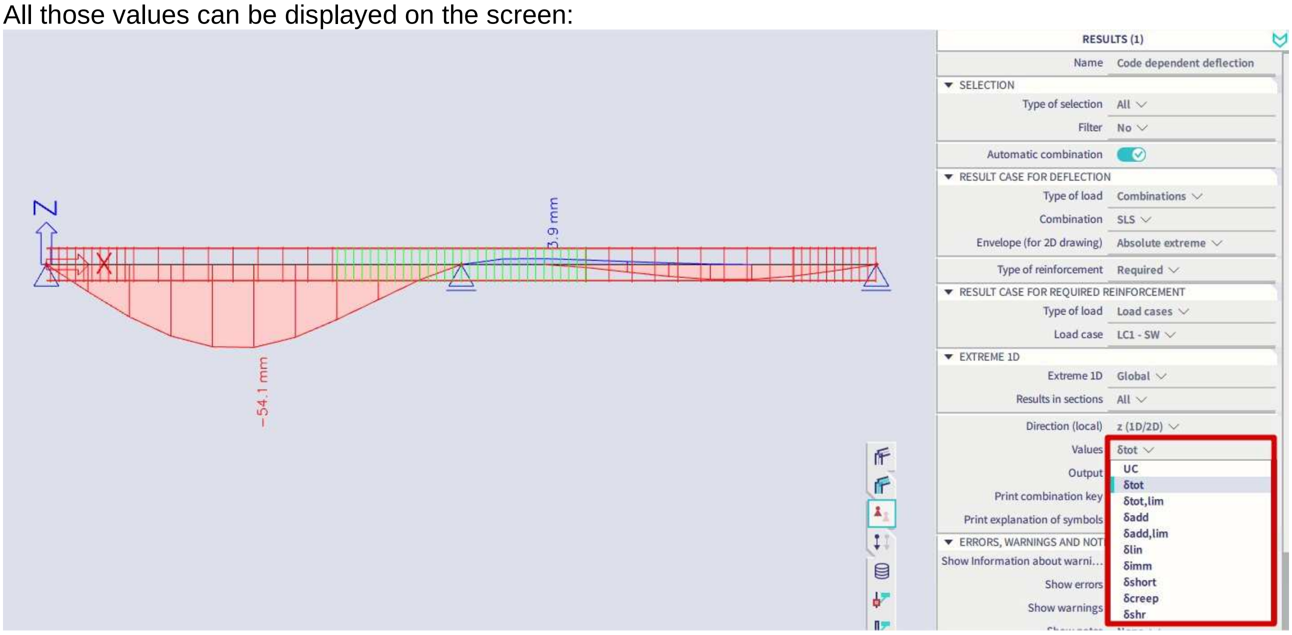
2.6.7 Calculation of deflection

The following deflections are calculated in the CDD check:

- δ_{lin} linear (elastic) deflection, calculated for the total combination and for linear stiffness.
- Δ_{imm} immediate deflection, the deflection after applying permanent and long-term variable loads which means calculated for short-term stiffness and immediate combination
- δ_{short} short-term deflection, the deflection which considers cracking of cross-section calculated for short-term stiffness and total combination
- δ_{creep} creep deflection, calculated as the difference between deflection calculated for long-term and short-term stiffness for the creep combination. $\Delta_{creep} = \delta_{creep,\, long} - \delta_{creep,\, short}$
- δ_{shr} deflection caused by drying and autogenous shrinkage. The long-term stiffness is calculated from strain and curvature caused by shrinkage using total combination.
- δ_{add} additional deflection, the deflection after applying a variable load and considering creep calculated as the difference between total and immediate deflection. $\Delta_{add} = \delta_{tot} - \delta_{imm}$
- δ_{tot} total deflection, the deflection which considers creep and cracking calculated as the sum of short-term deflection and deflection caused by creep. $\Delta_{tot} = \delta_{short} + \delta_{creep}$



All those values can be displayed on the screen:



Chapter 3: Modification of results

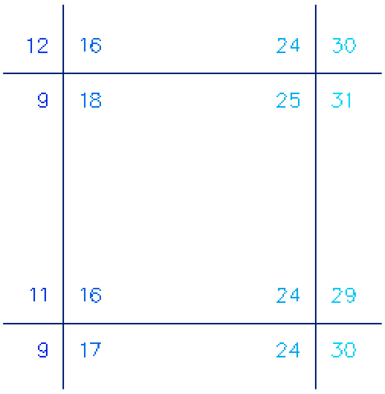
3.1 Location

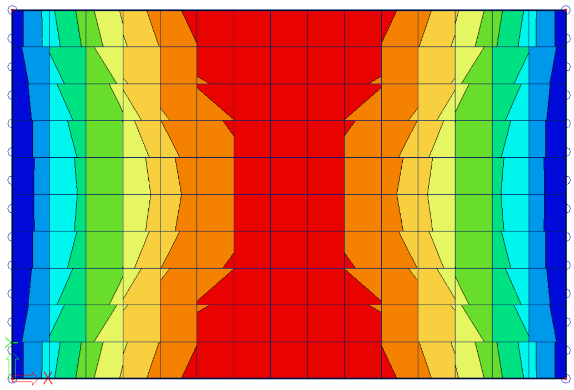
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions and are therefore in the Finite Elements Method always less accurate.

The Finite Elements Mesh in SCIA Engineer exists of linear 3- and/or 4-angular elements. Per mesh element 3 or 4 results are calculated, one in each node. When asking the results on 2D members, the option ‘Location’ in the Properties window gives the possibility to display these results in 4 ways.

3.1.1 In nodes, no average

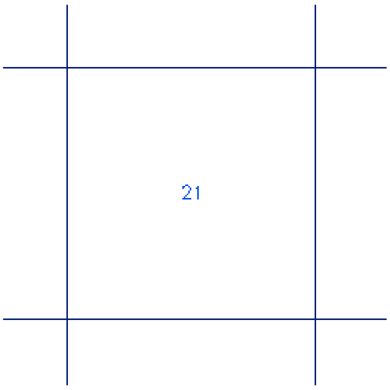
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.
This display of results therefore gives a good idea of the discretisation error in the calculation model.

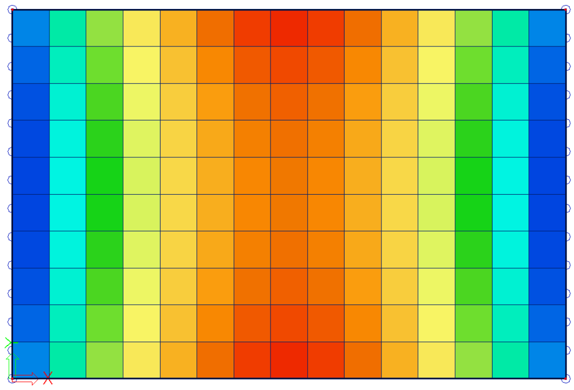




3.1.2 In centers

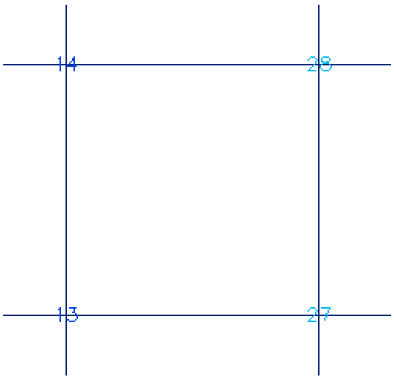
Per finite element, the mean value of the results in the nodes of that element is calculated. Since there is only 1 result per element, the display of isobands becomes a mosaic. The course over a section is a curve with a constant step per mesh element.

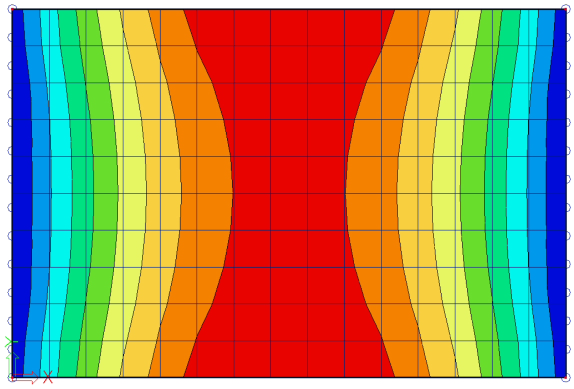




3.1.3 In nodes, average

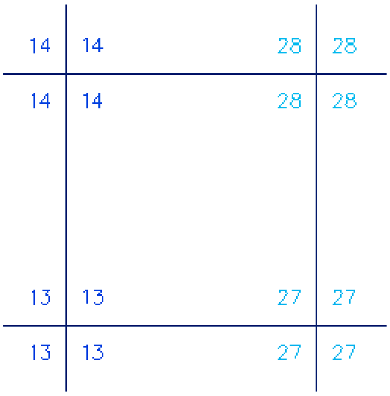
The values of the results of adjacent finite elements are averaged in the common node. Because of this, the graphical display is a smooth course of isobands.
In certain cases, it is not permissible to average the values of the results in the common node:
- At the transition between 2D members (plates, walls, shells) with different local axes.
- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.

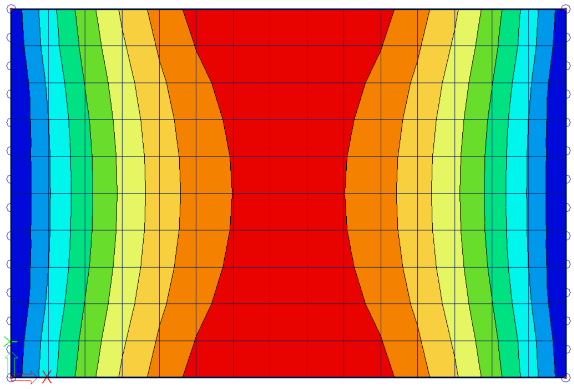




3.1.4 In nodes, average on macro

The values of the results are averaged per node *only* over mesh elements which belong to the same 2D member and which have the same directions of their local axes. This resolves the problems mentioned at the option 'In nodes, average'.



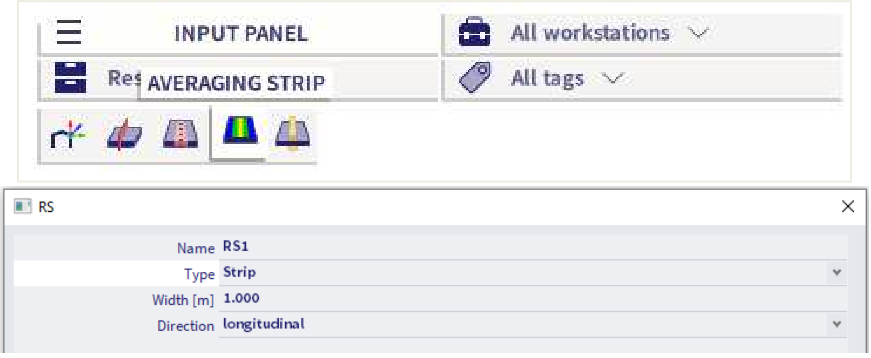


3.1.5 Accuracy of the results

If the results according to the 4 locations differ a lot, then the results are inaccurate and the mesh has to be refined. A basic rule for a good size of the mesh elements, is to take 1 to 2 times the thickness of the plate.

3.2 Averaging strip

An averaging strip averages peak values over a zone. You can find the averaging strip in the Input Panel in the "Result tools" category :

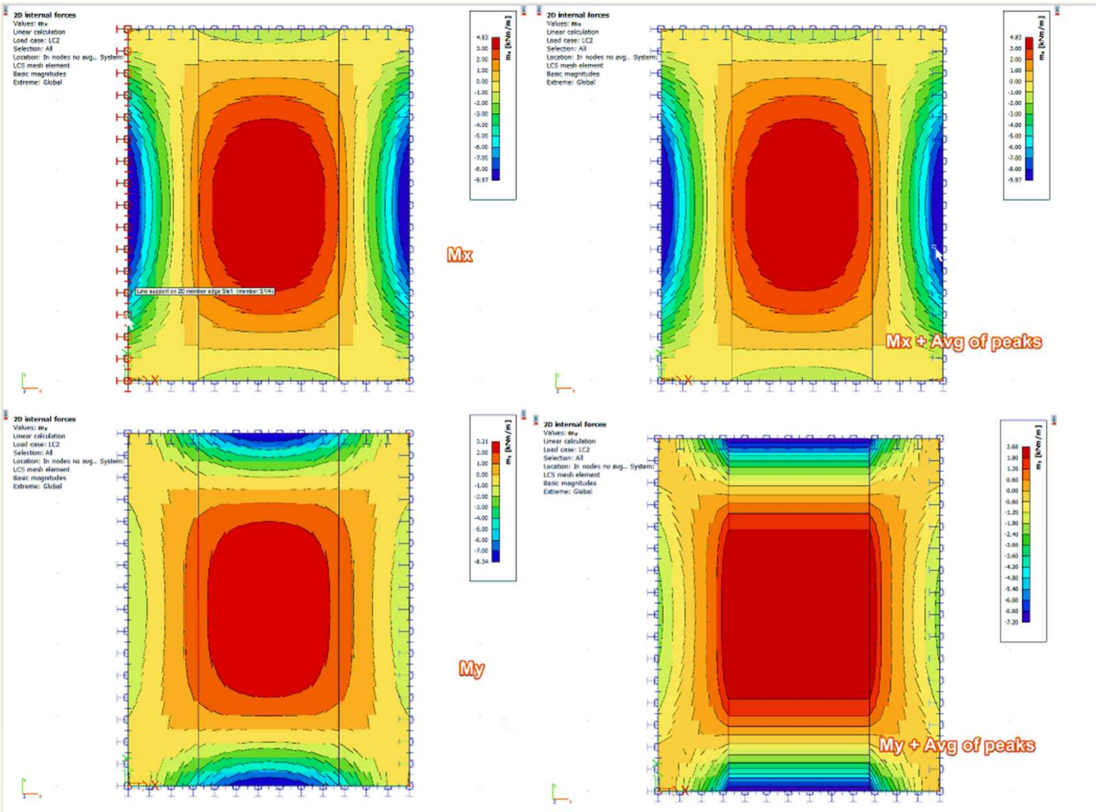


Type: a point or a strip can be chosen.

Dimensions: here the dimensions of the point/strip need be set.

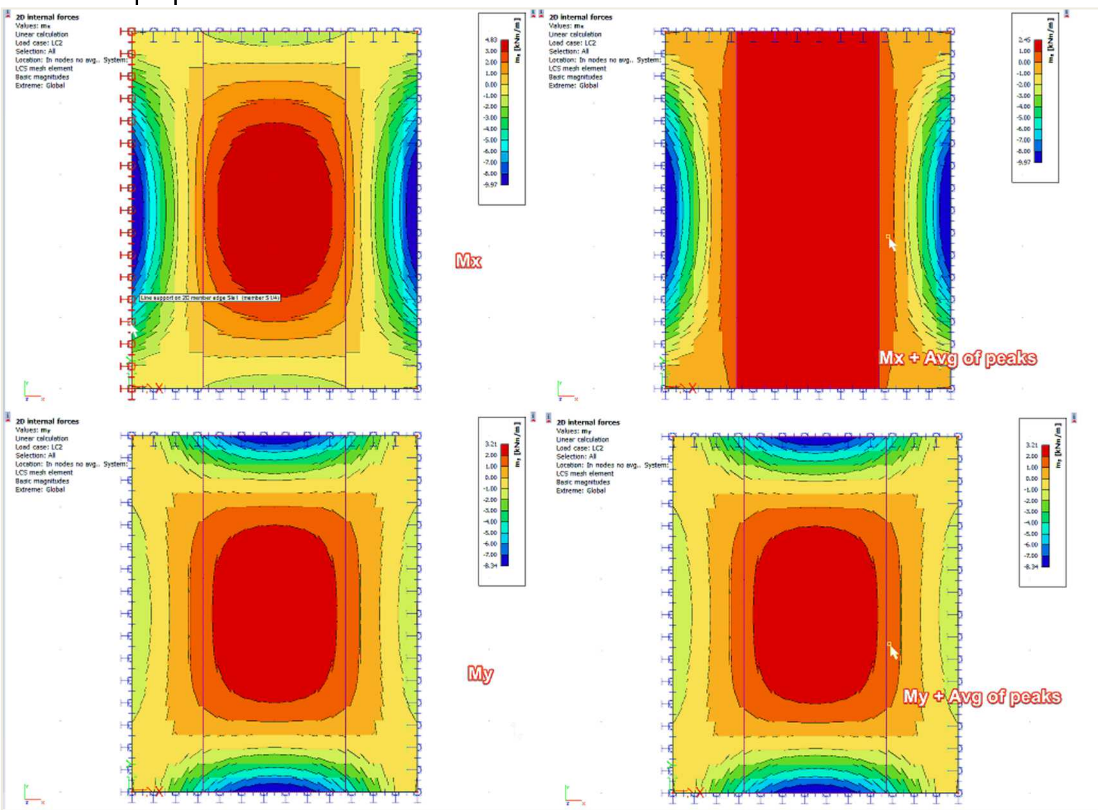
Direction:

1) Direction = Longitudinal



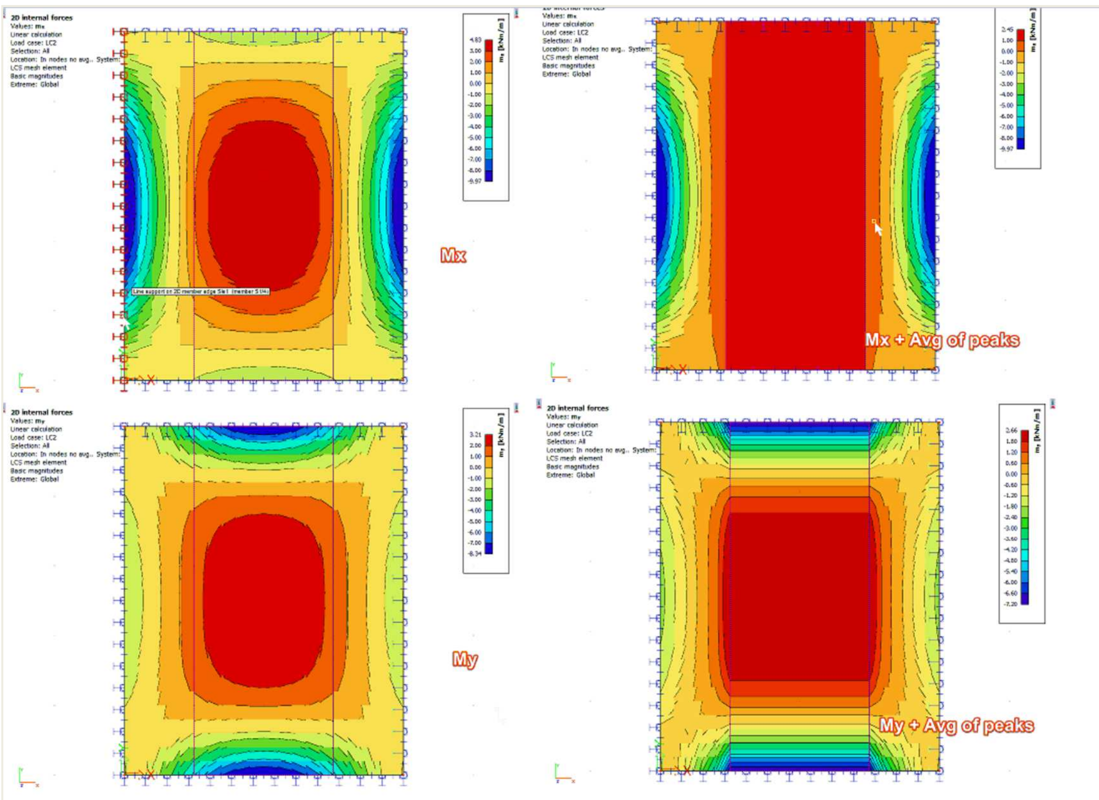
Longitudinal means that the averaging is done in the longitudinal direction of the strip. In the example above this is the y-direction. This means that the averaging is done for my. The values my are averaged in the x-direction.

2) Direction = perpendicular



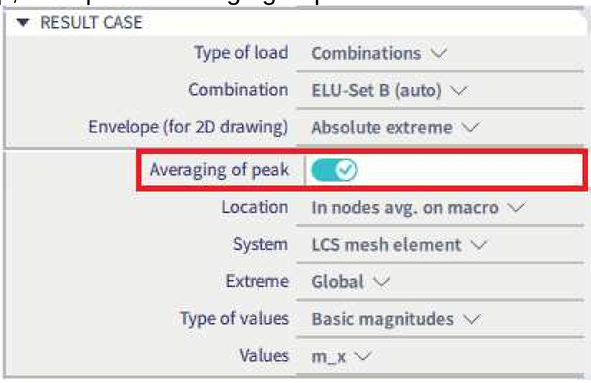
Perpendicular means that the averaging is done perpendicular to the longitudinal direction of the strip. In the example above this is the x-direction. This means that the averaging is done for mx. The values mx are averaged in the y-direction.

3) Direction = Both



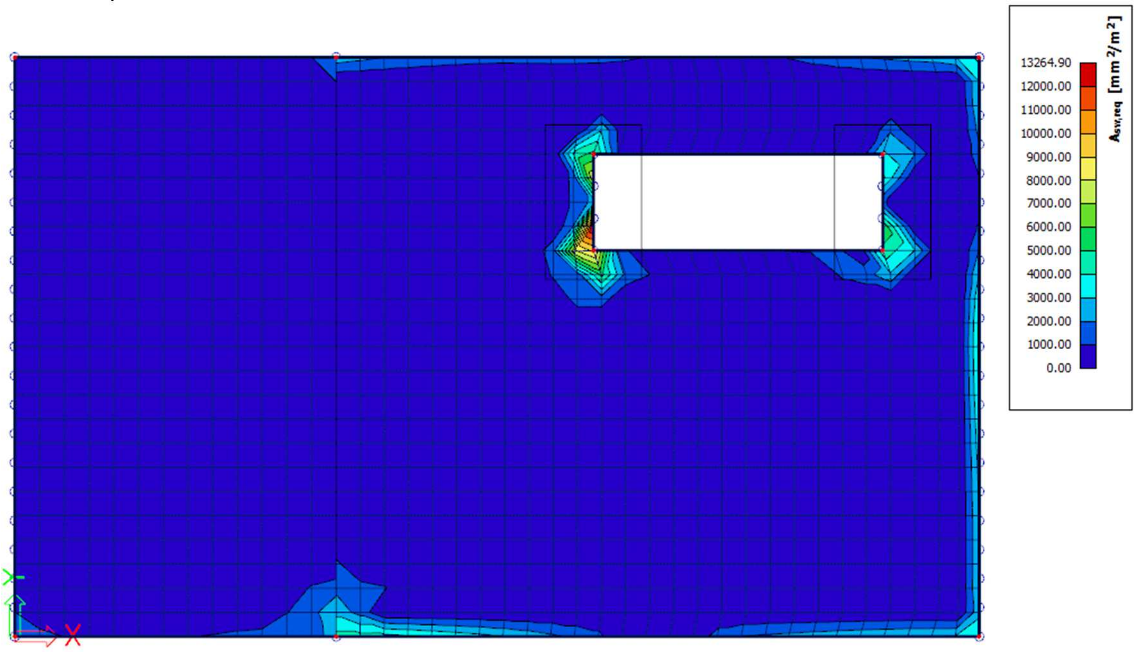
Both means that the averaging is done in both directions of the averaging strip. This means the values are averaged for mx as well as my in the direction perpendicular to mx and my.

To activate the averaging strip, the option ‘Averaging of peak’ needs to be checked in the properties panel.

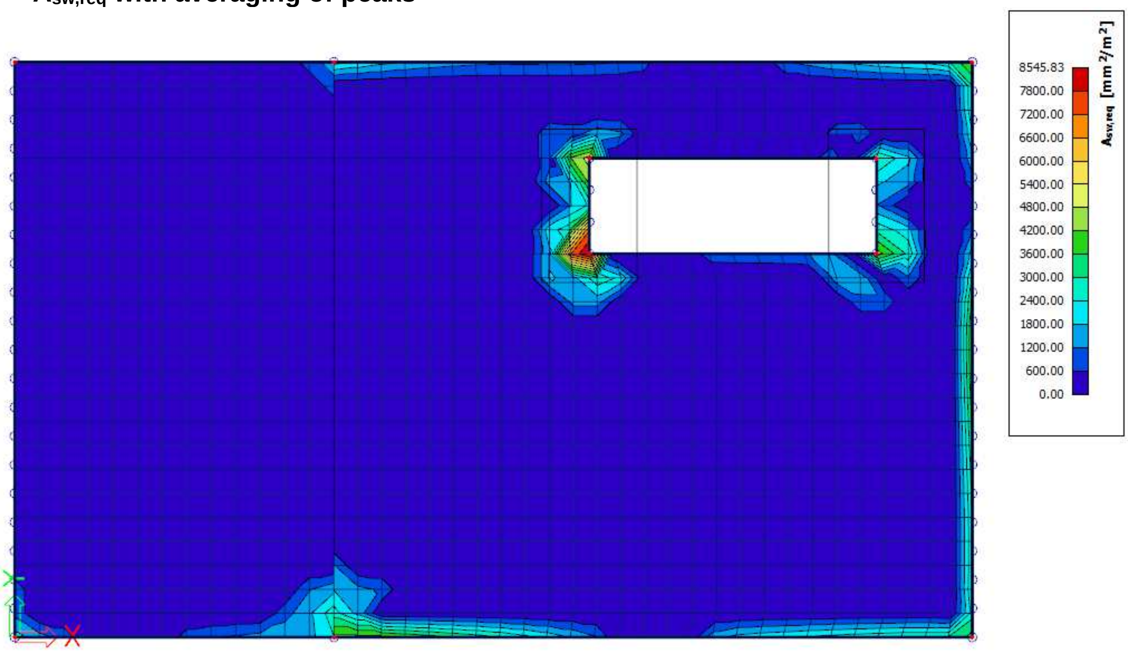


As an example, we will apply averaging strips to the model of the chapter “*2D concrete members”* for the value $A_{sw,req}$.

- **$A_{sw,req}$ without Averaging of peaks**



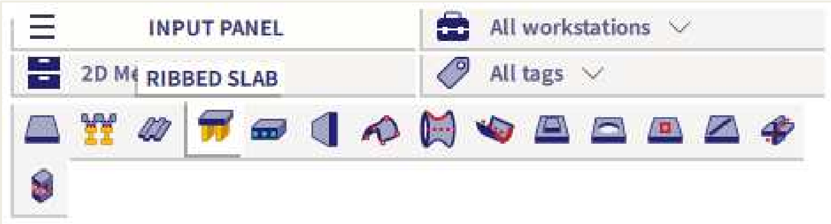
- **$A_{sw,req}$ with averaging of peaks**



3.3 Rib

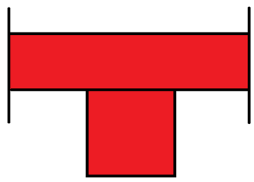
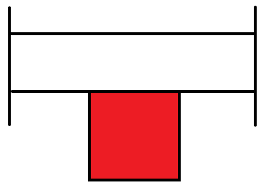
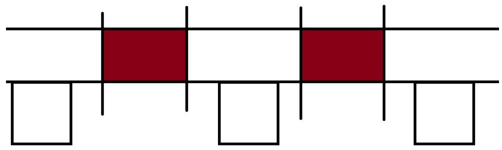
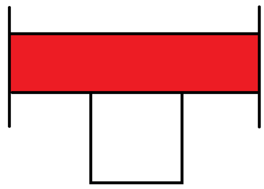
A rib can be added to a plate in the Input Panel in the “1D Members” category:

But also in the Input Panel in the “2D Members” category:



3.3.1 Results in ribs

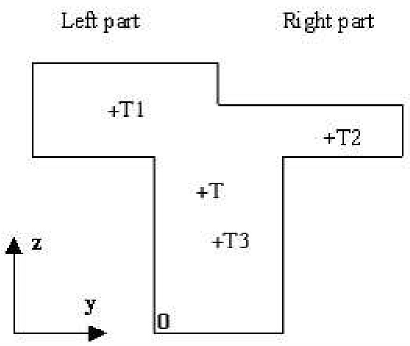
When a rib is added to the model there will be an option rib available in the result properties of 1D and 2D members. This option has an influence on what results you view.

	RIB Activated	RIB Not Activated
1D-Results		
2D-Results		

Link between the internal forces calculated for the entire T-section, and for the beam and slab separately

When calculating the internal forces in a rib, the substitute T-section is used to calculate the results. The web of this T-section is formed by the rib-beam itself, the flange of the T-section is made with the effective width of the slab. The effective width of the slab has to be used to determine the internal forces of the slab that have to be added to the internal forces calculated in the rib itself.

T	the heart of the entire substitute T-section
T1	the heart of the left part of the effective width
T2	the heart of the right part of the effective width
T3	the heart of the original rib



The coordinates of the hearts are used as lever arms in Y and Z direction:

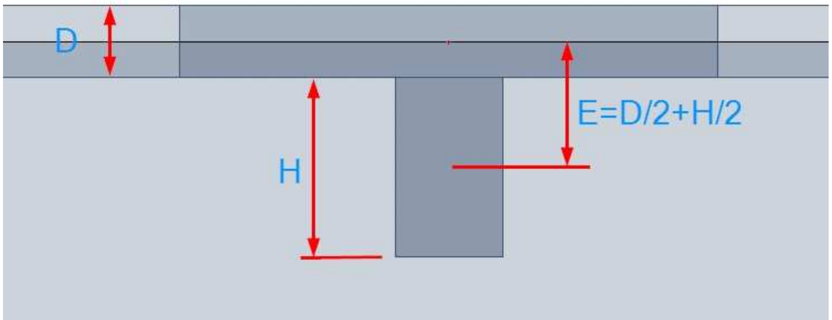
Lever Arm Z1 = T1z - Tz	Lever Arm Y1 = T1y - Ty
Lever Arm Z2 = T2z - Tz	Lever Arm Y2 = T2y - Ty
Lever Arm Z3 = T3z - Tz	Lever Arm Y3 = T3y - Ty
Lever Arm Z = Tz - 0z	Lever Arm Y = Ty - 0y

The final internal forces in the rib can be calculated with the formulas below:

- N = N beam + N slab, left + N slab, right
- V_y = V_y beam + V_y slab, left + V_y slab, right
- V_z = V_z beam + V_z slab, left + V_z slab, right
- M_x = M_x beam + M_x slab, left + M_x slab, right
- M_y = M_y beam + M_y slab, left + M_y slab right + N slab, left * (Lever Arm Z_1) – N slab, right * (Lever Arm Z_2) + N beam * Lever Arm Z3;
- M_z = M_z beam + M_z slab, left + M_z slab, right + N slab, left * (Lever Arm Y_1) – N slab, right * (Lever Arm Y_2) + N beam * Lever Arm Y_3;

Why is there an axial force in the rib?

SCIA Engineer integrates the ribs as eccentric beams attached to slabs. The eccentricity is calculated from the half of the slab thickness and half of the height of the cross section of the beam.

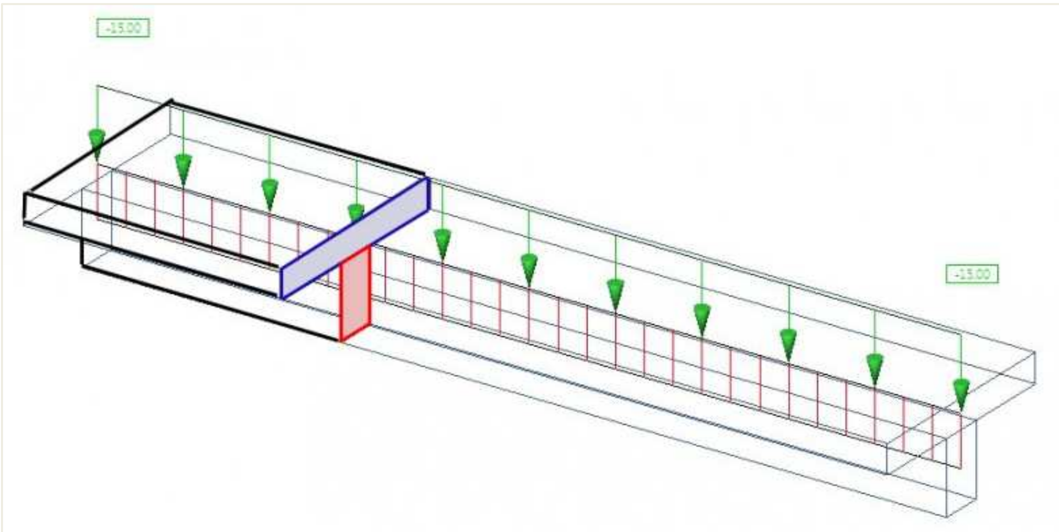


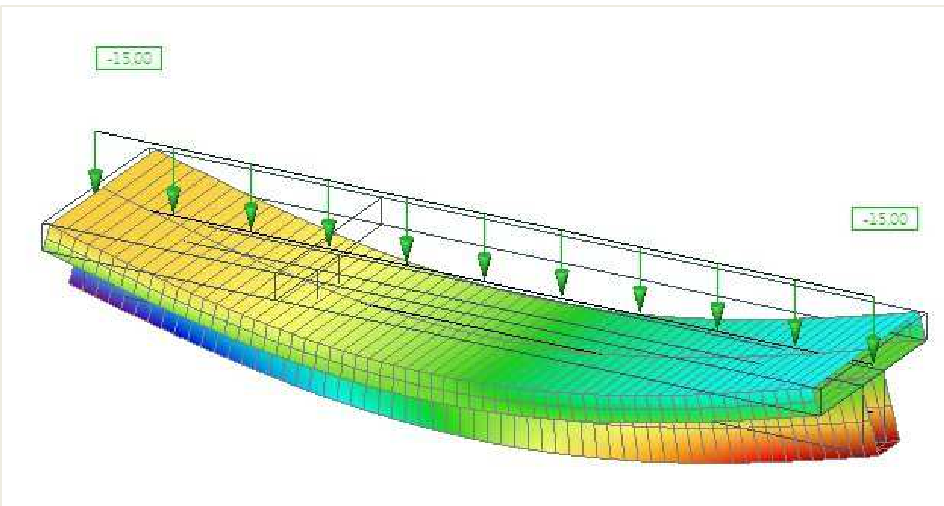
During the input of the cross section of the beam, the height of the cross section is defined as a distance between the bottom of the slab and the bottom of the beam. In the picture, the height is marked as “H”.

Due to the shift of the neutral axis, the internal forces in the whole system change. In a simple system subject to a bending moment only, we get a structure with an internal bending moment as well as axial force.

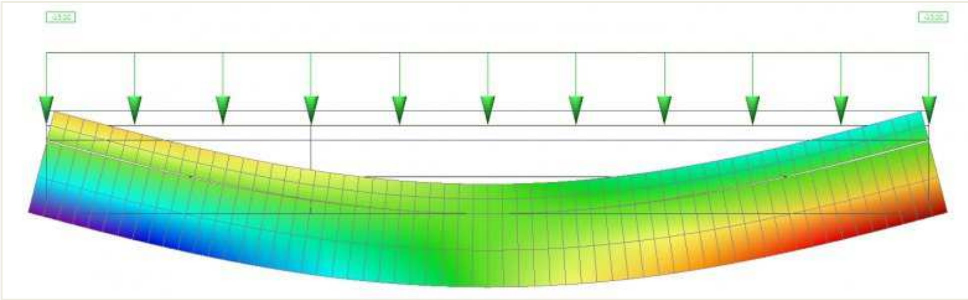
Usually, if the beam is below the slab, we get compression in the slab and tension in the beam.

The eccentric beam causes axial forces in the slab. This results from the deformation of the whole slab+beam system. The picture shows the horizontal deformation “u_x” to explain graphically the behaviour of the system. This system is composed of two beams of a rectangular cross section connected by rigid links. The horizontal displacement of the support is free to prevent the constraint.

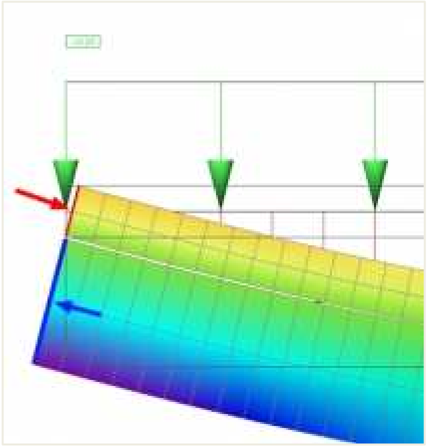




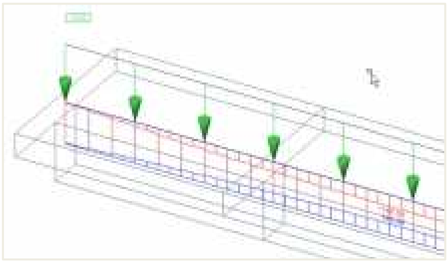
The horizontal deformation in a side view:



If we look at the beginning of the beam, we can see compression in the slab and tension in the beam:



Of course, the whole system must be in equilibrium and the total axial force equal to the sum of the axial force in the slab and in the beam must be zero.

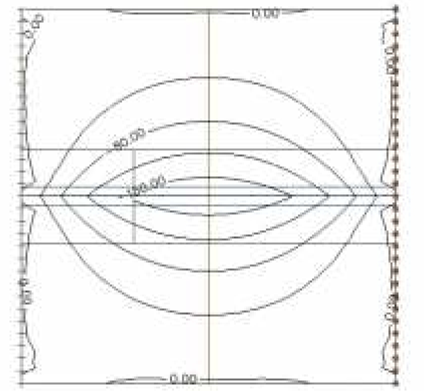


In our model, we have only one beam and all the internal forces of the top part are integrated in the axial force in the rib. Practically, the effective width of the slab is smaller than the whole width of the slab. Only exceptionally are the ribs arranged in such a way that there are no gaps in between the effective widths and all internal forces in the slab can be summed up into the rib. This happens if the distance between the ribs is smaller or equal to the effective width of the slab calculated from the national code.

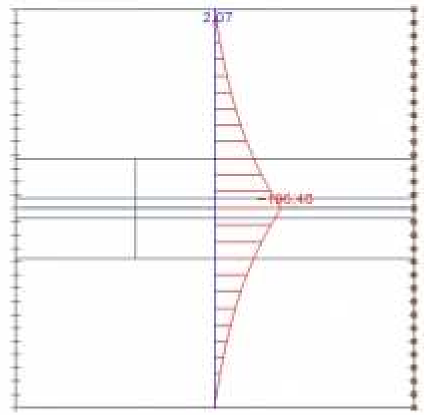
Behaviour of a rib in a wide slab

Now we can investigate a system where the width of the slab is greater than the effective width of the slab. The equilibrium condition must be fulfilled. If we integrate all the axial forces in the whole slab and the beam, we - of course - get a zero result.

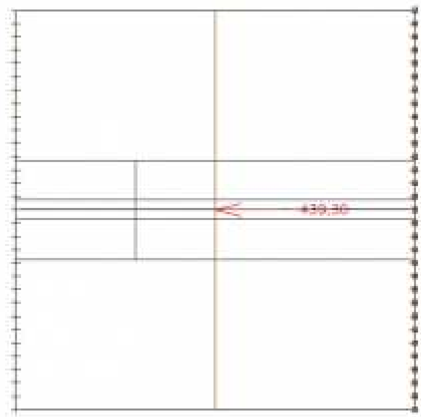
Distribution of the axial force in the slab. This is independent on the defined effective width of the slab. Only the stiffness of the slab and beam is responsible for the shape of the distribution of internal forces.



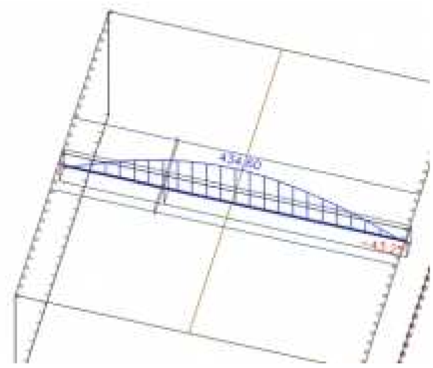
This is a section across the middle of the slab showing the distribution of the axial force.



We can integrate the axial force in the section across the whole width of the slab. We get 439kN.



Compared to the axial force in the beam, which is 435kN. We see the whole system is in equilibrium. The small difference results from the size of the finite elements.

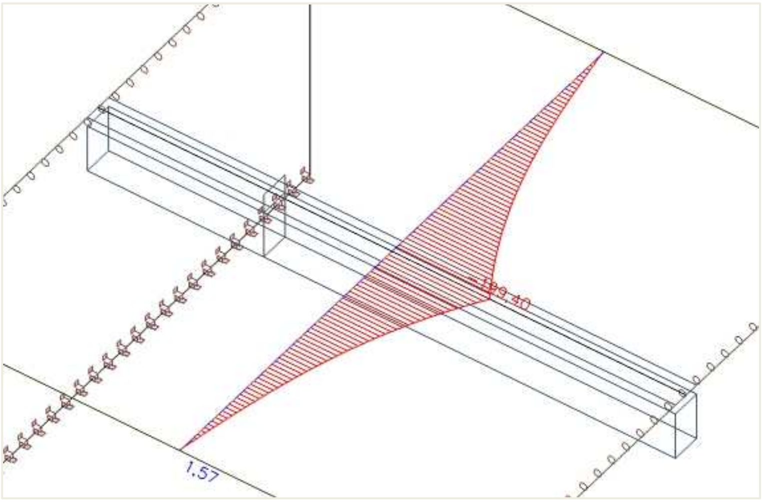


Comparison of different effective widths

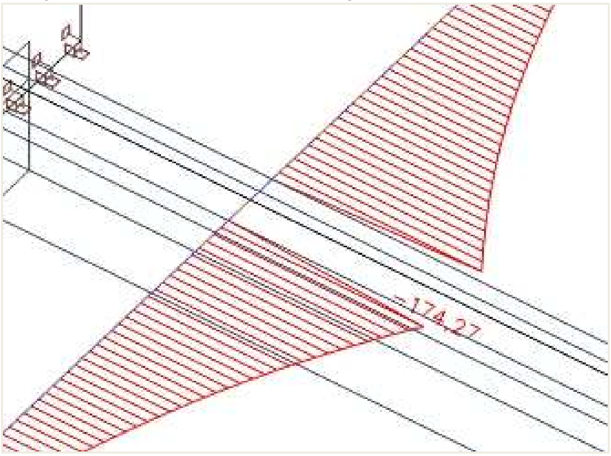
However, if we extend the effective width of the slab to the whole width of the slab, we neglect the distribution of the internal forces over the slab and the concentration over the beam. (In fact, there are two limit values: the minimum effective width is equal to the width of the beam and the maximum one is equal to the whole width of the slab.)

The internal forces in the slab are excluded from the slab and integrated into a new virtual T section. This virtual section consists of the effective slab width and the beam.

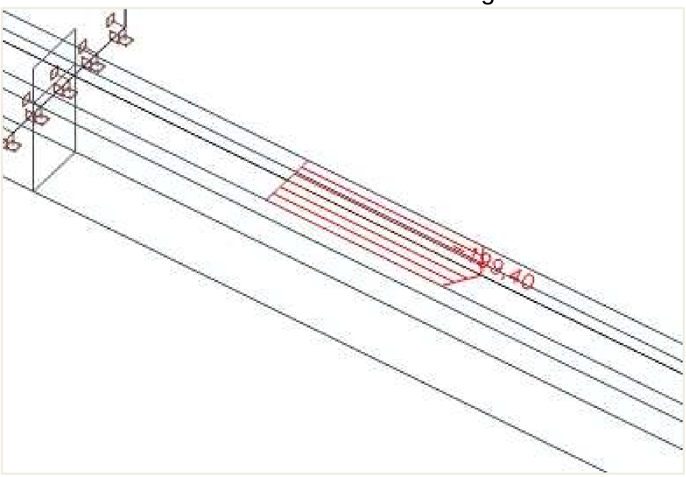
Distribution of the axial force in the slab. We can see that the distribution is equal to the one in the pictures above where the effective width of the slab was defined according to the code.



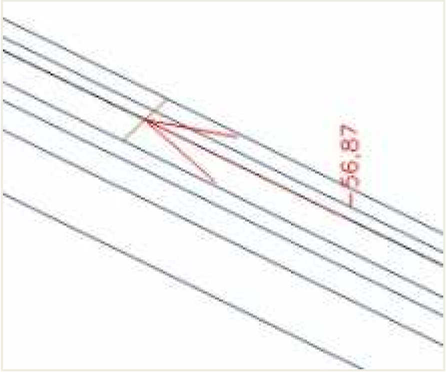
In the picture we can see the axial force after the forces within the effective width of the slab were excluded from the slab. In SCIA Engineer you can achieve this using the checkbox "RIB" in the results.



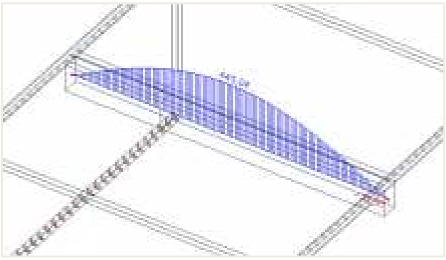
These axial forces within the effective width of the slab can be integrated.



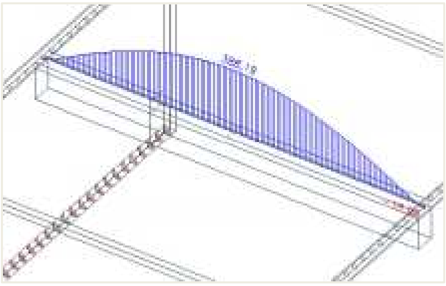
We get axial force equal to 56kN, which is in the slab. The total axial force in the slab was 435kN. Therefore, in the part outside the effective width we have axial force 435 - 56 = 379kN.



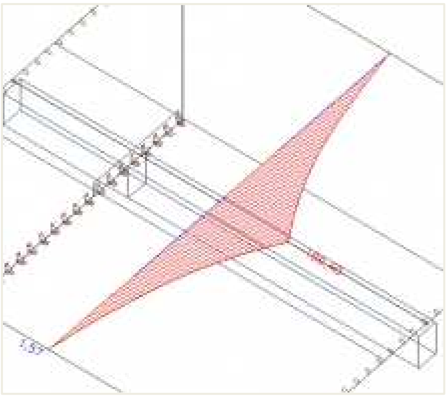
In the beam, we still have the same 445kN. (The difference to the previous pictures results from the changed size of the 2D finite elements).



If we create the sum of the integrated axial force in the slab and in the beam, we must get 445 – 57 = 388kN.

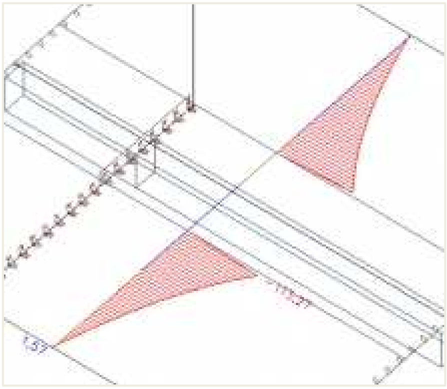


Look what happens if we increase the effective width of the slab to 1500mm. This results from the following formula: 2 * (0,1 * L) + b_w = 2*0,6+0,3

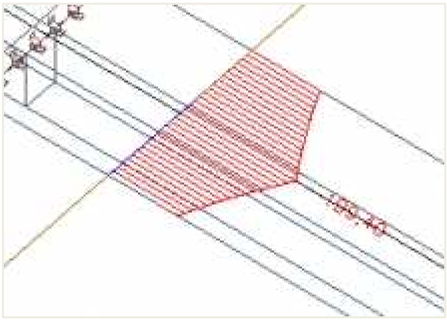


As we can see, the axial force in the slab is still the same. It must be, because the effective width of the slab has no influence on the distribution of the axial force in the finite element calculation. It only affects the split of the forces after the calculation between the slab and the virtual T section.

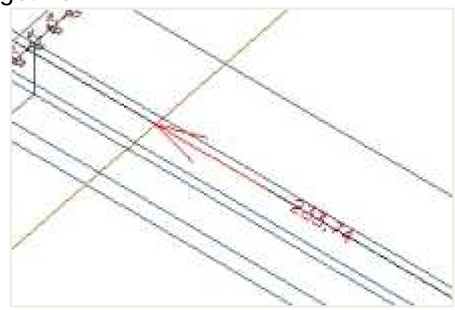
The area of the effective width of the slab will be removed from the slab and the forces will be integrated into the T section. The internal forces outside of the slab will remain in the slab.



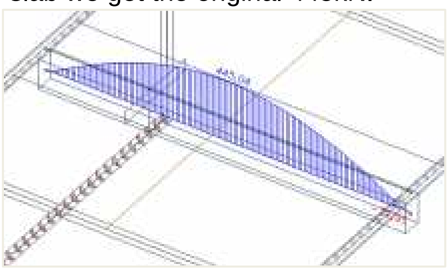
These internal forces will be moved to the T section.



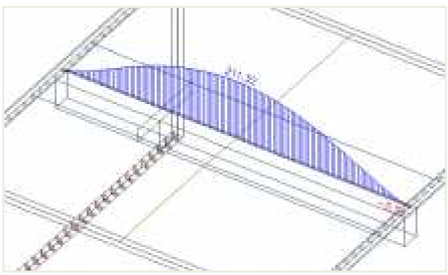
If we integrate the axial forces, we get 234kN.



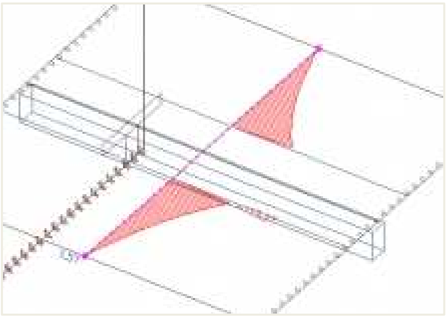
In the rectangular section below the slab we get the original 445kN.



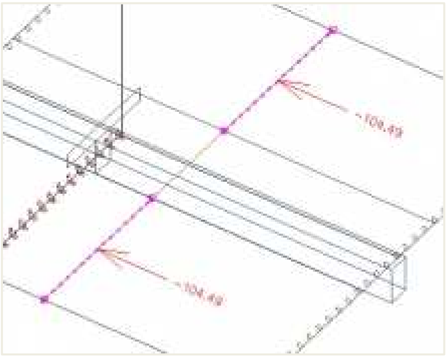
If we reduce this axial force of the beam by 234kN, which is the sum of the axial forces from the effective width of the slab, we get 211kN

.

The axial force outside the effective width remains in the slab.



If we integrate the forces (left and right) outside the effective width, we get axial force equal to 210kN, which is in equilibrium with the tension in the rib as a T Section.



3.3.2 Stiffness of ribs in CDD calculation

The calculation of the stiffness of the rib depends on the checkbox "**Rib".**

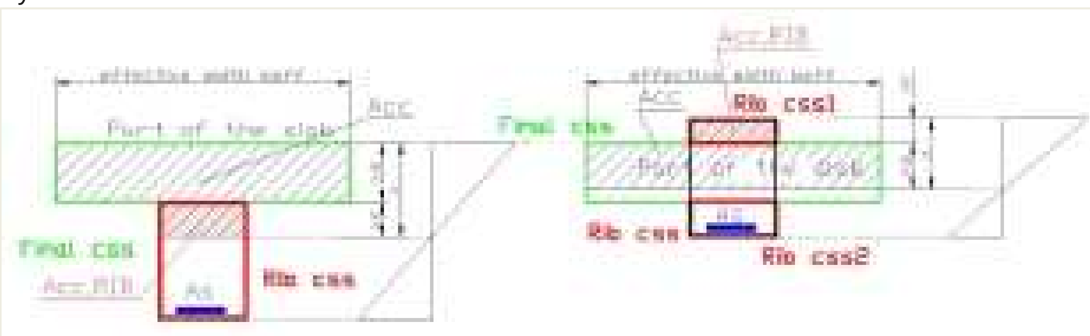
- ***Check box is OFF***

The stiffness of the beam and the plate will be calculated separately. If there is 1D reinforcement in part of the slab it is not taken into account for the calculation of the stiffness of the plate.

- ***Check box is ON***

1) Equilibrium for the final cross-section is calculated for each dangerous combination and each type of stiffness.

2) The stiffness of the rib, only taking into account the rib cross-section, is calculated with the height of compression zone from equilibrium on the whole (final) section. Stiffnesses are calculated to the centre of gravity of the transformed final cross-section.



3) Stiffness of the 2D element outside of the effective width is calculated by the standard procedure. Stiffness of the 2D element inside effective width is calculated in two directions: direction of the rib (α_{rib})and direction perpendicular to the rib. (α_{rib} + 90)

4) The stiffness perpendicular to the rib is calculated by the standard procedure.

5) The stiffness in the direction of the rib is calculated according to following procedure:
   - The 1D reinforcement which is designed or inputted in part of the slab of the final cross-section is taken into account for the calculation of the stiffness of the 2D element. This reinforcement is transformed to 2D reinforcement and is added to the standard 2D reinforcement.
   - Uncracked stiffnesses (EA_I, $EIy_{,I}$, $EIz_{,I}$) will be calculated for the whole thickness of the 2D element with standard 2D reinforcement (required/provided/user) and with transformed reinforcement from the 1D member. The stiffness is calculated to the transformed centre of gravity of the uncracked section.
   - Cracked stiffness is calculated in case that ($\sigma_{ct} <= \sigma_{cr}$). The stiffness ($EA_{II}$, $EIy_{,II}$, $EIz_{,II}$) will be calculated taking into account parameters from the calculation of the 1D element which is nearest to centre of gravity of the 2D element. The height of compression zone is calculated according to formula:

$$x_s = \frac{A_{cc} - A_{cc,Rib}}{b_{eff}}$$

     Where:
     - A_{cc} – compressive area of whole cross-section for cracked CSS
     - $A_{cc,RIB}$ – compressive area of part of cross-section (rib cross-section) for cracked CSS
     - b_{eff} – effective width of the slab for check
     - σct - is maximum tensile strength calculated for final cross-section (rib cross-section + part of the slab) and for characteristic combination

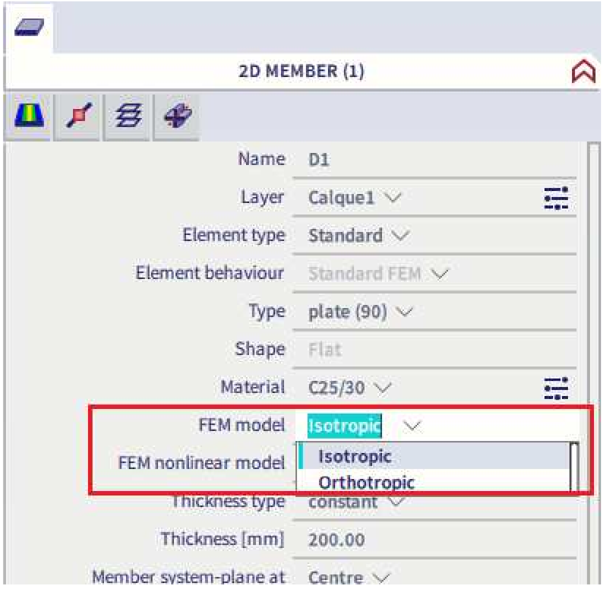
     The stiffness is calculated to the transformed centre of gravity of the cracked section.
   - The average stiffness will be calculated from the cracked and the uncracked stiffness using the distribution coefficient, which is calculated from the stresses calculated for the whole cross-section of the 1D element which is nearest to centre of gravity of the 2D element.
     - bending stiffness around y-axis (EIy) = $1/[\zeta/(EIy)_{II} + (1-\zeta)/ (EIy)_I]$
     - bending stiffness around z-axis (EIz) = $1/[\zeta/(EIz)_{II} + (1-\zeta)/ (EIz)_I]$
     - axial stiffness (EA) = $1/[(\zeta/(EA)_{II} + (1-\zeta)/(EA)_I]$

3.4 Orthotropy

In engineering practice, you may often come across a situation when the slab (or wall) to be designed has different characteristics (stiffness) in the longitudinal and transverse direction and thus, shows different behaviour in these two directions. Such a behaviour may result from the geometry (e.g. ribbed slabs) or from physical assumptions for a particular situation, for example, when determining deformations in a cracked plate or when excluding vertical members from a horizontal stiffening system (e.g. masonry walls).

Whenever you need to adjust the finite element model accordingly to reflect such a behaviour in SCIA Engineer, the orthotropic properties can be used. These orthotropic properties can be defined in two ways.

Orthotropy in the properties of a 2D member



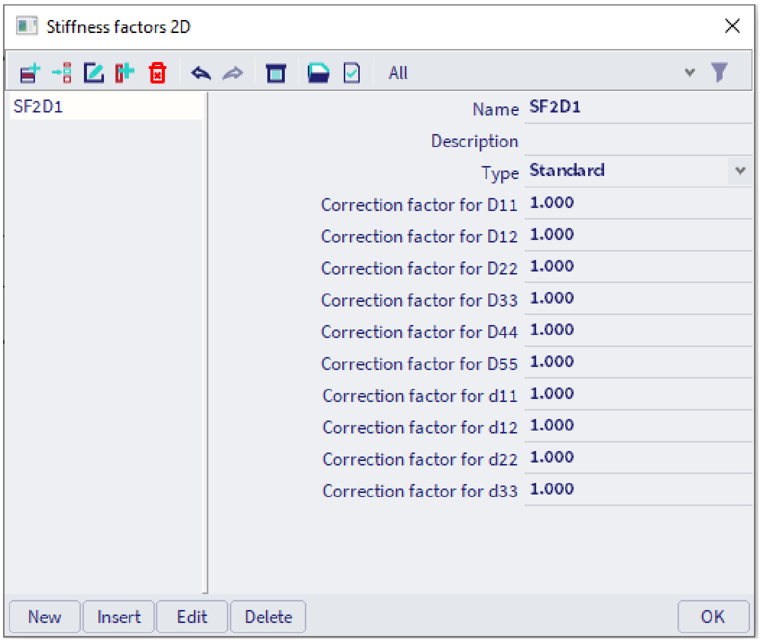
Property modifier

The difference is in the data you need to enter. In the orthotropy, the stiffnesses are defined directly, while in the property modifier, a factor is specified by which the isotropic stiffnesses are multiplied.

The property modifier is a bit more flexible because it does not depend directly on the properties of the modified part. If you want to enter an uniaxially stretched plate, then you can do that for a 20cm thick plate and also for a 30cm thick plate using the same values. The orthotropic properties require that you define separate properties for each of the plates (20cm and 30cm one).

On the other hand, also the orthotropy has its advantages. It can be parameterized, and the program includes a set of generators to help you with the input.

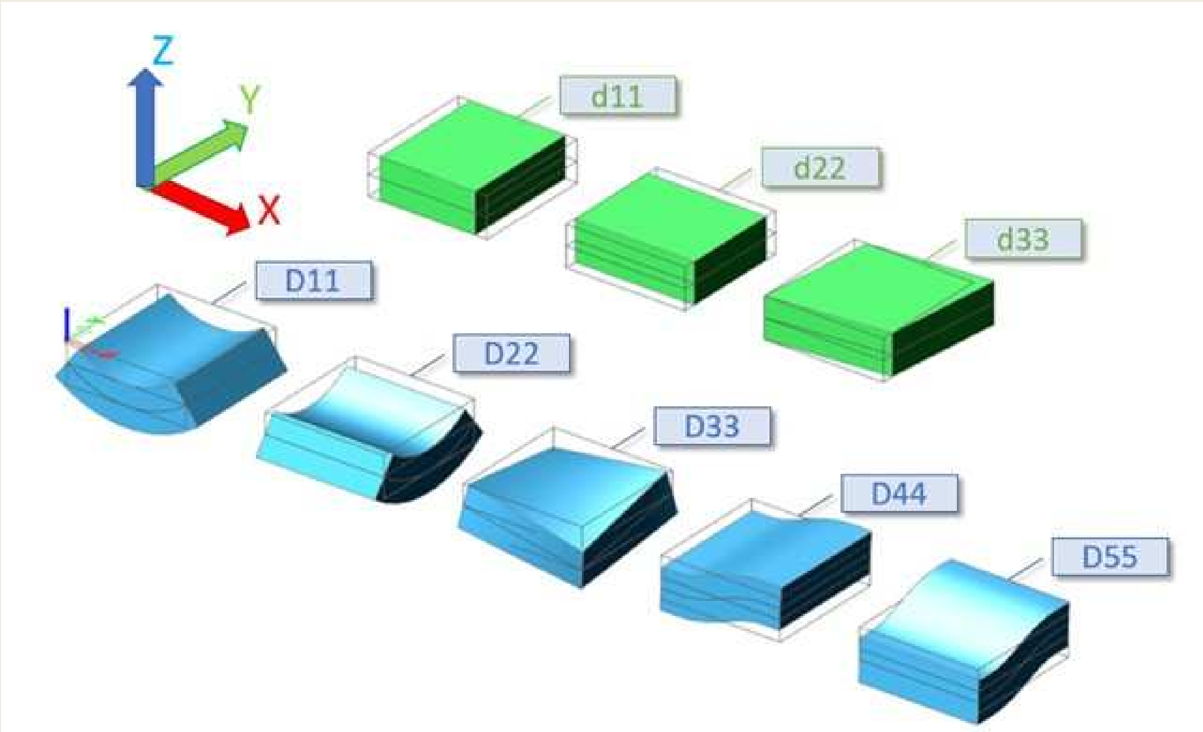
However, it is important to understand individual orthotropic parameters. The stiffnesses are defined with parameters starting with a "D" or "d". The property modifiers ask for the following parameters for a shell element:



The parameters beginning with "D" represent plate stiffnesses. The parameters starting with "d" are membrane stiffnesses. The direction is derived from the direction of the local coordinate system.

- D11: Flexural stiffness in the "x" direction (bending)
- D22: Flexural stiffness in the "y" direction
- D12: Mixed stiffness of D11 and D22 (transverse contraction)
- D33: Torsional stiffness
- D44: Shear flexural stiffness in the "x" direction
- D55: Shear flexural stiffness in the "y" direction
- d11: Normal membrane stiffness in the "x" direction (stretching)
- d22: Normal membrane stiffness in the "y" direction
- d12: Mixed stiffness of "d11" and "d22" (transversal contraction)
- d33: Shear membrane stiffness

$$\begin{bmatrix} M_x \\ M_y \\ M_{xy} \\ T_x \\ T_y \end{bmatrix} = \left[\begin{array}{ccc:cc} D_{11} & D_{12} & 0 & & \\ D_{21} & D_{22} & 0 & \multicolumn{2}{c}{\mathbf{0}} \\ 0 & 0 & D_{33} & & \\ \hdashline & & & D_{44} & 0 \\ & \mathbf{0} & & 0 & D_{55} \end{array}\right] \begin{bmatrix} w_{xx} \\ w_{yy} \\ 2w_{xy} \\ w_x + \varphi_y \\ w_y - \varphi \end{bmatrix}$$



In case of a simple, isotropic plate, the stiffness can be expressed using the following formulas:

Plate direction	Membrane stiffness
$D_{11} = D_{22} = \dfrac{E \cdot h^3}{12(1-\nu^2)}$ $D_{12} = \nu \cdot \sqrt{D_{11} \cdot D_{22}}$ $D_{33} = G \cdot \dfrac{h^3}{12}$ $G = \dfrac{E}{2 \cdot (1+\nu)}$ $D_{44} = D_{55} = G \cdot h$	$d_{11} = d_{22} = \dfrac{E \cdot h}{1-\nu^2}$ $d_{12} = \nu \cdot \sqrt{d_{11} \cdot d_{22}}$ $d_{33} = \dfrac{1}{2} \cdot (1-\nu) \cdot \sqrt{d_{11} \cdot d_{22}}$

How to model a one-way slab in SCIA Engineer

A one-way slab is a slab that bears the load in one direction mainly. It can be a slab supported on two edges only or a slab supported on four edges for which the bigger span length L_y is at least twice the smaller span L_x. The design of a one-way slab will lead to reinforcement mainly in the bearing direction.

In a Finite Element software like SCIA Engineer, when the slab is supported on its four edges, the software will by default consider it as a two-way slab. Since there is no predefined main direction for the bearing of the load, the bending stiffness of the slab will participate in both x and y directions. In SCIA Engineer you can easily define a one-way-bearing slab.

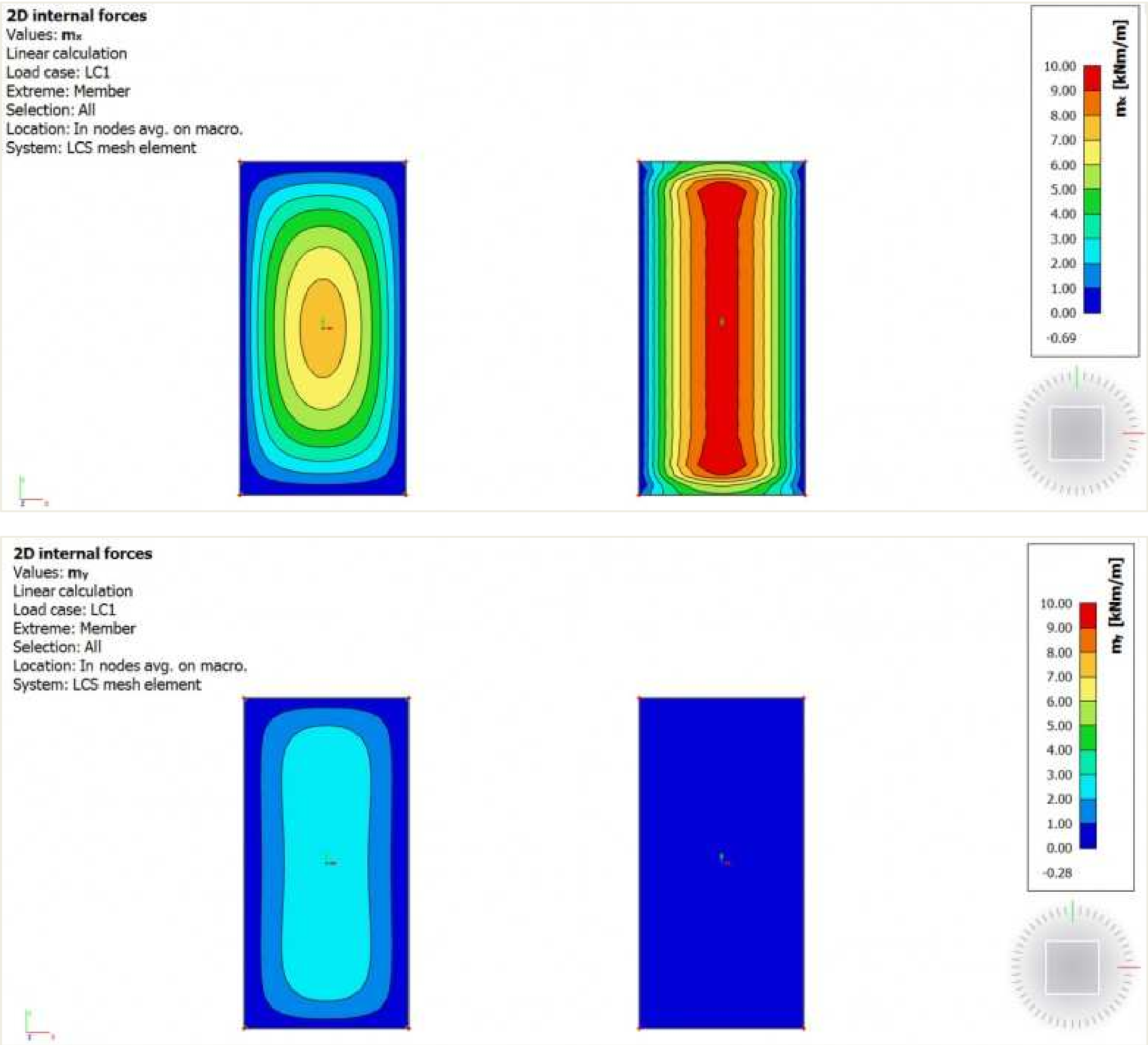


Figure 1: Bending moments in a two-way slab (on the left) and one-way slab (on the right)

In SCIA Engineer, the input of a one-way slab can be done with orthotropy properties. Two types can be used and are explained below.

One-way slab using "two heights" orthotropy type

The example is made of a slab supported by beams and columns. In the slab properties, change the FEM model to "Orthotropy", edit the orthotropy property and select the type of orthotropy "two heights". The input data are the thickness of the plate for the calculation of the flexural stiffness in the x-direction, h_1, and the y-direction, h_2. For a slab bearing mainly in the x direction (smaller span length in the example), h_1 should be kept equal to the actual plate thickness (180 mm) and h_2 (thickness in y-direction) should be reduced.

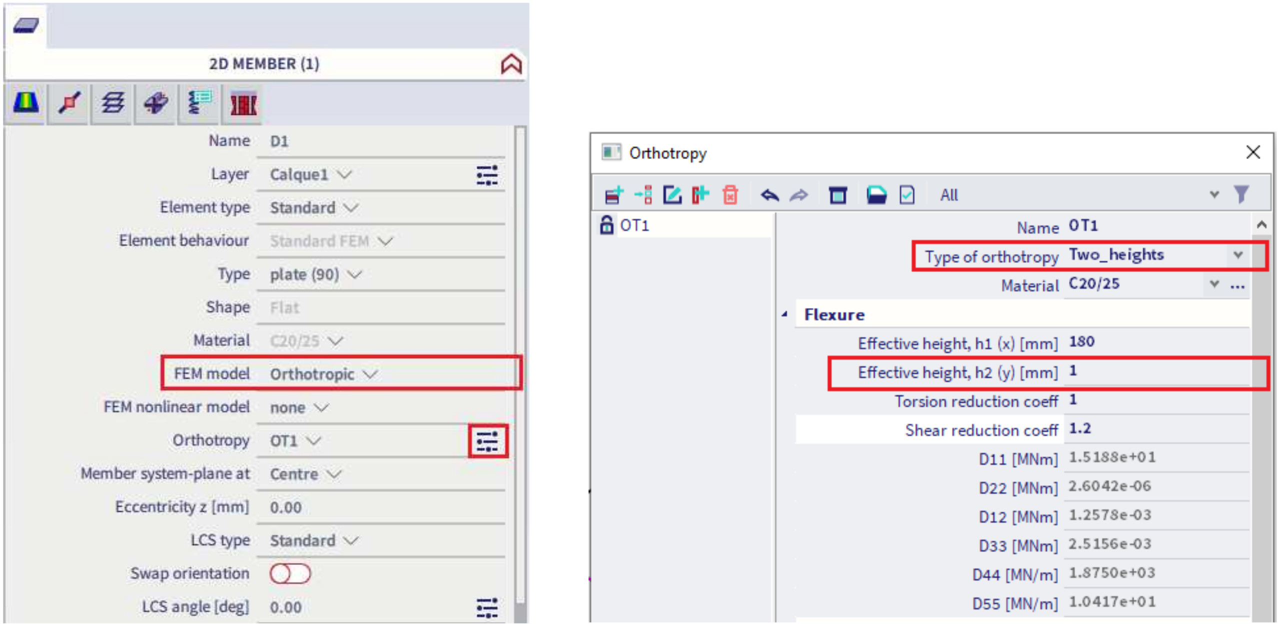


Figure 2: Parameters for a one-way slab using "two heights" orthotropy type

There is no specific rule regarding the value of h_2. With smaller values of h_2, the results will be close to the following load distribution:

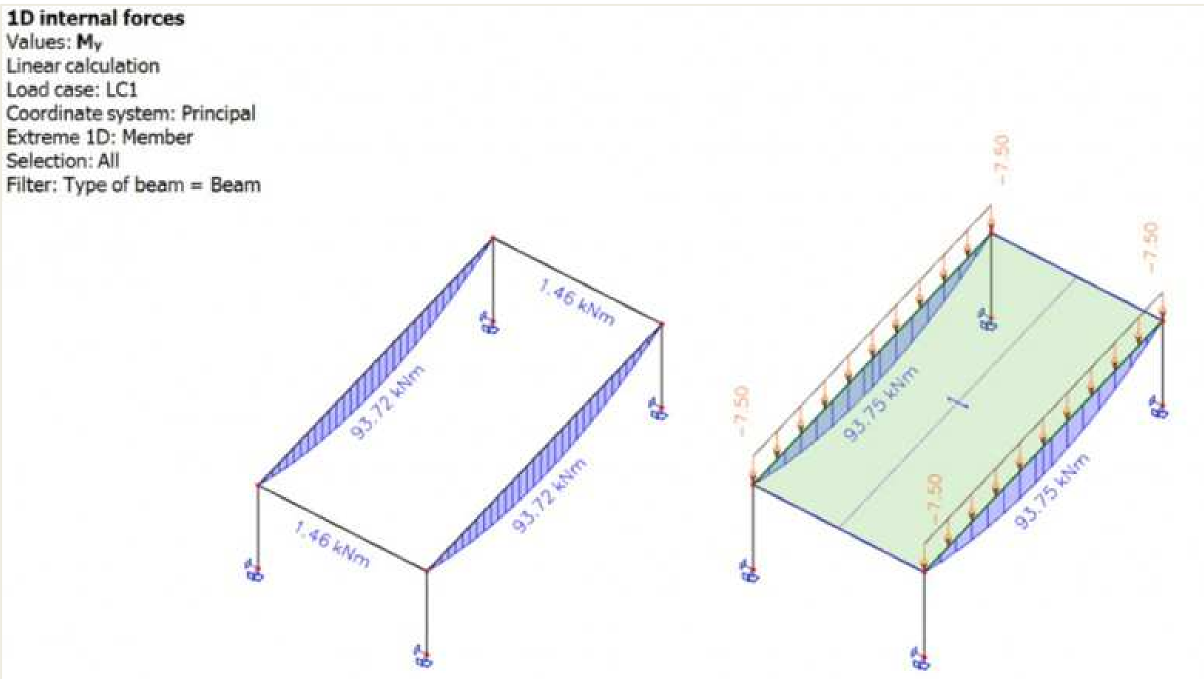


Figure 3: Bending moment in the supporting beams of a one-way slab (on the left) and of a one-way load panel (on the right)

The resulting moment mx in the slab is then close to a 1m-wide simply supported beam:

$$m_x = \frac{q * L_x^2}{8} = \frac{3 * 5^2}{8} = 9{,}4\mathrm{kNm/ml}$$

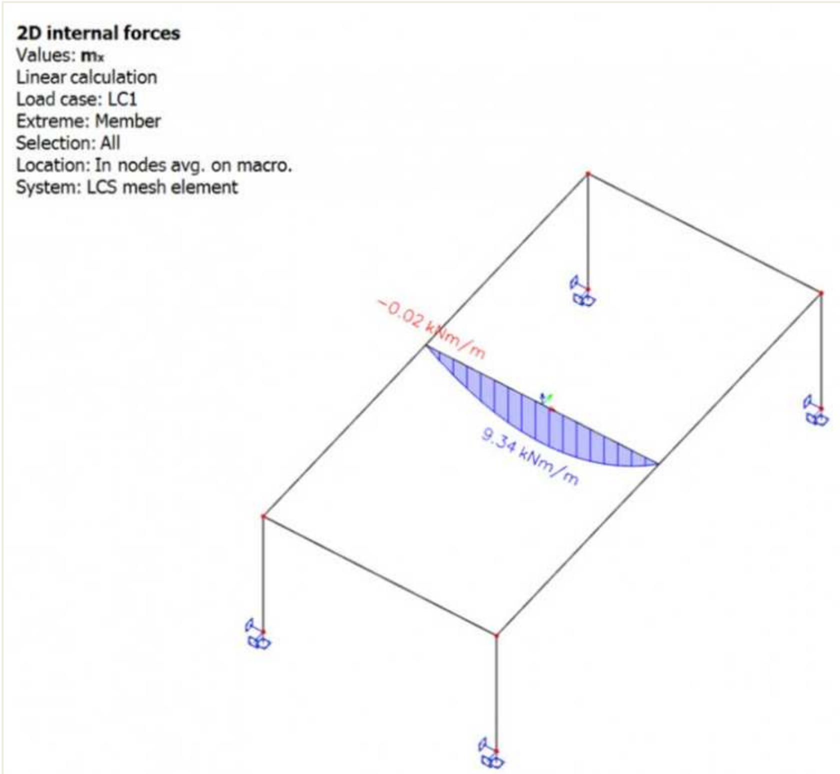


Figure 4: bending moment mx in a one-way slab

One-way slab using “one direction” orthotropy type

This type of orthotropy requires three input parameters and can also be used for modelling hollow core slabs: the equivalent beam cross-section CSS, the spacing L used for the calculation of the flexural bending stiffness in direction 1 (or x) and the concrete topping height h used for the calculation of the flexural bending stiffness in direction 2 (or y). To model a one-way slab, a small value of h can be used. However, keep in mind, that h is also used for the calculation of the self-weight of the slab.

For the equivalent cross-section, a slab-equivalent shape is used: “thickness of the slab” x “width of the beam”, i.e. 180 x 1000mm. For the spacing parameter, as the slab is plain, the same value as for the width of the beam is used, i.e. 1000mm.

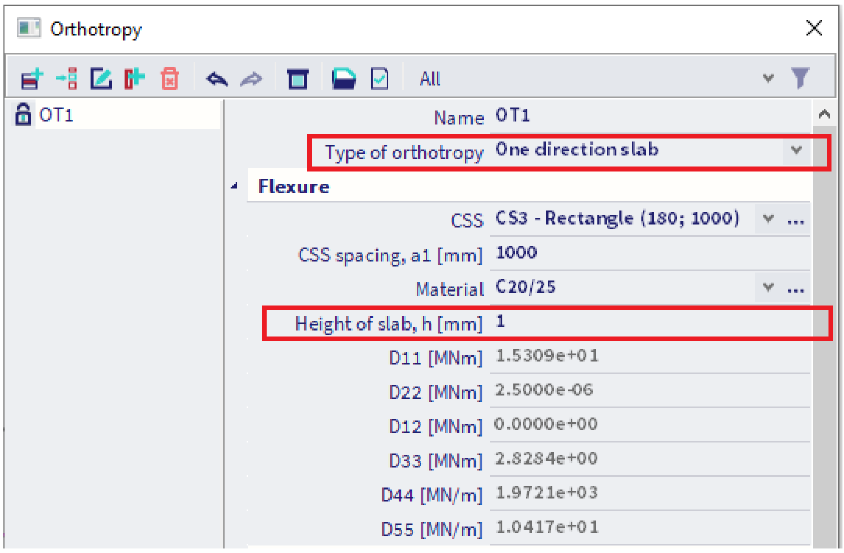


Figure 5: Parameters for a one-way slab using "one direction" orthotropy type

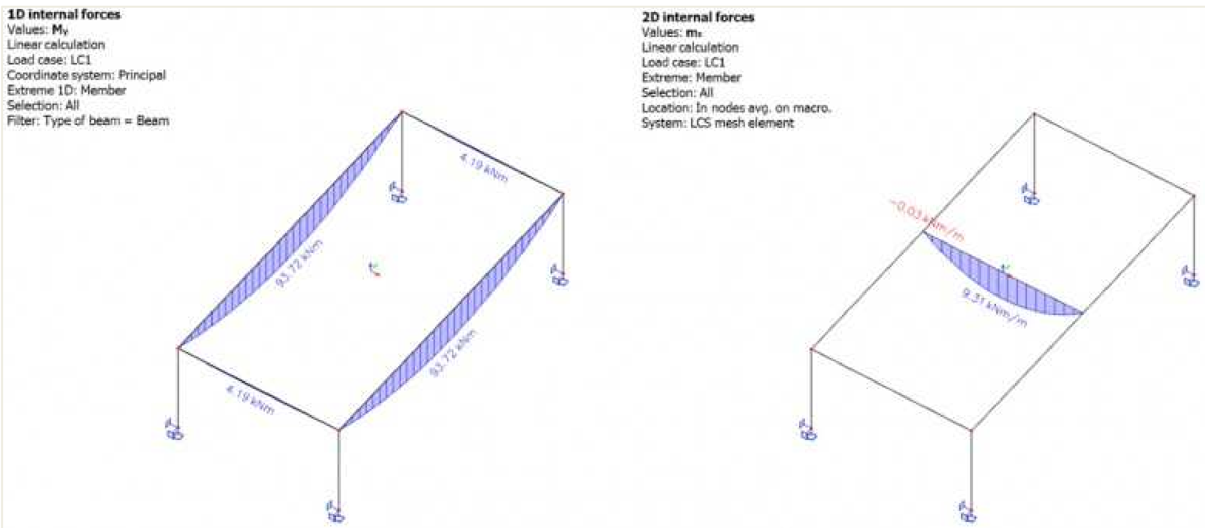


Figure 6: Bending moment in the supporting beams and in the one way slab using the type "one direction"

For small values of h_2 or h, both types give the same results for the bending moment in the bearing direction and the load transferred to the supported beams.

There are still some differences between both orthotropy types. First, using “one direction” type leads to higher values of bending moment on the secondary beams (parallel to the bearing direction). This is due to the torsional moment component of the plate (D33) that is different between both types. Secondly, with “one direction” orthotropy type, the self-weight of the slab is calculated based on the concrete topping thickness h only. The total height of the slab is thus not accounted for and you have to add the missing part of the self-weight manually in a permanent load case.