

**Version**  
**September 2011**

**Program**

# **RF-CONCRETE Members**

**Reinforced Concrete Design  
According to ACI 318-08**

## **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](mailto:info@dlubal.com)  
Web: [www.dlubal.com](http://www.dlubal.com)



# Contents

	Contents	Page		Contents	Page
<b>1.1</b>	<b>Add-on Module RF-CONCRETE Members</b>	<b>4</b>	5.1.4	Required Reinforcement by x-Location	46
<b>1.2</b>	<b>RF-CONCRETE Members Team</b>	<b>5</b>	5.1.5	Required Reinforcement Not Designable	47
<b>1.3</b>	<b>Using the Manual</b>	<b>5</b>	<b>5.2</b>	<b>Provided Reinforcement</b>	<b>48</b>
<b>1.4</b>	<b>Open RF-CONCRETE Members</b>	<b>6</b>	5.2.1	Longitudinal Reinforcement Provided	48
<b>2.1</b>	<b>Strength Analysis</b>	<b>8</b>	5.2.2	Shear Reinforcement Provided	52
2.1.1	Flexure and Axial Loads	8	5.2.3	Reinforcement provided by x-Location	55
2.1.2	Shear Design	11	5.2.4	Steel Schedule	56
<b>2.2</b>	<b>Serviceability Limit State Design</b>	<b>12</b>	<b>5.3</b>	<b>Serviceability Limit State Design</b>	<b>58</b>
2.2.1	Provided Reinforcement	12	5.3.1	Serviceability Proof by Cross-section	58
2.2.2	Limitation of Crack Widths	13	5.3.2	Serviceability Proof by Set of Members	60
2.2.3	Limitation of Deformations	14	5.3.3	Serviceability Proof by Member	60
<b>2.3</b>	<b>Development Length</b>	<b>15</b>	5.3.4	Serviceability Proof by x-Location	61
<b>3.1</b>	<b>General Data</b>	<b>18</b>	<b>6.1</b>	<b>Reinforcement Proposal</b>	<b>62</b>
3.1.1	Ultimate Limit State	19	<b>6.2</b>	<b>3D Rendering of Reinforcement</b>	<b>63</b>
3.1.2	Serviceability Limit State	20	<b>6.3</b>	<b>Results in the RFEM Model</b>	<b>65</b>
<b>3.2</b>	<b>Materials</b>	<b>21</b>	<b>6.4</b>	<b>Result Diagrams</b>	<b>68</b>
<b>3.3</b>	<b>Cross-sections</b>	<b>23</b>	<b>6.5</b>	<b>Filter for Results</b>	<b>69</b>
<b>3.4</b>	<b>Ribs</b>	<b>25</b>	<b>7.1</b>	<b>Printout Report</b>	<b>70</b>
<b>3.5</b>	<b>Supports</b>	<b>27</b>	<b>7.2</b>	<b>Graphic Printout</b>	<b>71</b>
<b>3.6</b>	<b>Reinforcement</b>	<b>29</b>	<b>8.1</b>	<b>Design Cases in RF-CONCRETE Members</b>	<b>72</b>
3.6.1	Longitudinal Reinforcement	30	<b>8.2</b>	<b>Cross-section Optimization</b>	<b>74</b>
3.6.2	Links	32	<b>8.3</b>	<b>Units and Decimal Places</b>	<b>75</b>
3.6.3	Reinforcement Layout	33	<b>8.4</b>	<b>Export of Results</b>	<b>76</b>
3.6.4	Min Reinforcement	35	<b>9.1</b>	<b>Input Data</b>	<b>78</b>
3.6.5	Standard	36	<b>9.2</b>	<b>Flexural Strength</b>	<b>79</b>
3.6.6	Tapered	38	<b>9.3</b>	<b>Shear Strength</b>	<b>80</b>
<b>4.1</b>	<b>Plausibility Check</b>	<b>40</b>	<b>9.4</b>	<b>Development of Reinforcement</b>	<b>81</b>
<b>4.2</b>	<b>Start Calculation</b>	<b>40</b>	<b>9.5</b>	<b>Results in RF-CONCRETE Members</b>	<b>82</b>
<b>5.1</b>	<b>Required Reinforcement</b>	<b>42</b>	<b>A</b>	<b>Literature</b>	<b>83</b>
5.1.1	Required Reinforcement by Cross-section	42	<b>B</b>	<b>Index</b>	<b>86</b>
5.1.2	Required Reinforcement by Set of Members	45			
5.1.3	Required Reinforcement by Member	45			

# 1. Introduction

## 1.1 Add-on Module RF-CONCRETE Members

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

The add-on module imports all relevant structure parameters from RFEM, such as material, cross-sections, members, sets of members, ribs, 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.

RF-CONCRETE Members analyzes the ultimate and the serviceability limit state. The analysis for cracks and deflections are performed by calculating crack widths and deformations directly. Optionally, the program checks if the requirements of the fire protection design according to EN 1992-1-2:2004 are fulfilled.

The influence of creeping and shrinkage can be taken into account additionally when analyzing the deformed system.

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 318-08

CEN	EU
	BS United Kingdom
	CSN Czech Republic
	DIN Germany
	DK Denmark
	NEN Netherlands
	NF France
	PN Poland
	SFS Finland
	SIST Slovenia
	SS Singapore
	SS Sweden
	STN Slovakia
	UNI Italy
	ÖNORM Austria

National annexes for EC 2

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 RFEM printout report like all other input and results data of the add-on module.

We hope you will enjoy working with the add-on module RF-CONCRETE Members.

Your team from ING.-SOFTWARE DLUBAL

## 1.2 RF-CONCRETE Members Team

The following people were involved in the development of RF-CONCRETE Members:

### 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  
Bc. 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  
Bc. Jana Vlachová

### Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel  
Dipl.-Ing. (BA) Sandy 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. Frank Lobisch  
Dipl.-Ing. (FH) Alexander Meierhofer  
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier  
M.Eng. Dipl.-Ing. (FH) Walter Rustler  
M.Sc. 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 RFEM. The present manual focuses on typical features of the add-on module RF-CONCRETE Members.

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 RF-CONCRETE Members

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

### Menu

To start the program in the menu bar,

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

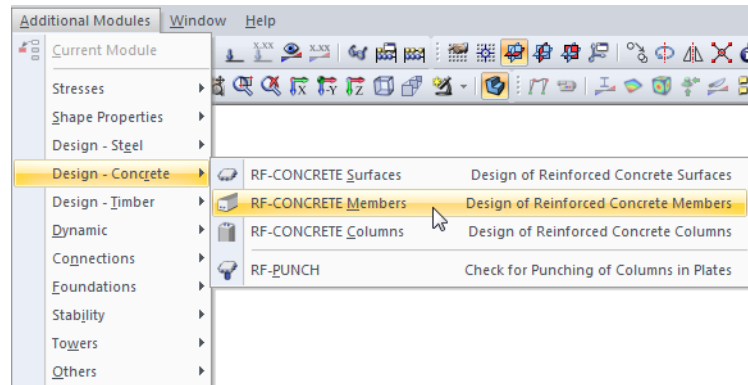


Figure 1.1: Menu *Additional Modules* → *Design - Concrete* → *RF-CONCRETE Members*

### Navigator

To start RF-CONCRETE Members in the *Data* navigator,

open the **Additional Modules** folder and select **RF-CONCRETE Members**.

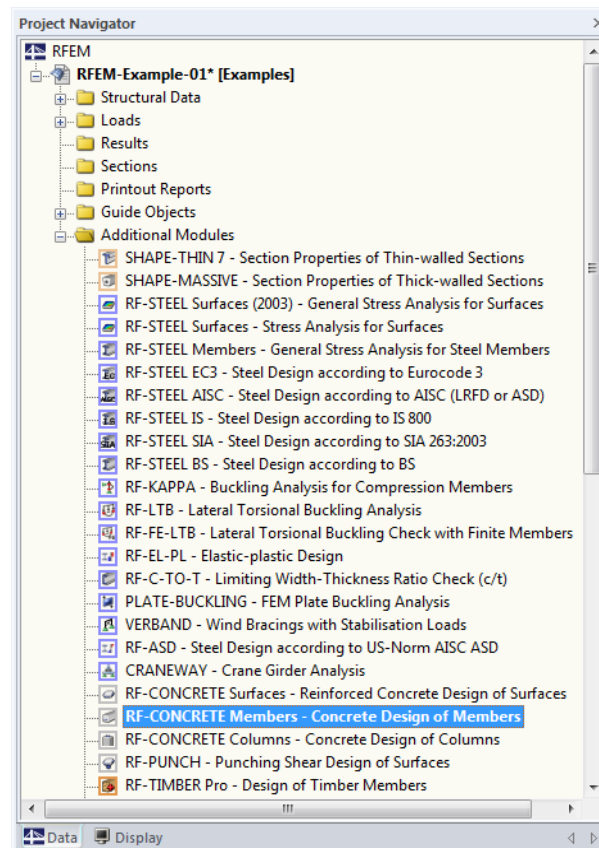
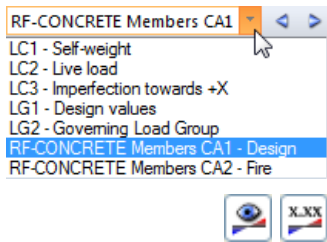


Figure 1.2: Data navigator: *Additional Modules* → *RF-CONCRETE Members*



RF-CONCRETE Members

## Panel

In case RF-CONCRETE Members results are already available in the RFEM structure, you can set the relevant design case in the load case list of the RFEM 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 [RF-CONCRETE Members] which you can use to access the design module.

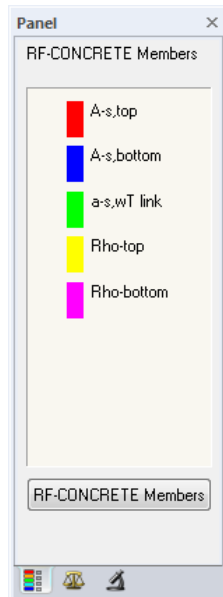


Figure 1.3: Panel button [RF-CONCRETE Members]

## 2. Theoretical Background

### 2.1 Strength Analysis

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 Flexure and Axial Loads

The standard ACI 318-08 describes in detail the calculation basis for the strength 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 strengths are reached. According to this, the design method was called ultimate-strength design. But the strength for particular reinforced concrete member is a value given by the Code and is not necessarily the true ultimate strength of the member. Therefore, the more general term strength design is used.

The design strength of a cross-section is taken as the product of the ultimate strength and a strength reduction factor. Strength reduction factors are used to account for the uncertainties of material strengths, inaccuracies in the design equations, approximations in analysis, possible variations in dimensions of the concrete sections and placement of reinforcement. The design strength of a member or connection must always be greater than the design action produced by the most severe factored load combination.

$$R_u \leq \phi R_n$$

where  $R_u$  Required strength  
 $\phi R_n$  Design strength

$\phi$  are the strength reduction factors. The design strength is achieved by multiplying the strength reduction factor  $\phi$  by nominal strength which is the theoretical ultimate strength. The Code specifies different values of  $\phi$  depending on the state of knowledge (the accuracy with which various strengths can be calculated). The prescribed strength reduction factors according to ACI Section 9.3.2 are in the following table.

Strength Condition	Strength Reduction Factor $\phi$
Tension-controlled beams and slabs	0.9
Shear and torsion in beams	0.75
Columns	0.65 or 0.75
Columns supporting very small axial loads	0.65 or 0.75 to 0.9
Bearing on concrete	0.65

Table 2.1: Prescribed  $\phi$  values for most situations

ACI 318-08 recognized three types of cross-sections based on net tensile strain. Tension-controlled members are members whose computed tensile strains are equal to or greater than 0.005; at the same time the concrete strain is 0.003. The compression-controlled member has net tensile strain less than 0.002. When  $\epsilon_t$  values fall between 0.002 and 0.005, they are said to be in the transition range between tension-controlled and compression-controlled sections. Figure 2.1 shows the variation of  $\phi$  based on  $\epsilon_t$ .



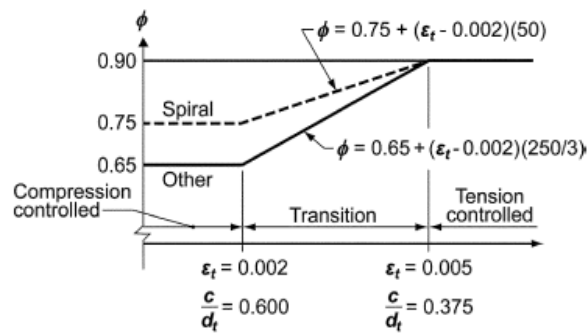


Figure 2.1: Variation of strength reduction factor with net tensile strain

For reinforced concrete structures at loads close to and at failure both of the materials, concrete and steel should be in their nonlinear inelastic range. The inelastic stress distribution is difficult and impractical, so it is convenient to use some simplifications. The concrete compressive stress distribution is replaced by an equivalent one of simple rectangular outline, see Figure 2.2 a. The simplified steel stress-strain diagram can be seen in Figure 2.2 b.

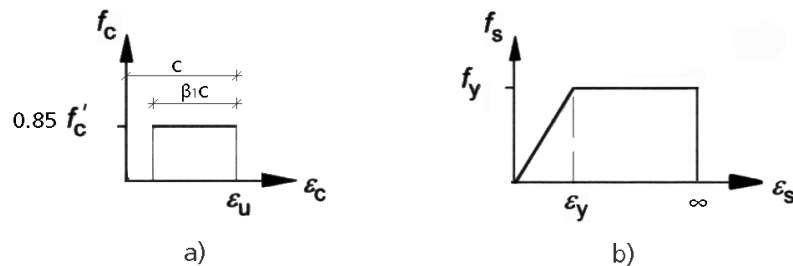


Figure 2.2: Simplified stress-strains diagrams

The following picture shows the allowable strain distributions for bending with and without axial force.

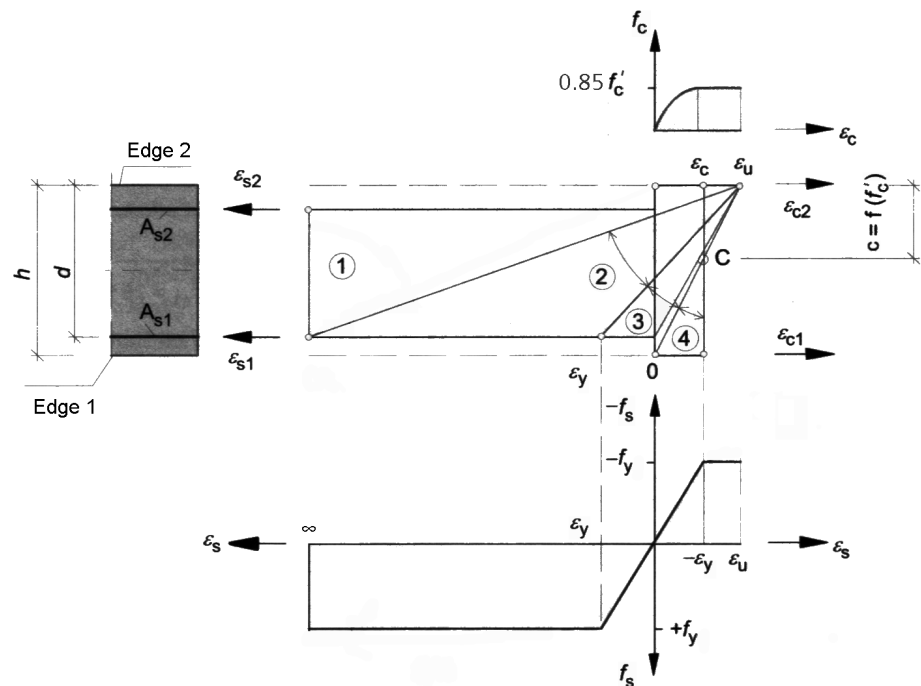


Figure 2.3: Possible strain distributions in strength limit state

According to [17] and the ACI 318-08, the different areas for strain distributions shown in Figure 2.3 have the following meanings:

### 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 after the yield strength of the reinforcement is reached.

### Area 2

This area appears in case of bending only and of bending with axial force (compression). The load-bearing capacity of the steel is higher than the one of the concrete. The concrete fails because its maximum compressive strain is reached.

The concrete's failure is announced by cracks like in the areas 1 because the steel exceeds the yield point (failure with announcement).

### Area 3

Area 3 appears in case of bending with a longitudinal compression force. It represents the transition of a cross-section mainly affected by bending and 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 3 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 4

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  $0 > \epsilon_{c1} > \epsilon_{c2}$ . All compression strain distributions intersect in point C.

## 2.1.2 Shear Design

The design for shear force resistance is only performed in the strength limit state. The actions and resistances are considered with their design values. The general design requirement according to ACI 318-08 11.1.1 is the following:

$$V_u \leq \phi V_n$$

where

$V_u$  Factored shear force at the section

$V_n = V_c + V_s$  Nominal shear strength

$V_c$  Shear strength provided by concrete

$V_s$  Shear strength provided by shear reinforcement

$\phi = 0.75$  Strength reduction factor for shear

When the factored load exceeds one-half the shear design strength of concrete, the Code requires provision of at least a minimum area of web reinforcement equal to:

$$A_{v,min} = 0.75 \sqrt{f'_c} \frac{b_w s}{f_{yt}} \geq 50 \frac{b_w s}{f_{yt}} \quad \text{ACI 318-08, eq. (11-13)}$$

where  $\sqrt{f'_c} \leq 100 \text{ psi}$  Square root of compressive strength of concrete

$b_w$  Web width in inches

$s$  Longitudinal spacing of web reinforcement in inches

$f_{yt}$  Yield strength of web steel in psi

### Shear strength provided by concrete

For members subject to shear and flexure is the concrete contribution to shear:

$$V_c = \left( 1.9 \lambda \sqrt{f'_c} + 2500 \rho_w \frac{V_u d}{M_u} \right) b_w d \leq 3.5 \lambda \sqrt{f'_c} b_w d \quad \text{ACI 318-08, eq. (11-5)}$$

where  $\sqrt{f'_c} \leq 100 \text{ psi}$  Square root of compressive strength of concrete

$\lambda$  Lightweight concrete modification factor

$\rho_w$  Ratio of longitudinal reinforcement

Ratio  $\frac{V_u d}{M_u}$  shall not be greater than 1.0

Alternative simplified equation for  $V_c$  is permitted by ACI 318-08, 11.2.1.1:

$$V_c = 2 \lambda \sqrt{f'_c} b_w d \quad \text{ACI 318-08, eq. (11-3)}$$

### Shear strength provided by shear reinforcement

Web reinforcement is provided to ensure that the full flexural capacity can be developed. For structural components with design shear reinforcement perpendicular to the component's axis ( $\alpha = 90^\circ$ ) the following can be applied:

$$V_s = \frac{A_v f_{yt} d}{s} \quad \text{ACI 318-08, eq. (11-15)}$$

where	$A_v$	Total cross-sectional area of web steel within distance $s$ in square inches
	$d$	Effective depth of bending reinforcement in inches
	$s$	Longitudinal spacing of web reinforcement in inches
	$f_{yt}$	Yield strength of web steel in psi

Where inclined stirrups are used:

$$V_s = \frac{A_v f_{yt} (\sin \alpha + \cos \alpha) d}{s} \quad \text{ACI 318-08, eq. (11-15)}$$

where	$\alpha$	Angle between stirrups and longitudinal axis of the member
-------	----------	--

According to ACI Section 11.4.7.9  $V_s$  is not allowed to be greater than  $8\lambda\sqrt{f_c}b_w d$ .

Maximum spacing of vertical stirrups is the smallest of  $\frac{d}{2}$  and 24 in. Inclined stirrups shall be spaced so that every  $45^\circ$  line is crossed by at least one line of web reinforcement. When  $V_s$  exceeds  $4\lambda\sqrt{f_c}b_w d$ , these maximum spacings are reduced by one-half.

## 2.2 Serviceability Limit State Design

The provision for adequate strength does not necessarily ensure acceptable behavior of the member at service load level. Therefore, the code includes additional requirements to provide satisfactory service load performance.

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

- Limitation of crack widths: ACI 318-08, 10.6
- Limitation of deformation: ACI 318-08, 9.5

### 2.2.1 Provided Reinforcement

Before the program designs the serviceability limit state, it checks the provided reinforcement. First, RF-CONCRETE Members 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 Crack Widths

### Permissible crack width

The maximum crack widths are considered to be acceptable vary from approximately 0.004 to 0.016 in., depending on the location of the member in question, the type of structure, the surface texture of the concrete, illumination, and other factors. ACI Committee 224 provides a guide for crack widths in the tensile face of reinforced-concrete structures for typical exposure conditions.

Exposure Condition	Tolerable Crack Width
Dry air or protective membrane	0.016 in
Humidity, moist air, soil	0.012 in
Deicing chemicals	0.007 in
Seawater and seawater spray	0.006 in
Use in water-retaining structures	0.004 in

Table 2.2: Tolerable crack widths

### The Gergely and Lutz equation

The following equation has been developed to predict the maximum width of crack at the tension face of the beam:

$$w = 0.076 \beta f_s \sqrt[3]{d_c A}$$

where	w	The estimated cracking width in thousandth of inches
	$f_s$	Steel tensile stress at service load in kips per square inch
	$d_c$	Thickness of concrete cover measured from tension face to center of bar closest to that face in inches
	$\beta$	Ratio of distances from tension face and from steel centroid to neutral axis
	A	Effective tension area of concrete surrounding tension bars and having the same centroid, divided by number of bars in square inches

### Design of rebar spacing

The maximum center-to-center spacing for the reinforcement closest to the surface of the tension member is specified in ACI Code 10.6.4. The rebar spacing should not exceed the value computed with the following expression:

$$s = 15 \left( \frac{40,000}{f_s} \right) - 2.5c_c \leq 12 \left( \frac{40,000}{f_s} \right) \quad \text{ACI 318-08, eq. (10-4)}$$

where	$f_s$	Steel tensile stress at service load in kips per square inch
	$c_c$	Clear cover from the nearest surface in tension to the surface of the flexural tension reinforcement in inches

The stress  $f_s$  is calculated by dividing the unfactored bending moment by the beam's internal moment arm. Alternatively, the ACI Code permits  $f_s$  to be taken simply equal to  $2/3 f_y$ .

These ACI maximum bar-spacing provisions are sufficient for beams and one-way slabs. The ACI equation (10-4) does not apply to beams with extreme exposure or to structures that are supposed to be watertight. In such cases special investigations and provisions are required. These include the use expressions such as Gergely-Lutz equation.

### 2.2.3 Limitation of Deformations

The reinforced concrete specification usually limits the deflections by specifying a certain minimum thickness or maximum permissible computed deflection. The first is simple and satisfactory in many cases and load distributions when member sizes and proportions are within the usual ranges.

#### Minimum thicknesses

The minimum depth-span ratio of one-way constructions is specified in ACI 318-08, 9.5.2, Table 9.5(a). This table applies only to members that are not supporting or not attached to partitions or other constructions likely to be damaged by excessive deflections.

	Minimum thickness, $h$			
	Simply supported	One end continuous	Both ends continuous	Cantilever
Member	Members not supporting or attached to partitions or other construction likely to be damaged by large deflections			
Solid one-way slabs	$\ell/20$	$\ell/24$	$\ell/28$	$\ell/10$
Beams or ribbed one-way slabs	$\ell/16$	$\ell/18.5$	$\ell/21$	$\ell/8$

Notes:  
 Values given shall be used directly for members with normalweight concrete and Grade 60 reinforcement. For other conditions, the values shall be modified as follows:  
 a) For lightweight concrete having equilibrium density,  $w_c$ , in the range of 90 to 115 lb/ft<sup>3</sup>, the values shall be multiplied by  $(1.65 - 0.005w_c)$  but not less than 1.09.  
 b) For  $f_y$  other than 60,000 psi, the values shall be multiplied by  $(0.4 + f_y/100,000)$ .

Figure 2.4: Minimum thickness of nonprestressed beams or one-way slabs according to ACI 318-08, Table 9.5(a)

#### Calculation of deflection

When there is need to use member depths shallower than are permitted by Table 9.5(a) or when members support a construction that is likely to be damage by large deflections, the deflection must be calculated and compared with limit values, see the following figure.

Type of member	Deflection to be considered	Deflection limitation
Fat roofs not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to live load $L$	$\ell/180^*$
Floors not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to live load $L$	$\ell/360$
Roof or floor construction supporting or attached to nonstructural elements likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements (sum of the long-term deflection due to all sustained loads and the immediate deflection due to any additional live load) <sup>†</sup>	$\ell/480^\ddagger$
Roof or floor construction supporting or attached to nonstructural elements not likely to be damaged by large deflections		$\ell/240^\S$

<sup>\*</sup>Limit not intended to safeguard against ponding. Ponding should be checked by suitable calculations of deflection, including added deflections due to ponded water, and considering long-term effects of all sustained loads, camber, construction tolerances, and reliability of provisions for drainage.  
<sup>†</sup>Long-term deflection shall be determined in accordance with 9.5.2.5 or 9.5.4.3, but may be reduced by amount of deflection calculated to occur before attachment of nonstructural elements. This amount shall be determined on basis of accepted engineering data relating to time-deflection characteristics of members similar to those being considered.  
<sup>‡</sup>Limit may be exceeded if adequate measures are taken to prevent damage to supported or attached elements.  
<sup>§</sup>Limit shall not be greater than tolerance provided for nonstructural elements. Limit may be exceeded if camber is provided so that total deflection minus camber does not exceed limit.

Figure 2.5: Maximum permissible computed deflection according to ACI 318-08, Table 9.5(b)

The deflection of concrete members can be calculated with the usual elastic deflection expression.

$$\delta = \frac{f(q, L, \dots)}{EI}$$

where  $EI$  Flexural stiffness  
 $f$  Function  $f$  depends on loading, geometry, etc.

Loads used in deflection calculation are unfactored loads. In some cases, only live load is considered, while in others, both live and dead loads are considered.

## 2.3 Development Length

The development length concept for anchorage of deformed bars and deformed wire in tension is based on the attainable average bond stress over the length of embedment of the reinforcement. This concept requires the specified minimum lengths or extensions of reinforcement beyond all locations of peak stress in the reinforcement. Such peak stresses generally occur in flexural members at the locations of maximum stress and where adjacent reinforcement terminates or is bent.

The strength reduction factor  $\phi$  is not used in development length calculation. The specified development lengths already include an allowance for understrength.

It is necessary that the calculated stress in the steel at each section is developed by the adequate embedded length or end anchorage, or a combination of the two. For the usual case, with no special end anchorage, this means that the full development length  $l_d$  must be provided beyond the critical sections at which peak stresses exist in the bars. These critical sections are located at points of maximum moment and at points where adjacent terminated reinforcement is no longer needed to resist bending.

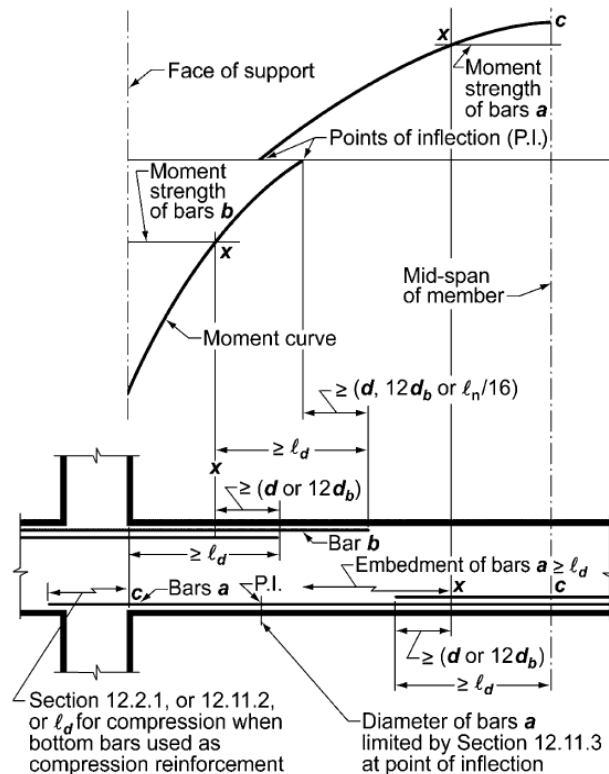


Figure 2.6: Development of flexural reinforcement according to ACI Code 12.10.3, Figure R12.10.2

### Development of tension bars

The development length of straight deformed bars and wires in tension is given in 12.2.3 of the Code by the general equation:

$$l_d = \left( \frac{3}{40} \frac{f_y}{\lambda \sqrt{f'_c}} \frac{\psi_t \psi_e \psi_s}{\left( \frac{c_b + K_{tr}}{d_b} \right)} \right) \cdot d_b \quad \text{ACI 318-08, eq. (12-1)}$$

where	$\sqrt{f'_c} \leq 100 \text{ psi}$	Square root of compressive strength of concrete
	$f_y$	Specified yield strength of reinforcement in psi
	$d_b$	Diameter of bar in inches
	$c_b$	Smaller of minimum cover or one-half of bar spacing measured to centre of bar in inches
	$\lambda$	Lightweight concrete modification factor
	$K_{tr}$	Transverse reinforcement index $K_{tr} = \frac{40 \cdot A_{tr}}{s \cdot n}$ . It shall be permitted to use $K_{tr} = 0$ as a designed simplification even if transverse reinforcement is present.
	$A_{tr}$	Total cross-section area of all transverse reinforcement within the spacing $s$ that crosses the potential plane of splitting through the reinforcement being developed in square inches
	$s$	Spacing of transverse reinforcement in inches
	$n$	Number of bars developed or spliced at the same location
	$\psi_t$	Reinforcement location factor
	$\psi_e$	Coating factor
	$\psi_s$	Reinforcement size factor

Note that the term  $\left( \frac{c_b + K_{tr}}{d_b} \right)$  cannot be taken greater than 2.5 to safeguard against pullout type failures.

When the amount of flexural reinforcement provided exceeds the theoretical amount required and where the specification being used does not specifically require that the development lengths is based on  $f_y$ , the value of  $l_d$  may be multiplied by  $\frac{A_{s,required}}{A_{s,provided}}$  as specified in ACI 318-08, 12.2.5.

The development length of tension reinforcement shall not be less than 12 in.



### Development of compression bars

Shorter development lengths are required for bars in compression than in tension since the weakening effect of flexural tension cracks in the concrete is not present.

According to ACI 318-08, 12.3 the development length in compression can be determined from the following formula:

$$l_{dc} = \frac{0.02f_y}{\lambda\sqrt{f'_c}} d_b \leq 0.0003f_y d_b$$

where	$\sqrt{f'_c} \leq 100\text{psi}$	Square root of compressive strength of concrete
	$f_y$	Specified yield strength of reinforcement in psi
	$d_b$	Diameter of bar in inches
	$\lambda$	Lightweight concrete modification factor

If more compression reinforcement is used than is required by analysis,  $l_{dc}$  may be multiplied by  $\frac{A_{s,required}}{A_{s,provided}}$  as per ACI Section 12.3.3.

### Development of standard hooks

When sufficient space is not available to anchor tension bars by running them straight for their development lengths, it is necessary to provide special anchorage at the end of the bars, usually by means of a 90° or a 180° hook. For compression bars for development length purposes are hooks considered useless.

The basic development length for standard hooks in tension is given in ACI Section 12.5.2:

$$l_{dh} = \frac{0.02\psi_e f_y}{\lambda\sqrt{f'_c}} d_b$$

where	$\sqrt{f'_c} \leq 100\text{psi}$	Square root of compressive strength of concrete
	$f_y$	Specified yield strength of reinforcement
	$d_b$	Diameter of bar
	$\lambda$	Lightweight concrete modification factor
	$\psi_e$	Coating factor

If more reinforcement is provided than that required by analysis, the development length  $l_{dh}$  may be reduced by the ratio of  $\frac{A_{s,required}}{A_{s,provided}}$ .

## 3. Input Data

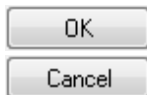


All data required for the definition of design cases is entered in tables. The [Pick] function allows for a graphical selection of the objects that you want to design.

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 72).

If you open RF-CONCRETE Members in an RFEM structure for the first time, the module imports the following design relevant data automatically:

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



To select a table, click the corresponding entry in the RF-CONCRETE Members 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 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 and the serviceability limit state by using the respective tab.

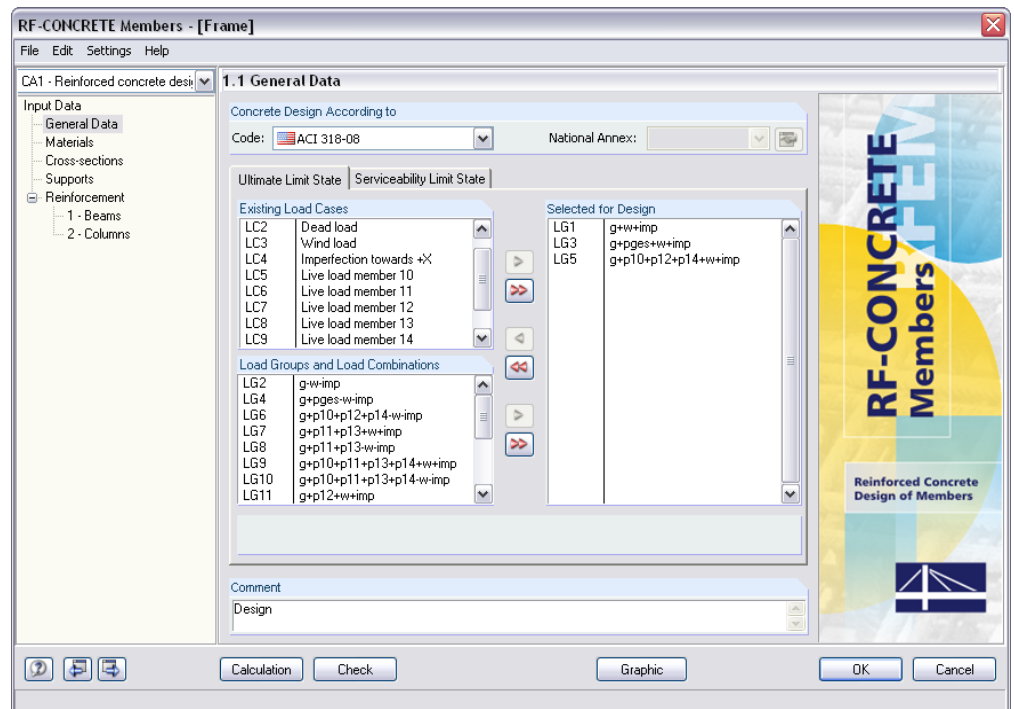


Figure 3.1: Table 1.1 *General Data*, tab *Ultimate Limit State*



### Concrete Design According to Code / National Annex

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

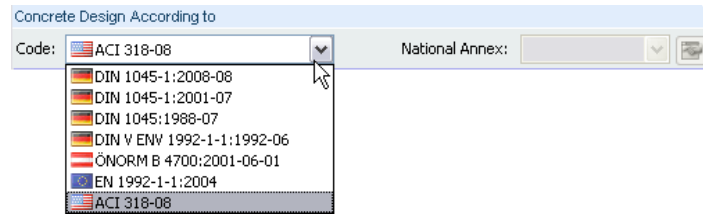


Figure 3.2: Selection of design standard

### 3.1.1 Ultimate Limit State

The first tab of table 1.1 *General Data* is shown in Figure 3.1 on page 18.

#### Existing Load Cases / Load Groups and Load Combinations

These two dialog sections list all actions and load combinations defined in RFEM that are relevant for the design. Use the [►] button to transfer the selected load cases (LC), groups (LG) or combinations (CO) 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 items. 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 is less transparent for a CO design.

#### 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

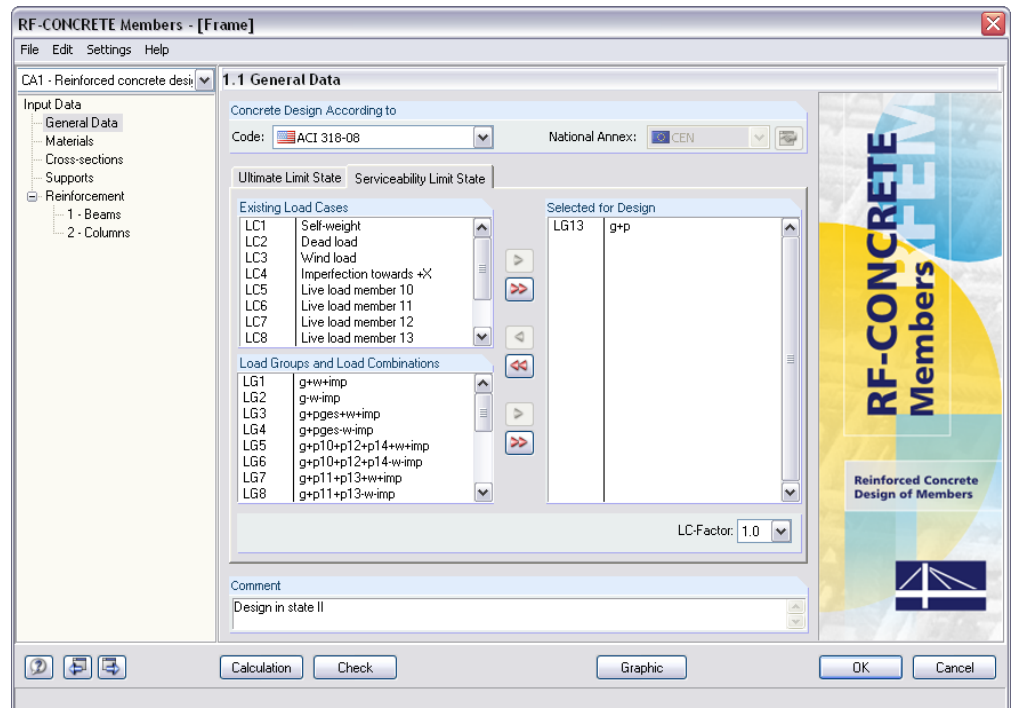


Figure 3.3: Table 1.1 General Data, tab Serviceability Limit State

#### Existing Load Cases / Load Groups and Load Combinations

These two dialog sections list all actions and load combinations defined in RFEM. Use the [▶] button to transfer the selected load cases (LC), groups (LG) or combinations (CO) 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 items. With the button [◀◀], you can transfer the entire list to the left.

## 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 RFEM are described in detail in chapter 5.3 of the RFEM 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 75).

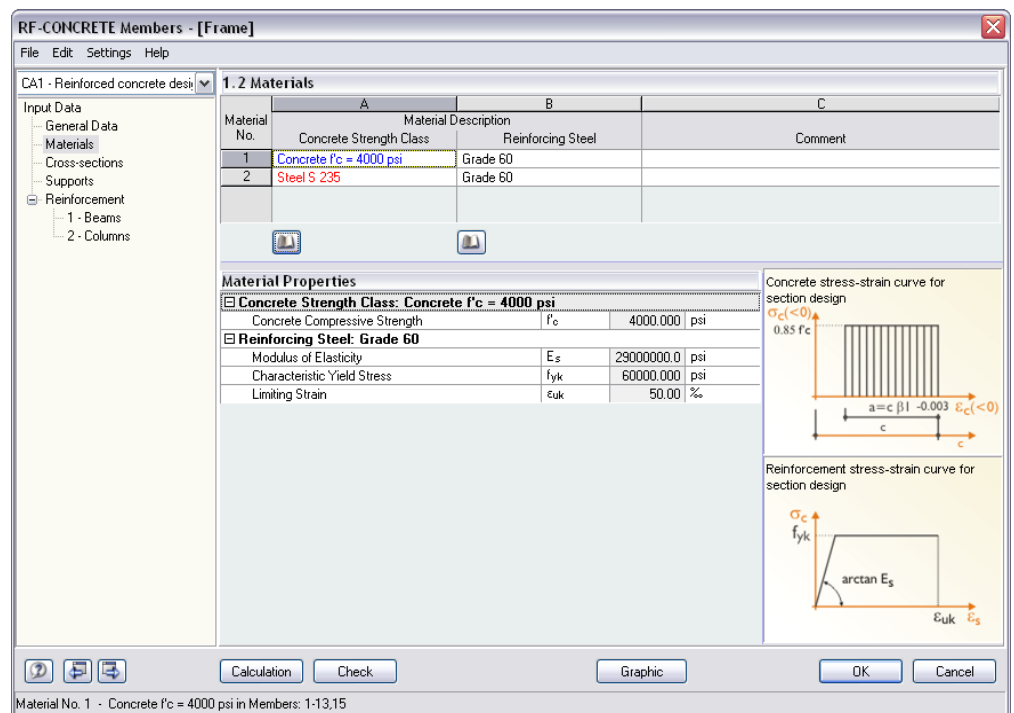


Figure 3.4: Table 1.2 *Materials*

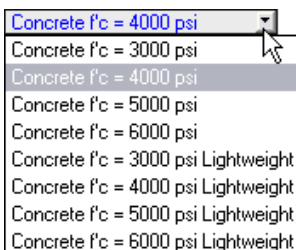
### Material Description

#### Concrete Strength Class

The concrete materials defined in RFEM 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, RF-CONCRETE Members 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.

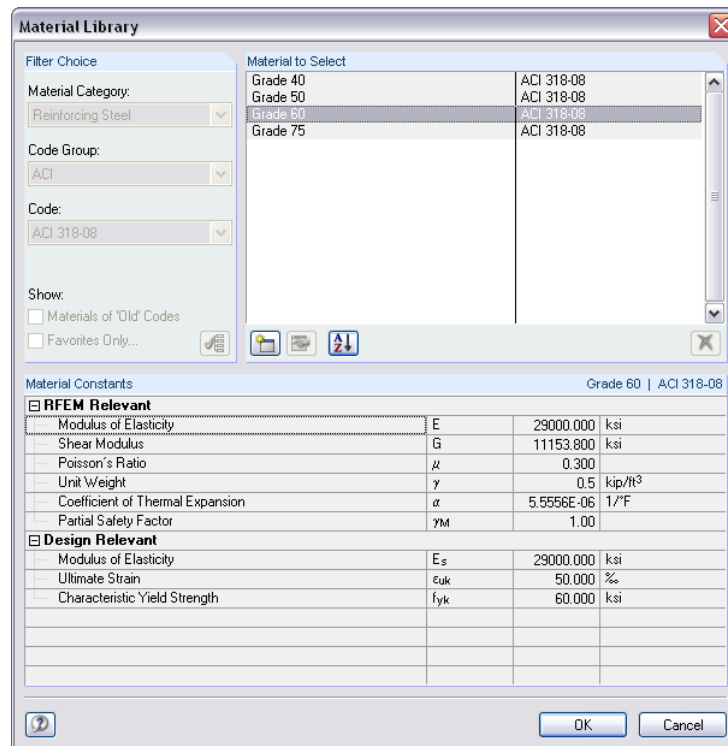
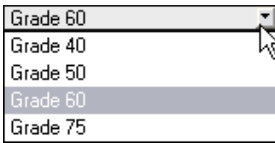


Figure 3.5: Dialog box *Material Library*

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 5.3 in the RFEM 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.

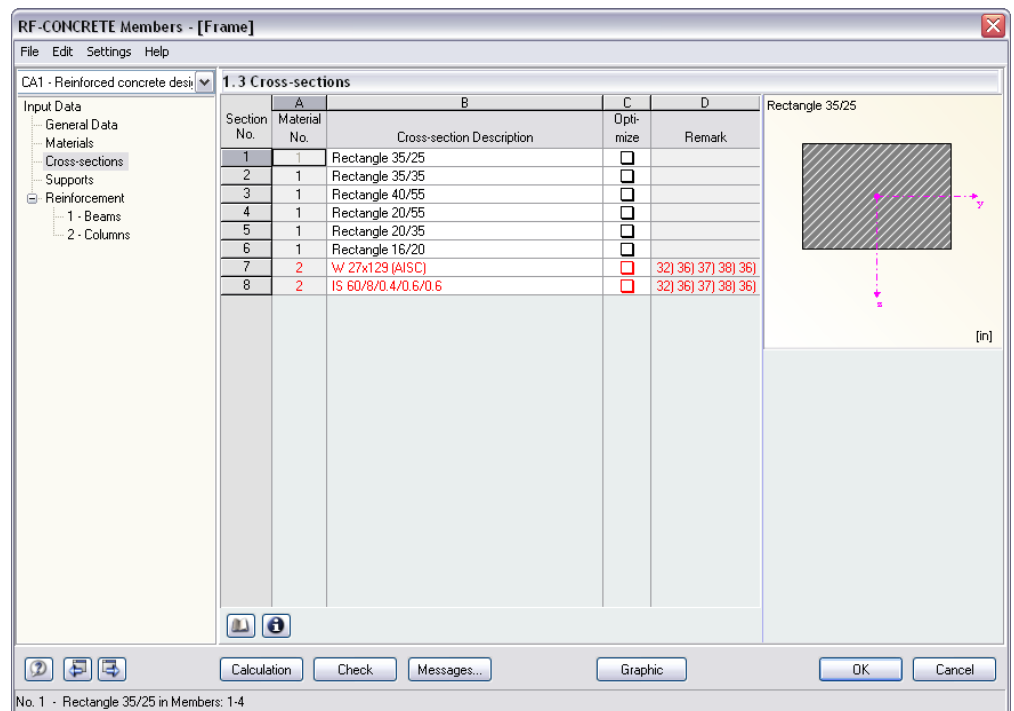


Figure 3.6: Table 1.3 Cross-sections

### Cross-section Description

When you open the table, the cross-sections used in RFEM 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 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 RFEM appears. For the design in RF-CONCRETE Members 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 5.13 of the RFEM manual.

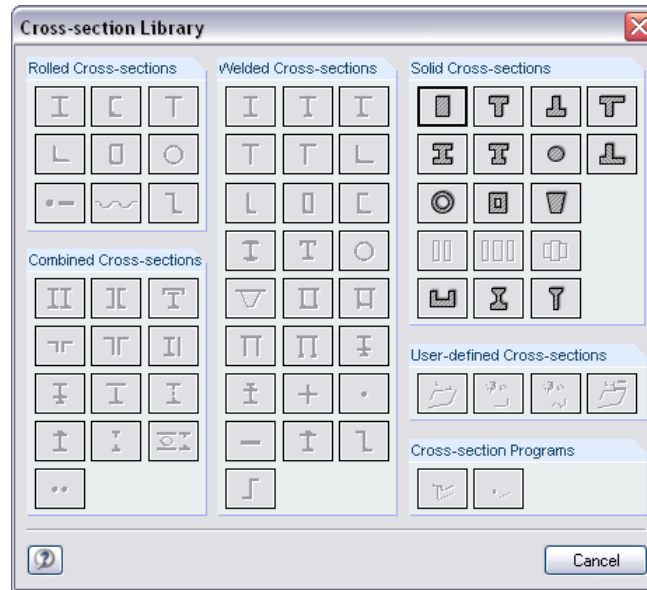
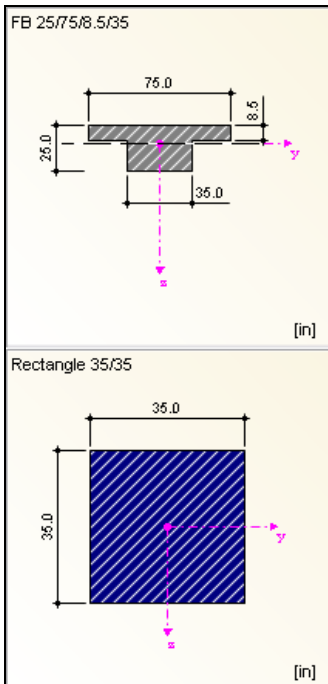


Figure 3.7: Cross-section Library

If the cross-sections in RF-CONCRETE Members are different from the ones used in RFEM, 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 RFEM 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 74).

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 74.

### Remark

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



## 3.4 Ribs

The ribs defined in RFEM are already preset. Ribs represent a special type of member consisting of a beam and a plate cross-section that is also effective (cf. chapter 5.18 of the RFEM manual). The program takes the rib internal forces from RFEM and uses them for the design.

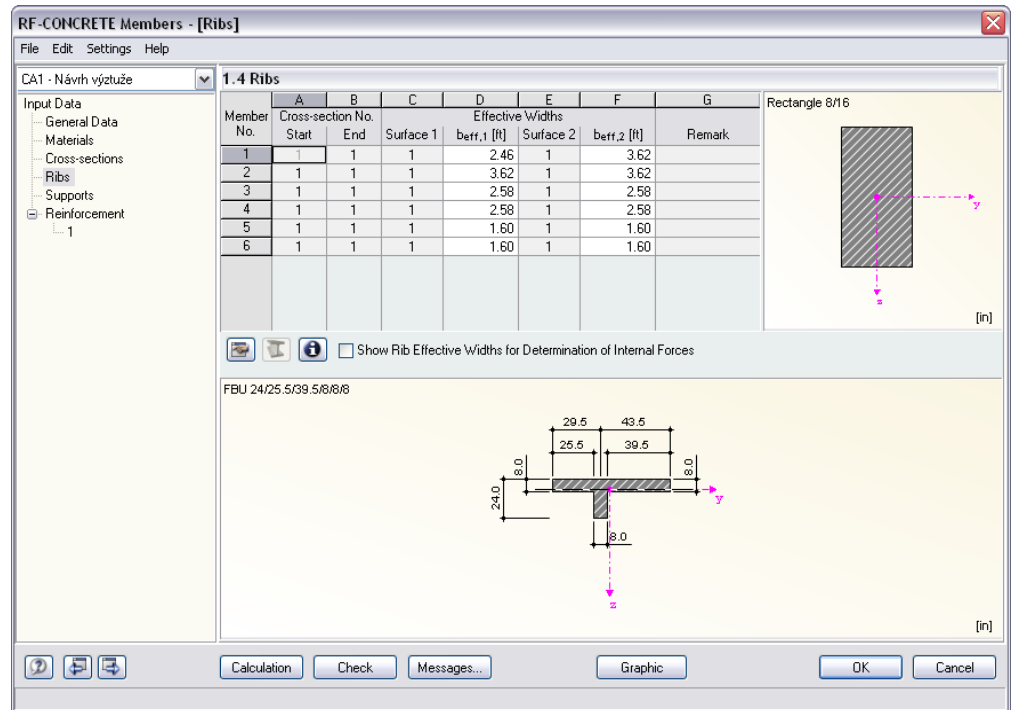


Figure 3.8: Table 1.4 Ribs

The effective widths in this table can be modified either directly by entering values in column D and F or indirectly by using the [Edit Rib] button. A recalculation in RFEM is not required as the system's stiffness is not changed. The calculation of the cross-section properties and the integration of the rib internal forces are always carried out automatically when the effective widths are modified.

### Member No.

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

### Cross-section No. Start / End

Column A and B are indicating the cross-section numbers (see chapter 3.3). In case different numbers are displayed, the member is a tapered member.

### Effective Widths $b_{eff}$

Column D and F show the effective widths for the left and the right side of the member. The values are identical with the specifications entered in the RFEM dialog box *New Rib* (cf. RFEM manual, figure 5.113 on page 161). Based on the integration widths for the pro rata internal forces in surfaces the rib internal forces are determined.

The effective width is decisive for the cross-section design in the form of an equivalent cross-section. Therefore, you can adjust the values for  $b_{eff}$  (but increasing the integration width is not allowed). To check the data, tick the check box *Show Rib Effective Widths for Determination of Internal Forces*: The table will be extended by two further columns and the button [Edit Rib] will be activated (see Figure 3.9).

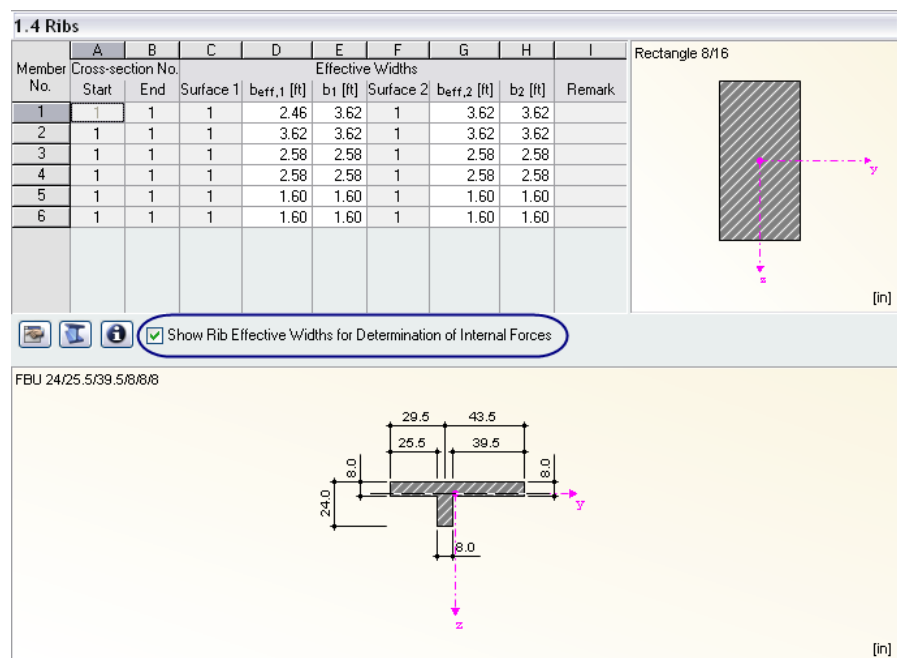


Figure 3.9: Table 1.4 Ribs

Modifications are shown by dynamic display in the cross-section graphic below the table. The graphic shows the equivalent cross-section that is used for the design.

Reduced effective widths result in reduced member internal forces affecting the design in RF-CONCRETE Members.

### Remark

In case a rib is a problem for the design, a note is indicated.

The buttons below the table are reserved for the following functions:




Button	Description	Function
	Edit	Opens the dialog box <i>Solid Sections - Floor Beam Unsymmetric</i> including parameters of equivalent cross-section
	Edit Rib	Opens the dialog box <i>New Rib</i> including rib parameters (cf. RFEM manual, figure 5.113 on page 161)
	Info about Cross-section	Shows cross-section properties of equivalent cross-section (type: floor beam unsymmetric)

Table 3.1: Buttons in table 1.4 Ribs



### For a correct design of ribs consider the following requirements:

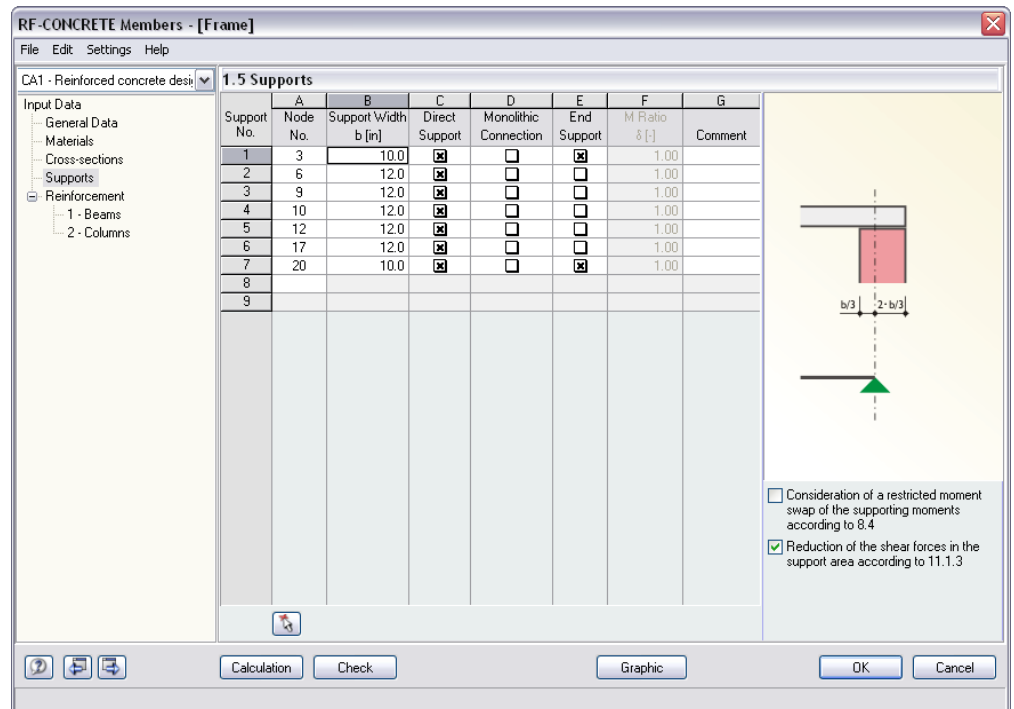
- The rib's local z-axis must be parallel to the local z-axis of the surface.
- The rib's local z-axis must be orthogonal to the xy-axis of the surface plane.
- The connected surface must be defined as surface type *Plane*.
- The cross-section type of the rib member must be a *Rectangle*.
- When sets of members are used, a uniform rib type must be defined for the entire set of members.
- The rib member must have the same cross-section at its start and its end. A taper is not allowed.

## 3.5 Supports

This table provides information about the support conditions of the members that you want to design. The nodal supports defined on horizontal members in RFEM are already preset. If necessary, it is possible to adjust them. RF-CONCRETE Members 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, i. e. no columns!



Support No.	A Node No.	B Support Width b [in]	C Direct Support	D Monolithic Connection	E End Support	F M Ratio δ [-]	G Comment
1	3	10.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.00	
2	6	12.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
3	9	12.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
4	10	12.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
5	12	12.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
6	17	12.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1.00	
7	20	10.0	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	1.00	
8							
9							

☐ Consideration of a restricted moment swap of the supporting moments according to 8.4  
☒ Reduction of the shear forces in the support area according to 11.1.3

Figure 3.10: 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 to select additional nodes graphically in the RFEM 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 only represented by a singular support in the RFEM model.

### 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 $\delta$

For continuous structural components you can define the ratio  $\delta$  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  $\delta$ -values can be determined according to the standard, e.g. ACI 318-08, 8.4.

## 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  $\delta$  that must be observed in order to ensure the ability for rotation in the critical areas without special designs, for example ACI 318-08, 8.4.

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

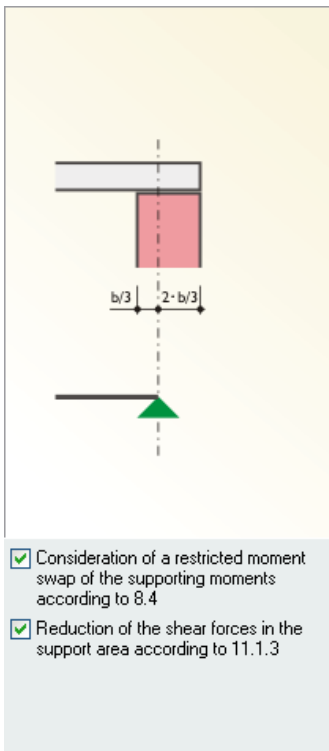
### Reduction of the shear forces in the support area

For a direct support it is possible to reduce the design value of the shear force, cf. ACI 318-08, 11.1.3.

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 the form of 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.

1.00
0.65
0.7
0.75
0.8
0.85
0.9
0.95



## 3.6 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 design case of RF-CONCRETE Members. The reinforcement specifications can be defined for members as well as for sets of members.

### Reinforcement groups

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.

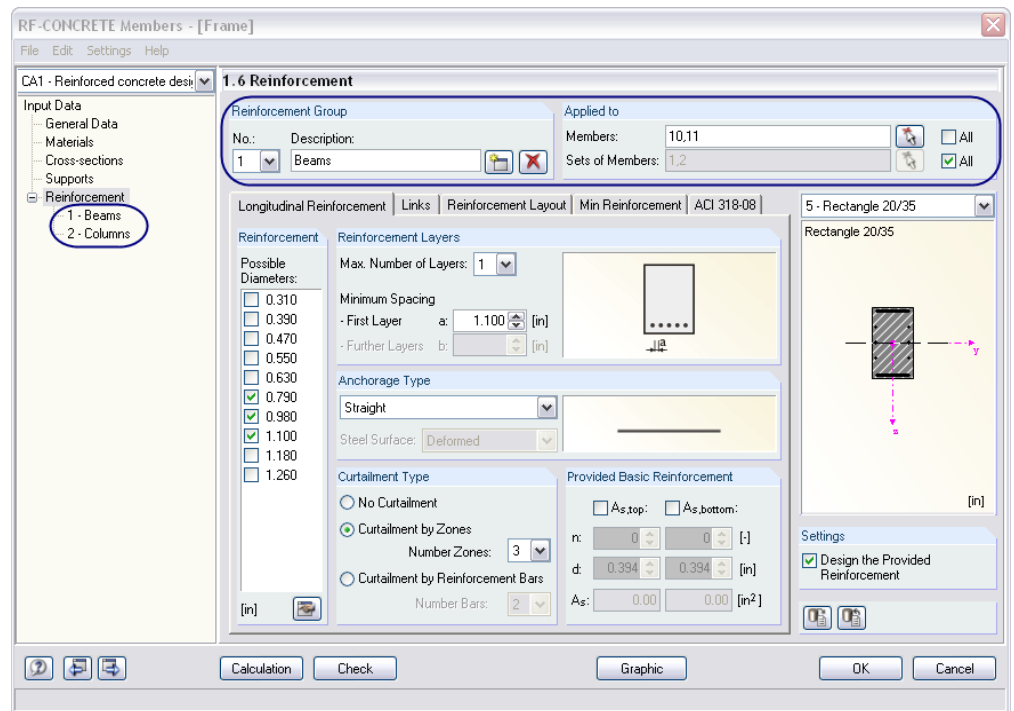


Figure 3.11: Table 1.6 *Reinforcement* with two reinforcement groups

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 RF-CONCRETE Members 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 according to 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 RFEM 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 another 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, RF-CONCRETE Members 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.

If you activate in table 1.1 *General Data* the design for the serviceability limit state or for fire protection, it is not possible to clear the check box because it is not active: 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.

### 3.6.1 Longitudinal Reinforcement

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

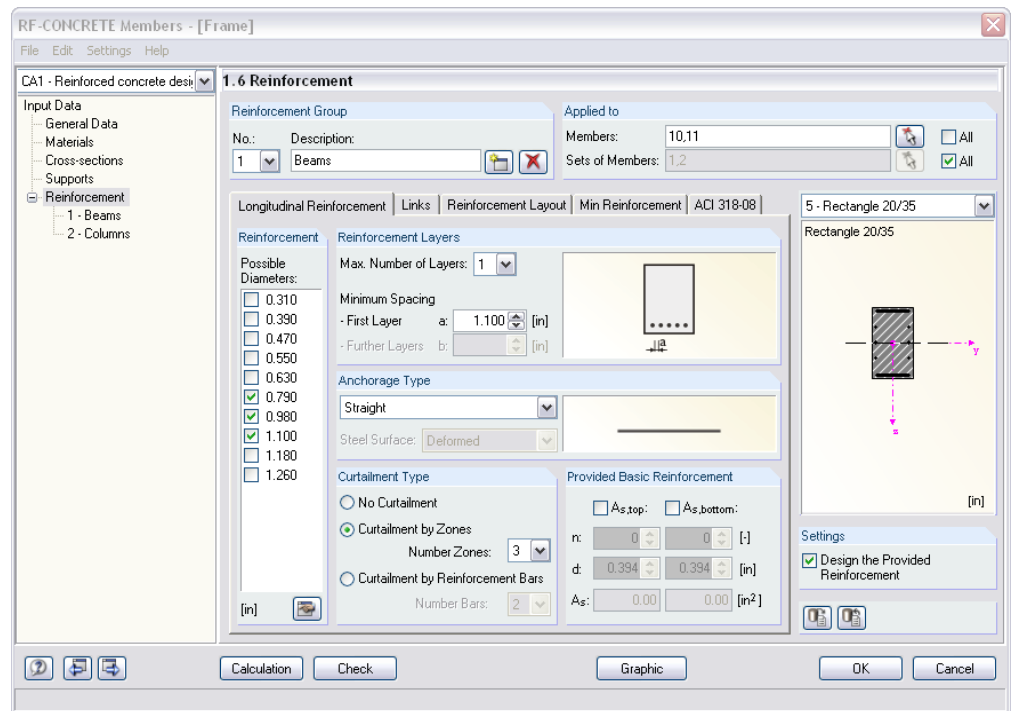
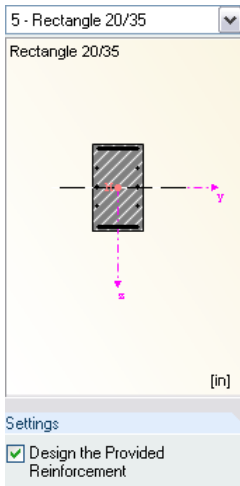


Figure 3.12: Table 1.6 *Reinforcement*, tab *Longitudinal Reinforcement*

### 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.

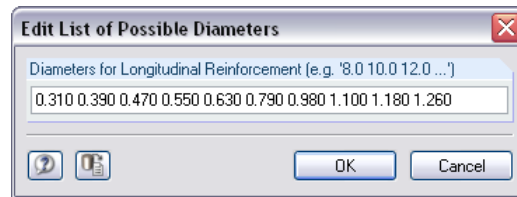
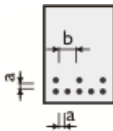


Figure 3.13: Dialog box *Edit List of Possible Diameters*

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

## Reinforcement Layers



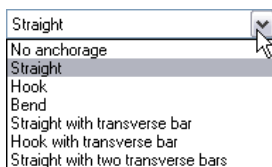
For the reinforcement proposal RF-CONCRETE Members takes into account also multi-layer arrangements of rebars. Use the list to specify the *Max. Number of Layers*. It is possible to define up to three reinforcement layers. The input fields below allow for entering specifications concerning the *Minimum 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 type affects 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 RF-CONCRETE Members 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 the bottom reinforcement layer. When the check boxes are ticked, the input fields below become active. Enter the number of rebars  $n$  and the rebar diameter  $d$  to define the reinforcement. The input field  $A_s$  will display the corresponding reinforcement areas.

The user-defined basic reinforcement is taken into account when the reinforcement proposal is created. It will be inserted across the entire length of the member or set of members. If the required reinforcement cannot be covered by the basic reinforcement, RF-CONCRETE will determine the rebars additionally required and insert them into the cross-section.

### 3.6.2 Links

This tab contains reinforcement specifications for the shear reinforcement.

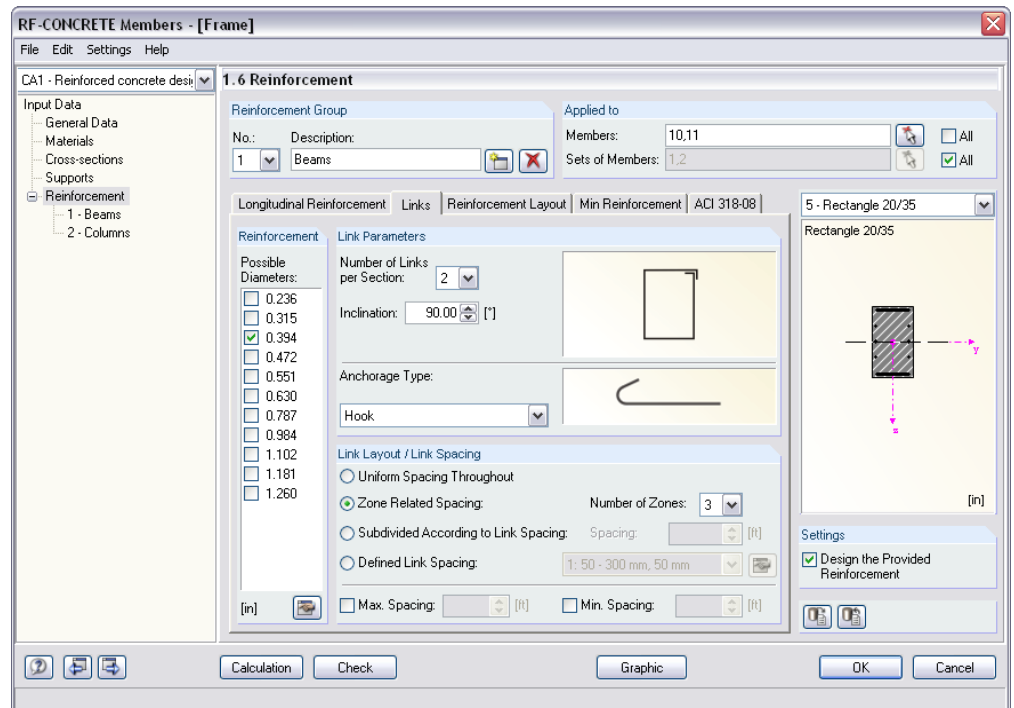


Figure 3.14: Table 1.6 Reinforcement, tab Links

#### 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 3.13).

#### 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 for nonpre-stressed members: ACI 318-08, 11.4.1.2 only allows for angles between 45° and 90°.

#### Anchorage Type

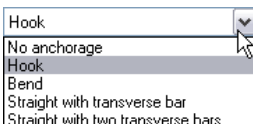
The list provides numerous possibilities for link anchorages affecting the determination of the anchorage length. The graphic to the right is dynamic, this means that modified settings are updated and displayed immediately in the graphic.

#### Link Layout / Link Spacing

This dialog section is only available if you create a reinforcement proposal.

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

If you select a *Zone Related Spacing*, you can use the list to the right to define how many zones with the same link layout can be assumed for the reinforcement proposal. In case only one





zone is specified, the program creates 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. When you specify two zones for example, RF-CONCRETE Members 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. The subdivision into further link zones is similar.

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

If you specify a *Defined Link Spacing*, you can select an entry from the list shown on the left. The [Edit] button allows you to adjust the entries or to create a new entry with user-defined link spacings.



Figure 3.15: 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 determined by the program and shown in the reinforcement proposal can be modified or added subsequently in table 3.2 *Shear Reinforcement Provided* (see chapter 5.2.2, page 52).



### 3.6.3 Reinforcement Layout

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

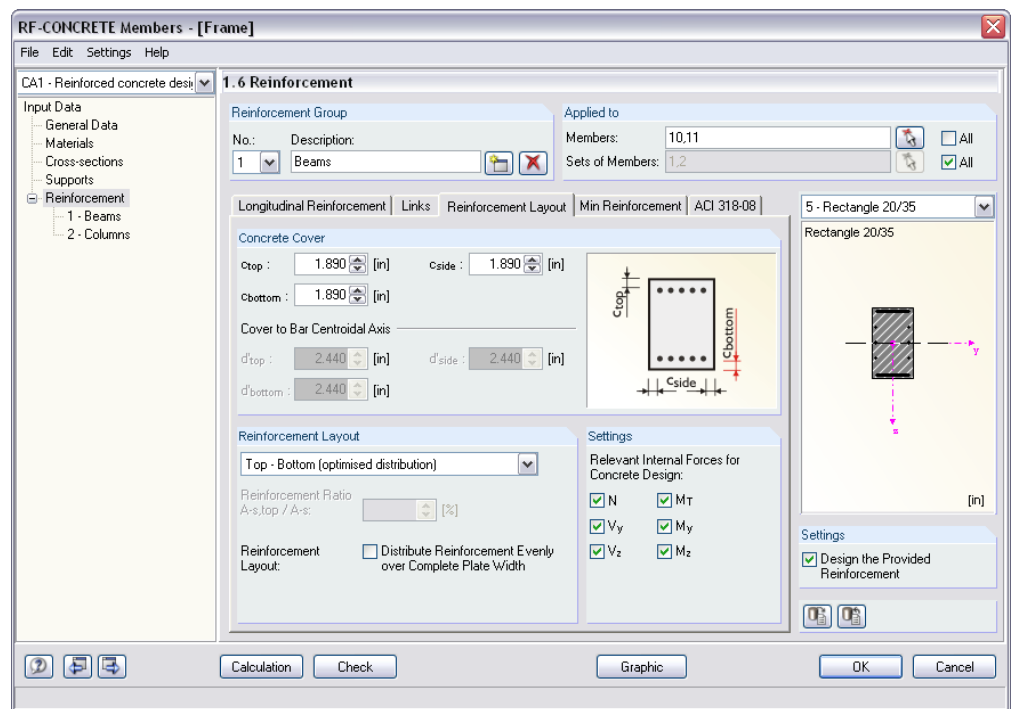


Figure 3.16: Table 1.6 *Reinforcement*, tab *Reinforcement Layout*

## 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 upper or lower input fields of the dialog section.

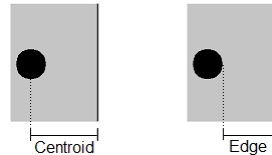
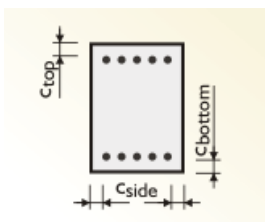


Figure 3.17: Relations of concrete cover



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 values  $c_c$  of the clear concrete cover. Based on these specifications and taking into account the applied rebar diameters, RF-CONCRETE Members determines the lever arm of internal forces.

"Top" and "bottom" are clearly defined by the position of the local member axes in RFEM. Specifications concerning the cover  $c_{side}$  are required to determine 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.

## 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

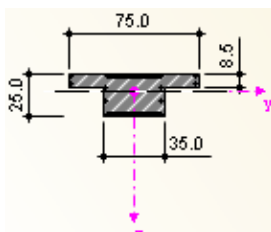
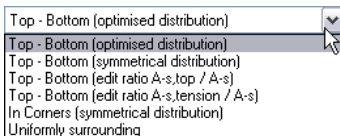
For the reinforcement layout *Top - Bottom (optimized distribution)* RF-CONCRETE Members also performs an optimization for biaxial bending.

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.



### 3.6.4 Min Reinforcement

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

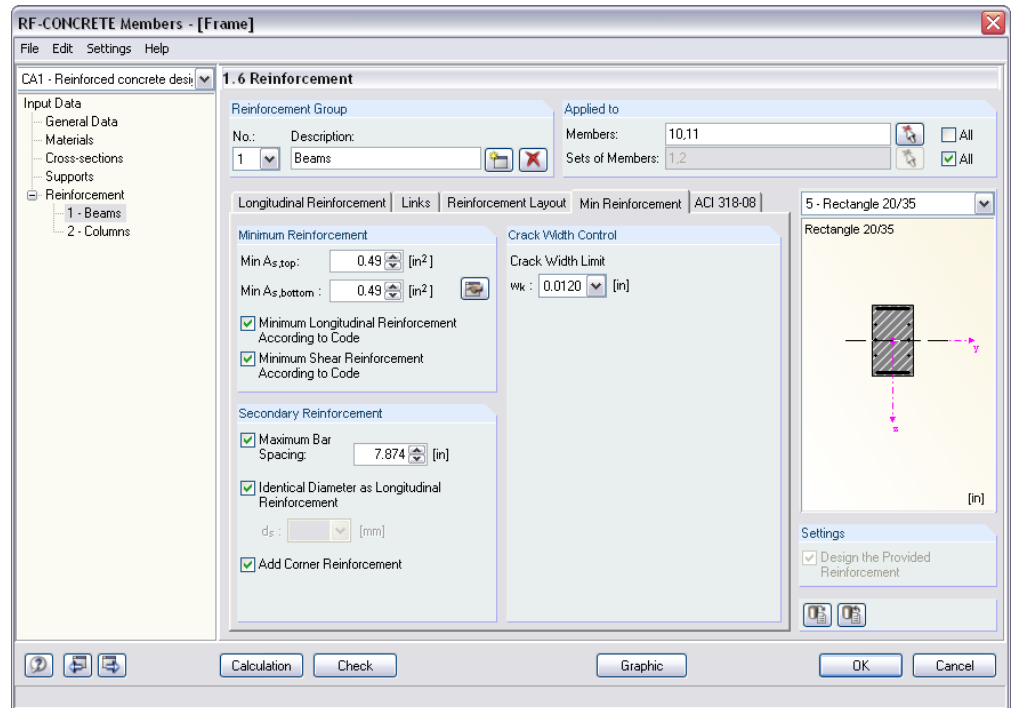


Figure 3.18: Table 1.6 Reinforcement, tab Min Reinforcement

#### 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 determine these cross-sectional areas from 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

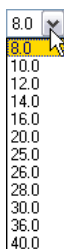
This dialog section is active only if you want to create a reinforcement proposal.

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 even possible to define a reinforcement outside the web of 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 *Crack Width Limit*  $w_{max}$  can be selected in the list. Depending on the environmental conditions, the permissible maximum crack widths for reinforced concrete members were presented by ACI Committee 224 in a report on cracking.

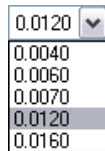


Figure 3.19: Permissible maximum crack widths

The crack width is controlled in the ACI Code by establishing a maximum center-to-center spacing  $s$  for the reinforcement closest to a tension surface. This spacing  $s$  is specified in ACI 318-08, 10.6.4, see page 13.

For structures with very aggressive exposure or structures that are supposed to be watertight, special considerations are required – such as Gergely-Lutz equation, see page 13.

### 3.6.5 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 19). This tab contains the standard-specific reinforcement specifications. In the following, the specifications for ACI 318-08 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.

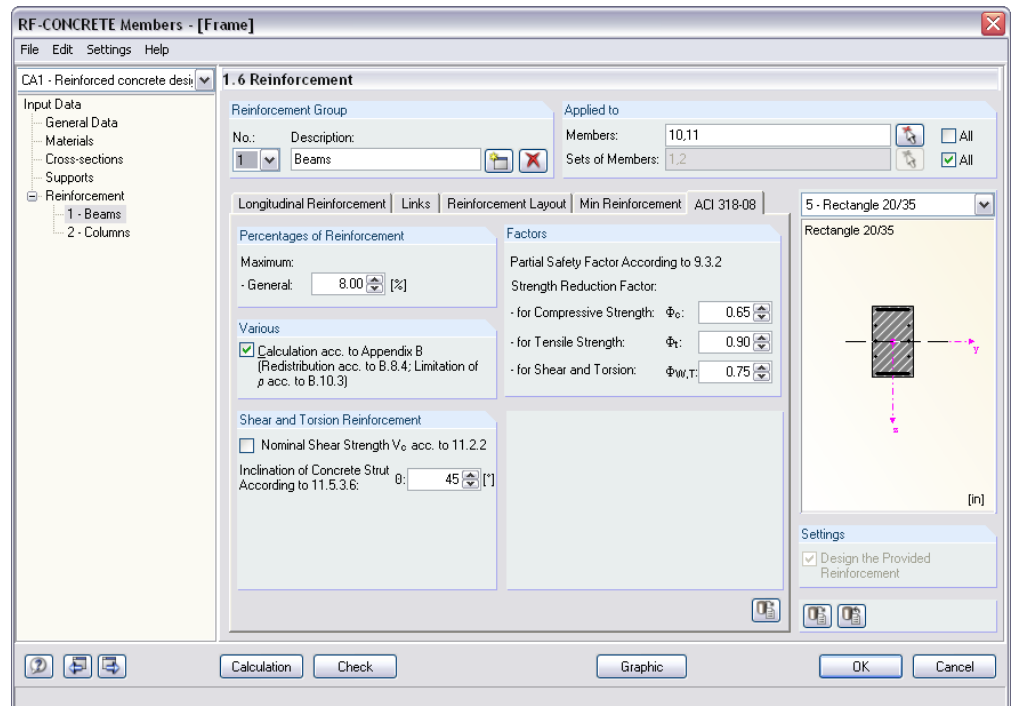


Figure 3.20: Table 1.6 Reinforcement, tab ACI 318-08

### Percentages of Reinforcement

The entry in this input field defines the general maximum reinforcement ratio for beams. The standard ACI 318-08, 10.9.1 recommends to use the value of  $A_{s,max} = 0.08 A_g$  for compression members, where  $A_g$  is the gross area of the cross section.

For flexural members and members whose axial load is less than  $0.10 f_c' A_g$ , the maximum reinforcement is limited by the net tensile strains  $\epsilon_t$ , which shall not be less than 0.004 at nominal strength.

### Various

For continuous beams there are two allowed methods of moment redistribution. The first one is described in ACI Code Section 8.4. This method permits factored moments calculated by elastic theory to be decreased at sections of maximum negative or maximum positive moments in any span of continuous members for any loading arrangement. The amount by which moments can be decreased cannot exceed 1000  $\epsilon_t$  percent, with maximum of 20 percent. The redistribution for values of  $\epsilon_t < 0.0075$  is conservatively prohibited.

Tick the check box *Calculation acc. to Appendix B* to use alternative provisions. The maximum allowable percentages to increase or decrease of negative moments are given in ACI 318-08, B.8.4.1 by the expression:

$$20 \left( 1 - \frac{\rho - \rho'}{\rho_b} \right)$$

### Shear Reinforcement

With the option *Nominal Shear Strength  $V_c$  acc. to 11.2.2* the program computes the shear strength provided by the concrete using the following equation:

$$V_c = \left( 1.9 \lambda \sqrt{f_c'} + 2500 \rho_w \frac{V_u d}{M_u} \right) b_w d \leq 3.5 \lambda \sqrt{f_c'} b_w d \quad \text{ACI Code, eq. (11-5)}$$

Otherwise the simpler equation  $V_c = 2 \lambda \sqrt{f_c'} b_w d$  is used according to ACI 318-08, 11.2.1.1. For more details see page 11.

The input field defines the allowed value of the inclination of concrete struts. In case user-defined angles are beyond the allowed limits mentioned in the standard, an error message appears.

### Factors

The three input fields define the *Strength Reduction Factor* for compressive strength  $\phi_c$ , for tensile strength  $\phi_t$  and for shear and torsion  $\phi_{w,t}$ . The values according to ACI 318-08, 9.3.2 are preset.

The strength reduction factor for members in the transition zone (where the net tensile strain in the extreme tension steel at nominal strength  $\epsilon_t$  is between the limits for compression-controlled and tension-controlled sections) is determined by a linear interpolation between  $\phi_c$  and  $\phi_t$ .

### 3.6.6 Tapered

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

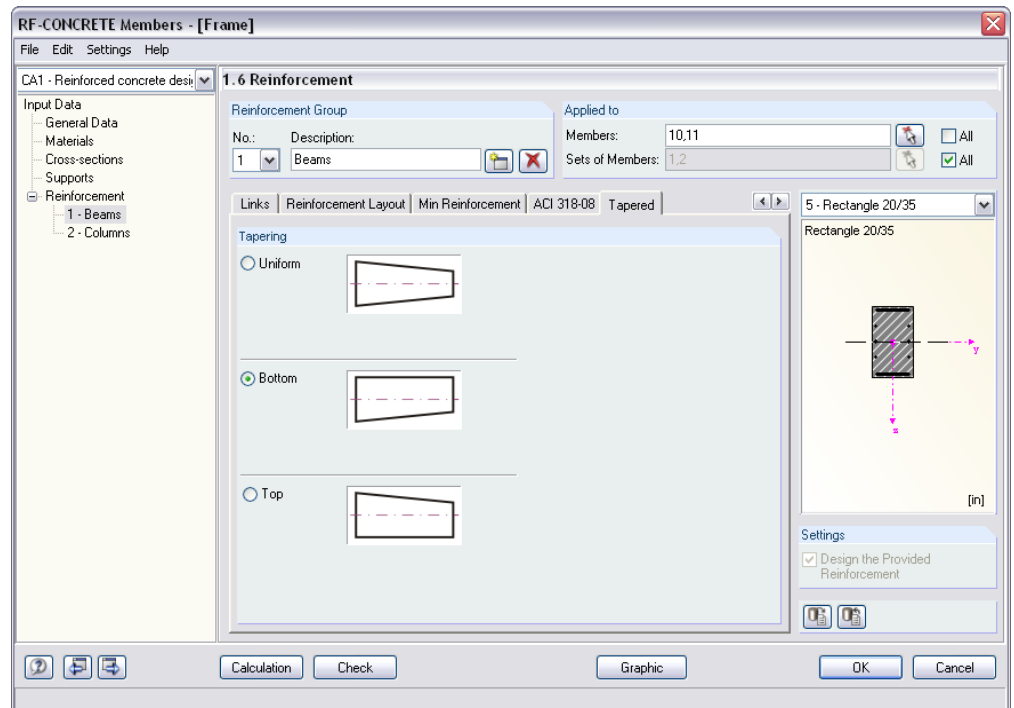


Figure 3.21: Table 1.6 Reinforcement, tab Tapered

RF-CONCRETE Members 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 RFEM displays a corresponding error message before the RFEM calculation is started.

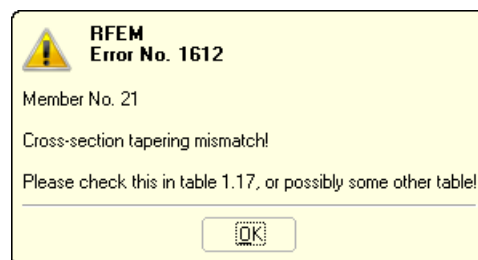


Figure 3.22: Error message in case of incompatible tapered cross-sections

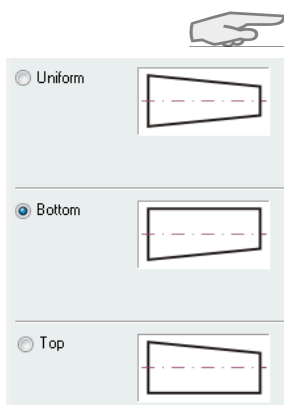
Tapered sets of members will be designed only if the entire set of members has a linear cross-sectional profile.

#### Tapering

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

- Uniform
- Bottom
- Top

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



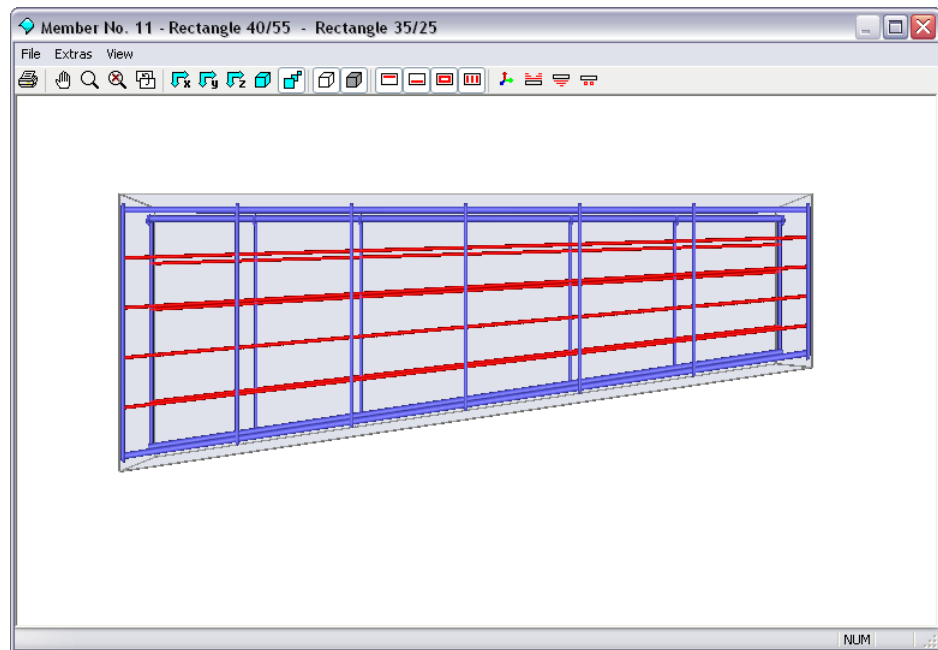


Figure 3.23: Taper with inclined bottom side

## 4. Calculation

Calculation

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

### 4.1 Plausibility Check

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 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 RF-CONCRETE Members.

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

You can also start the calculation for RF-CONCRETE Members out of the RFEM 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 RFEM,

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

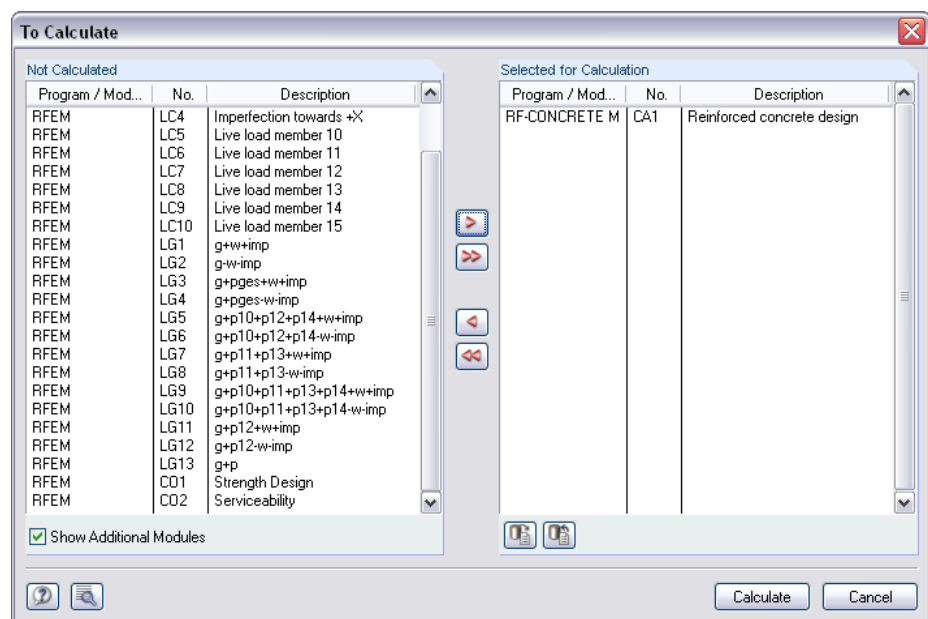


Figure 4.2: Dialog box *To Calculate*



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

To transfer the selected RF-CONCRETE Members 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 RFEM toolbar to calculate a selected RF-CONCRETE Members case directly: Select the relevant design case and click the button [Results on/off].

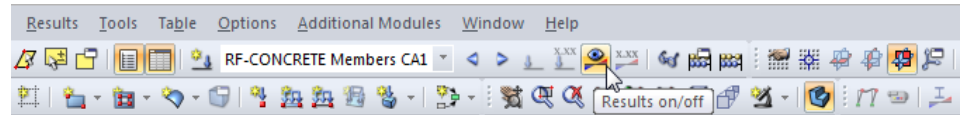


Figure 4.3: Direct calculation of an RF-CONCRETE Members design case in RFEM

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

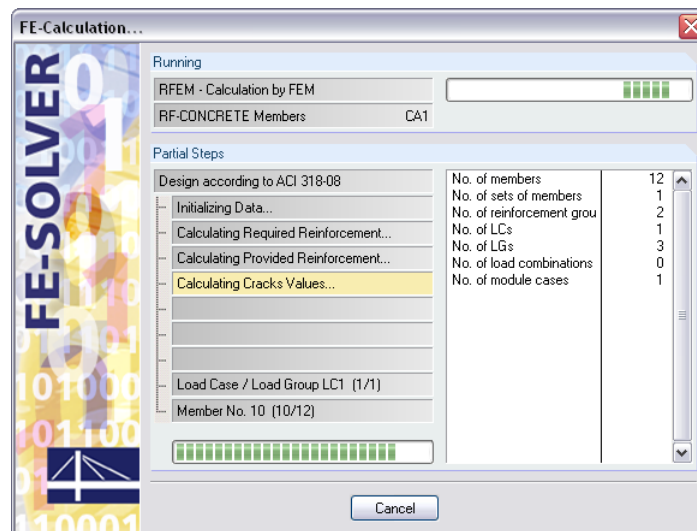


Figure 4.4: Design with RF-CONCRETE Members

## 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 serviceability limit state 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 navigator of RF-CONCRETE Members. 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 RF-CONCRETE Members, 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 62.

### 5.1 Required Reinforcement

#### 5.1.1 Required Reinforcement by Cross-section

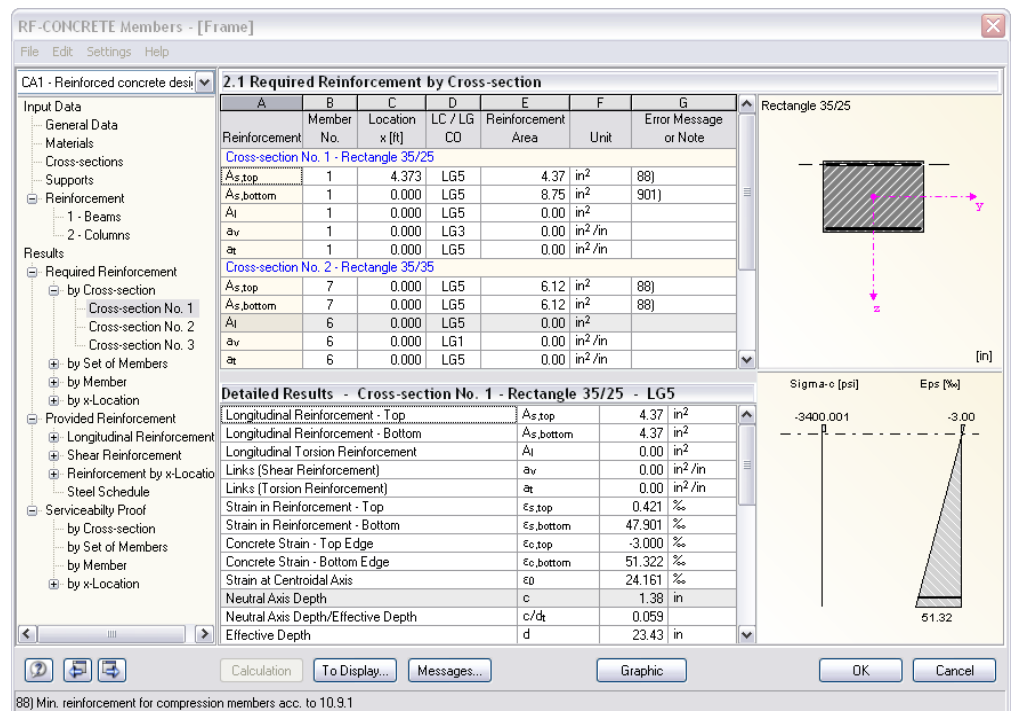


Figure 5.1: Table 2.1 *Required Reinforcement by Cross-section*

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* (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 in
$a_{sw,T \text{ link}}$	Area of required link reinforcement for absorption of torsional moment, referring to standard length of 1 in

Table 5.1: Longitudinal and link reinforcements

  
Top and Bot-  
tom layer

To Display...

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 RFEM 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.

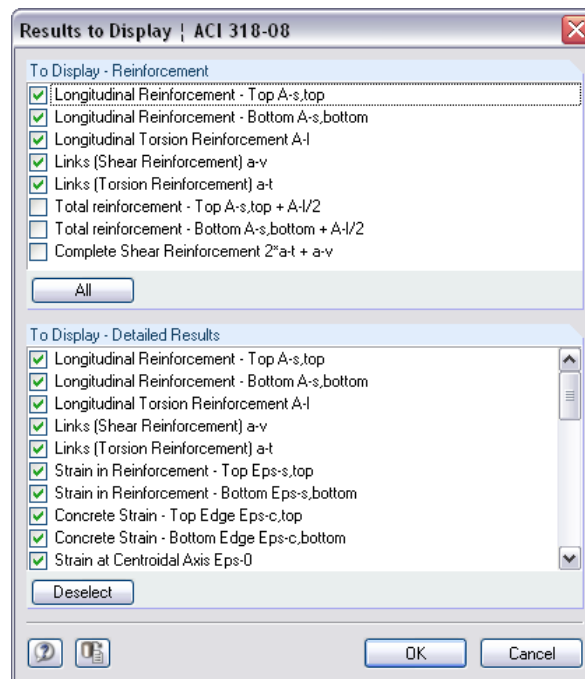


Figure 5.2: Dialog box *Results to Display*

### 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 RFEM 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 or load 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 75 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...

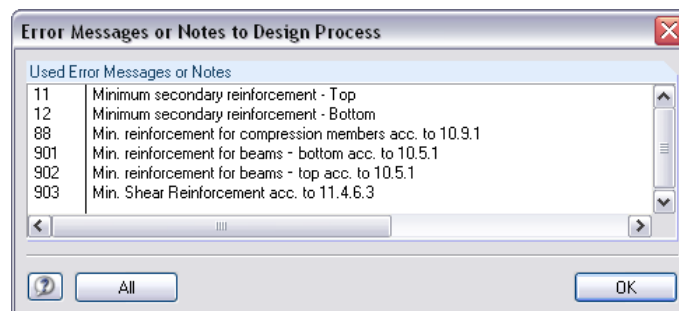


Figure 5.3: Dialog box *Error Messages or Notes*

### 5.1.2 Required Reinforcement by Set of Members

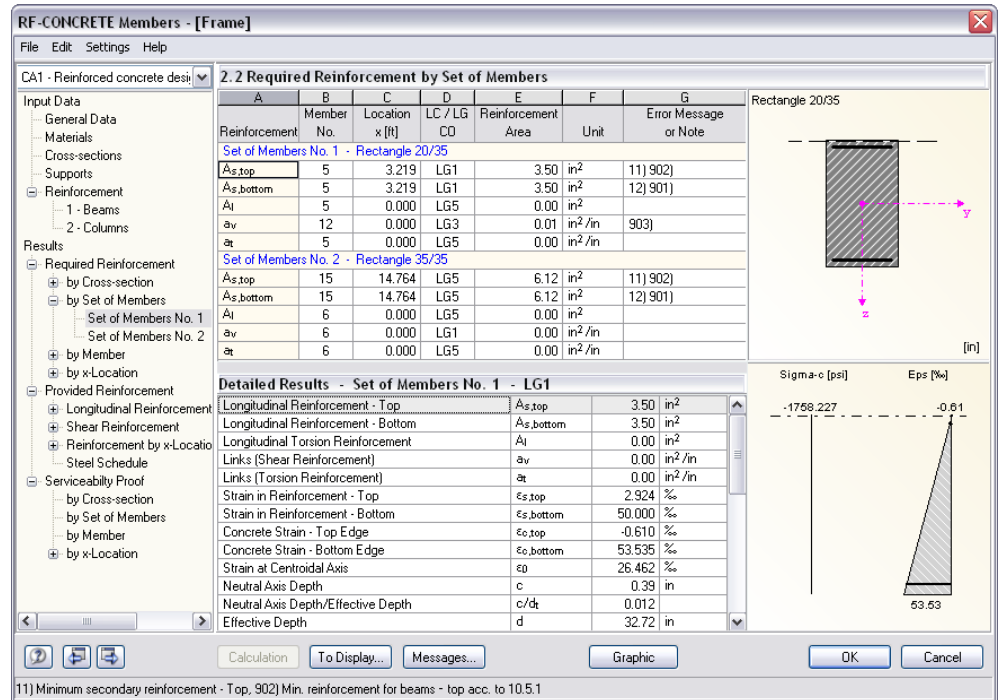


Figure 5.4: Table 2.2 Required Reinforcement by Set of Members

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

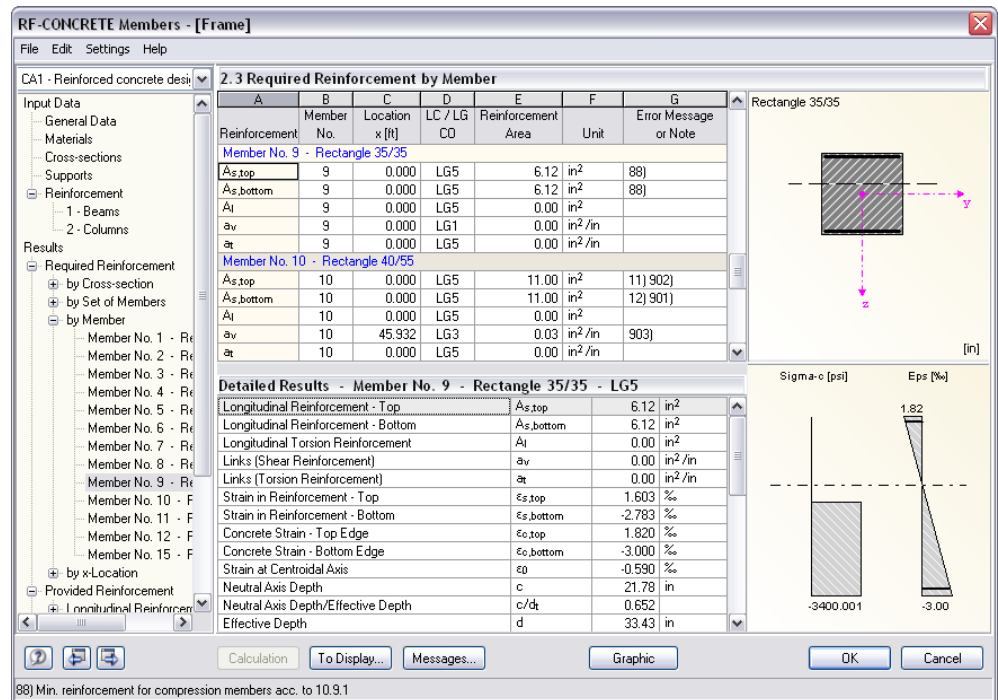


Figure 5.5: Table 2.3 Required Reinforcement by Member

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

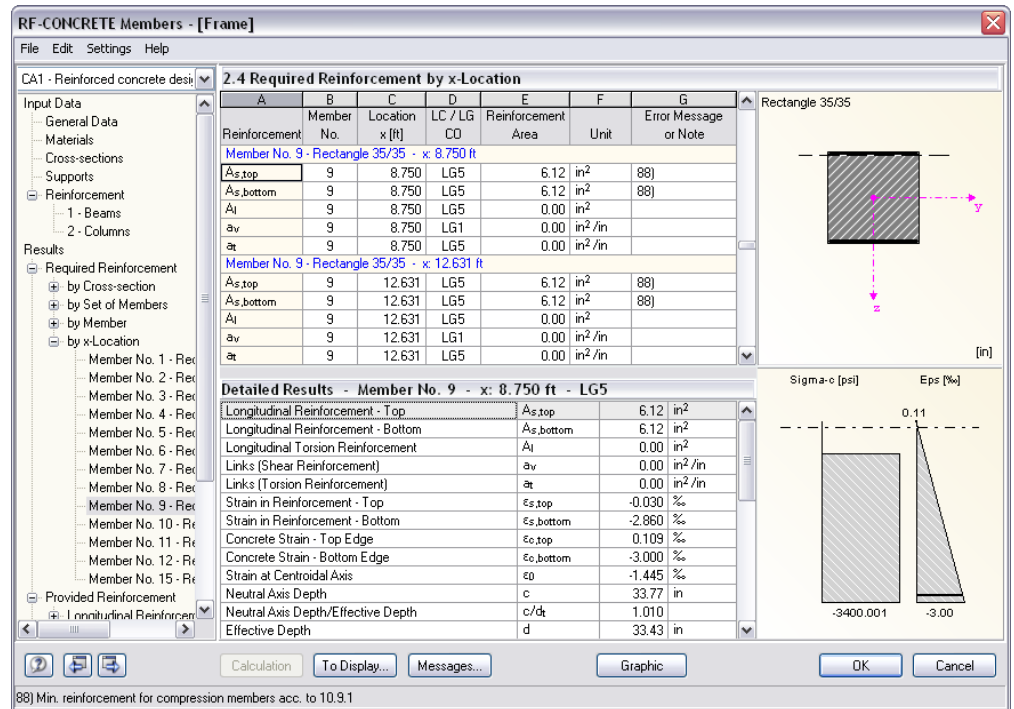


Figure 5.6: Table 2.4 Required Reinforcement by x-Location

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

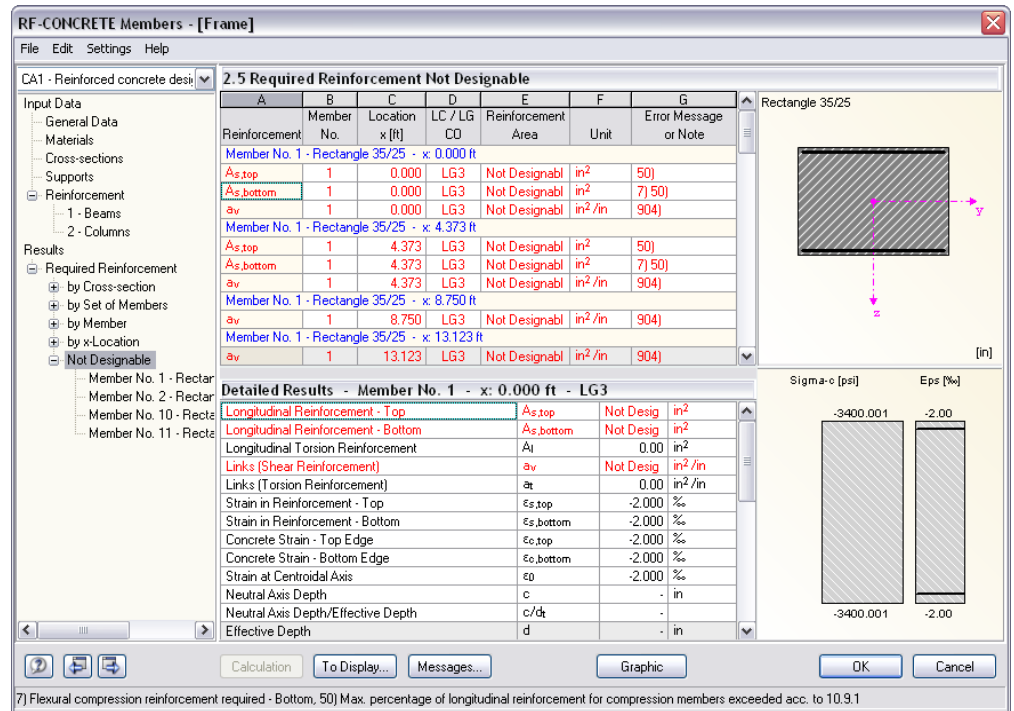


Figure 5.7: Table 2.5 Required Reinforcement Not Designable

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

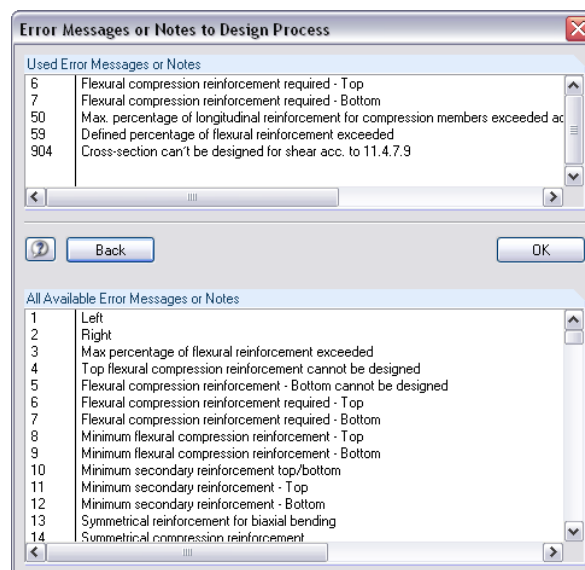


Figure 5.8: Dialog box Error Messages or Notes to Design Process

All

Click the [All] button in this dialog box to show all notes available for RF-CONCRETE Members.

## 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 30), and if no design problems were detected by the program (see chapter 5.1.5, page 47). Also the serviceability limit state design as well as the non-linear calculation require the determination of a provided reinforcement.

RF-CONCRETE Members 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 tables of the *Reinforcement Provided* so that you can adjust the diameter, number, position and length of the individual reinforcement groups to the respective requirements.

### 5.2.1 Longitudinal Reinforcement Provided

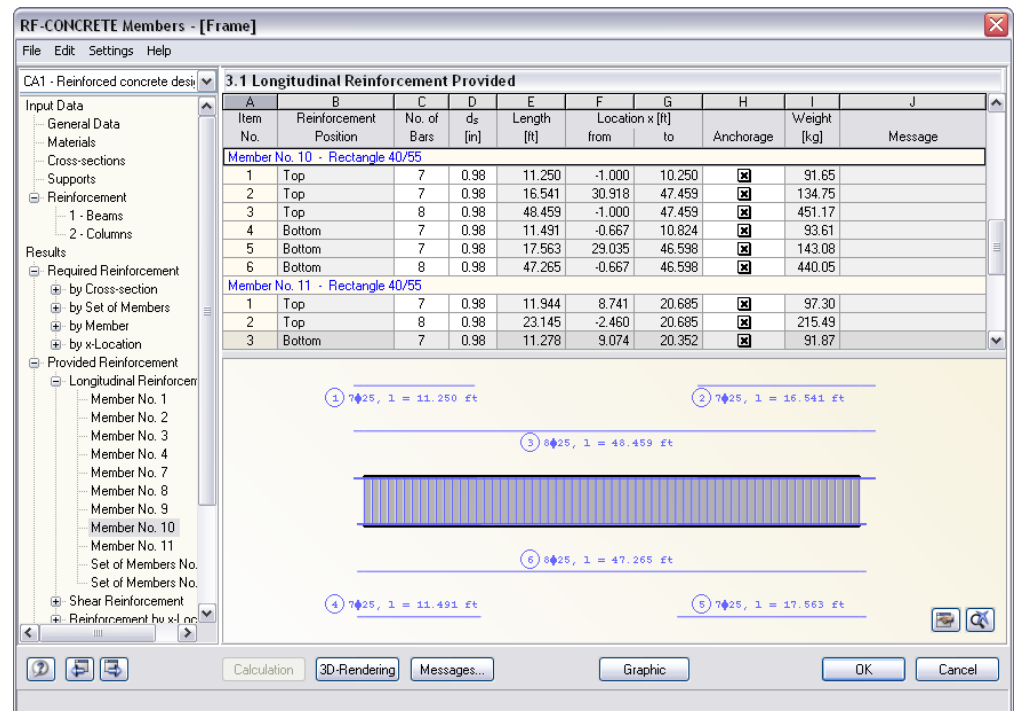


Figure 5.9: Table 3.1 Longitudinal Reinforcement Provided

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 ACI 318-08, 12.11.1, it is required to arrange a minimum reinforcement to supports that are assumed to be pinned, and at least one-third of the positive moment steel in simple members and one-fourth in continuous members must be continued uninterrupted along the same face of the beam a distance at least 6 in.



### 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 RF-CONCRETE Members takes into account the user specifications entered in table 1.6 *Reinforcement*, tab *Reinforcement Layout* (see chapter 3.6.3, page 34).

### 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.

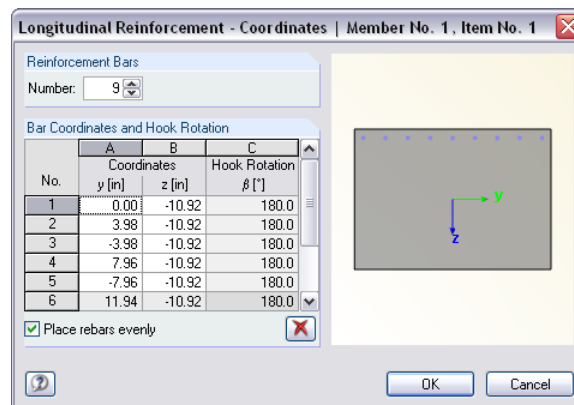
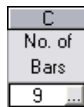


Figure 5.10: Dialog box *Longitudinal Reinforcement - Coordinates*

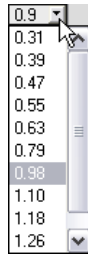


The *Number of Reinforcement Bars* can be set manually by using the spin buttons or by entering another number. The numerous input rows in the dialog section below allow you to adjust the position of each rebar. To delete a row selected in the lower section, click the [Delete] button.

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  $\beta$  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.



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

**d<sub>s</sub>**

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 of RFEM ( $x = 0$ ). 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 51).

**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.

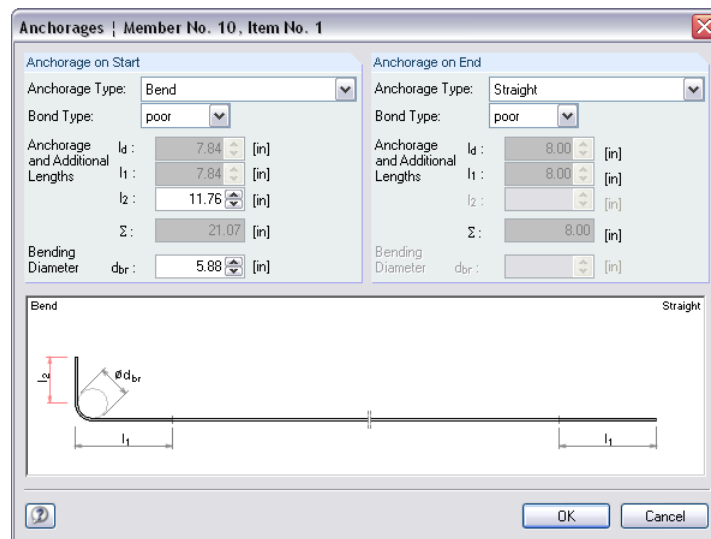


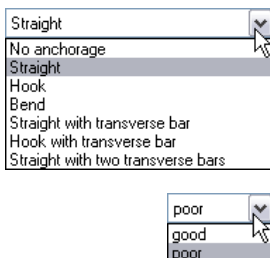
Figure 5.11: Dialog box *Anchorage*

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 3.6.1 on page 31. RF-CONCRETE Members 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 affect reinforcement location factor  $\Psi_t$  according to ACI Code, 12.2.4 (a).

For control reasons the program displays the design value of the anchorage length  $l_d$ . The lengths  $l_1$  of the *Anchorage and Additional Lengths* is determined by equation (12-1) according to ACI 318-08, 12.2.3, but it cannot be modified.

$l_2$  is the length of anchorage for hooks and bends, in accordance with ACI 318-08, 12.5. For example for 90-degree hook it should be at least  $12 d_b$ .



The required *Bending Diameter*  $d_{br}$  is specified according to ACI 318-08, 7.2, and can be adjusted, if necessary.

The total anchorage length  $\Sigma$  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

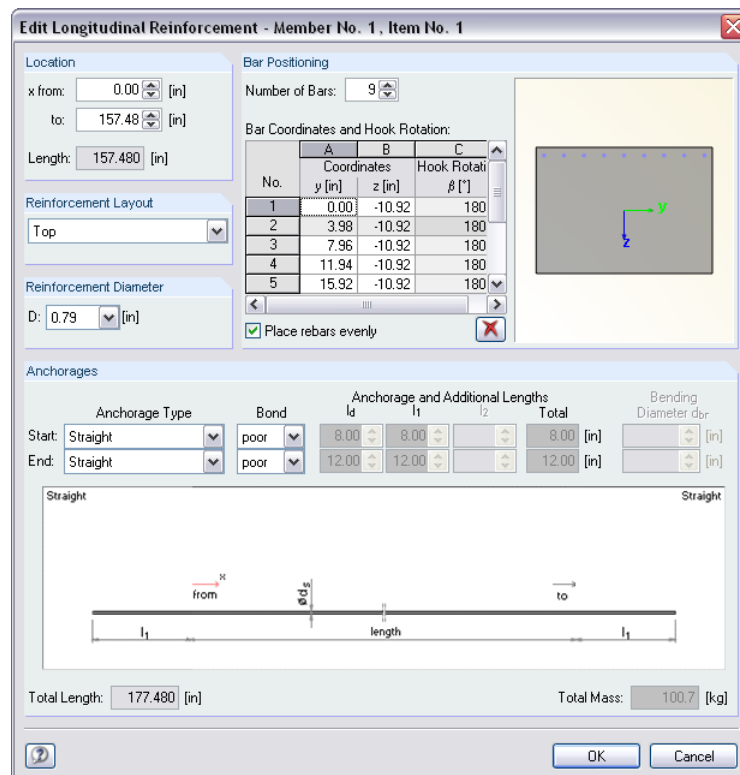
If a number is displayed in the final column, a special condition is the reason. The footers are explained in the status bar.

Messages...

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 44).

## Change reinforcement proposal

The graphic in the lower table section of table 3.1 represents the reinforcement including item members. 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.

No.	Coordinates		Hook Rotati $\beta$ [°]
	y [in]	z [in]	
1	0.00	-10.92	180
2	3.98	-10.92	180
3	7.96	-10.92	180
4	11.94	-10.92	180
5	15.92	-10.92	180

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 *Anchorages*.

Calculation

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

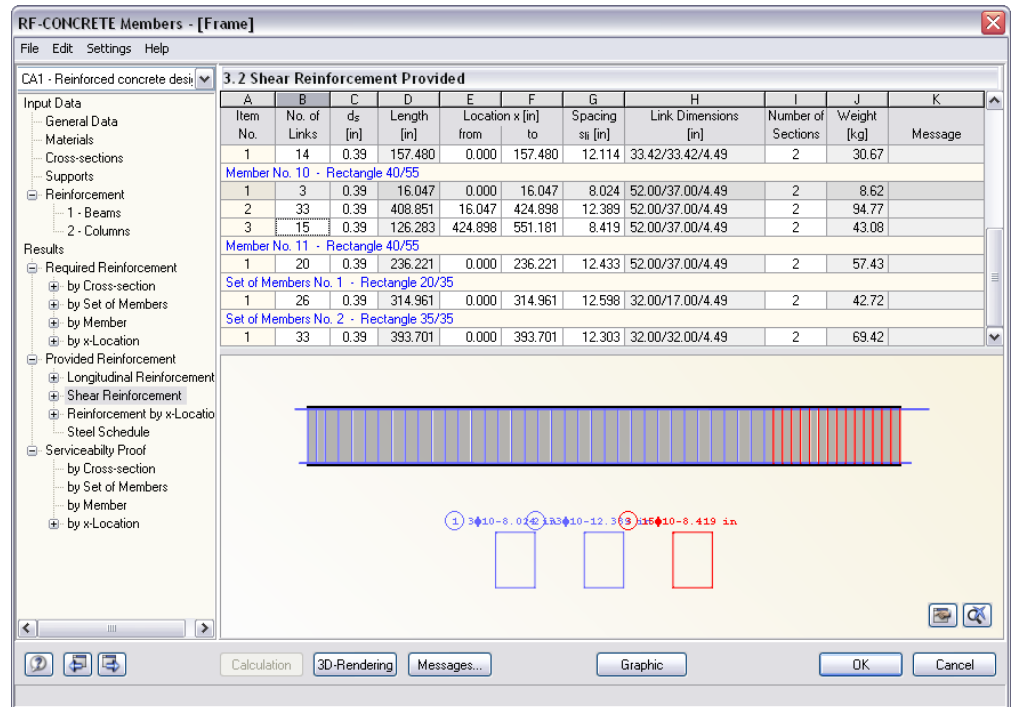


Figure 5.13: Table 3.2 Shear Reinforcement Provided

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 ACI 318-08, 11.4.5.1, the maximum spacing of stirrups shall not exceed  $d/2$ , nor 24 in.

### 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*.

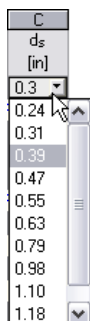
### No. of Links

When RF-CONCRETE Members determines the link reinforcement, it takes into account the user specifications entered in table 1.6 *Reinforcement*, tab *Links* (see chapter 3.6.2, page 32).

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 54).



### Location $x$ from ... to

These values represent the start and end positions of the link zones. They refer to the member's start node of RFEM ( $x = 0$ ). 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. RF-CONCRETE Members will automatically create a new link zone.

### Spacing $s_{li}$

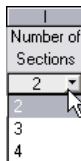
The proposed link spacing takes into account the specifications defined in table 1.6 *Reinforcement*, tab *Link* (see chapter 3.6.2, page 32). This value can be edited: Click in the corresponding cell and enter another spacing. The number of links (column B) will be adjusted automatically. The exact 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". RF-CONCRETE Members 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 3.6.2, page 32). Use the list to change the number of sections.

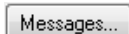


### Weight

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

### Message

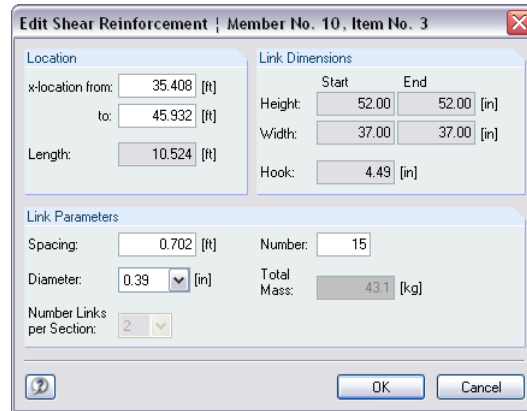
If a number is displayed in the final column, a special condition is the reason. The footers are explained in the status bar.



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 44).

### 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.



Location		Link Dimensions	
x-location from:	35.408 [ft]	Start	End
to:	45.932 [ft]	Height:	52.00 [in]
Length:	10.524 [ft]	Width:	37.00 [in]
		Hook:	4.49 [in]

Link Parameters	
Spacing:	0.702 [ft]
Diameter:	0.39 [in]
Number:	15
Total Mass:	43.1 [kg]
Number Links per Section:	2

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 the *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.

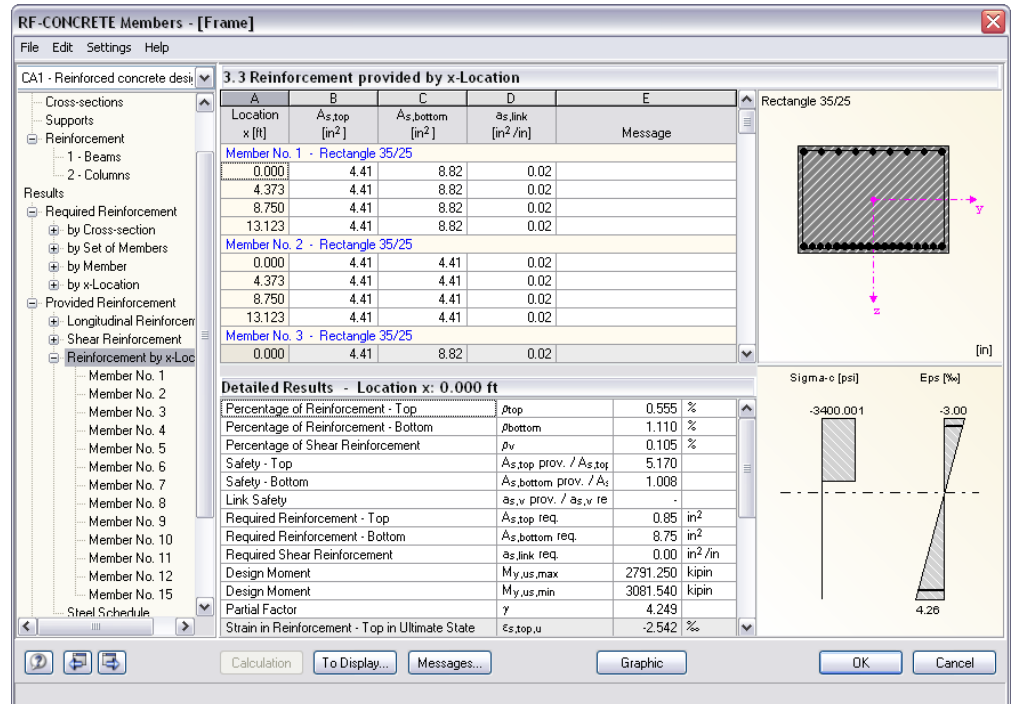


Figure 5.15: Table 3.3 Reinforcement provided by x-Location

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.

#### $a_{s,link}$

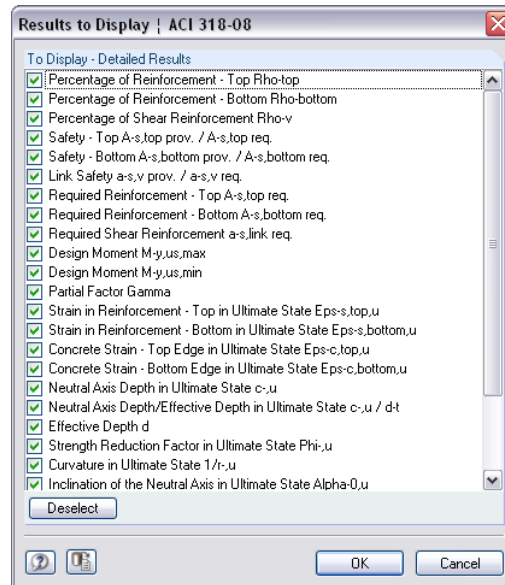
This column shows the area of the provided link reinforcement.



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 (i.e. the active location selected in the upper dialog section) including all design-relevant parameters.

To Display...

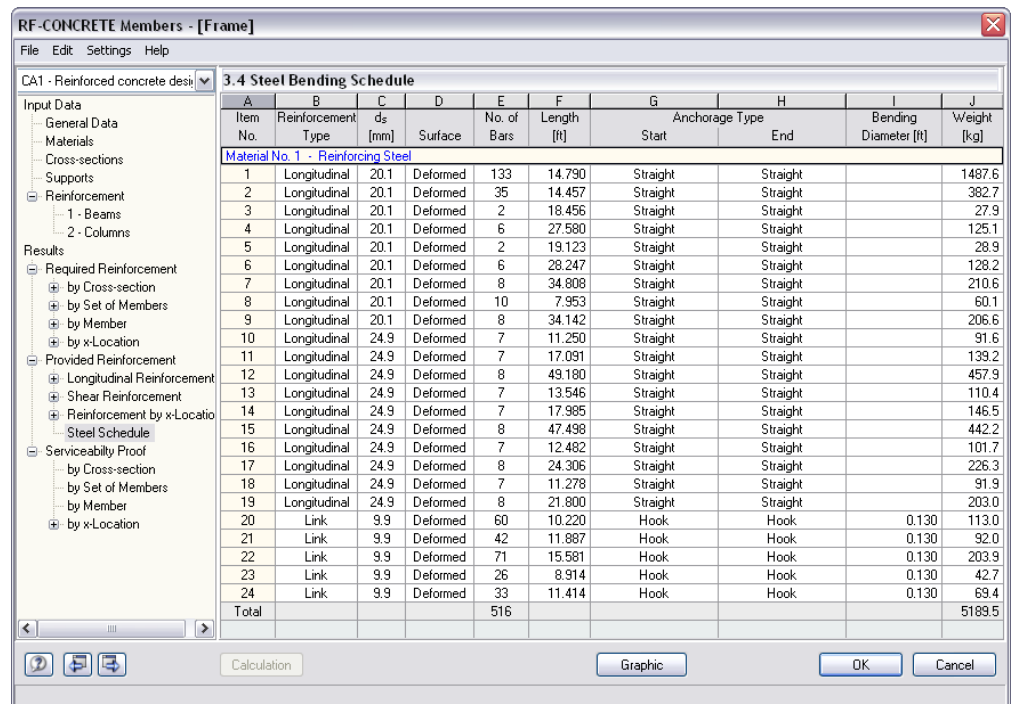
To reduce the displayed result parameters, use the button [To Display].

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 Schedule

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



3.4 Steel Bending Schedule									
A	B	C	D	E	F	G	H	I	J
Item No.	Reinforcement Type	$d_s$ [mm]	Surface	No. of Bars	Length [lt]	Anchorage Type		Bending Diameter [lt]	Weight [kg]
<b>Material No. 1 - Reinforcing Steel</b>									
1	Longitudinal	20.1	Deformed	133	14.790	Straight	Straight		1487.6
2	Longitudinal	20.1	Deformed	35	14.457	Straight	Straight		382.7
3	Longitudinal	20.1	Deformed	2	18.456	Straight	Straight		27.9
4	Longitudinal	20.1	Deformed	6	27.580	Straight	Straight		125.1
5	Longitudinal	20.1	Deformed	2	19.123	Straight	Straight		28.9
6	Longitudinal	20.1	Deformed	6	28.247	Straight	Straight		128.2
7	Longitudinal	20.1	Deformed	8	34.808	Straight	Straight		210.6
8	Longitudinal	20.1	Deformed	10	7.953	Straight	Straight		60.1
9	Longitudinal	20.1	Deformed	8	34.142	Straight	Straight		206.6
10	Longitudinal	24.9	Deformed	7	11.250	Straight	Straight		91.6
11	Longitudinal	24.9	Deformed	7	17.091	Straight	Straight		139.2
12	Longitudinal	24.9	Deformed	8	49.180	Straight	Straight		457.9
13	Longitudinal	24.9	Deformed	7	13.546	Straight	Straight		110.4
14	Longitudinal	24.9	Deformed	7	17.985	Straight	Straight		146.5
15	Longitudinal	24.9	Deformed	8	47.498	Straight	Straight		442.2
16	Longitudinal	24.9	Deformed	7	12.482	Straight	Straight		101.7
17	Longitudinal	24.9	Deformed	8	24.306	Straight	Straight		226.3
18	Longitudinal	24.9	Deformed	7	11.278	Straight	Straight		91.9
19	Longitudinal	24.9	Deformed	8	21.800	Straight	Straight		203.0
20	Link	9.9	Deformed	60	10.220	Hook	Hook	0.130	113.0
21	Link	9.9	Deformed	42	11.887	Hook	Hook	0.130	92.0
22	Link	9.9	Deformed	71	15.581	Hook	Hook	0.130	203.9
23	Link	9.9	Deformed	26	8.914	Hook	Hook	0.130	42.7
24	Link	9.9	Deformed	33	11.414	Hook	Hook	0.130	69.4
Total				516					5189.5

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<sub>s</sub>**

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**

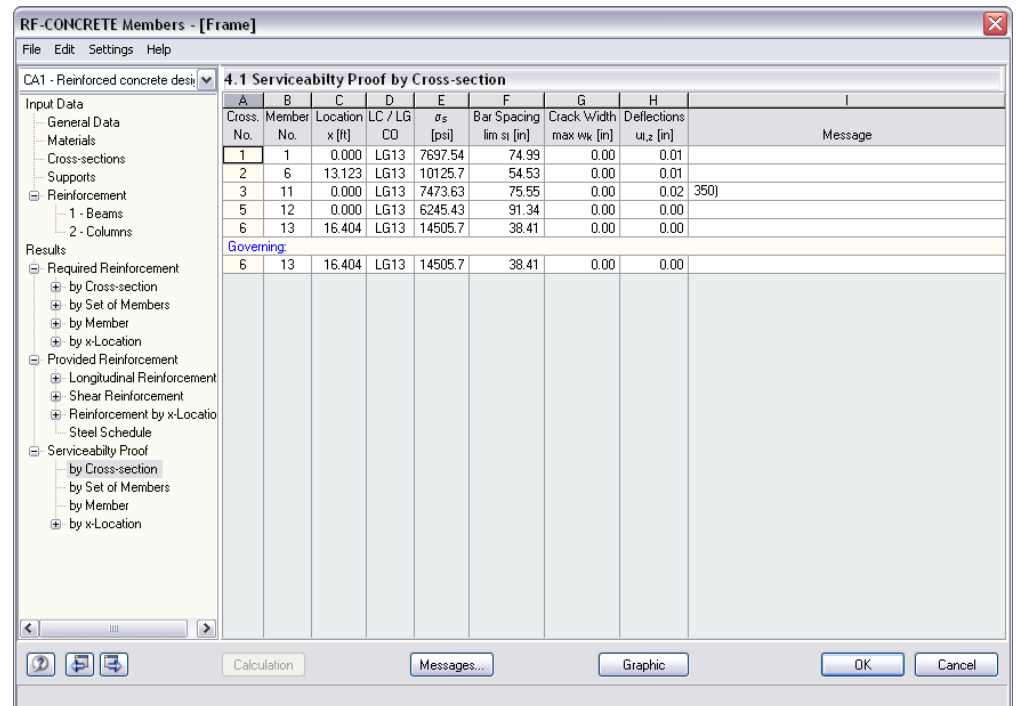
The final table row of the steel schedule shows the total number of rebars and the mass of steel that is totally required. It is determined from the values of the individual items above.

## 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 20), and if no design problems were detected by the program (see chapter 5.1.5, page 47 and chapter 5.2.3, page 55).

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
Cross. No.	Member No.	Location x [ft]	LC / LG / CO	$\sigma_s$ [psi]	Bar Spacing lim s <sub>1</sub> [in]	Crack Width max w <sub>k</sub> [in]	Deflections u <sub>1,2</sub> [in]	Message
1	1	0.000	LG13	7697.54	74.99	0.00	0.01	
2	6	13.123	LG13	10125.7	54.53	0.00	0.01	
3	11	0.000	LG13	7473.63	75.55	0.00	0.02	350)
5	12	0.000	LG13	6245.43	91.34	0.00	0.00	
6	13	16.404	LG13	14505.7	38.41	0.00	0.00	
<b>Governing:</b>								
6	13	16.404	LG13	14505.7	38.41	0.00	0.00	

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 3.6.4, page 35), the provided reinforcement and the internal forces of the governing actions.

#### 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 or load combinations that are decisive for the individual designs.

**$\sigma_s$** 

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:

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

**Bar Spacing lim  $s_l$** 

The allowable rebar spacings, in accordance with the simplified crack control method described in ACI 318-08, 10.6.4, are limited by the equation (10-4), see page 13 in this manual.

**Crack Width max  $w_k$** 

The characteristic crack width is determined according to the GERGELY-LUTZ equation (see page 13).

$$w = 0.076 \beta f_s \sqrt[3]{d_c A}$$

**Deflections  $u_{l,z}$** 

The penultimate table column shows the absolute values of the calculated deformations in direction of the local member axes  $z$ .

**Message**

The final column indicates design problems or shows notes referring to difficulties occurred during the analysis. The footers are explained in the status bar.

Messages...

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.

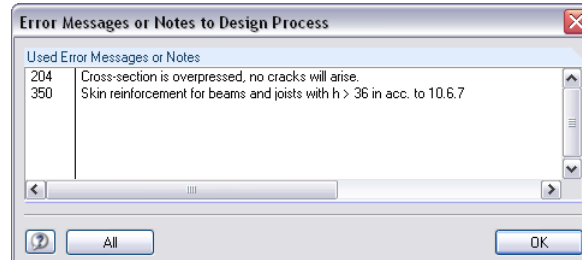
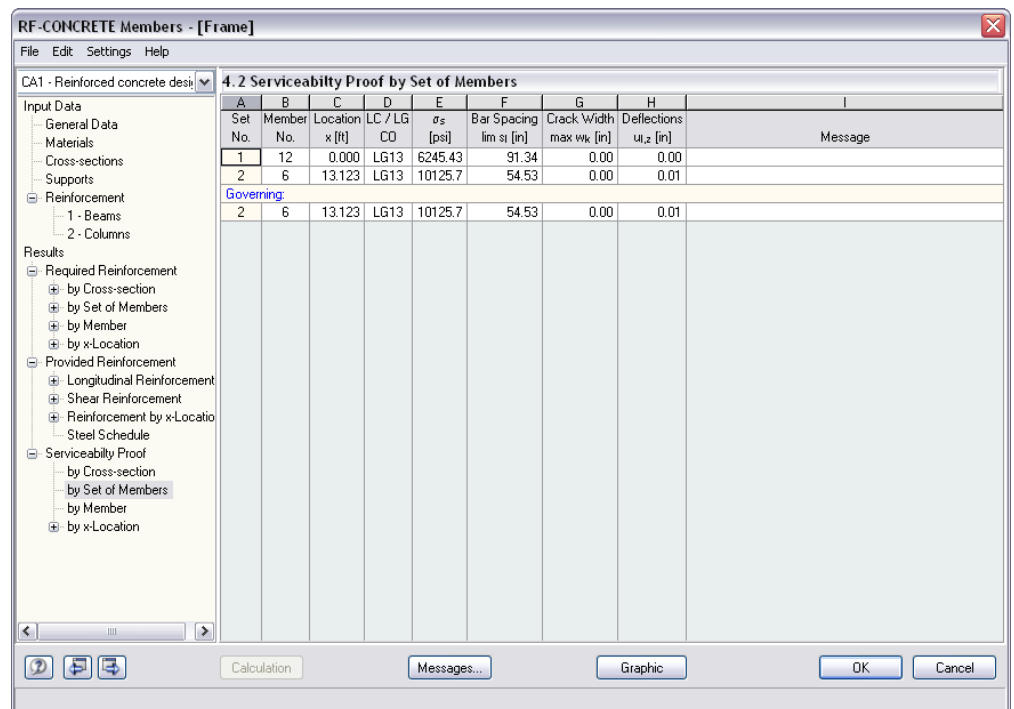


Figure 5.19: Dialog box *Error Messages or Notes to Design Process*

### 5.3.2 Serviceability Proof by Set of Members



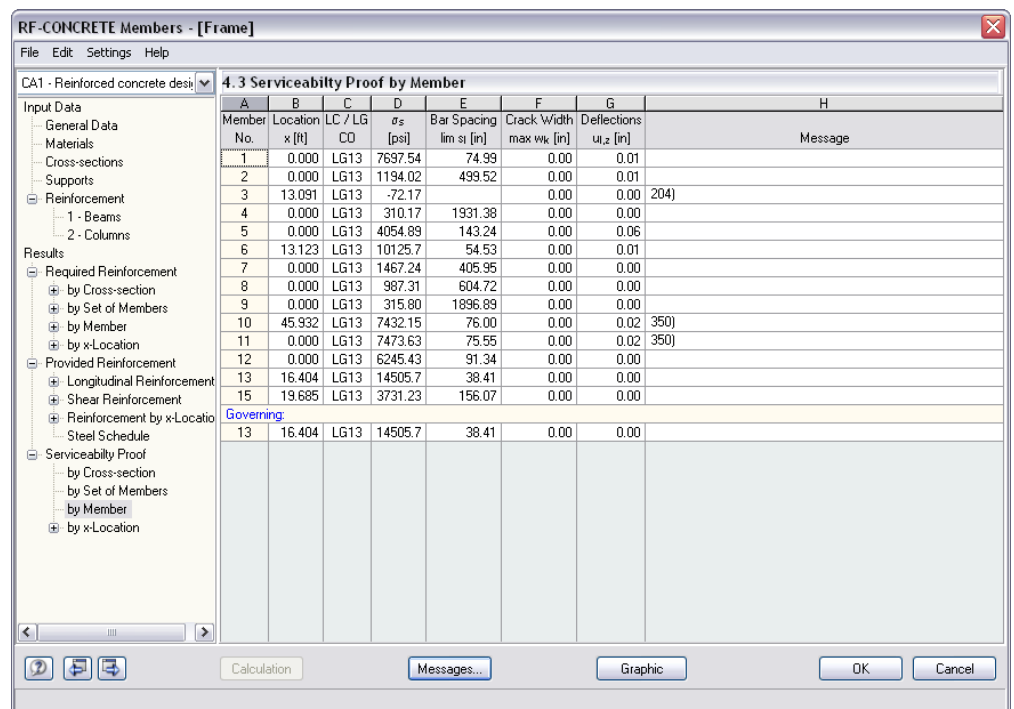
A	B	C	D	E	F	G	H	I
Set No.	Member No.	Location x [ft]	LC / LG CO	$\sigma_s$ [psi]	Bar Spacing lim s1 [in]	Crack Width max wk [in]	Deflections u1,2 [in]	Message
1	12	0.000	LG13	6245.43	91.34	0.00	0.00	
2	6	13.123	LG13	10125.7	54.53	0.00	0.01	
<b>Governing:</b>								
2	6	13.123	LG13	10125.7	54.53	0.00	0.01	

Figure 5.20: 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

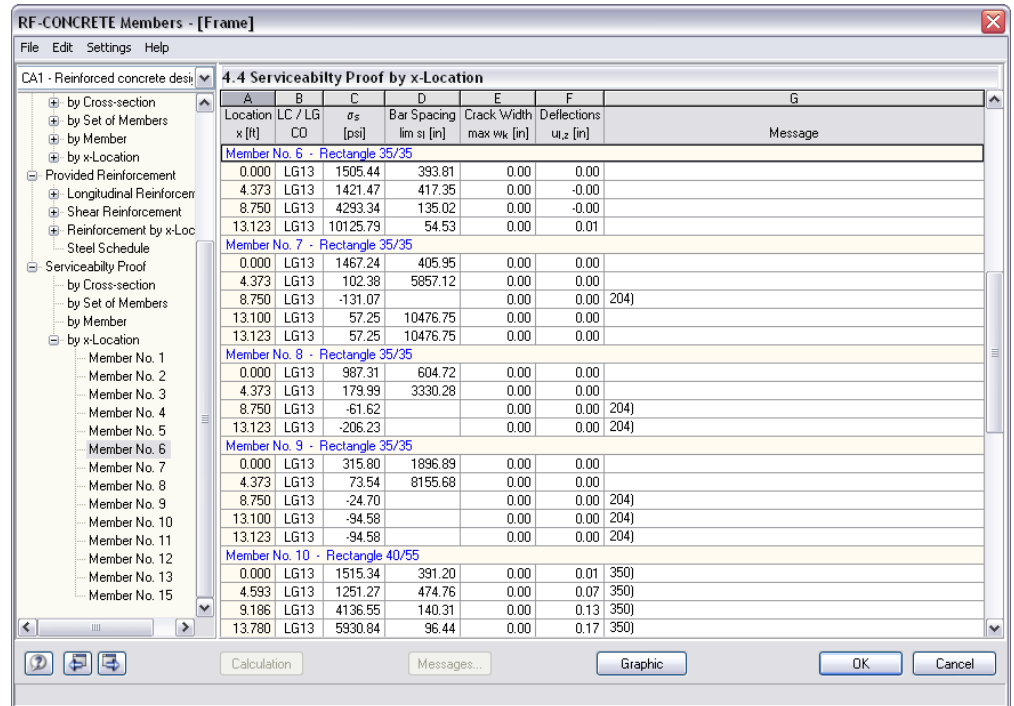


A	B	C	D	E	F	G	H
Member No.	Location x [ft]	LC / LG CO	$\sigma_s$ [psi]	Bar Spacing lim s1 [in]	Crack Width max wk [in]	Deflections u1,2 [in]	Message
1	0.000	LG13	7697.54	74.99	0.00	0.01	
2	0.000	LG13	1194.02	499.52	0.00	0.01	
3	13.091	LG13	-72.17		0.00	0.00	204)
4	0.000	LG13	310.17	1931.38	0.00	0.00	
5	0.000	LG13	4054.89	143.24	0.00	0.06	
6	13.123	LG13	10125.7	54.53	0.00	0.01	
7	0.000	LG13	1467.24	405.95	0.00	0.00	
8	0.000	LG13	987.31	604.72	0.00	0.00	
9	0.000	LG13	315.80	1896.89	0.00	0.00	
10	45.932	LG13	7432.15	76.00	0.00	0.02	350)
11	0.000	LG13	7473.63	75.55	0.00	0.02	350)
12	0.000	LG13	6245.43	91.34	0.00	0.00	
13	16.404	LG13	14505.7	38.41	0.00	0.00	
15	19.685	LG13	3731.23	156.07	0.00	0.00	
<b>Governing:</b>							
13	16.404	LG13	14505.7	38.41	0.00	0.00	

Figure 5.21: Table 4.3 Serviceability Proof by Member

This results table shows the output for the crack width designs sorted by members. The 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



A	B	C	D	E	F	G
Location x [ft]	LC / LG CD	$\sigma_s$ [psi]	Bar Spacing lim s1 [in]	Crack Width max wk [in]	Deflections u1,z [in]	Message
<b>Member No. 6 - Rectangle 35/35</b>						
0.000	LG13	1505.44	393.81	0.00	0.00	
4.373	LG13	1421.47	417.35	0.00	-0.00	
8.750	LG13	4293.34	135.02	0.00	-0.00	
13.123	LG13	10125.79	54.53	0.00	0.01	
<b>Member No. 7 - Rectangle 35/35</b>						
0.000	LG13	1467.24	405.95	0.00	0.00	
4.373	LG13	102.38	5857.12	0.00	0.00	
8.750	LG13	-131.07		0.00	0.00	204)
13.100	LG13	57.25	10476.75	0.00	0.00	
13.123	LG13	57.25	10476.75	0.00	0.00	
<b>Member No. 8 - Rectangle 35/35</b>						
0.000	LG13	987.31	604.72	0.00	0.00	
4.373	LG13	179.99	3330.28	0.00	0.00	
8.750	LG13	-61.62		0.00	0.00	204)
13.123	LG13	-206.23		0.00	0.00	204)
<b>Member No. 9 - Rectangle 35/35</b>						
0.000	LG13	315.80	1896.89	0.00	0.00	
4.373	LG13	73.54	8155.68	0.00	0.00	
8.750	LG13	-24.70		0.00	0.00	204)
13.100	LG13	-94.58		0.00	0.00	204)
13.123	LG13	-94.58		0.00	0.00	204)
<b>Member No. 10 - Rectangle 40/55</b>						
0.000	LG13	1515.34	391.20	0.00	0.01	350)
4.593	LG13	1251.27	474.76	0.00	0.07	350)
9.186	LG13	4136.55	140.31	0.00	0.13	350)
13.780	LG13	5930.84	96.44	0.00	0.17	350)

Figure 5.22: 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.

## 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 48 and Figure 5.13, page 52).

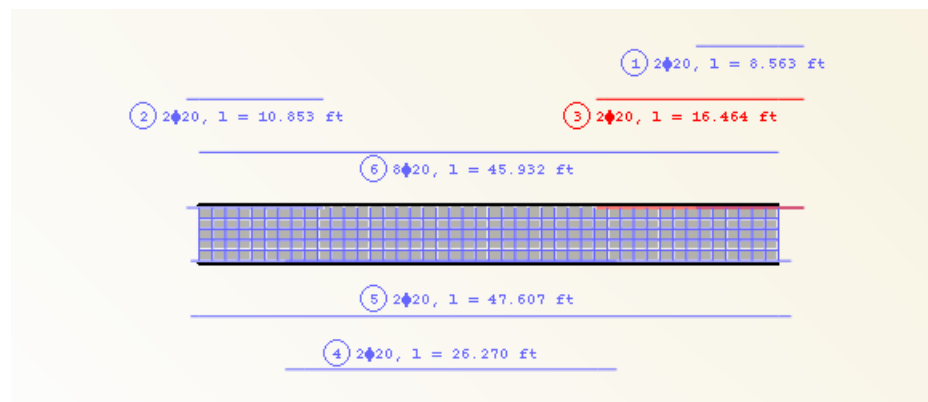


Figure 6.1: Reinforcement drawing in table 3.1 *Longitudinal Reinforcement Provided*

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 51 or in Figure 5.14 on page 54. 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).

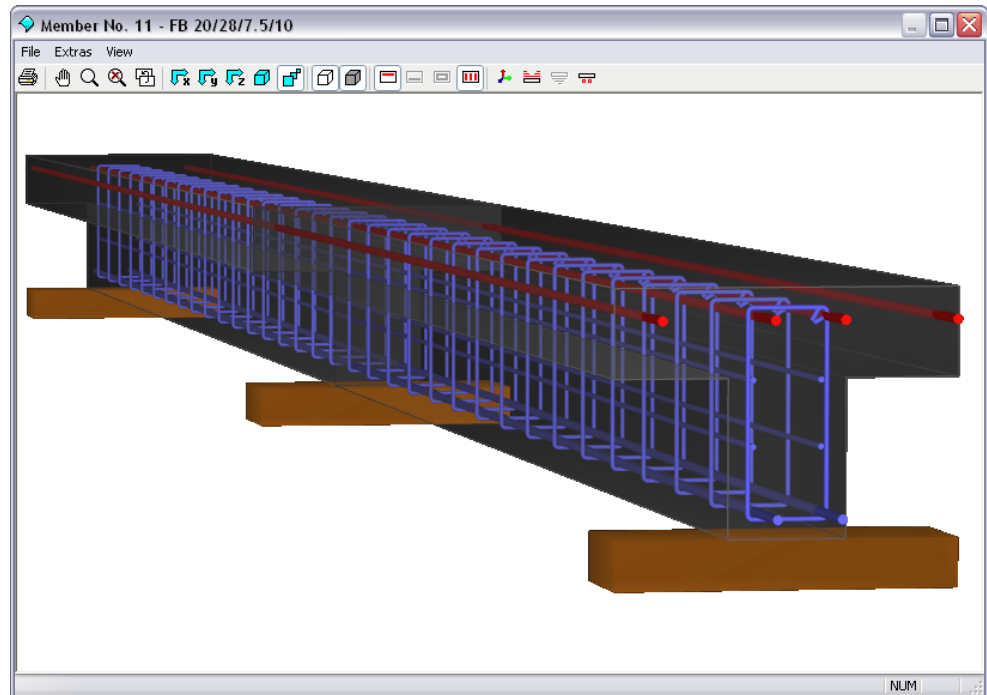


Figure 6.2: 3D rendering of provided longitudinal and link reinforcement

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 RFEM, 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:





















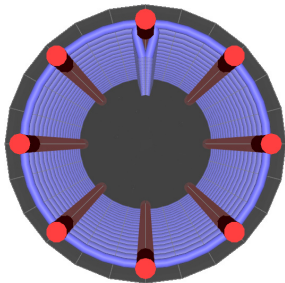
Button	Description	Function
	Print	Opens the dialog box <i>Graphic Printout</i> (Figure 7.4, page 71) providing the printout settings
	Shift	Allows for shifting the view by the mouse (zooming/rotating by pressing [Shift] or [Ctrl] key)
	Zoom	Allows for increasing a particular region in the graphic by 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



Reinforcement of column:  
perspective view in X





## 6.3 Results in the RFEM Model

To evaluate the design results graphically, you can also use the RFEM work window.

### RFEM background graphic

The RFEM 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 results table of RF-CONCRETE Members, the corresponding member is highlighted in the RFEM background graphic. In addition, an arrow indicates the member's x-location that is displayed in the currently selected table row.

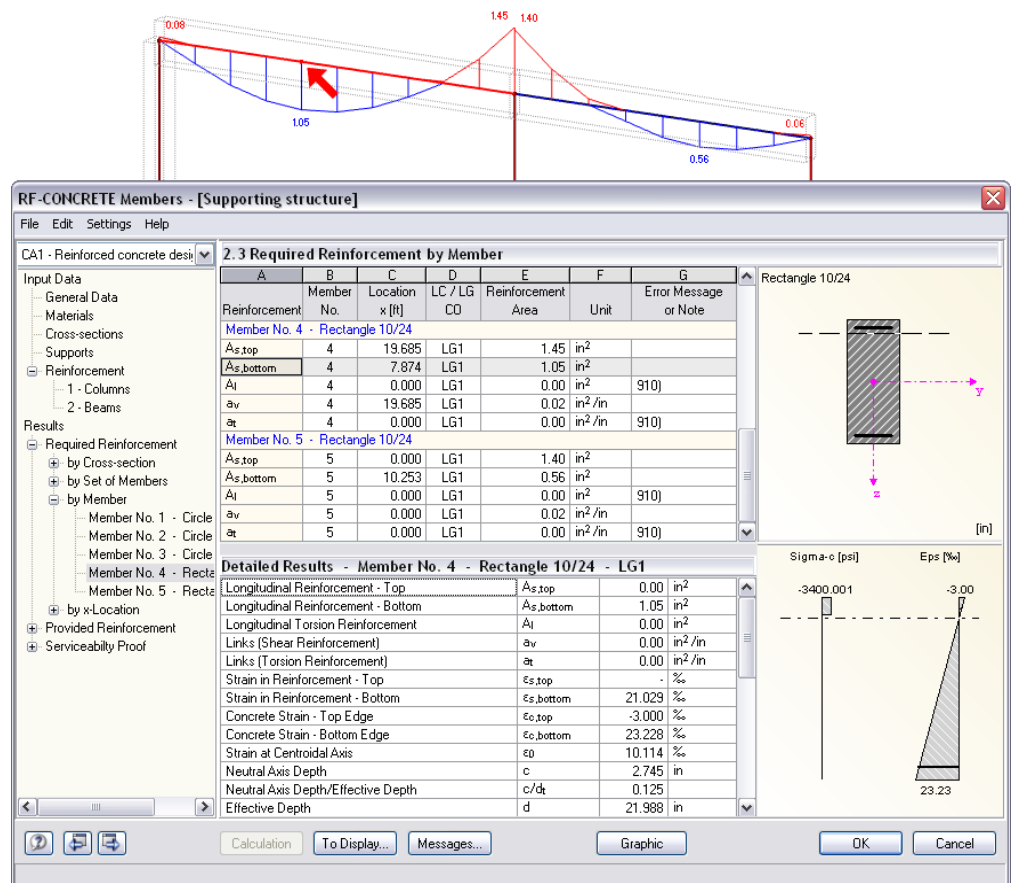


Figure 6.3: Indication of the member and the current Location x in the RFEM model

Graphic

RF-CONCRETE Members

This function, however, is only available if the results of the current RF-CONCRETE Members case are set in the RFEM graphical user interface. You can use the [Graphic] button to switch to RFEM and the button [RF-CONCRETE Members] in the control panel to return to the add-on module.

Graphic

## RFEM work window

All reinforcement areas and intermediate results can be visualized on the RFEM structural model: First, click the [Graphic] button to close the add-on module RF-CONCRETE Members. Now, the various reinforcements and design values are displayed in the graphic of the RFEM work window like the internal forces or deformations of a RFEM load case. The display of the different result types can be set in the *Results* navigator of RF-CONCRETE Members.

The *Results* navigator is aligned with the designs of the add-on module RF-CONCRETE Members. You find all the different reinforcement types for the ultimate and the serviceability limit state design as well as the fire protection design including all intermediate results available for selection.

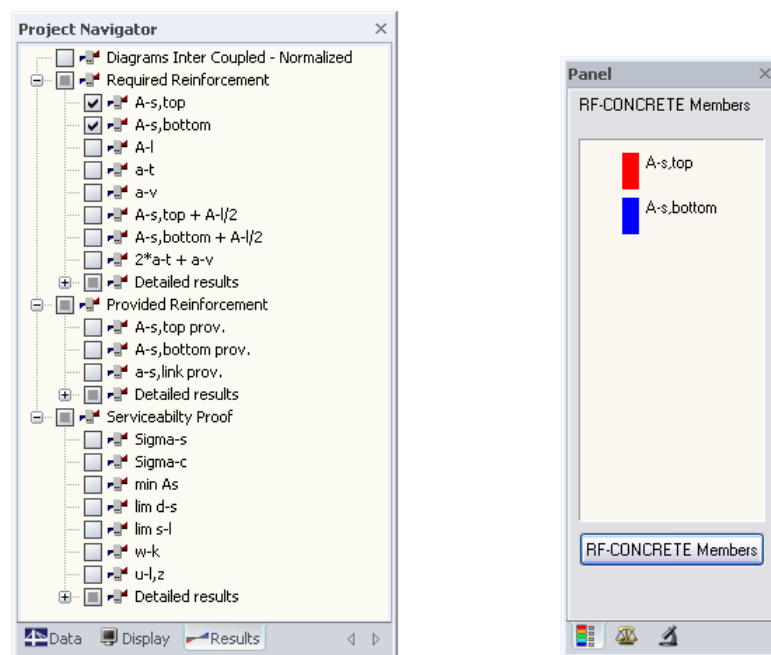


Figure 6.4: Results navigator of RF-CONCRETE Members and panel with selected result types

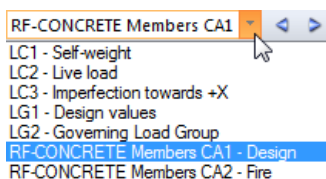
The *Results* navigator allows you to display several reinforcement types or designs at the same time. In this way, it is possible, for example, to compare graphically the required longitudinal reinforcement with the provided longitudinal reinforcement. The panel will be synchronized with the selected types of results.

Due to this multiple selection and the automatic color assignment, the options available in the RFEM *Display* navigator for the representation of member results are of no relevance.

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.

As the RFEM tables are of no relevance for the evaluation of the RF-CONCRETE Members results, you may deactivate them.

The design cases are selected as usual by means of the list in the RFEM 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.

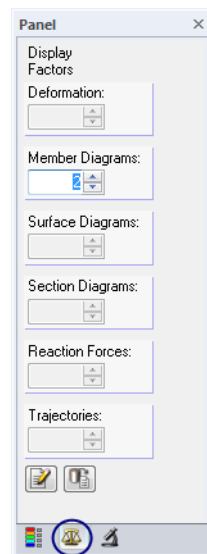


Figure 6.5: Panel tab *Display Factors*

In addition to the required and the provided reinforcement, it is possible to evaluate the intermediate results of all designs graphically.

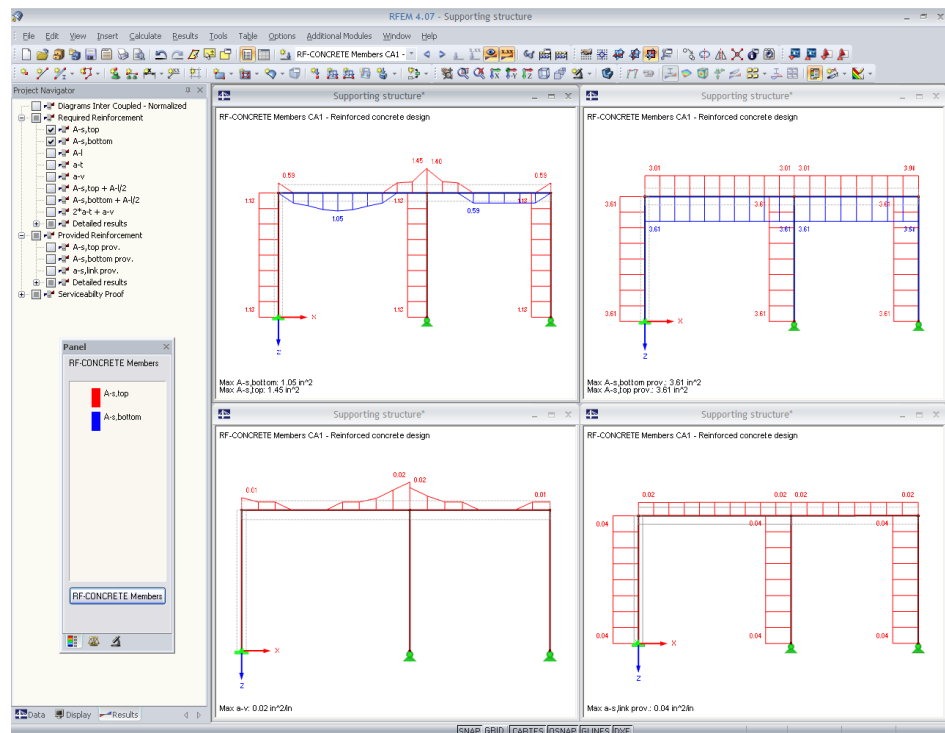


Figure 6.6: Graphical output of required and provided reinforcement as well as fire protection

All results graphics can be transferred like RFEM graphics to the global printout report (see chapter 7.2, page 71).

RF-CONCRETE Members

It is always possible to return to the design module by clicking the button [RF-CONCRETE Members] in the panel.

## 6.4 Result Diagrams

The result diagrams are available in the RFEM graphic. To display the diagrams, select **Result Diagrams on Selected Members** on the **Results** menu, or use the button in the RFEM 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.

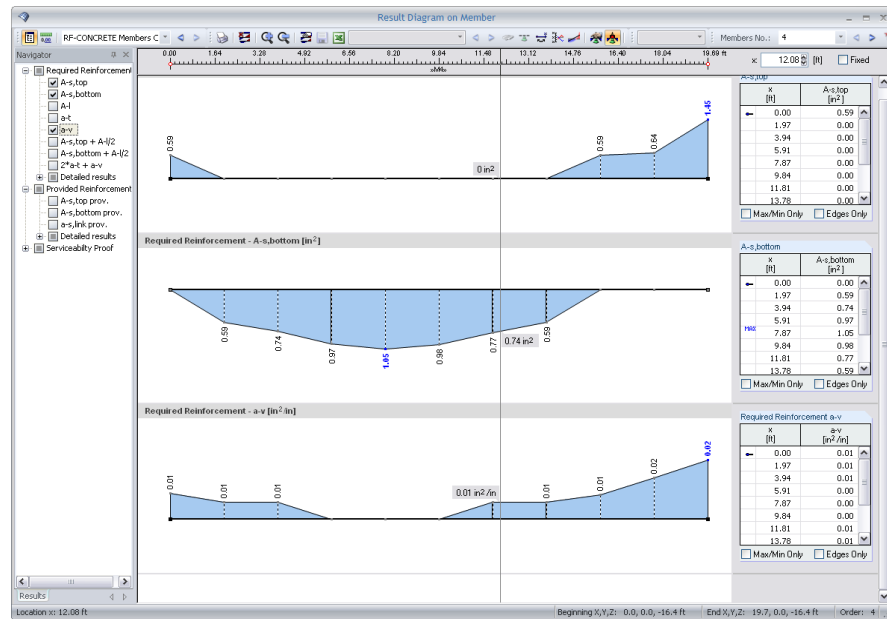


Figure 6.7: Dialog box *Result Diagram on Member*

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 design case of RF-CONCRETE Members as well as specific members or sets of members for the display.

For more detailed information on the dialog box *Result Diagram on Member*, see the RFEM manual, chapter 10.5, page 311.

## 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 RFEM manual to evaluate the designs graphically.



On the one hand, you can take advantage of already defined partial views (see RFEM manual, chapter 10.9, page 321) 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 RFEM manual, chapter 4.4.6, page 77.

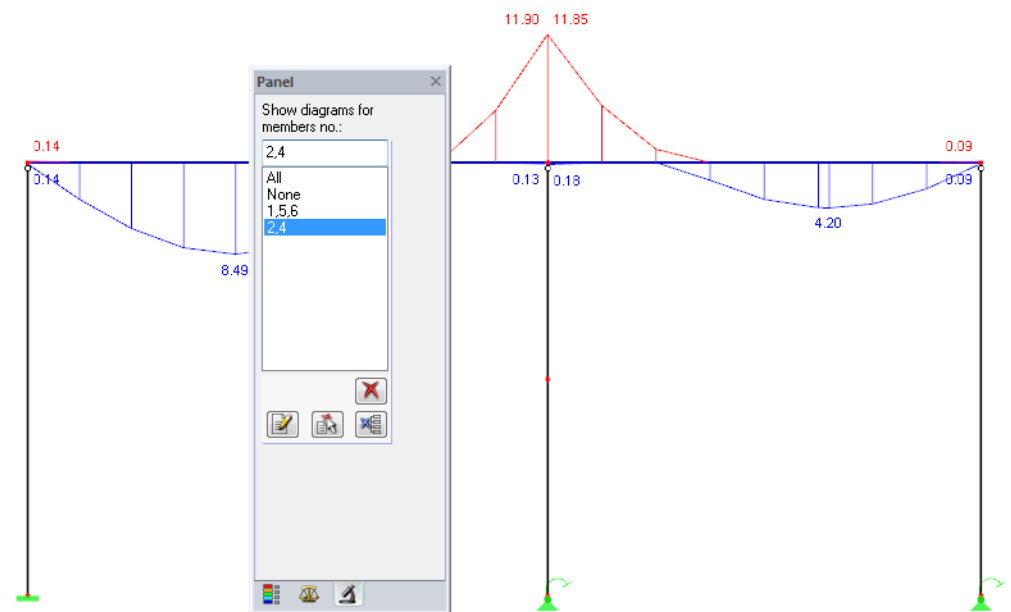


Figure 6.8: Filtering members in the panel

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 printout procedure in RFEM. First, the program generates a printout report for the results of RF-CONCRETE Members. 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 printout report for RF-CONCRETE Members, this printout report will be generated relatively quickly.

The printout report is described in detail in the RFEM manual. In particular, chapter 11.1.3.4 *Selecting Data of Add-on Modules* on page 338 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 RF-CONCRETE Members.

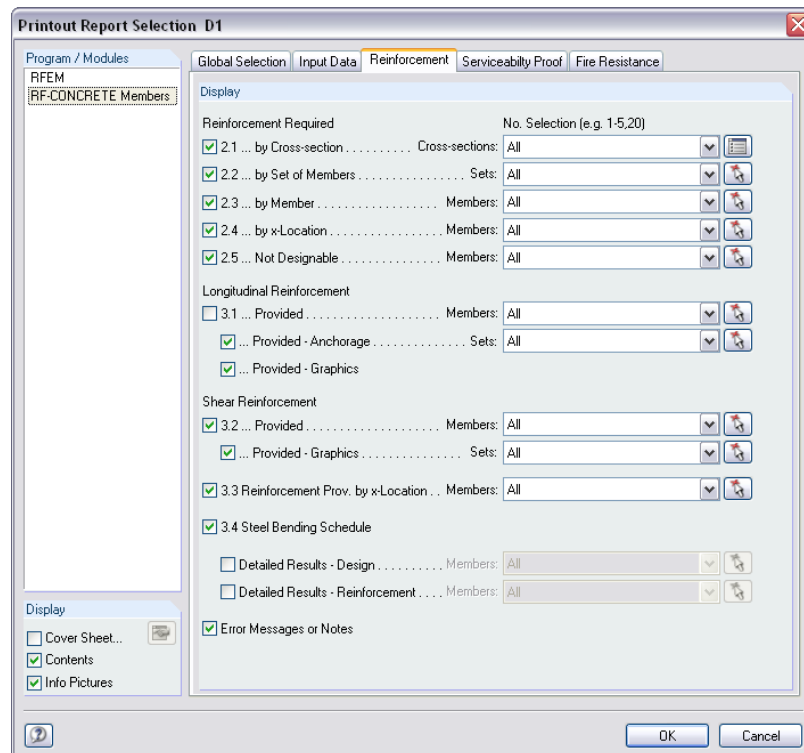


Figure 7.1: Printout report selection of results of RF-CONCRETE Members, tab *Reinforcement*

## 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 RFEM manual, chapter 11.2.



Every picture that is displayed in the graphic window of the main program RFEM can be included into 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 current RF-CONCRETE Members graphic displayed in the RFEM work window, select **Print** on the **File** menu or use the toolbar button shown on the left.

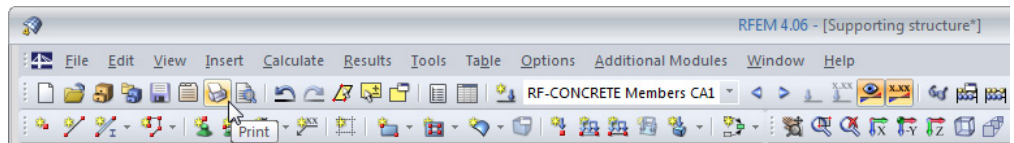


Figure 7.2: Button *Print* in the toolbar of the main window

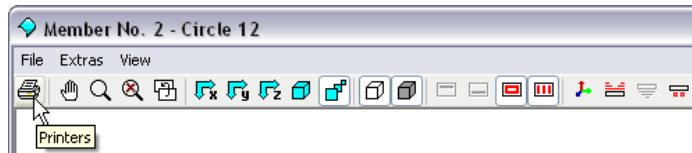


Figure 7.3: Button *Print* in the toolbar of the 3D rendering window

The following dialog box opens:

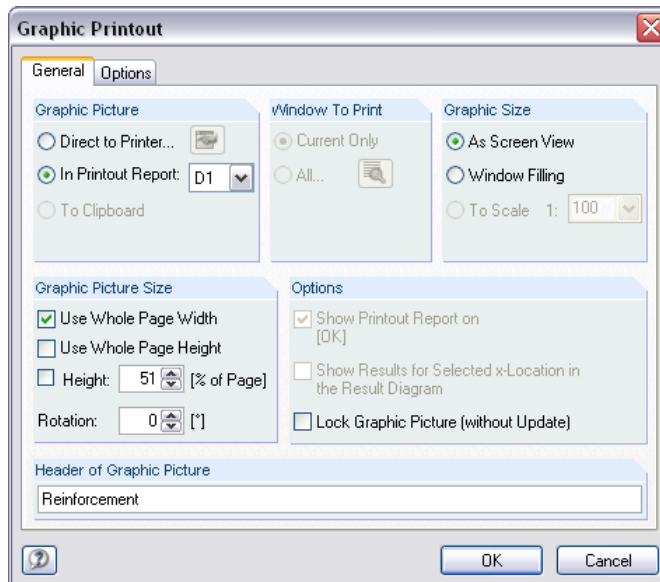
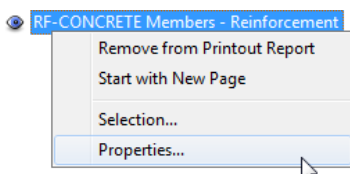


Figure 7.4: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in the RFEM manual, chapter 11.2, page 354. The RFEM manual also describes the *Options* and *Color Spectrum* tab.

A graphic from RF-CONCRETE Members, 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

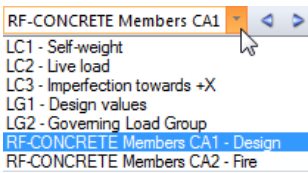
This chapter describes some menu functions as well as export options for the design results.

### 8.1 Design Cases in RF-CONCRETE Members

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 member in different design cases.

The design cases of RF-CONCRETE Members are available in the RFEM work window and can be displayed like a load case or load group by means of the toolbar list.



#### Create a new RF-CONCRETE Members case

To create a new design case,

select **New Case** on the **File** menu in the RF-CONCRETE Members add-on module.

The following dialog box appears.

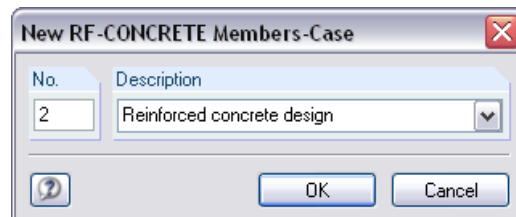


Figure 8.1: Dialog box *New RF-CONCRETE Members-Case*

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 RF-CONCRETE Members case

To change the description of a design case subsequently,

select **Rename Case** on the **File** menu in the RF-CONCRETE Members add-on module.

The dialog box *Rename RF-CONCRETE Members-Case* appears.

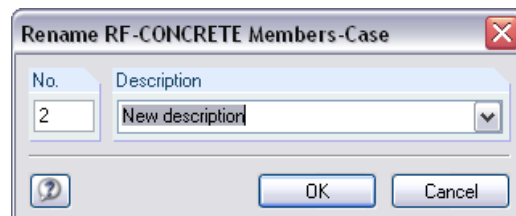


Figure 8.2: Dialog box *Rename RF-CONCRETE Members-Case*



### Copy a RF-CONCRETE Members case

To copy the input data of the current design case,

select **Copy Case** on the **File** menu in the RF-CONCRETE Members add-on module.

The dialog box *Copy RF-CONCRETE Members-Case* appears where you can specify the number and description of the new case.

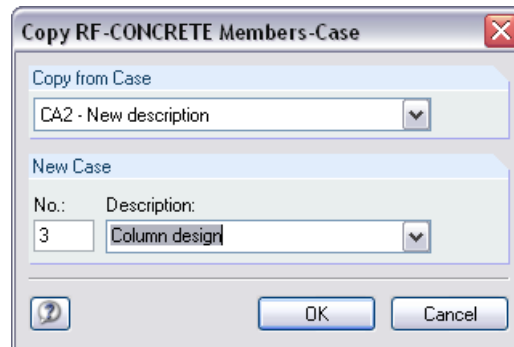


Figure 8.3: Dialog box *Copy RF-CONCRETE Members-Case*

### Delete a RF-CONCRETE Members case

To delete a design case,

select **Delete Case** on the **File** menu in the RF-CONCRETE Members 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].

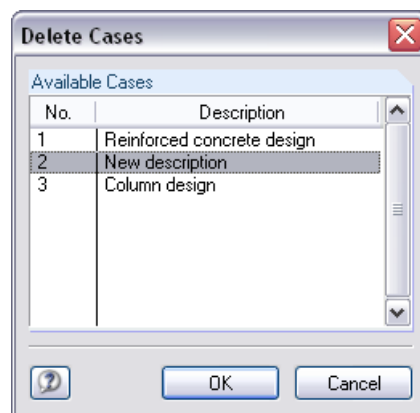


Figure 8.4: Dialog box *Delete Cases*

## 8.2 Cross-section Optimization

As mentioned in chapter 3.3, RF-CONCRETE Members offers you the possibility to optimize cross-sections. Tick the check box of the corresponding cross-section in column C of table 1.3 *Cross-sections* (see Figure 3.6, page 23). Simply click into the little box to open the following dialog box where you can define detailed specifications.

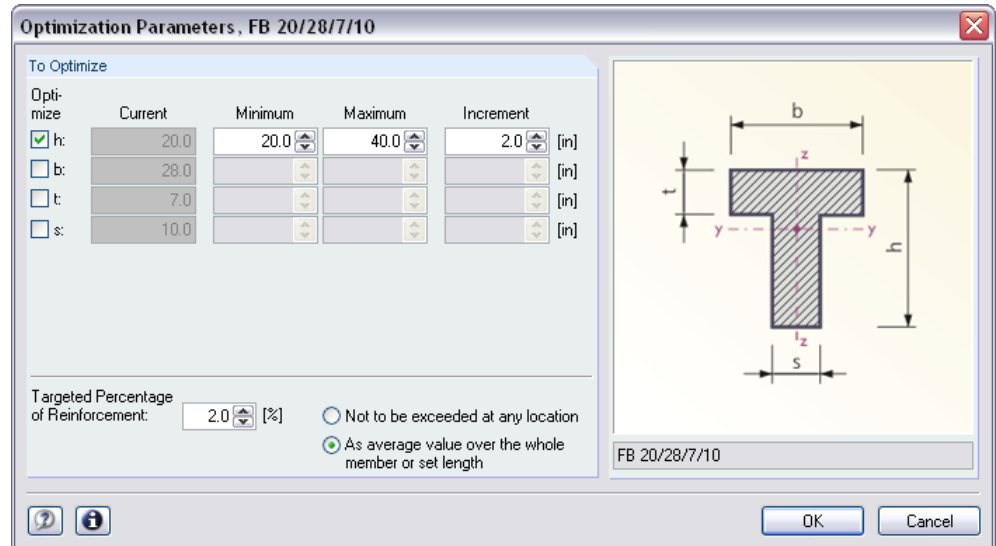


Figure 8.5: Dialog box *Optimization Parameters* of a T-beam

By ticking the check boxes 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.

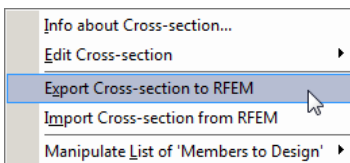
The criterion for optimization is specified by the fact that the *Targeted Percentage of Reinforcement* either won't be exceeded at any location or is available as average value across the entire member or set of member. The reinforcement ratio can be defined in the input field.

In the course of the optimization process, RF-CONCRETE Members 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 RFEM. 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.

You do not need to transfer the modified cross-sections to RFEM manually: Set table 1.3 *Cross-sections*, and then

select **Export Cross-section to RFEM** 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 RFEM.



Calculation

Before the changed cross-sections are transferred to RFEM, a security query appears, because the transfer requires the deletion of results. When you confirm the query and then start the [Calculation] in RF-CONCRETE Members, the RFEM internal forces as well as the areas of reinforcement are determined in one single calculation run.

To reimport the original RFEM cross-section to RF-CONCRETE Members, 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 RFEM and all add-on modules are managed in one global dialog box. In the module RF-CONCRETE Members, 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 RFEM. The add-on module RF-CONCRETE Members is preset.

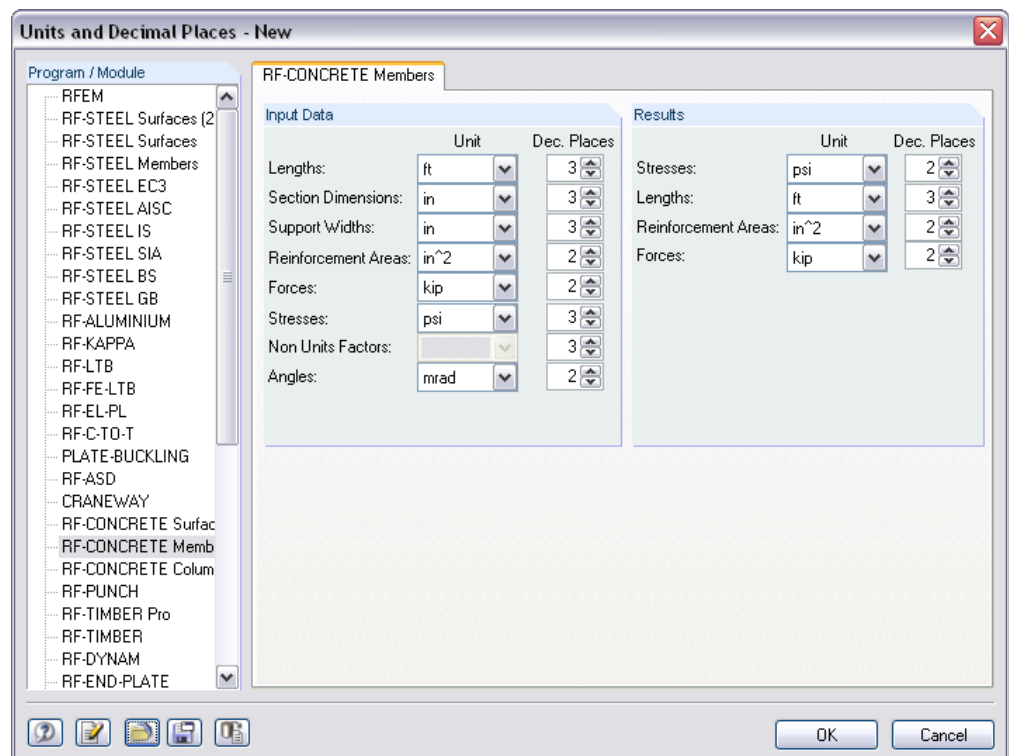


Figure 8.6: Dialog box *Units and Decimal Places*



The settings can be saved as user profile to reuse them in other structures. The functions are described in the RFEM manual, chapter 12.6.2, page 453.

## 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 results tables of RF-CONCRETE Members 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 data of RF-CONCRETE Members can be printed into the global printout report (cf. chapter 7.1, page 70) to export them subsequently. In the printout report,

select **Export to RTF File or BauText** on the **File** menu.

The function is described in detail in the RFEM manual, chapter 11.1.11, page 350.

### Excel / OpenOffice

RF-CONCRETE Members provides a function for the direct data export to MS Excel, OpenOffice.org Calc or as a CSV file. To open the corresponding dialog box,

select **Export Tables** on the **File** menu in the RF-CONCRETE Members add-on module.

The following export dialog box appears.

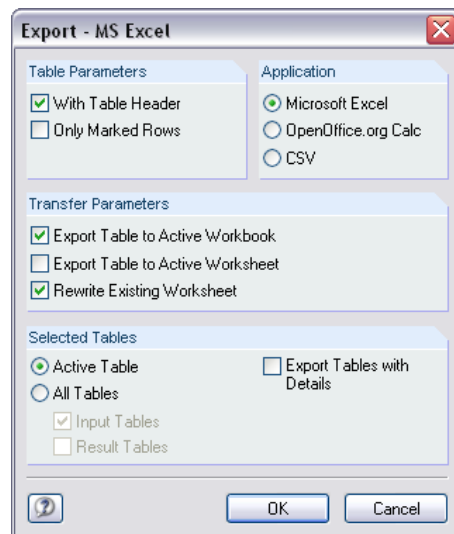


Figure 8.7: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking [OK]. MS Excel or OpenOffice will be started automatically. It is not necessary to run the programs in the background.

Sheet1 - Microsoft Excel

	A	B	C	D	E	F	G	H	I
1		Member	Location	LC / LG	Reinforcement		Error Message		
2	Reinforcement	No.	x [m]	CO	Area	Unit	or Note		
3	Cross-section No. 1 - Rectangle 900/600								
4	$A_{s,top}$	1	0,000	LG1	5,40	cm <sup>2</sup>	25)		
5	$A_{s,bottom}$	1	0,000	LG1	5,40	cm <sup>2</sup>	25)		
6	$A_{s,T}$	1	0,000	LG1	0,00	cm <sup>2</sup>			
7	$a_{s,wly}$ link	1	0,000	LG1	0,00	cm <sup>2</sup> /m	58)		
8	$a_{s,wly}$ link	1	0,000	LG1	0,00	cm <sup>2</sup> /m			
9	Cross-section No. 2 - Rectangle 900/900								
10	$A_{s,top}$	5	0,000	LG1	8,10	cm <sup>2</sup>	25)		
11	$A_{s,bottom}$	5	0,000	LG1	8,10	cm <sup>2</sup>	25)		
12	$A_{s,T}$	5	0,000	LG1	0,00	cm <sup>2</sup>			
13	$a_{s,wly}$ link	5	0,000	LG1	0,00	cm <sup>2</sup> /m	58)		
14	$a_{s,wly}$ link	5	0,000	LG1	0,00	cm <sup>2</sup> /m			
15	Cross-section No. 3 - Rectangle 1000/1400								
16	$A_{s,top}$	10	16,000	LG1	31,17	cm <sup>2</sup>			
17	$A_{s,bottom}$	10	8,000	LG1	39,86	cm <sup>2</sup>			
18	$A_{s,T}$	10	0,000	LG1	0,00	cm <sup>2</sup>			
19	$a_{s,wly}$ link	10	0,000	LG1	8,76	cm <sup>2</sup> /m	58) 69)		
20	$a_{s,wly}$ link	10	0,000	LG1	0,00	cm <sup>2</sup> /m			

2.1 Required Reinforcement by C 2.2 Required Reinforcement by T

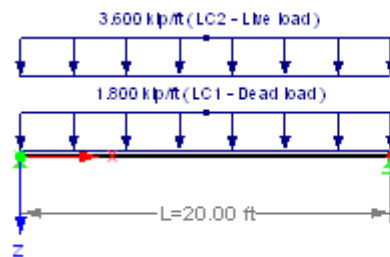
Figure 8.8: Result in MS Excel

## 9. Example

The final chapter describes an example where the doubly reinforced beam is designed for strength limit state according to ACI 318-08.

### 9.1 Input Data

#### System



#### Cross-section

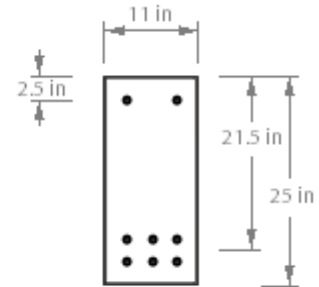


Figure 9.1: System, loads and cross-section

Cross-section	Rectangle
	$h = 25 \text{ in}$ , $b = 11 \text{ in}$
Material	Concrete $f'_c = 5\,000 \text{ psi}$
	Reinforcing steel $f_y = 60\,000 \text{ psi}$
Reinforcement	Top 2#8 ( $A_s = 1.57 \text{ in}^2$ ), $d = 21.5 \text{ in}$
	Bottom 6#8 ( $A'_s = 4.71 \text{ in}^2$ ), $d' = 2.5 \text{ in}$
	Stirrups #4

#### Loads

Dead load	
(including self-weight)	$w_{DL} = 1.80 \text{ kips/ft}$
Live load	$w_{LL} = 3.60 \text{ kips/ft}$

#### Factored internal forces

Load factor	1.2 (dead load)
Load factor	1.6 (live load)
Factored load combination	$w_u = 1.2w_{DL} + 1.6w_{LL} = 1.2 \cdot 1.80 + 1.6 \cdot 3.6 = 7.92 \text{ kips/ft}$
Factored moment	$M_u = \frac{1}{8} w_u L^2 = \frac{1}{8} 7.92 \cdot 20^2 = 396 \text{ kips} \cdot \text{ft}$
Factored shear force	$V_u = \frac{1}{2} w_u L = \frac{1}{2} 7.92 \cdot 20 = 79.2 \text{ kips}$

## 9.2 Flexural Strength

### Check steel percentage

$$\rho = \frac{A_s}{b \cdot d} = \frac{4.71}{11 \cdot 21.5} = 0.01992$$

$$\rho' = \frac{A'_s}{b \cdot d} = \frac{1.57}{11 \cdot 21.5} = 0.00664$$

$$\rho_{\min} = \max \left\{ \begin{array}{l} \frac{200}{f_y} = \frac{200}{60,000} = 0.00333 \\ \frac{3\sqrt{f'_c}}{f_y} = \frac{3\sqrt{5000}}{60,000} = 0.00354 \end{array} \right. \Rightarrow \rho_{\min} = 0.00354$$

### Check if compression steel yields

$$\text{Equilibrium equation} \quad A_s f_y = 0.85 f'_c b \cdot a + A'_s f_y$$

$$a = \frac{(A_s - A'_s) f_y}{0.85 f'_c b} = \frac{(4.71 - 1.57) 60}{0.85 \cdot 5 \cdot 11} = 4.03 \text{ in}$$

$$\beta_1 = 0.80 \text{ for concrete } 5000 \text{ psi}$$

$$c = \frac{a}{\beta_1} = \frac{4.03}{0.80} = 5.04 \text{ in}$$

$$\text{Strain in the compression steel} \quad \varepsilon'_s = \left( \frac{c - d'}{c} \right) 0.003 = \left( \frac{5.04 - 2.5}{5.04} \right) 0.003 = 0.00151$$

$$\text{Yield strain of steel} \quad \varepsilon_y = \frac{f_y}{E_s} = \frac{60}{29,000} = 0.00207$$

$$\varepsilon'_s < \varepsilon_y \Rightarrow \text{Compression reinforcement does not yield, assumption is not valid.}$$

### Define rectangular concrete stress distribution

$$\text{Equilibrium equation} \quad A_s f_y = 0.85 f'_c b \cdot \beta_1 c + A'_s \frac{c - d'}{c} 0.003 \cdot f_y$$

$$4.71 \cdot 60 = 0.85 \cdot 5 \cdot 11 \cdot 0.80 \cdot c + 1.57 \frac{c - 2.5}{c} 0.003 \cdot 29,000$$

$$\text{Solution of the quadratic equation: } c = 5.55 \text{ in}$$

$$a = \beta_1 c = 0.80 \cdot 5.55 = 4.44 \text{ in}$$

**Strains and stresses**

$$\varepsilon'_s = \left( \frac{c - d'}{c} \right) 0.003 = \left( \frac{5.55 - 2.5}{5.55} \right) 0.003 = 0.00165 < \varepsilon_y$$

$$f'_s = \varepsilon'_s E_s = 0.00165 \cdot 29,000,000 = 47,810 \text{ psi}$$

$$\varepsilon_t = \left( \frac{d - c}{c} \right) 0.003 = \left( \frac{21.5 - 5.55}{5.55} \right) 0.003 = 0.00862 > 0.005 \Rightarrow \phi = 0.9$$

**Design moment strength**

$$\begin{aligned} \phi \cdot M_n &= \phi \left( A_s f_y - A'_s f'_s \right) \left( d - \frac{a}{2} \right) + A'_s f'_s (d - d') = 0.9 \left( (4.71 \cdot 60 - 1.57 \cdot 47.81) \left( 21.5 - \frac{4.44}{2} \right) + \right. \\ &\quad \left. + 1.57 \cdot 47.81 (21.5 - 2.5) \right) = \\ &= 4885 \text{ kips} \cdot \text{in} = 407.1 \text{ kips} \cdot \text{ft} \end{aligned}$$

$$\phi \cdot M_n = 407.1 \text{ kips} \cdot \text{ft} > M_u = 396 \text{ kips} \cdot \text{ft} \quad \text{OK}$$

**9.3 Shear Strength****Shear strength provided by concrete**

$$M_{u, \text{support}} = 0$$

$$\frac{V_u d}{M_{u, \text{support}}} \leq 1.0$$

$$\begin{aligned} &\left( 1.9 \lambda \sqrt{f'_c} + 2500 \rho_w \frac{V_u d}{M_{u, \text{support}}} \right) b_w d = \\ &= (1.9 \cdot 1.0 \sqrt{5000} + 2500 \cdot 0.01992 \cdot 1) 11 \cdot 21.5 \frac{1}{1000} = 43.55 \text{ kips} \end{aligned}$$

$$3.5 \lambda \sqrt{f'_c} b_w d = 3.5 \sqrt{5000} \cdot 11 \cdot 21.5 \frac{1}{1000} = 58.53 \text{ kips}$$

$$V_c = \min \begin{cases} 43.55 \text{ kips} \\ 58.53 \text{ kips} \end{cases} = 43.55 \text{ kips}$$

$$V_u = 79.2 \text{ kips} > \frac{1}{2} \phi V_c = \frac{1}{2} 0.75 \cdot 43.55 = 16.33 \text{ kips} \Rightarrow \text{Stirrups needed}$$

**Shear strength provided by shear reinforcement**

$$V_s = \frac{V_u - \phi V_c}{\phi} = \frac{79.2 - 0.75 \cdot 43.55}{0.75} = 62.05 \text{ kips}$$

Theoretical spacing

$$s = \frac{A_v f_y d}{V_s} = \frac{2 \cdot 0.20 \cdot 60 \cdot 21.5}{62.05} = 8.32 \text{ in}$$



Maximum spacing to provide minimum  $A_v$ :

$$s = \frac{A_v f_y}{0.75 \sqrt{f'_c} b_w} = \frac{2 \cdot 0.20 \cdot 60,000}{0.75 \sqrt{5000} \cdot 11} = 41.14 \text{ in}$$

$$s = \frac{A_v f_y}{50 b_w} = \frac{2 \cdot 0.20 \cdot 60,000}{50 \cdot 11} = 43.63 \text{ in}$$

Spacing limits acc. to 11.4.5.1:

$$V_s = 62.05 \text{ kips} \leq 4 \sqrt{f'_c} b_w d = 4 \sqrt{5000} \cdot 11 \cdot 21.5 \frac{1}{1000} = 66.89 \text{ kips}$$

$$s = \frac{d}{2} = \frac{21.5}{2} = 10.75 \text{ in} \leq 24 \text{ in}$$

⇒ bars #4@8 in

### Design shear strength

$$V_s = \frac{A_v f_y d}{s} = \frac{2 \cdot 0.20 \cdot 21.5}{8} = 64.5 \text{ kips}$$

$$V_n = \phi(V_c + V_s) = 0.75(43.55 + 64.5) = 81.04 \text{ kips} \geq V_u = 79.2 \text{ kips} \quad \text{OK}$$

## 9.4 Development of Reinforcement

### Development length of tension reinforcement

Spacing of longitudinal reinforcement

$$s = 3 \text{ in}$$

$$c_b = \min \left\{ c_c + \frac{d_b}{2} = 2 + \frac{1}{2} = 2.5 \right. \\ \left. \frac{s}{2} = \frac{3}{2} = 1.5 \right. = 1.5 \text{ in}$$

$$K_{tr} = \frac{40 A_{tr}}{s n} = \frac{40 \cdot 2 \cdot 0.20}{8 \cdot 6} = 0.333$$

$$\frac{c_b + K_{tr}}{d_b} = \frac{1.5 + 0.333}{1.0} = 1.833 \leq 2.5$$

$$l_d = \left( \frac{3}{40} \frac{f_y}{\lambda \sqrt{f'_c}} \frac{\psi_t \psi_e \psi_s}{\left( \frac{c_b + K_{tr}}{d_b} \right)} \right) \cdot d_b = \left( \frac{3}{40} \frac{60,000}{1.0 \sqrt{5000}} \frac{1.0 \cdot 1.0 \cdot 1.0}{1.833} \right) \cdot 1.0 = 34.72 \text{ in}$$

### Development length of compression reinforcement

$$l_{dc} = \frac{0.02 f_y}{\lambda \sqrt{f'_c}} d_b = \frac{0.02 \cdot 60,000}{1.0 \sqrt{5000}} = 16.97 \text{ in}$$

$$\leq 0.0003 f_y d_b = 0.0003 \cdot 60,000 \cdot 1.0 = 18 \text{ in}$$

## 9.5 Results in RF-CONCRETE Members

The following figures show the result output of the designed longitudinal and shear reinforcement.

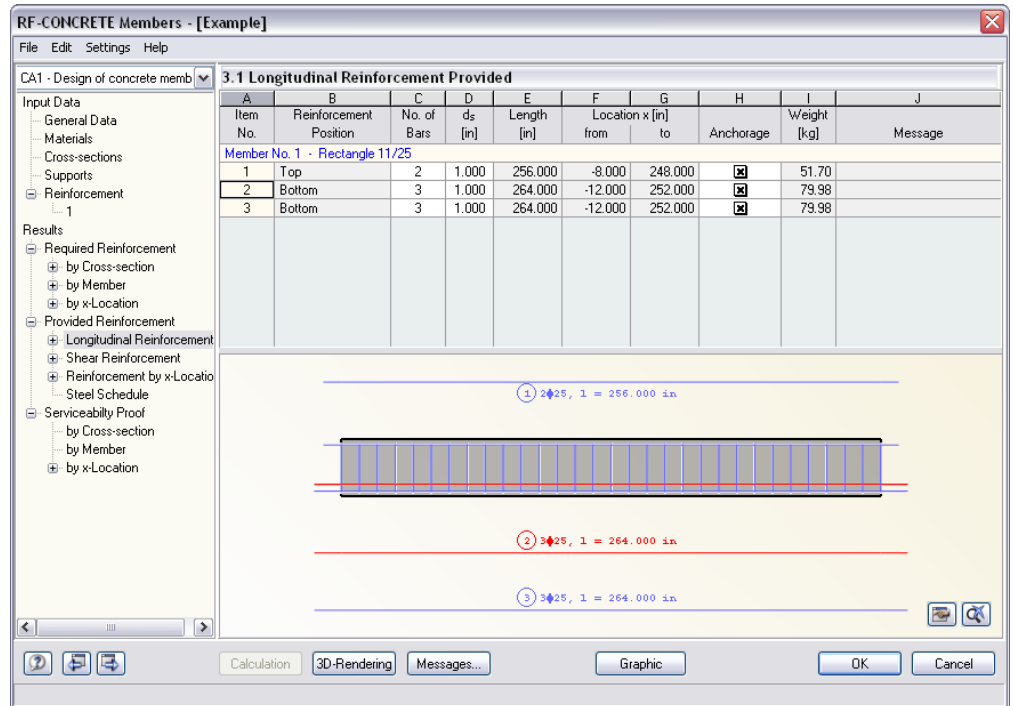


Figure 9.2: Table 3.1 Longitudinal Reinforcement Provided

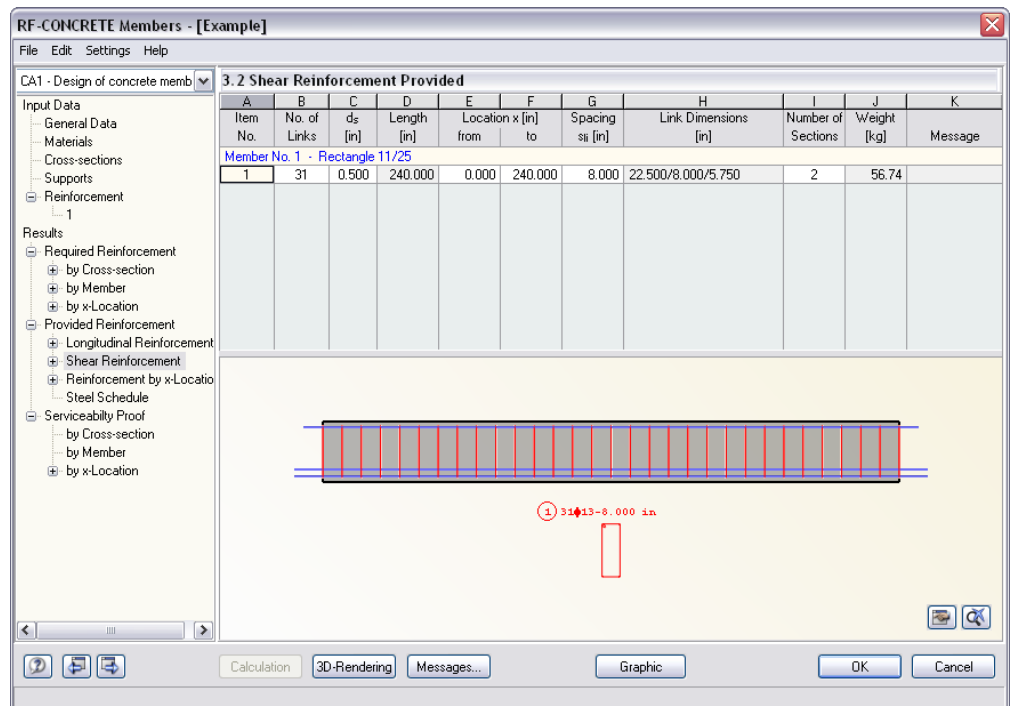


Figure 9.3: Table 3.2 Shear Reinforcement Provided

# 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., QUAST, 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
- [16] ACI 318-08: Building Code Requirements for Structural Concrete (ACI 318-08 and Commentary), ACI Committee 318, January 2008

## Miscellaneous

- [17] AVAK, R.: Stahlbetonbau in Beispielen, DIN 1045, Teil 1 „Grundlagen der Stahlbeton-Bemessung - Bemessung von Stabtragwerken“, Werner Verlag, 5. Auflage 2007
- [18] BARTH, C., RUSTLER, W.: „Finite Elemente in der Baustatik-Praxis“, Bauwerk Verlag 2010
- [19] CURBACH, M. et. al.: „Nichtlineare Berechnung alter Bogenbrücken auf Grundlage neuer Vorschriften“, Beton- und Stahlbetonbau 99 (04/2004)
- [20] HEYDEL G., KRINGS W., HERRMANN H.: „Stahlbeton im Hochbau nach EC 2“, Ernst und Sohn Verlag 1995
- [21] HOSSER, D. and 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
- [22] KLEINSCHMITT, J.: „Die Berechnung von Stahlbetonstützen nach DIN 1045-1 mit nicht-linearen Verfahren“, Beton- und Stahlbetonbau 02/2005
- [23] KORDINA, K. and QUAST, U.: „Bemessung von schlanken Bauteilen für den durch Tragwerksverformung beeinflussten Grenzzustand der Tragfähigkeit – Stabilitätsnachweis“, Betonkalender 2002/Teil 1, Ernst & Sohn Verlag 2002
- [24] LEONHARDT, F.: „Vorlesungen über Massivbau“, Teil 1 bis 4, Springer Verlag, 3. Auflage 1984
- [25] NOAKOWSKI, P. and Schäfer, H. G.: „Steifigkeitsorientierte Statik im Stahlbetonbau“, Ernst & Sohn, 2003
- [26] NOAKOWSKI, P. and SCHÄFER, H. G.: „Die Schnittgrößen in Stahlbetontragwerken einfach richtig berechnen“, Beton- und Stahlbetonbau 96 (06/2001)
- [27] 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
- [28] PFEIFFER, U. and 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
- [29] PFEIFFER, U. and QUAST, U.: „Nichtlineares Berechnen stabförmiger Bauteile“, Beton- und Stahlbetonbau 98 (09/2003)
- [30] QUAST, U.: „Zur Kritik an der Stützenbemessung“, Beton- und Stahlbetonbau 95 (05/2000)
- [31] QUAST, U.: „Zum nichtlinearen Berechnen im Stahlbeton- und Spannbetonbau“, Beton- und Stahlbetonbau 89 (09/1994)
- [32] 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
- [33] QUAST, U.: „Versagen Stahlbetonstützen anders als Stahlstützen?“, Veröffentlicht auf der Homepage der TU Hamburg-Harburg  
Download unter <http://www.mb.tu-harburg.de> möglich
- [34] VATER, C.: „Rechnerisch-theoretische Untersuchungen zur Schnittgrößenumlagerung in verschieblichen und unverschieblichen Stahlbetonrahmen“, Dissertation an der TU Hamburg-Harburg 1999

- [35] ZILCH, K. and 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
- [36] NILSON, A. H., DARWIN, D. and DOLAN, C. W.: „Design of Concrete Structures – 13th ed.“, McGraw-Hill, International Edition 2003
- [37] MCCORMAC, J. C. and BROWN, R. H.: „Design of Reinforced Concrete – Eighth ed.“, John Wiley & Sons, Inc. 2009

# B Index

## 3

3D Rendering..... 63, 71

## A

Anchorage..... 31, 50

Anchorage type ..... 50, 57

Axial force ..... 9

## B

Background graphic..... 65

Bar spacing ..... 31

Basic reinforcement..... 31

Bearing capacity ..... 19, 66

Bend..... 50

Bending ..... 9

Bending diameter ..... 51, 57

Bond ..... 50

Bottom reinforcement..... 43

Buttons ..... 26, 64

## C

Calculation..... 40

Centroidal axis..... 34

Code ..... 19

Comment ..... 19

Compression force..... 10

Compression reinforcement ..... 10

Concrete cover..... 34

Concrete strength class..... 21

Corner ..... 35

Crack width ..... 13, 59

Crack width control ..... 36

Crack width limit ..... 36

Cross-section ..... 23

Cross-section description..... 23

Cross-section library..... 23

Cross-section optimization ..... 74

CSV export..... 76

Curtailment ..... 31

Curtailment of reinforcement..... 31

## D

Decimal places ..... 21, 75

Deflection ..... 14

Deformation..... 4, 59

Design case ..... 66, 72, 73

Design details..... 43, 55

Detailed results..... 43, 68

Development length ..... 15, 81

Diameter ..... 31

Direct support..... 27

Distribution ..... 34

## E

Edge distance..... 34

Effective width ..... 25

End support ..... 28

Error message ..... 44, 47

Excel ..... 76

Export of cross-sections..... 74

Export of results ..... 76

## F

Failed design ..... 47

Filter ..... 69

Filtering members ..... 69

Fire protection design..... 66

## G

General data ..... 18

Global offset ..... 56

Graphic ..... 66

Gregely and Lutz equation ..... 13

## H

Hook..... 50

## I

Inclination ..... 32, 49

Indirect support..... 27

Installation ..... 5

Intermediate results..... 55, 66

Internal forces ..... 74

Item ..... 48, 52, 57, 62

Item link ..... 52, 54

Item member ..... 48, 51, 62, 64

## L

Length ..... 50

Length of anchorage ..... 31, 32

Link ..... 32

Link dimensions ..... 53

Link parameter..... 32

Link reinforcement..... 42, 43, 55, 62, 64

Link spacing .....	33, 53
Link zone .....	53
Load case .....	19, 20
Load combination .....	19, 20, 28
Load group .....	19, 20
Load near support .....	28
Location x .....	44, 46, 49, 50, 53, 55, 58, 61
Longitudinal reinforcement .....	30, 42, 43, 48, 55, 62, 64

## **M**

Material .....	21
Material description .....	21
Material Library .....	22
Material properties .....	21
Maximum reinforcement ratio .....	37
Member .....	29, 45, 60
Message .....	51, 53, 59
Minimum reinforcement .....	35
Minimum ticknesses .....	14
Moment swap .....	28
Monolithic connection .....	27
M-ratio $\delta$ .....	28

## **N**

National annex .....	19
Navigator .....	18
Nominal value .....	34
Non-linear calculation .....	48

## **O**

Open RF-CONCRETE Members .....	6
OpenOffice .....	76
Optimization .....	24, 74

## **P**

Panel .....	7
Partial safety factor for concrete .....	37
Partial view .....	69
Percentage of reinforcement .....	37, 56, 74
Plate width .....	34
Plausibility check .....	40
Print .....	71
Printing the graphic .....	71
Printout report .....	70, 76
Provided reinforcement .....	48

## **Q**

Quit RF-CONCRETE Members .....	18, 42
--------------------------------	--------

## **R**

Rebar .....	49
Rebar diameter .....	35, 50, 52, 57
Rebar spacing .....	13, 59
Reinforcement .....	29, 30, 32
Reinforcement area .....	42, 44
Reinforcement drawing .....	62
Reinforcement group .....	29
Reinforcement layer .....	31
Reinforcement layout .....	33, 34
Reinforcement position .....	49
Reinforcement proposal .....	30, 48, 51, 54, 62
Reinforcing steel .....	22
Remark .....	24, 44
Required reinforcement .....	42
Result diagrams .....	68, 71
Result values .....	66
Results evaluation .....	62
Results navigator .....	66
Results tables .....	42
RF-CONCRETE Members case .....	41, 72
RFEM work window .....	65
Rib .....	25, 26

## **S**

Safety .....	56
Scaling .....	67
Section .....	32, 53
Selecting tables .....	18
Selection for printout .....	70
Serviceability .....	12, 20, 48, 58, 66
Serviceability limit state .....	12, 30, 36, 58
Set of members .....	26, 29, 38, 45, 60
Shear force .....	28
Shear force resistance .....	11
Shear reinforcement .....	32, 37, 43, 52
Standard .....	4, 36
Start program .....	6
Steel schedule .....	56
Steel stress .....	59
Strength limit state .....	8, 9, 11
Strength reduction factors .....	8
Structural reinforcement .....	35
Support .....	27
Support width .....	27
Surface .....	57

Surrounding reinforcement ..... 34

**T**

Tables ..... 18

Taper..... 26, 45, 75

Tapered member..... 38

Tapering ..... 38

Tension force ..... 10

Top reinforcement..... 43

Torsion..... 43

Type of anchorage..... 31, 32

**U**

Units ..... 21, 75

**V**

Visualization ..... 63, 66

**W**

Weight..... 51, 53, 57