

Version
January 2012

Add-on Module

CONCRETE

**Linear and Non-linear Reinforced
Concrete Design
Fire Protection Design**

Program Description

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of ING.-SOFTWARE DLUBAL.

© **Ing.-Software Dlubal**
Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

Contents		Page	Contents		Page
1.	Introduction	5	2.4.8	Serviceability Limit State	48
1.1	Add-on Module CONCRETE	5	2.4.9	Convergence	50
1.2	CONCRETE Team	6	3.	Input Data	52
1.3	Using the Manual	6	3.1	General Data	52
1.4	Open the Add-on Module CONCRETE	7	3.1.1	Ultimate Limit State	54
2.	Theoretical Background	9	3.1.2	Serviceability Limit State	55
2.1	Ultimate Limit State Design	9	3.1.3	Details	56
2.1.1	Bending and Axial Force	9	3.1.4	Fire Resistance	57
2.1.2	Shear Force	10	3.2	Materials	58
2.2	Serviceability Limit State Design	12	3.3	Cross-sections	60
2.2.1	Provided Reinforcement	12	3.4	Supports	63
2.2.2	Limitation of Stresses	12	3.5	Reinforcement	65
2.2.3	Minimum Reinforcement	13	3.5.1	Longitudinal Reinforcement	66
2.2.4	Limitation of Crack Widths	14	3.5.2	Links	68
2.2.5	Limitation of Deformations	16	3.5.3	Reinforcement Layout	70
2.3	Fire Protection Design	17	3.5.4	Minimum Reinforcement	72
2.3.1	Subdivision of Cross-section	17	3.5.5	Standard	75
2.3.2	Reduction of Cross-section	18	3.5.6	Tapered	77
2.3.3	Stress-strain Curve of Concrete	20	3.5.7	Fire Resistance	79
2.3.4	Stress-strain Curve of Reinforcing Steel	22	4.	Calculation	81
2.4	Non-linear Design	24	4.1	Plausibility Check	81
2.4.1	Method	24	4.2	Details for Non-linear Analysis	81
2.4.2	Strain and Curvature	26	4.2.1	Analysis Approach	81
2.4.3	Tension-Stiffening	28	4.2.2	Tension Stiffening Effect	83
2.4.3.1	Model: Tensile Strength of Concrete	29	4.2.3	Iteration Parameters	85
2.4.3.2	Modified Characteristic Curve for Steel	31	4.3	Start Calculation	87
2.4.4	Mean Moment-Curvature Relation	32	5.	Results	89
2.4.5	Determination of Element Stiffnesses	33	5.1	Required Reinforcement	89
2.4.5.1	Flexural Resistance	33	5.1.1	Required Reinforcement by Cross-section	89
2.4.5.2	Longitudinal, Shear and Torsional Stiffness	34	5.1.2	Required Reinforcement by Set of Members	92
2.4.6	Creep and Shrinkage	37	5.1.3	Required Reinforcement by Member	92
2.4.6.1	Determination of Initial Values	37	5.1.4	Required Reinforcement by x-Location	93
2.4.6.2	Consideration of Creep/Shrinkage by Calculation	43	5.1.5	Required Reinforcement Not Designable	94
2.4.7	Ultimate Limit State	44	5.2	Provided Reinforcement	95
2.4.7.1	Material Properties	45	5.2.1	Longitudinal Reinforcement Provided	95
2.4.7.2	Safety Design	46	5.2.2	Shear Reinforcement Provided	99

Contents

Contents		Page	Contents		Page
5.2.3	Reinforcement Provided by x-Location	102	8.2	Cross-section Optimization	133
5.2.4	Steel Bending Schedule	104	8.3	Units and Decimal Places	134
5.3	Serviceability Limit State Design	105	8.4	Export of Results	135
5.3.1	Serviceability Proof by Cross-section	105	9.	Examples	137
5.3.2	Serviceability Proof by Set of Member	108	9.1	Direct Deformation Analysis	137
5.3.3	Serviceability Proof by Member	109	9.1.1	Input Data	137
5.3.4	Serviceability Proof by x-Location	109	9.1.2	Initial Values of Deformation Analysis	138
5.4	Fire Protection Design	110	9.1.3	Curvature for Uncracked Sections (State I)	138
5.4.1	Fire Protection Design by Cross-section	110	9.1.4	Curvature for Cracked Sections (State II)	139
5.4.2	Fire Protection Design by Set of Members	112	9.1.5	Determination of Deflection	140
5.4.3	Fire Protection Design by Member	113	9.1.6	Result in CONCRETE	142
5.4.4	Fire Protection Design by x-Location	113	9.2	Deformation Analysis NL in SLS	143
5.4.5	Fire Protection Design Not Designable	114	9.2.1	Input Data	143
5.5	Nonlinear Calculation	115	9.2.2	Basic Input in CONCRETE	144
5.5.1	Nonlinear Calculation - Ultimate Limit State	115	9.2.3	Checking the Reinforcement	147
5.5.2	Nonlinear Calculation - Serviceability Limit State	117	9.2.4	Specifications for Non-linear Calculation	147
5.5.3	Nonlinear Calculation - Fire Resistance	119	9.2.5	Results Display and Evaluation	151
5.5.4	Nonlinear Calculation - Design Details	120	9.2.5.1	Material Properties for Deformation Analysis	152
6.	Results Evaluation	121	9.2.5.2	State I	152
6.1	Reinforcement Proposal	121	9.2.5.3	State II	154
6.2	3D Rendering of Reinforcement	122	9.2.5.4	Mean Curvatures	160
6.3	Results in the RSTAB Model	124	9.2.6	Results Evaluation	161
6.4	Result Diagrams	127	9.3	Stability Analysis for Bracket	164
6.5	Filter for Results	128	9.3.1	Overview and Input in RSTAB	164
7.	Printout	129	9.3.2	Non-linear Calculation of Column	166
7.1	Printout Report	129	9.3.2.1	Design according to EN 1992-1-1, 5.7	166
7.2	Graphic Printout	130	9.3.2.2	Design according to EN 1992-1-1, 5.6.8	170
8.	General Functions	131	A	Literature	174
8.1	Design Cases in CONCRETE	131	B	Index	176

1. Introduction

1.1 Add-on Module CONCRETE

The add-on module CONCRETE for reinforced concrete design is completely integrated in the RSTAB user interface. Thus, a continuous analysis process is guaranteed for the design of framework models consisting of reinforced concrete elements.

The add-on module imports all relevant structure parameters from RSTAB, such as material, cross-sections, members, sets of members, supports as well as internal forces of defined actions and load combinations. The program allows also for alternative designs with modified cross-sections, including cross-section optimization.

CONCRETE analyzes the ultimate and the serviceability limit state. The designs can be performed by a conventional linear method or by a non-linear analysis. Optionally, the program checks if the requirements of the fire protection design according to EN 1992-1-2:2004 are fulfilled.

When the non-linear calculation is applied, it is possible to take into account the influence of creep and shrinkage for the analysis of crack formation during deformation. Increasing the stiffness of structural components due to the concrete's effectiveness for tension between the cracks (*Tension Stiffening*) can be controlled by user-defined parameters.

The reinforced concrete design is carried out according to the following national and European standards.

- DIN 1045:1988-07
- DIN 1045-1:2001-07
- DIN 1045-1:2008-08
- DIN V ENV 1992-1-1:1992-06
- ÖNORM B 4700:2001-06
- EN 1992-1-1:2004
- ACI 31808

The list shown on the left includes the national annexes available for EN 1992-1-1:2004 and is constantly being expanded.

The required reinforcement that is determined contains a reinforcement proposal taking into account all user specifications concerning the rebars in the longitudinal and link reinforcement. This reinforcement layout can always be adjusted. The designs related to the modifications will be updated automatically.

It is possible to visualize the inserted reinforcement by photo-realistic display. This close-to-reality representation of the reinforcement cage can be documented in the global RSTAB printout report like all other input and results data of the add-on module.

We hope you will enjoy working with CONCRETE.

Your team from ING.-SOFTWARE DLUBAL

CEN	EU
	United Kingdom
	Czech Republic
	Germany
	Denmark
	Netherlands
	France
	Portugal
	Poland
	Finland
	Slovenia
	Singapore
	Sweden
	Slovakia
	Spain
	Italy
	ÖNORM Austria

National annexes for EC 2

1.2 CONCRETE Team

The following people were involved in the development of CONCRETE:

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Alexander Meierhofer

Dipl.-Ing. (FH) Younes El Frem

Programming

Ing. Michal Balvon
Jaroslav Bartoš
Ing. Ladislav Ivančo
Ing. Alexandr Průcha

Ing. Roman Svoboda
Dis. Jiří Šmerák
RNDr. Stanislav Škovran

Program supervision

Dipl.-Ing. (FH) Alexander Meierhofer
Ing. Jan Fráňa
Ing. Pavel Gruber

Ing. Bohdan Šmid
Jana Vlachová

Manual, help system and translation

Dipl.-Ing. (FH) Alexander Meierhofer
Dipl.-Ing. (FH) Robert Vogl
Mgr. Petra Pokorná

Dipl.-Ing. Frank Faulstich
Dipl.-Ü. Gundel Pietzcker

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Andreas Hörold
Dipl.-Ing. (FH) Bastian Kuhn

M.Sc. Dipl.-Ing. (FH) Frank Lobisch
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ing. (FH) Andreas Wopperer

1.3 Using the Manual

Topics like installation, graphical user interface, results evaluation and printout are described in detail in the manual of the main program RSTAB. The present manual focuses on typical features of the CONCRETE add-on module.

Graphic

The descriptions in this manual follow the sequence of the module's input and results tables as well as their structure. The text of the manual shows the described **buttons** in square brackets, for example [Graphic]. At the same time, they are pictured on the left. In addition, **expressions** used in dialog boxes, tables and menus are set in *italics* to clarify the explanations.

At the end of the manual, you find the index. However, if you don't find what you are looking for, please check our website www.dlubal.com where you can go through our *FAQ pages*.

1.4 Open the Add-on Module CONCRETE

RSTAB provides the following options to start the add-on module CONCRETE.

Menu

To start the program in the menu bar,

point to **Design - Concrete** on the **Additional Modules** menu, and then select **CONCRETE**.

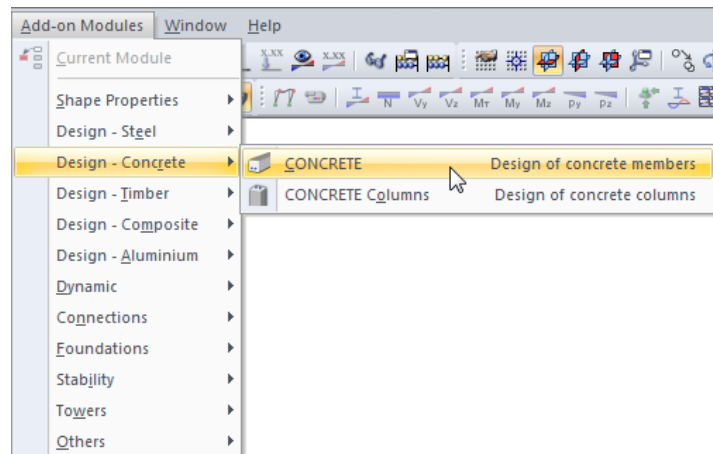


Figure 1.1: Menu *Additional Modules* → *Design - Concrete* → *CONCRETE*

Navigator

To start CONCRETE in the *Data* navigator,

open the **Additional Modules** folder and select **CONCRETE**.

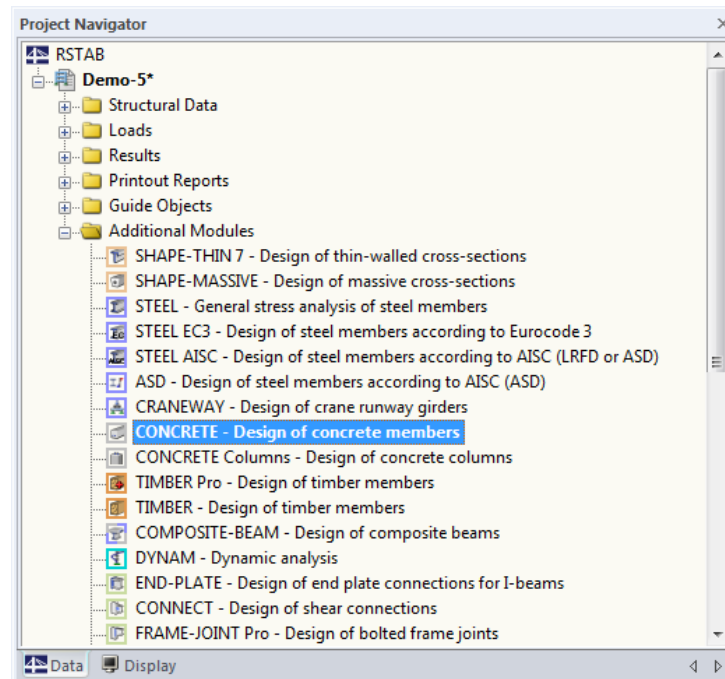
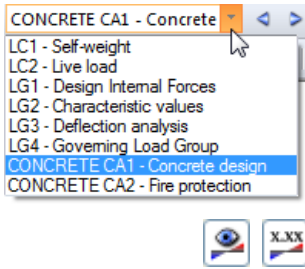


Figure 1.2: Data navigator: *Additional Modules* → *CONCRETE*



Panel

In case CONCRETE results are already available in the RSTAB structure, you can set the relevant design case in the load case list of the RSTAB toolbar (see on the left). If necessary, activate the graphical results display first by using the button [Results on/off].

When the results display is activated, the panel appears showing the button [CONCRETE] which you can use to access the design module.



Figure 1.3: Panel button [CONCRETE]

2. Theoretical Background

2.1 Ultimate Limit State Design

In the following, the module's theoretical basis is described in detail. However, this chapter shall not represent a substitute for the contents found in corresponding reference books.

2.1.1 Bending and Axial Force

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

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

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

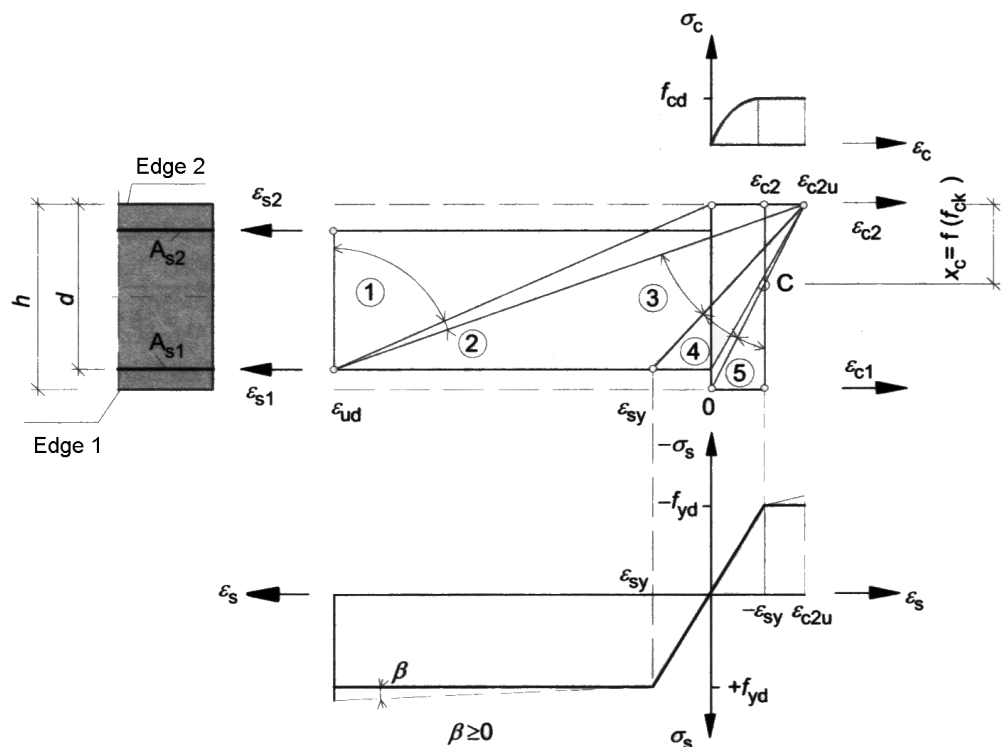


Figure 2.1: Possible strain distributions in ultimate limit state

According to [16] the different areas for strain distributions shown in the figure above have the following meaning:

Area 1

This area appears in 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 case of bending only and of bending with axial force (compression and tension force). The neutral axis lies within the cross-section. The bending-tension reinforcement is fully used, that means the steel fails when the ultimate strain is reached. Normally, the concrete cross-section is not fully used because the compression strains do not reach the ultimate strain ε_{c2u} .

Area 3

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

The concrete's failure is announced by cracks like in the areas 1 and 2 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 affected by 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. Area 4 implicates a strongly reinforced cross-section. Therefore, 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 compression force with a slight eccentricity (for example a column) or of a centric compression force. Only compression strains occur on the entire cross-section. The compression strain on the edge that is less compressed is between $0 > \varepsilon_{c1} > \varepsilon_{c2}$. All compression strain distributions intersect in point C.

2.1.2 Shear Force

The design for shear force resistance is only performed in the ultimate limit state. The actions and resistances are considered with their design values. The general design requirement according to EN 1992-1-1, 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 yield strength of shear reinforcement (failure of tie)
 $V_{Rd,max}$ Design shear resistance limited by strength of concrete compression strut

If the applied shear force V_{Ed} remains below the value of $V_{Rd,cr}$ no shear reinforcement is mathematically required and the design is fulfilled.

If the applied shear force V_{Ed} is higher than the value of $V_{Rd,cr}$ a shear reinforcement must be designed. The shear reinforcement must absorb the entire shear force. In addition, the bearing capacity of the concrete compression strut must be analyzed.

$$V_{Ed} \leq V_{Rd,s} \quad \text{and} \quad V_{Ed} \leq V_{Rd,max}$$

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 by:

$$V_{Rd,c} = \left[C_{Rd,c} \cdot k \cdot (100 \cdot \rho_l \cdot f_{ck})^{\frac{1}{3}} - k_1 \cdot \sigma_{cp} \right] \cdot b_w \cdot d \quad \text{EN 1992-1-1, Eq. (6.2a)}$$

where

$C_{Rd,c}$ Recommended value: $0.18 / \gamma_c$

$k = 1 + \sqrt{\frac{200}{d}} \leq 2.0$ Scaling factor for considering cross-section depth
 d Mean static depth in [mm]

$\rho_l = \frac{A_{sl}}{b_w \cdot d} \leq 0.02$ Ratio of longitudinal reinforcement

A_{sl} Area for tension reinforcement extended by minimum $(l_{bd} + d)$ beyond corresponding cross-section

f_{ck} Characteristic value of concrete compressive strength in N/mm^2

k_1 Recommended value: 0.15

b_w Minimum width of cross-section within tension zone in [mm]

d Effective depth of bending reinforcement in [mm]

$\sigma_{cp} = \frac{N_{Ed}}{A_c} < 0.2 \cdot f_{cd}$ Design value of concrete longitudinal stress in $[\text{N/mm}^2]$

We may apply, however, a minimum value of the shear force resistance $V_{Rd,c,min}$.

$$V_{Rd,c,min} = [v_{min} + k_1 \cdot \sigma_{cp}] \cdot b_w \cdot d \quad \text{EN 1992-1-1, Eq. (6.2b)}$$

where

$$v_{min} = \frac{k_1}{\gamma_c} \cdot \sqrt{k^3 \cdot f_{ck}}$$

Design shear resistance with shear reinforcement

For structural components with design shear reinforcement perpendicular to the component's axis ($\alpha = 90^\circ$) the following is applied:

$$V_{Rd,s} = \left(\frac{A_{sw}}{s} \right) \cdot z \cdot f_{ywd} \cdot \cot \theta \quad \text{EN 1992-1-1, Eq. (6.8)}$$

where

A_{sw} Cross-sectional area of shear reinforcement

s Distance between links

z Lever arm of internal forces assumed for $0.9 d$

f_{ywd} Design value for 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. In this way, the equation can take into account the fact that a part of the shear force is absorbed by crack friction. Thus, the structural system is less stressed. The following limits are recommended in equation (6.7) of EN 1992-1-1:

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

So the inclination of the concrete strut θ may vary between the following values:

	Minimum inclination	Maximum inclination
θ	21.8°	45.0°
$\cot\theta$	2.5	1.0

Table 2.1: Recommended limits for inclination of concrete strut

Design shear resistance of concrete compression strut

For structural components with design shear reinforcement perpendicular to the component's axis ($\alpha = 90^\circ$) the following can be applied:

$$V_{Rd,max} = \frac{\alpha_{cw} \cdot b_w \cdot z \cdot v_1 \cdot f_{cd}}{\cot\theta + \tan\theta} \quad \text{EN 1992-1-1, Eq. (6.9)}$$

where

α_{cw}	Factor for considering stress conditions in compression chord
b_w	Width of cross-section
z	Lever arm of internal forces (exactly calculated in bending design)
v_1	Reduction factor for concrete strength in case of shear cracks
f_{cd}	Design value of concrete strength
θ	Inclination of concrete compression strut

2.2 Serviceability Limit State Design

The serviceability limit state design consists of various individual designs that are specified in the following Eurocode chapters:

- Limitation of stresses: EN 1992-1-1, 7.2
- Limitation of crack widths: EN 1992-1-1, 7.3
- Limitation of deformations: EN 1992-1-1, 7.4

2.2.1 Provided Reinforcement

Before the program designs the serviceability limit state, it checks the provided reinforcement. First, CONCRETE uses the internal forces of the serviceability to perform a design similar to the design of the ultimate limit state. The design results in a structurally required reinforcement which is then compared to the user-defined provided reinforcement.

If the provided reinforcement is smaller than the statically required reinforcement, or if the analysis reveals any non-designable situations, the serviceability limit state design won't be performed.

2.2.2 Limitation of Stresses

Concrete compressive stresses

The concrete compressive stresses must be limited according to EN 1992-1-1, 7.2 (1) in order to avoid cracks or strong creep in case they would affect the structure's functioning. Therefore, chapter 7.2 (2) recommends to apply 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 non-elastic strains, unallowable crack formations and deformations, it is required to limit the tension stresses in the reinforcement according to EN 1992-1-1, 7.2 (4). Chapter 7.2 (5) recommends some reduction factors for the characteristic tensile strength which depend on the type of load combination.

$$\sigma_s = k_3 \cdot f_{yk} \quad \text{for characteristic load combination}$$

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

The recommended values for k_3 and k_4 are 0.8 and 1.

2.2.3 Minimum Reinforcement

The minimum area of reinforcement used to limit the crack width is determined in accordance with EN 1992-1-1, 7.3.2 (2), eq. (7.1) according to the following simplification:

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

where

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

σ_s Allowable stress of reinforcement steel according to Figure 2.2

k_c Factor for considering stress distribution in tension zone

$k_c = 1.0$ for tension only

$k_c = 0.4$ for bending or bending with axial force

k Factor for considering non-linearly distributed self-equilibrating stresses

$k = 1.0$ for webs with $h \leq 300$ mm

$k = 0.65$ for $h \geq 800$ mm

$k = 1.0$ for restraint externally caused (for example column settlement), only NA for Germany

$f_{ct,eff}$ Mean value of the concrete's effective tensile strength when cracks occur

$$f_{ct,eff} = f_{ctm}$$

A_{ct} Area of concrete within tensile 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 2.2: Limit diameter \varnothing^* , for reinforcing steels according to EN1992-1-1, table 7.2

2.2.4 Limitation of Crack Widths

Checking the rebar diameter

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

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

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

where

\varnothing_s^* Limit diameter according to Figure 2.2

$f_{ct,eff}$ Effective tensile strength of concrete at relevant point of time, here f_{ctm}

k_c Factor for considering stress distribution in tension zone, here $k_c = 0.4$

h_{cr} Depth of tensile zone immediately before cracking occurs

h Total depth of cross-section

d Statically effective depth to centroid of outside reinforcement

Design of rebar spacing

The maximum rebar spacing max s_r is specified in EN 1992-1-1, table 7.3.

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 2.3: Maximum values for rebar spacings according to EN 1992-1-1, table 7.3

Design of crack width

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

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

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 concerning tension between cracks

ε_{cm} Mean strain of concrete between cracks

Maximum crack spacing $s_{r,max}$

In case the rebar spacing is not more than $5 \cdot (c + \varnothing/2)$ in the tension zone, the maximum crack spacing for the final crack state may be determined according to EN 1992-1-1, 7.3.4 (3), eq. (7.11):

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

where

- k_3 Factor (recommended value: 3.4)
- c Concrete cover of longitudinal reinforcement
- k_1 Factor to consider bond properties
 $k_1 = 0.8$ for bars with high bond properties
 $k_1 = 1.6$ for bars with plain surface
- k_2 Factor to consider distribution of strain
 $k_2 = 0.5$ for bending
 $k_2 = 1.0$ for tension only
- k_4 Factor (recommended value: 0.425)
- ρ_{eff} Effective reinforcement ratio

If the spacing of rebars lying in the bond exceeds $5 \cdot (c + \varnothing/2)$, or if no reinforcement is available within the bond in the tension zone, the following crack width limit may be assumed:

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

Difference of mean strain ($\varepsilon_{sm} - \varepsilon_{cm}$)

The difference of mean strain for concrete and reinforcing steel is determined in accordance with EN 1992-1-1, 7.3.4 (2), eq. (7.9) as follows:

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

where

- σ_s Stress in tension reinforcement under assumption of cracked 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, here f_{ctm}
- α_e Relation of elastic moduli E_s / E_{cm}
- ρ_{eff} Effective reinforcement ratio

2.2.5 Limitation of Deformations

EN 1992-1-1, 7.4.3 allows for a simplified design to limit deformations by a direct calculation. The deflections must be determined close to reality: The calculation method has to match with the real structural material performance presenting a certain accuracy that corresponds to the design purpose.

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

Therefore, the deflection is determined by the principle of virtual work for the location of the maximum deformation. For the curvature an approximate line is applied connecting the extreme values of the curvature with a line that is affine with the moment distribution.

When calculating manually, three values of deflection are determined according to [16]:

Lower calculation value of deflection

The 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

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

Probable value of deflection

It is assumed 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 state I it shows some cracks. This assumption results in the probable value of deflection f existing between the lower and the upper calculated value. According to EN 1992-1-1, 7.4.3 (3), eq. (7.18) the value can be derived from the following relation:

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

The values α_I and α_{II} represent general deflection parameters (for example f_I or f_{II}). This can be a strain, curvature, deflection or rotation. ζ is the distribution value between state I and state II and, according to EN 1992-1-1, eq. (7.19), is between $0 \leq \zeta < 1$. To determine a probable deflection, a quasi-permanent combination of action is used for the calculation of internal forces.



Chapter 9.1 on page 137 shows an example where the manually performed calculation of a deformation analysis is compared with the CONCRETE analysis.

2.3 Fire Protection Design

The fire protection design with CONCRETE is performed according to the simplified calculation method in accordance with EN 1992-1-2, 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 the material stiffnesses. The damaged concrete zones directly exposed to fire are not taken into account for the equivalent cross-section that is used for the fire protection design. The fire protection design is performed with the reduced cross-section and the reduced material properties similar to the ultimate limit state design at normal temperature.

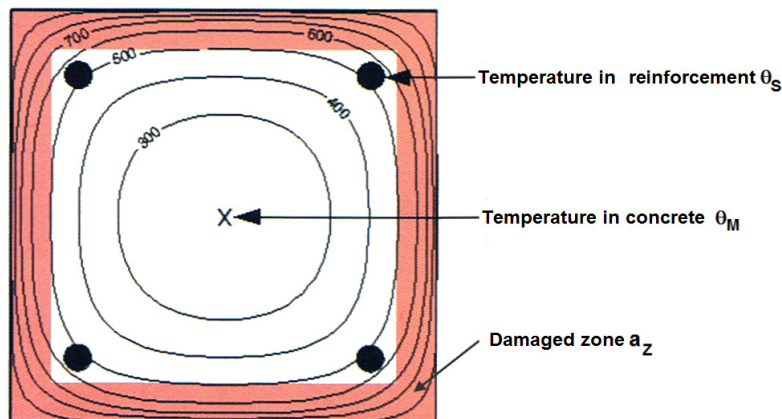


Figure 2.4: Cross-section exposed to fire with damaged zones

2.3.1 Subdivision of Cross-section

The cross-section is subdivided into a certain number of parallel ($n \geq 3$) zones having 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.

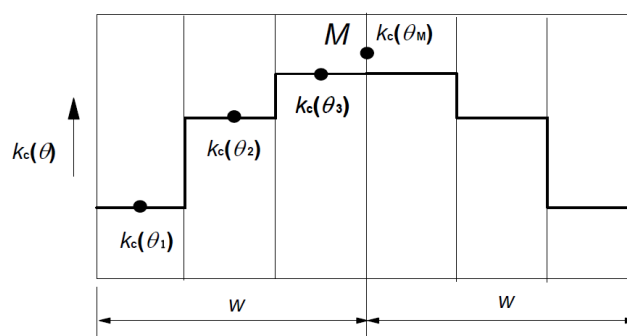


Figure 2.5: Subdivision of a wall with both sides exposed to fire into zones according to [7], figure B.4

The cross-section exposed to fire is compared with a wall. The width of the equivalent wall is $2 \cdot w$. The equivalent width is subdivided symmetrically in several zones as shown in Figure 2.5.

Half of the equivalent width w depends on the fire load acting on the structural component. The following table gives an overview about the determination of equivalent widths conforming to standards:

Fire load	Half of equivalent width w
Component with one side exposed to fire	Width of component in direction of fire effects
Column or wall with both sides (opposed) exposed to fire	0.5 * width of component in direction of fire effects
Column with four sides exposed to fire	0.5 * smallest external cross-section dimension

Table 2.2: Determination of equivalent widths

2.3.2 Reduction of Cross-section

Determination of temperature θ_i in center of zone

Subsequent to the cross-section's subdivision into zones, the temperature θ_i is determined in the center of each zone i . The determination is facilitated by temperature courses in accordance with EN 1992-1-2, annex A, based on the following assumptions:

- The concrete's specific heat corresponds to the specifications according to EN 1992-1-2, 3.2.2.
- The moisture is 1.5 % (for moistures > 1.5 % the specified temperatures are on the safe side).
- The concrete's thermal conductivity is the lower limit value mentioned in EN 1992-1-2, 3.3.3.
- The emission value for the concrete surface is 0.7.
- The convective heat-transmission coefficient is 25 W/m²K.

Determination of reduction factor $k_c(\theta_i)$

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

According to EN 1992-1-2, figure 4.1, graph 1 shown in the diagram below is to be used for normal concrete with aggregates containing quartz. Graph 2 represents normal concrete with aggregates containing limestone.

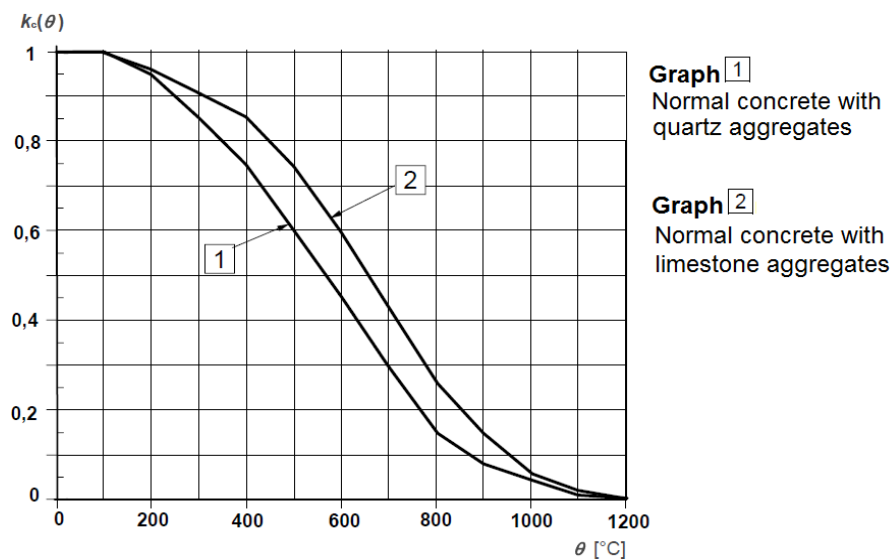


Figure 2.6: Factor $k_c(\theta)$ to consider decrease of concrete compressive strength according to [7], figure 4.1

Determination of damaged zone with thickness a_z

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

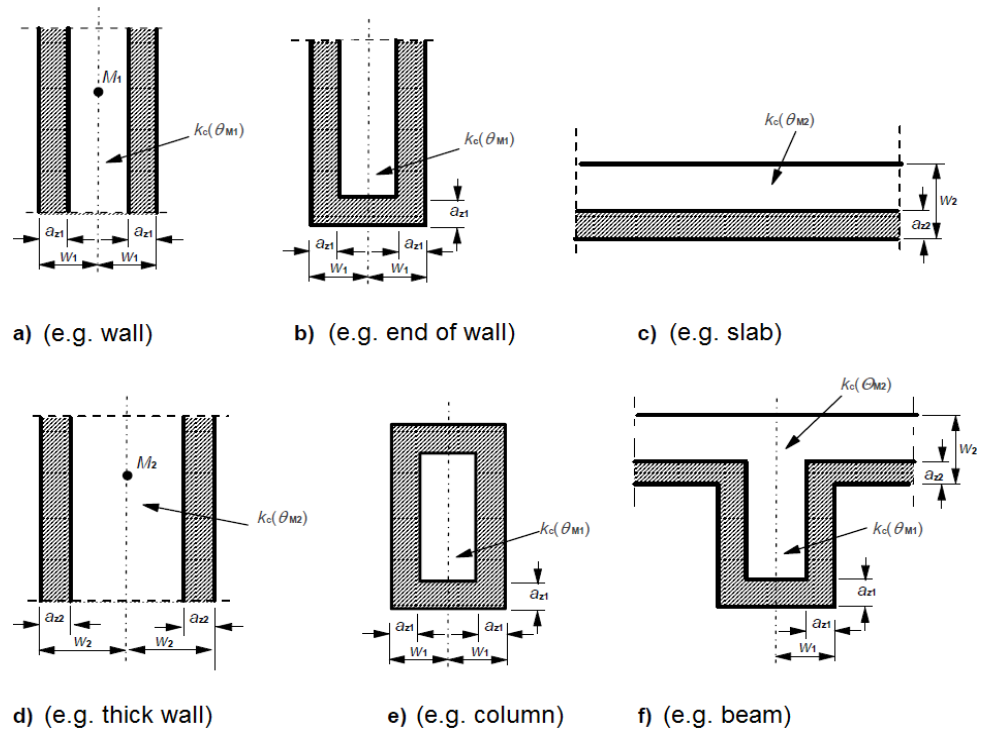


Figure 2.7: Reduction of strength and cross-section in case of fire according to [7], figure B.3

The calculation of the damaged zone thickness a_z depends on the 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 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 of width of equivalent wall

$k_{c,m}$ Mean reduction coefficient for a specific cross-section

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

n Number of parallel zones in w

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

$k_c(\theta_M)$ Reduction coefficient for concrete at point M (cf. Figure 2.6)

2.3.3 Stress-strain Curve of Concrete

Point M is decisive for the reduction of the concrete's material properties. M is a point on the central line of the equivalent wall (cf. Figure 2.5, page 17). It is used to determine the reduction factor $k_c(\theta_M)$. The reduced material properties of the concrete are to be used for the entire reduced cross-section (without the damaged zone a_z) in the ultimate limit state design in case of fire.

Compressive strength of concrete for fire protection design

The stress-strain curve for the concrete compressive strength is determined depending on the temperature in point M as well as on the type of aggregates. The values of the compression strain $\epsilon_{cu1,\theta}$ 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 2.6, page 18)

f_{ck} Characteristic compressive strength of concrete for normal temperature

Concrete temp. θ	Quartz aggregates			Limestone aggregates		
	$f_{c,\theta} / f_{ck}$	$\epsilon_{c1,\theta}$	$\epsilon_{cu1,\theta}$	$f_{c,\theta} / f_{ck}$	$\epsilon_{c1,\theta}$	$\epsilon_{cu1,\theta}$
[°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 2.8: Parameters of stress-strain relation for concrete in case of fire according to [7], table 3.1

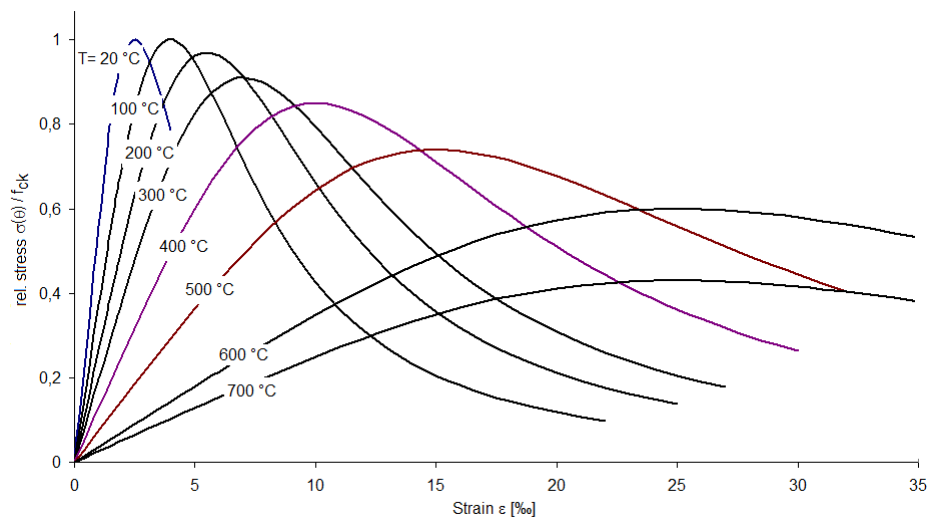


Figure 2.9: Stress-strain diagram for concrete with aggregates containing limestone, depending on temperature



As you can see in the diagram (Figure 2.9), the stress-strain relation of normal concrete with aggregates containing limestone is changing depending on the temperature. The decreasing graph is not taken into account for the fire protection design.

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

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

where

$k_c(\theta_M)$ Reduction coefficient for concrete at point M (see Figure 2.6, page 18)

E_c Elastic modulus of concrete for normal temperature (20 °C)

Determination of reduced tensile strength of concrete

In order to be on the safe side, the concrete's tensile strength is not considered for the exclusive cross-section design. But when compression elements are designed according to the non-linear approach, the tensile strength of concrete is taken into account for the determination of internal forces as well as for the cross-section design (cf. chapter 2.4.3).

For the fire protection design we determine the reduced tensile strength of concrete as follows:

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

where

$k_{c,t}(\theta_M)$ Reduction coefficient for tensile strength of concrete acc. to Figure 2.10

$f_{ck,t}$ Characteristic tensile strength of concrete for normal temperature

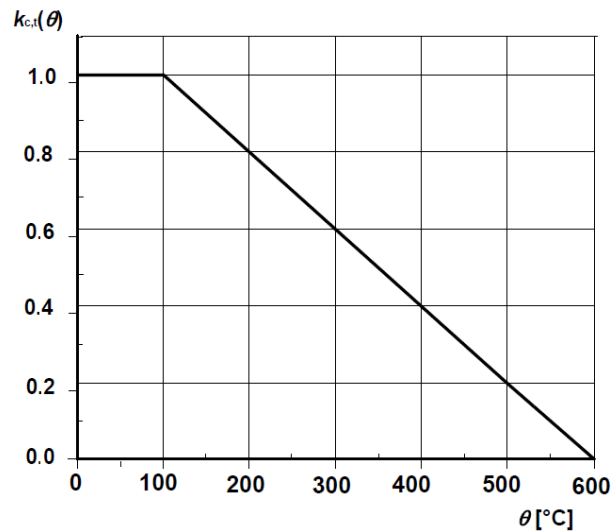
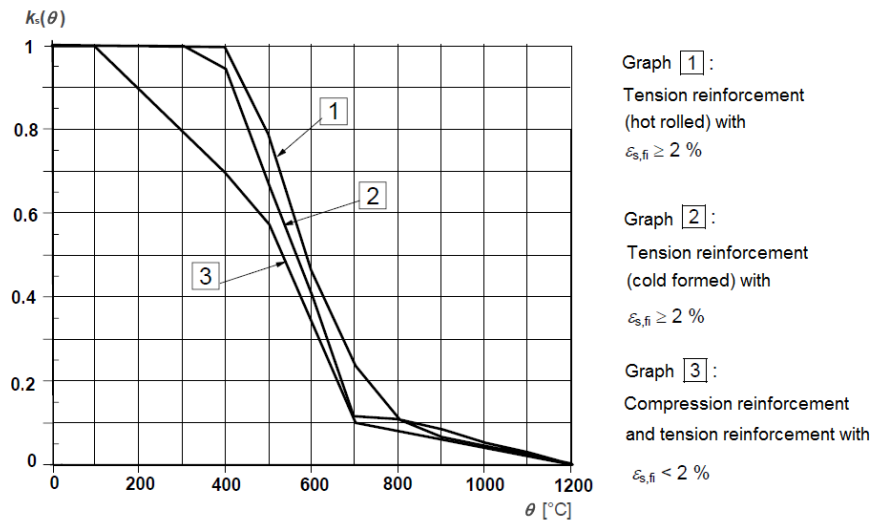


Figure 2.10: Reduction factor $k_{c,t}(\theta)$ to consider temperature-dependent tensile strength of concrete f_{ct} according to [7], figure 3.2

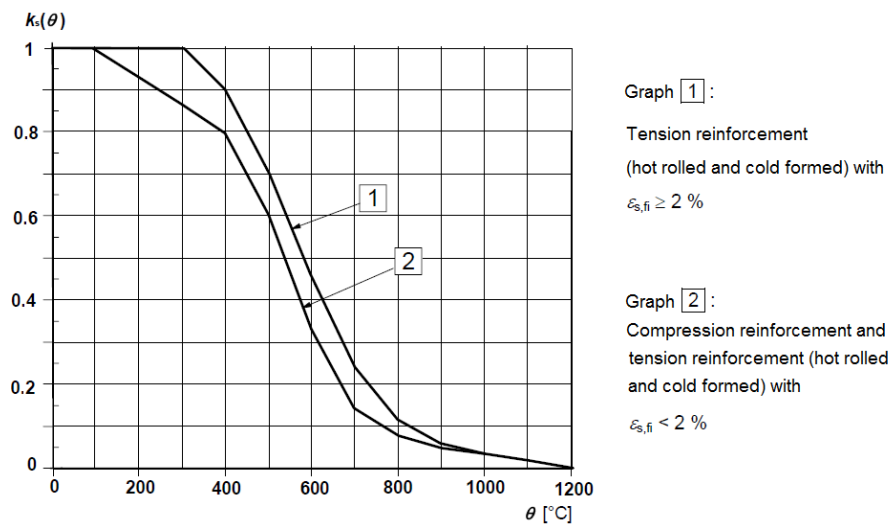
2.3.4 Stress-strain Curve of Reinforcing Steel

Determination of reduction factor $k_s(\theta)$ for tensile strength of steel

To determine the reduction factor $k_s(\theta)$, 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 X) and how much it is strained, the reduction factor $k_s(\theta)$ is defined.



Class N



Class X

Figure 2.11: Reduction factor $k_s(\theta)$ to consider temperature-dependent tensile strength of steel according to [7], figure 4.2a/b

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

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

- Slope in linear-elastic zone $E_{s,\theta}$
- Proportionality limit $f_{sp,\theta}$
- Maximum stress level $f_{sy,\theta}$

The maximum strength of the reinforcing steel that is to be applied for the fire protection design is determined as follows:

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

where

$k_s(\theta)$ Reduction coefficient for reinforcing steel (see Figure 2.11)

f_{yk} Characteristic strength of reinforcing steel for normal temperature

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

If the reinforcing steel can be assigned to graph 1 or graph 2 of figure 4.2a or 4.2b shown in EN 1992-1-2 (cf. Figure 2.11), it is possible to take the reinforcing steel's reduced elastic modulus, depending on the steel temperature and type of production, from EN 1992-1-2, table 3.2a or 3.2b.

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

Class N

Steel temperature θ [°C]	$f_{sy,\theta} / f_{yk}$	$f_{sp,\theta} / f_{yk}$	$E_{s,\theta} / E_s$
	hot rolled and cold formed	hot rolled and cold formed	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

Class X

Figure 2.12: Parameters of stress-strain relation for steel in case of fire according to [7], table 3.2a/b

For reinforcing steels that are 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

$k_s(\theta)$ Reduction coefficient for reinforcing steel (see Figure 2.11)

E_s Elastic modulus of reinforcing steel for normal temperature (20 °C)

2.4 Non-linear Design

EN 1992 and DIN 1045-1 allow for a non-linear determination of internal forces in the limit states for bearing capacity and serviceability. The internal forces and deformations are determined by taking into account the non-linear behavior of internal forces and deformations (physical).

2.4.1 Method

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

To avoid serious input mistakes and a certain oscillation of elements, we use an adaptive member division on the one hand, and a damped "stiffness application" on the other hand. In this way we try to avoid extremely large differences in stiffness between adjacent elements and a resulting oscillation to the next iteration loop. Chapter 2.4.9 describes the subject of convergence in detail.

The following figure shows the distribution of stiffnesses depending on the moment diagram in a scheme.

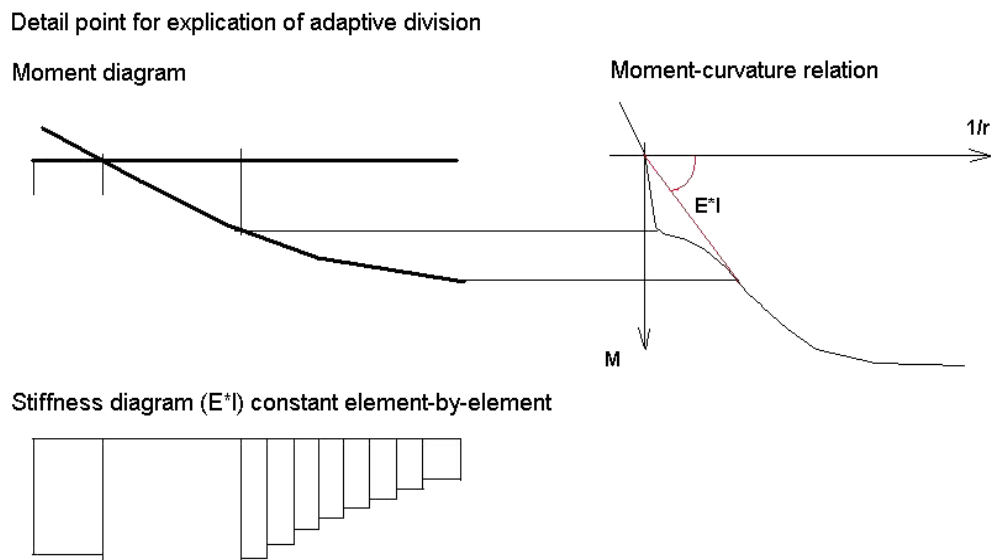


Figure 2.13: Principle of adaptive division by means of a detail point

The following flowchart shows the general process of a non-linear calculation:

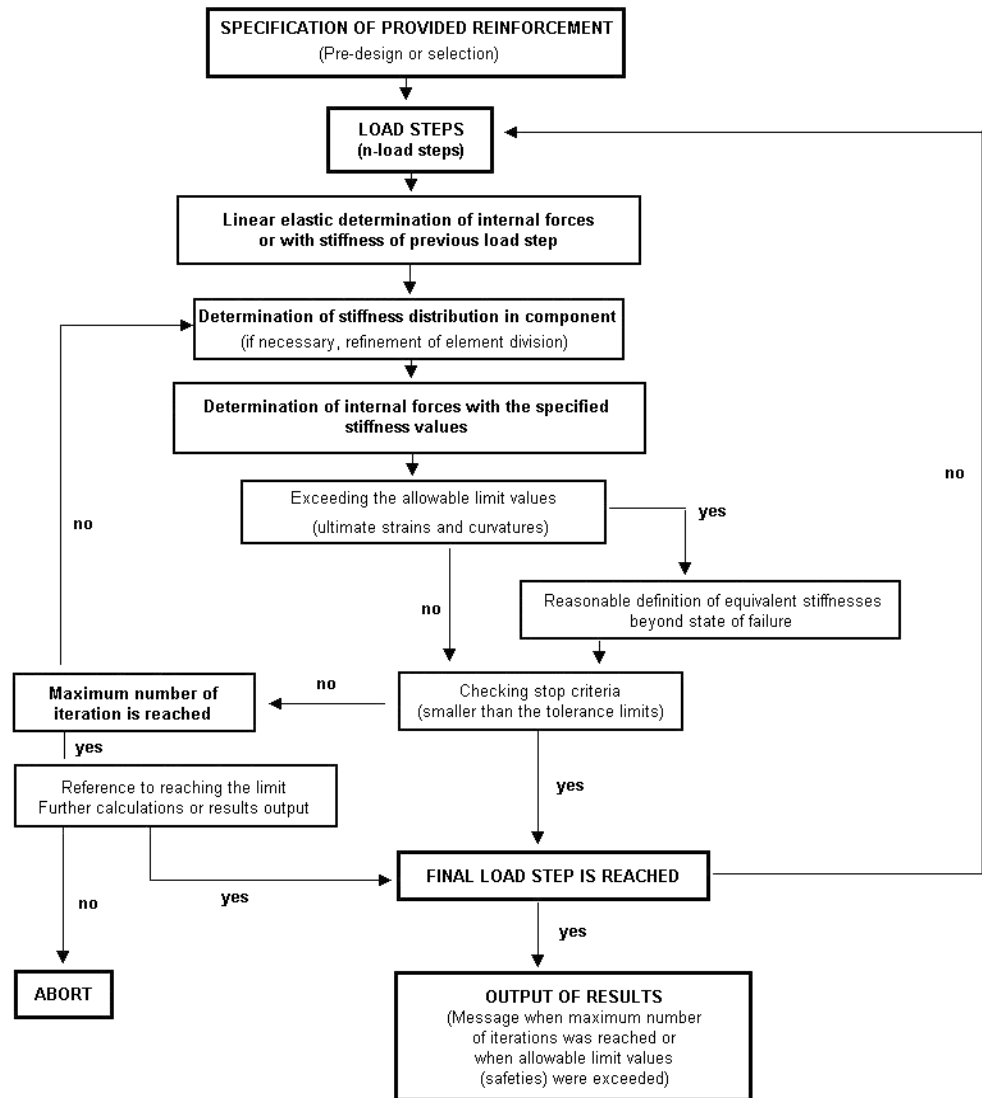


Figure 2.14: Usual calculation process of a non-linear analysis

The individual steps and procedures 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 relation moment-curvature (axial force) mirroring most clearly the stiffness development depending on the action is completely described. This means that the moment-curvature diagram depends on the cross-section's loading due to axial force.

Both chapters 2.4.7.1 and 2.4.8 describe in detail the material properties that are applied to the limit states for bearing capacity and serviceability.

The following essential relations exist between strain and curvature.

Linearized representation

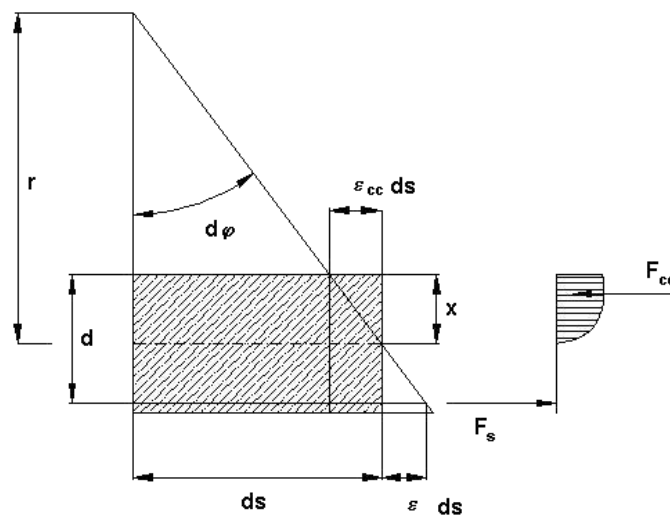


Figure 2.15: Relation between strain and curvature on the infinitesimal element

Referring to the relations represented above, the following conditions can be specified:

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

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

The equation is the following:

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

where ε_{cc} negative for compression strain of concrete

Taking 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 the one of the curvature graph gets lost. The value $E \cdot I$ (flexural resistance of secant) depends on the loading, and thus it is no longer constant where identical geometric boundary conditions are given.

The following figure shows the basic difference between secant stiffening and tangent stiffening.

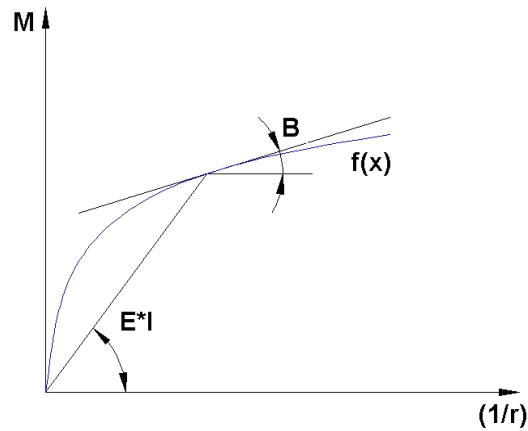


Figure 2.16: Secant stiffening (E^*I) and tangent bending rigidity (B)

When deformations are calculated, the approach depends strongly on the used method. In [29] 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 practicable with regard to the mentioned method or for "manual calculations" when deformations or release rotations are to be determined by the principle of virtual work.

When the finite element method is used, the calculation based on constant equivalent stiffnesses is recommended. In order to determine the non-linear diagram of the cross-section's moment-curvature relation in a sufficiently accurate way even there where abrupt changes of the tangential bending rigidity occur, a finer division is compulsory in such transition zones (M_{cr} , M_y). It is performed in the program's background by limiting the differences in stiffness of adjacent elements.

2.4.3 Tension-Stiffening

We know from the design in the ultimate limit state that tension forces, when parts of the reinforced concrete are cracked, must be absorbed within the crack by the reinforcement only. Between two cracks, however, tension forces are introduced into the concrete across the (displaceable) bond. Thus, relating to the length of the structural component, the concrete is participating in the absorption of internal tension forces, which leads to an increased component stiffness. This effect is called *effectiveness of concrete for tension between cracks* or *Tension Stiffening*.

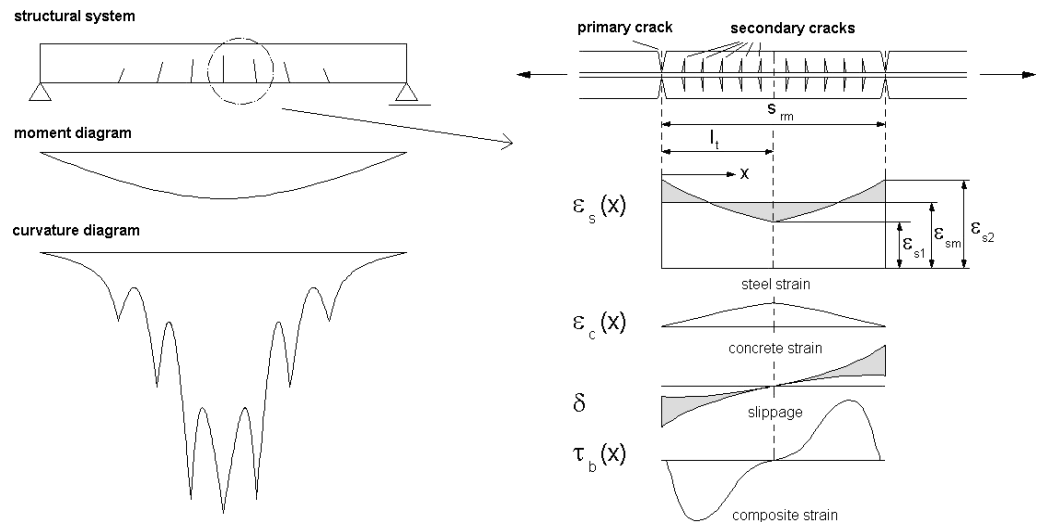


Figure 2.17: Stress and strain behavior between two primary cracks

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

- Subsequent to the crack formation, a constant residual tension stress is represented in the concrete's stress-strain diagram. The residual tensions stress is much smaller than the tensile strength of the concrete. Alternatively, it is possible to establish modified stress-strain relations for the tension zone that take into account the concrete's effect on tension between cracks in the form of a decreasing graph after the tensile strength is reached. Often, this procedure proves to be reasonable for numerical calculations.
- The approach that is more conventional for practical designs and even easier to comprehend is the modification of the "pure" stress-strain diagram of steel. In this case, a reduced steel strain ϵ_{sm} is applied in the relevant cross-section. This strain results from ϵ_{s2} and a reduction term due to the tension stiffening.

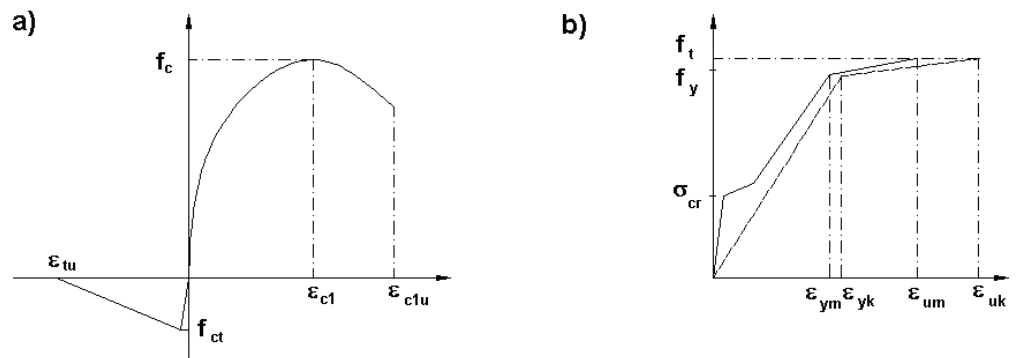


Figure 2.18: Different approaches to consider tension stiffening effect
 a) by characteristic concrete curve b) modified characteristic steel curve

In the add-on module CONCRETE, you can consider the effect of tension stiffening by a modified characteristic curve for steel according to [13] as well as by a stress-strain curve for concrete in the tensile zone according to [11] and [25].

Advantages and disadvantages of the respective approaches as well as the appropriate application of the individual methods are described in detail in corresponding reference books (for example [25]).

2.4.3.1 Model: Tensile Strength of Concrete

This model used to determine the effect of concrete on tension between cracks is based on a defined stress-strain curve of the concrete in the tension zone (parabola-rectangle diagram). The calculational tensile strength is **no** fixed value but refers to the given strain in the decisive steel (tension) fibre. The approach was taken on according to the specifications in [11] that the maximum tensile strength f_{ctR} decreases linearly to zero, starting at the defined crack strain until a strain of 2 ‰ is reached in the governing steel fibre.

As described in [11], the limit value 2 ‰, which is slightly below the actually assumed yield strain, represents an approach that is sufficiently accurate for practical needs. In several research projects (i.a. [25]) efforts were made to improve or modify the approach of QAST and to adjust it on the basis of evaluated experiments.

The following figure shows the approach in a scheme:

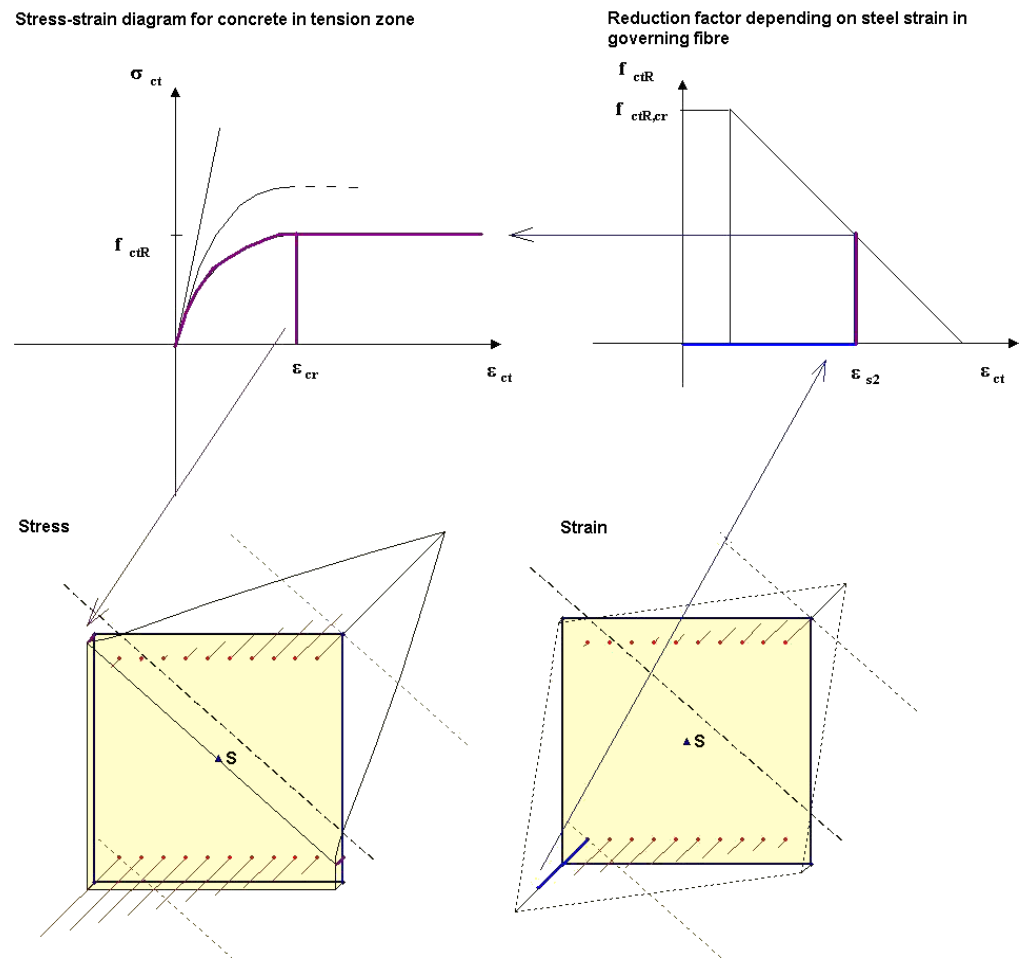


Figure 2.19: Calculation of residual tensile strength for the Tension-Stiffening model according to QAST

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

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

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

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

$$n_{ct} = 1.1 \cdot E_{ctm} \cdot \frac{\varepsilon_{cr}}{f_{ct,R}}$$

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

where:	α_{red}	Reduction factor of basic value of tensile strength
	$f_{ct,grund}$	Basic value of tensile strength (e.g. f_{ctm})
	$f_{ct,R}$	Calculational tensile strength
	v	Ratio of compressive and tensile strength
	ε_{cr}	Calculational strain when $f_{ct,R}$ 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 (simplified in computational model assumed with 2 ‰)
	ε_{s2}	Strain of decisive steel fibre

2.4.3.2 Modified Characteristic Curve for Steel

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

Stress-strain curve of steel

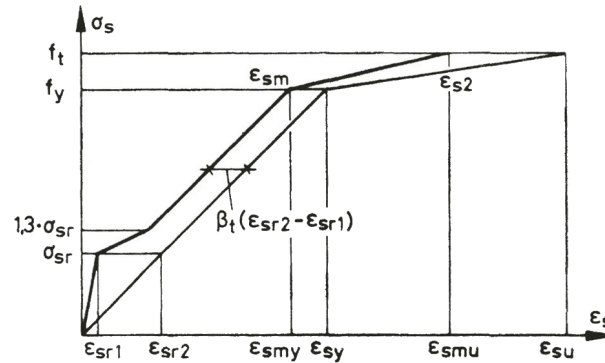


Figure 2.20: Modified stress-strain curve of reinforcing steel from [13]

Explanation

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

$$\epsilon_{sm} = \epsilon_{s1}$$

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

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

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

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

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

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

Descriptions:

ϵ_{sm}	Mean steel strain
ϵ_{su}	Ultimate strain at failure of reinforcing 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 to consider loading period or load repetitions
	0.40 short-term loading
	0.25 permanent loading or frequent load changes
σ_{sr}	Stress in tension reinforcement, calculated on the basis of a cracked section for the load combination that results in the first crack formation
σ_s	Steel stress in cracked state (in crack) in N/mm ²
δ_d	Factor to consider 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 into account the concrete's tension stiffening effect. By means of discrete conditions of strain (curvatures) it is possible to determine a corresponding moment. Normally, based on the ultimate strain at failure, the ultimate curvature is divided, which can vary according to the task. The disadvantage of this approach is that it requires a very fine division in order to represent also the transition zones for significant yield points. By connecting the respective single points we get a continuous (polygonal) line representing the characteristic moment-curvature diagram. Its graphs are also affected by or depending on the axial force that is acting. However, in most practical situations the approach of a moment-curvature relation linearized in particular areas is sufficient.

Depending on the selected approach (double bending, no constant axial force), CONCRETE determines the stiffness on every element node directly from the design force of the previous iteration. One of the differences existing between the two approaches of the Tension-Stiffening method is the following: in the approach according to QUASt the mean stiffness arises directly from the stress calculation. For the approach concerning the modified characteristic curve for steel, it is required to determine once again the mean curvature separately, which may lead to a certain loss of velocity depending on geometry and system.

For compression elements we generally have to use the model from QUASt [11] to consider the concrete's effectiveness. The reason is the simplified calculation in the uncracked state for the model referring to the modified characteristic curve for steel (see chapter 2.4.3.2 and 4.2.2).

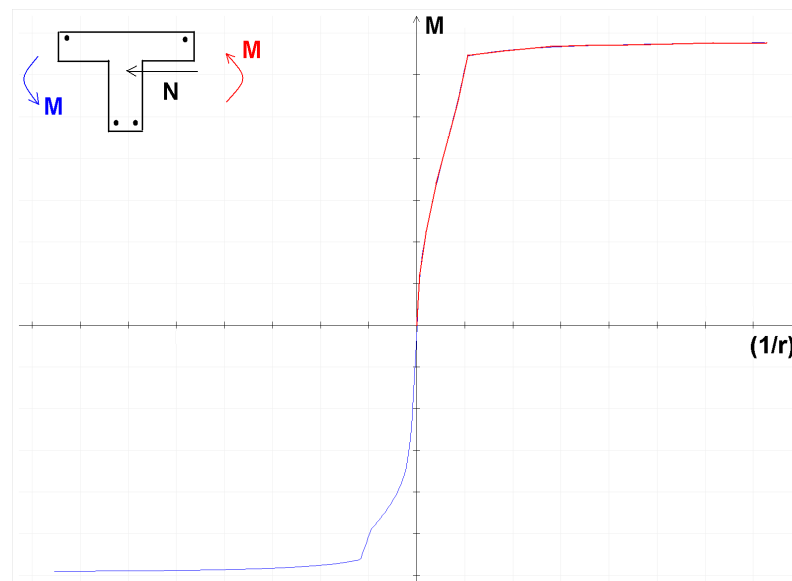


Figure 2.21: Exemplary diagram of a moment-curvature relation

2.4.5 Determination of Element Stiffnesses

2.4.5.1 Flexural Resistance

As described in chapter 2.4, the calculation is based on constant element-by-element equivalent stiffnesses. It is important for this approach to have a very fine division in zones of significant stiffness changes (tearing, yielding). In case the element division is too coarse, it may happen that the stiffness conditions will be misinterpreted for some parts or the stiffnesses will oscillate. Therefore, CONCRETE performs an adaptive member division to prevent these effects. The program also checks the difference in stiffness between the element nodes. If the difference is too large, intermediate nodes will be inserted to minimize the stiffness differences.

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

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

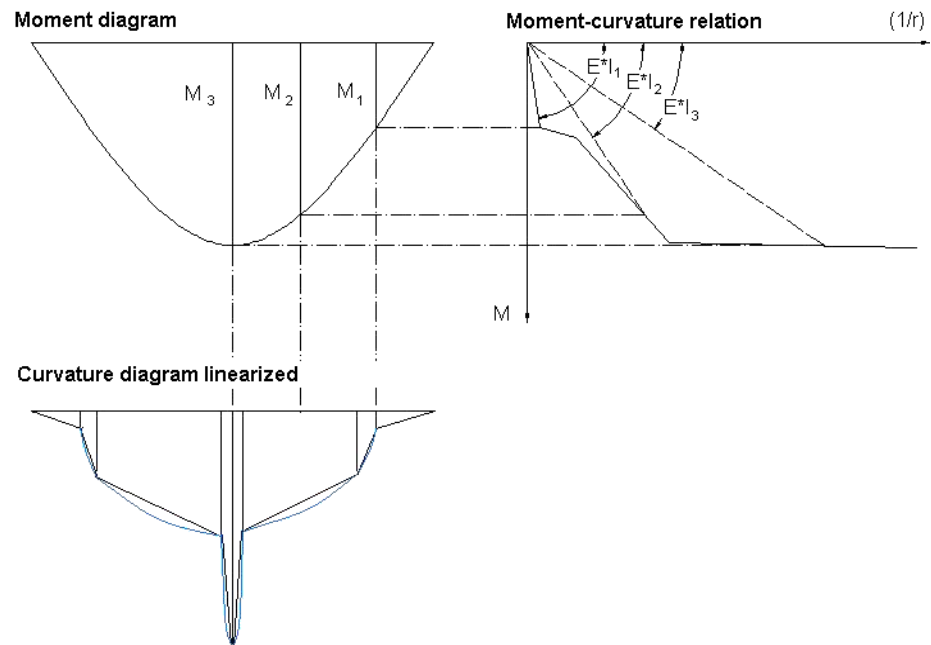


Figure 2.22: Scheme representation of relation between moment, curvature and stiffness

In accordance with the relations described before, the secant stiffening which depends on the action 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

How the flexural resistance is determined as initial value for the non-linear calculation is described in the previous chapters. The remaining stiffness parameters can be determined as follows.

Longitudinal stiffness

The longitudinal stiffness $E \cdot A$ is determined similar to bending from the relation 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 consequently. This results from the simplified analysis not considering the shifting of the neutral axis for strain. For non-linear calculations the axis is not matching the centroid of the cross-section any more. 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 relation between moment and axial force in the terms of the stiffness matrix. CONCRETE does not take into account the axis strain that arises from crack formation or physical non-linearity.

When we look at the relation between axial force and bending moment, we can see a direct relation between both stiffness terms. To make it clear, we can think of an example assuming a column with constant compression force: If an increasing moment is now acting in addition to the axial force, we don't only have a pure constant strain diagram but also a curvature leading to a displacement of the resulting axial force from the centroid. Thus, seen from a plastic point of view, also the effective area of the resultant force is reduced, which leads necessarily to larger strains and so to decreasing stiffnesses. Therefore, the approach providing the consideration for the affinity between bending rigidity and strain stiffness in case of bending with axial force represents a reasonable and practicable solution.

Shear stiffness

Determining the shear stiffness in detail is very difficult for the design of reinforced concrete structures, and with regard to particular geometry and load arrangements it represents a problem that is hard to solve. The beam theory soon reaches the limits because the bearing capacity is supposed to be determined by the truss effect to represent the stiffness for a moderate shear loading. In the past such models were used to develop different methods that in general case are not or only partially sufficient when it comes to their application.

In a simplified method, PFEIFFER [25] reduces the shear stiffness in accord with the available flexural resistance. Even if this approach seems to be strange at first view, it hides a quite simple and plausible idea. We can think of bending stress and shearing stress as independent values. With the modified loading of moment and axial force the flexural resistance changes as well according to the strain and curvature diagram. This does not only affect the stiffness in the beam's longitudinal direction but also in transversal direction used to transfer shear forces.

We should understand this method to be an approach that implies a sufficient shear capacity but does not (or only roughly) determine slanting cracks, increase of tension force etc. In spite of these simplifications we can consider the method according to PFEIFFER for moderately slender beams to be a sufficiently accurate approach. Alternatively, we can also take the linear elastic shear stiffness as a basis for the calculation in CONCRETE.

Torsional stiffness

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

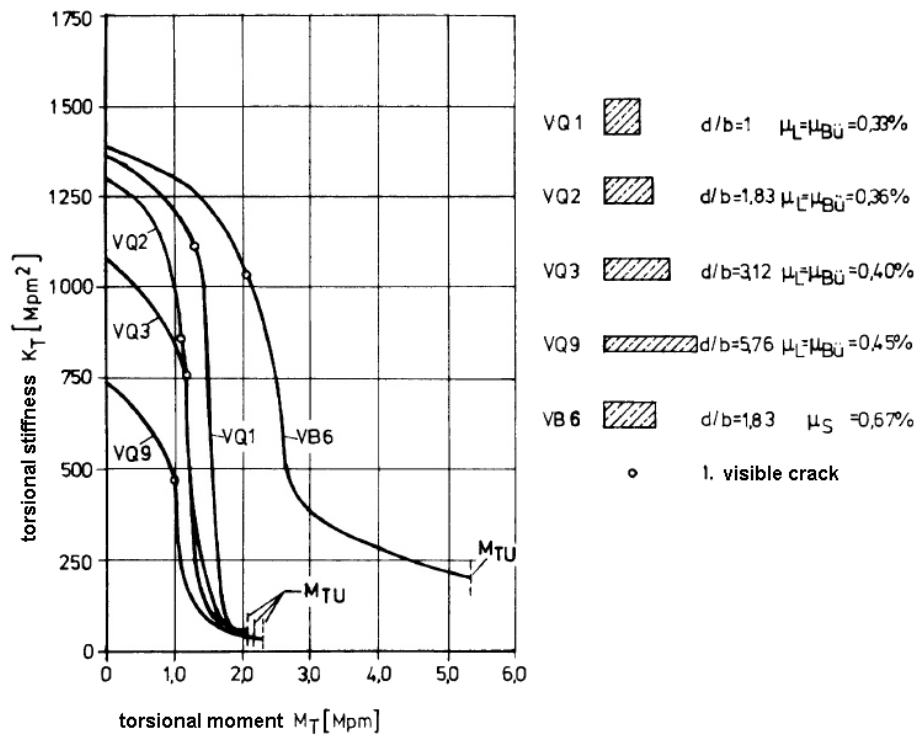


Figure 2.23: Scheme from LEONHARDT [22] about decrease of torsional stiffness in case of cracking

The CONCRETE calculation provides two different approaches to consider torsional stiffness.

- **Torsional stiffness according to LEONHARDT [22]**

Torsional stiffness in uncracked sections (state I)

For the torsional stiffness in uncracked sections (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. To some extent a micro crack formation is also involved in the reduction.

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

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 concrete compression strut below 45°. According to LEONHARDT this assumption is true when the ratio of the longitudinal reinforcement and the one of the transverse reinforcement are not equal. Minor strut inclinations arise 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 us that the planer inclination of cracks that we assume only occurs for high stress.

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

Link inclinations of 90°:

$$(G_c \cdot I_T(x))_{II} = \frac{4 \cdot E_c \cdot A_k^3}{u_k^2} \cdot \frac{1}{k_T(1/\mu_L + 1/\mu_{B\ddot{u}}) + \frac{4 \cdot \alpha \cdot A_k \cdot (1 + \varphi)}{u_k \cdot t}}$$

Link inclinations of 45°:

$$(G_c \cdot I_T(x))_{II} = \frac{E_c \cdot A_k^2 \cdot t}{u_k} \cdot \frac{1}{k_T/\mu_{B\ddot{u}} + \frac{\alpha}{4} \cdot (1 + \varphi)}$$

where $k_L = 1 - \frac{T_{Ed} - 0.7 \cdot T_{cr}}{T_{Rd,sy} - 0.7 \cdot T_{cr}}$ for 90° inclination of concrete strut

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

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

$\mu_{B\ddot{u}} = \frac{a_{sw} \cdot u_k}{A_k}$ ratio of transverse reinforcement related to kern

$$T_{Rd,sy} = \min \left\{ \begin{array}{l} A_{sw}/s_w \cdot f_y \cdot 2 \cdot A_k \\ A_{sl}/u_k \cdot f_y \cdot 2 \cdot A_k \end{array} \right.$$

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 the steel stress in the truss model reaches the yield point (torsional moment that can be absorbed)

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

$$T_{Rd,sy} = \min \left\{ \begin{array}{l} W_T f_{ctr1} \\ 2 \cdot A_k \cdot t \cdot f_{ctr1} \end{array} \right.$$

A_k Area enclosed by center line of walls

A_{sl} Cross-sectional area of longitudinal reinforcement

A_{sw} Cross-sectional area of link reinforcement

α Ratios of moduli of elasticity E_s/E_c

u_k Perimeter of area A_k

s_w Link spacing

t Effective thickness of wall

φ Factor for creeping to consider

A mutual influence of torsional stiffness and flexural resistance is not effected.



- **Global reduction of torsional stiffness**

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

2.4.6 Creep and Shrinkage

2.4.6.1 Determination of Initial Values

This chapter gives an overview about the time-dependent stresses and strains due to creeping and shrinkage.

Creeping represents the time-dependent deformation of concrete stressed by loading within a specific period of time. The essential influence values are similar to the values of shrinkage. But additionally the so-called creep-producing stress has 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 represented by the creep coefficient $\varphi(t, t_0)$ at the relevant point of time t .

Shrinkage describes a time-dependent modification of volume without effects due to external loads or temperature. The present documentation avoids describing further explanations concerning shrinkage problems and their individual types of appearance (drying shrinkage, autogenous shrinkage, plastic shrinkage and carbonation shrinkage). Significant influence values of shrinkage are the relative humidity, the effective thickness of structural components, the aggregate, the concrete strength, the water-cement ratio, the temperature as well as the type and duration of curing. The shrinkage determining value is represented by the shrinkage strain $\varepsilon_{c,s}(t, t_s)$ at the relevant point of time t .

Before the manual describes how creeping and autogenous shrinkage are taken into account by a non-linear calculation, it presents the determination of the creep coefficient $\varphi(t, t_0)$ and the shrinkage strain $\varepsilon_{c,s}(t, t_s)$ according to EN 1992-1-1, section 3.1.4 and Annex B.

Creep coefficient $\varphi(t, t_0)$

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

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

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

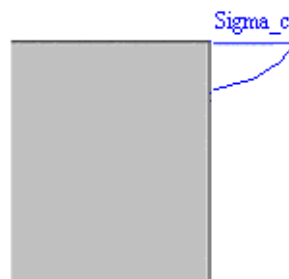


Figure 2.24: Creep-producing stress

In case a linear creep behavior ($\sigma_c \leq 0.45f_{ckj}$) is assumed, the concrete's creep can be determined by a reduction of the elastic modulus for concrete (see equation on the following page).

$$E_{c,eff} = \frac{1.1 \cdot E_{cm}}{1.1 + \varphi(t, t_0)}$$

where	E_{cm}	Mean elastic modulus according to EN 1992-1-1, table 3.1
	$\varphi(t, t_0)$	Creep coefficient
	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 $\varphi(t, t_0)$ at the analyzed point of time t may be calculated as follows:

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

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

RH Relative humidity in [%]

h_0 Effective thickness of structural component [mm]

$$h_0 = \frac{2 \cdot A_c}{u}$$

A_c Cross-sectional area

u Cross-section perimeter

α_1, α_2 Coefficients to consider influence of concrete strength

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

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

f_{cm} Mean value of compressive cylinder strength

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

f_{cm} Mean value of compressive cylinder strength of concrete in [N/mm²]

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

t_0 Age of concrete in days when load application starts

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

t Age of concrete in days at relevant point of time

t_0 Age of concrete in days when load application starts

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

RH Relative humidity [%]

h_0 Effective thickness of structural component [mm]

α_3 Coefficient to consider influence of concrete strength

$$\alpha_3 = \left(\frac{35}{f_{cm}} \right)^{0.5} \leq 1.0$$



The following input is required to calculate the creep coefficient:

- RH Relative humidity [%]
- t_0 Age of concrete in days when load application starts
- t Age of concrete in days at relevant point of time

The influence of high or low temperature on the concrete's curing degree in a range of 0 °C to 80 °C can be taken into account by a correction of the age of concrete 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 same temperature
 - $T(\Delta t_i)$ Temperature in °C during period Δt_i
 - Δt_i Number of days where temperature T prevails

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

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

- 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

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

Example

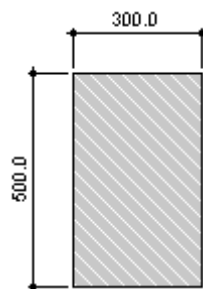


Figure 2.25: Cross-section

Concrete C25/30
 Cement CEM 42.5 N
 RH: 50%
 Two changes in temperature:

Duration	Temperature
6 days	15 °C
8 days	7 °C

Considered age of concrete t_k : 365 days

Age of concrete when creeping starts:

$$t_T = \sum_{i=1}^n e^{-\left[\frac{4000}{273+T(\Delta t_i)} - 13.65\right]} \cdot \Delta t_i = e^{-\left[\frac{4000}{273+15} - 13.65\right]} \cdot 6 + e^{-\left[\frac{4000}{273+7} - 13.65\right]} \cdot 8 = 8.96 \text{ days}$$

Age of concrete under influence of type of cement:

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

Effective thicknesses of structural component:

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

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

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

$$\alpha_1 = \left(\frac{35}{f_{cm}} \right)^{0.7} = \left(\frac{35}{33} \right)^{0.7} = 1.042 \quad \alpha_2 = \left(\frac{35}{f_{cm}} \right)^{0.2} = \left(\frac{35}{33} \right)^{0.2} = 1.012$$

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

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

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

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

$$\beta_H \leq 1500 \cdot \alpha_3 = 1500 \cdot 1.030 = 1545$$

$$\alpha_3 = \left(\frac{35}{33} \right)^{0.5} = 1.030$$

Shrinkage Strain $\varepsilon_{c,s}(t, t_s)$

Total shrinkage is subdivided into the components of autogenous shrinkage and drying shrinkage. Then shrinkage can be defined by specifying the shrinkage strain $\varepsilon_{c,s}(t, t_s)$ as follows:

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

where $\varepsilon_{cd}(t, t_s)$ Drying shrinkage strain
 $\varepsilon_{ca}(t)$ Autogenous shrinkage strain

Drying shrinkage is calculated from

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

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

t Age of concrete in days at relevant point of time
 t_s Age of concrete in days at the beginning of drying shrinkage
 h_0 Effective thickness of structural component in [mm]

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

f_{cm} Mean value of compressive cylinder strength of concrete in [N/mm²]
 $f_{cm0} = 10 \text{ N/mm}^2$

Type of cement	α_{ds1}	α_{ds2}
Class S	3	0.13
Class N	4	0.12
Class R	6	0.11

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

RH Ambient relative humidity in %
 $RH_0 = 100 \%$

h_0 [mm]	k_h
100	1.0
200	0.85
300	0.75
≥ 500	0.70

Autogenous shrinkage is calculated from

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

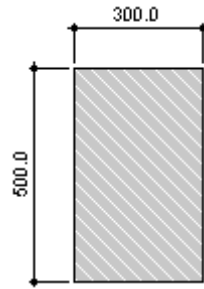
$$\text{where } \beta_{as}(t) = 1 - e^{-0.2t^{0.5}}$$

t Age of concrete in days at relevant point of time

$$\varepsilon_{ca}(\infty) = 2.5 (f_{ck} - 10) \cdot 10^{-6}$$

f_{ck} Characteristic value of compressive strength of concrete in [N/mm²]

Example



Concrete C25/30
 Cement CEM 42.5 N
 RH: 50%

Figure 2.26: Cross-section

Effective thicknesses of structural component:

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

Autogenous shrinkage strain

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

where

$$\beta_{as}(t) = 1 - e^{-0.2t^{0.5}} = 1 - e^{-0.2 \cdot 365^{0.5}} = 0.978$$

$$\epsilon_{ca}(\infty) = 2.5(f_{ck} - 10) \cdot 10^{-6} = 2.5(25 - 10) \cdot 10^{-6} = 0.0000375$$

Drying shrinkage strain

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

where

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

$$\begin{aligned} \epsilon_{cd,0} &= 0.85 \left[(220 + 110 \alpha_{ds1}) e^{-\alpha_{ds2} \frac{f_{cm}}{f_{cm0}}} \right] \cdot 10^{-6} \beta_{RH} = \\ &= 0.85 \left[(220 + 110 \cdot 4) e^{-0.12 \frac{33}{10}} \right] \cdot 10^{-6} \cdot 1.356 = 0.000512 \end{aligned}$$

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

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

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

Shrinkage strain

$$\epsilon_{cs}(t, t_s) = \epsilon_{cd}(t, t_s) + \epsilon_{ca}(t) = 0.0000367 + 0.000341 = 0.000378 = 0.378 \text{ ‰}$$

2.4.6.2 Consideration of Creep/Shrinkage by Calculation

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

Creep

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 condition 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 are rising disproportionately, resulting in loss of the linearly assumed reference.

For the calculation in CONCRETE the program uses a common and reasonable solution intended for practical construction purposes: The stress-strain curve of concrete is distorted by the factor $(1 + \varphi)$.

Distortion of stress-strain curve taking into account creeping

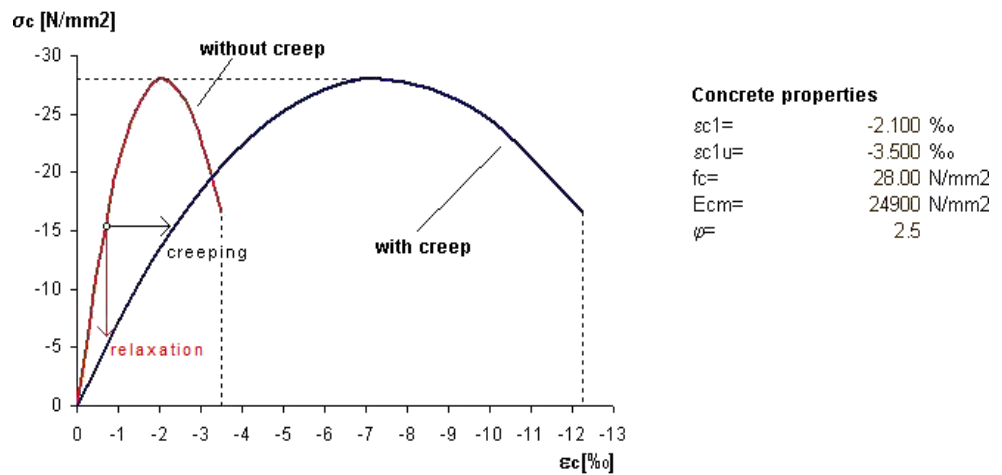


Figure 2.27: Distortion of stress-strain curve for determination of creep effect

When taking into account creeping, as shown in the figure above, uniform creep-producing stresses are assumed during the period of load application. 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 change in strains (relaxation) is not taken into account: In case a linear elastic behavior is assumed, it would be possible to imply a proportionality and the horizontal distortion would reflect the relaxation at a ratio of $(1 + \varphi)$. This context, however, is lost for the non-linear stress-strain diagram.

Thus, this procedure represents an approximation. A reduction of stresses due to relaxation as well as non-linear creep cannot be or can be only approximately represented.

The creep coefficient φ_t that is applied in CONCRETE must be considered as 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 2.4.6.1 must be adjusted as shown in the following equation.

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

Shrinkage

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

Therefore, the shrinkage can be represented by a pre-strain of the concrete or the steel. This means in detail that the "free strain" of the steel is restrained by a positive pre-strain of the concrete. In the same way, it would be possible to model the component with a negative pre-strain of the steel so that the concrete restrains the free strain of the pre-strained steel. Both models show an identical stress distribution, taking into account the relevant pre-strain, but they differ significantly in the plane of strain: When the steel has been pre-strained, it is immediately evident from the strain's condition where zones of tension and compression due to shrinkage occur. When the concrete has been pre-strained, it is possible to make statements from the strain's condition concerning the concrete's actual contraction.

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



The add-on module CONCRETE takes into account strain due to shrinkage by negative pre-strain of the reinforcing steel.

2.4.7 Ultimate Limit State

A non-linear analysis in the ultimate limit state is used to determine the limit of failure (mechanism) close to reality. The design involves the following difficulty: Realistic estimations require realistic initial and computational parameters.

The material properties are no 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 to determine deformations and internal forces.

Another important point when we determine the behavior of structural components close to reality 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, 5.7 non-linear methods must be used leading to realistic stiffnesses and considering uncertainties concerning failing. Design methods that are valid in the governing application areas are allowed to be used. An appropriate non-linear method for the determination of internal forces including cross-section design is represented by the approach with the average values of the material properties and the application of a global partial safety factor γ_r , which can be found in both the national annex for Germany referring to EN 1992-1-1, 5.7 and the German standard DIN 1045-1, 8.5. This approach is described in the following with regard to the *method according to EN 1992-1-1, 5.7*.

According to EN 1992-1-1, 5.7 (5), we can apply for structural components for which it is not allowed to ignore effects according to the second-order analysis the approach according to EN 1992-1-1, 5.8.6.

CONCRETE provides both non-linear methods of calculation described above (cf. Figure 4.2, page 81).

2.4.7.1 Material Properties

Method according to EN 1992-1-1, 5.7

The design according to EN 1992-1-1, 5.7 is based on mean material properties that have been calibrated to realize a global safety factor. As a result, we get a reduced concrete compressive strength representing a controversial subject because of the distortion of the averaged characteristic curve for concrete.

Calculational average values of the material stiffnesses

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

$$f_{yR} = 1.1 * f_{yk}$$

$$f_{tR} = 1.08 * f_{yR} \quad \text{Reinforcing steel - high ductility}$$

$$f_{tR} = 1.05 * f_{yR} \quad \text{Reinforcing steel - normal ductility}$$

$$E_s = 200,000 \text{ N/mm}^2 \quad \text{Modulus of elasticity for steel}$$

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

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

$$E_{cm} \quad \text{Mean modulus of elasticity (secant)}$$

We can see the following relation between the global safety factor γ_R and the mean strength of materials:

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

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

The following figure shows us how the reduced concrete compressive stress f_{cR} is represented with the calculational mean values in comparison to the concrete's stress-strain diagram. The strong distortion of the characteristic concrete curve is clearly identified. It results in an overestimation of strains, particularly in areas that are highly utilized, and thus it leads to overestimated curvatures.

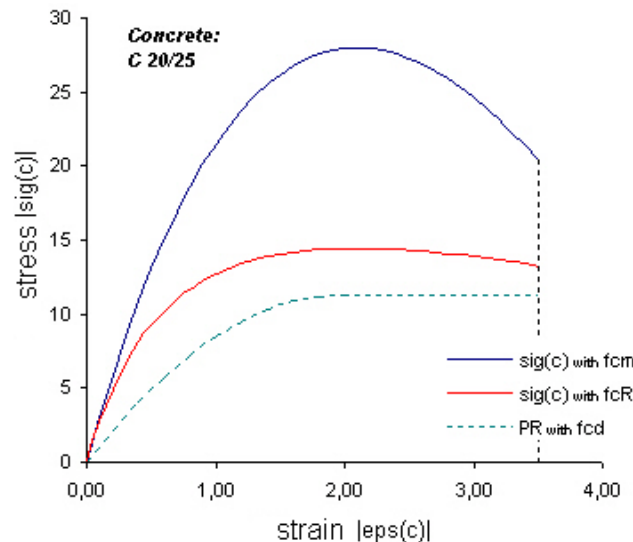


Figure 2.28: Stress-strain relation for internal forces and deformation analysis

When we look at the concrete's characteristic values, we see the following: though the theory is based on reduced stresses ($0.85 * \alpha * f_{ck}$), the modulus of elasticity corresponds to the mean value in accordance with EN 1992-1-1, 3.1.5.

Method according to EN 1992-1-1, 5.8.6

Section 5.8.6 of the Eurocode standard describes the non-linear calculation of structural components prone to instability risks. According to EN 1992-1-1, 5.8.6 (3) the stress-strain curves must be defined on the basis of design values.

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

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

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

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

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

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

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

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

2.4.7.2 Safety Design

Method according to EN 1992-1-1, 5.7

According to EN 1992-1-1, 5.7 we have to design the safety of non-linear calculations by means of a global safety factor γ_R . We can do this by modifying the mean stiffnesses of structural components (f_{cR} , f_{yR} etc.), though it is disputed. 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 $\gamma_R = 1.3$ (or 1.1 for extraordinary combinations of action).

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

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

where

E_d Design value of governing combination of action

R_d Design value of design resistance

γ_R Uniform partial safety factor on side of ultimate load

CONCRETE calculates with a γ_R -fold loading. 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.

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

In this way, also the reduction of imposed internal forces is determined sufficiently.

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 an easier handling as well as to economy of time when calculating because the determination of internal forces and the design are performed in one fell swoop.

The disadvantage will only be visible when we assume that the terms

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

are compatible. For non-linear calculations, of course, it is not given without restrictions. An

example showing that such an approach can be quite on the unsafe side represents the consideration of imposed internal forces. The use of material properties that are 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 make clear the problem of the reduced elastic modulus for steel.

The direct reduction of the stiffnesses is described in detail by QUASt [28] and is evaluated critically with regard to slender compression elements.

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

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

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

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

When we set $\sigma_{sR} = E_s * \varepsilon$, 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 quite alright to calculate with the reduced stiffnesses. In this case, the diagram of internal forces is affected anyway by the relation of the stiffnesses from different areas to each other.

However, the concept shows problems when compression elements are designed 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 are generally failing when the yield strain is reached in the reinforcement. So it is obvious that the deformations are overestimated because of the reduced modulus of elasticity and the larger curvatures resulting when the yielding starts. This leads to a smaller allowable load for the column or the reinforcement must be increased accordingly. QUASt [28] sees no reason for that.

Method according to EN 1992-1-1, 5.8.6

According to EN 1992-1-1, 5.8.6 (3) it is possible to perform the design for the sufficient structural safety directly on the basis of design values (f_{cd} , f_{yd} , ...) of the material properties. To determine the internal forces and deformations, the stress-strain curves which are also defined on the basis of design values have to be used in accordance with section (3).

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

According to the national annex for Germany EN 1992-1-1, 5.8.6 (NDP 5.8.6 (3)) it is allowed to determine the internal forces and deformations by means of average material properties (f_{cm} , f_{ctm} , ...). However, the design for the ultimate load capacity in the governing cross-sections must be performed with the design values (f_{cd} , f_{yd} , ...) of the material properties.

This "inconsistent" approach was disputed among experts. The problem 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 absorbed with the design values to be applied for the design. Increasing the reinforcement results in an increase of stiffness for the respective parts and areas, which requires again an increase of reinforcement in the subsequent iteration step. In addition, it is important to know 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) won't reach the value of the yield moment M_y (mean material properties).

CONCRETE performs the safety design according to the standard by contrasting provided reinforcement with required reinforcement determined for the design values of the material properties. This must always be observed when manual corrections are made to the reinforcement (keyword: increase of stiffness).

2.4.8 Serviceability Limit State

With the standards DIN 1045-1 and EN 1992-1-1 detailed designs for the serviceability limit state find their ways into more engineering offices.

The serviceability limit state is divided into three groups:

- Limitation of **stresses** (DIN 1045-1, 11.1 or EN 1992-1-1, 7.2)
- Limitation of **crack widths** (DIN 1045-1, 11.2 or EN 1992-1-1, 7.3)
- Limitation of **deformations** (DIN 1045-1, 11.3 or EN 1992-1-1, 7.4)

In the following, only the limitation of the deformation is described, taking into account the influence of creep and shrinkage.

The reason for the detailed description of deformations can be found again in the non-linear behavior of reinforced concrete as a composite material. As a result of crack formation, the stiffness is reduced significantly in particular areas compared to state I (uncracked sections). If the cracking is not taken into account, occurring deformations will be underestimated. By considering creeping and shrinkage, the deformation may be 3 to 8 times larger, depending on the stress and boundary conditions.

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

The correct interpretation of results from non-linear calculations requires the knowledge of the most important factors. Therefore, we compare the most important parameters affecting the stiffnesses in uncracked sections (state I) and cracked sections (state II) in the table below:

Influencing value	State I	State II
Creeping (here as reduction of the elastic modulus for concrete)	The stiffness is controlled mainly by the concrete. Thus, a reduced modulus of elasticity leads to a considerable reduction of stiffness.	Minor influence
Reinforcement ratio	Minor influence (reasons see <i>Creeping</i>)	The stiffness in state II is controlled mainly by the reinforcement. Therefore, there is a major influence.
Axial force	Influence hardly given (In case of simplified linear-elastic analysis there is no influence at all.)	A tensile force is reducing the stiffness considerably. Where necessary, this fact must be considered for the modeling of shrinkage as it leads to tension stresses within the concrete.

Table 2.3: Influence values and their impact in the 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 be taken into account as well by appropriate approaches (see chapter 2.4.3) because otherwise a realistic deformation analysis won't be possible.

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

Calculational average values of the material stiffnesses

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

$$f_y = f_{yk}$$

$$f_t = f_{yk} \quad \text{for serviceability considerations}$$

$$E_{sm} = 200,000 \text{ N/mm}^2 \quad \text{mean modulus of elasticity for steel}$$

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

$$f_{cm} \quad \text{mean concrete compressive strength}$$

$$E_{cm} \quad \text{mean modulus of elasticity (secant)}$$

2.4.9 Convergence

How fast and safe a non-linear calculation is converging depends on a variety of facts and can be specified for the general case only by trend.

Main-starting point of the convergence evaluation represents 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 determining an iterative improvement by means of secant stiffenings. However, secant methods are generally more stable seen from a numerical point of view, especially in the area of very flat gradients near the limit of failure (tangential stiffness runs to zero). Of course it is not possible to find a generalization because the convergence is affected by incremental load application, various iteration methods (NEWTON-RAPHSON, RIKS/WEMPNER/WESSELS etc.) and many other parameters.

In the following, the convergence behavior of the used algorithm is presented shortly. CONCRETE performs the actual iteration of the state of strain on the cross-section level. This means: based on a diagram of internal forces within an iteration loop more and more new, current strain-stress conditions are calculated. The convergence is reached when a state of equilibrium is established, i.e. the diagram of internal forces in two successive iteration steps remains within a given threshold.

This method performed exclusively 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, which cannot be prevented even by adaptive division. To avoid the non-convergence, a damped stiffness reduction was implemented into the calculation. The change between the stiffnesses of two iteration steps will be damped according to user specifications. Thus, the calculation slows down a little but it is numerically more stable. Finally, we know that a damping for statically determinate systems makes no sense.

Thus, the two controllable stop criteria of non-linear calculations are the following:

$$\varepsilon_1 = |(1/\gamma)_i - (1/\gamma)_{i-1}| \leq \text{Tolerance 1}$$

γ is an indicator for the ratio of ultimate moment and acting moment.

In this way, the break-off criterion ε_1 takes into account the change of internal forces.

$$\varepsilon_2 = (EI_i - EI_{i-1})^2 / (EI_i)^2 \leq \text{Tolerance 2}$$

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 in the program's background:

$$\varepsilon_3 = |u_i - u_{i-1}| \leq \text{Tolerance 3 (fix)}$$

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



If the non-linear calculation is not converging, the dialog box *Settings for Non-linear Calculation* offers you several possibilities to improve the convergence behavior.

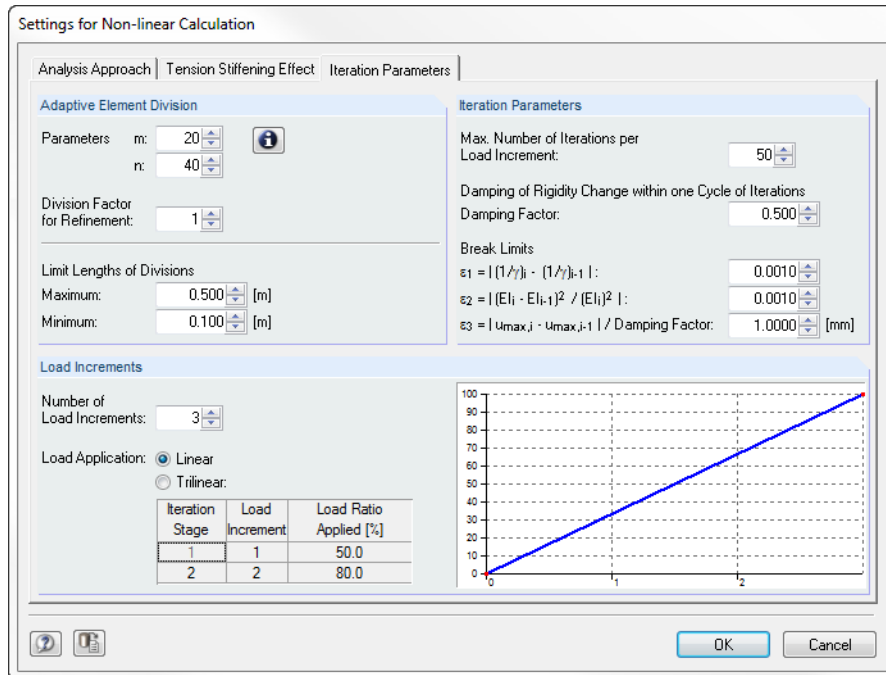


Figure 2.29: Dialog box *Settings for Non-linear Calculation*, tab *Iteration Parameters*

Increasing the number of iterations

The iteration process strongly depends on the cross-section shape, the structural system and the loading. This may lead to a different convergence behavior. Generally, structural components that are highly stressed by compression are converging a bit more slowly. As the current deviations ϵ_1 and ϵ_2 are displayed permanently during the calculation, we 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. This may result in a very large difference in stiffness in the first iteration loop when the calculation is done with one load step only. In this case, it could be useful to apply the load step-by-step.

Limit length of member division and adaptive element division



With this option it is possible to influence the stiffness diagram efficiently. As CONCRETE calculates with constant mean stiffnesses within a finite element, it is advisable to select a finer division of elements where major changes in stiffness occur (see Figure 2.13, page 24). With an adaptive division CONCRETE controls the stiffness differences on the end nodes of an element. If the specified tolerance limit will be exceeded, intermediate divisions will be inserted additionally. To access more details about the functionality, click the [Info] button that is directly available in the dialog box.

Reduction of damping factor

By a specific reduction of stiffness changes between two iteration steps it is possible to work against the calculation's oscillation. In case of two successive iteration steps, the program determines the stiffness difference 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.

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

This means: The higher the damping factor, the smaller the damping's influence. If the factor is 1, the damping has no influence on the iterative calculation.

3. Input Data



All data required for the definition of design cases is entered in tables. The [Pick] function allows for a graphical selection of the objects that you want to design.

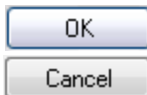
When you have started the add-on module, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently. The pull-down list above the navigator contains the design cases that are already available (see chapter 8.1, page 131).

When you open CONCRETE in an RSTAB structure for the first time, the module will import the following design relevant data automatically:

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



To select a table, click the corresponding entry in the CONCRETE navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.



To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the add-on module but without saving the data.

3.1 General Data

In table 1.1 *General Data*, you select the actions that you want to design. The relevant load cases, groups and combinations can be assigned to the ultimate limit state, the serviceability limit state and the fire protection design by using the respective tab.

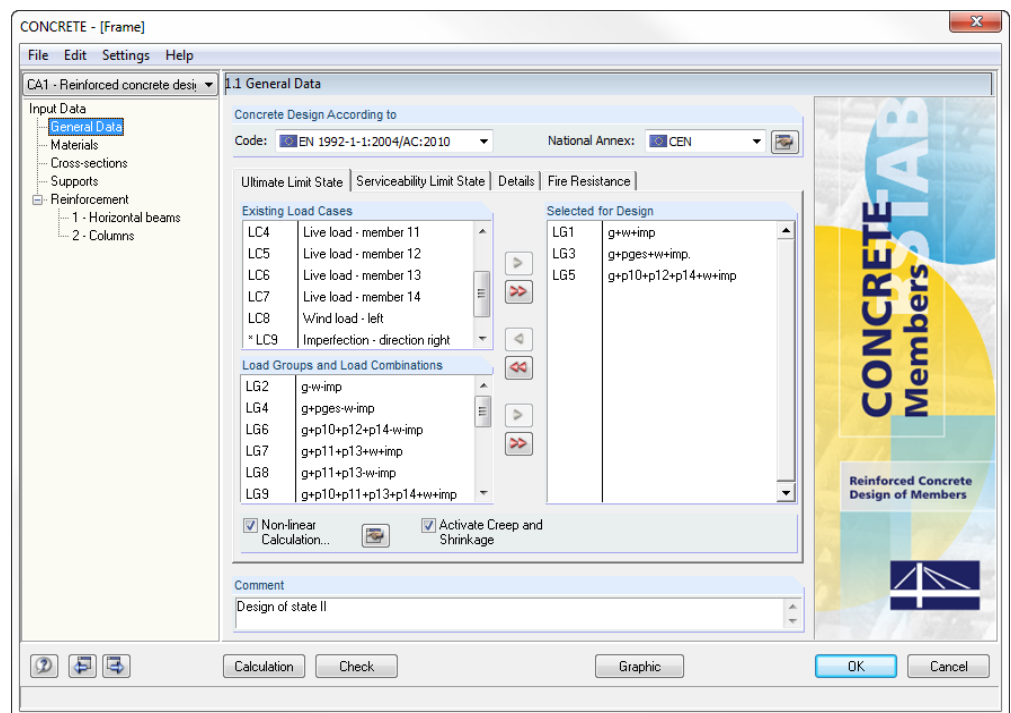


Figure 3.1: Table 1.1 *General Data*, tab *Ultimate Limit State*



Concrete Design According to Code / National Annex

The design standard is defined uniformly for all types of design available in this table. The following standards for reinforced concrete design can be selected.

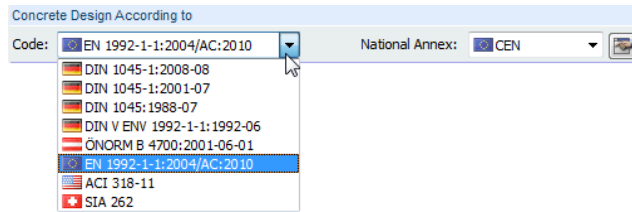


Figure 3.2: Selection of design standard

When EN 1992-1-1:2004 is set, you can select the *National Annex* from the list to the right.



Figure 3.3: Selection of national annex



Use the [Edit] button to check the factors of the selected national annex. If necessary, you can adjust the coefficients.

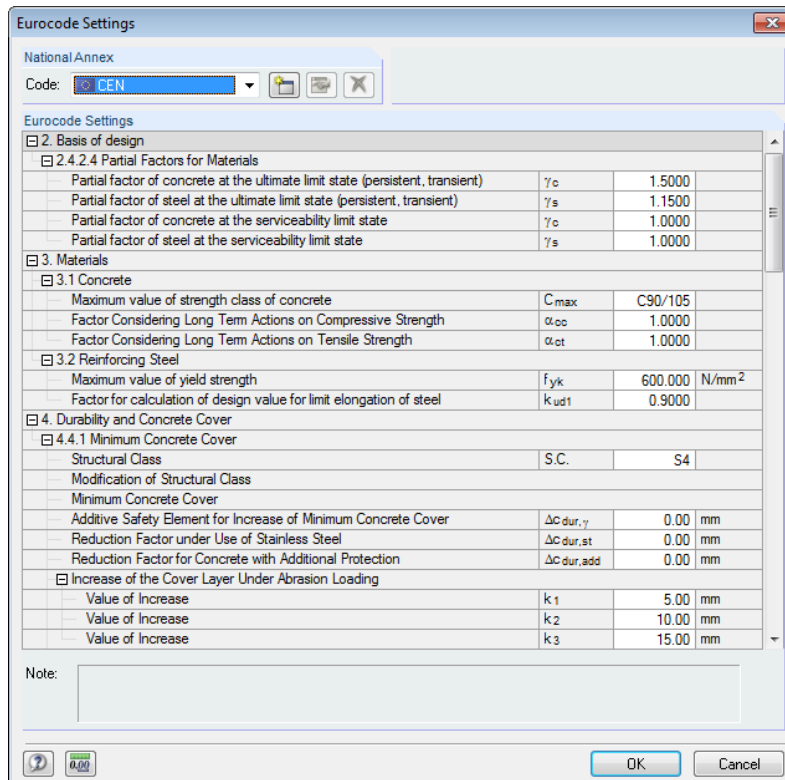


Figure 3.4: Dialog box Eurocode Settings



To create a copy of the annex currently selected, click the [New] button in the dialog box *Eurocode Settings*. When you have changed the parameters, you can save the copy using another name. The user-defined parameters are then globally available for all structures in the *National Annex* list.

Name	Country
Code: CEN (copied)	Sicily

Figure 3.5: Creating a user-defined national annex

3.1.1 Ultimate Limit State

The first tab of table 1.1 *General Data* is shown in Figure 3.1 on page 52.

Existing Load Cases / Load Groups and Load Combinations



These two dialog sections list all actions and load combinations defined in RSTAB that are relevant for the design. Use the [▶] button to transfer the selected load cases, groups or combinations as well as super combinations to the list *Selected for Design* on the right. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].

If a load case is marked by an asterisk (*) like load case 9 in Figure 3.1, it is not possible to calculate it. This may be the case when no loads are defined or, as you can see in the example, the load case contains only imperfections.

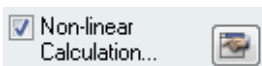
Selected for Design



The column on the right lists the loads selected for the design. Use the button [◀] to remove selected load cases, groups or combinations from the list. You can also double-click the entries. With the button [◀◀], you can transfer the entire list to the left.

The analysis of an enveloping *Or* load combination is often carried out more quickly than the design of all load cases and groups that have been globally set. On the other hand, the influence of the actions contained in a load combination are less transparent for a CO design.

Non-linear Calculation

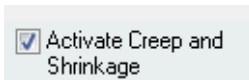


To perform the ultimate limit state design according to the non-linear design method, tick the check box below the list *Load Groups and Load Combinations*. The non-linear approach is described in the theory chapter 2. Chapter 2.4.7 on page 44 presents the non-linear analysis for the ultimate limit state in detail.



A non-linear analysis performed according to DIN 1045-1 or EN 1992-1-1 can be carried out only for load cases and load groups but not for load combinations. The analysis is possible for both the ultimate and the serviceability limit state, combined or separately.

To open the dialog box *Settings for Non-linear Calculation*, use the [Edit] button. The dialog box provides three tabs that are described in chapter 4.2, page 81.



Furthermore, table 1.1 offers you the option to *Activate Creep and Shrinkage*. For more information, see chapter 2.4.6 on page 37. When the check box is ticked, you can specify the modulus of creep and the shrinkage strain in table 1.3 *Cross-sections* (see Figure 3.13, page 62).

Comment

In this input field, you can enter user-defined notes, for example to describe in detail the current design case.

3.1.2 Serviceability Limit State

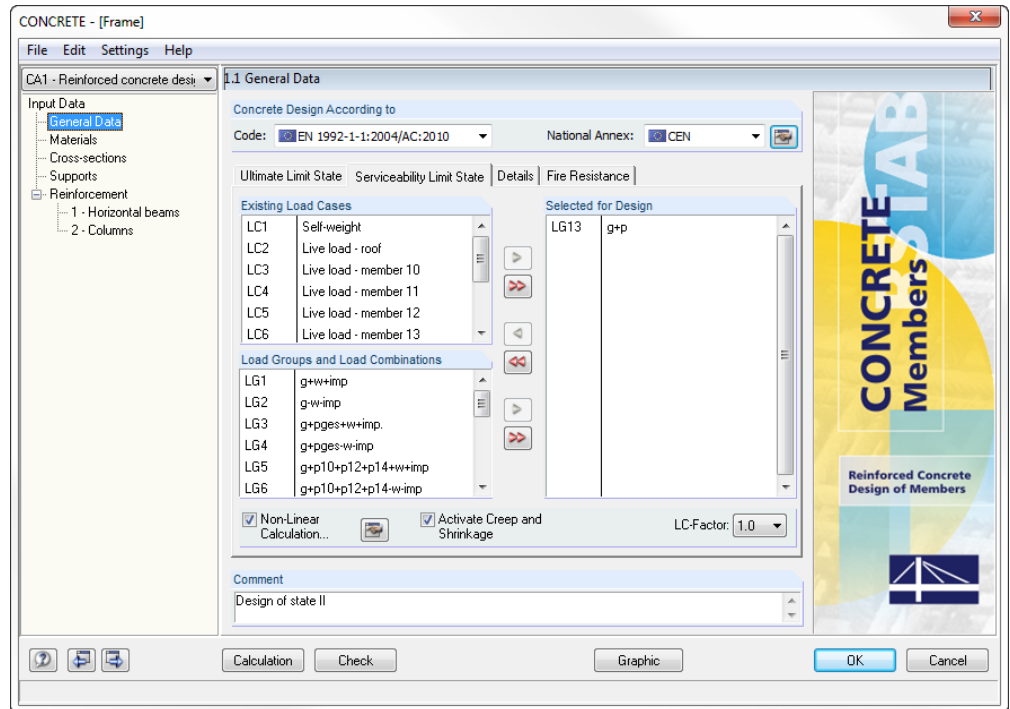


Figure 3.6: Table 1.1 General Data, tab Serviceability Limit State

Existing Load Cases / Load Groups and Load Combinations

These two dialog sections list all actions and load combinations defined in RSTAB. Use the [▶] button to transfer the selected load cases, groups and combinations or super combinations to the list *Selected for Design* on the right. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].



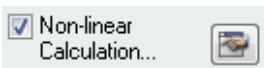
Selected for Design

The column on the right lists the loads selected for the serviceability limit state design. Use the button [◀] to remove selected load cases, groups or combinations from the list. You can also double-click the entries. With the button [◀◀], you can transfer the entire list to the left.



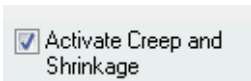
Non-linear Calculation

To perform the serviceability limit state design according to the non-linear design method, tick the check box below the list *Load Groups and Load Combinations*. Chapter 2.4.8 on page 48 presents the non-linear analysis for the serviceability limit state in detail.



A non-linear analysis performed according to DIN 1045-1 or EN 1992-1-1 can be carried out only for load cases and load groups but not for load combinations. The analysis is possible for both the ultimate and the serviceability limit state, combined or separately.

To open the dialog box *Settings for Non-linear Calculation*, use the [Edit] button. The three tabs available in the dialog box are described in chapter 4.2, page 81.



Furthermore, table 1.1 offers you the option to *Activate Creep and Shrinkage*. For more information, see chapter 2.4.6 on page 37. When the check box is ticked, you can specify the modulus of creep and the shrinkage strain in table 1.3 *Cross-sections* (see Figure 3.13, page 62).

3.1.3 Details

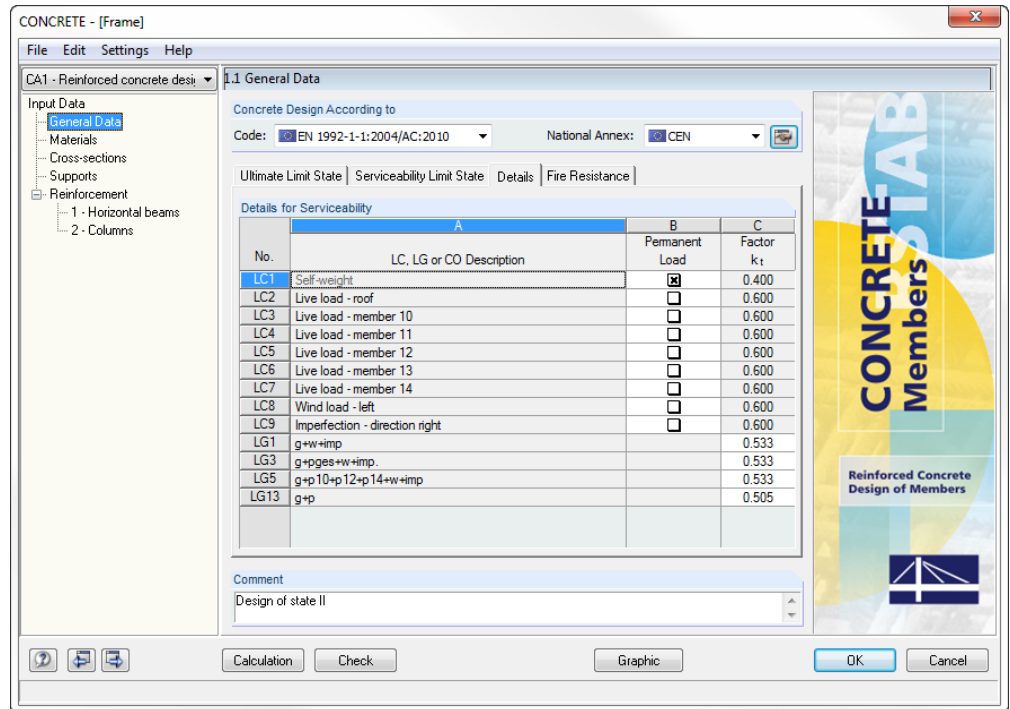


Figure 3.7: Table 1.1 General Data, tab Details



If you perform the design according to EN 1992-1-1 or one of the national EN application documents, the *Details* tab is displayed additionally.

This tab is unnecessary for serviceability limit state designs according to DIN 1045-1 because the factor k_t is generally defined with 0.4 in equation (136), section 11.2.4 (2).

LC, LG or CO Description

Column A lists all load cases, groups and combinations that have been selected for design in the *Serviceability Limit State* tab. The load cases that are contained in the selected load groups and combinations are also displayed.

Permanent Load

This column indicates the load cases that represent permanent loads. If a load case is specified as permanent load, the factor k_t is automatically set to 0.4 in the final column.

Factor k_t

The load duration factor k_t is used to determine the loading period. The factor k_t is 0.4 for long-term load actions and 0.6 for short-term actions.

For load groups and combinations, the factor k_t represents the average of the respective k_t values of the load cases contained in the corresponding load group or combination.

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

Equation 3.1

3.1.4 Fire Resistance

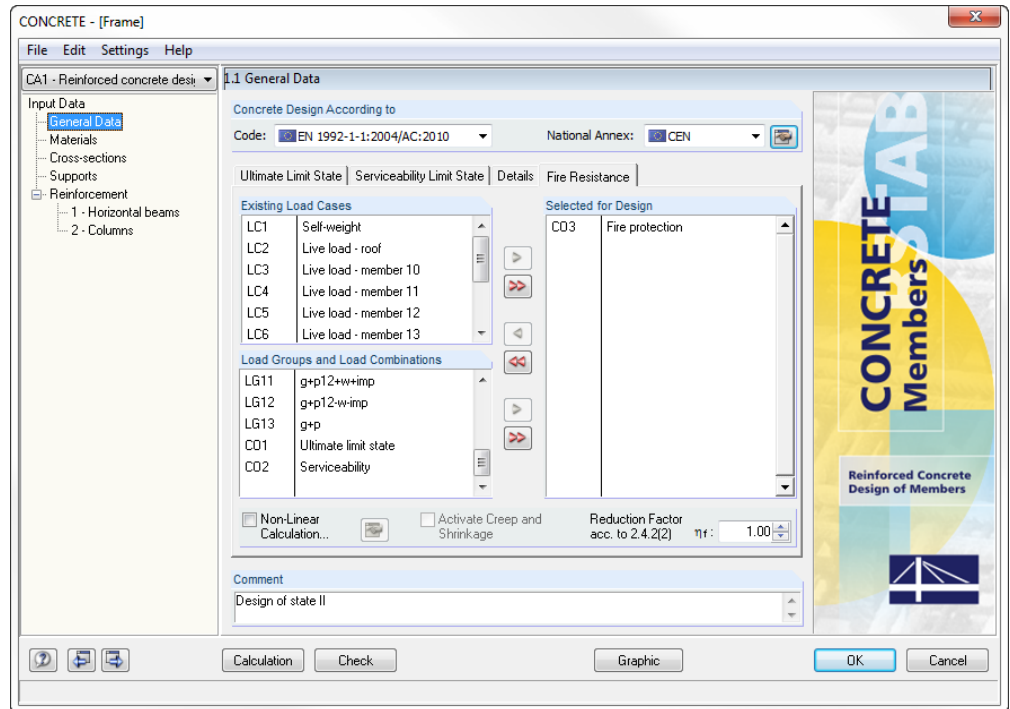


Figure 3.8: Table 1.1 General Data, tab Fire Resistance

Existing Load Cases / Load Groups and Load Combinations

These two dialog sections list all actions and load combinations defined in RSTAB. Use the [▶] button to transfer the selected load cases, groups and combinations or super combinations to the list *Selected for Design* on the right. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].



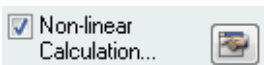
Selected for Design

The column on the right lists the loads selected for the fire protection design. Use the button [◀] to remove selected items from the list. You can also double-click the entries. With the button [◀◀], you can transfer the entire list to the left.



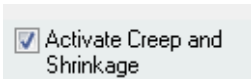
Non-linear Calculation

To perform the fire protection design according to the non-linear design method, tick the check box below the list *Load Groups and Load Combinations*. The non-linear approach is described in the theory chapter 2.



To open the dialog box *Settings for Non-linear Calculation*, use the [Edit] button. The three tabs available in the dialog box are described in chapter 4.2, page 81.

Furthermore, table 1.1 offers you the option to *Activate Creep and Shrinkage*. For more information, see chapter 2.4.6 on page 37.



The option *Reduction Factor acc. to 2.4.2 (2)* allows for a simplified transfer of loadings from the design for normal temperature to reduce these actions by the reduction factor η_{fi} . The reduction factor is determined as suggested in EN 1992-1-2 according to 2.4.2 (3). As a simplification you can use the recommended value $\eta_{fi} = 0.7$.

3.2 Materials

The table is subdivided into two parts. The upper part lists the concrete and steel grades used for the design. In the *Material Properties* section below, the properties of the current material, i.e. the table row currently selected in the upper section, are displayed.

Materials that are not used in the design appear gray in color. Materials that are not allowed are highlighted red. Modified materials are displayed in blue.

The material properties required for the determination of internal forces in RSTAB are described in detail in chapter 5.2 of the RSTAB manual. The design relevant material properties are stored in the global material library and preset automatically.

To adjust the units and decimal places of material properties and stiffnesses, select **Units and Decimal Places** in the module's **Settings** menu (see Figure 8.6, page 134).

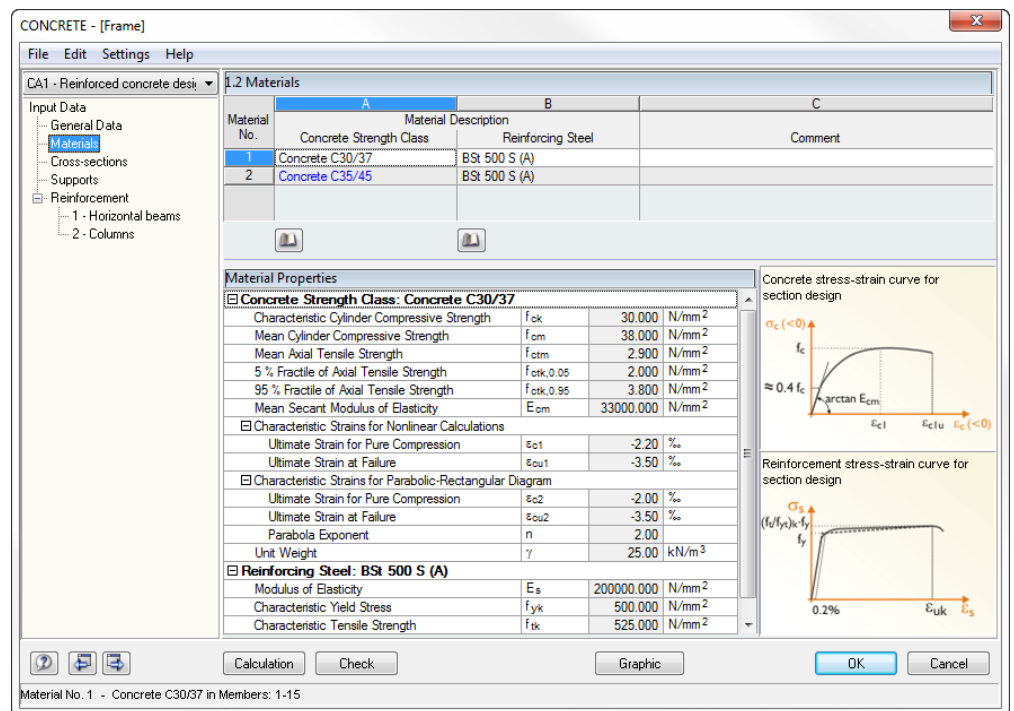


Figure 3.9: Table 1.2 *Materials*

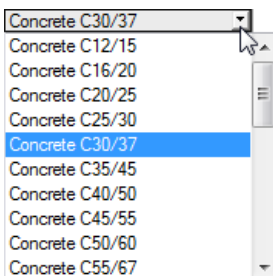
Material Description

Concrete Strength Class

The concrete materials defined in RSTAB are already preset. Materials of a different material type are highlighted red. When a manually entered *Material Description* corresponds to an entry of the material library, CONCRETE will import the appropriate material properties.

It is possible to select another material by using the list: Place the pointer in a table row of column A, and then click the button [▼] or use the function key [F7]. The list shown on the left opens. Subsequent to the transfer, the properties will be updated.

The list contains only materials of the *Concrete* category complying with the design concept of the selected standard. The import of materials from the library is described below.

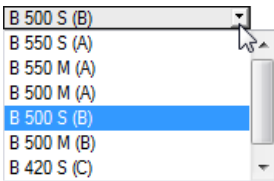


Reinforcing Steel

In this column, the program presets a common steel grade that corresponds to the design concept of the selected standard.

It is possible to select another reinforcing steel by using the list: Place the pointer in a table row of column B, and then click the button [▼] or use the function key [F7]. The list shown on the left opens. Subsequent to the transfer, the properties will be updated.

The import of materials from the library is described below.



Material Library

Numerous concrete and reinforcing steel materials are stored in the library. To open the library, use the button shown on the left. The button can be found below column A and B.

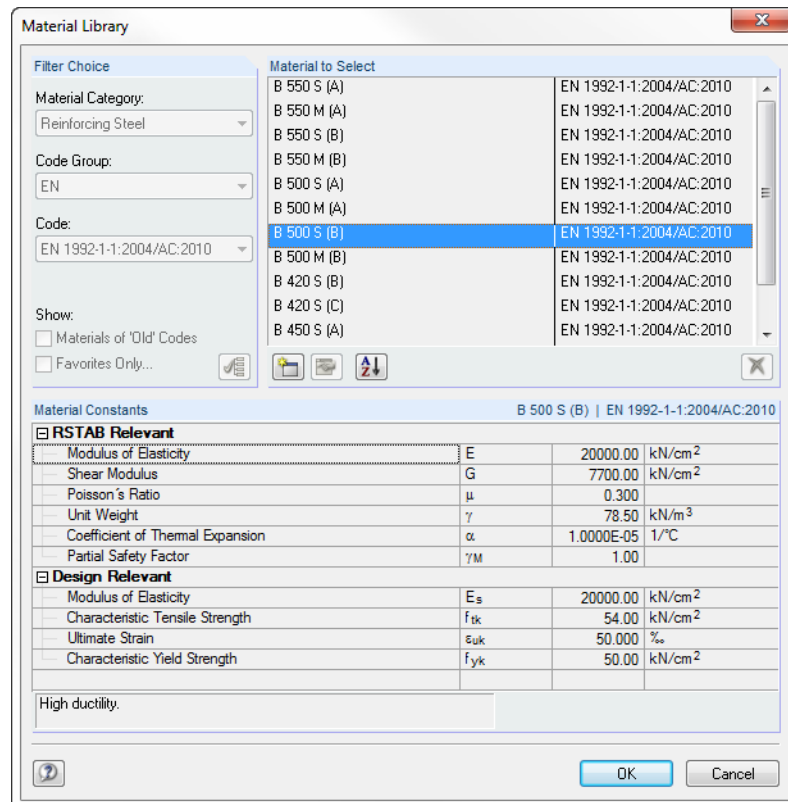


Figure 3.10: Dialog box *Material Library*

The standard relevant materials are already preset in a preselection so that no other categories or standards are available in the *Filter Choice* dialog section. Select a material from the list *Material to Select* and check the corresponding parameters in the lower part of the dialog box. As a matter of principle, it is not possible to edit the material properties in this dialog box.

Click [OK] or use the [↵] button to import the selected material to table 1.2 of the add-on module.



Chapter 5.2 in the RSTAB manual describes in detail how materials can be added or rearranged. By using the [Create New Material] button, you can create new types of concrete or reinforcing steel with user-defined material properties and store them for later use.

3.3 Cross-sections

This table lists the design relevant cross-sections.

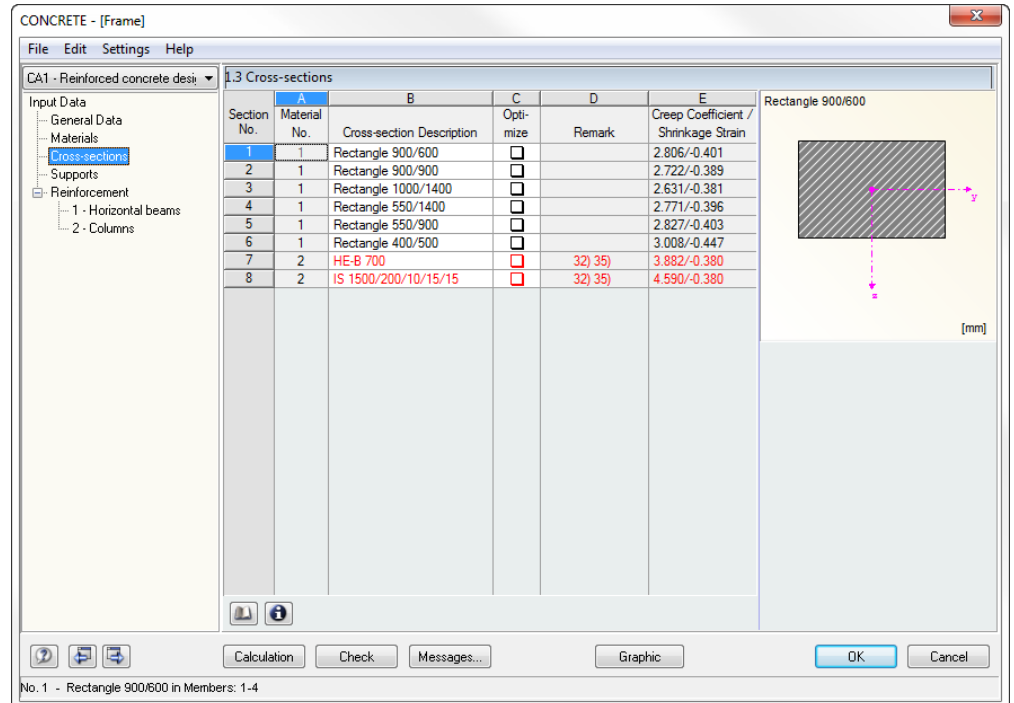


Figure 3.11: Table 1.3 Cross-sections

Cross-section Description

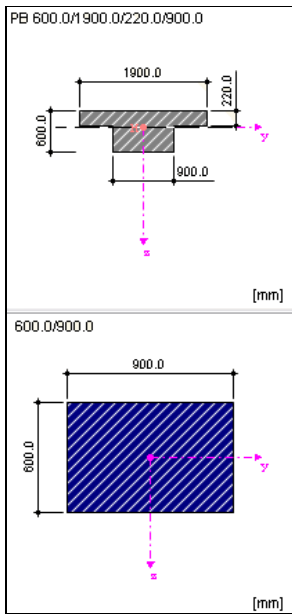
When you open the table, the cross-sections used in RSTAB are preset together with the assigned material numbers.

It is always possible to modify the preset cross-sections for the design. The description of a modified cross-section is highlighted in blue.

To modify a cross-section, enter the new cross-section description directly into the corresponding table row. You can also select the new cross-section from the library. To open the library, use the button [Import Cross-section from Library] below the table. Alternatively, place the pointer in the respective table row and click the [...] button, or use the function key [F7]. The cross-section library that you already know from RSTAB appears. For the CONCRETE design only few buttons are enabled in the dialog section *Solid Cross-sections*:

- 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)
- Channel (symmetric)

The selection of cross-sections from the library is described in detail in chapter 5.3 of the RSTAB manual.



Creep Coefficient / Shrinkage Strain
2.806/-0.401
2.722/-0.389

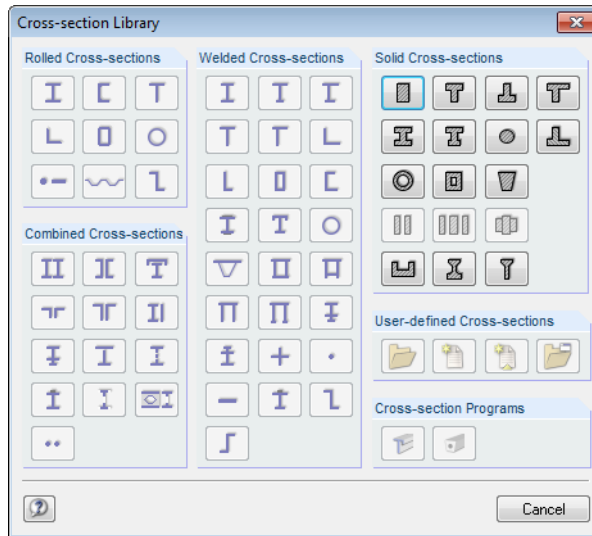


Figure 3.12: Cross-section Library

If the cross-sections defined in CONCRETE are different from the ones used in RSTAB, both cross-sections are displayed in the graphic in the right part of the table.

Optimize

For each cross-section it is possible to perform an optimization analysis. By using the internal forces from RSTAB the program determines within the same cross-section table the cross-section that meets the reinforcement requirements specified in the dialog box *Optimization Parameters* with the least possible dimensions (see Figure 8.5, page 133).

To optimize a particular cross-section, tick its check box in column C. Recommendations for optimizing cross-sections can be found in chapter 8.2 on page 133.

Remark

This column shows remarks in the form of footers that are described in detail below the cross-section list.

Modulus of creep / shrinkage strain

Column E displays the values for the creep coefficient and the strain due to shrinkage determined according to the preset method. Use the context button shown on the left to adjust the values. A dialog box (see Figure 3.13) opens where you can specify new data.

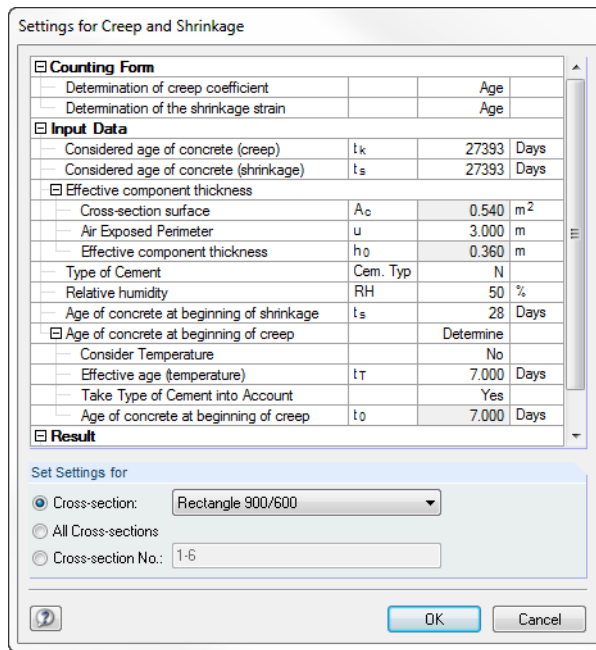
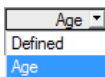


Figure 3.13: Dialog box *Settings for Creep and Shrinkage*



The first item *Counting Form* comprises two possibilities to define the modulus of creep and the coefficient of shrinkage:

- Age Modulus of creep and coefficient of shrinkage are calculated by means of parameters.
- Defined Modulus of creep and coefficient of shrinkage must be specified directly.

How the creep coefficient and the shrinkage strain are determined is described in chapter 2.4.6, page 37.

At the end of the table the *Result* is displayed showing the determined modulus of creep $\varphi(t, t_0)$ and the determined coefficient of shrinkage $\varepsilon_{c,s}(t, t_s)$.

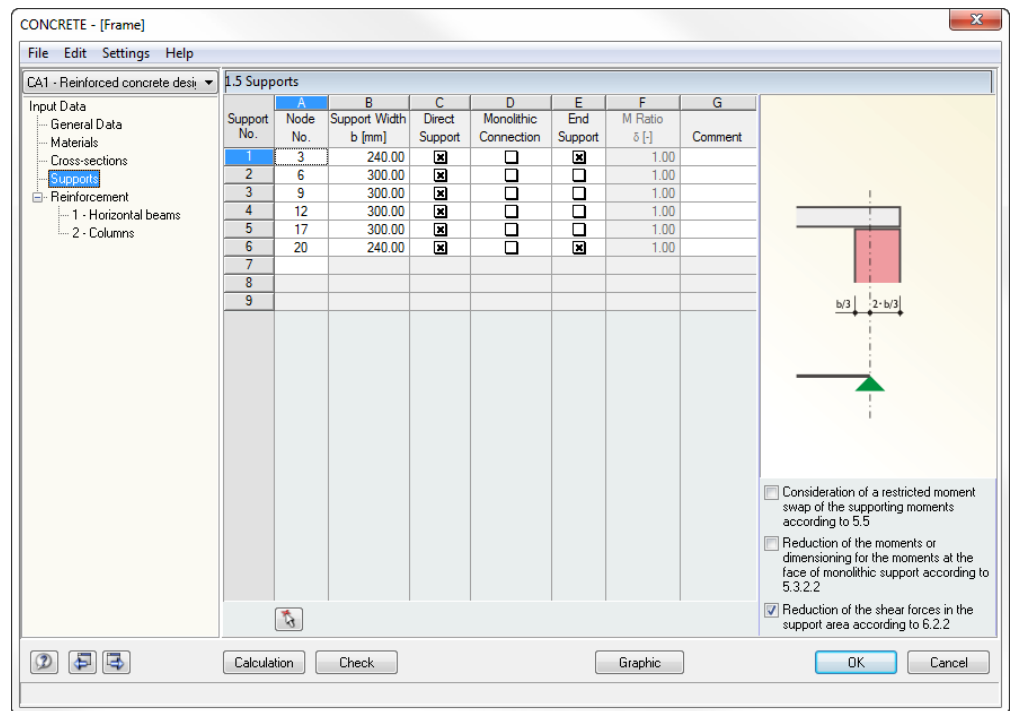
In the dialog section *Set Settings for*, you can decide whether the entered specifications are applied to a single cross-section, to all cross-sections or to selected cross-sections.

3.4 Supports

This table defines in detail the support conditions of the members that you want to design. The nodal supports defined on horizontal members in RSTAB are already preset. If necessary, it is possible to adjust them. CONCRETE also recognizes if intermediate or end supports are available.



Non-zero support widths affect the design (redistribution of moments, moment reduction, reduction of shear force) and the reinforcement proposal (length of anchorage). However, only members in horizontal or slight inclined position are affected. Columns are not taken into account!



Support No.	Node No.	Support Width b [mm]	Direct Support	Monolithic Connection	End Support	M Ratio δ [:]	Comment
1	3	240.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.00	
2	6	300.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
3	9	300.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
4	12	300.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
5	17	300.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
6	20	240.00	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.00	
7							
8							
9							

Figure 3.14: Table 1.5 Supports

Node No.



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

Support Width b

In column B you define the effective width of the corresponding nodal support. In this way you can determine, for example, a wall's wide bearing area which is represented in the RSTAB model only by a singular support.

Direct Support

The data in this column specifies the support type of the beam. When the load of an adjoining beam is introduced to a main beam, the support is an indirect support and you should clear the check box.

The specifications of this table column affect the lengths of anchorage and the shear design.

Monolithic Connection

In column D you decide if a flexurally rigid connection with the column or a free rotary support including reduction options for the supporting moments is available.

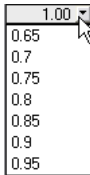
End Support

An end support has a different influence on the design moment and the lengths of anchorage as an intermediate support. Column E provides a check box for the appropriate assignment.

M Ratio δ

For continuous structural components you can define the ratio δ for the redistributed moment and the elastically determined initial moment in column F. The column is only available if you have ticked the check box *Consideration of a restricted moment swap* below the graphic on the right.

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



Comment

In the final column, you can enter a comment for each support, describing the selected support conditions in detail.

Taking into account the support widths

Below the interactive graphic in the table you find three check boxes whose specifications, depending on the design standard, have different effects on the required reinforcement. The settings are globally valid for the currently selected design case.

Consideration of a restricted moment swap

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. The standards describe the moment ratios δ that must be observed in order to ensure the ability for rotation in the critical areas without special designs, for example EN 1992-1-1, 5.5 (4).

CONCRETE determines this limit value and compares it with the value that is specified in column F. The higher value is used for the redistribution.

Reduction of the moments or dimensioning for the moments

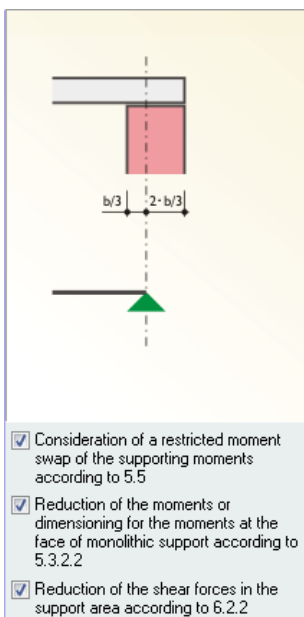
Optionally, the program performs a moment reduction, for example according to EN 1992-1-1, 5.3.2.2, or DIN 1045-1, 7.3.2, when all of the following requirements are met:

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

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

Reduction of the shear forces in the support area

For a direct support it is possible to reduce the design value of the shear force, cf. EN 1992-1-1, 6.2.1 (8) and 6.2.2 (6).



Loads near supports are taken into account, regardless of whether the check box for shear force reduction is ticked, provided that they are designed in load cases or groups.

In case of load combinations the shear force is designed generally on the support's edge because the standard's requirement for a "uniformly distributed load" by means of envelopes is not verifiable in detail.

3.5 Reinforcement

This table consists of several tabs where all reinforcement data is specified. As the individual members often require different reinforcement settings, you can create several reinforcement groups in each CONCRETE design case. The reinforcement specifications can be defined for members as well as for sets of members.

Reinforcement Group

To create a new reinforcement group, use the [New] button in the dialog section *Reinforcement Group*. The number is allocated by the program. The user-defined *Description* helps you to overlook all reinforcement groups created in the design case.

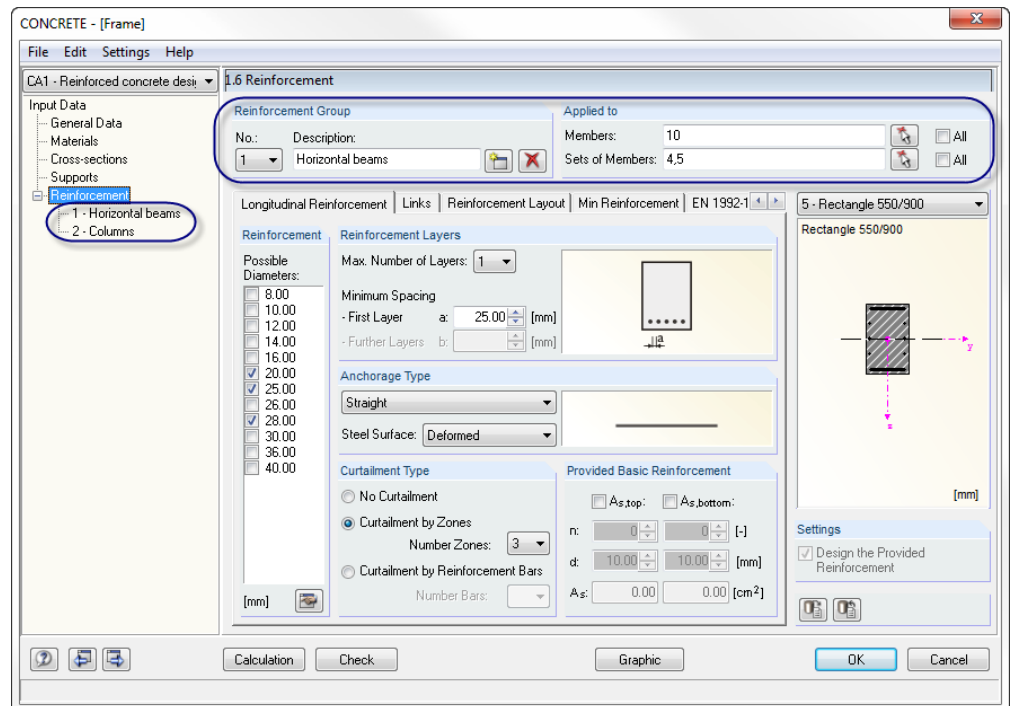


Figure 3.15: Table 1.6 Reinforcement with two reinforcement groups



To select a reinforcement group, use the *No.* list or the entries in the navigator.



By using the [Delete] button, the currently selected reinforcement group is deleted from the CONCRETE case without any additional warning. Members and sets of members that were contained in such a reinforcement group won't be designed. You must reassign them to a new or an existing reinforcement group if you want to design them.

In the dialog section *Applied to*, you define the members or sets of members to which the current reinforcement group is applied. *All* members and *All* sets of members are preset. If the check boxes are ticked, no further reinforcement group can be created because members and sets of members cannot be designed by different reinforcement specifications within the same design case. Therefore, in order to use the possibility for several reinforcement groups, clear at least one of the *All* check boxes.



Enter the numbers of the relevant *Members* or *Sets of Members* into the input field. The reinforcement specifications defined in the tabs of the table will be valid for all selected objects. You can also use the [Pick] button to select the objects graphically in the RSTAB work window. Thus, the button [Create New Reinforcement Group] becomes active. For the new group you may only select members and sets of members that are not yet assigned to any other reinforcement group.

Single members contained in the sets of members will be deactivated automatically for the design.

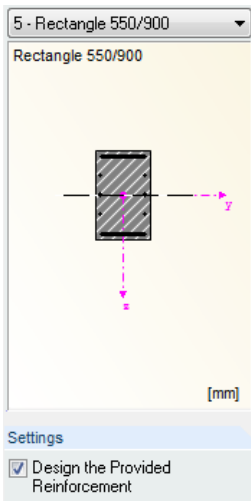
Reinforcement proposal

The graphic in the right part of the table shows how the input in the tabs affect the cross-section. Use the list above the graphic to select another cross-section. The graphic is dynamic: Modifications to the reinforcement specifications are updated and displayed immediately.

If the check box *Design the Provided Reinforcement* is ticked, CONCRETE will use the specifications entered in the various tabs to calculate a rebar reinforcement. If you clear the check box, some of the input fields cannot be accessed. In this case, the program will determine only the required reinforcement areas.

When you have activated the design for the serviceability limit state or for fire protection in table 1.1 *General Data*, it is not possible to clear the check box because it is not enabled: The SLS design is based on an actually provided reinforcement. Crack widths, crack spacings etc. can be determined only by the applied rebar diameters and spacings. The same applies to a design according to the non-linear method.

The individual tabs of table 1.6 are described in the following.



3.5.1 Longitudinal Reinforcement

In this tab you enter the specifications defining the longitudinal reinforcement.

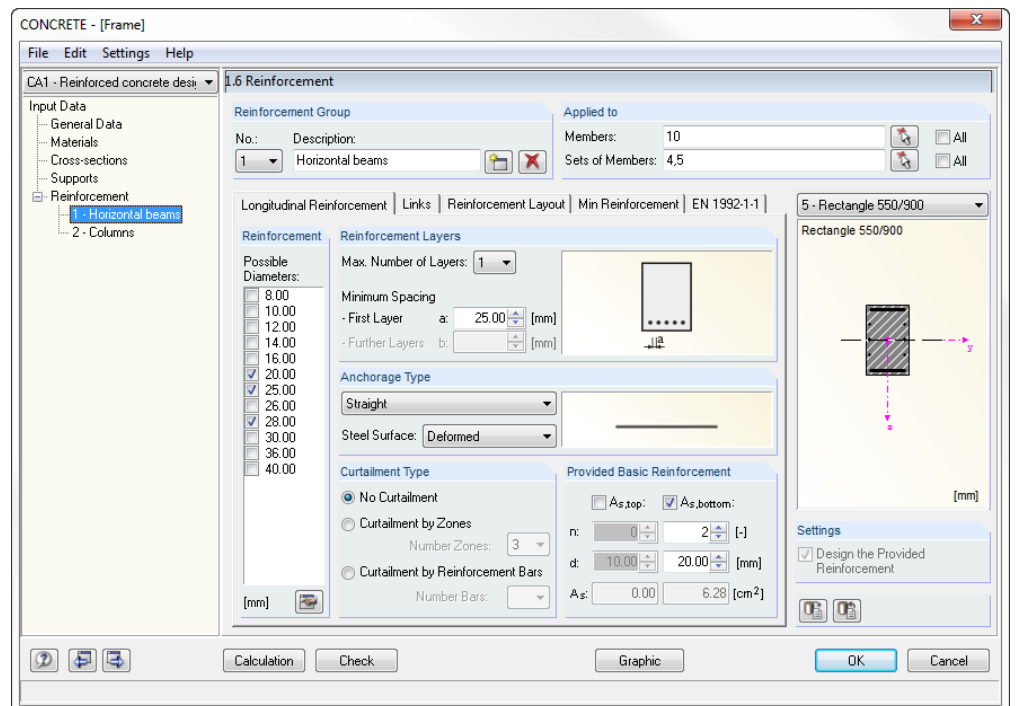


Figure 3.16: Table 1.6 Reinforcement, tab Longitudinal Reinforcement

Reinforcement

In addition to the nominal diameters for rebars mentioned in DIN 488, the list for *Possible Diameters* includes some diameters used outside Germany. A multiple selection is no problem for the design.

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

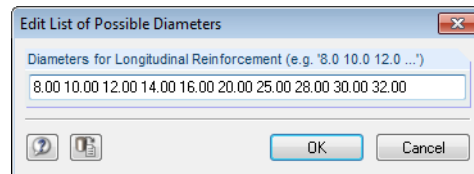
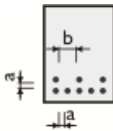


Figure 3.17: Dialog box *Edit List of Possible Diameters*

The input line allows for modifying, removing or adding entries.

Reinforcement Layers

For the reinforcement proposal CONCRETE takes into account also multi-layer arrangements of rebars. Use the list to specify the *Max. Number of Layers*. It is possible to define up to three reinforcement layers. The input fields below allow for entering specifications concerning the *Minimum Spacing a* for rebars of the first layer and *b* for further layers, if necessary.



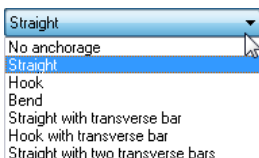
When the program creates the provided reinforcement, these structural specifications are considered. They affect the number of possible rebars inserted in each layer and the lever arm of internal forces.

If an arrangement of several reinforcement layers is defined, a curtailment of reinforcements is not possible.



Anchorage Type

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



The anchorage as well as the *Steel Surface* (plain, deformed) affect the required length of anchorage.



Curtailment Type

No Curtailment is preset. If you have specified several reinforcement layers, the other two options are disabled.

If you select a *Curtailment by Zones*, you can use the list to the right to define how many zones with the same reinforcement are allowed in the reinforcement proposal. With this setting CONCRETE finds out how to cover the required steel cross-sectional areas with the available rebars in an optimal way.

If *Curtailment by Reinforcement Bars* is selected, a new zone will be available only when the maximum number of rebars specified is reached. Use the list to the right to define the number of rebars.

Provided Basic Reinforcement

In this dialog section you can specify a basic reinforcement separately for the top and bottom layer. When the check boxes are ticked, the input fields below can be accessed. They allow you to define the number of rebars n and the rebar diameter d . The field A_s indicates the corresponding reinforcement areas.

The user-defined basic reinforcement is taken into account when the reinforcement proposal is created. It will be inserted over the entire length of the member or set of members. In case the required reinforcement is not covered by the basic reinforcement, CONCRETE determines the rebars that are additionally required and inserts them into the cross-section.

3.5.2 Links

This tab contains reinforcement specifications for the shear reinforcement.

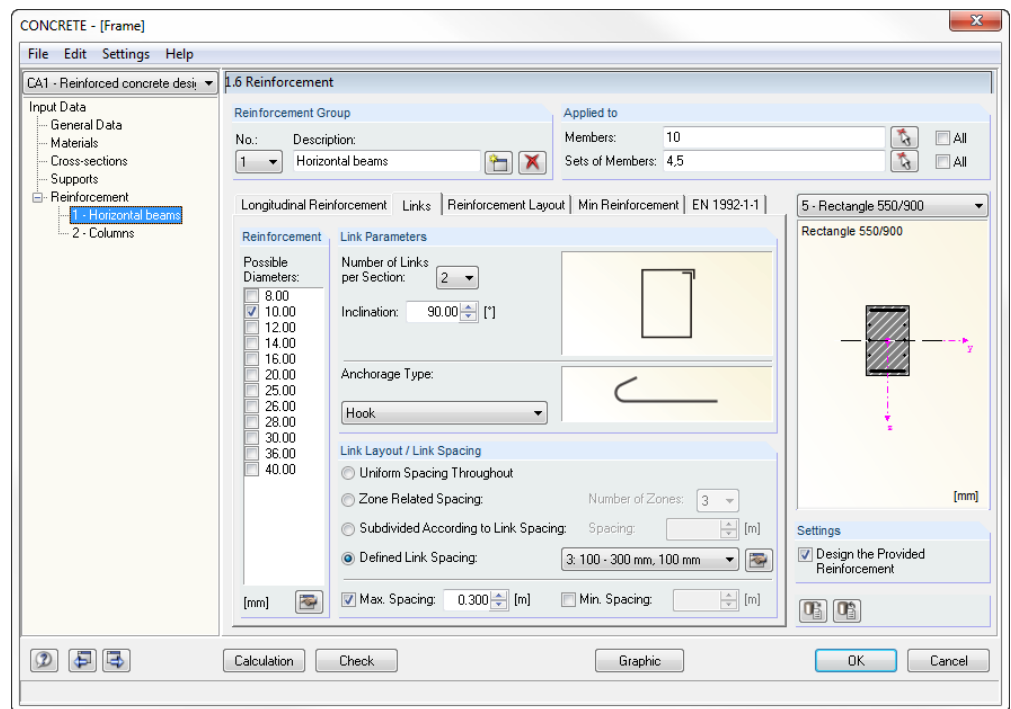


Figure 3.18: Table 1.6 Reinforcement, tab Links

Reinforcement

In addition to the nominal diameters for rebars mentioned in DIN 488, the list for *Possible Diameters* includes some diameters used outside Germany. A multiple selection is no problem for the design.

Use the [Edit] button to adjust the list of the displayed rebar diameters (see Figure 3.17).

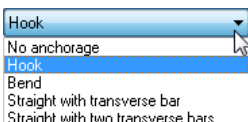
Link Parameters

With the list *Number of Links per Section* you define the link sections. Two sections are preset. To adjust the presetting, use the list. It is possible to define up to four sections.

The *Inclination* of the shear reinforcement is defined by the angle between the longitudinal and the shear reinforcement. The program presets 90°, this means that the links are perpendicular. If you want to modify the inclination, observe the standard specifications: EN 1992-1-1, 9.2.2 (1) only allows for angles between 45° and 90°. Furthermore, the code requires in chapter 9.2.2 (4) that bent bars acting as shear reinforcement may be used only together with links. Moreover, it is recommended that at least 50 % of the shear force to be absorbed is covered by links.

Anchorage Type

The list provides numerous possibilities for link anchorages affecting the determination of the anchorage lengths. The graphic to the right is dynamic, this means that modified settings are updated and displayed immediately in the graphic.



Link Layout / Link Spacing

This dialog section can be accessed only when a reinforcement proposal is created.

For all members and sets of members a *Uniform Spacing Throughout* is preset.

When you select a *Zone Related Spacing*, you have to specify the number of zones with the same link layout in the list. In case one zone is specified, the program will create a further zone in addition to the zone with the maximum link spacing (minimum reinforcement). The additional zone covers the maximum value of the required link reinforcement. If you specify two zones, CONCRETE will determine the mean value from the required minimum and maximum reinforcement and applies the corresponding x-locations on the member as further zone limits.

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

When a *Defined Link Spacing* is selected, you can choose an entry in the list shown on the left. Use the [Edit] button to the right to adjust the entries or to create a new entry with user-defined distances between links.

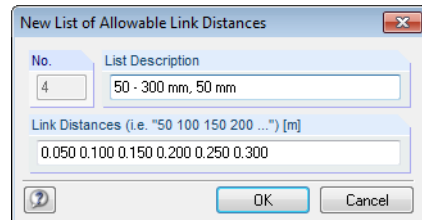
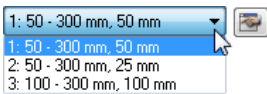


Figure 3.19: Dialog box *New List of Allowable Link Distances*



The *Max. Spacing* and the *Min. Spacing* of the link reinforcement can be defined directly.

The zones presented in the reinforcement proposal can be modified or completed subsequently in table 3.2 *Shear Reinforcement Provided* (see chapter 5.2.2, page 99).

3.5.3 Reinforcement Layout

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

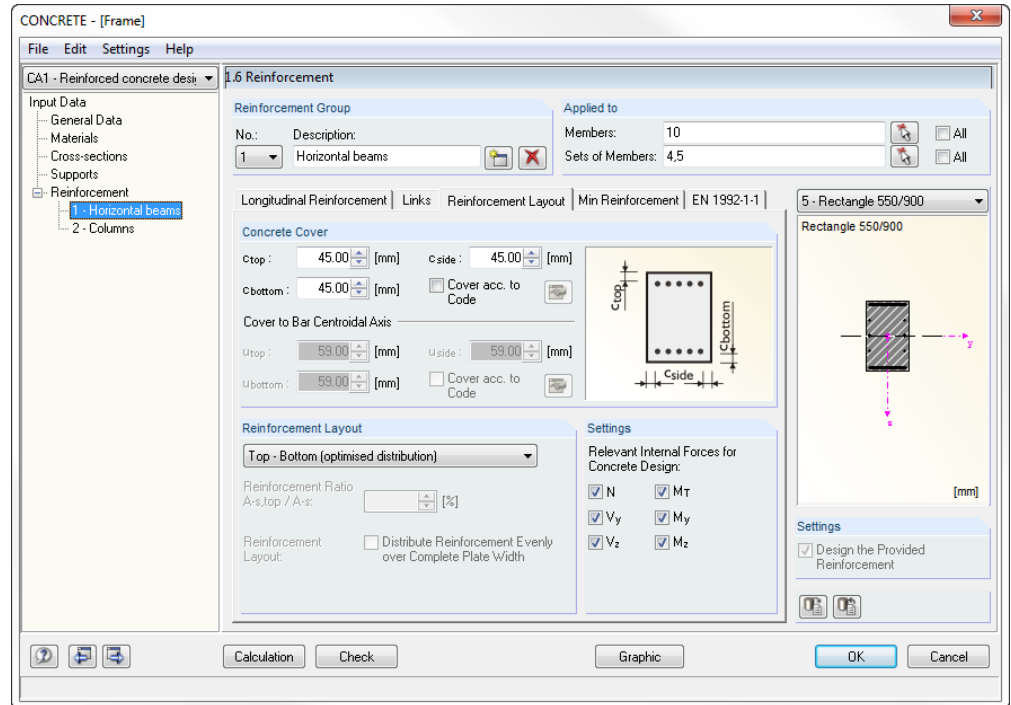


Figure 3.20: Table 1.6 Reinforcement, tab Reinforcement Layout

Concrete Cover

The settings for concrete covers interact with the specification for a reinforcement proposal: If a reinforcement proposal is created, the covers refer to the dimensions of the rebars' edge distance c . But if the program won't calculate a provided reinforcement, the specifications refer to the dimensions of the rebars' centroidal axis u . The following picture shows the difference.

Depending on your specifications, you can access the input fields in the upper or lower part of the dialog section.

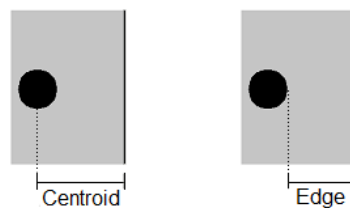
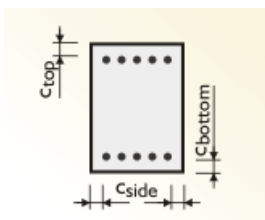


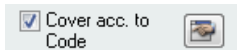
Figure 3.21: Relations of concrete cover



In the input field c_{top} , you enter the concrete cover of the longitudinal reinforcement at the top. In the input field c_{bottom} , you specify the cover of the longitudinal reinforcement at the bottom. These values represent the nominal values c_{nom} of the concrete cover, for example according to EN 1992-1-1, 4.4.1.1. Based on these specifications and taking into account the applied rebar diameters, CONCRETE determines the lever arm of the internal forces.

"Top" and "bottom" are clearly defined by the position of the local member axes in RSTAB. The cover c_{side} is required for the equivalent wall thickness for the torsional design.

For multilayer reinforcements, when entering the cover to the centroidal axis u , you have to take care that the distance refers to the reinforcement's centroid.



With the check box *Cover acc. to Code* you set the standard values of the selected design code. Use the [Edit] button to adjust the parameters. A dialog box opens offering three tabs used to control the determination of values.

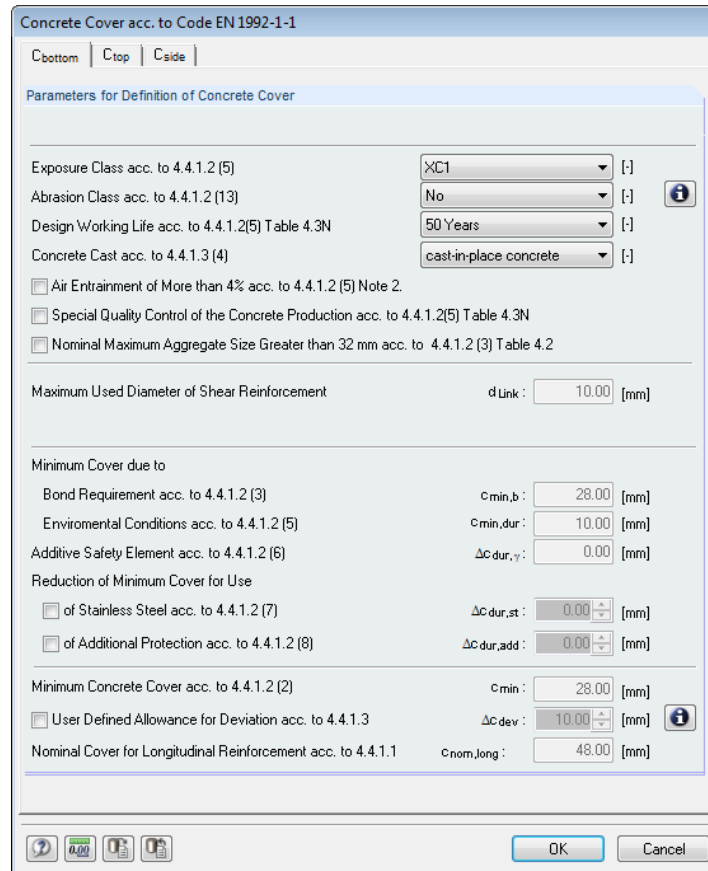


Figure 3.22: Dialog box *Concrete Cover acc. to Code*

Reinforcement Layout

The list contains a variety of possibilities how to lay out the reinforcement in the cross-section:

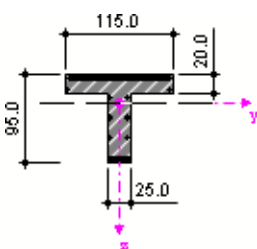
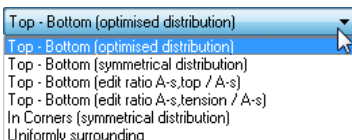
- Top - Bottom (optimized distribution)
- Top - Bottom (symmetrical distribution)
- Top - Bottom (edit ratio $A_{s,top} / A_s$)
- Top - Bottom (edit ratio $A_{s,tension} / A_s$)
- In Corners (symmetrical distribution)
- Uniformly surrounding

For the reinforcement layout *Top - Bottom (optimized distribution)* CONCRETE performs also an optimization for biaxial bending.

Due to this option, it is also possible to define the reinforcement by means of the ratio for top reinforcement and total reinforcement or for tension reinforcement and total reinforcement. The value for the ratio is entered in the input field below the list. In this way, the program facilitates an efficient replication of existing constructions.

Furthermore, if T-beams and I-sections are used, you can *Distribute Reinforcement Evenly over Complete Plate Width*. With this option the program releases a part of the rebars.

Modifications to the reinforcement layout are shown by dynamic display in the graphic to the right.





If the moment distribution is $M_y = 0$ and $M_z > 0$ for a *Top - Bottom* layout, the program's output shows increased areas of reinforcement: The design moment is not acting in the distributing direction specified for the reinforcement. In this case, select the *In Corners* layout in order to perform the design correctly.

3.5.4 Minimum Reinforcement

This tab includes the specifications concerning minimum and structural reinforcement as well as the parameters for crack width control.

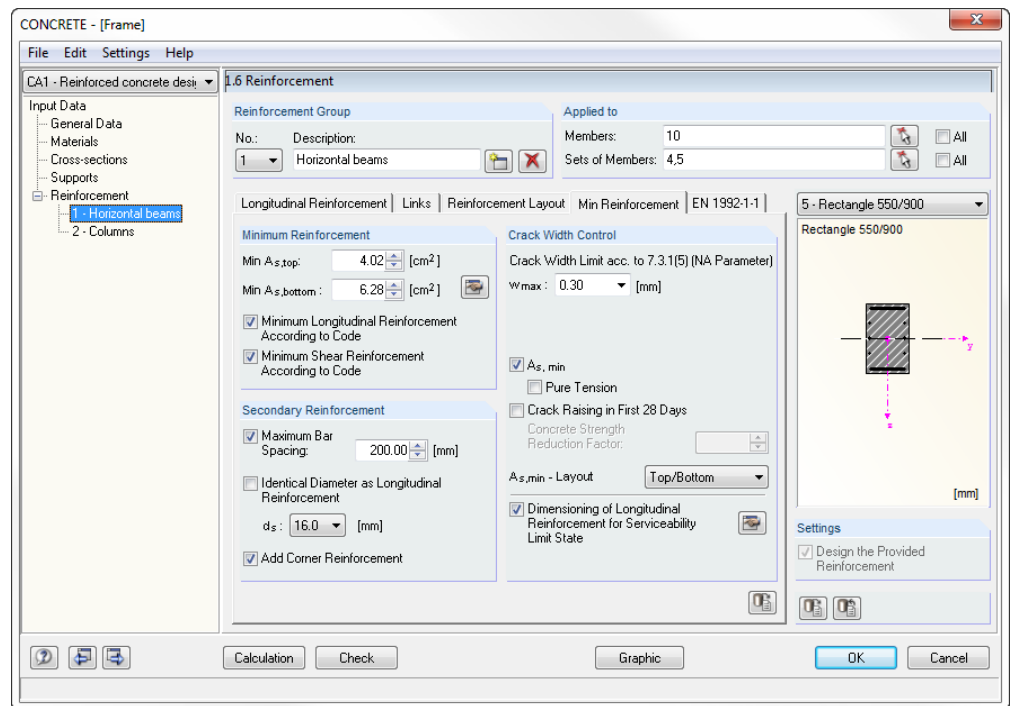


Figure 3.23: Table 1.6 Reinforcement, tab Min Reinforcement

Minimum Reinforcement

For the definition of a global minimum reinforcement two input fields are available. You can enter the steel cross-sections for $Min A_{s,top}$ and $Min A_{s,bottom}$. Use the [Edit] button to specify the cross-sectional areas based on the number of rebars and the rebar diameters in a separate dialog box.

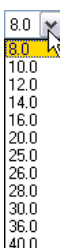
For the calculation of the required reinforcement it is possible to take into account or to exclude, optionally and independently of each other, the *Minimum Longitudinal Reinforcement* and the *Minimum Shear Reinforcement* according to the corresponding standard.

Secondary Reinforcement

This dialog section is active only if you want to create a reinforcement proposal.

The *Maximum Bar Spacing* of the secondary rebars, i.e. the rebars that are not structurally required in the cross-section, is defined by specifying a maximum value. The reinforcement proposal will try to realize a uniform distribution of rebars (for example for webs of T-beams or for slender rectangular cross-sections).

With the option *Identical Diameter as Longitudinal Reinforcement* the program aligns the secondary reinforcement with the rebar diameter of the required reinforcement. Alternatively, you can use the list to define a specific diameter d_s for the secondary reinforcement.



If you tick the check box *Add Corner Reinforcement*, a secondary reinforcement is generally arranged in all corners of the cross-section. In this way it is possible to define a reinforcement outside the web even for I-shaped cross-sections.

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

Crack Width Control

The access to the input fields in this dialog section is only available if the serviceability limit state design has been activated in table 1.1 *General Data*. In addition, the dialog section is adjusted to the selected standard. The description in the following refers to EN 1992-1-1.

The *Crack Width Limit* w_{max} can be selected in the list. Depending on the environmental conditions, the specifications mentioned in EN 1992-1-1, 7.3.1 (5) must be applied.

Crack Width [mm]	Exposure Class	Structural Element
0.20	XC0 - XC1	Prestressing
0.20	XC2 - XC4	Prestressing
0.30	XC2 - XC4	Reinf. Concrete, Unbonded Prestressing
0.30	XD1 - XD2	Reinf. Concrete, Unbonded Prestressing
0.30	XS1 - XS3	Reinf. Concrete, Unbonded Prestressing
0.40	XC0 - XC1	Reinf. Concrete, Unbonded Prestressing

Figure 3.24: Crack widths depending on exposure class (EN 1992-1-1)

When designing the limitation of crack widths, you have to distinguish between load actions and effects due to restraint. Effects due to restraint are reduced considerably by crack formations within the structural component so that a minimum reinforcement $A_{s,min}$ with sufficient dimensions results in a distribution of the entire component reduction on several cracks having respectively small crack widths. The crack widths caused by a load action, however, depend on the available steel stress and the reinforcement layout.

Concerning the factor k used to consider non-linearly distributed self-equilibrating stresses, the German national annex (NA) of EN 1992-1-1, 7.3.2 (2) distinguishes tension stresses due to

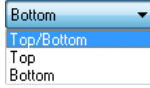
- restraint caused in the structural component itself (for example from drain of hydration heat)
- restraint caused outside of the structural component (for example column settlement).

CONCRETE calculates the crack width according to EN 1992-1-1, 7.3.4 for the respective load stress directly. For effects due to restraint the program designs the minimum reinforcement according to EN 1992-1-1, 7.3.2 to limit the specified crack width.

If you want to define the effects due to restraint to be *Pure Tension*, tick the corresponding check box: The definition affects the factor k_c for EN 1992-1-1, eq. (7.1), determining the stress distribution in the cross-section before the first crack is formed. In case of pure tension $k_c = 1.0$. For bending or bending with axial force k_c is determined according to eq. (7.2) or (7.3). The mean concrete stress σ_c is determined from the loads because without load it is not possible to calculate the concrete stress. In case of exclusive bending stress, σ_c is equal to zero in the structural component's axis and thus, according to eq. (7.2), $k_c = 0.4$. In addition to the stress distribution, the factor k_c takes into account approximately the increase of the inner lever arm for crack formation.

When you assume *Crack Raising in First 28 Days*, the effective tensile strength of concrete $f_{ct,eff}$ must be reduced according to EN 1992-1-1, 7.3.2 (2). Enter the corresponding *Concrete Strength Reduction Factor* into the input field to the right. The national annex for Germany requires $f_{ct,eff} = 0.50 \cdot f_{ctm}(28d)$. If it is not possible to determine definitely the point of time within the first 28 days for the cracking's start, it would be appropriate, in accordance with the national annex for Germany, to assume at least a tensile strength of 3 N/mm² for normal concrete.

In case of direct restraint, the extent of reduction is affected by the thickness of the structural component because higher self-equilibrating stresses occur when the cross-section dimensions are increased. However, deformations imposed from outside (for example support deformations) do not cause any self-equilibrating stresses. In such a case the reduction factor is 1.0.



Use the selection list to define the reinforcement layout to which you want to assign the minimum reinforcement.



To check and, if necessary, to adjust the *Dimensioning of Longitudinal Reinforcement for Serviceability Limit State*, use the [Edit] button to the right. The following dialog box opens.

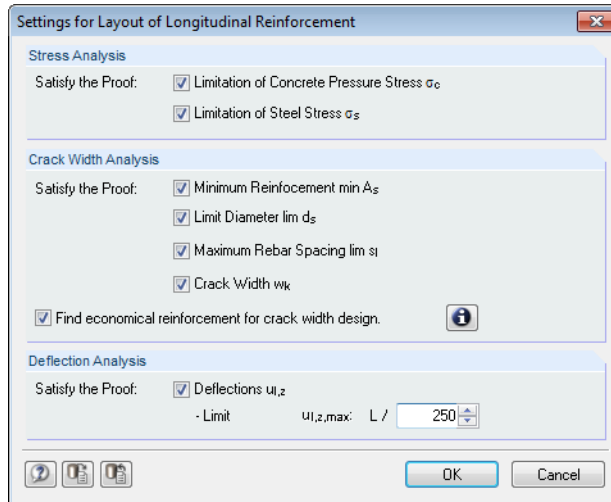


Figure 3.25: Dialog box *Settings for Layout of Longitudinal Reinforcement*

For the serviceability limit state design you can select various criteria for the stress design and the crack width analysis according to which the reinforcement is designed:

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)
Minimum reinforcement with A_s	7.3.2 (2)
Limit diameter d_s	Table 7.2 (see Figure 2.2, page 13)
Maximum rebar spacing $lim s_l$	Table 7.3 (see Figure 2.3, page 14)
Limitation of crack width w_k	7.3.1 (5) and 7.3.4

Table 3.1: Stress designs and crack width analyses



Generally, only one of the criteria, that means $lim d_s$, $lim s_l$ or w_k , must be fulfilled for the crack width analysis. If you select the option *Find economical reinforcement for crack width design*, it is possible to check which one of the three criteria can be covered by using the least necessary reinforcement area. The program will add individual rebars to the provided reinforcement until the design is successfully fulfilled. Click the [Info] button to get more information on the reinforcement's layout.



The criterion of the minimum reinforcement $min A_s$ must always be fulfilled in case of effects due to restraint.

Furthermore, you can specify a control of deflections for the *Deflection Analysis*. The program presets a sag of 1/250 of the span length, the value is recommended in EN 1992-1-1, 7.4.1 (4). The input field allows for defining another *Limit*.

The design criterion of the deformation $u_{l,z}$ considers the displacement in direction of the local member axis z. The sag is related to the undeformed system, this means that the displacements of the start and end nodes are not considered. The reinforcement's layout for the deformation analysis is carried out according to the simplified method in accordance with EN 1992-1-1, 7.4.3.

3.5.5 Standard

The fifth table tab is reserved for the settings of the standard that you have selected in table 1.1 *General Data* (see Figure 3.2, page 53). This tab contains the standard-specific reinforcement specifications. In the following, the specifications for EN 1992-1-1 are described.

In the tab's bottom section the button [Set Default Values] is displayed. Use this button to reset the initial values of the currently set standard.

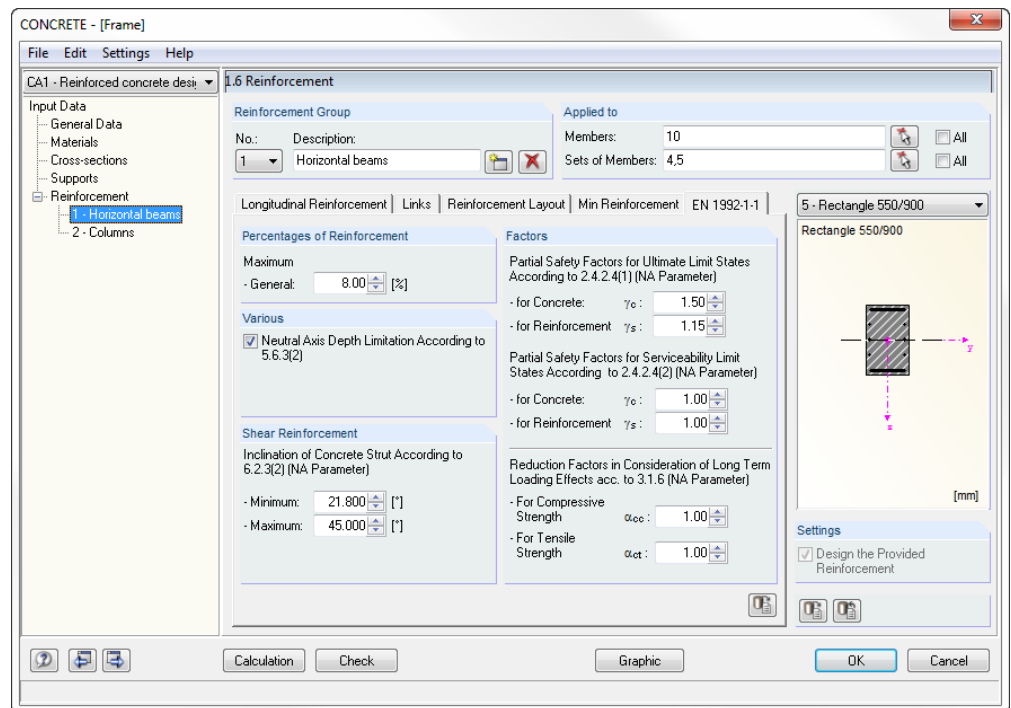


Figure 3.26: Table 1.6 Reinforcement, tab EN-1992-1-1

Percentages of Reinforcement

The entry in this input field defines the general maximum reinforcement ratio for beams. The standard EN 1992-1-1, 9.2.1.1 (3) recommends to use the 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, also in the zones of overlapping joints.

Various

If the concrete compression area is no longer able to absorb the compression forces, a compression reinforcement is required. This case occurs when the bending moment resulting from the concrete edge's compression strain of -3.50 ‰ and the strain ϵ_{yd} reaching the yield strength of the reinforcing steel is exceeded. For B 500 a related neutral axis depth of

$x/d = 0.617$ appears. If the model includes continuous beams, horizontal beams with un-displaceable frames and structural components mainly stressed by bending, it is recommended not to use this limiting bending moment to its full capacity in order to ensure a sufficient ability for rotation.

Tick the check box to 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 from strength class C55/67.

Shear Reinforcement

The two input fields define the allowable zone for the inclination of concrete struts. In case user-defined angles are outside the allowed limits mentioned in the standard, an error message appears.

EN 1992-1-1 describes an integrated model for the calculation of the shear force resistance. For structural components with shear reinforcement perpendicular to the component's axis ($\alpha = 90^\circ$) the following can be applied:

$$V_{Rd,s} = \frac{A_{sw}}{s} \cdot z \cdot f_{ywd} \cdot \cot \theta \quad \text{EN 1992-1-1, Eq. (6.8)}$$

Equation 3.2

where

A_{sw}	Cross-sectional area of shear reinforcement
s	Distance between links
f_{ywd}	Design value for yield strength of shear reinforcement
z	Lever arm of internal forces (assumed for 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. In this way, the equation can take into account the fact that a part of the shear force is absorbed by crack friction. Thus, the structural system is 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

The inclination of the concrete strut θ can vary between the following values:

	Minimum inclination	Maximum inclination
θ	20.8°	45°
$\cot \theta$	2.5	1

Table 3.2: Limits for inclination of concrete strut

The national annex for Germany allows for a flatter inclination of concrete struts with 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

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

Factors

The two input fields in the upper part define the *Partial Safety Factor* for concrete γ_c and for reinforcing steel γ_s , which is used for the ultimate limit state design. The values according to EN 1992-1-1, table 2.1N are preset.

Accordingly, the two input fields in the lower part define the two partial safety factors for the serviceability limit state design. The program presets the values of 1.0 which can also be found in the national annex for Germany.

The *Reduction Factor* α used to consider long-term effects on the concrete strength can be specified separately for the compressive and the tensile strength. The values recommended in EN 1992-1-1, 3.1.6 are preset in both input fields.

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

3.5.6 Tapered

This tab appears only in case a tapered member has been defined in the RSTAB model.

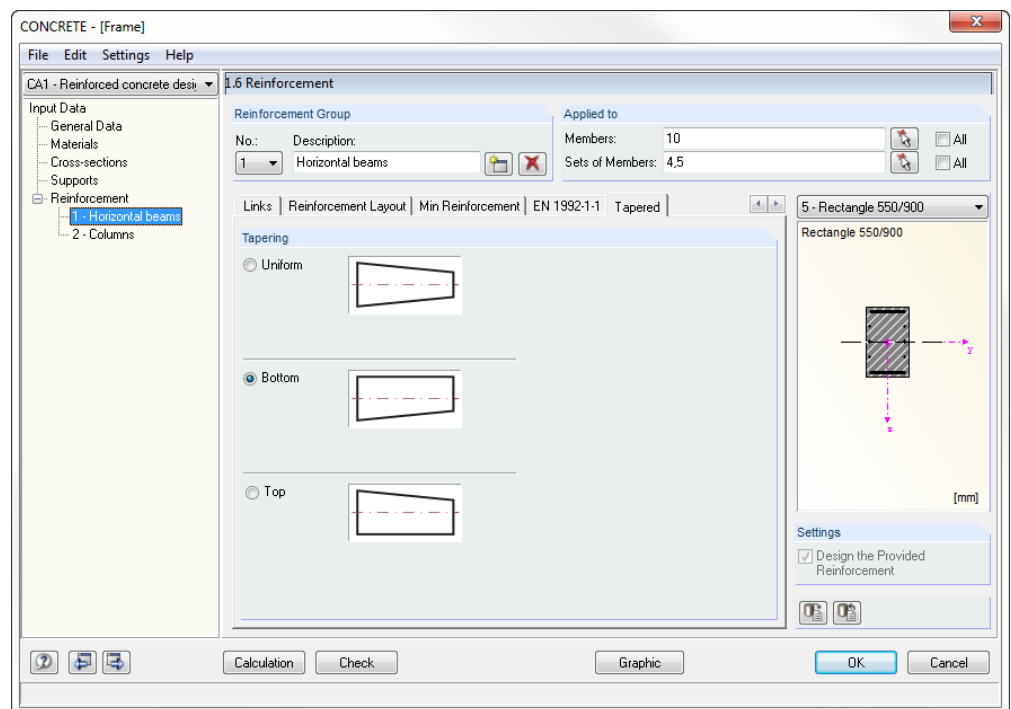


Figure 3.27: Table 1.6 Reinforcement, tab Tapered

CONCRETE also designs tapered members provided that the same cross-section type is defined for the member start and the member end. Otherwise it is not possible to interpolate intermediate values and RSTAB displays a corresponding error message before the RSTAB calculation is started.

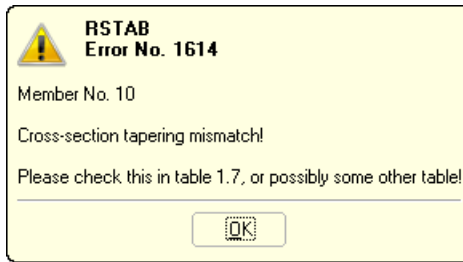


Figure 3.28: Error message in case of incompatible tapered cross-sections



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

Tapering

The following three options can be selected to describe the taper specifically:

- Uniform
- Bottom
- Top

The specification affects the design as well as the arrangement of the longitudinal reinforcement.

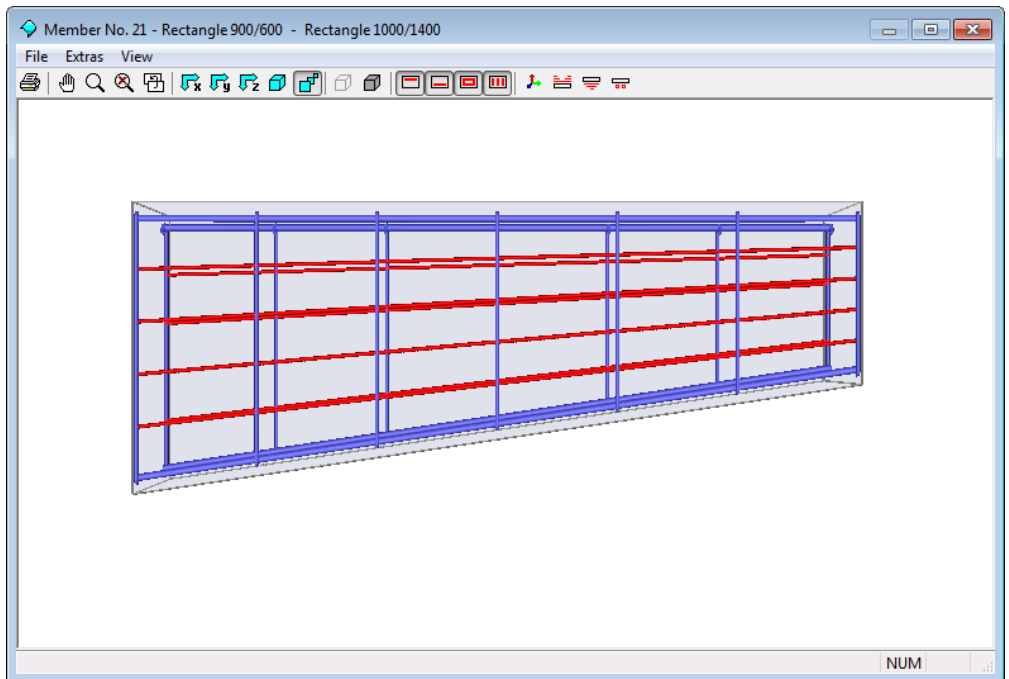
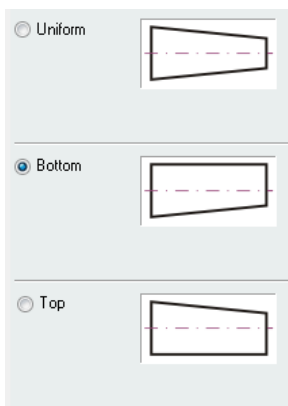


Figure 3.29: Taper with inclined bottom side

3.5.7 Fire Resistance

The final tab of this table is only available if you have entered data for the fire protection design in table 1.1 *General Data* (see chapter 3.1.4, page 57). In this tab you can define the fire-specific design settings.



In the tab's bottom section the button [Set Default Values] is displayed. Use this button to reset the initial values. The fire protection design is performed according to the simplified calculation method described in EN 1992-1-2, 4.2 (see chapter 2.3, page 17).

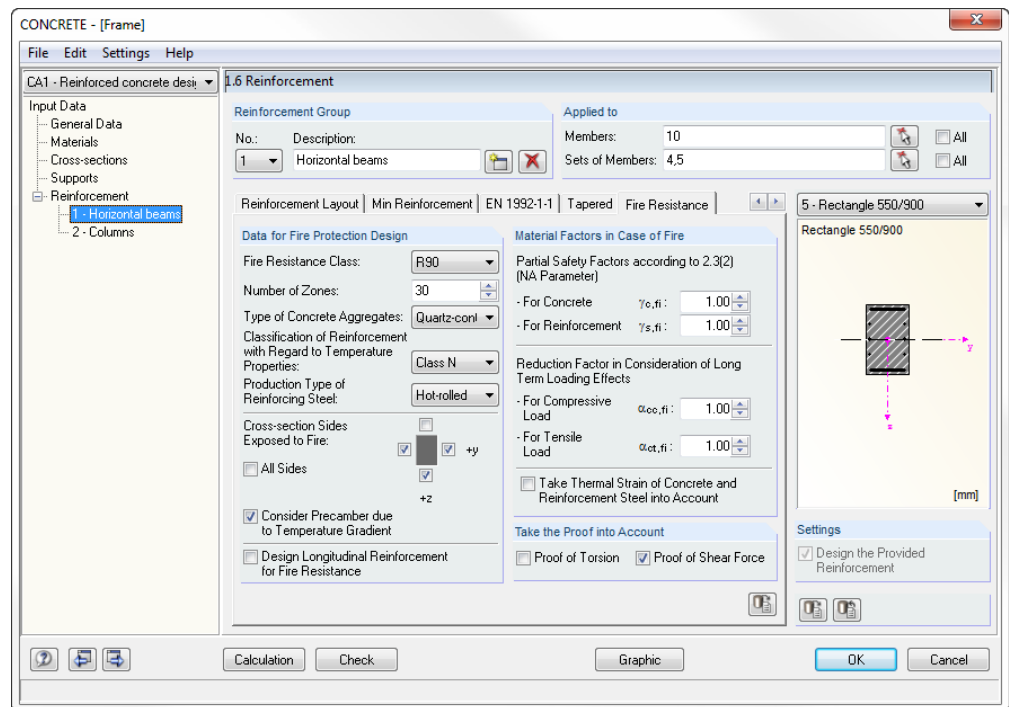


Figure 3.30: Table 1.6 Reinforcement, tab Fire Resistance

Data for Fire Protection Design

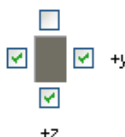
The following five selection lists control the parameters, affecting the fire protection design significantly:

- Fire resistance class (according to EN 1992-1-2, 1.6.1 (1))
- Number of zones (zone method according to EN 1992-12, annex B.2)
- Type of concrete aggregate (cf. Figure 2.6, page 18 and Figure 2.8, page 20)
- Classification of reinforcement (cf. Figure 2.11, page 22)
- Production type of reinforcing steel (cf. Figure 2.12, page 23)

For more theoretical details on these parameters, see the description in chapter 2.3, page 17.

In this dialog section, you also define the *Cross-section Sides Exposed to Fire*. If not *All Sides* are affected by charring, clear the check box. Then the check boxes around the cross-section symbol to the right become active and you can specify the individual sides directly. The directions refer to the local member axes.

In case of asymmetrical effects of fire, the cross-section is stressed by an additional thermal *Precamber* due to the temperature gradient that must be taken into account for the calculation according to EN 1992-1-2, 2.4.2 (4). This thermal precamber affects the ultimate limit state of structural components like brackets calculated according to the second-order analysis. Internally, the program creates a member load in the form of a precamber of the cross-section and superimposes the load with the design loads.



By ticking the check box *Design Longitudinal Reinforcement for Fire Resistance*, the reinforcement proposal will take into account the effects of fire in addition to the bearing capacity.

Material Factors in Case of Fire

The two input fields in the upper part define the *Partial Safety Factor* for concrete γ_c and for reinforcing steel γ_s , which is used for the fire protection design. The values recommended in EN 1992-1-2, 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 the compressive and the tensile strength. The values of 1.0 recommended in EN 1992-1-1, 3.1.6 are preset in both input fields.

With the option *Take Thermal Strain of Concrete and Reinforcement Steel into Account*, it is possible to consider the difference between the strain of the "hot" reinforcement and the regular thermal strain of the concrete cross-section in the form of a pre-shortening of the rebar: In case of loading due to temperature, thermal longitudinal strains occur in the concrete and in the reinforcing steel varying within the cross-section because of different temperature distributions. The thermal strains cannot arise everywhere freely 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-shortened. The zone method according to EN 1992-1-2 includes only a calculation of structural components, this means that the thermal additional strains in the centroid are not taken into account by the standard. However, according to HOSSEY [19] it is required to take into account those thermal strains for calculations according to the second-order analysis. The concrete's thermal strain is calculated across the entire concrete cross-section using the temperature's average value.

Take the Proof into Account

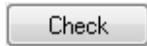
Annex D of EN 1992-1-2 contains a calculation method for the shear and the torsional design of structural components exposed to fire. This method of calculation is implemented in the add-on module CONCRETE and can be activated separately.

As this calculation method for the shear and torsional design is not allowed in Germany, both options are not available for the design according to German standards.

4. Calculation



To start the calculation, click the [Calculation] button that is available in all input tables.



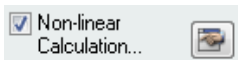
4.1 Plausibility Check

Before you start the calculation, it is recommended to check the input data. The [Check] button is available in each input table of the module and is used to start the check. If no input error is detected, the following message will be displayed.



Figure 4.1: Plausibility check is ok

4.2 Details for Non-linear Analysis



For the analysis according to the non-linear design method the program provides control parameters affecting the calculation method and the convergence behavior. They are managed in the dialog box *Settings for Non-linear Calculation* that you open by clicking the [Settings] button available in table 1.1 (see Figure 3.1, page 52 or Figure 3.6, page 55). The dialog box offers three tabs.

4.2.1 Analysis Approach

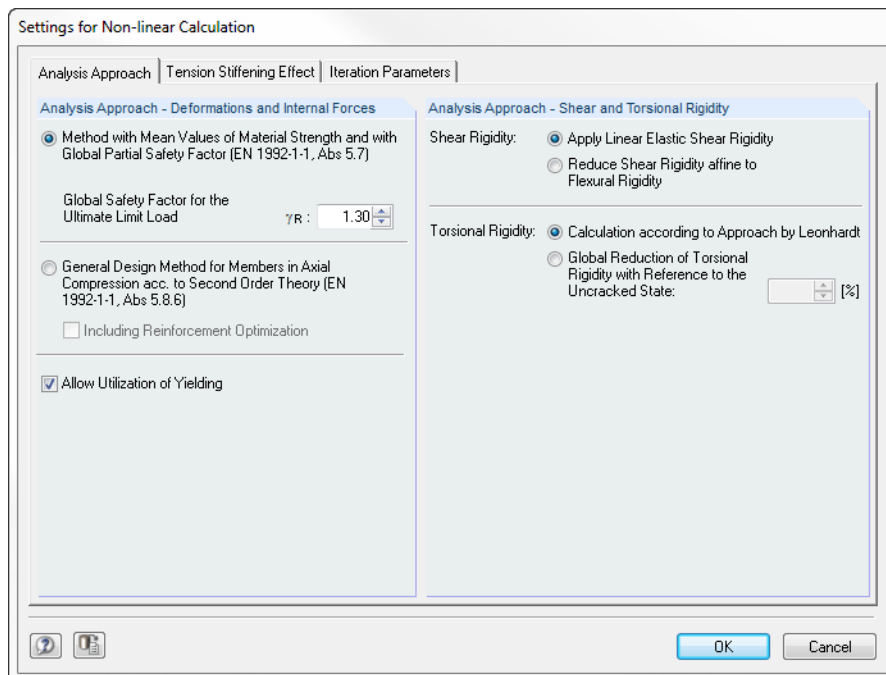


Figure 4.2: Dialog box *Settings for Non-linear Calculation*, tab *Analysis Approach* for EN 1992-1-1

The contents in the tab are in line with the selected standard.

Deformations and Internal Forces

Method with Mean Values of Material Strength

As described in chapter 2.4.7, two methods for the non-linear calculation are standardized according to EN 1992-1-1. The program presets the *Method with Mean Values of Material Strength and Global Partial Safety Factor* according to EN 1992-1-1, 5.7.

Global Safety Factor for the Ultimate Limit Load

The method according to EN 1992-1-1, 5.7 was standardized for the uniform application of only one safety concept (see chapter 2.4.7, page 45).

According to EN 1992-1-1, 5.7 (NA.10) for Germany the value is defined as follows:

- $\gamma_R = 1.3$ for permanent and temporary design situations and analysis for fatigue
- $\gamma_R = 1.1$ for extraordinary design situations

General Design Method

Alternatively, you can use the *General Design Method for Members in Axial Compression acc. to Second Order Theory* in accordance with EN 1992-1-1, 5.8.6. This method is especially useful for calculating slender compression elements because it is possible to perform a cost-efficient design due to the determination of deformations and internal forces with verified mean values. Background knowledge and further descriptions can be found in chapter 2.4.7.2.

Including Reinforcement Optimization

It is possible to optimize the reinforcement for simple structures. However, please note the indications to be found in [32], page 233 concerning the non-linear design concept: It may be that a reinforcement optimization cannot reach a convergent result due to permanent stiffness change.

Allow Utilization of Yielding

The check box *Allow Utilization of Yielding* is enabled for both calculation methods according to EN 1992-1-1, 5.7 and 5.8.6 described above. The reason for it can be found in section 8.6.1 (5) of the German standard DIN 1045-1 not allowing plastic releases (curvatures $(1/r)_m > (1/r)_y$) for structural components stressed by longitudinal compression. As a result of the abrupt decrease of stiffness when plastic zones or releases are created, they often lead to loss of stability in slender compression elements and, thus, to failure of the column.

The check box is deactivated by default: The criterion is taken into account for the calculation of cross-section curvatures. No plastic curvatures are possible.

Shear and Torsional Rigidity

Apply Linear Elastic Shear Rigidity

The shear areas are calculated linear-elastically and without consideration of a reduction due to cracking.

Reduce Shear Rigidity affine to Flexural Rigidity

The linear-elastic shear stiffness diagram is reduced in line with the diagram for flexural resistance. The theoretical background is described in chapter 2.4.5.2 on page 34.

Calculation according to Approach by LEONHARDT

The torsional stiffness is calculated by taking into account cracking according to the method of LEONHARDT [22] (cf. chapter 2.4.5.2, page 35).

Global Reduction of Torsional Rigidity

The torsional stiffness in case of cracking is reduced to a user-defined residual value. The program presets a residual torsional stiffness of 10 % resulting from the high decrease of torsional rigidity (cf. Figure 2.23, page 35).

4.2.2 Tension Stiffening Effect

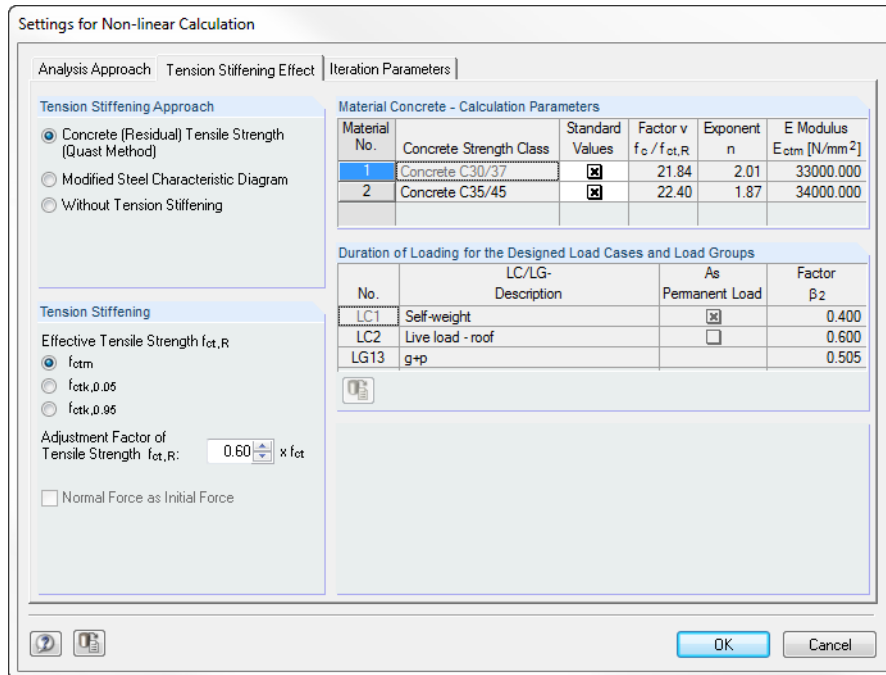


Figure 4.3: Dialog box *Settings for Non-linear Calculation*, tab *Tension Stiffening Effect*

The specifications for Tension-Stiffening (effectiveness of concrete between the 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 the residual tensile strength of concrete according to QUAST [11] that is defined depending on the decisive strain of the steel fibre within the tension zone. The approach is represented graphically in chapter 2.4.3.1 on page 29.

Modified Steel Characteristic Diagram

As described in chapter 2.4.3.2 on page 31 the Tension-Stiffening effect can also be considered by a modified characteristic steel curve. The computing time will be increased a little because the analysis requires in addition to the sole calculation in cracked sections (state II) a calculation in the uncracked state as well as a determination of crack internal forces.

Without Tension Stiffening

In case the tension stiffening effect is not taken into account, the program will distinguish only cracked and uncracked zones: In uncracked zones CONCRETE will calculate with the concrete's corresponding stiffness in uncracked sections (state I, considering provided longitudinal reinforcement). In cracked zones it will calculate with stiffnesses available in pure state II.

Tension Stiffening

The calculated values of the concrete tensile strength determine the exponent of the parabola area in such a way that the result will be an increase in line with the compression zone ($E_{cm} = E_{ctm}$).

Effective Tensile Strength $f_{ct,R}$

To take into account the appropriate safety level, it is possible to select one of the following strengths for the concrete tensile strength to be applied:

- f_{ctm} Mean value of axial tensile strength
- $f_{ctk;0.05}$ Characteristic value of 5%-quantile of axial tensile strength
- $f_{ctk;0.95}$ 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 by calculation can be influenced by means of an adjustment factor. Thus, it is possible to consider boundary conditions such as an existing damage.

PFEIFFER [25] 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 ticked, the axial force will be kept constant for the calculation of crack moments. This is useful for example when you apply a prestress. If it is not ticked, the entire load vector will be taken into account for the calculation of crack internal forces.

Material Concrete - Calculation Parameters

The concrete's *Standard Values* of the calculation parameters are preset (cf. chapter 2.4.3.1, page 29). If you clear the check box (third table column), you can directly influence the stress-strain curve of the tension zone. As the values depend on each other, the relevant values will be adjusted accordingly after a modification.

Duration of Loading for the Designed Load Cases and Load Groups

This section manages the load duration factors β for applying the reduction term $(\varepsilon_{sr2} - \varepsilon_{sr1})$, this means the strains of the governing steel fibres for the crack internal forces in the cracked or uncracked state (cf. chapter 2.4.3.2, page 31). The factor β depends on the time of load duration:

- | | |
|-----|------------------------------------|
| 0.4 | Permanent load or repeated loading |
| 0.6 | Temporary loading |

When using the approach of a modified characteristic steel curve you can decide if a load case is considered *As Permanent Load* by ticking the check box.

For load groups the applied factor β_2 represents the average of the respective β_2 -values of the load cases contained in the LC group.



Note for compression elements

Generally, the Tension-Stiffening model according to QUAST (*Concrete (Residual) Tensile Strength*) has to be used for the design of compression elements. The residual tension force can be influenced by means of the *Adjustment Factor of Tensile Strength*.

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 area CONCRETE calculates linear-elastically with a constant modulus of elasticity for concrete ($E_{cm,eff}$). In case of predominant compression, however, considerably expanded curvatures occur for minor moment loadings due to the non-linear diagram of the concrete's stress-strain curve. Thus, results may be remarkably on the unsafe side.

4.2.3 Iteration Parameters

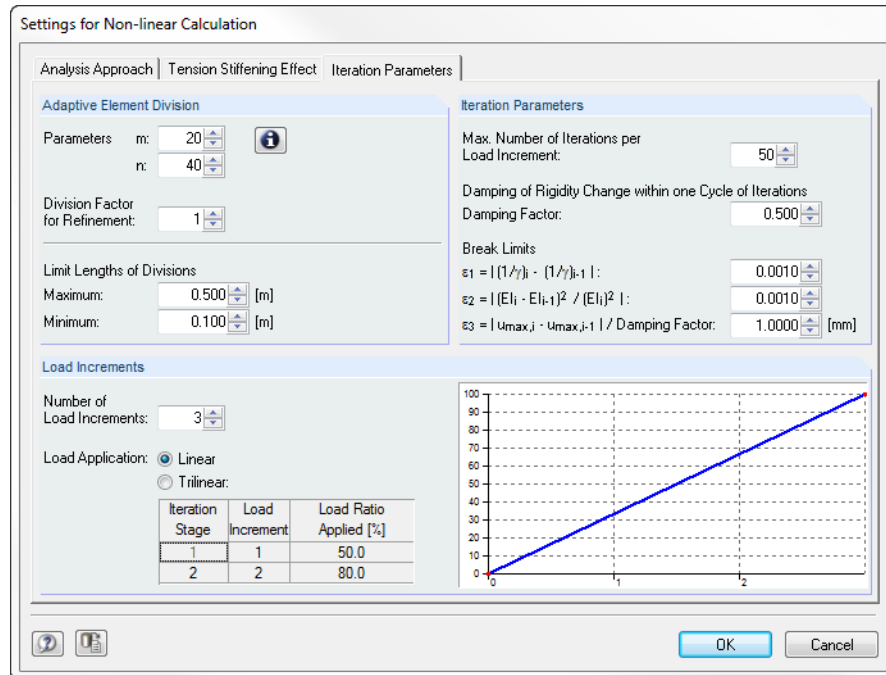


Figure 4.4: Dialog box *Settings for Non-linear Calculation*, tab *Iteration Parameters*

Information on this tab can also be found in chapter 2.4.9 *Convergence*, page 51.

Adaptive Element Division

The division of elements has a significant impact on the non-linear calculation: When the stiffness changes within an element are too large, you may get inaccurate design results or a non-convergent calculation may be the result. The *Adaptive Element Division* controls the differences in stiffness at the element nodes and applies appropriate intermediate divisions if differences are too large.

Parameters m / n

As CONCRETE calculates with constant mean stiffnesses within a finite element, it is recommended to select a finer division of elements where major stiffness changes occur (cf. Figure 2.13, page 24). To access more details about the formal context of the parameters m and n , click the [Info] button in the dialog box.

According to [25] values for $m \cong 20$ and $n \cong 40$ have proved to be acceptable. However, it must be mentioned that PFEIFFER calculates with tangential bending rigidities in line with the used transfer matrix method. Those rigidities show a considerably higher gradient in the area where cracking and yielding starts. For this reason it is possible that different values for m and n are reasonable in particular cases.

Division Factor for Refinement

With this input field you specify the number of nodes by which the finite element is divided.

Limit Lengths of Divisions

The *Maximum* limit length of the member division specifies the largest possible length of an FE beam element. The *Minimum* limit length represents the smallest allowable length for a finite element.

The limit lengths represent also the limits for the adaptive element division. Comparative calculations showed us that the preset lengths of $0.5 m$ and $0.1 m$ are reasonable values. Lengths that are further reduced affect the results only insignificantly but they slow down

the calculation considerably due to the increased number of finite elements. However, in case of very large stiffness changes (for example for underreinforced cross-sections from state I to state II) it may be reasonable to refine the minimum limit length.

Iteration Parameters

The iteration parameters allow you to control the iteration process.

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 defined 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 Rigidity Change within one Cycle of Iterations

In the course of two successive iteration steps, the program determines the stiffness difference 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. By a specific reduction of stiffness changes between two iteration steps it is possible to work against the calculation's oscillation.



The higher the damping factor is, the smaller is the damping's influence. If the factor is equal to 1, the damping has no influence on the iterative calculation.

Break Limits $\varepsilon_1 / \varepsilon_2 / \varepsilon_3$

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 the linear static analysis (for example beam deformations in SLS), it is nevertheless recommended to refine the tolerances for stability analyses ($\varepsilon_1 = \varepsilon_2 \leq 0.001$). Example 3 in chapter 9.3 shows the effect in a clear presentation.

Additionally, the break limit ε_3 controls the deformation change. This criterion observes how the size of the maximum deformation is changing. The specified damping factor is also taken into account.

Load Increments

The loading can be applied step by step in order to avoid or attenuate an abrupt stiffness change within the individual finite elements ("adapting" the system to the loading). The objective is to avoid the generation of major stiffness changes during an iteration. When the loading is applied step by step, it is always possible in the iteration step of a load increment to fall back on the relevant final stiffness of the element from the previous load increment.

Number of Load Increments

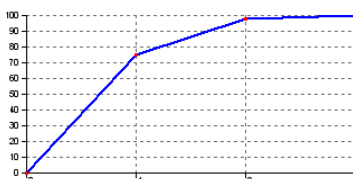
The input field determines the number of individual load increments for the non-linear calculation.

Load Application

- *Linear*
The load is applied in linear steps.

- *Trilinear*
As you can react to the load-dependent stiffness development only with a correspondingly fine graduation when loads are applied linearly, CONCRETE offers a trilinear load application as alternative. Thus, it is possible to respond accordingly to boundary conditions like creeping near the state of failure.

The trilinear load application is managed by the table: You have to specify two intermediate points characterizing the load quantity applied until then.



Trilinear load application:
75 % - 98 % - 100 %

4.3 Start Calculation

Calculation

You can start the calculation by clicking the [Calculation] button that is available in each input table of the add-on module CONCRETE.

CONCRETE searches for the results of the load cases, load groups and load combinations that should be designed. If they cannot be found, the RSTAB calculation starts to determine the design relevant internal forces. In this determination process, the calculation parameters preset in RSTAB are applied.

You can also start the CONCRETE calculation out of the RSTAB user interface. All add-on modules are listed in the dialog box *To Calculate* like load cases or load groups. To open the dialog box in RSTAB,

select **To Calculate** on the **Calculate** menu.

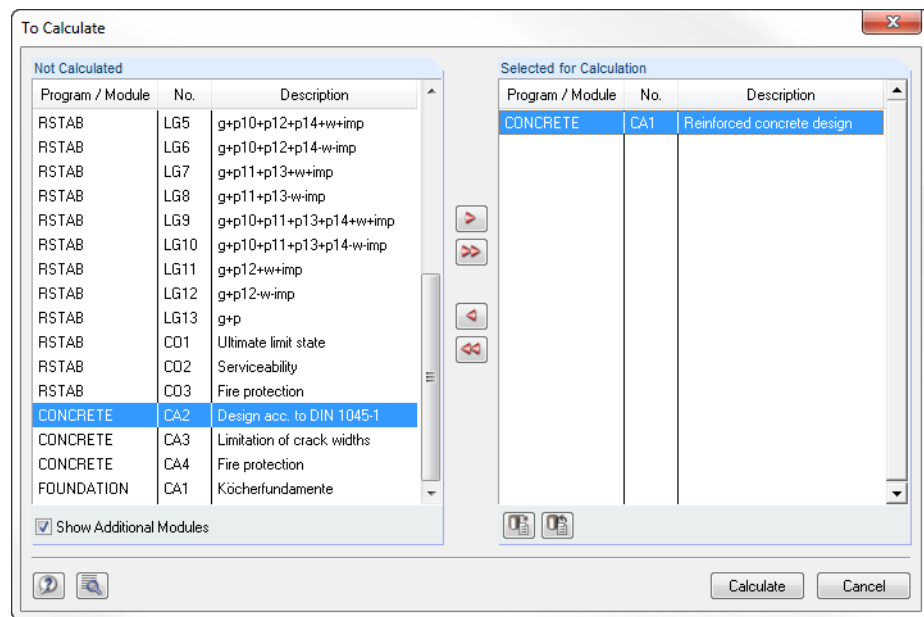


Figure 4.5: Dialog box *To Calculate*

If the CONCRETE design cases are missing in the *Not Calculated* list, tick the check box *Show Additional Modules* below the list.

To transfer the selected CONCRETE cases to the list on the right, use the button [▶]. Start the calculation by using the [Calculate] button.

You can also use the load case list in the RSTAB toolbar to calculate a selected CONCRETE case directly: Select the relevant design case and click the button [Results on/off].

Calculate



Figure 4.6: Direct calculation of a CONCRETE design case in RSTAB

Subsequently, you can observe the calculation process in a separate dialog box.

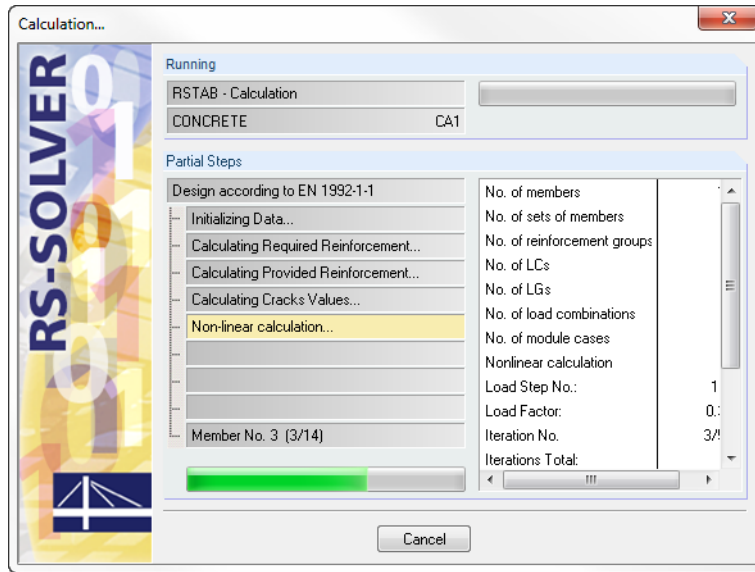


Figure 4.7: CONCRETE design process

5. Results

Table 2.1 *Required Reinforcement by Cross-section* is displayed immediately after the calculation.

The reinforcement areas required for the ultimate limit state design are listed in the results tables 2.1 to 2.4. If the program created a reinforcement proposal, the provided reinforcement including steel schedule is displayed in the results tables 3.1 to 3.4. The results for the crack width design are represented in the results tables 4.1 to 4.4. Tables 5.1 to 5.4 show the results of the fire protection design. Tables 6.1 to 6.4 are reserved for the results determined by a non-linear design.



All tables can be selected directly in the CONCRETE navigator. You can also use the two buttons shown on the left or the function keys [F2] and [F3] to select the previous or subsequent table.

To save the results and quit the add-on module CONCRETE, click [OK].

In the following, the different results tables are described in sequence. Evaluating and checking results is described in chapter 6 *Results Evaluation*, page 121.

5.1 Required Reinforcement

5.1.1 Required Reinforcement by Cross-section

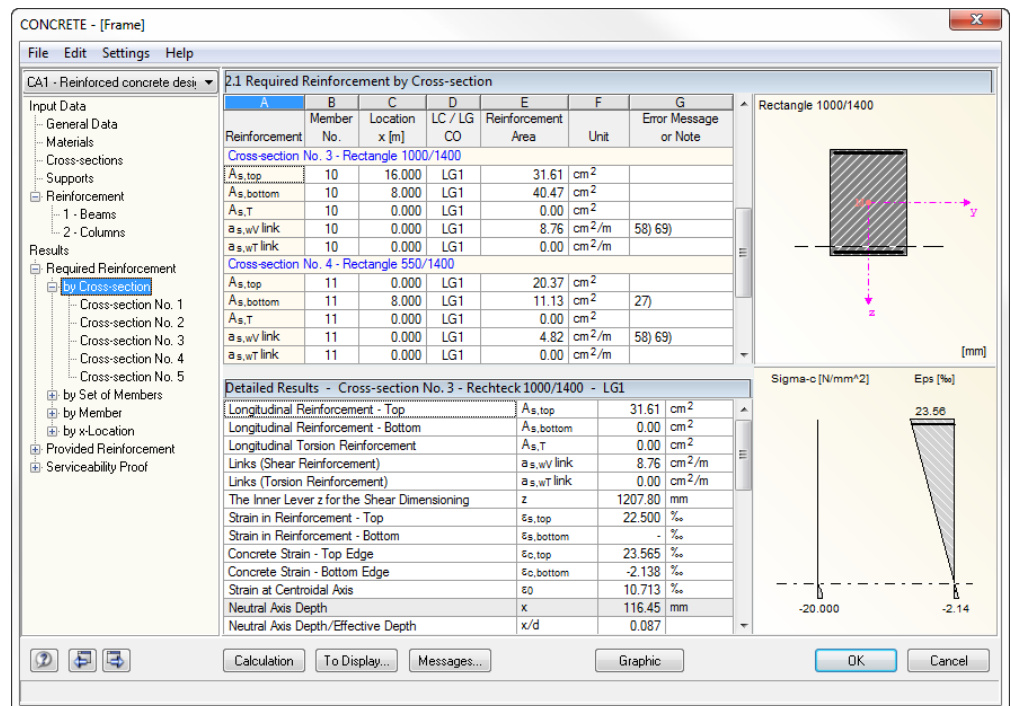


Figure 5.1: Table 2.1 *Required Reinforcement by Cross-section*

For all designed cross-sections the program displays the maximum required reinforcement areas resulting from the parameters of the reinforcement groups and the internal forces of the governing actions.

The reinforcement areas of the longitudinal and the link reinforcement are listed according to cross-sections. In both parts of the table the program displays the reinforcement types

and design details that have been activated in the dialog box *Results to Display* (see Figure 5.2).

In the lower part of the table the *Detailed Results* for the entry selected in the table row above are shown. Thus, due to the design details, it is possible to evaluate the results specifically. The output of the intermediate results in the lower part will be updated automatically as soon as another row is selected in the upper part.

Reinforcement

The following longitudinal and link reinforcements are preset:

Reinforcement	Explanation
$A_{s,top}$	Reinforcement area of required top longitudinal reinforcement due to bending with or without axial force or axial force only
$A_{s,bottom}$	Reinforcement area of required bottom longitudinal reinforcement due to bending with or without axial force or axial force only
$A_{s,T}$	Reinforcement area of required longitudinal torsion reinforcement, if applicable
$a_{sw,V \text{ link}}$	Area of required shear reinforcement for absorption of shear force, referring to standard length of 1 m
$a_{sw,T \text{ link}}$	Area of required link reinforcement for absorption of torsional moment, referring to standard length of 1 m

Table 5.1: Longitudinal and link reinforcements



The bottom reinforcement is defined on the member side in direction of the positive local member axis z . Accordingly, the top reinforcement is defined in direction of the negative z -axis. To display the local member axes, use the *Display* navigator in the RSTAB graphical user interface or the context menu of the member.

Click the button [To Display] to open a dialog box where you can specify the reinforcement and intermediate results that you want to display in both parts of the table. The settings also define the types of results appearing in the printout report.

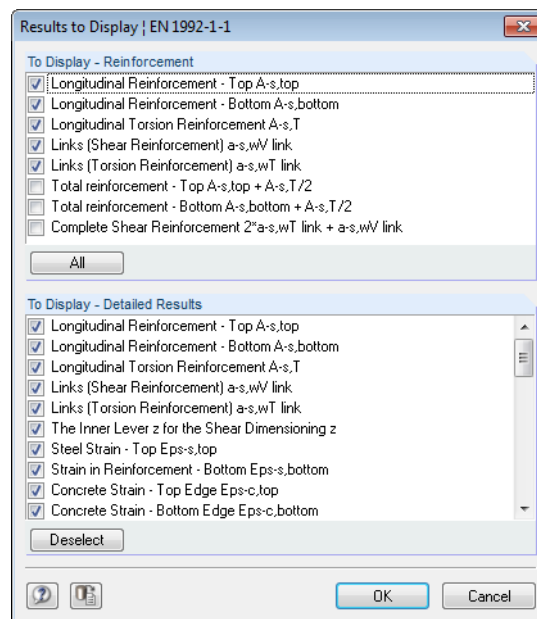


Figure 5.2: Dialog box *Results to Display*

Member No.

For each cross-section and each reinforcement type, the table shows the number of the member with the maximum reinforcement area.

Location x

The column shows the respective x-location for which the program has determined the member's maximum reinforcement. For the table output, the program uses the following RSTAB member locations x:

- Start and end node
- Partition points according to possibly defined member division
- Extreme values of internal forces

LC / LG / CO

This column displays the numbers of the load cases, load groups, load combinations or super combinations that are decisive for the respective design.

Reinforcement Area

Column E contains information about the maximum reinforcement areas required for each reinforcement type. They are necessary to fulfill the ultimate limit state design.

The reinforcements' *Unit* specified in column F can be adjusted. To modify the corresponding settings,

select **Units and Decimal Places** on the **Settings** menu.

The dialog box shown in Figure 8.6 on page 134 opens.

Error Message or Note

The final column indicates non-designable situations or notes referring to design problems. The numbers are explained in the status bar.

To display all messages of the currently selected design case, use the [Messages] button shown on the left. A dialog box with relevant information appears.

Messages...

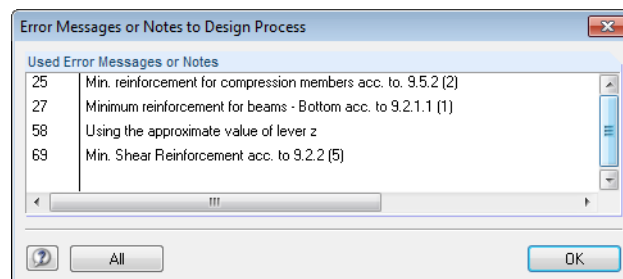


Figure 5.3: Dialog box *Error Messages or Notes*

5.1.2 Required Reinforcement by Set of Members

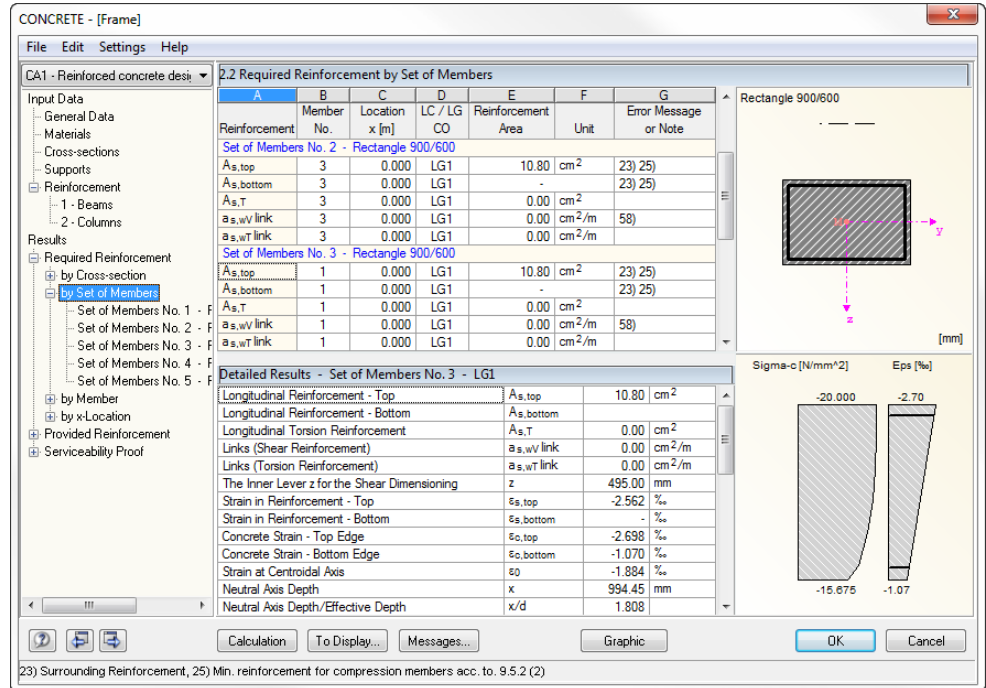


Figure 5.4: Table 2.2 Required Reinforcement by Set of Members

This table presents the maximum reinforcement areas that are required for the individual sets of members. Details on the columns can be found in the previous chapter 5.1.1.

5.1.3 Required Reinforcement by Member

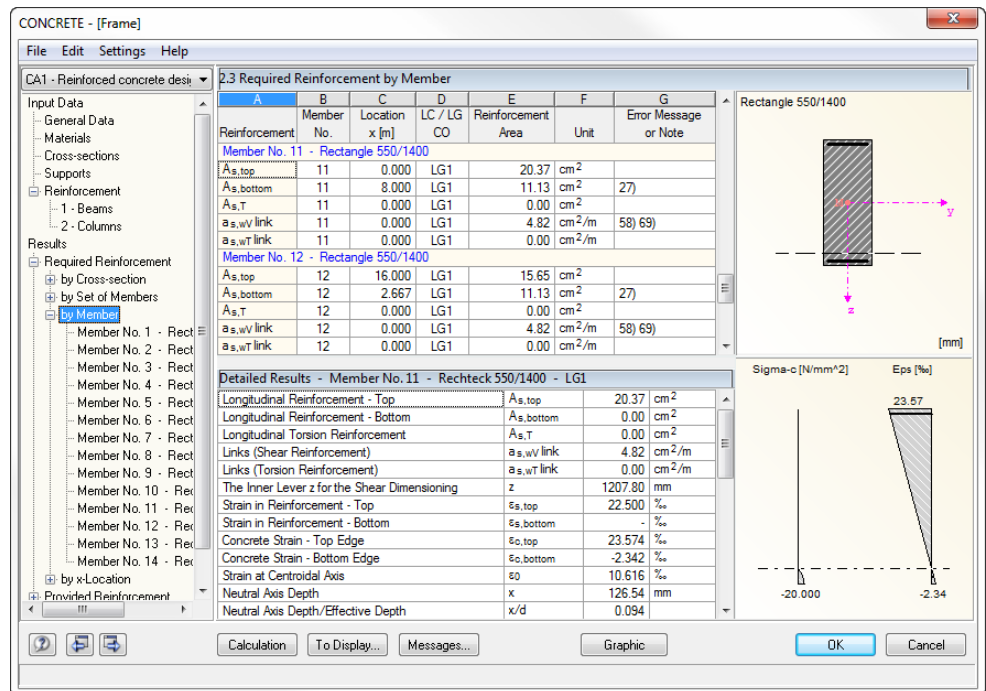


Figure 5.5: Table 2.3 Required Reinforcement by Member

The maximum reinforcement areas are listed according to members. For tapered beams both cross-section descriptions are displayed to the right of the member numbers.

5.1.4 Required Reinforcement by x-Location

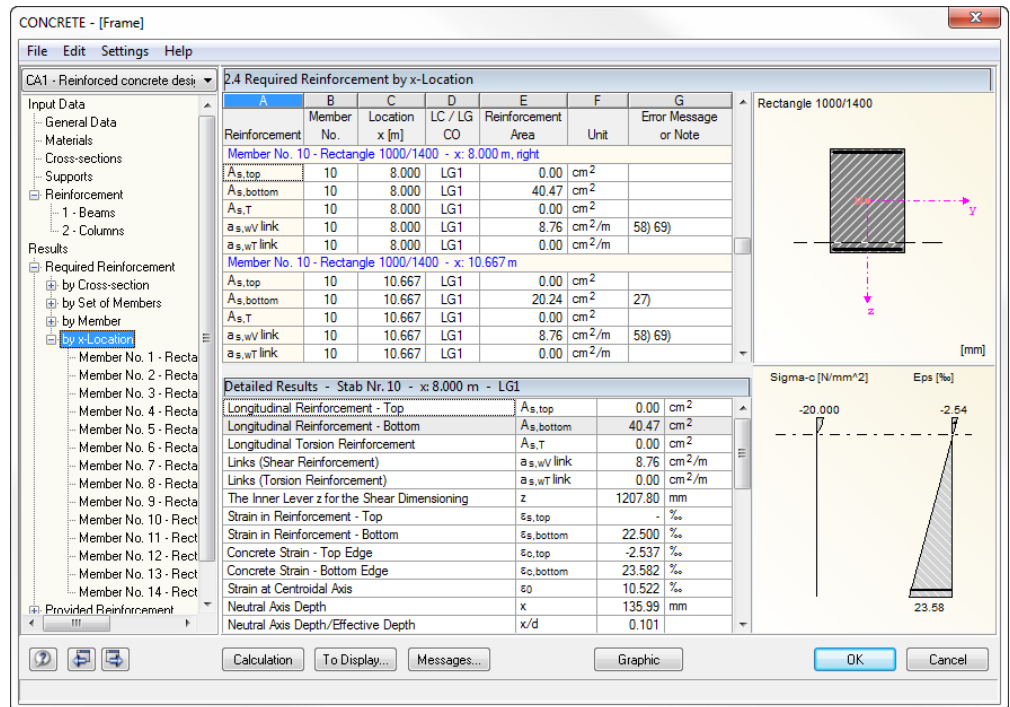


Figure 5.6: Table 2.4 Required Reinforcement by x-Location

This table shows for each member the required reinforcement areas including intermediate results listed by x-location:

- Start and end node
- Partition points according to possibly defined member division
- Extreme values of internal forces

Locations of discontinuity are indicated separately.

The table offers you the special opportunity to display specific information concerning the design results. In this way, it is possible for example to check the required link reinforcement with the corresponding details for a particular location on the member (designed location).

The different columns are described in detail in chapter 5.1.1.

5.1.5 Required Reinforcement Not Designable

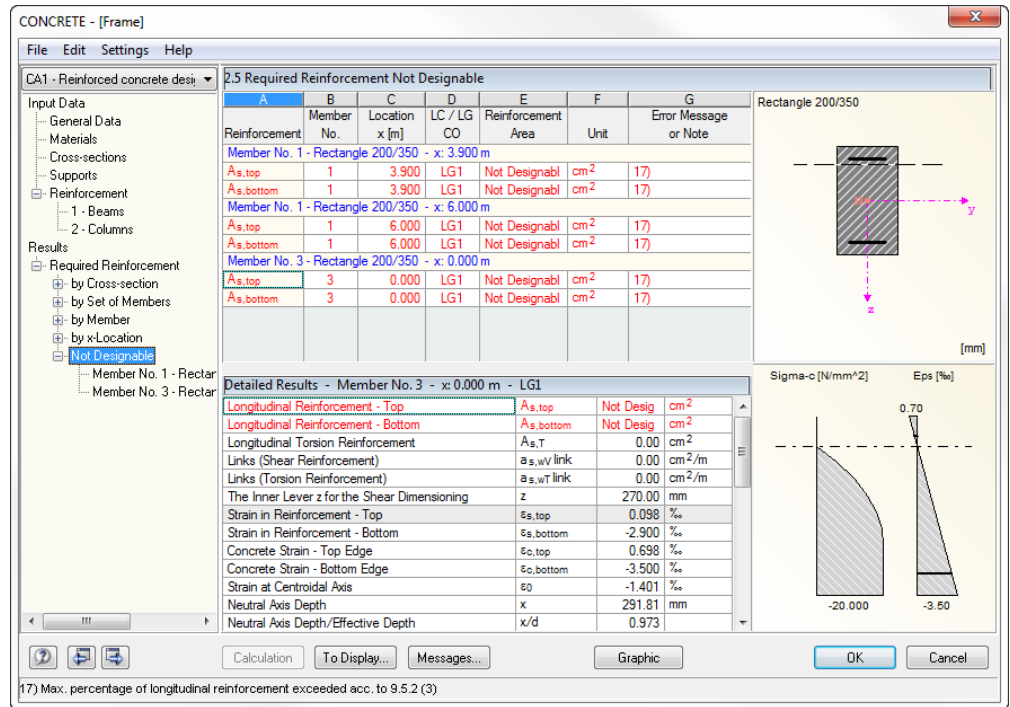


Figure 5.7: Table 2.5 Required Reinforcement Not Designable

This table is displayed only if the program has detected failed designs or any other problems during the CONCRETE analysis. 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...

Click the [Messages] button to display all specific conditions that have occurred during the design of the current x-location.

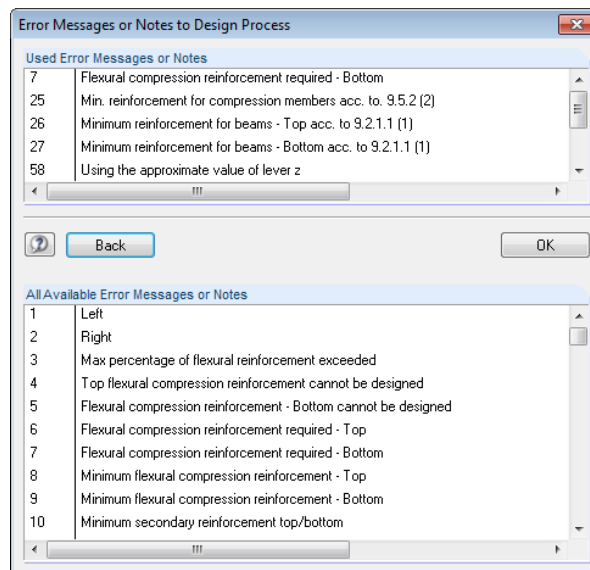


Figure 5.8: Dialog box Error Messages or Notes to Design Process

All

Use the [All] button in this dialog box to show all messages available for CONCRETE.

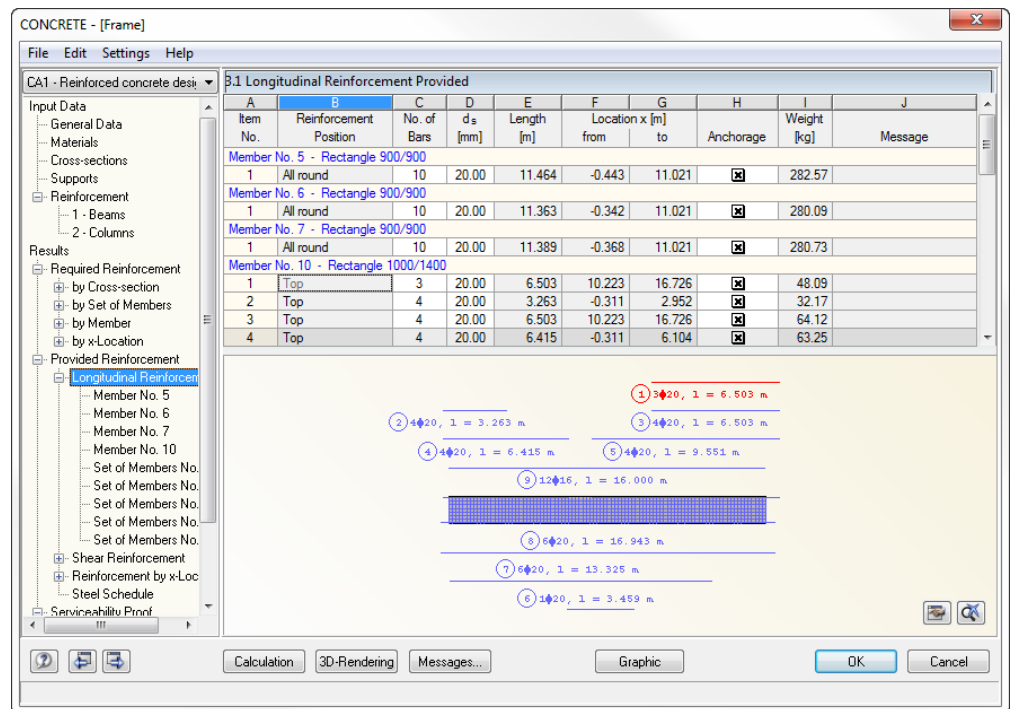
5.2 Provided Reinforcement

The results tables 3.1 to 3.4 appear only in case the option *Design the Provided Reinforcement* was activated in table 1.6 *Reinforcement* (see page 66), and if no design problems were detected by the program (see chapter 5.1.5, page 94). Also the serviceability limit state design as well as the non-linear calculation require the determination of a provided reinforcement.

CONCRETE determines the reinforcement proposal for the longitudinal and the link reinforcement by using the specifications defined in table 1.6. The program tries to cover the required reinforcement, taking into account the corresponding parameters (specified rebar diameter, possible number of reinforcement layers, curtailment, type of anchorage) by means of the least possible amount of rebars or reinforcement areas.

The proposed reinforcement can be edited in the *Reinforcement Provided* tables in order to adjust the diameter, number, position and length of the individual reinforcement groups to the respective requirements.

5.2.1 Longitudinal Reinforcement Provided



The screenshot displays the 'CONCRETE - [Frame]' application window. The main window title is '3.1 Longitudinal Reinforcement Provided'. The interface includes a menu bar (File, Edit, Settings, Help) and a toolbar. On the left, there is a tree view for 'Input Data' and 'Results'. The 'Results' section is expanded to show 'Provided Reinforcement' and 'Longitudinal Reinforcement'. The main area contains a table with columns A through J, and a graphic below it showing a reinforcement layout with numbered callouts (1-9) and their corresponding parameters.

Item No.	Reinforcement Position	No. of Bars	d_s [mm]	Length [m]	Location x [m] from	Location x [m] to	Anchorage	Weight [kg]	Message
Member No. 5 - Rectangle 900/900									
1	All round	10	20.00	11.464	-0.443	11.021	<input checked="" type="checkbox"/>	282.57	
Member No. 6 - Rectangle 900/900									
1	All round	10	20.00	11.363	-0.342	11.021	<input checked="" type="checkbox"/>	280.09	
Member No. 7 - Rectangle 900/900									
1	All round	10	20.00	11.389	-0.368	11.021	<input checked="" type="checkbox"/>	280.73	
Member No. 10 - Rectangle 1000/1400									
1	Top	3	20.00	6.503	10.223	16.726	<input checked="" type="checkbox"/>	48.09	
2	Top	4	20.00	3.263	-0.311	2.952	<input checked="" type="checkbox"/>	32.17	
3	Top	4	20.00	6.503	10.223	16.726	<input checked="" type="checkbox"/>	64.12	
4	Top	4	20.00	6.415	-0.311	6.104	<input checked="" type="checkbox"/>	63.25	

The graphic below the table shows a reinforcement layout with numbered callouts (1-9) and their corresponding parameters:

- 1) $3\phi 20, l = 6.503 \text{ m}$
- 2) $4\phi 20, l = 3.263 \text{ m}$
- 3) $4\phi 20, l = 6.503 \text{ m}$
- 4) $4\phi 20, l = 6.415 \text{ m}$
- 5) $4\phi 20, l = 9.551 \text{ m}$
- 6) $12\phi 16, l = 16.000 \text{ m}$
- 7) $6\phi 20, l = 16.943 \text{ m}$
- 8) $6\phi 20, l = 13.325 \text{ m}$
- 9) $1\phi 20, l = 3.459 \text{ m}$

Figure 5.9: Table 3.1 *Longitudinal Reinforcement Provided*

The output results for the provided reinforcement are sorted by members and sets of members according to *Item* numbers (reinforcement groups).

The graphic in the lower table section represents the reinforcement including item members. The currently selected item (the row in the upper table section in which the pointer is placed) is highlighted in red. Modifications to the parameters entered in the upper section are updated and displayed immediately in the graphic.

The reinforcement proposal also takes into account structural regulations. For example, in accordance with EN 1992-1-1, 9.2.1.2, it is required to arrange a minimum reinforcement to supports that are assumed to be pinned, and it must cover at least 25 % of the maximum adjacent moment of span and be available across 25 % of the length of the final span (values for national annex Germany).

Item No.

The results are listed by *Items* having each the same properties (diameter, length).

The items of all members and sets of members are summarized in table 3.4 *Steel Schedule*.

Reinforcement Position

This table indicates the position of the reinforcement in the cross-section:

- Top
- Bottom
- Corners
- All round
- Secondary

For the reinforcement's arrangement CONCRETE takes into account the user specifications entered in table 1.6 *Reinforcement*, tab *Reinforcement Layout* (see chapter 3.5.3, page 71).

No. of Bars

The number of rebars of an item can be edited: Select the corresponding cell and click the button [...] to open the edit dialog box.

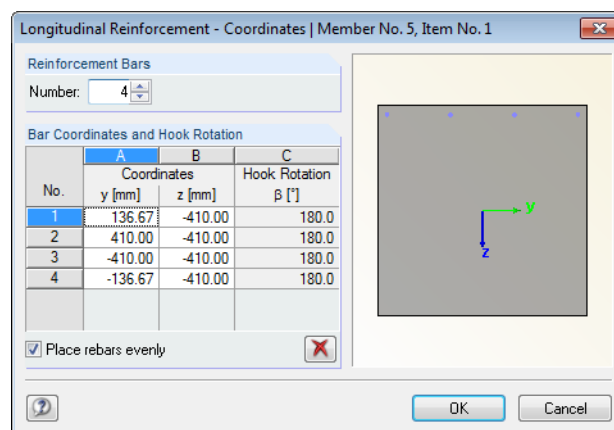


Figure 5.10: Dialog box *Longitudinal Reinforcement - Coordinates*



The *Number of Reinforcement Bars* can be set manually by using the spin buttons or by entering another number. The numerous input rows in the dialog section below allow you to adjust the position of each rebar. To delete a row selected in the lower section, click the [Delete] button.

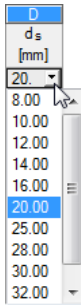
The position of a rebar is defined by means of its *Bar Coordinates*: The coordinates y and z determine the global distance from the cross-section's centroid. The angle β describes the inclination against the longitudinal member axis for the anchorage types "Hook" and "Bend". For example, a *Hook Rotation* about the angle $\beta = 90^\circ$ results in a downward rotation (i.e. in direction z) for the top reinforcement. The angle $\beta = 270^\circ$ rotates the anchorage end of the bottom reinforcement upwards. For the anchorage type "Straight", column C is of no importance.

3D-Rendering

When modifying the hook rotation, it is recommended to check the data subsequently in the rendering mode by clicking the [3D-Rendering] button.

d_s

The used rebar diameters affect the calculation of the inner lever arm of the forces and the number of rebars depending on the position. Use the list to change the rebar diameter for the current item number.



Length

This column displays for each item the total length of a representative rebar. The entry, composed by the required member length and the lengths of anchorage at both member ends, cannot be edited in this table.

Location x from ... to

These values represent the mathematical start and end positions of the rebar. They refer to the member's start node in RSTAB ($x = 0$). When the program determines the dimensions, it takes into account the support conditions and the lengths of anchorage l_1 and l_2 .

The specifications in both columns cannot be modified. To change data, use the [Edit] button in the bottom right corner of the graphic (see Figure 5.12, page 98).



Anchorage

The lengths of anchorage of the reinforcement proposal can be changed by using the list. The *Details* option opens the following edit dialog box.

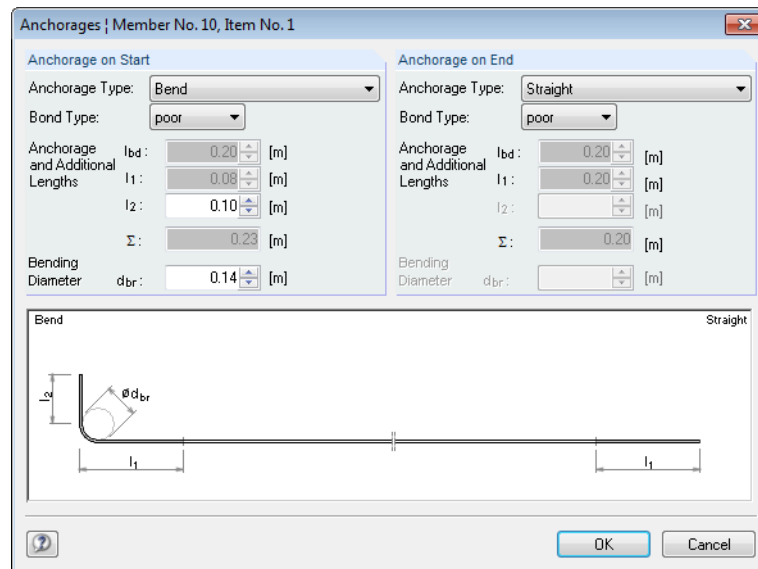
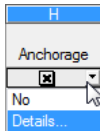


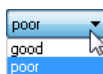
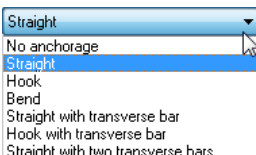
Figure 5.11: Dialog box *Anchorage*

This dialog box contains the parameters of the *Anchorage on Start* and *on End* of the rebar.

Use the lists to adjust the *Anchorage Type* and the *Bond Type*. Please find a description of the anchorage type in chapter 3.5.1 on page 67. CONCRETE recognizes automatically the bond conditions resulting from the cross-section geometry and the position of rebars. However, it is also possible to enter user-defined specifications. "Good" and "poor" bond conditions are shown in figure 8.2 of EN 1992-1-1, 8.4.2.

The lengths l_1 of the *Anchorage and Additional Lengths* is determined by equation (8.4) according to EN 1992-1-1, 8.4.4 (1), taking into account table 8.2, but it cannot be modified.

The length of anchorage l_2 for hooks and bends, in accordance with EN 1992-1-1, 8.4.1 (2), should be at least $5 d_s$.



The required *Bending Diameter* d_{br} is specified according to EN 1992-1-1, table 8.1, and can be adjusted, if necessary.

The total anchorage length Σ at each end of the member is comprised of the respective ratios.

Weight

Column I of table 3.1 indicates for each item the mass of all rebars.

Message

If a number is displayed in the final column, a special condition is the reason. The numbers are explained in the status bar.



To display all messages of the currently selected item, use the [Messages] button shown on the left. A dialog box with relevant information appears (cf. Figure 5.3, page 91).

Change reinforcement proposal

The graphic in the lower table section of table 3.1 represents the reinforcement including item bars. The currently selected reinforcement position (the row in the upper table section in which the pointer is placed) is highlighted in red. To open the edit dialog box for the selected item, click the [Edit] button in the bottom right corner of the graphic.

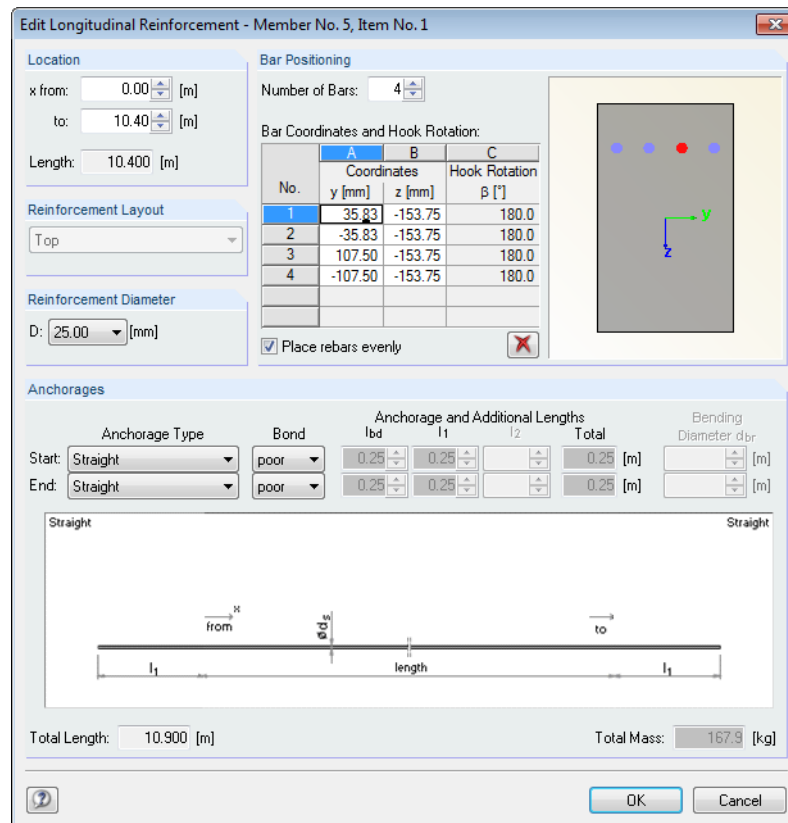


Figure 5.12: Dialog box *Edit Longitudinal Reinforcement*

This dialog box summarizes the reinforcement parameters already described above. Use the dialog box to control or, if necessary, modify the specifications concerning *Location*, *Bar Positioning*, *Reinforcement Diameter* and *Anchorage*.



When you have modified data, the program automatically recalculates the design, using the values of the new provided reinforcement. The results of a non-linear analysis represent an exception: The program deletes these results and a new manual [Calculation] is required.

5.2.2 Shear Reinforcement Provided

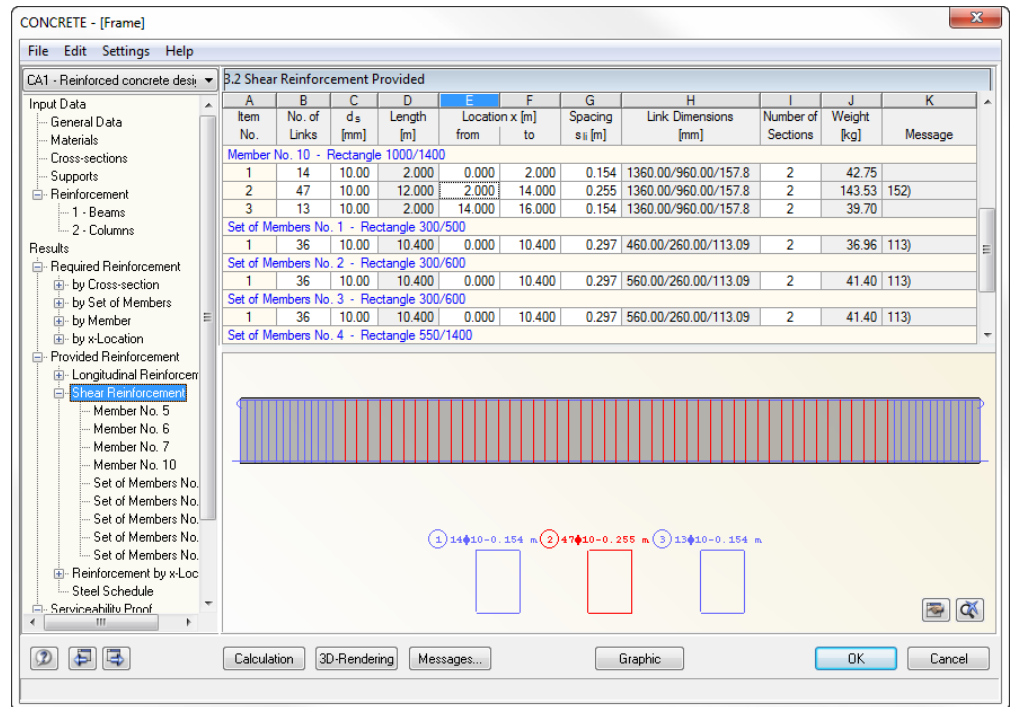


Figure 5.13: Table 3.2 Shear Reinforcement Provided

Similar to the output for the longitudinal reinforcement, the results for the link reinforcement are sorted by members and sets of members according to *Item* numbers (reinforcement groups).

The graphic in the lower table section represents the reinforcement including item links. The currently selected item (the row in the upper table section in which the pointer is placed) is highlighted in red. Modifications to the parameters entered in the upper section are updated and displayed immediately in the graphic.

The reinforcement proposal also takes into account structural regulations. For example, in accordance with EN 1992-1-1, 9.2.2 (6), it is recommended to define the maximum spacing for vertical links with $s_{l,max} = 0.75d$. Table NA 9.1 of the national annex for Germany only allows for reduced longitudinal spacings depending on the shear force ratio and the concrete strength.

Item No.

The results are listed by *Items* having each the same properties (diameter, spacing).

The items of all members and sets of members are summarized in table 3.4 *Steel Schedule*.

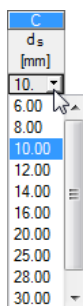
No. of Links

When CONCRETE determines the link reinforcement, it takes into account the user specifications entered in table 1.6 *Reinforcement*, tab *Links* (see chapter 3.5.2, page 68).

The number of links for an item can be edited: Click in the corresponding cell and enter another value. The link spacing (column G) will be adjusted automatically.

d_s

The reinforcement proposal is based on the specifications defined in table 1.6 *Reinforcement*, tab *Links*. Use the list to change the rebar diameter for the current item number.



Length



Column D displays for each item the total length of the link zone. It is determined by the start and end locations x , but cannot be edited in this table. To change data, use the [Edit] button in the bottom right corner of the graphic (see Figure 5.14, page 101).

Location x from ... to

These values represent the start and end positions of the link zones. They refer to the member's start node in RSTAB ($x = 0$). The entries in both columns can be edited so that you can shift the zone limits by modifying the values.



To subdivide a zone, enter a location x at the start or end position which lies between both values. CONCRETE will automatically create a new link zone.

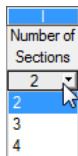
Spacing s_{ii}

The proposed link spacing takes into account the specifications defined in table 1.6 *Reinforcement*, tab *Link* (see chapter 3.5.2, page 68). This value can be edited: Click in the corresponding cell and enter another spacing. The number of links (column B) will be adjusted automatically. The exact link spacing, however, is calculated on the basis of an amount of links defined by an integer.

Link Dimensions

In this column, the link dimensions are specified following the sequence "height/width/anchorage length". CONCRETE takes into account the defined rebar diameters and concrete covers. The values cannot be edited.

Number of Sections



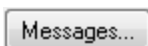
The links' sections are based on the specifications defined in table 1.6 *Reinforcement*, tab *Links* (see chapter 3.5.2, page 68). Use the list to change the number of sections.

Weight

Column J of table 3.2 indicates for each item the mass of all link rebars.

Message

If a number is displayed in the final column, a special condition is the reason. The numbers are explained in the status bar.



To display all messages of the currently selected item, use the [Messages] button shown on the left. A dialog box with relevant information appears (cf. Figure 5.3, page 91).

Change reinforcement proposal

The graphic in the lower table section of table 3.2 represents the reinforcement including item links. The currently selected reinforcement position (the row in the upper table section in which the pointer is placed) is highlighted in red. To open the edit dialog box for the selected item, click the [Edit] button in the bottom right corner of the graphic.

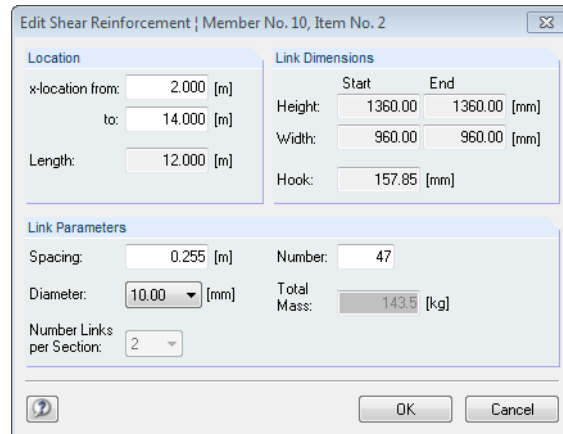


Figure 5.14: Dialog box *Edit Shear Reinforcement*

This dialog box summarizes the reinforcement parameters already described above. Use the dialog box to control or, if necessary, modify the specifications concerning *Location*, *Link Dimensions* and *Link Parameters*.

Calculation

When you have modified data, the program automatically recalculates the design, using the values of the new provided link reinforcement. The results of a non-linear analysis represent an exception: The program deletes these results and a new manual [Calculation] is required.

5.2.3 Reinforcement Provided by x-Location

This table contains information about the ultimate limit state designs that have been fulfilled or failed. The dynamic of the safety designs represents a great advantage: When modifications are made to the designed reinforcements, the program updates the designs automatically.

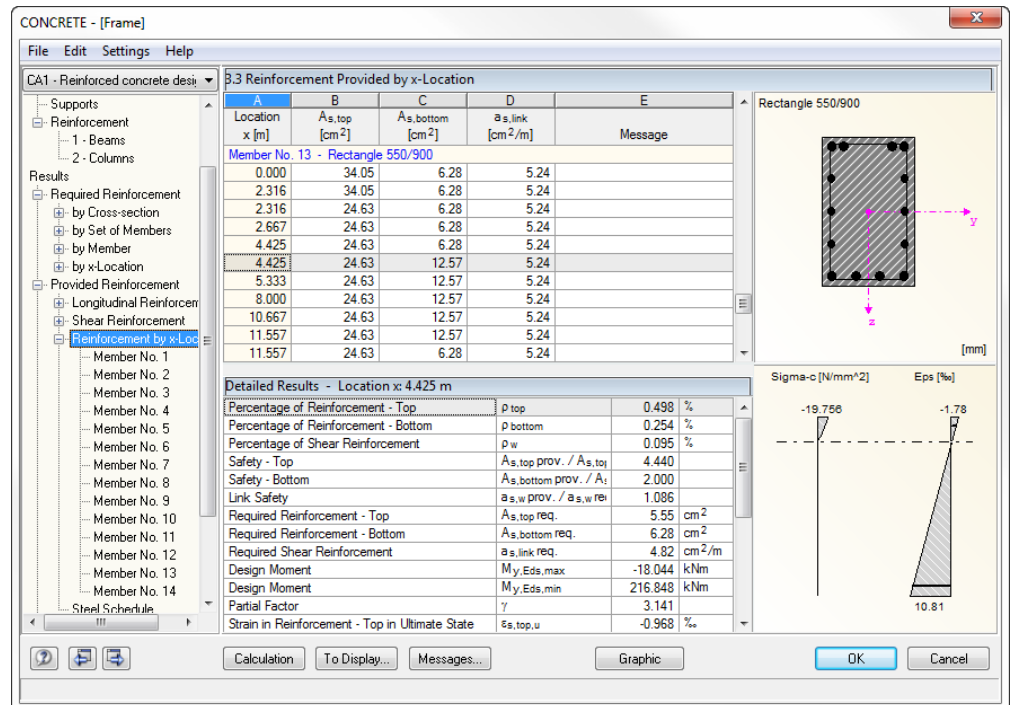


Figure 5.15: Table 3.3 Reinforcement Provided by x-Location

The upper dialog section lists the longitudinal and link reinforcement areas for each member location x.

Location x

The provided reinforcement areas for each member are listed by x-location:

- Start and end node
- Partition points according to possibly defined member division
- Extreme values of internal forces

In case of reinforcements by curtailment, the output shows the x-locations twice for the zone limits.

$A_{s,top}$

This value represents the reinforcement area of the provided top longitudinal reinforcement.

$A_{s,bottom}$

This value represents the reinforcement area of the provided bottom longitudinal reinforcement.

$a_{s,link}$

This column shows the area of the provided link reinforcement.



To Display...

The *Detailed Results* displayed in the lower dialog section allow for a detailed evaluation of the performed designs. The output includes the design details of the current location x (that means the active location selected in the upper dialog section) including all design-relevant parameters.

To reduce the displayed result parameters, use the button [To Display].

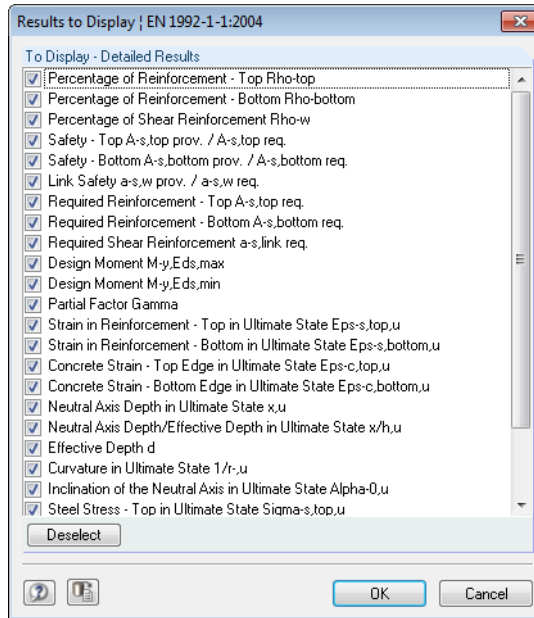
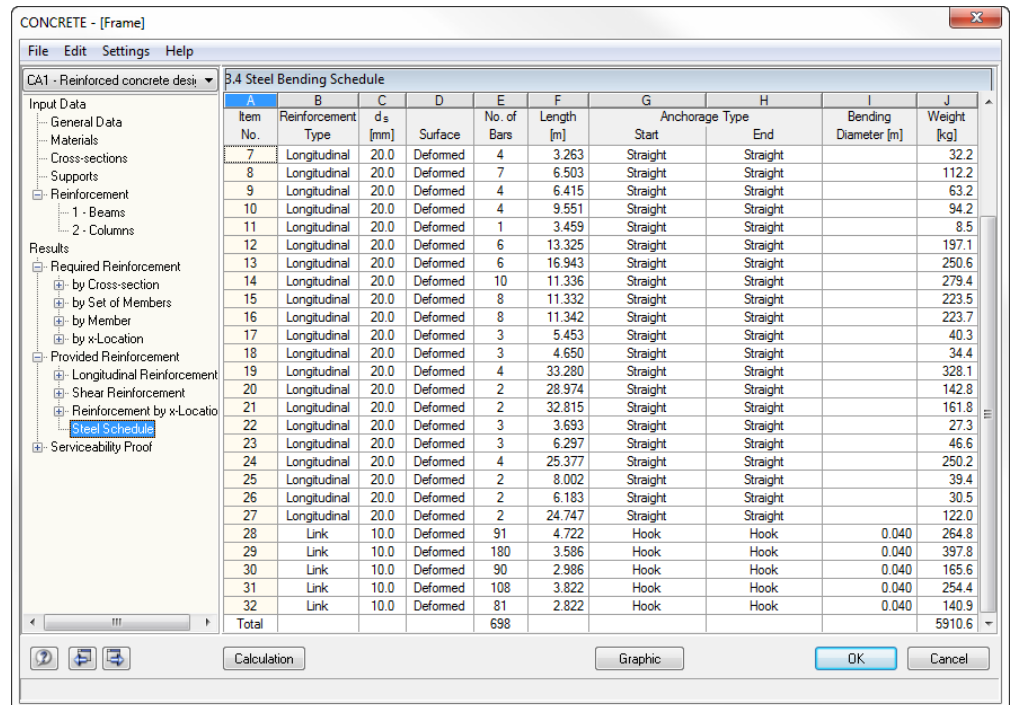


Figure 5.16: Dialog box *Results to Display*

The intermediate results contain information about the *Percentage of Reinforcement* and the *Safety* of the selected reinforcement, i.e. the ratio of the provided to the required reinforcement. The safety for the longitudinal reinforcement is designed by an increased moment taking into account the global offset.

5.2.4 Steel Bending Schedule

The schedule displays the provided rebars combined in a single overview. The table cannot be edited.



Item No.	Reinforcement Type	d _s [mm]	Surface	No. of Bars	Length [m]	Anchorage Type Start	Anchorage Type End	Bending Diameter [m]	Weight [kg]
7	Longitudinal	20.0	Deformed	4	3.263	Straight	Straight		32.2
8	Longitudinal	20.0	Deformed	7	6.503	Straight	Straight		112.2
9	Longitudinal	20.0	Deformed	4	6.415	Straight	Straight		63.2
10	Longitudinal	20.0	Deformed	4	9.551	Straight	Straight		94.2
11	Longitudinal	20.0	Deformed	1	3.459	Straight	Straight		8.5
12	Longitudinal	20.0	Deformed	6	13.325	Straight	Straight		197.1
13	Longitudinal	20.0	Deformed	6	16.943	Straight	Straight		250.6
14	Longitudinal	20.0	Deformed	10	11.336	Straight	Straight		279.4
15	Longitudinal	20.0	Deformed	8	11.332	Straight	Straight		223.5
16	Longitudinal	20.0	Deformed	8	11.342	Straight	Straight		223.7
17	Longitudinal	20.0	Deformed	3	5.453	Straight	Straight		40.3
18	Longitudinal	20.0	Deformed	3	4.650	Straight	Straight		34.4
19	Longitudinal	20.0	Deformed	4	33.280	Straight	Straight		328.1
20	Longitudinal	20.0	Deformed	2	28.974	Straight	Straight		142.8
21	Longitudinal	20.0	Deformed	2	32.815	Straight	Straight		161.8
22	Longitudinal	20.0	Deformed	3	3.693	Straight	Straight		27.3
23	Longitudinal	20.0	Deformed	3	6.297	Straight	Straight		46.6
24	Longitudinal	20.0	Deformed	4	25.377	Straight	Straight		250.2
25	Longitudinal	20.0	Deformed	2	8.002	Straight	Straight		39.4
26	Longitudinal	20.0	Deformed	2	6.183	Straight	Straight		30.5
27	Longitudinal	20.0	Deformed	2	24.747	Straight	Straight		122.0
28	Link	10.0	Deformed	91	4.722	Hook	Hook	0.040	264.8
29	Link	10.0	Deformed	180	3.586	Hook	Hook	0.040	397.8
30	Link	10.0	Deformed	90	2.986	Hook	Hook	0.040	165.6
31	Link	10.0	Deformed	108	3.822	Hook	Hook	0.040	254.4
32	Link	10.0	Deformed	81	2.822	Hook	Hook	0.040	140.9
Total				698					5910.6

Figure 5.17: Table 3.4 Steel Bending Schedule

Item No.

The rebars are listed by *Items* having each the same properties (diameter, length, type of anchorage etc.).

Normally, the item numbers are not identical with the numbers of the tables 3.1 and 3.2.

Reinforcement Type

This column shows you if the reinforcement is a *Longitudinal* or a *Link* reinforcement.

d_s

Column C indicates the used rebar diameters.

Surface

This column displays the surface type of the reinforcing steel which can be *Deformed* or *Plain*.

No. of Bars

The number of similar rebars of each item is displayed in column E.

Length

This column shows for each item the total length of a representative rebar.

Anchorage Type Start / End

The two columns provide information about the types of anchorage at the start and the end of the rebars (*No anchorage, Straight, Hook, Bend* etc.).

Bending Diameter

In case links and hooks are used, you find the bending diameter d_{br} specified in column I.

Weight

The final column indicates for each item the mass of all rebars.

Total

The final table row of the steel schedule shows the total number of rebars and the mass of steel that is totally required. It is determined from the values of the individual items above.

5.3 Serviceability Limit State Design

The results tables 4.1 to 4.4 appear only in case the design of the *Serviceability Limit State* was activated in table 1.1 (see chapter 3.1.2, page 55), and if no design problems were detected by the program (see chapter 5.1.5, page 94 and chapter 5.2.3, page 102).

The design of the serviceability limit state is performed by means of the reinforcement layout that is available as *Reinforcement Provided* in the tables 3.1 and 3.2.

5.3.1 Serviceability Proof by Cross-section

A	B	C	D	E	F	G	H	I	J	K	L	M
Cross. No.	Member No.	Location x [m]	LC / LG CO	σ_s [N/mm ²]	σ_c [N/mm ²]	min A_s [cm ²]	lim d_s [mm]	Bar Spacing lim s_l [mm]	Crack Spacing $s_{r,max}$ [mm]	Crack Width max w_k [mm]	Deflections $u_{1,z}$ [mm]	Message
1	2	4.400	LG1	-3.3	-6.0	0.00	36.0			0.00	4.28	204)
2	8	0.000	LG1	-0.6	-5.9	0.00	36.0			0.00	0.00	204)
3	10	8.000	LG1	279.2	-11.1	75.41	12.1	150.97	80.99	0.09	51.43	201) 202) 331)
4	11	0.000	LG1	235.6	-10.0	39.38	17.0	205.54	58.07	0.06	1.24	201)
5	13	16.000	LG1	263.8	-12.6	30.84	13.6	170.22	71.76	0.08	0.00	201) 202)
Governing:												
3	10	8.000	LG1	279.2	-11.1	75.41	12.1	150.97	80.99	0.09	51.43	201) 202) 331)

Figure 5.18: Table 4.1 *Serviceability Proof by Cross-section*

The table contains the extreme values of different criteria that must be proved for the serviceability limit state. The values result from the parameters of the reinforcement groups for the crack width control (see chapter 3.5.4, page 72), the provided reinforcement and the internal forces of the governing actions.



When you evaluate the results of this output table, please consider the descriptions for the dialog box *Settings for Layout of Longitudinal Reinforcement* on page 74.

Cross. No.

The designs are sorted by cross-section numbers. The final row of the table shows the *Governing* cross-section for the serviceability limit state design.

Member No.

This column displays the number of the member that provides the extreme values for each cross-section type.

Location x

The column shows the respective x-location where the most unfavorable values occur. The distances refer to the start node of the governing member.

LC / LG / CO

This column displays the numbers of the load cases, load groups, load combinations or super combinations that are decisive for the individual designs.

σ_s

The values represent the stresses within the reinforcement in case the tensile zone is cracked. They are determined from the product of the steel strain and the modulus of elasticity:

$$\sigma_s = \varepsilon_s \cdot E_s$$

σ_c

This table column displays the concrete stresses in the serviceability limit state.

min A_s

The minimum area of the rebar reinforcement according to EN 1992-1-1, 7.3.2, eq. (7.1) is:

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

- where k_c Factor to take into account the stress distribution within the cross-section before first crack formation
- k Factor to take into account self-equilibrating stresses distributed non-linearly across the cross-section
- $f_{ct,eff}$ Mean value of effective tensile strength of concrete when cracks occur
- A_{ct} Tension zone of concrete in cracked state in case of first crack formation
- $f_{ct,eff}$ Effective tensile strength of concrete at decisive point of time
- σ_s Allowable steel stress directly after crack formation (depending on limit diameter or maximum value of rebar spacings, where applicable)

lim d_s

The allowable diameters of the reinforcing steels, in accordance with the simplified design method described in EN 1992-1-1, 7.3.3 are limited to the dimensions mentioned in table 7.2.

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 5.19: Limit diameter according to EN 1992-1-1, table 7.2

Bar Spacing s_1

The allowable rebar spacings, in accordance with the simplified design method described in EN 1992-1-1, 7.3.3, are limited to the dimensions mentioned in table 7.3.

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 5.20: Maximum values for rebar spacings according to EN 1992-1-1, table 7.3

Crack Spacing $s_{r,max}$

This column displays the maximum crack spacing according to EN 1992-1-1, 7.3.4, eq. (7.11) that is available in the final crack state:

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

- where k_3 Recommended factor: 3.4 (national annex for Germany: 0)
- c Concrete cover of longitudinal reinforcement
- k_1 Factor to take into account the reinforcement's bond properties: 0.8 for ribbed steel bars or 1.6 for rebars with plain surface
- k_2 Factor to consider distribution of strain:
0.5 for bending or 1.0 for tension only
- k_4 Recommended factor: 0.425 (national annex for Germany: 1 / 3.6)
- \emptyset Rebar diameter
- ρ_{eff} Effective reinforcement ratio according to eq. (7.10): $A_s / A_{c,eff}$

Crack Width max w_k

The characteristic crack width is determined according to EN 1992-1-1, 7.3.4, eq. (7.8):

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

- where $s_{r,max}$ Maximum crack spacing in final crack state
- ε_{sm} Mean strain of reinforcement considering contribution of concrete concerning tension between cracks
- ε_{cm} Mean strain of concrete between cracks

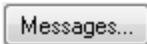
Deflections $u_{l,z}$

The penultimate column displays respectively the absolute value of the deformation available in direction of the local member axis z .

The settings for the allowable relative deflection are managed in the dialog box *Settings for Layout of Longitudinal Reinforcement* (see Figure 3.25, page 74).

Message

The final column indicates design problems or shows notes referring to difficulties occurred during the analysis. The numbers are explained in the status bar.



To display all messages of the current serviceability limit state design, use the [Messages] button shown on the left. A dialog box with relevant information appears.

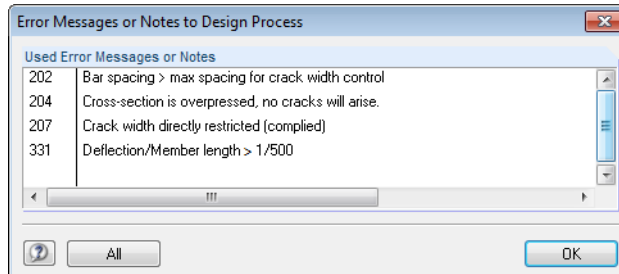


Figure 5.21: Dialog box *Error Messages or Notes to Design Process*

5.3.2 Serviceability Proof by Set of Member

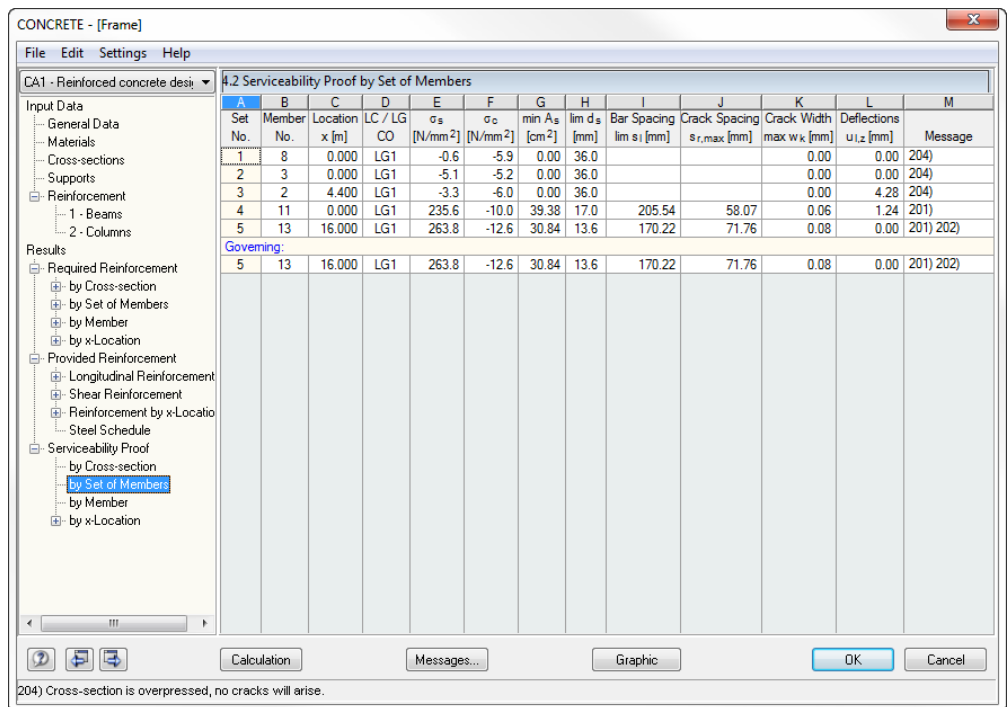


Figure 5.22: Table 4.2 *Serviceability Proof by Set of Members*

When sets of members have been selected for design, the governing serviceability limit state designs are sorted by sets of members in this results table.

Details on the table columns can be found in the previous chapter 5.3.1.

5.3.3 Serviceability Proof by Member

Member No.	Location x [m]	LC / LG CO	σ_s [N/mm ²]	σ_c [N/mm ²]	min A _s [cm ²]	lim d _s [mm]	Bar Spacing lim s ₁ [mm]	Crack Spacing s _{r,max} [mm]	Crack Width max w _k [mm]	Deflections u _{1,z} [mm]	Message
1	6.000	LG1	-9.5	-6.4	0.00	36.0			0.00	0.97	204)
2	4.400	LG1	-3.3	-6.0	0.00	36.0			0.00	4.28	204)
3	0.000	LG1	-5.1	-5.2	0.00	36.0			0.00	0.00	204)
4	3.740	LG1	-6.3	-3.3	0.00	36.0			0.00	4.19	204)
5	0.000	LG1	-1.9	-8.2	0.00	36.0			0.00	0.00	204)
6	0.000	LG1	-7.8	-6.2	0.00	36.0			0.00	0.00	204)
7	0.000	LG1	-8.3	-6.7	0.00	36.0			0.00	0.00	204)
8	0.000	LG1	-0.6	-5.9	0.00	36.0			0.00	0.00	204)
9	4.400	LG1	-0.9	-4.8	0.00	36.0			0.00	4.13	204)
10	8.000	LG1	279.2	-11.1	75.41	12.1	150.97	80.99	0.09	51.43	201) 202) 331)
11	0.000	LG1	235.6	-10.0	39.38	17.0	205.54	58.07	0.06	1.24	201)
12	16.000	LG1	181.7	-7.7	38.06	28.2	272.89	58.07	0.04	10.60	
13	16.000	LG1	263.8	-12.6	30.84	13.6	170.22	71.76	0.08	0.00	201) 202)
14	0.000	LG1	112.3	-5.1	36.03	32.0	300.00	71.76	0.02	6.67	
Governing:											
10	8.000	LG1	279.2	-11.1	75.41	12.1	150.97	80.99	0.09	51.43	201) 202) 331)

Figure 5.23: Table 4.3 Serviceability Proof by Member

This results table shows the output for the crack width designs sorted by members. The individual table columns correspond to the columns of table 4.1. They are described in chapter 5.3.1.

5.3.4 Serviceability Proof by x-Location

Location x [m]	LC / LG CO	σ_s [N/mm ²]	σ_c [N/mm ²]	min A _s [cm ²]	lim d _s [mm]	Bar Spacing lim s ₁ [mm]	Crack Spacing s _{r,max} [mm]	Crack Width max w _k [mm]	Deflections u _{1,z} [mm]	Message	
Member No. 8 - Rectangle 900/900											
0.000	LG1	-0.6	-5.9	0.00	36.0			0.00	0.00	204)	
3.000	LG1	-11.3	-3.8	0.00	36.0			0.00	1.01	204)	
6.000	LG1	-12.3	-3.7	0.00	36.0			0.00	2.88	204)	
Member No. 9 - Rectangle 900/900											
0.000	LG1	-9.3	-3.1	0.00	36.0			0.00	2.88	204)	
4.400	LG1	-0.9	-4.8	0.00	36.0			0.00	4.13	204)	
Member No. 10 - Rectangle 1000/1400											
0.000	LG1	24.5	-1.3	85.82	32.0	300.00	98.18	0.01	0.00		
2.667	LG1	102.6	-4.4	81.63	32.0	300.00	85.39	0.03	26.26		
5.333	LG1	215.9	-9.5	82.87	21.4	230.12	85.39	0.07	45.69	331)	
8.000	LG1	279.2	-11.1	75.41	12.1	150.97	80.99	0.09	51.43	201) 202) 331)	
8.000	LG1	279.2	-11.1	75.41	12.1	150.97	80.99	0.09	51.43	201) 202) 331)	
10.667	LG1	143.4	-5.9	82.67	32.0	300.00	85.39	0.04	40.55	331)	
13.333	LG1	19.7	-1.2	91.63	32.0	300.00	81.27	0.00	20.36		
16.000	LG1	251.7	-9.4	83.35	14.8	185.37	81.27	0.08	2.16	201)	
Member No. 11 - Rectangle 550/1400											
0.000	LG1	235.6	-10.0	39.38	17.0	205.54	58.07	0.06	1.24	201)	
2.667	LG1	94.9	-4.4	36.30	32.0	300.00	58.07	0.02	-3.21		
5.333	LG1	5.4	-0.5	32.25	32.0	300.00	85.28	0.00	-2.20		
8.000	LG1	35.1	-1.6	40.21	32.0	300.00	83.09	0.01	-0.43		
8.800	LG1	40.1	-1.8	40.21	32.0	300.00	83.09	0.01	-0.32		
10.667	LG1	26.9	-1.3	39.74	32.0	300.00	83.09	0.01	-0.08		
13.333	LG1	29.7	-1.4	32.25	32.0	300.00	74.79	0.01	-0.86		
16.000	LG1	180.0	-6.5	39.83	28.5	274.99	66.61	0.05	0.00		
Member No. 12 - Rectangle 550/1400											
0.000	LG1	129.7	-4.7	42.33	32.0	300.00	66.61	0.03	9.87		

Figure 5.24: Table 4.4 Serviceability Proof by x-Location

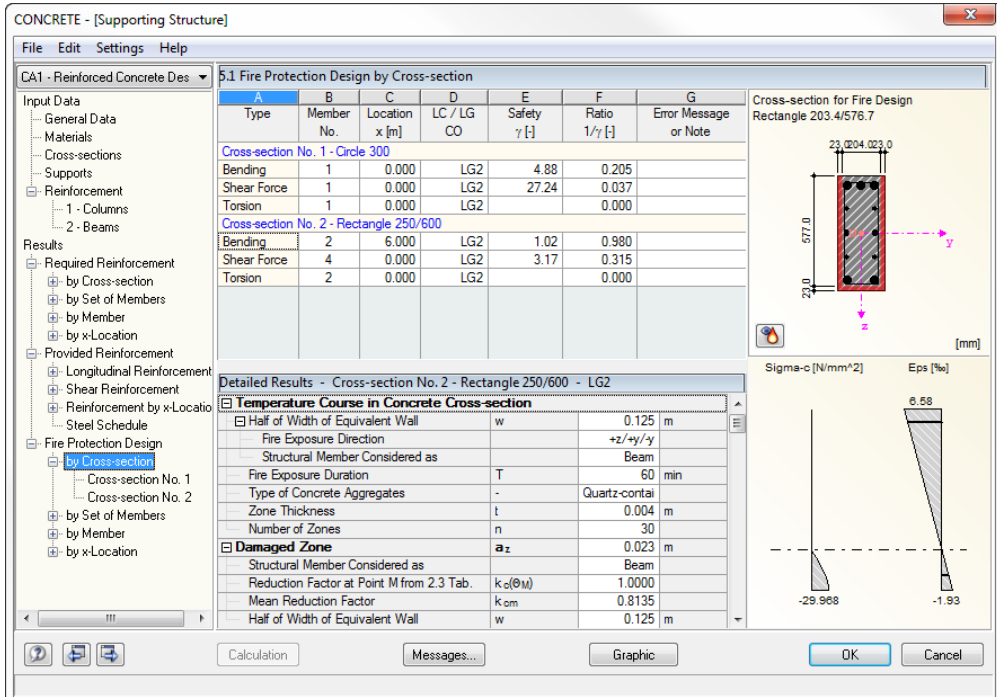
This table lists in detail the various designs (see chapter 5.3.1) according to x-locations.

5.4 Fire Protection Design

The results tables 5.1 to 5.4 appear only in case the design of the *Fire Resistance* was activated in table 1.1 (see chapter 3.1.4, page 57), and if no design problems were detected by the program (see chapter 5.1.5, page 94 and chapter 5.2.3, page 102).

The fire protection design is performed by means of the reinforcement layout that is available as *Reinforcement Provided* in the tables 3.1 and 3.2.

5.4.1 Fire Protection Design by Cross-section



The screenshot shows the 'CONCRETE - [Supporting Structure]' window with the '5.1 Fire Protection Design by Cross-section' dialog open. The dialog is divided into several sections:

- Table 5.1:** A table with columns A (Type), B (Member No.), C (Location x [m]), D (LC / LG CO), E (Safety γ []), F (Ratio $1/\gamma$ []), and G (Error Message or Note). It lists results for two cross-sections: 'Circle 300' and 'Rectangle 250/600'.
- Detailed Results:** A sub-table for 'Cross-section No. 2 - Rectangle 250/600 - LG2' showing parameters like 'Half of Width of Equivalent Wall' (w = 0.125 m), 'Fire Exposure Duration' (T = 60 min), and 'Number of Zones' (n = 30).
- Graphical View:** A diagram of a rectangular cross-section (250x600 mm) with reinforcement bars. It shows stress (σ) and strain (ϵ) distributions across the height of the section.

Figure 5.25: Table 5.1 *Fire Protection Design by Cross-section*

This table shows for each designed cross-section the decisive safety values determined during the fire protection design. The values result from the parameters of the reinforcement groups for the fire designs (see chapter 3.1.4, page 57), the provided reinforcement and the internal forces of the relevant actions.

In the lower part of the table the *Detailed Results* for the entry selected in the table row above are shown. Thus, due to the design details, it is possible to evaluate the results specifically. The output of the intermediate results in the lower part will be updated automatically as soon as another row is selected in the upper part.

The theoretical background explaining the fire protection design can be found in chapter 2.3, page 17.

Type

The safety values are listed according to cross-sections. In addition to the design values for *Bending*, the table shows the safeties for *Shear Force* and *Torsion*, provided that these designs have been specified in the *Fire Resistance* tab of table 1.6 *Reinforcement* (see Figure 3.30, page 79).

Member No.

For each cross-section and each design type, the table shows the number of the member providing the minimum safety.

Location x

The column shows the respective x-location on the member which is decisive for the fire protection design. For the table output, the program uses the following RSTAB member locations x:

- Start and end node
- Partition points according to possibly defined member division
- Extreme values of internal forces

LC / LG / CO

This column displays the numbers of the load cases, load groups, load combinations or super combinations that are decisive for the respective designs.

Safety γ

Column E informs you about the minimum safety factors γ for each type of design. If the safety is less than 1, the fire protection design has failed and the result row is highlighted in red.

Ratio $1/\gamma$

The entries in this column represent the safeties' reciprocal values (column E). In this way, it is possible to evaluate quickly the resources which are available until the limit value 1 is reached in the cross-section.

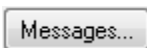
Among the detailed results, you can find all initial values of the designs. The design safeties result from the ratio of the ultimate internal force in case of fire and the analyzed internal force acting on the cross-section.

Design			
<input type="checkbox"/>	Factor of Utilization for Bending Rein	$1/\gamma_{SL,fi}$	0.0355
<input type="checkbox"/>	Existing Safety to Bending	$\gamma_{SL,fi}$	28.1597
<input type="checkbox"/> Influencing Internal Forces			
	Moment around y-Axis	$M_{y,Ed,fi}$	0.00 kN
	Moment around z-Axis	$M_{z,Ed,fi}$	0.00 kN
	Axial Force	$N_{ed,fi}$	-34.58 kN
<input type="checkbox"/> Ultimate Internal Forces			
	Ultimate Moment around y-Axis	$M_{yu,fi}$	0.00 kN
	Ultimate Moment around z-Axis	$M_{zu,fi}$	0.00 kN
	Ultimate Axial force	$N_{u,fi}$	-973.68 kN
<input type="checkbox"/>	Factor of Utilization for Shear Reinfor	$1/\gamma_{sw,fi}$	0.0000
<input type="checkbox"/>	Factor of Utilization for Torsion Reinf	$1/\gamma_{T,fi}$	0.0000

Figure 5.26: Detailed results, section *Design*

Error Message or Note

The final column indicates non-designable situations or notes referring to design problems. The numbers are explained in the status bar.



To display all messages of the currently selected design case, use the [Messages] button shown on the left. A dialog box with relevant information appears.

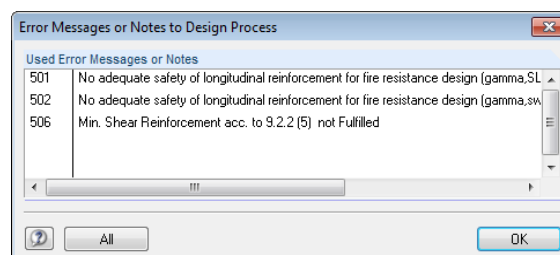


Figure 5.27: Dialog box *Error Messages or Notes to Design Process*

Temperature course

The cross-section used for the fire protection design is graphically displayed to the right of the table. The damaged zone is highlighted in red.

Use the button [Temperature Course] to show the distribution of temperature in the currently selected cross-section. The following dialog box appears:

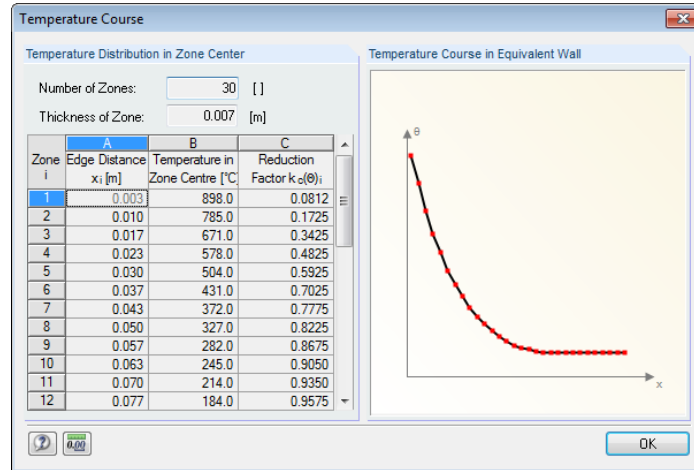
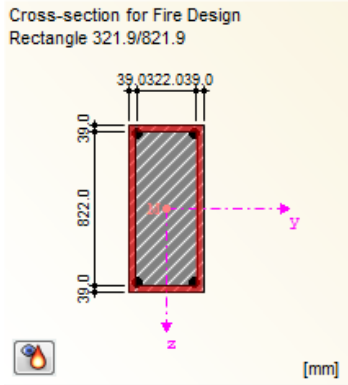


Figure 5.28: Dialog box *Temperature Course*

A table displays the *Temperature Distribution in Zone Center* (cf. chapter 2.3.2, page 18). The graphic shows the *Temperature Course in Equivalent Wall* according to EN 1992-1-2, annex A.

5.4.2 Fire Protection Design by Set of Members

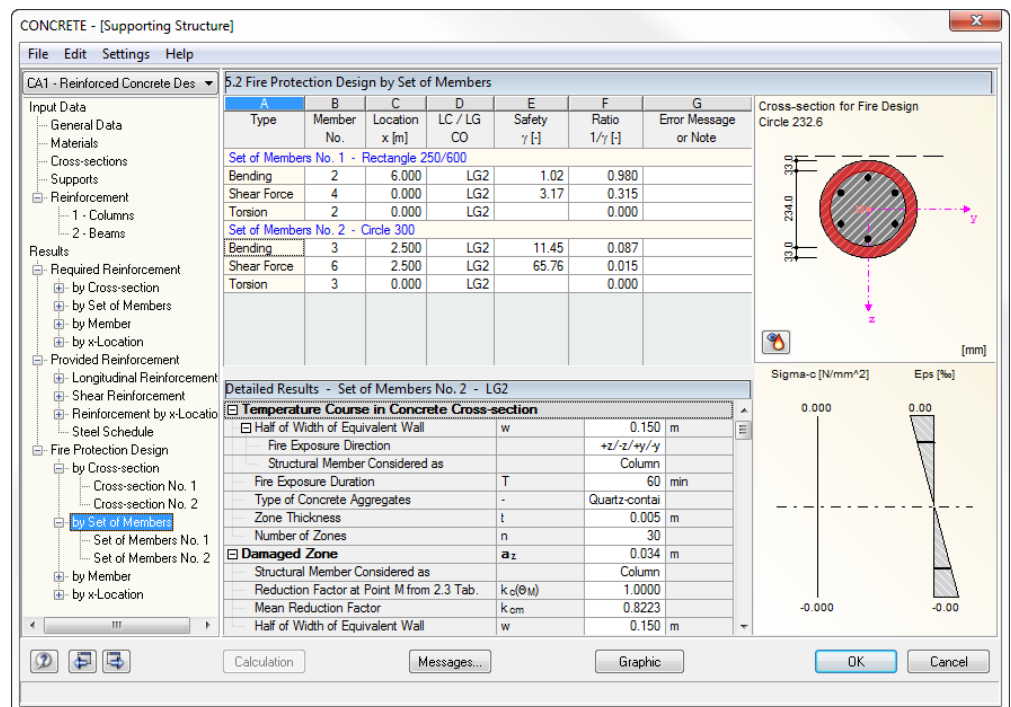


Figure 5.29: Table 5.2 *Fire Protection Design by Set of Members*

When sets of members have been selected for design, the fire protection designs are sorted by sets of members in this table. Details on the columns can be found in the previous chapter 5.4.1.

5.4.3 Fire Protection Design by Member

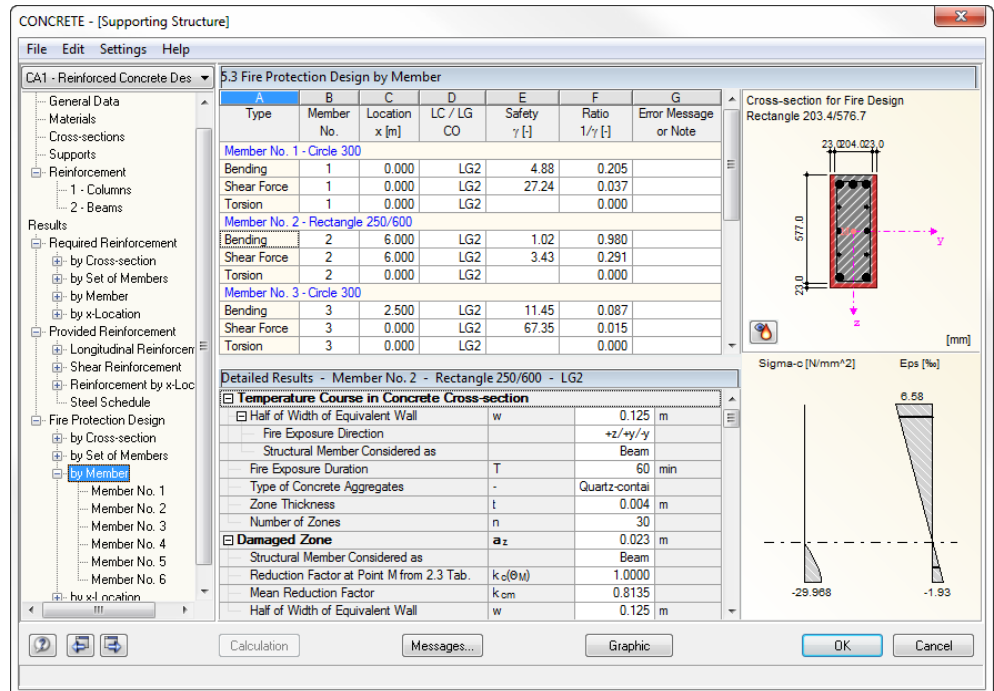


Figure 5.30: Table 5.3 Fire Protection Design by Member

This table shows the results output of the fire protection designs listed by members. The individual table columns correspond to the columns of table 5.1 as described in chapter 5.4.1.

5.4.4 Fire Protection Design by x-Location

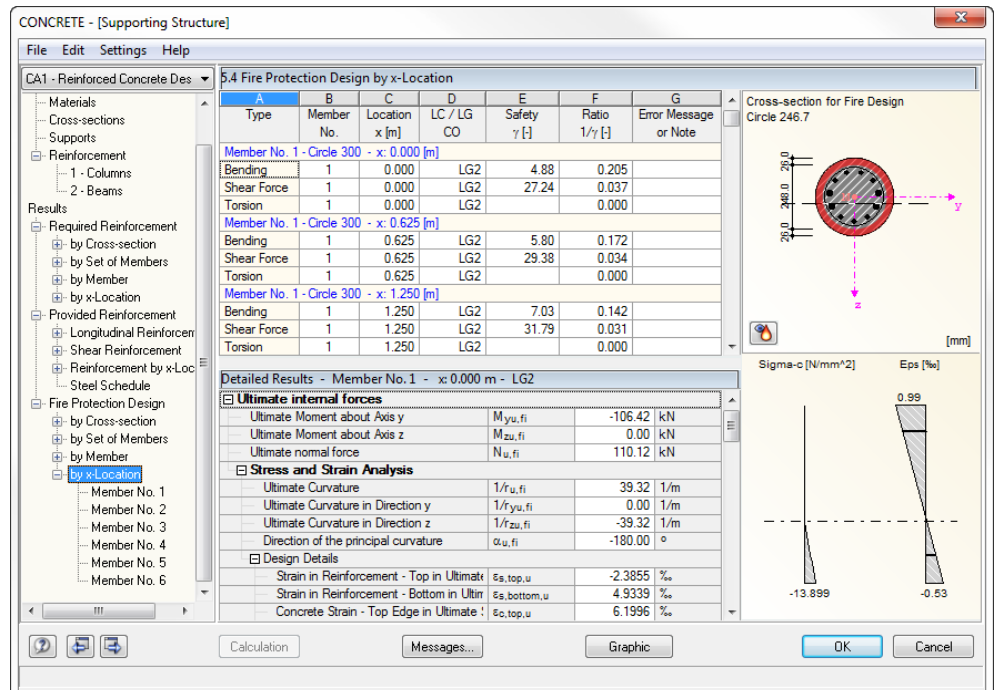


Figure 5.31: Table 5.4 Fire Protection Design by x-Location

This results table displays in detail the fire protection designs (cf. chapter 5.4.1) sorted by x-location.

5.4.5 Fire Protection Design Not Designable

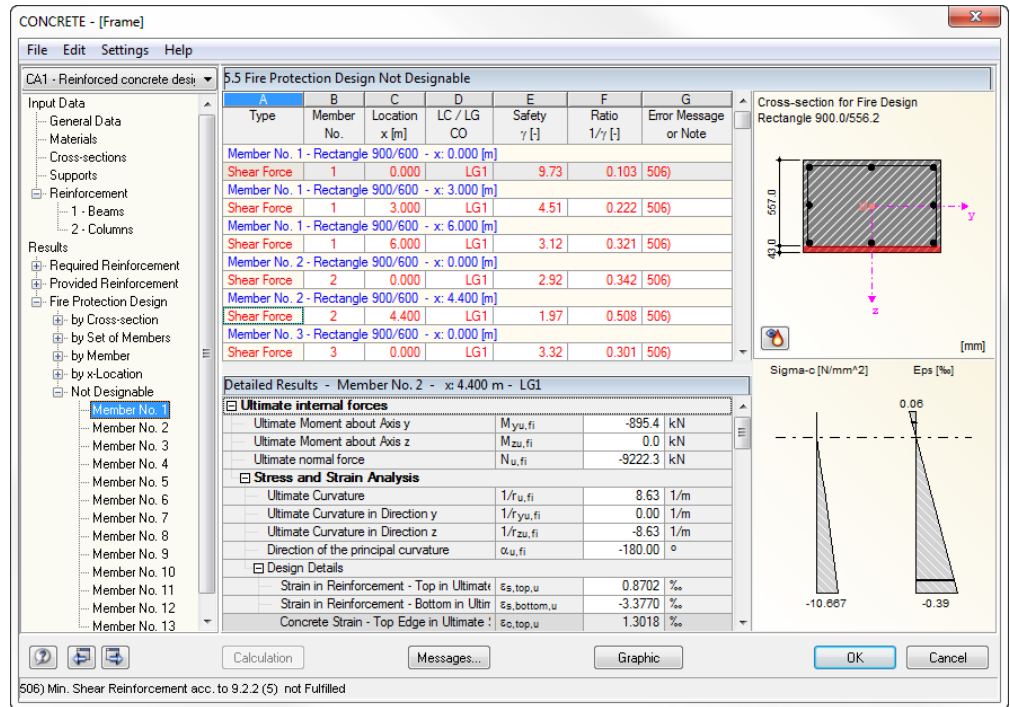
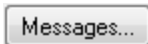


Figure 5.32: Table 5.5 Fire Protection Design Not Designable

This table is displayed only if CONCRETE has detected failed designs or any other problems occurring during the fire protection 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.



Click the [Messages] button to display all specific conditions that have occurred during the design of the current x-location.

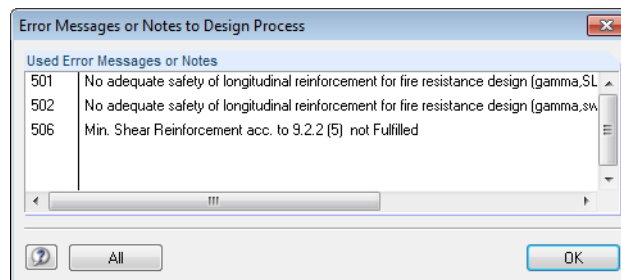
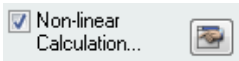


Figure 5.33: Dialog box Error Messages or Notes to Design Process



Use the [All] button in this dialog box to show all messages available for CONCRETE.

5.5 Nonlinear Calculation



The results tables 6.1 to 6.4 appear only in case the check box for the *Non-Linear Calculation* was activated in the single tabs available in table 1.1 (see chapter 3.1, page 52), and if no design problems were detected by the program (see chapter 5.1.5, page 94 and chapter 5.2.3, page 102).

The designs are performed with the reinforcement layout that is available as *Reinforcement Provided* in tables 3.1.

5.5.1 Nonlinear Calculation - Ultimate Limit State

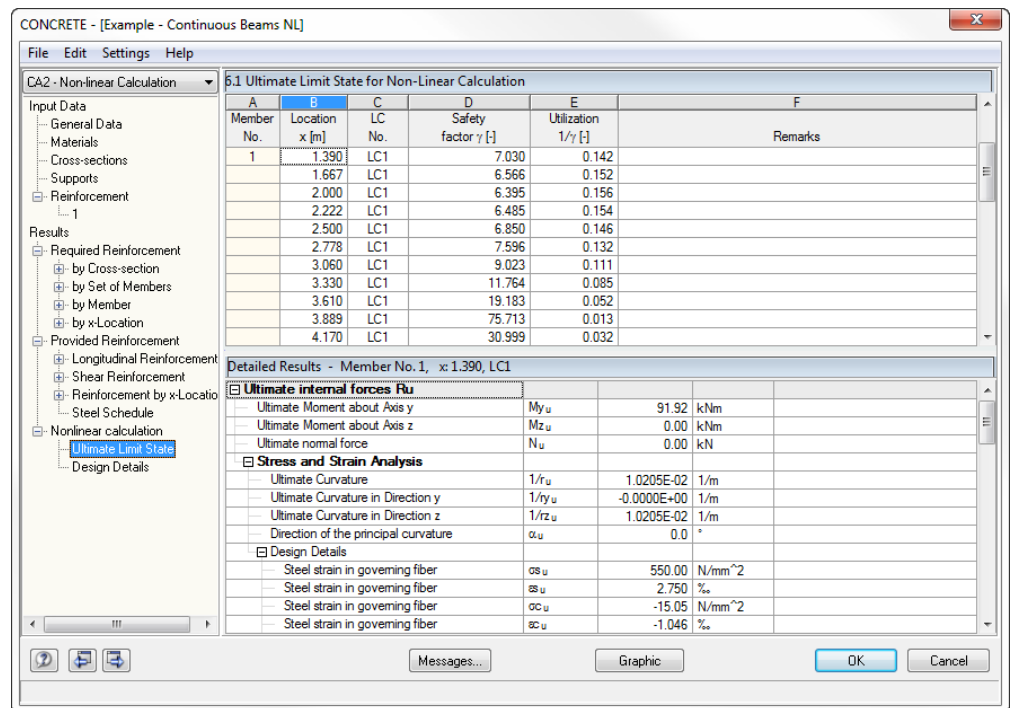


Figure 5.34: Table 6.1 *Ultimate Limit State for Non-Linear Calculation*

For the designed members this table shows the decisive safety values determined during the non-linear ultimate limit state design. The values result from the parameters set for the reinforcement groups, the internal forces of the actions and the specifications in the dialog box *Settings for Non-linear Calculation* (see Figure 4.2, page 81 and Figure 4.3, page 83).

In the lower part of the table the *Detailed Results* for the entry selected in the table row above are shown. Thus, due to the design details, it is possible to evaluate the results specifically. The output of the intermediate results in the lower part will be updated automatically as soon as another row is selected in the upper part.

The theoretical background explaining the non-linear design is described in detail in chapter 2.4, page 24.

Member No.

The displayed output for safety factors and utilizations is sorted by members.

Location x

The x-locations represent the FE nodes in the member used for the non-linear calculation. They are based on the settings specified for the adaptive element division in the dialog box *Settings for Non-linear Calculation*, tab *Iteration Parameters* (see Figure 2.29, page 51).

LC No.

The numbers of the load cases or load groups that are decisive on the respective locations x are displayed.

Safety factor γ

This column indicates the safety factor γ on each location x. The factor represents the relation of limit load and available loading.

$$\gamma = \frac{R_d}{E_d} \geq 1.0$$

where

$$R_d = \begin{pmatrix} N_{Rd} \\ M_{y,Rd} \\ M_{z,Rd} \end{pmatrix} \quad \text{Design value of design resistance}$$

$$E_d = \begin{pmatrix} N_{Ed} \\ M_{y,Ed} \\ M_{z,Ed} \end{pmatrix} \quad \text{Design value of action}$$

If the safety factor at any location is minor than 1, or if no convergence is reached, the ultimate limit state design has failed. The entire table is presented in red color.

Utilization $1/\gamma$

The entries in this column represent the reciprocal values of the safety factors γ . So it is easy to evaluate resources within the cross-section quickly.

The *Detailed Results* in the lower part of the table provide information about all design details. They are listed like in a tree structure.

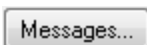
[-] Stress and Strain Analysis			
[-] Mean Main Curvature	1/r _m	1.1423E-04	1/m
[-] Middle curvature in y-direction	1/ty _m	-3.6025E-12	1/m
[-] Mean curvature in z-direction	1/rz _m	-1.1423E-04	1/m
[-] Direction of the principal curvature	α _m	-180.0	°
[-] Design Details			
[-] Steel strain in governing fiber	σs _m	4.23	N/mm ²
[-] Steel strain in governing fiber	εs _m	0.021	‰
[-] Steel strain in governing fiber	σc _m	-0.69	N/mm ²
[-] Steel strain in governing fiber	εc _m	-0.021	‰
[-] Assessed Concrete Residual Tensile Strength	σct _m	0.56	N/mm ²
[-] Maximum Concrete Tensile Strain	εct _m	0.024	‰
[-] Cross-section properties			
[-] Mean moment of inertia over the y-axis	Iy _m	1.40631E+09	mm ⁴

Figure 5.35: Detailed results, section *Stress and Strain Analysis*

As different approaches for Tension-Stiffening may be used for the calculation (see chapter 2.4.3, page 28), also the details of the individual models differ a bit from each other.

Remarks

The final table column informs you about failed designs or contains notes about specific situations occurring in the course of the design (for example when the maximum number of iterations was reached in the final load step without fulfilling the break-off criterion). The numbers indicated are explained in the status bar.



To display the messages of the currently selected design case, use the [Messages] button shown on the left. A dialog box with relevant information appears (cf. Figure 5.27, page 111).

5.5.2 Nonlinear Calculation - Serviceability Limit State

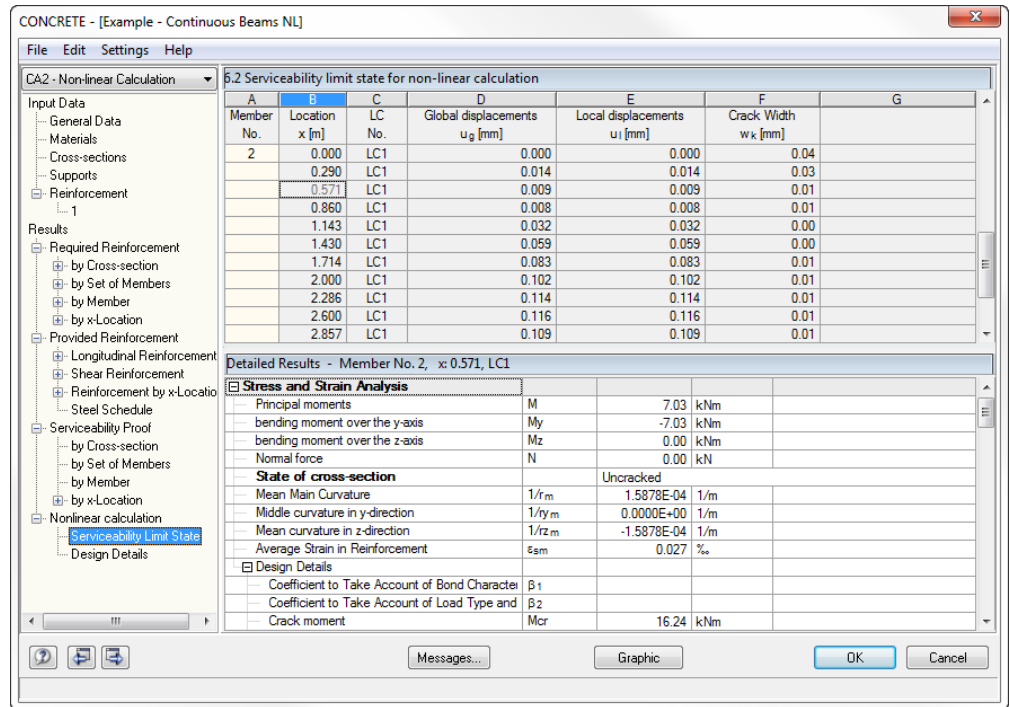


Figure 5.36: Table 6.2 Serviceability limit state for non-linear calculation

For the designed members this table shows the decisive deformations and crack widths determined during the non-linear serviceability limit state design. The values result from the parameters set for the reinforcement groups, the internal forces of the actions and the specifications in the dialog box *Settings for Non-linear Calculation* (see Figure 4.2, page 81 and Figure 4.3, page 83).

In the lower part of the table the *Detailed Results* for the entry selected in the table row above are shown. Thus, due to the design details, it is possible to evaluate the results specifically.

The theoretical background explaining the non-linear design is described in detail in chapter 2.4, page 24.

Member No.

The displayed output for deformations and crack widths is sorted by members.

Location x

The x-locations represent the FE nodes in the member used for the non-linear calculation. They are based on the settings specified for the adaptive element division in the dialog box *Settings for Non-linear Calculation*, tab *Iteration Parameters* (see Figure 2.29, page 51).

LC No.

The numbers of the load cases or load groups that are decisive on the respective locations x are shown.

Global displacements u_g

The column displays the resulting displacements that refer to the global coordinate system XYZ.

Among the intermediate results under *Global Node Deformations* you can check the deformation portions of the FE nodes in direction of each global axis.

Local displacements u_i

The entries in this column represent the resulting displacements in direction of the local member axis system xyz.

Among the intermediate results under *Local Node Deformations* you can check the deformation portions of the FE nodes in direction of the single member axes.

Crack Width w_k

This column shows the characteristic crack widths according to EN 1992-1-1, 7.3.4, eq. (7.8).

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

- where $s_{r,max}$ Maximum crack spacing in final crack state
- ϵ_{sm} Mean strain of reinforcement considering contribution of concrete concerning tension between cracks
- ϵ_{cm} Mean strain of concrete between cracks

To determine crack spacings and strains, CONCRETE uses the internal forces of the non-linear calculation.

The *Detailed Results* in the lower part of the table provide information about all design details. They are listed like in a tree structure.

Zustand II			
Cross-section properties			
—	Moment of inertia about the y-axis	$I_{y II}$	3.34737E+08 mm ⁴
—	Moment of inertia about the z-axis	$I_{z II}$	5.20833E+08 mm ⁴
—	Surface	A_{II}	25105.30 mm ²
Stress and Strain Analysis			
—	main curvature	$1/r_{II}$	7.0048E-04 1/m
—	curvature in y-direction	$1/r_{y II}$	7.9712E-11 1/m
—	curvature in z-direction	$1/r_{z II}$	-7.0048E-04 1/m
—	Direction of the principal curvature	α_{II}	-180.0 °
—	Steel strain in governing fiber	$\sigma_{s II}$	40.28 N/mm ²
—	Steel strain	$\epsilon_{s II}$	0.201 ‰
—	Steel strain in governing fiber	$\sigma_{cc II}$	-1.91 N/mm ²
—	Steel strain in governing fiber	$\epsilon_{cc II}$	-0.059 ‰

Figure 5.37: Detailed results, section *Zustand II* (state II)

5.5.3 Nonlinear Calculation - Fire Resistance

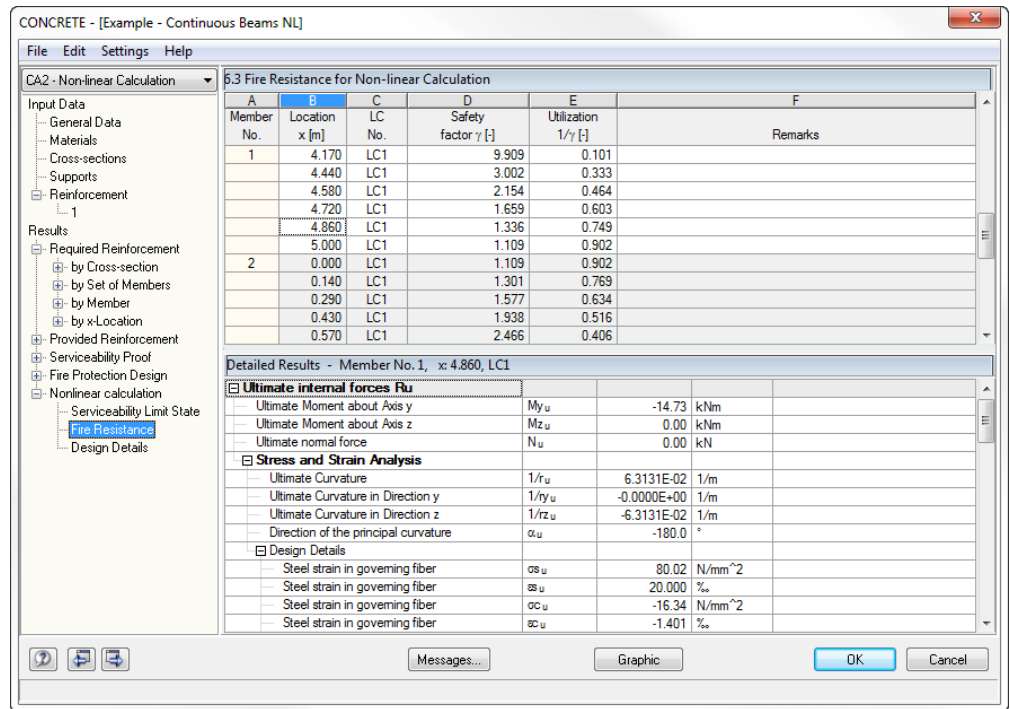


Figure 5.38: Table 6.3 Fire Resistance for Non-linear Calculation

The table contains the governing safety factors determined during the non-linear fire protection design. They result from the parameters set for the reinforcement groups, the internal forces of the actions and the specifications in the dialog box *Settings for Non-linear Calculation* (see Figure 4.2, page 81 and Figure 4.3, page 83).

In the lower part of the table the *Detailed Results* for the entry selected in the table row above are shown. Thus, due to the design details, it is possible to evaluate the results specifically.

The individual table columns correspond to the columns of table 6.1. They are described in chapter 5.5.1.

5.5.4 Nonlinear Calculation - Design Details

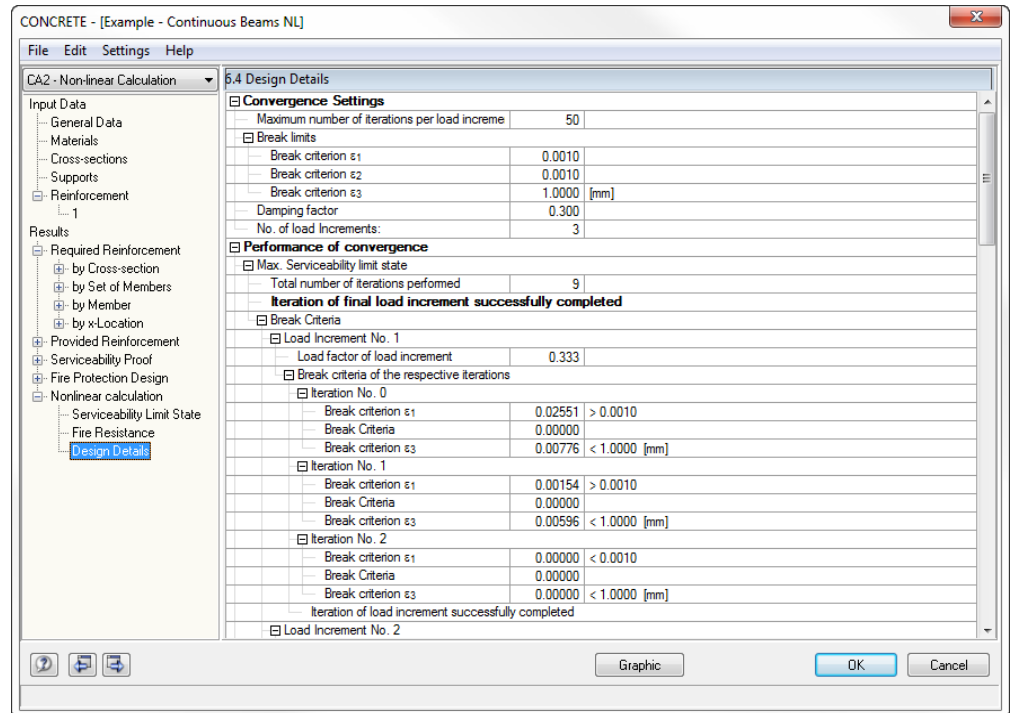


Figure 5.39: Table 6.4 Design Details

The final results table represents an important control option for the non-linear calculation process. The output is subdivided into two main parts.

Convergence Settings

This section lists all global specifications significant for the non-linear calculation (see descriptions for Figure 2.29, page 51).

Performance of convergence

The convergence behavior is an important starting point in order to evaluate the results from the non-linear calculation by looking at the iteration process. The stop criteria of the single iteration steps are shown separately for the ultimate limit state, the serviceability limit state and the fire protection design as well as for each load increment.

In general, a non-linear calculation is converging because the deviations concerning internal forces, stiffnesses and deformations are decreasing continuously. This effect can be checked by means of the values ε_1 and ε_2 from subsequent iterations: It is easier to understand peaks or increasing deviations (for example in case of stability analyses) in this way.

With the help of the convergence performance you can evaluate the calculation run appropriately. It is also possible to make conclusions about how calculation parameters are to be influenced, if necessary (see chapter 2.4.9 *Convergence*, page 50).

6. Results Evaluation

After the design has been carried out, the results can be evaluated in various ways. The individual results tables are described in detail in chapter 5. This chapter describes the graphical evaluation.

6.1 Reinforcement Proposal

The results tables 3.1 and 3.2 show you how the required areas of reinforcement can be covered with rebars to fulfill the relevant design, for example the serviceability limit state design. The reinforcement proposal is displayed graphically in the form of a drawing in the lower part of table 3.1 *Longitudinal Reinforcement Provided* and table 3.2 *Shear Reinforcement Provided* (see Figure 5.9, page 95 and Figure 5.13, page 99).

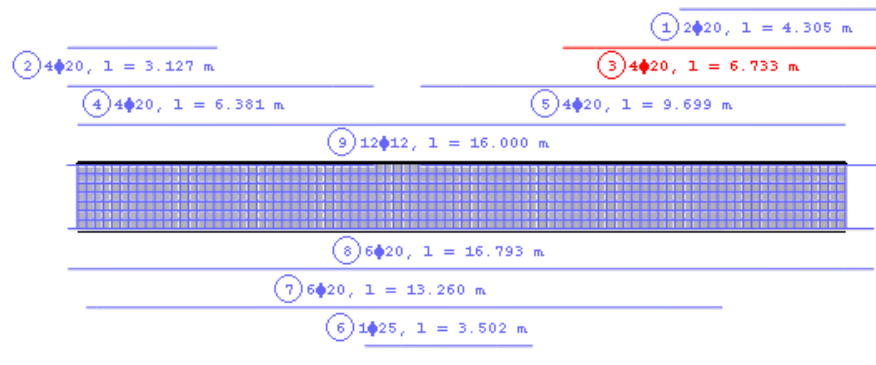


Figure 6.1: Reinforcement drawing in table 3.1 *Longitudinal Reinforcement Provided*

The currently selected item (the row in the table above in which the pointer is placed) is highlighted in red. The graphic allows you to display the position and arrangement of the individual item members in order to evaluate them appropriately.

To open the edit dialog box for the selected reinforcement position, click the [Edit] button in the bottom right corner of the drawing. The dialog box is shown in Figure 5.12 on page 98 or in Figure 5.14 on page 101. In this dialog box, you can check the numerous parameters of the selected longitudinal or link reinforcement and, if necessary, adjust them.

6.2 3D Rendering of Reinforcement

3D-Rendering

Both results tables 3.1 *Longitudinal Reinforcement* and 3.2 *Shear Reinforcement* provide the button [3D-Rendering] enabling a photo-realistic visualization of the provided reinforcement. A new window opens showing a rendered graphical representation of the reinforcement cage of the current member or set of members (i.e. the table row of the object where the pointer is placed).

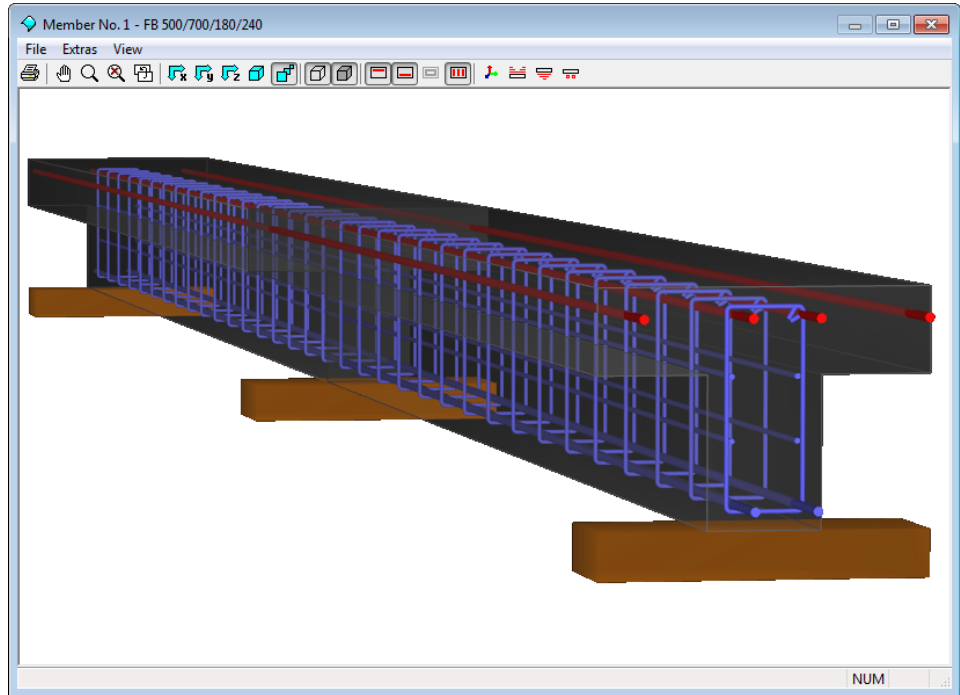


Figure 6.2: 3D rendering of provided longitudinal and link reinforcement

By means of the graphic you can check the selected reinforcement close to reality.

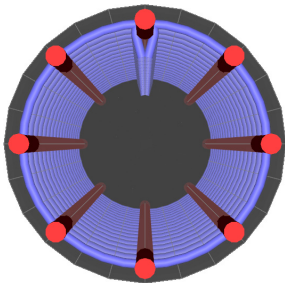


To set the graphic display appropriately, use the *View* pull-down menu or the corresponding buttons (see Table 6.1). Similar to the display in RSTAB, you can use the control functions: Shifting, zooming and rotating the object by keeping the [Shift] or the [Ctrl] key pressed.



The current graphic can also be sent directly to the printer, the printout report or the clipboard.

The buttons in the toolbar are reserved for the following functions:



Reinforcement of column:
perspective view in X























Button	Description	Function
	Print	Opens the dialog box <i>Graphic Printout</i> (Figure 7.4, page 130) providing the printout settings
	Shift	Allows for shifting the view by the mouse (zooming/rotating by pressing [Shift] or [Ctrl] key)
	Zoom	Allows for increasing a particular region in the graphic by drawing a window with the left mouse button
	Show whole structure	Resets the graphic's full view
	Previous view	Shows the view previously selected
	View in X	Shows the view in plane YZ
	View in Y	Shows the view in plane XZ
	View in Z	Shows the view in plane XY
	Isometric view	Shows the object in 3D
	Perspective view	Shows the object in a perspective view (can be combined with all four types of view)
	Line model	Hides the concrete material
	Solid model	Represents the concrete in the member or set of member
	Top reinforcement	Displays the longitudinal reinforcement defined in top of member
	Bottom reinforcement	Displays the longitudinal reinforcement defined in bottom of member
	Peripheral reinforcement	Displays the peripheral or secondary longitudinal reinforcement
	Link reinforcement	Shows the link reinforcement
	Member axis system	Controls the display of the local member axes x,y,z
	Longitudinal reinforcement - top	Shows the item members of the top reinforcement above the member
	Longitudinal reinforcement - bottom	Shows the item members of the bottom reinforcement below the member
	Shear reinforcement	Displays the item members of the link reinforcement

Table 6.1: Buttons for 3D rendering

6.3 Results in the RSTAB Model

To evaluate the design results graphically, you can also use the RSTAB work window.

RSTAB background graphic

The RSTAB graphic in the background may be useful when you want to check the position of a particular member in the model. When you select a table row in the CONCRETE results table, the corresponding member is highlighted in the RSTAB background graphic. In addition, an arrow indicates the location x on the member that is displayed in the active table row.

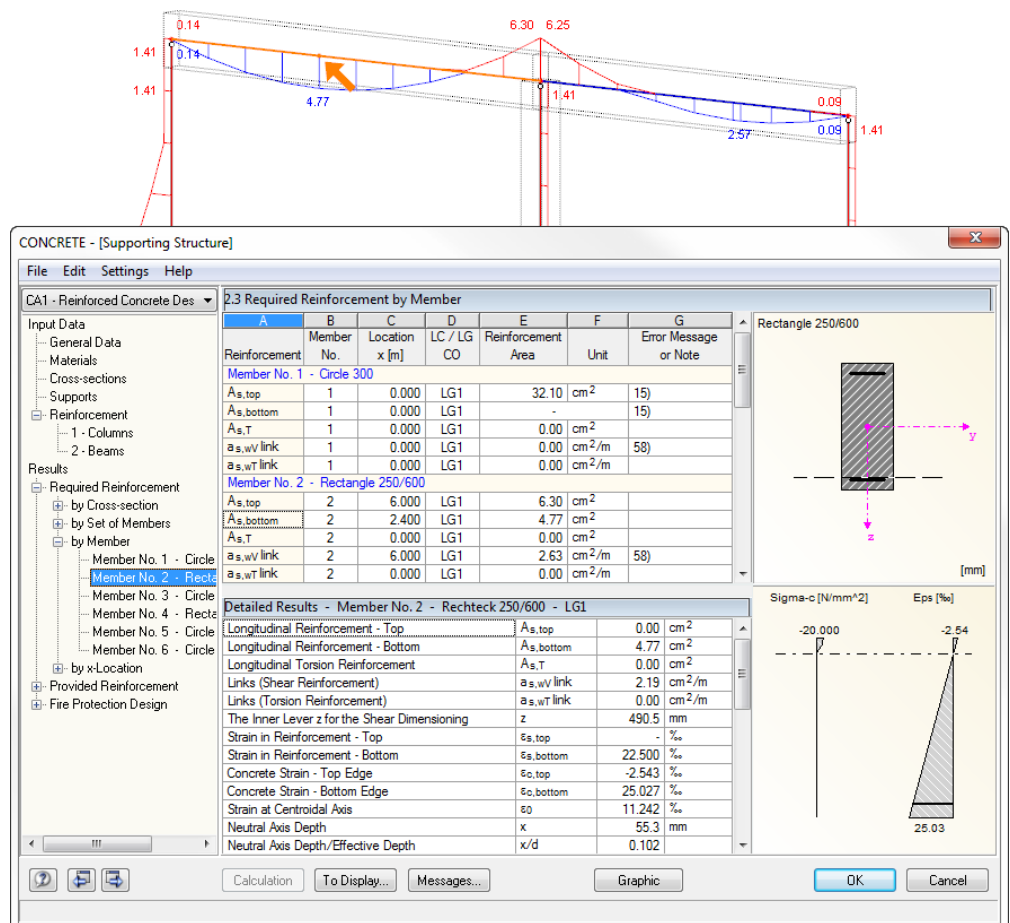


Figure 6.3: Indication of member and current Location x in the RSTAB model



This function, however, is only available if the results of the current CONCRETE case are set in the RSTAB graphical user interface. You can use the [Graphic] button to switch to RSTAB and the button [CONCRETE] in the control panel to return to the add-on module.

Graphic

RSTAB work window

All reinforcement areas and intermediate results can be visualized in the RSTAB structural model: First, click the [Graphic] button to close the add-on module CONCRETE. Now, the various reinforcements and design values are displayed in the graphic of the RSTAB work window like the internal forces or deformations of an RSTAB load case. The display of the different result types can be set in the *Results* navigator of CONCRETE.

The *Results* navigator is aligned with the designs of the CONCRETE module. The reinforcement types for the ultimate and the serviceability limit state design as well as the fire protection design and the non-linear calculation including all intermediate results are available for selection.

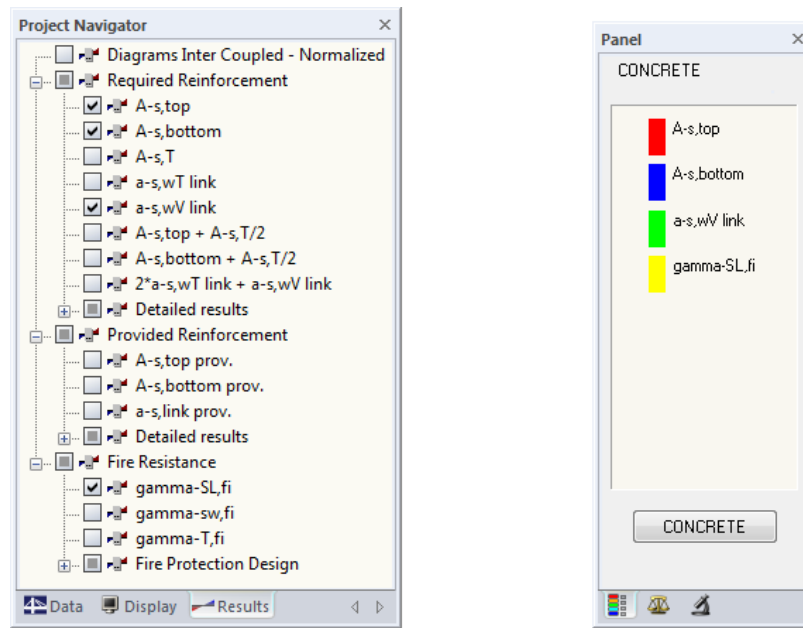


Figure 6.4: Results navigator of CONCRETE and panel with selected result types

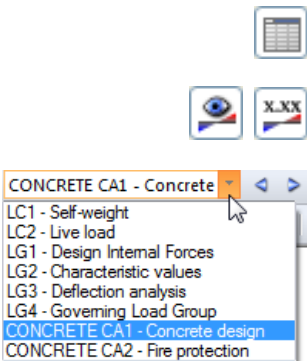
The *Results* navigator allows you to display several reinforcement types or designs at the same time. In this way, it is possible to compare graphically, for example, the required longitudinal reinforcement with the provided longitudinal reinforcement. The panel will be synchronized with the selected types of results.

Due to multiple selection and automatic color assignment, the options offered in the RSTAB *Display* navigator for the representation of member results are without effect.

As the RSTAB tables are of no relevance for the evaluation of CONCRETE results, you may deactivate them.

To turn the display of design results on or off, use the button [Results on/off] shown on the left. To display the result values in the graphic, use the toolbar button [Show Result Values] to the right.

The design cases are selected as usual by means of the list in the RSTAB menu bar.



As you already know it from the member internal forces, a scaling of member diagrams can be set in the second panel tab *Display Factors*. This means that you can scale the design results for the evaluation (and the printout) graphically.

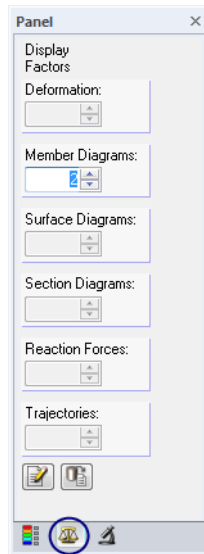


Figure 6.5: Panel tab *Display Factors*

The results of non-linear analyses can be evaluated specifically as well.

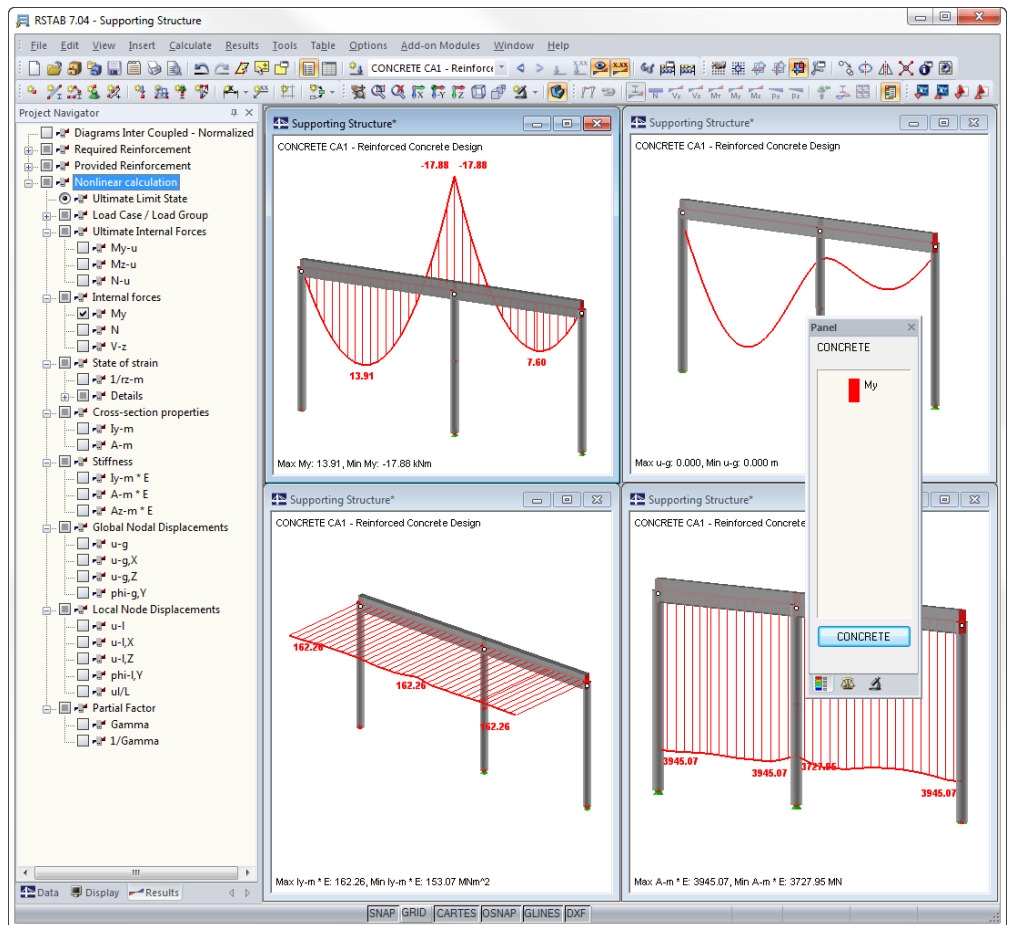


Figure 6.6: Graphical output of non-linear design results

CONCRETE

All results graphics can be transferred like RSTAB graphics to the global printout report (see chapter 7.2, page 130).

It is always possible to return to the design module by clicking the [CONCRETE] button in the panel.

6.4 Result Diagrams

The result diagrams can be accessed in the RSTAB graphic. To display the diagrams,



select **Result Diagrams on Selected Members** on the **Results** menu, or use the button in the RSTAB toolbar shown on the left.

A window opens showing the distribution of the reinforcement areas and detailed results on the selected member or set of members.

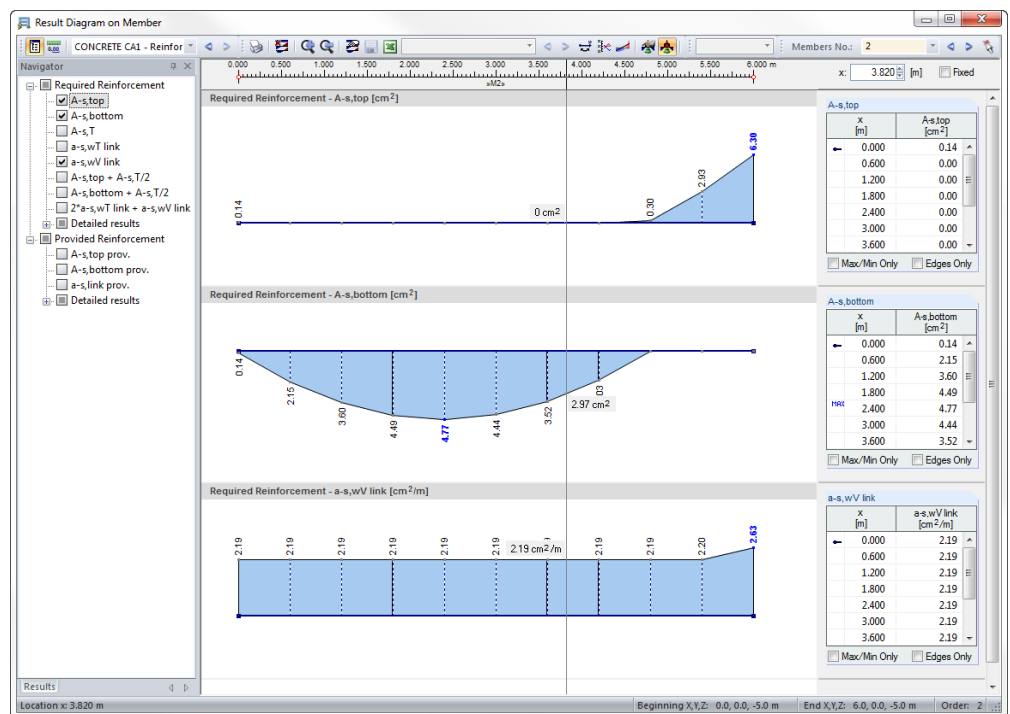


Figure 6.7: Dialog box *Result Diagram on Member*



In the navigator to the left, select the reinforcements and detailed results that you want to display in the result diagram. By means of the lists in the toolbar above you can set a particular CONCRETE design case as well as specific members or sets of members for display.

For more detailed information on the dialog box *Result Diagram on Member*, see the RSTAB manual, chapter 9.8.4, page 205.

6.5 Filter for Results

In addition to the results tables which already allow for a particular selection according to certain criteria because of their structure, you can use the filter options described in the RSTAB manual to evaluate the designs graphically.



On the one hand, you can take advantage of already defined partial views (see RSTAB manual, chapter 9.8.6, page 209) used to group objects appropriately.

On the other hand, you can define the numbers of the members, whose results should be shown exclusively in the graphic, in the *Filter* tab of the control panel. A description of this function can be found in the RSTAB manual, chapter 4.4.6, page 70.

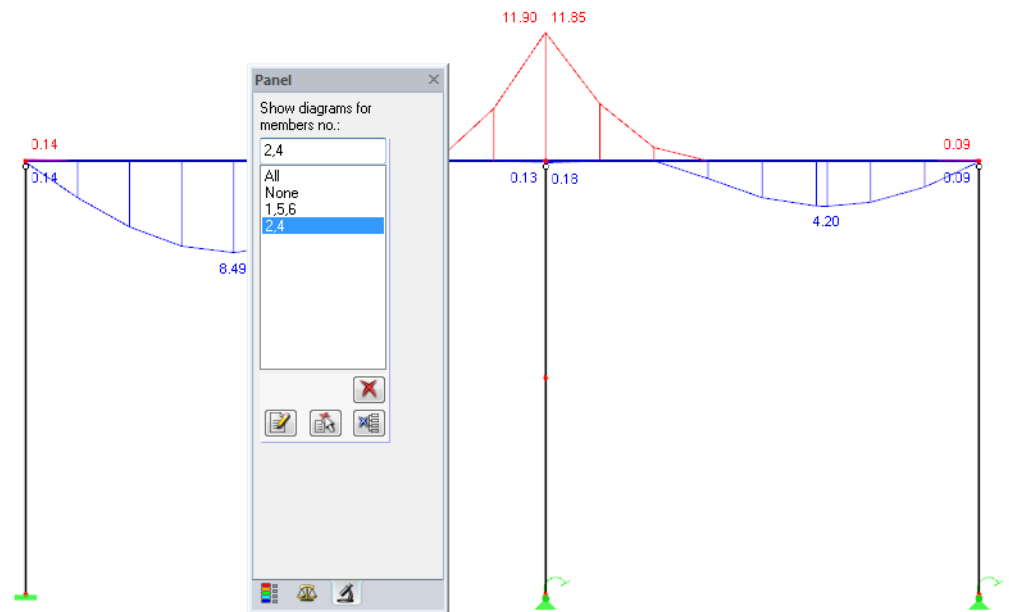


Figure 6.8: Filtering members in the panel

In contrast to the partial view function, the model is now displayed completely in the graphic.

7. Printout

7.1 Printout Report

The creation of printouts is similar to the procedure in RSTAB. First, the program generates a printout report for the CONCRETE results. Graphics and descriptions can be added. Moreover, this print preview requires the definition of results which you want to print for the reinforced concrete design.



When your structure is quite extensive, it is advisable to split the data into several small reports. If you create a separate report for CONCRETE, the printout report will be generated relatively quickly.

The printout report is described in detail in the RSTAB manual. In particular, chapter 10.1.3.4 *Selecting Data of Add-on Modules* on page 227 provides information concerning the selection of input and output data in add-on modules.

All general selection options are available in order to print the design cases as well as input and results data of CONCRETE.

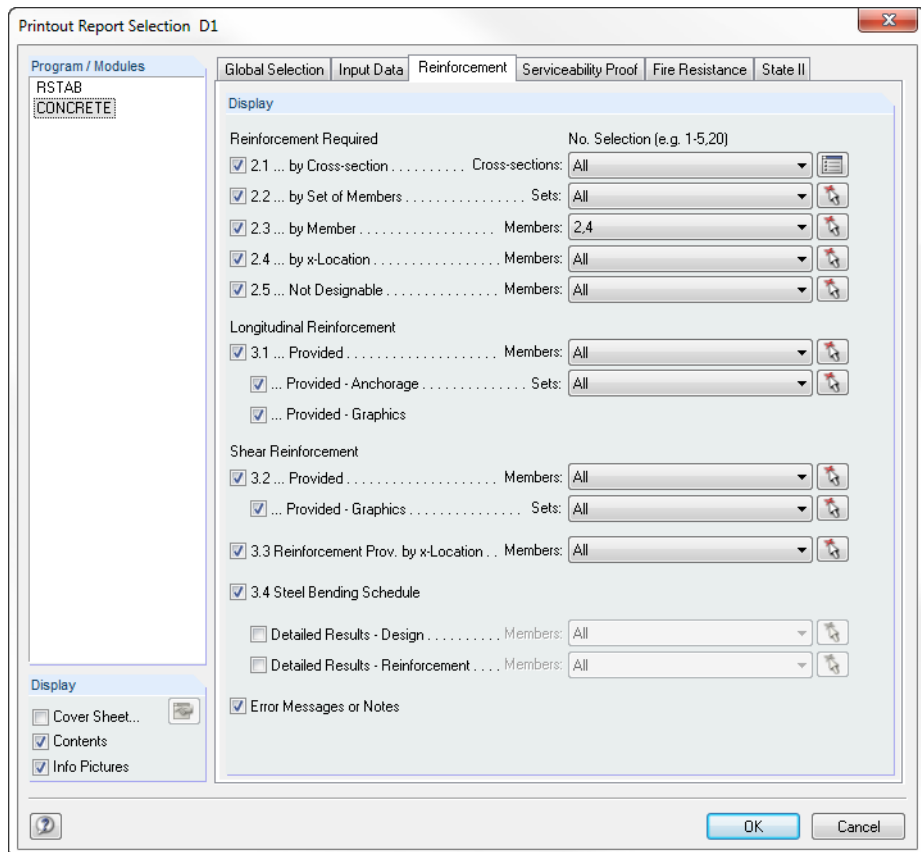


Figure 7.1: Printout report selection of CONCRETE results, tab Reinforcement

7.2 Graphic Printout

The design graphics can be either integrated in the printout report or sent directly to the printer. Printing graphics is described in detail in the RSTAB manual, chapter 10.2.



Every picture that is displayed in the graphic window of the main program RSTAB can be included in the printout report. Furthermore, it is possible to include the 3D rendering graphics and the member result diagrams in the printout report by using the [Print] button.

To print the CONCRETE graphic currently displayed in the RSTAB work window, select **Print** on the **File** menu or use the toolbar button shown on the left.



Figure 7.2: Button *Print* in the toolbar of the main window

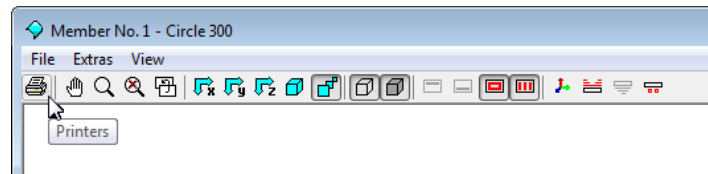


Figure 7.3: Button *Printers* in the toolbar of the 3D rendering window

The following dialog box opens:

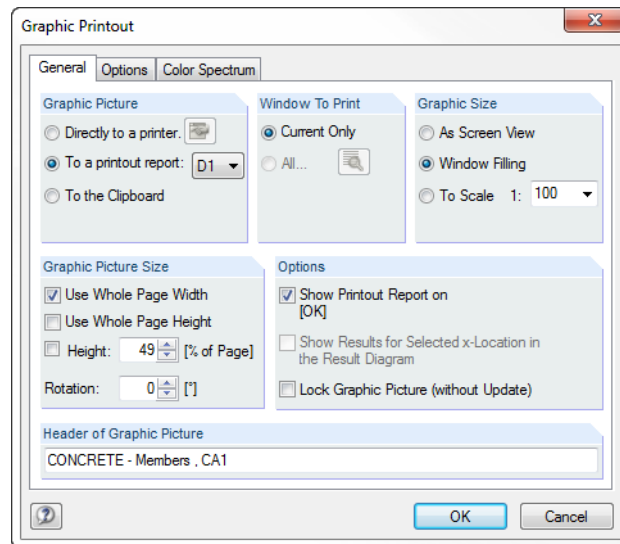
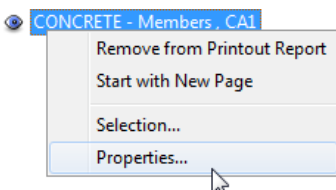


Figure 7.4: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in the RSTAB manual, chapter 10.2, page 243. The RSTAB manual also describes the *Options* and *Color Spectrum* tab.

A graphic from CONCRETE that has been integrated in the printout report can be moved anywhere within the report by using the drag-and-drop function. In addition, it is possible to adjust inserted graphics subsequently: Right-click the relevant entry in the navigator of the printout report and select *Properties* in the context menu. The dialog box *Graphic Printout* opens again, offering different options for modification.



8. General Functions

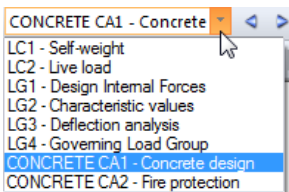
This chapter describes some menu functions as well as export options for the design results.

8.1 Design Cases in CONCRETE

Members and sets of members can be arranged in groups for different design cases. In this way, you can define particular design specifications (materials, cross-sections, reinforcement layout etc.) for example for groups of structural components.

It is no problem to analyze the same member or set of members in different design cases.

The CONCRETE design cases are available in the RSTAB work window and can be displayed like a load case or load group by means of the toolbar list.



Create a new CONCRETE case

To create a new design case,

select **New Case** on the **File** menu in the CONCRETE add-on module.

The following dialog box appears.

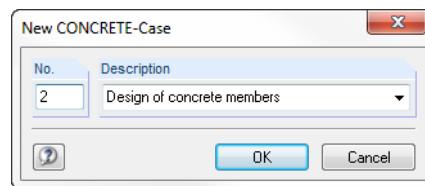


Figure 8.1: Dialog box *New CONCRETE-Case*

In this dialog box, enter a *No.* (which is not yet assigned) and a *Description* for the new design case. When you click [OK], table 1.1 *General Data* opens where you can enter the new design data.

Rename a CONCRETE case

To change the description of a design case subsequently,

select **Rename Case** on the **File** menu in the CONCRETE add-on module.

The dialog box *Rename CONCRETE-Case* appears.

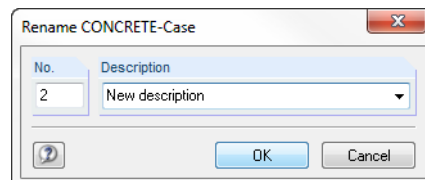


Figure 8.2: Dialog box *Rename CONCRETE-Case*

Copy a CONCRETE case

To copy the input data of the current design case,

select **Copy Case** on the **File** menu in the CONCRETE add-on module.

The dialog box *Copy CONCRETE-Case* appears where you can specify the number and description of the new case.

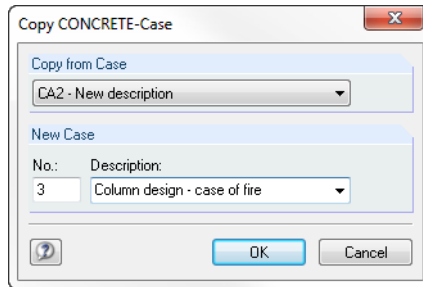


Figure 8.3: Dialog box *Copy CONCRETE-Case*

Delete a CONCRETE case

To delete a design case,

select **Delete Case** on the **File** menu in the CONCRETE add-on module.

In the dialog box *Delete Cases*, you can select a design case in the *Available Cases* list to delete it by clicking [OK].

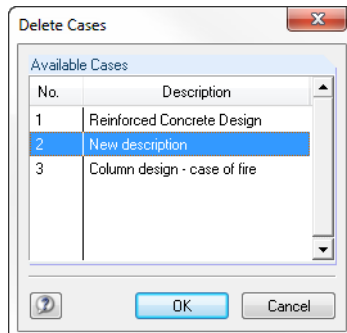


Figure 8.4: Dialog box *Delete Cases*

8.2 Cross-section Optimization

As mentioned in chapter 3.3, CONCRETE offers you the possibility to optimize cross-sections. Tick the check box of the relevant cross-section in column C of table 1.3 *Cross-sections*. Simply click into the little box to open the following dialog box where you can define detailed specifications.

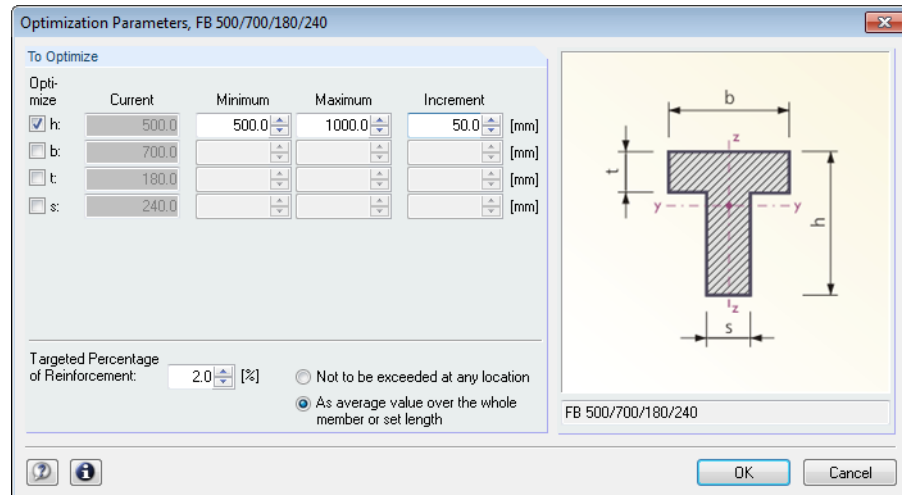


Figure 8.5: Dialog box *Optimization Parameters* of a T-beam

By ticking the check box(es) in the *Optimize* column, you decide which parameter(s) you want to modify. The ticked check box enables the *Minimum* and *Maximum* columns in order to define the upper and lower limit of the corresponding parameter for optimization. The *Increment* column determines the interval in which the dimensions of this parameter vary during the optimization process.

The criterion for optimization is specified by the fact that the *Targeted Percentage of Reinforcement* either won't be exceeded at any location or is available as average value across the entire member or set of members. The reinforcement ratio can be defined in the input field.

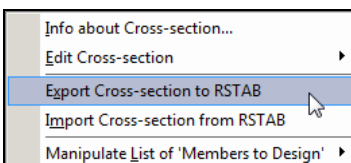


In the course of the optimization process, CONCRETE finds out which dimensions of the allowable parameter should be used in order to still fulfill the design. Please note that the internal forces won't be recalculated automatically with the changed cross-sections. It is up to you to decide when to transfer the optimized cross-sections for a new calculation run to RSTAB. As a result of optimized cross-sections, internal forces may vary considerably because of the changed stiffnesses in the structural system. Therefore, it is recommended to recalculate the internal forces after the first optimization and then to modify the cross-sections once again, if necessary.

You do not need to transfer the modified cross-sections to RSTAB manually: Set table 1.3 *Cross-sections*, and then

select **Export Cross-section to RSTAB** on the **Edit** menu.

Also the context menu of the table rows in table 1.3 shown on the left provides options to export modified cross-sections to RSTAB.



Calculation

Before the changed cross-sections are transferred to RSTAB, a security query appears, because the transfer requires the deletion of results. When you confirm the query and then start the [Calculation] in CONCRETE, the RSTAB internal forces as well as the areas of reinforcement are determined in one single calculation run.

To reimport the original RSTAB cross-section to CONCRETE, use the menu function already described above. Please note that this option is only available in table 1.3 *Cross-sections*.

If you optimize a tapered member, the program modifies the member's start and end and interpolates the second moments of area for the intermediate locations linearly. As these moments are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-section differ considerably. In this case, it is recommended to divide the taper into several single members whose start and end cross-sections have minor differences in depth.

8.3 Units and Decimal Places

The units and decimal places for RSTAB and all add-on modules are managed in one global dialog box. In the add-on module CONCRETE, you can use the menu to define the units. To open the corresponding dialog box,

select **Units and Decimal Places** on the **Settings** menu.

The following dialog box opens, which you already know from RSTAB. The add-on module CONCRETE is preset.

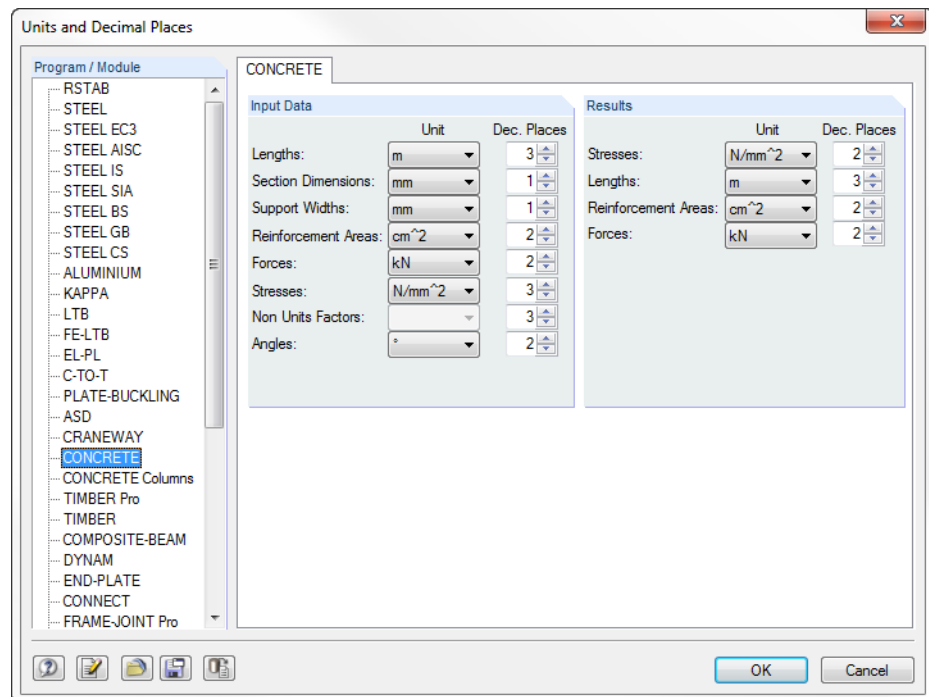


Figure 8.6: Dialog box *Units and Decimal Places*



The settings can be saved as user profile to reuse them in other structures. The corresponding functions are described in the RSTAB manual, chapter 11.6.2, page 336.

8.4 Export of Results

The results of the reinforced concrete design can also be used in other programs.

Clipboard

To copy cells selected in the CONCRETE results tables to the clipboard, use the keyboard keys [Ctrl]+[C]. To insert the cells, for example in a word processing program, press [Ctrl]+[V]. The headers of the table columns won't be transferred.

Printout report

The CONCRETE data can be printed into the printout report (cf. chapter 7.1, page 129) to export it subsequently. In the printout report,

select **Export to RTF File or BauText** on the **File** menu.

The function is described in detail in the RSTAB manual, chapter 10.1.11, page 239.

Excel / OpenOffice

CONCRETE provides a function for the direct data export to MS Excel and OpenOffice.org Calc. To open the corresponding dialog box,

select **Export Tables** on the **File** menu in the CONCRETE add-on module.

The following export dialog box appears.

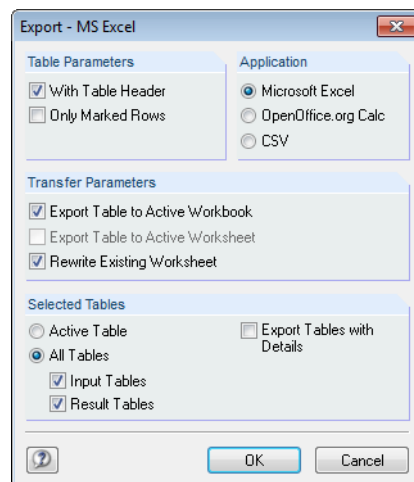


Figure 8.7: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking [OK]. Excel or OpenOffice will be started automatically. It is not necessary to run the programs in the background.

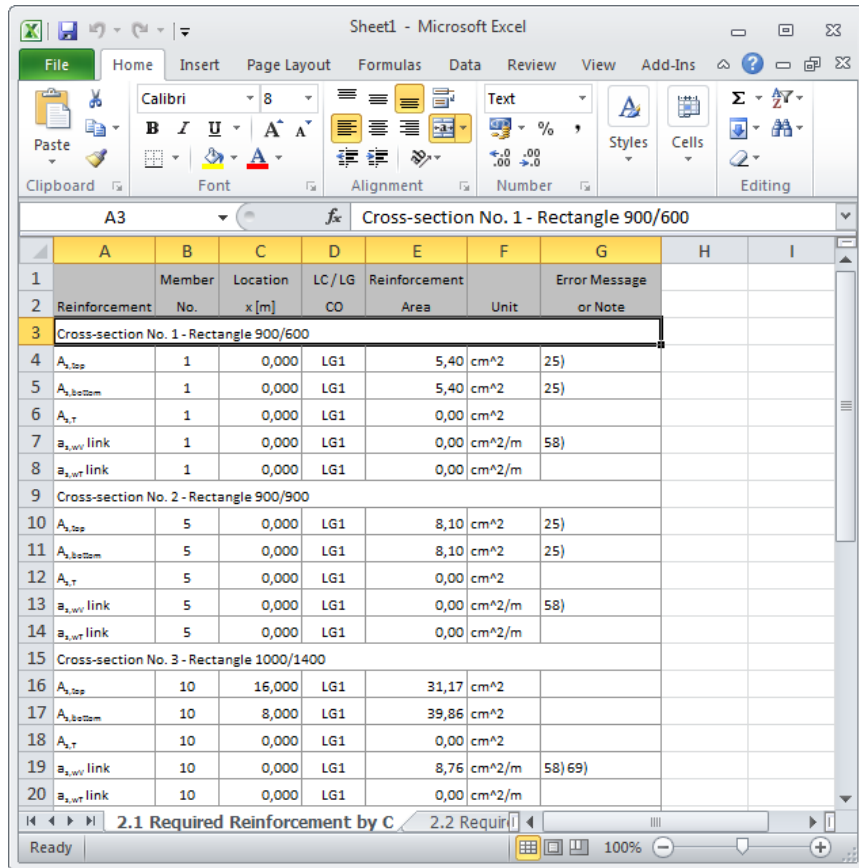


Figure 8.8: Results in Excel

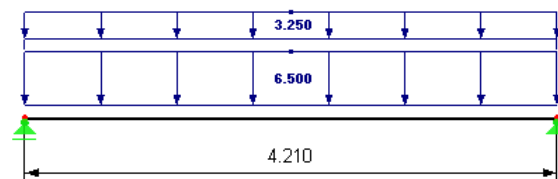
9. Examples

9.1 Direct Deformation Analysis

This chapter describes an example where the limitation of deformations is designed according to EN 1992-1-1, 7.4.3 by means of a direct calculation.

9.1.1 Input Data

System



Cross-section

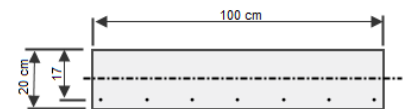


Figure 9.1: System, loads and cross-section

Thickness of plate	20 cm
Material	Concrete C25/30 (characteristic values: see chapter 9.1.2) BSt 500
Reinforcement	$A_{s,prov} = 4.43 \text{ cm}^2$, $d = 17 \text{ cm}$

Loads

Self-weight	$0.20 \cdot 25.0 = 5.00 \text{ kN/m}$
Plaster and flooring	1.50 kN/m
	<hr/>
	$g_k = 6.50 \text{ kN/m}$
Live load of office	2.00 kN/m
Load of partition walls	1.25 kN/m
	<hr/>
	$q_k = 3.25 \text{ kN/m}$

Maximum moment for quasi-permanent load

Combination coefficient	$\psi_2 = 0.3$ (live load of office)
Combination coefficient	$\psi_2 = 1.0$ (load of partition walls)
Quasi-permanent load	$6.50 + 0.30 \cdot 2.00 + 1.0 \cdot 1.25 = 8.35 \text{ kN/m}$
Maximum moment	$M_{\text{quasi-permanent}} = 8.35 \cdot 4.21^2 / 8 = 18.50 \text{ kNm}$

9.1.2 Initial Values of Deformation Analysis

Parameters

Mean modulus of elasticity	$E_{cm} = 29000 \text{ MN/m}^2$
Mean tensile strength	$f_{ctm} = 2.2 \text{ MN/m}^2$
Final creep ratio	$\varphi = 1.8$ (interior room)
Strain due to shrinkage	$\varepsilon_s = -0.5 \text{ ‰}$

Ratio of longitudinal reinforcement

$$\rho_e = \frac{A_s}{b \cdot d} = \frac{4.43 \text{ cm}^2}{100 \cdot 17 \text{ cm}^2} = 0.0026$$

$$\alpha_e = \frac{E_s}{E_c} = \frac{200000}{10360} = 19.31$$

Effective modulus of elasticity of concrete

$$E_{c,eff} = \frac{E_{cm}}{1 + \varphi} = \frac{29000}{1 + 1.8} = 10360 \text{ N/mm}^2$$

The influence of creeping is taken into account by the final creep ratio φ .

9.1.3 Curvature for Uncracked Sections (State I)

Cross-section properties

$$W_{c,I} = 1.00 \cdot 0.20^2 / 6 = 0.00667 \text{ m}^3$$

$$I_{c,I} = 1.00 \cdot 0.20^3 / 12 = 0.000667 \text{ m}^4$$

$$S_I = A_s \cdot z_s = 0.000443 \cdot 0.070 = 0.000031 \text{ m}^3$$

Curvature due to load

$$\left(\frac{1}{r}\right)_M = \frac{M_{Ed}}{E_{c,eff} \cdot I_c} = \frac{0.01850 \text{ MNm}}{10360 \text{ MN/m}^2 \cdot 0.000667 \text{ m}^4} = 0.00267 \text{ 1/m}$$

Curvature due to shrinkage

$$\left(\frac{1}{r}\right)_{CS} = \varepsilon_{cs\infty} \cdot \alpha_e \cdot \frac{S_I}{I_c} = 0.0005 \cdot 19.31 \cdot \frac{0.000031 \text{ m}^3}{0.000667 \text{ m}^4} = 0.00045 \text{ 1/m}$$

Total curvature

$$\left(\frac{1}{r}\right)_{tot,I} = \left(\frac{1}{r}\right)_M + \left(\frac{1}{r}\right)_{CS} = 0.00267 + 0.00045 = 0.00312 \text{ 1/m}$$

9.1.4 Curvature for Cracked Sections (State II)

Curvature due to load

When characteristic loads are applied, concrete is showing a linear elastic behavior. The distribution of the concrete stress across the compression zone is assumed to be triangular.

The height of the concrete compression area can be determined as follows:

$$x = \rho \cdot \alpha_e \cdot d \cdot \left[-1 + \sqrt{1 + \frac{2}{\rho \cdot \alpha_e}} \right] = 0.0026 \cdot 19.31 \cdot 17 \text{ cm} \cdot \left[-1 + \sqrt{1 + \frac{2}{0.0026 \cdot 19.31}} \right] = 4.53 \text{ cm}$$

The tension stress within the reinforcement is determined with $M_{Ed} = 18.50 \text{ kNm}$ as follows:

$$\sigma_s = \frac{M}{A_s \cdot (d - \frac{x}{3})} = \frac{18.5 \cdot 10^{-3}}{4.43 \cdot 10^{-4} \cdot \left(0.17 - \frac{0.0453}{3}\right)} = 269.60 \text{ N/mm}^2$$

The curvature in the final crack state is determined as follows:

$$\left(\frac{1}{r}\right)_{M,II} = \frac{\varepsilon_s}{d - x} = \frac{1.34 \cdot 10^{-3}}{170 - 42.9} = 0.01054 \text{ 1/m}$$

$$\text{where } \varepsilon_s = \frac{\sigma_s}{E_s} = \frac{268.19}{200000} = 1.34 \cdot 10^{-3}$$

Curvature due to shrinkage

The curvature for cracked sections (state II) is determined in manual calculations by means of a table from [18] (see Figure 9.2).

$$\omega_1 = \alpha_e \cdot \frac{A_s}{b \cdot d} = 19.31 \cdot \frac{4.43 \text{ cm}^2}{100 \text{ cm} \cdot 17 \text{ cm}} = 0.050 \quad \rightarrow \quad \beta = 1.10$$

$$\left(\frac{1}{r}\right)_{CS,II} = \varepsilon_{CS\infty} \cdot \alpha_e \cdot \frac{S_{II}}{I_{II}} = \varepsilon_{CS\infty} \cdot \beta \cdot \frac{1}{d} = 0.0005 \cdot 1.10 \cdot \frac{1}{0.17 \text{ m}} = 0.00324 \text{ 1/m}$$

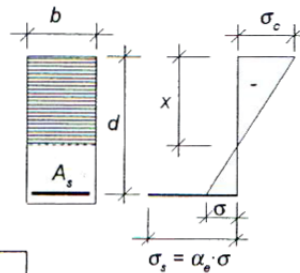
Total curvature

$$\left(\frac{1}{r}\right)_{\text{tot},II} = \left(\frac{1}{r}\right)_{M,II} + \left(\frac{1}{r}\right)_{CS,II} = 0.01054 + 0.00324 = 0.01378 \text{ 1/m}$$

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 : $\alpha_s = E_s / E_c = 15$
 Reinforcing steel: $E_s = 2 \cdot 10^5 \text{ MN/m}^2$



ω_1	$\xi = k_x$	$\zeta = k_\zeta$	$2/(\xi \cdot \zeta)$	$2/(\xi^2 \cdot \zeta)$	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:

$$\omega_1 = \alpha_s \cdot \frac{A_s}{b \cdot d}$$

Ratio of elastic moduli:

$$\alpha_s = \frac{E_s}{E_c}$$

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:

$$\kappa = \frac{1}{r} = \frac{2}{\xi^2 \cdot \zeta} \cdot \frac{\alpha_s \cdot M}{E_s \cdot b \cdot d^3}$$

Formula A 4.4 from EC2:

$$\alpha_s \cdot \left(\frac{S}{J} \right)_{II} = \beta \cdot \frac{1}{d}$$

Figure 9.2: Calculation table for cracked sections only (state II) from [18]

9.1.5 Determination of Deflection

As described in chapter 2.2.5 on page 16, it is possible to determine the probable value of the deformation according to equation (7.18) of EN 1992-1-1.

Coefficient of distribution

The distribution coefficient ζ between state I (uncracked sections) and state II (cracked sections) can be determined as follows:

$$\zeta = 1 - \beta_1 \cdot \beta_2 \cdot \left(\frac{\sigma_{s,cr}}{\sigma_s} \right)^2 = 1 - 1.0 \cdot 0.5 \cdot \left(\frac{213.78}{269.60} \right)^2 = 0.686$$

where $\beta_1 = 1.0$ Ribbed steel

$\beta_2 = 0.5$ Permanent load

The first cracking moment M_{cr} is:

$$M_{cr} = f_{ctm} \cdot W_I = 2.2 \cdot 0.00667 \cdot 10^3 = 14.67 \text{ kNm}$$

The stress $\sigma_{s,cr}$ immediately after cracking is determined with M_{cr} as follows:

$$\sigma_{s,cr} = \frac{M_{cr}}{A_s \cdot \left(d - \frac{x}{3} \right)} = \frac{14.67 \cdot 10^{-3}}{4.43 \cdot 10^{-4} \cdot \left(0.17 - \frac{0.0453}{3} \right)} = 213.78 \text{ N/mm}^2$$

Mean curvature

The mean curvature is determined with the distribution coefficient ζ as follows:

$$\frac{1}{r_m} = \zeta \cdot \frac{1}{r_{II}} + (1 - \zeta) \cdot \frac{1}{r_I} = 0.686 \cdot 0.01378 + (1 - 0.686) \cdot 0.00312 = 0.01043 \frac{1}{m}$$

Deformation

Thus, the deflection f in the beam center can be determined as follows:

$$f = k \cdot l_{\text{eff}}^2 \cdot \frac{1}{r_m} = \frac{5}{48} \cdot 4.21^2 \text{ m}^2 \cdot 0.01043 \frac{1}{m} = 19.3 \text{ mm}$$

9.1.6 Result in CONCRETE

In the results output, CONCRETE shows also 19.3 mm for the deformation value in the beam center.

A	B	C	D	E	F	G	H	I	J	K	L
Member No.	Location x [m]	LC / LG CO	σ_s [MPa]	σ_c [MPa]	min A_s [mm ²]	lim d_s [mm]	Bar Spacing lim s_1 [mm]	Crack Spacing $s_{r,max}$ [mm]	Crack Width max w_k [mm]	Deflections $u_{1,z}$ [mm]	Message
Governing:											
1	2.105	LC3	259.47	-7.72	0.00	14.1	175.7	257.8	0.2	19.3	331

Figure 9.3: Table 4.3 Serviceability Proof by Member

The following figure compares the deformations for state I (uncracked sections) and state II (cracked sections).

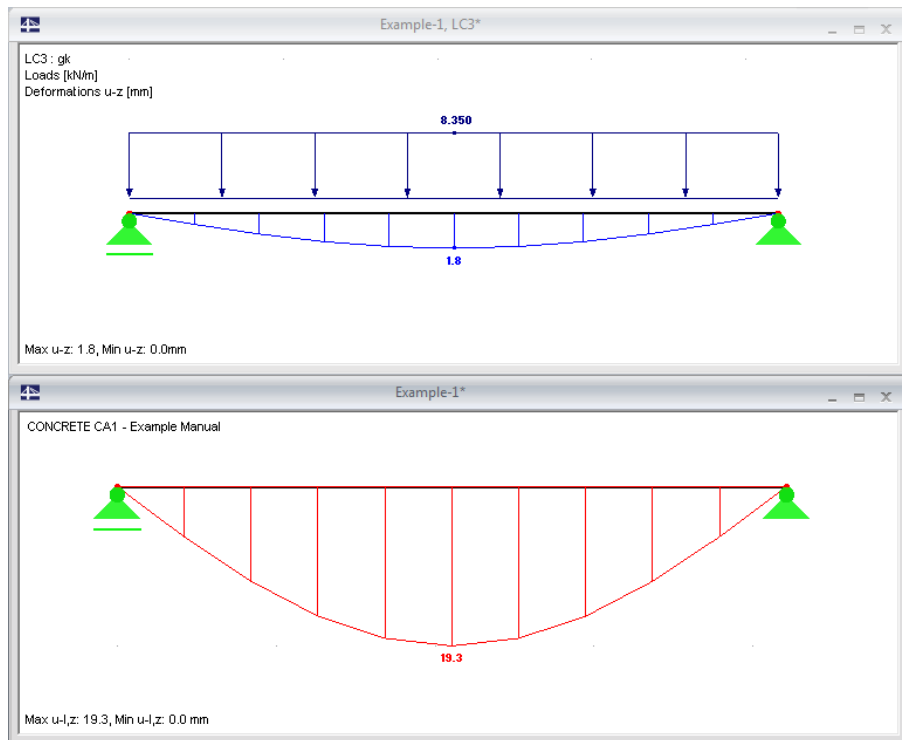


Figure 9.4: Deformations in state I (uncracked sections) and state II (cracked sections)

9.2 Deformation Analysis NL in SLS

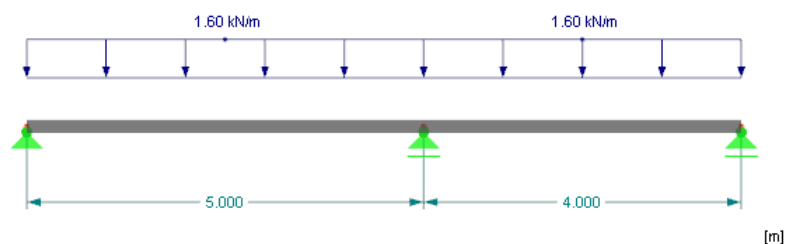
This example shows the basic relations of a non-linear calculation. For the manually performed reference calculation a two-span system is analyzed. The example from [14] is designed according to DIN 1045-1:2008-08.

Moreover, we look more closely at the approach used to limit the degree of bending slenderness as well as the tensile strength that is applied.

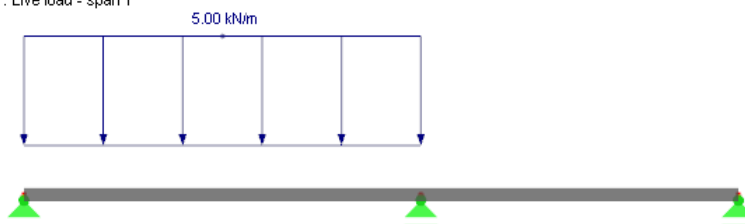
9.2.1 Input Data

System and loads

LC1: Permanent load



LC2: Live load - span 1



LC3: Live load - span 2

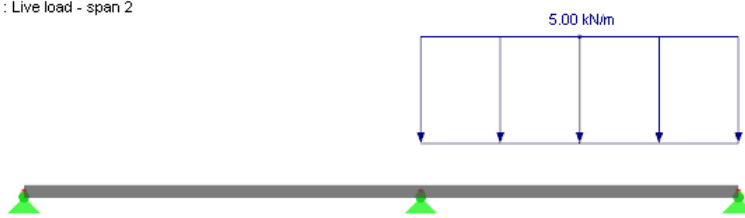


Figure 9.5: System and loads

Thickness of plate	16 cm
Material	Concrete C20/25 Reinforcing steel 500 S (A) and BSt 500 M (A)
Exposure class	XC1
Concrete cover	$c_{nom} = 20$ mm

Combination for ultimate limit state

In RSTAB we create a load combination that will be used for the design in the ultimate limit state. It is defined as follows:

$$1.35 * LC1/permanent + 1.5 * LC2 + 1.5 * LC3$$

Combination for serviceability limit state

The combination factor $\psi_{2,1}$ for the quasi-permanent combination of action is applied with 0.6. As superimposing load cases is not possible for non-linear calculations, three load groups with the following combination factors are defined for the design in the serviceability limit state:

- LG1: LC1 + 0.6 * LC2 + 0.6 * LC3
- LG2: LC1 + 0.6 * LC2
- LG3: LC1 + 0.6 * LC3

In addition, we have to deactivate the reduction of stiffness in the *Calculation Parameters* of each load group.

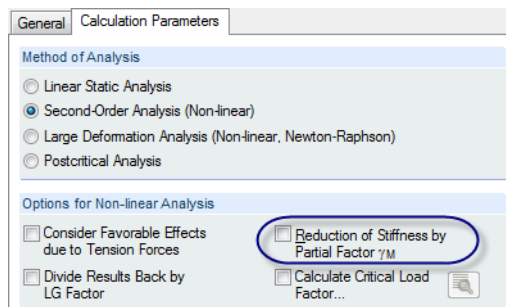


Figure 9.6: Clear check box of stiffness reduction

9.2.2 Basic Input in CONCRETE

To determine the reinforcement, we only look at the ultimate limit state. In the CONCRETE table 1.1 *General Data*, we select CO1 for the design.

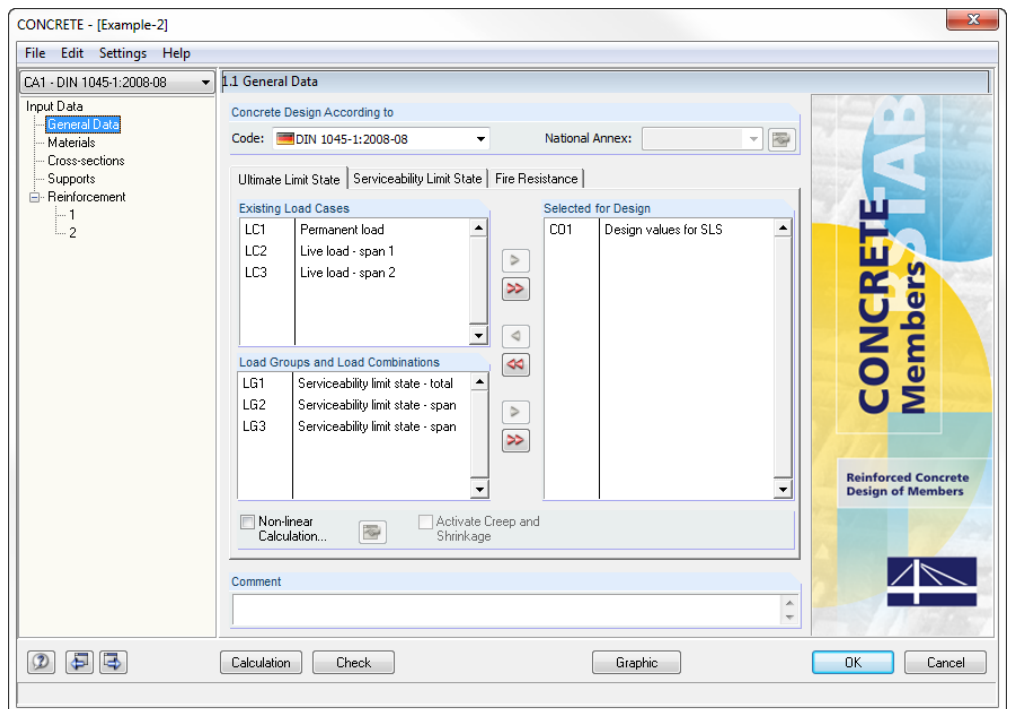


Figure 9.7: Table 1.1 *General Data*, tab *Ultimate Limit State*

In table 1.2 *Materials*, the materials *Concrete C20/25* and *B 500 S (A)* are taken from the library.

Table 1.3 *Cross-sections* is described in detail later when we consider creeping (see Figure 9.15, page 149).

As the calculation of the example in [14] is performed considering moment swap and moment reduction, we have to specify some settings in table 1.5 *Supports*.

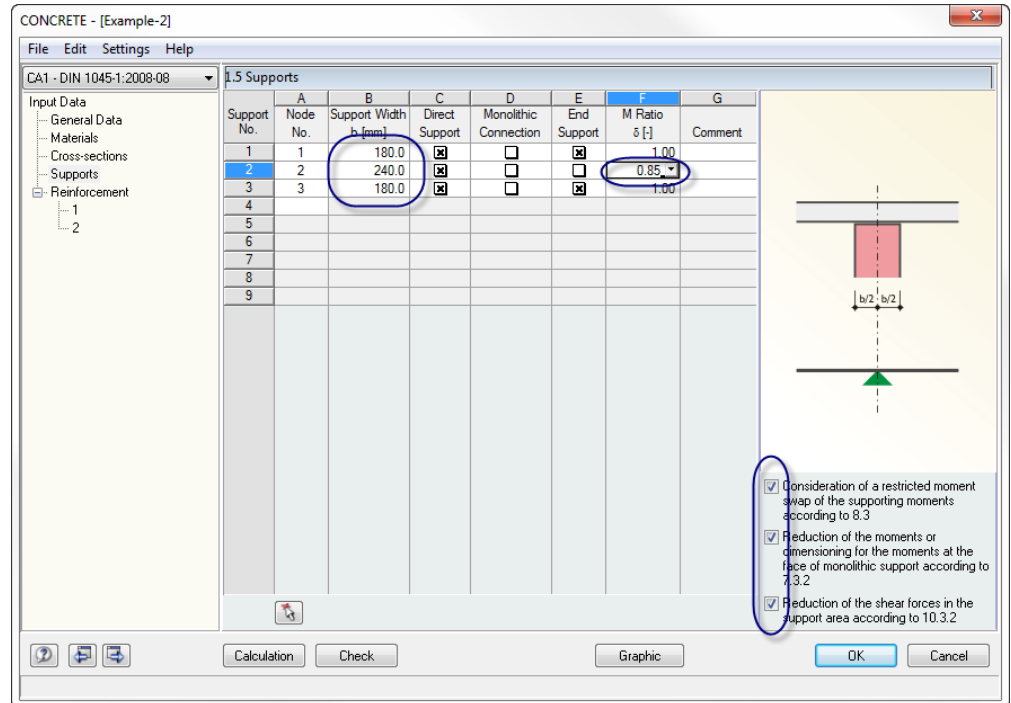


Figure 9.8: Table 1.5 *Supports*

First, we define the support widths shown in the picture. Then, in table column F, we reduce the maximum column moment for the intermediate support to 85 % of the linear elastic value. We have to tick the check boxes below the graphic so that CONCRETE will take into account the specifications.

Imaging the provided reinforcement is of major importance for the non-linear calculation because it represents a decisive value of influence for the determination of curvatures in cracked state. For our example it is necessary to define an additional reinforcement diameter of 6 mm in table 1.6 *Reinforcement* (see figure below).

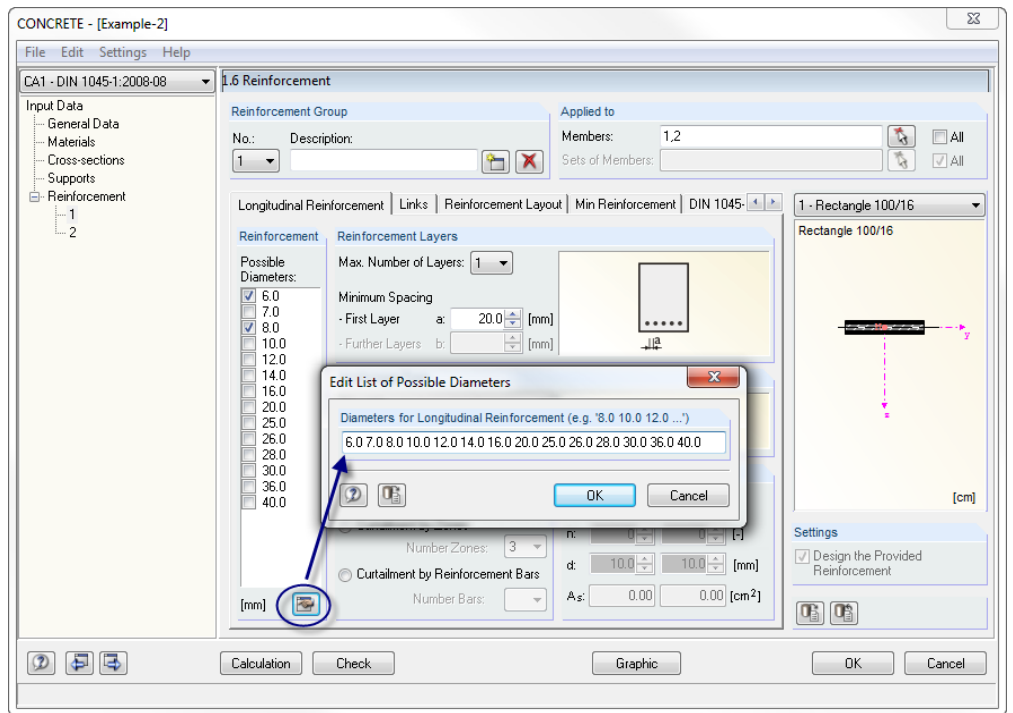


Figure 9.9: Table 1.6 Reinforcement, tab Longitudinal Reinforcement

In the table tab *Reinforcement Layout* of table 1.6 Reinforcement, we define the concrete covers with 21 mm in order to ensure the bar centroidal axis assumed with 25 mm in the design in [14].

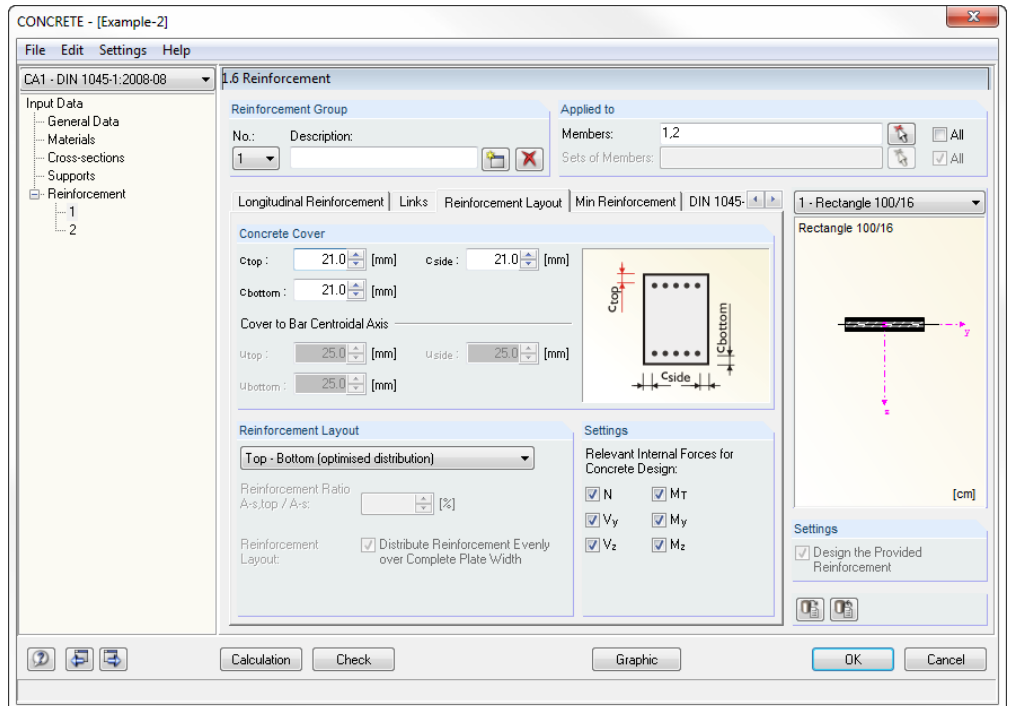


Figure 9.10: Table 1.6 Reinforcement, tab Reinforcement Layout

Now the input is complete and we can start the calculation.

9.2.3 Checking the Reinforcement

With regard to the effective cross-section area, the longitudinal reinforcement displayed in the reinforcement proposal corresponds to [14]. The non-continuous top reinforcement in the plate deviates from the reinforcement drawing in the example but we do not modify any specifications. In any case, the curvatures due to shrinkage will have a reducing effect.

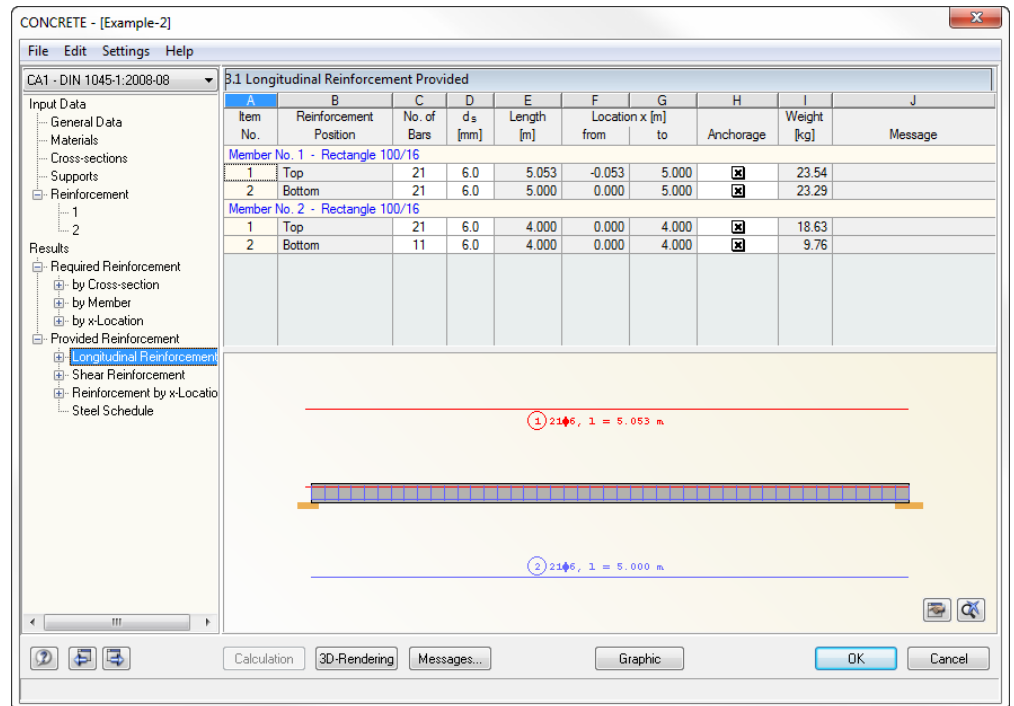


Figure 9.11: Table 3.1 Longitudinal Reinforcement Provided

9.2.4 Specifications for Non-linear Calculation

The non-linear calculation for the serviceability limit state is prepared in table 1.1 General Data, tab Serviceability Limit State.

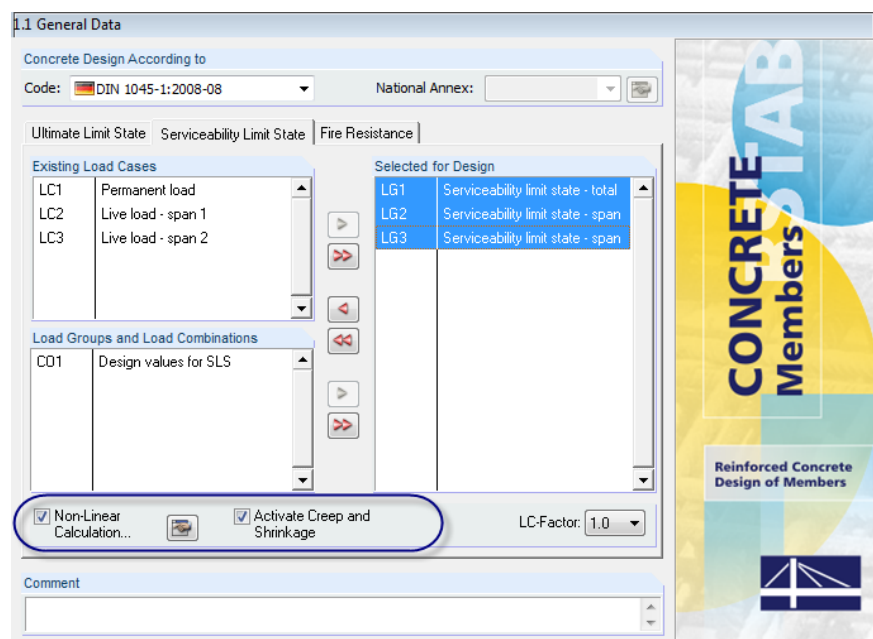
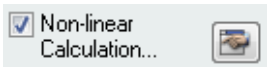


Figure 9.12: Table 1.1 General Data, tab Serviceability Limit State

For the design we select the load groups that are defined for the design in the serviceability limit state. In addition, we tick both check boxes for *Non-Linear Calculation* and *Activate Creep and Shrinkage*.



We use the button [Settings for Non-linear Calculation] to open the corresponding dialog box. The presettings in the tab *Analysis Approach* remain unchanged. In the tab *Tension Stiffening Effect* we specify the following settings.

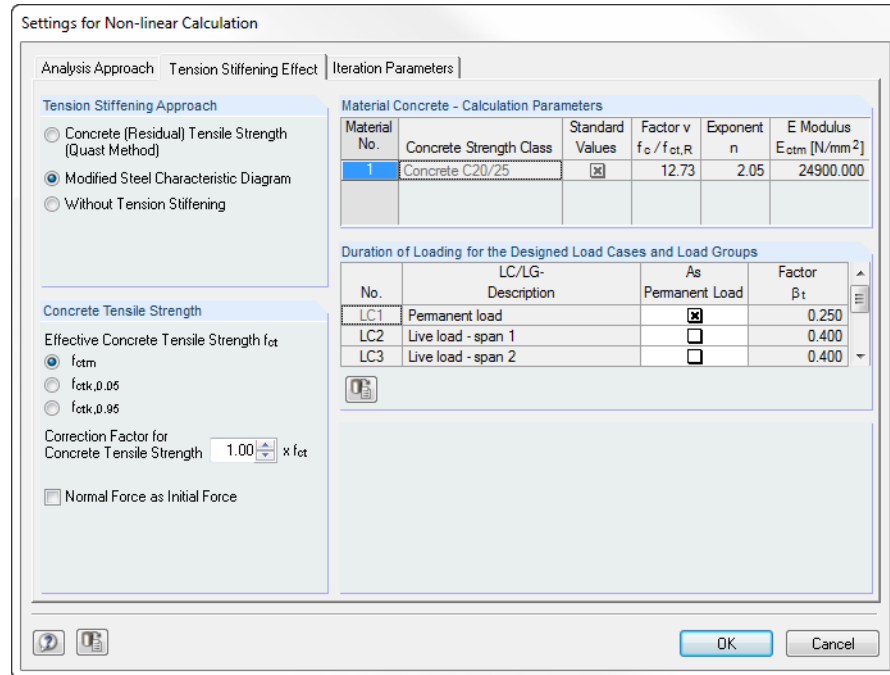


Figure 9.13: Dialog box *Settings for Non-linear Calculation*, tab *Tension Stiffening Effect*

We select the approach of the modified characteristic steel curve. For now we take the correction factor of the tensile strength $1.00 \cdot f_{ct}$ as it is already preset. The calculation is done with the mean axial tensile strength of concrete that is specified in DIN 1045-1, table 9.

The load duration factor β_t is calculated depending on the load cases of the relevant load groups between the limit values 0.25 and 0.4. Default settings for the axial forces are of no relevance for pure bending.

We can keep the default values set in the tab *Iteration Parameters*. We only reduce the limit lengths of the element divisions to values between 0.05 and 0.10 m.

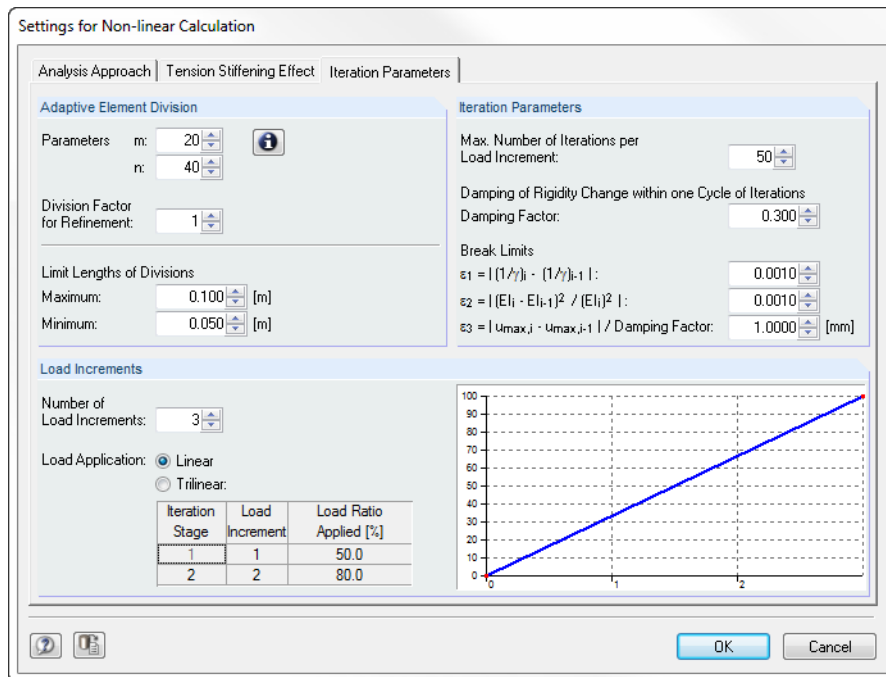


Figure 9.14: Dialog box *Settings for Non-linear Calculation*, tab *Iteration Parameters*

Finally, we define the settings for creep and shrinkage in table 1.3 *Cross-sections*.

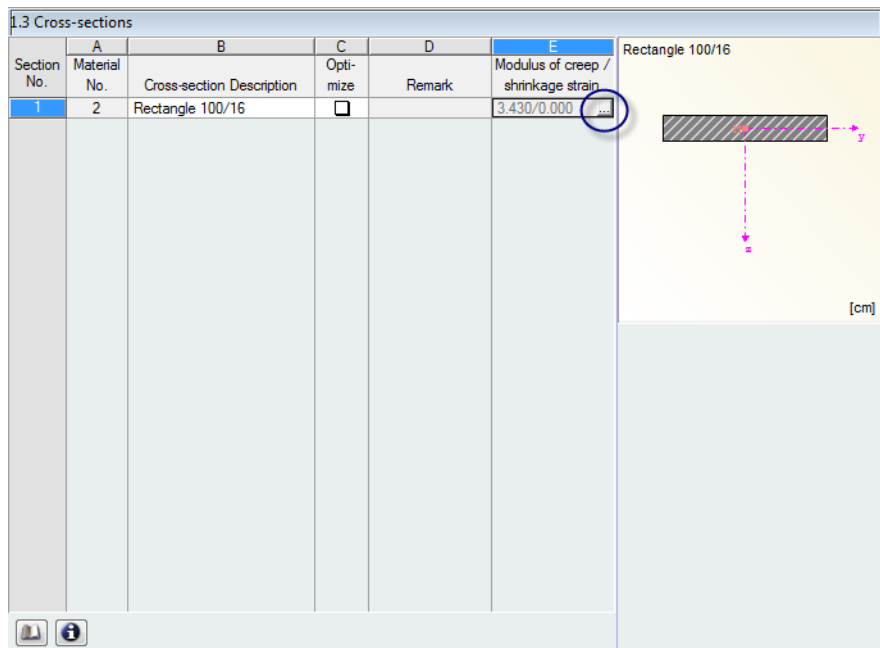


Figure 9.15: Table 1.3 *Cross-sections*

The button available in table cells of column E opens a dialog box required for entering creep and shrinkage parameters.

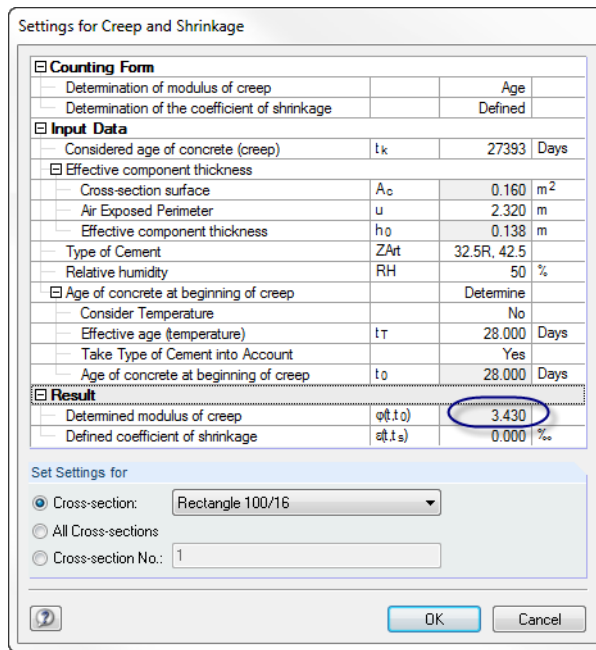


Figure 9.16: Dialog box *Settings for Creep and Shrinkage*

To compare: From the diagrams presented in DIN 1045-1, figure 18, a creeping factor of $\phi_{\infty} = 3.5$ is determined approximately for dry interior rooms (RH 50 %). This creep coefficient determined according to DIN 1045-1 must be understood as the pure final creep ratio and usually must still be converted into an effective creep coefficient in accordance with the relation of creep-producing and effective loading.

Shrinkage is not analyzed further: Due to the symmetrical reinforcement in span 1 and the minor reinforcement difference in span 2 the shrinkage curvatures are not contributing significantly to the total deformation.

Now the input for the non-linear calculation is complete and we can start the calculation.

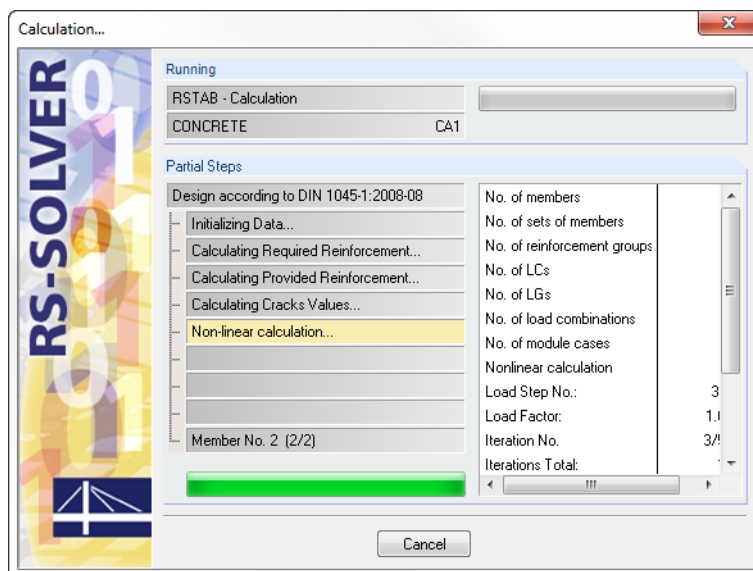


Figure 9.17: Calculation details of non-linear analysis

9.2.5 Results Display and Evaluation

A	B	C	D	E	F	G
Member No.	Location x [m]	LC No.	Global displacements u_g [mm]	Local displacements u_l [mm]	Crack Width w_k [mm]	
1	2.140	LG1	22.586	22.586	0.1	
		LG2	25.297	25.297	0.1	
		LG3	7.843	7.843	0.0	
	2.210	LG1	22.591	22.591	0.1	
		LG2	25.339	25.339	0.1	
		LG3	7.837	7.837	0.0	
	2.310	LG1	22.493	22.493	0.1	
		LG2	25.287	25.287	0.1	
		LG3	7.796	7.796	0.0	
	2.400	LG1	22.303	22.303	0.1	
		LG2	25.129	25.129	0.1	

Figure 9.18: Table 6.2 Serviceability limit state for non-linear calculation

In table 6.2 *Serviceability limit state for non-linear calculation* we can display the deformations on each location x. The maximum deformation occurs on the location $x = 2.21$ m for LG 2 (traffic load in span 1). The absolute value of $u = 25.34$ mm is equal to a limit value of $1/197 * l$, and thus lies below the recommended value of $1/250 * l$.

Stress and Strain Analysis			
Principal moments	M	17.66	kNm
— bending moment over the y-axis	My	17.66	kNm
— bending moment over the z-axis	Mz	0.00	kNm
— Normal force	N	0.00	kN
State of cross-section		Crack Formation	Completed
— Mean Main Curvature	1/r _m	1.3147E-02	1/m
— Middle curvature in y-direction	1/r _y	0.0000E+00	1/m
— Mean curvature in z-direction	1/r _z	1.3872E-02	1/m
— Average Strain in Reinforcement	ε _{sm}	1.068	%
Design Details			
— Coefficient to Take Account of Load Duration	β	0.306	
— Coefficient to Take Account of Yield Strain	delta	0.600	
— Crack moment	M _{cr}	12.93	kNm
— crack moment related to the y-axis	M _{cry}	12.93	kNm
— crack moment related to the z-axis	M _{crz}	0.00	kNm
— Crack normal force	N _{cr}		kN
Zustand I			
Cross-section properties			
— Moment of inertia about the y-axis	I _{yI}	1.06113E+08	mm ⁴
— Moment of inertia about the z-axis	I _{zI}	3.77977E+09	mm ⁴
— Surface	A _I	45385.6	mm ²
Stress and Strain Analysis			
— main curvature	1/r _I	6.6820E-03	1/m
— curvature in y-direction	1/r _{yI}	0.0000E+00	1/m
— curvature in z-direction	1/r _{zI}	6.6820E-03	1/m
— Direction of the principal curvature	α _I	0.0	°
— Steel strain in governing fiber	σ _{sI}	74.84	N/mm ²
— Steel strain in governing fiber	ε _{sI}	-3.00	%
— steel stress for crack moment	σ _{srI}	54.79	N/mm ²
— steel strain for the crack moment	ε _{srI}	0.274	%
Zustand II			
Cross-section properties			
— Moment of inertia about the y-axis	I _{yII}	48497500.0	mm ⁴
— Moment of inertia about the z-axis	I _{zII}	1.33333E+10	mm ⁴
— Surface	A _{II}	22733.2	mm ²
Stress and Strain Analysis			
— main curvature	1/r _{II}	1.4620E-02	1/m
— curvature in y-direction	1/r _{yII}	1.1061E-09	1/m
— curvature in z-direction	1/r _{zII}	1.4620E-02	1/m
— Direction of the principal curvature	α _{II}	0.0	°
— Steel strain in governing fiber	σ _{sII}	253.64	N/mm ²
— Steel strain	ε _{sII}	1.268	%
— Steel strain in governing fiber	σ _{crII}	-0.720	%
— steel stress for crack moment	σ _{srII}	185.61	N/mm ²
— steel strain for the crack moment	ε _{srII}	0.928	%
Cross-section properties			
Cross-sectional rigidities (Secant Values)			
Global Node Deformations			
Local Node Deformations			
Crack Width			
Verification of Safety			
Partial Factor	γ	2.353	
Percentage of utilization	1/γ	0.425	

Figure 9.19: Intermediate results for member 1 on location x = 2.21 m, LG 2

For the location $x = 2.21$ m the results are now recalculated step by step. Of primary interest is the calculation of the plane of strain and stress which is decisive as basis for the stiffness properties for the non-linear determination of deformations and internal forces. During manual calculations we sometimes use simplified approaches which may lead to minor differences.

9.2.5.1 Material Properties for Deformation Analysis

Concrete C 20/25

$$f_c = f_{cm} = 20 + 8 = 28 \text{ N/mm}^2$$

$$E_c = E_{cm} = 24900 \text{ N/mm}^2$$

$$\varepsilon_{c1} = -2.1 \text{ ‰}$$

$$\varepsilon_{c1u} = -3.5 \text{ ‰}$$

Distorted for creep with $(1 + \varphi) = 4.43$:

$$E_c = E_{cm} = 5620.8 \text{ N/mm}^2$$

$$\varepsilon_{c1} = -9.30 \text{ ‰}$$

$$\varepsilon_{c1u} = -15.51 \text{ ‰}$$

Reinforcing steel BSt 500 S (A)

$$f_{ym} = f_{yk} = 500 = 500.00 \text{ N/mm}^2$$

$$f_{tm} = f_{tk} = 550 = 550 \text{ N/mm}^2$$

$$E_s = 200000 \text{ N/mm}^2$$

$$\varepsilon_{su} = 25 \text{ ‰}$$

$$\alpha_e = 200000 / 24900 = 8.03$$

Distorted for creep with $(1 + \varphi) = 4.43$:

$$\alpha_e = 200000 / 5620.8 = 35.58$$

9.2.5.2 State I

When the cross-section properties are determined, the available steel area is taken into account. The missing area for concrete in the zone where rebars are lying 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 resulting directly:

$$a_c = 0 \text{ cm}$$

$$a_{s1} = 8 - 2.5 = 5.5 \text{ cm}$$

$$a_{s2} = 5.5 \text{ cm}$$

Second moment of area

$$I_{y,I} = \frac{b \cdot h^3}{12} + 2 \cdot (A_{s1/s2} \cdot a_2^2 \cdot \alpha_e) = \frac{100 \cdot 16^3}{12} + 2 \cdot (5.93 \cdot 5.5^2 \cdot 35.58) = 46898 \text{ cm}^4$$

Ideal cross-section area

$$A - (I) = A_c + A_s \cdot \alpha_e = 16 \cdot 100 + 11.86 \cdot 35.58 = 2022 \text{ cm}^2$$

Crack moment M_{cr}

We assume that the cross-section will crack when the tensile strength f_{ctm} in the most external fiber is reached.

$$\sigma = \frac{M_{cr}}{I} \cdot z_{ct} = f_{ctm}$$

$$M_{cr} = \frac{f_{ctm} \cdot I}{z_{ct}} = \frac{0.22 \cdot 46898}{8} = 1289.7 \text{ kNcm} = 12.9 \text{ kNm}$$

Steel stress σ_{sr1} and steel strain ϵ_{sr1} for crack moment

$$\sigma_{sr1,l} = f_{ctm} \cdot \frac{5.5}{8} \cdot \alpha_e = 2.2 \cdot \frac{5.5}{8} \cdot 35.53 = 53.74 \text{ N/mm}^2$$

$$\epsilon_{sr1,l} = \frac{\sigma_{sr1}}{E_s} = \frac{53.73}{200000} = 2.687 = 0.2687 \%$$

Notional steel and concrete stress for effective moment $M = 17.66 \text{ kNm}$

$$\sigma_{s1} = \frac{M}{I} \cdot z_{s1} \cdot \alpha_e = \frac{1766}{46898} \cdot 5.5 \cdot 35.58 = 7.4 \text{ kN/cm}^2 = 74 \text{ N/mm}^2$$

$$\sigma_c = -\frac{M}{I} \cdot z_{cc} = -\frac{1766}{46898} \cdot 8 = -0301 \text{ kN/cm}^2 = -3.01 \text{ N/mm}^2$$

Curvature in state I ($M = 17.66 \text{ kNm}$) – $(1/r)_{z,1} = (1/r)_1$

$$\left(\frac{1}{r}\right)_{z,1} = \frac{M}{E \cdot I} = \frac{0.01766}{5620 \cdot 4.6898e-4} = 6.758e-3 \text{ 1/m}$$

CONCRETE results

Design Details			
— Coefficient to Take Account of Load Duration	β	0.306	
— Coefficient to Take Account of Yield Strain	δ	0.600	
— Crack moment	M_{cr}	12.93	kNm
— crack moment related to the y-axis	M_{cry}	12.93	kNm
— crack moment related to the z-axis	M_{crz}	0.00	kNm
— Crack normal force	N_{cr}		kN
Zustand I			
Cross-section properties			
— Moment of inertia about the y-axis	I_{y1}	1.06113E+08	mm ⁴
— Moment of inertia about the z-axis	I_{z1}	3.77977E+09	mm ⁴
— Surface	A_1	45385.6	mm ²
Stress and Strain Analysis			
— main curvature	$1/r_1$	6.6820E-03	1/m
— curvature in y-direction	$1/r_{y1}$	0.0000E+00	1/m
— curvature in z-direction	$1/r_{z1}$	6.6820E-03	1/m
— Direction of the principal curvature	α_1	0.0	°
— Steel strain in governing fiber	σ_{s1}	74.84	N/mm ²
— Steel strain in governing fiber	σ_c	-3.00	N/mm ²
— steel stress for crack moment	σ_{sr1}	54.79	N/mm ²
— steel strain for the crack moment	ϵ_{sr1}	0.274	‰

Figure 9.20: Detailed results for state I

9.2.5.3 State II

Cross-section properties in state II

In contrast to the cross-section properties in the uncracked state (state I) determining the cross-section properties in state II (cracked sections) manually is quite difficult. Determining the strain distribution (general case: $\varepsilon_0 + (1/r)_y \cdot y + (1/r)_z \cdot z$) for a particular action combination by means of the stress-strain relations defined in the standards for non-linear methods already represents a problem. To determine the conditions of strain, the tangent method is normally used like for example the Newton-Raphson method or the modified Newton-Raphson method (tangent won't be recalculated in each single load step). For further information we refer to the corresponding literature [11].

Steel stress and strain for crack moment (σ_{srII} and ε_{srII})

To determine the stresses and strains during crack formation, we normally can start from simplified assumptions (linear elastic material rules). We can justify this approach by the fact that the relation of stress and strain for concrete is appearing nearly linearly until a stress of $\sigma_c \cong 0.4 \cdot f_c$ is reached. For the reinforcing steel we can roughly assume this fact anyway until yielding is reached. Thus, if we have a structural component with a crack moment in the characteristic load level, we can calculate stresses and strains by means of these simplified approaches with sufficient accuracy.

Without the action of an axial force, the solution for a triangular compression zone is leading 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 we get a decoupling of the neutral axis depth from the applied moment.

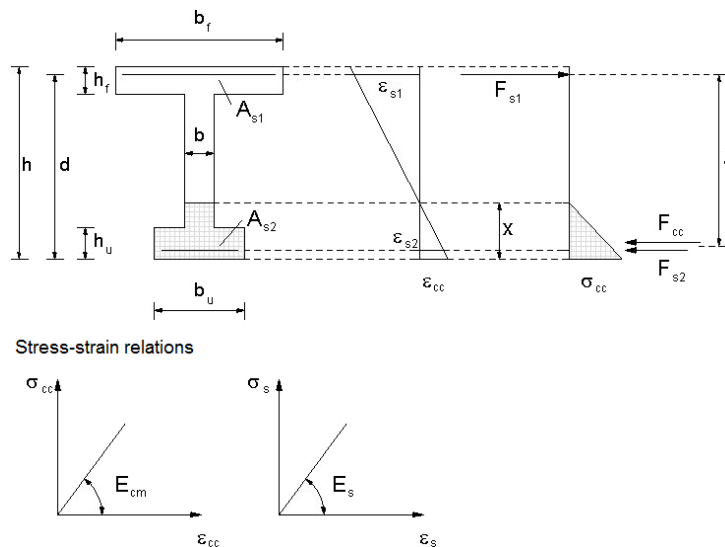


Figure 9.21: Relations for calculation of stresses and strains for characteristic loads

Calculation of neutral axis depth x

For the rectangular cross-section without loading due to axial force the derivation from the quadratic equation (without subtraction of the missing concrete areas) results in the following neutral axis depth (height of compression zone).

$$\begin{aligned}
 0 &= \frac{b}{2} \cdot x^2 + \left(\frac{2}{3} \cdot \alpha_e \cdot A_{s2} + \alpha_e \cdot A_{s1} \right) \cdot x - \alpha_e \cdot A_{s1} \cdot d \\
 &= \frac{100}{2} \cdot x^2 + \left(\frac{2}{3} \cdot 35.58 \cdot 5.93 + 35.58 \cdot 5.93 \right) \cdot x - 35.58 \cdot 5.93 \cdot 13.5 = 50 \cdot x^2 + 351.65 \cdot x - 2848.35
 \end{aligned}$$

$$x_1 = 4.81 \text{ cm}$$

$$x_2 = -11.84 \text{ cm}$$

Second moment of area

$$\begin{aligned}
 I_{y,II} &= \frac{1}{3}bx^3 + \alpha_e A_{s1}(d-x)^2 + \alpha_e A_{s2}(x-d_2)^2 \\
 &= \frac{1}{3} \cdot 100 \cdot 4.81^3 + 35.58 \cdot 5.93(13.5 - 4.81)^2 + 35.58 \cdot 5.93(4.81 - 2.5)^2 = 20768 \text{ cm}^4
 \end{aligned}$$

Stresses for crack internal forces

$$\sigma_{cr,II} = \frac{M}{I_{y,II}} \cdot x = \frac{1290}{20768} \cdot 4.81 \cdot 10 = 2.98 \text{ N/mm}^2$$

$$\sigma_{sr1,II} = \alpha_e \cdot \frac{M}{I_{y,II}} \cdot (d-x) = 35.58 \cdot \frac{1290}{20768} \cdot 8.69 \cdot 10 = 192.05 \text{ N/mm}^2$$

$$\sigma_{sr2,II} = \sigma_{sr2II} \cdot \frac{(x-d_2)}{(d-x)} = 192.05 \cdot \frac{(4.81-2.5)}{(13.5-4.81)} = 51.05 \text{ N/mm}^2$$

Steel strain for crack internal forces

$$\varepsilon_{sr1,II} = \frac{\sigma_{sr1,II}}{E_s} = \frac{192.05}{200000} \cdot 1000 = 0.96025 \%$$

Steel and concrete stress for effective moment

A simplified calculation of stresses and strains, as it was done for the crack moment, cannot be applied just like that. The stresses and strains for the effective moment $M = 17.66 \text{ kNm}$ required for the calculation of the curvatures and stiffnesses are determined in the comparative calculation with the exact stress-strain curves for concrete and reinforcing steel according to DIN 1045-1, figure 22 or 26.

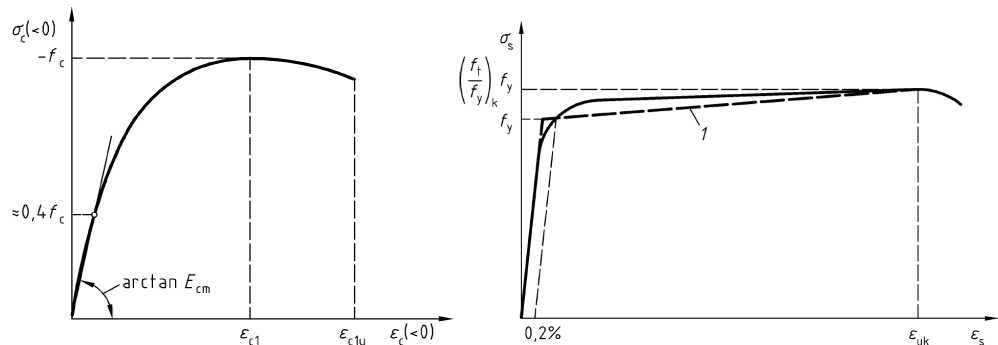


Figure 9.22: Relations for calculation of stresses and strains for characteristic loads according to DIN 1045-1

To evaluate the stress-strain relation for the concrete stresses and strains, the curvature factor α_v and the centroidal position are described by $k_a = a/x$. The iterative convergence of the state of stress and strain is performed according to the Newton-Raphson method. Because of the uniaxial loading the iteration is reduced to a two-parametric problem, this means ε_0 and $(1/r)_z$.

The curvature factor α_v and the centroidal position with $k_a = a/x$ are not specified now in detail, but descriptions can be found in the corresponding literature [25].

The accurate calculation is performed with the help of an Excel application used for stress integration. To make it clearer, we have a closer look at it with regard to an iteration step.

The figure below shows the definition of the strain ϵ_0 / the curvature $(1/r)_z$ and the stress-strain curves that are applied.

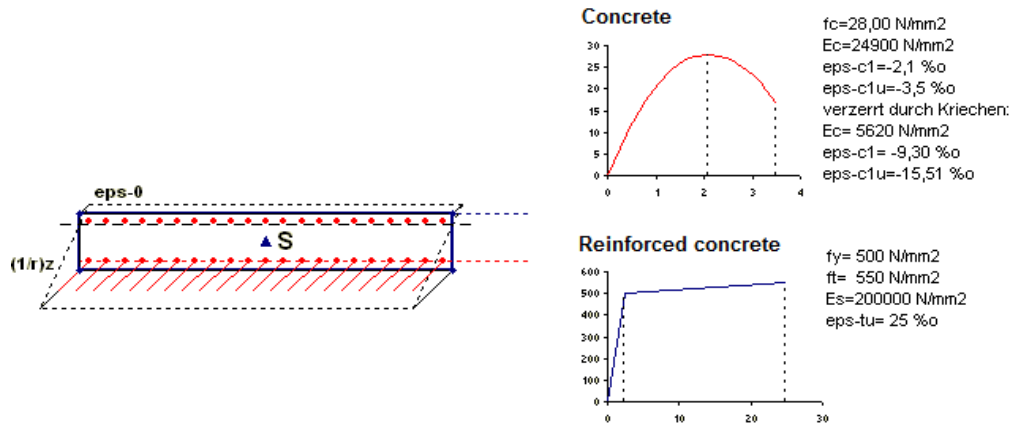


Figure 9.23: Applied material and strain parameters

Initial value used to calculate plane of strain

First, we find an appropriate initial value for the calculation. There are hardly any restrictions for the selection of an appropriate state of initial strain. However, we should avoid ultimate strain conditions in zones where yielding and failure occurs because without corresponding intervention (damping or something similar) coefficients of the Jacobian matrix may lead to a poorly performed or even no convergence.

For our example we apply the following state of initial strain:

$$\begin{aligned} \epsilon_{0,1} &= -0.3 \text{ ‰} & (1/r)_{z,1} &= 5.00E-3 \text{ 1/m} \\ M_y &= 7.016 \text{ kNm} & N &= -31.48 \text{ kN} \end{aligned}$$

Difference quotients of matrix

Instead of differential quotients we use difference quotients converging the tangential stiffness very well when a sufficiently small increment is selected. In this way, the condition of strain should be represented more precisely.

$$\begin{bmatrix} \frac{dN}{d\epsilon_0} & \frac{dN}{d(1/r)_z} \\ \frac{dM_y}{d\epsilon_0} & \frac{dM_y}{d(1/r)_z} \end{bmatrix} \cdot \begin{bmatrix} d\epsilon_0 \\ d(1/r)_z \end{bmatrix} = \begin{bmatrix} dN \\ dM_y \end{bmatrix}$$

With $d\epsilon_0 = -0.01 \text{ ‰}$ (because $\epsilon_{s0} < 0$) results in:

$$\begin{aligned} \frac{dN}{d\epsilon_0} &= \frac{(-37.35 - (-31.27))}{-0.01} = 607.86 \\ \frac{dM_y}{d\epsilon_0} &= \frac{(7.198 - 7.016)}{-0.01} = -18.86 \end{aligned}$$

With $d(1/r)_z = 0.01 \text{ ‰ / m}$ (because $(1/r)_z > 0$) results in:

$$\begin{aligned} \frac{dN}{d(1/r)_z} &= \frac{(-30.97 - (-31.27))}{0.01} = 30 \\ \frac{dM_y}{d(1/r)_z} &= \frac{(7.0187 - 7.016)}{0.01} = 0.27 \end{aligned}$$

So the system of equations to be solved is the following:

$$\begin{bmatrix} 607.86 & 30 \\ -18.86 & 0.27 \end{bmatrix} \cdot \begin{bmatrix} d\varepsilon_0 \\ d(1/r)_z \end{bmatrix} = \begin{bmatrix} 0 - (-31.27) \\ 17.63 - 7.016 \end{bmatrix}$$

$$\begin{bmatrix} 3.787E-4 & -4.2231E-2 \\ 2.5902E-2 & 8.6374E-1 \end{bmatrix} \cdot \begin{bmatrix} 31.27 \\ 10.614 \end{bmatrix} = \begin{bmatrix} d\varepsilon_0 \\ d(1/r)_z \end{bmatrix} = \begin{bmatrix} -4.364E-1 \\ 9.97769 \end{bmatrix}$$

First calculation of a new state of strain:

$$\begin{bmatrix} \varepsilon_0 \\ (1/r)_z \end{bmatrix}_1 = \begin{bmatrix} \varepsilon_0 \\ (1/r)_z \end{bmatrix}_0 + \begin{bmatrix} d\varepsilon_0 \\ d(1/r)_z \end{bmatrix} = \begin{bmatrix} -0.3 \\ 5 \end{bmatrix} + \begin{bmatrix} -0.4363 \\ 9.97769 \end{bmatrix} = \begin{bmatrix} -0.7363 \\ 14.97769 \end{bmatrix}$$

With this state of strain the following internal forces are the result:

$$\begin{bmatrix} N \\ M_y \end{bmatrix}_1 = \begin{bmatrix} 0.81 \\ 18.044 \end{bmatrix}$$

Additional enhancement of strain condition

In an analogue way, the condition of strain is to be enhanced further. According to needs, it is possible to keep the matrix of the difference quotients (modified Newton-Raphson method) or to calculate it again in each step. The calculational advantage represented by the fact that the coefficients of the Jacobian matrix do not need to be recalculated in each step is "compensated" in the modified Newton-Raphson method by a higher number of iterations. This can also be seen in the progress of the iteration process shown below.

Nr.	$\Delta\varepsilon_0$ [%o]	$\Delta(1/r)_z$ [mm/m]	ε_0 [%o]	$(1/r)_z$ [mm/m]	N [kN]	My [kNm]	ΔN [kN]	ΔMy [kNm]
1	-0,436396	9,9776976	-0,736396	14,977698	0,8137791	18,044738	-0,813779	-0,414738
2	0,0166102	-0,366692	-0,719785	14,611006	-0,03207	17,630852	0,0320702	-0,000852
3	5,542E-05	0,0001007	-0,71973	14,611106	-0,000108	17,630003	0,0001077	-3,01E-06
4	1,922E-07	2,135E-07	-0,71973	14,611107	-3,55E-07	17,63	3,552E-07	1,402E-07
5	-5,56E-09	1,259E-07	-0,71973	14,611107	1,053E-08	17,63	-1,05E-08	-3,34E-07

Figure 9.24: Calculation according to large deformation analysis (Newton-Raphson)

Nr.	$\Delta\varepsilon_0$ [%o]	$\Delta(1/r)_z$ [mm/m]	ε_0 [%o]	$(1/r)_z$ [mm/m]	N [kN]	My [kNm]	ΔN [kN]	ΔMy [kNm]
1	-0,436396	9,9776976	-0,736396	14,977698	0,8137791	18,044738	-0,813779	-0,414738
2	0,0172065	-0,379305	-0,719189	14,598393	-0,047881	17,616211	0,0478809	0,013789
3	-0,000564	0,0131503	-0,719753	14,611543	-0,000928	17,630552	0,0009284	-0,000552
4	2,364E-05	-0,000452	-0,71973	14,611091	-0,000302	17,62999	0,000302	9,672E-06
5	-2,94E-07	1,618E-05	-0,71973	14,611107	-3,27E-05	17,630001	3,268E-05	-1,44E-06
6	7,307E-08	-3,95E-07	-0,71973	14,611107	-4,39E-06	17,63	4,391E-06	-1,7E-07

Figure 9.25: Calculation according to the modified Newton-Raphson method

In the modified Newton-Raphson method the coefficient matrix is calculated only in the first step. This procedure proves to be very efficient for our example as the result is already lying near the expected solution after the first iteration step. To achieve nearly equal break-off tolerances, only one more step is required. In this way, we can save time because the matrix of coefficients has to be calculated only once.

Decisive plane of strain

$$\epsilon_z = \epsilon_0 + (1/r)_z \cdot z = -0.729557 + 14.8042 \cdot z$$

Thus, the steel strain is calculated as follows:

$$\epsilon_{z,s1} = \epsilon_0 + (1/r)_z \cdot z_{s1} = -0.729557 + 14.8042 \cdot 0.135 = 1.269 \text{ ‰}$$

Eingabe:			
Schätzungen:			
$\epsilon_{c2}=\epsilon_0=$	-0,735 ‰	$\geq -15,31 \text{ ‰}$	der Betondruckspannung
$\epsilon_{s1}=$	1,279 ‰	$\leq 25 \text{ ‰}$	der Stahlspannung
Festwerteingabe:			
Beton C 20/25			
$\epsilon_{c1}=$	-9,300 ‰	<i>(-) Dehnung bei Erreichen der Spannung f_c</i>	
$\epsilon_{c1u}=$	-15,310 ‰	<i>(-) Dehnung bei Betonversagen</i>	
$f_c=$	28,00 N/mm²	<i>(+) Höchstwert der ertragenen Beton druck spannungen)</i>	
$E_{cm}=$	5620,76 N/mm²		
$b=$	100 cm	<i>(Breite)</i>	
$h=$	16 cm	<i>(Höhe)</i>	
$d=$	13,5 cm	<i>(statische Höhe)</i>	
BSt 500			
$E_s=$	200000 N/mm²		
$A_{s1}=$	5,94 cm²	$A_{s2}=$	5,94 cm² <i>(Druckbewehrung)</i>
$f_y=$	500 N/mm²	$d_2=$	2,5 cm
$f_t=$	525 N/mm² <i>(bei einer Grenzdehnung</i>		25 ‰
Auswertung:			
$k=$	2,05358481	$-1,1E_{cm} \cdot \epsilon_{c1} / f_c$	Spannungs-Dehnungs-Beziehung Beton (DIN 1045-1 nichtlinearer Ansatz)
$p=$	0,07908433	$\epsilon_{c2} / \epsilon_{c1}$	
$\alpha_v=$	-0,07890	<i>Volligkeitsbeiwert</i>	
$K_a=$	0,33577	$K_a = a/x$	
$x=$	4,93 cm	<i>Intern aus Dehnungen ermittelt</i>	
$a=$	1,65 cm		
$z=$	11,85 cm		
$\sigma_s=$	255,839192 N/mm²		
$F_{s1}=$	151,91 kN	<i>(resultierend aus Stahldehnung und Stahlquerschnitt)</i>	
$F_{s2}=$	-43,04 kN	$\epsilon_{s2}=$	-0,362 ‰
$F_c=$	-108,87 kN	$\sigma_{s2}=$	-72,479 N/mm²
$F_{s1}+F_{s2}+F_c=$	0,00 kN	<i>(Bei guter Näherung sollte sich ~ 0 einstellen)</i>	
$M_{rd}=$	17,6300 kNm		
$(1/r)_z=$	0,01492356 1/m		

Figure 9.26: Iteratively determined condition of strain, excerpt from German verification

Calculation of ultimate moment

The ultimate moment for a calculation in the serviceability limit state represents the safety considering the average values of the material properties. The safety against failure in the ultimate limit state is not designed with this calculation.

Differing from the determination of the state of strain we can calculate the state of failure by varying a strain parameter. We analyze the limit of failure separately for the concrete compression zone and the steel fibre. When applying the ultimate strains we have to consider that they are distorted with regard to creeping. Therefore, the ultimate strains according to the stretched stress-strain curve are to be used.

With the calculation it is only possible to design the state of equilibrium for steel failure. All specifications required to calculate the moment of rupture can be seen in the figure below.

Eingabe:			
Schätzungen:			
$\epsilon_{c2} = \epsilon_0 =$	5,513 ‰	$\geq -15,31 \text{ ‰}$	der Betondruckspannung
$\epsilon_{s1} =$	25,000 ‰	$\leq 25 \text{ ‰}$	der Stahlspannung
Festwerteingabe:			
Beton C 20/25			
$\epsilon_{c1} =$	-9,300 ‰	(•) Dehnung bei Erreichen der Spannung f_c	
$\epsilon_{c1u} =$	-15,310 ‰	(•) Dehnung bei Betonversagen	
$f_c =$	28,00 N/mm ²	(•) Höchstwert der ertragenen Beton druck spannungen)	
$E_{cm} =$	5620,76 N/mm ²		
$b =$	100 cm	(Breite)	
$h =$	16 cm	(Höhe)	
$d =$	13,5 cm	(statische Höhe)	
BSt 500			
$E_s =$	200000 N/mm ²		
$A_{s1} =$	5,94 cm ²	$A_{s2} =$	5,94 cm ² (Druckbewehrung)
$f_y =$	500 N/mm ²	$d_2 =$	2,5 cm
$f_t =$	525 N/mm ² (bei einer Grenzdehnung		25 ‰
Auswertung:			
$k =$	2,05358481	$-1,1 E_{cm} \epsilon_{c1} / f_c$	
$\rho =$	0,59281242	$\epsilon_{c2} / \epsilon_{c1}$	
$\alpha_v =$	-0,48168	Volligkeitsbeiwert	
$K_a =$	0,35500	$K_a = a/x$	
$x =$	2,44 cm	intern aus Dehnungen ermittelt	
$a =$	0,87 cm		
$z =$	12,63 cm		
$\sigma_s =$	525 N/mm ²		
$F_{s1} =$	311,72 kN	(resultierend aus Stahldehnung und Stahlquerschnitt)	
$F_{s2} =$	16,32 kN	$\epsilon_{s2} =$	0,137 ‰
$F_c =$	-328,98 kN	$\sigma_{s2} =$	27,486 N/mm ²
$F_{s1} + F_{s2} + F_c =$	-0,93 kN	(Bei guter Näherung sollte sich ~ 0 einstellen)	
$M_{rd} =$	39,7682 kNm		
$(1/r)_z =$	0,22602337 1/m		

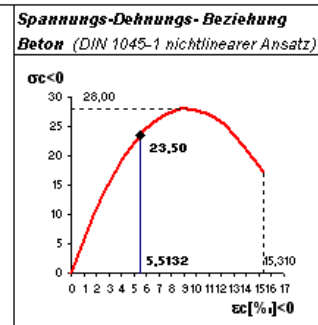


Figure 9.27: Calculation of ultimate moment, excerpt from German verification

CONCRETE results

Detailed Results - Member No. 1, x: 2.210, LG2			
Zustand II			
Cross-section properties			
— Moment of inertia about the y-axis	I_{yII}	48497500.0	mm ⁴
— Moment of inertia about the z-axis	I_{zII}	1.33333E+10	mm ⁴
— Surface	A_{II}	22733.2	mm ²
Stress and Strain Analysis			
— main curvature	$1/r_{II}$	1.4620E-02	1/m
— curvature in y-direction	$1/r_{yII}$	1.1061E-09	1/m
— curvature in z-direction	$1/r_{zII}$	1.4620E-02	1/m
— Direction of the principal curvature	α_{II}	0.0	°
— Steel strain in governing fiber	σ_{sII}	253.64	N/mm ²
— Steel strain	ϵ_{sII}	1.268	‰
— Steel strain in governing fiber	σ_{cII}	-4.27	N/mm ²
— Steel strain in governing fiber	ϵ_{cII}	-0.720	‰
— steel stress for crack moment	σ_{srII}	185.61	N/mm ²
— steel strain for the crack moment	ϵ_{srII}	0.928	‰
Cross-section properties			
Cross-sectional rigidities (Secant Values)			
Global Node Deformations			
Local Node Deformations			
Crack Width			
Verification of Safety			
— Ultimate internal forces R_u	M_u	41.54	kNm
— Ultimate Moment about Axis y	M_{y_u}	41.54	kNm

Figure 9.28: Detailed results for state II

9.2.5.4 Mean Curvatures

The average curvatures arising with the selected approach for Tension-Stiffening are determined from the calculations for pure state I and pure state II.

The Tension Stiffening model described in book 525 [13] which is taken as a basis takes into account the concrete's tension stiffening effect occurring between the cracks by a reduction of the steel strain. The required parameters are determined as follows.

Decisive state of cracking

Steel stress in state II in case of crack formation: $\sigma_{sr1,II} = 192.05 \text{ N/mm}^2$

Steel stress in state II: $\sigma_{s1,II} = 255.8 \text{ N/mm}^2$

$$\sigma_{s1,II} = 255.8 \text{ N/mm}^2 \geq 1.3 \cdot \sigma_{sr1,II} = 249.67 \text{ N/mm}^2$$

Thus, we have a closer look at the final crack state.

Mean steel strain

$$\varepsilon_{sm} = \varepsilon_{s2,II} - \beta_t (\varepsilon_{sr1,II} - \varepsilon_{sr1,I})$$

$$\varepsilon_{sm} = 1.279 - 0.306(0.96025 - 0.2687) = 1.067 \text{ ‰}$$

where

$$\varepsilon_{s2,II} = 1.279 \text{ ‰}$$

Steel strain in state II

$$\varepsilon_{sr1,II} = 0.96025 \text{ ‰}$$

Steel strain for crack internal force in state II

$$\varepsilon_{sr1,I} = 0.2687 \text{ ‰}$$

Steel strain for crack internal force in state I

$$\beta_t = 0.306$$

Load duration factor of available action

Mean curvature

$$\left(\frac{1}{r}\right)_{z,m} = \frac{(\varepsilon_{sm} - \varepsilon_c)}{d} = \frac{(1.067 + 0.735)}{0.135} = 13.35 \frac{\text{mm}}{\text{m}} = 1.335e-2 \frac{1}{\text{m}}$$

Mean flexural resistance

From the mean curvature $(1/r)_{z,m}$ together with the relation

$$\left(\frac{1}{r}\right)_{z,m} = \frac{M}{I_{y,m} \cdot E}$$

arises the secant stiffening in the corresponding node.

$$I_{y,m} \cdot E = \frac{M_y}{\left(\frac{1}{r}\right)_{z,m}} = \frac{0.01766}{1.335e-2} = 1.3228 \text{ MNm}^2 = 1322.80 \text{ kNm}^2$$

where

$$M_y = 17.66 \text{ kNm}$$

Provided moment

$$\left(\frac{1}{r}\right)_{z,m} = 1.335e-2 \frac{1}{\text{m}}$$

Steel strain for crack internal force in state II

CONCRETE results

Detailed Results - Member No. 1, x: 2.210, LG2				
Stress and Strain Analysis				
Principal moments	M	17.66	kNm	
bending moment over the y-axis	My	17.66	kNm	
bending moment over the z-axis	Mz	0.00	kNm	
Normal force	N	0.00	kN	
State of cross-section				
Mean Main Curvature	1/r _m	1.3147E-02	1/m	Crack Formation Completed
Middle curvature in y-direction	1/r _{ym}	0.0000E+00	1/m	
Mean curvature in z-direction	1/r _{zm}	1.3872E-02	1/m	
Average Strain in Reinforcement	ε _{sm}	1.068	‰	
Design Details				
Cross-section properties				
Cross-sectional rigidities (Secant Values)				
Strain resistance	A _m * E	574.36	MN	
Mean Bending Rigidity about y-Axis	I _{ym} * E	1.34	MNm ²	
Mean Bending Rigidity about z-Axis	I _{zm} * E	94.12	MNm ²	
Shear rigidity in y-direction	A _{ym} * E	1394.40	MN	
Shear rigidity in z-direction	A _{zm} * E	1394.40	MN	
Torsional Rigidity	I _t * G	3.99	MNm ²	

Figure 9.29: Detailed results of mean curvatures

9.2.6 Results Evaluation

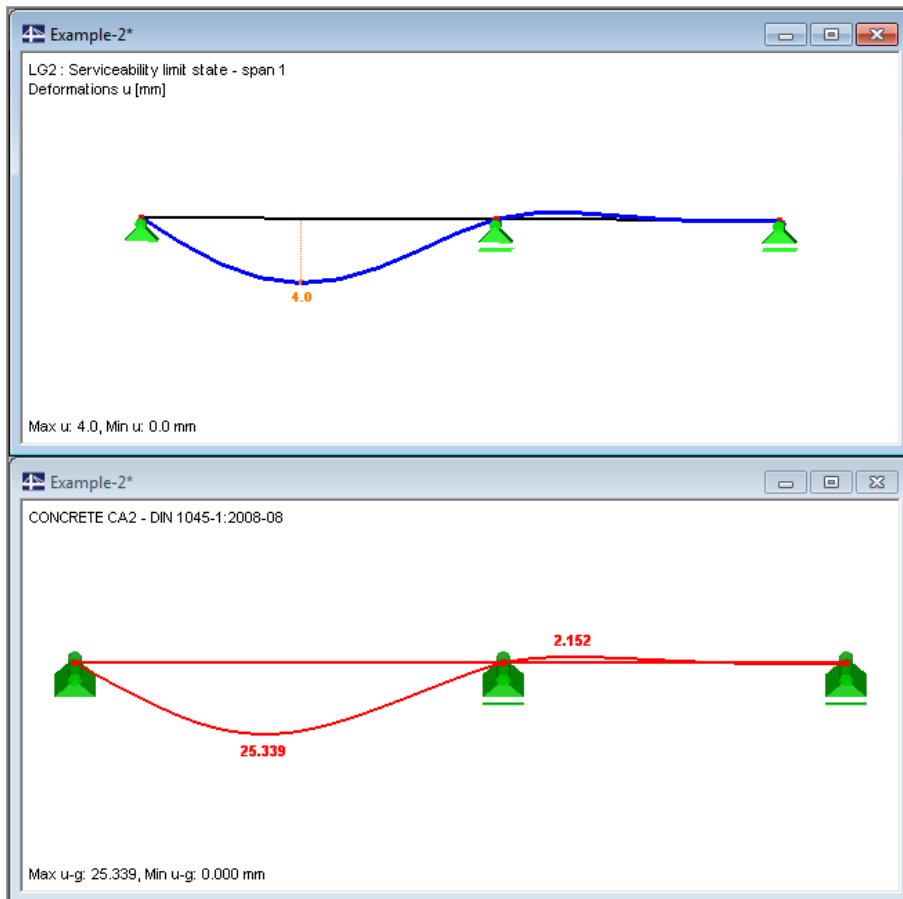


Figure 9.30: Deformations according to linear elastic calculation (above) and non-linear calculation with creep (below)

The deformation from the non-linear calculation where the creep effect is also taken into account proves to be significantly higher than the deformation from the pure linear elastic calculation without creep effect. As described on page 151, the calculated deformation exceeds the recommended limit value of $l/250$.

The deformations in state II are affected considerably 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. A comparison with DIN V ENV 1992-1-1:1992-06 appears to be interesting because there the floor thickness is determined with $h \geq 18$ cm by considering the same boundary conditions. It's a moot question whether this value is necessarily to be called too conservative.

Increasing the floor thickness to $h = 18$ cm is a possibility to reduce the deformation considerably ($u_{max} \leq 16$ mm for an accordingly little reinforcement and $\varphi_{eff} = 3.40$).

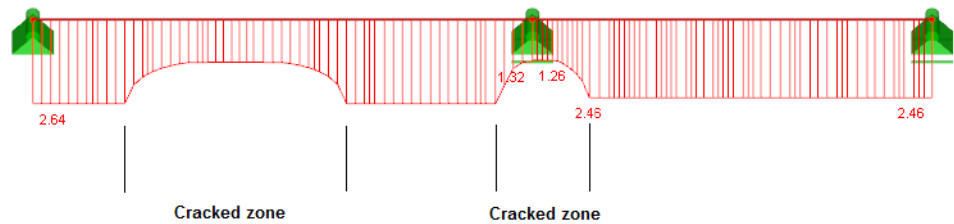
Creep

The creep coefficient assumed with $\varphi_{\infty} = 3.43$ appears to be relatively high but meets the requirements according to DIN 1045-1, figure 18, with the assumed environmental conditions and the cross-section geometry.

By means of the ψ factor ($\psi_{2,1} = 0.6$) used to calculate the quasi-permanent action combination it would be possible to effect some reduction from creep-producing to acting load.

Concrete tensile strength

The diagram of stiffnesses shows us that a large area in span 1 is cracked in the serviceability limit state.



Max $I_{y,m} * E$: 2.64, Min $I_{y,m} * E$: 1.26 MNm²

Figure 9.31: Stiffness diagram $I_{y,m} * E$ along the beam length

For the concrete tensile strength the value f_{ctm} (axial tensile strength) according to DIN 1045-1, table 9, was assumed in the calculation.

Effects such as the gradient of the stresses have a great influence on the concrete's effective tensile strength. For example, a large stress gradient increases the tensile strength because the corresponding high stresses are acting only in very few fibres. More information about the different influencing factors acting on the tensile strength can be found in [23].

For our example the tensile strength is calculated again according to [23], 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/mm², which is different to DIN 1045-1.

$$C_V = 0.85 - 0.2 \cdot 0.16 = 0.818 \geq 0.65$$

Pre-damage of structural component is taken into account.

$$C_h = \frac{2.6 + 24 \cdot 0.16}{1.0 + 40 \cdot 0.16} = 0.870$$

Influence of structural component thickness

$$C_{\eta} = 1$$

Influence of eccentricity $\eta = M/(N \cdot h) \rightarrow \infty$ for $N \rightarrow 0$

To consider the influence of an increased tensile strength, the model is calculated with the adjustment factor $3.14 / 2.2 = 1.42$.

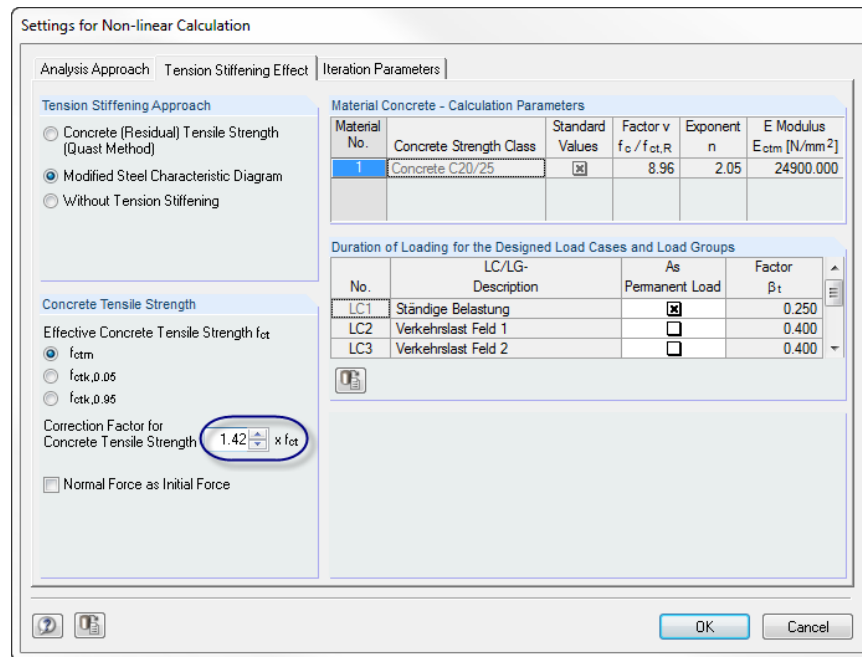


Figure 9.32: Dialog box *Settings for Non-linear Calculation*, tab *Tension Stiffening Effect*

The calculation shows us a strong reduction of the cracked zones that leads also to a reduction of the deformation to $u_1 = 15.27$. This value is lying clearly below the reference value of $l/250 = 5/250 = 20$ mm. The following figure illustrates the relation between deformation and stiffness reduction. The cross-section is passing locally into the cracked state only in the column's support area.

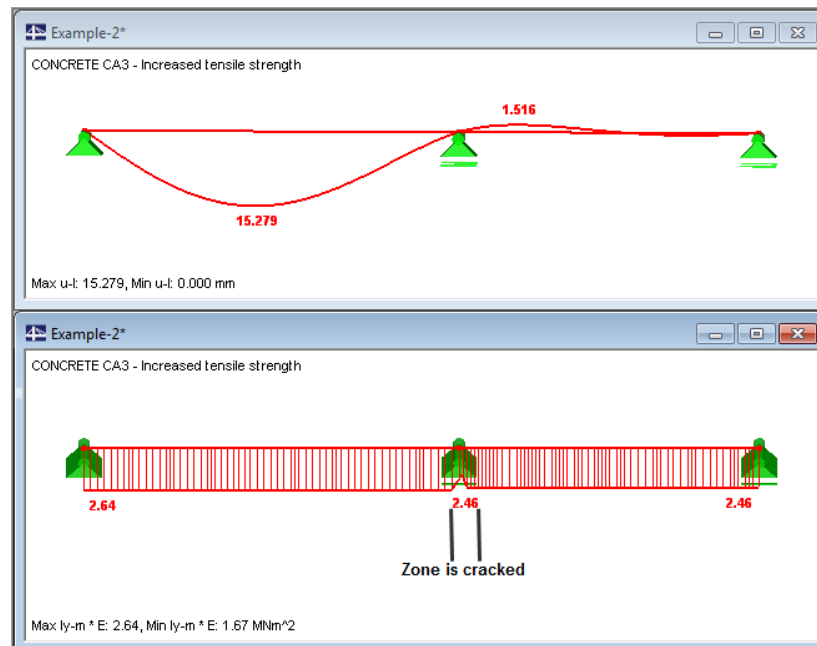


Figure 9.33: Deformation and flexural resistance for calculation with increased concrete tensile strength (stress gradient)

We have seen how sensitively the non-linear calculation is reacting when calculation parameters are modified. The difference is especially significant in structural components with large stiffness changes between cracked and uncracked state.

9.3 Stability Analysis for Bracket

9.3.1 Overview and Input in RSTAB

By describing the stability analysis of a restrained column we look at the basic differences of both approaches for the non-linear calculation according to EN 1992-1-1, 5.7 and 5.8.6.

The bracket model is presented as example 1 in [20].

System and loads

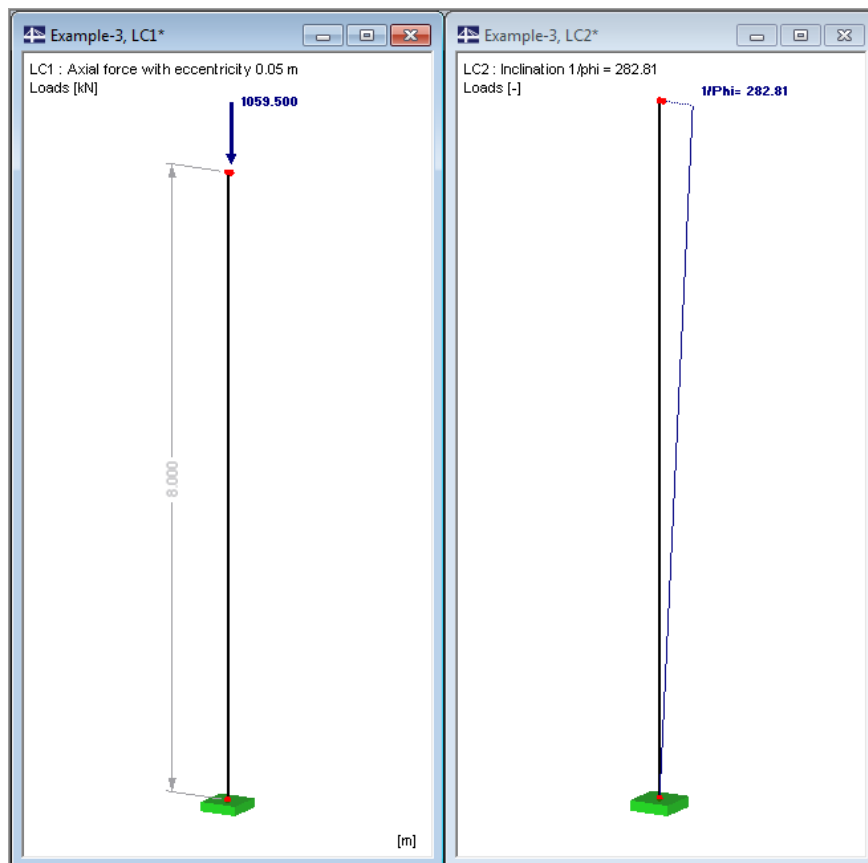


Figure 9.34: System and loads

The loading corresponds to the specifications in [20]. The design value $N_{Ed} = 1059.5$ kN is already taken into account in load case 1. As shown in the figure above the loading is entered eccentrically. The eccentricity can be determined geometrically or by an additional moment $M_{sd} = 1059.5 * 0.05 = 52.98$ kNm. In our example the load is introduced eccentrically by means of a short member.

The inclination of the system is considered as imperfection in load case 2 and it is calculated with $1/\varphi = 1/0.003536 = 282.81$.

Several load groups are defined to represent the load-deformation behavior of slender columns consisting of reinforced concrete.

Load groups

Design-relevant load arrangement

LG 1 LC1 + LC2

Intermediate steps

LG 2 0.90*LC1 + LC2

LG 3 1.05*LC1 + LC2

LG 4 1.10*LC1 + LC2

The stiffness is not reduced by the partial safety factor γ_M for the LG calculation parameters.

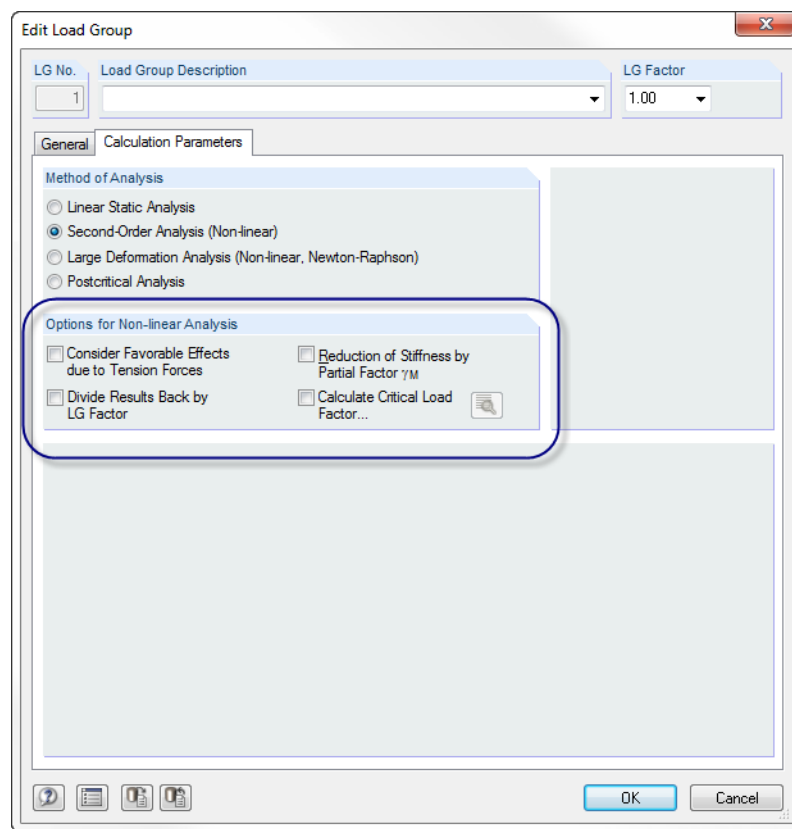


Figure 9.35: Calculation parameters of load groups

Before we perform the non-linear calculation of the column, we calculate all load groups with RSTAB. The following internal forces and deformations are the result:

Load Group	Axial Force N [kNm]	Moment Linear Static Analysis M _I [kNm]	Moment II. Order Analysis M _{II} [kNm]	Column Head Deformation u [mm]
LG 1	-1059.50	82.94	155.45	68.48
LG 2	-953.55		128.99	57.03
LG 3	-1112.47		170.39	74.93
LG 4	-1165.45		186.67	81.94

Table 9.1: Summary of RSTAB results output

9.3.2 Non-linear Calculation of Column

The column is designed with both methods according to EN 1992-1-1, 5.7 and 5.8.6.

9.3.2.1 Design according to EN 1992-1-1, 5.7

The first design case performs the analysis according to the holistic concept of the European standard EC 2.

Input in CONCRETE

The basic input is shown in the figures below.

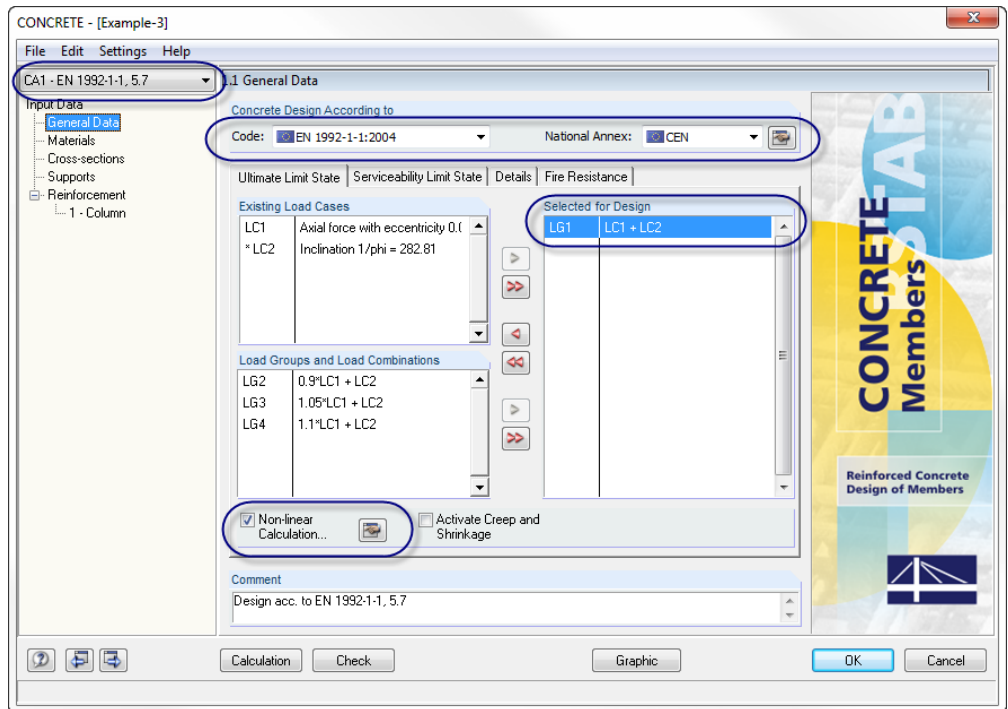


Figure 9.36: General data for non-linear calculation according to EN 1992-1-1, 5.7

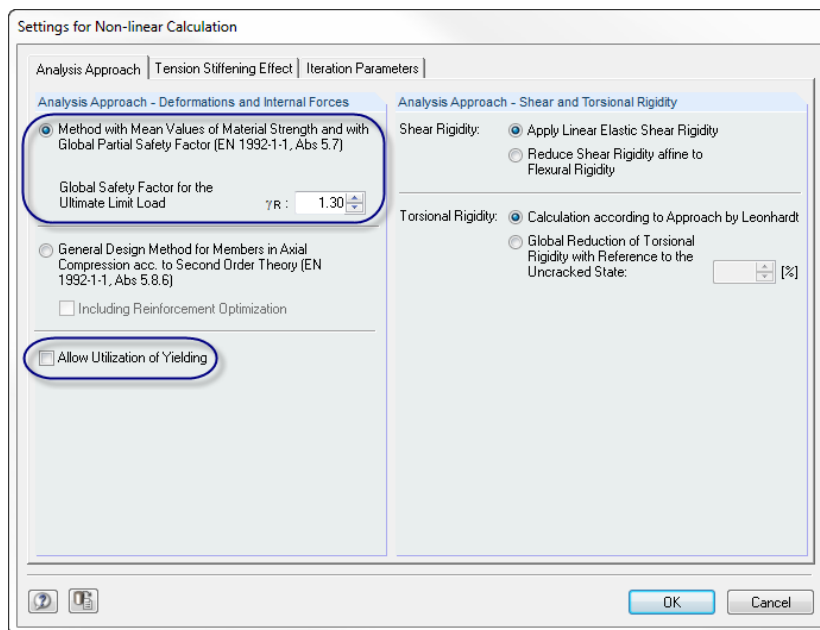
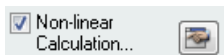


Figure 9.37: Analysis approach according to EN 1992-1-1, 5.7

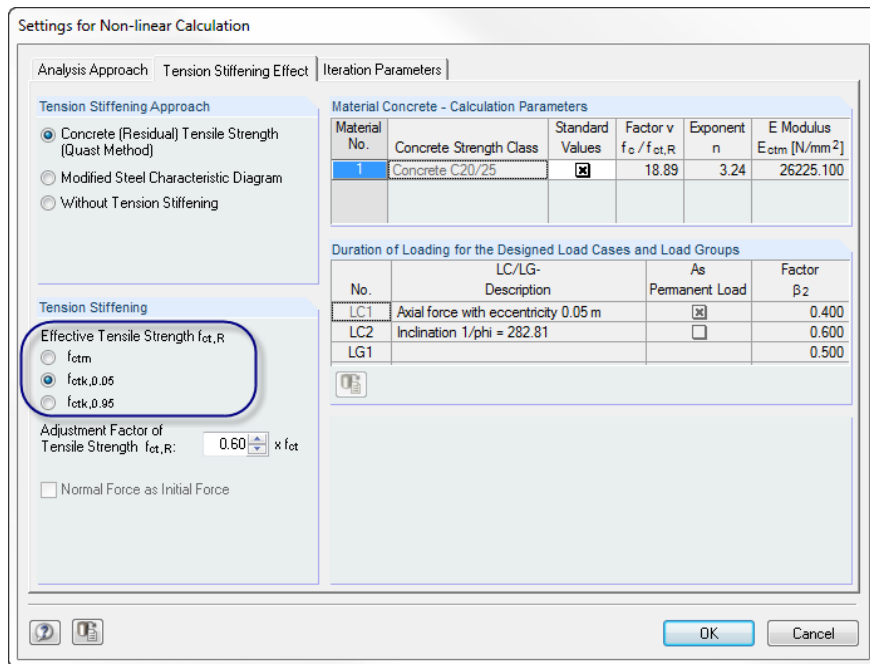


Figure 9.38: Effective tension stress of concrete for Tension Stiffening

To achieve results comparable to the calculation in [20], we have to modify the *Tension-Stiffening* model according to QUAST as shown above. As the calculation of the allowable compression stress f_{cR} is based on a low quantile, the value $f_{ctk,0.05}$ is used also for the determination of the allowable concrete tension stress.

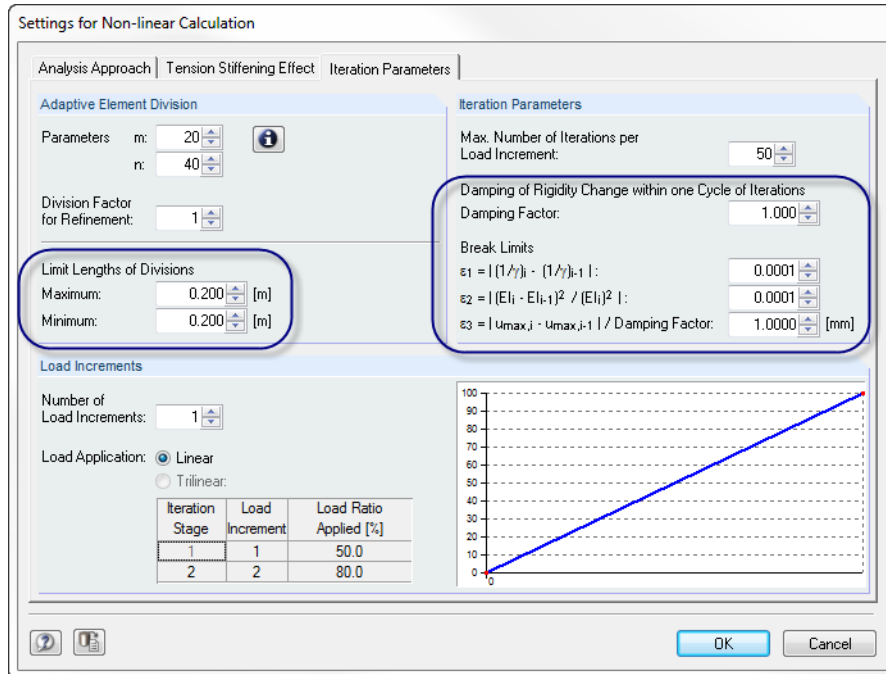


Figure 9.39: Limit lengths of adaptive member division and break-off limits

As our structure is a statically determinate system, we can keep the damping factor set to 1.0. In order to represent the distribution of stiffnesses accurately, we limit the maximum member length to 0.20 m.

We have to note the following for the non-linear calculation of structures prone to instability risks concerning the selection of appropriate break-off limits ϵ_1 and ϵ_2 : Though a calcula-

tion is converging continuously according to the linear static analysis, it is possible that compression elements may see a "reversal point" where deviations ε will increase again. The described effect occurs when the system cannot compensate or absorb anymore the increase of internal forces caused by the increase of the deformation according to the second-order analysis with the reduced stiffnesses. The example specifies $\varepsilon_1 = \varepsilon_2 = 0.0001$.

In [20] a required reinforcement of $A_{s,tot} = 66.10 \text{ cm}^2$ is determined by means of the similar design method according to DIN 1045-1, 8.5. In order to compare these results with the CONCRETE 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 table 1.6 *Reinforcement*. The diameter is defined with 25 mm in the tab *Longitudinal Reinforcement*.

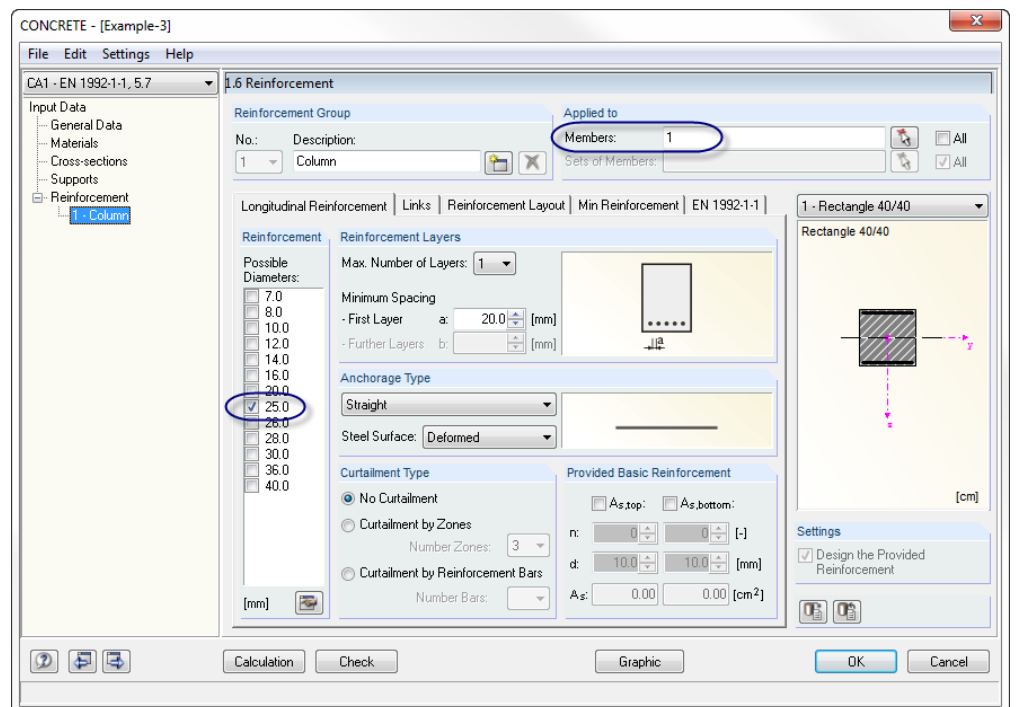


Figure 9.40: Specification of rebar diameter

The concrete cover is selected with $c_{nom} = 27.5 \text{ mm}$ to have a center distance of 40 mm for the distance from the center to the edge.

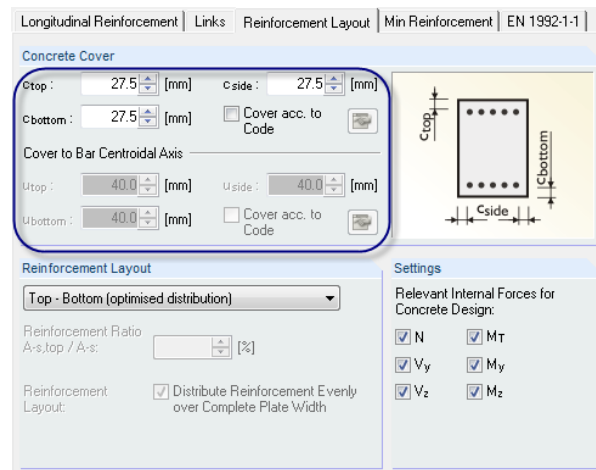


Figure 9.41: Settings for concrete cover

In order to perform the design with the specified reinforcement from [20], a minimum reinforcement of $A_{s,top} = A_{s,bottom} = 32 \text{ cm}^2$ is defined.

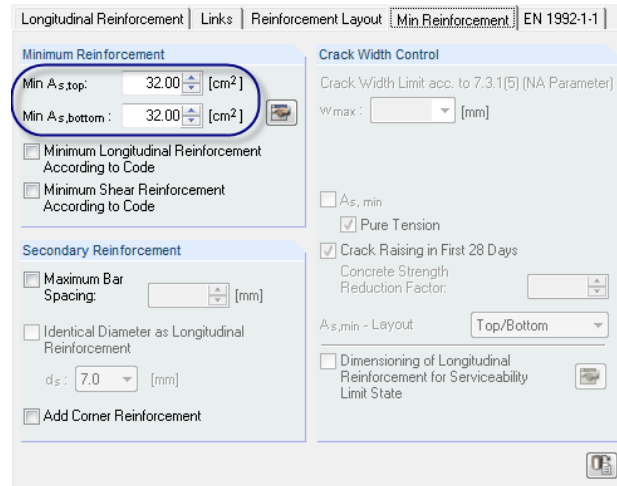


Figure 9.42: Specification of minimum reinforcement

Now the input is complete and we can start the [Calculation].

Results of non-linear calculation

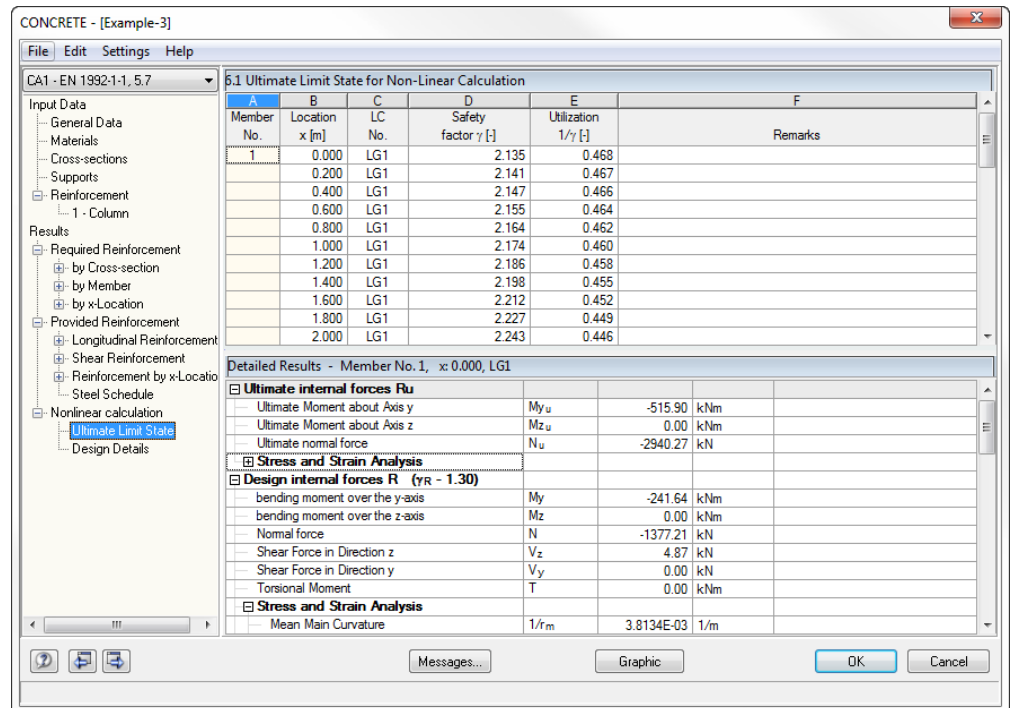


Figure 9.43: Table 6.1 Ultimate Limit State for Non-Linear Calculation

Interpreting results is described in detail in chapter 9.2.5, page 151.

For this example we want to demonstrate that another load increment will lead to the system's instability. We select LG 4 which is stable for a calculation according to the physically linear second-order analysis in table 1.1 General Data.

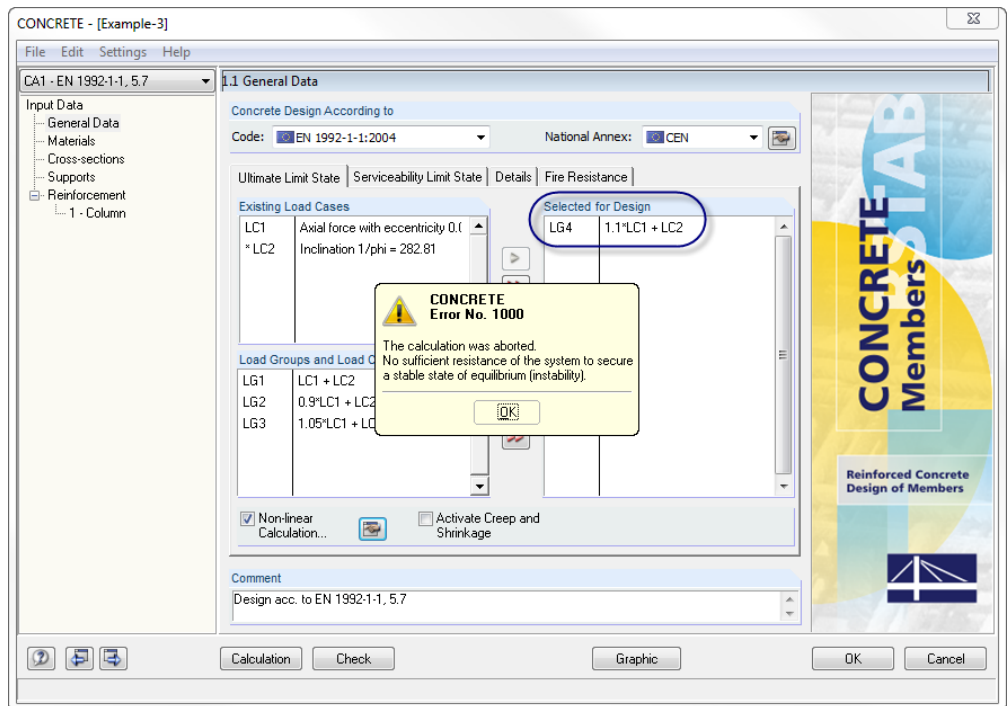


Figure 9.44: Break-off of calculation for LG 4 because of instability

The 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 when a load increment of 10 % is applied.

The following column calculation according to EN 1992-1-1, 5.8.6 will show that the column fails before the cross-section resistance is reached.

9.3.2.2 Design according to EN 1992-1-1, 5.6.8

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.

Input in CONCRETE

In order to compare the results, a new concrete case is created for EN 1992-1-1, 5.6.8. As we need to change only few input data, we simply copy the first design case by using the CONCRETE menu. To open the corresponding dialog box,

we select **Copy Case** on the **File** menu in the CONCRETE add-on module.

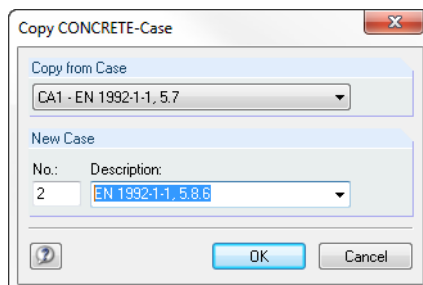
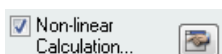


Figure 9.45: Copying the first design case

In table 1.1 *General Data* we adjust the comment accordingly.

We use the [Edit] button to open the dialog box *Settings for Non-linear Calculation* and adjust the conditions as follows:



We select the general method as the analysis approach for members with compression according to the second-order analysis. Plastic releases (plastic curvatures) are again excluded.

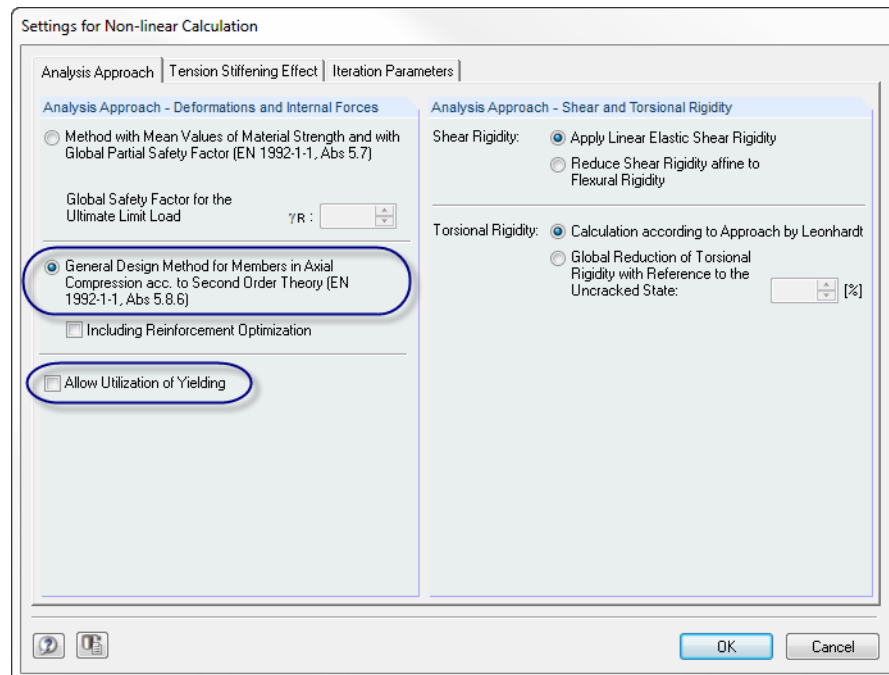


Figure 9.46: Analysis approach according to EN 1992-1-1, 5.8.6

The non-linear 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, page 48). Therefore, we have to calculate with the average values of the material properties also for the approach of Tension Stiffening. The partial safety factor γ_c flows directly into the applied tensile strength: $f_{ct,R} = f_{ct} / \gamma_c$. This also applies to the concrete's modulus of elasticity.

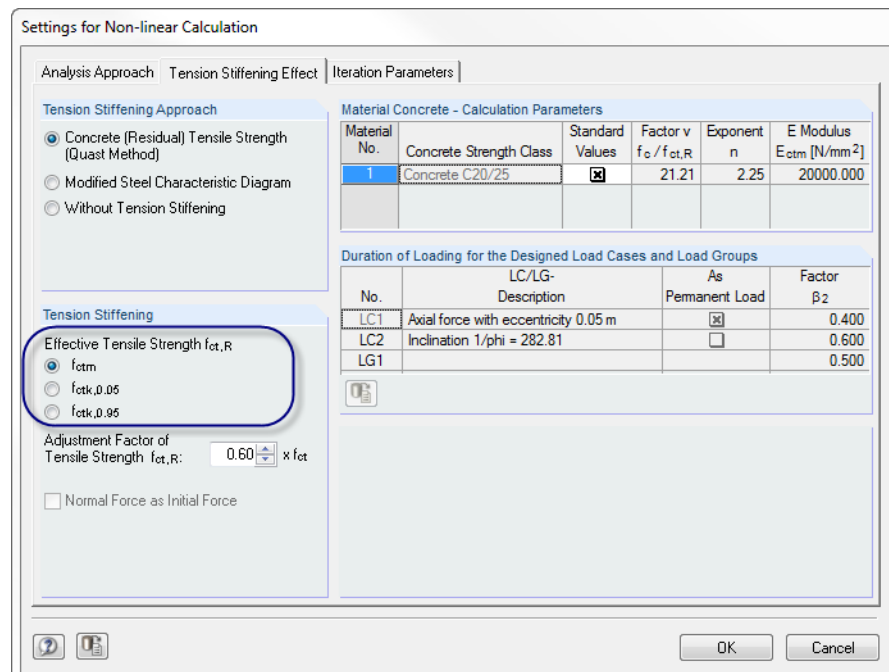


Figure 9.47: Effective tension stress of concrete for Tension Stiffening

The parameters of the table tab *Iteration Parameters* remain unchanged.

In [20] a required reinforcement of $A_{s,tot} = 40.0 \text{ cm}^2$ is determined by means of the similar design method according to DIN 1045-1, 8.6.1. In order to compare these results with the CONCRETE calculation performed according to EN 1992-1-1, 5.8.6, we define a minimum reinforcement of $A_{s,top} = A_{s,bottom} = 20 \text{ cm}^2$ in the tab *Minimum Reinforcement* of table 1.6 *Reinforcement*.

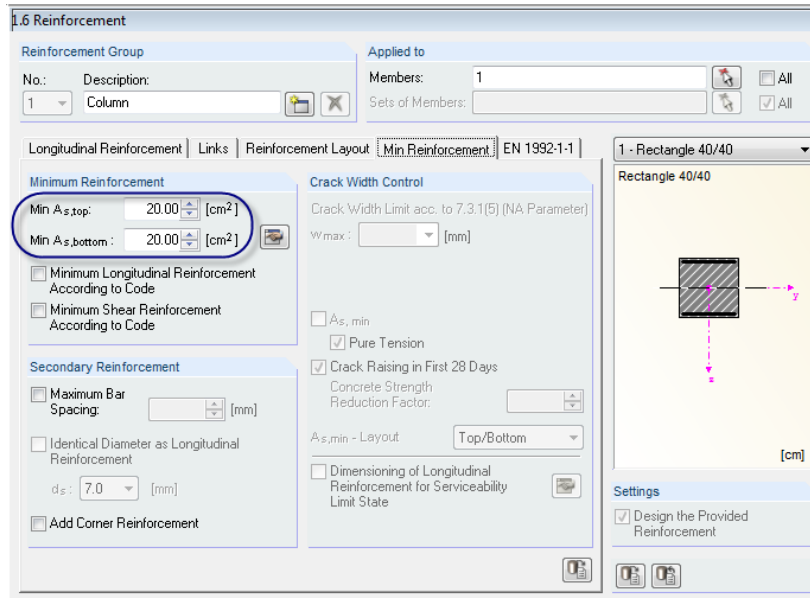


Figure 9.48: Changing the minimum reinforcement

Now the changes are set and we can start the [Calculation].

Results of non-linear calculation

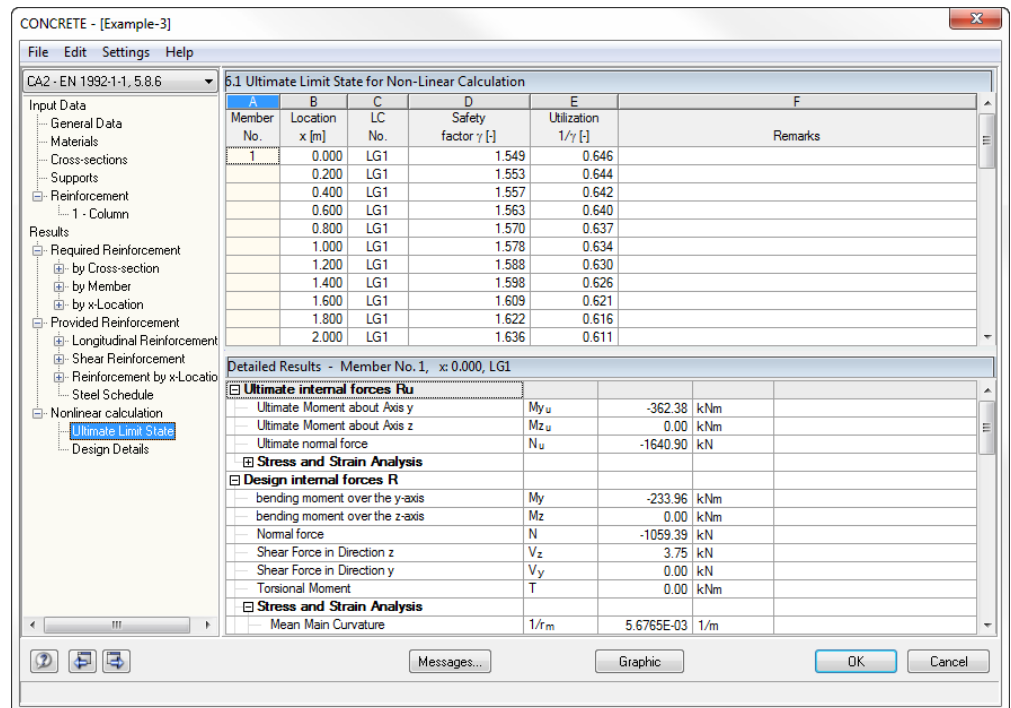


Figure 9.49: Table 6.1 *Ultimate Limit State for Non-Linear Calculation*

With the selected reinforcement we get a safety factor γ of 1.549 for the restrained location (to compare: $\gamma = 2.135$ for the design according to EN 1992-1-1, 5.7).

The following figure compares the deformations according to the second-order analysis with the ones according to both non-linear calculation methods.

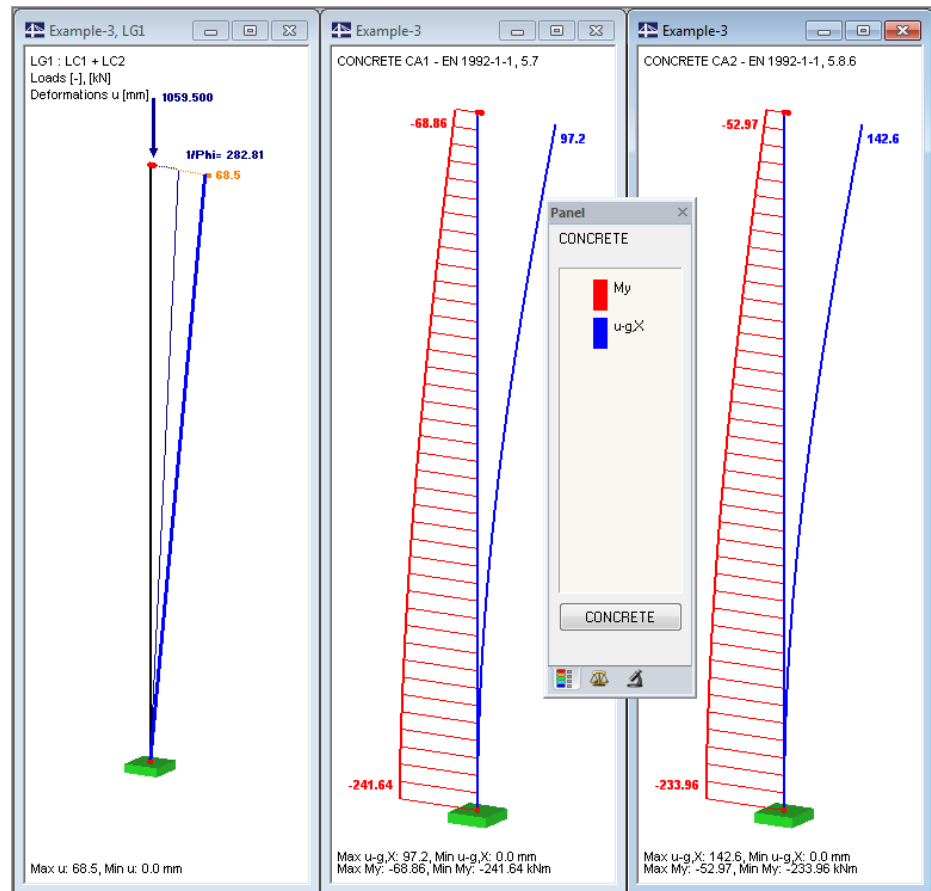


Figure 9.50: Comparison of results according to second-order analysis and non-linear calculation

For our slender compression element the calculation according to the second-order analysis is already deviating from the calculation according to the linear static analysis if a low load level is applied. The physical non-linearity 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.

When the material-dependent non-linearity is not taken into account, the pure cross-section design of the internal forces of LG1 provides a required reinforcement of $A_{s,tot} = 2 * 4.10 = 8.20 \text{ cm}^2$ according to the second-order analysis (physically linear).

2.1 Required Reinforcement by Cross-section						
A	B	C	D	E	F	G
Reinforcement	Member No.	Location x [m]	LC / LG CO	Reinforcement Area	Unit	Error Message or Note
Cross-section No. 1 - Rectangle 40/40						
$A_{s,top}$	1	0.000	LG1	4.10	cm ²	
$A_{s,bottom}$	1	0.000	LG1	4.10	cm ²	

Figure 9.51: Required reinforcement for physically linear design

Thus, the effective required reinforcement is clearly undervalued. But also the design of moment and axial force from the physically non-linear calculation would lead to an under-designed reinforcement. In this case, the result for the required reinforcement would be $A_{s,tot} = 2 * 10.1 = 20.2 \text{ cm}^2$ for $M_y = 233.96 \text{ kNm}$ and $N = -1059.39 \text{ kN}$. The reason is that the internal forces are calculated on the basis of the provided reinforcement. However, the column fails before the ultimate load capacity of the cross-section is reached. In our example this would happen for a moment of $\sim 362 \text{ kN}$ which leads in interaction with the axial force to a required reinforcement of $A_{s,tot} = 2 * 19.7 = 39.4 \text{ cm}^2$.

A Literature

Standards and other materials

- [1] DIN 1045-1 (Juli 2001) Tragwerke aus Beton, Stahlbeton und Spannbeton – Teil 1: Bemessung und Konstruktion, Beuth Verlag, Berlin Wien Zürich 2001
- [2] DIN 1045-1 (Juli 2001, 2. Auflage) Tragwerke aus Beton, Stahlbeton und Spannbeton, Teil 1 Bemessung und Konstruktion, Kommentierte Kurzfassung, Beuth Verlag, Berlin Wien Zürich 2005
- [3] DIN 1045-1 (Juni 2008) Tragwerke aus Beton, Stahlbeton und Spannbeton – Teil 1: Bemessung und Konstruktion, Beuth Verlag, Berlin Wien Zürich 2008
- [4] DIN 1045-88 Tragwerke aus Beton, Stahlbeton und Spannbeton, Teil 1 Bemessung und Konstruktion, Beuth Verlag, Berlin Wien Zürich 1988
- [5] DIN V ENV 1992-1-1:1992: Planung von Stahlbeton- und Spannbetontragwerken – Teil 1: Grundlagen und Anwendungsregeln für den Hochbau. Juni 1992.
- [6] DIN EN 1992-1-1:2005: Planung von Stahlbeton- und Spannbetontragwerken – Teil 1-1: Allgemeine Bemessungsregeln und Regeln für den Hochbau. Februar 2010.
- [7] DIN EN 1992-1-2:2005: Planung von Stahlbeton- und Spannbetontragwerken – Teil 1-2: Allgemeine Regeln – Tragwerksbemessung für den Brandfall. Oktober 2006.
- [8] ÖNORM B 4700 (Juni 2001): „Stahlbetontragwerke EUROCODE-nahe Berechnung, Bemessung und konstruktive Durchbildung“, Österreichisches Normungsinstitut, Wien
- [9] Heft 220 DAfStb. „Bemessung von Beton- und Stahlbetonbauteilen nach DIN 1045 – Biegung mit Längskraft, Schub, Torsion“, Beuth Verlag, Berlin Wien Zürich, 2. Auflage 1978
- [10] Heft 240 DAfStb. „Hilfsmittel zur Berechnung der Schnittgrößen und Formänderungen von Stahlbetontragwerken nach DIN 1045-88“, Beuth Verlag, Berlin Wien Zürich, 3. Auflage 1991
- [11] Heft 415 DAfStb., BUSJÄGER, D., QUAFT, U.: „Programmgesteuerte Berechnung beliebiger Massivbauquerschnitte unter zweiachsiger Biegung mit Längskraft“, Beuth Verlag, Berlin Wien Zürich 1990
- [12] Heft 425 DAfStb., KORDINA, K. et. al.: „Bemessungshilfsmittel zu Eurocode 2 Teil 1 – Planung von Stahlbeton- und Spannbetontragwerken“, Beuth Verlag, Berlin Wien Zürich 1992
- [13] Heft 525 DAfStb. „Erläuterungen zu DIN 1045-1“, Beuth Verlag, Berlin Wien Zürich 2003
- [14] Deutscher Beton- und Bautechnik-Verein E.V.: „Beispiel zur Bemessung nach DIN 1045-1, Band 1: Hochbau“, Ernst & Sohn Verlag 2005, 2. Auflage
- [15] Deutscher Beton- und Bautechnik-Verein E.V.: „Beispiel zur Bemessung nach DIN 1045-1, Band 2: Ingenieurbau“, Ernst & Sohn Verlag 2003, 1. Auflage

General

- [16] AVAK, R.: Stahlbetonbau in Beispielen, DIN 1045, Teil 1, „Grundlagen der Stahlbeton-Bemessung - Bemessung von Stabtragwerken“, Werner Verlag, 5. Auflage 2007

- [17] CURBACH, M. et. al.: „Nichtlineare Berechnung alter Bogenbrücken auf Grundlage neuer Vorschriften“, Beton- und Stahlbetonbau 99 (04/2004)
- [18] HEYDEL G., KRINGS W., HERRMANN H.: „Stahlbeton im Hochbau nach EC 2“, Ernst und Sohn Verlag 1995
- [19] HOSSER, D. und RICHTER, E.: 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, Stuttgart, Fraunhofer IRB 2007
- [20] KLEINSCHMITT, J.: „Die Berechnung von Stahlbetonstützen nach DIN 1045-1 mit nicht-linearen Verfahren“, Beton- und Stahlbetonbau 02/2005
- [21] KORDINA, K. und QUAST, U.: „Bemessung von schlanken Bauteilen für den durch Tragwerksverformung beeinflussten Grenzzustand der Tragfähigkeit – Stabilitätsnachweis“, Betonkalender 2002/Teil 1, Ernst & Sohn Verlag 2002
- [22] LEONHARDT, F.: „Vorlesungen über Massivbau“, Teil 1 bis 4, Springer Verlag, 3. Auflage 1984
- [23] NOAKOWSKI, P. und Schäfer, H. G.: „Steifigkeitsorientierte Statik im Stahlbetonbau“, Ernst & Sohn, 2003
- [24] NOAKOWSKI, P. und SCHÄFER, H. G.: „Die Schnittgrößen in Stahlbetontragwerken einfach richtig berechnen“, Beton- und Stahlbetonbau 96 (06/2001)
- [25] PFEIFFER, U.: „Die nichtlineare Berechnung ebener Rahmen aus Stahl- oder Spannbeton mit Berücksichtigung der durch das Aufreißen bedingten Achsdehnung“, Dissertation an der TU Hamburg-Harburg 2004, Cuvillier Verlag Göttingen
- [26] PFEIFFER, U. und QUAST, U.: „Some advantages of 1D- instead of 2D- or 3D- modelling for non-linear analysis of reinforced concrete frames“, Proceedings of the EURO-C Conference 2003, St. Johann im Pongau, 17-20 March 2003, 805-815. Lisse, Abingdon, Exton (PA), Tokyo: A. A. Balkema Publishers, 2003
Download unter <http://www.mb.tu-harburg.de> möglich
- [27] PFEIFFER, U. und QUAST, U.: „Nichtlineares Berechnen stabförmiger Bauteile“, Beton- und Stahlbetonbau 98 (09/2003)
- [28] QUAST, U.: „Zur Kritik an der Stützenbemessung“, Beton- und Stahlbetonbau 95 (05/2000)
- [29] QUAST, U.: „Zum nichtlinearen Berechnen im Stahlbeton- und Spannbetonbau“, Beton- und Stahlbetonbau 89 (09/1994)
- [30] QUAST, U.: „Nichtlineare Verfahren, normungsreif oder nicht?“, Betonbau - Forschung, Entwicklung und Anwendung, 223-232. Braunschweig: Institut für Baustoffe, Massivbau und Brandschutz
Download unter <http://www.mb.tu-harburg.de> möglich
- [31] QUAST, U.: „Versagen Stahlbetonstützen anders als Stahlstützen?“, Veröffentlicht auf der Homepage der TU Hamburg-Harburg
Download unter <http://www.mb.tu-harburg.de> möglich
- [32] VATER, C.: „Rechnerisch-theoretische Untersuchungen zur Schnittgrößenumlagerung in verschieblichen und unverschieblichen Stahlbetonrahmen“, Dissertation an der TU Hamburg-Harburg 1999
- [33] ZILCH, K. und ROGGE, A.: „Bemessung der Stahlbeton- und Spannbetonbauteile nach DIN 1045-1“, Betonkalender 2002/Teil 1 und 2004/Teil 2, Ernst & Sohn Verlag 2002 bzw. 2004

B Index

3

3D Rendering..... 122, 130

A

Ability for rotation..... 76

Adaptive element division 51, 85

Adaptive member division 24

Adjustment factor..... 84

Age of concrete 39

Analysis approach..... 81

Anchorage 67, 97

Anchorage type 97, 104

Axial force..... 9

B

Background graphic 124

Bar spacing 67

Basic reinforcement 67

Bearing capacity 46, 115, 125

Bend 97

Bending 9

Bending diameter 98, 105

Bond 97, 107

Bottom reinforcement 90

Break off limit..... 86

Buttons..... 123

C

Calculation..... 81, 87

Cement type 39

Centroidal axis 70

Charring..... 79

Classification..... 79

Code 53, 71

Column settlement 73

Comment..... 54

Compression element 82, 84

Compression force..... 10

Compression reinforcement..... 10, 75

Concrete aggregate 79

CONCRETE case..... 131

Concrete compression area 75

Concrete compression strut..... 11, 76

Concrete compressive strength77

Concrete compressive stress..... 12

Concrete cover70

CONCRETE design case.....87

Concrete strength class58

Concrete stress106

Concrete tensile strength83

Convergence50, 85, 120

Convergence settings.....120

Corner71, 73

Crack formation31

Crack spacing 15, 107

Crack width14, 73, 107, 118

Crack width analysis.....74

Crack width control.....73

Creep.....43, 54, 55, 61

Creep coefficient37, 38

Creeping.....37, 49

Cross-section60, 112

Cross-section description60

Cross-section library60

Cross-section optimization.....133

Cross-section side.....79

Curtailment67

Curtailment of reinforcement.....67

Curvature 16, 26, 45, 48, 139, 160

D

Damaged zone112

Damping33, 51, 86

Decimal places58, 134

Deflection.....16, 75, 107, 140

Deformation..... 16, 48, 75, 86, 107, 117, 120, 137, 143, 161

Design case125, 131, 132, 170

Design details.....90, 103, 115, 120

Design moment.....64

Designing longitudinal reinforcement.....80

Detailed results90, 115, 117, 127

Direct support63

Displacement117

Division85

Ductility	31	Iteration	50, 86, 120
Duration of loading	84	Iteration parameter	86
E			
Edge distance	70	L	
Effective tensile strength $f_{ct,R}$	84	Layout	71
Effects of fire	79	Layout of longitudinal reinforcement	74
End support	64	Length	97
Error message	91, 94, 111	Length of anchorage	67, 68
Eurocode	54, 56	Limit diameter	14, 106
Excel	135	Limit length	85
Export of cross-sections	133	Link	68
Export of results	135	Link dimensions	100
F			
Factor k_t	56	Link parameter	68
Failed design	94, 114	Link reinforcement	89, 90, 102, 121, 123
Filter	128	Link spacing	69, 100
Filtering members	128	Link zone	100
Fire protection	17	Load application	86
Fire protection design	57, 79, 110, 113, 125	Load case	54, 55, 111, 116, 117
Fire resistance	57, 79	Load combination	54, 55, 65
Fire resistance class	79	Load duration factor	84
Fire resistance design	119	Load group	54, 55
G			
General data	52	Load increment	51, 86
General design method	82	Load near support	65
Global offset	103	Location x. 91, 93, 96, 97, 100, 102, 106, 109, 111, 113, 115, 117	
Global partial safety factor	82	Longitudinal reinforcement	66, 89, 90, 95, 102, 121, 123
Graphic	125	Long-term effects	77, 80
H			
Hook	97	Loss of stability	82
Hydration heat	73	M	
I			
Inclination	68, 96	M ratio δ	64
Inclination of concrete strut θ	12, 76	Material	45, 49, 58
Indirect support	63	Material description	58
Initial force	84	Material library	59
Installation	6	Material properties	58
Intermediate results	103, 110, 125	Material stiffness	45, 46, 49
Internal forces	133	Maximum reinforcement ratio	75
Item	95, 99, 104, 121	Mean values of material	82
Item bar	98	Member	65, 92, 109
Item link	99, 101	Member division	51
Item member	95, 121, 123	Message	98, 100, 108, 116
		Minimum reinforcement	13, 72, 106
		Modified characteristic steel curve	84
		Modified steel characteristic diagram	83
		Modulus of creep	61

Moment reduction.....	64	Reinforcement layout.....	70, 71
Moment swap	64	Reinforcement optimization.....	82
Monolithic connection.....	64	Reinforcement position.....	96
N		Reinforcement proposal...66, 95, 98, 101, 121	
National annex	53	Reinforcing steel.....	59, 79
Navigator.....	52	Relative humidity.....	38
Neutral axis depth	75	Relaxation	43
Nominal value.....	70	Remark.....	61, 116
Non-linear calculation. 54, 55, 57, 81, 95, 115, 117, 126, 143		Required reinforcement	89
Note.....	91, 111	Restraint.....	73, 74
O		Result diagrams.....	127, 130
OpenOffice	135	Result values	125
Optimization.....	61, 133	Results evaluation	121
P		Results navigator.....	125
Panel.....	8	Results tables	89
Partial safety factor for concrete.....	77, 80	RSTAB work window	124
Partial safety factor for reinforcing steel	77, 80	S	
Partial view	128	Safety.....	103, 111, 115, 119
Percentage of reinforcement	75, 103, 133	Safety factor γ	116
Permanent load	56, 84	Scaling.....	126
Plastic release	82	Secant stiffening	27
Plate width	71	Section	68, 100
Plausibility check.....	81	Selecting tables.....	52
Precamber.....	79	Selection for printout.....	129
Prestress.....	84	Self-equilibrating stresses.....	73
Print.....	130	Serviceability 12, 48, 55, 74, 95, 105, 117, 125	
Printing graphics.....	130	Serviceability limit state.....12, 48, 66, 73, 105, 117	
Printout report.....	129, 135	Set of members.....65, 78, 92, 108, 112	
Provided reinforcement	95	Shear	80
Q		Shear force.....	64
Quit CONCRETE.....	52, 89	Shear force resistance	10
R		Shear reinforcement.....68, 76, 90, 99	
Ratio	111	Shear resistance	76
Rebar	96	Shear rigidity.....	82
Rebar diameter	67, 72, 97, 99, 104	Shrinkage	37, 43, 44, 54, 55, 61
Rebar spacing	14, 107	Shrinkage strain	37, 41, 61
Reduction factor α	77, 80	Standard	5, 75
Reinforcement	65, 67, 68	Start CONCRETE	7
Reinforcement area	89, 91	Start program	7
Reinforcement drawing	121	State II..... 16, 49, 54, 55, 83, 115, 117, 154	
Reinforcement group.....	65	Steel schedule	104
Reinforcement layer.....	67	Steel stress	13, 106
		Steel surface.....	67

Stiffness.....	33, 47, 86, 120	Tension force.....	9
Stop criteria.....	120	Tension Stiffening.....	28, 83, 84
Strain.....	26, 45	Thermal strain.....	80
Strain distribution.....	107	Time of load duration.....	84
Stress design.....	74	Top reinforcement.....	90
Structural reinforcement.....	72	Torsion.....	80, 90
Super combination.....	54	Torsional stiffness.....	82
Support.....	63	Type of anchorage.....	67, 68
Support width.....	63	U	
Supporting moment.....	64	Ultimate limit state.....	9, 10, 44, 54, 115
Surface.....	104	Units.....	58, 134
Surrounding reinforcement.....	71	Utilization.....	116
T		V	
Tables.....	52	Visualization.....	122, 125
Tangent stiffening.....	27	W	
Taper.....	92, 134	Weight.....	98, 100, 105
Tapered member.....	77	Y	
Tapering.....	78	Yielding.....	82
Temperature course.....	112	Z	
Tensile strength of concrete.....	29, 73, 106	Zones.....	79, 112
Tension.....	73		