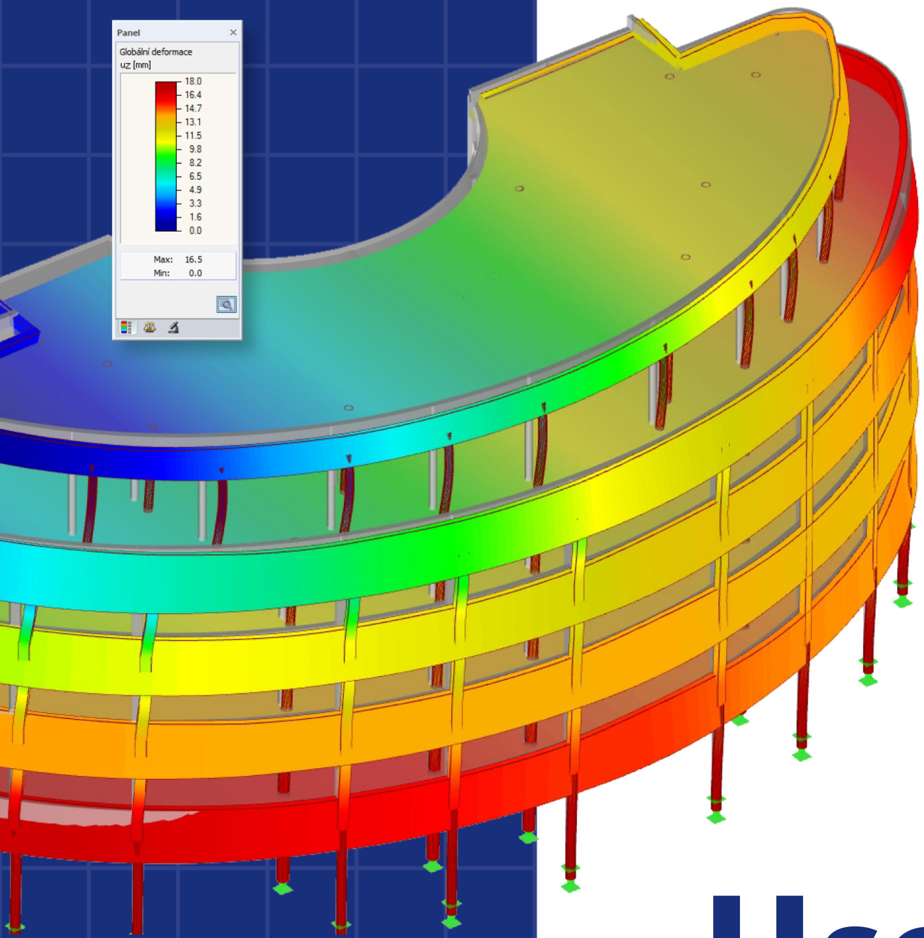
RF-CONCRETE Members

Reinforced Concrete Design



User Manual

Version

June 2020

Dlubal Software

Short Overview

Dlubal Software GmbH
Am Zellweg 2
93464 Tiefenbach
Germany

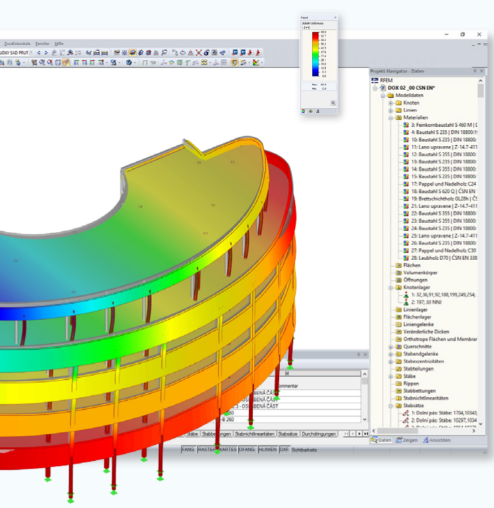
Telephone: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com

Dlubal Software, Inc.
The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Phone: +1 267 702-2815
E-mail: info@dlubal.com

Using the Manual

The program description is organized in chapters that follow the order and structure of the input and result windows. The chapters present the individual windows column by column. They help to better understand the functioning of the add-on module. General functions are described in the manual of the main program RFEM.



Hint

The text of the manual shows the described buttons in square brackets, for example [OK]. In addition, they are pictured on the left. Expressions appearing in dialog boxes, tables, and menus are set in *italics* to clarify the explanation. You can also use the search function for the Knowledge Base and FAQs to find a solution in the posts about add-on modules.

Topicality

The high quality standards placed on the software are guaranteed by a continuous development of the program versions. This may result in differences between program description and the current software version you are using. Thank you for your understanding that no claims can be derived from the figures and descriptions. We always try to adapt the documentation to the current state of the software.

Table of Contents

1 Introduction

1.1 Add-on Module RF-CONCRETE Members

RF-CONCRETE Members is an add-on module for RFEM. It is completely integrated into the RFEM user interface, which ensures a continuous calculation and design of reinforced concrete components available in RFEM in the form of framework structure elements.

The add-on module imports all relevant model parameters such as materials, cross-sections, members, sets of members, ribs, supports, as well as internal forces of load cases and combinations from RFEM. The program also allows for design alternatives with modified cross-sections including optimization.

RF-CONCRETE Members analyzes the ultimate and serviceability limit states. The analyses for cracks and deflections are performed by calculating crack widths and deformations directly.

Designs are possible according to the following standards:

- EN 1992-1-1:2004/A1:2014
- DIN 1045-1:2008-08
- DIN 1045:1988-07
- ACI 318-19
- ACI 318-14
- ACI 318-11
- CSA A23.3-19
- CSA A23.3-14 (R2015)
- SIA 262:2013
- SIA 262:2017
- GB 50010-2010

CEN	EU
BDS:2011	Bulgaria
BS:2005	United Kingdom
CSN:2016	Czech Republic
CYS:2009	Cyprus
DIN:2015	Germany
DK:2013	Denmark
LST:2011	Lithuania
LVS:2014	Latvia
MS:2010	Malaysia
NBN:2010	Belgium
NEN:2016	Netherlands
NF:2016	France
NP:2010	Portugal
NS:2008	Norway
PN:2010	Poland
SFS:2007	Finland
SingaporeS:2008	Singapore
SIST:2006	Slovenia
SR:2008	Romania
STN:2008	Slovakia
SvenskS:2008	Sweden
TKP:2009	Belarus
UNE:2013	Spain
UNI:2007	Italy
ÖNORM:2018	Austria

National Annexes for EN 1992-1-1

The figure on the left shows the National Annexes for EN 1992-1-1 [1] that are currently implemented in RF-CONCRETE Members. Optionally, the program checks if the requirements of the fire protection design according to EN 1992-1-2:2004 [2] are fulfilled.

The determined required reinforcement contains a reinforcement concept, which takes user specifications for rebars in the longitudinal and shear reinforcement into account. This reinforcement layout can be adjusted at any time. The designs related to the modifications will be updated automatically.

RF-CONCRETE Members also allows for a nonlinear analysis (state II) considering *Tension Stiffening*. In addition, the influence of creep and shrinkage can be determined.

The inserted reinforcement is visualized photo-realistically — both in the add-on module and on the concrete cross-sections of the RFEM model. This realistic representation of the reinforcement cage can be documented in the printout report with the design's remaining input and output data.

We hope you will enjoy working with RF-CONCRETE Members.

Your Dlubal Software team

1.2 Using the Manual

Topics such as installation, graphical user interface, result evaluation, and printout are described in detail in the manual of the main program RFEM. This manual focuses on typical features of the add-on module RF-CONCRETE Members.

The descriptions in this manual follow the sequence and structure of the module's input and result windows. In the text, the described **buttons** are given in square brackets, for example [View Mode]. At the same time, they are pictured on the left. **Expressions** that appear in dialog boxes, tables, windows, and menus are set in *italics* to clarify the explanation.

If you do not find what you are looking for, use the search function of the Knowledge Base to find a solution among the articles. You may also consult the FAQs on our website.

1.3 Starting RF-CONCRETE Members

RFEM provides the following options to start the RF-CONCRETE Members add-on module.

Menu

To start the add-on module, use the RFEM menu item

Add-on Modules → Design - Concrete → RF-CONCRETE Members.

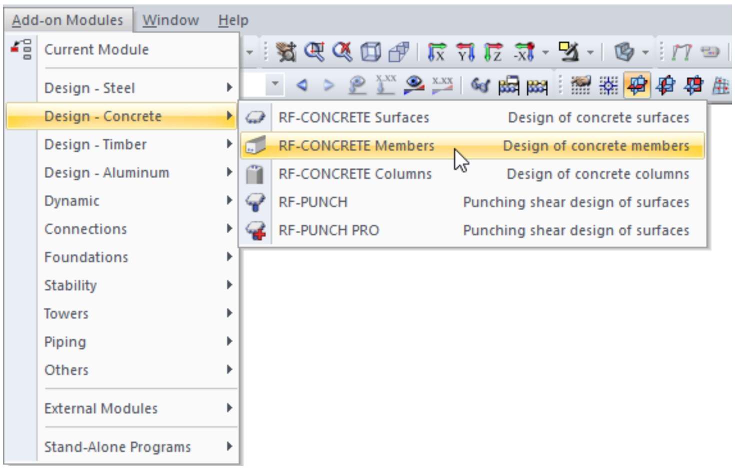


Figure 1.1 *Menu Add-on Modules → Design - Concrete → RF-CONCRETE Members*

Navigator

Alternatively, you can open the add-on module in the *Data* navigator by double-clicking

Add-on Modules → RF-CONCRETE Members.

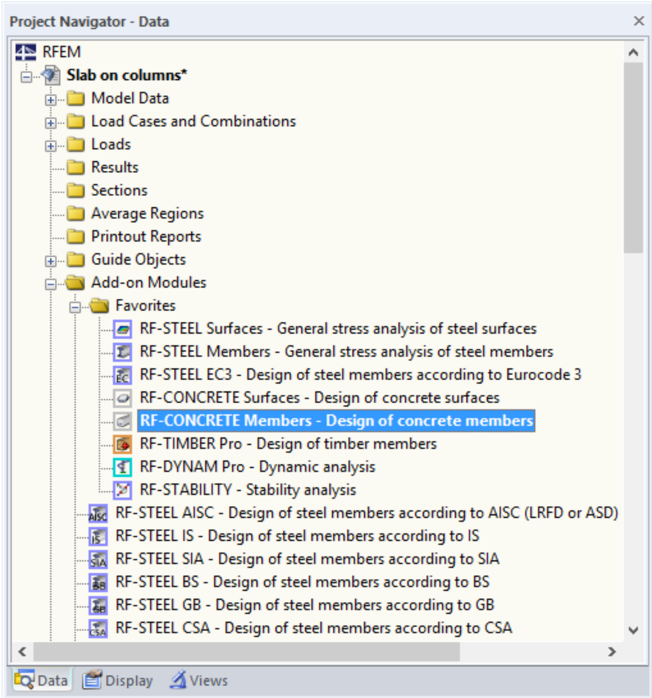
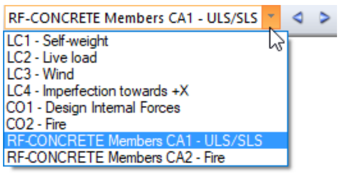


Figure 1.2 Data navigator: *Add-on Modules* → *RF-CONCRETE Members*

Panel



If results from RF-CONCRETE Members are already available in the RFEM model, you can start the design module in the panel:

Set the relevant design case of RF-CONCRETE Members in the load case list of the menu bar. Click the [Show Results] button to display the reinforcements graphically.

You can now click the RF-CONCRETE Members button in the panel to start the add-on module.

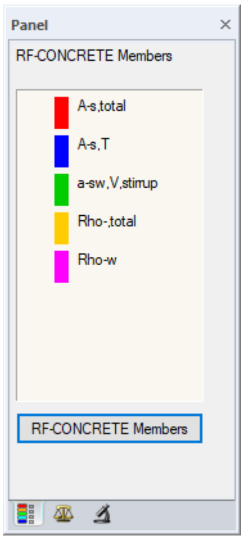


Figure 1.3 [RF-CONCRETE Members] panel button

2 Theoretical Background

2.1 Ultimate Limit State Design

We forgo a detailed description of linear design methods, because the manual is not meant to replace reference books.

2.1.1 Bending and Axial Force

The standards EN 1992-1-1, clause 6.1 or DIN 1045-1, clause 10.2 describe the calculation basis for the ultimate limit state design in detail. These regulations apply to bending with or without axial force, as well as to axial force only.

The mathematical state of failure occurs when the ultimate strains are reached. Depending on where these ultimate strains occur, the failure can be caused by the concrete or the reinforcing steel.

The following figure shows the allowable strain distributions for bending with and without axial force according to EN 1992-1-1, clause 6.1.

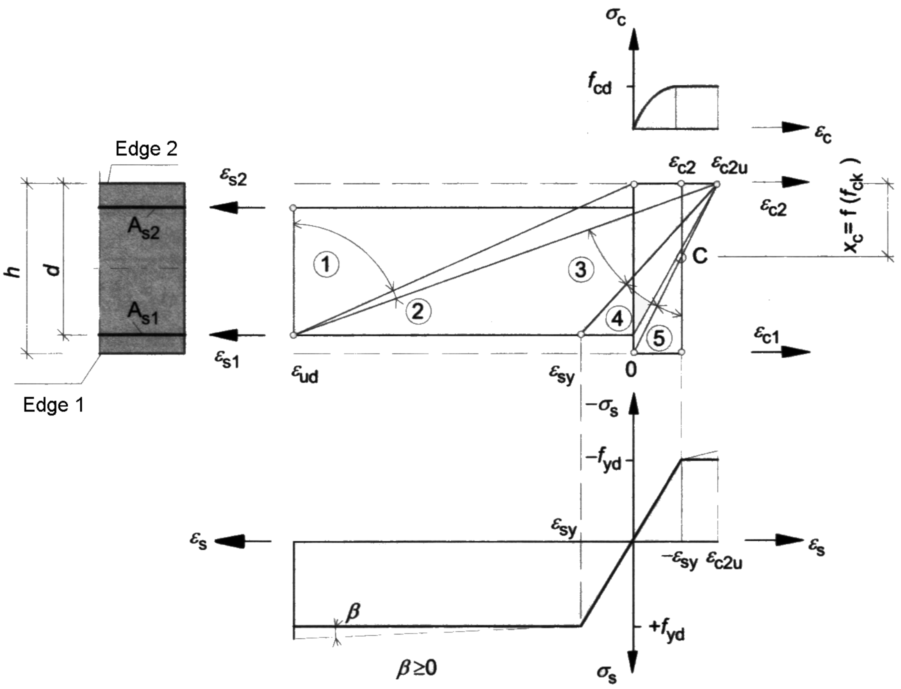


Figure 2.1 Mathematically possible strain distributions in ultimate limit state

According to [3], the areas for strain distributions shown in the figure have the following meaning:

Area 1

This area appears in the case of a central tension force or a tension force with slight eccentricity. Only strains occur on the entire cross-section. The statically effective cross-section consists only of the two reinforcement layers A_{s1} and A_{s2}. The reinforcement fails because the ultimate strain ε_{ud} is reached.

Area 2

Area 2 appears in the case of pure bending and bending with axial force (compression and tension force). The neutral axis lies within the cross-section. The bending-tension reinforcement is fully used, meaning the steel fails when the ultimate strain is reached. Generally, the concrete cross-section is not fully used: The compressive strains do not reach the ultimate strain ε_{c2u}.

Area 3

This area appears only in case of pure bending and bending with axial force (compression). The steel's load-bearing capacity is higher than the load-bearing capacity of the concrete. The concrete fails because its ultimate strain ε_{c2u} is reached.

As in the areas 1 and 2, the concrete's failure is announced by cracks because the steel exceeds the yield point (failure with announcement).

Area 4

Area 4 appears in case of bending with a longitudinal compression force. It represents the transition of a cross-section mainly subjected to bending to a cross-section affected by compression. The concrete fails before the steel's yield point is reached because the possible strains are very small. This area results in a strongly reinforced cross-section. To avoid such a cross-section, a compression reinforcement is inserted.

Small steel strains in the tension zone result in failure without announcement (the bending-tension reinforcement does not start to yield).

Area 5

This area appears in case of a compression force with slight eccentricity (a column, for example) or a centric compression force. On the entire cross-section, only compressive strains occur.

The compressive strain on the edge that is less compressed is between $0 > \varepsilon_{c1} > \varepsilon_{c2}$. All compressive strain distributions intersect in point C.

2.1.2 Shear Force

The check of shear force resistance is to be performed only in the ultimate limit state (ULS). The actions and resistances are considered with their design values. The general design requirement according to EN 1992-1-1, clause 6.2.1 is the following:

$$V_{Ed} \leq V_{Rd}$$

where

V_{Ed} : design value of applied shear force

V_{Rd} : design value of shear force resistance

Depending on the failure mechanism, the design value of the shear force resistance is determined by one of the following three values.

$V_{Rd,c}$: design shear resistance of a structural component without shear reinforcement

$V_{Rd,s}$: design shear resistance of a structural component with shear reinforcement, limited by the yield point of shear reinforcement (failure of tie)

$V_{Rd,max}$: design shear resistance limited by strength of concrete compression strut

If the acting shear force V_{Ed} remains below the value of $V_{Rd,c}$, no calculated shear reinforcement is necessary and the check is verified.

If the applied shear force V_{Ed} is higher than the value of $V_{Rd,c}$, a shear reinforcement must be designed.

The shear reinforcement must resist the entire shear force. In addition, the bearing capacity of the concrete compression strut must be analyzed.

$V_{Ed} \le V_{Rd,s}$ and $V_{Ed} \le V_{Rd,max}$

The various types of shear force resistance are determined according to EN 1992-1-1 as follows.

Design shear resistance without shear reinforcement

The design value for the design shear resistance $V_{Rd,c}$ may be determined with:

$$V_{Rd,c} = \left[C_{Rd,c} \cdot k\,(100\sigma_l \cdot f_{ck})^{\frac{1}{3}} - k_1 \cdot \sigma_{cp}\right] b_w \cdot d$$

Equation 2.1 EN 1992-1-1, Eq. (6.2a)

where

$C_{Rd,c}$	recommended value: 0.18 / γ_c
$k = 1 + \sqrt{\frac{200}{d}} \le 2.0$	scaling factor for considering cross-section depth d : mean static depth in [mm]
$\sigma_l = \frac{A_{sl}}{b_w \cdot d} \le 0.02$	ratio of longitudinal reinforcement A_{sl} : - area of tensile reinforcement, which extends by at least (l_{bd} + d) beyond the considered cross-section
f_{ck}	characteristic value of concrete compressive strength in [N/mm^2]
k_1	recommended value: 0.15
b_w	minimum cross-section width within tension zone in [mm]
d	static effective depth of bending reinforcement in [mm]
$\sigma_{cp} = \frac{N_{Ed}}{A_c} < 0.2 f_{cd}$	design value of concrete longitudinal stress in [N/mm^2]

It is allowed, however, to apply a minimum value of the shear force resistance $V_{Rd,c,min}$.

$$V_{Rd,c,min} = \left[v_{min} + k_1 \cdot \sigma_{cp}\right] \cdot b_w \cdot d$$

Equation 2.2 EN 1992-1-1, Eq. (6.2b)

where

$$v_{min} = 0.035 \cdot \sqrt{k^3 \cdot f_{ck}}$$

Design shear resistance with shear reinforcement

The following applies for structural components with shear reinforcement running perpendicular to the component's axis (α = 90°):

$$V_{Rd,s} = \frac{A_{sw}}{s} \cdot z \cdot f_{ywd} \cdot \cot\theta$$

Equation 2.3 EN 1992-1-1, Eq. (6.8)

where

A_{sw}	cross-sectional area of shear reinforcement
s	spacing of links
z	lever arm of the internal forces assumed with 0.9 d
f_{ywd}	design yield strength of shear reinforcement
θ	inclination of concrete compression strut

The inclination of the concrete compression strut θ may be selected within certain limits depending on the loading. This way, the equation can take into account the fact that a part of the shear force is resisted by crack friction and the virtual truss is thus less stressed. The following limits are recommended in equation (6.7) of EN 1992-1-1:

$1 \leq \cot\theta \leq 2.5$

Thus, the compression strut inclination θ can vary between the following values:

	Minimum inclination	**Maximum inclination**
θ	21.8°	45.0°
cot θ	2.5	1.0

Table 2.1 Recommended limits for inclination of compression strut

Design shear resistance of concrete compression strut

The following applies for structural components with shear reinforcement running perpendicular to the component's axis (α = 90°):

$$V_{Rd,max} = \frac{\alpha_{cw} \cdot b_w \cdot z \cdot v_1 \cdot f_{cd}}{\cot\theta + \tan\theta}$$

Equation 2.4 EN 1992-1-1, Eq. (6.9)

where

α_{cw}	coefficient for considering stress state in compression flange

b_w cross-section width

z lever arm of the internal forces (precisely calculated in bending design)

ν_1 reduction factor for concrete strength in case of shear cracks

f_{cd} design value of concrete strength

θ inclination of concrete compression strut

2.1.3 Shear Forces Between Web and Flanges of T-Beams

The longitudinal shear stress $v_{Ed,f}$ at the junction between flange and web is determined by the longitudinal force difference $\Delta F_{d,f}$ in the flange's governing part according to EN 1992-1-1, clause 6.2.4 (3), equation (6.20).

$$v_{Ed,f} = \frac{\Delta F_{d,f}}{h_f \cdot \Delta x_f}$$

Equation 2.5

where

h_f flange thickness at junction

Δx_f considered length

$\Delta F_{d,f}$ longitudinal force difference in flange over length Δx

The maximum value that may be assumed for the length Δx_f is half the distance between the maximum and the zero point of moments. Where concentrated loads are applied, the distance between the concentrated loads should not be exceeded.

The determination of $\Delta F_{d,f}$ is done optionally with a control available in the module details according to two different methods that are described below.

1. Simplified method via inner lever arm z = 0.9d without considering $M_{z,Ed}$

$$F_{d,i} = \left(\frac{M_{y,Ed}}{z} - \frac{N_{Ed}}{z} \cdot z_s\right) \cdot \frac{b_{eff,i}}{b_{eff}}$$

for compression flanges

$$F_{d,i} = \left(\frac{M_{y,Ed}}{z} - \frac{N_{Ed}}{z} \cdot z_s + N_{Ed}\right) \cdot \frac{A_{s,a}}{A_s}$$

for tension flanges

where

z_s distance between centroid of cross-section and tension reinforcement

z lever arm of internal forces 0.9 d

$b_{eff,i}$ width of adjacent flange (compression flange) or width of reinforcement distribution in adjacent flange (tension flange) considering the *Distribute reinforcement evenly over complete slab width* option (see Figure 3.30)

b_{eff} flange width

A_{sa} reinforcement exposed in connected tension flange

A_s total area of tension reinforcement

2. Calculation of F_d from general stress integration in partial areas of cross-section

The required tension flange reinforcement due to shear forces per unit length a_{sf} may be determined according to equation (6.21).

$$a_{sf} \geq \frac{v_{Ed,f} \cdot h_f}{\cot \theta_f \cdot f_{yd}}$$

where

$1.0 \leq \cot \theta_f \leq 2.0$ inclination of concrete compression strut for compression flanges

$1.0 \leq \cot \theta_f \leq 1.25$ inclination of concrete compression strut for tension flanges

f_{yd} design yield strength of reinforcement

At the same time, the compression struts in the flange must be prevented from failing, which is ensured if the following requirement is met:

$$v_{Ed} \leq v_1 \cdot f_{cd} \cdot \sin \theta_f \cdot \cos \theta_f$$

Equation 2.6 EN 1992-1-1, Eq. (6.22)

where

f_{cd} design value of concrete strength

ν_1 reduction factor for concrete strength in case of shear cracks

2.1.4 Shear Force Transfer in Joints

When concrete components are added retroactively, the transfer of shear force between the different casting zones must be designed. These so-called shear joints occur for concrete structural components of different ages. In such a case, for example, you need to consider joints between precast parts and cast-in-place concrete additions, or connection joints between construction stages during reconstruction, new construction, or renovation.

The shear force transfer should be designed as follows:

$$v_{Edi} \leq v_{Rdi}$$

Equation 2.7 EN 1992-1-1, Eq. (6.23)

There are two possibilities for calculating the shear stress in the shear joint:

1. Calculation from $V_{z,Ed}$ and factor β according to EN 1992-1-1, equation (6.24) without considering $M_{z,Ed}$

In this case, v_{Edi} is the design value of the shear force to be resisted per unit of length in the joint. This value is determined by [1] equation (6.24).

$$v_{Edi} = \frac{\beta \cdot V_{Ed}}{z \cdot b_i}$$

Equation 2.8 EN 1992-1-1, Eq. (6.24)

where

β	quotient from longitudinal force in concrete topping and total longitudinal force in compression or tension zone in considered cross-section
V_{Ed}	design value of applied shear force
z	lever arm of combined cross-section
b_i	width of joint

The design value of the shear resistance v_{Rdi} is determined by the following equation:

$$v_{Rdi} = c \cdot f_{ctd} + \mu \cdot \sigma_n + \rho \cdot f_{yd} \cdot (\mu \cdot \sin \alpha + \cos \alpha) \leq 0.5\nu \cdot f_{cd}$$

Equation 2.9 EN 1992-1-1, Eq. (6.25)

where

c, μ	coefficients that depend on roughness of joint according to [1] 6.2.5 (2)
f_{ctd}	design value of concrete tensile strength according to [1] 3.1.6 (2)P
σ_n	smallest stress perpendicular to joint, which acts simultaneously with shear force (positive for compression) where $\sigma_n < 0.6 \cdot f_{cd}$
ρ	A_s / A_i where A_s cross-sectional area of reinforcement crossing the joint A_i area of connection joint
α	inclination angle of joint reinforcement
ν	strength reduction factor according to [1] 6.2.2 (6)

2. Calculation from difference of axial forces in added concrete part from general stress integration

The rigid bond assumed for the design of shear joints in the ULS should be primarily reached by an adhesive bond, that is, adhesion and micromechanical gearing. Hence, the joint reinforcement is responsible for the transfer of forces after overcoming the rigid bond, as well as for the ductility of the connection, while the shear joint would have to be designed solely for the adhesive bond.

In the current standards, this approach is accommodated only to a minor degree. Though a movable bond is allowed, it is conservatively differentiated and complemented by construction rules, to be on the safe side.

For shear joints, which are designed for the ultimate limit state for a moveable bond as planned, serviceability limit state designs must additionally be performed. For this case, the moveable bond must be consistently included in the determination of the internal forces and stresses in the ULS and SLS.

Self-equilibrating stresses normally involving shear stresses in the joint (due to the varying shrinkage behavior of two concrete components of different ages, for example) are generally not considered. The acting shear force v_{Edi} is solely calculated from internal forces on the cross-section.

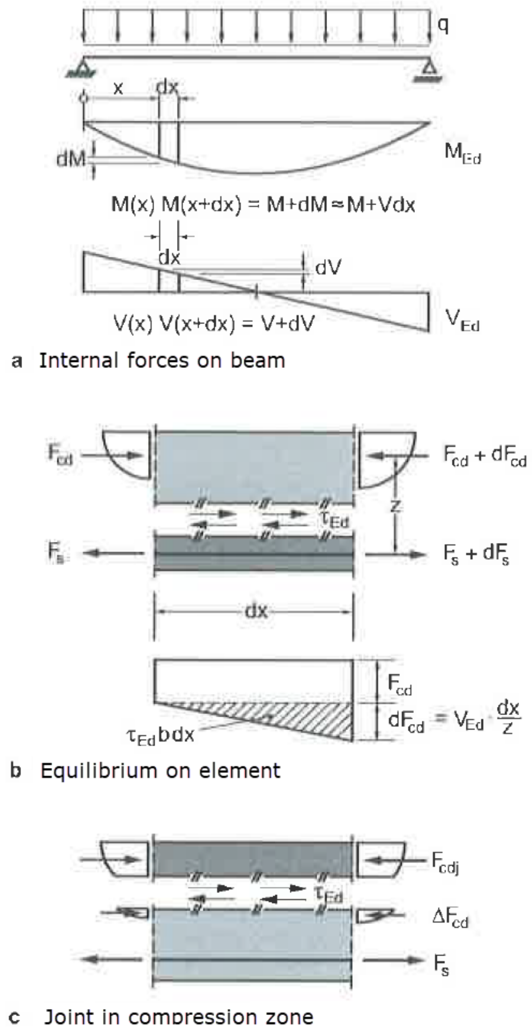


Figure 2.2 Shear stresses in joints according to [4]

Figure 2.2 shows a section of the length *dx* from a beam with a shear joint parallel to the structural component's axis. The variable bending moment causes a change in the flange forces along the length. For example, the following applies to the compression flange:

$$dF_{cd} = \frac{dM_{Ed}}{z} = \frac{V_{Ed}dx}{z}$$

There is an equilibrium between the compression force change and the shear stresses in the joint.

$$\tau_{Ed} = \frac{dF_{cd}}{bdx} = \frac{V_{Ed}dx}{bzdx} = \frac{V_{Ed}}{bz}$$

Thus, for a constant lever arm z, the stress of the shear joint is in proportion to the shear force V_{Ed}, with a constant axial force having no influence on the shear force in the joint parallel to the component axis.

If the shear joint lies within the compression zone, only the portion of the flange force difference between joint and compression flange edge must be transferred. As a result, τ_{Ed} becomes:

$$\tau_{Ed} = \frac{F_{cdi}}{F_{cd}} \cdot \frac{V_{Ed}}{bz}$$

2.2 Serviceability Limit State Design

The serviceability limit state designs consist of various individual designs that are specified, for example, for the Eurocode in the following clauses:

- Stress limitation: EN 1992-1-1, clause 7.2
- Crack control: EN 1992-1-1, clause 7.3
- Deflection control: EN 1992-1-1, clause 7.4

2.2.1 Provided Reinforcement

In the serviceability limit state design, it is first analyzed whether the cross-section can be reinforced, and if the reinforcement proposal can be placed in the cross-section. If this is not the case, no serviceability will be designed for this member.

The serviceability limit state designs are performed with the reinforcement areas available in Window *3.1 Provided Longitudinal Reinforcement*.

2.2.2 Limitation of Stresses

Concrete compressive stresses

The concrete compressive stresses must be limited according to EN 1992-1-1, clause 7.2 (1), in order to avoid cracks or strong creep in case they could affect the structure's function. Therefore, clause 7.2 (2) recommends applying a reduction factor for the characteristic concrete compressive strength.

$$\sigma_c = k_1 \cdot f_{ck}$$

The recommended value for k_1 is 0.6.

Reinforcing steel stresses

To avoid inelastic strains, unallowable cracking, and deformations, the tension stresses in the reinforcement must be limited according to EN 1992-1-1, clause 7.2 (4). Clause 7.2 (5) recommends reduction factors for the characteristic tensile strength, which depend on the type of action combination.

$\sigma_s = k_3 \cdot f_{yk}$ for characteristic action combination

$\sigma_s = k_4 \cdot f_{yk}$ for indirect action (restraint)

The recommended values for k_3 and k_4 are 0.8 and 1.0.

2.2.3 Minimum Reinforcement

The minimum reinforcement area for crack control is determined according to EN 1992-1-1, clause 7.3.2 (2), Eq. (7.1), simplified as follows:

$$A_{s,min} \cdot \sigma_s = k_c \cdot k \cdot f_{ct,eff} \cdot A_{ct}$$

Equation 2.10 EN 1992-1-1, Eq. (7.1)

where

$A_{s,min}$: minimum area of reinforcing steel in tension zone

σ_s : allowable stress of reinforcing steel according to Figure 2.3

k_c : factor for considering stress distribution in tension zone

k_c = 1.0 for pure tension

k_c = 0.4 for bending

For bending with axial force, k_c is determined as follows:

$$k_c = 0.4 \cdot \left[1 - \frac{\sigma_c}{k_1 \cdot (h/h^*) \cdot f_{ct,eff}}\right] \leq 1$$

Equation 2.11 EN 1992-1-1, Eq. (7.2)

where

σ_c : mean concrete stress acting on part of section under consideration
$\sigma_c = N_{Ed} / (b \cdot h)$

N_{Ed} : axial force acting on part of cross-section under consideration

$h^* = h < 1.0$ m

k_1 : coefficient for considering effects of axial forces on stress distribution:

k_1=1.5 for N_{Ed} = compressive force
$k_1 = 2h^*/3h$ for N_{Ed} = tensile force

k : coefficient to consider nonlinearly distributed self-equilibrating stresses

k = 1.0 for webs with h ≤ 300mm

k = 0.65 for h ≥ 800 mm

k = 1.0 for restraint caused outside (e.g. column settlement)

$f_{ct,eff} = f_{ctm}$: mean value of effective tensile strength of concrete when cracks occur

A_{ct} : area of concrete tension zone

Steel stress [MPa]	Maximum bar size [mm]		
	w_k= 0.4 mm	w_k= 0.3 mm	w_k= 0.2 mm
160	40	32	25
200	32	25	16
240	20	16	12
280	16	12	8
320	12	10	6
360	10	8	5
400	8	6	4
450	6	5	-

Figure 2.3 Limit diameter $\varnothing_s^*$ for reinforcing steels according to EN 1992-1-1, Table 7.2

2.2.4 Crack Width Control

Checking rebar diameter

The limit diameter of reinforcing bars with max $\varnothing_s$ is checked in accordance with EN 1992-1-1, clause 7.3.3 (2) as follows.

$$\varnothing_s = \varnothing_s^* \cdot \frac{f_{ct,eff}}{2.9} \cdot \frac{k_c \cdot h_{cr}}{2(h-d)} \quad \text{for bending}$$

$$\varnothing_s = \varnothing_s^* \cdot \frac{f_{ct,eff}}{2.9} \cdot \frac{h_{cr}}{8(h-d)} \quad \text{for uniformly distributed tensile normal stresses}$$

where

$\varnothing_s^*$: limit diameter according to Figure 2.3

$f_{ct,\,eff}$: effective tensile strength of concrete at relevant point of time, in this case f_{ctm}

k_c : factor for considering stress distribution in tension zone, see Chapter 2.2.3

h_{cr} : depth of tension zone immediately before cracking occurs

h : overall depth of cross-section

d : effective depth up to the centroid of outside reinforcement

Design of rebar spacing

The maximum rebar spacing max s_l is specified according to EN 1992-1-1, Table 7.3 (see Figure 2.4).

Steel stress [MPa]	Maximum bar spacing [mm] w_k=0.4 mm	w_k=0.3 mm	w_k=0.2 mm
160	300	300	200
200	300	250	150
240	250	200	100
280	200	150	50
320	150	100	-
360	100	50	-

Figure 2.4 Maximum values for rebar spacings according to EN 1992-1-1, Table 7.3

Design of crack width by direct calculation

The characteristic crack width w_k is determined according to EN 1992-1-1, clause 7.3.4, Eq. (7.8).

$$w_k = s_{r,max} \cdot (\varepsilon_{sm} - \varepsilon_{cm})$$

Equation 2.12 EN 1992-1-1, Eq. (7.8)

where

$s_{r,max}$ maximum crack spacing for final crack state according to Eq. (7.11) or (7.14)

ε_{sm} mean strain of reinforcement considering contribution of concrete to tension between the cracks

ε_{cm} mean strain of concrete between the cracks

Maximum crack spacing $s_{r,max}$

If the rebar spacing in the tension zone is not greater than 5 · (c + Ø/2), the maximum crack spacing for the final crack state may be determined as follows according to EN 1992-1-1, clause 7.3.4 (3):

$$s_{r,max} = k_3 \cdot c + \frac{k_1 \cdot k_2 \cdot k_4 \cdot \varnothing}{\rho_{p,eff}}$$

Equation 2.13 EN 1992-1-1, Eq. (7.11)

where

k_3 recommended value: 3.4 (German National Annex: 0)

c concrete cover of longitudinal reinforcement

k_1 coefficient for considering the bond properties of the reinforcement (0.8 for ribbed steel bars and 1.6 for rebars with a plain surface)

k_2 coefficient for considering strain distribution (0.5 for bending and 1.0 for pure tension)

k_4 recommended value: 0.425 (German National Annex: 1/3.6)

$\rho_{p,eff}$ effective reinforcement ratio

If the spacing of rebars within the bond exceeds 5 · (c + Ø/2) or if there is no reinforcement within the bond in the tension zone, the following limit value of the crack width may be assumed:

$$s_{r,max} = 1.3 \cdot (h - x)$$

Equation 2.14 EN 1992-1-1, Eq. (7.14)

Applying equations (7.11) and (7.14) are "optional" rules within the meaning of the Eurocode. Internal study of these two crack spacing equations has shown that the explicit differentiation when applying equation (7.14) to rebars with a larger spacing than 5 · (c + Ø/2) does not always lead to the desired crack width. We analyzed cross-sections with slightly different rebar spacings in the range of 5 · (c + Ø/2). For T-beam-like cross-sections and a bar spacing of 1.01 · [5 · (c + Ø/2)] using Eq. (7.14), the result was a smaller crack spacing than with Eq. (7.11) and a bar spacing of 0.99 · [5 · (c + Ø/2)]. This would mean that when you increase the reinforcement content, the crack width increases as soon as you fall below the limit value of the rebar spacing 5 · (c + Ø/2). To put it clearly: The calculated crack width in a zone without reinforcement is smaller than in a reinforced zone!

In the program, the crack spacing is calculated using equation (7.11) by default. Optionally, it is possible to activate $s_{r,max}$ as the upper limit according to equation (7.14). As a result of the circumstance described above, the upper limit value is always taken into account, regardless of the available rebar spacing in the tension reinforcement.

Difference of mean strain (ε_{sm} - ε_{cm})

The difference of the mean strain of concrete and reinforcing steel is determined as follows according to [1] 7.3.4 (2), Eq. (7.9).

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_s - k_t \cdot \dfrac{f_{ct,\mathrm{eff}}}{\rho_{p,\mathrm{eff}}} \cdot \left(1 + \alpha_e \cdot \rho_{p,\mathrm{eff}}\right)}{E_s} \geq 0.6 \cdot \frac{\sigma_s}{E_s}$$

Equation 2.15 EN 1992-1-1, Eq. (7.9)

where

σ_s : stress in tension reinforcement assuming a cracked cross-section

k_t : factor for creep of bond

k_t = 0.6 for short-term loading

k_t = 0.4 for long-term loading

$f_{ct,eff}$: effective tensile strength of concrete at relevant point of time (in this case f_{ctm})

α_e : ratio of moduli of elasticity E_s / E_{cm}

ρ_{eff} : effective reinforcement ratio

2.2.5 Limitation of Deformations

EN 1992-1-1, clause 7.4.3 allows for a simplified design of the limitation of deformations via direct calculation. The deflections must be determined realistically: The calculation method has to match the real structural material performance with an accuracy that corresponds to the design purpose.

The deflection is determined by double integration from the differential equation of the bending line. However, as the stiffness of a reinforced concrete cross-section changes in parts due to cracking, the moment-curvature diagram is nonlinear. There are big differences in curvature, and thus in deflection, for uncracked (state I) and cracked sections (state II).

Therefore, the deflection is determined with the principle of virtual work for the location of the maximum deformation. An approximation line is used for the curvature, connecting the extreme values of the curvature with a line that is affine to the moment distribution.

When calculating manually, three values of the deflection are determined according to [3]:

Lower calculation value of deflection

Minimum deflection is achieved when the calculation is performed for a completely uncracked cross-section (state I). This type of deflection is described as f_I.

Upper calculation value of deflection

Maximum deflection is achieved when the calculation is performed for a completely cracked cross-section (state II). This type of deflection is referred to as f_{II}.

Probable value of deflection

It is fair to assume that some parts of the cross-section are uncracked, and other, highly stressed parts are cracked. The moment-curvature relation runs up to the first crack after state I, after which it shows some cracks. This assumption results in the probable value of the deflection f, which lies between the lower and upper calculated value. According to EN 1992-1-1, clause 7.4.3 (3), Eq. (7.18), the value can be derived from the following relation:

$$\alpha = \zeta \cdot \alpha_{II} + (1 - \zeta) \cdot \alpha_1$$

Equation 2.16 EN 1992-1-1, Eq. (7.18)

The values α_I and α_{II} represent general deflection parameters (e.g. f_I or f_{II}). This can be a strain, curvature, deflection, or rotation. ζ is the distribution coefficient between state I and state II, and lies between $0 \leq \zeta < 1$, as shown in EN 1992-1-1, Eq. (7.19). Generally, the deformation calculation is to be performed with a quasi-permanent combination (see EN 1992-1-1, clause 7.4.3 (4)).

Chapter 9.1 describes an example where the manually performed calculation of a deformation analysis is compared with the results of the program.

2.2.6 Creep and Shrinkage

Determination of initial values

This chapter gives an overview of the time-dependent stresses and strains due to creep and shrinkage. The influence of creep and shrinkage is used in the analytical serviceability limit state design for the determination of the deformation. The approach of creep and shrinkage in the nonlinear calculation is described in Chapter 2.4.6.

Creep is the time-dependent deformation of concrete under loading over a specific period of time. The essential influence values are similar to those of shrinkage, with the so-called creep-producing stress having considerable effects on the creep deformation. Special attention must be paid to the load duration, the point of time of load application, as well as to the extent of actions. The creep determining value is the creep coefficient $\varphi\ (t, t_0)$ at the relevant point of time *t*.

Shrinkage describes a time-dependent modification of volume without influence due to external loads or temperature. We will not elaborate on further expansion of the shrinkage problem into individual types (drying shrinkage, autogenous shrinkage, plastic shrinkage, and carbonation shrinkage). Significant influence values of shrinkage are relative humidity, effective thickness of structural components, aggregate, concrete strength, water-cement ratio, temperature, as well as the type and duration of curing. The shrinkage determining value is the shrinkage strain $\varepsilon_{c,s}\ (t, t_s)$ at the relevant point of time *t*.

Hereafter, the determination of the creep coefficient $\varphi\ (t, t_0)$ and shrinkage strain $\varepsilon_{c,s}\ (t, t_s)$ according to EN 1992-1-1, clause 3.1.4 and annex B is described.

Creep coefficient φ (t, t_0)

Using the following formulas requires the creep-producing stress σ_c of the acting permanent load to not exceed the following value:

$$\sigma_c \leq 0.45 \cdot f_{ckj}$$

where

f_{ckj} : cylinder compressive strength of concrete at point of time when creep-producing stress is applied

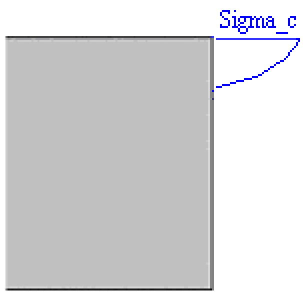


Figure 2.5 Creep-producing stress

Under the assumption of a linear creep behavior ($\sigma_c \leq 0.45 \cdot f_{ckj}$), the concrete's creep can be determined by a reduction of the modulus of elasticity for concrete.

$$E_{c,eff} = \frac{E_{cm}}{1 + \varphi_{eff}(t, t_0)}$$

where

E_{cm} : mean modulus of elasticity according to EN 1992-1-1, Table 3.1

φ_{eff} (t, t_0) : effective creep coefficient, φ_{eff} (t, t_0) = φ (t, t_0) · M_{QP} / M_{Ed}

t : age of concrete in days at relevant point of time

t_0 : age of concrete in days when load application starts

The creep coefficient φ (t, t_0) at the analyzed point of time *t* may be calculated as follows:

$$\varphi(t, t_0) = \varphi_{RH} \cdot \beta(f_{cm}) \cdot \beta(t_0) \cdot \beta(t, t_0)$$

where

$$\varphi_{RH} = \left[1 + \frac{1 - \frac{RH}{100}}{0.1 \cdot \sqrt[3]{h_0}} \cdot \alpha_1\right] \cdot \alpha_2$$

RH : relative humidity in [%]

h_0 : effective thickness of structural component in [mm]

h_0 = 2 • A_c / u

A_c : cross-sectional area

u : cross-section perimeter

α_1, α_2 : adjustment factors

$\alpha_1 = (35 / f_{cm})^{0.7}$

$\alpha_2 = (35 / f_{cm})^{0.2}$

f_{cm} : mean value of cylinder compressive strength

$$\beta(f_{cm}) = \frac{16.8}{\sqrt{f_{cm}}}$$

f_{cm} : mean value of cylinder compressive strength of concrete in [N/mm^2]

$$\beta_{(t_0)} = \frac{1}{0.1 + t_0^{0.20}}$$

t_0 : age of concrete in days when load application starts

$$\beta(t, t_0) = \left[\frac{t - t_0}{\beta_H + t - t_0}\right]^{0.3}$$

t : age of concrete in days at relevant point of time

t_0 : age of concrete in days when load application starts

$\beta_H = 1.5 \cdot [1 + (0.012 \cdot RH)^{18}] \cdot h_0 + 250 \cdot \alpha_3 \leq 1500 \cdot \alpha_3$

RH : relative humidity in [%]

h_0 : effective thickness of structural component [mm]

α_3 : adjustment factor

$\alpha_3 = (35 / f_{cm})^{0.5} \leq 1.0$

The following input is required to calculate the creep coefficient:

- RH : relative humidity in [%]
- t_0 : age of concrete in days when load application starts
- t : age of concrete in days at relevant point of time (optionally ∞)

The influence of high or low temperature ranging from 0 °C to 80 °C on the concrete's maturity can be taken into account by correcting the concrete age with the following equation:

$$t_T = \sum_{i=1}^{n} e^{-\left[\frac{4000}{273 + T(\Delta t_i)} - 13.65\right]} \cdot \Delta t_i$$

where

n	number of periods with identical temperature
T (Δt_i)	temperature in [°C] during time period Δt_i
Δt_i	number of days with this temperature T

The influence of the type of cement on the concrete's creep coefficient can be taken into account by modifying the load application age t_0 with the following equation:

$$t_0 = t_{0,T} \cdot \left(1 + \frac{9}{2 + (t_{0,T})^{1.2}}\right)^{\alpha} \geq 0.5$$

where

$t_{0,T} = t_T$	effective age of concrete when load application starts considering influence of temperature
α	exponent, depends on type of cement, see Table 2.2

α	Type of Cement
−1	slow-hardening cements of class S
0	normal- or rapid-hardening cements of class N
1	rapid-hardening, high-strength cements of class R

Table 2.2 Exponent α

Example

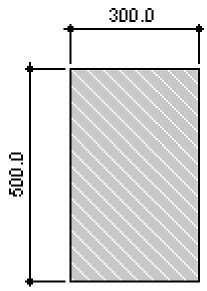


Figure 2.6

concrete C25/30
cement CEM 42.5 N
RH: 50%
Two temperature changes:
6 days - temperature 15 °C
8 days - temperature 7 °C
Considered age of concrete t_k: 365 days

Age of concrete when creeping starts:

$$t_\tau = \sum_{i=1}^{n} e^{-\left[\frac{4000}{273+T(\Delta t_i)} - 13.65\right]} \cdot \Delta t_i = e^{-\left[\frac{4000}{273+T(\Delta t_i)} - 13.65\right]} \cdot 6 + e^{-\left[\frac{4000}{273+T(\Delta t_i)} - 13.65\right]} \cdot 8$$

$$= 8.96 \text{ days}$$

Age of concrete under influence of type of cement:

$$t_0 = t_{0,\tau} \cdot \left(1 + \frac{9}{2 + (t_{0,\tau})^{1.2}}\right)^{\alpha} = 8.96 \cdot \left(1 + \frac{9}{2 + (8.96)^{1.2}}\right)^{0} = 8.96 \text{ days}$$

Effective structural component thicknesses:

$$h_0 = \frac{2 \cdot A_c}{u} = \frac{2 \cdot 0.3 \cdot 0.5}{2 \cdot (0.3 + 0.5)} = 0.1875\, cm$$

Creep coefficient:

$$\varphi(t,t_0) = \varphi_{RH} \cdot \beta(f_{cm}) \cdot \beta(t_0) \cdot \beta_c(t,t_0) = 1.933 \cdot 2.923 \cdot 0.606 \cdot 0.758 = 2.595$$

where

$$\varphi_{RH} = \left[1 + \frac{1 - \frac{RH}{100}}{0.1 \cdot \sqrt[3]{h_0}} \cdot \alpha_1\right] \cdot \alpha_2 = \left[1 + \frac{1 - \frac{50}{100}}{0.1 \cdot \sqrt[3]{187.5}} \cdot 1.042\right] \cdot 1.012 = 1.933$$

$$\alpha_1 = \left(\frac{35}{f_{cm}}\right)^{0.7} = \left(\frac{35}{33}\right)^{0.7} = 1.042$$

$$\alpha_2 = \left(\frac{35}{f_{cm}}\right)^{0.2} = \left(\frac{35}{33}\right)^{0.2} = 1.012$$

$$\beta(f_{cm}) = \frac{16.8}{\sqrt{f_{cm}}} = \frac{16.8}{\sqrt{33}} = 2.923$$

$$\beta_{(t_0)} = \frac{1}{0.1 + t_0^{0.2}} = \frac{1}{0.1 + 8.96^{0.2}} = 0.606$$

$$\beta_c(t,t_0) = \left[\frac{t - t_0}{\beta_H + t - t_0}\right]^{0.3} = \left[\frac{365 - 8.96}{538.779 + 365 - 8.96}\right]^{0.3} = 0.758$$

$$\beta_H = 1.5\,[1 + (0.012 \cdot RH)^{18}] \cdot h_0 + 250 \cdot \alpha_3 =$$
$$= 1.5 \cdot \left[1 + (0.012 \cdot 50)^{18}\right] \cdot 187.5 + 250 \cdot 1.030 = 538.779$$

$$\beta_H \leq 1500 \cdot \alpha_3 = 1500 \cdot 1.030 = 1545$$
$$\alpha_3 = \left(\frac{35}{33}\right)^{0.5} = 1.030$$

Coefficient of shrinkage ε (t, t_s)

When determining the coefficient of shrinkage ε (t, t_s) according to EN 1992-1-1, clause 3.1.4, the shrinkage strain ε_{cs} (t) can be calculated from the sum of the components of the autogenous shrinkage ε_{ca} (t) and drying shrinkage ε_{cd} (t, ts).

$$\varepsilon_{cs}(t) = \varepsilon_{ca}(t) + \varepsilon_{cd}(t, t_s)$$

The autogenous shrinkage strain ε_{ca} at the relevant point of time (t) is determined as follows:

$$\varepsilon_{ca}(t) = \beta_{as}(t) \cdot \varepsilon_{ca}(\infty)$$

where

$$\beta_{as}(t) = 1 - e^{-0.2 \cdot \sqrt{t}}$$

$$\varepsilon_{ca}(\infty) = 2.5 \cdot (f_{ck} - 10) \cdot 10^{-6} \qquad f_{ck} \text{ in } [\mathrm{N/mm^2}]$$

The component from drying shrinkage ε_{cd} is determined as follows:

$$\varepsilon_{cd}(t, t_s) = \beta_{ds}(t, t_s) \cdot k_h \cdot \varepsilon_{cd,0}(f_{cm})$$

where

$$\beta_{ds}(t, t_s) = \frac{t - t_s}{t - t_s + 0.04 \cdot \sqrt{h_0^3}}$$

t age of concrete in days at relevant point of time

t_s age of concrete in days when shrinkage starts

h_0 effective cross-section thickness in [mm] : $h_0 = 2 \cdot A_c / u$

$$\varepsilon_{cd,0} = 0.85 \cdot \left[(220 + 110 + \alpha_{ds1}) \cdot e^{-\alpha_{ds2} \frac{f_{cm}}{f_{cm0}}}\right] \cdot 10^{-6} \cdot \beta_{RH}$$

f_{cm} : mean cylinder compressive strength of concrete in [N/mm^2]

f_{cm0} : 10 N/mm^2

Cement class	Property	α_{ds1}	α_{ds2}
S	slow-hardening	3	0.13
N	normal-hardening	4	0.12
R	rapid-hardening	6	0.11

Table 2.3 α_{ds1} and α_{ds2}

$$\beta_{RH} = 1.55 \cdot \left[1 - \left(\frac{RH}{RH_0}\right)^3\right]$$

RH relative humidity of environment in [%]

RH_0 100 %

Example

concrete C25/30

cement CEM 42.5 N

RH: 50 %

Age of concrete t_s when shrinkage starts: 28 days

Considered age of concrete t: 365 days

Effective cross-section thickness:

$$h_0 = \frac{2 \cdot A_c}{u} = \frac{2 \cdot 0.3 \cdot 0.5}{2 \cdot (0.3 + 0.5)} = 0.1875\,\text{m}$$

Autogenous shrinkage:

$$\varepsilon_{ca}(t) = \beta_{as}(t) \cdot \varepsilon_{ca}(\infty) = 0.978 \cdot 0.0000375 = 0.0000367$$

where

$$\beta_{as}(t) = 1 - e^{-0.2t^{0.5}} = 1 - e^{-0.2 \cdot \sqrt{365}} = 0.978$$
$$\varepsilon_{ca}(\infty) = 2.5 \cdot \left(f_{ck} - 10\right) \cdot 10^{-6} = 2.5 \cdot (25 - 10) \cdot 10^{-6} = 0.0000367$$

Drying shrinkage:

$$\varepsilon_{cd}(t, t_s) = \beta_{ds}(t, t_s) \cdot k_h \cdot \varepsilon_{cd,0} = 0.766 \cdot 0.87 \cdot 0.000512 = 0.000341$$

where

$$\beta_{ds}(t, t_s) = \frac{t - t_s}{t - t_s + 0.04 \cdot \sqrt{h_0^3}} = \frac{365 - 28}{365 - 28 + 0.04 \cdot \sqrt{187.5^3}} = 0.766$$

$$h_0 = 187.5\,\text{mm} \Rightarrow k_h = 0.87$$

$$\varepsilon_{cd,0} = 0.85 \cdot \left[\left(220 + 110 \cdot \alpha_{ds1}\right) \cdot e^{-\alpha_{ds2} \frac{f_{cm}}{f_{cm0}}} \right] \cdot 10^{-6} \cdot \beta_{RH} =$$

$$= 0.85 \cdot \left[(220 + 110 \cdot 4) \cdot e^{-0.12 \frac{33}{10}} \right] \cdot 10^{-6} \cdot 1.356 = 0.000512$$

$$\beta_{RH} = 1.55 \cdot \left[1 - \left(\frac{RH}{RH_0} \right)^3 \right] = 1.55 \cdot \left[1 - \left(\frac{50}{100} \right)^3 \right] = 1.356$$

Cement class N $\Rightarrow \alpha_{ds1} = 4$; $\alpha_{ds2} = 0.12$

Total coefficient of shrinkage:

$$\varepsilon(t,t_s) = \varepsilon_{cd}(t,t_s) + \varepsilon_{ca}(t) = 0.0000367 + 0.000341 = 0.000378 = 0.378\,‰$$

2.3 Fire Resistance Design

The fire resistance design in RF-CONCRETE Members is performed by using the simplified calculation method according to EN 1992-1-2, clause 4.2. The program uses the *zone method* described in annex B.2:

In case of exposure to fire, the bearing capacity is reduced due to a reduction of the component's cross-section and a decrease of material strengths. The concrete zones that are directly exposed to fire and thus damaged are not taken into account for the equivalent cross-section used for the fire resistance design. The fire protection design is performed with the reduced cross-section and the reduced material properties analogous to the ultimate limit state design at normal temperature.

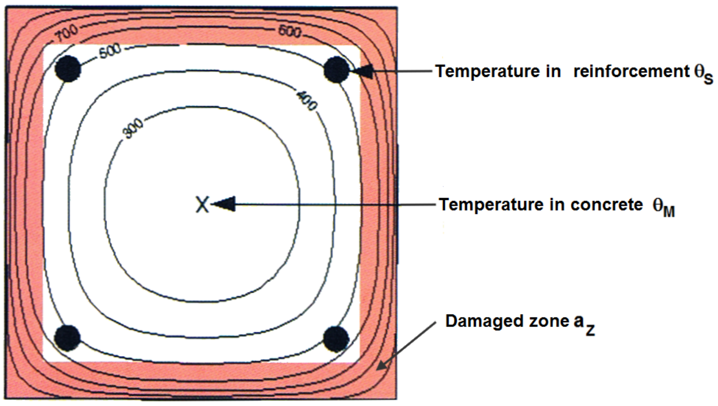


Figure 2.7 Cross-section exposed to fire with damaged zones

2.3.1 Subdivision of Cross-Section

The cross-section is subdivided into a certain number of parallel (n ≥ 3) zones of the same thickness. For each zone, the program determines the mean temperature, the corresponding compressive strength $f_{c,\theta}$, and, if necessary, the modulus of elasticity.

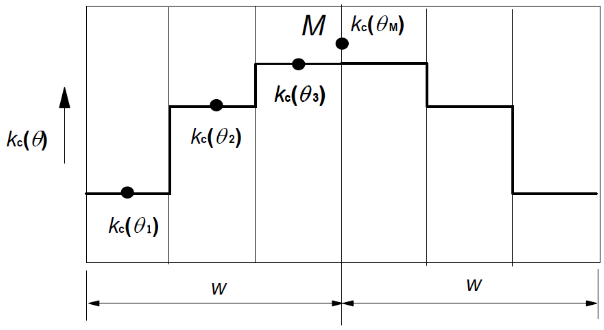


Figure 2.8 Subdivision of wall with both sides exposed to fire into zones according to [2], Figure B.4

The cross-section exposed to fire is compared to an equivalent wall. The width of the equivalent wall is 2 · w. The equivalent width is subdivided symmetrically in several zones as shown in Figure 2.8.

Half of the equivalent width w depends on the fire load acting on the structural component. The following Table 2.4 gives an overview of the determination of equivalent widths conforming to the standard.

Fire load	Half of equivalent width w
Structural component with one side exposed to fire	Structural component width in direction of fire effects
Column or wall with both sides (facing each other) exposed to fire	0.5 · structural component width in direction of fire effects
Column with four sides exposed to fire	0.5 · smaller cross-section dimension

Table 2.4 Determination of equivalent widths

2.3.2 Reduction of Cross-Section

Determination of temperature θ_i in center of zone

After the cross-section's subdivision into zones, the temperature θ_i in the center of each *i* zone is determined. This occurs based on the temperature courses according to EN 1992-1-2, Annex A, which are based on the following assumptions:

- The specific heat of concrete corresponds to the specifications according to EN 1992-1-2, clause 3.2.2.
- The moisture is 1.5 % (the specified temperatures are on the safe side for moistures > 1.5 %).
- The thermal conductivity of concrete is the lower limit value mentioned in EN 1992-1-2, clause 3.3.3.
- The emission value for the concrete surface is 0.7.
- The convective heat-transmission coefficient is 25 W/m^{2}K.

Determination of reduction factor k_c (θ_i)

The reduction factor k_c (θ_i) is determined for the temperature found in the *i* zone's center in order to take the decrease of the characteristic concrete compressive strength f_{ck} into account. This reduction factor k_c (θ_i) depends on the concrete's aggregates:

According to EN 1992-1-2, Figure 4.1, graph 1 is to be used for normal concrete with aggregates containing quartz, and graph 2 for normal concrete with aggregates containing limestone.

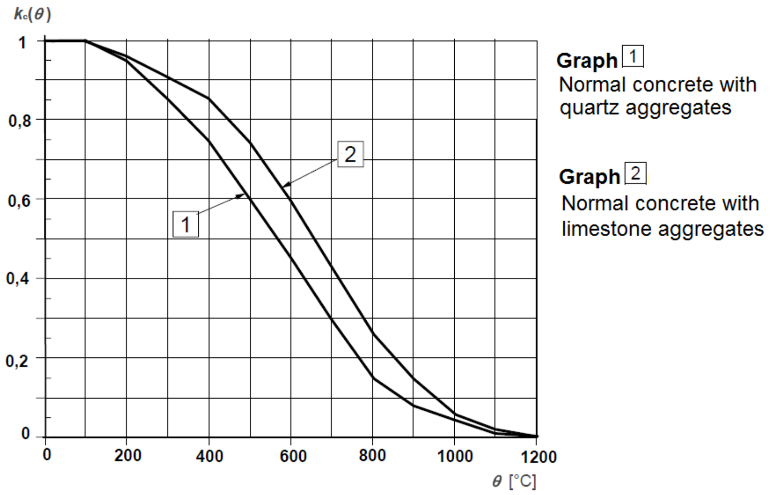


Figure 2.9 Factor k_c (θ_i) for considering decrease of concrete compressive strength according to [2] Figure 4.1

Determination of damaged zone with thickness a_z

The cross-section damaged by fire is represented by a reduced cross-section. Consequently, a damaged zone of thickness a_z on the sides exposed to fire is not taken into account for the ultimate limit state design.

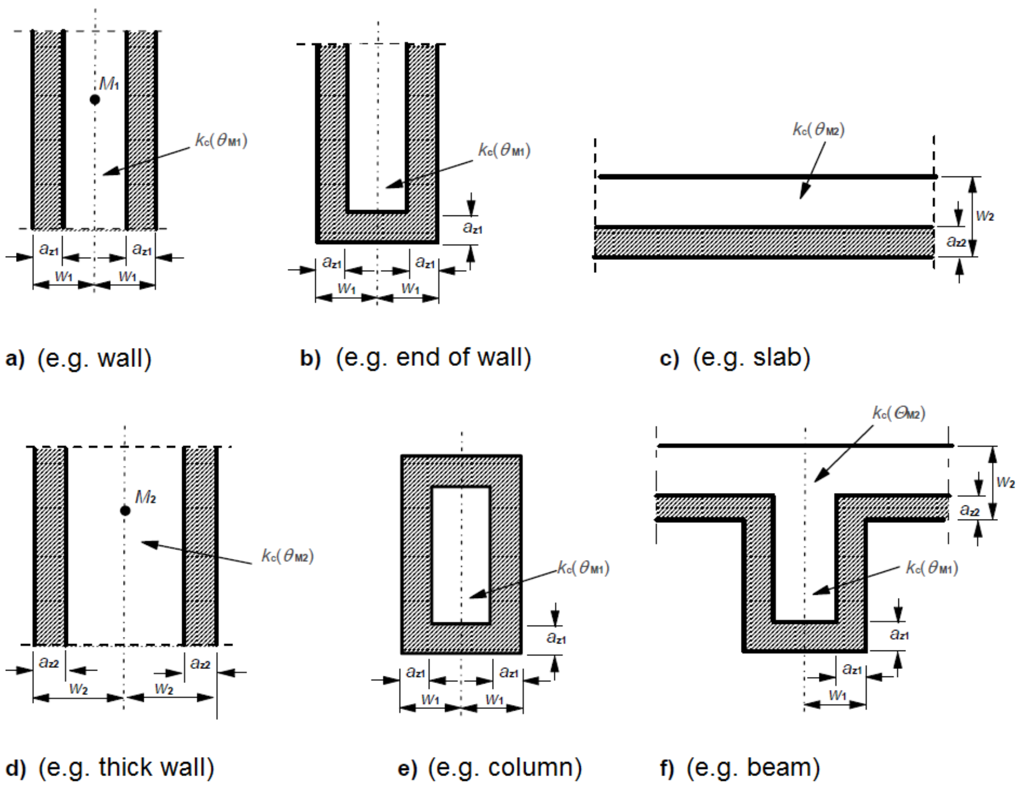


Figure 2.10 Reduction of strength and cross-section in case of fire exposure according to [2] Figure B.3

The calculation of the damaged zone thickness a_z depends on the structural component type:

- **Beams, plates**

$$a_z = w \cdot \left[1 - \frac{k_{c,m}}{k_c(\theta_M)}\right]$$

- **Columns, walls**, and other structural components for which effects due to the second-order analysis must be taken into account

$$a_z = w \cdot \left[1 - \left(\frac{k_{c,m}}{k_c(\theta_M)}\right)^{1.3}\right]$$

where

w	half the width of the equivalent wall
$k_{c,m}$	mean reduction coefficient for a specific cross-section

$$k_{c,m} = \frac{1 - \frac{0.2}{n}}{n} \sum_{i=1}^{n} k_c(\theta_i)$$

n : number of parallel zones in w

The temperature change in each zone is taken into account with the factor (1 - 0.2/n).

$k_c(\theta_M)$	reduction coefficient for concrete at point M (see Figure 2.9)

2.3.3 Stress-Strain Curve of Concrete

Point M, a point on the central line of the equivalent wall (cf. Figure 2.8), is governing for the reduction of the concrete's material properties. It is used to determine the reduction factor k_c (θ_M). The concrete's reduced material properties are to be used for the entire reduced cross-section (without damaged zone a_z) for the ultimate limit state design in case of fire.

Compressive strength of concrete for fire resistance design

The stress-strain curve for the concrete's compressive strength is determined depending on the temperature in point M and the type of aggregates. The values of the compressive strain $\varepsilon_{cu1,\theta}$ for the compression strength $f_{c,\theta}$ can be found in EN 1992-1-2, Table 3.1.

$$f_{c,\theta} = k_c(\theta_M) \cdot f_{ck}$$

where

$k_c(\theta_M)$: reduction coefficient for concrete at point M (see Figure 2.9)

f_{ck} : characteristic compressive strength of concrete at normal temperature

Concrete temp. θ	Quartz aggregates $f_{c,\theta} / f_{ck}$	Quartz aggregates $\varepsilon_{c1,\theta}$	Quartz aggregates $\varepsilon_{cu1,\theta}$	Limestone aggregates $f_{c,\theta} / f_{ck}$	Limestone aggregates $\varepsilon_{c1,\theta}$	Limestone aggregates $\varepsilon_{cu1,\theta}$
[°C]	[–]	[–]	[–]	[–]	[–]	[–]
1	2	3	4	5	6	7
20	1,00	0,0025	0,0200	1,00	0,0025	0,0200
100	1,00	0,0040	0,0225	1,00	0,0040	0,0225
200	0,95	0,0055	0,0250	0,97	0,0055	0,0250
300	0,85	0,0070	0,0275	0,91	0,0070	0,0275
400	0,75	0,0100	0,0300	0,85	0,0100	0,0300
500	0,60	0,0150	0,0325	0,74	0,0150	0,0325
600	0,45	0,0250	0,0350	0,60	0,0250	0,0350
700	0,30	0,0250	0,0375	0,43	0,0250	0,0375
800	0,15	0,0250	0,0400	0,27	0,0250	0,0400
900	0,08	0,0250	0,0425	0,15	0,0250	0,0425
1 000	0,04	0,0250	0,0450	0,06	0,0250	0,0450
1 100	0,01	0,0250	0,0475	0,02	0,0250	0,0475
1 200	0,00	–	–	0,00	–	–

Figure 2.11 Parameters of stress-strain relation for concrete in case of fire according to [2] Table 3.1

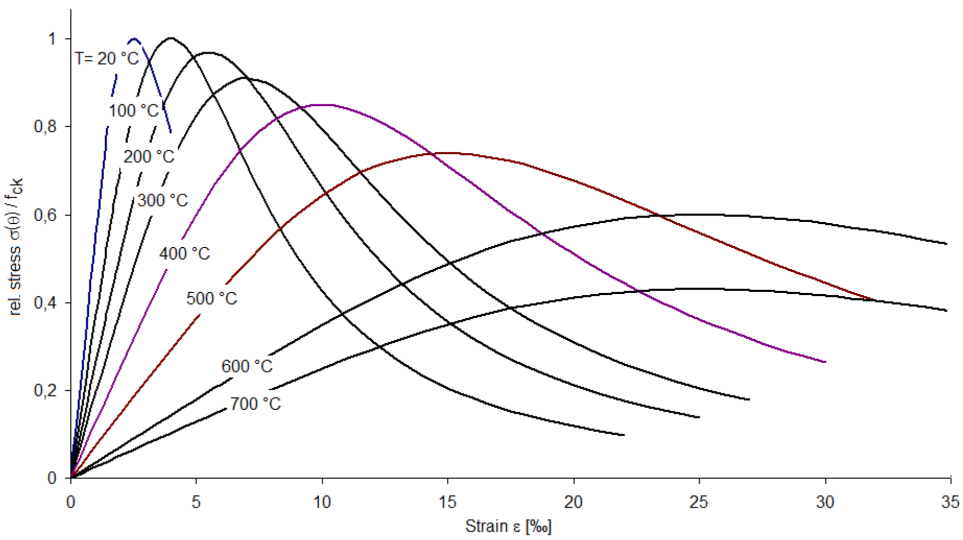


Figure 2.12 Stress-strain diagram for concrete with aggregates containing limestone, dependant on temperature

The diagram (Figure 2.12) shows how the stress-strain relation of normal concrete with aggregates containing limestone changes depending on the temperature. The graph's descending branch is not taken into account for the fire resistance design.

The concrete's reduced modulus of elasticity is determined for the fire protection design according to the following equation:

$$E_{cd,\theta} = \left[k_c(\theta_M)\right]^2 \cdot E_c$$

where

$k_c(\theta_M)$: reduction coefficient for concrete at point M (see Figure 2.9)

E_c : modulus of elasticity of concrete at normal temperature (20 °C)

Tensile strength of concrete for fire resistance design

Being on the safe side, the concrete's tensile strength is not applied for either the cross-section design or the fire protection design. For the sake of completeness, however, the values can be found in the description of the material properties (cf. Chapter 3.2).

According to [2] Figure 3.2, the tensile strength of concrete is generally to be reduced for the fire resistance design:

$$f_{ck,t}(\theta) = k_{c,t}(\theta_M) \cdot f_{ck,t}$$

where

$k_{c,t}\ (\theta_M)$: reduction coefficient for concrete tensile strength according to Figure 2.13

$f_{ck,t}$: characteristic tensile strength of concrete at normal temperature (20 °C)

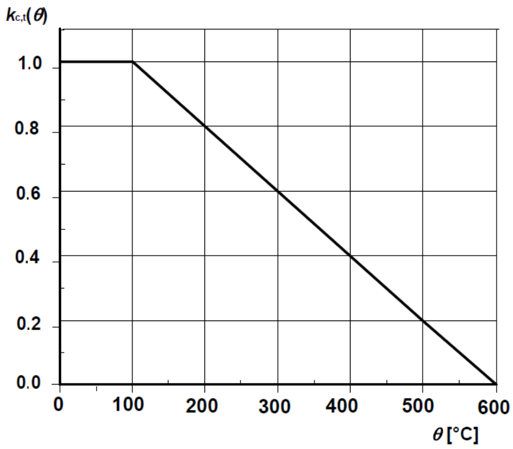


Figure 2.13 Reduction factor $k_{c,t}(\theta)$ for considering temperature-dependent tensile strength of concrete f_{ct} according to [2] Figure 3.2

2.3.4 Stress-Strain Curve of Reinforcing Steel

Reduction factor k_s (θ) for tensile strength of steel

To determine the reduction factor k_s (θ), the temperature in the center of the most unfavorable reinforcing member must be determined first. Depending on how the reinforcing steel is produced and classified (class N or class X), and how much it is strained, the reduction factor k_s (θ) is determined.

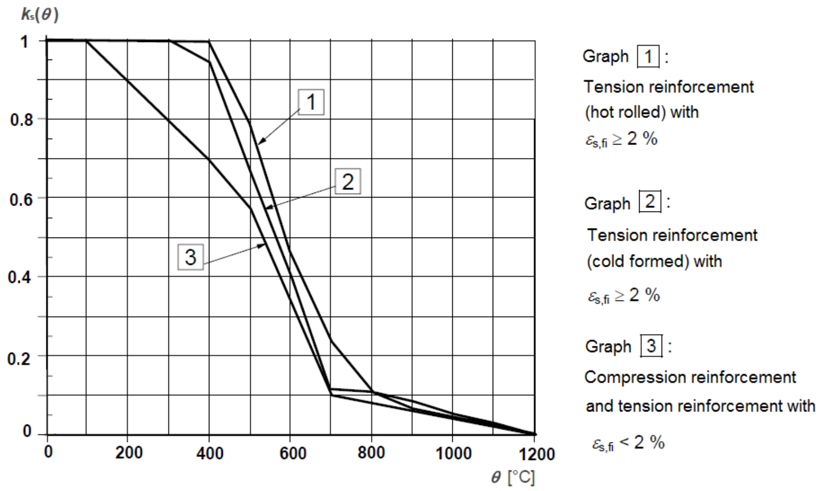


Figure 2.14 Class N – reduction factor k_s (θ) according to [2] Figure 4.2a

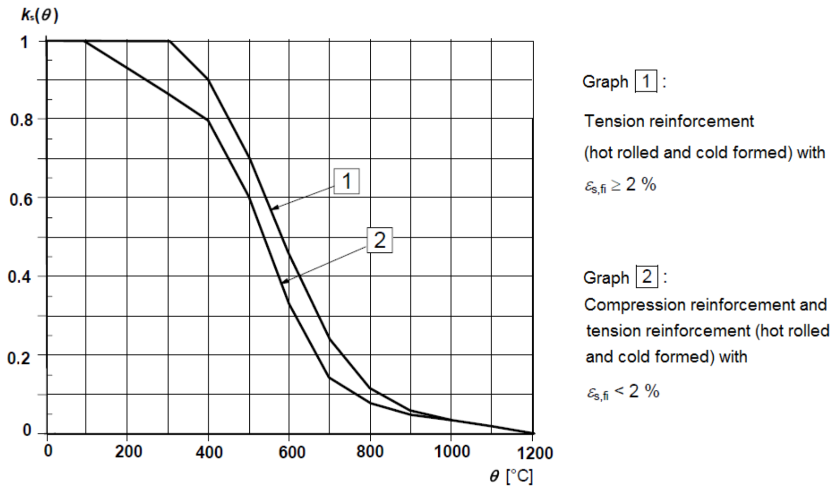


Figure 2.15 Class X – reduction factor k_s (θ) according to [2] Figure 4.2b

Reduction of reinforcing steel strength $f_{sy,\theta}$

The stress-strain relation of reinforcing steel is defined by the following parameters:

- slope in linear-elastic range $E_{s,\theta}$
- proportionality limit $f_{sp,\theta}$
- maximum stress level $f_{sy,\theta}$

The maximum reinforcing steel strength to be applied in the fire design is determined as follows:

$$f_{sy,\theta} = k_s(\theta) \cdot f_{yk}$$

where

k_s (θ) : reduction coefficient for reinforcing steel (see Figure 2.14 or Figure 2.15)

f_{yk} : characteristic strength of reinforcing steel at normal temperature

Determination of reduced modulus of elasticity $E_{s,\theta}$ of reinforcing steel

If the reinforcing steel can be assigned to graph 1 or graph 2 of Figures 4.2a or 4.2b in EN 1992-1-2 (cf. Figure 2.14 and Figure 2.15), it is possible to take the reduced modulus of elasticity of the reinforcing steel depending on the reinforcing steel temperature and the steel's type of production from EN 1992-1-2, Table 3.2a or 3.2b. They are displayed in the following figures.

Steel temperature θ [°C]	$f_{sy,\theta} / f_{yk}$		$f_{sp,\theta} / f_{yk}$		$E_{s,\theta} / E_s$	
	hot rolled	cold formed	hot rolled	cold formed	hot rolled	cold formed
1	2	3	4	5	6	7
20	1.00	1.00	1.00	1.00	1.00	1.00
100	1.00	1.00	1.00	0.96	1.00	1.00
200	1.00	1.00	0.81	0.92	0.90	0.87
300	1.00	1.00	0.61	0.81	0.80	0.72
400	1.00	0.94	0.42	0.63	0.70	0.56
500	0.78	0.67	0.36	0.44	0.60	0.40
600	0.47	0.40	0.18	0.26	0.31	0.24
700	0.23	0.12	0.07	0.08	0.13	0.08
800	0.11	0.11	0.05	0.06	0.09	0.06
900	0.06	0.08	0.04	0.05	0.07	0.05
1 000	0.04	0.05	0.02	0.03	0.04	0.03
1 100	0.02	0.03	0.01	0.02	0.02	0.02
1 200	0.00	0.00	0.00	0.00	0.00	0.00

Figure 2.16 Class N – parameters of stress-strain relation of steel according to [2] Table 3.2a

Steel temperature θ [°C]	$f_{sy,\theta} / f_{yk}$ hot rolled and cold formed	$f_{sp,\theta} / f_{yk}$ hot rolled and cold formed	$E_{s,\theta} / E_s$ hot rolled and cold formed
20	1.00	1.00	1.00
100	1.00	1.00	1.00
200	1.00	0.87	0.95
300	1.00	0.74	0.90
400	0.90	0.70	0.75
500	0.70	0.51	0.60
600	0.47	0.18	0.31
700	0.23	0.07	0.13
800	0.11	0.05	0.09
900	0.06	0.04	0.07
1 000	0.04	0.02	0.04
1 100	0.02	0.01	0.02

Figure 2.17 Class X – parameters of stress-strain relation of steel according to [2] Table 3.2b

For reinforcing steels assigned to graph 3 according to EN 1992-1-2, Figure 4.2a, the reduced modulus of elasticity is calculated as follows:

$$E_{sy,\theta} = k_s(\theta) \cdot E_s$$

where

k_s (θ) : reduction coefficient for reinforcing steel (see Figure 2.14 or Figure 2.15)

E_s : modulus of elasticity of reinforcing steel at normal temperature (20 °C)

2.4 Nonlinear Design

EN 1992-1-1 allows for a nonlinear determination of internal forces in the ultimate and serviceability limit states. The internal forces and deformations are determined while considering the nonlinear behavior of internal forces and deformations (physical).

2.4.1 Method

The principle for the analysis of nonlinear problems is presented by describing an example of uniaxial bending. To determine the nonlinear diagram for deformation of internal forces, the finite element method is used with constant equivalent stiffnesses that are element-by-element. For this reason, the selected division of elements has a significant influence on both the results and the calculation's convergence.

The following flowchart shows the general procedure of a nonlinear calculation.

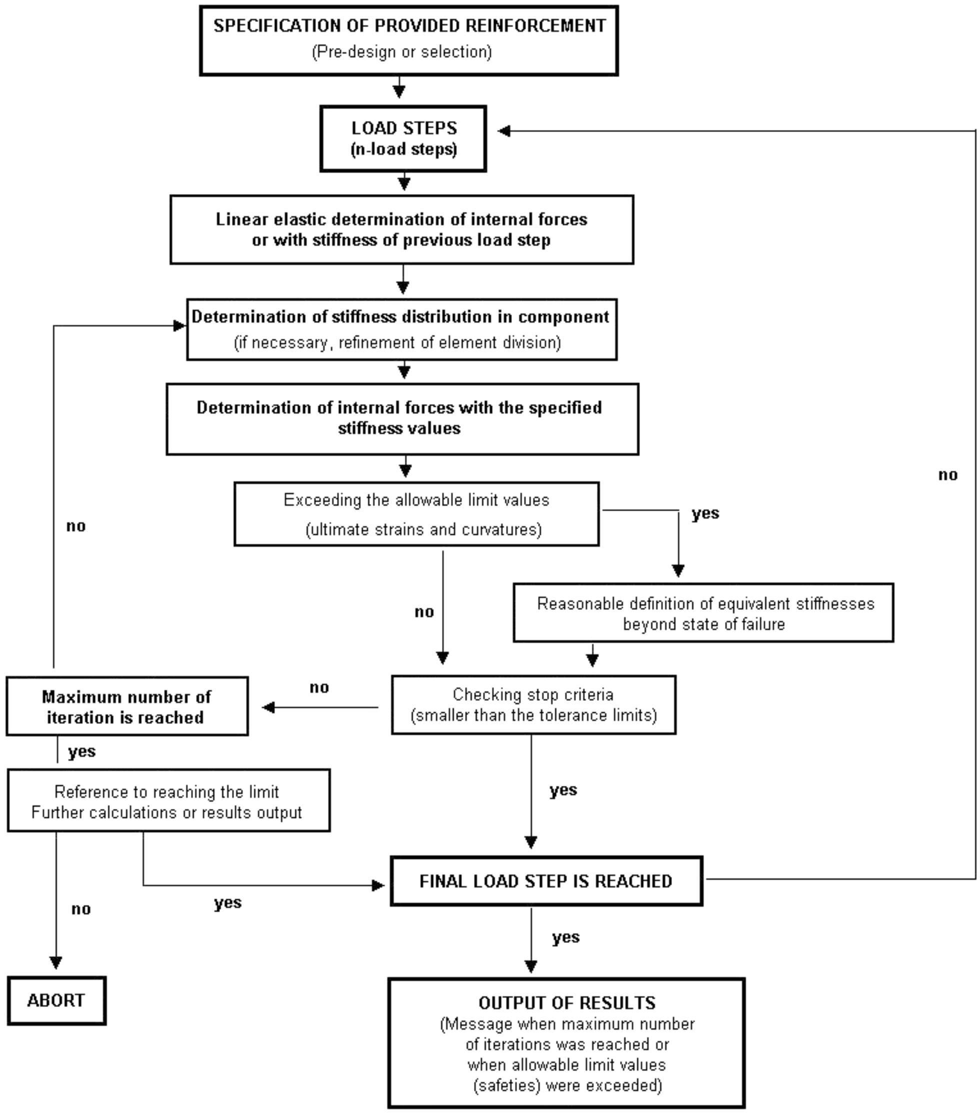


Figure 2.18 General calculation process of nonlinear design

The individual steps are described in the following chapters.

2.4.2 Strain and Curvature

This chapter describes the determination of significant parameters on the cross-section level. The description is reduced to a simple rectangular cross-section affected by uniaxial bending. The advantage is that the moment-curvature (axial force) relation, which mirrors the stiffness development depending on the loading the most clearly, is specified completely. The moment-curvature diagram is therefore dependent on the cross-section's loading due to axial force.

Chapter 2.4.7.1 and Chapter 2.4.8 describe the material properties that are applied for the ultimate and serviceability limit states in detail.

The following essential relations exist between strain and curvature:

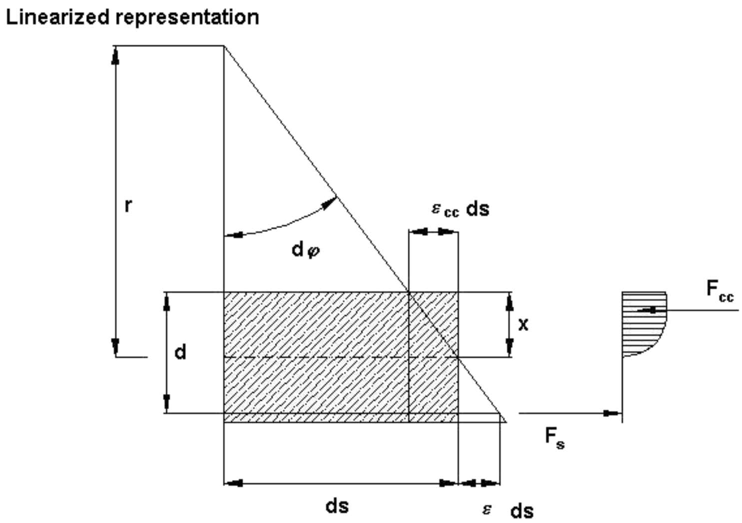


Figure 2.19 Relation between strain and curvature on infinitesimal element

The following conditions result based on the relations shown above.

$$d_{\varphi} \approx \tan(d\varphi) = \frac{ds}{r}$$

$$d_{\varphi} \approx \tan(d\varphi) = \frac{\varepsilon_s \cdot ds - \varepsilon_{cc} \cdot ds}{d} = \frac{\varepsilon_s - \varepsilon_{cc}}{d} ds$$

The following results by equating

$$\left(\frac{1}{r}\right) = \frac{\varepsilon_s - \varepsilon_{cc}}{d}$$

where

ε_{cc} : negative for compressive strain of concrete

Taking the linear elastic material behavior as a basis, the relation between moment and curvature for uncracked sections (state I) is as follows.

$$\left(\frac{1}{r}\right) = \frac{M}{E \cdot I}$$

For cracked sections (state II), the direct affinity between the course of the moment graph and curvature graph is lost. The value E · I (flexural resistance of secant) depends on the loading and is thus no longer constant where identical geometric boundary conditions are given.

The following figure illustrates the basic difference between secant and tangent stiffness.

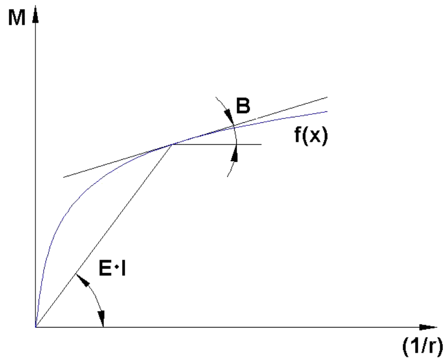


Figure 2.20 Secant stiffness (**E** · **I**) and tangent flexural stiffness (**B**)

When calculating deformations, the approach depends strongly on the used method. In [5], Quast points out the advantages of using the transfer matrix method applying the approach of the tangential flexural resistances (for area-by-area linearization $(1/r)_0 + M/B_{II}$). This may be very practical in regards to the mentioned method or for "manual calculations" when deformations or release rotations are to be determined with the principle of virtual work.

When the finite element method is used, calculation based on constant equivalent stiffnesses is recommended. In order to also determine the nonlinear diagram of the cross-section's moment-curvature relation in the area where abrupt changes of the tangential flexural stiffness occur with a sufficient level of detail, a finer division in such transition zones (M_{cr}; M_y) is required. This is done in the program's background by limiting the differences in stiffness of adjacent elements.

2.4.3 Tension Stiffening

When parts of the reinforced concrete are cracked, we know from the design in the ultimate limit state that the tension forces occurring in the crack must be absorbed by the reinforcement only. Between two cracks, however, tension stresses are transferred into the concrete by means of the (movable) bond. Thus, in relation to the length of the structural component, the concrete participates in the absorption of internal tension forces, which leads to increased structural component stiffness. This effect is called *effectiveness of concrete for tension between cracks* or *Tension Stiffening*.

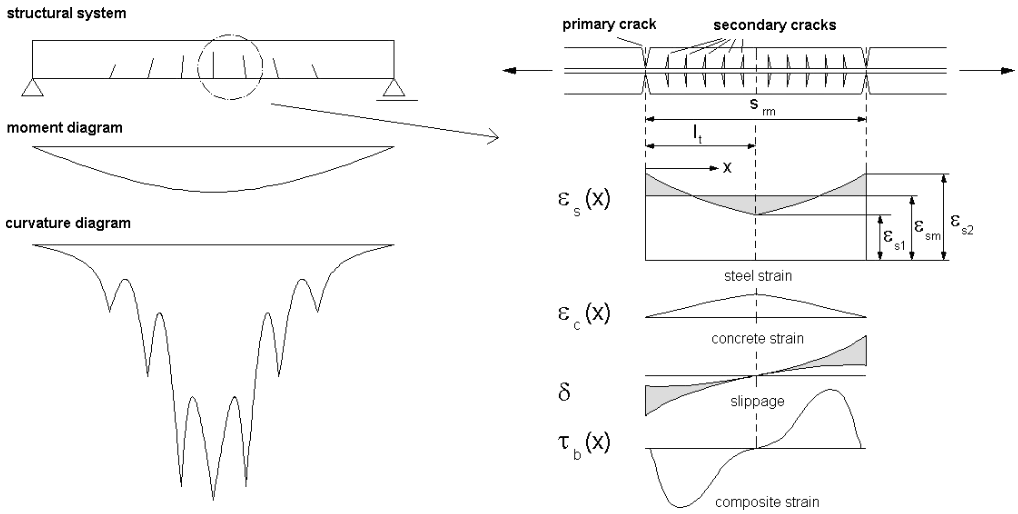


Figure 2.21 Stress and strain behavior between two primary cracks

This increase in the structural component stiffness due to tension stiffening can be considered in two ways:

- After the crack formation, a remaining constant residual tension stress is represented in the concrete's stress-strain diagram. The residual tension stress is notably smaller than the concrete's tensile strength. Alternatively, it is possible to introduce modified stress-strain relations for the tension zone, which consider the concrete's effect on tension between cracks in the form of a descending branch in the graph after the tensile strength is reached. This procedure often proves to be sensible for numerical calculations.
- The approach that is clearer and more conventional for practical designs is the modification of the "pure" stress-strain diagram of steel. A reduced steel strain ε_{sm} is applied in the considered cross-section, resulting from ε_{s2} and a reduction term due to tension stiffening.

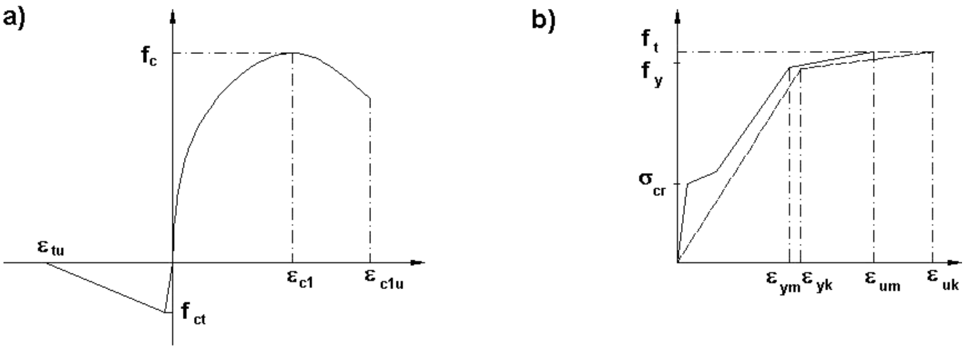


Figure 2.22 Considering the tension stiffening effect via a) characteristic concrete curve or b) modified characteristic steel curve

In RF-CONCRETE Members, it is possible to consider the effect of *Tension Stiffening* by means of a modified characteristic curve for steel according to [6], as well as through a stress-strain curve for concrete in the tensile zone according to [7] and [8].

Advantages and disadvantages of these approaches and the functional application of the individual methods are described in detail in corresponding reference books (for example [8]).

2.4.3.1 Model: Tensile Strength of Concrete

This model that is used to determine the effectiveness of concrete on tension between cracks is based on a defined stress-strain curve of concrete in the tension zone (parabola-rectangle diagram). The mathematical tensile strength is **no** fixed value but refers to the given strain in the governing steel (tension) fiber. The approach has been taken up in accordance with the specifications in [7] to the effect that the maximum tensile strength f_{ctR} decreases linearly to zero, starting at the defined crack strain until a yield strain ε_{sy} is reached in the governing steel fiber.

In several research projects (i.a. [8]), efforts were made to refine or modify the approach of Quast and adjust it on the basis of evaluated experiments.

The following figure illustrates the schematic approach.

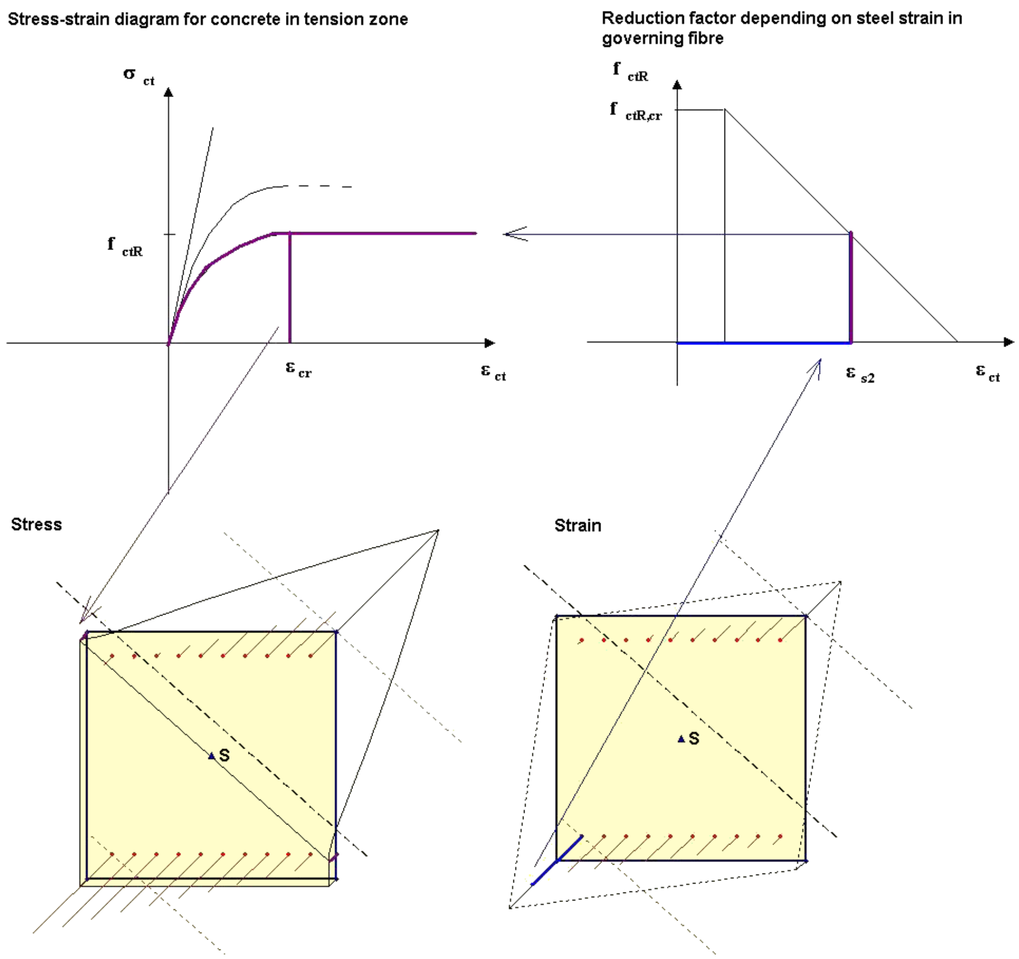


Figure 2.23 Calculation of residual tensile strength for tension-stiffening model according to Quast

The parabola-rectangle diagram for the tensile zone is determined according to the following formal relations:

$$f_{ct,R} = \alpha_{red} \cdot f_{ct,basic}$$

$$v = \frac{f_{ct}}{f_{ct,R}}$$

$$\varepsilon_{cr} = \left| \frac{\varepsilon_{c1}}{v} \right|$$

$$n_{ct} = 1.05 \cdot E_{ctm} \cdot \frac{\varepsilon_{cr}}{f_{ct,R}}$$

$$\sigma_{ct} = f_{ct,R} \cdot \frac{\varepsilon_{sy} - \varepsilon_{s2}}{\varepsilon_{sy} - \varepsilon_{cr}} \quad \text{with } \varepsilon_{cr} \leq \varepsilon_{s2} \leq \varepsilon_{sy}$$

where

α_{red} : reduction factor of basic value of tensile strength

$f_{ct,basic}$: basic value of tensile strength (e.g. f_{ctm})

$f_{ct,R}$: calculational tensile strength v : ratio of compressive to tensile strength

ε_{cr} : calculational strain when $f_{cr,R}$ is reached

n_{ct} : exponent of parabola in tension zone

$\sigma_{ct,R}$: calculational stress depending on governing strain of steel fibre

ε_{sy} : calculational yield strain

ε_{s2} : strain of governing steel fiber

2.4.3.2 Modified Characteristic Steel Curve

The *Tension Stiffening* effect can also be taken into account by means of a modified characteristic steel curve. The approximate minor tangential stiffness (abrupt change in case of re-cracking) during crack development is determined by distinguishing between crack formation and final crack state.

Stress-strain curve of steel

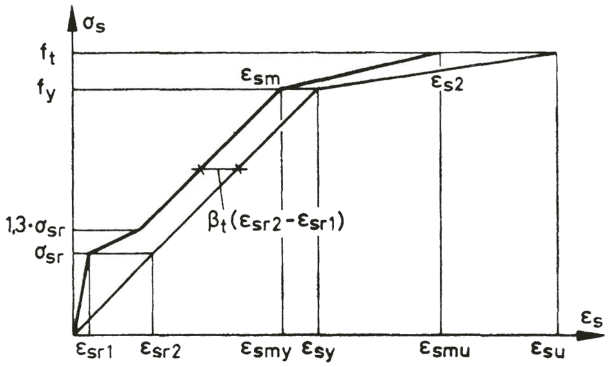


Figure 2.24 Modified stress-strain curve of reinforcing steel according to [6]

Explanation

Uncracked — state I ($0 < \sigma_s \leq \sigma_{sr}$)

$$\varepsilon_{sm} = \varepsilon_{s1}$$

State of first crack formation ($\sigma_{sr} < \sigma_s \leq 1.3\sigma_{sr}$)

$$\varepsilon_{sm} = \varepsilon_{s2} - \frac{\beta_t(\sigma_s - \sigma_{sr}) + (1.3\,\sigma_{sr} - \sigma_s)}{0.3\,\sigma_{sr}}(\varepsilon_{sr2} - \varepsilon_{sr1})$$

State of final crack state ($1.3\,\sigma_{sr} < \sigma_s \leq f_y$)

$$\varepsilon_{sm} = \varepsilon_{s2} - \beta_t(\varepsilon_{sr2} - \varepsilon_{sr1})$$

Plastic steel yielding until failure ($f_y < \sigma_s \leq f_t$)

$$\varepsilon_{sm} = \varepsilon_{sy} - \beta_t(\varepsilon_{sr2} - \varepsilon_{sr1}) + \delta_d\left(1 - \frac{\sigma_{sr}}{f_y}\right)(\varepsilon_{s2} - \varepsilon_{sy})$$

Descriptions:

ε_{sm} : mean steel strain

ε_{su} : ultimate strain at failure of reinforced steel

ε_{s1} : steel strain in uncracked state

ε_{s2} : steel strain in cracked state (in crack)

ε_{sr1} : steel strain in uncracked state with crack internal forces

ε_{sr2} : steel strain in crack with crack internal forces

β_t : factor for considering loading period or load repetitions

0.40 short-term loading

0.25 permanent load or frequent load changes

σ_{sr} : stress in tension reinforcement, calculated based on cracked cross-section for action combination resulting in first crack formation

σ_s : steel stress in cracked state (in crack) in [N/mm^2]

σ_d : factor for considering ductility of reinforcement

0.8 highly ductile steel

0.6 normally ductile steel

2.4.4 Mean Moment-Curvature Relation

The mean moment-curvature relation describes the relation between moment and curvature by taking the concrete's tension stiffening effect into account. By means of discrete conditions of strain (curvatures), it is possible to determine a corresponding moment. On the basis of the ultimate strain at failure, the ultimate curvature is generally divided varyingly depending on the task. The disadvantage of this approach is that it requires a very fine division in order to also represent the transition zones for significant yield points. By connecting the respective single points, you get a continuous (polygonal) line as the characteristic moment-curvature diagram. The diagram curve is also affected by or dependent on the acting axial force. However, in most practical situations, it is sufficient to apply a moment-curvature relation linearized in particular areas.

RF-CONCRETE Members determines the stiffness in a process-related way (double bending, no constant axial force) on every element node directly from the internal force of the previous iteration. One of the differences between the two approaches of Tension Stiffening is that in the approach by Quast, the mean stiffness arises directly from the stress calculation. In contrast, in the approach with the modified characteristic steel curve, the mean curvature is to be determined separately once more, which may lead to certain losses in velocity depending on geometry and system.

For compression elements, we generally have to use the model by Quast [7] to consider the concrete's effectiveness. The reason is the simplified calculation in the uncracked state for the model via the modified characteristic steel curve (see Chapter 2.4.3.2 and Chapter 4.2.2).

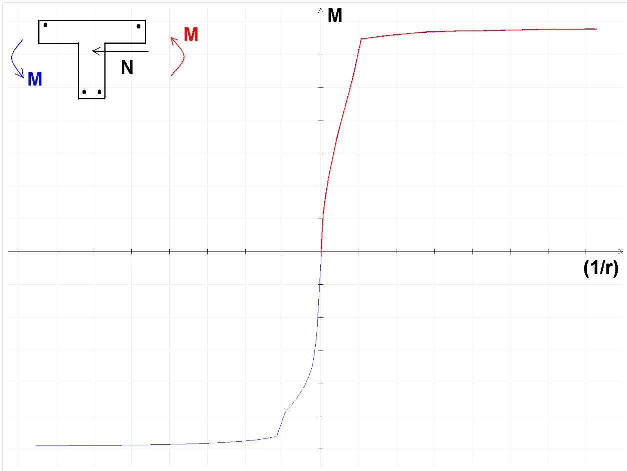


Figure 2.25 Examplary diagram of moment-curvature relation

2.4.5 Determination of Element Stiffnesses

2.4.5.1 Bending Stiffness

As described in Chapter 2.4.1, the calculation is based on constant element-by-element equivalent stiffnesses. For this approach, is important to have a very fine division in zones of significant stiffness changes (tearing, yielding). If the element division is too coarse, the stiffness conditions may be misinterpreted for some parts and stiffnesses may oscillate. In this case, the FE mesh must be refined in RFEM to prevent these effects.

Another important aspect for minimizing a non-convergence of statically indeterminate systems is damping the stiffness change. A change of stiffnesses occurring too abruptly may lead to the iteration "breaking off", especially in cases where the stiffness decreases strongly due to cracking and yielding (slightly reinforced cross-sections).

The following figure illustrates the relation between moment, curvature, and stiffness.

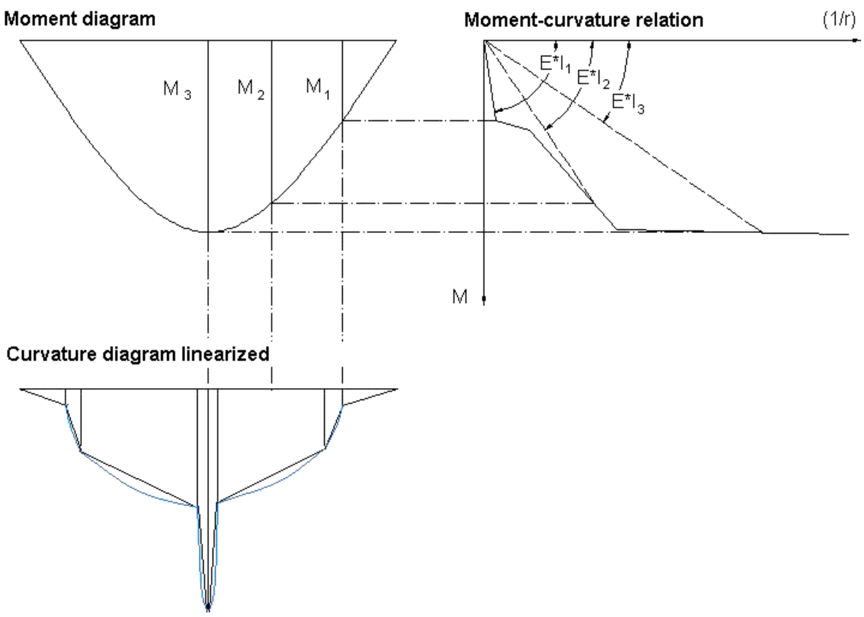


Figure 2.26 Schematic representation of relation between moment, curvature, and stiffness

In accordance with the relations described above, the load-dependent secant stiffness arises according to the following equation:

$$E \cdot I(x) = \frac{M(x)}{(1/r)(x)}$$

2.4.5.2 Longitudinal, Shear and Torsional Stiffness

The determination of the bending stiffness as the input parameter for the nonlinear calculation is described in previous chapters. The remaining stiffness parameters can be determined as follows.

Longitudinal stiffness

Similar to the procedure for bending, the longitudinal stiffness E · A is determined from the ratio of the strain ε_0 to the acting axial force. When bending moment and axial force occur at the same time, it is no longer possible to apply this relation directly because this would result in negative stiffnesses in particular areas, provided that the approach is performed consistently. This results from the simplified analysis not considering the shifting of the neutral axis for strain. When performing nonlinear calculations, this axis no longer matches the centroid of the cross-section. Generally, it is possible to take this fact into account by uncoupling the stiffness matrix from the centroid. However, this will result in a direct correlation between moment and axial force in the terms of the stiffness matrix. RF-CONCRETE Members does not consider axis strain due to crack formation or physical nonlinearity.

Looking at the relation between axial force and bending moment, we can see a direct correlation between both stiffness terms. To clarify, imagine a column with a constant compression force: If an increasing moment acts in addition to the axial force, a curvature leading to a displacement of the resulting axial force from the centroid is added to the pure constant strain diagram. Seen from a plastic point of view, the effective area of the resultant force is thus reduced as well, which by necessity leads to larger strains and thereby to decreasing stiffnesses. Therefore, the approximate consideration for an affinity between bending stiffness and strain stiffness in case of bending with axial force is a practical solution.

Shear stiffness

Determining the shear stiffness in detail is very difficult for the design of reinforced concrete structures, and is an endeavor that is barely manageable in regards to various geometry and load arrangements. The beam theory quickly reaches its limits because the bearing capacity should be determined by the truss effect in order to represent the stiffness for moderate shear loading. In the past, such models were used to develop different methods which are generally not or only partially sufficient in their application.

In a simplified method, Pfeiffer [8] reduces the shear stiffness in accordance with the available bending stiffness. Even if this approach seems to be somewhat strange at first, it is the result of a basic idea that is quite simple and plausible. Imagine that bending load and shearing stress are independent values. When looking at the modified loading of moment and axial force, the bending stiffness changes according to the strain and curvature diagram. However, this does not only affect the stiffness in the beam's longitudinal direction, but also in transversal direction used to transfer shear forces.

This approach is meant as an approximation, which assumes a sufficient shear capacity, but does not (or only roughly) determine slanting cracks, increase of tension force, etc. In spite of these simplifications, the method according to Pfeiffer for moderately slender beams can be considered a sufficiently accurate approach. Alternatively, we can also take the linear elastic shear stiffness as a basis for the calculation in RF-CONCRETE Members.

Torsional stiffness

Compared to bending stiffness, torsional stiffness is reduced strongly in case of cracking. On one hand this is positive, as torsion moments from restraint which frequently occur in building construction are almost completely reduced for load increments until failure is reached. On the other hand, there is the so-called equilibrium torsion where the strong decrease of the torsional stiffness may already lead to remarkable torsions in the serviceability state and thus to a reduction of the serviceability.

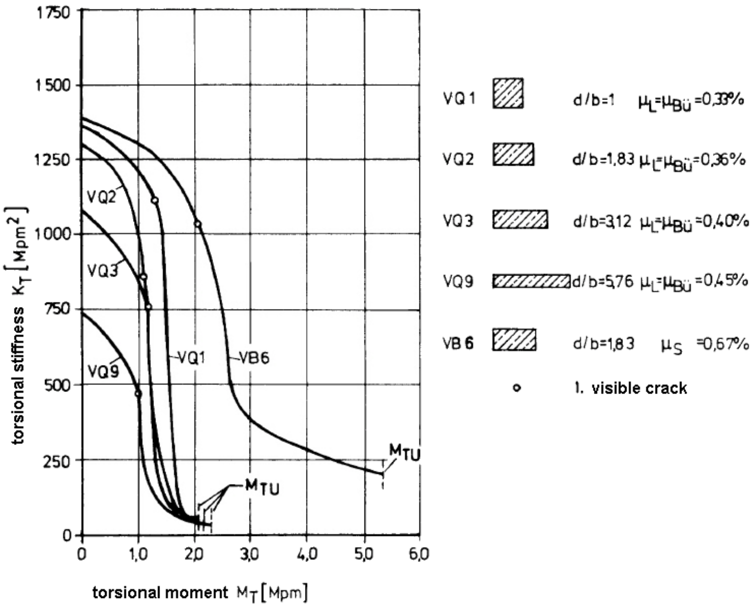


Figure 2.27 Figure from [9] about decrease of torsional stiffness in case of cracking

There are two different approaches for considering torsional stiffness available for the calculation with RF-CONCRETE Members.

Torsional stiffness according to Leonhardt [9]

Torsional stiffness in uncracked sections (state I)

For the torsional stiffness in state I, the program takes into account that the stiffness is reduced by 30 and 35 % until the crack moment is reached. Reasons indicated by Leonhardt are the following: The concrete core escapes the loading and the stresses are displaced to the outside. A micro crack formation is also involved in the reduction to some extent.

$$\left(G_c \cdot I_T(x)\right)_I = 0.8 \cdot G_c \cdot I_{T,0}(x) \quad \text{as mean value}$$

where

I_T : torsional constant

G_c : shear modulus

Torsional stiffness in cracked sections (state II)

The torsional stiffness in state II is derived from a spatial truss model. For simplification, we can assume the inclination of the compression strut to be below 45°. According to Leonhardt, this assumption is also true when the ratios of the longitudinal and transverse reinforcement are not equal. Minor strut inclinations result from the equilibrium analysis or from the design assumption, if the reinforcement ratio of the links is less than the one of the longitudinal reinforcement. However, tests showed that the assumed planer inclination of cracks only occurs for high stress.

Tests also showed that the truss model provides a good algorithm for determining the torsional stress for the limit of failure. However, for the serviceability state we can observe that the steel stresses in the shear and longitudinal reinforcement do not reach the values according to the truss analogy even after several load repetitions.

Link inclinations of 90°:

$$\left(G_c \cdot I_T(x)\right)_{II} = \frac{4\,E_c \cdot A_k^3}{u_k^2} \cdot \frac{1}{k_T\left(\frac{1}{\mu_L} + \frac{1}{\mu_{Li}}\right) + \frac{4\alpha \cdot A_k}{u_k \cdot t} \cdot (1+\varphi)}$$

Link inclinations of 45°:

$$\left(G_c \cdot I_T(x)\right)_{II} = \frac{E_c \cdot A_k^2 \cdot t}{u_k^2} \cdot \frac{1}{\frac{k_T}{\mu_{Li}} + \frac{\alpha}{4} \cdot (1+\varphi)}$$

where

$$k_T = 1 - \frac{T_{ed} - 0.7 \cdot T_{cr}}{T_{Rd,sy} - 0.7 \cdot T_{cr}}$$ for compression strut inclination of 90°

$$k_T = 1 - \frac{T_{Ed} - 0.9 \cdot T_{cr}}{T_{Rd,sy} - 0.9 \cdot T_{cr}}$$ for compression strut inclination of 45°

$$\mu_L = \frac{A_{sl}}{A_k}$$ longitudinal reinforcement ratio related to kern

$$\mu_{Li} = \frac{\alpha_{sw} \cdot u_k}{A_k}$$ transverse reinforcement ratio related to kern

$$T_{Rd,sy} = \min \begin{cases} A_{sw}/s_w \cdot f_y \cdot 2 \cdot A_k \\ A_{sl}/u_k \cdot f_y \cdot 2 \cdot A_k \end{cases}$$

Determination of crack moment for solid cross-section:

Start: $f_{ctr1} = 0.55 \cdot f_{ck}^{2/3}$

End: $f_{ctr2} = 0.65 \cdot f_{ck}^{2/3}$

Determination of crack moment for hollow cross-section:

Start: $f_{ctr1} = 0.45 \cdot f_{ck}^{2/3}$

End: $f_{ctr2} = 0.55 \cdot f_{ck}^{2/3}$

$T_{Rd,sy}$ torsional moment for which steel stress in truss model reaches yield point (torsional moment that can be absorbed)

T_{cr} torsional moment for transition to state II (crack moment)

$$\min \begin{cases} W_T \cdot f_{ctr1} \\ 2 \cdot A_k \cdot t \cdot f_{ctr1} \end{cases}$$

A_k area enclosed by center line of walls

A_{sl} cross-sectional area of longitudinal reinforcement

A_{sw} cross-sectional area of shear reinforcement

α ratios of moduli of elasticity E_s / E_c

u_k perimeter of area A_k

s_w spacing of links

t effective thickness of wall

φ creeping coefficient to consider

A mutual influence of torsional and bending stiffness is not effected.

Global reduction of torsional stiffness

As an alternative, it is possible to calculate with a linear elastic torsional stiffness that is reduced on a percentage basis in the cracked area.

2.4.6 Creep and Shrinkage

Determining the coefficients for creep φ (t,t0) and shrinkage $\varepsilon_{c,s}$ (t,ts) according to EN 1992-1-1, Annex B is described in Chapter 2.2.6.

Creep and shrinkage in the model are considered by calculation as described below.

Creeping

If the strains are known at the point of time t = 0 as well as any later point of time t, the factor for creeping φ_t can be specified as follows.

$$\varphi_t = \frac{\varepsilon_t}{\varepsilon_{t=0}} - 1$$

The equation is converted to the strain at the point of time t. This results in the following correlation, which is valid for uniform stresses (less than approx. 0.4 f_{ck}).

$$\varepsilon_t = \varepsilon_{t=0} \cdot (\varphi_t + 1)$$

For stresses higher than approx. 0.4 f_{ck}, the strains increase disproportionately, resulting in the loss of the linearly assumed reference.

The calculation in RF-CONCRETE Members uses a common solution that is reasonable for construction purposes: The concrete's stress-strain curve is distorted by the factor (1+ φ).

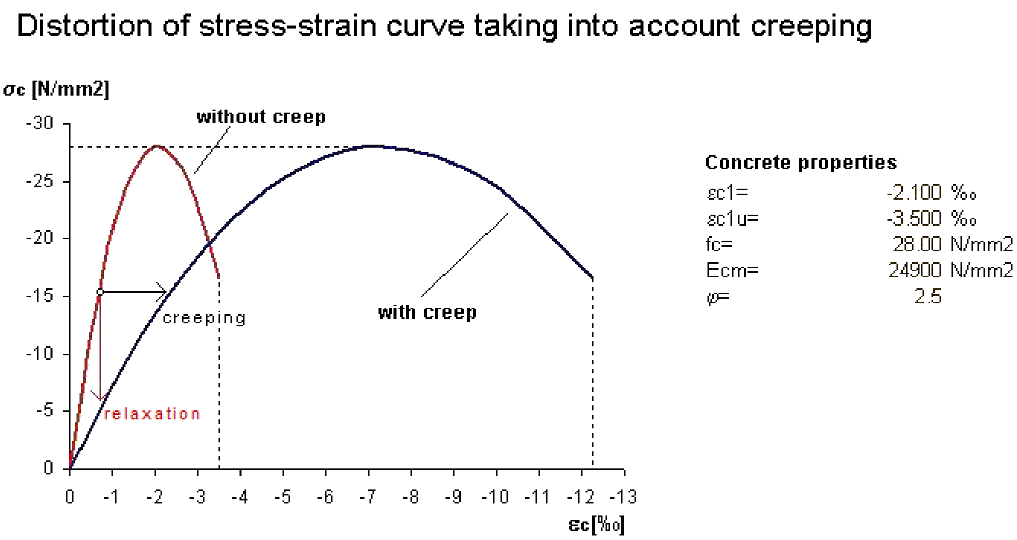


Figure 2.28 Distortion of stress-strain relation to determine creep effect

When taking account of creeping, uniform creep-producing stresses are assumed during the period of load application as shown in the figure above. Due to the neglect of stress redistributions, the deformation is slightly overestimated by this approach. In addition, this model comprises stress reduction only in parts as the change in strains (relaxation) is not taken into account: If we assume a linear elastic behavior, it would be possible to presume a proportionality and the horizontal distortion would also reflect the relaxation at a ratio of (1+ φ). This context, however, is lost for the nonlinear stress-strain relation.

This procedure thus represents an approximation. A reduction of stresses due to relaxation as well as nonlinear creep cannot or can only approximately be represented.

The creep coefficient φ_t applied in RF-CONCRETE Members is to be considered an effective creep coefficient. For calculations in the ultimate limit state, this means that the ratio of creep-producing and acting load must be taken into account. Therefore, the creep coefficients determined according to Chapter 2.2.6 must be adjusted as shown in the following equation.

$$\varphi_{t,eff} = \frac{\text{creep}-\text{producing load}}{\text{acting load}} \cdot \varphi_t$$

Shrinkage

The question arises how distortions of the structural component relevant for the calculation are caused. The reason for this is the concrete's restrained reduction due to reinforcement. If the boundary conditions of common "slender" components with uniformly distributed shrinkage strain are assumed, component curvatures will occur only for asymmetric reinforcement distribution.

Therefore, the shrinkage can be represented by a pre-strain of concrete or steel. In detail, this means that the steel's "free strain" is restrained by a positive pre-strain of concrete. In the same way, it would be possible to model the component with a negative pre-strain of steel so that the concrete restrains the free strain of the pre-strained steel. Both variants show identical stress distributions while taking into account the respective pre-strain, but they differ significantly on the strain level: If steel is pre-strained, it is immediately evident from the strain's condition where zones of tension and compression due to shrinkage occur. If concrete is pre-strained, it is possible to make statements from the strain's condition concerning the concrete's actual reduction.

As the determination of deformations is most important for the calculation, it is of no interest whether the modeling in the stiffness determination is carried out by a positive pre-strain of concrete or a negative pre-strain of reinforcement.

RF-CONCRETE Members takes shrinkage strain by negative pre-strain of the reinforcing steel into account.

2.4.7 Ultimate Limit State

Nonlinear analyses in the ultimate limit state serve to determine the limit of failure (mechanism) realistically. However, the design involves the following difficulty: Realistic estimations require realistic initial and computational parameters.

The material properties are not deterministic parameters. In contrast to the discrete cross-section design where the concept of "local defects" is always applied, mean material properties have to be used for the determination of deformations and internal forces.

Another aspect when determining the behavior of structural components realistically is the consideration of the concrete's effectiveness for tension between cracks (Tension Stiffening, see Chapter 2.4.3). The influence of creep and shrinkage is especially significant for compression elements.

According to EN 1992-1-1, clause 5.7, we have to use nonlinear methods leading to a realistic stiffness and considering uncertainties concerning failing. Design methods that are valid in the governing application areas may be used. An appropriate nonlinear method for determining internal forces including cross-section design is the approach with average values of material properties and the application of a global partial safety factor γ_r described both in the National Annex for Germany to EN 1992-1-1, clause 5.7 as well as the German DIN standard 1045-1, clause 8.5. This approach is described in the following as the *method according to EN 1992-1-1, clause 5.7*.

According to EN 1992-1-1, clause 5.7 (5), it is possible to use the approach as per EN 1992-1-1, clause 5.8.6 for structural components where effects according to the second-order analysis may not be neglected.

RF-CONCRETE Members provides both nonlinear methods of calculation (cf. Figure 4.4).

2.4.7.1 Material Properties

Method according to EN 1992-1-1, clause 5.7

The design according to EN 1992-1-1, clause 5.7 is based on mean material properties that have been calibrated to realize a global safety factor. The result is a reduced concrete compressive strength that represents a controversial subject due to the distortion of the average characteristic curve for concrete.

Calculational mean values of material strengths

- Stress-strain curve for **steel** according to EN 1992-1-1, Figure NA.3.8.1

$f_{yR} = 1.1 \cdot f_{yk}$

$f_{tR} = 1.08 \cdot f_{yR}$	Reinforced steel high ductility
$f_{tR} = 1.05 \cdot f_{yR}$	Reinforced steel normal ductility
$E_s = 200\,000\ N/mm^2$	Modulus of elasticity for steel

- Stress-strain curve for **concrete** according to EN 1992-1-1, Figure 3.2

$f_{cR} = 0.85 \cdot \alpha \cdot f_{ck}$

E_{cm}	mean modulus of elasticity for concrete (secant)

The following relation between the global safety factor R and the mean material strengths applies:

Concrete ($\gamma_c = 1.5$) : $1.5 \cdot 0.85 = 1.275 \sim \gamma_R = 1.3$

Reinforcing steel ($\gamma_s = 1.15$) : $1.15 \cdot 1.1 = 1.265 \sim \gamma_R = 1.3$

Figure 2.29 shows how the reduced concrete compressive stress f_{cR} is represented with the calculational mean values in comparison with the concrete's stress-strain diagram. The strong distortion of the characteristic curve for concrete is clearly recognizable. It results in an overestimation of strains, particularly in highly utilized areas, thus leading to overestimated curvatures.

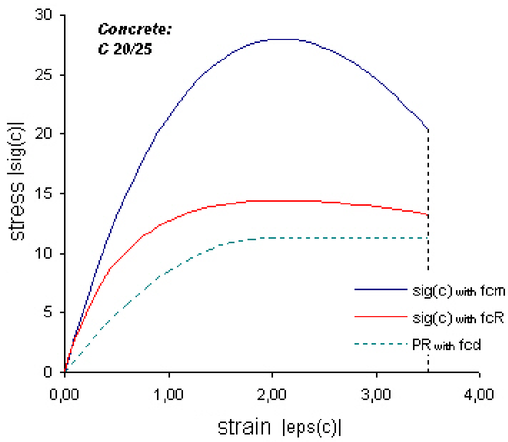


Figure 2.29 Stress-strain relation for internal forces and deformation analysis

Looking at the concrete's characteristic values, we can see the following: Though the theory is based on reduced stresses ($0.85 \cdot \alpha \cdot f_{ck}$), according to EN 1992-1-1, clause 3.1.5, the modulus of elasticity corresponds to the mean value.

Method according to EN 1992-1-1, clause 5.8.6

Clause 5.8.6 of the Eurocode standard describes the nonlinear calculation of structural components prone to instability risks. According to EN 1992-1-1, clause 5.8.6 (3), we need to define the stress-strain curves on the basis of design values.

Design values of the material strengths for the calculation of internal forces and deformations, as well as for design on cross-section level

- Stress-strain curve for steel according to EN 1992-1-1, clause 3.2.7

  $f_{yd} = f_{yk} / \gamma_s$

  $f_{td} = k \cdot f_{yk} / \gamma_s$

  E_{sm} = mean modulus of elasticity for steel (200 000 N/mm^2)

- Stress-strain curve for concrete according to EN 1992-1-1, clause 3.1.5

  $f_{cm} = f_{cd} = \alpha \cdot f_{ck} / \gamma_c$

  $E_c = E_{cd} = E_{cm} / \gamma_{cE}$

2.4.7.2 Safety Design

Method according to EN 1992-1-1, clause 5.7

According to EN 1992-1-1, clause 5.7, we have to design the safety of nonlinear calculations by means of a global safety factor γ_R. We can do this with a "trick", though it is disputed: modifying the mean stiffnesses of structural components (f_{cR}, f_{yR}, etc.). The calculational steel stress has been increased and the calculational concrete stress has been reduced, which allows for a return to the global safety factor γ_R = 1.3 (or 1.1 for extraordinary action combinations).

To ensure sufficient bearing capacity, the following conditions are required:

$$E_d \leq R_d = \frac{R}{\gamma_R}(f_{cR}, f_{yR}, f_{tR} \ldots)$$

where

E_d : design value of governing action combination

R_d : design value of load-bearing capacity

γ_R : uniform partial safety factor on side of ultimate load

RF-CONCRETE Members calculates with a γ_R-fold action. It can be applied in load steps, corresponding to an incremental calculation of the ultimate load.

The design is fulfilled when the γ_R-fold action is higher than the ultimate load. This corresponds to a conversion of the equation above.

$$\gamma_R \cdot E_d \leq R_d = R(f_{cR}, f_{yR}, f_{tR} \ldots)$$

This also takes account of the aspect for determining the reduction of imposed internal forces.

Advantages and disadvantages of the method

The most important advantage of this approach is obvious: Only one material rule is used for the entire calculation. This leads to easier handling as well as economy of time when calculating, because the determination of internal forces and the design are performed in one go.

The disadvantage is only explicitly visible when we assume that the terms

$$\frac{R}{\gamma_R}\left(f_{cR}, f_{yR}, f_{tR}, \dots\right) = R\left(\frac{f_{cR}}{\gamma_R}, \frac{f_{yR}}{\gamma_R}, \frac{f_{tR}}{\gamma_R}, \dots\right)$$

are compatible. In nonlinear calculations, of course, this compatibility is **not** given without restrictions. An example, which shows that such an approach can be very much on the unsafe side, is the consideration of imposed internal forces. The use of material properties divided by γ_R leads to strongly reduced stiffnesses resulting in a strong reduction of the imposed internal forces. However, this representation is quite useful to illustrate the problem of the reduced elastic modulus for steel.

The direct reduction of the stiffnesses is described in detail by Quast [10] and is evaluated critically with regard to slender compression elements.

To clarify the correlations, we simplify the conditions and assume a horizontal branch of the characteristic curve for reinforcing steel ($f_{yd} = f_{td}$). This results in the reduced design resistance R_d for:

$$R_d = \frac{R}{\gamma_R} = \frac{1}{\gamma_R}\int a \cdot \sigma_R[\varepsilon(y,z)]\, dA \quad \text{where } a = \begin{Bmatrix} 1 \\ z \\ -y \end{Bmatrix}$$

$$R_d = \frac{1}{\gamma_R}\int a\left[-f_{cR} \leq \sigma_{cR}(\varepsilon, f_{cR}) \leq 0;\ -f_{yR} \leq \sigma_{sR}(\varepsilon) \leq f_{yR}\right] dA$$

$$R_d = \int a\left[\frac{-f_{cR}}{\gamma_R} \leq \frac{\sigma_{cR}(\varepsilon, f_{cR})}{\gamma_R} \leq 0;\ \frac{-f_{yR}}{\gamma_R} \leq \frac{\sigma_{sR}(\varepsilon)}{\gamma_R} \leq \frac{f_{yR}}{\gamma_R}\right] dA$$

If we set $\sigma_{sR} = E_s \cdot \varepsilon$, the result is the following:

$$R_d = \int a\left[\frac{-f_{cR}}{\gamma_R} \leq \frac{\sigma_{cR}(\varepsilon, f_{cR})}{\gamma_R} \leq 0;\ \frac{-f_{yR}}{\gamma_R} \leq \frac{E_s}{\gamma_R}\varepsilon \leq \frac{f_{yR}}{\gamma_R}\right] dA$$

For a practical determination of internal forces according to the linear static analysis without imposed internal forces, it is absolutely legitimate to calculate with the reduced stiffnesses. In this case, the diagram of internal forces is affected by the relation of the stiffnesses from different areas to each other anyway.

However, this concept proves to be problematic when designing slender compression elements according to the second-order analysis. The deformations are overestimated because of the reduced stiffness in the system. This results in an overestimation of internal forces for calculations according to the second-order analysis.

Slender compression elements generally fail when the yield strain in the reinforcement is reached. Hence, it becomes obvious that deformations are overestimated due to the reduced modulus of elasticity and the resulting larger curvatures when the yielding starts. This leads to a smaller allowable column load, or the reinforcement must be increased accordingly. Quast [10] sees no reason for that.

Method according to EN 1992-1-1, clause 5.8.6

According to EN 1992-1-1, clause 5.8.6 (3), it is possible to directly perform the design for sufficient structural safety on the basis of design values (f_{cd}, f_{yd}, ...) of the material properties. In accordance with clause (3), the stress-strain curves defined on the basis of design values must also be used for the determination of internal forces and deformations. The modulus of elasticity E_{cd} to be applied must be calculated with the safety factor γ_{CE} ($E_{cd} = E_{cm} / \gamma_{CE}$).

Note concerning German NAD DIN EN 1992-1-1, clause 5.8.6

According to the National Annex for Germany EN 1992-1-1, clause 5.8.6 (NDP 5.8.6 (3)), the internal forces and deformations may be determined by means of average material properties (f_{cm}, f_{ctm}, ...). However, the design for the ultimate load capacity in the governing sections must be performed with the design values (f_{cd}, f_{yd}, ...) of the material properties.

The problem with this approach is that it is impossible for some parts in statically indeterminate systems to reach a convergence of results: The internal forces calculated with the mean values of the material properties cannot be taken over in the design with the design values to be applied. Increasing the reinforcement results in an increase of stiffness of the respective parts and areas, which again requires an increase of reinforcement in the subsequent iteration step. It is also important to note that a utilization of the plastic resources in the ultimate limit state is hardly possible, as the calculational design moment M_{Ed} (design values for strengths of materials) will not reach the value of the yield moment M_y (mean material properties).

RF-CONCRETE Members performs the safety design according to the standard by contrasting the provided reinforcement with the required reinforcement that is determined for the design values of the material properties. This must always be observed when manually correcting the reinforcement (keyword: "increase in stiffness").

2.4.8 Serviceability Limit State

With EN 1992-1-1, more detailed designs for the serviceability limit state have found their way into engineering offices.

The serviceability limit state is divided into three groups:

- Limitation of **stresses** (EN 1992-1-1, clause 7.2)
- Limitation of **crack widths** (EN 1992-1-1, clause 7.3)
- Limitation of **deformations** (EN 1992-1-1, clause 7.4)

Hereafter, only the limitation of deformations is described, also taking the influence of creep and shrinkage into account.

The reason for the more detailed analysis of deformations can be found again in the nonlinear behavior of reinforced concrete as a composite material. As a result of crack formation, the stiffness is reduced significantly in particular areas compared to the pure state I (uncracked sections). If the cracking is not taken into account, occurring deformations will be underestimated. By considering creep and shrinkage, the deformation may be three to eight times larger, depending on the stress state and boundary conditions.

The governing curvatures are determined as basis for deformations. It is important not to forget the concrete's effectiveness for tension between the cracks, otherwise unrealistic results are to be expected.

A correct interpretation of results from nonlinear calculations requires knowledge of the most important factors. Therefore, we compare the most important parameters that affect the stiffnesses in uncracked sections (state I) and cracked sections (state II) in the table below:

Influencing value	State I (uncracked)	State II (cracked)
Creep (here as reduction of elastic modulus for concrete)	The stiffness is mainly controlled by concrete. Thus, a reduced modulus of elasticity leads to a considerable reduction of stiffness.	Minor influence
Reinforcement ratio	Minor influence (see *Creep* for reasons)	The stiffness in state II is mainly controlled by the reinforcement. The influence is therefore enormous.
Axial force	Influence hardly given (In case of simplified linear-elastic analyses, there is no influence at all.)	A tensile force reduces the stiffness significantly. This must be considered when modeling the shrinkage, as it leads to tension stresses in the concrete.

Table 2.5 Influence values and their impact in uncracked and cracked state

Material properties

Generally, the mean material properties are used to calculate the deformation. The effectiveness of concrete on tension between cracks (*Tension Stiffening*) must also be taken into account by appropriate approaches (see Chapter 2.4.3) because otherwise no realistic deformation analysis is possible.

The mean material properties according to DIN 1045-1 and EN 1992-1-1 for determining the deformations do not differ from each other (or only marginally).

Calculational mean values of material strengths

– Stress-strain curve for **steel** according to EN 1992-1-1, Figure NA.3.8.1

$f_y = f_{yk}$

$f_t = f_{yk}$ for serviceability considerations

E_{sm} = 200 000 N/mm^2 mean modulus of elasticity for steel

– Stress-strain curve for **concrete** according to EN 1992-1-1, 3.1.5 and 5.7

f_{cm} mean concrete compressive strength

E_{cm} mean modulus of elasticity for concrete (secant)

2.4.9 Convergence

How fast and safely a nonlinear calculation converges depends on a variety of factors and can be specified for the general case only as a tendency.

Main starting point of the convergence evaluation is the method that is used. We know that methods based on tangential improvements (tangential stiffness matrix) often converge faster (square convergence in area of searched solution) than methods that determine an iterative improvement by means of secant stiffnesses. However, secant methods are generally numerically more stable, especially in the area of very flat gradients near the limit of failure (tangential stiffness approaches zero). Of course, this cannot be generalized because the convergence is affected by incremental load application, various iteration methods (Newton-Raphson, Riks/Wempner/Wessels, etc.), and other parameters.

Hereafter, the convergence behavior of the used algorithm is presented briefly. RF-CONCRETE Members performs the actual iteration of the state of strain on the cross-section level. This means that based on a diagram of internal forces within one iteration cycle, more and more new and current strain-stress conditions are calculated. The convergence is reached when a state of equilibrium is established, meaning the diagram of internal forces in two successive iteration steps remains within a given threshold.

This method alone is very stable in case of minor stiffness fluctuations in statically indeterminate frameworks. However, problems occur in case of abrupt changes or major changes in stiffness. The calculation may oscillate. To avoid this non-convergence, a damped stiffness reduction has been implemented in the calculation. The change between the stiffnesses of two iteration steps will be damped according to user specifications. The calculation thereby slows down a little, but is numerically more stable. Finally, we know that a damping for statically determinate systems makes no sense.

Hence, the two controllable termination criteria of nonlinear calculations are the following:

$$\varepsilon_1 = \left|(1/\gamma)_i - (1/\gamma)_{i-1}\right| \leq \text{tolerance}\,1$$

γ is an indicator for the ratio of ultimate moment to acting moment. This way, the termination criterion ε_1 takes into account the change of internal forces.

$$\varepsilon_2 = \left(EI_i - EI_{i-2}\right)^2 / \left(EI_i\right)^2 \leq \text{tolerance}\,2$$

This design criterion controls the stiffness difference of two successive iteration steps on the nodes.

In addition, the deformation difference between two iterations is checked:

$$\varepsilon_3 = \left|u_i - u_{i-1}\right| \leq \text{tolerance}\,3\,\text{(fixed)}$$

The maximum deformation difference is set fixed to the value ≤ 0.1 mm.

If the nonlinear calculation does not converge, some possibilities are offered in the *Settings for Nonlinear Calculation* dialog box (see Figure 2.30) to improve the convergence behavior.

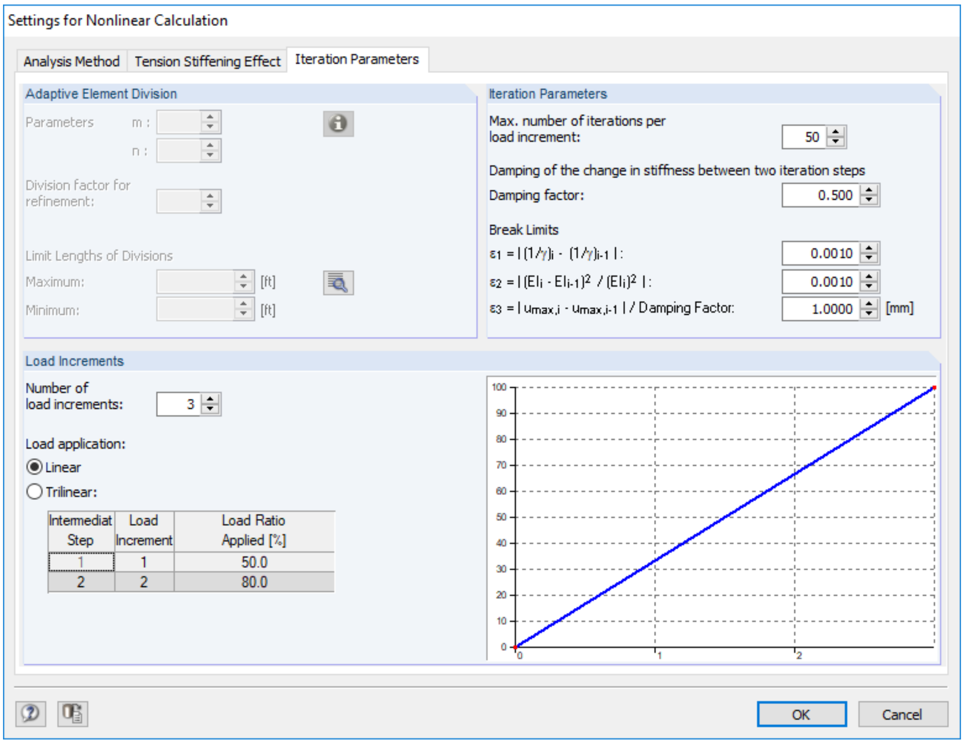


Figure 2.30 *Settings for Nonlinear Calculation* dialog box, *Iteration Parameters* tab

Increasing the number of iterations

The iteration process strongly depends on cross-section shape, structural system, and loading. This may lead to a different convergence behavior. Generally, structural components that are highly stressed by compression converge a bit more slowly. As the current deviations ε_1 and ε_2 are displayed permanently during calculation, you can easily decide if increasing the number of iterations (slow but continuous convergence) makes sense.

Increasing the number of load increments, trilinear if necessary

In the first load step, the linear-elastic stiffness is used as initial value. Calculating with only one load step in the first iteration cycle may result in a very large difference in stiffness, which interferes with the convergence. In this case, it can be practical to apply the load gradually.

Reduction of damping factor

By a specific reduction of stiffness changes between two iteration steps, it is possible to counteract the calculation's oscillation. In two successive iteration steps, the program determines the difference in stiffness on one node. The damping factor represents the part of the stiffness difference that is considered for the new stiffness applied in the subsequent iteration step:

$$E \cdot I_{i,\text{damped}} = E \cdot I_{i-1} \cdot \left(1 - \text{damping factor}\right) + E \cdot I_i \cdot \text{damping factor}$$

This means: The higher the damping factor, the smaller the damping's influence. If the factor is 1, the damping does not affect the iterative calculation.

3 Input Data

When you start the add-on module, a new window appears. On the left, a navigator that manages the available module windows is displayed. The drop-down list above the navigator contains the design cases (see Chapter 8.1).

The design-relevant data is defined in several input windows. When you open RF-CONCRETE Members for the first time, the following parameters are imported automatically:

- Load cases, load combinations, and result combinations
- Materials
- Members and sets of members
- Cross-sections
- Internal forces (in background, if calculated)

To select a window, click the corresponding entry in the navigator. To go to the previous or the next module window, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] window.

To save the entered data, click [OK]. You will exit RF-CONCRETE Members and return to the main program. To exit the add-on module without saving the data, click [Cancel].

3.1 General Data

In Window *1.1 General Data*, you specify the design standard and the actions. The tabs manage the load cases, load combinations, and result combinations for the designs in the ultimate and the serviceability limit state, as well as for the fire resistance design.

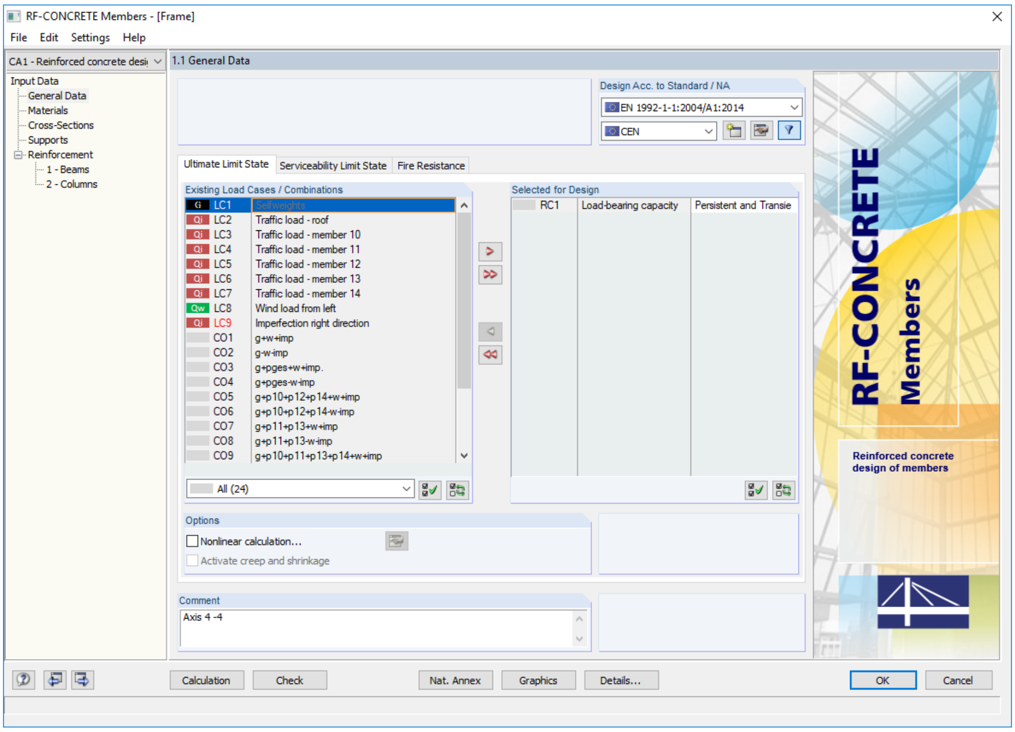


Figure 3.1 Window *1.1 General Data*

Design Acc. to Standard / NA

Standard

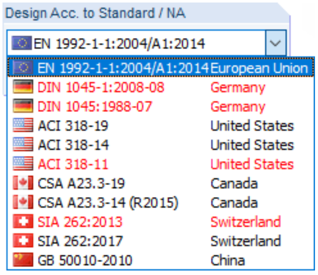


Figure 3.2 Available standards

You have to specify the standard according to which you want to perform the reinforced concrete design. Different standards are available for selection in the list. You can purchase each standard separately.

Standards marked in red are no longer valid, but can, for example, be used for recalculations of existing structural buildings. You can hide the "old" standards by using the [Filter] button.

National Annex

For the design according to Eurocode (EN 1992-1-1:2004/A1:2014), you have to specify the National Annex whose parameters apply to the checks.

Figure 3.3 National Annexes for EN 1992-1-1

Click the button to view the default parameters (see Figure 3.4).

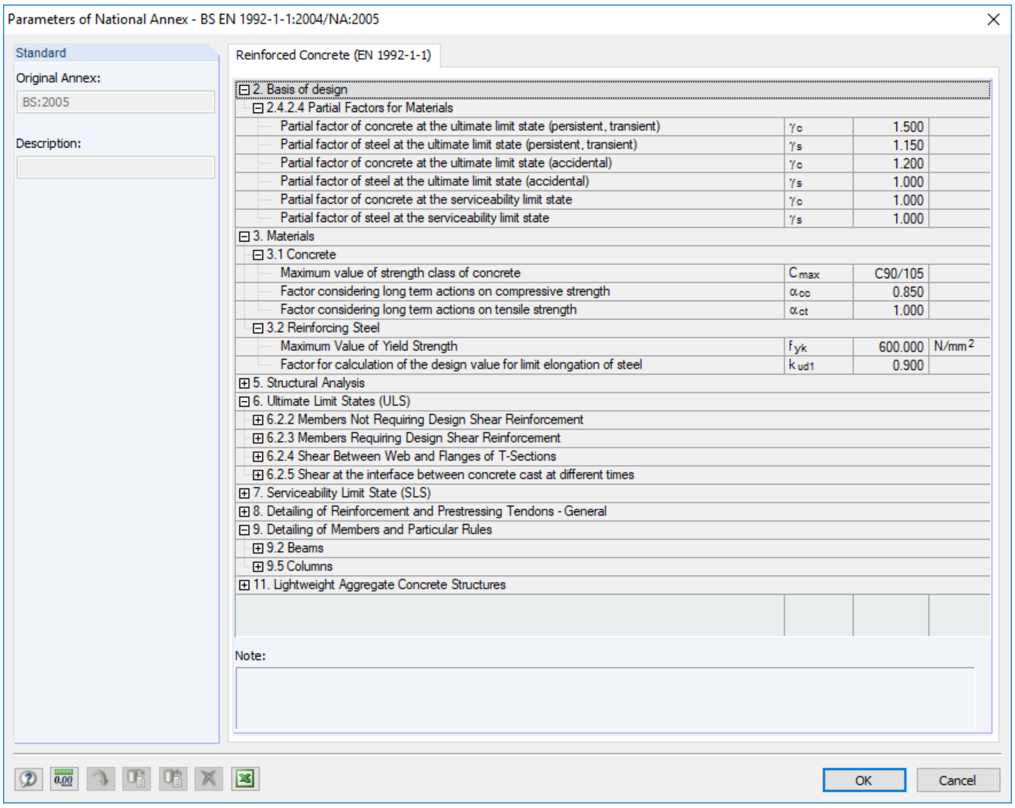
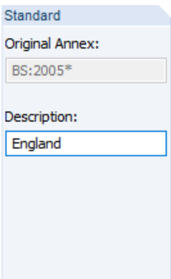


Figure 3.4 *Parameters of National Annex* dialog box



In this dialog box, you can find all design-relevant coefficients specified in the National Annexes. They are listed by the Eurocode's clause numbers.

If other specifications apply to partial safety factors, reduction factors, angles of concrete compression strut, etc., it is possible to adjust these parameters: First, click the button to create a copy of the current National Annex. In this user-defined annex, you can specify the parameters individually.

Comment

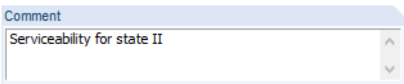


Figure 3.5 User-defined comment

In this input field, you can enter user-defined notes describing the current design case, for example.

3.1.1 Ultimate Limit State

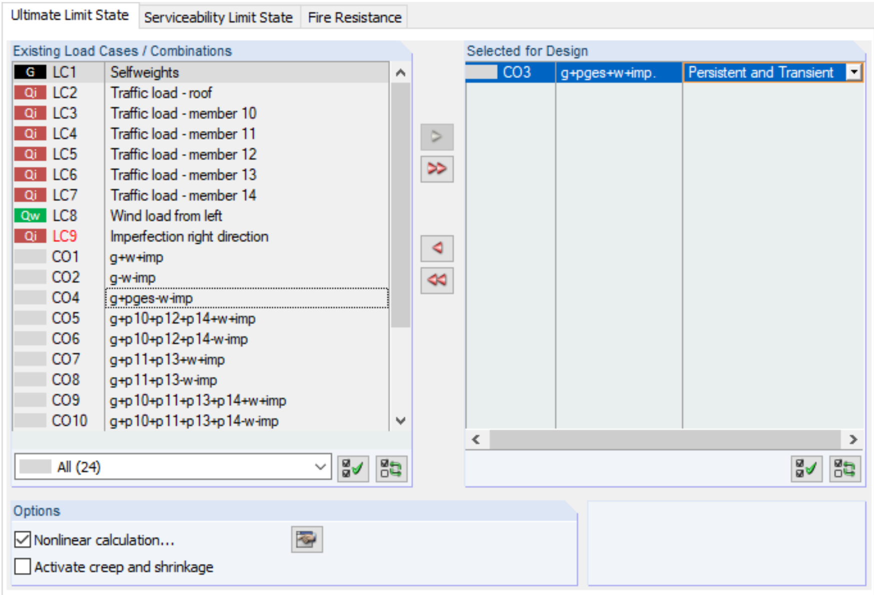


Figure 3.6 Window *1.1 General Data, Ultimate Limit State* tab

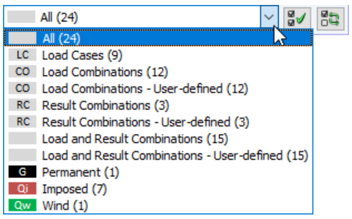
Existing Load Cases / Combinations

This column lists all load cases, load combinations, and result combinations that have been defined in RFEM.

To transfer selected entries to the *Selected for Design* list on the right, click the button. You can also double-click the items. To transfer the entire list, click .

As common for Windows applications, selecting several load cases is possible by clicking them one by one while holding down the [Ctrl] key. This way, you can transfer several load cases at once.

Load case numbers marked in red like LC 9 in Figure 3.6 cannot be designed: It indicates a load case without load data or a load case containing imperfections. A warning appears if you try to transfer it.



Below the list, various filter options are available. They help you to assign the entries sorted by load case, combination, or action category. The buttons have the following functions:

	Selects all load cases in the list.
	Inverts the selection of load cases.

Table 3.1 Buttons in *Existing Load Cases / Combinations* column

Selected for Design

This column lists the load cases, load combinations, and result combinations that have been selected for design. To remove selected items from the list, click or double-click the entries. To empty the entire list, click .

You can assign the load cases, load and result combinations to the following design situations:

- *Persistent and Transient*
- *Accidental*

This classification controls the partial safety factors γ_c and γ_s according to EN 1992-1-1, Table 2.1 (see Figure 3.4 and Figure 3.37).

To change the design situation, use the list, which you can access by clicking the button at the end of the text box.

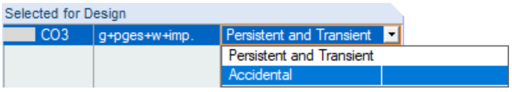


Figure 3.7 Assigning the design situation

For a multiple selection, press the [Ctrl] button and click the corresponding entries. This way, you can change several entries at once.

Result combination

If more than 16 load combinations are available for a design situation, analyzing an enveloping max/min result combination is faster than the analysis of all contained load cases and load combinations. The reason is that a design is performed on every x-location for 16 extreme values for result combinations. If less than 16 different load combinations per design situation are available for design, the load combinations should be selected individually for the design. However, when analyzing a result combination, it is difficult to discern the influence of the included actions.

Options

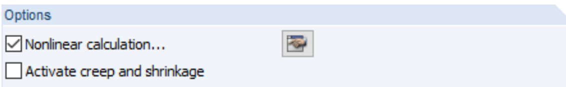


Figure 3.8 Options for nonlinear calculation

Nonlinear calculation

A license of the add-on module **RF-CONCRETE NL** is required for the nonlinear design method. The program performs a physical and geometrical nonlinear calculation. The internal forces are generally determined according to the second-order analysis. The nonlinear analysis for the ultimate limit state is described in Chapter 2.4.7.

The nonlinear design method is based on the assumption of an interaction between model and action-effects, requiring a clear distribution of internal forces. Therefore, only load cases and load combinations can be analyzed, but not result combinations (RC): In a result combination, there is a maximum and a minimum value per x-location available for each internal force.

To open the *Settings for Nonlinear Calculation* dialog box, use the button. This dialog box consists of three tabs. They are described in Chapter 4.2.

Nonlinear analyses are possible for both the ultimate limit state and the serviceability limit state.

Creep /Shrinkage

In the nonlinear calculation, it is possible to take the influence of creep and shrinkage into account. For more information, see Chapter 2.2.6.

If the check box is selected, you can define the creep coefficient φ (t, t_0) and shrinkage strain ε (t, t_s) in Window *1.3 Cross-Sections* (see Figure 3.19).

3.1.2 Serviceability Limit State

The serviceability limit state design depends on the results of the ultimate limit state design (reinforcement). It is not possible to perform the serviceability limit state design alone.

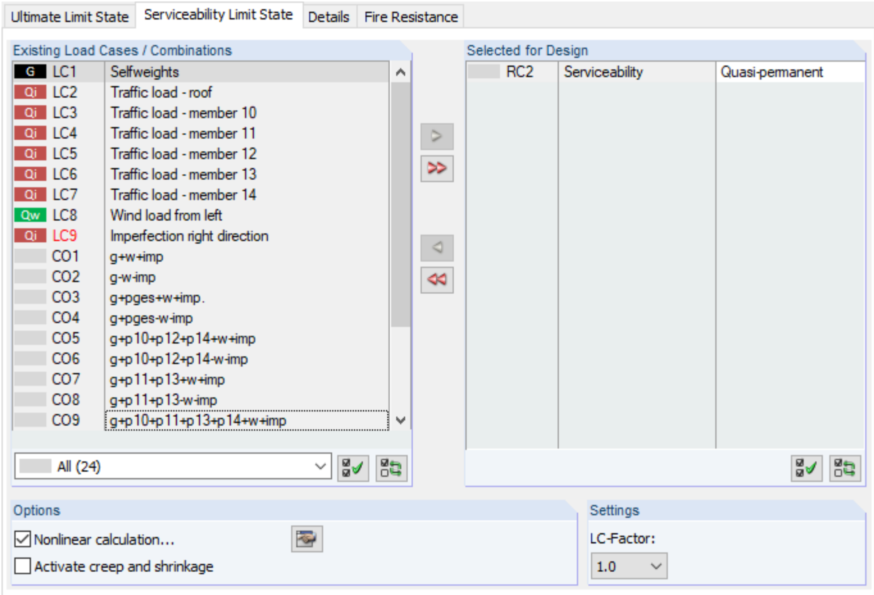


Figure 3.9 Window 1.1 *General Data, Serviceability Limit State* tab

Existing Load Cases / Combinations

This window section lists all load cases, load and result combinations defined in RFEM.

Normally, the actions and partial safety factors relevant for the serviceability limit state (SLS) design are different from the ones considered for the ultimate limit state. The corresponding combinations can be created in RFEM.

Selected for Design

You can add or remove load cases, as well as load and result combinations as described in Chapter 3.1.1.

For EN 1992-1-1, it is possible to assign different limit values for deflection to the individual load cases, load and result combinations. The following design situations are available:

- *Characteristic with direct load*
- *Characteristic with imposed deformation*
- *Frequent*
- *Quasi-permanent*

To change the design situation, use the list that you can access by clicking the button at the end of the text box.

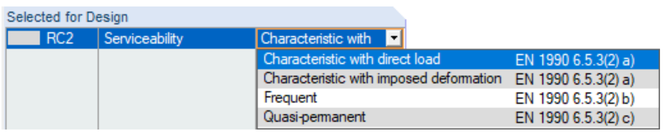


Figure 3.10 Assigning the design situation

Details...

With the [Details] button you can access the setting options for the individual design situations (see Chapter 4.1.2).

Options

Figure 3.11 Options for nonlinear calculation

Nonlinear calculation

A license of the add-on module **RF-CONCRETE NL** is required for the nonlinear design method. The nonlinear analysis for the serviceability limit state is described in Chapter 2.4.8.

Nonlinear analyses performed according to EN 1992-1-1 or DIN 1045-1 can only be carried out for load cases and load combinations, but not for result combinations.

To open the *Settings for Nonlinear Calculation* dialog box, use the button. This dialog box consists of three tabs. They are described in Chapter 4.2.

Nonlinear analyses are possible for both the ultimate limit state and the serviceability limit state.

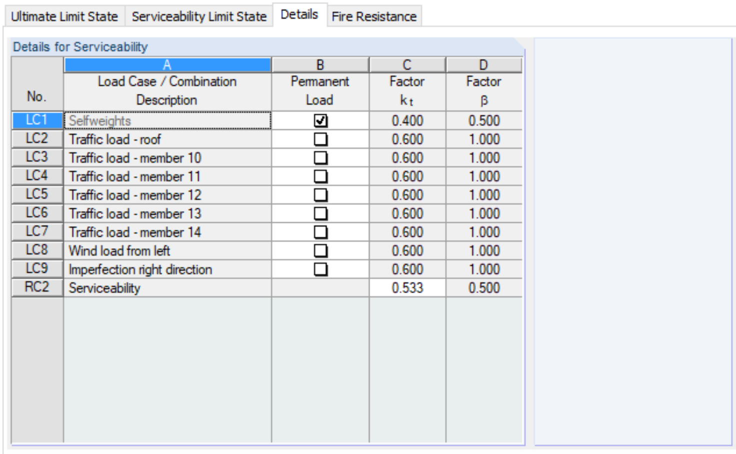
Creep/Shrinkage

In the nonlinear calculation, it is possible to take the influence of creep and shrinkage into account. For more information, see Chapter 2.2.6.

If the check box is selected, you can define the creep coefficient φ (t, t_0) and shrinkage strain ε (t, t_s) in Window *1.3 Cross-Sections* (see Figure 3.19).

3.1.3 Details

This tab appears when load cases or combinations are selected for the serviceability limit state design, and the analysis is performed according to EN 1992-1-1. There is no need to display this tab for DIN 1045-1 because the factor k_t is specified as 0.4 in Eq. (136).



No.	A Load Case / Combination Description	B Permanent Load	C Factor k_t	D Factor β
LC1	Selfweights	☑	0.400	0.500
LC2	Traffic load - roof	☐	0.600	1.000
LC3	Traffic load - member 10	☐	0.600	1.000
LC4	Traffic load - member 11	☐	0.600	1.000
LC5	Traffic load - member 12	☐	0.600	1.000
LC6	Traffic load - member 13	☐	0.600	1.000
LC7	Traffic load - member 14	☐	0.600	1.000
LC8	Wind load from left	☐	0.600	1.000
LC9	Imperfection right direction	☐	0.600	1.000
RC2	Serviceability		0.533	0.500

Figure 3.12 Window *1.1 General Data, Details* tab

In the crack width design, the program calculates the differences in the mean strains of concrete and reinforcing steel (see Chapter 2.2.4). According to EN 1992-1-1, 7.3.4 (2), Eq. (7.9), the load duration factor k_t must be specified for this.

Load Case / Combination Description

This column lists all load cases, load combinations, and result combinations that have been selected for design in the *Serviceability Limit State* tab. For load and result combinations, the included load cases are shown, too.

Permanent load

This column indicates the load cases to be applied as permanent loads. When selecting an entry, the corresponding factor k_t is automatically set to 0.4 in column C.

Factor k_t

The load duration factor k_t is used to consider the duration of the load. The factor k_t is 0.4 for long-term load actions and 0.6 for short-term actions.

For load and result combinations, the mean is taken from the k_t values of the load cases contained in the CO or RC.

$$k_t = \frac{\sum_{i=1}^{n} \gamma_i(\mathrm{LC}) \cdot k_{t,i}(\mathrm{LC})}{\sum_{i=1}^{n} \gamma_i(\mathrm{LC})}$$

Equation 3.1 Load duration factor k_t

Factor β

This column is displayed if the analysis of *Deflection* $u_{l,z}$ is set in Window 1.6 (see Figure 3.38). The factor β is 1.0 for short-term loading and 0.5 for long-term loading.

3.1.4 Fire Resistance

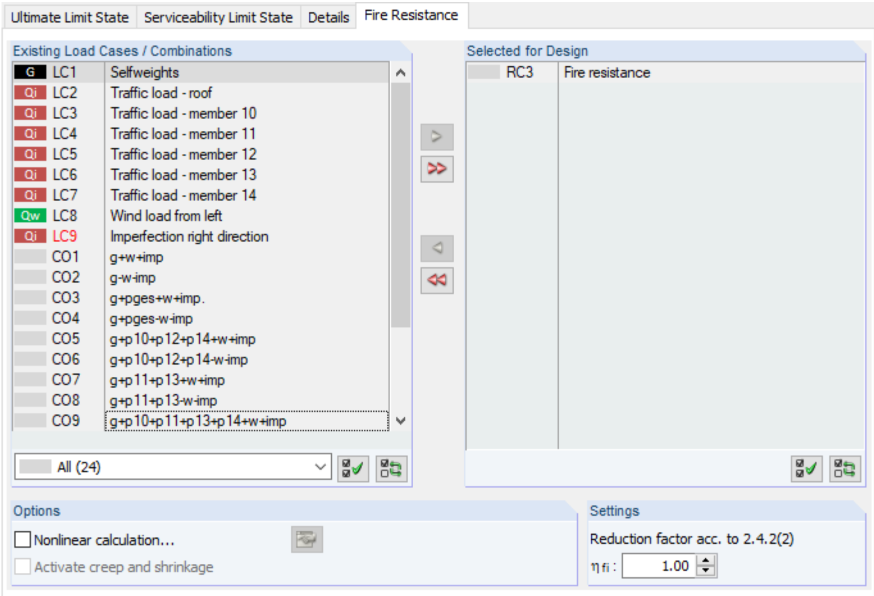


Figure 3.13 Window *1.1 General Data, Fire Resistance* tab

Existing Load Cases / Combinations

This window section lists all load cases, load and result combinations defined in RFEM.

Normally, the actions and partial safety factors relevant for the fire resistance designs are different from the ones considered for the ultimate limit state. The corresponding combinations can be created in RFEM.

Selected for Design

You can add or remove load cases, as well as load and result combinations for the fire protection design as described in Chapter 3.1.1.

Options

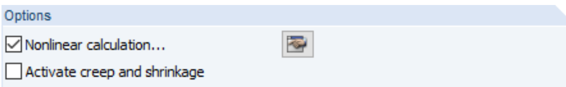


Figure 3.14 Options for nonlinear calculation

Nonlinear calculation

A license of the add-on module **RF-CONCRETE NL** is required for the nonlinear design method. The nonlinear design method is described in Chapter 2.4.

To open the *Settings for Nonlinear Calculation* dialog box, use the [Settings] button. This dialog box consists of three tabs. They are described in Chapter 4.2.

Creep/Shrinkage

In the nonlinear calculation, it is possible to take the influence of creep and shrinkage into account. For more information, see Chapter 2.2.6.

If the check box is selected, you can define the creep coefficient φ (t, t_0) and shrinkage strain ε (t, t_s) in Window *1.3 Cross-Sections* (see Figure 3.19).

Settings

The *Reduction factor acc. to 2.4.2 (2)* option allows for a simplified transfer of loadings from the design for normal temperature and to reduce these actions by the reduction factor η_{fi}.

The reduction factor η_{fi} is determined as per the equations according to EN 1992-1-2, 2.4.2 (3). As a simplification, the recommended value η_{fi} = 0.7 can be used.

3.2 Materials

The window is divided into two parts. The upper section lists the concrete classes and steel grades relevant for the design. All materials of the 'concrete' category used for members in RFEM are preset. In the *Material Properties* section, the properties of the current material, that is the material whose table row is selected in the upper section, are shown.

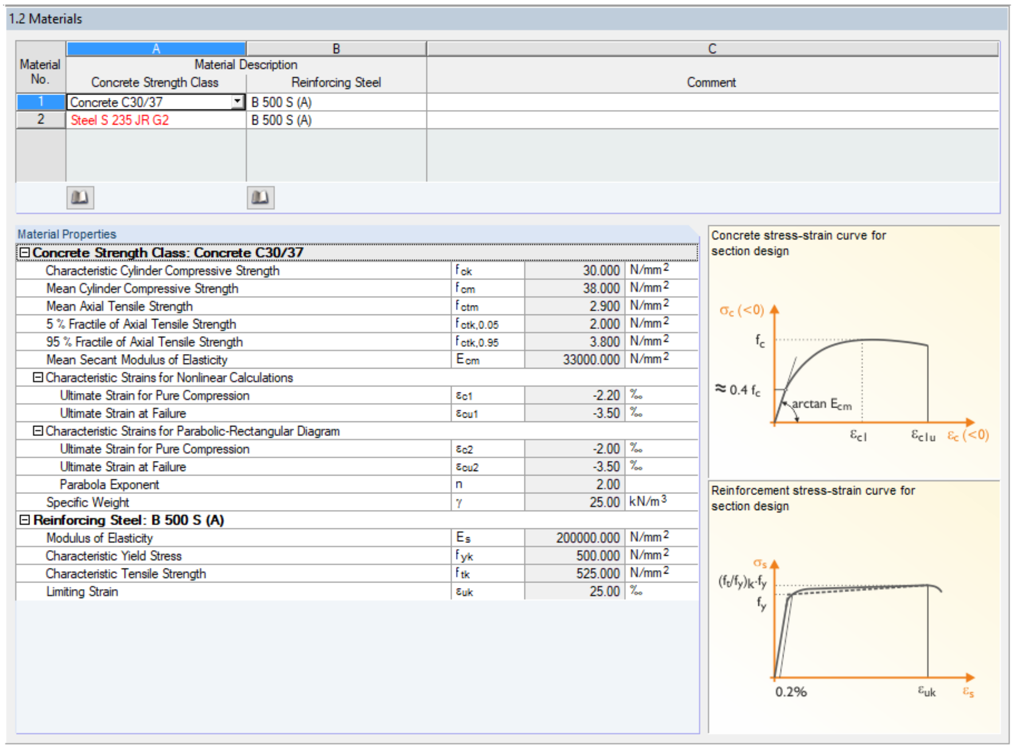


Figure 3.15 Window *1.2 Materials*

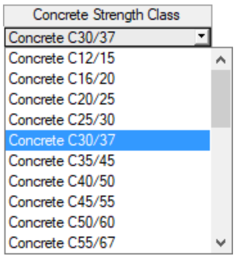
Column A lists the materials relevant for the design of the members. Materials that are not allowed are highlighted in red, modified materials in blue.

Chapter 4.3 of the RFEM manual describes the material properties used for the determination of the internal forces. The design-relevant material properties are also stored in the global material library. The values are preset for the *Concrete Strength Class* and for *Reinforcing Steel*.

To adjust the units and decimal places of the characteristic values and stresses, select **Settings → Units and Decimal Places** in the menu (see Chapter 8.3).

Material description

Concrete Strength Class



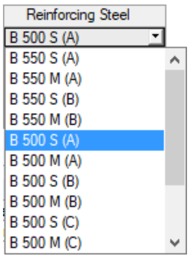
The materials used in RFEM are already preset.

The strength class can be changed at any time: Click the material in column A to select the field. Then, click the button, or press the [F7] function key to open the list of strength classes. The list contains only strength classes in accordance with the design concept of the selected standard.

After importing a material, the *Material Properties* are updated in the section below.

The material properties are generally not editable in RF-CONCRETE Members.

Reinforcing Steel



In this column, the program presets a steel grade in accordance with the design concept of the selected standard.

As with the concrete strength class, you can select a different reinforcing steel from the list: Click the material in column B to select the field. Then, click the button or press the [F7] key to open the list of reinforcing steels. As with the concrete strength classes, the list contains only steel grades that are relevant for the selected standard.

After importing a material, the *Material Properties* are updated in the section below.

Material library

Many materials are stored in a database. To open the library, select the menu item

Edit → Import Material from Library

or use the button shown on the left. You can find the library buttons below columns A and B.

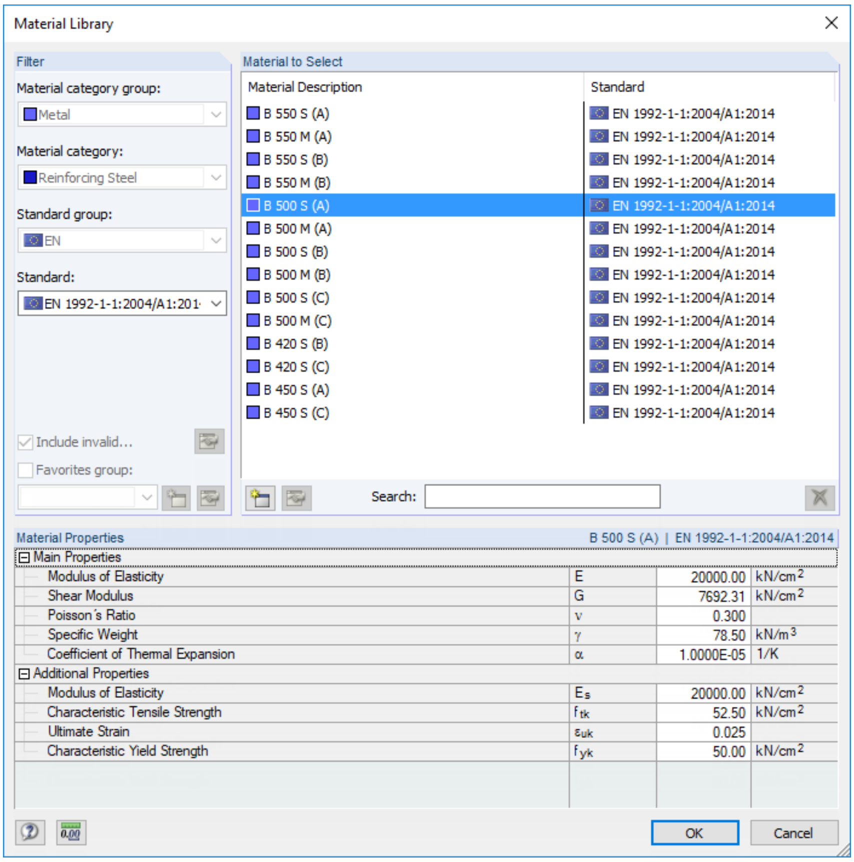


Figure 3.16 *Material Library* dialog box

OK

In the *Filter* section, the materials relevant to the standard are already preset, thus excluding all other categories or standards. You can select the desired concrete strength class or steel grade from the *Material to Select* list; then you can check the properties in the section below.

Click [OK] or press Enter to import the selected material into Window 1.2 of RF-CONCRETE Members.

Chapter 4.3 of the RFEM manual describes how to filter, add, or reorganize materials.

3.3 Cross-Sections

This window lists the cross-sections used for the design. In addition, you can make specifications for optimization.

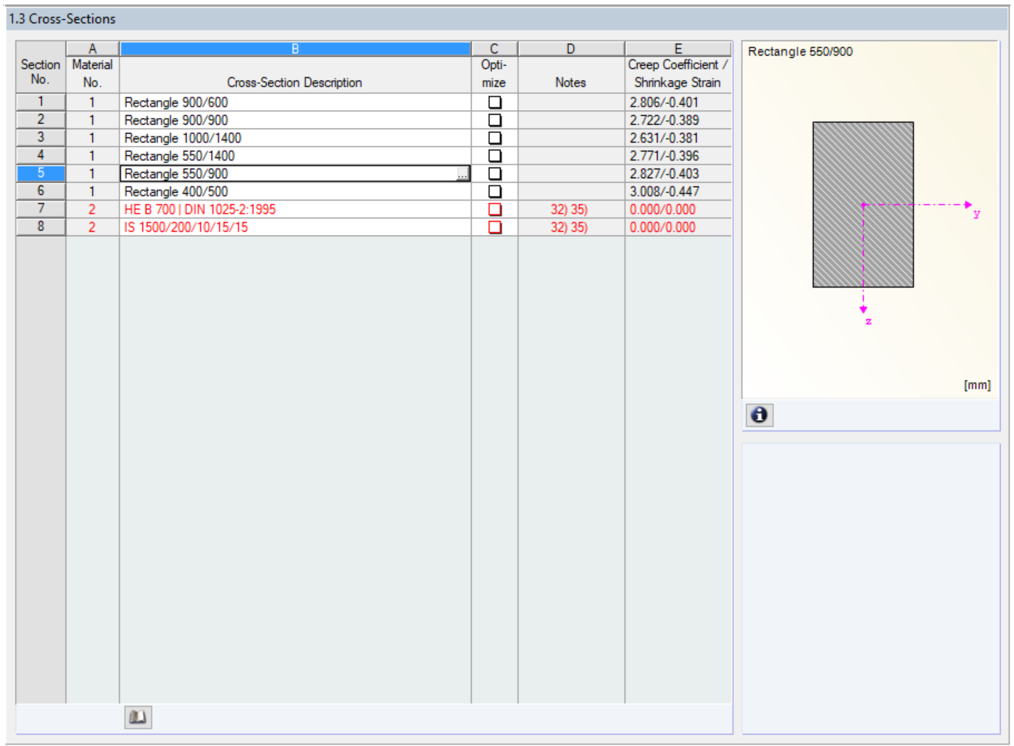
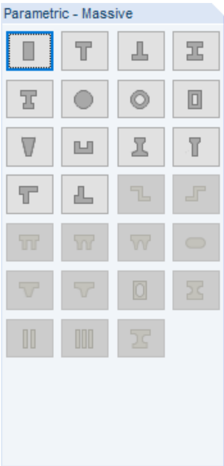


Figure 3.17 Window *1.3 Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM are preset along with the assigned material numbers.

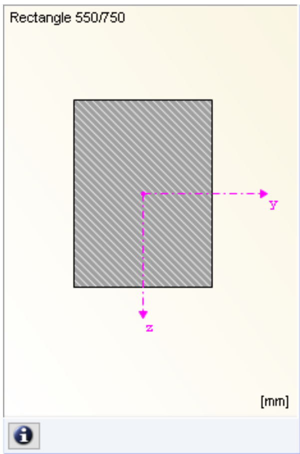
To modify a cross-section, click the entry in column B to activate the field. Then, use the [Cross-Section Library] button, click ... in the text box, or press the [F7] key to open the cross-section table of the current input field (see Figure 3.18).

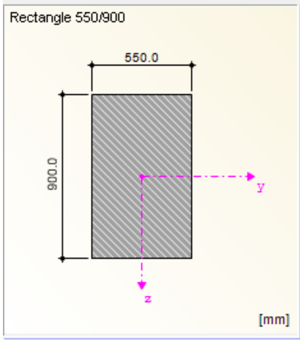


A dialog box opens where you can select a different cross-section or a different cross-section table. The following *Solid Cross-Sections* are enabled for the design in RF-CONCRETE Members:

- Rectangle
- Floor beam (symmetric, unsymmetric, or conic)
- Rotated floor beam (symmetric or unsymmetric)
- I-shape (symmetric, unsymmetric, or conic)
- Circle
- Ring
- Hollow rectangle (Z-symmetric)
- Conic shape (symmetric)
- U-section (symmetric)

Selecting cross-sections from the library is described in chapter 4.13 of the RFEM manual.





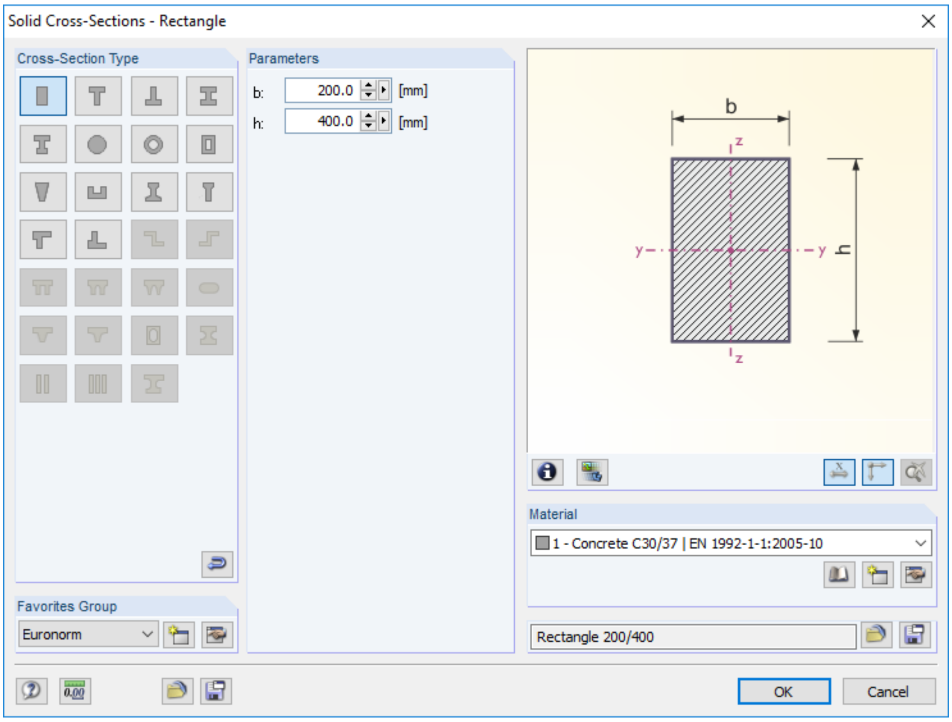


Figure 3.18 Parametric solid cross-sections of rectangular shape in cross-section library

You can also enter the new cross-section description directly into the text box. If the entry is already listed in the database, RF-CONCRETE Members will import the properties.

A modified cross-section is highlighted in blue.

If the cross-sections in RF-CONCRETE Members are different from the ones used in RFEM, both are displayed in the graphic to the right of the table. The designs will be performed for the cross-section selected in RF-CONCRETE Members, using the internal forces from RFEM.

Optimize

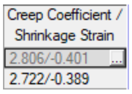
Each allowable cross-section can run through an optimization process. For the internal forces determined in RFEM, the program finds the cross-section that meets the specifications of the *Optimization Parameters* dialog box with the least possible dimensions (see Figure 8.5).

To optimize a certain cross-section, select its check box in column C. Recommendations for optimizing cross-sections can be found in Chapter 8.2.

Notes

This column shows remarks in the form of footnotes. They are explained in the status bar.

Creep Coefficient / Shrinkage Strain



Column E shows the values of the creep coefficients φ (t,t_0) and shrinkage strains ε (t,t_s). The values are determined from preset parameters. They can be adjusted with the ... button that appears after clicking into the text box. A new dialog box opens.

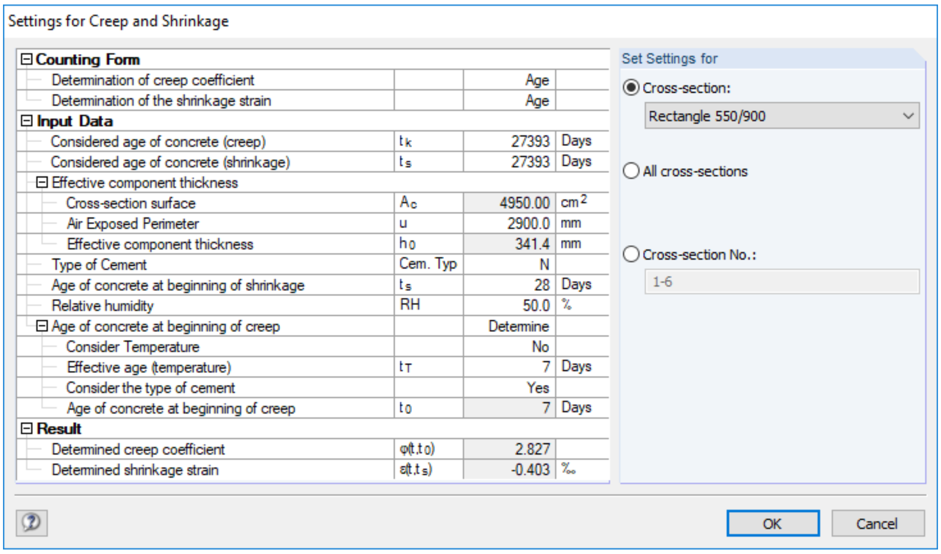
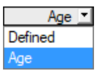


Figure 3.19 *Settings for Creep and Shrinkage* dialog box



The *Counting Form* for the creep coefficient and the shrinkage strain is possible in two ways:

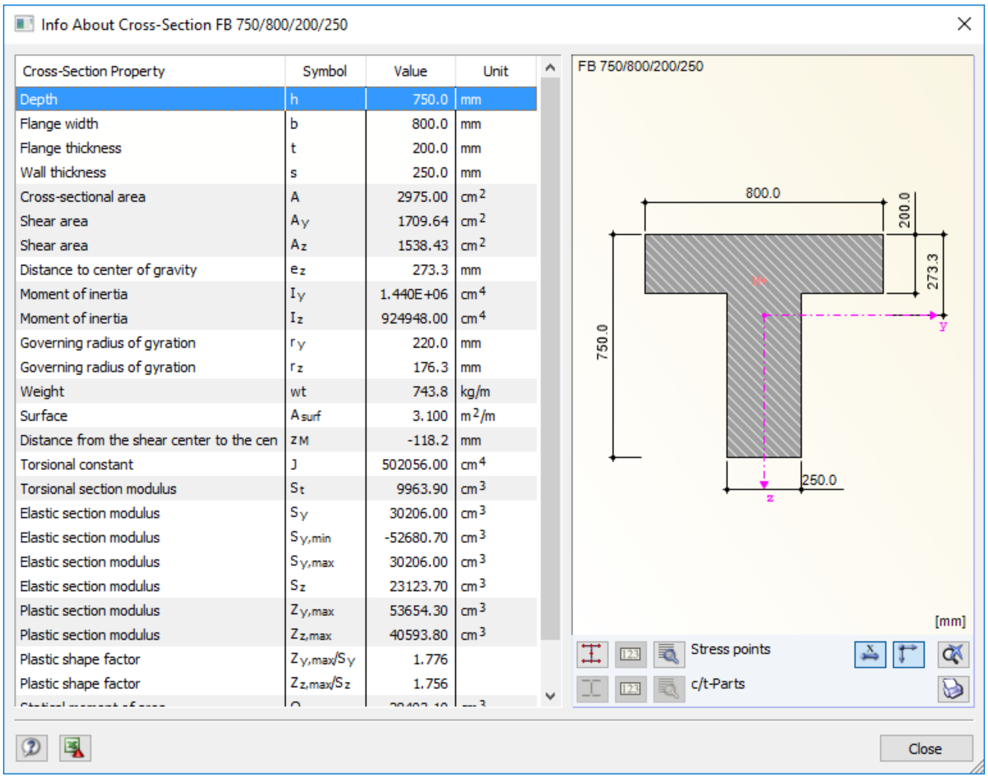
- *Age* : The values are calculated by the program using parameters.
- *Defined* : The values must be specified directly.

At the end of the table, the *Result* determined from the parameters is displayed for the creep coefficient φ (t,t_0) and the shrinkage strain ε (t,t_s). The determination of the creep coefficient and shrinkage strain is described in Chapter 2.2.6.

In the *Set Settings for* dialog section, you can define whether the specifications apply to a particular cross-section, all cross-sections, or selected cross-sections.

Info About Cross-Section

The *Info About Cross-Section* dialog box provides information on the cross-section's properties.



Cross-Section Property	Symbol	Value	Unit
Depth	h	750.0	mm
Flange width	b	800.0	mm
Flange thickness	t	200.0	mm
Wall thickness	s	250.0	mm
Cross-sectional area	A	2975.00	cm^2
Shear area	A_y	1709.64	cm^2
Shear area	A_z	1538.43	cm^2
Distance to center of gravity	e_z	273.3	mm
Moment of inertia	I_y	1.440E+06	cm^4
Moment of inertia	I_z	924948.00	cm^4
Governing radius of gyration	r_y	220.0	mm
Governing radius of gyration	r_z	176.3	mm
Weight	wt	743.8	kg/m
Surface	A_{surf}	3.100	m^2/m
Distance from the shear center to the cen	z_M	-118.2	mm
Torsional constant	J	502056.00	cm^4
Torsional section modulus	S_t	9963.90	cm^3
Elastic section modulus	S_y	30206.00	cm^3
Elastic section modulus	$S_{y,min}$	-52680.70	cm^3
Elastic section modulus	$S_{y,max}$	30206.00	cm^3
Elastic section modulus	S_z	23123.70	cm^3
Plastic section modulus	$Z_{y,max}$	53654.30	cm^3
Plastic section modulus	$Z_{z,max}$	40593.80	cm^3
Plastic shape factor	$Z_{y,max}/S_y$	1.776	
Plastic shape factor	$Z_{z,max}/S_z$	1.756	

Figure 3.20 *Info About Cross-Section* dialog box

3.4 Ribs

The ribs defined in RFEM are preset. Ribs represent a special type of member consisting of a beam and an effective plate cross-section (see RFEM manual, chapter 4.18). RF-CONCRETE Members imports the rib internal forces from RFEM and uses them for the design.

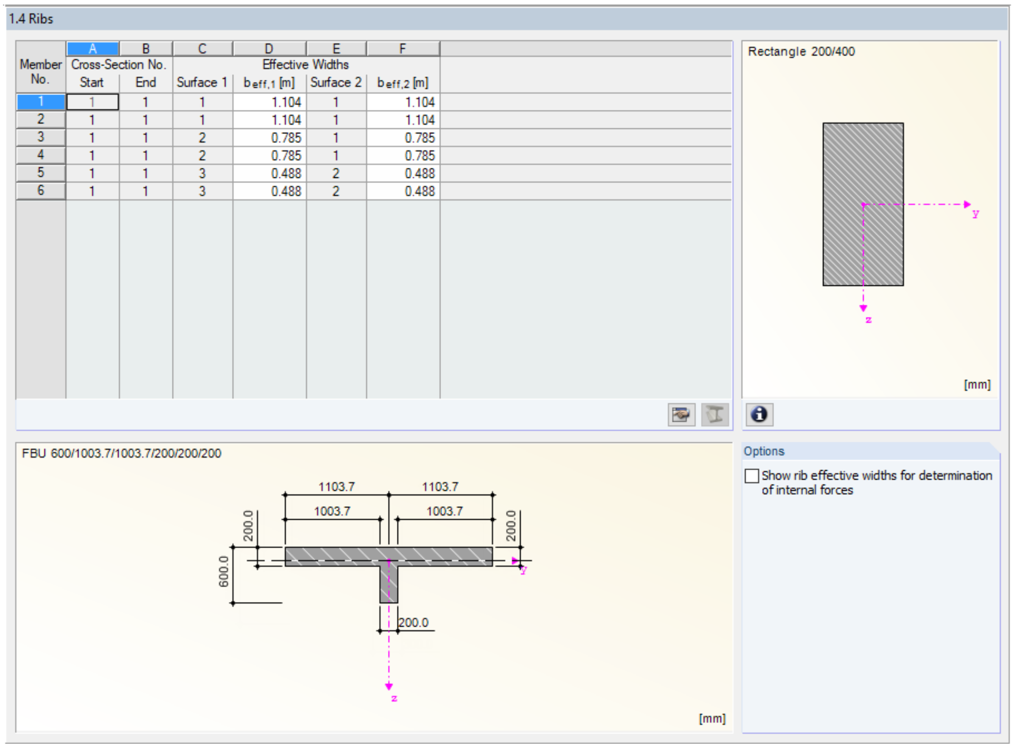


Figure 3.21 Window *1.4 Ribs*

You can adjust the effective widths b_{eff} in columns D and F. A recalculation in RFEM is not required because the system stiffness remains unchanged. The calculation of the cross-section properties and the integration of the rib internal forces are carried out automatically for each change of the effective widths.

Member No.

This column shows the numbers of the members defined as *Rib* member type in RFEM.

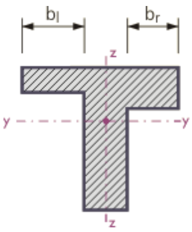
Cross-Section No. Start / End

Columns A and B show the cross-section numbers (see Chapter 3.3). If different numbers are displayed, the member is a tapered member.

Effective Width b_{eff}

Columns D and F indicate the effective widths for the left and the right side of the member. The values for b_1 and b_2 taken from the *New Rib* dialog box in RFEM (see RFEM manual, chapter 4.18) are preset here. The rib internal forces are determined based on the integration widths for the pro rata internal forces in surfaces.

The effective width controls the cross-section design in the form of an equivalent cross-section. Therefore, you can adjust the values for b_{eff}. Reduced effective widths result in reduced member internal forces, which affect the design in RF-CONCRETE Members. Increasing the integration widths is not allowed.



The effective widths can be modified directly. Alternatively, it is possible to adjust the widths with the button that opens a dialog box managing the parameters of the equivalent cross-section called *Unsymmetric Floor Beam*. When entering the widths b_l and b_r, follow the sketch of the cross-section.

To check the effective widths, select the *Show rib effective widths for determination of internal forces* check box. The table will be extended by two additional columns.

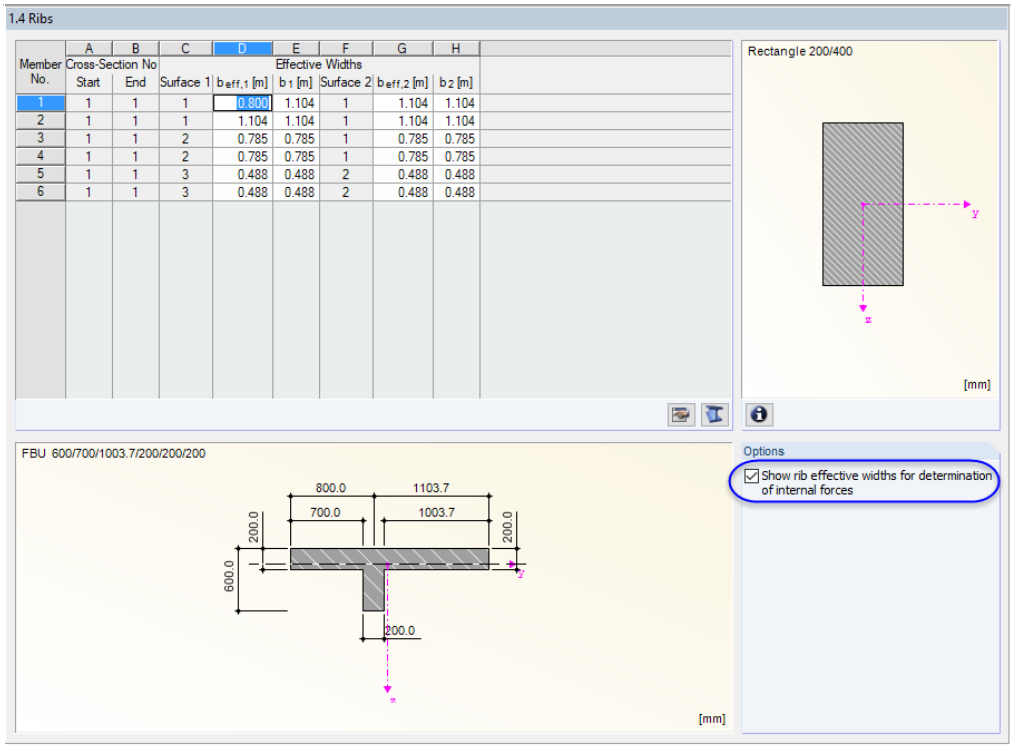


Figure 3.22 Window *1.4 Ribs*

By selecting the check box, the [Edit Rib] button becomes accessible. It enables you to adjust the rib parameters of RFEM, thus affecting the system stiffness. However, this also means that the internal forces must be recalculated.

In the cross-section graphic below the table, changes are represented dynamically. The graphic shows the equivalent cross-section used for the design.

Notes

If the rib leads to problems in the design, a corresponding note appears in this column.

The following items must be considered for a correct design of ribs:

- The rib's local z-axis must be parallel to the local z-axis of the surface.
- The rib's local z-axis must be orthogonal to the xy-plane of the surface.
- The surface type of the connected surface must be *Plane*.
- The cross-section type of the rib member must be a *Rectangle*.
- When sets of members are used, a uniform rib type must be defined for the entire set of members.
- The material of the rib must be identical to the material of the surface.

3.5 Supports

In this module window, you can define the support conditions for the members. The support nodes of **horizontal** members taken from RFEM are preset. The program recognizes whether the support is located at the end of a member or between two members (continuous beam).

Support widths greater than zero affect the design (redistribution of moments, moment reduction, reduction of shear force) and the reinforcement proposal (length of anchorage). However, this only applies to members in horizontal or slightly inclined position, not to columns!

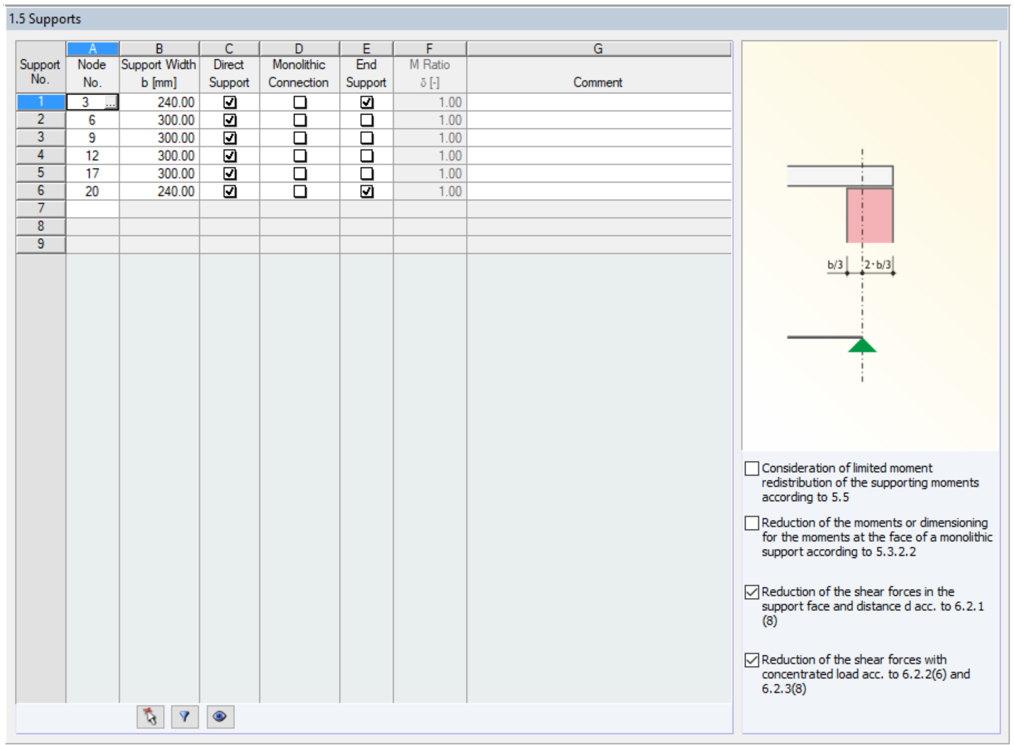


Figure 3.23 Window *1.5 Supports*

Node No.

This column lists the supported nodes of all members in a horizontal position or a position that is inclined up to 15°. Use the [...] button in this column to select additional nodes graphically in the RFEM work window.

Support Width b

In this column, the widths of the supports must be defined. This way, you can determine, for example, a wall's wide bearing area that is represented as singular support in the RFEM model.

Direct Support

This column controls the support type of the beam. If there is an adjoining beam introducing its load to another beam, the support is an indirect support. In this case, you have to clear the check box.

The specifications affect the lengths of anchorage and the shear force design.

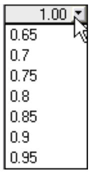
Monolithic Connection

In column D, you can specify whether there is a flexurally rigid connection with the support, or a rotary support allowing for a reduction of the supporting moment.

End Support

The geometrical conditions for end supports are different than the ones for intermediate supports, in order to determine the design moments and anchorage lengths (see graphic on the right in window).

M Ratio δ



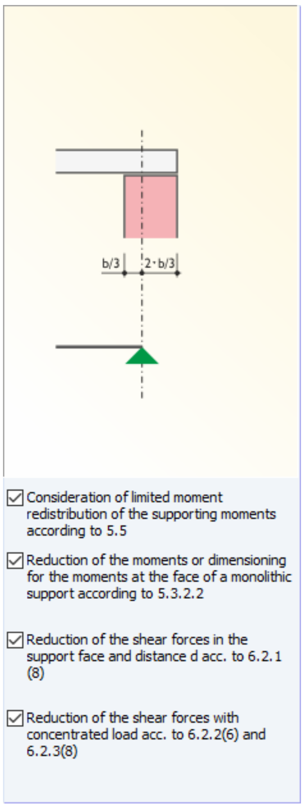
For **continuous** structural components, you can define the ratio δ of redistributed moment and elastically determined initial moment in column F. You can access the column as soon as the option *Consideration of limited moment redistribution* below the graphic is selected.

The δ-values can be determined according to a standard, for example EN 1992-1-1, 5.5 (4).

Comment

Here, you can enter a text describing the selected support conditions.

Moment redistribution / Shear force design



Below the graphic, you can find four check boxes whose settings affect the required reinforcement. These settings are generally effective for the current design case (see Chapter 8.1).

Consideration of limited moment redistribution of the supporting moments

For continuous beams, it is possible to apply the linear-elastic methods with limited redistribution of the supporting moments. The resulting distribution of internal forces must be at equilibrium with the acting loads. In standards such as EN 1992-1-1, 5.5 (4), the moment ratios δ that must be observed in order to ensure the ability for rotation in critical areas without special designs are described.

RF-CONCRETE Members determines the limit value and compares it with the value specified in column F. Then, the higher value is used for the redistribution.

Reduction of the moments or dimensioning for the moments at the face of a monolithic support

Optionally, RF-CONCRETE Members performs a moment reduction according to, for example, EN 1992-1-1, 5.3.2.2, if all of the following requirements are met:

- No end support
- Support width > 0
- Support is restrained in Z-direction
- Support force acts positive in Z
- Member in horizontal position or with max. inclination of 15°
- Negative moment distribution in entire support zone

The decision whether to reduce the moment or to apply the moment at the face of the support depends on the support's definition in column D: The moment at the face of the support is used for a monolithic connection; the supporting moment is reduced for a support with no rotational restraint.

Shear force design at the support face

A shear force design is possible for the moment at the face of the support. In case of a direct support, it is possible to reduce the design value of shear force according to EN 1992-1-1, 6.2.1 (8).

For result combinations, the shear force design is always carried out on the support's edge because a "uniformly distributed load" cannot be presumed for an envelope.

Reduction of shear forces with concentrated loads near support

Use this check box to control whether the shear force component of concentrated loads close to the support is considered according to EN 1992-1-1, 6.2.2 (6) or 6.2.3 (8). A reduction is only carried out for load cases and load combinations, not for result combinations.

3.6 Reinforcement

This module window consists of several tabs that manage the settings for the reinforcement. As the members to be designed often require different specifications, it is possible to define what are known as "reinforcement groups" in each design case (*RF-CONCRETE Members case*): Each reinforcement group contains the reinforcement's parameters that apply to particular members or sets of members.

Reinforcement Group

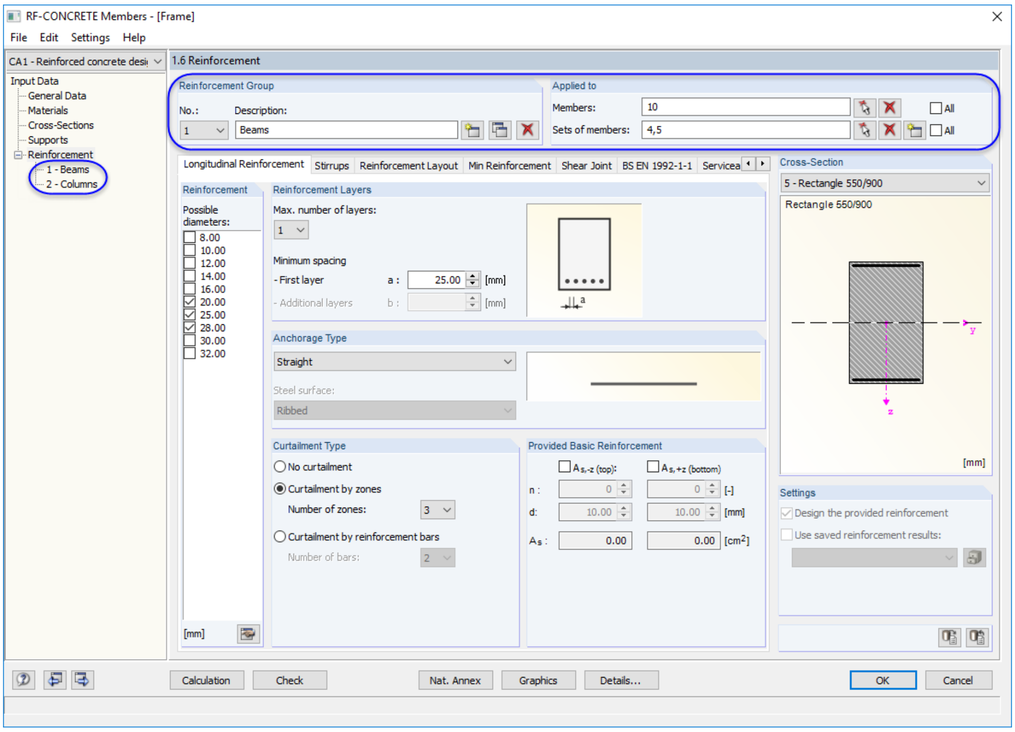
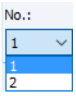


Figure 3.24 Window *1.6 Reinforcement* with two reinforcement groups

Click the [New] button to create a new reinforcement group. The button is accessible if the *All* check box in the *Applied to* section is deactivated so that the members can be assigned clearly.

The number of the reinforcement group is defined automatically; it cannot be changed. A user-defined *Description* helps you to keep track of all reinforcement groups available in the current design case.

With the [Delete] button, you can remove the current reinforcement group from the design case without any further warning. As a result, members that were included in that reinforcement group will not be designed. To design them, they must be reassigned to a new or existing reinforcement group.



To select the desired reinforcement group, use the *No.* list, or click the entry in the navigator.

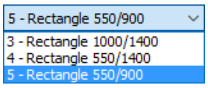
Applied to members/sets of members

In this dialog section, you can specify the members and sets of members the parameters of the current reinforcement group apply to. *All* members and *All* sets of members are preset. If both check boxes are selected, you cannot create another reinforcement group because it is not possible to design a member using different rules (this is only possible by using "design cases", see Chapter 8.1). To use reinforcement groups, you have to clear the *All* check box.

In the input fields, enter the numbers of the members and sets of members the reinforcement parameters of the tabs below apply to. You can also use the button to select them graphically in the RFEM work window. Only numbers of members and sets of members that have not yet been assigned to other reinforcement groups can be entered into the fields.

Members contained in continuous members are automatically deactivated for the design.

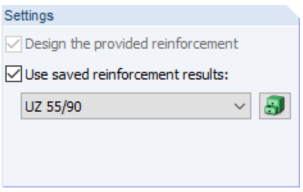
Cross-Section



The cross-section graphic shows how the data entered in the various tabs affects the cross-section. Changes in the reinforcement specifications are carried out dynamically.

The relevant cross-section can be set in the list above the graphic.

Settings



With the *Design the provided reinforcement* option, RF-CONCRETE Members takes the reinforcement specifications to calculate a rebar reinforcement. If you clear this check box, some input fields in the tabs of this module window will be locked. Then, RF-CONCRETE Members will determine only the required reinforcement areas.

If you have set the design for the serviceability limit state or for fire resistance in Window *1.1 General Data*, it is not possible to prevent the calculation of the provided reinforcement: The SLS designs are based on an available bar reinforcement because crack widths, crack spacings, etc. can only be determined on the basis of rebar diameters and rebar spacings. The same applies to the design according to the nonlinear method.

Select the *Use saved reinforcement results* check box to apply a reinforcement stored in the user-defined library of reinforcement templates. Reinforcement templates can be defined and saved in module Window 3.1 (see Chapter 5.2.1). The check box will also be enabled.

The relevant reinforcement template can be selected in the list. Use the button to access the database of reinforcement templates where you can choose among the stored templates (see Figure 3.25).

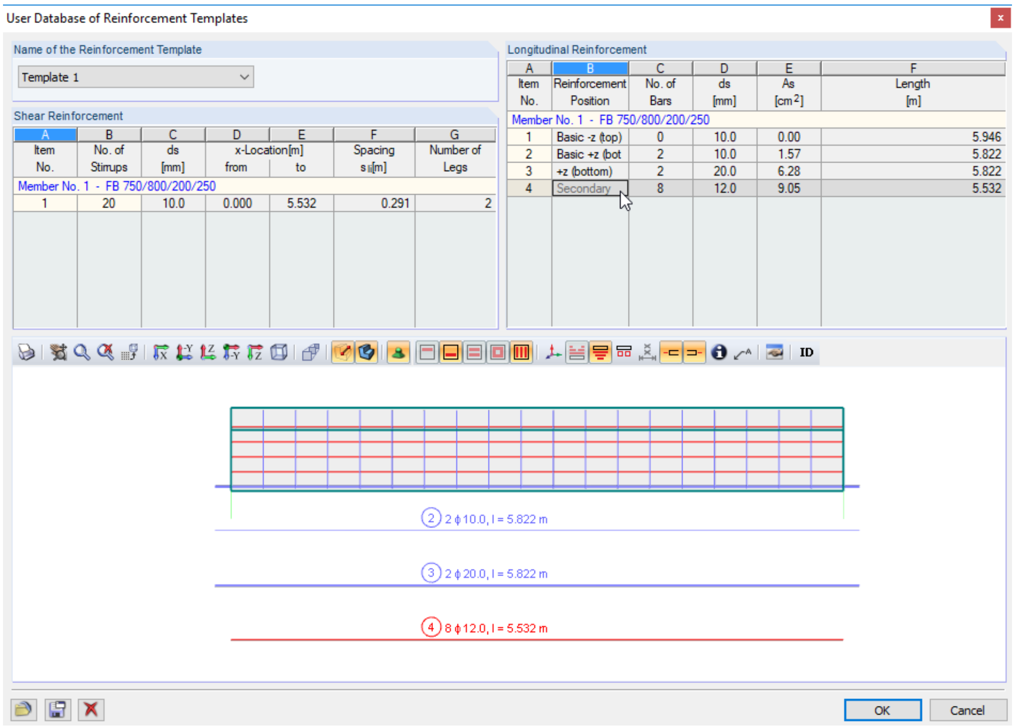


Figure 3.25 User Database of Reinforcement Templates

User-defined reinforcement templates allow for keeping the reinforcing bars and using them for design even if the input data is changed.

The reinforcement stored in the reinforcement template is not dynamic, which means that the reinforcements' position and length are fixed and assigned to a particular member. If the original length of a member changes in RFEM, it is **not** adjusted in the reinforcement template.

3.6.1 Longitudinal Reinforcement

In this tab, you can enter the specifications for the longitudinal reinforcement.

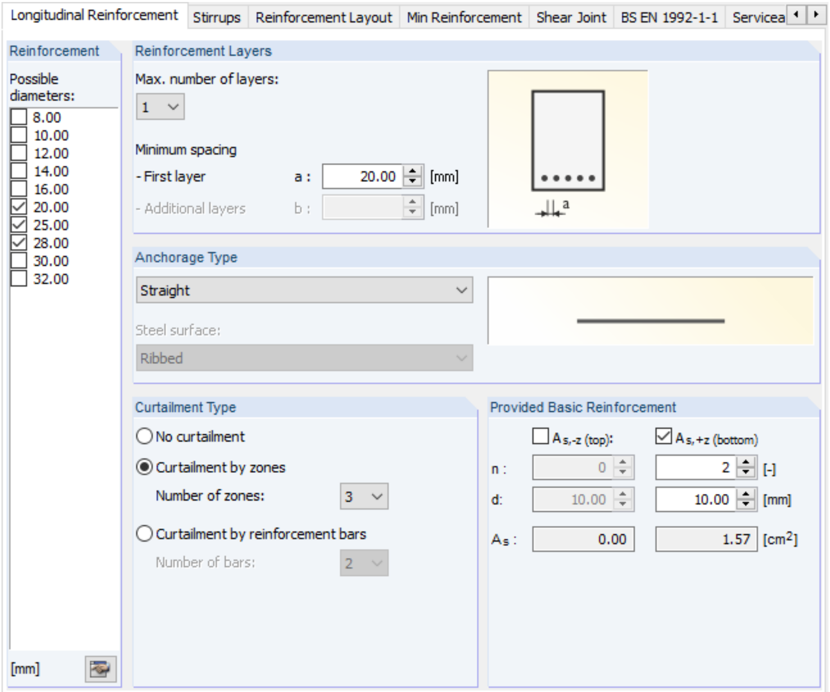


Figure 3.26 Window 1.6 *Reinforcement, Longitudinal Reinforcement* tab

Reinforcing Steel

The list of possible diameters provides commonly used options for reinforcing bars. You can set different diameters for the provided reinforcement.

Use the [Edit] button to adjust the list of available rebar diameters.

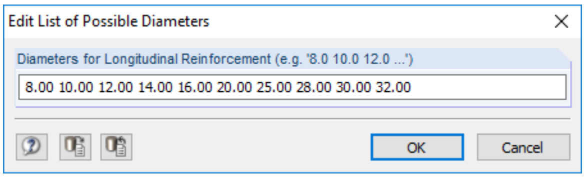
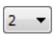


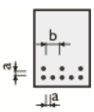
Figure 3.27 *Edit List of Possible Diameters* dialog box

You can modify, remove, or add diameters in the input line.

Reinforcement Layers



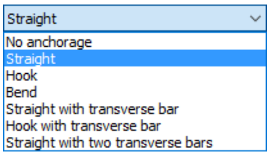
RF-CONCRETE Members also considers multi-layered arrangements of rebars for the provided reinforcement. Use the list to specify the *Max. number of layers*.

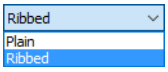


The *Minimum spacing* of rebars *a* for the first layer and, if necessary, *b* for additional layers can be defined in the two input fields.

These structural specifications are considered for the provided reinforcement: They affect the number of possible rebars of each layer and the lever arm of internal forces.

If several reinforcement layers are arranged, a curtailment of the reinforcement is not possible.





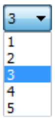
Anchorage Type

Both lists in this dialog section provide a variety of anchorage possibilities. The graphic to the right is dynamic, which means that modified settings are displayed immediately in the graphic.

Anchorage and *Steel surface* (solely ribbed for ACI 318 and EN 1992) affect the required length of anchorage.

Curtailment Type

No curtailment is preset. If several reinforcement layers are specified, the remaining two options are disabled.



If you select a *Curtailment by zones*, use the list to define how many zones with the same reinforcement are allowed in the provided reinforcement. Then, RF-CONCRETE Members will find out how to optimally cover the required steel cross-section areas with the available rebars.

If you select a *Curtailment by reinforcement bars*, the program will only open a new zone when the user-defined maximum number of rebars is reached. Use the list to define this rebar number.

Provided Basic Reinforcement

In this window section, you can specify a basic reinforcement $A_{s,-z}$ for the top layer and $A_{s,+z}$ for the bottom layer. After selecting the check boxes, you can access the input fields to define the number of rebars *n* and the rebar diameters *d*. Based on these specifications, RF-CONCRETE Members determines the reinforcement areas A_s of the basic reinforcement.

When the provided reinforcement is created, the user-defined basic reinforcement is taken into account. It will be inserted over the entire length of the member or set of members. If the required reinforcement cannot be covered by the basic reinforcement, the program determines the additionally required rebars and inserts them into the cross-section.

3.6.2 Stirrups

This tab manages the specifications for the shear reinforcement.

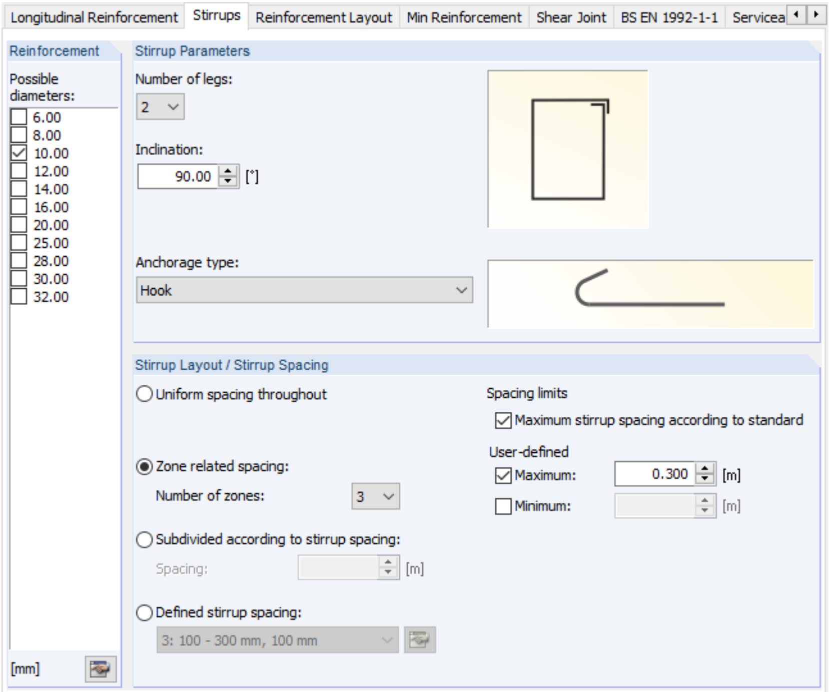
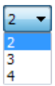


Figure 3.28 Window 1.6 *Reinforcement, Stirrups* tab

Reinforcing Steel

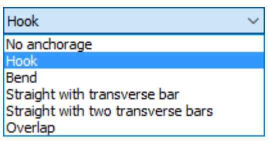
The list of possible diameters provides commonly used options for reinforcing bars. Use the button to adjust the list of available rebar diameters (see Figure 3.27).

Stirrup Parameters



With the *Number of legs* list, you can define how many stirrups are available in the shear force direction. It is possible to specify up to four legs; two legs are default. The changes are represented dynamically in the graphic to the right.

The *Inclination* of the shear reinforcement is defined by the angle between longitudinal and shear reinforcement. The default is 90°, which means perpendicular stirrups. When changing the angle, observe the standard specifications: EN 1992-1-1, 9.2.2 (1) only allows for angles between 45° and 90° and stipulates in 9.2.2 (4) that bent bars acting as shear reinforcement may be used only together with stirrups. At least 50 % of the shear force to be absorbed should be covered by stirrups.

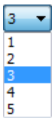


The *Anchorage type* list provides various possibilities for stirrup anchorages, which affect the determination of the anchorage lengths. The changes are represented dynamically in the graphic to the right.

Stirrup Layout / Stirrup Spacing

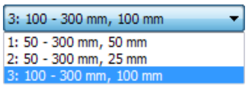
This window section is only accessible if a provided reinforcement is created.

A *Uniform spacing throughout* is default for all members and continuous members.



If you select a *Zone related spacing*, use the list to specify the number of zones with the same stirrup layout. Specifying one zone leads to the creation of another zone in addition to the zone with the maximum stirrup spacing (minimum reinforcement). The additional zone covers the maximum value of the required stirrup reinforcement. In case of two zones, RF-CONCRETE Members determines the mean value from the required minimum and maximum reinforcement and applies the corresponding x-locations in the member as additional zone limits.

If the layout is *Subdivided according to stirrup spacing*, you have to define a spacing for the stirrup zones. The zones will change in the spacing intervals that are also determined from the required minimum and maximum reinforcement with an interpolation method.



When a *Defined stirrup spacing* is selected, you can choose an entry in the list shown on the left. Use the button to adjust these entries or create a new entry with user-defined stirrup spacings.

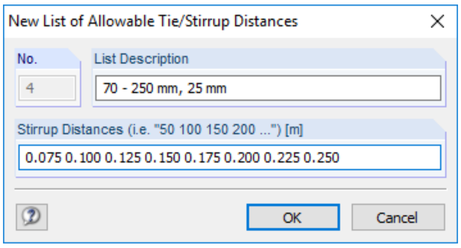


Figure 3.29 *New List of Allowable Tie/Stirrup Distances* dialog box

An example of user-defined stirrup spacings can be found in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000491

It is also possible to define the *Spacing limits* according to the standard or to specify them with user-defined limit values.

The zones presented in the provided reinforcement can be modified or complemented in Window 3.2 *Provided Shear Reinforcement* at any time (see Chapter 5.2.2).

3.6.3 Reinforcement Layout

This tab defines how the reinforcement is inserted and which of the RFEM internal forces will be designed.

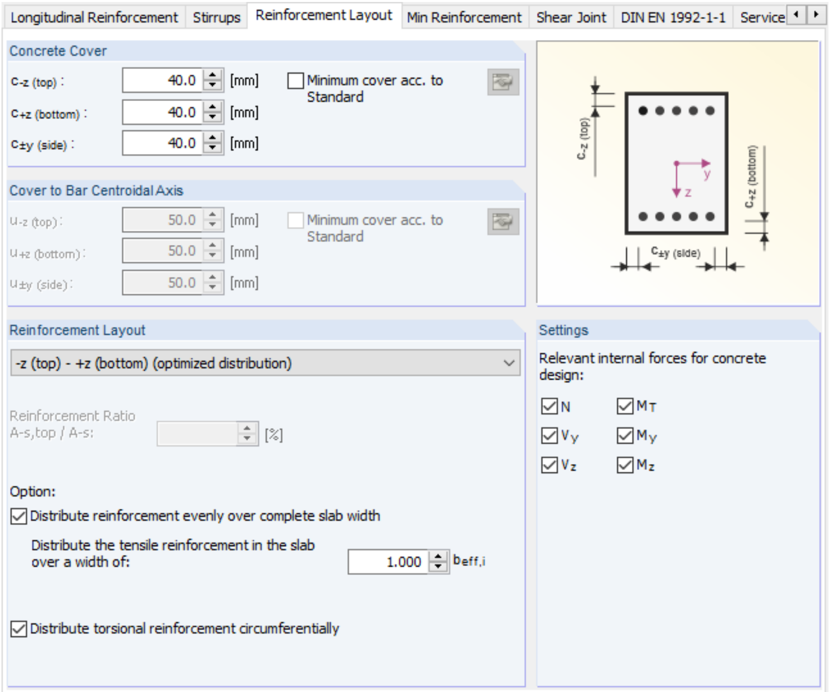


Figure 3.30 Window 1.6 *Reinforcement, Reinforcement Layout* tab

Concrete Cover / Cover to Bar Centroidal Axis

Both window sections are interactive with *Design the provided reinforcement* (see Chapter 3.6): If the check box is selected, the covers refer to the edge distances c. They can be defined in the *Concrete Cover* section. If the provided reinforcement is not desired, the specifications refer to the dimensions of the rebars' centroidal axis *u*. They can be entered in the section *Cover to Bar Centroidal Axis*.

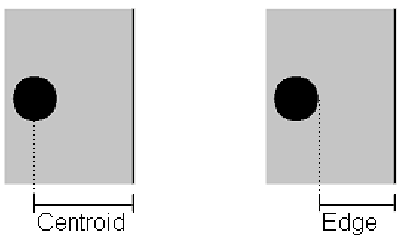
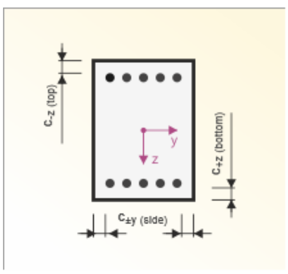


Figure 3.31 Centroidal axis *u* and edge distance c



In the input field $c_{-z(top)}$, you can enter the concrete cover of the longitudinal reinforcement at the top; in the input field $c_{+z(bottom)}$, specify the cover of the longitudinal reinforcement at the bottom. These values represent the nominal values of the concrete cover c_{nom} (e.g. EN 1992-11, 4.4.1.1). Based on these specifications and taking into account the applied rebar diameters, RF-CONCRETE Members determines the lever arm of internal forces.

"Top" and "bottom" are clearly defined by the position of the local member axes in RFEM. The cover $c_{\pm y(side)}$ is needed for the equivalent wall thickness in the torsional design.

The position of the member axes is shown in the cross-section graphic to the right. In the RFEM work window you can display the local member axes by using the member's shortcut menu or the *Display* navigator (see Figure 3.32).

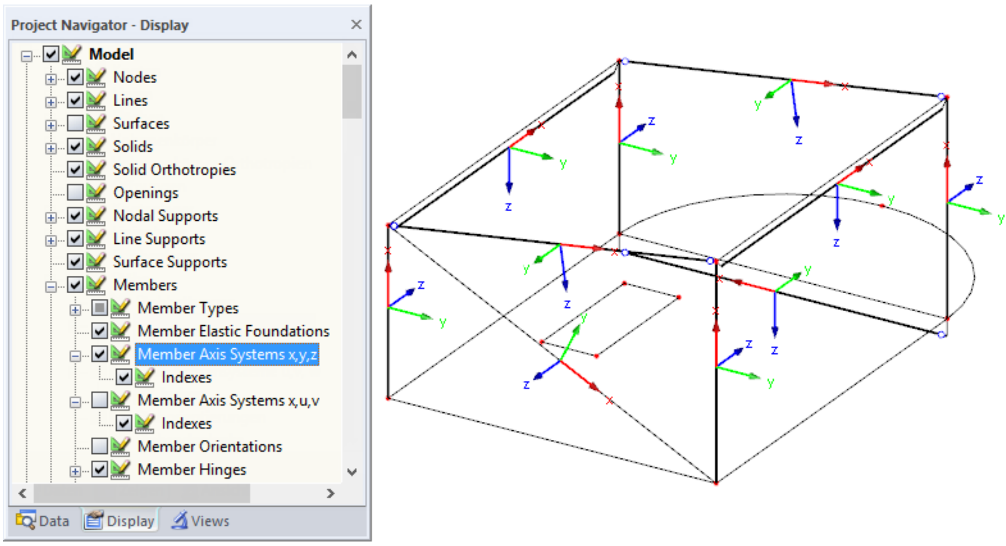


Figure 3.32 Activating the member axis systems in the RFEM *Display* navigator

When entering the edge distances *u* for multilayer reinforcements, the distances must relate to the centroid of the entire reinforcement layer!

☑ Minimum cover acc. to Standard

If you select the *Minimum cover acc. to Standard* check box, the default values of the selected design standard are set. It is also possible to [Edit] the parameters for determining the concrete cover. Clicking the button opens a dialog box with three tabs where you can adjust the specifications for the layouts individually.

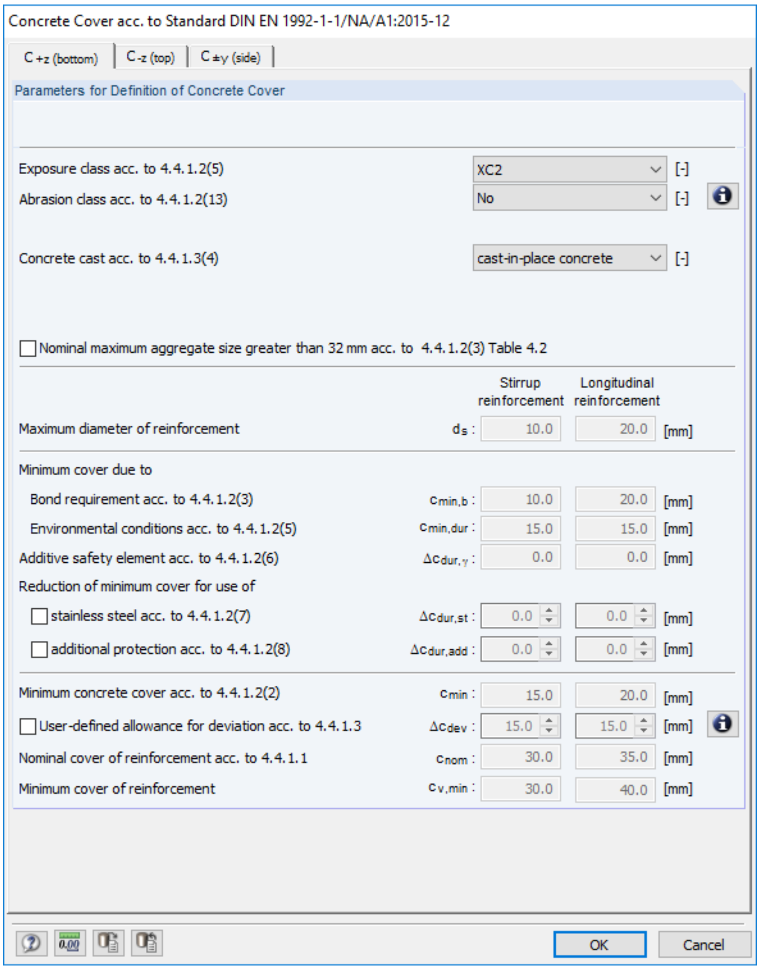
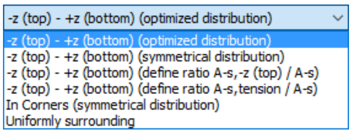


Figure 3.33 *Concrete Cover acc. to Standard* dialog box

Reinforcement Layout

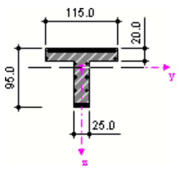


The list contains various ways to arrange the reinforcement in the cross-section:

- top - bottom (optimized distribution)
- top - bottom (symmetrical distribution)
- top - bottom (define ratio $A_{s,top}$ / A_s)
- top - bottom (define ratio $A_{s,tension}$ / A_s)
- in corners (symmetrical distribution)
- uniformly surrounding

For the option *-z (top) - +z (bottom) (optimized distribution)*, RF-CONCRETE Members also performs an optimization for biaxial bending load.

The reinforcement layout can also be defined by a user-defined *Ratio* of top reinforcement or tension reinforcement at the total reinforcement. You can enter the reinforcement ratio in the input field below. This enables you to analyze existing structural buildings.



For T-beams and I-sections, you can *Distribute reinforcement evenly over complete slab width*. With this option, the program releases a part of the rebars. It is also possible to reduce the distribution of the tension reinforcement over the effective width b_{eff} individually with a factor.

If a *torsional reinforcement* is required, it is distributed over the circumference by default.

Modifications to the reinforcement layout are shown dynamically in the graphic to the right.

In case of pure bending about the minor axis (M_y = 0; M_z > 0), increased reinforcement areas result for *top - bottom* reinforcement layouts: The design moment does not act in the specified distributing direction of the reinforcement. In this case, it is recommended to use the reinforcement layout *In Corners*.

Settings

All internal forces (axial and shear forces, bending and torsional moments) are taken into account for the design. This window section allows you to suppress particular internal forces in the performance of the design, for example, if very small torsional moments lead to problems in the shear analysis.

Modifying the default settings is the user's responsibility!

3.6.4 Min Reinforcement

This tab manages the specifications for the minimum and the secondary reinforcement, as well as the parameters for crack width control. The entered data is considered for the ultimate and serviceability limit state designs, but not for the fire resistance design.

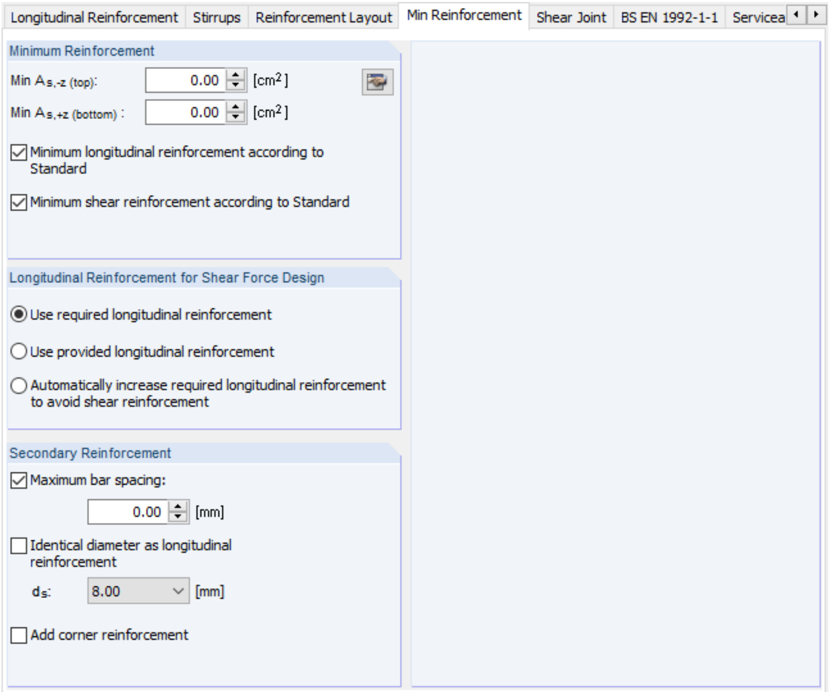


Figure 3.34 Window *1.6 Reinforcement, Min Reinforcement tab*

Minimum Reinforcement

Two input fields are available for specifying a general minimum reinforcement. You can enter the steel cross-sections for *Min* $A_{s,-z\ (top)}$ and *Min* $A_{s,+z\ (bottom)}$. The reinforcement's cross-sectional areas can also be determined from the number of rebars and the rebar diameters. Click the button to open the corresponding dialog box.

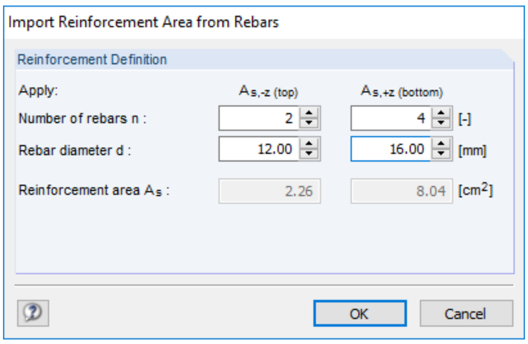


Figure 3.35 *Import Reinforcement Area from Rebars dialog box*

When determining the required reinforcement, it is possible to independently take into account or exclude the *Minimum longitudinal reinforcement* and the *Minimum shear reinforcement* required by the standard.

Longitudinal Reinforcement for Shear Force Design

You can choose among three options to decide, which longitudinal reinforcement is applied in the shear force design without shear reinforcement.

- Use required longitudinal reinforcement

  The design of shear resistance is carried out with the available tension reinforcement.

- Use provided longitudinal reinforcement

  For the design of shear resistance, either the user-defined longitudinal reinforcement or the one suggested by the program is used.

- Automatically increase required longitudinal reinforcement to avoid shear reinforcement

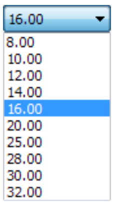
  If the required longitudinal reinforcement is not sufficient for the shear force resistance, the longitudinal reinforcement is increased (at most up to the longitudinal reinforcement ratio of 0.02) until the shear force design is satisfied without any shear reinforcement.

You can find more information about these three options in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000655

Secondary Reinforcement

This window section is enabled if you design a provided reinforcement.

The *Maximum bar spacing* of secondary rebars (which are not structurally required) in the cross-section can be defined by a peak value. The aim of this setting is to find a uniform distribution of bars in the cross-section leading to a secondary reinforcement of T-beam webs or slender rectangular cross-sections.



If you select the *Identical diameter as longitudinal reinforcement* option, the same (minimum) rebar diameter as for the required reinforcement is used. Alternatively, after clearing the check box, you can select another diameter d_s in the list.

Optionally, you can *Add corner reinforcement* in order to set a secondary reinforcement in all corners of the cross-section. This way, it is possible, for example for I-shaped sections, to arrange a reinforcement outside the web.

Similar to the minimum reinforcement, the secondary reinforcement, if sufficiently anchored, is taken into account for the ultimate limit state design and the calculation of crack widths.

3.6.5 Shear Joint

In the design according to EN 1992-1-1, it is possible to design shear joints of casting segments, as well as shear forces between beam web and flange.

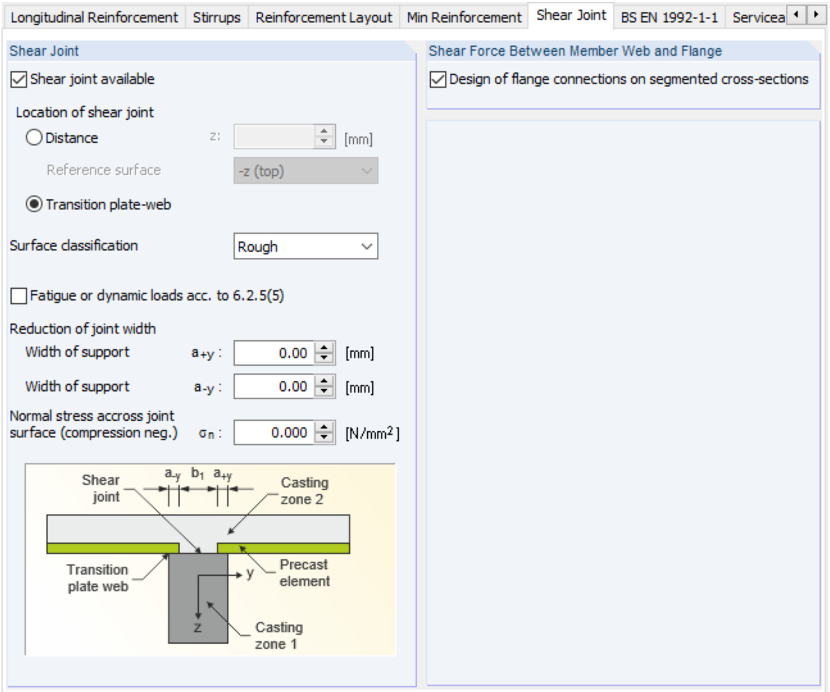


Figure 3.36 *Window 1.6 Reinforcement, Shear Joint tab*

Shear Joint

By selecting the *Shear joint available* check box, you can activate the design of joints, for example in concrete additions for precast members. The other input fields will be enabled for entering the parameters according to EN 1992-1-1, 6.2.5.

The *Location of shear joint* must be defined by the distance from the top or bottom side of the plate. Alternatively, it can be arranged automatically between plate and web.

Rough

Surface	c	μ
Indented	0.500	0.900
Rough	0.400	0.700
Smooth	0.200	0.600
Very smooth	0.025	0.500

The *Surface classification* can be selected in the list. EN 1992-1-1, 6.2.5 (2) describes the different surface categories.

If there is *Fatigue or dynamic loads*, the roughness factors c must be halved according to EN 1922-1-1, 6.2.5 (5). For this, specify the support widths a_{+y} and a_{-y} of the precast elements, as well as the stress σ_n, if applicable, due to the minimum axial force perpendicular to the joint.

Shear Force Between Member Web and Flange

The check box provides the possibility of activating the *Design of flange connections on segmented cross-sections*.

Details

Details...

The [Details] button opens the *Details* dialog box with general calculation settings (see Figure 4.1). In the *Ultimate Limit State* tab, you can define the calculation method for determining the shear stress in shear joints, as well as the longitudinal force difference between member web and flange. The theoretical background is described in Chapter 2.1.4.

3.6.6 Standard

The parameters in this tab depend on the standard set in Window *1.1 General Data* (see Figure 3.2). Here, you can specify standard-specific reinforcement data, which is described in the following for **EN 1992-1-1**.

Use the [Default] button in the bottom right corner of this tab to restore the initial values of the selected standard.

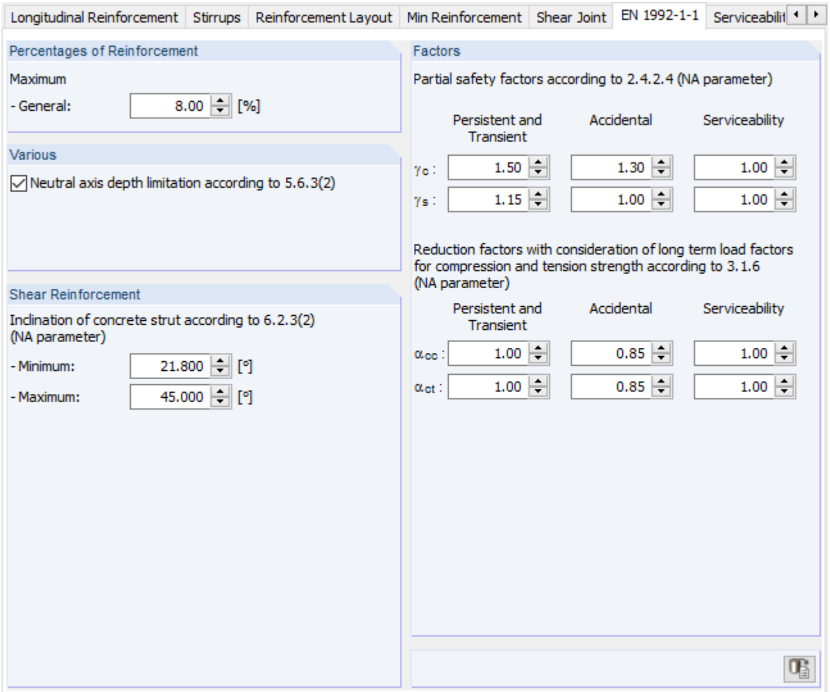


Figure 3.37 Window *1.6 Reinforcement, EN 1992-1-1* tab

Percentages of Reinforcement

The input field controls the general maximum percentage of reinforcement for the beam. EN 1992-1-1, 9.2.1.1 (3) recommends a value of $A_{s,max}$ = 0.04 A_c for tension or compression reinforcement and refers to country-specific regulations. The National Annex for Germany defines the maximum value of the sum resulting from tension and compression reinforcement with $A_{s,max}$ = 0.08 A_c, which must not be exceeded, even in zones of overlapping joints.

Various

If the concrete compression area is no longer able to absorb compression forces, a compression reinforcement is required. This occurs if the bending moment is exceeded, which results from the concrete edge's compression strain of −3.50 ‰ and the strain ε_{yd} when reaching the yield strength of the reinforcing steel. For rebars of the material type 500, a related neutral axis depth of x/d = 0.617 ensues. In case of continuous beams, horizontal beams of non-sway frames, and structural components mainly stressed by bending, you should not use this limiting bending moment to its full capacity in order to ensure a sufficient ability for rotation.

With the check box you can limit the depth of the compression area according to EN 1992-1-1, 5.6.3 (2). In this case, the maximum ratio is x_d/d = 0.45 for concrete up to strength class C50/60 and x_d/d = 0.35 for concrete starting from strength class C55/67.

Shear Reinforcement

The two input fields define the allowable range of the compression strut inclination. If there are user-defined angles beyond the allowed limits of the standard, a corresponding error message appears.

EN 1992-1-1 provides an integrated model for calculating the shear force resistance. For structural components with shear reinforcement perpendicular to the component's axis (α= 90°), the following applies:

$$V_{Rd,s} = \frac{A_{sw}}{s} \cdot z \cdot f_{ywd} \cdot \cot\theta$$

Equation 3.2 Shear resistance according to EN 1992-1-1, Eq. (6.8)

where

A_{sw}	cross-sectional area of shear reinforcement
s	spacing of links
f_{ywd}	design yield strength of shear reinforcement
z	lever arm of internal forces (assumed as 0.9 · d)
θ	inclination of concrete compression strut

The inclination of the concrete compression strut θ may be selected within certain limits depending on the loading. This way, the equation can take into account the fact that a part of the shear force is resisted by crack friction and the virtual truss is thus less stressed. These limits are recommended in EN 1992-1-1, Eq. (6.7N) as follows:

$$1 \leq \cot\theta \leq 2.5$$

Equation 3.3 Compression strut inclination

The compression strut inclination θ can vary between these values:

	Minimum inclination	**Maximum inclination**
θ	21.8°	45°
cot θ	2.5	1

Table 3.2 Limits for compression strut inclination

The National Annex for Germany allows for a flatter compression strut inclination of 18.4°.

$$1.0 \leq \cot\theta \leq \frac{1.2 + 1.4 \cdot \frac{\sigma_{cd}}{f_{cd}}}{1 - \frac{V_{Rd,cc}}{V_{Ed}}} \leq 3.0$$

Equation 3.4 Compression strut inclination NA Germany

A flatter concrete compression strut results in reduced tension forces within the shear reinforcement and thus in a reduced required area of reinforcement.

Factors

The upper input fields control the *Partial safety factors according to 2.4.2.4* for concrete γ_c and reinforcing steel γ_s. The values according to EN 1992-1-1, Table 2.1N are preset for the load-bearing capacity. In the same way, the recommended values are preset for the serviceability. They can be adjusted as needed.

The *Reduction factor* α for considering long-term effects on the concrete strength can be specified separately for compression and tension loads. Again, a differentiation by design situations is possible. The values recommended in EN 1992-1-1, 3.1.6 are preset.

According to the National Annex for Germany, the reduction factor for the concrete compressive strength to be applied is α_{cc} = 0.85, the one for the concrete tensile strength is also α_{ct} = 0.85. According to EN 1992-1-1, remark to 3.1.7 (3), the value $\eta \cdot f_{cd}$ must additionally be reduced by 10 % if the width of the compression area decreases towards the compressed edge of the cross-section. If this condition is given, RF-CONCRETE Members will perform the reduction automatically.

3.6.7 Serviceability Limit State

This tab is displayed if at least one load case or load combination is selected for the serviceability limit state design in Window *1.1 General Data*.

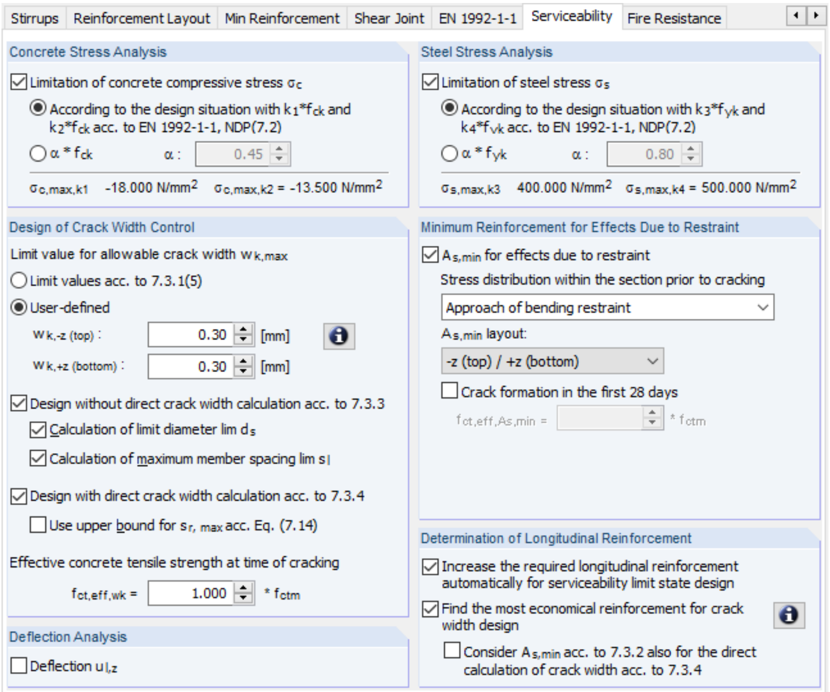


Figure 3.38 Window *1.6 Reinforcement, Serviceability* tab

For information on the theoretical background of serviceability limit state designs, see Chapter 2.2.

For the serviceability limit state design, you can specify various criteria concerning stress design and crack width analysis. Table 3.3 provides an overview of the relevant clauses in the standard.

Design	Normative specifications in EN 1992-1-1
Limitation of concrete stress σ_c	7.2 (1)
Limitation of steel stress σ_s	7.2 (4)
Limitation of crack widths w_k	7.3.1 (5) and 7.3.4
Limit diameter limit d_s	Table 7.2 (see Figure 2.3)
Maximum rebar spacing limit s_l	Table 7.3 (see Figure 2.4)
Minimum reinforcement min A_s	7.3.2 (2)

Table 3.3 Stress designs and crack width analyses

Not all of these designs have to be fulfilled. The design of concrete and steel stresses, for example, can be dispensed with if the internal forces are determined according to the elasticity theory, not more than 15 % are redistributed in the ULS design, and the rules according to EN 1992-1-1, clause 9 are followed.

By deactivating individual designs (e.g. stress designs), they are not considered for the determination of the longitudinal reinforcement. The available results (e.g. concrete and steel stresses under loading in serviceability limit state), however, are still displayed in the result windows 4.1 to 4.4.

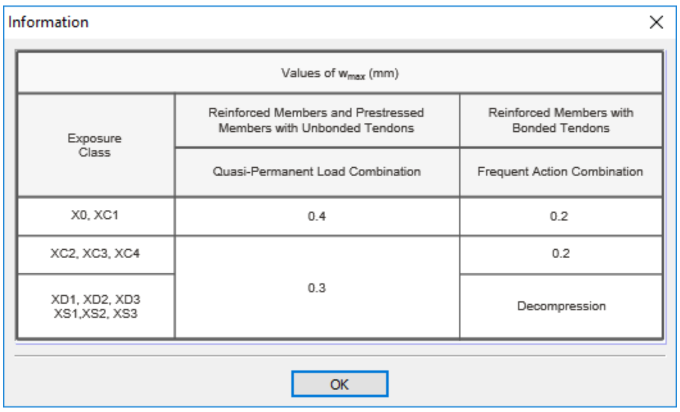
Concrete Stress Analysis

EN 1992-1-1, 7.2 (1) requires the *Limitation of concrete compressive stress* σ_c in order to avoid function-affecting longitudinal cracks, micro cracks, or strong creep.

The concrete compressive stresses can be reduced according to clause 7.2 (2) and 7.2 (3) with the factors k_1 (0.6 recommended) and k_2 (0.45 recommended), or a user-defined factor α.

Design of Crack Width Control

The *Limit value for allowable crack width* $w_{k,max}$ can be specified for the top and bottom side of the member according to EN 1992-1-1, 7.3.1 (5). The [Info] button opens a dialog box with information about the surrounding conditions.



Exposure Class	Values of w_{max} (mm): Reinforced Members and Prestressed Members with Unbonded Tendons – Quasi-Permanent Load Combination	Values of w_{max} (mm): Reinforced Members with Bonded Tendons – Frequent Action Combination
X0, XC1	0.4	0.2
XC2, XC3, XC4	0.3	0.2
XD1, XD2, XD3 XS1,XS2, XS3		Decompression

Figure 3.39 Crack widths depending on exposure class according to EN 1992-1-1

It is also possible to specify the crack widths individually for the top and bottom reinforcement.

For the limitation of crack widths, three different criteria are available according to which the reinforcement is designed:

Design	**Normative specifications in EN 1992-1-1**
Limit diameter limit d_s	Table 7.2 (see Figure 2.3)
Maximum rebar spacing limit s_l	Table 7.3 (see Figure 2.4)
Direct calculation of crack width w_k	7.3.1 (5) and 7.3.4

Table 3.4 Crack width designs

These design criteria are described in Chapter 2.2.4.

Generally, only **one** of the *limit* d_s, *limit* s_l, or w_k criteria must be fulfilled for the crack width design!

With the *Find the most economical reinforcement for crack width design* option in the *Determination of Longitudinal Reinforcement* section (see below), you can check, which of the three criteria can be covered by the least necessary reinforcement area. The program will add individual rebars to the provided reinforcement until the design is successfully fulfilled.

For the direct crack width calculation of w_k, you can define an upper limit for the maximum crack spacing $s_{r,max}$ according to EN 1992-1-1, Eq. (7.14) (see Chapter 2.2.4).

The effective concrete tensile strength $f_{ct,eff,wk}$ at the moment of cracking used for the crack width analysis can be influenced by a factor for the mean concrete tensile strength f_{ctm}. The effective concrete tensile strength $f_{ct,eff,wk}$ is also used in the analytical serviceability limit state design to analyze whether the cross-section is cracked or uncracked. The general settings in the *Serviceability* tab of the *Details* dialog box also allow you to analyze the crack width on the uncracked cross-section, meaning $f_{ct,eff,wk}$ is not reached (see Figure 4.2).

Deflection Analysis

If the check box is selected, the additional module Window *1.7 Deflection Data* is available for entering the member parameters (see Chapter 3.7).

Steel Stress Analysis

EN 1992-1-1, 7.2 (4) requires a *Limitation of steel stress* σ_s in order to avoid non-elastic strains, unallowable crack formations, and deformations.

The steel stresses can be reduced according to clause 7.2 (5) by the factors k_3 (0.8 recommended) and k_4 (1.0 recommended), or by a user-defined factor α.

Minimum Reinforcement for Effects Due to Restraint

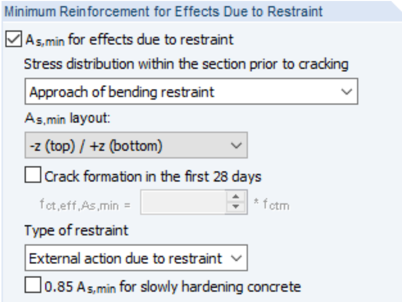
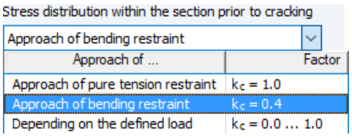


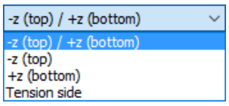
Figure 3.40 Window section *Minimum Reinforcement for Effects Due to Restraint* (for NA DIN and ÖNORM)

When designing the limitation of crack widths, you have to distinguish between load actions and effects due to restraint. An *effect due to restraint* is considerably reduced by crack formation in the structural component. A sufficiently dimensioned minimum reinforcement $A_{s,min}$ provides for an allocation of the entire component reduction to several cracks with accordingly small crack widths. The crack widths due to load actions, however, depend on the available steel stress and the reinforcement layout.

For effects due to restraint, it is necessary for the criterion of the minimum reinforcement $A_{s,min}$ to always be fulfilled.

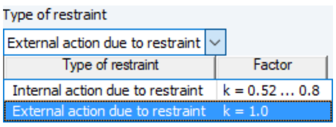


The *Stress distribution within the section prior to cracking* affects the factor k_c according to EN 1992-1-1, Eq. (7.1). Several options are available in the list. $k_c = 1.0$ is applied to centric restraint with pure tension load. In the case of bending restraint with pure bending load, σ_c is equal to zero in the component axis, and thus $k_c = 0.4$ according to Eq. (7.2). Alternatively, it is possible to determine k_c according to Eq. (7.2) or (7.3) dependent on the loading, whereas the mean concrete stress σ_c is determined from the loads. In addition to the stress distribution, the factor k_c approximately considers the increase of the inner lever arm for crack formation.



The *$A_{s,min}$ layout* list defines the reinforcement layer the minimum reinforcement is assigned to.

If you expect a *Crack formation in the first 28 days*, you should reduce the effective concrete tensile strength $f_{ct,eff}$ according to EN 1992-1-1, 7.3.2 (2). You can enter the appropriate reduction factor into the input field. The German National Annex recommends to use $f_{ct,eff} = 0.50 \cdot f_{ctm}$ (28d). If you cannot determine a definite point of time for crack formation within the first 28 days, a tensile strength of at least 3 N/mm^2 for normal concrete should be assumed in accordance with the German National Annex.



The National Annex for Germany for EN 1992-1-1, clause 7.3.2 (2) distinguishes between the types of effect due to restraint for the factor *k* that is used to consider nonlinearly distributed self-equilibrating stresses. You have to specify whether tensile stresses are caused due to

- restraint caused in the structural component itself (e.g. from drain of hydration heat) or
- restraint caused outside of the structural component (e.g. column settlement).

In RF-CONCRETE Members, the crack width is calculated directly for the respective load action according to EN 1992-1-1, 7.3.4. For effects due to restraint, the program designs the minimum reinforcement for limiting the specified crack width according to EN 1992-1-1, clause 7.3.2.

The *0.85 $A_{s,min}$ for slowly hardening concrete* check box allows you to reduce the minimum reinforcements for concretes with r ≤ 0.3 according to the National Annex for Germany or Austria. You can find further information about this in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000889

Determination of Longitudinal Reinforcement

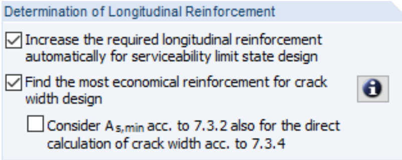


Figure 3.41 *Determination of Longitudinal Reinforcement* window section

The *Increase the required longitudinal reinforcement automatically for serviceability limit state design* check box allows you to design the longitudinal reinforcement so that the design for serviceability is fulfilled. If this option is deactivated, the program uses the provided reinforcement for the SLS design that results from the ultimate limit state design or from manually defined specifications.

The reinforcement's dimensioning for the SLS design is determined by increasing the reinforcement iteratively. The required ULS reinforcement serves as the initial value for iterations. The program analyzes whether it is sufficient to resist the characteristic load. If not, it is increased gradually. The dimensioning process ends without results if the rebar spacing s_l of the reinforcement is as large as the rebar diameter d_{sl}. The result windows will indicate that the respective point cannot be designed.

For the design according to EN 1992-1-1, it is possible to *Find the most economical reinforcement for crack width design*. Click the [Info] button to display information about this procedure. The *Information* dialog box describes when the check of crack width can be considered as being fulfilled.

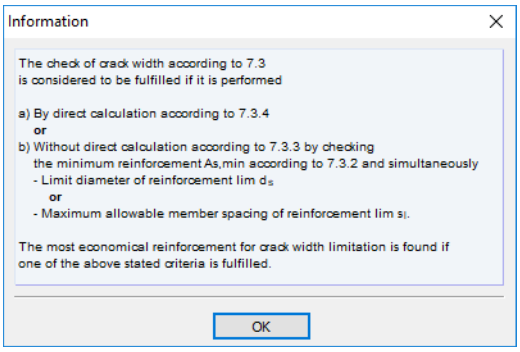


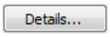
Figure 3.42 *Information* dialog box for determining the most economical reinforcement

Clause 7.2 of EN 1992-1-1 describes, under which conditions the stresses shall be limited. This means that not **all** design ratios shown in Window 4.1 have to be less than 1 in order to fulfill the serviceability limit state design!

The following article on our website describes how the program determines the most economic reinforcement for the crack width analysis:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000506

If the check box to *Consider* $A_{s,min}$ *acc. to 7.3.2 also for the direct calculation of crack width acc. to 7.3.4* is selected, $A_{s,min}$ is also considered if the crack width design is performed in the most economical way by using direct calculation according to 7.3.4. Hence, if the check box is clear, the reinforcement will only be considered if the crack width design is carried out without direct calculation.

Details



The [Details] button opens the *Details* dialog box. In the *Serviceability* tab, you can specify additional settings for the SLS designs.

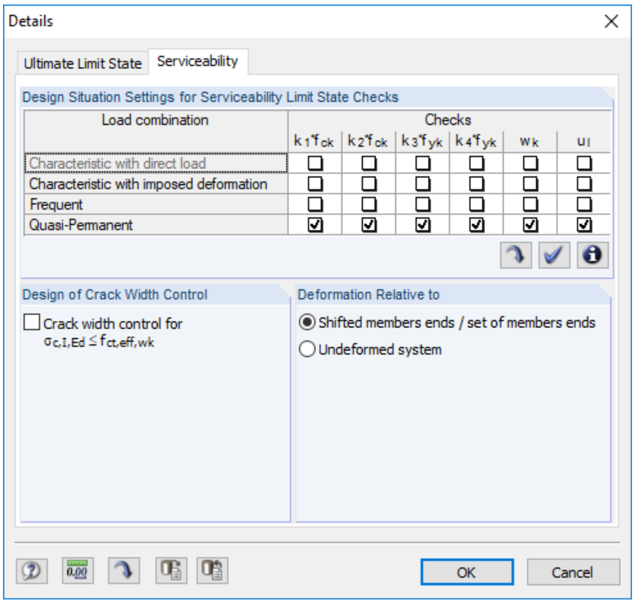


Figure 3.43 *Details* dialog box, *Serviceability* tab

This dialog tab is described in Chapter 4.1.2.

3.6.8 Tapered

This tab only appears if the RFEM model includes tapered members.

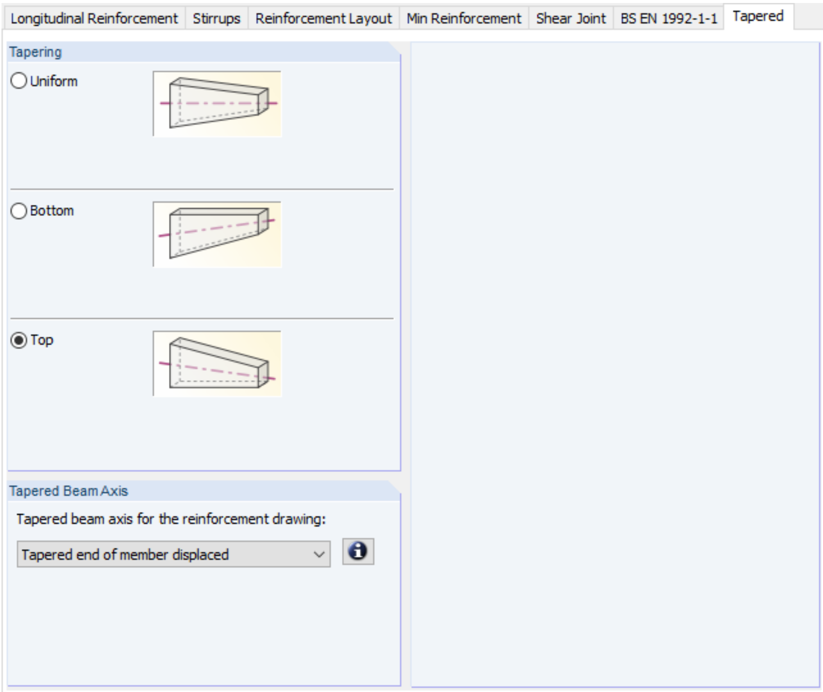


Figure 3.44 *Window 1.6 Reinforcement, Tapered tab*

RF-CONCRETE Members is also able to design tapered members if the same cross-section type is defined for the member start and end. Otherwise it is not possible to interpolate intermediate values, and RFEM displays a corresponding error message before starting the calculation.

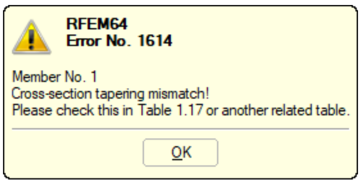
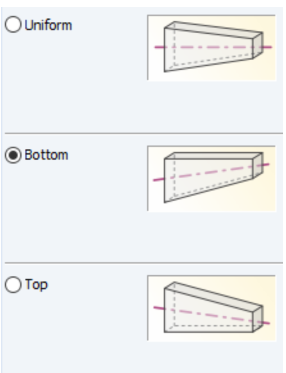


Figure 3.45 Error message in case of incompatible tapered cross-sections

Tapered sets of members are only designed if the entire set of members has a linear cross-sectional profile.



Tapering

The following three options are available to describe the taper in detail:

- uniform
- bottom
- top

This specification affects the design as well as the arrangement of the longitudinal reinforcement (see Figure 3.46).

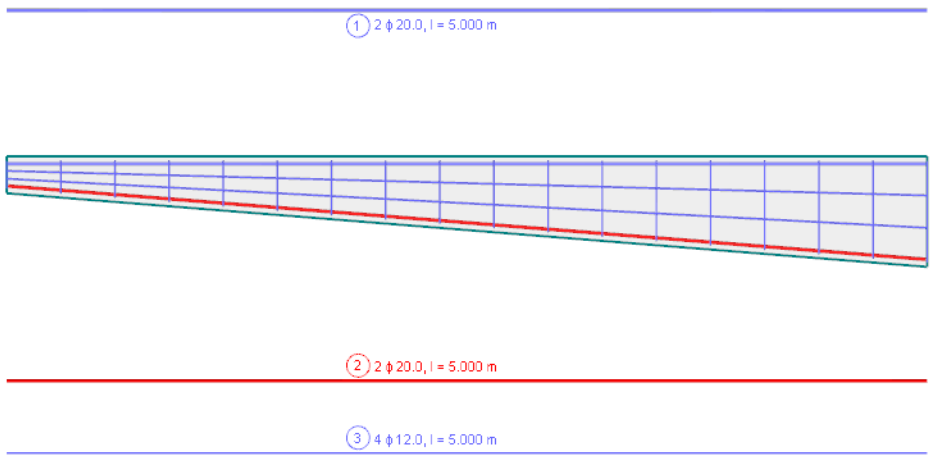
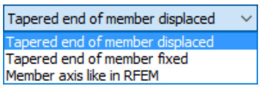


Figure 3.46 Taper with inclined bottom side

Tapered Beam Axis



To display the reinforcement in the 3D rendering of RFEM, you must specify the position of the tapered member axis. Normally, tapers in the structural system are defined centrically in RFEM. However, tapers in RF-CONCRETE Members are usually designed and calculated with displaced taperings. In order to correctly display the reinforcement's connection to the continuing members in the RFEM rendering, you have to specify the member end of the tapered member that will be displaced in construction - provided that it has not yet been taken into account by member eccentricities in RFEM.

Use the [Info] button to see more information.

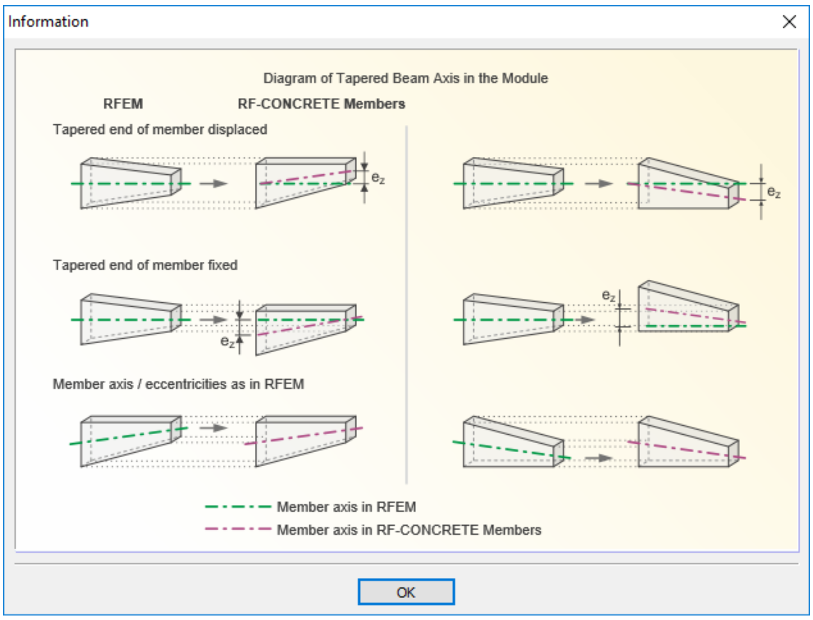


Figure 3.47 *Information* dialog box describing taper member axes

3.6.9 Fire Resistance

The final tab of Window 1.6 is available if at least one load case or load combination is selected for the fire resistance design in Window *1.1 General Data* (see Chapter 3.1.4). The "fire protection design" is performed as per the simplified calculation method according to EN 1992-1-2, clause 4.2 (see Chapter 2.3).

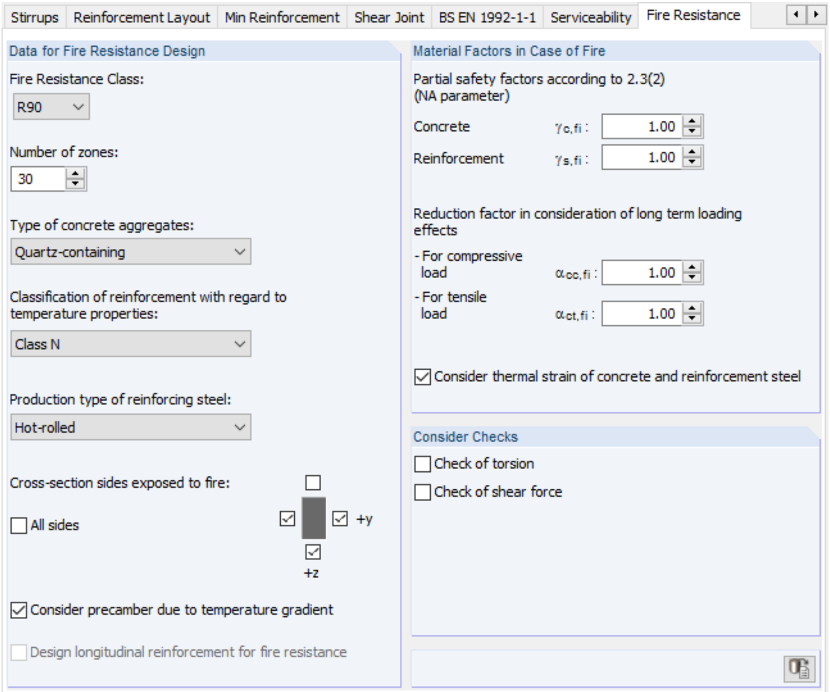
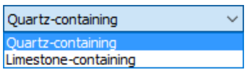
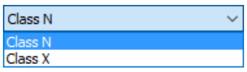


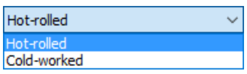
Figure 3.48 *Window 1.6 Reinforcement, Fire Resistance tab*

In the bottom right corner of this tab, you can find the [Default] button for restoring the initial values.

Data for Fire Resistance Design







Five drop-down lists control the parameters that have a decisive influence on the fire resistance design:

- Fire resistance class (according to EN 1992-1-2, clause 1.6.1 (1))
- Number of zones (zone method according to EN 1992-12, Annex B.2)
- Type of concrete aggregates (see Figure 2.9 and Figure 2.11)
- Classification of reinforcement (see Figure 2.14 and Figure 2.15)
- Production type of reinforcing steel (see Figure 2.16 and Figure 2.17)

These parameters are described in the theory Chapter 2.3.

The *Cross-section sides exposed to fire* have to be defined as well. If not *All sides* are affected by charring, clear the corresponding check box (see Figure 3.48). The check boxes around the cross-section symbol to the right become accessible, allowing for specific settings. The directions refer to the local member axes.

In the case of asymmetrical effects of fire, the cross-section is stressed by an additional thermal *precamber* due to the temperature difference that must be considered in the calculation according to EN 1992-1-2, clause 2.4.2 (4). This thermal precamber affects the load-bearing capacity of structural components such as brackets calculated according to the second-order analysis. The program internally creates a member load as precamber of the cross-section and superimposes it with the design loads.

The *Design longitudinal reinforcement for fire resistance* check box controls whether the provided reinforcement also considers effects of fire in addition to the ultimate limit state.

Material Factors in Case of Fire

The two upper input fields control the *Partial safety factors* for concrete γ_c and reinforcing steel γ_s that must be applied for the fire resistance design. The values recommended in EN 1992-1-2, clause 2.3 (2) are preset.

The *Reduction factor* α used to consider long-term effects on the concrete strength in case of fire can be specified separately for compression and tension loads. The value 1.0 recommended in EN 1992-1-1, clause 3.1.6 is preset in both input fields.

With the *Consider thermal strain of concrete and reinforcement steel* option, it is possible to consider the difference between the strain of the "hot" reinforcement and the concrete cross-section's regular thermal strain in the form of a pre-compression strain of the rebar. For loading due to temperature, thermal longitudinal strains occur in concrete and reinforcing steel, varying within the cross-section because of the non-uniform temperature distribution. The thermal strains cannot freely arise everywhere in the cross-section as they are influenced by the adjacent areas. Generally, it may be assumed that the cross-sections remain plane. As the thermal strain of the reinforcement in the cross-section's edge area is restricted, the reinforcement is pre-compressed.

The zone method according to EN 1992-1-2 includes only a calculation of structural components, which means the thermal additional strains in the centroid are not taken into account by the standard. According to Hosser [11], however, these thermal strains must not be neglected for calculations according to the second-order analysis. In his approach, the concrete's thermal strain is calculated across the entire concrete cross-section by using the temperature's mean value.

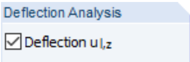
Consider Checks

Annex D to EN 1992-1-2 contains a calculation method for shear and torsional design of structural components exposed to fire. This method can be activated separately for both internal force types.

As this calculation method for shear and torsional design is not allowed in Germany, both options are disabled for a design according to German standards.

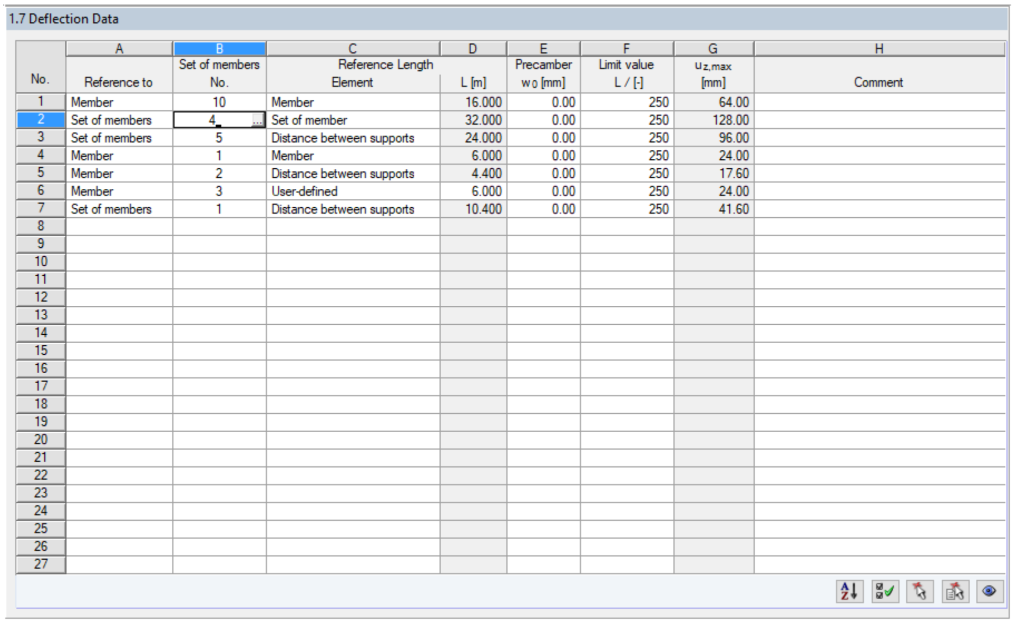
3.7

Deflection Data



Window 1.6, *Serviceability* tab

This window is only available if the deflection analysis is activated in the bottom left in the *Serviceability* tab of Window *1.6 Reinforcement*.



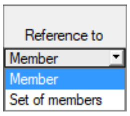
1.7 Deflection Data

No.	A Reference to	B Set of members No.	C Reference Length Element	D L [m]	E Precamber w_0 [mm]	F Limit value L / [-]	G $u_{z,max}$ [mm]	H Comment
1	Member	10	Member	16.000	0.00	250	64.00	
2	Set of members	4	Set of member	32.000	0.00	250	128.00	
3	Set of members	5	Distance between supports	24.000	0.00	250	96.00	
4	Member	1	Member	6.000	0.00	250	24.00	
5	Member	2	Distance between supports	4.400	0.00	250	17.60	
6	Member	3	User-defined	6.000	0.00	250	24.00	
7	Set of members	1	Distance between supports	10.400	0.00	250	41.60	
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								

Figure 3.49 Window *1.7 Deflection Data*

The design criterion of the deflection $u_{l,z}$ considers the displacement in direction of the local member axis z. The reinforcement's layout for the deformation analysis is carried out as per the simplified method according to EN 1992-1-1, clause 7.4.3.

Reference to

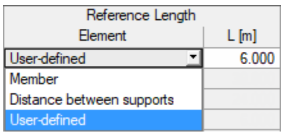


Column A shows if the deflection refers to single members or sets of members.

Member/Set of members No.

In this text box, you can enter the numbers of the members or sets of members to be designed. Click the [...] button for a graphical selection in the RFEM work window. Then, enter the length of the member or set of members as the *Reference Length* in column D.

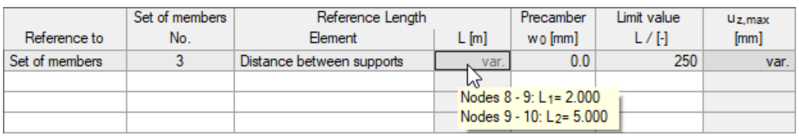
Reference Length



The list in column C allows you to manipulate the reference lengths of the deflections. With the *Member* default setting, the distance between start and end nodes is used. The *Distance between supports* option uses the reduced span length resulting from the support widths specified in Window *1.5 Supports* (see Chapter 3.5).

Use the *User-defined* option to individually specify the reference length in column D.

For sets of members with different field lengths, the program determines the *variable* span lengths automatically. They can be displayed in the tooltip.



Reference to	Set of members No.	Reference Length Element	L [m]	Precamber w_0 [mm]	Limit value L / [-]	$u_{z,max}$ [mm]
Set of members	3	Distance between supports	var.	0.0	250	var.

Figure 3.50 Reference lengths of a set of members with different span lengths

Precamber

In this column, you can enter a *Precamber* w_0.

The shape of the precamber is calculated as follows:

$$w_{c,x} = w_0 \cdot \sin\left(\pi \cdot \frac{x}{L}\right)$$

where

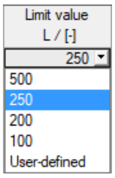
$w_{c,x}$: precamber at location x

w_0 : precamber specified in column E

x : location x

L : length of member or set of members

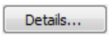
Limit value / $u_{z,max}$



Enter the relative limit value of the deflection in column F. A sag of 1/250 of the span length is preset as recommended in EN 1992-1-1, clause 7.4.1 (4). In the list, you can select another limit value or specify a *User-defined* value.

The maximum allowable deflection is indicated in column G. It is determined from the limit value (column F) and the reference length (column D).

Details



The [Details] button opens the *Details* dialog box. In the *Serviceability* tab, you can set the relation of deformations.

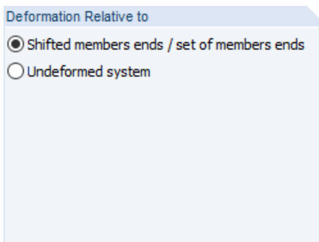


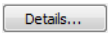
Figure 3.51 Relation of deformations in *Details* dialog box

Both options are described in Chapter 4.1.2.

4 Calculation

RF-CONCRETE Members uses the RFEM internal forces for the design. If results are not yet available in RFEM, the internal forces are calculated automatically.

4.1 Details



The *Details* dialog box manages global settings for the analysis approaches.

4.1.1 Ultimate Limit State

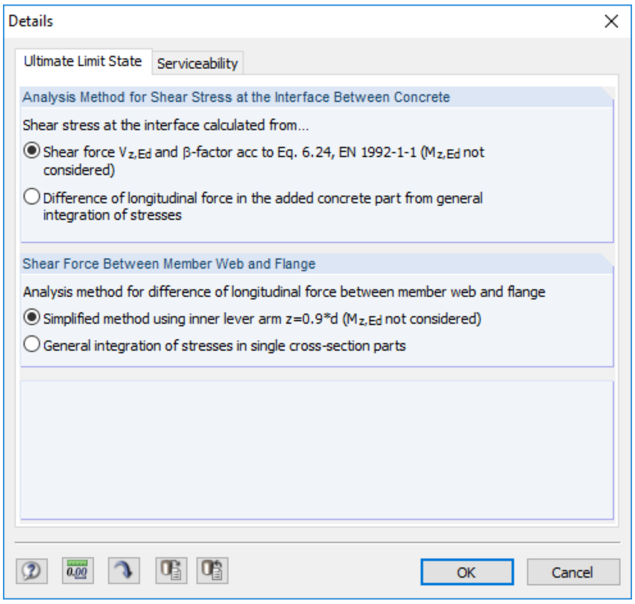


Figure 4.1 *Details dialog box, Ultimate Limit State tab*

Analysis Method for Shear Stress at Interface Between Concrete

In this dialog section, you can specify the calculation method according to which the shear stresses in the shear joint are determined. Both calculation types are described in Chapter 2.1.4.

Shear Force Between Member Web and Flange

Plates on T-beams or box sections acting as compression or tension chords must be connected to the web by a shear-resisting connection. It must be verified that on the one hand the bearing capacity of the concrete compression strut is not exceeded, and on the other hand the tension strut force of the shear reinforcement can be absorbed. The theoretical basis for transferring shear forces between beam web and flanges is described in EN 1992-1-1, clause 6.2.4.

The *Simplified method* is preset for the design. In this method, only moments about the structural component's y-axis with the inner lever arm $z = 0.9 \cdot d$ are considered.

Alternatively, it is possible to determine the longitudinal force difference by a *General integration of stresses in single cross-section parts*. However, this option requires more calculation time. This procedure is described in Chapter 2.1.4.

4.1.2 Serviceability

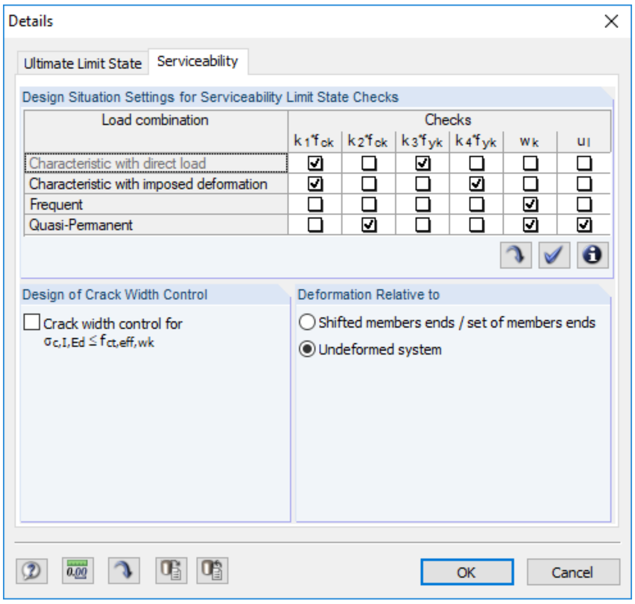


Figure 4.2 *Details* dialog box, *Serviceability* tab

Design Situation Settings for Serviceability Limit State Checks

By selecting the check boxes in the table, you can decide which serviceability limit state designs are performed in the individual design situations. Thus, it is possible to calculate different limit values per design situation directly in a concrete case. For example, with the settings shown in Figure 4.2, the program will analyze the crack widths w_k only with loads of the *Frequent* and *Quasi-Permanent* design situations.

Click the [Info] button to display information describing the requirements the limit values of the serviceability limit state designs are based on.

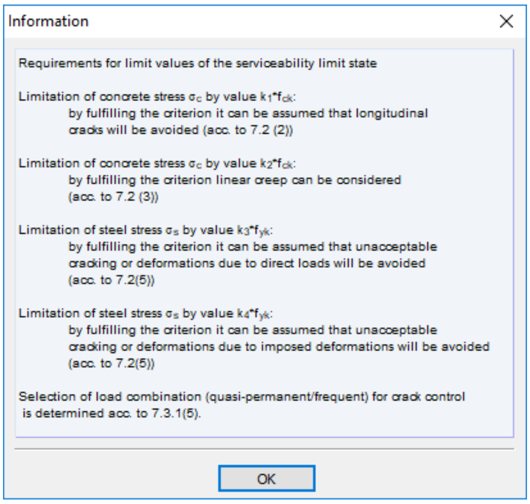


Figure 4.3 *Information* dialog box

Use the [Select or Clear All for Selected Line] button to quickly activate or suppress all checks for a particular design situation.

Design of Crack Width Control

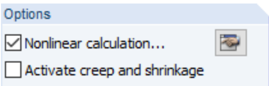
After selecting the *Crack width control for* $\sigma_{c,I.Ed} < f_{ct,eff,wk}$ check box, the crack width analysis will also be performed on the locations where the concrete's effective tensile strength $f_{ct,eff,wk}$ is not reached.

Deformation relative to

The two options control whether the maximum deformations are related to the *Shifted members ends or set of members end* (connection line between start and end nodes of deformed system) or to the *Undeformed system*. Generally, the deformations have to be designed relative to the displacements in the entire structural system.

An example of the relation of member deformations can be found in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001081

4.2 Details for Nonlinear Calculation



It is possible to control the nonlinear calculation by means of parameters that affect the analysis method and the convergence behavior. They are managed in the *Settings for Nonlinear Calculation* dialog box. You can open the dialog box with the button available in Window 1.1 (see Figure 3.1 and Figure 3.9). The dialog box consists of three tabs.

4.2.1 Analysis Method

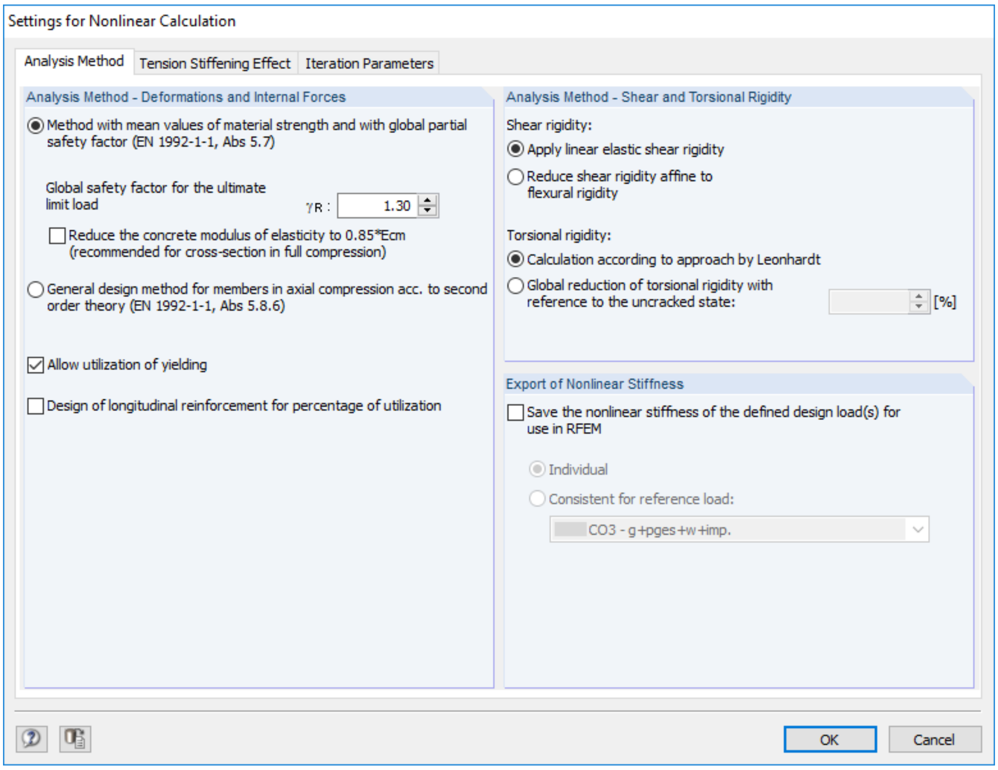


Figure 4.4 *Settings for Nonlinear Calculation* dialog box, *Analysis Method* tab for EN 1992-1-1

The sections in this tab differ depending on the selected design standard (ACI 318 does not provide for nonlinear calculations). The following description refers to EN 1992-1-1.

Analysis Method - Deformations and Internal Forces

Method with mean values of material strength and with global partial safety factor

As described in Chapter 2.4.7, two methods for the nonlinear calculation are specified in EN 1992-1-1. The *Method with mean values* according to EN 1992-1-1, clause 5.7 is preset.

The procedure has been modified in order to consistently use one safety concept only. According to EN 1992-1-1, clause 5.7 (NA.10) for Germany, the global partial safety factor on the ultimate load side is to be applied as follows:

γ_R = 1.3 for permanent and temporary design situations and analysis for fatigue

γ_R = 1.1 for extraordinary design situations

The concrete's modulus of elasticity can be reduced by a factor of 0.85 for the analysis. This is recommended for cross-sections that are fully compressed.

General design method for members in axial compression acc. to second order theory

The *General design method* according to EN 1992-1-1, clause 5.8.6 is mainly suited for the design of slender compression elements. In most cases, the determination of deformations and internal forces using verified mean values leads to more efficient designs. Chapter 2.4.7.2 provides more information about this method.

Allow utilization of yielding

The check box is enabled for both analysis methods (EN 1992-1-1, clauses 5.7 or 5.8.6). The reason for it is that clause 8.6.1 (5) of the German DIN standard 1045-1 does not allow plastic releases (curvatures $(1/r)_m > (1/r)_y$) for structural components stressed by longitudinal compression. Because of the abrupt stiffness decrease when plastic zones or releases are created, the result is often a loss of stability for slender compression elements resulting in failure of the column.

If the check box is clear, no plastic curvatures are possible in the calculation of cross-section curvatures.

Design of longitudinal reinforcement for percentage of utilization

If this option is activated, the longitudinal reinforcement will be increased if the cross-section's load bearing capacity is exceeded. This is the case when the design ratio in the result window of the nonlinear calculation (see Chapter 5.5.1) is greater than 1.

Analysis Method - Shear and Torsional Rigidity

Shear rigidity

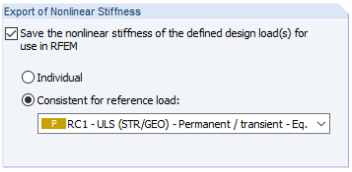
If you *Apply linear elastic shear rigidity*, the shear areas will be calculated linear-elastically. Reduction due to cracking is not taken into account.

Alternatively, you can *Reduce shear rigidity affine to flexural rigidity*. In this case, the linear-elastic shear stiffness diagram will be reduced in line with the diagram of bending stiffness. The theoretical basis is described in Chapter 2.4.5.2.

Torsional rigidity

By default, the torsional stiffness is calculated while considering cracking according to the *approach by Leonhardt* [9] (see Chapter 2.4.5.2).

The *Global reduction of torsional rigidity* allows you to reduce the stiffness for cracking to a user-defined residual value. A residual stiffness of 10 % is preset, which is based on the relatively high decrease of torsional stiffness (see Figure 2.27).



Export of Nonlinear Stiffness

In this dialog section, you can save the stiffness from nonlinear calculations (considering reinforcement and cracked state) to use it later in RFEM. This way, it is possible to also consider the reduced stiffnesses of reinforced concrete components in the cracked state for the determination of internal forces and the design of remaining structural components consisting of steel or timber. This is useful, for example, if the stiffening components of a model are designed in reinforced concrete.

In the *Extra Options* tab of the *Edit Load Cases and Combinations* dialog box in RFEM, you can also find settings for considering the nonlinear stiffness from RF-CONCRETE Members.

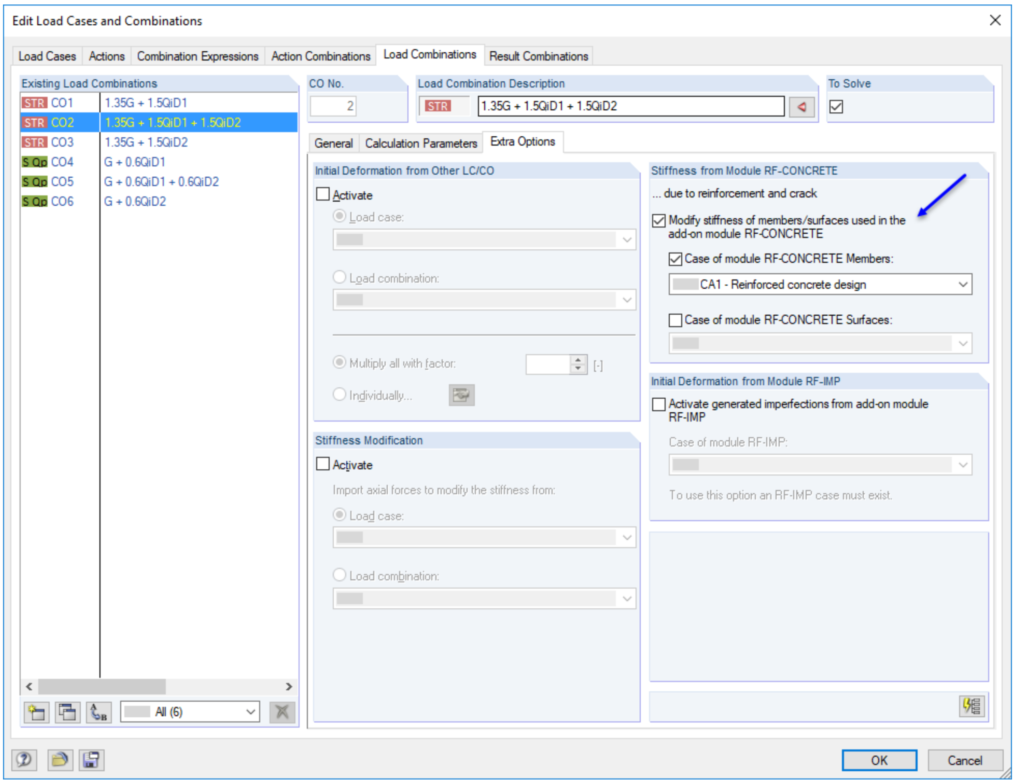


Figure 4.5 RFEM dialog box *Edit Load Cases and Combinations*, *Extra Options* tab

The two options in the add-on module's dialog box control the assignment of stiffnesses for RFEM: The stiffnesses of load combinations calculated linearly with RF-CONCRETE Members can be saved *Individually* (separately). Then, in RFEM, they will only be useable for the respective load combinations. With the *Consistent for reference load* option, however, the stiffness of a reference load is saved, which can then be assigned to any load combination in RFEM.

4.2.2 Tension Stiffening

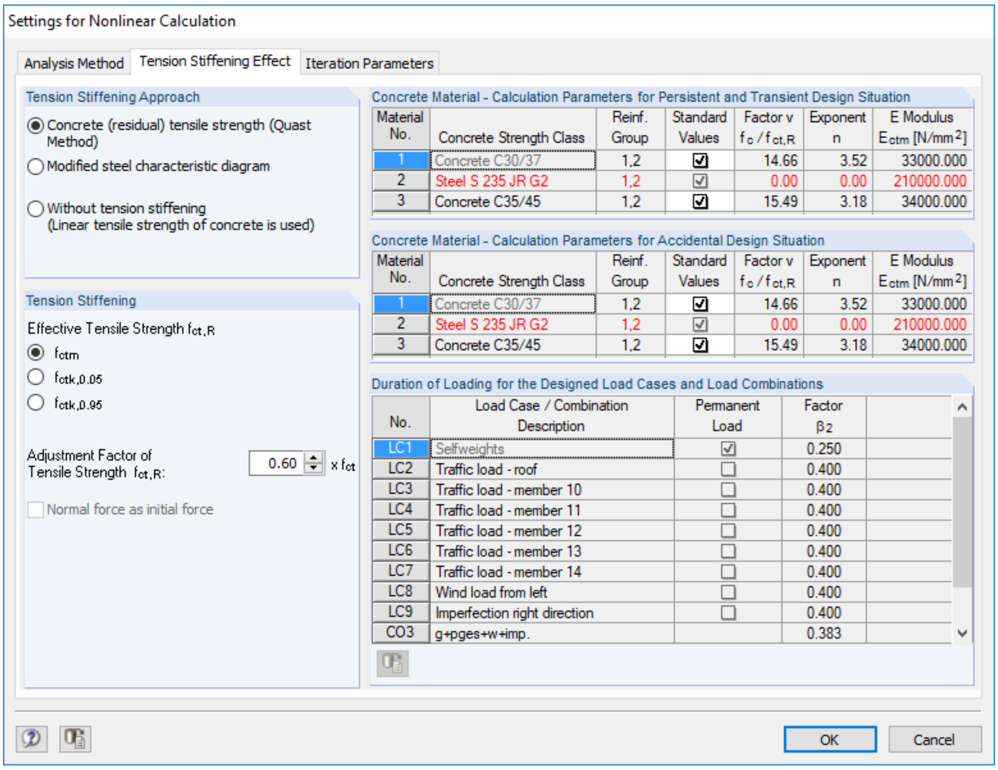


Figure 4.6 *Settings for Nonlinear Calculation* dialog box, *Tension Stiffening Effect* tab

The specifications for the *Tension Stiffening Effect* (effectiveness of concrete between cracks) can be defined separately for the ultimate limit state, the serviceability limit state, and for fire resistance.

Tension Stiffening Approach

Concrete (residual) tensile strength

The approach is based on a residual tensile strength of concrete described by Quast [7] that is defined depending on the governing strain of the steel fibre within the tension zone. This approach is represented graphically in Chapter 2.4.3.1.

Modified steel characteristic diagram

As described in Chapter 2.4.3.2, the *Tension Stiffening Effect* can also be considered through a modified characteristic steel curve. The computing time will be slightly increased, because in addition to the sole calculation in cracked sections (state II), the analysis requires a calculation in the uncracked state, as well as a determination of crack internal forces.

Without tension stiffening

If the *Tension Stiffening Effect* is not taken into account, the program will simply distinguish between cracked and uncracked zones: In uncracked zones, the program calculates using the concrete's corresponding stiffness in uncracked sections (state I, considering the provided longitudinal reinforcement). In cracked zones, it calculates with the stiffnesses available in pure state II.

Tension Stiffening

The calculation values of the concrete tensile strength determine the exponent of the parabola area in a way that results in an increase affine to the compression zone ($E_{cm} = E_{ctm}$).

Effective Tensile Strength $f_{ct,R}$

To take the appropriate safety level into account, it is possible to select one of the following strengths for the concrete tensile strength to be applied:

- f_{ctm} : mean axial tensile strength
- $f_{ctk;\,0.05}$: characteristic value of 5%-quantile of tensile strength
- $f_{tk;\,0.95}$: characteristic value of 95%-quantile of axial tensile strength

Adjustment Factor of Tensile Strength $f_{ct,R}$

The value for the concrete tensile strength $f_{ct,R}$ applied for calculation can be influenced by an adjustment factor. Thus, it is possible to consider boundary conditions such as existing damage.

Pfeiffer [8] suggests a reduction to 60 % of the tensile strengths (default setting).

Normal force as initial force

This check box is important for the calculation of crack internal forces: If it is selected (not possible for method by Quast [7]), the axial force will be kept constant for the calculation of crack moments. This case is applicable in case of an acting prestress, for example. If it is clear, the entire load vector will be considered for the calculation of the crack internal forces.

Concrete Material - Calculation Parameters

The *Standard Values* of the concrete parameters are preset (see Chapter 2.4.3.1). After clearing the check box (fourth table column), you can directly influence the stress-strain curve of the tension zone. As the values are interdependent, the corresponding columns are adjusted accordingly after a modification.

Duration of Loading for the Designed Load Cases and Load Combinations

This dialog section manages the load duration factors β for applying the reduction term (ε_{sr2} - ε_{sr1}), that is the strains of the governing steel fibres for the crack internal forces in the cracked or uncracked state (see Chapter 2.4.3.2). The factor β depends on the time of load duration:

- 0.25 : permanent load or repeated loading
- 0.4 : short-term loading

When applying a *Modified characteristic steel diagram*, you can use the check box to decide if a load case is considered a *Permanent Load* or short-time load.

For load combinations, the applied factor β_2 represents the average of the respective β_2-values of the load cases contained in the combination.

When designing **compression elements**, you generally have to use the *Tension Stiffening* model by Quast. The residual tension force can be influenced by the *Adjustment Factor of Tensile Strength* $f_{ct,R}$.

The *Tension Stiffening* model with the modified characteristic steel curve is based on a distinction between cracked ($M > M_{cr}$) and uncracked zones ($M < M_{cr}$): In the uncracked zone, the program calculates linear-elastically using a constant modulus of elasticity for concrete ($E_{cm,eff}$). In the case of predominant compression, however, considerably expanded curvatures occur for minor moment loadings due to the nonlinear diagram of the concrete's stress-strain curve. Thus, results may be very much on the unsafe side.

4.2.3 Iteration Parameters

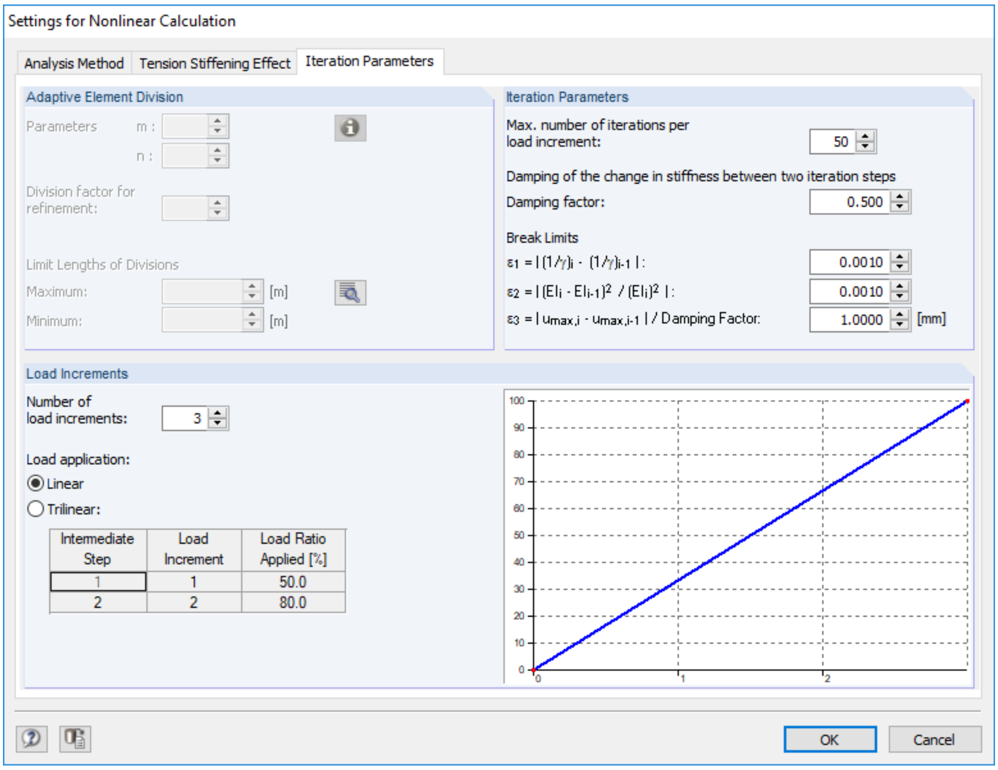


Figure 4.7 *Settings for Nonlinear Calculation* dialog box, *Iteration Parameters* tab

For more information about this tab, see Chapter 2.4.9.

Adaptive Element Division

The locked setting options of this dialog section are not required for RF-CONCRETE Members: The FE division allowing for a fine control of the convergence behavior is used for nonlinear calculations.

Click the button to open the *FE Mesh* dialog box where you can adjust the global target length of elements of the FE mesh and the division specifications for members.

Iteration Parameters

You can control the iteration process with the parameters in this dialog section.

Max. number of iterations per load increment

The iteration process strongly depends on the cross-section shape, the structural system, and the loading. Thus, the number of iterations required to reach the break-off limits is exposed to strong fluctuations. The preset value of *50* iterations is sufficient for most practical applications but can be adjusted, if necessary.

Damping of the change in stiffness between two iteration steps

The program determines the difference in stiffness on a node in the course of two successive iteration steps. The *Damping factor* represents the part of the stiffness difference that is considered for the new stiffness applied in the subsequent iteration step. By reducing the stiffness changes between two iteration steps, it is possible to counteract the calculation's oscillation.

The higher the damping factor, the smaller the damping's influence. If the factor is equal to 1, the damping does not affect the iterative calculation.

Break Limits ε_1 / ε_2 / ε_3

The break-off limits can be adjusted depending on purpose and function: Even if relatively roughly defined break limits ($\varepsilon_1 = \varepsilon_2 \leq 0.01$) lead to sufficiently accurate results when calculating according to linear static analysis (beam deformations in SLS, for example), it is nevertheless recommended to refine the tolerances used for stability analyses ($\varepsilon_1 = \varepsilon_2 \leq 0.001$). Example 3 in Chapter 9.3 illustrates the effect clearly.

With the break limit ε_3, you can additionally control the deformation change. This criterion observes how the size of the maximum deformation changes. The specified damping factor is also taken into account.

Load Increments

The loading can be gradually applied in order to avoid or attenuate an abrupt stiffness change within the individual finite elements ("adapting" the system to the loading). The aim is to avoid the generation of major stiffness changes during an iteration. When the loading is applied step-by-step, it is possible in the iteration step of a load increment to always fall back on the corresponding final stiffness of the element from the previous load increment.

Number of load increments

This input field determines the number of individual load increments for the nonlinear calculation.

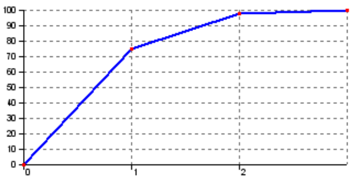
Load application

- Linear

  The load is applied in linear steps.

- Trilinear

  As you can only react to the load-dependent stiffness development with a correspondingly fine gradation when applying loads linearly, RF-CONCRETE Members provides a trilinear load application as an alternative. Thus, it is possible to respond accordingly to boundary conditions like creeping near the state of failure.



Trilinear load application: 75 % / 98 % / 100 %

The trilinear load application is managed by a table: You have to specify two intermediate points that characterize the respective applied load ratio.

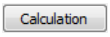
4.3 Check

Before you start the calculation, it is recommended to check if the input data is correct. The [Check] button is available in every input window of RF-CONCRETE Members.

The program checks if the data required for the design is complete and if the references of the data sets are defined sensibly. If the program does not detect any input errors, the following message is displayed.

Figure 4.8 Plausibility check

4.4 Starting the Calculation



In every input window of RF-CONCRETE Members, you can start the calculation with the [Calculation] button.

RF-CONCRETE Members searches for the results of the load cases, load combinations, and result combinations to be designed. If they cannot be found, the RFEM calculation initially starts in order to determine the design-relevant internal forces.

You can also start the calculation in the RFEM user interface: The *To Calculate* dialog box (menu **Calculate → To Calculate**) lists the design cases of the add-on modules such as load cases or load combinations.

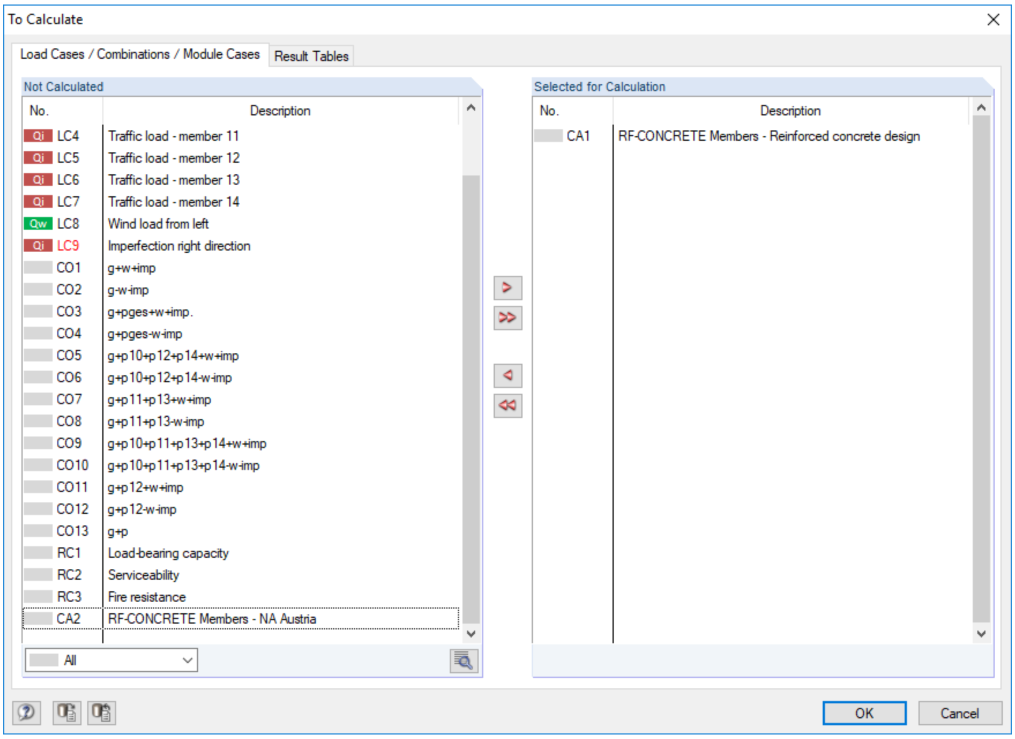
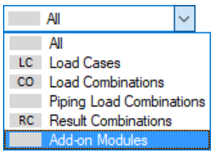


Figure 4.9 *To Calculate* dialog box



If the design cases of RF-CONCRETE Members are missing in the *Not Calculated* section, select *All* or *Add-on Modules* from the drop-down list below.

To transfer the selected RF-CONCRETE Members cases to the list on the right, click [>]. Then, click [OK] to start the calculation.

To calculate a design case directly, use the drop-down list in the RFEM toolbar: Set the RF-CONCRETE Members case and click the [Show Results] button.

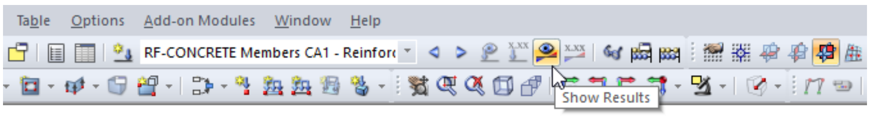


Figure 4.10 Direct calculation of an RF-CONCRETE Members design case in RFEM

You can subsequently observe the calculation process in the solver dialog box (see Figure 9.20).

5 Results

Window 2.1 *Required Reinforcement by Cross-Section* is displayed after the calculation.

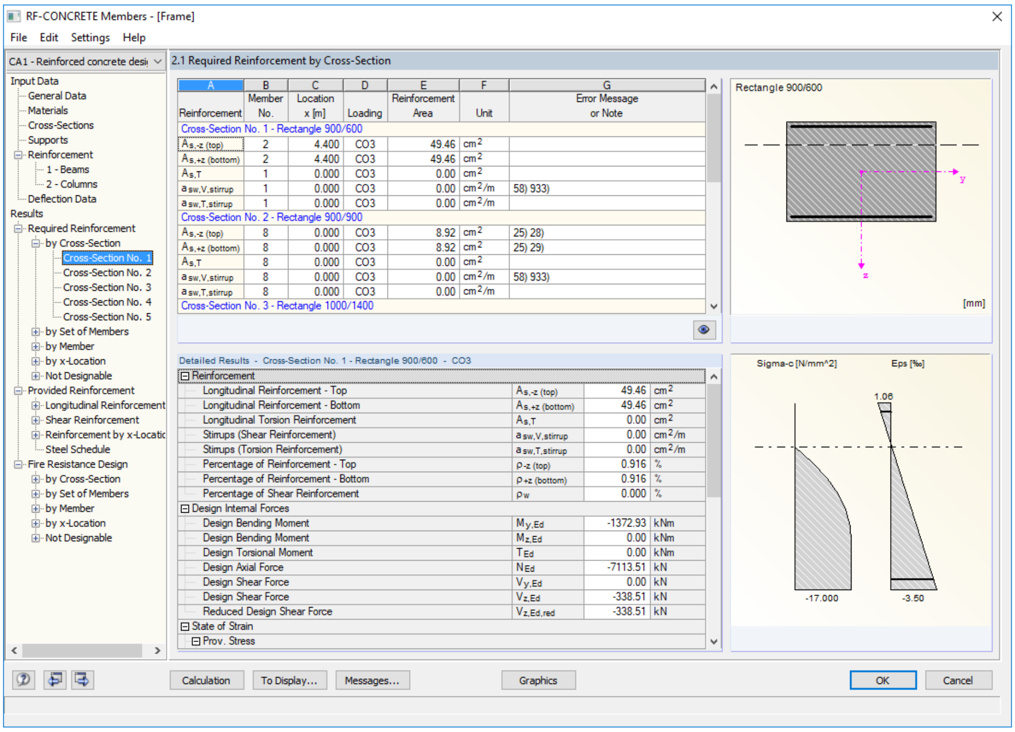


Figure 5.1 Result window

The reinforcement areas required for the ultimate limit state design are listed in the result windows 2.1 to 2.4. If the program created a reinforcement proposal, the provided reinforcement including steel schedule is displayed in the result windows 3.1 to 3.4.

Windows 4.1 to 4.4 contain the results of the serviceability limit state designs. Windows 5.1 to 5.4 show the results of the fire resistance designs.

Windows 6.1 to 6.4 are reserved for the results determined by a nonlinear design.

Every window can be selected directly by clicking the corresponding entry in the navigator. To go to the previous or subsequent module window, use the buttons shown on the left. You can also use the function keys [F2] and [F3] to go through the windows.

OK

Click [OK] to save the results. RF-CONCRETE Members closes and you return to the main program.

Chapter 5 describes the result windows one by one. Evaluating and checking results is described in Chapter 6.

5.1 Required Reinforcement

5.1.1 Required Reinforcement by Cross-Section

The table shows the maximum reinforcement areas of all analyzed members, which are determined from the internal forces of the load cases, load and result combinations selected for the ultimate limit state design.

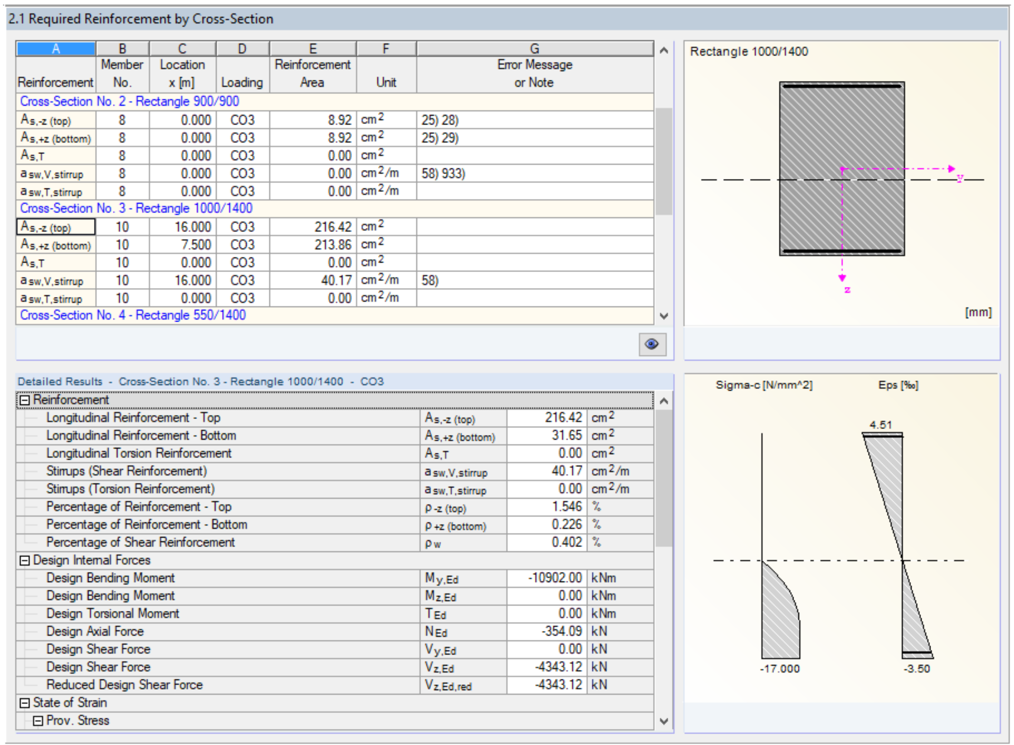


Figure 5.2 Window 2.1 *Required Reinforcement by Cross-Section*

The program displays the maximum required reinforcement areas resulting from the parameters of the reinforcement groups and the internal forces of the governing actions for all designed cross-sections.

The reinforcement areas of the longitudinal and the shear reinforcement are sorted by cross-sections. Both parts of the window show the reinforcement types and design details that are selected in the *Results to Display* dialog box (see Figure 5.3).

The lower part of the window lists all *Detailed Results* for the table row selected above. These design details allow for a specific evaluation of the results. If you select another table row in the upper part, the detailed results will automatically be updated in the lower part.

Reinforcement

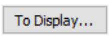
The following longitudinal and shear reinforcements are preset:

Reinforcement	Explanation
$A_{s,-z(top)}$	Reinforcement area of the required top longitudinal reinforcement due to axial force, or bending with or without axial force
$A_{s,+z(bottom)}$	Reinforcement area of the required bottom longitudinal reinforcement due to axial force, or bending with or without axial force
$A_{s,T}$	Reinforcement area of torsional axial reinforcement, if required
$a_{sw,V,stirrup}$	Area of required shear reinforcement for absorption of shear force, in relation to the standard length 1 m
$a_{sw,T,stirrup}$	Area of required shear reinforcement for absorption of torsional moment, in relation to the standard length 1 m
$a_{sf,-z(top)}$	Area of required shear reinforcement for absorption of shear forces between beam web and flanges on the -z side of the cross-section, in relation to the standard length 1 m
$a_{sw,T,stirrup}$	Area of required shear reinforcement for absorption of shear forces between beam web and flanges on the +z-side, in relation to the standard length 1 m

Table 5.1 Longitudinal and shear reinforcements

Top and *Bottom* layer

The top reinforcement is defined on the member side in direction of the negative local member axis z (-z), the bottom reinforcement correspondingly in the direction of the positive axis z (+z). To display the member axes, use the *Display* navigator in the RFEM graphical user interface or the shortcut menu of the member (see Figure 3.32).



Click the [To Display] button to open a dialog box where you can specify the reinforcement results to be displayed in the upper part of the window.

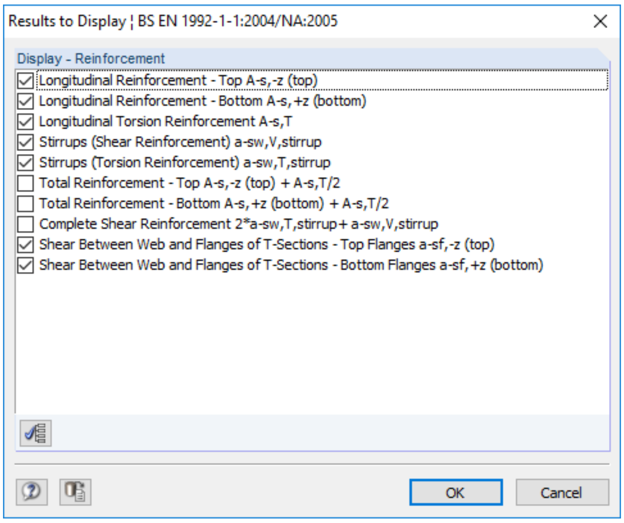


Figure 5.3 *Results to Display* dialog box

The settings in this dialog box also control the results output in the printout report!

Member No.

For each cross-section and each reinforcement type, the table shows the number of the member that has the maximum reinforcement area.

Location x

The column shows the x-location on the member for which the program has determined the maximum reinforcement. The following RFEM member locations *x* are used for the table output:

- Start and end node
- Division points according to member division, if specified (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Extreme values of internal forces

LC / CO / RC

This column shows the numbers of the load cases, load or result combinations that are governing for the respective designs.

Reinforcement Area

For each reinforcement type, column E informs about the maximum reinforcement areas required for the ultimate limit state design.

The *Units* of the reinforcements shown in column F can be adjusted with the menu option

Settings → Units and Decimal Places.

The dialog box described in Chapter 8.3 opens.

Error Message or Note

The final column indicates non-designable situations or notes referring to design issues. The numbers are explained in the status bar.

Use the button shown on the left to view all [Messages] of the current design case.

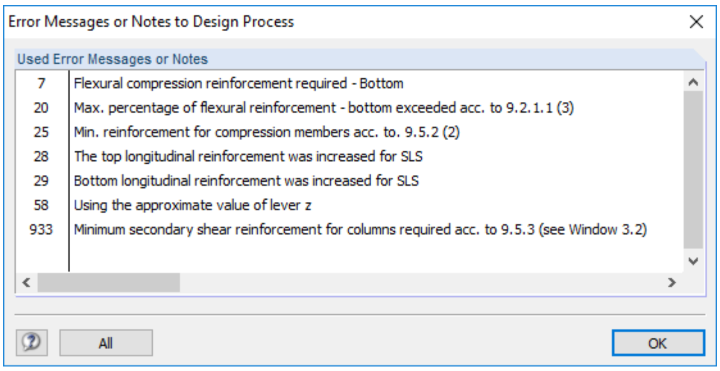


Figure 5.4 *Error Messages or Notes to Design Process dialog box*

5.1.2 Required Reinforcement by Set of Members

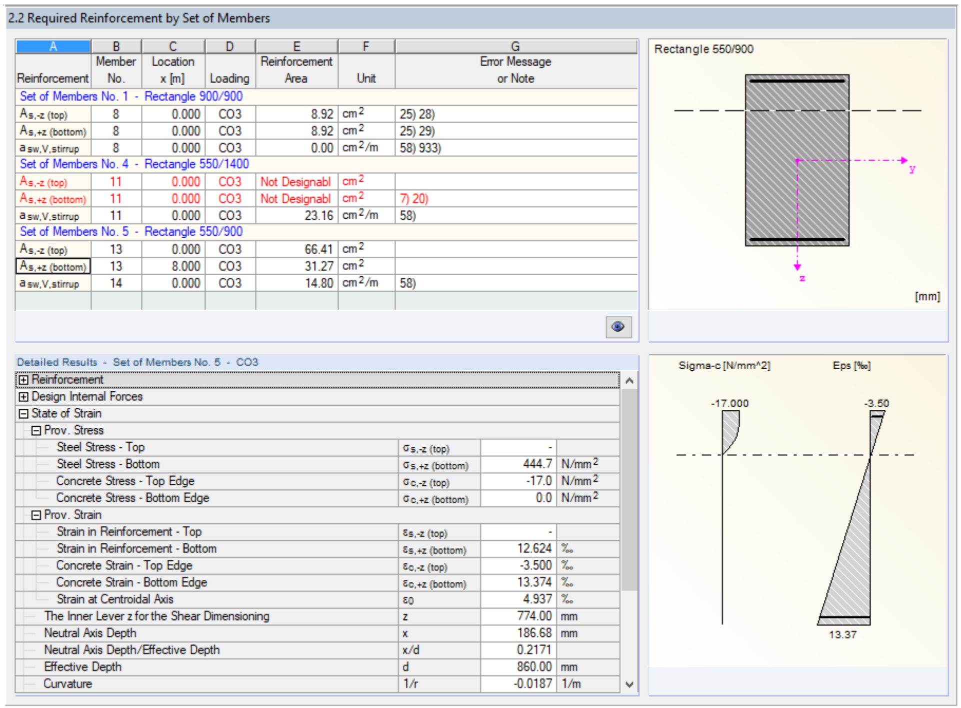


Figure 5.5 *Window 2.2 Required Reinforcement by Set of Members*

This window lists the maximum reinforcement areas that are required for each of the designed sets of members. The columns are described in detail in Chapter 5.1.1.

5.1.3 Required Reinforcement by Member

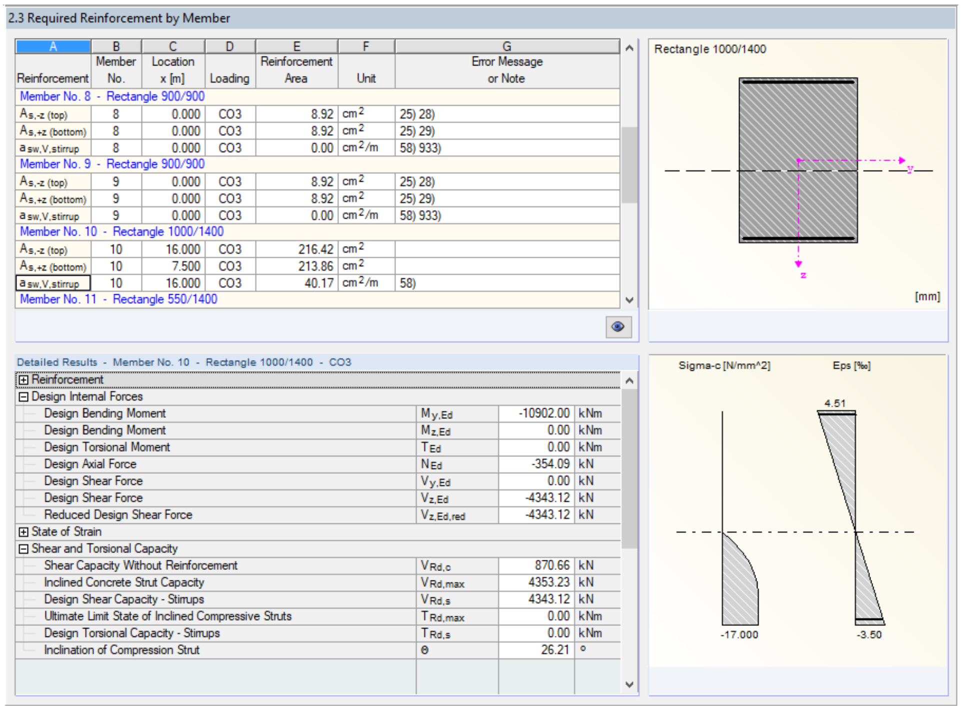


Figure 5.6 *Window 2.3 Required Reinforcement by Member*

The maximum reinforcement areas are sorted by members. In case of tapered members, the table shows the cross-section descriptions available for the member start and end.

5.1.4 Required Reinforcement by x-Location

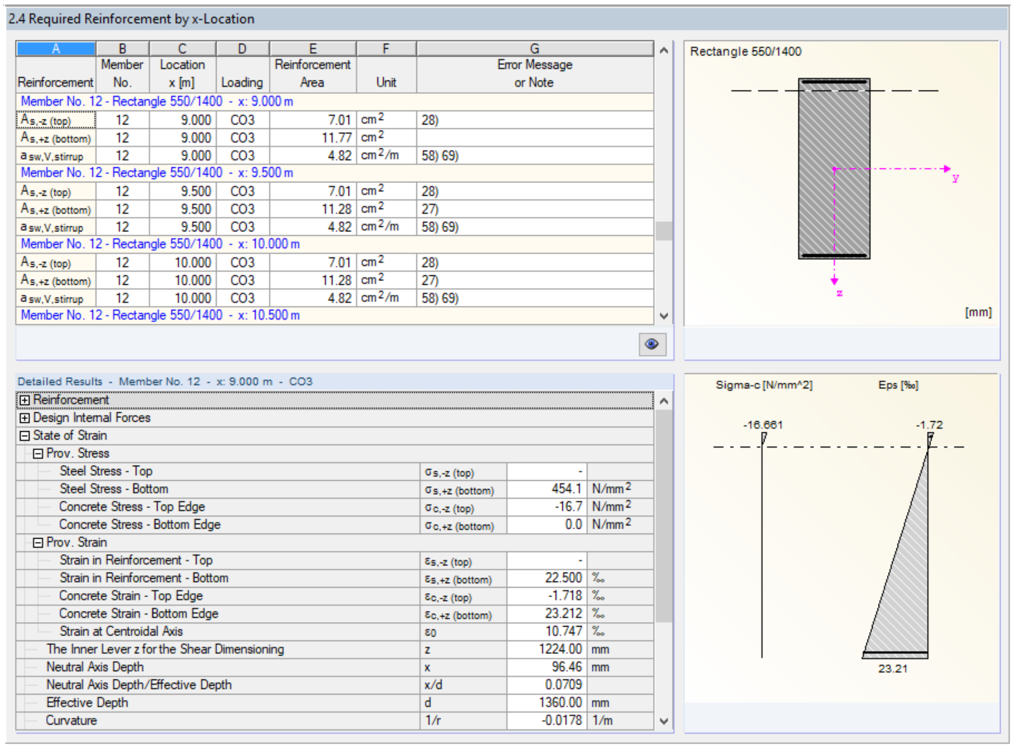


Figure 5.7 Window *2.4 Required Reinforcement by x-Location*

This table shows the required reinforcement areas including detailed results sorted by x-location for each member:

- Start and end node
- Division points according to member division, if specified (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Extreme values of internal forces

Locations of discontinuity are indicated separately.

The window provides the option to specifically access information about the design results. This way, you can check, for example, the required shear reinforcement with related details for a particular member location (designed location).

The individual columns are described in Chapter 5.1.1.

5.1.5 Required Reinforcement Not Designable

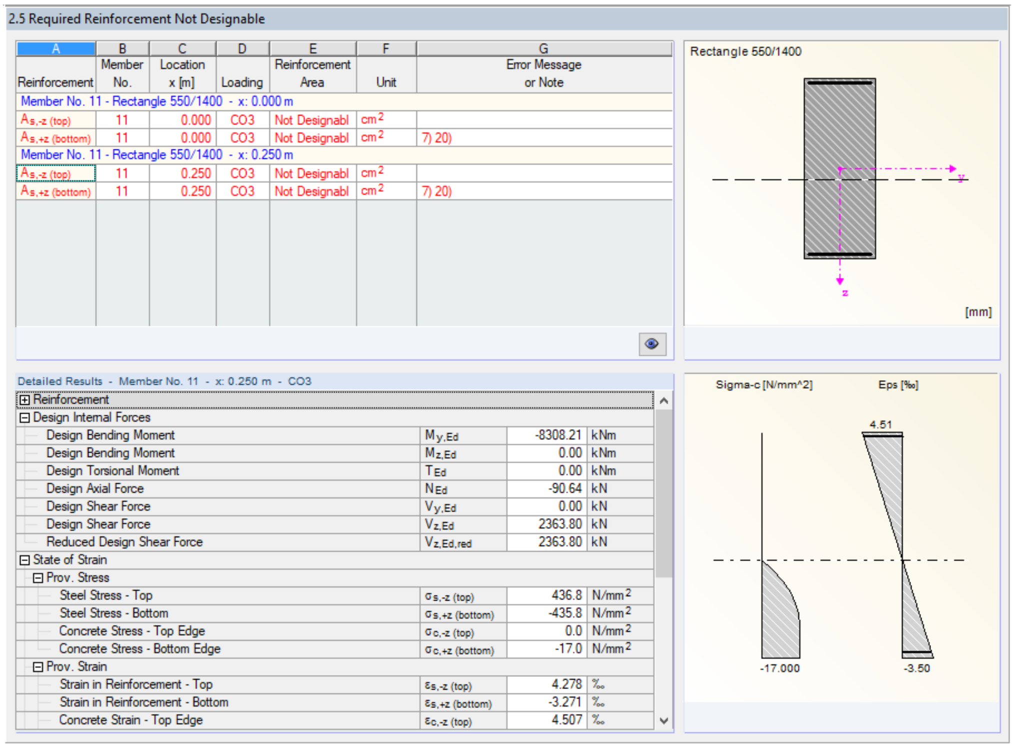


Figure 5.8 *Window 2.5 Required Reinforcement Not Designable*

This window is only displayed if the program has detected non-designable situations or any other problems during the calculation. The error messages are sorted by members and x-locations.

The number of the *Error Message* indicated in column G is described by comments in the footer.

Messages...

Use the [Messages] button to display all issues that have occurred in the design process.

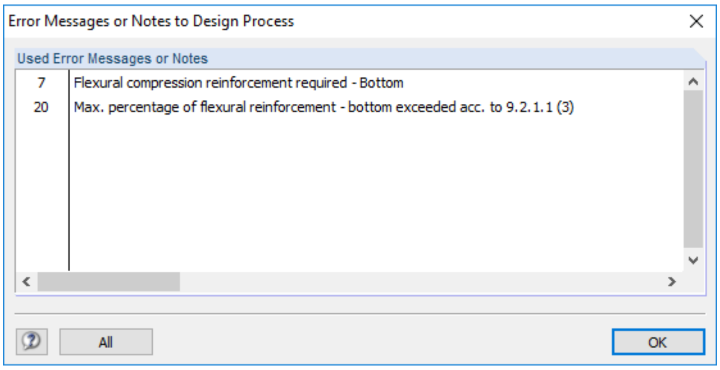


Figure 5.9 *Error Messages or Notes to Design Process dialog box*

All

Click the [All] button in the dialog box to get a list of all messages available in RF-CONCRETE Members.

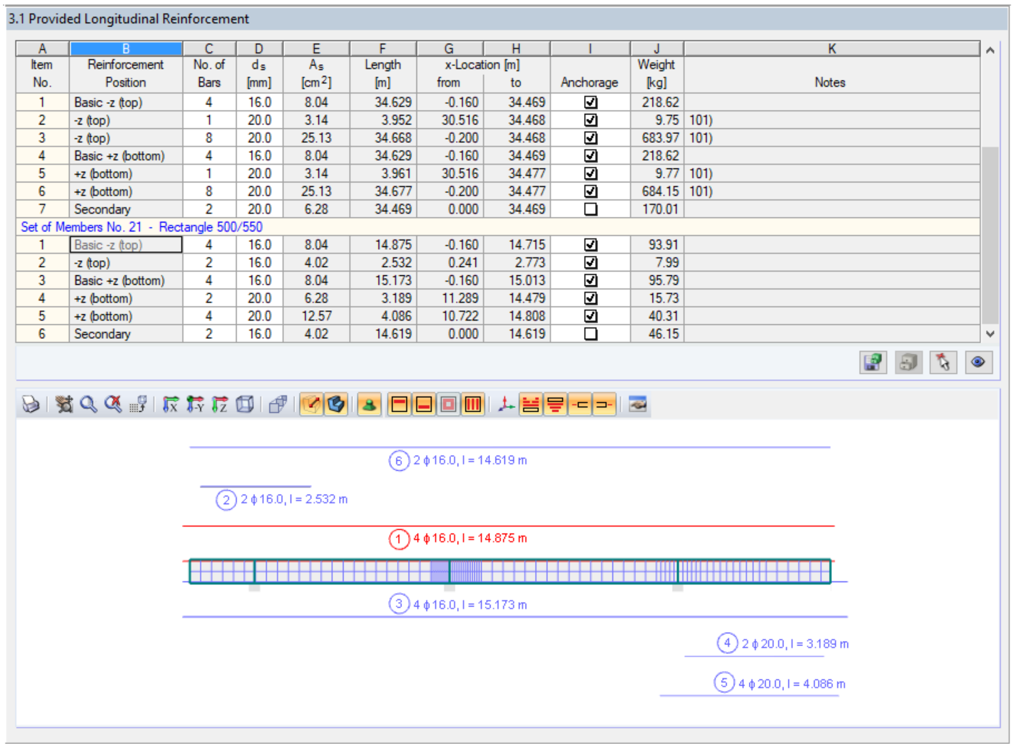
5.2 Provided Reinforcement

The result windows 3.1 to 3.4 are displayed if the *Design the provided reinforcement* option has been activated in window *1.6 Reinforcement* and if no non-designable situations have been found (see Chapter 5.1.5). Serviceability limit state designs and nonlinear calculations also require a reinforcement proposal, that is, a provided reinforcement.

By using the specifications of Window 1.6, RF-CONCRETE Members determines a proposal for the layout of the longitudinal and shear reinforcement. The program tries to cover the required reinforcement, taking into account the parameters (specified rebar diameters, possible number of reinforcement layers, curtailment, type of anchorage) with the least possible amount of rebars and reinforcement areas.

The provided reinforcement can be edited in the *Provided Reinforcement* windows: Diameter, number, position, and length of the reinforcement items can be adjusted individually.

5.2.1 Provided Longitudinal Reinforcement



3.1 Provided Longitudinal Reinforcement

A	B	C	D	E	F	G	H	I	J	K
Item No.	Reinforcement Position	No. of Bars	d_s [mm]	A_s [cm²]	Length [m]	x-Location [m] from	to	Anchorage	Weight [kg]	Notes
1	Basic -z (top)	4	16.0	8.04	34.629	-0.160	34.469	☑	218.62	
2	-z (top)	1	20.0	3.14	3.952	30.516	34.468	☑	9.75	101)
3	-z (top)	8	20.0	25.13	34.668	-0.200	34.468	☑	683.97	101)
4	Basic +z (bottom)	4	16.0	8.04	34.629	-0.160	34.469	☑	218.62	
5	+z (bottom)	1	20.0	3.14	3.961	30.516	34.477	☑	9.77	101)
6	+z (bottom)	8	20.0	25.13	34.677	-0.200	34.477	☑	684.15	101)
7	Secondary	2	20.0	6.28	34.469	0.000	34.469	☐	170.01	
Set of Members No. 21 - Rectangle 500/550										
1	Basic -z (top)	4	16.0	8.04	14.875	-0.160	14.715	☑	93.91	
2	-z (top)	2	16.0	4.02	2.532	0.241	2.773	☑	7.99	
3	Basic +z (bottom)	4	16.0	8.04	15.173	-0.160	15.013	☑	95.79	
4	+z (bottom)	2	20.0	6.28	3.189	11.289	14.479	☑	15.73	
5	+z (bottom)	4	20.0	12.57	4.086	10.722	14.808	☑	40.31	
6	Secondary	2	16.0	4.02	14.619	0.000	14.619	☐	46.15	

Figure 5.10 Window 3.1 *Provided Longitudinal Reinforcement*

The longitudinal reinforcement to be inserted is arranged by *Items* (reinforcement sets) and sorted by members and sets of members.

Below the table you can see the reinforcement represented graphically with item members. The current item (row selected in upper part where pointer is placed) is highlighted in red. Modifications to the parameters in the table are immediately displayed in the graphic. The buttons for controlling the reinforcement graphic are described in Chapter 6.1.

The reinforcement proposal also considers structural regulations. According to EN 1992-1-1, 9.2.1.2, for example, a minimum reinforcement on supports assumed to be hinged must be arranged in such a way that it covers at least 25 % of the maximum adjacent moment of span and is available over the 0.25-fold of the length of the final span (values for German National Annex).

Item No.

The results are listed by *Items* which have the same properties (diameter, length).

The items of all members and sets of members are summarized in the *3.4 Steel Schedule* window.

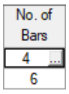
Reinforcement Position

This column indicates the position of the reinforcement in the cross-section:

- basic reinforcement -z (top)
- basic reinforcement +z (bottom)
- -z (top)
- +z (bottom)
- in corners
- all round
- secondary

RF-CONCRETE Members considers the specifications set in the *Reinforcement Layout* tab of Window *1.6 Reinforcement* for the reinforcement's arrangement (see Chapter 3.6.4).

No. of Bars



The number of rebars of an item can be edited: Click into the table cell to open the edit dialog box with the ... button.

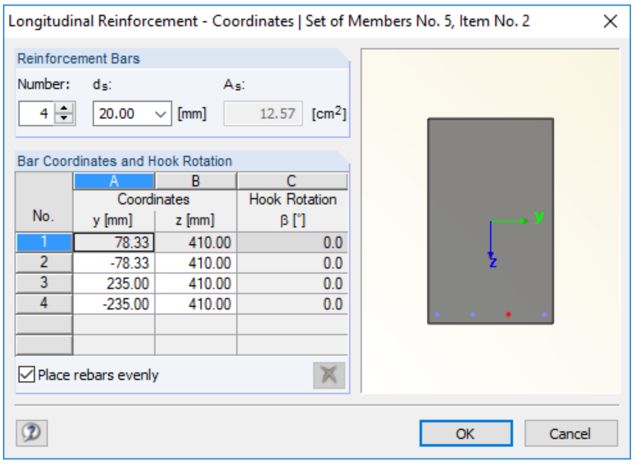


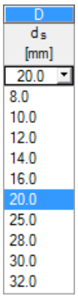
Figure 5.11 *Longitudinal Reinforcement - Coordinates dialog box*

The *Number* of rebars can be changed by using the spin buttons or by directly entering a number. In the section below, you can adjust the position of the members in the individual input rows.

The position of a rebar is defined by means of its *Bar Coordinates*: The coordinates *y* and *z* determine the global distance from the cross-section's centroid. The angle β describes the inclination against the longitudinal member axis for the anchorage types "Hook" and "Bend". For example, a *Hook Rotation* about the angle β = 90° results in a downward rotation (i.e. in direction +z) for the top reinforcement; the angle β = 270° rotates the anchorage end of the bottom reinforcement upwards. For the "Straight" anchorage type, column C is of no importance.

In order to delete the reinforcement of a row, clear the *Place rebars evenly* check box, which enables the [Delete] button.

d_s



The rebar diameters affect the calculation of the inner lever arm of forces as well as the number of rebars per position. Use the list to change the rebar diameter for the current item number.

A_s

Column E lists the respective total reinforcement area resulting from the number of rebars and the diameter.

Length

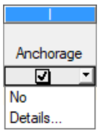
The total length of a representative rebar is shown for each position. The value is composed of the required rebar length and the lengths of anchorage at both member ends. It cannot be edited here.

x-Location from / to

The values represent the mathematical start and end positions of the rebars. They refer to the member's start node given in RFEM (x = 0). When the program determines the dimensions, it takes the support conditions and the anchorage lengths l_1 and l_2 into account.

The values cannot be modified in both columns. This is only possible with the [Edit Reinforcement] button, which is available above the graphic (see Figure 5.10 and Figure 5.13).

Anchorage



The anchorage lengths of the provided reinforcement can be changed using the list. The *Details* option opens the following edit dialog box.

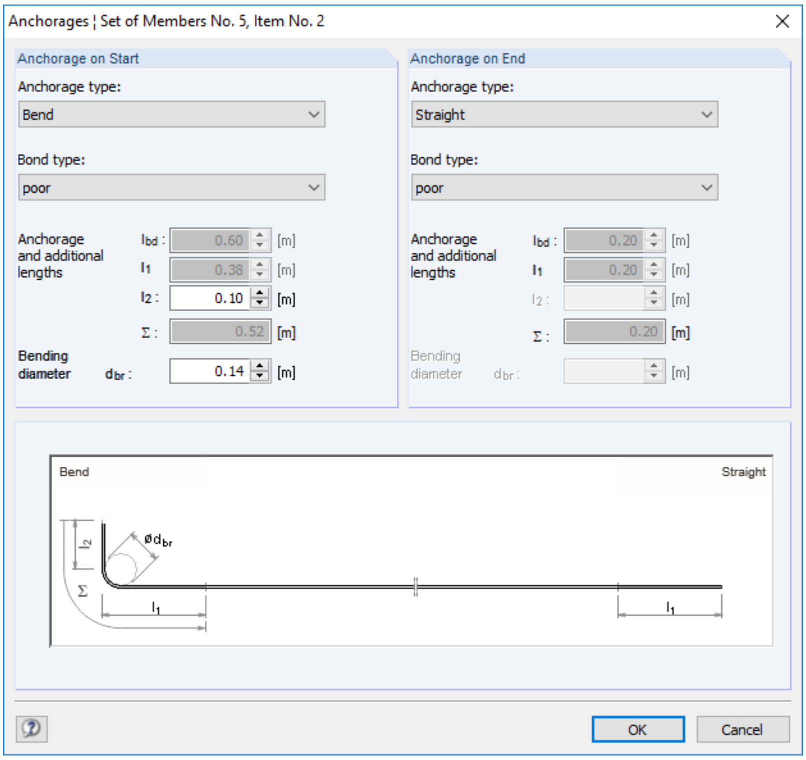
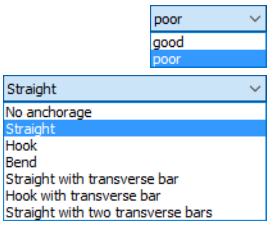


Figure 5.12 *Anchorages* dialog box

The dialog box manages the parameters of the *Anchorage on Start* and *on End* of the rebar.



You can adjust the *Anchorage type* and the *Bond type* via the lists. The anchorage type is described in Chapter 3.6.1. RF-CONCRETE Members automatically recognizes the bond conditions that result from the cross-section geometry and the rebar position. It is also possible to enter user-defined specifications. Figure 8.2 in EN 1992-1-1, 8.4.2 describes good and poor bond conditions.

The *Anchorage length* l_1 is determined with equation (8.4) according to EN 1992-1-1, 8.4.4 (1) considering Table 8.2. It cannot be modified.

The *Anchorage length* l_2 for hooks and bends should be at least 5 d_S according to EN 1992-1-1, 8.4.1 (2).

The required *Bending diameter* d_{br} is specified according to EN 1992-1-1, Table 8.1.

The entire anchorage length Σ for each rebar end results from the individual portions.

Weight

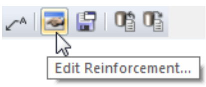
This column shows the total weight of the contained rebars for each position.

Notes

If a footer is indicated in the final column, the reason is a special condition. The numbers are explained in the status bar.

To access all [Messages] for the provided reinforcement, use the corresponding button. The *Error Messages or Notes to Design Process* dialog box appears (see Figure 5.4).

Editing the reinforcement proposal



In the graphic zone of the window, the longitudinal reinforcement is represented by item members. The current reinforcement item (row in upper table where pointer is placed) is highlighted in red. Clicking the [Edit Reinforcement] button above the graphic opens the edit dialog box for the selected item.

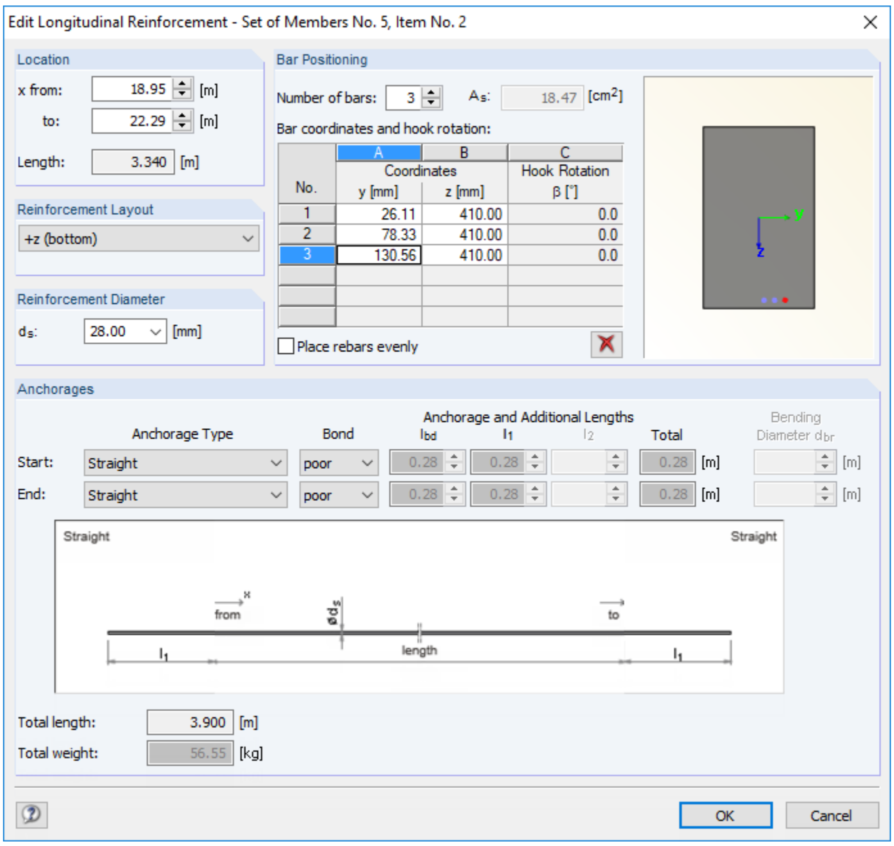


Figure 5.13 *Edit Longitudinal Reinforcement* dialog box

This dialog box summarizes the reinforcement parameters already described above. Use the dialog box to control or, if necessary, adjust the specifications for *Location*, *Bar Positioning*, *Reinforcement Diameter*, and *Anchorages*.

After modifying parameters, the designs must be calculated again with the new reinforcement. So, when leaving Window 3.1, a query appears that asks if you want to recalculate the safety.

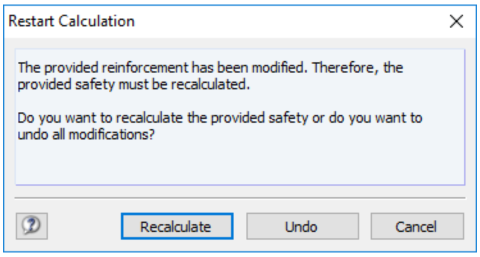
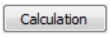


Figure 5.14 Query when leaving Window 3.1



An exception exists for the results of nonlinear analyses: They are generally deleted so that a manual [Calculation] is required.

A modified reinforcement proposal can be stored as a template with the [Save] button. In the following dialog box, the *Name of the Reinforcement Template* must be entered.

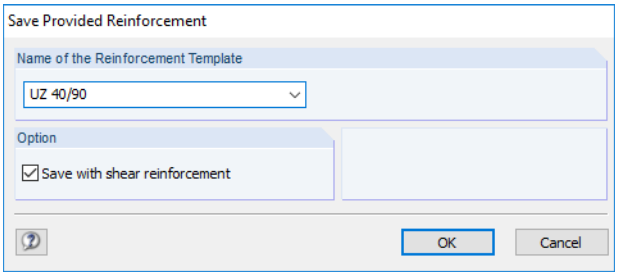
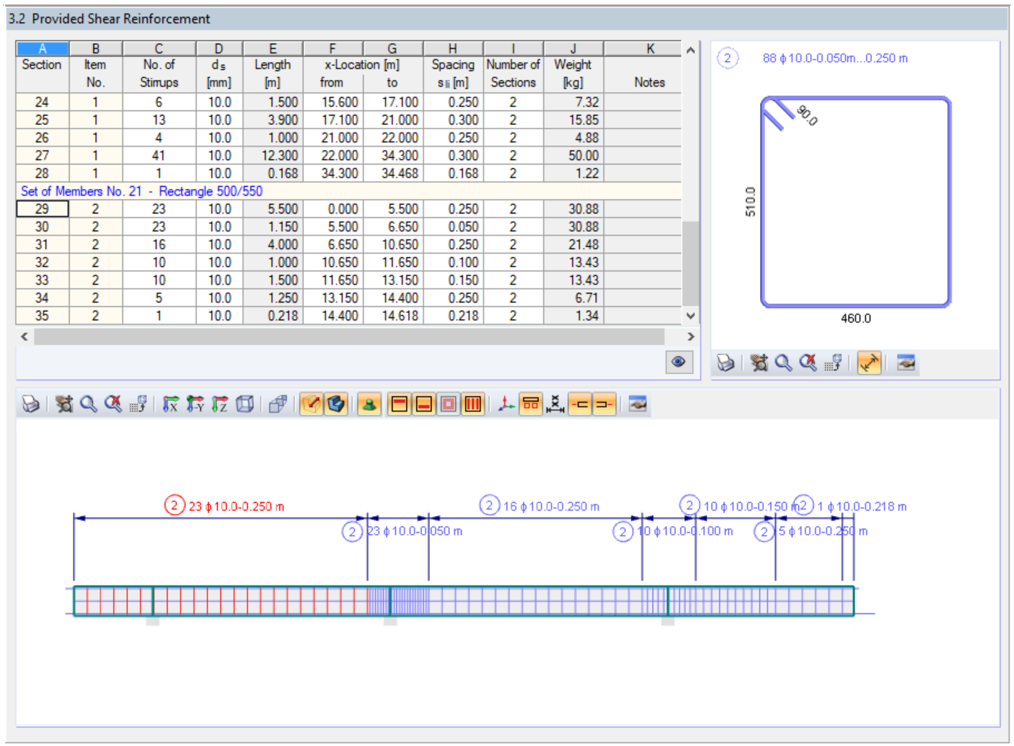


Figure 5.15 *Save Provided Reinforcement* dialog box

Using these templates, you can reset the user-defined reinforcement when the design specifications have been changed in Window 1.6. The changes are not lost if RF-CONCRETE Members creates a new provided reinforcement.

The possibility to import reinforcement templates is described in Chapter 3.6 (see Figure 3.25).

5.2.2 Provided Shear Reinforcement


3.2 Provided Shear Reinforcement

A Section	B Item No.	C No. of Stirrups	D d_s [mm]	E Length [m]	F x-Location [m] from	G x-Location [m] to	H Spacing s_{li} [m]	I Number of Sections	J Weight [kg]	K Notes
24	1	6	10.0	1.500	15.600	17.100	0.250	2	7.32	
25	1	13	10.0	3.900	17.100	21.000	0.300	2	15.85	
26	1	4	10.0	1.000	21.000	22.000	0.250	2	4.88	
27	1	41	10.0	12.300	22.000	34.300	0.300	2	50.00	
28	1	1	10.0	0.168	34.300	34.468	0.168	2	1.22	
Set of Members No. 21 - Rectangle 500/550										
29	2	23	10.0	5.500	0.000	5.500	0.250	2	30.88	
30	2	23	10.0	1.150	5.500	6.650	0.050	2	30.88	
31	2	16	10.0	4.000	6.650	10.650	0.250	2	21.48	
32	2	10	10.0	1.000	10.650	11.650	0.100	2	13.43	
33	2	10	10.0	1.500	11.650	13.150	0.150	2	13.43	
34	2	5	10.0	1.250	13.150	14.400	0.250	2	6.71	
35	2	1	10.0	0.218	14.400	14.618	0.218	2	1.34	

Figure 5.16 Window 3.2 *Provided Shear Reinforcement*

The shear reinforcement to be inserted is arranged by *Items* (reinforcement sets) and sorted by members and sets of members.

The reinforcement is represented graphically below the table. The current item (row selected in upper part where pointer is placed) is highlighted in red. The graphic to the right shows the position stirrup, including dimensioning. Modifications to the parameters in the table will be updated in the graphic.

The buttons for controlling the reinforcement graphic are described in Chapter 6.1.

The reinforcement proposal also considers structural regulations. EN 1992-1-1, 9.2.2 (6), for example, recommends a maximum spacing of $s_{l,max}$ = 0.75 d for vertical links. Table NA 9.1 of the National Annex for Germany also allows for reduced longitudinal spacings depending on the shear force ratio and the concrete strength.

Section

The sections indicated in this table divide the shear reinforcement into zones with the same diameters and spacings (see also Chapter 3.6.2). They are defined in columns F and G where they can be adjusted, if necessary (see below).

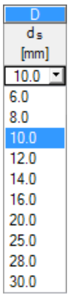
Item No.

The results are listed by *Items*, each having the same properties (diameter, spacing).

The items of all members and sets of members are summarized in Window 3.4 *Steel Schedule*.

No. of Stirrups

RF-CONCRETE Members considers the user specifications of the *Stirrups* tab in Window *1.6 Reinforcement* (see Chapter 3.6.2) when determining the shear reinforcement.



The link number can be modified quickly: After clicking into a cell, you can enter another value. The link spacing in column H will be converted automatically.

d_s

The reinforcement proposal uses the specifications of the *Stirrups* tab in Window *1.6 Reinforcement*. Use the list to change the rebar diameter in the current section.

Length

Column E shows the length of the link zone for each section. It is determined by the start and end locations x and cannot be edited in this column. This is only possible with the two subsequent table columns or the [Edit Reinforcement] button (see Figure 5.16 and Figure 5.17).

x-Location from / to

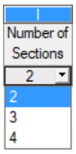
The values represent the start and end positions of the reinforcement section. They refer to the member's start node given in RFEM (x = 0). The entries in both columns can be edited so that the zone limits can be shifted by modifying the values.

You can subdivide a zone as follows: In column F or G, enter a new location *x* which is between both column values. Thus, a new link zone is automatically created.

Spacing s_{li}

RF-CONCRETE Members considers the specifications of the *Stirrups* tab in Window *1.6 Reinforcement* (see Chapter 3.6.2) when determining the link spacings. The values in this column can be edited. When changing a spacing manually, the number of links in column C are automatically adjusted. The exact link spacing, however, is then calculated on the basis of an integer amount of links.

Number of Sections



The links' sections are based on the specifications of the *Stirrups* tab in Window *1.6 Reinforcement* (see Chapter 3.6.2). The number of sections can be changed by using the list.

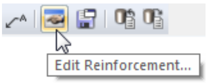
Weight

This column shows the total weight of stirrups for each link section.

Notes

If a footer is indicated in the final column, a special condition is the reason. The numbers are explained in the status bar.

To access all [Messages] for the provided reinforcement, use the corresponding button. The *Error Messages or Notes to Design Process* dialog box appears (see Figure 5.4).



Editing the reinforcement proposal

The graphic in the lower part of this window represents the shear reinforcement on the member or set of members. The current reinforcement section (i.e. the row in the upper table where the pointer is placed) is highlighted in red. Clicking the [Edit Reinforcement] button above the graphic opens the dialog box for editing the selected link section.

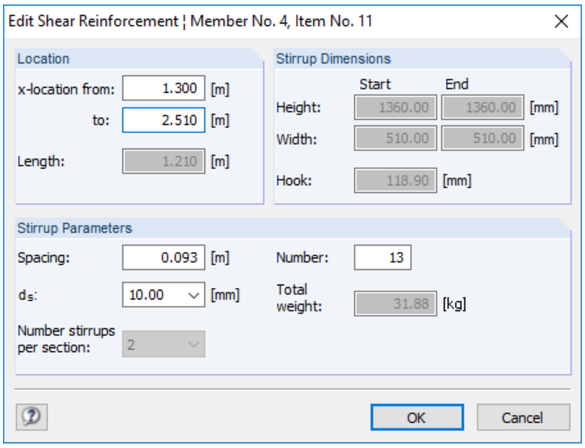


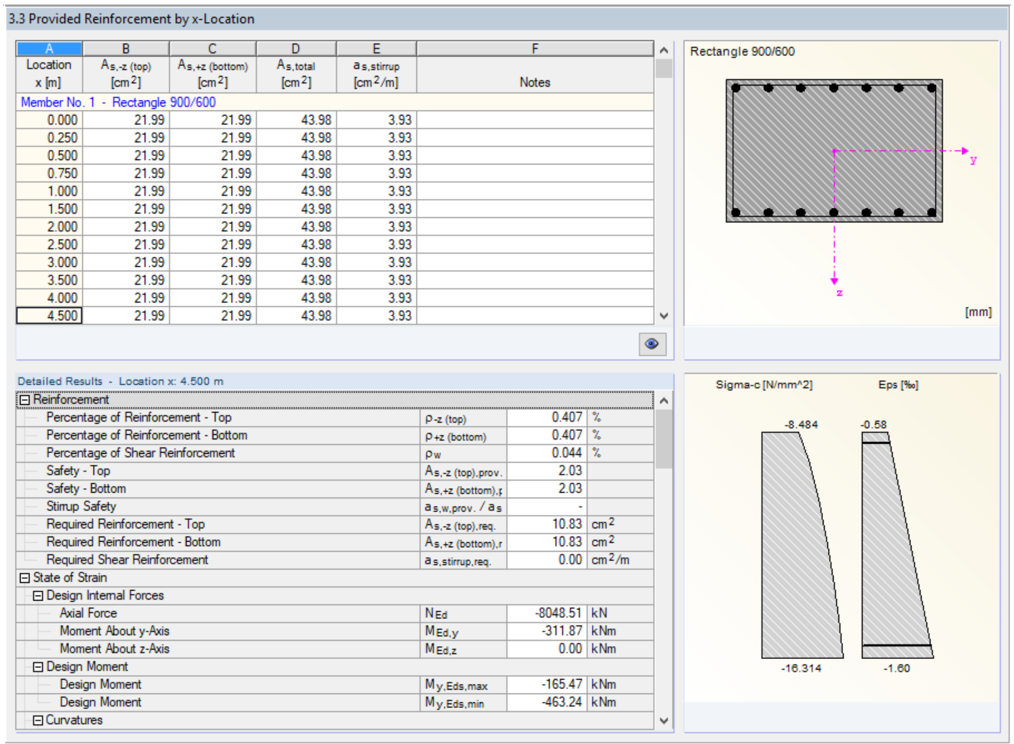
Figure 5.17 *Edit Shear Reinforcement* dialog box

This dialog box summarizes the reinforcement parameters already described above. Use the dialog box to control or, if necessary, adjust the specifications for *Location*, *Stirrup Dimensions*, and *Stirrup Parameters*.

After modifying parameters, the designs must be calculated again with the new reinforcement. So, when leaving Window 3.2, a query appears that asks if you want to recalculate the safety (see Figure 5.14).

5.2.3 Provided Reinforcement by x-Location

This window provides a list with reinforcement areas sorted by x-location including notes about design criteria. In case of a recalculation after modifying the reinforcement in windows 3.1 and 3.2, the results will be updated.



3.3 Provided Reinforcement by x-Location

A	B	C	D	E	F
Location x [m]	$A_{s,-z\,(top)}$ [cm²]	$A_{s,+z\,(bottom)}$ [cm²]	$A_{s,total}$ [cm²]	$a_{s,stirrup}$ [cm²/m]	Notes
Member No. 1 - Rectangle 900/600					
0.000	21.99	21.99	43.98	3.93	
0.250	21.99	21.99	43.98	3.93	
0.500	21.99	21.99	43.98	3.93	
0.750	21.99	21.99	43.98	3.93	
1.000	21.99	21.99	43.98	3.93	
1.500	21.99	21.99	43.98	3.93	
2.000	21.99	21.99	43.98	3.93	
2.500	21.99	21.99	43.98	3.93	
3.000	21.99	21.99	43.98	3.93	
3.500	21.99	21.99	43.98	3.93	
4.000	21.99	21.99	43.98	3.93	
4.500	21.99	21.99	43.98	3.93	

Detailed Results - Location x: 4.500 m

Reinforcement			
Percentage of Reinforcement - Top	$\rho_{-z\,(top)}$	0.407	%
Percentage of Reinforcement - Bottom	$\rho_{+z\,(bottom)}$	0.407	%
Percentage of Shear Reinforcement	ρ_w	0.044	%
Safety - Top	$A_{s,-z\,(top),prov.}$	2.03	
Safety - Bottom	$A_{s,+z\,(bottom),p}$	2.03	
Stirrup Safety	$a_{s,w,prov.} / a_s$	-	
Required Reinforcement - Top	$A_{s,-z\,(top),req.}$	10.83	cm²
Required Reinforcement - Bottom	$A_{s,+z\,(bottom),r}$	10.83	cm²
Required Shear Reinforcement	$a_{s,stirrup,req.}$	0.00	cm²/m
State of Strain			
Design Internal Forces			
Axial Force	N_{Ed}	-8048.51	kN
Moment About y-Axis	$M_{Ed,y}$	-311.87	kNm
Moment About z-Axis	$M_{Ed,z}$	0.00	kNm
Design Moment			
Design Moment	$M_{y,Eds,max}$	-165.47	kNm
Design Moment	$M_{y,Eds,min}$	-463.24	kNm
Curvatures			

Figure 5.18 Window 3.3 *Provided Reinforcement by x-Location*

The table in the upper part of this window shows the longitudinal and shear reinforcement areas at the individual member locations. The lower part of the window lists all *Detailed Results* for the table row selected above.

Location x

The provided reinforcement areas are sorted by x-location for each member:

- Start and end node
- Division points according to member division, if specified (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Extreme values of internal forces

In case of reinforcements by curtailment, two identical x-locations with both reinforcement values appear on the zone limit.

$A_{s,-z}$ (top)

This value represents the reinforcement area of the provided top longitudinal reinforcement.

$A_{s,+z}$ (bottom)

This value represents the reinforcement area of the provided bottom longitudinal reinforcement.

$A_{s,total}$

The provided total longitudinal reinforcement is determined from the values of columns B and C.

$a_{s,stirrup}$

This column shows the area of the provided shear reinforcement.

Notes

The meaning of the footers is explained in the status bar.

The *Detailed Results* allow you to check the designs specifically. The design details refer to the *x*-location selected in the table above. They are updated automatically when clicking into another table row.

The detailed results also provide information about the percentages of reinforcement and the safeties of the selected reinforcement, that is, the ratio of provided to required reinforcement. The safety of the longitudinal reinforcement is designed with an increased moment, taking the global offset into account.

5.2.4 Steel Schedule

Window 3.4 lists the scheduled rebars by items. The table cannot be edited.

3.4 Steel Schedule

A	B	C	D	E	F	G	H	I	J
Item No.	Reinforcement Type	d_s [mm]	Surface	No. of Bars	Length [m]	Anchorage Type Start	End	Bending Diameter [m]	Weight [kg]
Material No. 1 - Reinforcing Steel B 500 S (B)									
1	Longitudinal	12.0	Ribbed	6	11.000	No anchorage	No anchorage		58.60
2	Longitudinal	20.0	Ribbed	9	6.239	Straight	Straight		138.47
3	Longitudinal	20.0	Ribbed	4	11.400	Straight	Straight		112.46
4	Longitudinal	20.0	Ribbed	2	11.439	Straight	Straight		56.42
5	Longitudinal	20.0	Ribbed	12	5.400	Straight	Straight		159.81
6	Stirrup	10.0	Ribbed	38	1.738	Hook	Hook	0.040	40.71
7	Stirrup	10.0	Ribbed	66	1.055	Hook	Hook	0.040	42.91
Total				137					609.38

Figure 5.19 Window 3.4 *Steel Schedule*

Item No.

The rebars are listed by *Items* that have the same properties (diameter, length, type of anchorage, etc.).

The item numbers are not identical to the numbers of windows 3.1 and 3.2.

Reinforcement Type

This column indicates if the reinforcement is a *Longitudinal* or a $A_{s,Stirrup}$ reinforcement.

d_s

Column C informs about the used rebar diameters.

Surface

Column D shows the surface type of the reinforcing steel, which can be *Ribbed* or *Plain*.

No. of Bars

Here, you can see the number of similar rebars available in each item.

Length

The total length of a representative rebar is shown for each position.

Anchorage Type Start / End

Both columns inform about the types of anchorage at the start and end of the rebars (*No anchorage*, *Straight*, *Hook*, etc.).

Bending Diameter

The bending diameter d_{br} is indicated for stirrups and hooks.

Weight

This column shows the total weight of the contained rebars for each position.

Total

At the end of the steel schedule, the total number of rebars as well as the weight of the total required reinforcing steel is shown. The sums are determined from the values of the items above.

5.3 Serviceability Limit State Design

The result windows 4.1 to 4.4 are displayed if the design for the *Serviceability Limit State* has been set in Window 1.1 (see Chapter 3.1.2).

The serviceability limit state designs are performed with the reinforcement layout available as *Provided Reinforcement* in windows 3.1 and 3.2.

5.3.1 Serviceability Check by Cross-Section

The upper part of the window presents a summary of the governing serviceability limit state designs. The lower part shows the *Detailed Results* of the current member (row selected in table above) with all design-relevant parameters. You can expand or reduce the chapters by clicking [+] and [-].

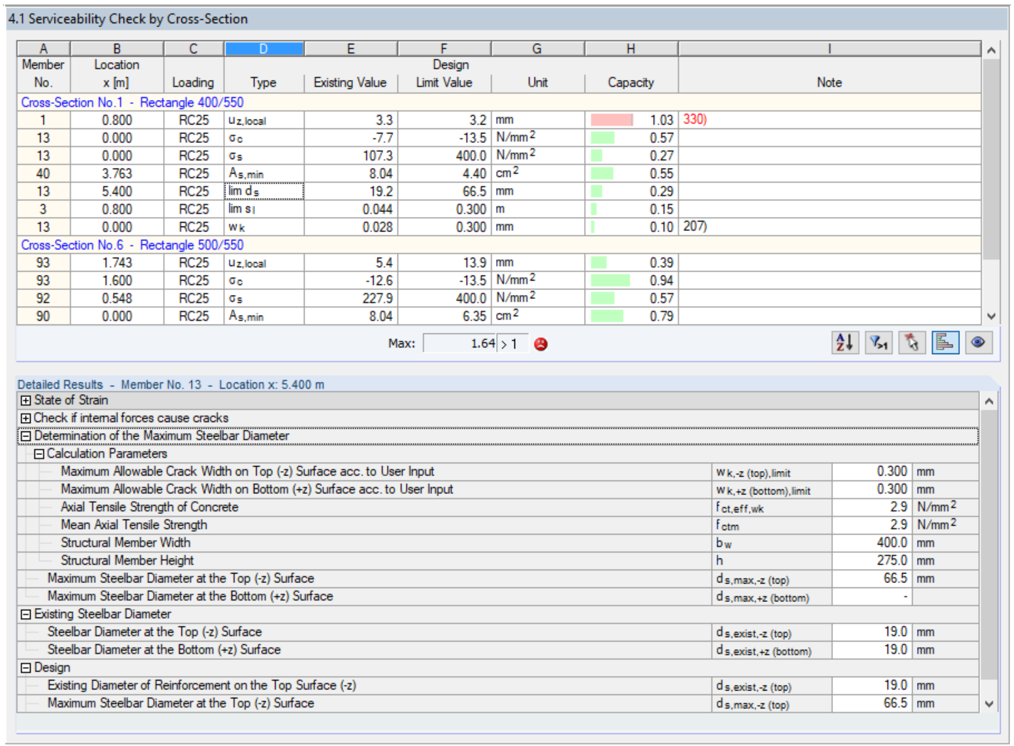


Figure 5.20 Window 4.1 *Serviceability Check by Cross-Section*

The designs are sorted by cross-sections. The table shows the most unfavorable values of the criteria to be considered for the serviceability limit state design. They are based on the reinforcement groups' parameters for crack width control (see Chapter 3.6.7), the provided reinforcement, and the internal forces for the serviceability limit state.

Generally, only **one** of the criteria *lim* d_s, *lim* s_l, or w_k must be fulfilled for the crack width design. Clause 7.2 of EN 1992-1-1 describes the conditions under which the stresses shall be limited. This means that not all design ratios shown in Window 4.1 have to be less than 1 in order to fulfill the serviceability limit state design!

Member No.

For each cross-section, the column shows the numbers of the members whose result values (column E) or design ratios (column H) are greatest. Use the [Sort] button to control the reference.

Location x

The column shows the x-location on the member where the most unfavorable values or design ratios occur. The distances refer to the start nodes of the members.

Loading

The column lists the numbers of the governing load cases, load and result combinations.

Type

Type
$u_{z,local}$
σ_c
σ_s
$A_{s,min}$
lim d_s
lim s_l
w_k

Column D indicates the criteria to be considered for the serviceability limit state design. Depending on the settings in the *Serviceability* tab of Window *1.6 Reinforcement*, up to seven design types are displayed.

$u_{z,local}$

This is the absolute value of the deflection available in the direction of the local member axis z.

The allowable relative deformations are managed in Window *1.7 Deflection Data* (see Chapter 3.7).

σ_c

This parameter describes the concrete stresses in the serviceability limit state.

σ_s

The stresses in the reinforcement for a cracked tensile zone are determined as the product of steel strain and modulus of elasticity:

$$\sigma_s = \varepsilon_s \cdot E_s$$

$A_{s,min}$

The minimum area of the rebar reinforcement according to EN 1992-1-1, 7.3.2, Eq. (7.1) is:

$$A_{s,\min} = k_c \cdot k \cdot f_{ct,\text{eff}} \cdot \frac{A_{ct}}{\sigma_s}$$

where

k_c : factor for considering stress distribution in cross-section before first crack formation

k : factor for considering self-equilibrating stresses distributed nonlinearly over cross-section

$f_{ct,\ eff}$: mean value of effective concrete tensile strength when cracks occur

A_{ct} : tension zone of concrete in uncracked state in case of first crack formation

σ_s : allowable steel stress directly after crack formation (depending on limit diameter or maximum value of rebar spacings, where applicable)

lim d_s

According to the simplified design method described in EN 1992-1-1, 7.3.3, the limit diameters of the reinforcing steel are limited to the dimensions stated in Table 7.2 (see Figure 2.3).

lim s_l

According to the simplified design method described in EN 1992-1-1, 7.3.3, the allowable bar spacings are limited to the maximum values stated in Table 7.3 (see Figure 2.4).

w_k

The characteristic crack width is determined according to EN 1992-1-1, 7.3.4, Eq. (7.8) as follows:

$$w_k = s_{r,max} \cdot (\varepsilon_{sm} - \varepsilon_{cm})$$

where

$s_{r,max}$: maximum crack spacing in final crack state (see Chapter 2.2.4)

ε_{sm} : mean strain of reinforcement considering contribution of concrete to tension between cracks

ε_{cm} : mean strain of concrete between cracks

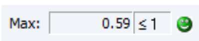
Existing Value

This column displays the values that are governing for each cross-section for the serviceability limit state designs.

Design Limit Value

The limit values are determined from the standard specifications and the load situation. The determination of limit values is described in Chapter 2.2.

Capacity



Column H shows the ratio of existing value (column E) to limit value (column F). Ratios greater than 1 mean that the design criterion is not fulfilled. The length of the colored bar represents the design ratio graphically.

Note

The final column indicates design problems or shows notes referring to difficulties that occurred during the analysis. The numbers are explained in the status bar.

To see all [Messages], use the button shown on the left. The *Error Messages or Notes to Design Process* dialog box appears.

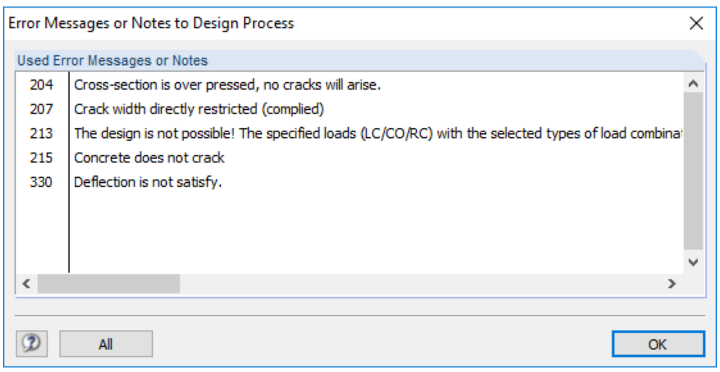


Figure 5.21 *Error Messages or Notes to Design Process dialog box*

The buttons below the table have the following functions:

Button	Description	Function
	Sort results	Sorts results by maximum ratios (column H) or maximum values (column E)
	Exceeding	Displays only rows with a ratio greater than 1
	Select member	Allows for graphical selection of member to display its results in the table
	Color bars	Displays or hides the colored relation scales
	View mode	Jumps to RFEM work window to change the view

Table 5.2 Buttons in result windows 4.1 to 4.4

5.3.2 Serviceability Check by Set of Members

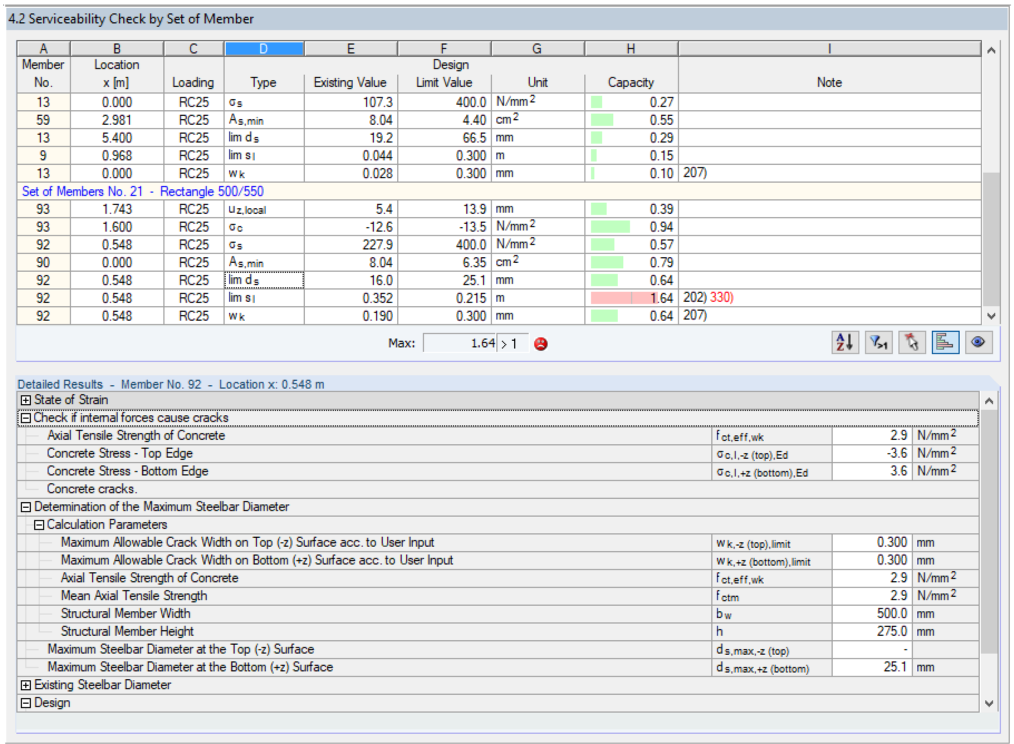


Figure 5.22 Window 4.2 *Serviceability Check by Set of Members*

This window lists the governing serviceability limit state designs for each of the designed sets of members. The columns are described in detail in Chapter 5.3.1.

5.3.3 Serviceability Check by Member

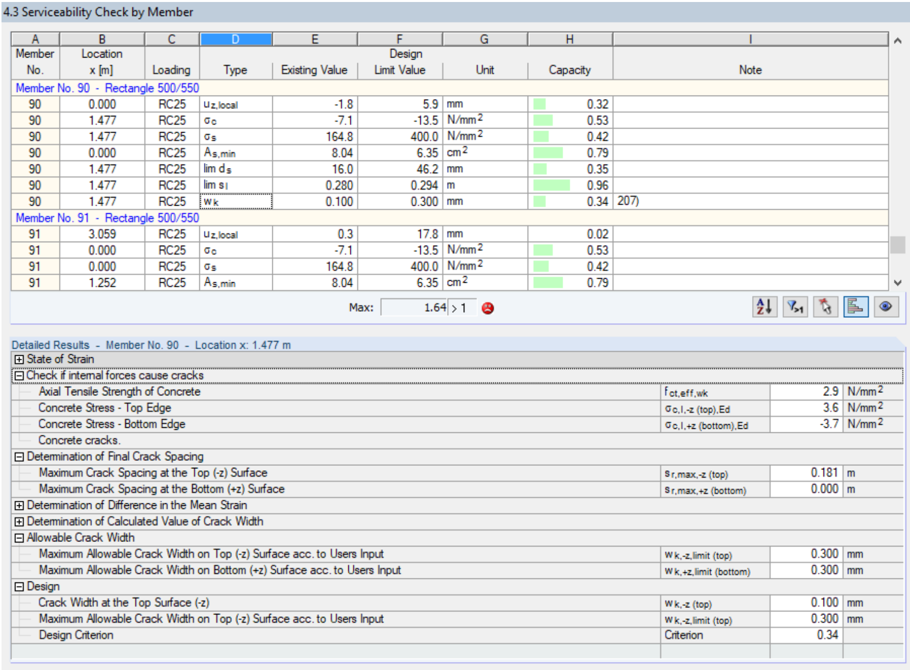


Figure 5.23 Window *4.3 Serviceability Check by Member*

In this window, the results of the SLS designs (see Chapter 5.3.1) are sorted by members.

5.3.4 Serviceability Check by x-Location

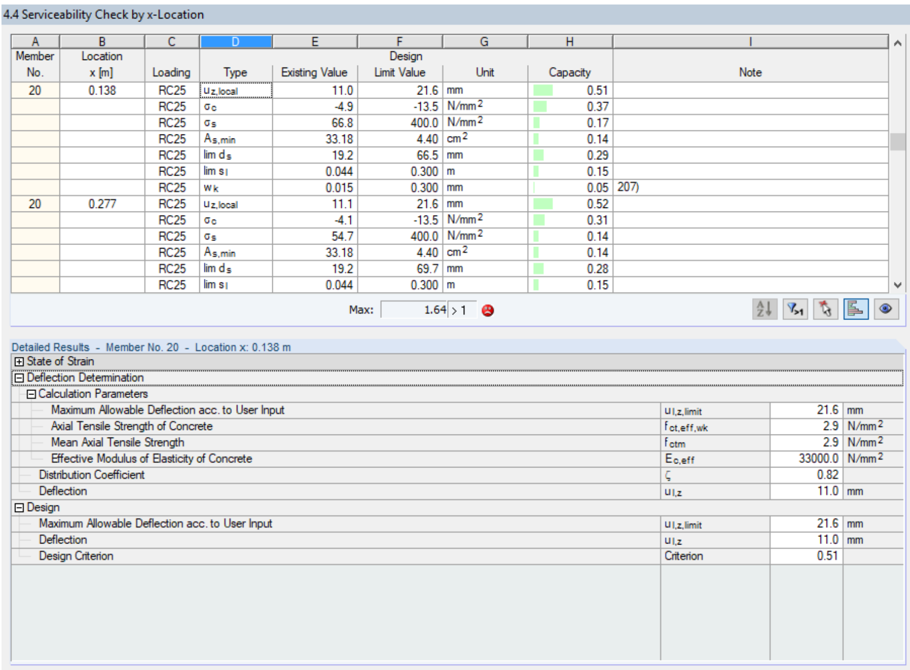


Figure 5.24 Window *4.4 Serviceability Check by x-Location*

This window lists the individual designs (see Chapter 5.3.1) by x-locations.

5.4 Fire Resistance Design

The result windows 5.1 to 5.4 are displayed if the design of the *Fire Resistance* was enabled in Window 1.1 (see Chapter 3.1.4).

The fire resistance design is performed with the reinforcement layout available as *Provided Reinforcement* in windows 3.1 and 3.2.

5.4.1 Fire Resistance Design by Cross-Section

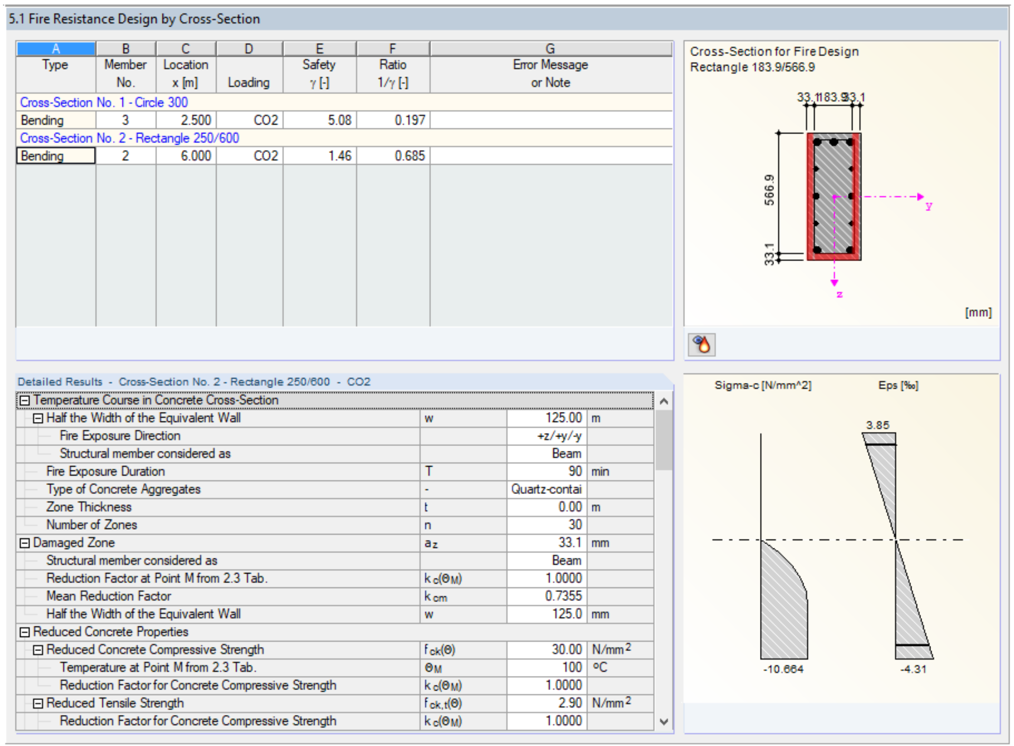


Figure 5.25 Window 5.1 *Fire Resistance Design by Cross-section*

This overview shows the smallest safeties and greatest ratios of the individual cross-sections in the case of fire. They result from the parameters of the reinforcement groups defined for the fire designs (see Chapter 3.1.4), the provided reinforcement, and the internal forces of the relevant actions.

The lower part of the window lists all *Detailed Results* for the table row selected above. These design details allow for a specific evaluation of the results. They are updated when clicking into a different table row.

For information on the theoretical background of the fire protection design, see Chapter 2.3.

Type

The safeties and ratios are listed by cross-sections. In addition to the designs for *Bending*, the table shows the safeties for *Shear Force* and *Torsion*, provided that these designs have been activated in the *Fire Resistance* tab of Window *1.6 Reinforcement* (see Figure 3.48).

Member No.

The column displays the numbers of the members that have the maximum design ratios.

Location x

The column shows the x-location on the member that is governing for the fire design. The following RFEM member locations x are used for the table output:

- Start and end node
- Division points according to member division, if specified (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Extreme values of internal forces

Loading

Column D shows the numbers of the load cases, as well as load and result combinations that are governing for the respective designs.

Safety γ

Column E provides information about the minimum safety factors γ for each type of design. If the safety is less than 1, the fire resistance design is not fulfilled. The results row is highlighted in red.

Capacity 1 / γ

The entries in this column represent the safeties' reciprocal values (column E). This way, you can quickly evaluate, which resources are available in the cross-section until the limit value 1 is reached.

The *Detailed Results* provides information about how the provided safeties are determined from the ratio of ultimate internal force in case of fire to effective internal force.

⊟ Design			
⊟ Factor for utilization of the bending reinforcement	$1/\gamma_{SL,fi}$	0.6847	
⊟ Existing Safety to Bending	$\gamma_{SL,fi}$	1.4606	
⊟ Influencing Internal Forces			
Moment About y-Axis	$M_{y,Ed,fi}$	-111.86	kNm
Moment About z-Axis	$M_{z,Ed,fi}$	0.00	kNm
Axial Force	$N_{Ed,fi}$	1.65	kN
⊟ Ultimate Internal Forces			
Ultimate Moment About y-Axis	$M_{yRd,fi}$	-163.38	kNm
Ultimate Moment About z-Axis	$M_{zRd,fi}$	0.00	kNm
Ultimate Axial force	$N_{Rd,fi}$	2.41	kN

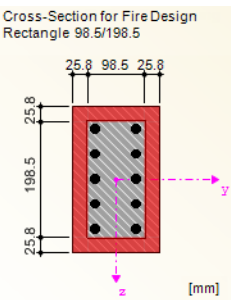
Figure 5.26 *Detailed Results* window section with design details

Error Message or Note

Messages...

The final column indicates non-designable situations or notes referring to design issues. The numbers are explained in the status bar. Use the [Messages] button to see all notes of the current design case.

Temperature course



To the right of the table, the equivalent cross-section used for the fire resistance design is graphically displayed. The damaged zone a_z is highlighted in red (cf. Figure 2.7).

To look at the distribution of temperature in the current cross-section, click the button below the graphic. The following dialog box appears.

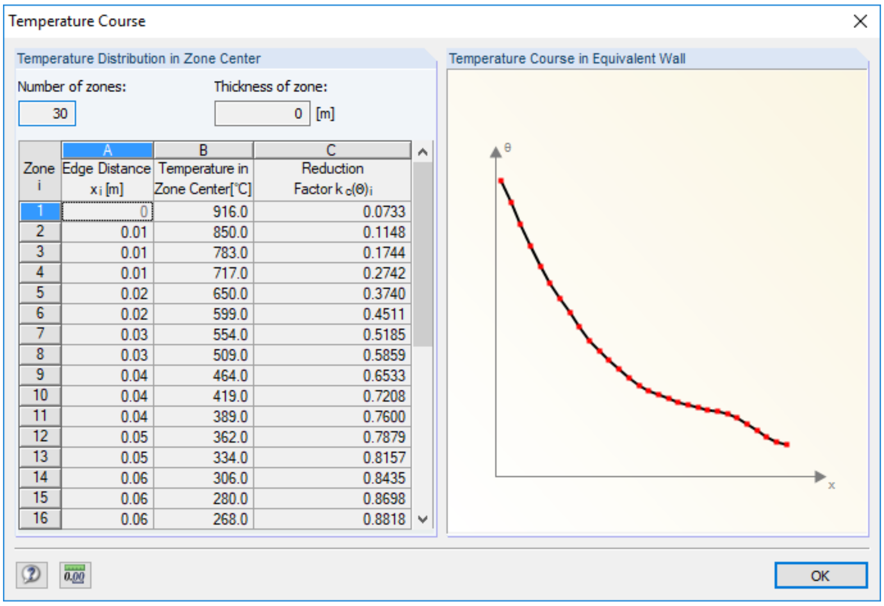


Figure 5.27 *Temperature Course* dialog box

The table describes the *Temperature Distribution in Zone Center* (see Chapter 2.3.2). The graphic shows the *Temperature Course in Equivalent Wall* according to EN 1992-1-2, Annex A.

5.4.2 Fire Resistance Design by Set of Members

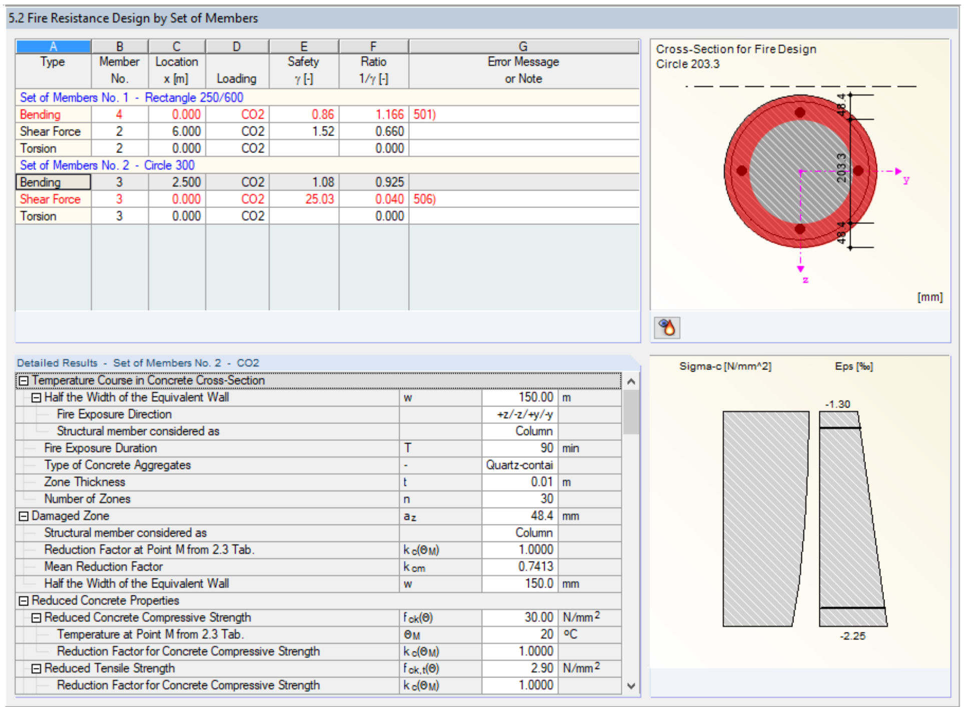


Figure 5.28 Window 5.2 *Fire Resistance Design by Set of Members*

If sets of members have been selected for design, this window shows the fire resistance designs sorted by sets of members. The columns are described in detail in Chapter 5.4.1.

5.4.3 Fire Resistance Design by Member

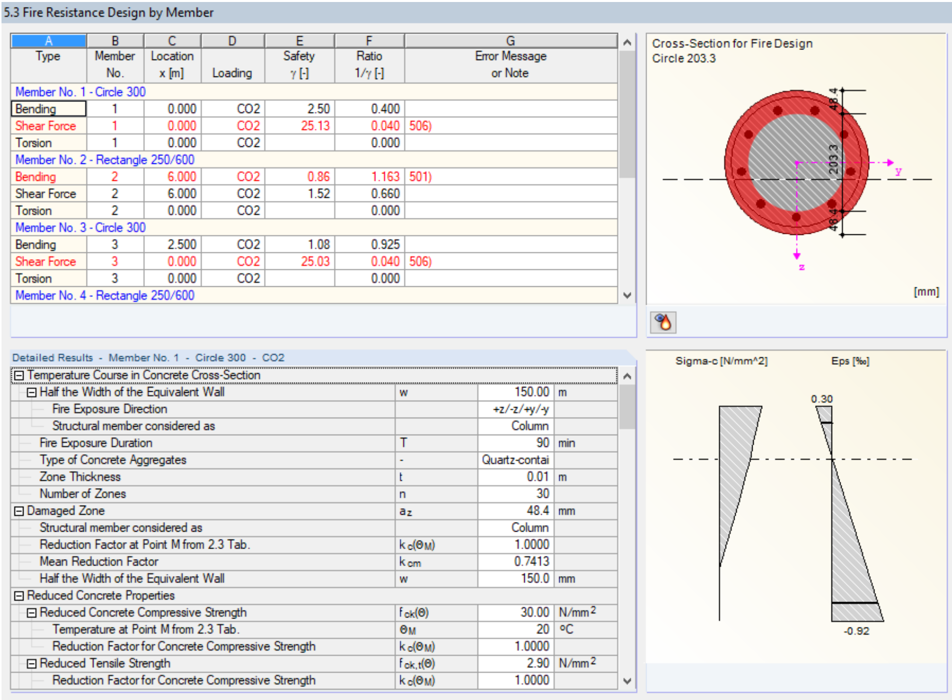


Figure 5.29 Window 5.3 *Fire Resistance Design by Member*

This window shows the results output of the fire protection designs listed by members.

5.4.4 Fire Resistance Design by x-Location

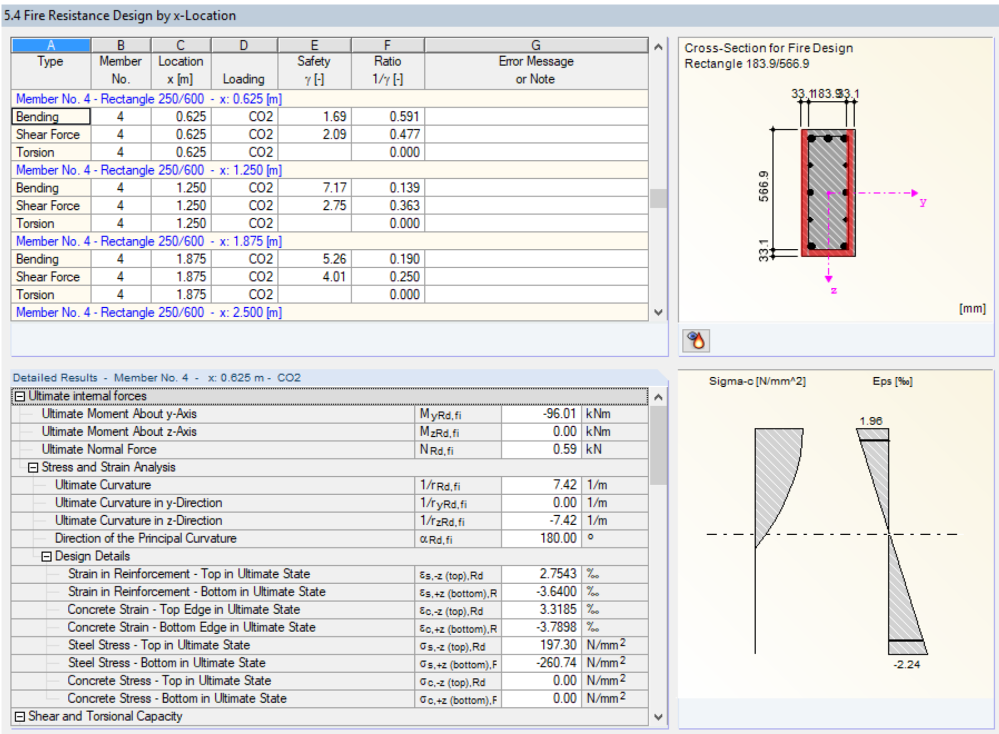


Figure 5.30 Window 5.4 *Fire Resistance Design by x-Location*

This result window displays the fire protection designs sorted by x-locations in detail.

5.4.5 Fire Resistance Design Not Designable

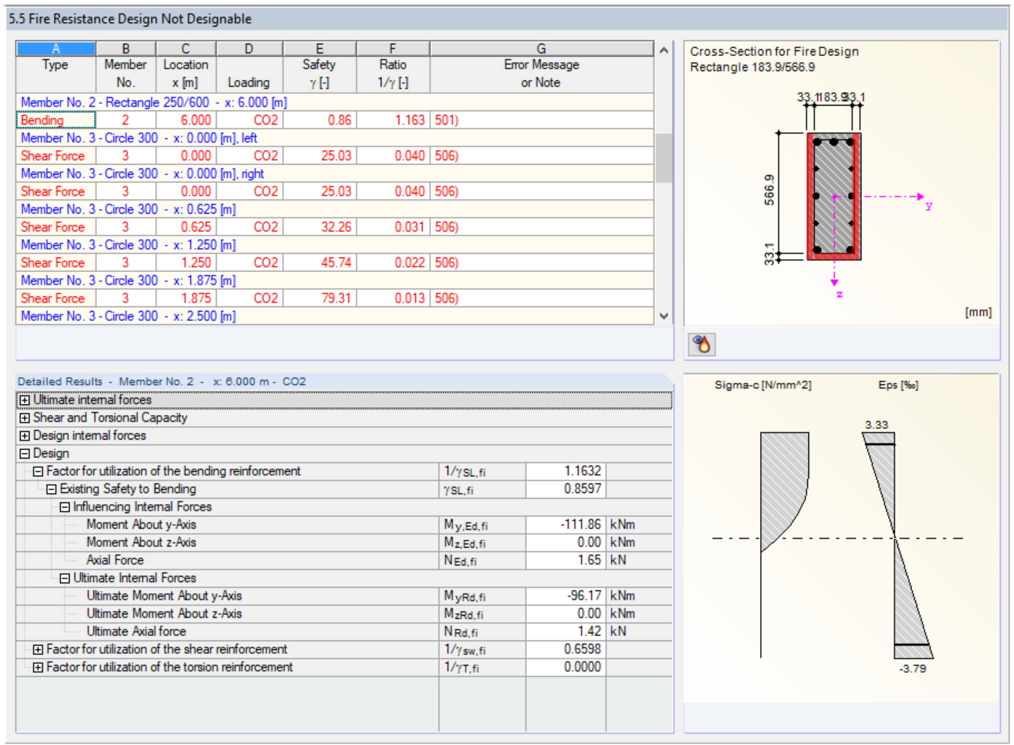


Figure 5.31 Window *5.5 Fire Resistance Design Not Designable*

This window is only displayed if the program has detected failed designs or any other problems in the fire design process. The error messages are sorted by members and x-locations.

The number of the *Error Message* indicated in column G is described by comments in the footer.

Messages...

Use the [Messages] button to display all issues that have occurred in the design process for the fire situation.

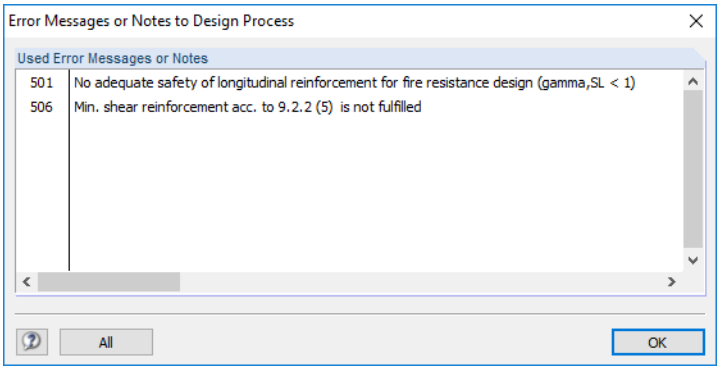
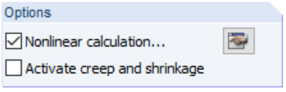


Figure 5.32 *Error Messages or Notes to Design Process* dialog box

5.5 Nonlinear Calculation



The result windows 6.1 to 6.4 are only displayed if the *Nonlinear calculation* option (state II) has been activated for the ultimate limit state, the serviceability limit state, or the fire resistance design in Window *1.1 General Data* (see Chapter 3.1). Furthermore, there should be no design problems (see Chapter 5.1.5).

The designs are performed with the reinforcement available in Window *3.1 Provided Longitudinal Reinforcement*.

The theory of nonlinear designs is described in Chapter 2.4.

The results of the nonlinear design are sorted by ultimate limit state, serviceability limit state, and fire resistance designs. In addition, they are sorted by cross-sections, members, sets of members, and x-locations in individual windows for each of the mentioned categories. The concept of these windows corresponds to one of the "usual" serviceability limit state designs (see Chapter 5.3.1).

5.5.1 Nonlinear Calculation - Ultimate Limit State

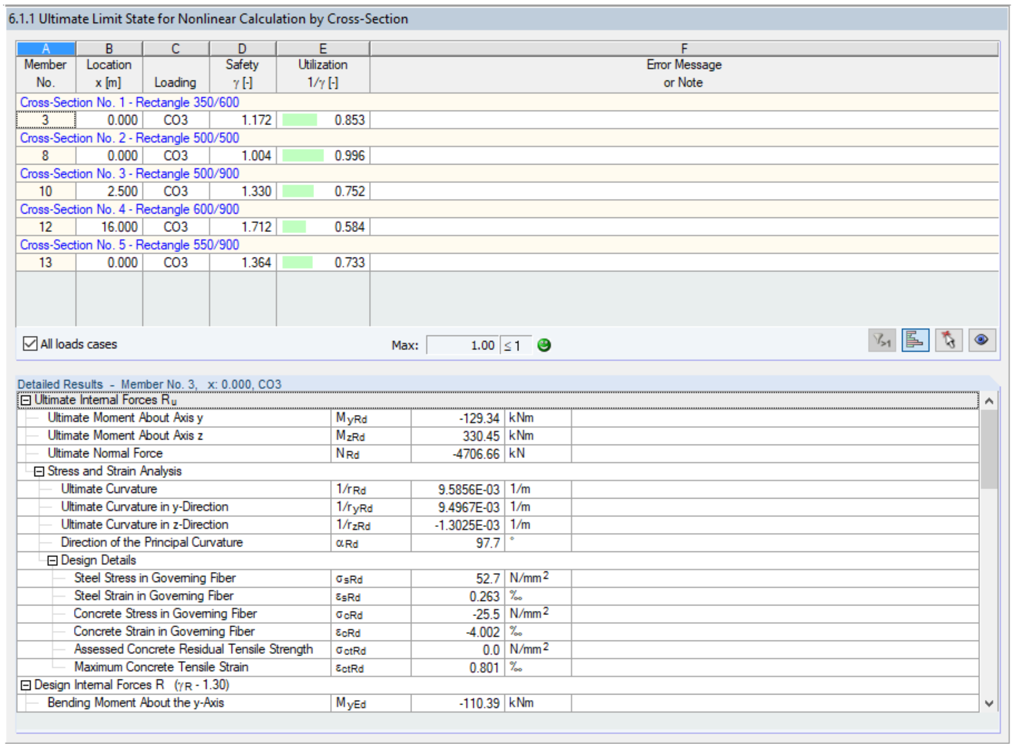
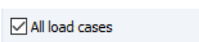


Figure 5.33 Window *6.1.1 Ultimate Limit State for Nonlinear Calculation by Cross-Section*

The table in the upper part contains the governing safety factors and utilizations determined in the nonlinear ultimate limit state design for each cross-section, member, or set of members. They result from the internal forces, the reinforcement parameters, and the specifications set in the *Settings for Nonlinear Calculation* dialog box (see Figure 4.4 and Figure 4.6).

The lower part lists all *Detailed Results* for the table row selected above. The design details are updated when clicking into another row.



With the *All load cases* check box, you can decide whether to only show the results of the governing action or (as shown in Figure 5.33) the results of all analyzed load cases and combinations.

Member No.

The safeties and design ratios are listed by cross-sections.

Location x

The x-locations refer to the FE nodes that have been created on the members for the nonlinear calculation. Use the [Details] button in the *Iteration Parameters* tab of the *Settings for Nonlinear Calculation* dialog box (see Figure 2.30) to access the *FE Mesh* dialog box of RFEM where you can adjust the FE mesh settings.

Loading

This column displays the numbers of the load cases or load combinations that are governing for the individual cross-sections, members, or sets of members.

If several load cases or combinations have been designed, it is possible to show the results of all analyzed actions by selecting the *All load cases* check box.

Safety γ

The column lists the safety factors γ for each location x. They represent the ratio of ultimate and acting internal forces.

$$\gamma = \frac{R_d}{E_d} \geq 1.0$$

where

$$R_d = \begin{vmatrix} N_{Rd} \\ M_{y,Rd} \\ M_{z,Rd} \end{vmatrix} \qquad \text{design value of load-bearing capacity}$$

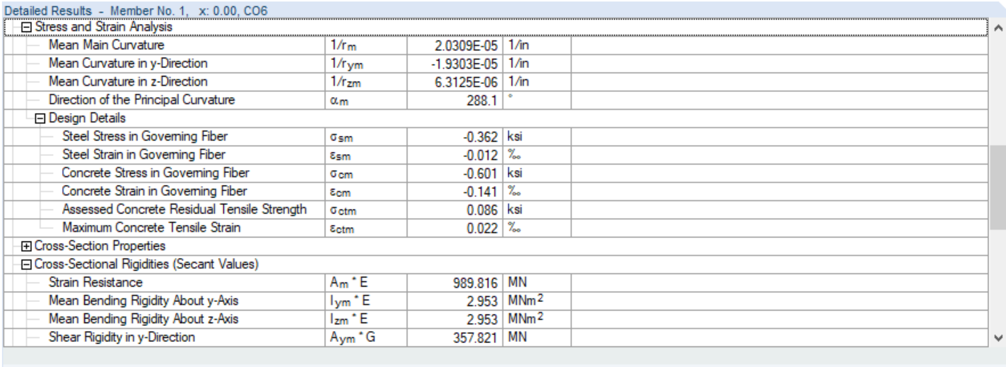
$$E_d = \begin{vmatrix} N_{Ed} \\ M_{y,Ed} \\ M_{z,Ed} \end{vmatrix} \qquad \text{design value of action}$$

If the safety factor at any x-location is less than 1, or if no convergence is reached, the ultimate limit state design is not fulfilled. The entire table is presented in red color.

Capacity 1/γ

The entries in this column represent the safety factors' reciprocal values γ. Thus, you can quickly evaluate the resources in the cross-section.

The *Detailed Results* in the lower part of the window (see Figure 5.34) provide information about all design details. They are organized in a tree structure.



Detailed Results - Member No. 1, x: 0.00, CO6

⊟ Stress and Strain Analysis			
Mean Main Curvature	$1/r_m$	2.0309E-05	1/in
Mean Curvature in y-Direction	$1/r_{ym}$	-1.9303E-05	1/in
Mean Curvature in z-Direction	$1/r_{zm}$	6.3125E-06	1/in
Direction of the Principal Curvature	α_m	288.1	°
⊟ Design Details			
Steel Stress in Governing Fiber	σ_{sm}	-0.362	ksi
Steel Strain in Governing Fiber	ε_{sm}	-0.012	‰
Concrete Stress in Governing Fiber	σ_{cm}	-0.601	ksi
Concrete Strain in Governing Fiber	ε_{cm}	-0.141	‰
Assessed Concrete Residual Tensile Strength	σ_{ctm}	0.086	ksi
Maximum Concrete Tensile Strain	ε_{ctm}	0.022	‰
⊞ Cross-Section Properties			
⊟ Cross-Sectional Rigidities (Secant Values)			
Strain Resistance	$A_m * E$	989.816	MN
Mean Bending Rigidity About y-Axis	$I_{ym} * E$	2.953	MNm^2
Mean Bending Rigidity About z-Axis	$I_{zm} * E$	2.953	MNm^2
Shear Rigidity in y-Direction	$A_{ym} * G$	357.821	MN

Figure 5.34 *Detailed Results* window section with details for stress and strain analysis

The detailed results are adjusted to the selected *Tension Stiffening Approach* (see Figure 4.6).

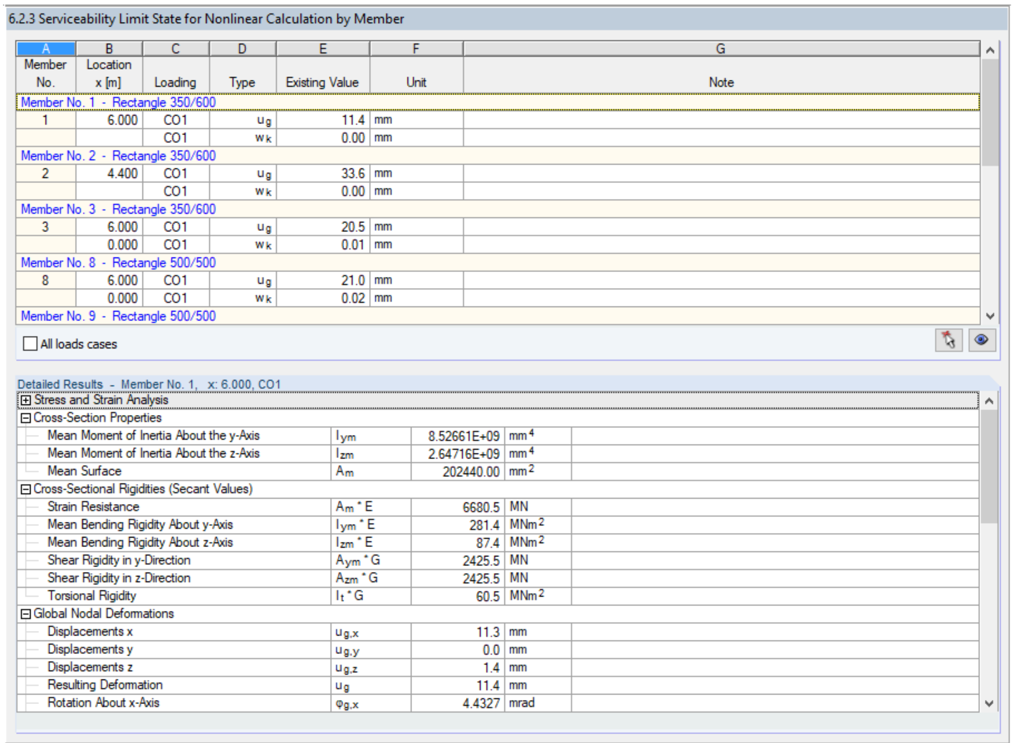
Error Message or Note

The final column refers to failed designs or specific situations occurring in the course of the design (e.g. when the maximum number of iterations was reached in the final load step of the calculation without fulfilling the break-off criterion). The numbers are explained in the status bar.

Messages...

Use the [Messages] button to display all issues that have occurred in the nonlinear calculation.

5.5.2 Nonlinear Calculation - Serviceability



6.2.3 Serviceability Limit State for Nonlinear Calculation by Member

A Member No.	B Location x [m]	C Loading	D Type	E Existing Value	F Unit	G Note
Member No. 1 - Rectangle 350/600						
1	6.000	CO1	u_g	11.4	mm	
		CO1	w_k	0.00	mm	
Member No. 2 - Rectangle 350/600						
2	4.400	CO1	u_g	33.6	mm	
		CO1	w_k	0.00	mm	
Member No. 3 - Rectangle 350/600						
3	6.000	CO1	u_g	20.5	mm	
	0.000	CO1	w_k	0.01	mm	
Member No. 8 - Rectangle 500/500						
8	6.000	CO1	u_g	21.0	mm	
	0.000	CO1	w_k	0.02	mm	
Member No. 9 - Rectangle 500/500						

All loads cases

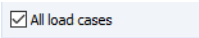
Detailed Results - Member No. 1, x: 6.000, CO1

⊞ Stress and Strain Analysis			
⊟ Cross-Section Properties			
Mean Moment of Inertia About the y-Axis	I_{ym}	8.52661E+09	mm^4
Mean Moment of Inertia About the z-Axis	I_{zm}	2.64716E+09	mm^4
Mean Surface	A_m	202440.00	mm^2
⊟ Cross-Sectional Rigidities (Secant Values)			
Strain Resistance	$A_m * E$	6680.5	MN
Mean Bending Rigidity About y-Axis	$I_{ym} * E$	281.4	MNm^2
Mean Bending Rigidity About z-Axis	$I_{zm} * E$	87.4	MNm^2
Shear Rigidity in y-Direction	$A_{ym} * G$	2425.5	MN
Shear Rigidity in z-Direction	$A_{zm} * G$	2425.5	MN
Torsional Rigidity	$I_t * G$	60.5	MNm^2
⊟ Global Nodal Deformations			
Displacements x	$u_{g,x}$	11.3	mm
Displacements y	$u_{g,y}$	0.0	mm
Displacements z	$u_{g,z}$	1.4	mm
Resulting Deformation	u_g	11.4	mm
Rotation About x-Axis	$\varphi_{g,x}$	4.4327	mrad

Figure 5.35 Window 6.2.3 *Serviceability Limit State for Nonlinear Calculation by Member*

The table in the upper part shows the deformations and crack widths that have been determined as governing in the nonlinear serviceability limit state design.

The lower part lists all *Detailed Results* for the table row selected above.



With the *All load cases* check box, you can decide whether to only show the deformations and crack widths of the governing action or (as shown in Figure 5.35) the results of all analyzed load cases and combinations.

Member No.

The deformations and crack widths are sorted by members.

Location x

The x-locations refer to the FE nodes that have been created on the members for the nonlinear calculation. Use the [Details] button in the *Iteration Parameters* tab of the *Settings for Nonlinear Calculation* dialog box (see Figure 2.30) to access the *FE Mesh* dialog box of RFEM where you can adjust the FE mesh settings.

Loading

This column displays the numbers of the load cases or load combinations that are governing for the individual cross-sections, members, or sets of members.

If several load cases or combinations have been designed, it is possible to show the results of all analyzed actions by selecting the *All load cases* check box.

Type

u_g

The column lists the resulting displacements u_g on the individual member locations. The total displacement is related to the global XYZ-coordinate system.

The deformation components in the direction of the global axes X, Y, and Z can be seen among the detailed results in the *Global Nodal Deformations* category. The components in the direction of the local member axes x, y, and z can be found in the *Local Nodal Deformations* category.

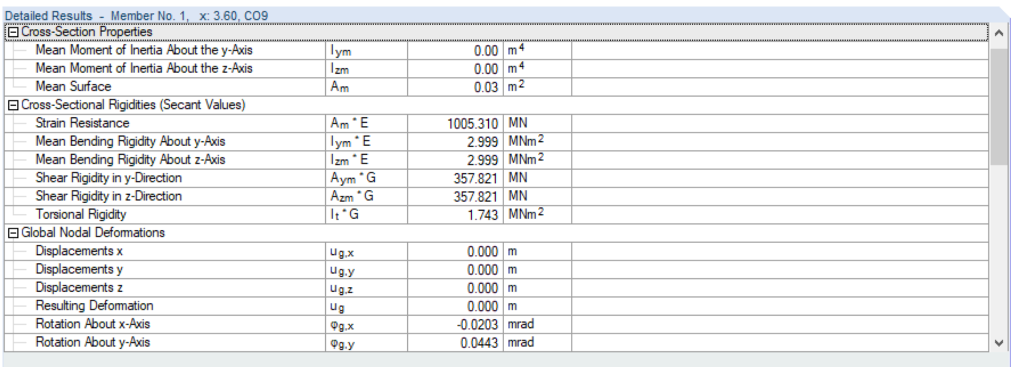
w_k

In this column, the characteristic crack widths according to EN 1992-1-1, 7.3.4, Eq. (7.8) are indicated (see Chapter 2.2.4). For the determination of crack spacing and strains, the program uses the internal forces of the nonlinear calculation.

Existing Value

This column displays the values that are governing for the deformation and crack width analyses.

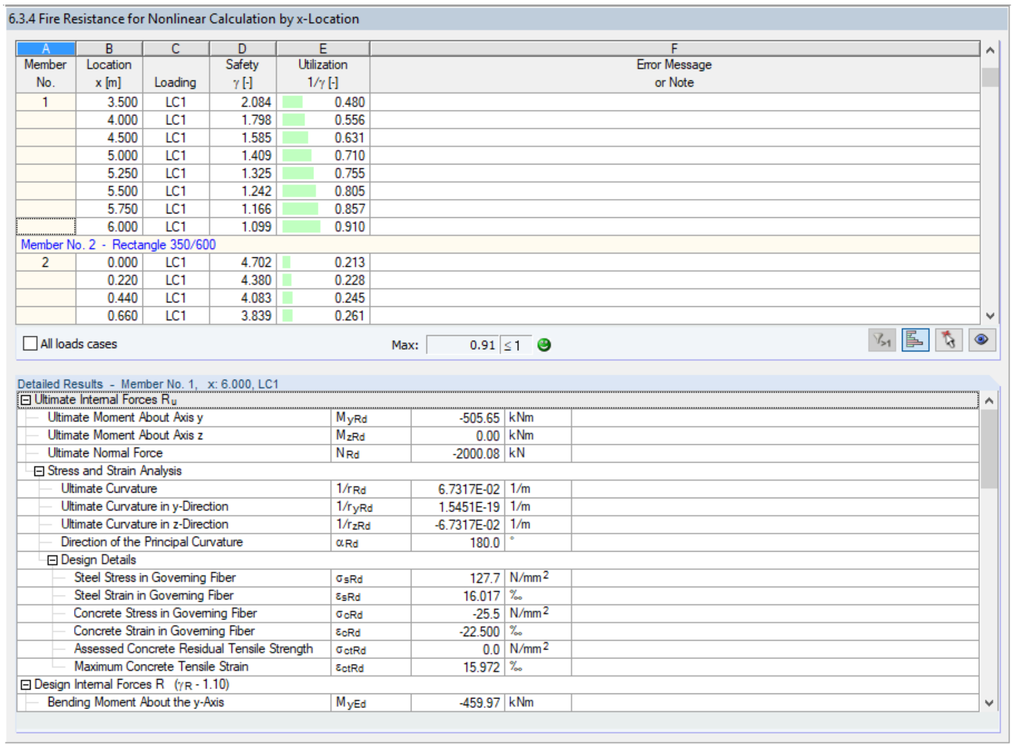
The *Detailed Results* in the lower part of the window (see Figure 5.36) provide information about all design details. They are organized in a tree structure.


Detailed Results - Member No. 1, x: 3.60, CO9

⊟ Cross-Section Properties			
Mean Moment of Inertia About the y-Axis	I_{ym}	0.00	m^4
Mean Moment of Inertia About the z-Axis	I_{zm}	0.00	m^4
Mean Surface	A_m	0.03	m^2
⊟ Cross-Sectional Rigidities (Secant Values)			
Strain Resistance	$A_m \cdot E$	1005.310	MN
Mean Bending Rigidity About y-Axis	$I_{ym} \cdot E$	2.999	MNm^2
Mean Bending Rigidity About z-Axis	$I_{zm} \cdot E$	2.999	MNm^2
Shear Rigidity in y-Direction	$A_{ym} \cdot G$	357.821	MN
Shear Rigidity in z-Direction	$A_{zm} \cdot G$	357.821	MN
Torsional Rigidity	$I_t \cdot G$	1.743	MNm^2
⊟ Global Nodal Deformations			
Displacements x	$u_{g,x}$	0.000	m
Displacements y	$u_{g,y}$	0.000	m
Displacements z	$u_{g,z}$	0.000	m
Resulting Deformation	u_g	0.000	m
Rotation About x-Axis	$\varphi_{g,x}$	-0.0203	mrad
Rotation About y-Axis	$\varphi_{g,y}$	0.0443	mrad

Figure 5.36 *Detailed Results* window section with details for cross-sectional rigidities and deformations

5.5.3 Nonlinear Calculation - Fire Resistance


6.3.4 Fire Resistance for Nonlinear Calculation by x-Location

A Member No.	B Location x [m]	C Loading	D Safety γ [-]	E Utilization 1/γ [-]	F Error Message or Note
1	3.500	LC1	2.084	0.480	
	4.000	LC1	1.798	0.556	
	4.500	LC1	1.585	0.631	
	5.000	LC1	1.409	0.710	
	5.250	LC1	1.325	0.755	
	5.500	LC1	1.242	0.805	
	5.750	LC1	1.166	0.857	
	6.000	LC1	1.099	0.910	
Member No. 2 - Rectangle 350/600					
2	0.000	LC1	4.702	0.213	
	0.220	LC1	4.380	0.228	
	0.440	LC1	4.083	0.245	
	0.660	LC1	3.839	0.261	

All loads cases   Max: 0.91 ≤ 1

Detailed Results - Member No. 1, x: 6.000, LC1

Ultimate Internal Forces R_u			
Ultimate Moment About Axis y	M_{yRd}	-505.65	kNm
Ultimate Moment About Axis z	M_{zRd}	0.00	kNm
Ultimate Normal Force	N_{Rd}	-2000.08	kN
Stress and Strain Analysis			
Ultimate Curvature	$1/r_{Rd}$	6.7317E-02	1/m
Ultimate Curvature in y-Direction	$1/r_{yRd}$	1.5451E-19	1/m
Ultimate Curvature in z-Direction	$1/r_{zRd}$	-6.7317E-02	1/m
Direction of the Principal Curvature	α_{Rd}	180.0	°
Design Details			
Steel Stress in Governing Fiber	σ_{sRd}	127.7	N/mm²
Steel Strain in Governing Fiber	ε_{sRd}	16.017	‰
Concrete Stress in Governing Fiber	σ_{cRd}	-25.5	N/mm²
Concrete Strain in Governing Fiber	ε_{cRd}	-22.500	‰
Assessed Concrete Residual Tensile Strength	σ_{ctRd}	0.0	N/mm²
Maximum Concrete Tensile Strain	ε_{ctRd}	15.972	‰
Design Internal Forces R (γ_R - 1.10)			
Bending Moment About the y-Axis	M_{yEd}	-459.97	kNm

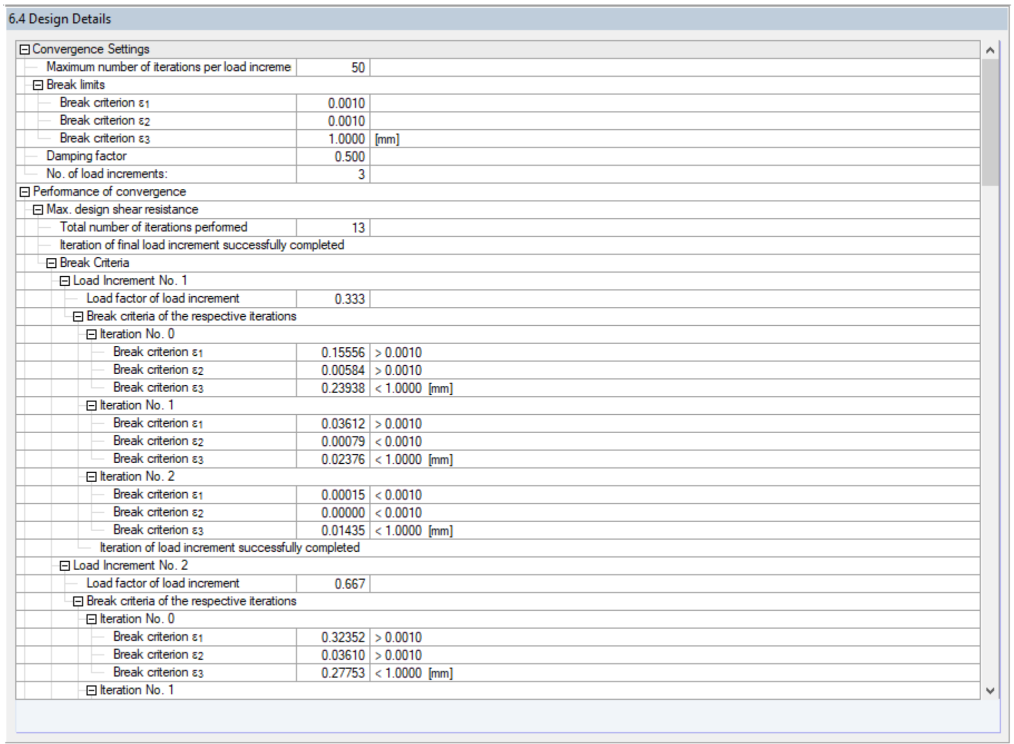
Figure 5.37 Window *6.3.4 Fire Resistance for Nonlinear Calculation by x-Location*

The window shows the governing safety factors and design ratios of the nonlinear fire resistance design. They result from the internal forces for the fire situation, the reinforcement parameters, and the specifications set in the *Settings for Nonlinear Calculation* dialog box (see Figure 4.4 and Figure 4.6).

The lower part lists all *Detailed Results* for the table row selected above.

The table columns correspond to the ones of Window 6.1.1. They are described in Chapter 5.5.1.

5.5.4 Nonlinear Calculation - Design Details



6.4 Design Details

⊟ Convergence Settings		
Maximum number of iterations per load increme	50	
⊟ Break limits		
Break criterion ε1	0.0010	
Break criterion ε2	0.0010	
Break criterion ε3	1.0000	[mm]
Damping factor	0.500	
No. of load increments:	3	
⊟ Performance of convergence		
⊟ Max. design shear resistance		
Total number of iterations performed	13	
Iteration of final load increment successfully completed		
⊟ Break Criteria		
⊟ Load Increment No. 1		
Load factor of load increment	0.333	
⊟ Break criteria of the respective iterations		
⊟ Iteration No. 0		
Break criterion ε1	0.15556	> 0.0010
Break criterion ε2	0.00584	> 0.0010
Break criterion ε3	0.23938	< 1.0000 [mm]
⊟ Iteration No. 1		
Break criterion ε1	0.03612	> 0.0010
Break criterion ε2	0.00079	< 0.0010
Break criterion ε3	0.02376	< 1.0000 [mm]
⊟ Iteration No. 2		
Break criterion ε1	0.00015	< 0.0010
Break criterion ε2	0.00000	< 0.0010
Break criterion ε3	0.01435	< 1.0000 [mm]
Iteration of load increment successfully completed		
⊟ Load Increment No. 2		
Load factor of load increment	0.667	
⊟ Break criteria of the respective iterations		
⊟ Iteration No. 0		
Break criterion ε1	0.32352	> 0.0010
Break criterion ε2	0.03610	> 0.0010
Break criterion ε3	0.27753	< 1.0000 [mm]
⊟ Iteration No. 1		

Figure 5.38 Window *6.4 Design Details*

In the final window, you can check the course of the nonlinear calculation. The output is partitioned into two main parts.

Convergence settings

This part lists the general specifications for the nonlinear calculation (see explanations to Figure 2.30).

Performance of convergence

The convergence behavior is an important criterion for evaluating the results. The iteration process allows conclusions to be drawn on the quality of the nonlinear calculation. In the table, you can see the break criteria of the iteration steps for each load increment. The output is displayed separately for the ultimate limit state, the *Serviceability limit state*, and the *Fire Resistance Design*.

In the majority of cases, a nonlinear calculation converges because the deviations concerning internal forces, stiffnesses, and deformations are steadily decreasing. This effect can be checked by means of the values ε_1 and ε_2 in successive iterations: Hence, peaks or increasing deviations (in case of stability analyses, for example) are easy to understand.

Looking at the convergence behavior, you can also see how the parameters can be influenced to control the calculation (see Chapter 2.4.9).

6 Result Evaluation

The design results can be evaluated and adjusted in different ways. The result windows are described in detail in Chapter 5. Chapter 6 describes the graphical evaluation of results and the possibilities for modifying the reinforcement proposal.

6.1 Reinforcement Proposal

The result windows 3.1 and 3.2 show how to cover the required reinforcement areas with rebars to fulfill the serviceability limit state design, for example.
The reinforcement proposal is graphically displayed in the lower part of Window 3.1 *Provided Longitudinal Reinforcement* and Window 3.2 *Provided Shear Reinforcement* in the form of a 3D drawing (see Figure 5.10 and Figure 5.16).

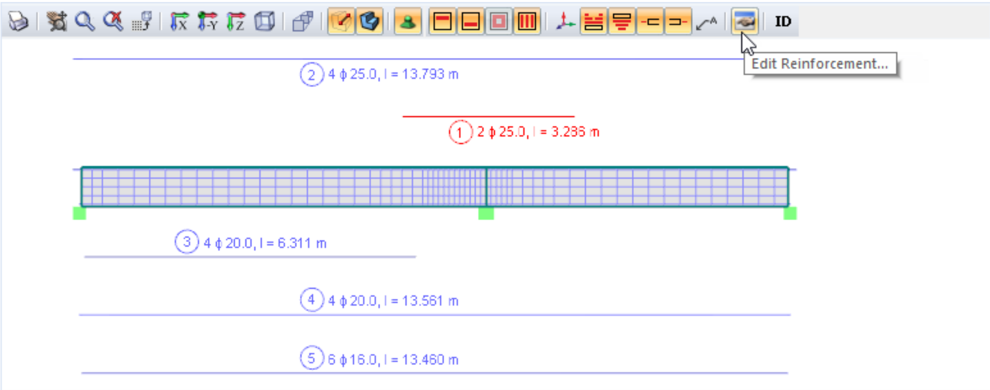


Figure 6.1 Reinforcement drawing in Window 3.1 *Provided Longitudinal Reinforcement*

The current item (row selected in table above, in which the pointer is placed) is highlighted in red. The graphic allows you to see and, if necessary, adjust the position and arrangement of the individual item members.

The graphic window provides mouse functions for zooming, rotating, and shifting the representation familiar from RFEM. By scrolling the wheel button you can maximize or minimize the model. By holding down the wheel button you can shift the reinforcement graphic. When you additionally hold down the [Ctrl] key, you can rotate the graphic (also possible with wheel button and by holding down the right mouse button). The pointer symbols indicate the selected function.

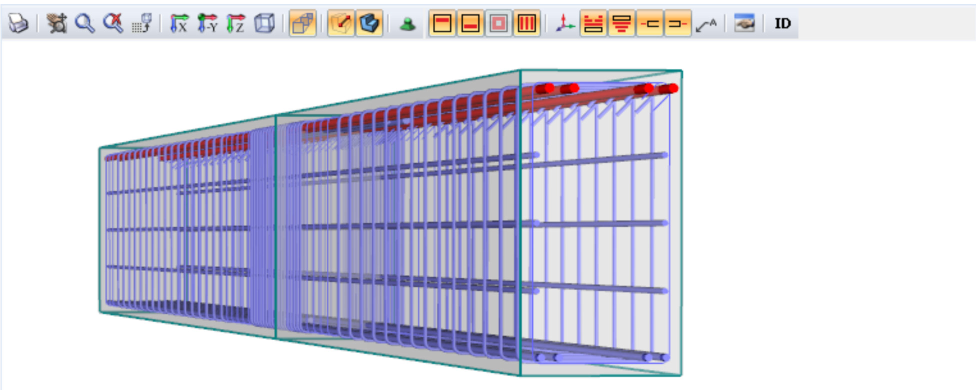
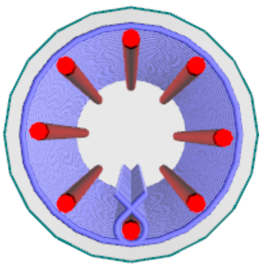


Figure 6.2 3D representation of reinforcement cage

The buttons have the following functions:

Button	Description	Function
	Print	Opens the *Graphic Printout* dialog box → Figure 7.5
	Move, zoom, rotate	Allows for shifting, zooming, and rotating the reinforcement graphic with the mouse
	Zoom	Allows for zooming by drawing a window
	Whole model	Resets to full view
	Previous view	Sets the previous view
	View in X / -Y / Z	Sets the view in direction of axes X and Z, or against axis Y
	Isometric view	Shows the object in 3D
	Perspective view	Shows reinforcement graphic in perspective view (can be combined with all view types)
	Line model	Displays or hides the cross-section outlines in line model
	Solid model	Turns the concrete represented in cross-section on and off
	Supports	Displays or hides the supports
	Top reinforcement	Displays or hides the top longitudinal reinforcement
	Bottom reinforcement	Displays or hides the bottom longitudinal reinforcement
	Peripheral reinforcement	Displays or hides the circumferential and the secondary reinforcement
	Stirrups	Displays or hides the stirrups
	Member axis system	Controls the display of local member axes x,y,z
	Longitudinal reinforcement - top	Displays or hides the item members of top reinforcement
	Longitudinal reinforcement - bottom	Displays or hides the item members of bottom reinforcement


Column reinforcement in perspective view

	Shear reinforcement	Displays or hides the position stirrups and the link zone descriptions
	Stirrup description	Shows stirrup descriptions parallel or perpendicular to the member
	Start of anchorage	Displays or hides the anchorage lengths on member start
	End of anchorage	Displays or hides the anchorage lengths on member end
	Description	Displays or hides the item descriptions in X-view (in preparation)
	Edit reinforcement	Opens the *Edit Longitudinal Reinforcement* (Figure 5.13) or *Edit Shear Reinforcement* (Figure 5.17) dialog box.
	Numbering	Displays or hides the numbering of members and sets of members

Table 6.1 Buttons in result windows 3.1 and 3.2

Edit reinforcement

The [Edit Reinforcement] button selected in Figure 6.1 opens the edit dialog box of the current reinforcement item. The dialog box is shown in Figure 5.13 and Figure 5.17. It can be used to check and, if necessary, modify the parameters of the longitudinal or shear reinforcement (e.g. reinforcement layout, rebar diameters, anchorages).

You can also access the edit dialog boxes in the shortcut menu of the marked item members or reinforcements.

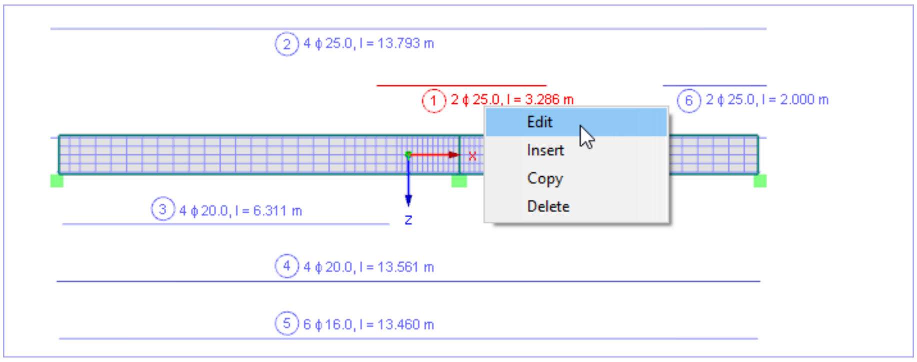
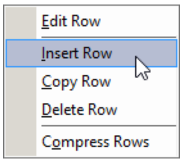


Figure 6.3 Shortcut menu of reinforcement

The following article describes how to edit a reinforcement proposal:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000651



Shortcut menu of table

Complete reinforcement

Using the shortcut menu shown in Figure 6.3, you can also *Insert* a new reinforcement item. The same is possible in the table of Window 3.1 (see shortcut menu shown on the left).

The *Edit Longitudinal Reinforcement* dialog box appears. The next *Item No.* is preset. Now, you can define number and position of the rebars, as well as diameter and anchorage lengths of the reinforcement. After clicking [OK], the new reinforcement item is added to the table and the graphic.

More editing options are described in the following chapter.

Save reinforcement

A modified reinforcement can be saved as a template by using the [Save] table button. You have to enter the *Name of the Reinforcement Template* in the *Save Provided Reinforcement* dialog box (see Figure 5.15).

Use this template to preset the user-defined reinforcement configuration when the design specifications in Window 1.6 are changed. The changes are not lost if RF-CONCRETE Members creates a new provided reinforcement.

Importing reinforcement templates is described in Chapter 3.6 (see Figure 3.25).

6.2 3D Rendering of Reinforcement

The [3D-Rendering] button is available in the windows *3.1 Provided Longitudinal Reinforcement* and *3.2 Provided Shear Reinforcement*, providing the possibility to display the reinforcement of the selected member photo-realistically and modify it, if required.

It is also possible to access the 3D rendering by double-clicking the reinforcement in the graphical area of windows 3.1 and 3.2.

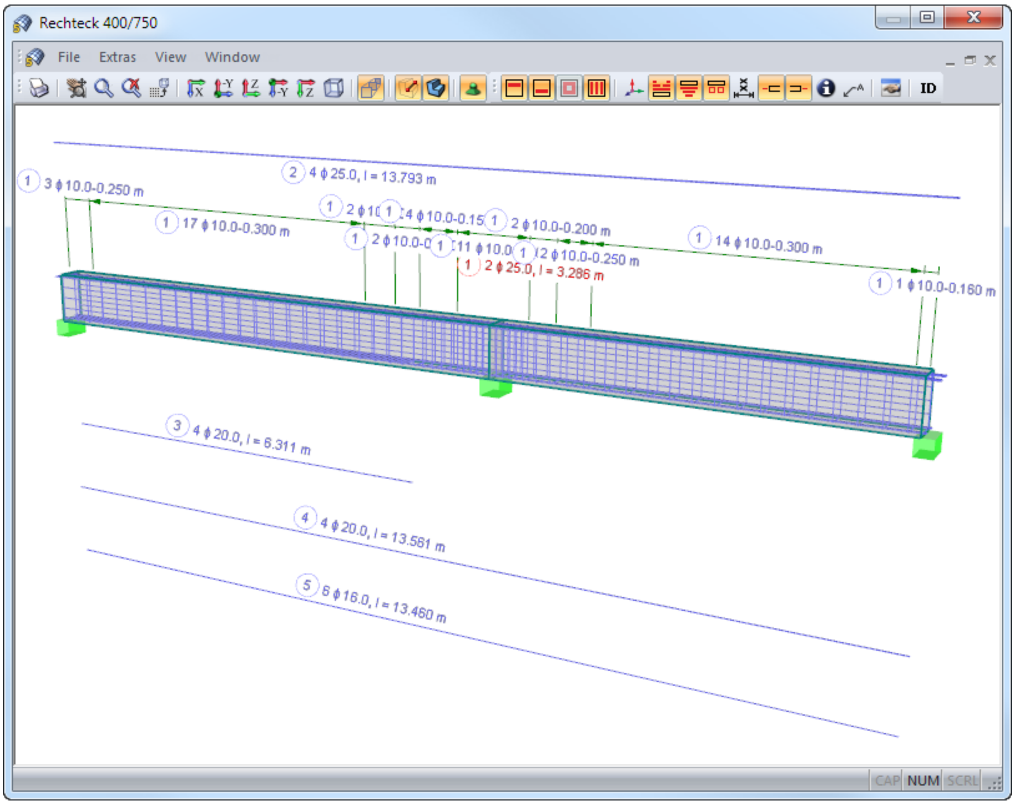
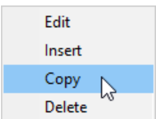


Figure 6.4 3D rendering of provided longitudinal and shear reinforcement

You can control the graphic in the *View* menu, or by using the corresponding buttons (see Table 6.1). In addition, you can use the mouse functions described in Chapter 6.1.

Copy reinforcement

The graphic window provides the option to copy reinforcements by item. This way, it is possible, for example, to quickly create new reinforcement layers from existing rebars or to generate secondary reinforcements.



Shortcut menu of reinforcement

First, select the reinforcement or item description that you want to copy. Then, right-click the red-highlighted object to open the shortcut menu of the reinforcement (see figure on the left). The *Copy* option opens the *Move or Copy* dialog box (see Figure 6.5).

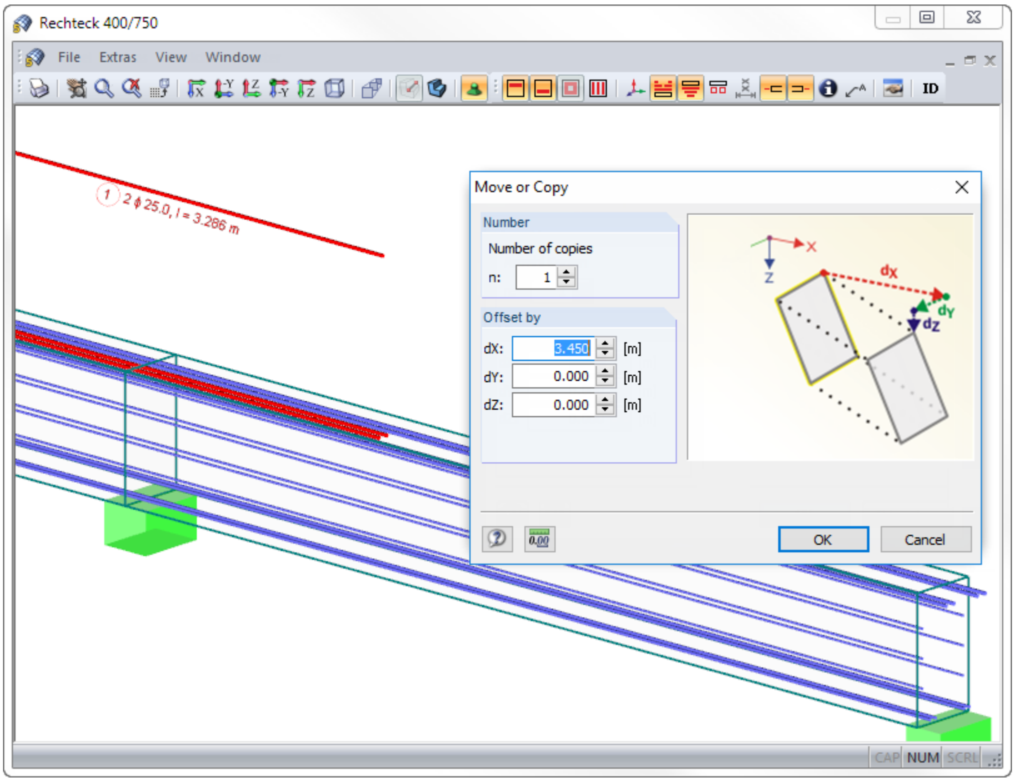


Figure 6.5 Reinforcement item created via *Copy* shortcut menu

In the dialog box, you can specify the *Number* of copies and the displacement vector in the direction of the global axes X, Y, and Z.

After clicking [OK], a new reinforcement item (or several, if applicable) is created.

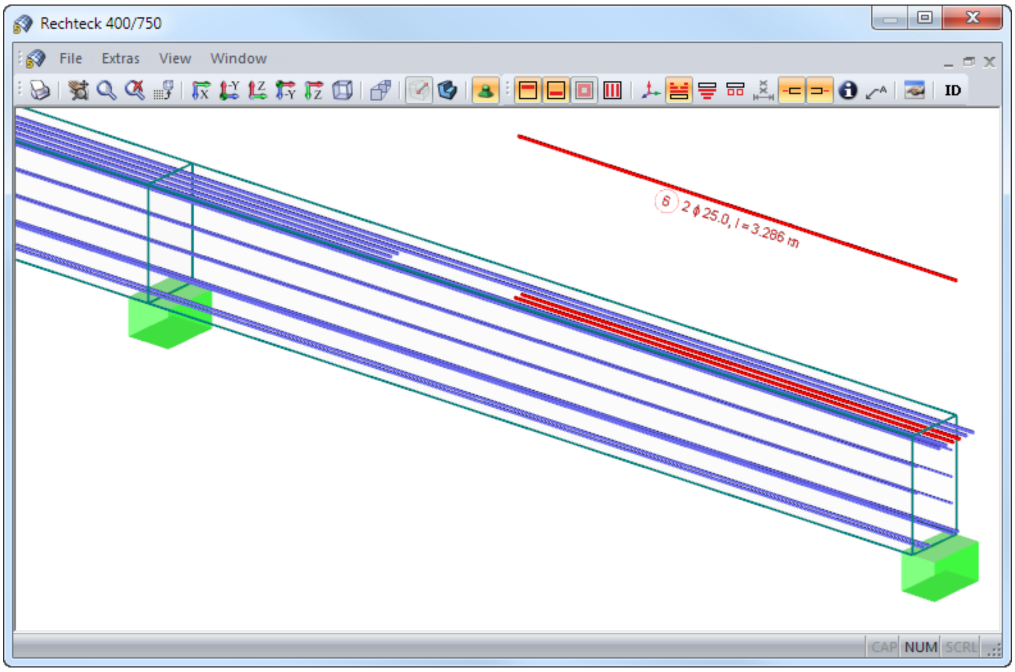


Figure 6.6 Copied reinforcement item

Windows display

The *Window* menu provides various ways to display the reinforcement in several windows. For spatial control, we recommend using the function

Window → Isometric and 3 Views.

The 3D rendering window is divided as follows.

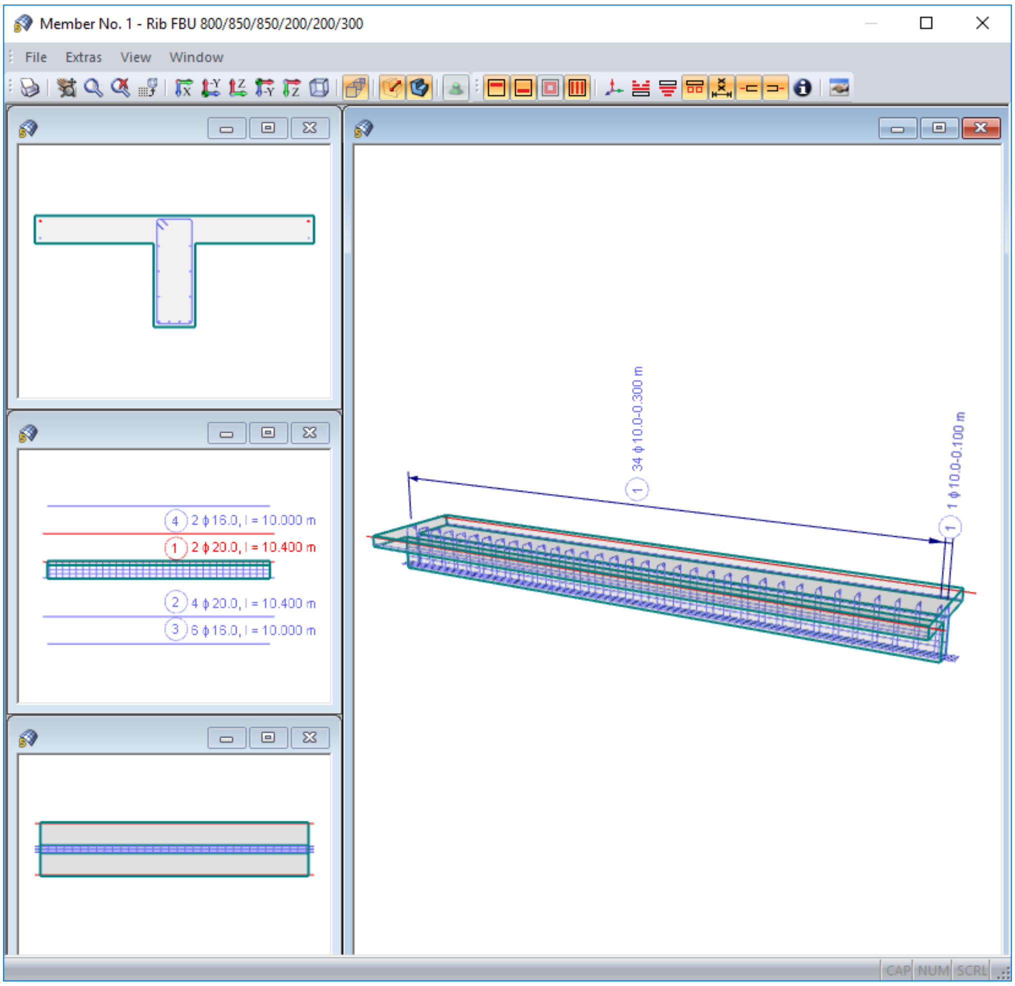


Figure 6.7 *Isometric and 3 Views window display*

In addition to the isometric view shown in the main window, the view in the longitudinal direction X, the lateral view in -Y, and the top view in Z are displayed.

If changes are made in one window, the views in the other windows are updated automatically.

With the [Print] button you can send the current graphic directly to the printer. You can also transfer it to the printout report or the clipboard. Printing several windows is not possible.

6.3 Results on RFEM Model

You can also evaluate the design results in the RFEM work window.

6.3.1 Background Graphic and View Mode

The RFEM work window in the background is useful when you want to find the position of a particular member in the model: The member selected in the result window of RF-CONCRETE Members is indicated by an arrow in the background graphic; the x-location is highlighted in the selection color.

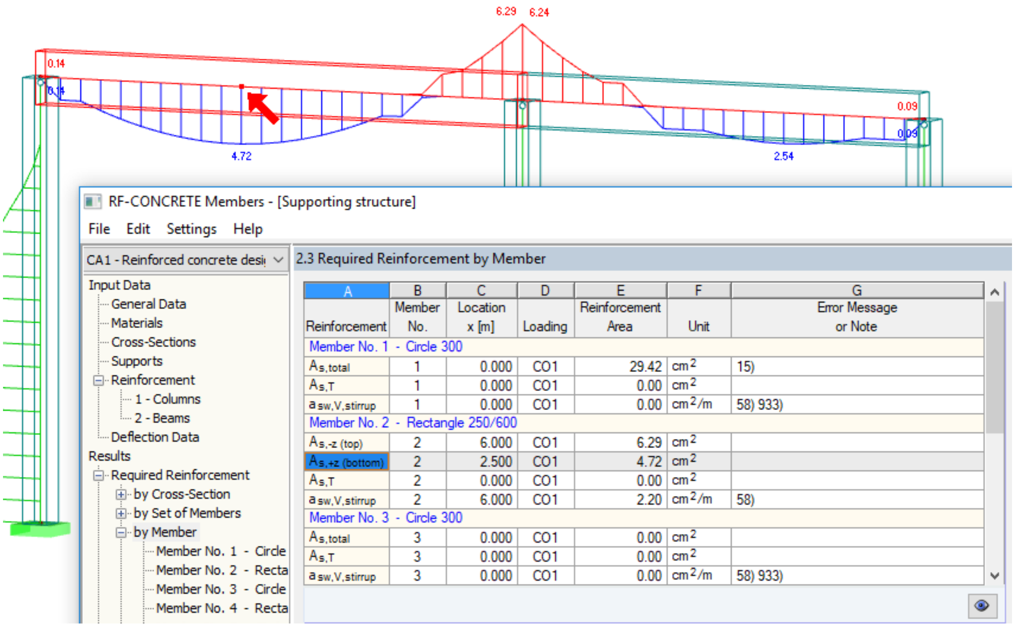
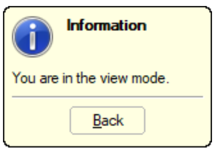


Figure 6.8 Indication of member and current x-location in RFEM model



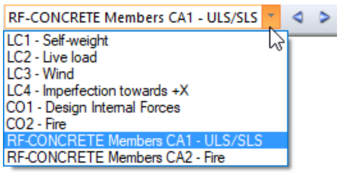
If the background graphic does not display any results of RF-CONCRETE Members, you can use the [Jump to graphic] button to activate *view mode*: The program hides the module window so that you can set the design case in the RFEM toolbar and activate the results with the button.

The view mode provides all functions of the *View* menu, for example zooming, moving, or rotating the model view so that you can adjust the representation accordingly.

Click [Back] to return to RF-CONCRETE Members.

6.3.2 RFEM Work Window

You can also check the reinforcements and detailed results graphically on the RFEM model: Click the [Graphics] button to exit the design module. The work window of RFEM will subsequently show all design results such as the internal forces of a load case.



You can set the design cases in the drop-down list of the RFEM menu bar.

Results navigator

The *Results* navigator is tailored to the designs of the RF-CONCRETE Members add-on module: You can select the design results for the ultimate and the serviceability limit state designs, the fire resistance design, and the nonlinear calculation including all intermediate results.

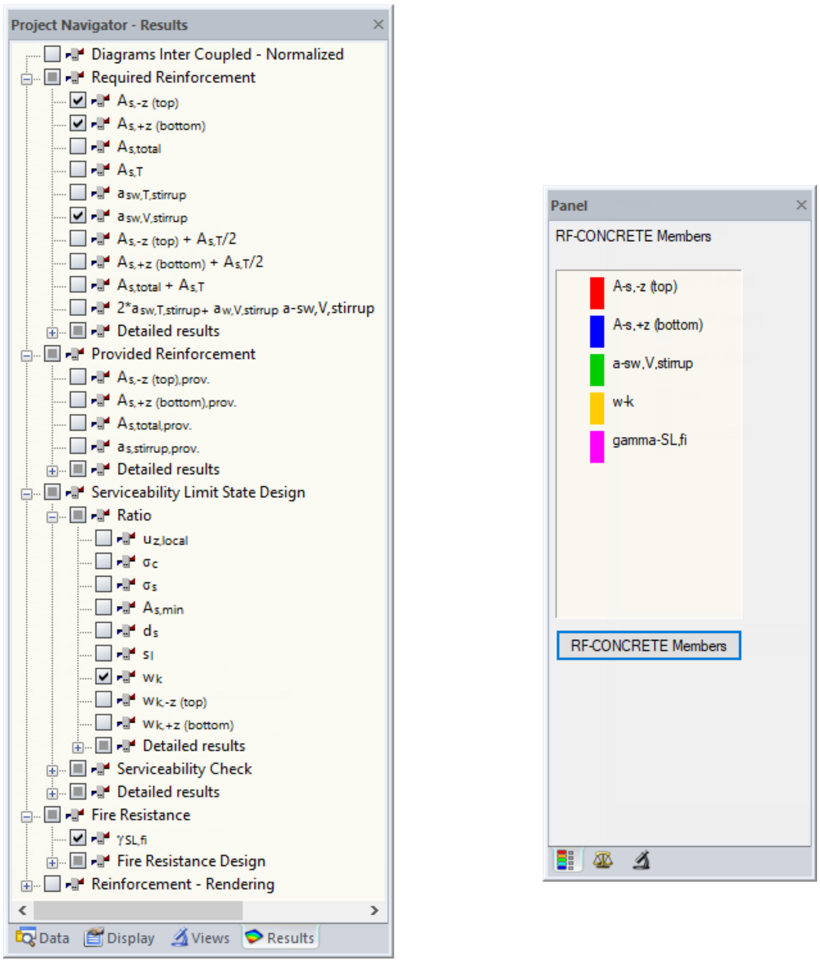


Figure 6.9 *Results* navigator and panel for RF-CONCRETE Members

The *Results* navigator allows you to display several reinforcement types or designs at the same time. This way you can graphically compare the required longitudinal reinforcement with the provided longitudinal reinforcement, for example. The panel is synchronized with the selected types of results.

An example of graphical documentation for shear analysis can be found in the following article: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000715

This article shows how the curtailment or reinforcement covering lines can be displayed: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001482

Due to the multiple selection and automatic color assignment, the display options for member results in the *Display* navigator are without effect.

The *Results* navigator also allows for graphically evaluating the design details of nonlinear analyses.

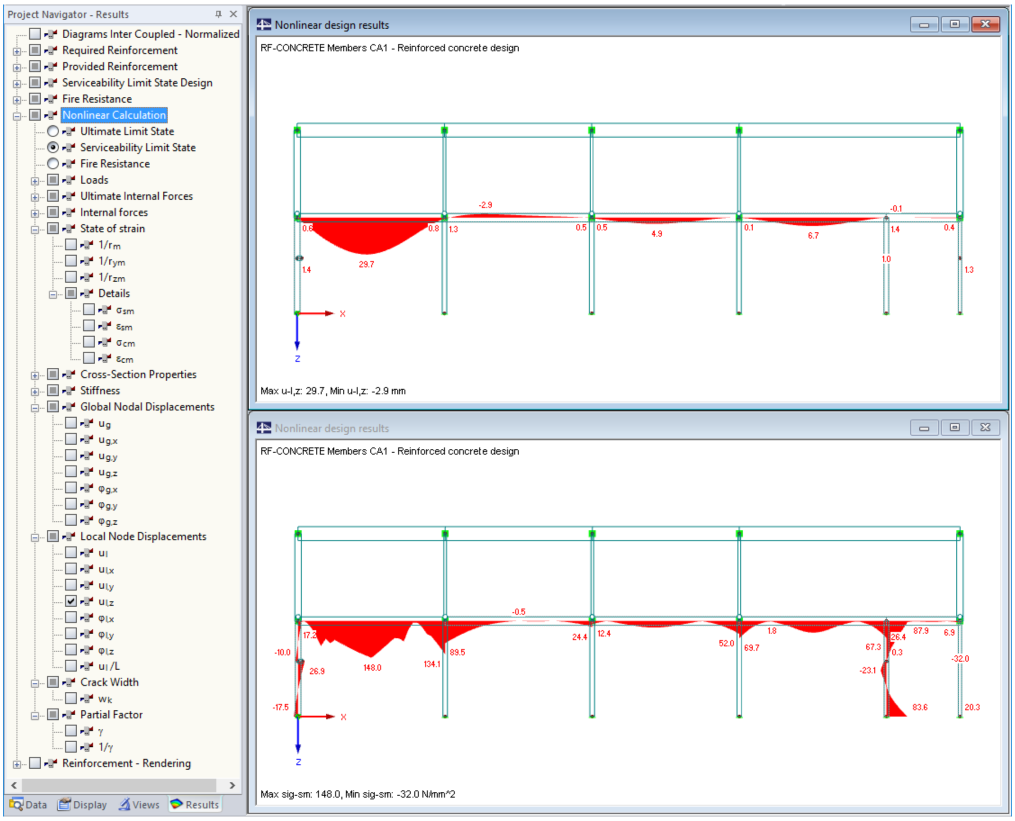


Figure 6.10 Graphical output of nonlinear design results

With the *Reinforcement - Rendering* option in the navigator, it is possible to display the rebars and stirrups in the RFEM work window.

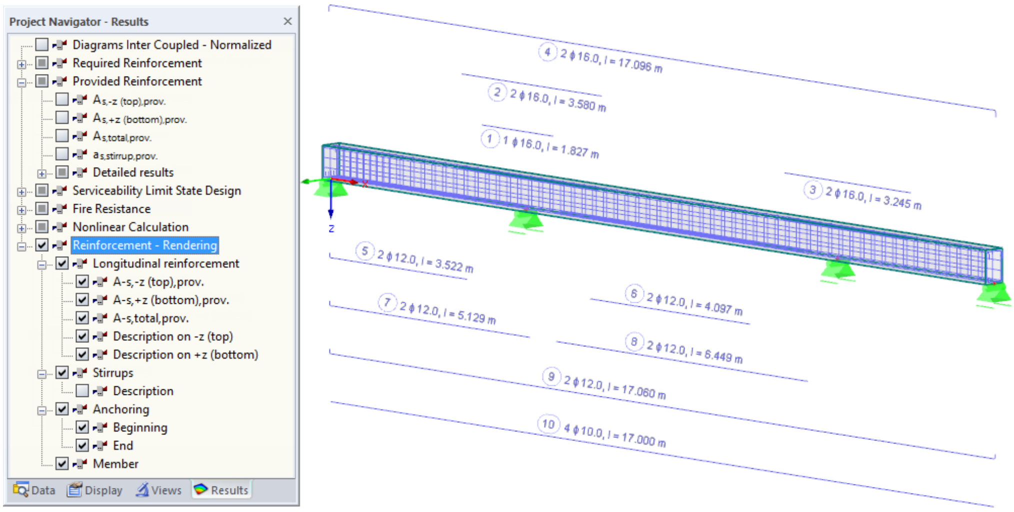
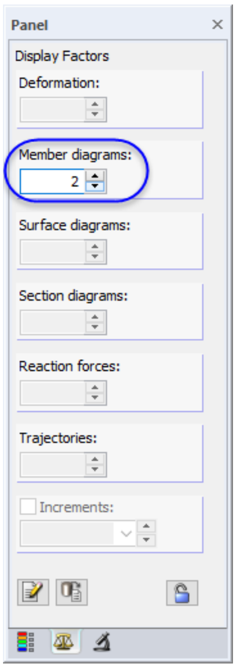


Figure 6.11 3D rendering of reinforcement in RFEM

As is usual in RFEM, you can use the [Print Graphic] button to send the graphics to the printer or transfer them to the printout report (see Chapter 7.2).

Panel

The panel with the usual control options is also available for the result evaluation. The functions are described in chapter 3.4.6 of the RFEM manual.



Tab Factors

In the second tab of the panel, you can set the *Display Factors* for the reinforcements, stresses, strains, or load-bearing capacities.

The *Filter* panel tab allows you to select the results of particular members for display. This function is described in chapter 9.9.3 of the RFEM manual.

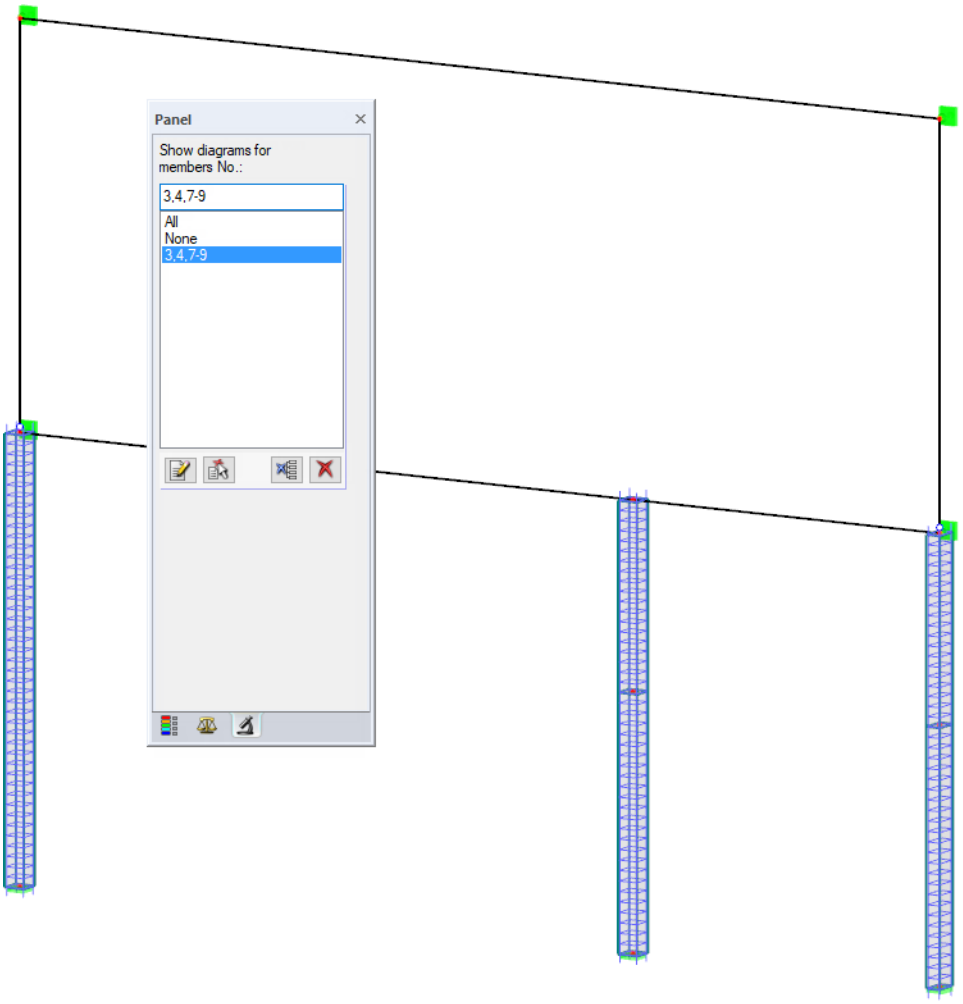
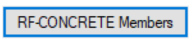


Figure 6.12 Display of selected members via *Filter* tab

The figure above shows the reinforcement of a building's concrete columns. The remaining members are shown in the model but are displayed without reinforcement.



To return to the add-on module, click the [RF-CONCRETE Members] button in the panel.

6.4 Result Diagrams

To access the result diagrams in the RFEM work window, select

Results → Result Diagrams for Selected Members

in the menu or use the corresponding button in the RFEM toolbar.

A window opens showing the distribution of reinforcement areas and detailed results on the selected member or set of members.

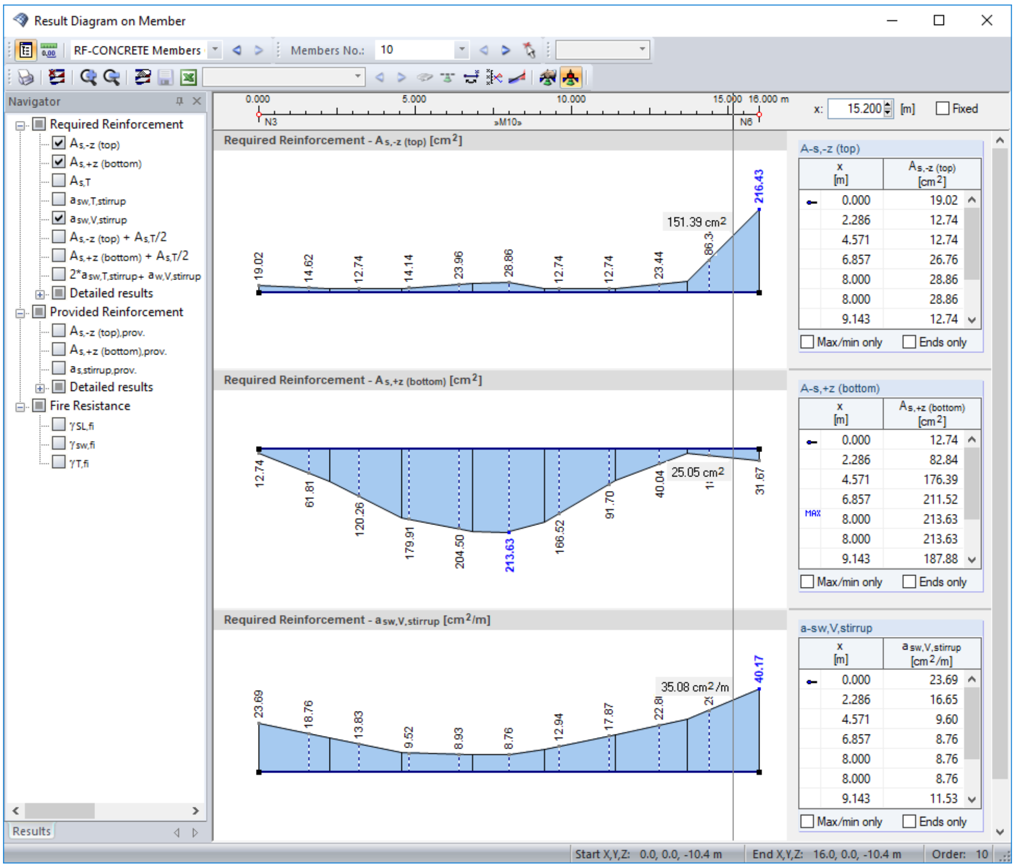


Figure 6.13 *Result Diagram on Member dialog box*

In the navigator on the left, you can select the reinforcements and detailed results. With the lists in the toolbar you can switch between the design cases of RF-CONCRETE Members, as well as members or sets of members.

In chapter 9.5 of the RFEM manual, you can find a detailed description of the *Result Diagram on Member* dialog box (i.a. with the smoothing options).

7 Printout

7.1 Printout Report

RFEM creates a printout report for the data of RF-CONCRETE Members, which can be complemented by graphics and descriptions. With the selection options in the printout report you can determine the design module's data that appears in the printout.

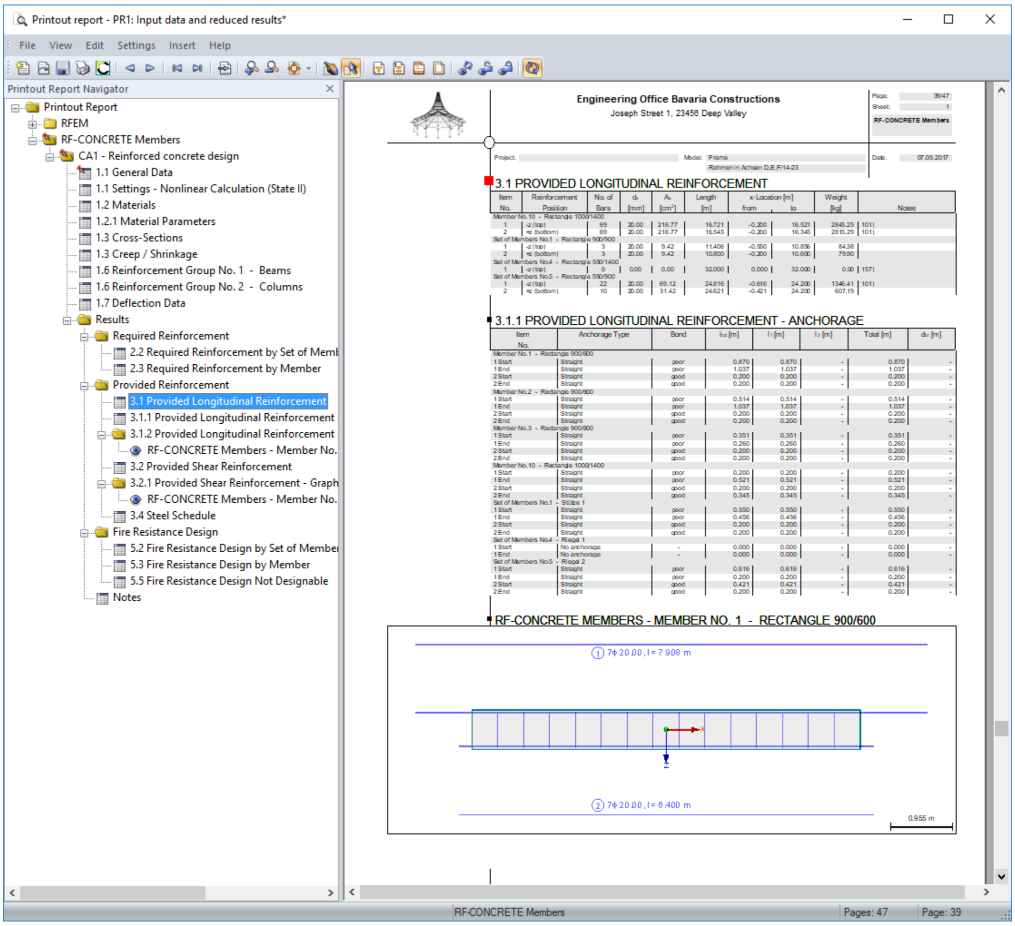


Figure 7.1 Design results in printout report

The reinforcement graphics of windows 3.1 and 3.2 are preset in the printout report (see Figure 7.1).

The printout report is described in detail in the RFEM manual. Chapter 10.1.3.5 *Selecting Data of Add-on Modules* explains how to prepare the input and output data of the add-on modules for the printout.

The *Printout Report Selection* dialog box provides various options for selecting the input and output data (see Figure 7.2). Thus, the reinforcements as well as the member results and detailed results can be individually prepared for documentation.

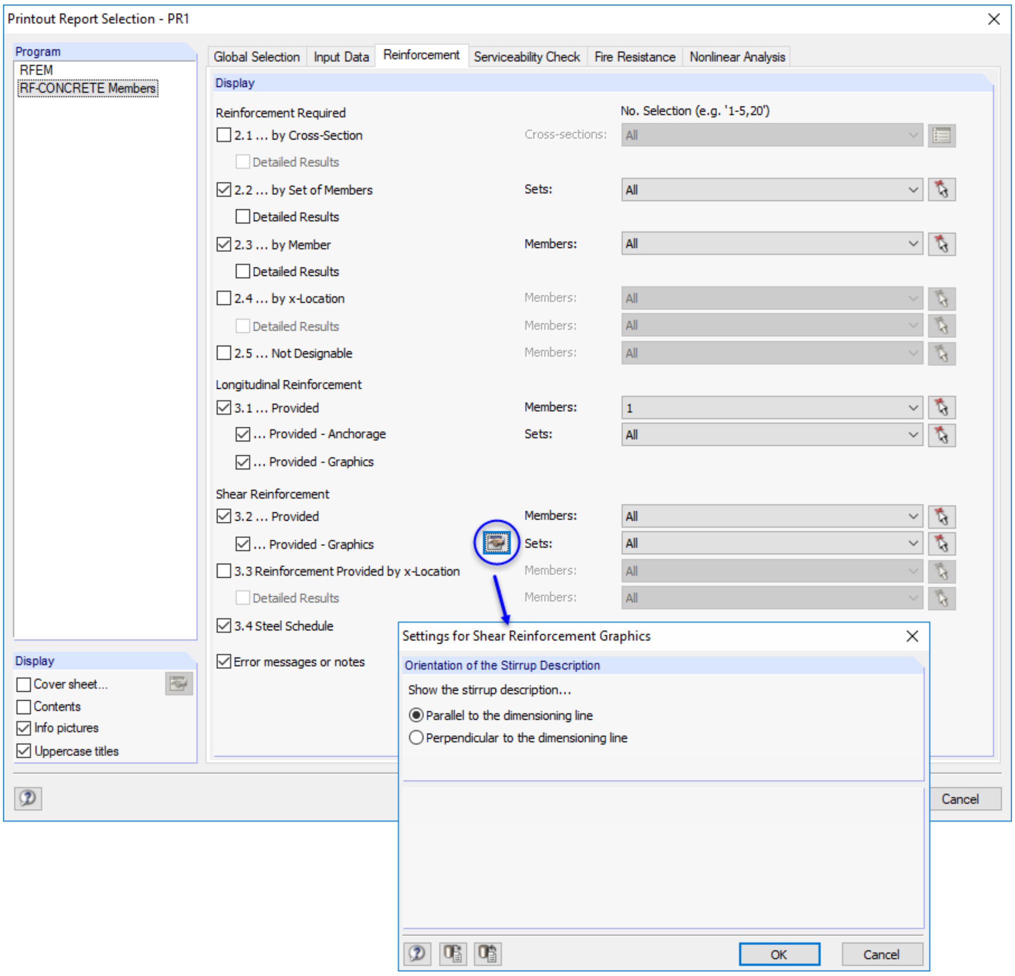


Figure 7.2 *Printout Report Selection* dialog box, *Reinforcement* tab

For large structural systems with many design cases, it is recommended to split the data into several printout reports, thus allowing for a clearly arranged overview.

7.2

Graphic Printout

In RFEM, you can transfer every image shown in the work window to the printout report or send it directly to a printer. Hence, you can also prepare the reinforcements and design criteria shown on the RFEM model for the printout report.

Printing graphics is described in chapter 10.2 of the RFEM manual.

To print the current reinforcement graphic, select

File → Print Graphic

on the menu or use the corresponding button in the toolbar.

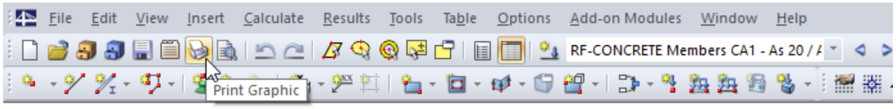
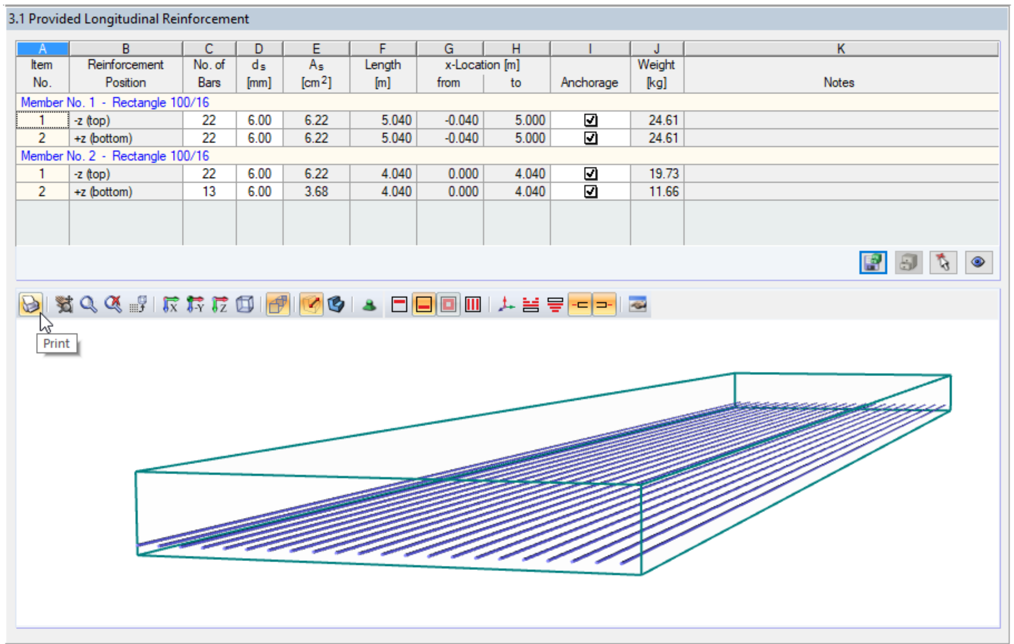


Figure 7.3 [Print Graphic] button in RFEM toolbar

The button is also available in windows 3.1 and 3.2 of RF-CONCRETE Members, as well as in the 3D rendering window (see Figure 6.4).


3.1 Provided Longitudinal Reinforcement

A Item No.	B Reinforcement Position	C No. of Bars	D d_s [mm]	E A_s [cm^2]	F Length [m]	G x-Location [m] from	H to	I Anchorage	J Weight [kg]	K Notes
Member No. 1 - Rectangle 100/16										
1	-z (top)	22	6.00	6.22	5.040	-0.040	5.000	☑	24.61	
2	+z (bottom)	22	6.00	6.22	5.040	-0.040	5.000	☑	24.61	
Member No. 2 - Rectangle 100/16										
1	-z (top)	22	6.00	6.22	4.040	0.000	4.040	☑	19.73	
2	+z (bottom)	13	6.00	3.68	4.040	0.000	4.040	☑	11.66	

Figure 7.4 [Print] button in window 3.1 of RF-CONCRETE Members

The *Graphic Printout* dialog box shown in Figure 7.5 appears.

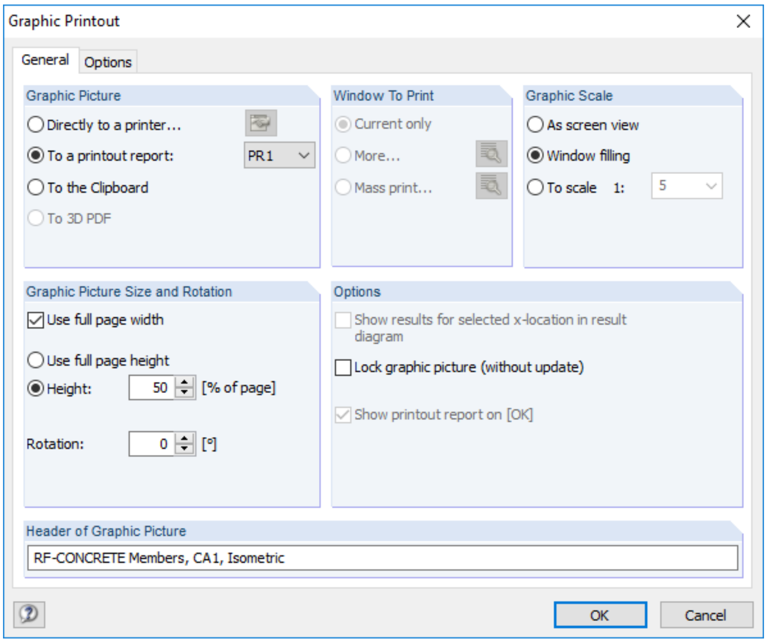
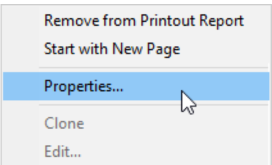


Figure 7.5 *Graphic Printout* dialog box, *General* tab

The dialog box is described in chapter 10.2 of the RFEM manual.

Use the drag-and-drop function to move a graphic within the report to another position.



To retroactively adjust a graphic in the printout report, right-click it in the report navigator. The *Properties* option in the shortcut menu opens the *Graphic Printout* dialog box where you can adjust the settings.

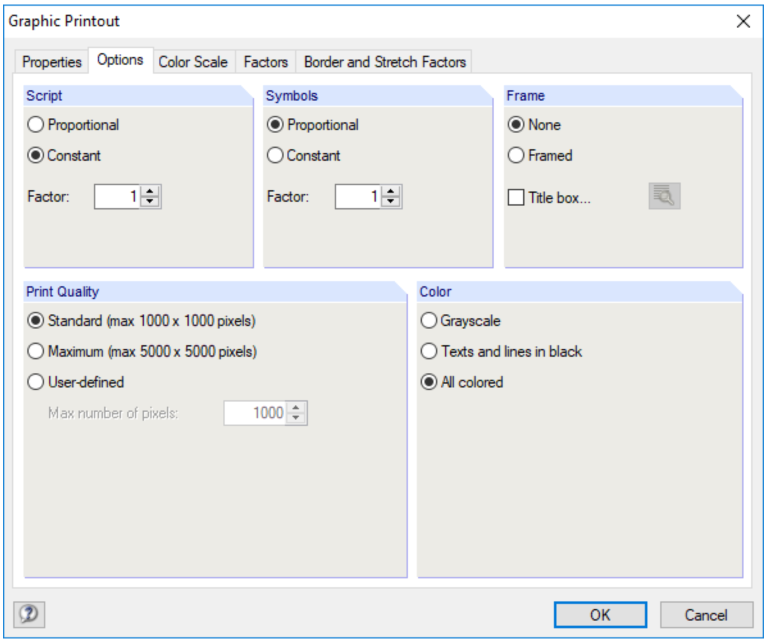


Figure 7.6 *Graphic Printout* dialog box, *Options* tab

Use the *Edit* option in the shortcut menu to change the view (angle of view, object and value display, etc.) in the RFEM work window.

The *General* tab of the *Graphic Printout* dialog box provides the option to save the graphic as a *3D PDF* (see Figure 7.5). Thus, the reinforcement representations and results graphics can be displayed interactively even without RFEM.

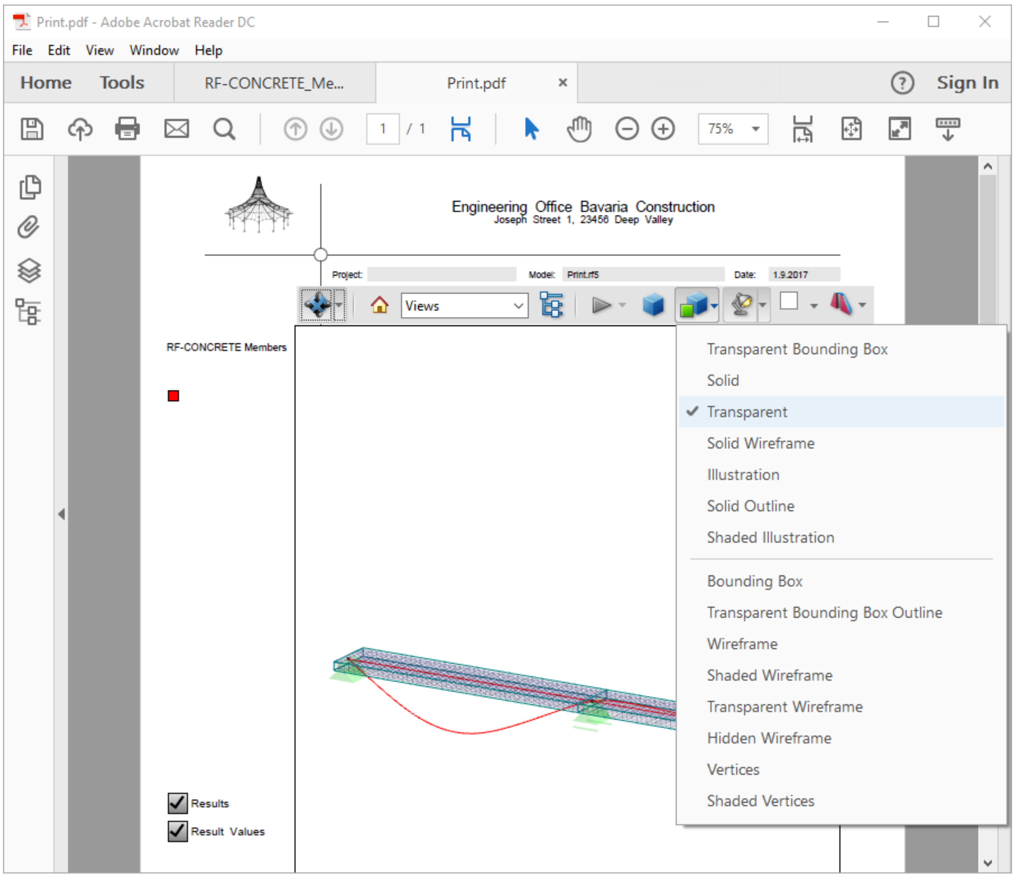


Figure 7.7 3D PDF from RF-CONCRETE Members

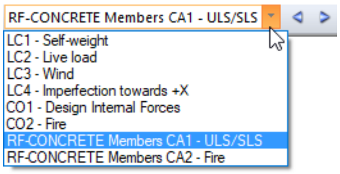
8 General Functions

This chapter describes some menu functions and export options for the design results.

8.1 Design Cases

Design cases allow you to group members and sets of members for the designs, or to design different variants (e.g. modified materials or reinforcement specifications, nonlinear analysis).

It is no problem to analyze the same member in different design cases.



In RFEM you can set the design cases of RF-CONCRETE Members in the load case list of the toolbar.

Creating a new design case

To create a new design case, select the RF-CONCRETE Members menu item

File → New Case.

The following dialog box appears.

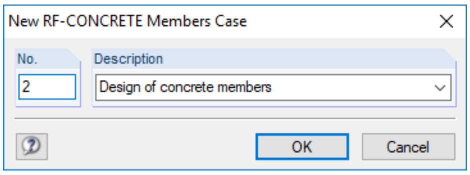


Figure 8.1 *New RF-CONCRETE Members Case dialog box*

In this dialog box, enter an (unassigned) *No.* for the new design case. A *Description* makes a selection in the load case list easier.

After clicking [OK], Window *1.1 General Data* of RF-CONCRETE Members opens where you can enter the design data.

Renaming a design case

To change the description of a design case, select the RF-CONCRETE Members menu entry

File → Rename Case.

The following dialog box appears.

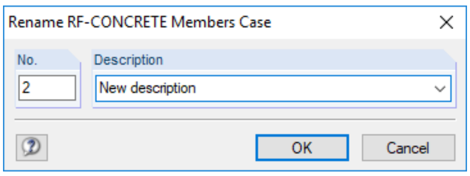


Figure 8.2 *Rename RF-CONCRETE Members Case dialog box*

In this dialog box, you can specify a different *Description* as well as a different *No.* for the design case.

Copying a design case

To copy the input data of the current design case, select the RF-CONCRETE Members menu item

File → Copy Case.

The following dialog box appears.

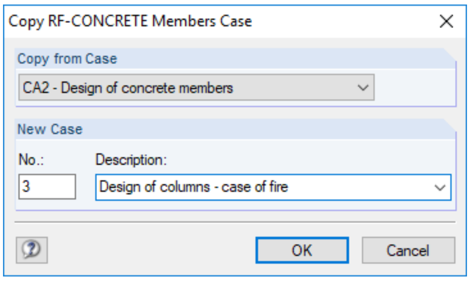


Figure 8.3 *Copy RF-CONCRETE Members Case dialog box*

Define the *No.* and, if necessary, a *Description* for the new case.

Deleting a design case

To delete a design case, select the RF-CONCRETE Members menu entry

File → Delete Case.

The following dialog box appears.

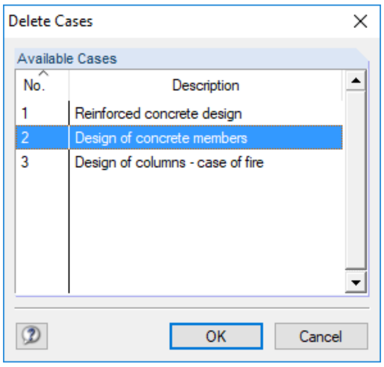


Figure 8.4 *Delete Cases dialog box*

You can select the design case in the *Available Cases* list. To delete the selected case, click [OK].

8.2 Cross-Section Optimization

As mentioned in Chapter 3.3, RF-CONCRETE Members provides a means of optimizing cross-sections. In column C of Window *1.3 Cross-Sections* (see Figure 3.17), specify the relevant cross-section by selecting the check box. The following dialog box appears.

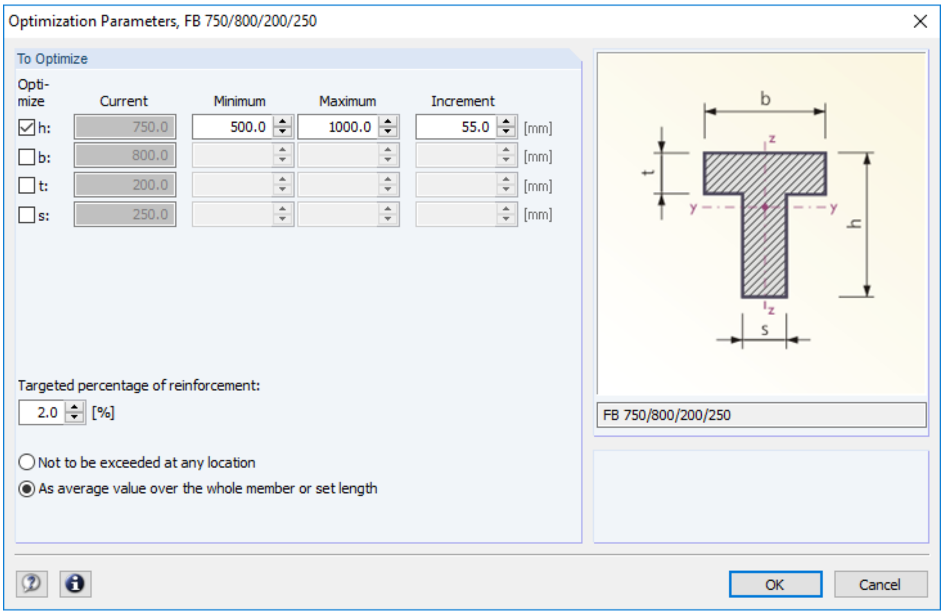


Figure 8.5 *Optimization Parameters* dialog box of a T-beam

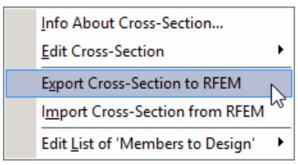
Select check boxes in the *Optimize* column to specify the parameter(s) you want to modify. This enables the *Minimum* and *Maximum* columns where you can define the parameter's upper and lower limits. The *Increment* column controls the interval in which the parameter's magnitude varies during the optimization process.

The optimization criterion specifies that a *Targeted percentage of reinforcement* either must not be exceeded at any location or is an available as average value over the entire member or set of members. The desired reinforcement ratio can be defined in the input field.

During the optimization process, RF-CONCRETE Members examines which dimensions should be used in order to still fulfill the design. Please note that the internal forces are not automatically recalculated with the changed cross-sections (the internal forces may vary significantly due to the changed stiffnesses in the structural system). In fact, it is up to you to decide when the optimized cross-sections are transferred to RFEM for recalculation. It is therefore recommended to recalculate the internal forces with the modified cross-sections after the first optimization and subsequently optimize the cross-sections once again.

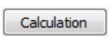
It is also possible to export the modified cross-sections to RFEM: Go to Window *1.3 Cross-Sections* and select the menu item

Edit → Export Cross-Section to RFEM.



You can also use the shortcut menu of the table row in Window 1.3 to export the modified cross-section.

Before a cross-section is transferred, a query appears that asks if the results from RFEM should be deleted.



After starting the [Calculation] in RF-CONCRETE Members, the internal forces and designs are determined in one calculation procedure.

If the cross-section has not yet been exported to RFEM, you can reimport the original *Cross-Section from RFEM* by using the corresponding option in the shortcut menu.

8.3 Units and Decimal Places

The units and decimal places for RFEM and the add-on modules are managed in one dialog box. In RF-CONCRETE Members, you can open the dialog box for adjusting the units by selecting the menu entry

Settings → Units and Decimal Places.

The following dialog box, familiar from RFEM, appears. RF-CONCRETE Members is preset in the *Program / Module* list.

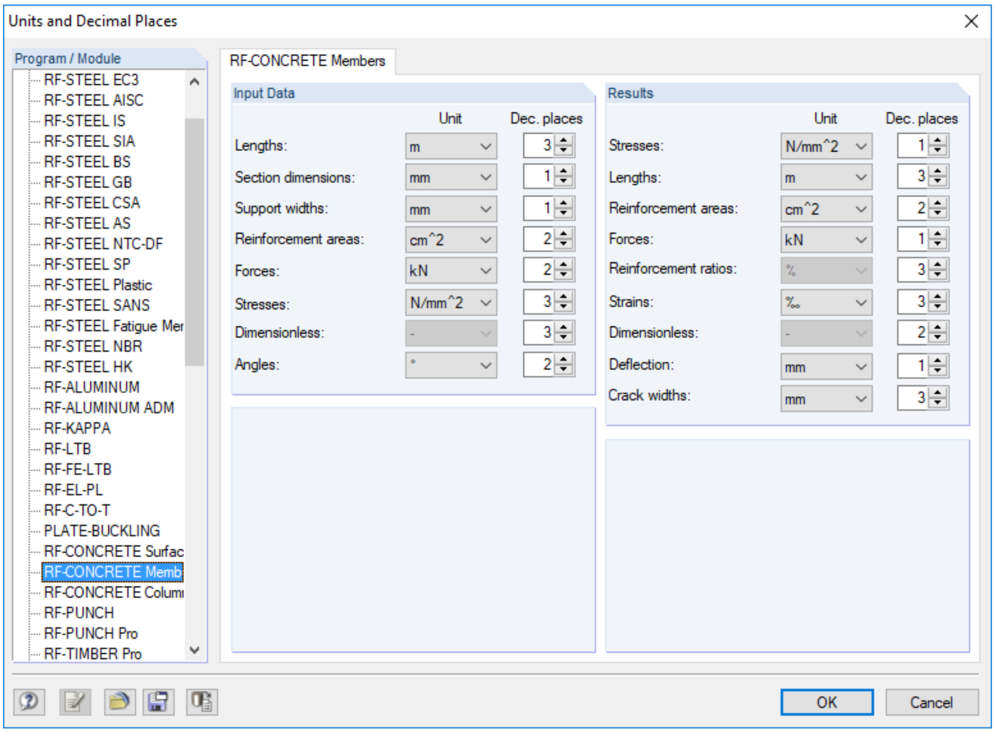


Figure 8.6 *Units and Decimal Places* dialog box

The settings can be saved as a user profile in order to reuse them in other models. This function is described in chapter 11.1.3 of the RFEM manual.

8.4 Export of Results

The results of RF-CONCRETE Members can also be used in other programs.

Clipboard

To copy cells selected in the results windows to the clipboard, use the key combination [Ctrl]+[C]. To insert them, for example in a word processing program, press [Ctrl]+[V]. The headers of the table columns are not transferred.

Printout report

The data of RF-CONCRETE Members can be printed into the printout report (see Chapter 7.1) where they can be exported by selecting the menu item

File → Export to RTF.

This function is described in chapter 10.1.11 of the RFEM manual.

MS Excel

RF-CONCRETE Members allows for the direct data export to MS Excel or into the CSV format. To open the corresponding dialog box, select the RF-CONCRETE Members menu entry

File → Export Tables.

The following export dialog box opens.

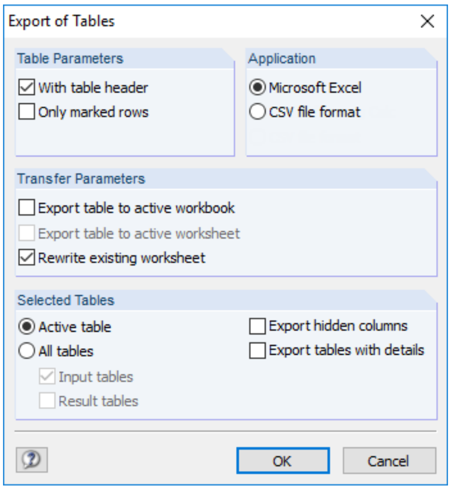


Figure 8.7 *Export of Tables* dialog box

When your selection is complete, click [OK] to start the export.

Excel will be started automatically, meaning you do not need to open the program first.

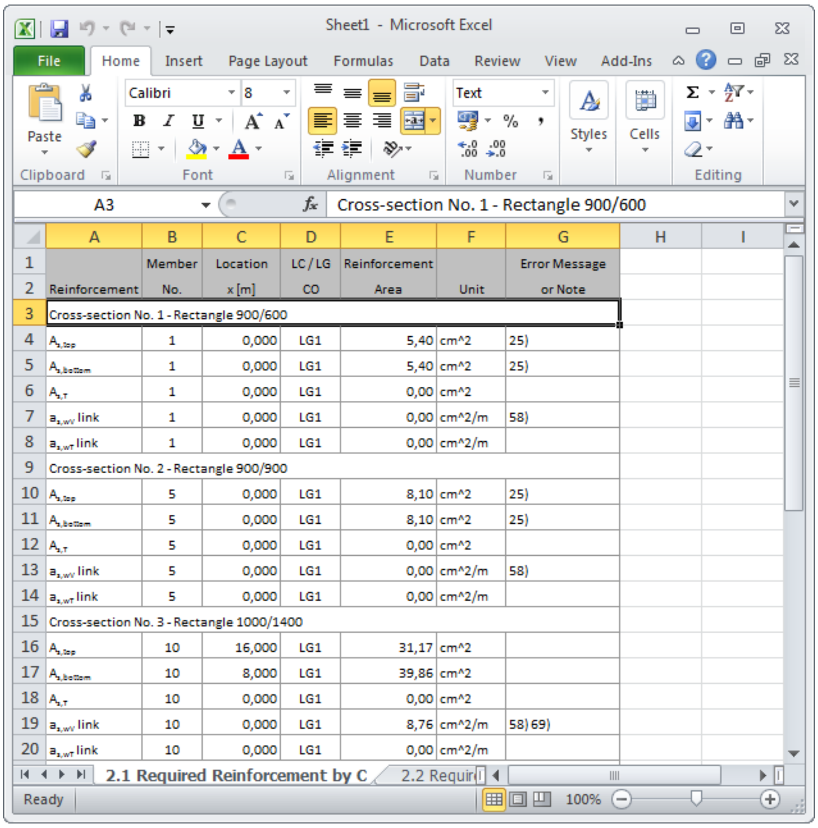


Figure 8.8 Results in Excel

9 Examples

9.1 Direct Deformation Analysis

This example describes the analysis for limitation of deformations by direct calculation according to EN 1992-1-1, clause 7.4.3.

9.1.1 Input Data

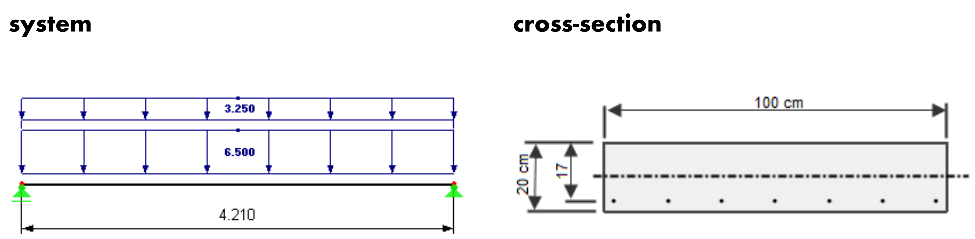


Figure 9.1 System, loads, and cross-section

Slab thickness	20 cm
Material	Concrete C20/25
	Reinforcement steel B 500
Reinforcement	$A_{s,prov}$ = 4.45 cm^2
	d = 17 cm

Actions

Self-weight	0.20 · 25.0 = 5.00 kN/m
Plaster and flooring	1.5 kN/m
	→ g_k = 6.50 kN/m
Live load for office	2.00 kN/m
Load of partition walls	1.25 kN/m
	→ q_k = 3.25 kN/m

Maximum moment for quasi-permanent load

Combination factor	ψ_2 = 0.3 (Live load for office)
Combination factor	ψ_2 = 1.0 (Load of partition walls)
Quasi-permanent load	6.50 + 0.30 · 2.00 + 1.0 · 1.25 = 8.35 kN/m
Maximum moment	$M_{quasi\text{-}permanent}$ = 8.35 · 4.21^2 / 8 = 18.50 kNm

9.1.2 Initial Values of Deformation Analysis

Parameters

Mean modulus of elasticity	E_{cm} = 30 000 MN / m^2
Mean tensile strength	f_{ctm} = 2.2 MN/m^2
Final creep ratio	φ = 1.8 (interior room)
Coefficient of shrinkage	ε_s = −0.5 ‰

Longitudinal reinforcement ratio

$$\rho_e = \frac{A_s}{b \cdot d} = \frac{4.45\ \text{cm}^2}{100 \cdot 20\ \text{cm}^2} = 0.002225$$

$$\alpha_e = \frac{E_s}{E_c} = \frac{200\,000}{10\,000} = 20.0$$

Effective modulus of elasticity for concrete

$$E_{c,\text{eff}} = \frac{E_{cm}}{1 + \varphi} = \frac{30\,000}{1 + 2.0} = 10\,000\ \text{MN}/\text{m}^2$$

The influence of creeping is taken into account with the final creep coefficient φ.

9.1.3 Curvature for Uncracked Sections (State I)

Cross-section values

Depth of concrete compression zone x_I

$$x_I = \frac{b \cdot h \cdot \frac{h}{2} + \alpha_e \cdot A_s \cdot d - A_s \cdot d}{\alpha_e \cdot A_s - A_s + b \cdot h} = \frac{100 \cdot 20 \cdot \frac{20}{2} + 20.0 \cdot 4.45 \cdot 17 - 4.45 \cdot 17}{20 \cdot 4.45 - 4.45 + 100 \cdot 20} = 10.28\,\text{cm}$$

$$\xi = x_I / h = 10.28 / 20 = 0.514$$

$$\kappa = 1 + 12 \cdot (0.5 - \xi)^2 + 12 \cdot (\alpha - 1) \cdot \rho \cdot \left(\frac{d}{h} - \xi\right)^2 =$$

$$= 1 + 12 \cdot (0.5 - 0.514)^2 + 12 \cdot (20 - 1) \cdot 0.002225 \cdot \left(\frac{17}{20} - 0.514\right)^2 = 1.06$$

$$I_{cI} = \kappa \cdot b \cdot \frac{h^3}{12} = 1.06 \cdot 100 \cdot \frac{20^3}{12} = 70\,667\,\text{cm}^4 = 0.0007067\,\text{m}^4$$

$$W_{c,I} = \frac{I_{c,I}}{h - x_I} = \frac{70\,667}{20 - 10.28} = 7\,270.2\,\text{cm}^3$$

$$S_I = A_s \cdot (d - x_I) = 4.45 \cdot (17 - 10.28) = 29.904\,\text{cm}^3 = 0.0000299\,\text{m}^3$$

Curvature due to loading

$$\left(\frac{1}{r}\right)_M = \frac{M_{Ed}}{E_{c,\text{eff}} \cdot I_c} = \frac{0.01850\,\text{MNm}}{10\,000\,\text{MN/m}^2 \cdot 0.0007067\,\text{m}^4} = 0.00262\,\text{m}^{-1}$$

Curvature due to shrinkage

$$\left(\frac{1}{r}\right)_{cs} = \varepsilon_{cs\infty} \cdot \alpha_e \cdot \frac{S_I}{I_c} = 0.0005 \cdot 20.0 \cdot \frac{0.0000299\,\text{m}^3}{0.0007067\,\text{m}^4} = 0.00042\,\text{m}^{-1}$$

Total curvature

$$\left(\frac{1}{r}\right)_{tot,I} = \left(\frac{1}{r}\right)_M + \left(\frac{1}{r}\right)_{cs} = 0.00262 + 0.00042 = 0.00304\,\text{m}^{-1}$$

9.1.4 Curvature for Cracked Sections (State II)

Curvature due to loading

When characteristic loads are applied, concrete shows linear elastic behavior. The concrete stress distributed over the compression zone is assumed to be triangular.

The depth of the concrete compression area can be determined as follows:

$$x = \rho \cdot \alpha_e \cdot d \cdot \left[-1 + \sqrt{1 + \frac{2}{\rho \cdot \alpha_e}}\right] =$$

$$= 0.0026 \cdot 20.0 \cdot 17\,\text{cm} \cdot \left[-1 + \sqrt{1 + \frac{2}{0.0026 \cdot 20.0}}\right] = 4.68\,\text{cm}$$

The tension stress in the reinforcement is determined with M_{Ed} = 18.50 kNm as follows:

$$\sigma_s = \frac{M}{A_s \cdot \left(d - \frac{x}{3}\right)} = \frac{18.5 \cdot 10^{-3}}{4.45 \cdot 10^{-4} \cdot \left(0.17 - \frac{0.0468}{3}\right)} = 269.60\,\text{N/mm}^2$$

The curvature in the final crack state is determined as follows:

$$\left(\frac{1}{r}\right)_{M,II} = \frac{\varepsilon_s}{d - x} = \frac{1.346 \cdot 10^{-3}}{170 - 46.8} = 0.010931\,\text{m}^{-1}$$

where

$$\varepsilon_s = \frac{\sigma_s}{E_s} = \frac{269.26}{200\,000} = 1.346 \cdot 10^{-3}$$

Curvature due to shrinkage

In manual calculations, the curvature for cracked sections (state II) is determined by means of a table from [12] (see Figure 9.2).

$$\omega_1 = \alpha_e \cdot \frac{A_s}{b \cdot d} = 20.0 \cdot \frac{4.45\,\text{cm}^2}{100\,\text{cm} \cdot 17\,\text{cm}} = 0.052 \quad \rightarrow \beta = 1.10$$

$$\left(\frac{1}{r}\right)_{cs,II} = \varepsilon_{cs\infty} \cdot \alpha_e \cdot \frac{S_{II}}{I_{II}} = \varepsilon_{cs\infty} \cdot \beta \cdot \frac{1}{d} = 0.0005 \cdot 1.10 \cdot \frac{1}{0.17\,\text{m}} = 0.00324\,\text{m}^{-1}$$

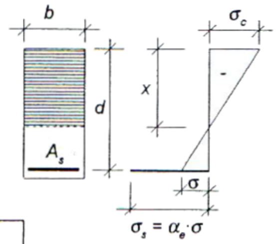
Total curvature

$$\left(\frac{1}{r}\right)_{tot,II} = \left(\frac{1}{r}\right)_{M,II} = \left(\frac{1}{r}\right)_{cs,II} = 0.01093 + 0.00324 = 0.01417\,\text{m}^{-1}$$

Serviceability Limit State

Calculation table for stresses, curvature and values for cracked sections only (state II) of a rectangular cross-section

EC2, 4.4.1.2 : $\alpha_e = E_s / E_c = 15$
Reinforcing steel: $E_s = 2 \cdot 10^5$ MN/m²



ω_1	$\xi = k_x$	$\zeta = k_z$	$2/(\xi \cdot \zeta)$	$2/(\xi^2 \cdot \zeta)$	β
0,001	0,044	0,985	46,41	1061	1,01
0,002	0,061	0,980	33,32	544	1,02
0,003	0,075	0,975	27,52	369	1,03
0,004	0,086	0,971	24,07	281	1,03
0,005	0,095	0,968	21,71	228	1,03
0,006	0,104	0,965	19,98	193	1,04
0,007	0,112	0,963	18,63	167	1,04
0,008	0,119	0,960	17,54	148	1,04
0,009	0,125	0,958	16,64	133	1,04
0,010	0,132	0,956	15,87	120	1,05
0,012	0,143	0,952	14,65	102	1,05
0,014	0,154	0,949	13,70	89,0	1,05
0,016	0,164	0,945	12,93	79,0	1,06
0,018	0,173	0,942	12,30	71,2	1,06
0,020	0,181	0,940	11,76	65,0	1,06
0,022	0,189	0,937	11,30	59,8	1,07
0,024	0,196	0,935	10,90	55,5	1,07
0,026	0,204	0,932	10,54	51,8	1,07
0,028	0,210	0,930	10,23	48,6	1,08
0,030	0,217	0,928	9,94	45,9	1,08
0,040	0,246	0,918	8,87	36,1	1,09
0,050	0,270	0,910	8,14	30,1	1,10
0,060	0,292	0,903	7,60	26,1	1,11
0,070	0,311	0,896	7,18	23,1	1,12
0,080	0,328	0,891	6,85	20,9	1,12
0,090	0,344	0,885	6,57	19,1	1,13
0,100	0,358	0,881	6,34	17,7	1,14

Table input value:

$$\omega_1 = \alpha_e \cdot \frac{A_s}{b \cdot d}$$

Ratio of elastic moduli:

$$\alpha_e = \frac{E_s}{E_c}$$

Steel stress:

$$\sigma_s = \frac{1}{\zeta} \cdot \frac{M}{A_s \cdot d}$$

Concrete stress:

$$\sigma_c = \frac{2}{\xi \cdot \zeta} \cdot \frac{-M}{b \cdot d^2}$$

Curvature:

$$\kappa = \frac{1}{r} = \frac{2}{\xi^2 \cdot \zeta} \cdot \frac{\alpha_e \cdot M}{E_s \cdot b \cdot d^3}$$

Formula A 4.4 from EC2:

$$\alpha_e \cdot \left(\frac{S}{J}\right)_{II} = \beta \cdot \frac{1}{d}$$

Figure 9.2 Calculation table for cracked sections only (state II) from [12]

9.1.5 Determination of Deflection

As described in Chapter 2.2.5, it is possible to determine the probable value of the deformation according to equation (7.18) of EN 1992-1-1.

Distribution coefficient

The distribution coefficient ζ between state I and state II is determined as follows:

$$\zeta = 1 - \beta_1 \cdot \beta_2 \cdot \left(\frac{\sigma_{s,cr}}{\sigma_s}\right)^2 = 1 - 1.0 \cdot 0.5 \cdot \left(\frac{232.87}{269.26}\right)^2 = 0.63$$

where

$\beta_1 = 1.0$: ribbed steel

$\beta_2 = 0.5$: permanent load

The first cracking moment M_{cr} is:

$$M_{cr} = f_{ctm} \cdot W_1 = 2.2 \cdot 0.007270 \cdot 10^3 = 16.0 \text{ kNm}$$

The stress $\sigma_{s,cr}$ immediately after cracking is determined with M_{cr} as follows:

$$\sigma_{s,cr} = \frac{M_{cr}}{A_s \cdot \left(d - \frac{x}{3}\right)} = \frac{16.0 \cdot 10^{-3}}{4.45 \cdot 10^{-4} \cdot \left(0.17 - \frac{0.0468}{3}\right)} = 232.87 \text{ N/mm}^2$$

Mean curvature

With the distribution coefficient ζ, the mean curvature is determined as follows:

$$\frac{1}{r_m} = \zeta \cdot \frac{1}{r_{II}} + (1 - \zeta) \cdot \frac{1}{r_1} = 0.63 \cdot 0.01417 + (1 - 0.63) \cdot 0.00304 = 0.01005\,\mathrm{m}^{-1}$$

Deformation

Thus, the deflection *f* in the beam center can be determined as follows:

$$f = k \cdot l_{\mathrm{eff}}^2 \cdot \frac{1}{r_m} = \frac{5}{48} \cdot 4.21^2\,\mathrm{m}^2 \cdot 0.01005\,\mathrm{m}^{-1} = 18.6\,\mathrm{mm}$$

9.1.6 Results in RF-CONCRETE Members

Some parameters must be defined for the comparative calculation with RF-CONCRETE Members.

In Window *1.1 General Data*, the governing load case must be selected for design in the *Ultimate Limit State* tab and the *Serviceability Limit State* tab. For the SLS design, the **Activate creep and shrinkage** check box must be selected as well.

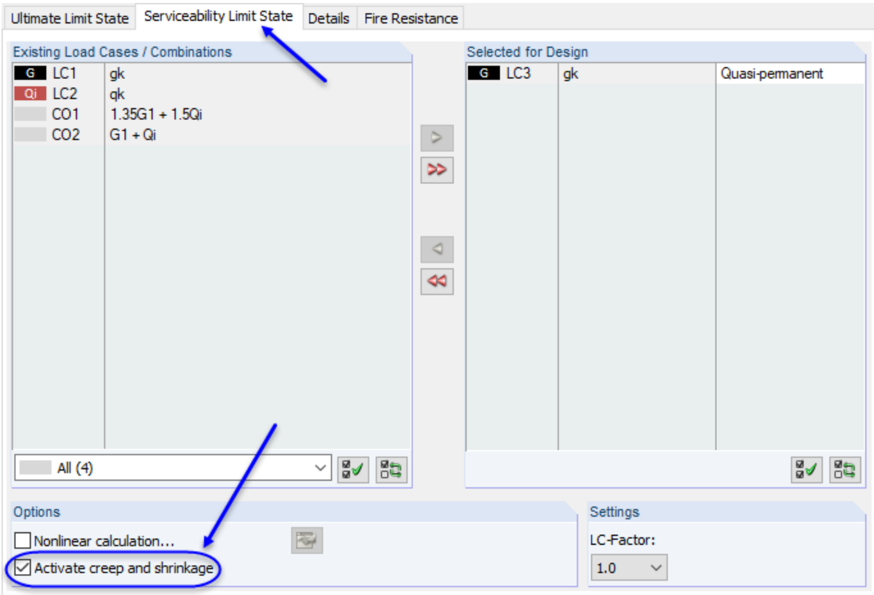


Figure 9.3 Settings for serviceability limit state design in Window *1.1 General Data*

Then, you can directly define the creep coefficient and the shrinkage strain in Window *1.3 Cross-Sections*.

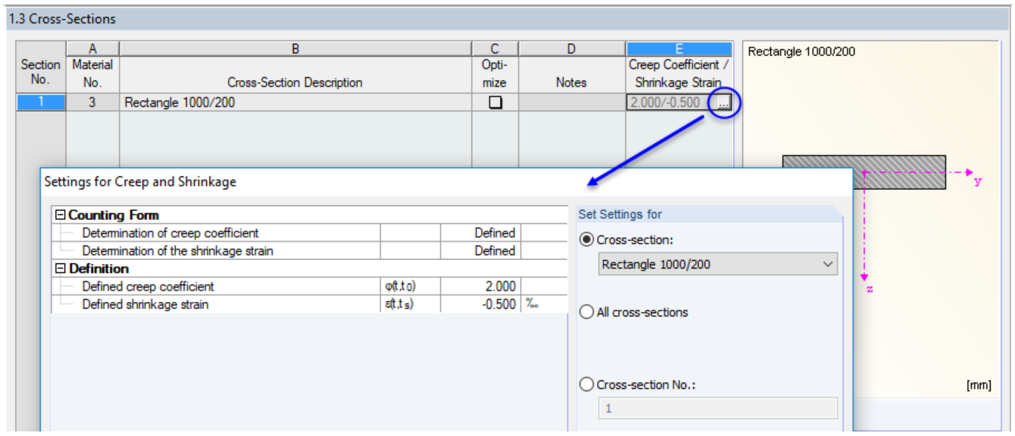


Figure 9.4 Defining creep coefficient and shrinkage strain in Window *1.3 Cross-Sections*

In Window *1.6 Reinforcement*, the reinforcement diameters and concrete cover must be specified as follows.

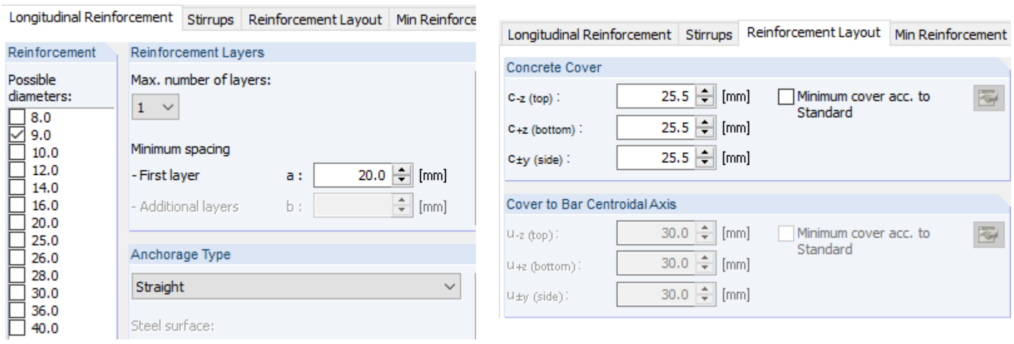
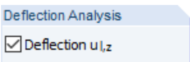


Figure 9.5 Defining reinforcement diameters and concrete cover in Window *1.6 Reinforcement*



Window 1.6, *Serviceability* tab

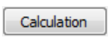
If the **Deflection Analysis** is selected (see figure on the left) in the *Serviceability* tab of Window *1.6 Reinforcement*, the additional Window *1.7 Deflection Data* appears.

1.7 Deflection Data

	A	B	C	D	E	F	G
No.	Reference to	Member No.	Reference Length Element	L [m]	Precamber w 0 [mm]	Limit value L / [-]	u z,max [mm]
1	Member	1	Distance between supports	4.210	0.0	250	16.8
2							
3							

Figure 9.6 Defining deformation parameters in Window *1.7 Deflection Data*

The recommended deflection of $\ell/250$ is the default limit value.



The [Calculation] provides the maximum deformation value of **18.9 mm** in the center of the member, confirming the manual calculation. The result is below the limit value of $\ell/250$.

4.3 Serviceability Check by Member

A	B	C	D	E	F	G	H
Member No.	Location x [m]	Loading	Type	Existing Value	Design Limit Value	Unit	Capacity
Member No. 1 - Rectangle 1000/200							
1	2.105	LC3	u z,local	18.9	16.8	mm	1.11

Figure 9.7 Window *4.3 Serviceability Check by Member*

The following figure compares the deformations for state I (uncracked sections) and state II (cracked sections). The crack formation leads to a significant increase in the deflection.

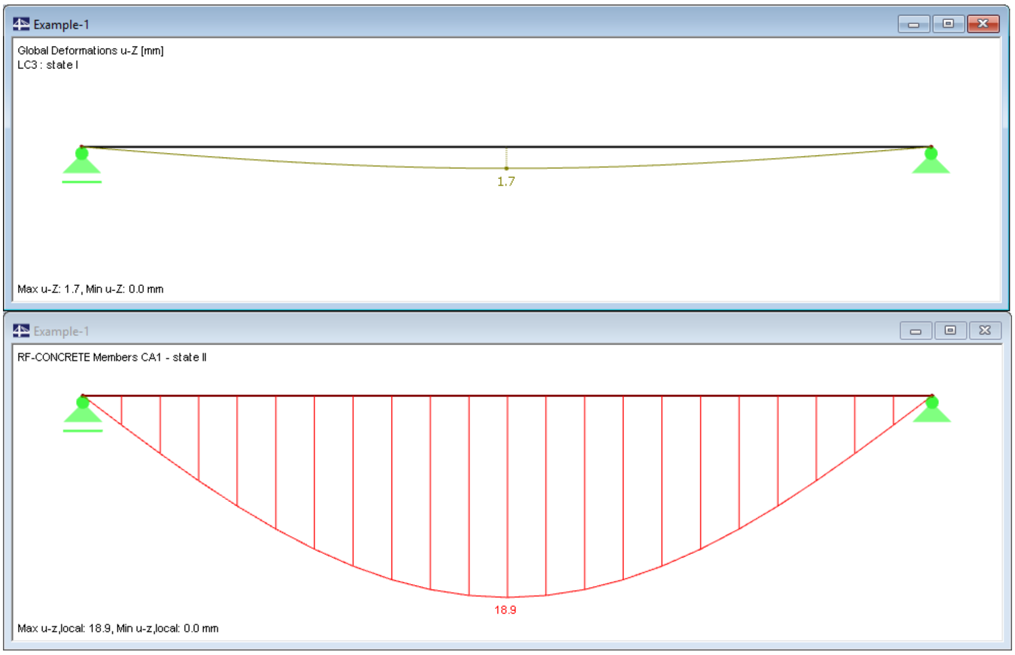


Figure 9.8 Deformations in state I (uncracked sections) and state II (cracked sections)

9.2 Nonlinear Deformation Analysis

The second example presents the basic principles of a nonlinear calculation for the serviceability limit state.

A comparative calculation is carried out for a two-span system. We look more closely at the approach used to limit the degree of bending slenderness and the applied tensile strength.

9.2.1 Input Data

System and loading

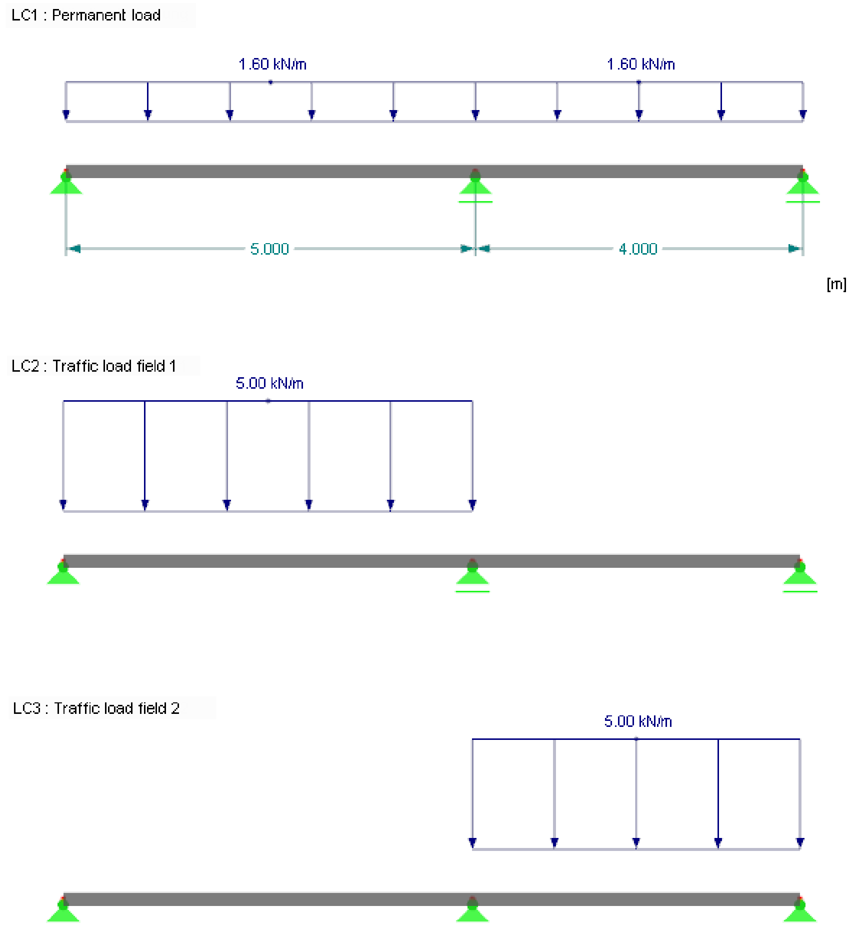


Figure 9.9 System and loading

Slab thickness	16 cm
Material	Concrete C20/25 Reinforcement steel B 500 S (A)
Exposure class	XC1
Concrete cover	c_{nom} = 22 mm

Combination for ultimate limit state

By selecting the corresponding *General Data* option, RFEM automatically creates a result combination that is used for the ultimate limit state designs. This result combination forms the envelope of the following load combinations:

CO1: $1.35 \cdot LC1 + 1.5 \cdot LC2 + 1.5 \cdot LC3$

CO2: $1.35 \cdot LC1 + 1.5 \cdot LC2$

CO3: $1.35 \cdot LC1 + 1.5 \cdot LC3$

Combination for serviceability limit state

The combination factor $\psi_{2,1}$ for the quasi-permanent action combination is applied with 0.6. As a superposition of load case results (RC) is excluded for nonlinear calculations, three load combinations with the following combination factors are used for the design in the serviceability limit state:

CO4: $LC1 + 0.6 \cdot LC2 + 0.6 \cdot LC3$

CO5: $LC1 + 0.6 \cdot LC2$

CO6: $LC1 + 0.6 \cdot LC3$

9.2.2 Input in RF-CONCRETE Members

To determine the reinforcement, only the ultimate limit state is considered. In Window *1.1 General Data*, select RC1 for the design.

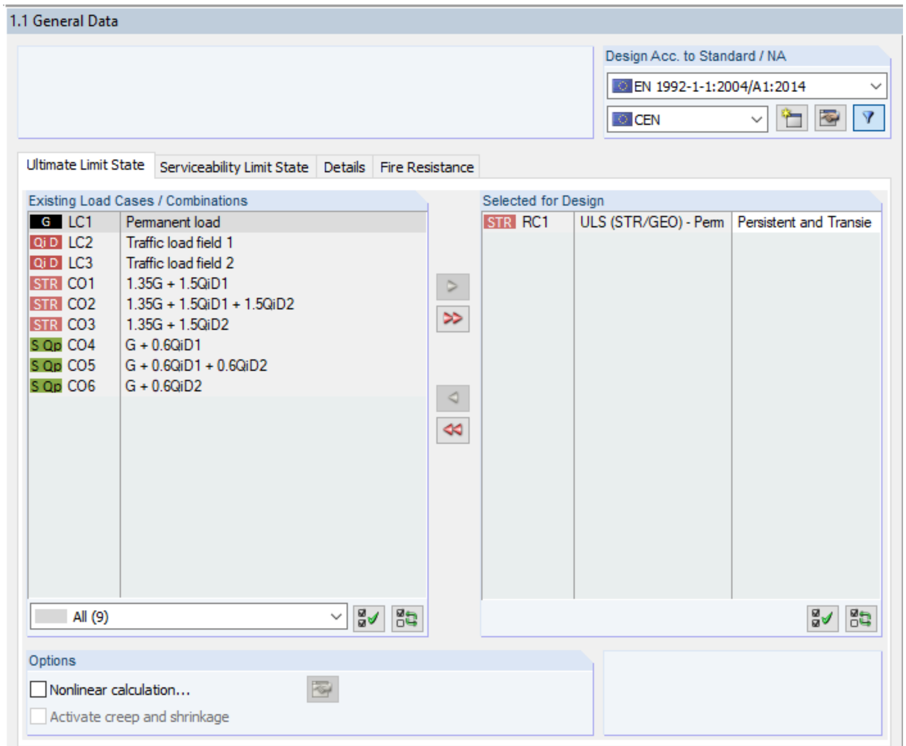


Figure 9.10 Window *1.1 General Data, Ultimate Limit State* tab

In Window *1.2 Materials*, the materials *Concrete C20/25* and *B 500 S (A)* are preset.

Window *1.3 Cross-Sections* is described elsewhere when creeping is considered (see Figure 9.18).

Because the calculation is performed by considering moment swap and moment reduction, we have to specify some settings in Window *1.5 Supports*.

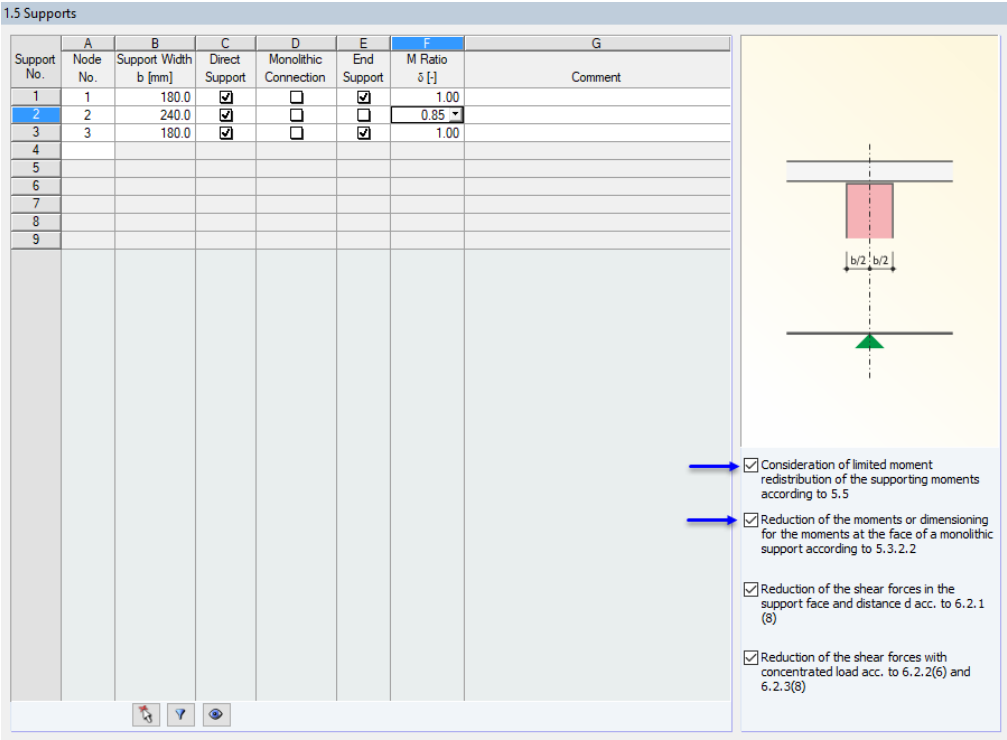


Figure 9.11 Window *1.5 Supports*

In column B, we enter the support widths shown in the figure above. Then, in table column F, we reduce the maximum column moment for the intermediate support to **85 %** of the linear elastic value. To enable the input field, select the check boxes below the graphic on the right so that the program will take the specifications for the design into account.

Showing the provided reinforcement is of major importance for the nonlinear calculation because it is a decisive influencing value when determining curvatures in the cracked state. In Window 1.6, we create an additional reinforcement diameter of **6 mm** (see Figure 9.12).

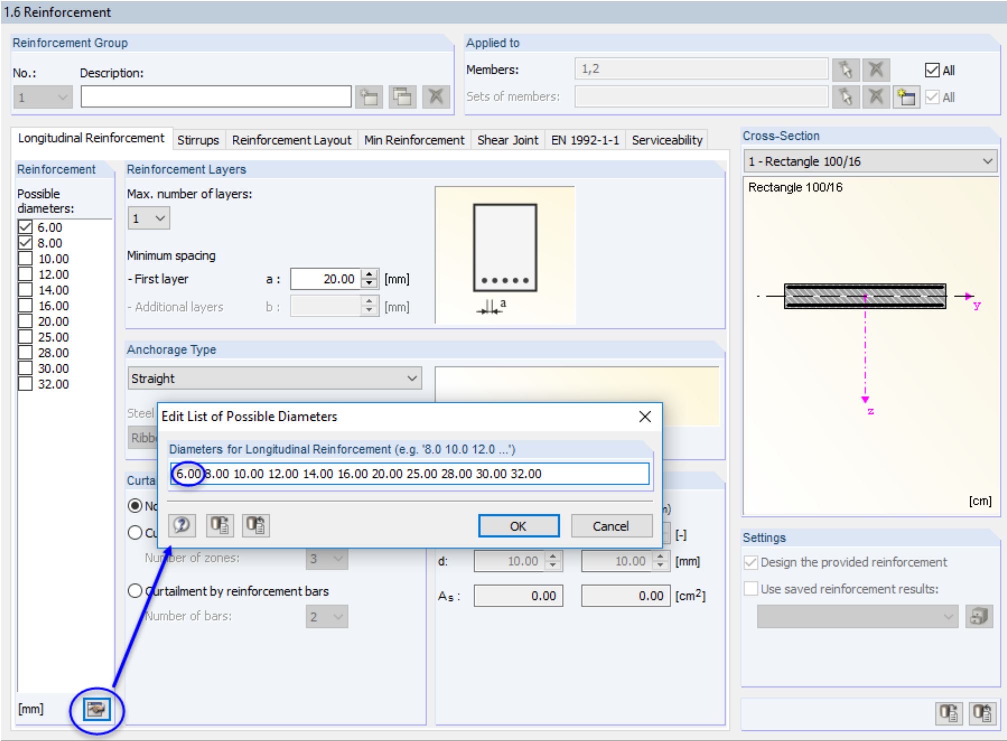


Figure 9.12 Window *1.6 Reinforcement, Longitudinal Reinforcement* tab

In the *Reinforcement Layout* tab of Window *1.6 Reinforcement*, we specify the concrete covers with **22 mm**. Thus, the bar centroidal axis applied for the design is 25 mm.

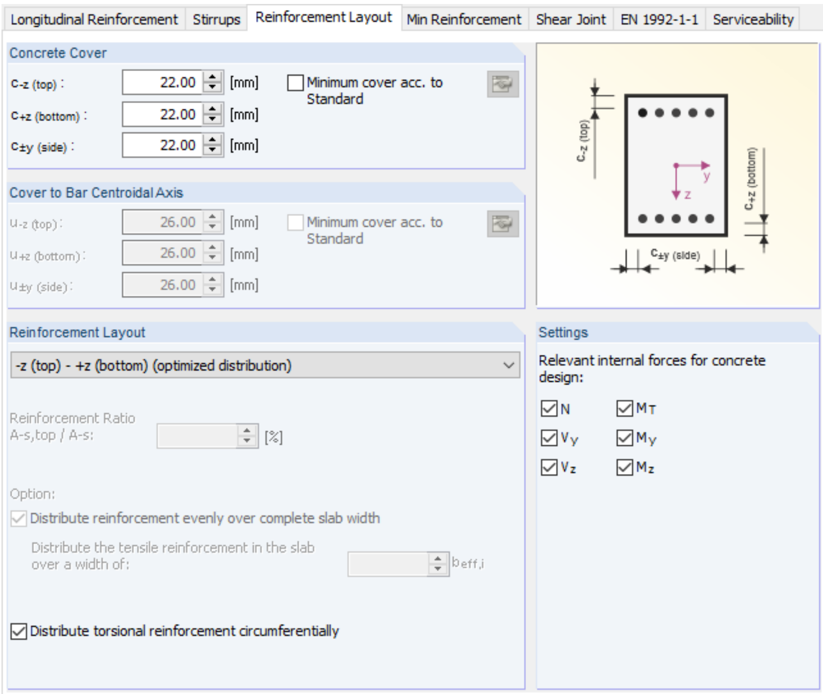
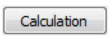


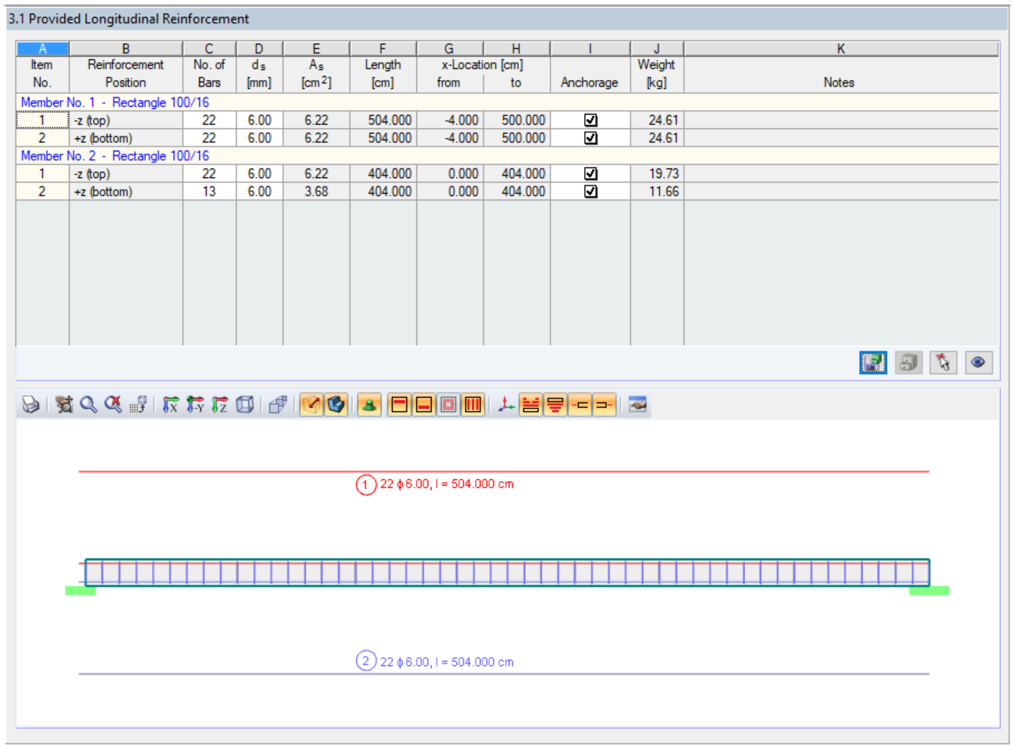
Figure 9.13 Window *1.6 Reinforcement, Reinforcement Layout* tab



The input is thus complete and the [Calculation] can be started.

9.2.3 Checking the Reinforcement

The longitudinal reinforcement given in result window 3.1 is the basis for the nonlinear calculation.



Item No.	Reinforcement Position	No. of Bars	d_s [mm]	A_s [cm²]	Length [cm]	x-Location [cm] from	x-Location [cm] to	Anchorage	Weight [kg]	Notes
Member No. 1 - Rectangle 100/16										
1	-z (top)	22	6.00	6.22	504.000	-4.000	500.000	☑	24.61	
2	+z (bottom)	22	6.00	6.22	504.000	-4.000	500.000	☑	24.61	
Member No. 2 - Rectangle 100/16										
1	-z (top)	22	6.00	6.22	404.000	0.000	404.000	☑	19.73	
2	+z (bottom)	13	6.00	3.68	404.000	0.000	404.000	☑	11.66	

Figure 9.14 Window *3.1 Provided Longitudinal Reinforcement*

9.2.4 Specifications for Nonlinear Calculation

The nonlinear calculation for the serviceability limit state is prepared in the *Serviceability Limit State* tab of Window *1.1 General Data*.

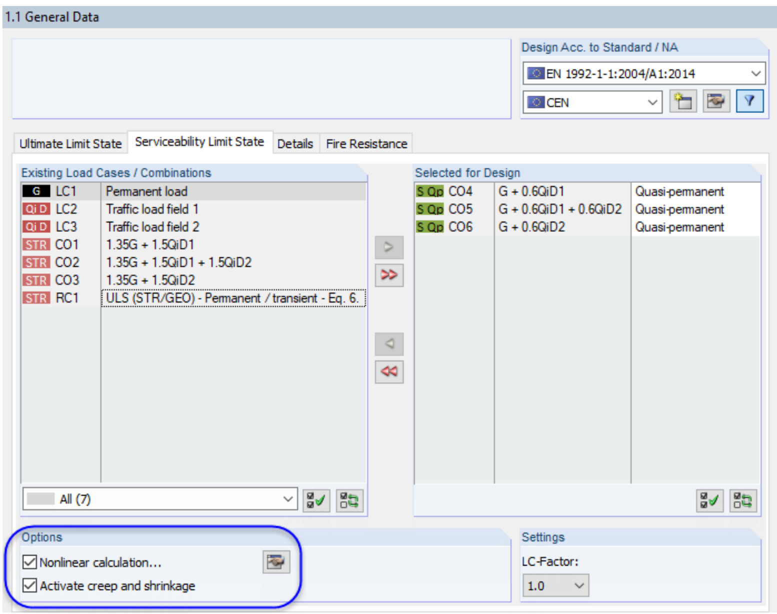


Figure 9.15 Window *1.1 General Data, Serviceability Limit State* tab

For the design, we select load combinations **CO4 to CO6** defined for the serviceability limit state design. In addition, we select the *Nonlinear calculation* and *Activate creep and shrinkage* check boxes.

The [Settings for Nonlinear Calculation] button opens the dialog box of the same name. The default settings of the *Analysis Method* tab remain unchanged. In the *Tension Stiffening Effect* tab, we specify the following settings.

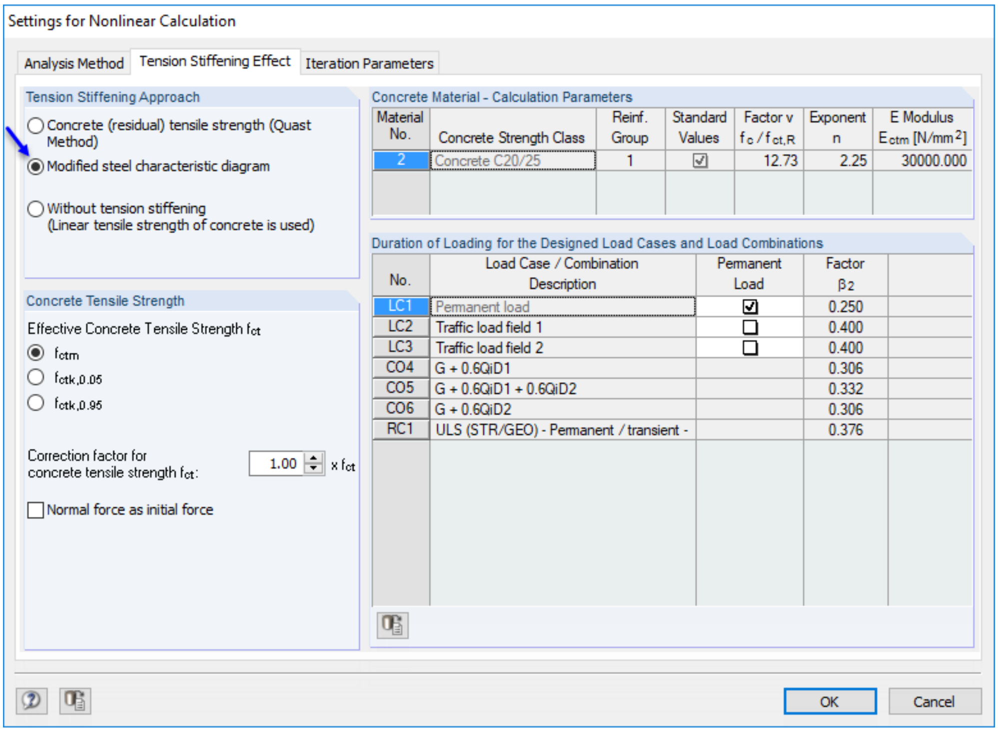


Figure 9.16 *Settings for Nonlinear Calculation dialog box, Tension Stiffening Effect tab*

We select the approach of the *Modified steel characteristic diagram*. For now we leave the adjustment factor of the tensile strength as 1.00 · f_{ct}. The calculation is performed with the mean axial tensile strength of concrete specified in EN 1992-1-1, table 3.1.

The load duration factor β_t is calculated depending on the load cases of the respective load combinations between the limit values 0.25 and 0.4. Specifications for the axial force are of no relevance for pure bending.

We can keep the default values in the *Iteration Parameters* tab. But we have to adjust the FE division of members with the button.

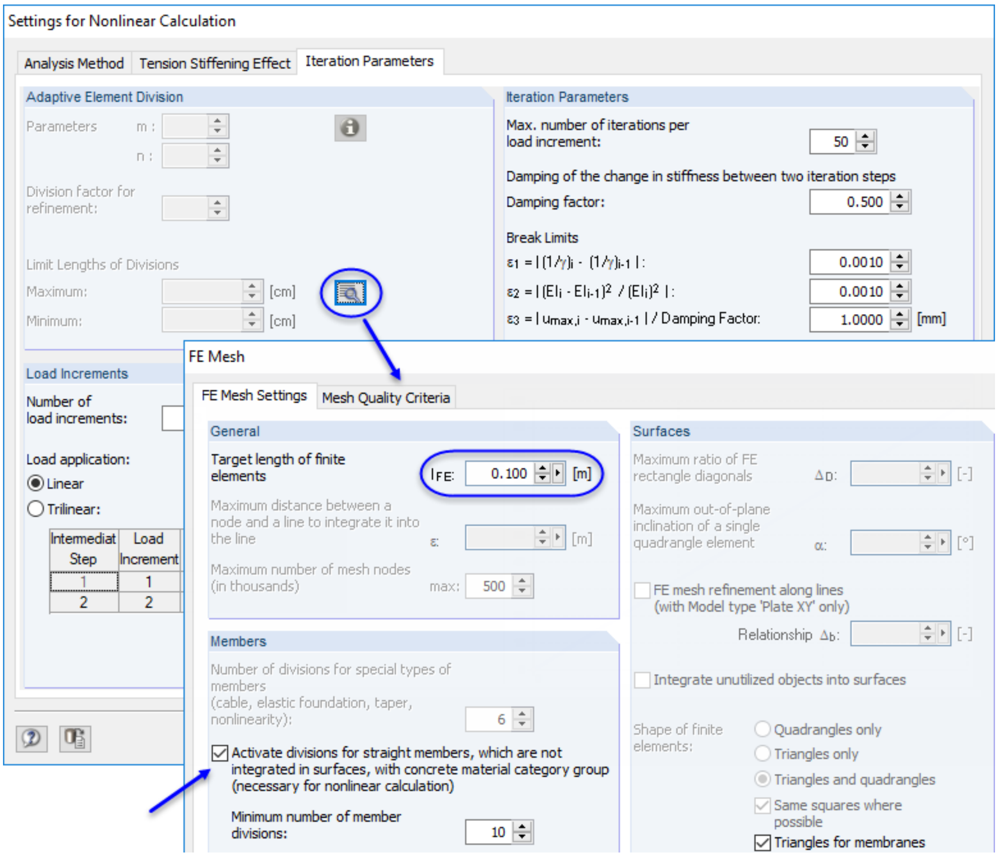


Figure 9.17 *Settings for Nonlinear Calculation and FE Mesh dialog boxes*

Finally, we define the settings for creep and shrinkage in Window *1.3 Cross-Sections*.

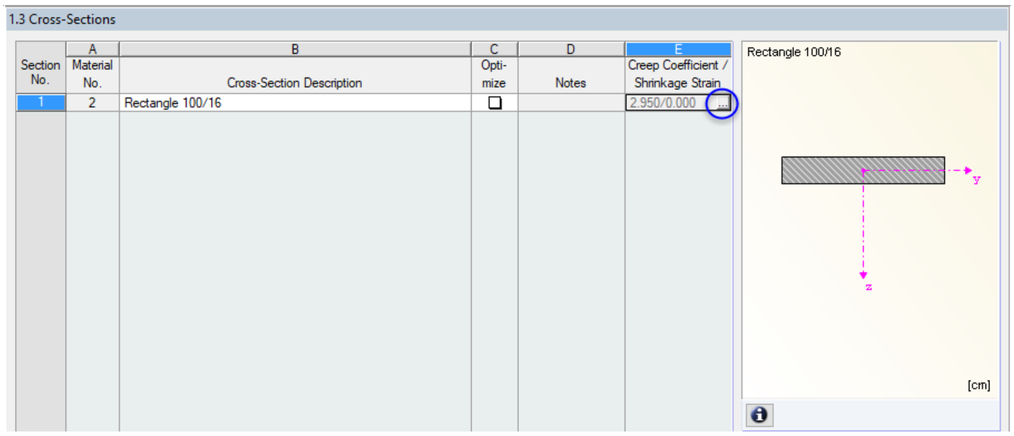


Figure 9.18 *Window 1.3 Cross-Sections*

The table cell button in column E opens the dialog box for entering the creep and shrinkage parameters.

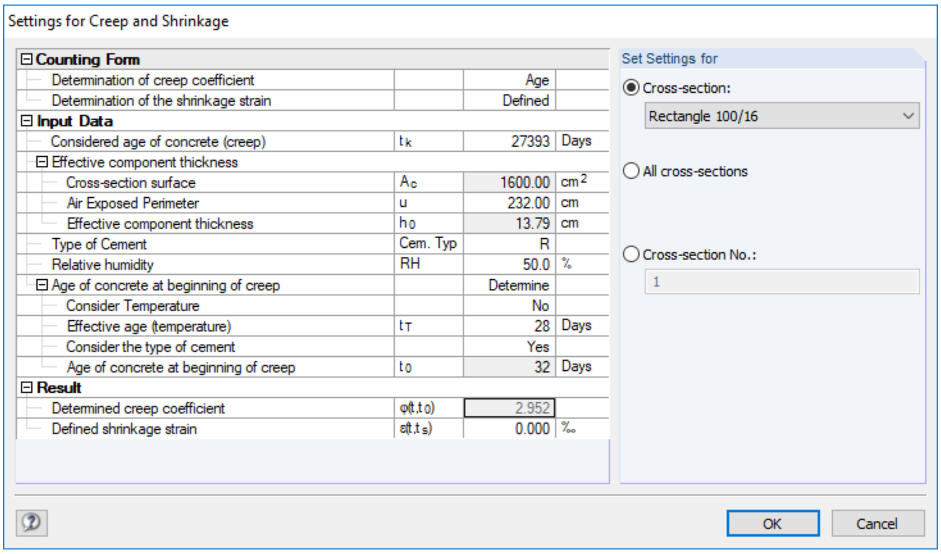
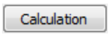


Figure 9.19 *Settings for Creep and Shrinkage* dialog box

We change the *Type of cement* to **R** and the *Effective age* to **28 days**. The creep coefficient according to EN 1992-1-1 shall be understood as the pure final creep. It usually must still be converted into an effective creep coefficient in accordance with the ratio of creep-producing load to effective load.

The shrinkage is not analyzed further: Due to the almost symmetrical reinforcement in span 1 and the minor reinforcement difference in span 2, the shrinkage curvatures do not significantly contribute to the total deformation. Therefore, we *Determine* the shrinkage strain as **0**.



The input for the nonlinear calculation is now complete. We can start the [Calculation] again.

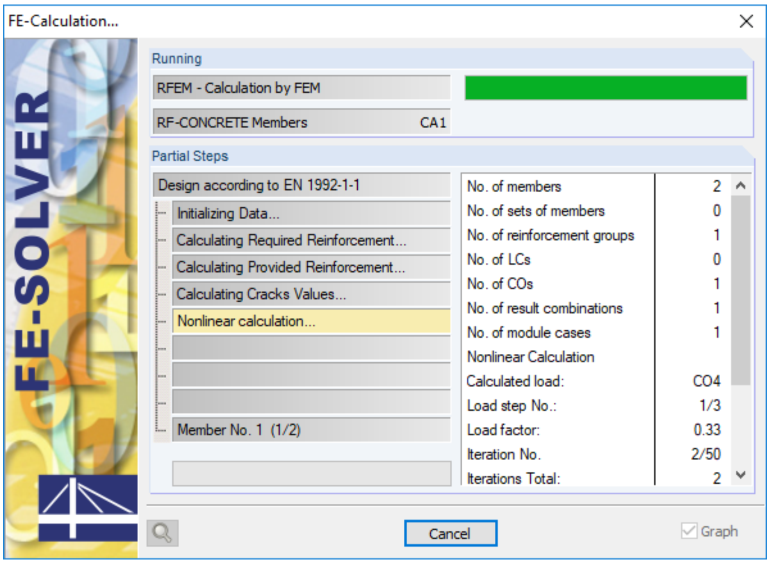


Figure 9.20 Nonlinear calculation

9.2.5 Results of RF-CONCRETE Members

In the *6.2.4 Serviceability Limit State for Nonlinear Calculation by x-Location* window, we can display the maximum deformations and crack widths on each x-location.

6.2.4 Serviceability Limit State for Nonlinear Calculation by x-Location

A Member No.	B Location x [m]	C Loading	D Type	E Existing Value	F Unit	G Note
1	2.200	CO4	u_g	23.94	mm	
		CO5	u_g	21.60	mm	
		CO6	u_g	6.52	mm	
		CO4	w_k	0.11	mm	
		CO5	w_k	0.10	mm	
		CO6	w_k	0.06	mm	
	2.300	CO4	u_g	23.89	mm	
		CO5	u_g	21.51	mm	
		CO6	u_g	6.49	mm	
		CO4	w_k	0.11	mm	
		CO5	w_k	0.09	mm	
		CO6	w_k	0.06	mm	
	2.400	CO4	u_g	23.71	mm	

☑ All loads cases

Figure 9.21 Window 6.2.4 *Serviceability Limit State for Nonlinear Calculation by x-Location*

The maximum deformation occurs at the location **x = 2.200 m** for CO4 (traffic load in span 1). The displacement u = 23.94 mm corresponds to a value of 1/209 · ℓ and is therefore larger than the recommended value of 1/250 · ℓ

⊟ Stress and Strain Analysis			
Principal Moments	M	17.66	kNm
Bending Moment About the y-Axis	M_{yEd}	17.66	kNm
Bending Moment About the z-Axis	M_{zEd}	0.00	kNm
Normal Force	N_{Ed}	0.00	kN
State of Cross-Section		Crack Formation Completed	
Mean Main Curvature	$1/r_m$	1.3095E-02	1/m
Mean Curvature in y-Direction	$1/r_{ym}$	5.2565E-16	1/m
Mean Curvature in z-Direction	$1/r_{zm}$	1.3095E-02	1/m
Average Strain in Reinforcement	ε_{sm}	1.021	‰
⊟ Design Details			
Coefficient for Consideration of Bond Characteristic	β_1	0.500	
Coefficient for Consideration of Load Type and Dur	β_2	0.306	
Crack Moment	Mcr	12.01	kNm
Crack Moment Relative to the y-Axis	Mcry	12.01	kNm
Crack Moment Relative to the z-Axis	Mcrz	0.00	kNm
Crack Normal Force	Ncr		kN
⊟ State I (uncracked)			
⊟ Cross-Section Properties			
Moment of Inertia About the y-Axis	Iy_I	1.10503E+08	mm^4
Moment of Inertia About the z-Axis	Iz_I	4.03092E+09	mm^4
Surface	A_I	48463.20	mm^2
⊟ Stress and Strain Analysis			
Main Curvature	$1/r_I$	5.3267E-03	1/m
Curvature in y-Direction	$1/r_{yI}$	5.2565E-16	1/m
Curvature in z-Direction	$1/r_{zI}$	5.3267E-03	1/m
Direction of the Principal Curvature	α_I	0.0	°
Steel Stress in Governing Fiber	σs_I	58.59	N/mm^2
Concrete Stress in Governing Fiber	σc_I	-3.23	N/mm^2
Steel Stress for Crack Moment	σsr_I	39.51	N/mm^2
Steel Strain for the Crack Moment	εsr_I	0.198	‰
⊟ State II (cracked)			
⊟ Cross-Section Properties			
Moment of Inertia About the y-Axis	Iy_{II}	42930500.00	mm^4
Moment of Inertia About the z-Axis	Iz_{II}	1.33333E+10	mm^4
Surface	A_{II}	20123.70	mm^2
⊟ Stress and Strain Analysis			
Main Curvature	$1/r_{II}$	1.3711E-02	1/m
Curvature in y-Direction	$1/r_{yII}$	-1.0707E-20	1/m
Curvature in z-Direction	$1/r_{zII}$	1.3711E-02	1/m
Direction of the Principal Curvature	α_{II}	0.0	°
Steel Stress in Governing Fiber	σs_{II}	242.51	N/mm^2
Steel strain	εs_{II}	1.213	‰
Concrete Stress in Governing Fiber	σc_{II}	-4.81	N/mm^2
Concrete Strain in Governing Fiber	εc_{II}	-0.638	‰
Steel Stress for Crack Moment	σsr_{II}	164.80	N/mm^2
Steel Strain for the Crack Moment	εsr_{II}	0.824	‰
⊞ Cross-Section Properties			
⊞ Cross-Sectional Rigidities (Secant Values)			
⊞ Global Nodal Deformations			
⊞ Local Nodal Deformations			
⊞ Crack Width			
⊞ Verification of Safety			
Partial Factor	γ	2.439	
Percentage of Utilization	$1/\gamma$	0.410	

Figure 9.22 Detailed results for member 1 at location x = 2.200 m, CO4

9.2.6 Manual Calculation

Now we check the results for the location x = 2.20 m step by step. Of primary interest is the calculation of the strain and stress level, which is decisive for the nonlinear determination of deformations and internal forces as a basis for the stiffness properties.

For manual calculations, we use simplified approaches to some extend, which lead to minor differences.

9.2.6.1 Material Properties for Deformation Analysis

Concrete C 20/25

$f_c = f_{cm} = 20 + 8 = 28\ \text{N/mm}^2$

$E_c = E_{cm} = 30000\ \text{N/mm}^2$

$\varepsilon_{c1} = -2.0\ ‰$

$\varepsilon_{c1u} = -3.5\ ‰$

Distorted for creep with $(1 + \varphi) = 3.95$:

$E_c = E_{cm} = 7594.9\ \text{N/mm}^2$

$\varepsilon_{2c1} = -7.9\ ‰$

$\varepsilon_{c1u} = -13.8\ ‰$

Reinforcement steel B 500 S (A)

$f_{ym} = f_{yk} = 500 = 500.00\ \text{N/mm}^2$

$f_{tm} = f_{tk} = 550 = 550.00\ \text{N/mm}^2$

$E_s = 200000\ \text{N/mm}^2$

$\varepsilon_{su} = 25\ ‰$

$\alpha_e = 200{,}000 / 30{,}000 = 6.67$

Distorted for creep with $(1 + \varphi) = 3.95$:

$\alpha_e = 200000 / 7594.9 = 26.33$

9.2.6.2 State I (uncracked)

When determining the cross-section properties, we take the available steel area into account. The missing area of concrete in the zone of rebars is neglected. Recalculating the centroid of the ideal cross-section is not necessary because the reinforcement is symmetric with the same edge distances on top and bottom side.

The following distances for the Steiner component (parallel axis theorem) are the direct result:

$a_c = 0\ \text{cm}$

$a_{s1} = 8 - 2.5 = 5.5\ \text{cm}$

$a_{s2} = 5.5\ \text{cm}$

Moment of inertia

$$I_{y,I} = \frac{b \cdot h^3}{12} + 2 \cdot \left(A_{s1/s2} \cdot a_2^2 \cdot \alpha_e\right) = \frac{100 \cdot 16^3}{12} + 2 \cdot (6.22 \cdot 5.5^2 \cdot 26.33) = 44\,041 \text{ cm}^4$$

Ideal cross-section area

$$A_I = A_c + A_s \cdot \alpha_e = 16 \cdot 100 + 12.44 \cdot 26.33 = 1927.5 \text{ cm}^2$$

Crack moment M_{cr}

We assume that the cross-section cracks when the tensile strength f_{ctm} in the most external fiber is reached.

$$\sigma = \frac{M_{cr}}{I} \cdot z_{ct} = f_{ctm}$$

$$M_{cr} = \frac{f_{ctm} \cdot I}{z_{ct}} = \frac{0.22 \cdot 44\,041}{8} = 1\,211 \text{ kNcm} = 12.1 \text{ kNm}$$

Steel stress σ_{srI} and steel strain ε_{srI} for crack moment

$$\sigma_{sr1,I} = f_{ctm} \cdot \frac{5.5}{8} \cdot \alpha_e = 2.2 \cdot \frac{5.5}{8} \cdot 26.33 = 39.82 \text{ N/mm}^2$$

$$\varepsilon_{sr1,I} = \frac{\sigma_{sr}}{E_s} = \frac{39.82}{200\,000} = 1.991 = 0.199 \text{ ‰}$$

Notional steel and concrete stress for effective moment M = 17.64 kNm

$$\sigma_{s1} = \frac{M}{I} \cdot z_{s1} \cdot \alpha_e = \frac{1764}{44041} \cdot 5.5 \cdot 26.33 = 5.77 \text{ kN/cm}^2 = 57.7 \text{ N/mm}^2$$

$$\sigma_c = -\frac{M}{I} \cdot z_{cc} = -\frac{1764}{44041} \cdot 8 = -0.32 \text{ kN/cm}^2 = -3.2 \text{ N/mm}^2$$

Curvature for uncracked section (state I) $(1/r)_{z,I} = (1/r)_I$

$$\left(\frac{1}{r}\right)_{z,I} = \frac{M}{E \cdot I} = \frac{0.01764}{7594.9 \cdot 4.4041 \cdot 10^{-4}} = 5.283 \cdot 10^{-3} \text{ m}^{-1}$$

Results of RF-CONCRETE Members

⊟ Design Details			
Coefficient for Consideration of Bond Characteristic	β_1	0.500	
Coefficient for Consideration of Load Type and Dur	β_2	0.306	
Crack Moment	Mcr	12.01	kNm
Crack Moment Relative to the y-Axis	Mcry	12.01	kNm
Crack Moment Relative to the z-Axis	Mcrz	0.00	kNm
Crack Normal Force	Ncr		kN
⊟ State I (uncracked)			
⊟ Cross-Section Properties			
Moment of Inertia About the y-Axis	$I_{y\,I}$	1.10503E+08	mm^4
Moment of Inertia About the z-Axis	$I_{z\,I}$	4.03092E+09	mm^4
Surface	A_I	48463.20	mm^2
⊟ Stress and Strain Analysis			
Main Curvature	$1/r_I$	5.3267E-03	1/m
Curvature in y-Direction	$1/r_{yI}$	5.2565E-16	1/m
Curvature in z-Direction	$1/r_{zI}$	5.3267E-03	1/m
Direction of the Principal Curvature	α_I	0.0	°
Steel Stress in Governing Fiber	$\sigma_{s\,I}$	58.59	N/mm^2
Concrete Stress in Governing Fiber	$\sigma_{c\,I}$	-3.23	N/mm^2
Steel Stress for Crack Moment	$\sigma_{sr\,I}$	39.51	N/mm^2
Steel Strain for the Crack Moment	$\varepsilon_{sr\,I}$	0.198	‰

Figure 9.23 Detailed results for state I

9.2.6.3 State II (cracked)

Cross-section properties in cracked state (state II)

In contrast to the cross-section properties in the uncracked state (state I), the cross-section properties in state II (cracked sections) are quite difficult to determine manually. Determining the strain distribution (general case: $\varepsilon_0 + (1/r)_y \cdot y + (1/r)_z \cdot z$) for a particular action constellation with the stress-strain relations defined in the standards for nonlinear methods already represents a problem. For further information, refer to the corresponding literature [7].

Steel stress σ_{srII} and steel strain ε_{srII} for crack moment

To determine the stresses and strains for crack formation, we can normally make simplified assumptions (linear elastic material rules). We can justify this approach by the fact that the ratio of stress to strain for concrete behaves nearly linearly up to a stress of $\sigma_c \cong 0.4 \cdot f_c$. For reinforcing steel, we can roughly assume this fact until yielding is reached anyway. Thus, if we have a structural component with a crack moment in the characteristic load level, we can calculate stresses and strains with sufficient accuracy using these simplified approaches.

Without an acting axial force, the solution for a triangular compression zone leads to a quadratic equation (with axial force: cubic equation) when calculating the neutral axis depth x (height of compression zone). Due to the assumed linearity of stresses and strains, the neutral axis depth is decoupled from the applied moment.

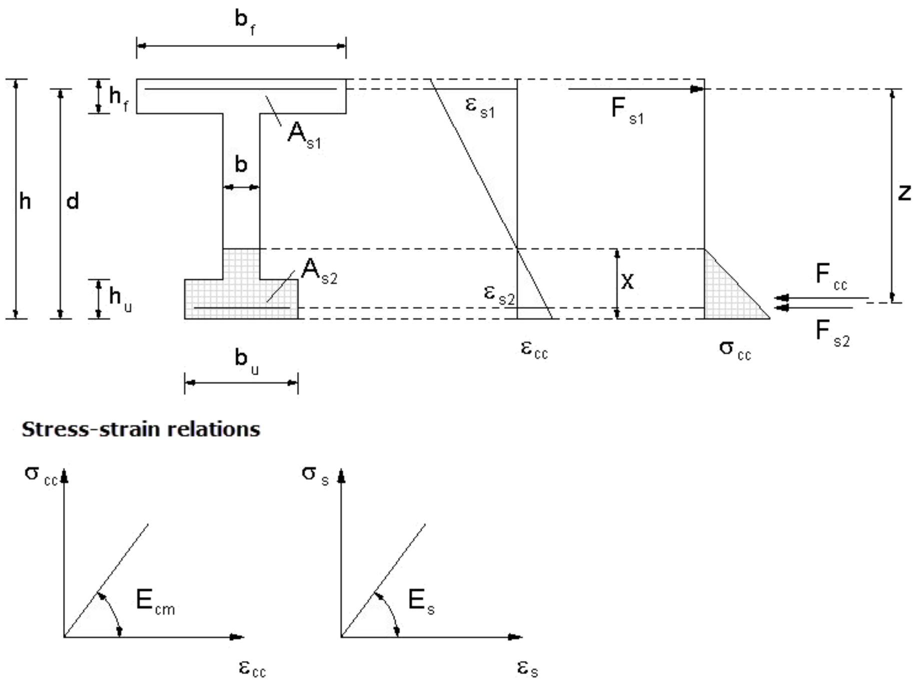


Figure 9.24 Relations for calculating the stresses and strains for characteristic loads

Calculation of neutral axis depth x_{II}

$$\rho = \frac{A_{s1}}{b \cdot d} = \frac{6.22\ \text{cm}^2}{100 \cdot 13.5\ \text{cm}^2} = 0.004607$$

$$\xi = -\alpha_e \cdot \rho \cdot \left(1 + \frac{A_{s2}}{A_{s1}}\right) + \sqrt{\alpha_e \cdot \rho \cdot \left(1 + \frac{A_{s2}}{A_{s1}}\right)^{\prime\prime} + 2 \cdot \alpha_e \cdot \rho \cdot \left(1 + \frac{A_{s2} \cdot d_2}{A_{s1} \cdot d}\right)} =$$

$$= -26.33 \cdot 0.004607 \cdot (1 + 1) +$$

$$+ \sqrt{[26.33 \cdot 0.004607 \cdot (1 + 1)]^{\prime\prime} + 2 \cdot 26.33 \cdot 0.004607 \cdot \left(1 + \frac{1 \cdot 2.5}{1 \cdot 13.5}\right)} =$$

$$= 0.3459$$

$$x_{II} = \xi \cdot d = 0.346 \cdot 13.5 = 4.67\ \text{cm}^4$$

Moment of inertia

$$\kappa = 4 \cdot \xi^3 + 12 \cdot \alpha_e \cdot \rho \cdot (1 - \xi)^2 + 12 \cdot \alpha_e \cdot \rho \cdot \frac{A_{s2}}{A_{s1}} \cdot \left(\xi - \frac{d_2}{2}\right)^2 =$$

$$= 4 \cdot 0.346^3 + 12 \cdot 26.33 \cdot 0.00460 \cdot (1 - 0.346)^2 + 12 \cdot 26.33 \cdot 0.00460 \cdot 1 \cdot \left(0.346 - \frac{2.5}{13.5}\right)^2 =$$

$$= 0.826$$

$$I_{c,II} = \kappa \cdot b \cdot \frac{d^3}{12} = 0.826 \cdot 100 \cdot \frac{13.5^3}{12} = 16\,935\ \text{cm}^4$$

Stresses for crack internal forces

$$\sigma_{cr,II} = \frac{M}{I_{y,II}} \cdot x = \frac{1210}{16935} \cdot 4.67 \cdot 10 = 3.34\ \mathrm{N/mm^2}$$

$$\sigma_{sr1,II} = \alpha_e \cdot \frac{M}{I_{y,II}} \cdot (d - x) = 26.33 \cdot \frac{1210}{16935} \cdot (13.5 - 4.7) \cdot 10 = 166.12\ \mathrm{N/mm^2}$$

$$\sigma_{sr2,II} = \sigma_{sr1,II} \cdot \frac{x - d_2}{d - x} = 166.12 \cdot \frac{4.67 - 2.5}{13.5 - 4.67} = 40.82\ \mathrm{N/mm^2}$$

Steel strain for crack internal forces

$$\varepsilon_{sr1,II} = \frac{\sigma_{sr1,II}}{E_s} = \frac{166.012}{200\,000} \cdot 1\,000 = 0.8306\ ‰$$

Steel stress and concrete stress for available moment

A simplified calculation of stresses and strains, as was done for the crack moment, cannot be applied without due consideration. The stresses and strains for the effective moment M = 17.64 kNm required for the calculation of the curvatures and stiffnesses are determined in a comparative calculation using the exact stress-strain curves for concrete and reinforcing steel according to EN 1992-1-1, Figure 3.2 or 3.3.

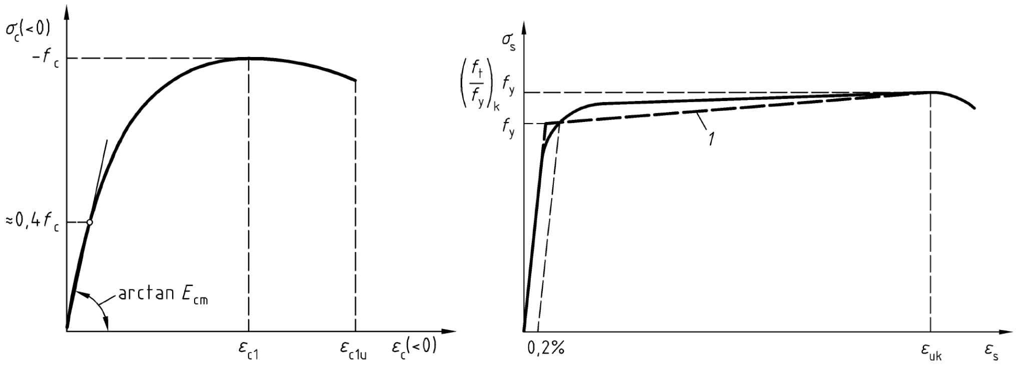


Figure 9.25 Relations for calculating stresses and strains for characteristic loads according to [1]

The accurate calculation of stresses in the cracked state is performed by means of a third party application used for exact stress integration, leading to the following results for M = 17.64 kNm:

$\sigma_{s1,II} = 242.27\ \mathrm{N/mm^2}$

$\sigma_{s2,II} = -59.07\ \mathrm{N/mm^2}$

$\varepsilon_{s1,II} = 1.211\ ‰$

$\varepsilon_{s2,II} = -0.638\ ‰$

$\varepsilon_{c,II} = -0.6378\ ‰$

Results of RF-CONCRETE Members

⊟ State II (cracked)			
⊟ Cross-Section Properties			
Moment of Inertia About the y-Axis	$I_{y\,II}$	42930500.00	mm^4
Moment of Inertia About the z-Axis	$I_{z\,II}$	1.33333E+10	mm^4
Surface	A_{II}	20123.70	mm^2
⊟ Stress and Strain Analysis			
Main Curvature	$1/r_{II}$	1.3711E-02	1/m
Curvature in y-Direction	$1/r_{yII}$	-1.0707E-20	1/m
Curvature in z-Direction	$1/r_{zII}$	1.3711E-02	1/m
Direction of the Principal Curvature	α_{II}	0.0	°
Steel Stress in Governing Fiber	$\sigma_{s\,II}$	242.51	N/mm^2
Steel strain	$\varepsilon_{s\,II}$	1.213	‰
Concrete Stress in Governing Fiber	$\sigma_{c\,II}$	-4.81	N/mm^2
Concrete Strain in Governing Fiber	$\varepsilon_{c\,II}$	-0.638	‰
Steel Stress for Crack Moment	$\sigma_{sr\,II}$	164.80	N/mm^2
Steel Strain for the Crack Moment	$\varepsilon_{sr\,II}$	0.824	‰
⊞ Cross-Section Properties			
⊞ Cross-Sectional Rigidities (Secant Values)			
⊞ Global Nodal Deformations			
⊞ Local Nodal Deformations			
⊞ Crack Width			
⊟ Verification of Safety			
⊟ Ultimate Internal Forces R_u	M_u	43.07	kNm
Ultimate Moment About Axis y	M_{yRd}	43.07	kNm
Ultimate Moment About Axis z	M_{zRd}	0.00	kNm
Ultimate Normal Force	N_{Rd}	0.00	kN
⊟ Design Internal Forces R	M	17.66	kNm

Figure 9.26 Detailed results for state II

9.2.6.4 Mean Curvatures

The mean curvatures arising with the selected Tension-Stiffening approach are determined from the calculations for pure state I and pure state II.

The underlying Tension Stiffening model described in book 525 [6] considers the concrete's tension stiffening effect between the cracks by means of a reduction of the steel strain. The required parameters are determined as follows.

Governing state of crack formation

Steel stress in state II for crack formation: $\sigma_{sr1,II} = 166.12\ N/mm^2$

Steel stress in state II: $\sigma_{s1,II} = 242.27\ N/mm^2$

$$\sigma_{s1,II} = 242.27\ N/mm^2 \geq 1.3 \cdot \sigma_{sr1,II} = 215.96\ N/mm^2$$

Hence, we will have a closer look at the final crack state.

Average steel strain

$$\varepsilon_{sm} = \varepsilon_{s2,II} - \beta_t \cdot (\varepsilon_{sr,II} - \varepsilon_{sr,I})$$

$$\varepsilon_{sm} = 1.211 - 0.306 \cdot (0.8306 - 0.199) = 1.0177\ ‰$$

where

$\varepsilon_{s2,II} = 1.211$ ‰ : steel strain in state II

$\varepsilon_{sr1,II} = 0.8306$: steel strain for crack internal force in state II

$\varepsilon_{sr1,I} = 0.199$ ‰ : steel strain for crack internal force in state I

$\beta_t = 0.306$: load duration factor of available action

Mean curvature

$$\left(\frac{1}{r}\right)_{z,m} = \frac{(\varepsilon_{sm} - \varepsilon_c)}{d} = \frac{1.0177 + 0.6378}{0.135} = 12.26\,\frac{\text{mm}}{\text{m}} = 1.226 \cdot 10^{-2}\,\text{m}^{-1}$$

Mean bending stiffness

From the mean curvature $(1/r)_{z,m}$ and the relation

$$\left(\frac{1}{r}\right)_{z,m} = \frac{M}{I_{y,m} \cdot E}$$

the secant stiffness in the corresponding node results.

$$I_{y,m} \cdot E = \frac{M_y}{(1/r)_{z,m}} = \frac{0.01764}{1.226 \cdot 10^{-2}} = 1.43883\ \text{MNm}^2 = 1438.83\ \text{kNm}^2$$

where

$M_y = 17.64$ kNm : available moment

$(1/r)_{z,m} = 1.226 \cdot 10^{-2}\text{m}^{-1}$: steel strain for crack internal force in state II

Results of RF-CONCRETE Members

⊟ Stress and Strain Analysis			
Principal Moments	M	17.66	kNm
Bending Moment About the y-Axis	M_{yEd}	17.66	kNm
Bending Moment About the z-Axis	M_{zEd}	0.00	kNm
Normal Force	N_{Ed}	0.00	kN
State of Cross-Section		Crack Formation Completed	
Mean Main Curvature	$1/r_m$	1.3095E-02	1/m
Mean Curvature in y-Direction	$1/r_{ym}$	5.2565E-16	1/m
Mean Curvature in z-Direction	$1/r_{zm}$	1.3095E-02	1/m
Average Strain in Reinforcement	ε_{sm}	1.021	‰
⊞ Design Details			
⊞ Cross-Section Properties			
⊟ Cross-Sectional Rigidities (Secant Values)			
Strain Resistance	A_m * E	630.15	MN
Mean Bending Rigidity About y-Axis	I_{ym} * E	1.44	MNm^2
Mean Bending Rigidity About z-Axis	I_{zm} * E	120.93	MNm^2
Shear Rigidity in y-Direction	A_{ym} * G	1680.00	MN
Shear Rigidity in z-Direction	A_{zm} * G	1680.00	MN

Figure 9.27 Detailed results of mean curvatures

9.2.7 Result Evaluation

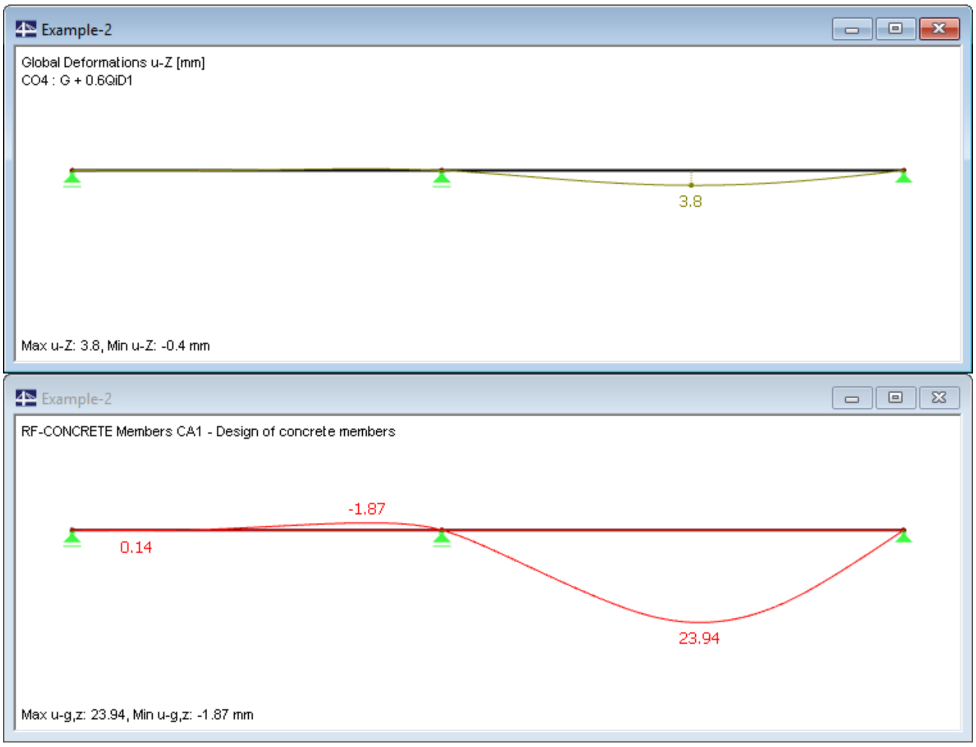


Figure 9.28 Deformations according to linear elastic calculation (above) and nonlinear calculation with creep (below)

The deformation from the nonlinear calculation with consideration of the creep effect proves to be significantly higher than the deformation from the pure linear elastic calculation without creep effect. As described in Chapter 9.2.5, the calculated deformation falls below the recommended limit value of $\ell/250$.

The deformations in state II are considerably affected by three factors:

Floor thickness

In our example, the floor thickness was determined by a limitation of the bending slenderness according to DIN 1045-1, 11.3.2, so that we could describe the calculation process. According to EN 1992-1-1, the result is a floor thickness of $h \geq 18$ cm calculated with the same boundary conditions. By increasing the thickness to 18 cm we can significantly reduce the deformations.

Creeping

The assumed creep coefficient appears to be relatively high at $\varphi_\infty = 2.95$ but meets the requirements according to EN 1992-1-1 for the assumed environmental conditions and the cross-section geometry.

By means of the factor ψ ($\psi_{2,1} = 0.6$) used to calculate the quasi-permanent action combination, it would be possible to effect a certain reduction from creep-producing load to acting load.

Concrete tensile strength

The distribution of stiffnesses in the figure below shows that a large area in span 1 is cracked in the serviceability limit state.

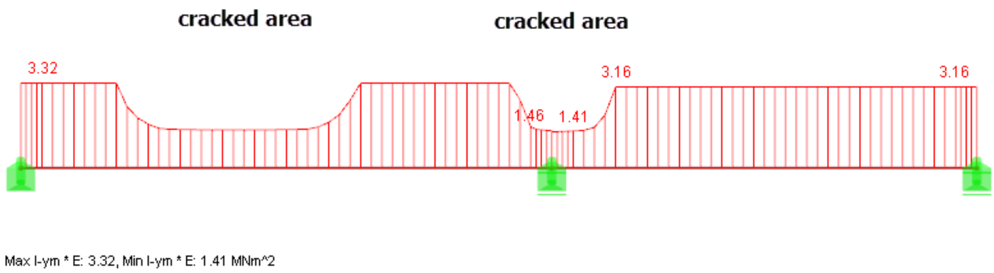


Figure 9.29 Stiffness diagram $I_{y,m} \cdot E$ over beam length

Alternative calculation with increased concrete tensile strength

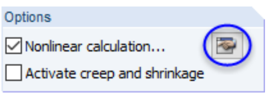
For the calculation, the concrete tensile strength was assumed with the value f_{ctm} (axial tensile strength) according to EN 1992-1-1. Parameters such as the gradient of stresses have a great influence on the concrete's effective tensile strength: A large stress gradient increases the tensile strength because the corresponding high stresses act only in few fibres. You can find more information about the different influencing factors that affect the tensile strength, inter alia, in [13].

For our example, the tensile strength is calculated once more according to [13], chapter 2.1.1:

$$f_{ctm} = 0.45 \cdot 0.818 \cdot 1 \cdot 25^{2/3} = 3.14 \text{ N/mm}^2$$

where

$f_{cm} = 20 + 5 = 25 \text{ N/mm}^2$	The mean value is taken into account by the summand 5 N/mm^2
$C_V = 0.85 - 0.2 \cdot 0.16 = 0.818 \geq 0.65$	Prior damage of structural component is taken into account
$C_h = (2.6 + 24 \cdot 0.16)/(1.0 + 40 \cdot 0.16) = 0.87$	Influence of structural component thickness
$C_\eta = 1$	Influence of eccentricity $\eta = M/(N \cdot h) \longrightarrow \infty$ für $N \longrightarrow 0$



To consider the influence of an increased tensile strength, the model is calculated in a second design case using the adjustment factor 3.14/2.2 = 1.42 (see Figure 9.30).

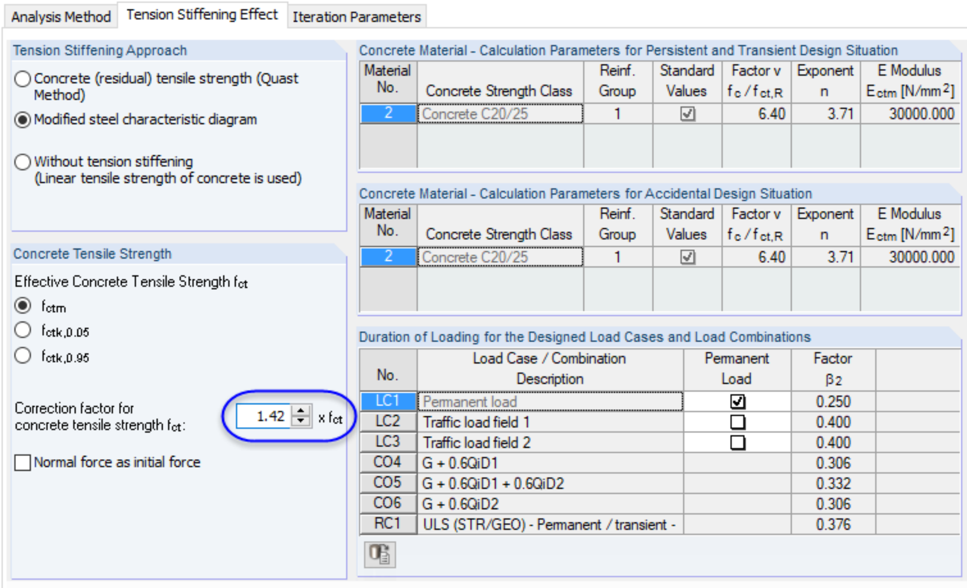


Figure 9.30 Adjusting the concrete tensile strength in the *Settings for Nonlinear Calculation* dialog box

The calculation shows a strong reduction of the cracked zones that also leads to a reduction of the deformation to u_l = 14.6 mm. This value clearly lies below the reference value of ℓ/250 = 5,000/250 = 20 mm.

The following figure illustrates the relation between deformation and stiffness reduction. In span 1, the start of the crack formation is discernible; only in the support area does the cross-section pass locally into the cracked state.

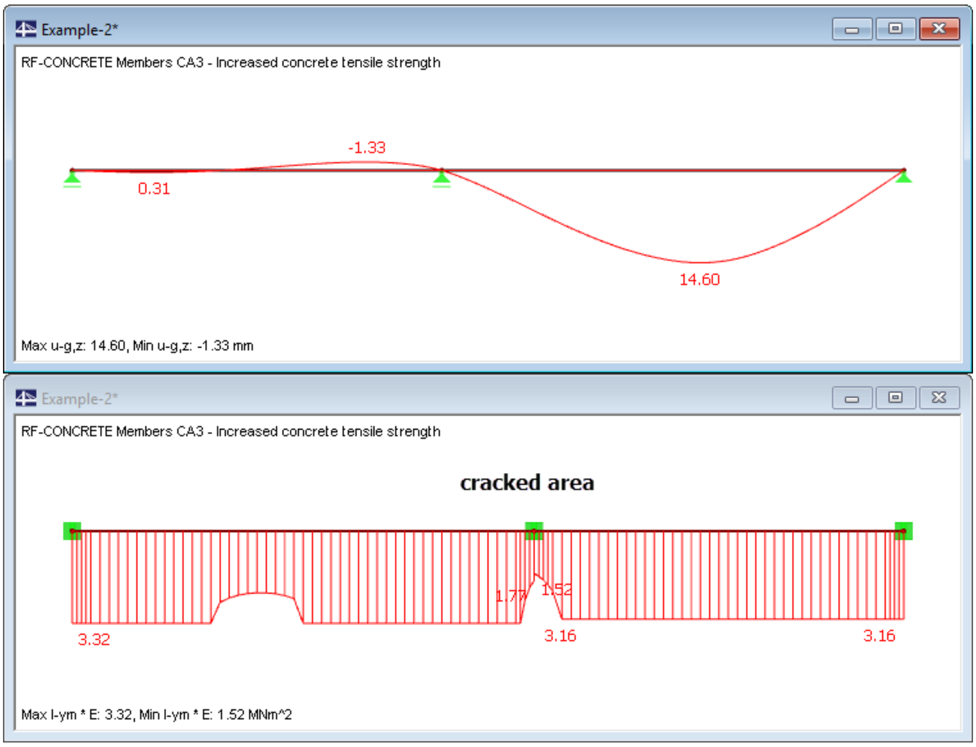


Figure 9.31 Deformation and bending stiffness for increased concrete tensile strength (stress gradient)

It becomes evident how sensitively the nonlinear calculation reacts to changed parameters. The difference is especially significant for structural components with large stiffness changes between cracked and uncracked state.

9.3 Stability Analysis for Bracket

9.3.1 Model in RFEM

By describing the stability analysis of a slender, restrained column, we look at the differences of both approaches regarding the nonlinear calculation according to EN 1992-1-1, 5.7 and 5.8.6.

This model is presented as example 1 in [14].

System and loading

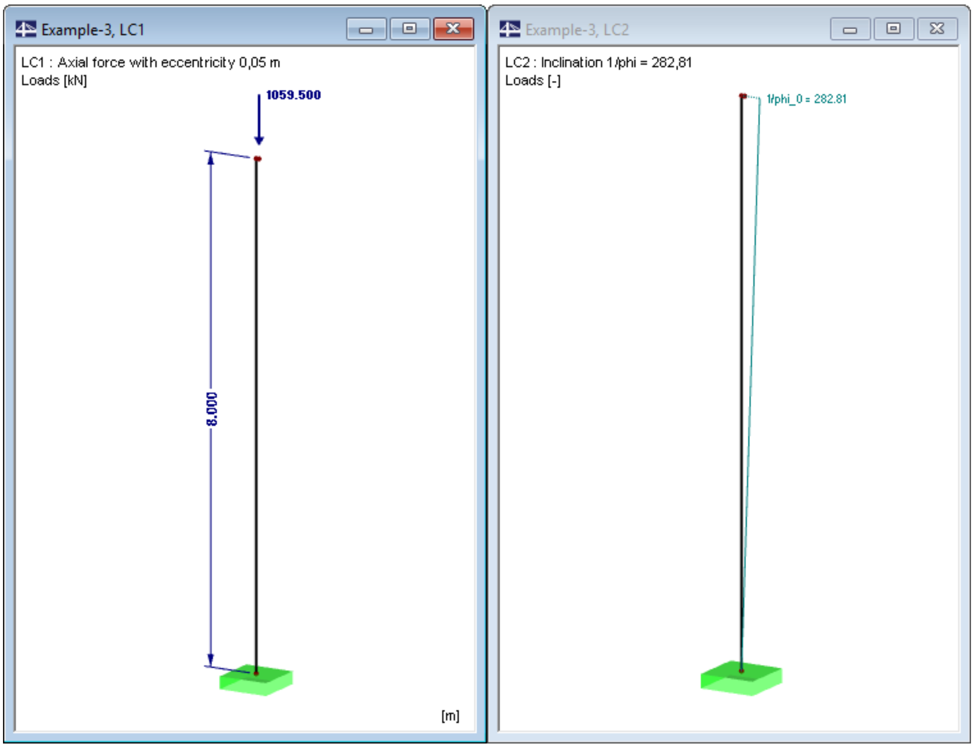


Figure 9.32 System and loading

The loading corresponds to the specifications from [14]. In load case 1, the design value N_{Ed} = 1059.5 kN is taken into account.

As shown in the figure above, the loading is entered eccentrically. The eccentricity can be determined geometrically or with an additional moment M_{Sd} = 1059.5 · 0.05 = 52.98 kNm. In our example, the load is introduced eccentrically through a short member.

The inclination of the column is considered as an imperfection in load case 2. The value of the inclination is calculated as $1/\varphi$ = 1/0.003536 = 282.81.

The concrete's modulus of elasticity is defined with 26230 N/mm^2 according to the specification in [14].

Load combinations

For analyzing the load-deformation behavior, we define the following load combinations:

- Design-relevant combination:

CO 1	LC1 + LC2

- Alternatives:

CO 2	0.20 • LC1 + LC2
CO 3	0.50 • LC1 + LC2
CO 4	0.70 • LC1 + LC2
CO 5	0.80 • LC1 + LC2
CO 6	0.90 • LC1 + LC2
CO 7	0.92 • LC1 + LC2
CO 8	0.94 • LC1 + LC2
CO 9	0.96 • LC1 + LC2
CO 10	0.97 • LC1 + LC2
CO 11	0.98 • LC1 + LC2
CO 12	0.99 • LC1 + LC2
CO 13	1.05 • LC1 + LC2
CO 14	1.10 • LC1 + LC2

No stiffness reduction by the partial safety factor γ_M is carried out for the calculation (RFEM default setting).

Results

The calculation with RFEM provides the following internal forces and deformations:

Load combination	Axial force N [kNm]	Moment I. Order Theory M_I [kNm]	Moment II. Order Theory M_{II} [kNm]	Column head displacement u [mm]
CO 1	**−1059.50**	**82.59**	**170.58**	**82.71**
CO 2	−211.90		18.55	9.27
CO 3	−529.75		56.18	27.77
CO 4	−741.65		91.27	44.77
CO 5	−847.60		113.28	55.36
CO 6	−953.55		139.33	67.83
CO 7	−974.74		145.12	70.59
CO 8	−995.93		151.12	73.45
CO 9	−1017.12		157.36	76.42
CO 10	−1027.71		160.57	77.95
CO 11	−1038.31		163.84	79.51
CO 12	−1048.91		167.18	81.09
CO 13	−1112.47		186.66	91.29
CO 14	−1165.45		208.71	100.80

Table 9.1 RFEM results

9.3.2 Nonlinear Calculation

The column is designed with both methods according to EN 1992-1-1, **5.7** and **5.8.6**.

9.3.2.1 EN 1992-1-1, 5.7

The first design case performs the analysis according to the holistic concept of the European standard EC 2.

Data entered in RF-CONCRETE Members

The basic input is shown in the following figures.

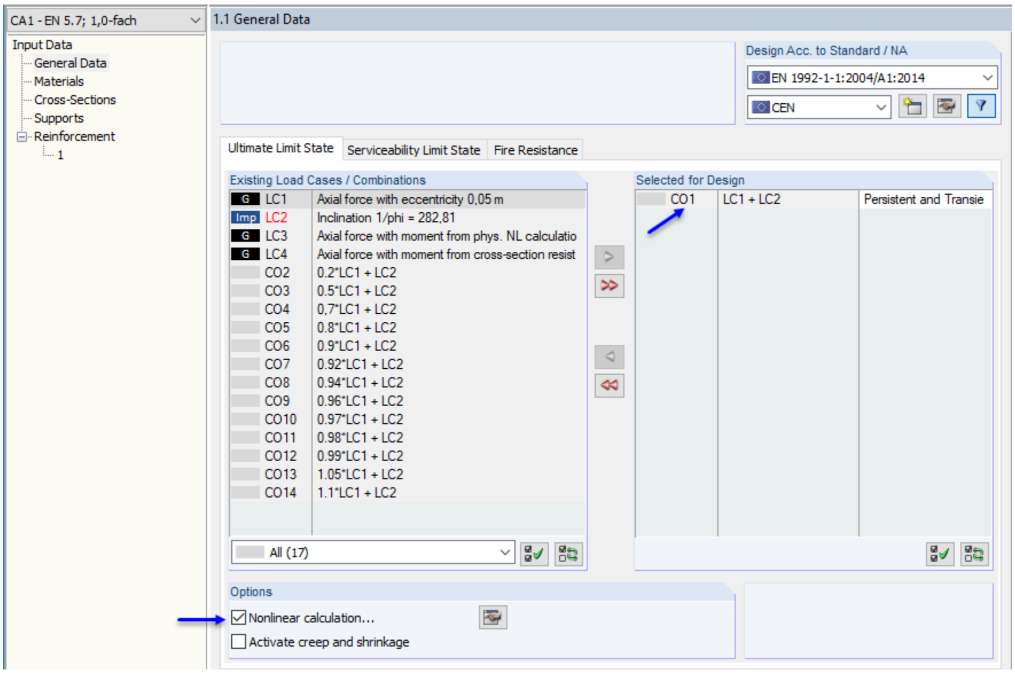
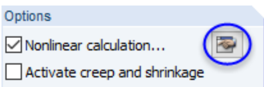


Figure 9.33 General data for nonlinear calculation according to EN 1992-1-1, 5.7



The [Settings] for the nonlinear calculation must be defined as shown in the following figures.

We select the method with mean values of material strength and global partial safety factor. Plastic releases (plastic curvatures) are excluded.

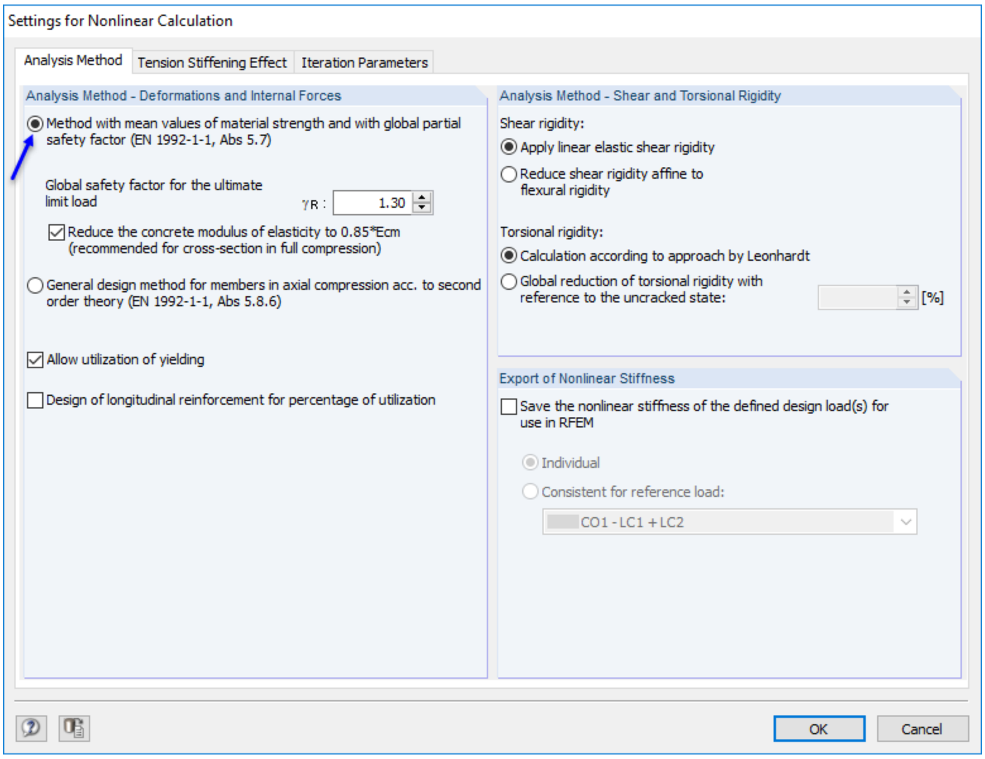


Figure 9.34 Analysis method according to EN 1992-1-1, 5.7

To achieve results comparable to the calculation in [14], we have to modify the *Tension Stiffening* model according to Quast: As the calculation of the allowable compression stress f_{cR} is based on a low quantile, $f_{ctk,0.05}$ is also used for the determination of the allowable concrete tension stress.

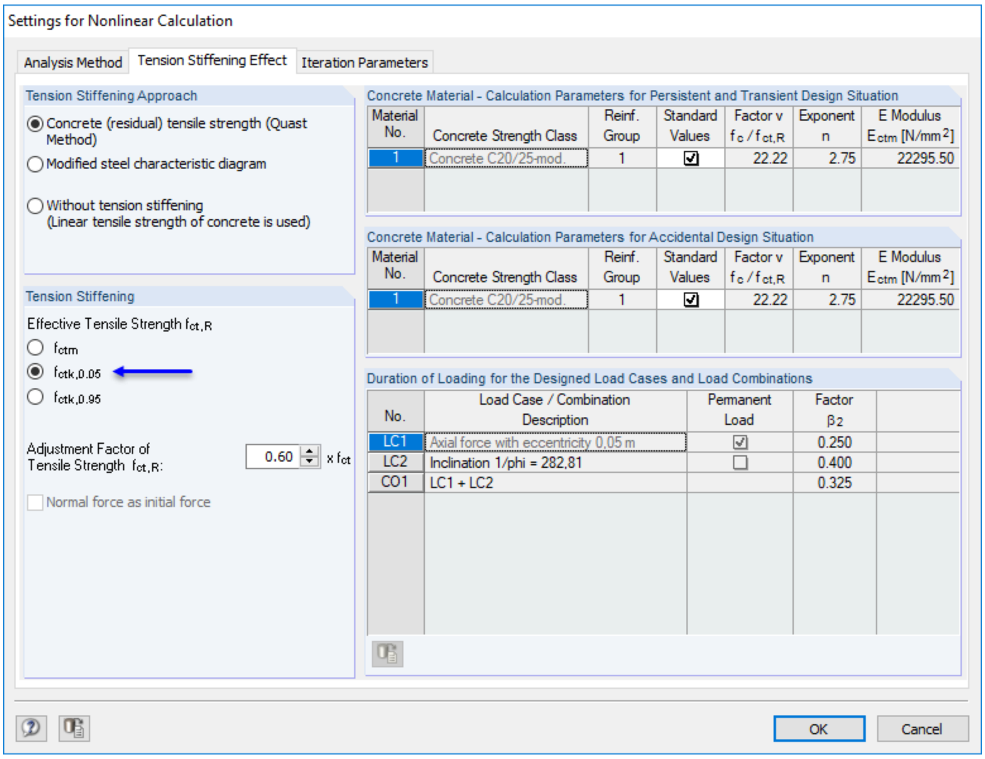


Figure 9.35 Effective tension stress of concrete for Tension Stiffening

As our structure is a statically determinate system, we can keep the damping factor set to *1.0*.

For the nonlinear calculation of models prone to instability risks, the break-off limits ε_1 and ε_2 are important: If a calculation according to the linear static analysis converges steadily, it is possible that compression elements may see a "reversal point" where the deviations ε increase again. This effect occurs when the system can no longer compensate or absorb the increase of internal forces through the decreasing stiffnesses, caused by the increase of the deformation according to the second-order analysis. We keep $\varepsilon_1 = \varepsilon_2 = 0.001$ unchanged in our example.

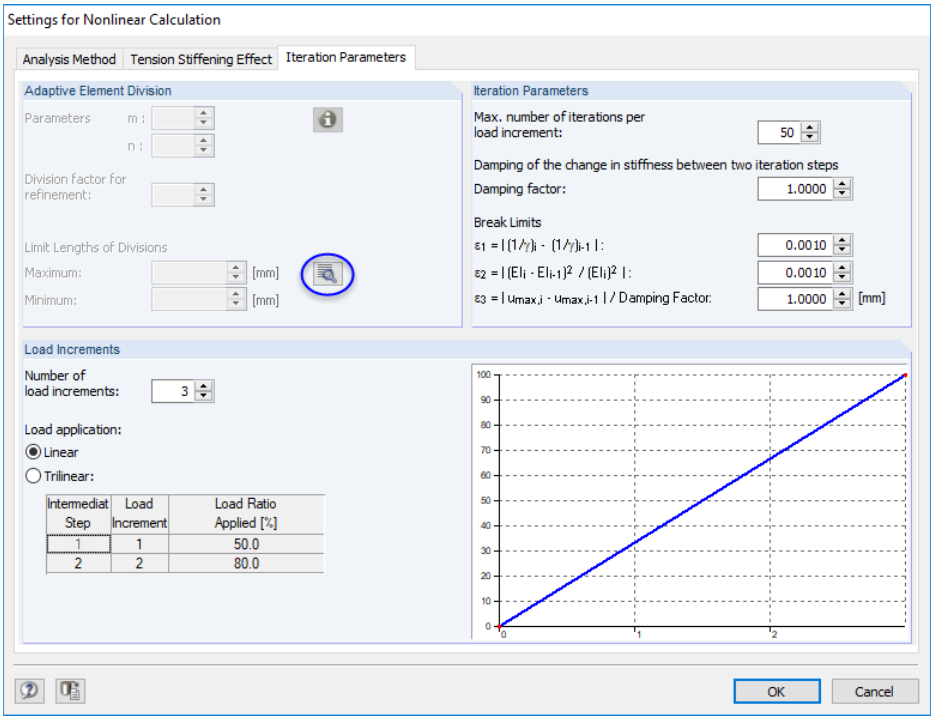


Figure 9.36 Iteration parameters

To represent the distribution of stiffnesses with sufficient accuracy, we limit the target length of the FE mesh to 0.20 m.

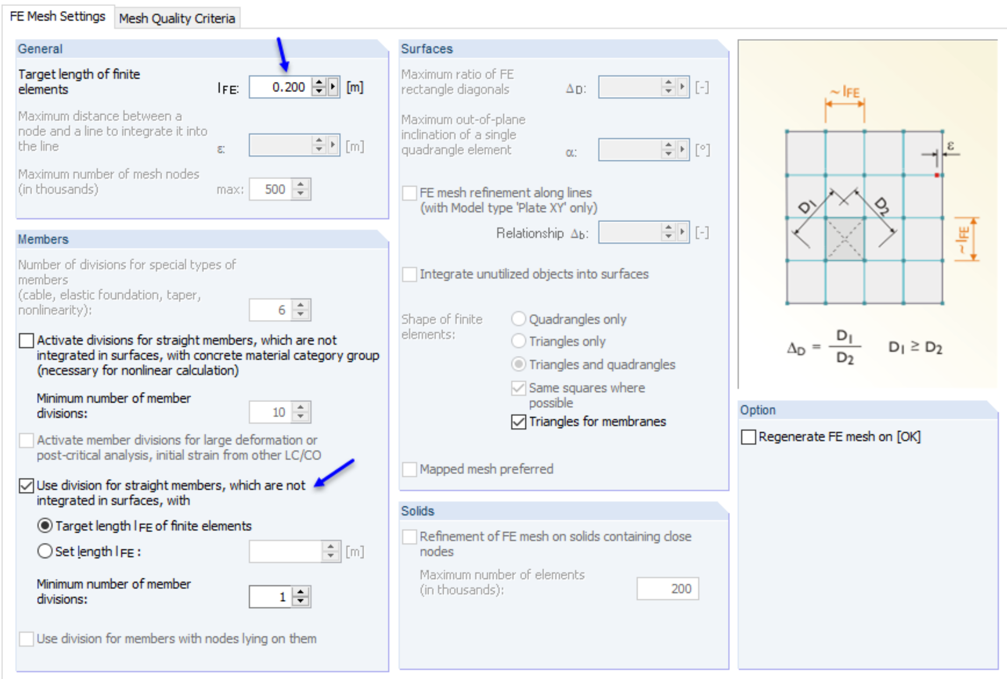


Figure 9.37 FE mesh settings

In [14], a required reinforcement of $A_{s,tot}$ = 66.10 cm^2 is determined using the similar design method according to DIN 1045-1, 8.5. In order to compare these results with the RF-CONCRETE Members calculation according to EN 1992-1-1, 5.7, we still have to specify other settings.

The design is performed with a provided reinforcement that is actually available. Thus, some specifications for diameter, concrete cover, and reinforcement amount are still required in Window *1.6 Reinforcement*. In the *Longitudinal Reinforcement* tab, we define the diameter as *25 mm*.

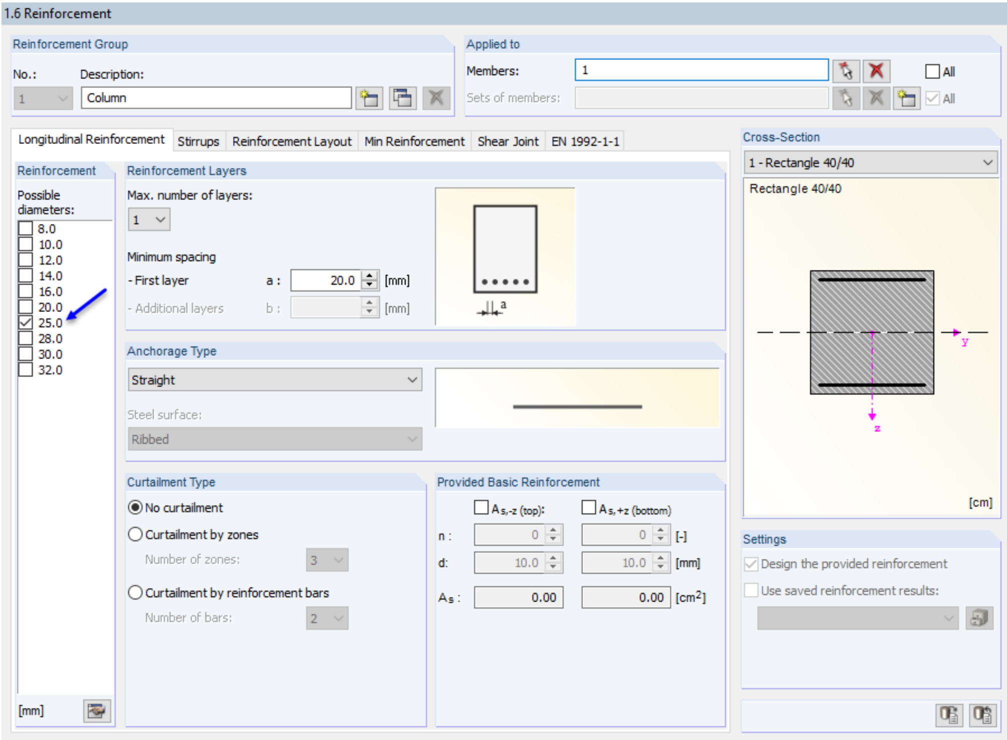


Figure 9.38 Specifying the rebar diameter

The concrete cover is selected as c_{nom} = 27.5 mm in order to get a center distance of 40 mm.

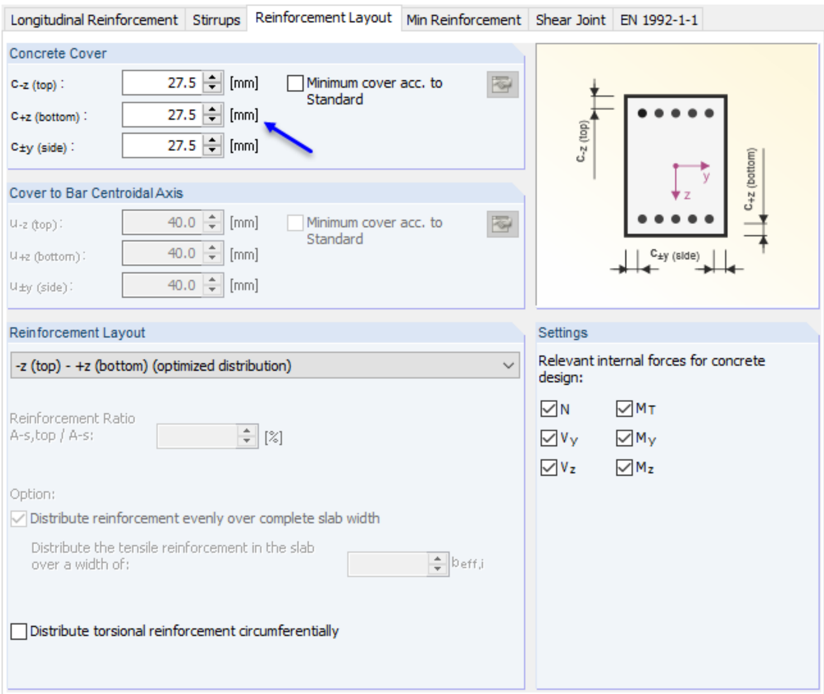


Figure 9.39 Specifying the concrete cover

In order to perform the design with the specified reinforcement from [14], a minimum reinforcement of $A_{s,top} = A_{s,bottom} = 32\ cm^2$ is defined.

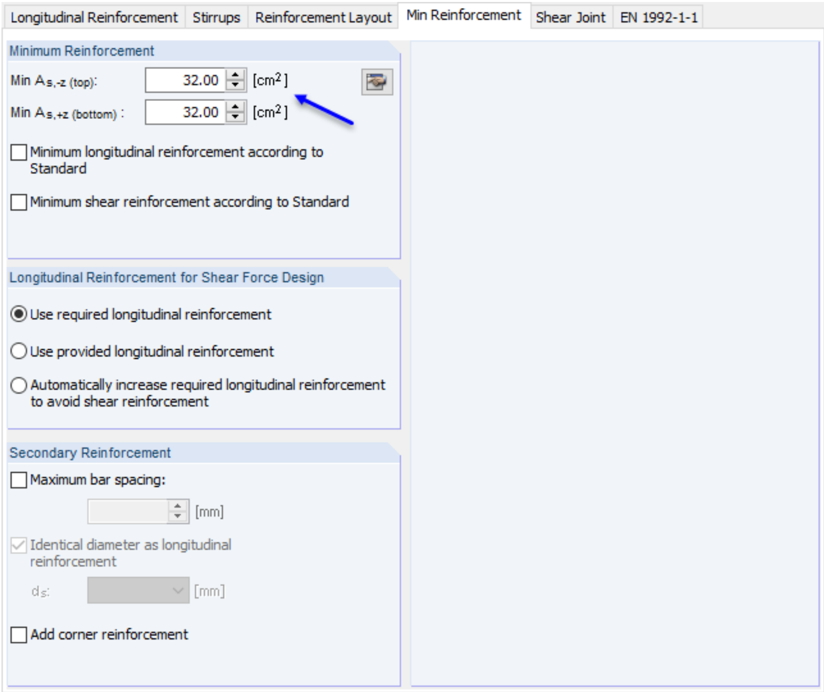
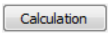


Figure 9.40 Specifying the minimum reinforcement



Now the input is complete and we can start the [Calculation].

Results of nonlinear calculation

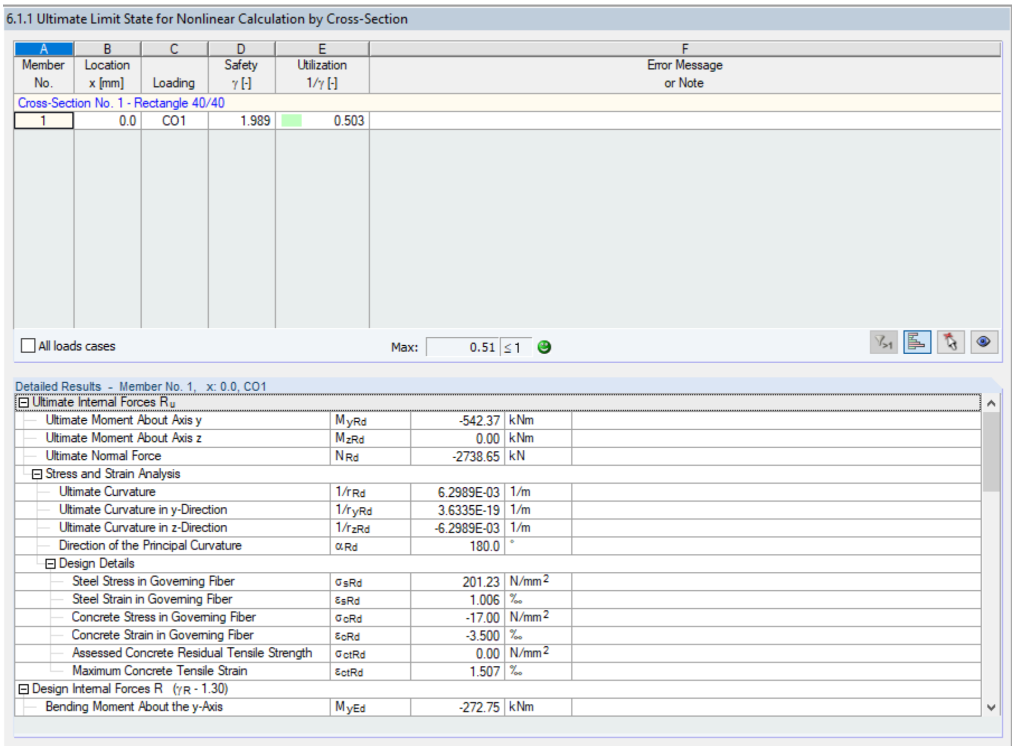
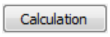


Figure 9.41 Window 6.1.1 *Ultimate Limit State for Nonlinear Calculation by Cross-Section*

The interpretation of results is explained in the previous example (Chapter 9.2).



With the safety factor γ = 1.989, the system apparently has sufficient reserves. However, we want to demonstrate that a small load increase will lead to the system's instability. In Window *1.1 General Data*, we select CO 14 for the design so that the loading is increased by 10%. According to the physically linear second order theory, there is no stability problem for this load combination.

Now the nonlinear [Calculation] is stopped by displaying a message telling us that it is not possible to design a sufficient resistance of the system with the selected reinforcement.

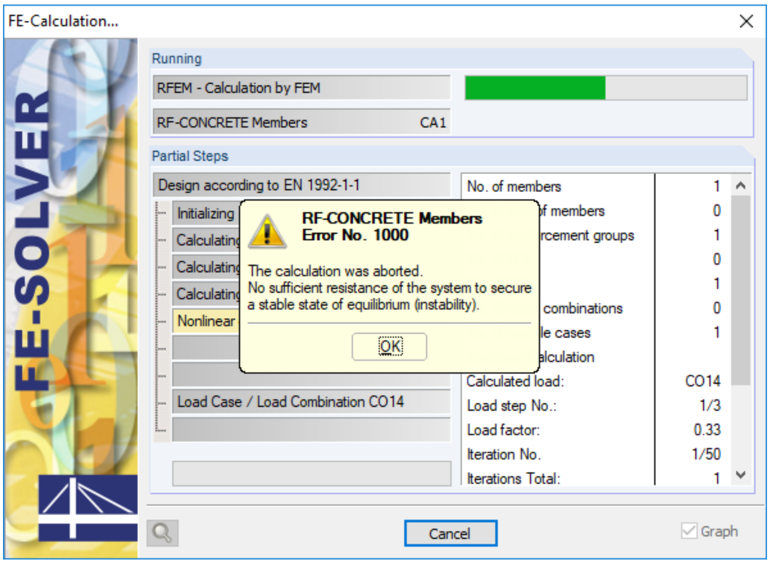


Figure 9.42 Break-off of calculation for CO 14 due to instability

Analyzing the model according to EN 1992-1-1, 5.8.6, described in the following chapter, shows that the column fails before the cross-section resistance is reached.

9.3.2.2 EN 1992-1-1, 5.8.6

The second design case performs the design in accordance with the general design method of EC 2 for compression members according to the second-order analysis.

Data entered in RF-CONCRETE Members

In order to compare the results, a new concrete case is created for EN 1992-1-1, 5.8.6. As we need to change only little input data, we simply copy the first design case on the RF-CONCRETE Members menu by selecting

File → Copy Case.

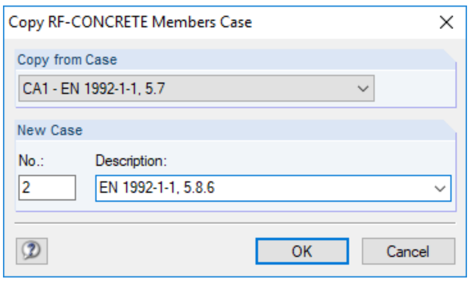
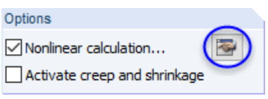


Figure 9.43 Copying the first design case



In Window *1.1 General Data*, we need to adjust the [Settings] for the calculation.

We select the general method for members in axial compression according to the second-order theory.

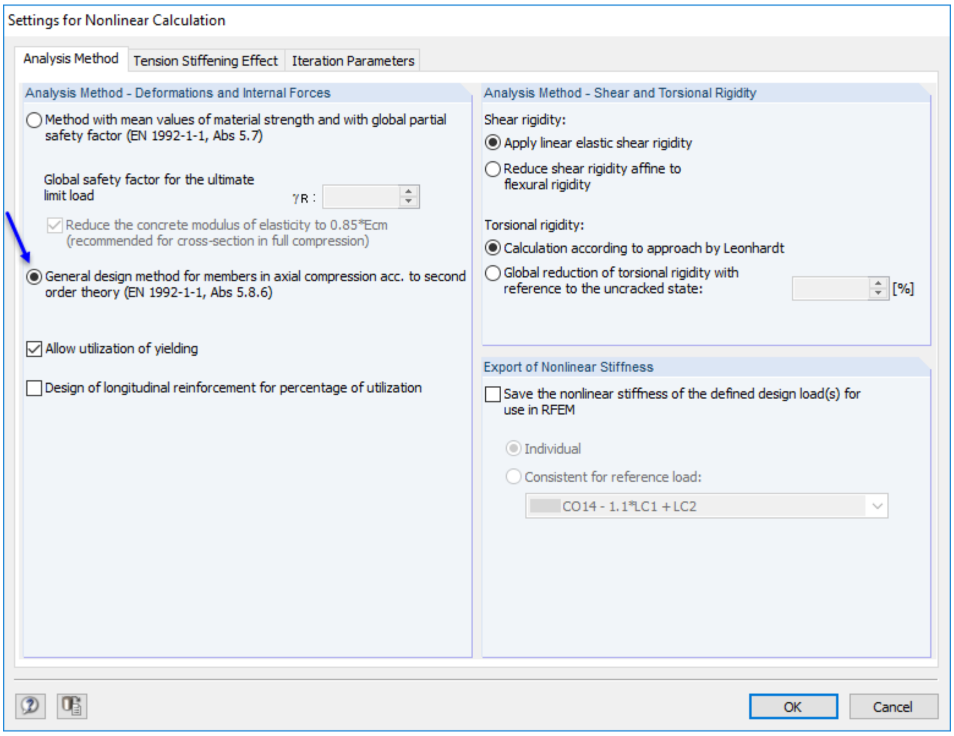


Figure 9.44 Analysis method according to EN 1992-1-1, 5.8.6

The nonlinear design of the ultimate limit state for compression elements according to EN 1992-1-1, 5.8.6 is based on a divided safety concept (see Chapter 2.4.7.2). Therefore, we also have to calculate with the average values of the material parameters for the approach of *Tension Stiffening*. The partial safety factor γ_c flows directly into the applied tensile strength: $f_{ct,R} = \alpha \cdot f_{ct} / \gamma_c$. This also applies for the concrete's modulus of elasticity.

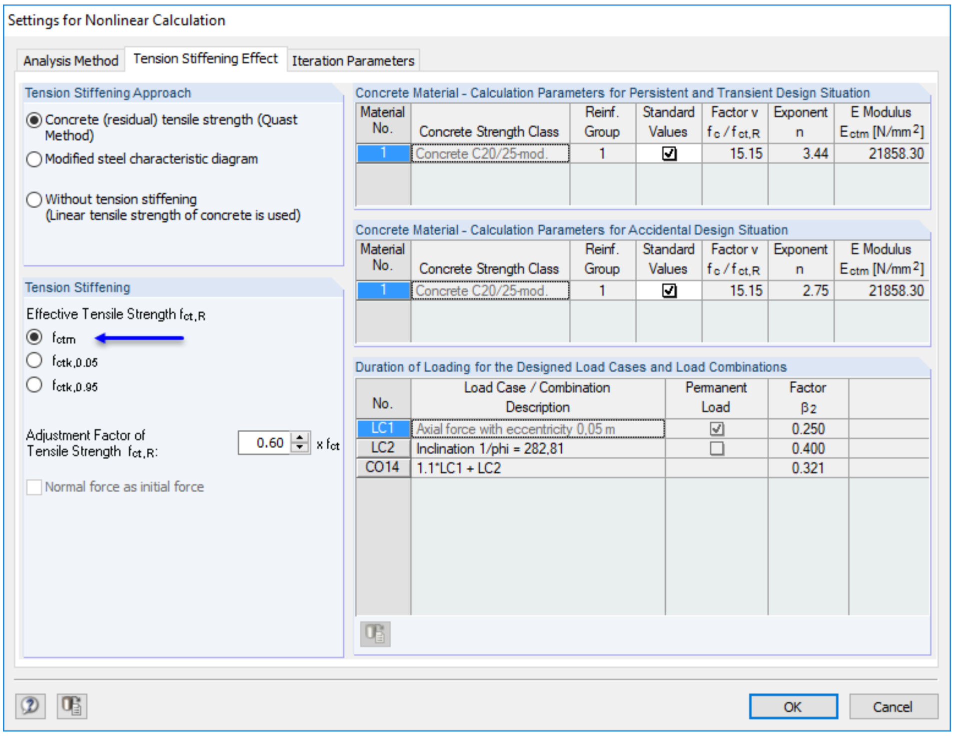


Figure 9.45 Effective tension stress of concrete for Tension Stiffening

The parameters of the *Iteration Parameters* tab remain unchanged.

In [14], a required reinforcement of $A_{s,tot}$ = 51.0 cm^2 is determined by using the similar design method according to DIN 1045-1, 8.6.1. In order to compare these results with the RF-CONCRETE Members calculation according to EN 1992-1-1, 5.8.6, we change the minimum reinforcement to $A_{s,top}$ = $A_{s,bottom}$ = 25 cm^2 in Window *1.6 Reinforcement*.

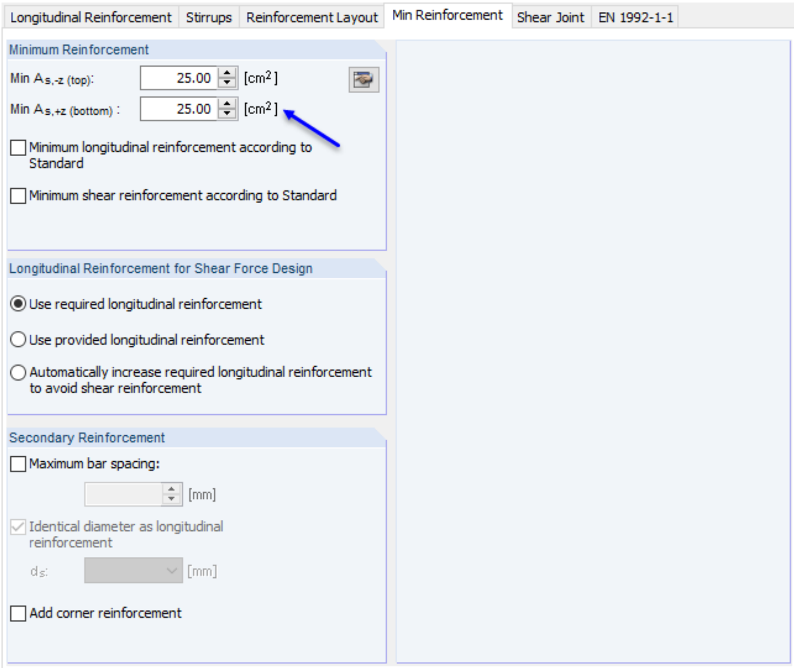
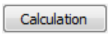


Figure 9.46 Modifying the minimum reinforcement



Now the modifications are complete and we can start the [Calculation].

Results of nonlinear calculation

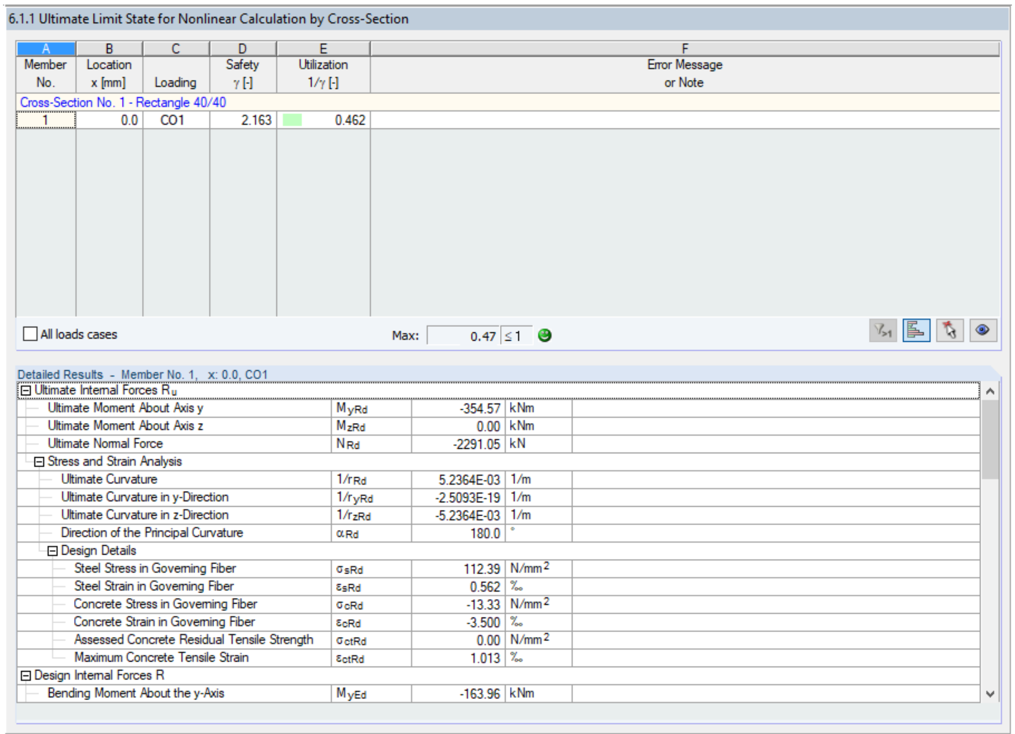


Figure 9.47 Window 6.1.1 *Ultimate Limit State for Nonlinear Calculation by Cross-Section*

With the selected reinforcement, we get a safety factor γ of 2.163 for the restrained location (in comparison: γ = 1.989 for the design according to EN 1992-1-1, 5.7).

The following figure compares the deformations determined according to the second-order analysis and both nonlinear calculation methods.

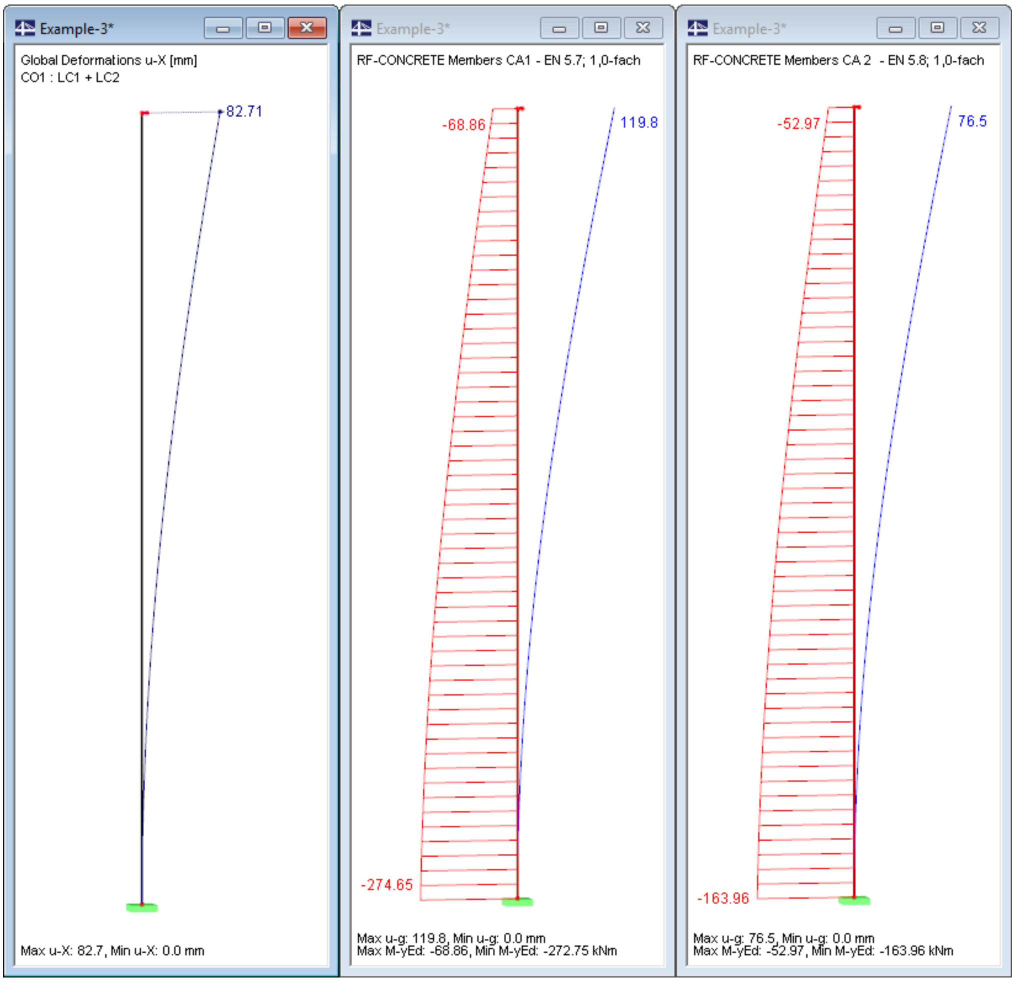


Figure 9.48 Comparison of results according to second-order analysis and nonlinear calculation

The results can be illustrated by a representation in the M-N interaction diagram. In addition to the cross-section resistance (verified quantile values), Figure 9.49 shows the capacity curves for calculation according to the linear static analysis and the second-order analysis for linear material behavior, as well as according to the second-order analysis for nonlinear material behavior.

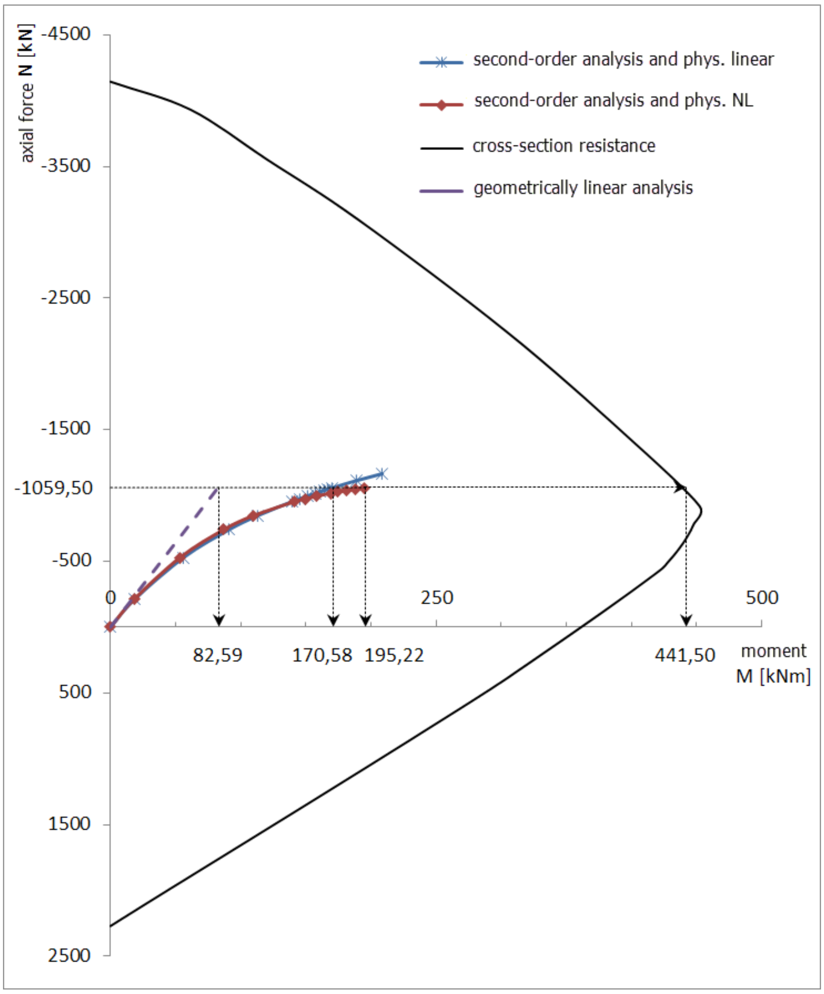


Figure 9.49 M-N interaction diagram

For our slender compression element, the calculation according to the second-order analysis already deviates from the calculation according to the linear static analysis when applying a low load level. The physical nonlinearity becomes noticeable only for a higher load level, but then it proceeds very quickly. Finally, the column fails due to loss of stability because of the strong stiffness reduction occurring in this process.

If the material-dependent nonlinearity is not taken into account, the pure cross-section design of the CO1 internal forces according to the second-order analysis (physically linear) provides a required reinforcement of $A_{s,tot}$ = 2 · 5.27 = **10.54 cm²**.

2.1 Required Reinforcement by Cross-Section

A Reinforcement	B Member No.	C Location x [mm]	D Loading	E Reinforcement Area	F Unit	G Error Message or Note
Cross-Section No. 1 - Rectangle 40/40						
$A_{s,-z\,(top)}$	1	0.0	CO1	5.27	cm²	
$A_{s,+z\,(bottom)}$	1	0.0	CO1	5.27	cm²	

Figure 9.50 Required reinforcement for physically linear design

Thus, the de facto required reinforcement is clearly underestimated. The design of moment and axial force from the physically nonlinear calculation would also lead to an under-designed reinforcement: The result for M_y = 195.22 kNm and N = −1059.39 kN would be a required reinforcement of $A_{s,tot}$ = 2 · 7.15 = **14.30 cm²**. The reason is that the internal forces are calculated depending on the provided reinforcement. However, the column fails before the ultimate load bearing capacity of the cross-section is reached. In our example, this happens for a moment of approximately 441.5 kN. In the interaction with the axial force, we get a required reinforcement of $A_{s,tot}$ = 2 · 25.40 = **50.80 cm²**.

10 Literature

[1] Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings; EN 1992-1-1:2011-01

[2] Eurocode 2: Design of concrete structures - Part 1-2: Structural fire design; EN 1992-1-2:2010-12

[3] Avak, Ralf. Stahlbetonbau in Beispielen, DIN 1045 — Teil 1 : Grundlagen der Stahlbeton-Bemessung - Bemessung von Stabtragwerken. Werner Verlag, 5. Auflage, 2007

[4] Zilch, Konrad u. Zehetmaier, Gerhard. Bemessung im konstruktiven Betonbau. Springer Verlag, 2. Auflage 2010

[5] Quast, Ulrich. Zum nichtlinearen Berechnen im Stahlbeton- und Spannbetonbau. Beton und Stahlbetonbau, Heft 9 und Heft 10, 1994.

[6] Deutscher Ausschuss für Stahlbeton (Hrsg.): Heft 525 — Erläuterungen zu DIN 1045-1. Beuth Verlag GmbH, 2003.

[7] Deutscher Ausschuss für Stahlbetonbau (Hrsg.) Heft 415 — Programmgesteuerte Berechnung beliebiger Massivbauquerschnitte unter zweiachsiger Biegung mit Längskraft. Beuth Verlag GmbH, Berlin, 1990.

[8] Pfeiffer, Uwe. Die nichtlineare Berechnung ebener Rahmen aus Stahl- oder Spannbeton mit Berücksichtigung der durch das Aufreißen bedingten Achsendehnung. Cuviller Verlag, Göttingen, 2004.

[9] Leonhardt, Fritz. Vorlesungen über Massivbau - Teil 1 bis 4. Springer Verlag, 3. Auflage, 1984

[10] Quast, Ulrich. Zur Kritik an der Stützenbemessung. Beton- und Stahlbetonbau 95 (05/2000)

[11] Hosser, Dietmar u. Richter, Ekkehard. Überführung von EN 1992-1-2 in EN-Norm und Bestimmung der national festzulegenden Parameter (NDP) im Nationalen Anhang zu EN 1992-1-2. Schlussbericht. Fraunhofer IRB, Stuttgart, 2007

[12] Heydel, Günter, Krings, Wolfgang u. Hermann, Horst. Stahlbeton im Hochbau nach EC2: Einführung und Anwendungsbeispiele. Ernst & Sohn Verlag, 1995

[13] Noakowski, Piotr u. Schäfer, Horst. Steifigkeitsorientierte Statik im Stahlbetonbau. Ernst & Sohn Verlag, 2003.

[14] Kleinschmitt, Jörrit. Die Berechnung von Stahlbetonstützen nach DIN 1045-1 mit nichtlinearen Verfahren. Beton- und Stahlbetonbau 100 (02/2005)