

Version
December 2017

Program

RF-CONCRETE Members

**Reinforced Concrete Design
According to ACI 318-14**

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.

© **Dlubal Software GmbH**
Am Zellweg 2 D-93464 Tiefenbach

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

Contents

Contents		Page	Contents		Page
1.	Introduction	4	5.1.4	Required Reinforcement by x-Location	48
1.1	Add-on Module RF-CONCRETE Members	4	5.1.5	Required Reinforcement Not Designable	49
1.2	Using the Manual	4	5.2	Provided Reinforcement	50
1.3	Open RF-CONCRETE Members	5	5.2.1	Longitudinal Reinforcement Provided	50
2.	Theoretical Background	7	5.2.2	Provided Shear Reinforcement	55
2.1	Strength Analysis	7	5.2.3	Provided Reinforcement by x-Location	57
2.1.1	Flexure and Axial Loads	7	5.2.4	Steel Schedule	58
2.1.2	Shear Design	10	5.3	Serviceability Limit State Design	60
2.2	Serviceability Limit State Design	11	5.3.1	Serviceability Check by Cross-Section	60
2.2.1	Provided Reinforcement	11	5.3.2	Serviceability Check by Set of Members	62
2.2.2	Limitation of Crack Widths	12	5.3.3	Serviceability Check by Member	62
2.2.3	Limitation of Deformations	13	5.3.4	Serviceability Check by x-Location	63
2.3	Development Length	14	6.	Results Evaluation	64
3.	Input Data	17	6.1	Reinforcement Proposal	64
3.1	General Data	17	6.2	3D Rendering of Reinforcement	65
3.1.1	Strength Limit State	18	6.3	Results on RFEM Model	67
3.1.2	Serviceability Limit State	19	6.3.1	RFEM Background Graphic	67
3.2	Materials	20	6.3.2	RFEM Work Window	68
3.3	Cross-Sections	22	6.4	Result Diagrams	71
3.4	Ribs	24	6.5	Filter for Results	72
3.5	Supports	26	7.	Printout	73
3.6	Reinforcement	28	7.1	Printout Report	73
3.6.1	Longitudinal Reinforcement	30	7.2	Graphic Printout	74
3.6.2	Ties and Stirrups	32	8.	General Functions	75
3.6.3	Reinforcement Layout	33	8.1	Design Cases	75
3.6.4	Min Reinforcement	35	8.2	Cross-section Optimization	77
3.6.5	ACI 318-14	36	8.3	Units and Decimal Places	78
3.6.6	Serviceability	37	8.4	Export of Results	79
3.6.7	Tapered	38	9.	Example	81
3.7	Deflection Data	40	9.1	Input Data	81
4.	Calculation	42	9.2	Flexural Strength	82
4.1	Plausibility Check	42	9.3	Shear Strength	83
4.2	Start Calculation	42	9.4	Development of Reinforcement	84
5.	Results	44	9.5	Results in RF-CONCRETE Members	85
5.1	Required Reinforcement	44	A	Literature	86
5.1.1	Required Reinforcement by Cross-section	44	B	Index	87
5.1.2	Required Reinforcement by Set of Members	47			
5.1.3	Required Reinforcement by Member	47			

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 strength limit state and the serviceability limit state. The analysis for cracks and deflections are performed by calculating crack widths and deformations directly.

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 standards.

- EN 1992-1-1:2004
- **ACI 318-14**
- CSA A23.3-14 (R2015)
- SIA 262:2013
- GB 50010-2010

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.

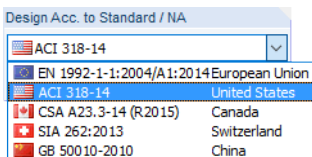
Your team from ING.-SOFTWARE DLUBAL

1.2 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.

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. If you cannot find what you are looking for, please check the **Knowledge Base** and **FAQs** on our website for answers to your questions.



Design acc. to ACI 318-14

Graphics

1.3 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**.

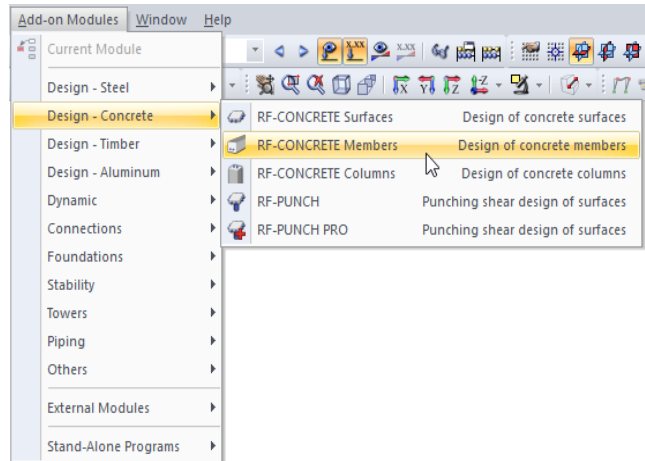


Figure 1.1: Menu *Additional Modules* → *Design - Concrete* → *RF-CONCRETE Members*

Navigator

To start RF-CONCRETE Members in the *Data* navigator, open the **Additional Modules** folder, and then select **RF-CONCRETE Members**.

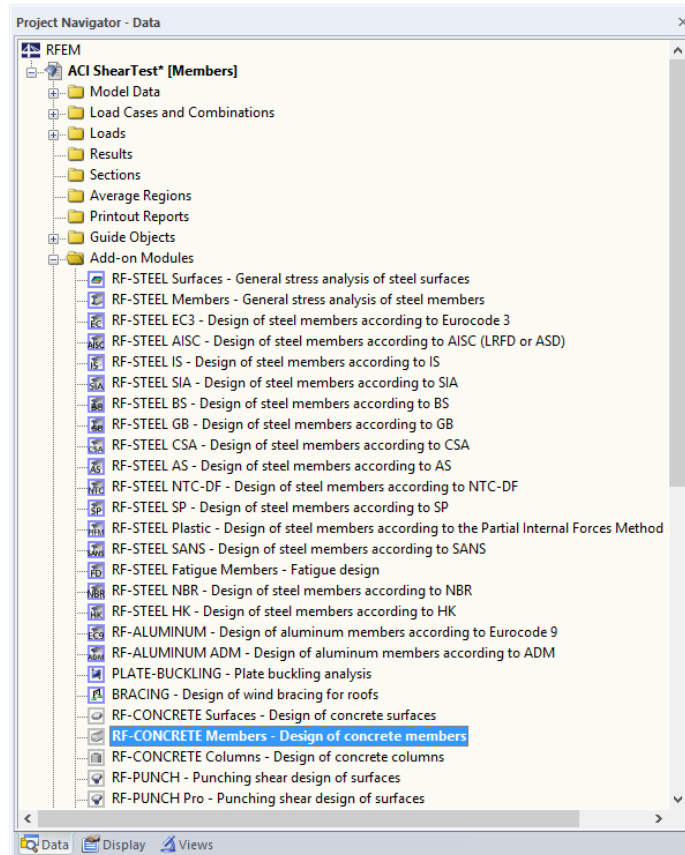
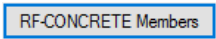
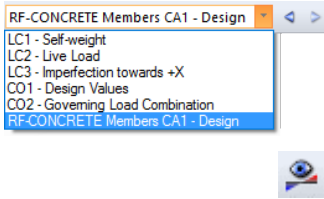


Figure 1.2: *Data* navigator: *Additional Modules* → *RF-CONCRETE Members*



Panel

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

When the results are shown, the panel appears showing the button [RF-CONCRETE Members]. You can use that button to access the design module.

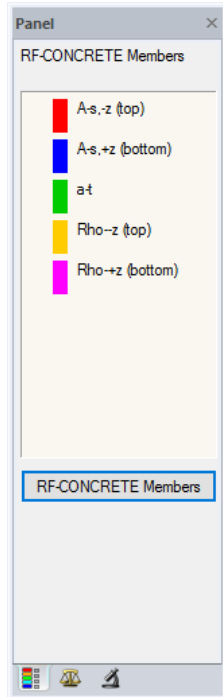


Figure 1.3: Panel button [RF-CONCRETE Members]

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 318-14 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
 ϕR_n Design strength

ϕ are the strength reduction factors. The design strength is achieved by multiplying the strength reduction factor ϕ by nominal strength which is the theoretical ultimate strength. The Code specifies different values of ϕ depending on the state of knowledge (the accuracy with which various strengths can be calculated). The prescribed strength reduction factors according to ACI Section 21.2 are in the following table.

Strength Condition	Strength Reduction Factor ϕ
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 ϕ values for most situations

ACI 318-14 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 the tensile strain in the extreme layer of longitudinal reinforcement ε_{ty} . When ε_t values fall between ε_{ty} 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 ϕ based on ε_t .

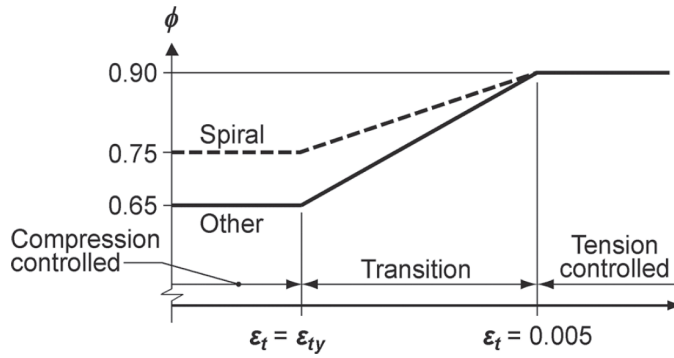


Figure 2.1: Variation of strength reduction factor with net tensile strain

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.

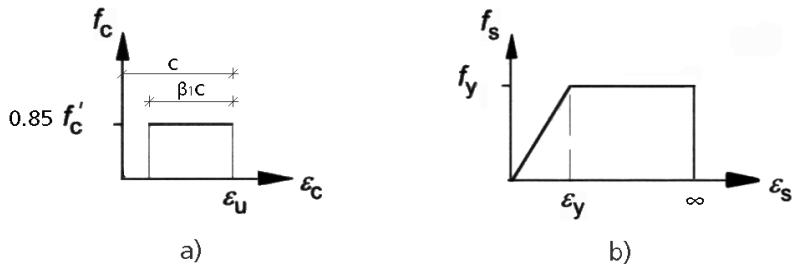


Figure 2.2: Simplified stress-strains diagrams

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

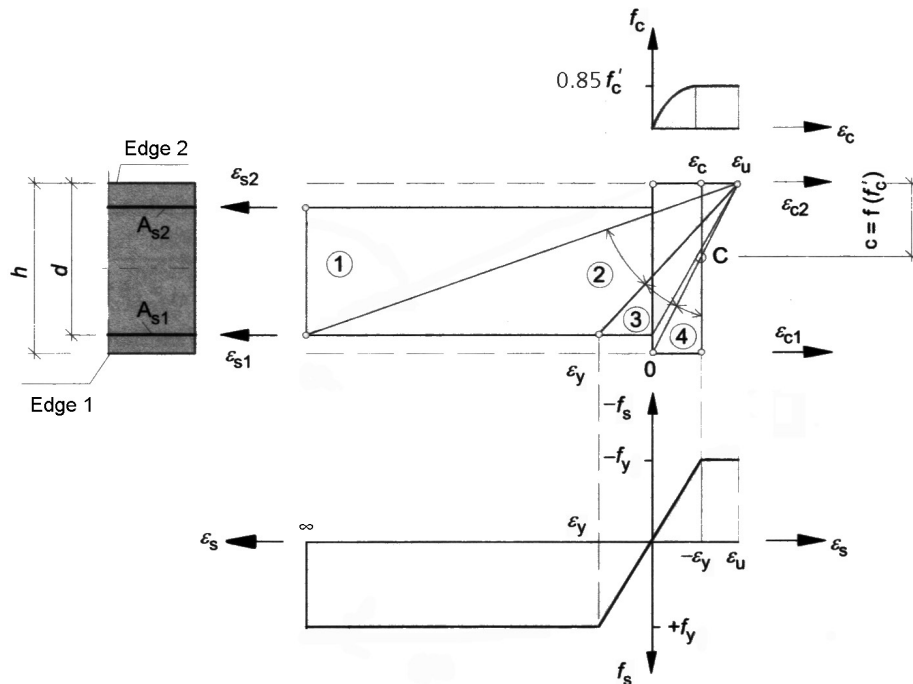


Figure 2.3: Possible strain distributions in strength limit state

According to ACI 318-14 and [3], 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 318-14, 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 ACI 318-14, Eq. (22.5.1.1)
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:

$$A_{v,min} = 0.75 \sqrt{f'_c} \frac{b_w s}{f_{yt}} \geq 50 \frac{b_w s}{f_{yt}} \quad \text{ACI 318-14, 9.6.3.3 / 10.6.2.2 / 15.4.2}$$

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:

$$V_c = \left(1.9 \lambda \sqrt{f'_c} + 2500 \rho_w \frac{V_u d}{M_u} \right) b_w d \leq 3.5 \lambda \sqrt{f'_c} b_w d \quad \text{ACI 318-14, Table 22.5.5.1}$$

where	$\sqrt{f'_c} \leq 100 \text{ psi}$	Square root of compressive strength of concrete
	λ	Lightweight concrete modification factor
	ρ_w	Ratio of longitudinal reinforcement
	Ratio $\frac{V_u d}{M_u}$	shall not be greater than 1.0

Alternative simplified calculation of V_c for nonprestressed members without axial force is permitted by ACI 318-14, 22.5.5.1:

$$V_c = 2 \lambda \sqrt{f'_c} b_w d \quad \text{ACI 318-14, Eq. (22.5.5.1)}$$

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:

$$V_s = \frac{A_v f_{yt} d}{s} \quad \text{ACI 318-14, Eq. (22.5.10.5.3)}$$

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:

$$V_s = \frac{A_v f_{yt} (\sin \alpha + \cos \alpha) d}{s} \quad \text{ACI 318-14, Eq. (22.5.10.5.4)}$$

where	α	Angle between stirrups and longitudinal axis of member
-------	----------	--

According to ACI 318-14, 22.5.1.2, 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° 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 318-14, 24.3
- Limitation of deformation: ACI 318-14, 24.2

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 strength 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, wetting and drying	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
	β	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 24.3.2. The rebar spacing should not exceeded the value computed with the following expression:

$$s = 15 \left(\frac{40,000}{f_s} \right) - 2.5c_c \leq 12 \left(\frac{40,000}{f_s} \right) \quad \text{ACI 318-14, Table 24.3.2}$$

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 Table 24.3.2 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 ratios of one-way constructions are specified in ACI 318-14, 7.3.1, Table 7.3.1.1 and 9.3.1, Table 9.3.1.1. These tables apply only to members that are not supporting or not attached to partitions or other constructions likely to be damaged by excessive deflections.

Member	Minimum thickness, <i>h</i>			
	Simply supported	One end continuous	Both ends continuous	Cantilever
	Members not supporting or attached to partitions or other construction likely to be damaged by large deflections			
Solid one-way slabs	<i>l</i> /20	<i>l</i> /24	<i>l</i> /28	<i>l</i> /10
Beams or ribbed one-way slabs	<i>l</i> /16	<i>l</i> /18.5	<i>l</i> /21	<i>l</i> /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/ft³, 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 318-14, Table 7.3.1.1 and Table 9.3.1.1

Calculation of deflection

When there is need to use member depths shallower than are permitted by Tables 7.3.1.1 and 9.3.1.1 or when members support a construction that is likely to be damaged by large deflections, the deflection must be calculated and compared with limit values, see the following figure.

Member	Condition	Deflection to be considered	Deflection limitation
Flat roofs	Not supporting or attached to nonstructural elements likely to be damaged by large deflections	Immediate deflection due to maximum of L , S , and R	<i>l</i> /180 ^[1]
Floors		Immediate deflection due to L	<i>l</i> /360
Roof or floors	Supporting or attached to non-structural elements	Likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements, which is the sum of the time-dependent deflection due to all sustained loads and the immediate deflection due to any additional live load ^[2]
		Not likely to be damaged by large deflections	

^[1]Limit not intended to safeguard against ponding. Ponding shall be checked by calculations of deflection, including added deflections due to ponded water, and considering time-dependent effects of sustained loads, camber, construction tolerances, and reliability of provisions for drainage.

^[2]Time-dependent deflection shall be calculated in accordance with 24.2.4, but shall be permitted to be reduced by amount of deflection calculated to occur before attachment of nonstructural elements. This amount shall be calculated on basis of accepted engineering data relating to time-deflection characteristics of members similar to those being considered.

^[3]Limit shall be permitted to be exceeded if measures are taken to prevent damage to supported or attached elements.

^[4]Limit shall not exceed tolerance provided for nonstructural elements.

Figure 2.5: Maximum permissible computed deflection according to ACI 318-14, Table 24.2.2

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 f 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.

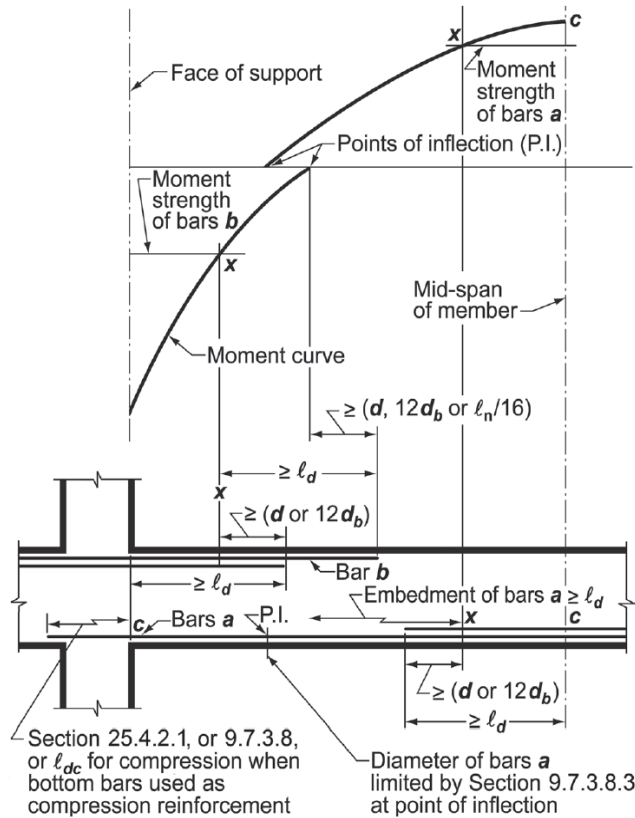


Figure 2.6: Development of flexural reinforcement according to ACI 318-14 9.7.3, Figure R9.7.3.2

Development of tension bars

The development length of straight deformed bars and wires in tension is given in 25.4.2.3 of the Code by the general equation:

$$l_d = \left(\frac{3}{40} \frac{f_y}{\lambda \sqrt{f_c'}} \frac{\Psi_t \Psi_e \Psi_s}{\left(\frac{c_b + K_{tr}}{d_b} \right)} \right) \cdot d_b \quad \text{ACI 318-14, Eq. (25.4.2.3a)}$$

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
	λ	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
	Ψ_t	Reinforcement location factor
	Ψ_e	Coating factor
	Ψ_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 318-14, 25.4.10.1.

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 318-14, 25.4.9 the development length in compression can be determined from the following formula:

$$l_{dc} = \frac{f_y \psi_r}{50 \lambda \sqrt{f_c'}} d_b \geq 0.0003 f_y \psi_r d_b$$

where	$\sqrt{f_c'} \leq 100$ psi	Square root of compressive strength of concrete
	f_y	Specified yield strength of reinforcement in psi
	d_b	Diameter of bar in inches
	λ	Lightweight concrete modification factor
	ψ_r	Development length modification factor based on confining reinforcement

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 318-14, 25.4.10.1.

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 318-14, 25.4.3.1:

$$l_{dh} = \frac{f_y \psi_e \psi_c \psi_r}{50 \lambda \sqrt{f_c'}} d_b \geq 8 d_b \geq 6 \text{ in.}$$

where	$\sqrt{f_c'} \leq 100$ psi	Square root of compressive strength of concrete
	f_y	Specified yield strength of reinforcement
	d_b	Diameter of bar
	λ	Lightweight concrete modification factor
	ψ_e	Development length modification factor based on coating
	ψ_c	Development length modification factor based on cover
	ψ_r	Development length modification factor based on confining reinforcement

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 windows. The [Select] function allows for a graphical selection of the objects that you want to design.

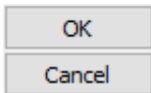
When you have started the add-on module, a new window opens where a navigator is displayed on the left, managing all windows that can be selected currently. The pull-down list above the navigator contains the design cases that are already available (see Chapter 8.1, page 75).

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

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



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



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 Window 1.1 *General Data*, select the actions that you want to design. The relevant load cases, load combinations and result combinations can be assigned to the strength limit state and the serviceability limit state by using the respective tab.

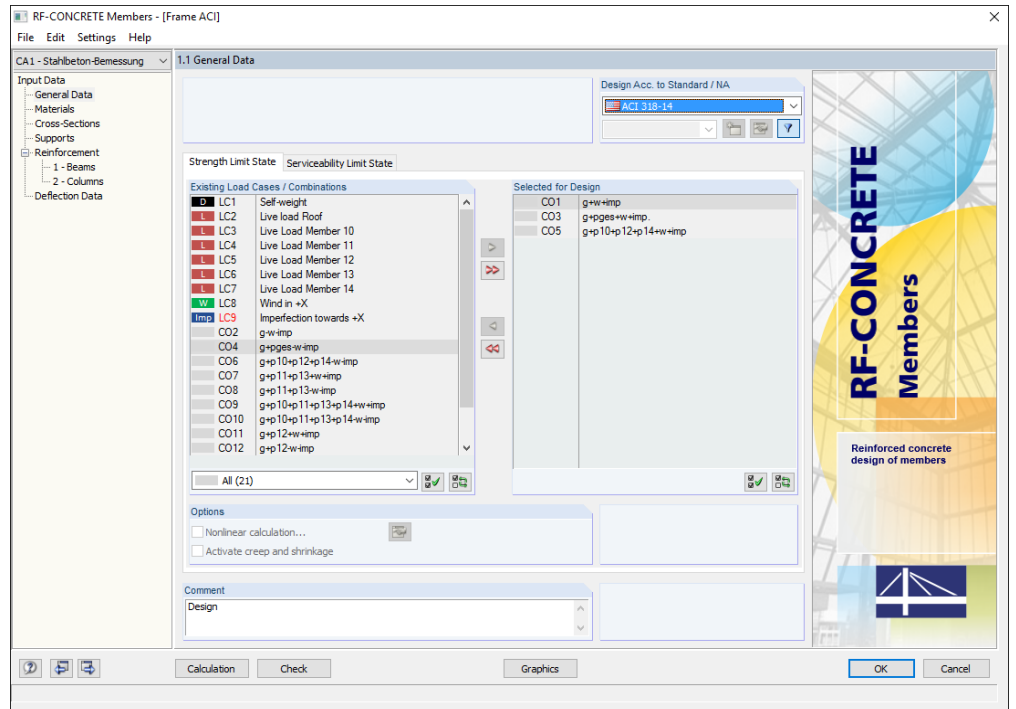


Figure 3.1: Window 1.1 *General Data*, tab *Strength Limit State*



Concrete Design According to Standard / NA

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

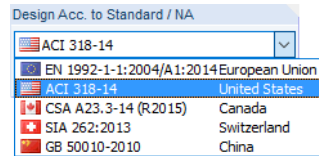


Figure 3.2: Selection of design standard

3.1.1 Strength Limit State

The first tab of Window 1.1 *General Data* is shown in Figure 3.1 on page 17.

Existing Load Cases / Combinations



This window section lists all load cases, load and result combinations defined in RFEM that are relevant for the design. Use the [▶] button to transfer the selected load cases (LC), load combinations (CO) or result combinations (RC) 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 red like LC 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, load or result 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* result combination is often carried out more quickly than the design of all load cases and load combinations that have been globally set. On the other hand, the influence of the actions contained in a result combination is less transparent when a RC is designed.

Comment

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

3.1.2 Serviceability Limit State

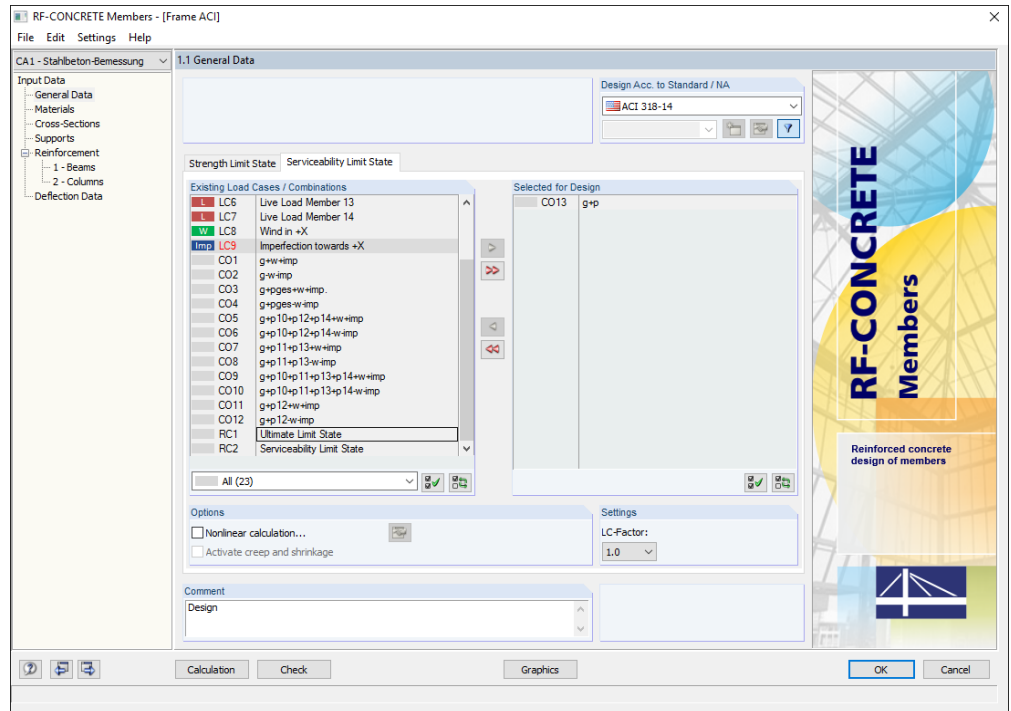


Figure 3.3: Table 1.1 General Data, tab Serviceability Limit State

Existing Load Cases / Combinations

This dialog section lists all load cases and combinations defined in RFEM. Use the [▶] button to transfer the selected load cases (LC), load combinations (CO) or result combinations (RC) to the list *Selected for Design* on the right. You can also double-click the items. To transfer the entire list to the right, use the button [▶▶].



Selected for Design

The column on the right lists the items selected for the serviceability limit state design. Use the button [◀] to remove selected load cases, load or result 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 window 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 4.3 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** on the **Settings** menu (see Figure 8.6, page 78).

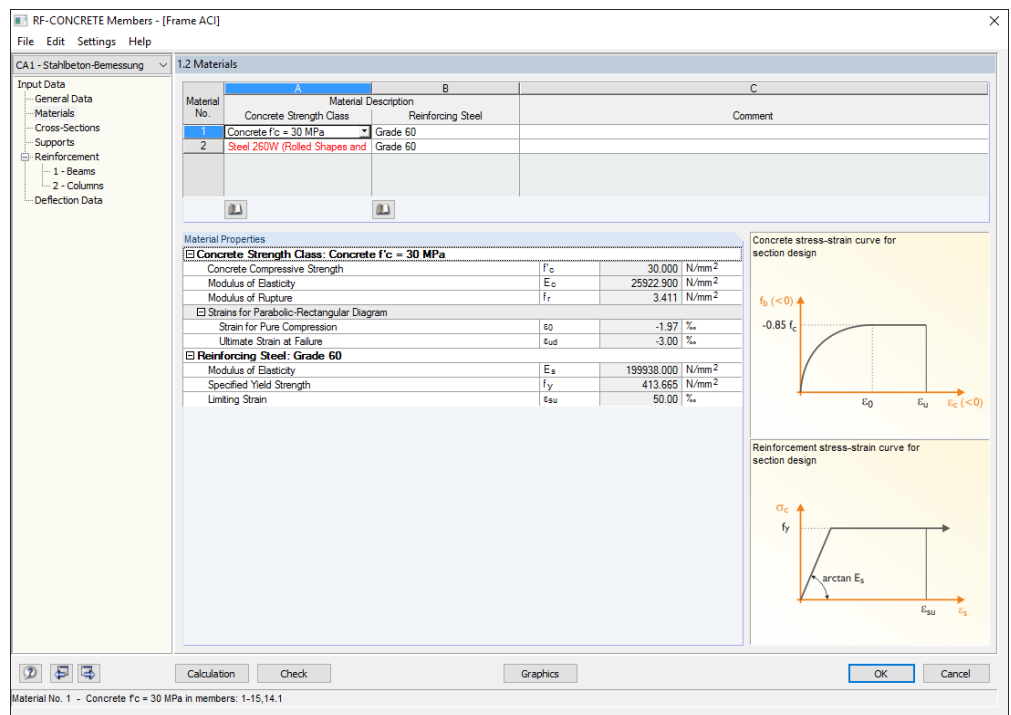


Figure 3.4: Window 1.2 Materials

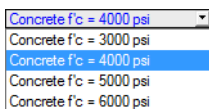
Material Description

Concrete Strength Class

The concrete materials defined in RFEM are preset. Materials of a different material type are highlighted in red. When a manually entered *Material Description* corresponds to an entry of the material library, RF-CONCRETE Members imports the relevant material properties.

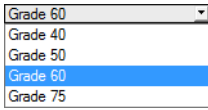
It is possible to select a different material by using the list: Place the pointer in a table row of column A, and then click the button [▼] or use the function key [F7]. The list shown on the left opens. Subsequent to the transfer, the properties will be updated.

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.

Material Library

Many 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.

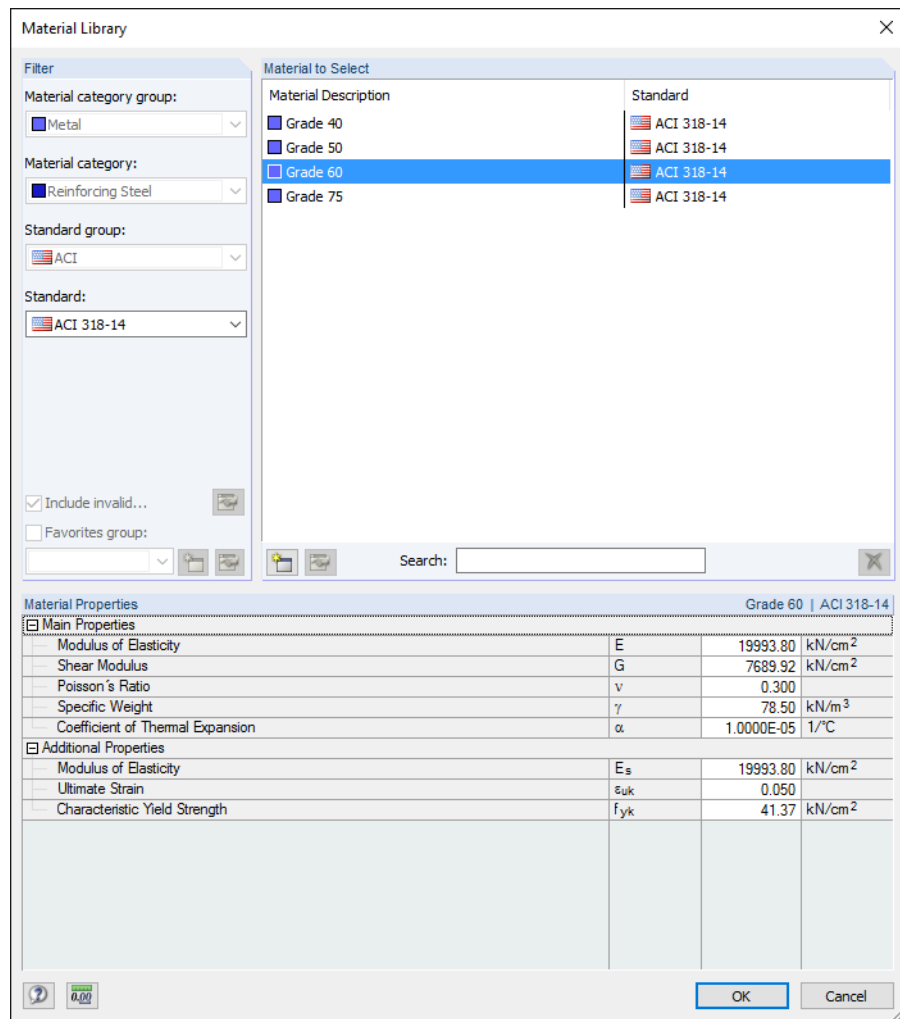


Figure 3.5: Dialog box *Material Library*

The material relevant for the standard are preset so that no other categories or standards are available in the *Filter* section. Select a material from the list *Material to Select* and check the corresponding parameters in the lower part of the dialog box. The material properties cannot be edited in this dialog box.

Click [OK], use the [...] button, or double-click the material to import the selected material to *Window 1.2 Materials* of the add-on module.



Chapter 4.3 of the RFEM manual describes how materials can be added or rearranged. Via 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 window lists the design relevant cross-sections.

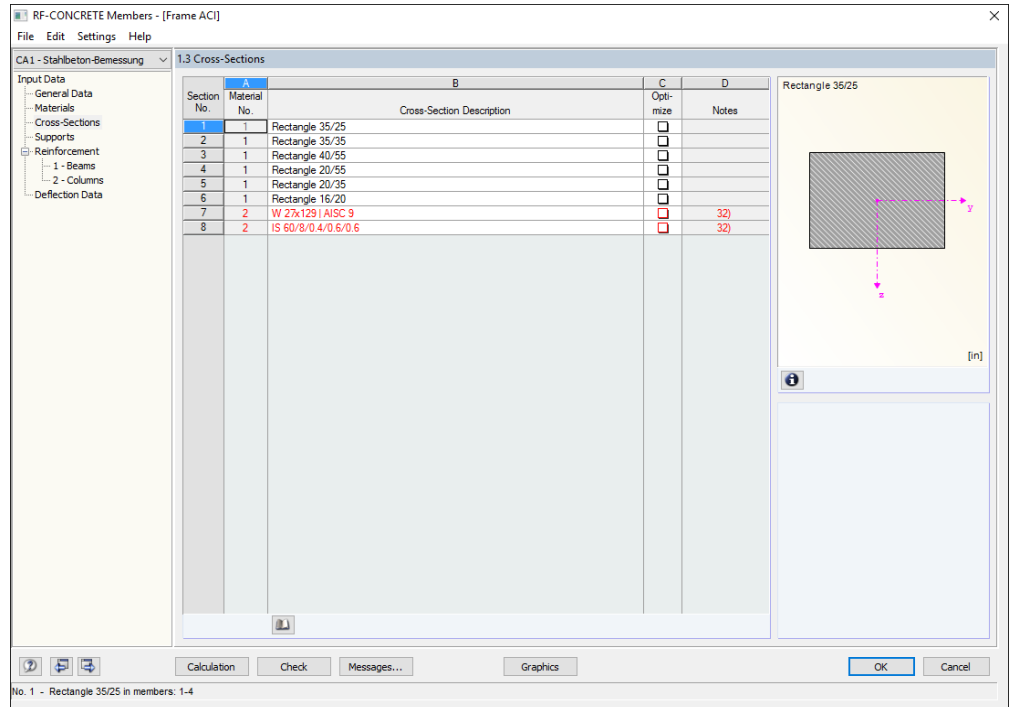


Figure 3.6: Window 1.3 Cross-Sections

Cross-Section Description

When you open this window, the cross-sections used in RFEM are preset 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 change a cross-section, enter the new description in the corresponding table row. You can also select the new cross-section from the [Library] by clicking the corresponding button 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, the following buttons are enabled in the *Parametric - Massive* section of the library (see Figure 3.7):

- 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 Chapter 4.13 of the RFEM manual.

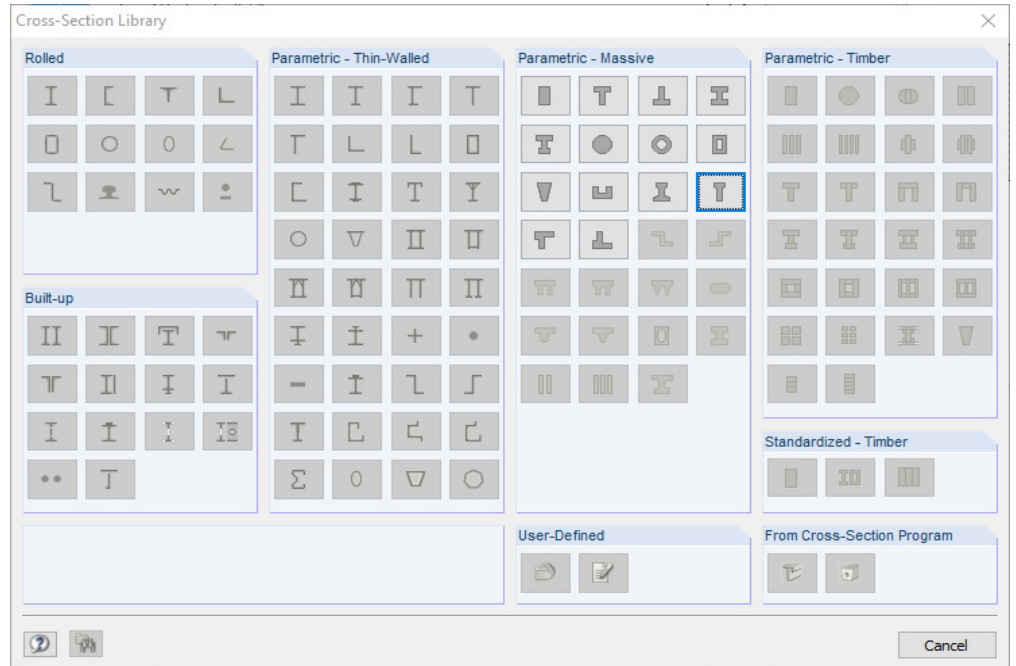
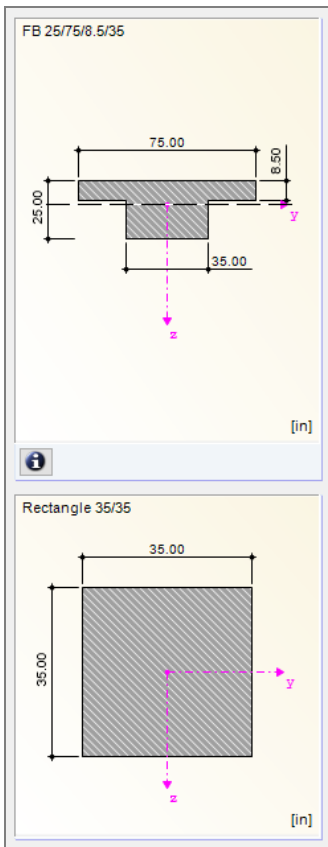


Figure 3.7: Cross-section Library

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 window.

Optimize

For each cross-section it is possible to perform an optimization analysis. Applying the internal forces from RFEM, the program determines the cross-section within the same section table that meets the reinforcement requirements specified in the *Optimization Parameters* dialog box with the least possible dimensions (see Figure 8.5, page 77).

To optimize a specific cross-section, select its check box in column C. Recommendations on the optimization can be found in Chapter 8.2 on page 77.

Notes

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

3.4 Ribs

The ribs defined in RFEM are preset. Ribs represent a special type of member consisting of a beam and a plate cross-section which is also effective (cf. Chapter 4.18 of the RFEM manual). The program takes the rib internal forces from RFEM and uses them for the design.

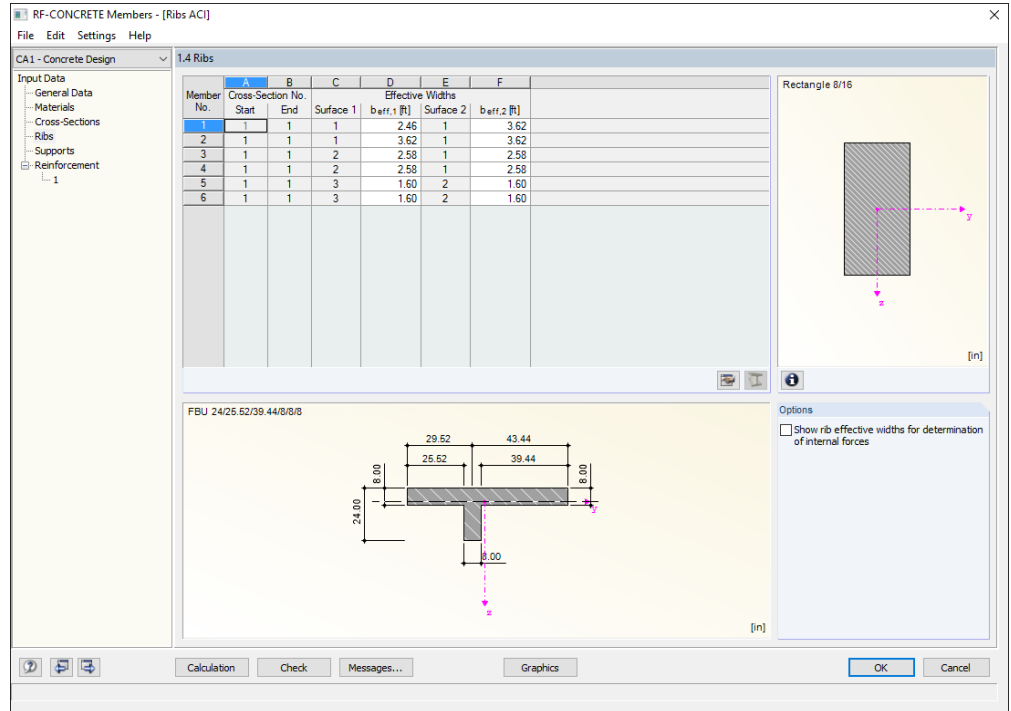


Figure 3.8: Window 1.4 Ribs

The effective widths in this table can be modified either directly by entering values in columns D and F or indirectly by using the [Edit Rib] button. A recalculation in RFEM is not required as the stiffness of the model 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

These columns show the cross-section numbers (see Chapter 3.3). If different numbers are displayed, the object is a tapered member.

Effective Widths b_{eff}

Column D and F show the effective widths for the left and the right sides of the member. The values are identical with the specifications entered in the RFEM dialog box *New Rib* (cf. RFEM manual, Figure 4.172). The rib internal forces are determined on the basis of the integration widths for the pro rata internal forces in surfaces.



The effective width is important for the design with respect to the equivalent cross-section. Therefore, you can adjust the values for b_{eff} (increasing the integration width is not allowed, however). To check the data, select the check box *Show rib effective widths for determination of internal forces*: The table is extended by two more columns, and the button [Edit Rib] becomes accessible (see Figure 3.9).



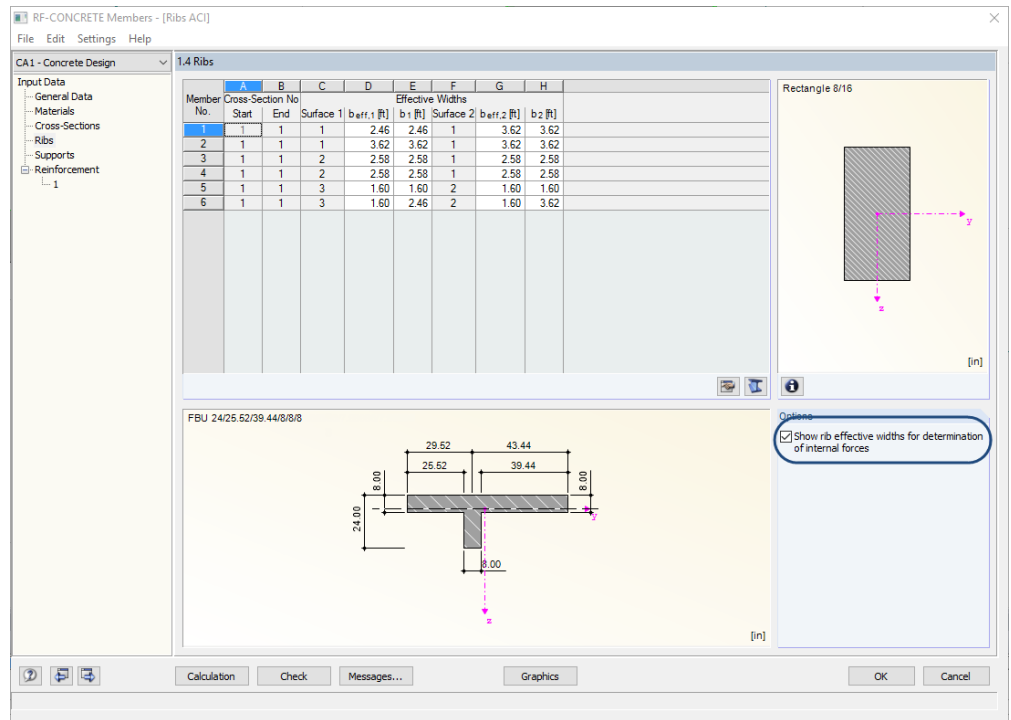


Figure 3.9: Window 1.4 Ribs

Modifications are shown dynamically 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 which affect the design in RF-CONCRETE Members.

Remark

If a rib represents a problem for the design, a note is given.

The buttons below the table have the following functions:

Button	Description	Function
	Edit	Opens <i>Solid Cross-Sections – Unsymmetric Floor Beam</i> dialog box with parameters of equivalent cross-section
	Edit Rib	Opens <i>Edit Rib</i> dialog box with rib parameters (cf. RFEM manual, Figure 4.172)
	Info about Cross-Section	Shows cross-section properties of equivalent cross-section (type: FBU - floor beam unsymmetric)

Table 3.1: Buttons in Window 1.4 Ribs



To design ribs in a correct manner, consider the following requirements:

- The local z-axis of the rib must be parallel to the local z-axis of the surface.
- The local z-axis of the rib 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 a set of members is used, a uniform rib type must be defined for the entire set.
- The rib member must have the same cross-section at its start and its end (i.e. no taper).

3.5 Supports

This window provides information about the support conditions of the members that are to be designed. The nodal supports defined on horizontal members in RFEM are preset. If necessary, it is possible to adjust them. RF-CONCRETE Members also distinguishes between intermediate and end supports.



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!

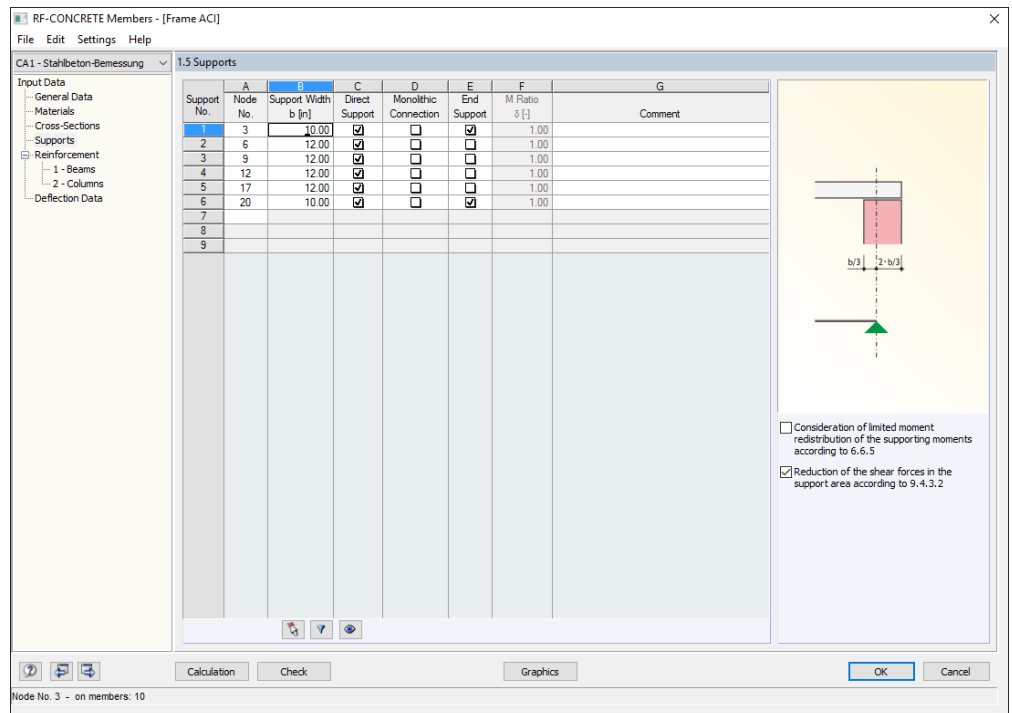


Figure 3.10: Window 1.5 Supports

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, the bearing area of a wall which is modeled 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 adjacent 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 is available, or if there is a free rotary support including reduction options for the supporting moments.

End Support

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

M-Ratio δ

For continuous structural components you can define the ratio δ for the redistributed moment and the elastically determined initial moment. Column F is only available if you have selected the option *Consideration of a limited moment redistribution of the supporting moments according to 6.6.5* below the graphic on the right.

The δ -values can be determined according to ACI 318-14, 6.6.5.

Comment

In the final column, you can enter comments, e.g. to describe the support conditions.

Taking into account the support widths

Below the interactive graphic in the window, you find two check boxes whose specifications, have different effects on the required reinforcement. Those settings are globally valid for the current design case.

Consideration of limited moment redistribution

For continuous beams it is possible to apply the linear-elastic methods with limited redistribution of the supporting moments. The resulting distribution of internal forces must be at equilibrium with the acting loads. The standards describe the moment ratios δ that must be observed in order to ensure the ability for rotation in the critical areas without special designs, for example ACI 318-14, 6.6.5.

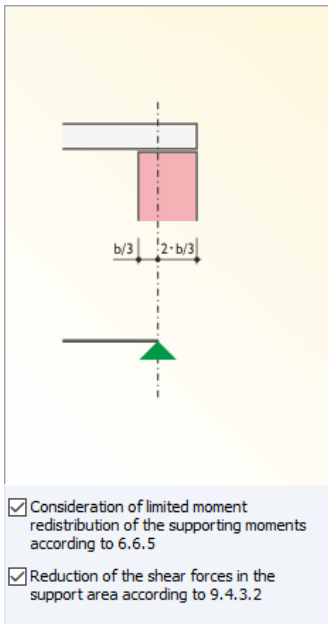
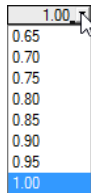
RF-CONCRETE Members determines this limit value and compares it with the value that is specified in column F. The higher value is used for the redistribution.

Reduction of the shear forces in the support area

For direct supports, the design value of the shear force can be reduced (cf. ACI 318-14, 9.4.3.2).

Loads near supports are taken into account, regardless of whether the check box for shear force reduction is selected – but only if they are included in load cases or load combinations.

If there is a result combination, the shear force is designed generally on the edge of the support because the requirement of the Code for a "uniformly distributed load" by means of envelopes is not verifiable in detail.



3.6 Reinforcement

This window consists of several tabs in which all reinforcement data is to be 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 Group

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

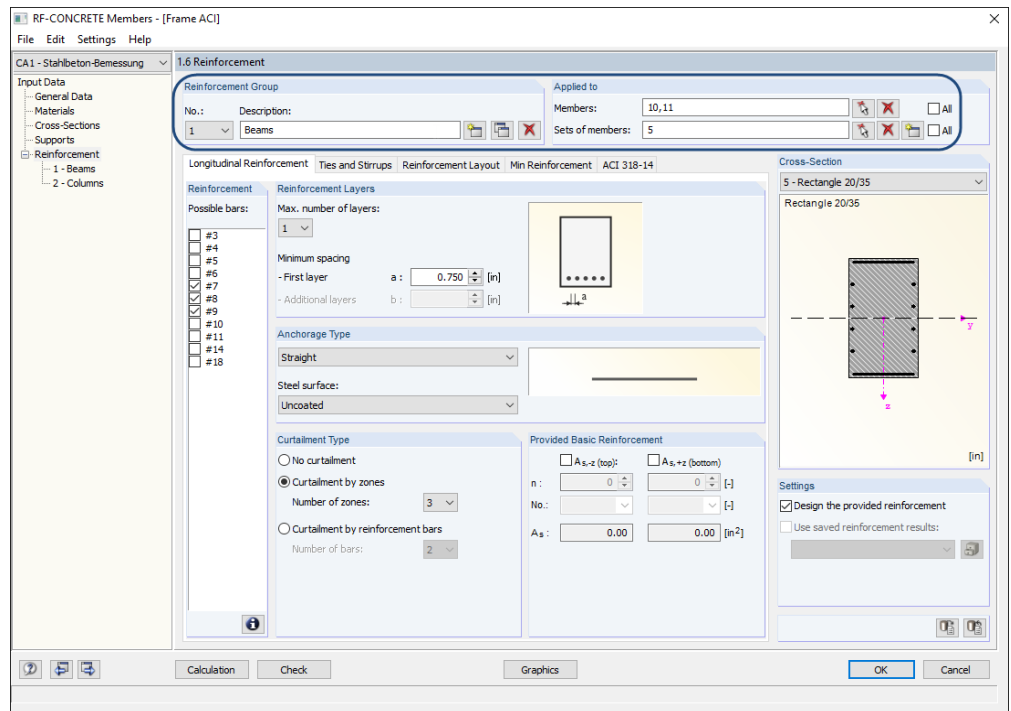


Figure 3.11: Window 1.6 Reinforcement with two reinforcement groups

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 that are part of the current reinforcement group. *All* members and *All* sets of members are preset. If the check boxes are selected, no further reinforcement group can be created: Members and sets of members cannot be designed more than once with 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.

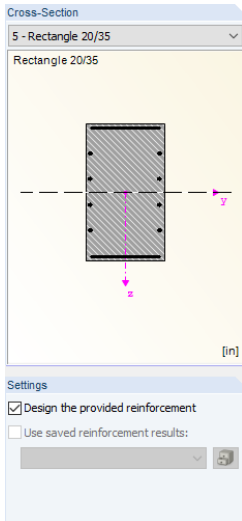
Enter the numbers of the relevant members or sets of members into the text boxes. The reinforcement specifications defined in the tabs of the window will be valid for all selected objects. You can also use the [Select] button to determine the objects graphically in the RFEM work window. You may only select members and sets of members that have not been assigned to an existing reinforcement group yet.

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



Cross-Section

The graphic in the right part of the window shows how the input in the tabs affect the cross-section. Use the list above to select a different cross-section. The graphic is dynamic: Modifications to the specifications of the reinforcement are instantly displayed.



Settings

If the check box *Design the provided reinforcement* is selected, 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 text boxes cannot be accessed. In this case, the program will determine only the required reinforcement areas.

If you activate the design for the serviceability limit state in Window 1.1 *General Data*, it is not possible to clear the above-mentioned check box because the SLS design is based on a provided reinforcement: Crack widths, crack spacings etc. can be determined only by the applied rebar diameters and spacings.

Select the check box *Use saved reinforcement results* to apply a reinforcement stored in the user-defined library of templates. Reinforcement templates can be defined and saved in module Window 3.1 (see Chapter 5.2.1, page 54). Then, the check box will be enabled.

The appropriate reinforcement template can be selected in the list. Use the button [Select template] to access the database of reinforcement templates where you can choose among the templates that you have stored.

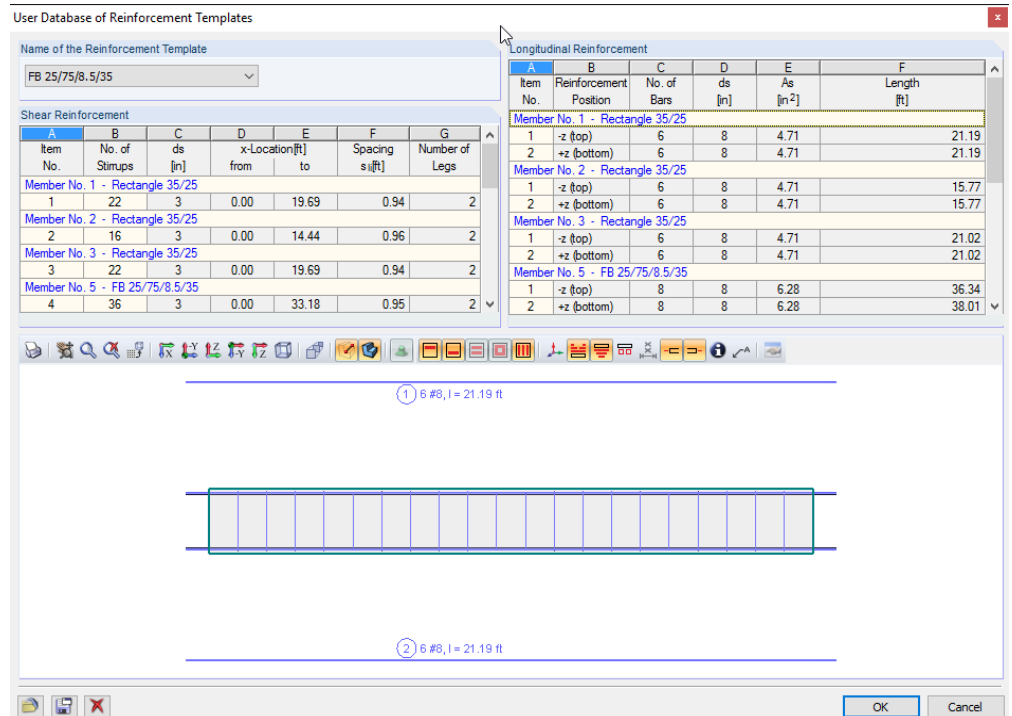
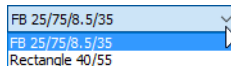


Figure 3.12: Dialog box *User Database of Reinforcement Templates*



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

The reinforcement stored in the reinforcement template is not dynamic. This means that the positions and lengths of the reinforcement are fixed and assigned to a particular member. So, if the original length of a member is changed in RFEM, it won't be adjusted in the template.

3.6.1 Longitudinal Reinforcement

In this tab, you enter the specifications to define the longitudinal reinforcement.

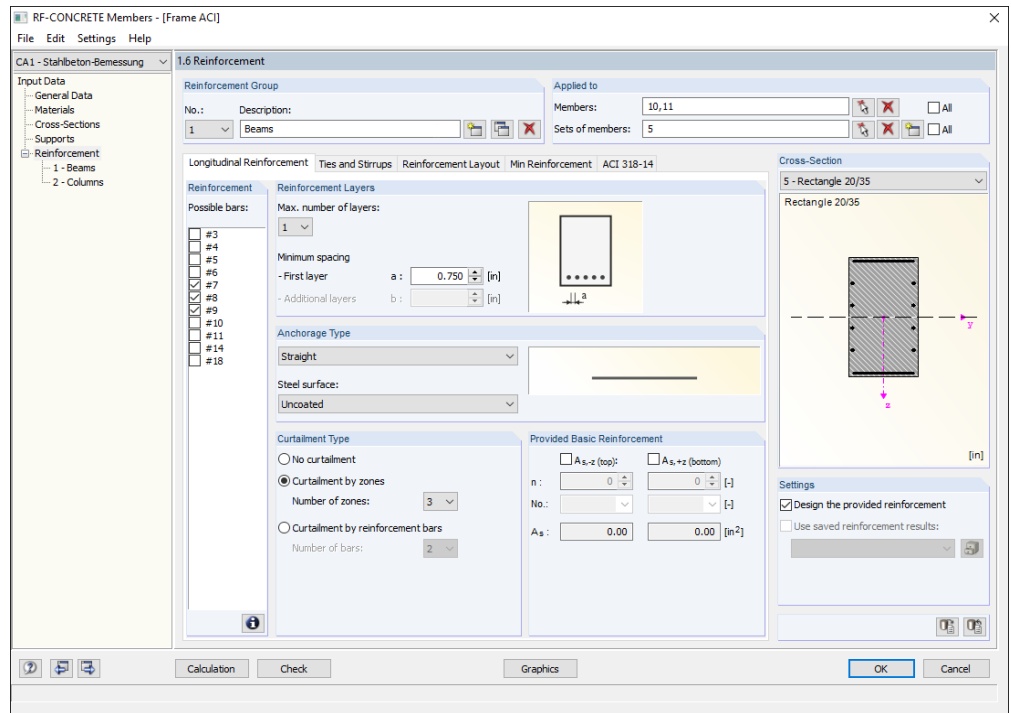


Figure 3.13: Window 1.6 Reinforcement, tab Longitudinal Reinforcement

Reinforcement

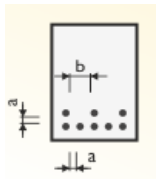
The list of *Possible bars* includes common ASTM Standard reinforcement bars. A multiple selection is no problem for the design.



Use the [Info] button to view the diameters and cross-sectional areas associated with the bar numbers.

A	B	C	D
Bars No.		Diameter	Cross-Sectional
Inch-Pound	SI	[in]	Area [in ²]
3	10	0.375	0.11
4	13	0.500	0.20
5	16	0.625	0.31
6	19	0.750	0.44
7	22	0.875	0.60
8	25	1.000	0.79
9	29	1.128	1.00
10	32	1.270	1.27
11	36	1.410	1.56
14	43	1.693	2.25
18	57	2.257	4.00

Figure 3.14: Info box of rebar properties



- Straight
 - No anchorage
 - Straight
 - Hook
 - Bend
 - Straight with transverse bar
 - Hook with transverse bar
 - Straight with two transverse bars
-
- Uncoated
 - Uncoated
 - Zink-coated
 - Epoxy-coated

Reinforcement Layers

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

When RF-CONCRETE Members creates the 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

The two lists provide a variety of possibilities with respect to the anchorage. The graphic to the right is dynamic, which means that modified settings are immediately displayed.

The anchorage type as well as the steel surface affect the required length of anchorage.

Curtailment Type

No curtailment is preset. If you have specified more than one reinforcement layer, 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, each with identical reinforcement, are allowed in the reinforcement proposal. With this setting, RF-CONCRETE Members determines 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 layers. When the check boxes are selected, the boxes below become active. Enter the number of rebars *n* and the rebar number *No.* to define the reinforcement. The *A_s* text boxes 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 along the entire length of the member or set of members. If the required reinforcement cannot be covered by the basic reinforcement, the program calculates the additional rebars and inserts them into the cross-section.

3.6.2 Ties and Stirrups

This tab manages the specifications concerning the shear reinforcement.

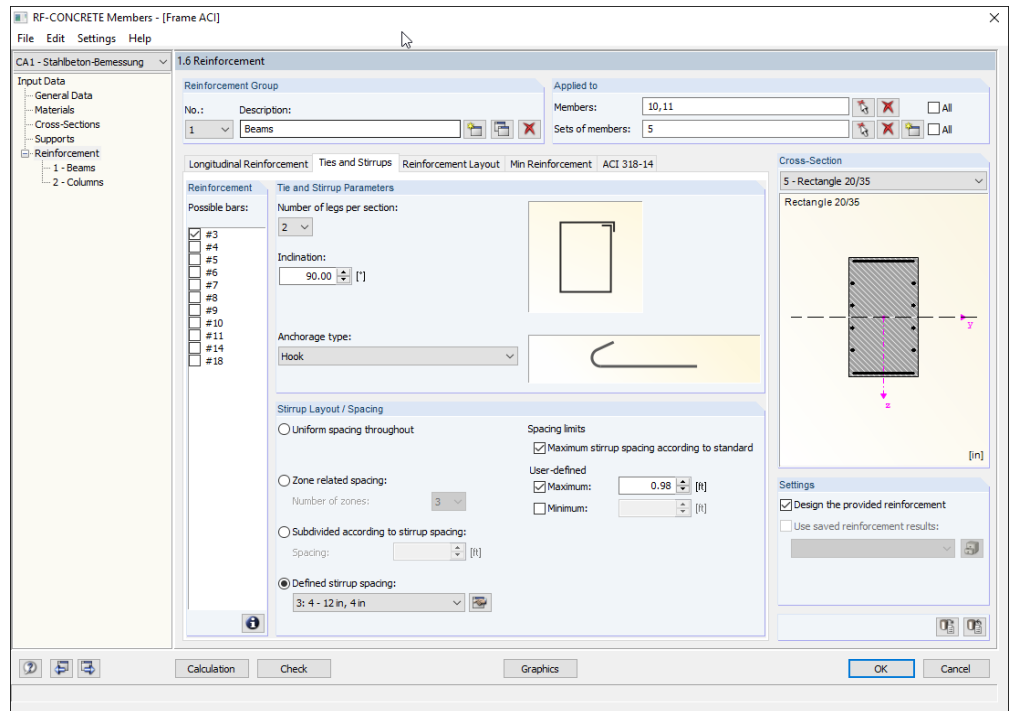


Figure 3.15: Window 1.6 Reinforcement, tab Ties and Stirrups

Reinforcement

The list of *Possible bars* includes common ASTM Standard reinforcement bars. A multiple selection is no problem for the design.



Use the [Info] button to view the areas and diameters associated with the bar designation numbers (see Figure 3.14).

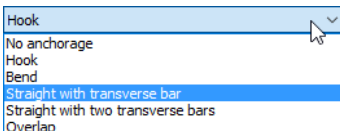
Tie and Stirrup Parameters

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

The *Inclination* of the shear reinforcement is defined by the angle between the longitudinal and the shear reinforcement. It is preset to 90°, which means that the stirrups are perpendicular to the longitudinal rebars. If you want to modify the inclination, respect the specifications of the Code for nonprestressed members: ACI 318-14, 22.5.10.5.2 only allows for angles between 45° and 90°.

Anchorage Type

The list provides numerous possibilities for stirrup anchorages which affect the anchorage length. The graphic to the right is dynamic, which means that modified settings are displayed immediately.



Stirrup Layout / Spacing

This dialog section is only available when a reinforcement proposal is created.

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 stirrup layout can be assumed for the reinforcement proposal. For one zone, the

program creates one extra zone in addition to the zone with the maximum stirrup spacing (i.e. the minimum reinforcement). The additional zone covers the maximum value of the required shear reinforcement. If two zones are set, RF-CONCRETE Members determines the mean value from the required minimum and maximum reinforcement and applies the corresponding member x-locations as additional zone limits. The division into three or more zones is similar.

If the layout is *Subdivided according to stirrup spacing*, you define a particular spacing for the stirrup 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 choose a *Defined stirrup spacing*, you can select an option 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 spacings.

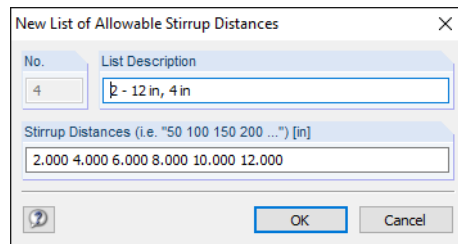
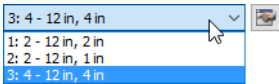


Figure 3.16: Dialog box *New List of Allowable Stirrup Distances*



The *Maximum* and the *Minimum* spacing of the stirrup reinforcement can be defined directly.

The zones determined by the program and shown in the reinforcement proposal can be modified or supplemented in Window 3.2 *Provided Shear Reinforcement* (see Chapter 5.2.2, page 55).

3.6.3 Reinforcement Layout

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

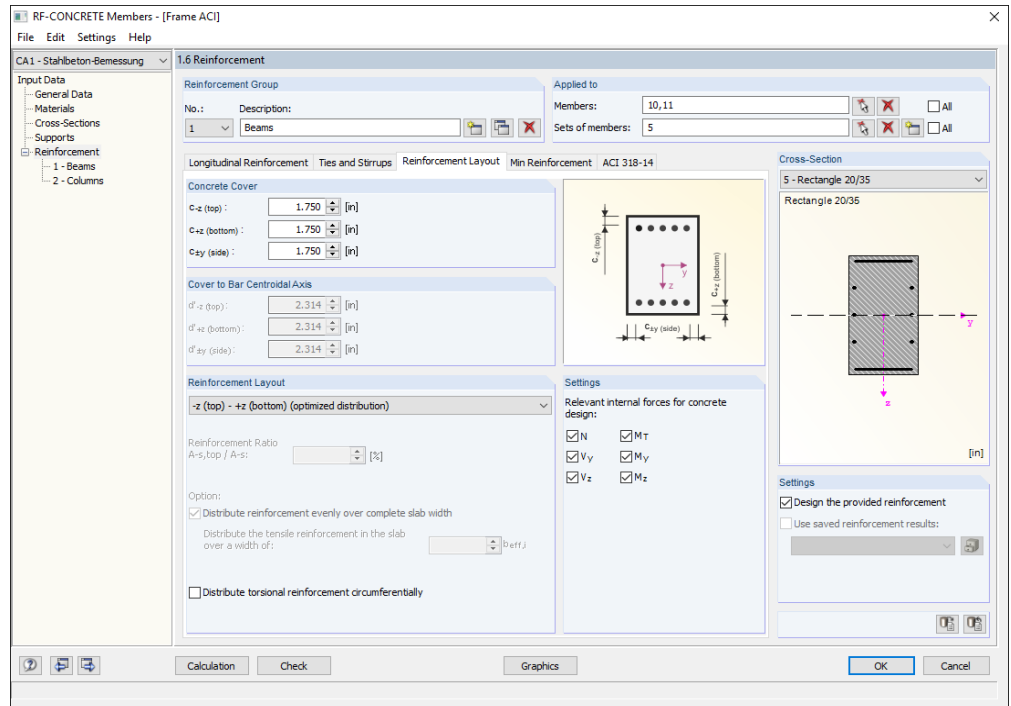


Figure 3.17: Window *1.6 Reinforcement*, tab *Reinforcement Layout*

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 edge distances c of the rebars. If no provided reinforcement is to be determined, the specifications refer to the dimensions of the centroidal axis u of the rebars. Figure 3.18 explains the difference.

Depending on the settings, you can access the upper or lower text boxes.

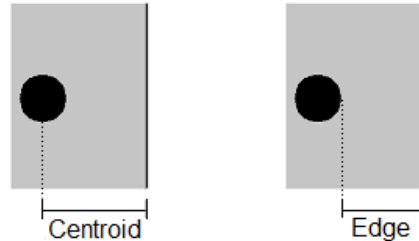
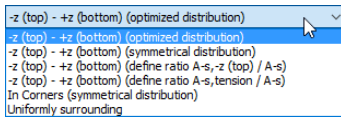
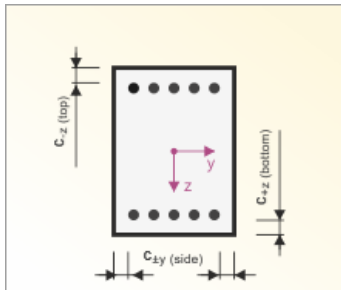


Figure 3.18: Relations of concrete cover

In the $c_{-z(top)}$ box, enter the concrete cover of the longitudinal reinforcement at the top side of the member. The $c_{+z(bottom)}$ box accordingly defines the cover of the longitudinal reinforcement at the bottom. Those values represent the values c_c of the clear concrete cover. On the basis of these specifications and the applied rebar diameters, RF-CONCRETE Members determines the lever arm of the internal forces of the rebars and concrete section.

"Top" and "bottom" are defined by the position of the local member axes in RFEM. The specifications concerning the cover $c_{\pm y(side)}$ are required to determine the equivalent thickness for the torsional design.

For multilayer reinforcements, the cover u refers to the centroid of all layers defined for the top or bottom side.



Reinforcement Layout

The list contains several options how to lay out the reinforcement in the cross-section:

- -z (top) - +z (bottom) (optimized distribution)
- -z (top) - +z (bottom) (symmetrical distribution)
- -z (top) - +z (bottom) (define ratio $A_{s,-z(top)} / A_s$)
- -z (top) - +z (bottom) (define ratio $A_{s,tension} / A_s$)
- In Corners (symmetrical distribution)
- Uniformly surrounding

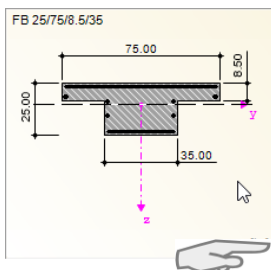
For the layout *-z (top) - +z (bottom) (optimized distribution)*, RF-CONCRETE Members performs an optimization concerning biaxial bending.

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 of those ratios can be entered in the text box below the list. In this way, existing constructions can be modeled for the design.

For T-beams and I-sections, you can *Distribute reinforcement evenly over complete slab width*. With this option, the program sources part of the rebars out (see figure to the left).

Modified layout settings are shown dynamically in the graphic of the window.

If the moment distribution is $M_y = 0$ and $M_z > 0$ for a *top - bottom* layout, the output will give increased values of reinforcement: The design moment (M_y) does not act in the defined direction of the target layout. In this case, choose the *In Corners* layout for a correct design.



3.6.4 Min Reinforcement

This tab controls the specifications concerning the minimum and structural reinforcement.

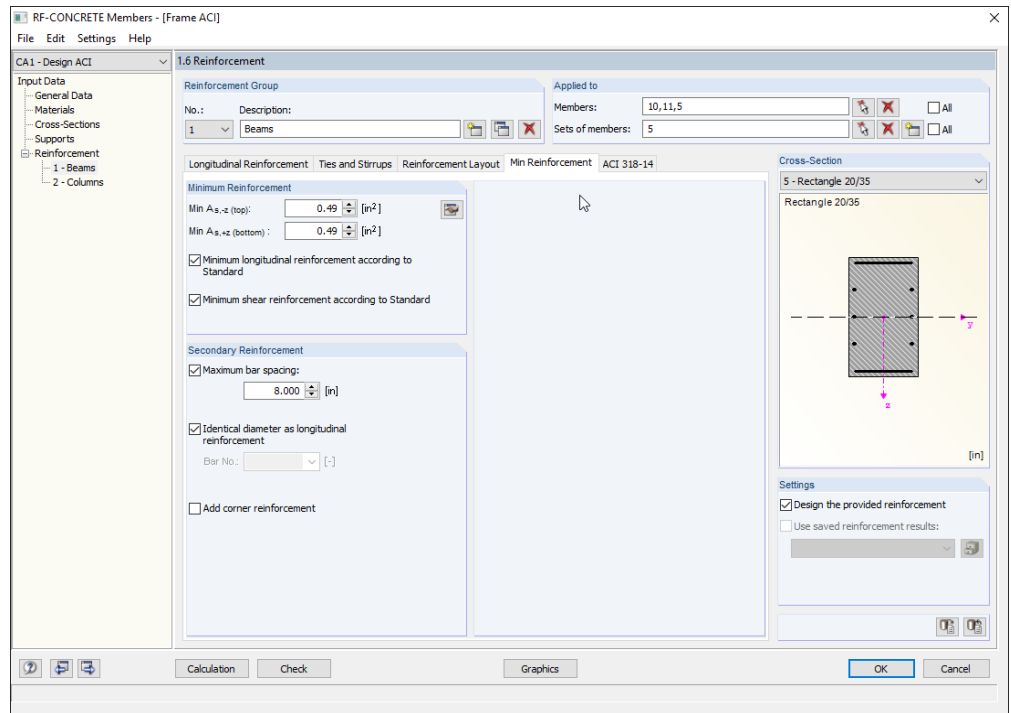


Figure 3.19: Window 1.6 Reinforcement, tab Min Reinforcement

Minimum Reinforcement

You can define a basic minimum reinforcement in the two text boxes. Enter the steel areas for *Min A_{s,-z (top)}* and *Min A_{s,+z (bottom)}*. Via the [Edit] button, you can define the cross-sectional areas in a separate dialog box from the number of rebars and the rebar diameters.

For the calculation of the required reinforcement, you can take into account – independently of each other – the *Minimum longitudinal reinforcement* and/or *Minimum shear reinforcement* according to the Code.

Secondary Reinforcement

This dialog section is accessible when a reinforcement proposal is to be created.

The *Maximum bar spacing* of the secondary reinforcement – the rebars that are not structurally required in the cross-section – can be defined in the text box. The reinforcement proposal then tries to realize a uniform distribution of rebars (for example for webs of T-beams or for slender rectangular 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 *Bar No.* for the secondary reinforcement.

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

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



Bars No. [Inch-Pound]	Diameter [in]
3	0.375
4	0.500
5	0.625
6	0.750
7	0.875
8	1.000
9	1.128
10	1.270
11	1.410
14	1.693
18	2.257

3.6.5 ACI 318-14

The fifth table tab manages the settings of the Code that has been selected in Window 1.1 *General Data* (see Figure 3.2, page 18). It contains the code-specific settings concerning the reinforcement. In the following, the specifications of ACI 318-14 are described.

In the bottom section of this tab, the button [Set Default Values] is displayed. Use this button to reset the initial values of the Code.

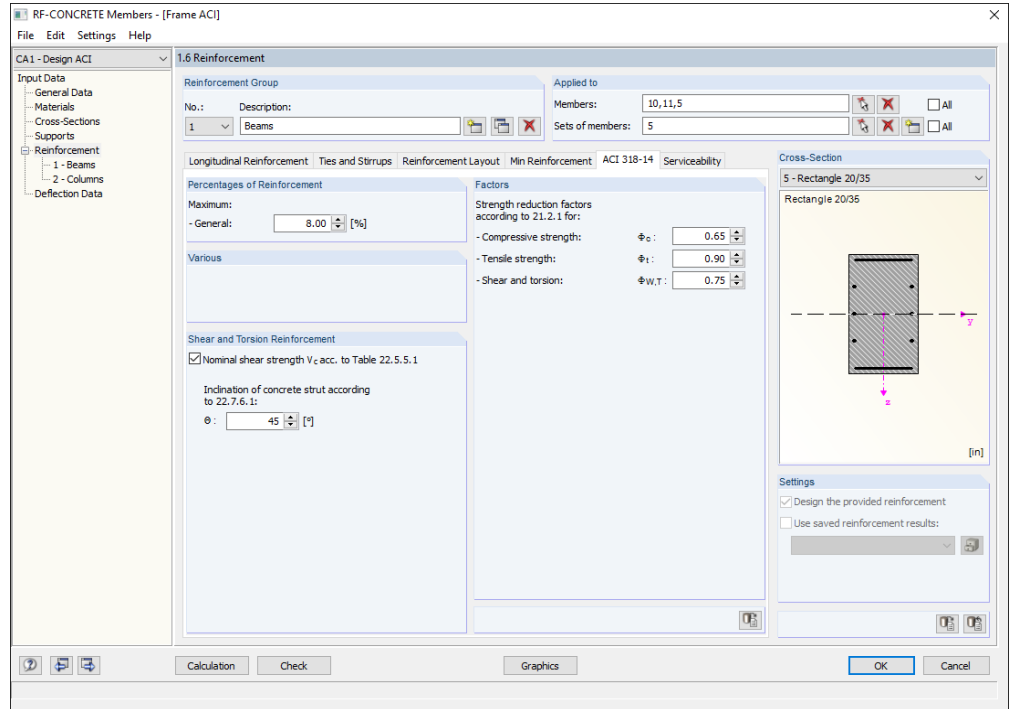


Figure 3.20: Window 1.6 Reinforcement, tab ACI 318-14

Percentages of Reinforcement

The entry in this text box defines the general maximum reinforcement ratio for beams. ACI 318-14, 10.6.1.1 recommends the value of $A_{s,max} = 0.08 A_g$ for compression members, where A_g is the gross area of the section.

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

Shear and Torsion Reinforcement

With the option *Nominal shear strength V_c acc. to Table 22.5.5.1*, the program computes the shear strength provided by the concrete using the following equation:

$$V_c = \left(1.9\lambda\sqrt{f_c'} + 2500\rho_w \frac{V_u d}{M_u} \right) b_w d \leq 3.5\lambda\sqrt{f_c'} b_w d \quad \text{ACI 318-14, Table 22.5.5.1}$$

where $V_u \cdot d / M_u \leq 1$

Otherwise the simplified equation $V_c = 2\lambda\sqrt{f_c'} b_w d$ is used according to ACI 318-14, 22.5.5.1. For more details, see page 10.

The text box defines the inclination of concrete struts according to ACI 318-14, 22.7.6.1. If user-defined angles are beyond the allowed limits mentioned in the Code, an error message appears.

Factors

The three text boxes define the *Strength reduction factors* of the compressive strength, ϕ_c , the tensile strength, ϕ_t , and for shear and torsion, $\phi_{w,t}$. The values according to ACI 318-14, 21.2.1 are preset.

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

3.6.6 Serviceability

This tab includes the parameters for crack control as well as the specifications concerning the deflection analysis. It is active only if loads for the SLS design have been selected in Window 1.1 *General Data*.

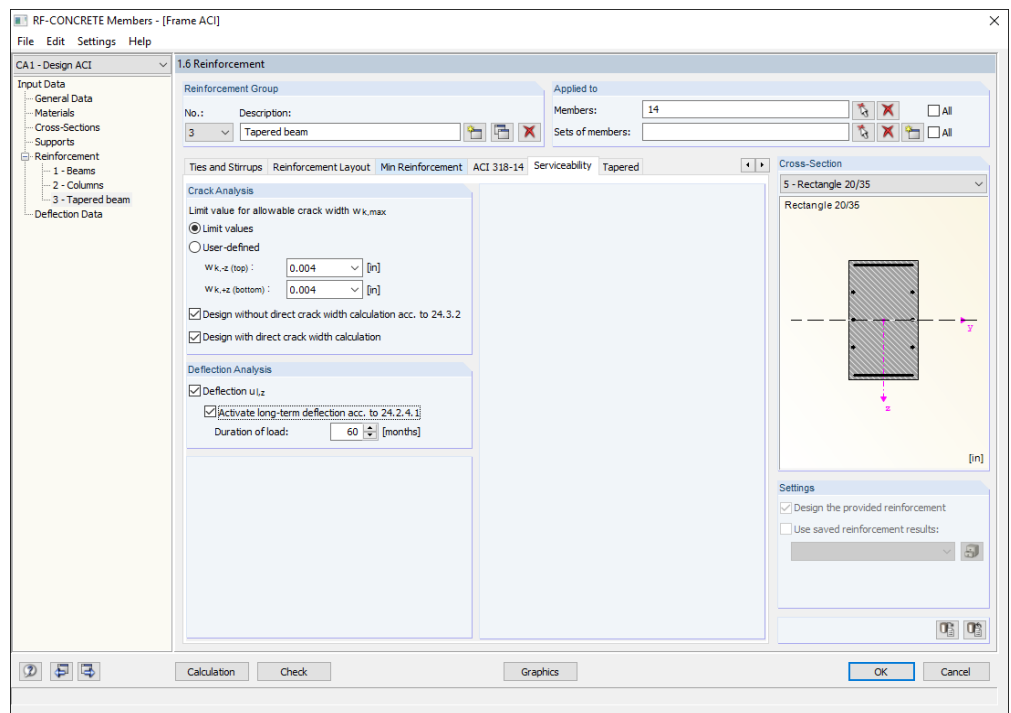


Figure 3.21: Window 1.6 Reinforcement, tab Serviceability

Crack Analysis

The maximum allowable *crack width* $w_{k,max}$ can be selected from the list. Depending on the exposure conditions, the allowable crack widths for reinforced concrete members are presented by ACI Committee 224 in a report on cracking.

0.012 [in]

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

Figure 3.22: Allowable crack widths

The crack widths are regulated in the Code by establishing the maximum center-to-center spacing, s , for the reinforcement which is closest to a tension face. This spacing is specified in ACI 318-14, 24.3.2 (see page 12).

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 12).

Deflection Analysis

This dialog section can be used to consider the deflection during the Serviceability Design. The first check has to be set if the immediate deflection is to be designed, while the second check considers the long-term deflections according to ACI 318-14, 24.2.4.1 using a time dependent factor, ξ , for sustained loads.

3.6.7 Tapered

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

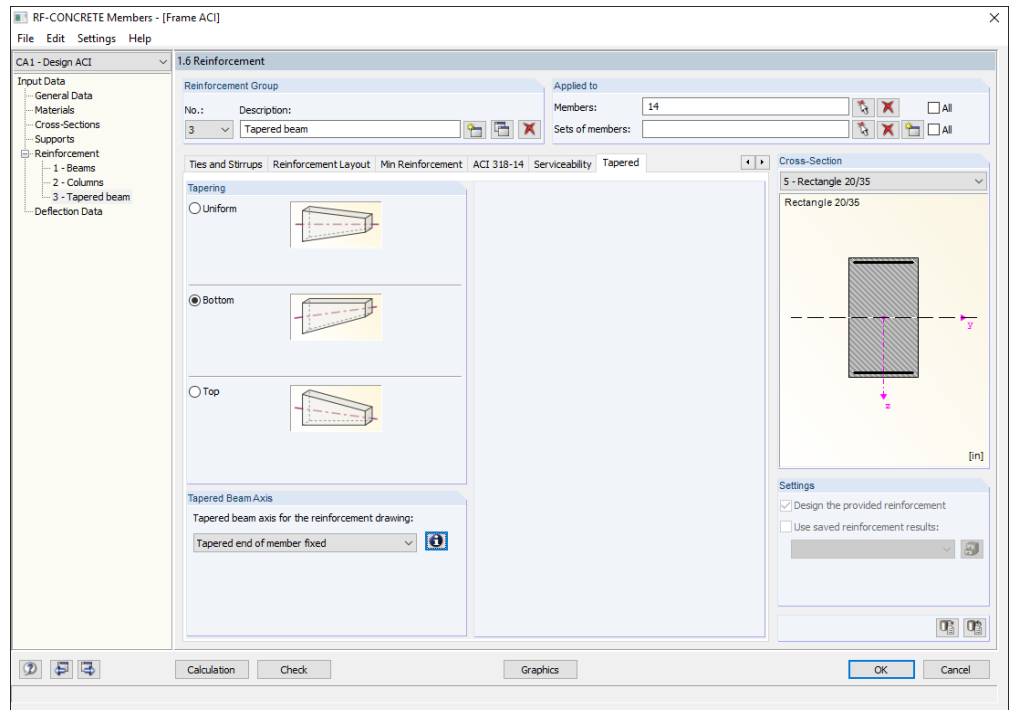


Figure 3.23: Window 1.6 Reinforcement, tab Tapered

RF-CONCRETE Members 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 an error message before the calculation is started.

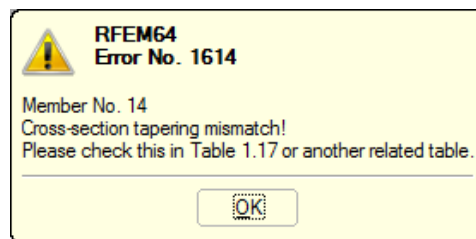
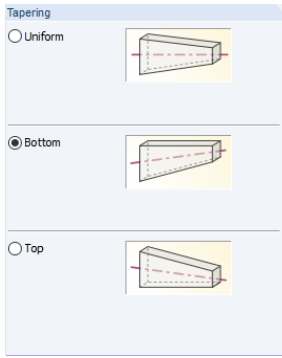


Figure 3.24: Error message for incompatible tapered cross-sections



Tapered sets of members cannot be designed. They have to be designed as single members.



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.

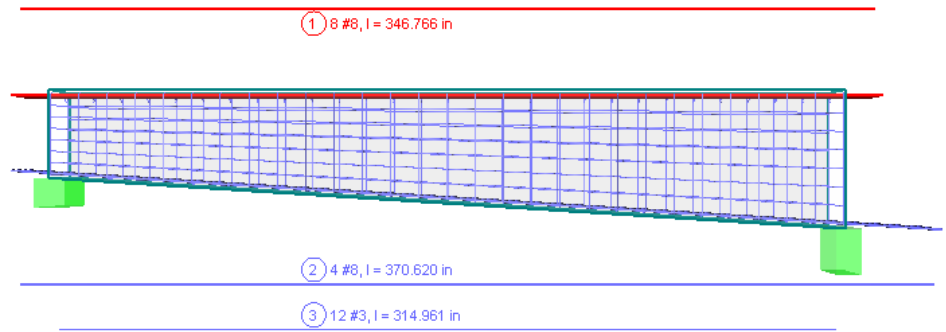


Figure 3.25: Taper with inclined bottom side

- Tapered end of member fixed
- Tapered end of member displaced
- Tapered end of member fixed
- Member axis like in RFEM



Tapered Beam Axis

For the display of the reinforcement in the 3D rendering of RFEM it is necessary to determine the slope of the tapered beam axis. In general, tapers are defined centrally in the structural model. In the RF-CONCRETE Members model, however, the tapers are usually calculated with an even top or bottom side, i.e. shifted out of the center line. For a correct view of the reinforcement connection to the adjacent members, the correct position of the tapered end has to be specified – unless this has already been done in RFEM by a member eccentricity.

Additional explanations are available via the [Info] button.

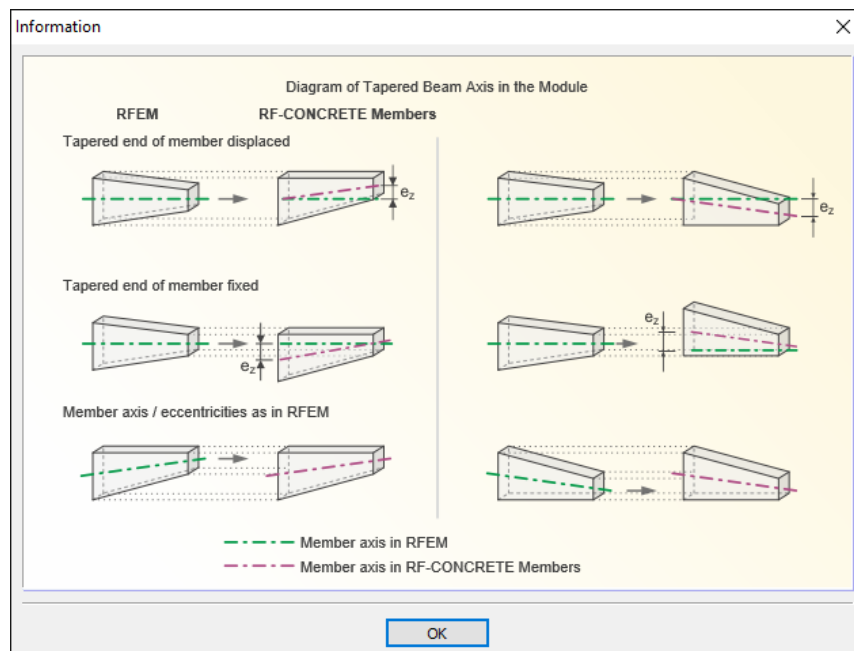
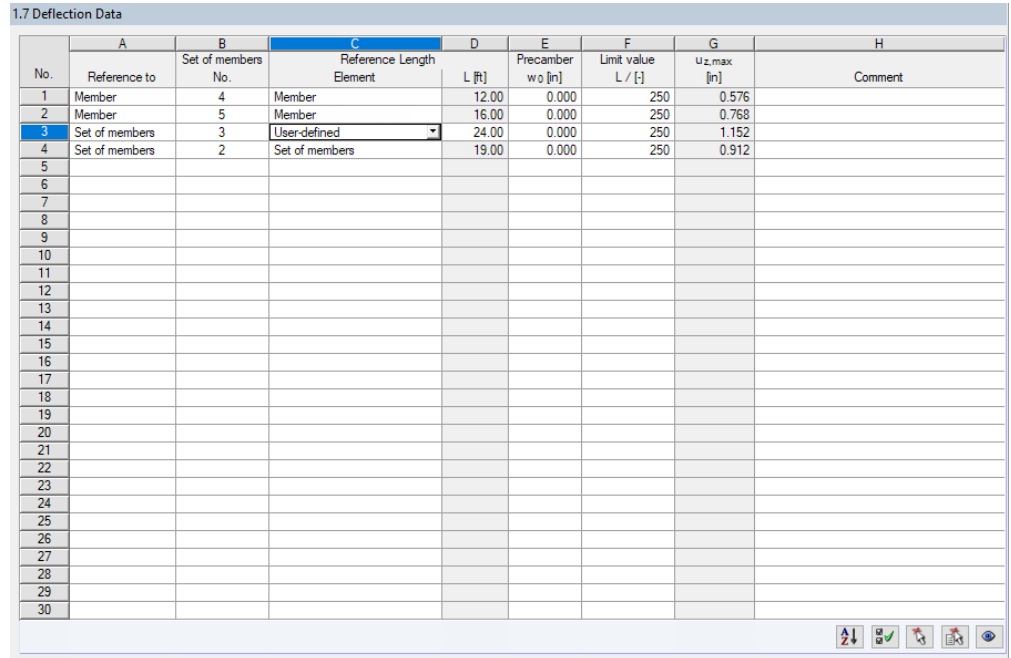


Figure 3.26: Dialog box *Information* for further explanation on Tapered Beam Axis

3.7 Deflection Data

This window is available only when in Window 1.6 Reinforcement, tab Serviceability the deflection analysis has been activated.



No.	A Reference to	B Set of members No.	C Reference Length Element	D L [ft]	E Preamber wo [in]	F Limit value L / [-]	G Uz,max [in]	H Comment
1	Member	4	Member	12.00	0.000	250	0.576	
2	Member	5	Member	16.00	0.000	250	0.768	
3	Set of members	3	User-defined	24.00	0.000	250	1.152	
4	Set of members	2	Set of members	19.00	0.000	250	0.912	
5								
6								
7								
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								

Figure 3.27: Window 1.7 Deflection Data

The design criterion for the deflection, $u_{I,z}$, considers the deformation in the direction of the local member axis z.

Reference to

Column A regulates if a member or a set of members is considered for the design.

Member / Set of Members No.

In this column, the numbers of the members or sets of members have to be entered. Via the [...] button, you can select the objects graphically in the RFEM work window. The length of the member or set of members is then preset as reference length in column D.

Reference Length

Via list in column C, you can adjust the reference lengths: The default setting *Member* refers to the distance between the nodes, while with the option *Distance between supports* considers the reduced span width which is based on the support dimensions of Window 1.5 Supports (see Chapter 3.5, page 26).

The option *User-defined* makes it possible to define the reference length manually in column D.

If sets of members with different segment lengths are used, the *variable* distances between the supports are determined automatically. They can be shown via the tool tip.

No.	A Reference to	B Set of members No.	C Reference Length Element	D L [ft]	E Preamber wo [in]	F Limit value L / [-]	G Uz,max [in]
1	Set of members	6	Distance between supports	var.	0.00	250	var.
2							
3					Nodes 15 - 16: L ₁ = 7.00		
4					Nodes 16 - 17: L ₂ = 6.00		
5					Nodes 17 - 18: L ₃ = 6.00		

Figure 3.28: Reference lengths on a set of members with different distances between supports

Precamber

In column E, a *Precamber* w_0 can be entered, if necessary.

The shape of the precamber can be determined as follows:

$$w_{c,x} = w_0 \cdot \sin(\pi \cdot x / L)$$

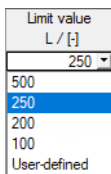
where

- $w_{c,x}$ precamber at location x
- w_0 precamber specified in column E
- x location x
- L length of member or set of members

Limit value / $u_{z,max}$

The relative limit of the deflection has to be specified in column F. Default is the limit of L/250 as specified in ACI 318-14, Table 24.2.2 for constructions supporting non-structural elements not likely to be damaged by large deflections. Different limits can be selected from the list or specified individually.

The maximum permissible deflection is stated in column G. This value is determined from the limit value (column F) and the reference length (column D).



4. Calculation

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 window of the module. If no input error is detected, the following message will be displayed.



Figure 4.1: Result of plausibility check

4.2 Start Calculation

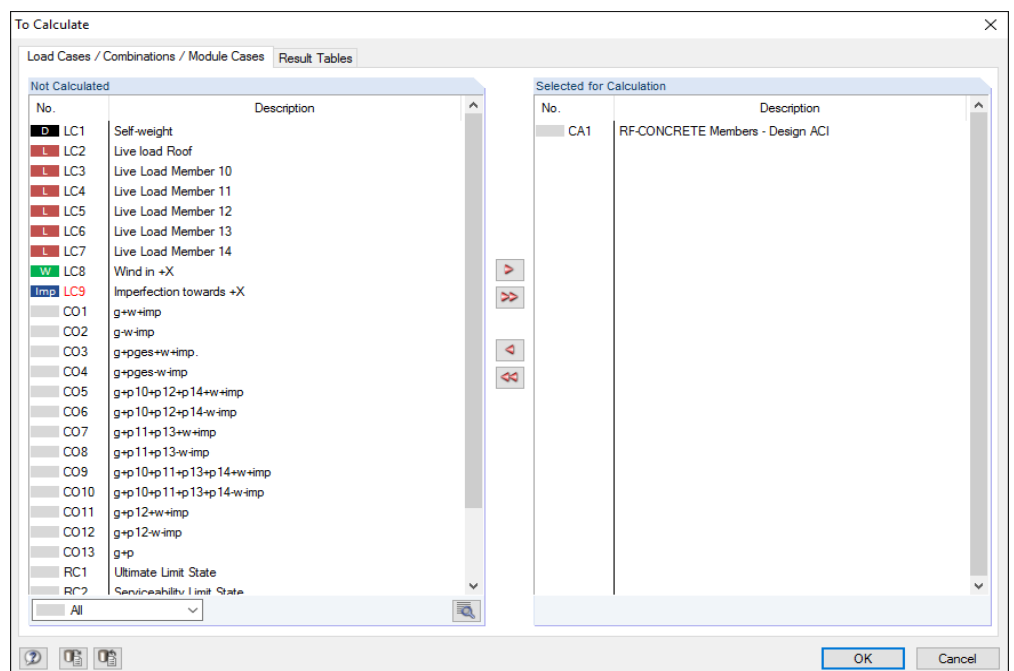
Calculation

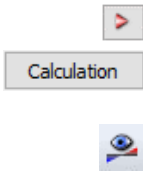
To start the calculation, click the [Calculation] button.

RF-CONCRETE Members searches for the results of the load cases, load and result combinations that are to be designed. If they are not yet available, the RFEM calculation starts to determine the internal forces relevant for the design. In this process, the calculation parameters set in RFEM are applied.

You can also start the design of RF-CONCRETE Members in the RFEM user interface. All add-on modules are listed in the dialog box *To Calculate*, like load cases or combinations. To open this dialog box in RFEM,

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


 Figure 4.2: Dialog box *To Calculate*



To transfer the selected RF-CONCRETE Members cases to the list on the right, use the button [▶]. Then start the calculation with [OK].

You can also use the load case list in the RFEM toolbar to calculate a RF-CONCRETE Members case: Select the relevant design case, and then click the button [Show Results].

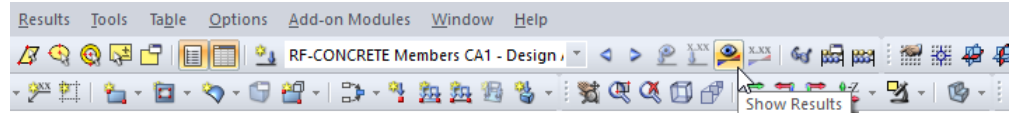


Figure 4.3: Calculating RF-CONCRETE Members design case in RFEM

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

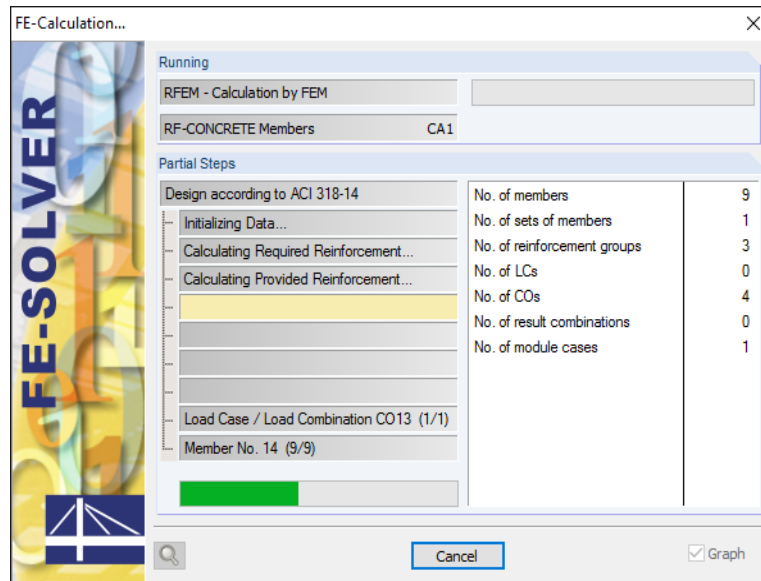


Figure 4.4: Design with RF-CONCRETE Members

5. Results

After the calculation, Window 2.1 *Required Reinforcement by Cross-Section* is displayed.

The reinforcement areas required for the strength limit state design are listed in the result windows 2.1 to 2.4. If the program created a reinforcement proposal, the provided reinforcement including steel schedule is displayed in the result windows 3.1 to 3.4. The results for the serviceability limit state design are represented in the result windows 4.1 to 4.4.



All windows 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 window.

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

In the following, the different result windows are described one by one. Evaluating and checking results is described in Chapter 6 *Results Evaluation*, page 64.

5.1 Required Reinforcement

5.1.1 Required Reinforcement by Cross-section

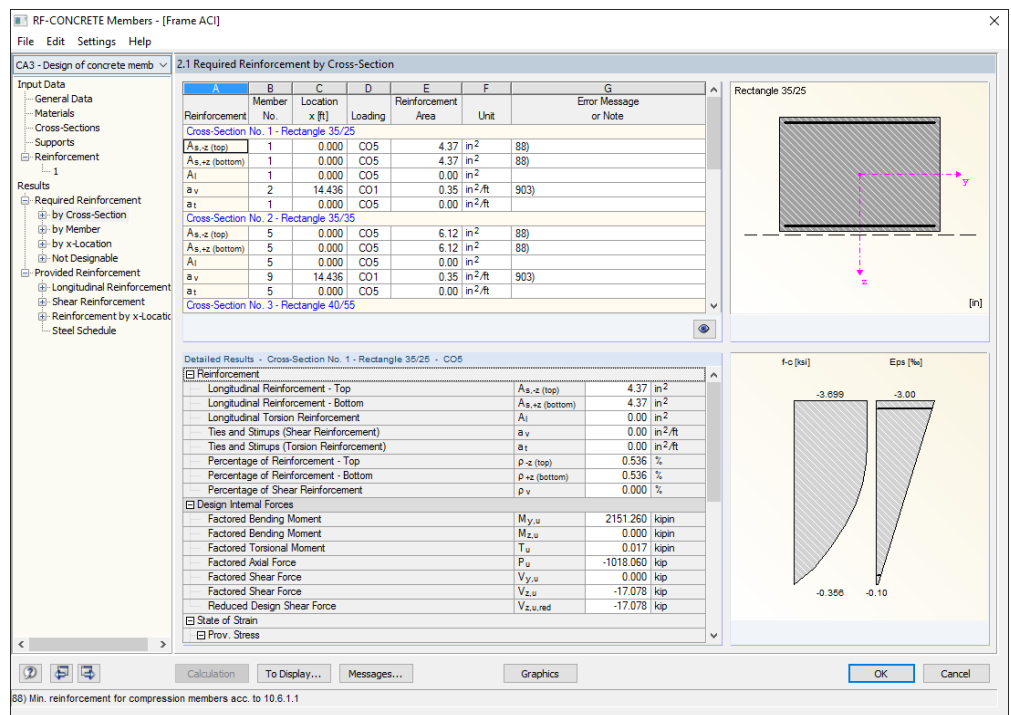


Figure 5.1: Window 2.1 *Required Reinforcement by Cross-Section*

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 shear reinforcement are listed according to cross-sections. The upper table of the window lists the reinforcement types and design details that have been activated in the dialog box *Results to Display* (see Figure 5.2).

In the lower part of the window, the *Detailed Results* of the item selected in the table row above are shown. Due to those design details, you can evaluate the results specifically. The output of the detailed results is updated automatically when a different row is selected above.

Reinforcement

The following types of longitudinal and stirrup reinforcement are preset:

Reinforcement	Explanation
$A_{s,z (top)}$	Reinforcement area of required top longitudinal reinforcement due to bending with or without axial force or axial force only
$A_{s,z (bottom)}$	Reinforcement area of required bottom longitudinal reinforcement due to bending with or without axial force or axial force only
A_l	Reinforcement area of required longitudinal torsion reinforcement, if applicable
a_v	Area of required shear reinforcement for absorption of shear force, referring to standard length of 1 ft
a_t	Area of required stirrup reinforcement for absorption of torsional moment, referring to standard length of 1 ft

Table 5.1: Longitudinal and stirrup reinforcements



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 shortcut menu of the member.

Click the button [To Display] to open a dialog box where you can specify the types of reinforcement that are to be displayed. These settings also define the types of results appearing in the printout report.

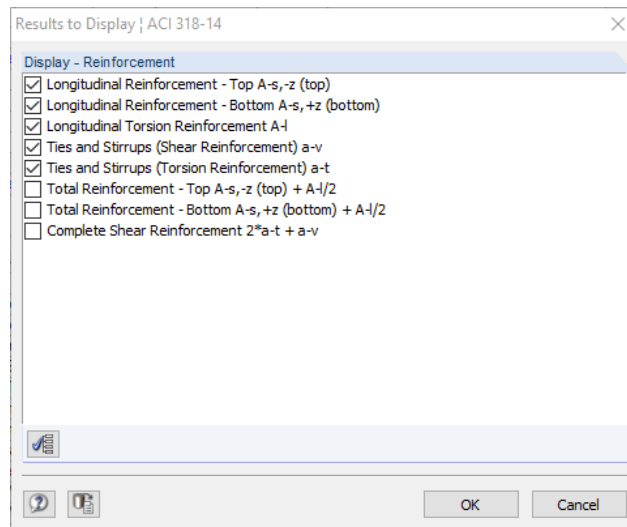


Figure 5.2: Dialog box *Results to Display*

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

This column specifies the x-locations of the respective members for which the maximum reinforcement have been determined. For the table output, the program uses these member locations x:

- Start and end nodes
- Partition points according to possibly defined member division (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Extreme values of internal forces

Loading

In this column, the numbers of the load cases, load or result combinations are displayed that are governing for the respective designs.

Reinforcement Area

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

The *Unit* of the reinforcements shown 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 78 opens.

Error Message or Note

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

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...

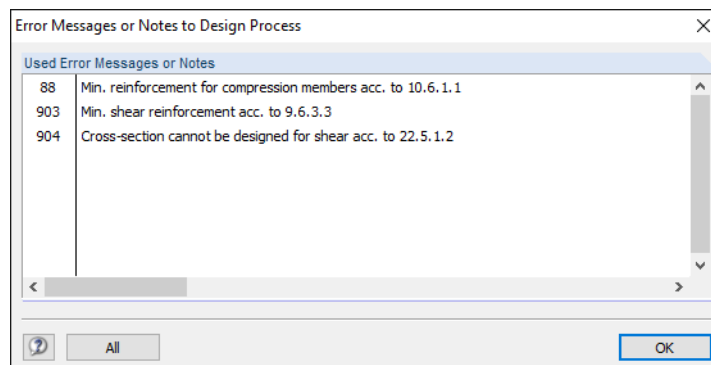


Figure 5.3: Dialog box *Error Messages or Notes to Design Process*

5.1.2 Required Reinforcement by Set of Members

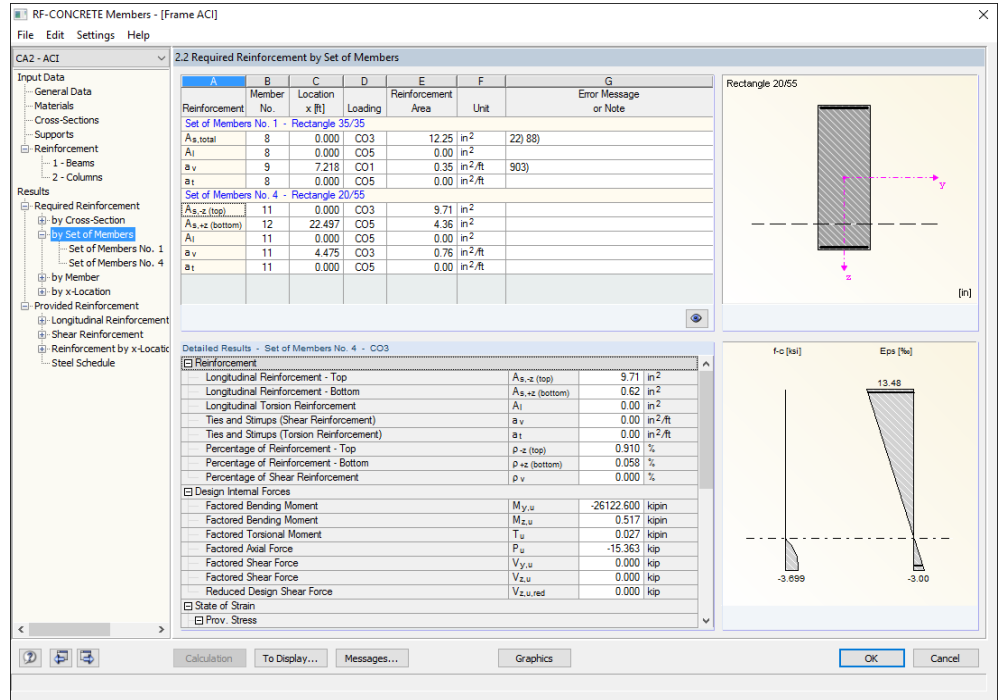


Figure 5.4: Window 2.2 Required Reinforcement by Set of Members

This window 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

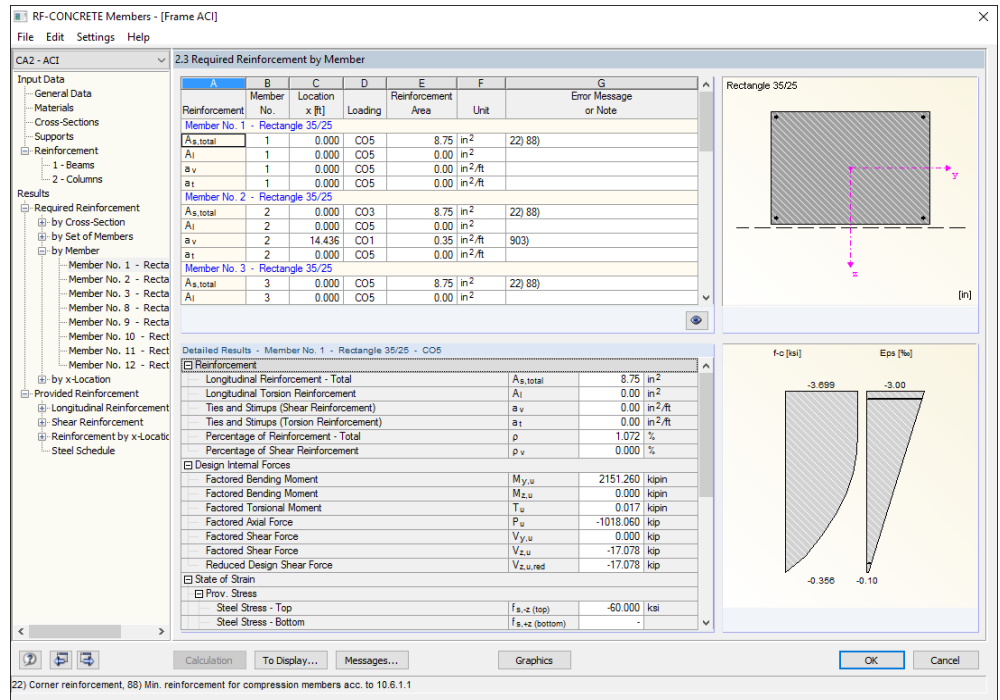


Figure 5.5: Window 2.3 Required Reinforcement by Member

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

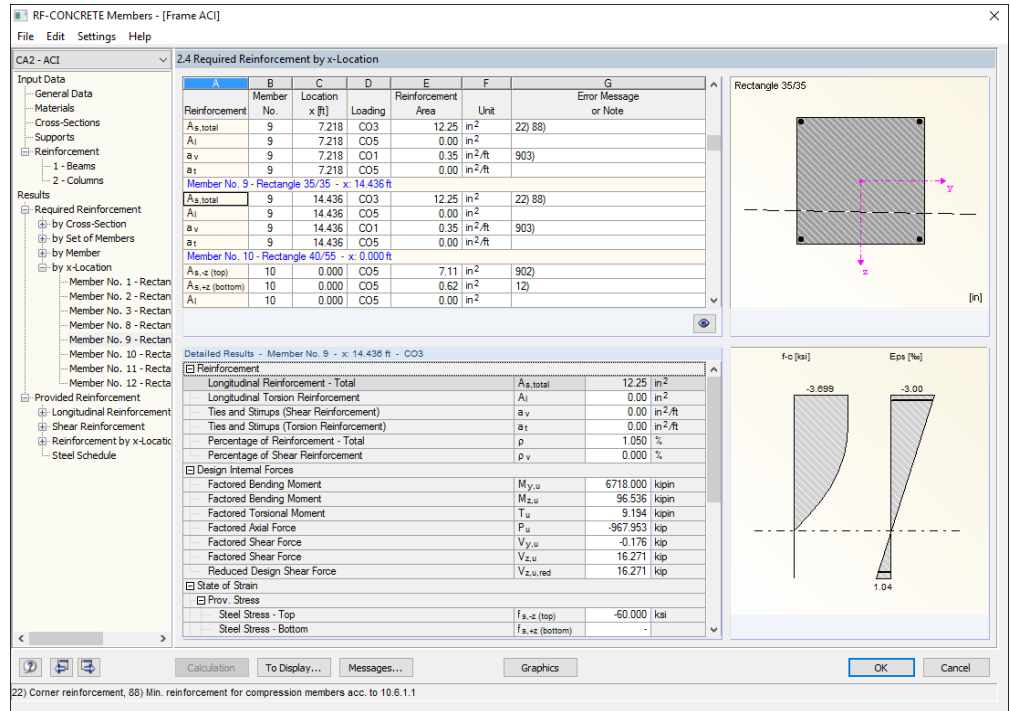


Figure 5.6: Window 2.4 Required Reinforcement by x-Location

This window 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 (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Extreme values of internal forces

Locations of discontinuity are indicated separately.

The window offers you the possibility to specifically access information on the design results. This way, you can check, for example, the required sheareinforcement with the related details of a particular member location (designed location).

The different columns are described in Chapter 5.1.1.

5.1.5 Required Reinforcement Not Designable

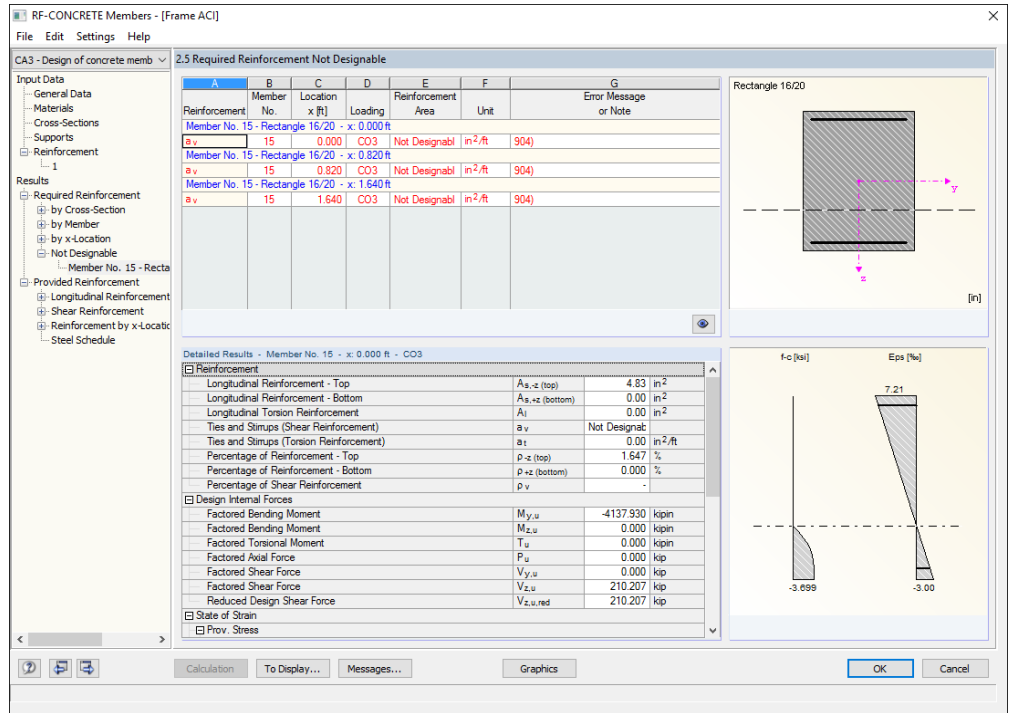


Figure 5.7: Window 2.5 Required Reinforcement Not Designable

This window is displayed when the program has detected failed designs or some other problem during the calculation. The error messages are sorted by member and x-location.

The number of the *Error Message* indicated in column G is described by comments in the footer.

Click the [Messages] button to display all issues that have occurred in the design process.

Messages...

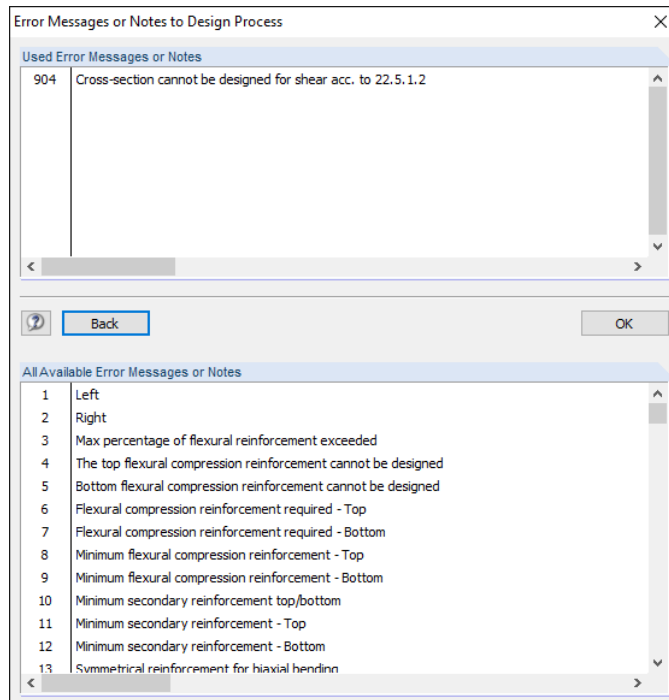


Figure 5.8: Dialog box Error Messages or Notes to Design Process

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

All

5.2 Provided Reinforcement

The result Windows 3.1 to 3.4 are displayed if the *Design the Provided Reinforcement* option has been activated in Window 1.6 *Reinforcement* (see page 29), and if no design problems have been detected (see Chapter 5.1.5, page 49). The serviceability limit state design as well as the non-linear calculation also require the determination of a provided reinforcement.

Based on the specifications of Window 1.6, RF-CONCRETE Members manages the reinforcement proposal of the longitudinal and shear reinforcement. 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 *Provided Reinforcement* windows 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

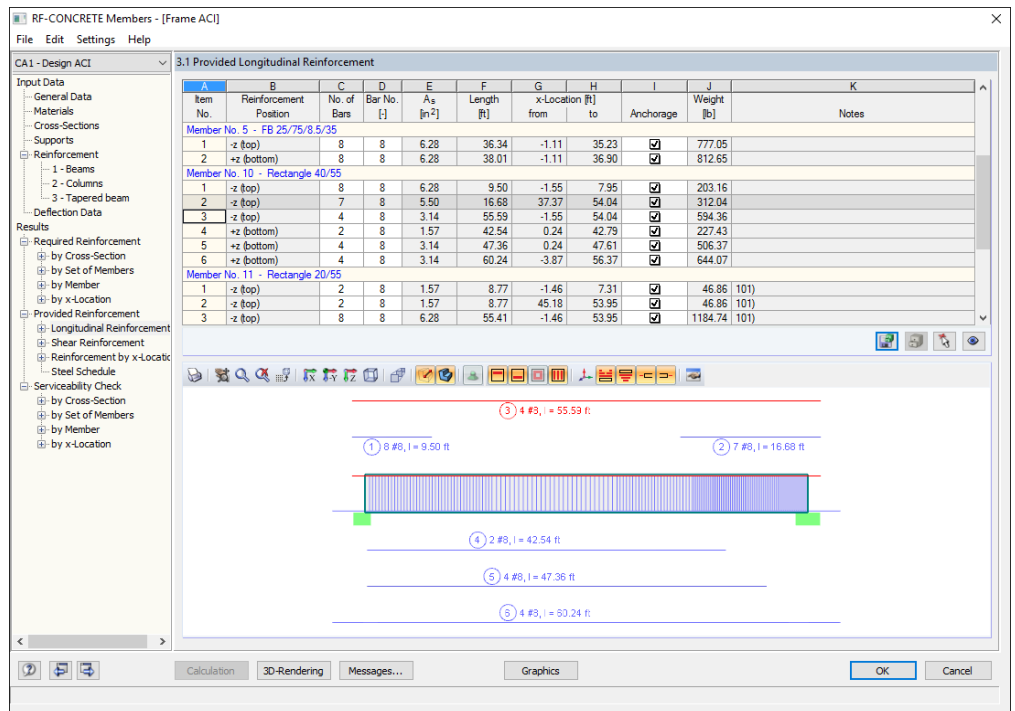


Figure 5.9: Window 3.1 Provided Longitudinal Reinforcement

The results for the provided reinforcement are sorted according to *Item* numbers (reinforcement groups) for every designed member and set of members.

The graphic below the table shows the reinforcement including item members. The current item (the row selected in the upper table) is highlighted in red. Modifications to the parameters entered in the table are updated and displayed immediately in the graphic.

The reinforcement proposal also takes into account structural regulations. For example, in accordance with ACI 318-14, 9.7.3.8, 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 reinforcement of single members and one-fourth of continuous members must be continued uninterrupted along the same face of the beam for a distance of at least 6 in.

Item No.

The results are listed by items which have the same properties (diameter, length).

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

Reinforcement Position

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

- Top
- Bottom
- Corners
- All round
- Secondary

RF-CONCRETE Members considers the specifications set in *Window 1.6 Reinforcement*, tab *Reinforcement Layout* (see Chapter 3.6.3, page 34) for the arrangement of the reinforcement.

No. of Bars

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

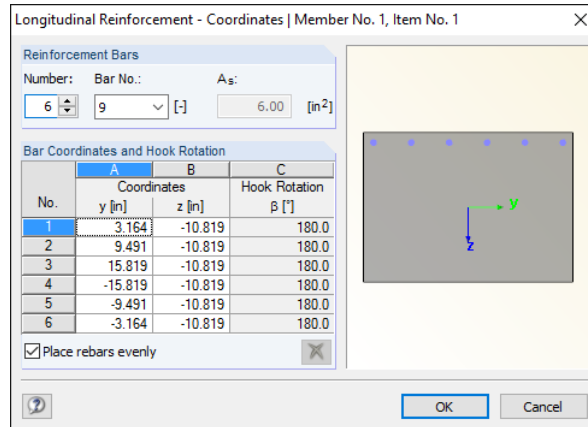
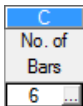


Figure 5.10: Dialog box *Longitudinal Reinforcement - Coordinates*

The *Number of Reinforcement Bars* can be set manually by using the spin buttons or by directly entering a number. In the section below, you can adjust the position of the rebars in the individual input rows. To delete a row selected in the lower section, click the [Delete] button.

The position of a rebar is defined by its *Bar Coordinates*: The coordinates *y* and *z* determine the global distances from the cross-section's centroid. The angle, β , describes the inclination against the longitudinal member axis for the anchorage types "Hook" and "Bend." For example, a *Hook Rotation* about $\beta = 90^\circ$ 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.

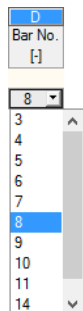
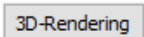
When modifying the hook rotation, it is recommended to check the data subsequently in the rendering mode by clicking the [3D-Rendering] button.

Bar No.

The used rebar diameters affect the calculation of the inner lever arm of the forces as well as the number of rebars per item. Use the list to change the rebar designation number for the current item number.

As

The total reinforcement area of each item is listed in column E. It can be modified by changing the number of rebars or the rebar designation number.



Length

For each item, the column displays 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.

x-Location from/to

These values represent the mathematical start and end positions of the rebars. They refer to the start node of each member as defined in 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 values of both columns cannot be modified. To change data, use the [Edit] button above the graphic to open the *Edit Longitudinal Reinforcement* dialog box (see Figure 5.12, page 53).

Anchorage

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

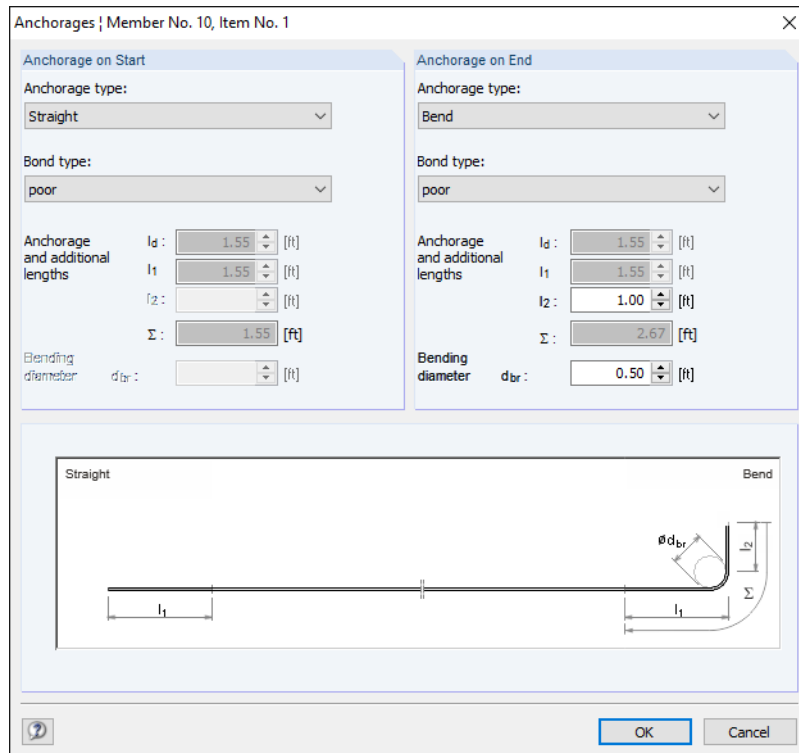
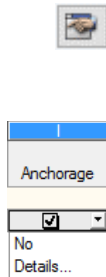


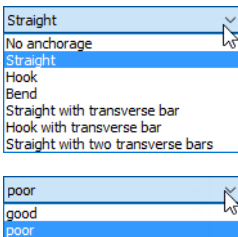
Figure 5.11: Dialog box *Anchorage*

This dialog box manages 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*. The anchorage type is described 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 Ψ_t according to ACI 318-14, 25.4.2.4.

For control reasons the program displays the design value of the anchorage length l_d . The length, l_1 , is determined according to ACI 318-14, Equation (25.4.2.3a). It cannot be modified.

Length l_2 is the length of anchorage for hooks and bends, in accordance with ACI 318-14, 25.4.3. For 90° hooks, for example, it should be at least 12 d_b .



The required *Bending Diameter* d_{br} is specified according to ACI 318-14, 25.3. It can be adjusted, if necessary.

The total anchorage length Σ at each end of the member results from the respective portions.

Weight

Column J of Window 3.1 indicates for each item the mass of all rebars.

Notes

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

To display all messages of the currently selected item, use the [Messages] button. A dialog box with relevant information appears (see Figure 5.3, page 46).

Editing the reinforcement proposal

The graphic in the lower section of Window 3.1 represents the reinforcement including item members. The currently selected reinforcement position (the row in the upper table where the pointer is placed) is highlighted in red. To open the edit dialog box for the selected item, click the [Edit Reinforcement] button above the graphic.

Messages...

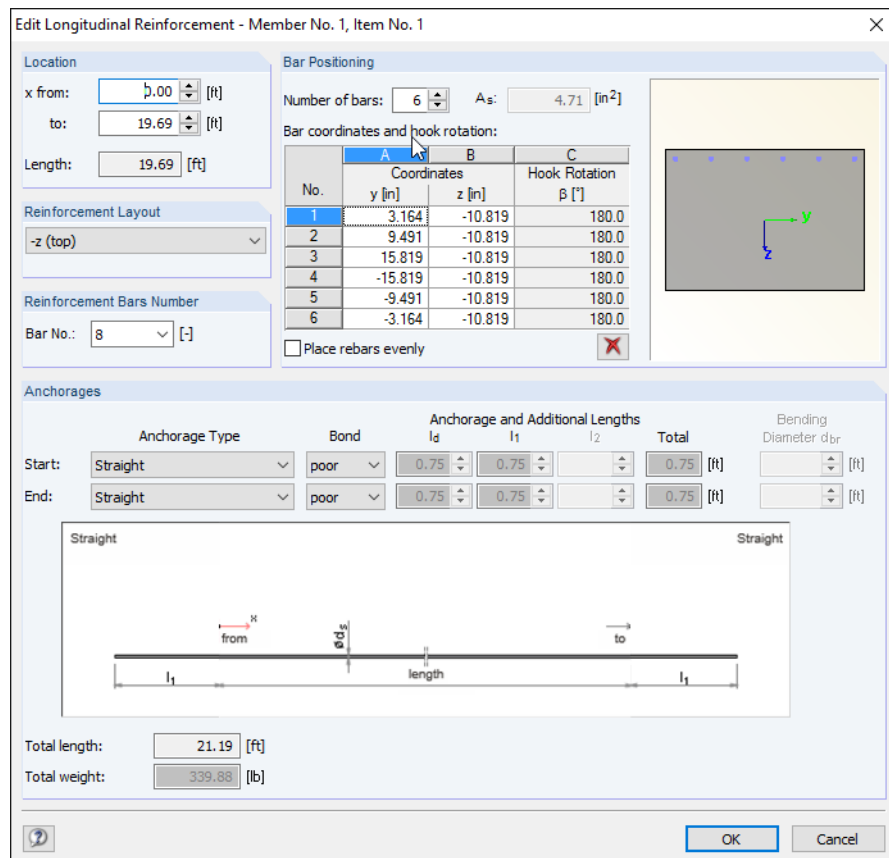


Figure 5.12: Dialog box *Edit Longitudinal Reinforcement*

This dialog box summarizes the reinforcement parameters already described above. Use the dialog box to control or adjust, if necessary, the specifications for *Location*, *Bar Positioning*, *Reinforcement Bars Number*, and *Anchorages*.

When you have modified data, the design must be calculated again with the new reinforcement. So, when leaving Window 3.1, the program asks you whether you want to recalculate the safety (see Figure 5.13).

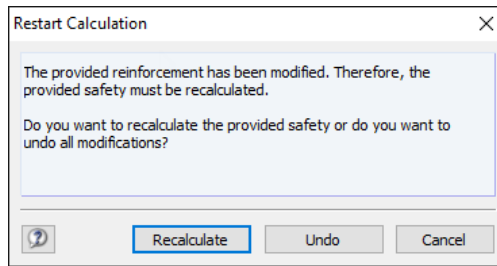


Figure 5.13: Query when leaving Window 3.1

Calculation

An exception exists for the results of nonlinear analyses: They are generally deleted so that a manual [Calculation] is required.

Reinforcement Templates

A modified reinforcement proposal can be stored as a template by using the [Save] button. In the following dialog box, the *Name of the Reinforcement Template* has to be specified.

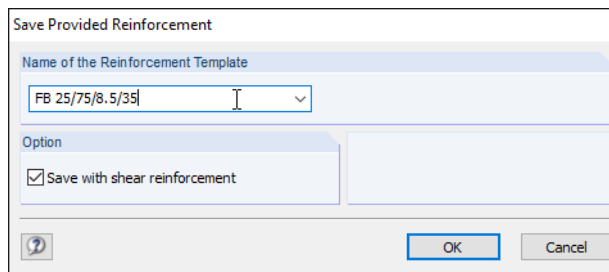


Figure 5.14: Dialog box *Save Provided Reinforcement*

These templates are useful to reset the user-defined reinforcement when the design specifications are modified in Window 1.6. Thus, your changes won't get lost when the program creates a new provided reinforcement.

The possibility to import reinforcement templates is described in Chapter 3.6 on page 29.

5.2.2 Provided Shear Reinforcement

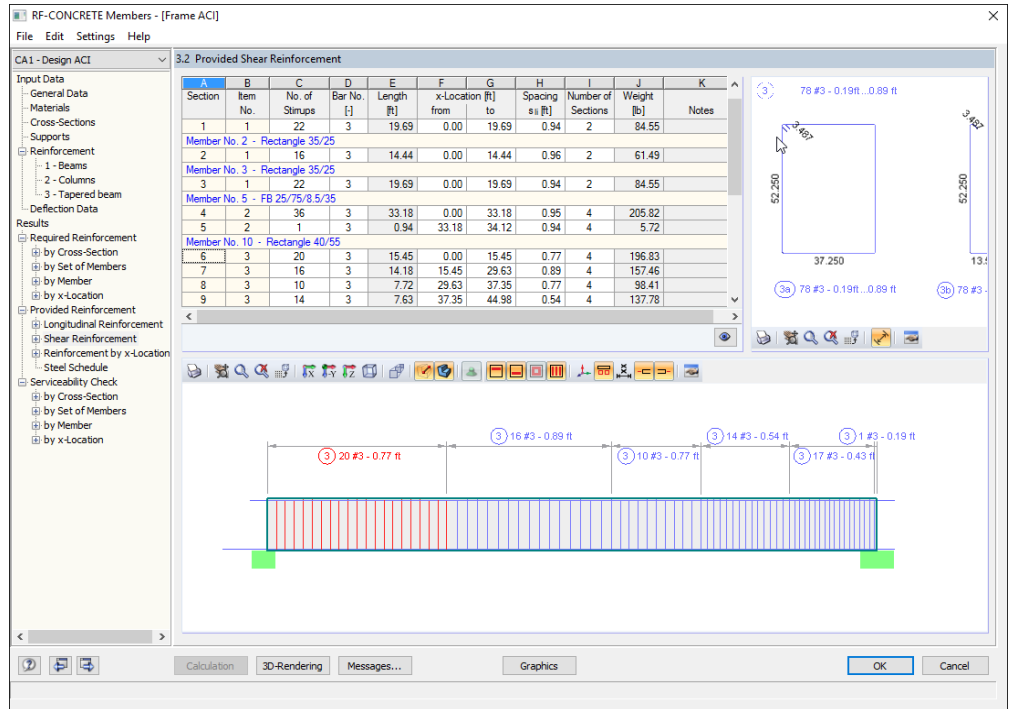


Figure 5.15: Window 3.2 Provided Shear Reinforcement

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

The graphic below the table represents the reinforcement including stirrups. The currently selected item (the row selected in the upper table) is highlighted in red. Modifications to the parameters entered in the upper table are displayed immediately in the graphic.

The reinforcement proposal also takes into account structural regulations. For example, in accordance with ACI 318-14, 9.7.6.2.2, the maximum spacing of stirrups shall neither exceed $d/2$ nor 24 in.

Section

The sections allocate the shear reinforcement to zones with the same diameters and spacings. The zones are defined in columns F and G where they can be adjusted, if necessary (see below).

Item No.

The results are listed by *Items* having each the same properties (diameter, dimensions). The items of all members and sets of members are summarized in Window 3.4 Steel Schedule.

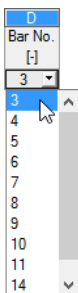
No. of Stirrups

When the program determines the stirrup reinforcement, it takes into account the user-defined specifications of Window 1.6 Reinforcement, tab Ties and Stirrups (see Chapter 3.6.2, page 32).

The number of stirrups of an item can be edited: Select the corresponding cell and enter a different value. The stirrup spacing in column H will be adjusted automatically.

Bar No.

The reinforcement proposal is based on the specifications defined in Window 1.6 Reinforcement, tab Ties and Stirrups. Use the list to change the rebar diameter for the current item.



Length

Column E displays the total length of the stirrup zone for each item. It is determined by the start and end locations x , but cannot be edited in the table. To change data, use the [Edit Reinforcement] button and open the *Edit Shear Reinforcement* dialog box (see Figure 5.16, page 56).

x-Location from/to

These values represent the start and end positions of the stirrup zones. They refer to the start node of the member given in RFEM ($x = 0$). The entries in both columns can be edited so that it is possible to shift the zone limits by modifying the values.

To subdivide a zone, enter a new location for the start or end which lies between both values. RF-CONCRETE Members will automatically create a new stirrup zone (section).

Spacing s_{ii}

The proposed stirrup spacing considers the specifications defined in Window 1.6 *Reinforcement*, tab *Ties and Stirrups* (see Chapter 3.6.2, page 32). This value can be edited: Click in the corresponding cell and enter the new spacing. The number of stirrups (column C) will be adjusted automatically. The exact stirrup spacing, however, is calculated by the amount of stirrups defined by an integer.

Number of Sections

The sections of the stirrups are based on the specifications defined in Window 1.6 *Reinforcement*, tab *Ties and Stirrups* (see Chapter 3.6.2, page 32). Use the list to change the number of sections.

Weight

For each item, column J of Window 3.2 shows the mass of all stirrup rebars.

Notes

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

To display all messages of the currently selected item, use the [Messages] button. A dialog box with relevant information appears (see Figure 5.3, page 46).

Editing the reinforcement proposal

The graphic in the lower section of Window 3.2 shows the reinforcement including stirrups. The currently selected reinforcement position (the row in the upper table where the pointer is placed) is highlighted in red. To open the edit dialog box for the selected item, click the [Edit Reinforcement] button above the graphic.

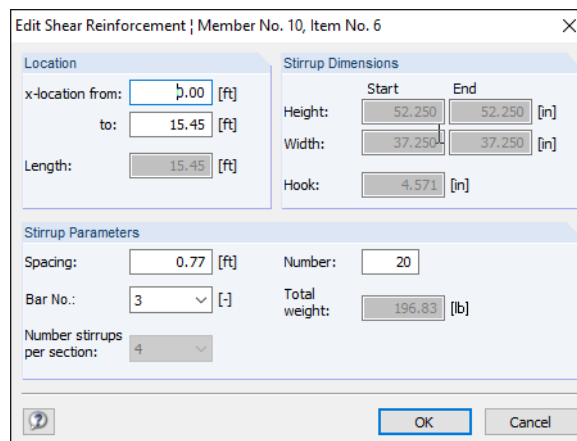


Figure 5.16: Dialog box *Edit Shear Reinforcement*

Calculation

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*.

When you have modified data, the program automatically recalculates the design as described in chapter 5.2.1, page 54, using the values of the new provided stirrup 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 Provided Reinforcement by x-Location

This window contains information on the strength limit state designs that have been fulfilled or failed. In the case of a recalculation after having modified the reinforcement in Window 3.1 or 3.2, the results are updated automatically.

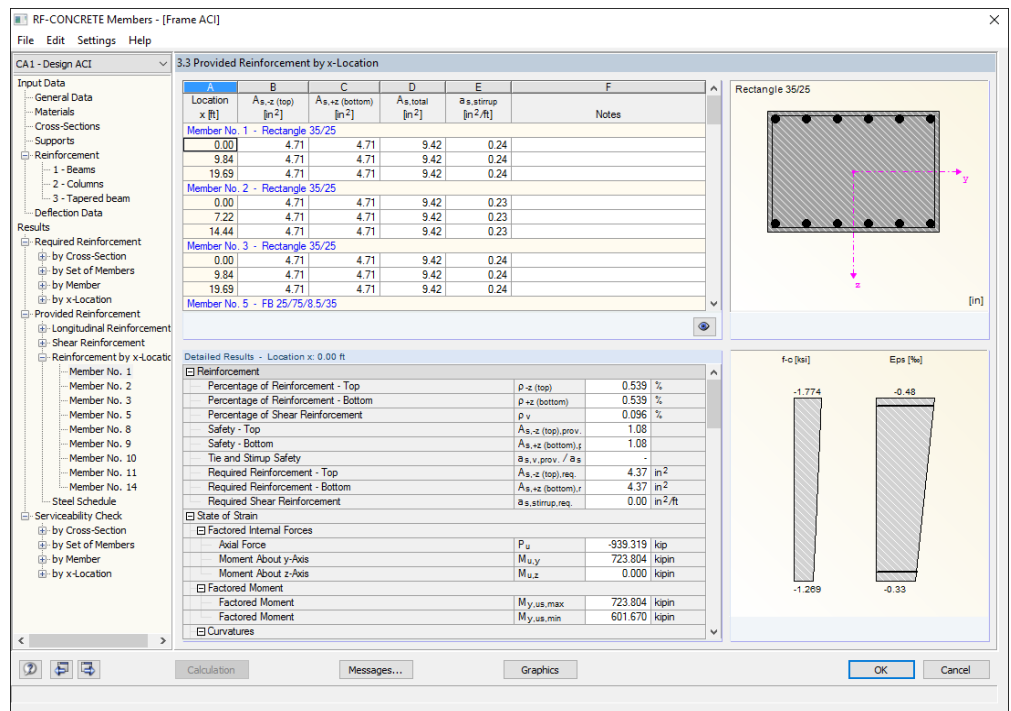


Figure 5.17: Window 3.3 Provided Reinforcement by x-Location

The table in the upper part of this window lists the longitudinal and shear reinforcement areas at the individual member location x. The lower part of the window lists all *Detailed Results* for the table row selected above.

Location x

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

- Start and end nodes
- Partition points according to possibly defined member division (see RFEM Table 1.16)
- Member division according to specification for member results (*Global Calculation Parameters* tab of *Calculation Parameters* dialog box in RFEM)
- Partition points according to possibly defined member division
- Extreme values of internal forces

In case of reinforcements by curtailment, the output shows two identical x-locations with both reinforcement values at the zone limit.

$A_{s,-z}$ (top)

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

$A_{s,+z}$ (bottom)

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

$A_{s,total}$

The provided total longitudinal reinforcement is determined from the values of columns B and C.

$a_{s,stirrup}$

This column shows the areas of the provided shear reinforcement per unit length.



The *Detailed Results* in the lower section allow for a specific evaluation of the designs. The design details refer to the x-location selected in the table above. They are updated automatically when clicking into a different table row.

The detailed 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 of the longitudinal reinforcement is designed by an increased moment taking the global offset into account.

5.2.4 Steel Schedule

The schedule displays the provided rebars in an overview. The table cannot be edited.

Item No.	Reinforcement Type	Bar No.	Surface	No. of Bars	Length [m]	Anchorage Type	End	Bending Diameter [m]	Weight [kg]
Material No. 1 - Reinforcing Steel Grade 60									
1	Longitudinal	3	Ribbed with epoxy resin coating	12	26.25	No anchorage	No anchorage		118.38
2	Longitudinal	8	Ribbed	12	21.19	Straight	Straight		679.76
3	Longitudinal	8	Ribbed	12	15.77	Straight	Straight		505.77
4	Longitudinal	8	Ribbed	12	21.02	Straight	Straight		674.14
5	Longitudinal	8	Ribbed	8	36.34	Straight	Straight		777.95
6	Longitudinal	8	Ribbed	8	39.39	Straight	Straight		842.32
7	Longitudinal	8	Ribbed	11	55.59	Straight	Straight		1634.38
8	Longitudinal	8	Ribbed	10	59.85	Straight	Straight		1599.58
9	Longitudinal	8	Ribbed	15	55.38	Straight	Straight		2220.23
10	Longitudinal	8	Ribbed with epoxy resin coating	8	28.92	Straight	Straight		618.40
11	Longitudinal	8	Ribbed with epoxy resin coating	4	32.60	Straight	Straight		348.51
12	Longitudinal	8	Ribbed	8	35.98	Straight	Straight		769.42
13	Longitudinal	8	Ribbed	8	38.54	Straight	Straight		824.18
14	Tie/Stirrup	3	Ribbed	37	13.55	Hook	Hook	0.13	211.54
15	Tie/Stirrup	3	Ribbed	78	24.39	Hook	Hook	0.13	767.63
16	Tie/Stirrup	3	Ribbed	67	21.05	Hook	Hook	0.13	561.44
17	Tie/Stirrup	3	Ribbed with epoxy resin coating	53	9.18	Hook	Hook	0.13	216.05
18	Tie/Stirrup	3	Ribbed	116	10.22	Hook	Hook	0.13	445.79
Total				479					13814.60

Figure 5.18: Window 3.4 Steel Schedule

Item No.

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

The item numbers are not identical with the numbers of the Windows 3.1 or 3.2.

Reinforcement Type

This column indicates whether the reinforcement is a *Longitudinal* or a *Stirrup* reinforcement.

Bar No.

Column C shows the used rebar designation numbers.

Surface

This column displays the surface type of the reinforcing steel which can be *Ribbed* or *Ribbed with zinc/epoxy resin coating*.

No. of Bars

In column E, you can see the number of identical rebars for each item.

Length

This column shows 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 end of the rebars (*No anchorage, Straight, Hook, Bend, etc.*).

Bending Diameter

For stirrups and hooks, you find the bending diameters d_{br} in column I.

Weight

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

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 result Windows 4.1 to 4.4 are only shown when the design of the *Serviceability Limit State* was activated in Window 1.1 (see Chapter 3.1.2, page 19) and no design problems were detected by the program (see Chapter 5.1.5, page 49 and Chapter 5.2.3, page 57).

The SLS design is based on the reinforcement layout, i.e. the *Reinforcement Provided* as defined in Window 3.1 and 3.2.

5.3.1 Serviceability Check by Cross-Section

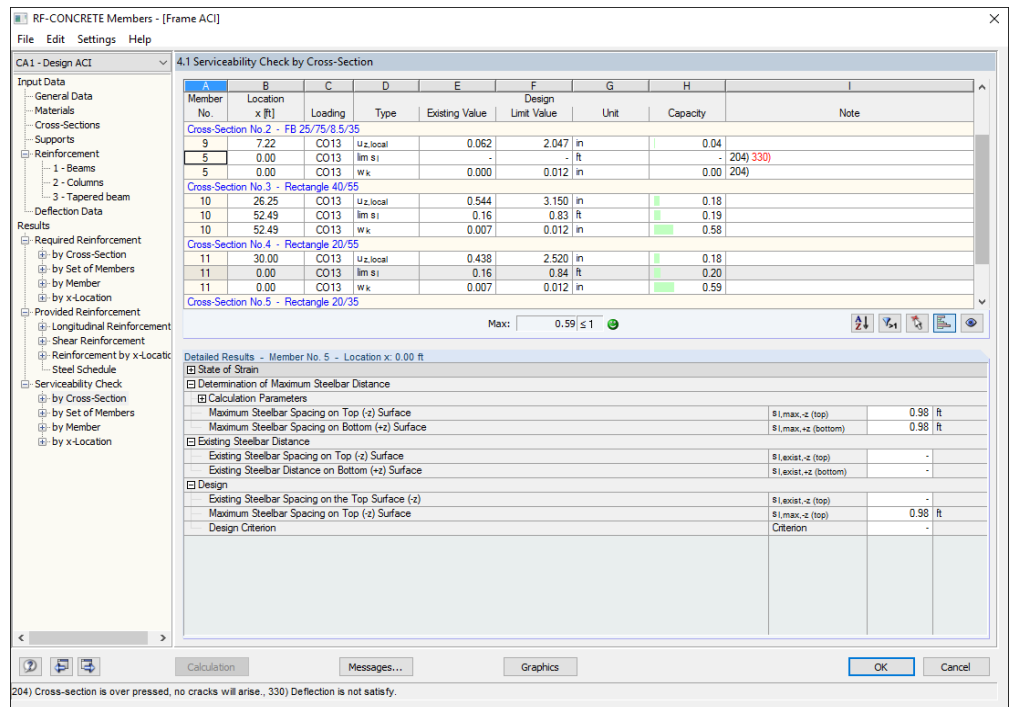


Figure 5.19: Window 4.1 *Serviceability Check by Cross-Section*

The serviceability design is sorted by cross-sections. The table shows the most unfavorable 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.

Member No.

For every cross-section, the number of the member with the greatest result values (column E) or capacity (column H) is shown. The reference can be changed using the [Sort] button.

Location x

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

Loading

This column shows the numbers of the load cases, load or result combinations that are governing for the SLS design.



Type

This column describes the criteria of the serviceability check. Those can be

- $u_{z,local}$ - Deflection in direction of the local member axis z
- $lim s_l$ - Crack width design without direct crack width calculation according to ACI 318-14, 24.3.2
- w_k - Crack width design with direct crack width calculation according to the GERGELY-LUTZ equation (see page 12).

Existing Value

Column E displays the values that are governing for each criterion of the serviceability limit state design.

Limit Value

The limit values are determined from the specifications of the ACI Standard.

Capacity

This column lists the ratios of the SLS design. They depend on the specifications made in Window 1.6 Reinforcement, tab Serviceability.

Note

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

Messages...

To display all notes of the serviceability limit state design, use the [Messages] button. A dialog box with relevant information appears.

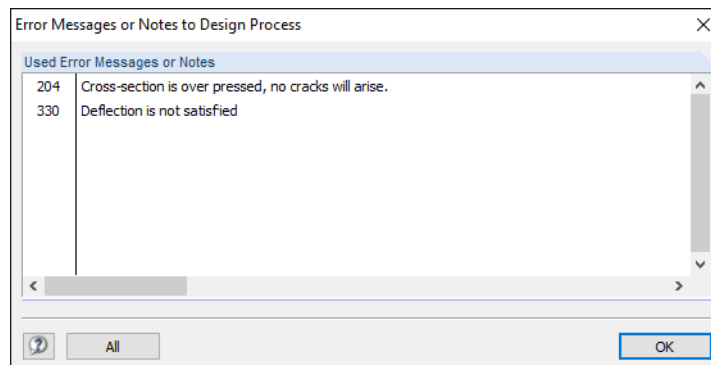


Figure 5.20: Dialog box *Error Messages or Notes to Design Process*

5.3.2 Serviceability Check by Set of Members

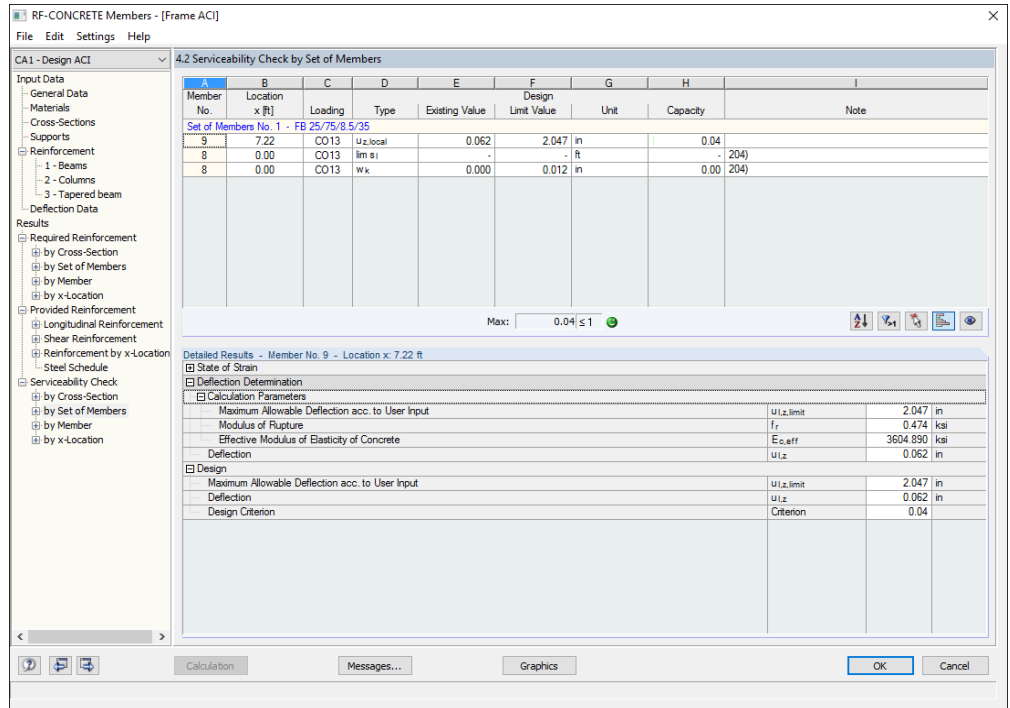


Figure 5.21: Window 4.2 Serviceability Check by Set of Members

When sets of members have been selected for design, the governing serviceability limit state designs are sorted by sets of members in this result window.

Details on the individual columns can be found in Chapter 5.3.1.

5.3.3 Serviceability Check by Member

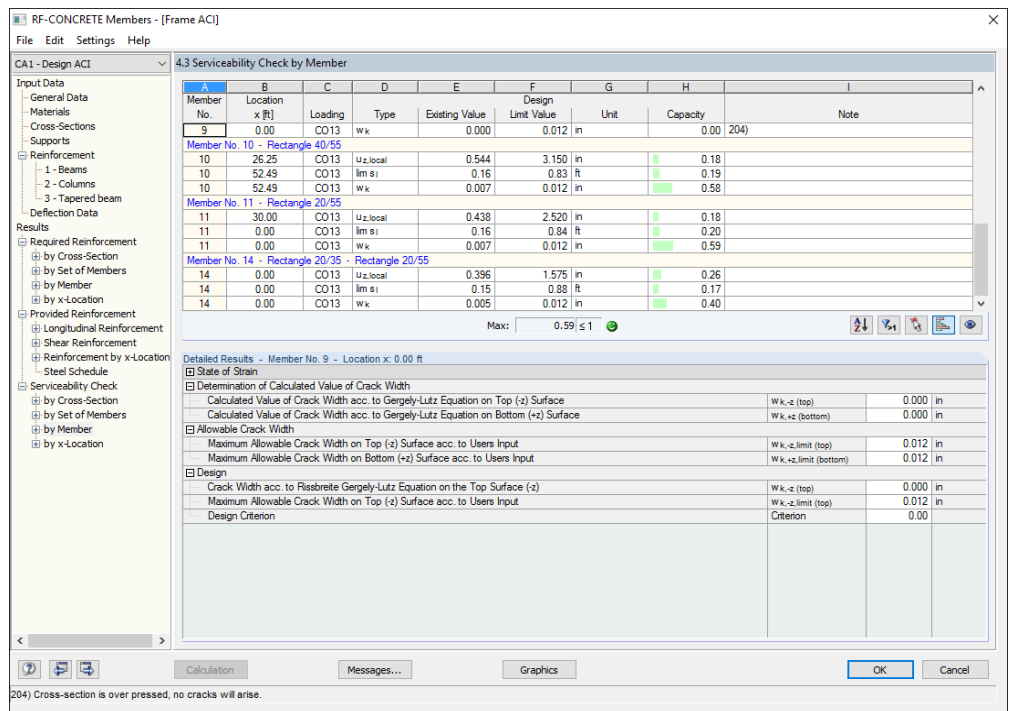
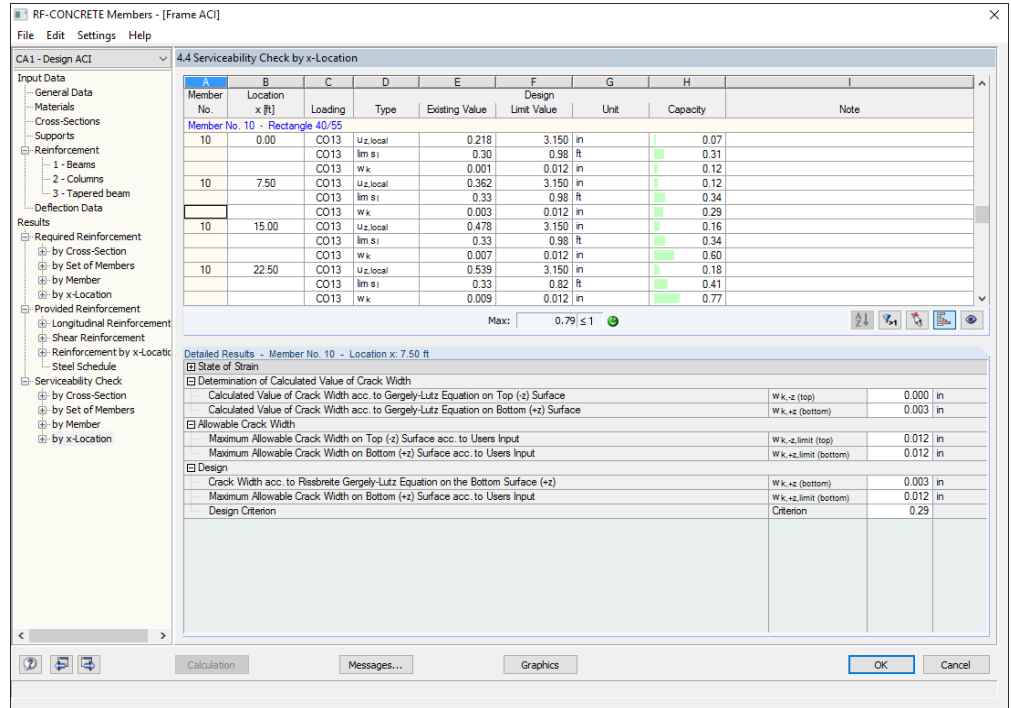


Figure 5.22: Window 4.3 Serviceability Check by Member

This result window shows the output for the crack width designs sorted by member. The table columns correspond to the columns of Window 4.1. They are described in Chapter 5.3.1.

5.3.4 Serviceability Check by x-Location



The screenshot shows a software window titled "4.4 Serviceability Check by x-Location" with a menu bar (File, Edit, Settings, Help) and a tree view on the left. The main area contains a table with the following data:

Member No.	Location x [ft]	Loading	Type	Existing Value	Design Limit Value	Unit	Capacity	Note
Member No. 10 - Rectangle 40/55								
10	0.00	CO13	Uz,local	0.218	3.150	in	0.07	
		CO13	lim s1	0.30	0.98	ft	0.31	
		CO13	wk	0.001	0.012	in	0.12	
10	7.50	CO13	Uz,local	0.362	3.150	in	0.12	
		CO13	lim s1	0.33	0.98	ft	0.34	
		CO13	wk	0.003	0.012	in	0.29	
10	15.00	CO13	Uz,local	0.478	3.150	in	0.16	
		CO13	lim s1	0.33	0.98	ft	0.34	
		CO13	wk	0.007	0.012	in	0.60	
10	22.50	CO13	Uz,local	0.539	3.150	in	0.18	
		CO13	lim s1	0.33	0.82	ft	0.41	
		CO13	wk	0.009	0.012	in	0.77	

Below the table, there is a "Detailed Results" section for Member No. 10 at Location x: 7.50 ft, showing state of strain, determination of calculated value of crack width, allowable crack width, and design criteria.

Figure 5.23: Window 4.4 Serviceability Check by x-Location

This window lists the SLS design results (see Chapter 5.3.1) for all locations x of the designed members.

6. Results Evaluation

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 result Windows 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 lower part of Window 3.1 *Provided Longitudinal Reinforcement* and Window 3.2 *Provided Shear Reinforcement* (see Figure 5.9, page 50 and Figure 5.15, page 55).

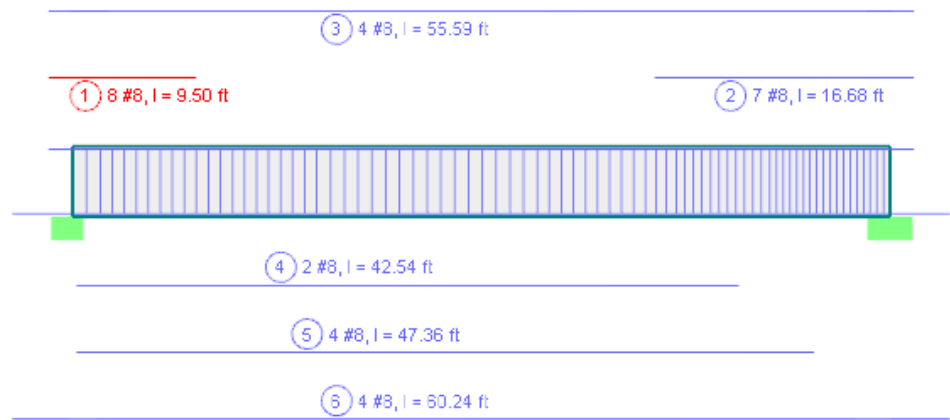


Figure 6.1: Reinforcement drawing in Window 3.1 *Provided Longitudinal Reinforcement*

The current item, i.e. the row selected in the table above, 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 item, click the [Edit Reinforcement] button in the toolbar above the graphics. The dialog box shown in Figure 5.12 on page 53 or in Figure 5.16 on page 56 appears. In this dialog box, you can check the individual parameters of the selected longitudinal or stirrup reinforcement and adjust them, if necessary.

6.2 3D Rendering of Reinforcement

3D-Rendering

In the result Windows 3.1 *Longitudinal Reinforcement* and 3.2 *Shear Reinforcement*, the button [3D-Rendering] is available. It enables a photo-realistic visualization of the provided reinforcement. A new window opens, showing a rendered graphical representation of the reinforcing cage of the current member or set of members (i.e. the table row of the object where the pointer is placed).

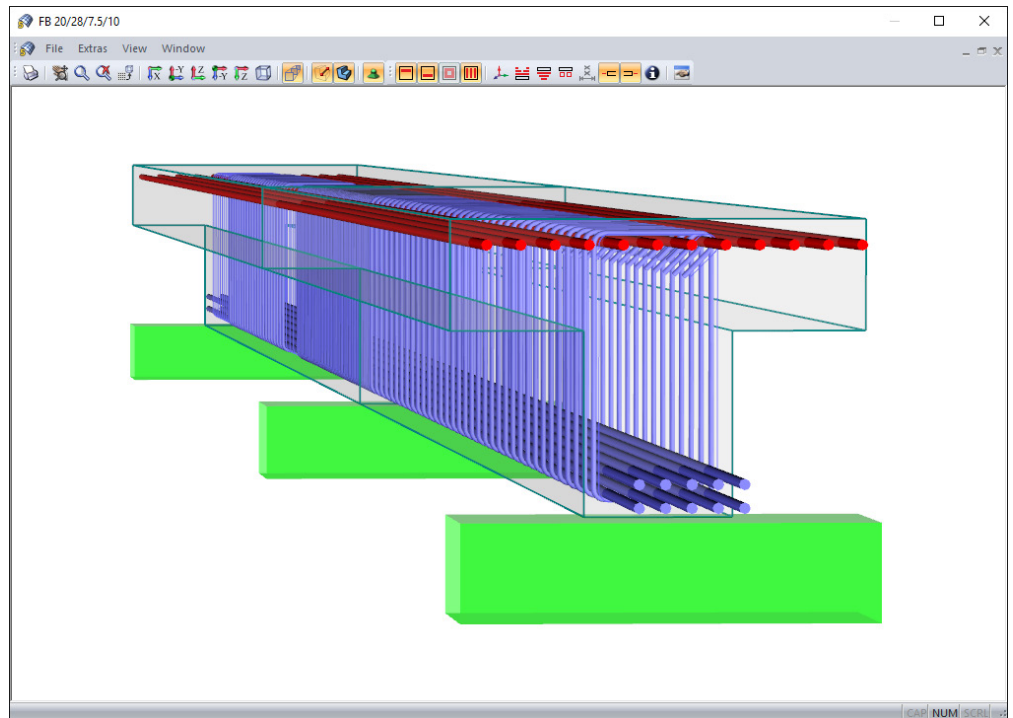


Figure 6.2: 3D rendering of provided longitudinal and stirrup reinforcement

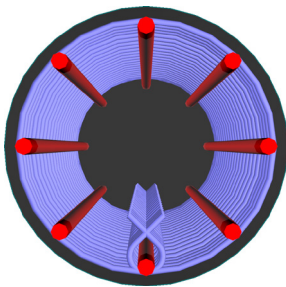
By means of the graphic, you can check the selected reinforcement close to reality.

To set the graphic display appropriately, use the *View* 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 to the printer, the printout report or the clipboard.

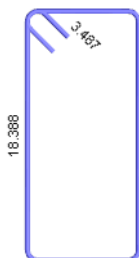
























The buttons in the toolbar have the following functions:



Reinforcement of column in perspective view

① 17 #3 - 0.75 ft



Button	Description	Function
	Print	Opens the dialog box <i>Graphic Printout</i> (see Figure 7.4, page 74) providing the printout settings
	Move, Zoom, Rotate	Allows for shifting the view by the mouse (zooming/rotating by pressing [Shift] or [Ctrl]/[Alt] key)
	Zoom	Allows for increasing a particular region in the graphic by drawing a window with the left mouse button
	Show whole model	Resets the graphic's full view
	Previous view	Shows the previously selected view
	View in X	Shows the view in YZ plane
	View in -Y	Shows the view in XZ plane (vertically/horizontally)
	View in Z	Shows the view in XY plane (vertically/horizontally)
	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)
	View support	Shows the supports defined in <i>Window 1.5 Supports</i>
	Line model	Hides the concrete material
	Solid model	Represents the concrete in the member or set of members
	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 longitudinal reinforcement
	Link reinforcement	Shows the stirrup reinforcement
	Member axis system	Controls the display of the local member axes x,y,z
	Longitudinal reinforcement - top	Shows the item of the top reinforcement above the member
	Longitudinal reinforcement - bottom	Shows the item of the bottom reinforcement below the member
	Shear reinforcement	Displays the items of the stirrup reinforcement
	Direction of description	Shows description of stirrup reinforcement perpendicular to member

	View start of anchorage	Shows anchorage of longitudinal reinforcement at start of member
	View end of anchorage	Shows anchorage of longitudinal reinforcement at end of member
	Show information	Shows additional information for reinforcement in lower left corner of window
	Edit reinforcement	Opens dialog box for editing reinforcement (see Figure 5.12, page 53 and Figure 5.16, page 56)

Table 6.1: Buttons for 3D rendering

6.3 Results on RFEM Model

You can also evaluate the design results in the RFEM work window.

6.3.1 RFEM Background Graphic

The RFEM graphic in the background is 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 background graphic. In addition, an arrow indicates the x-location of the selected member.

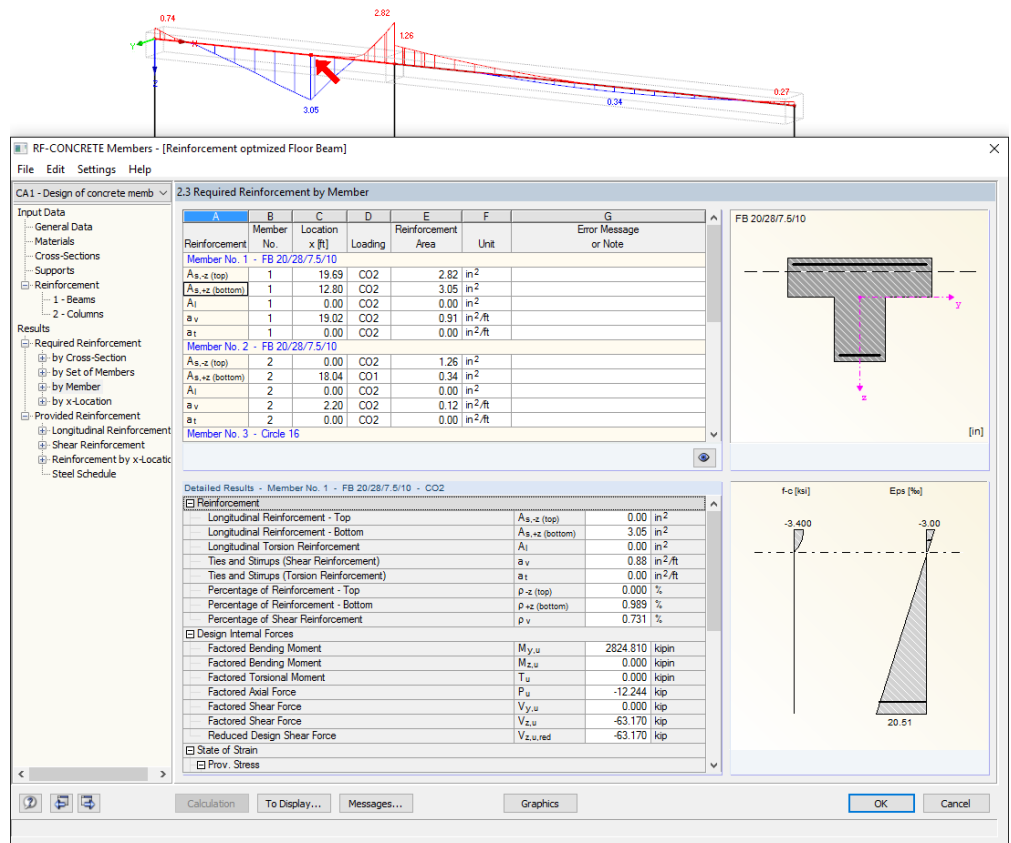


Figure 6.3: Indication of the member and the current Location x on RFEM model

Graphics

RF-CONCRETE Members

This function, however, is only available when the results of the RF-CONCRETE Members case are set in the RFEM 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.

6.3.2 RFEM Work Window

You can check the reinforcement areas and detailed results graphically on the RFEM model: First, click the [Graphic] button to close the add-on module. Then, all design results are displayed in the work window of RFEM, like the internal forces or deformations of a RFEM load case.

You can set the design case in the drop-down-list of the RFEM menu bar.

The *Results* navigator is aligned with the design results of the RF-CONCRETE Members module. It enables you to select the different reinforcement types for the strength limit state and the serviceability limit state designs including all detailed results.

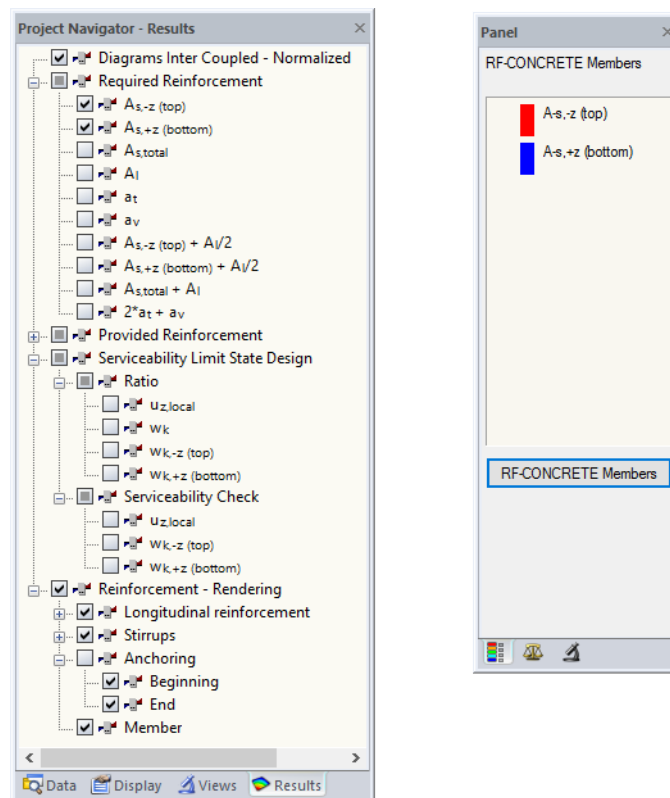


Figure 6.4: Results navigator and panel for RF-CONCRETE Members

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.

Find an example for the graphical documentation for a shear analysis in the following article: <https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000715>

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

To turn the display of design results on or off, use the toolbar button [Results on/off]. To display the result values in the graphic, use the [Show Result Values] button to the right of it.

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



Similarly to the member internal forces, the member diagrams can be scaled in the second panel tab *Display Factors* for the evaluation and the printout.

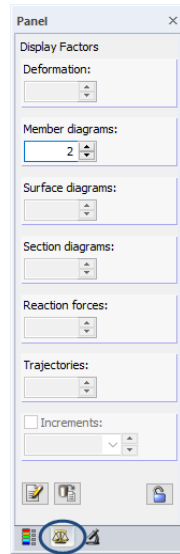


Figure 6.5: Panel tab *Display Factors*

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

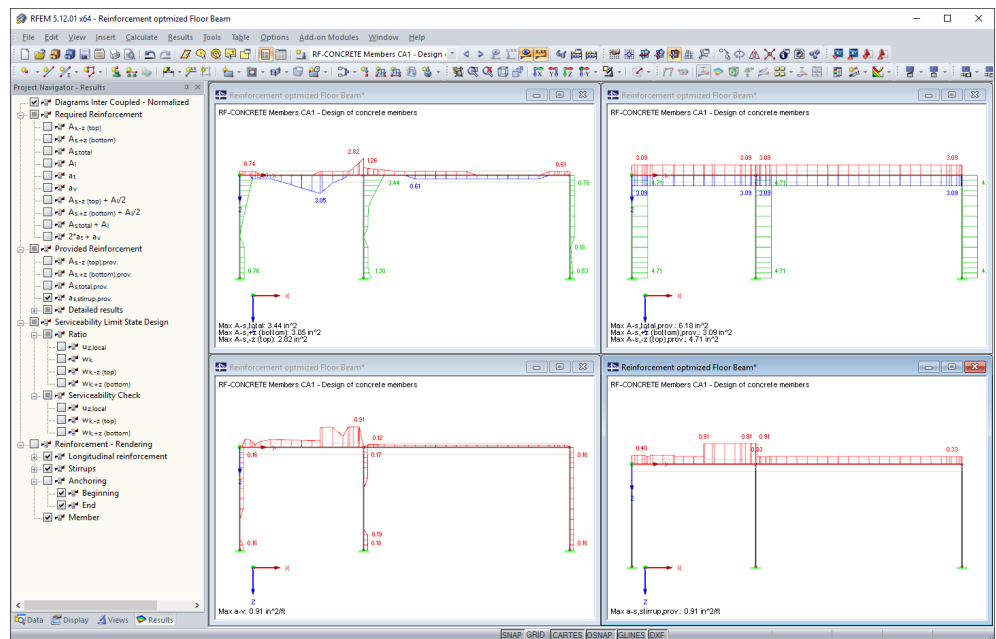


Figure 6.6: Graphical output of required and provided reinforcement

All results graphics can be transferred like RFEM graphics to the global printout report (see chapter 7.2, page 74).

With the *Reinforcement - Rendering* option in the navigator, it is possible to display the rebars and stirrups in the RFEM work window (see Figure 6.7).

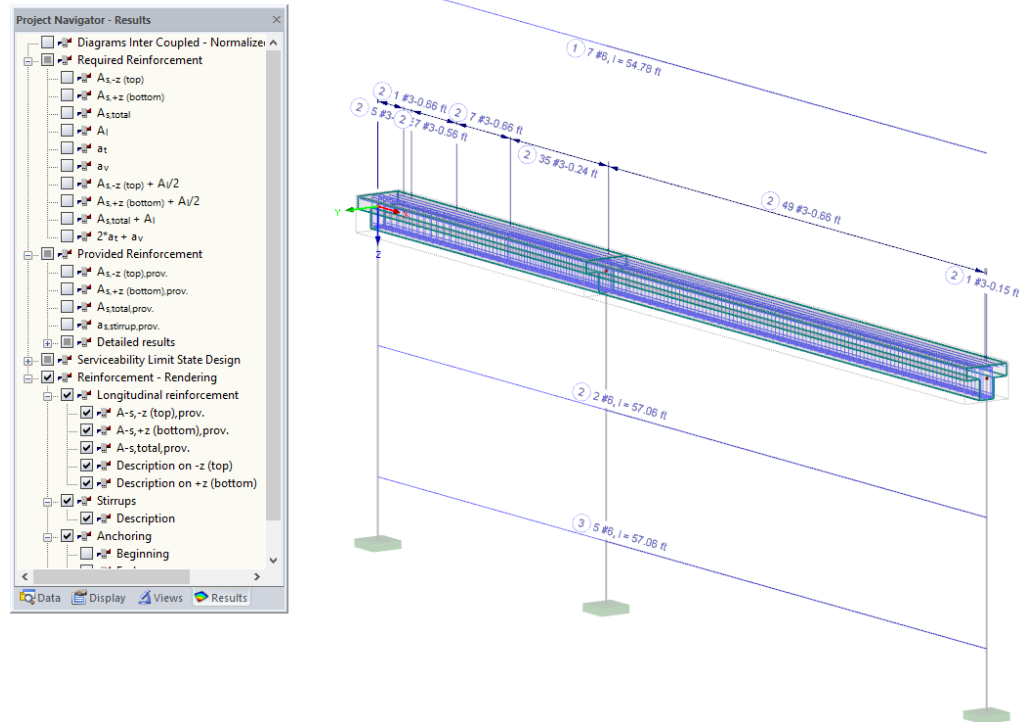


Figure 6.7: 3D rendering of reinforcement in RFEM



As usual in RFEM, you can use the [Print Graphic] button to send the graphic to the printer, or to transfer it to the printout report.

RF-CONCRETE Members

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

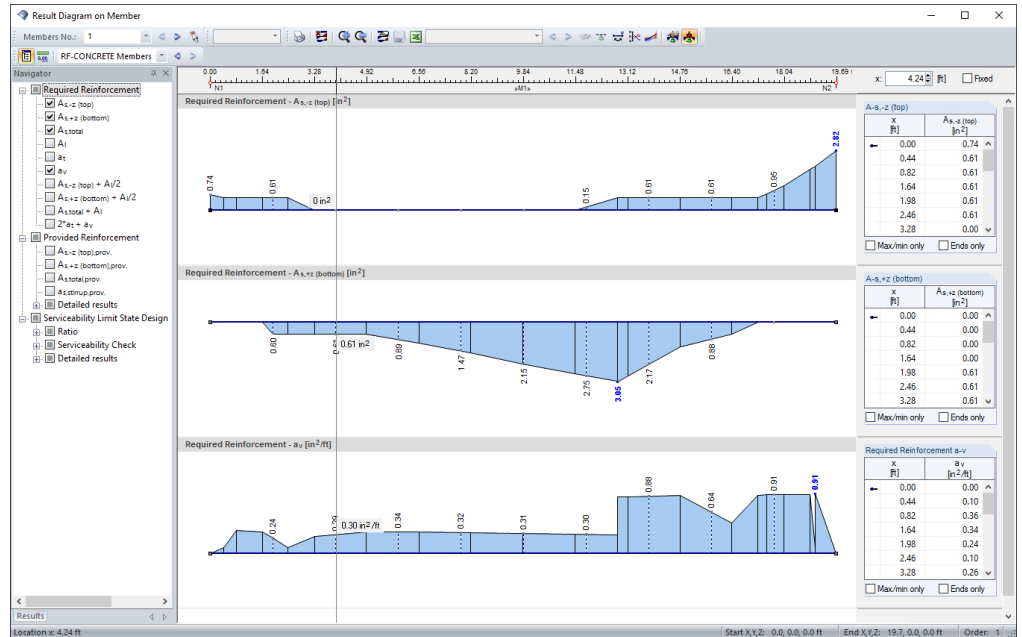


Figure 6.8: Dialog box *Result Diagram on Member*

In the Navigator on the left, select the reinforcements and detailed results. 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 Chapter 9.5 of the RFEM manual.

6.5 Filter for Results

In addition to the result windows which already allow for a particular selection according to certain criteria due to their structure, you can use the filter options for members or results to evaluate the designs graphically.



Furthermore, you can apply defined or new *Visibilities* (see Chapter 9.9.1 of the RFEM manual) to group objects for the evaluation.

To set the numbers of those members whose results are to be shown exclusively, use the *Filter* tab of the control panel. This function is described in Chapter 9.9.3 of the RFEM manual.

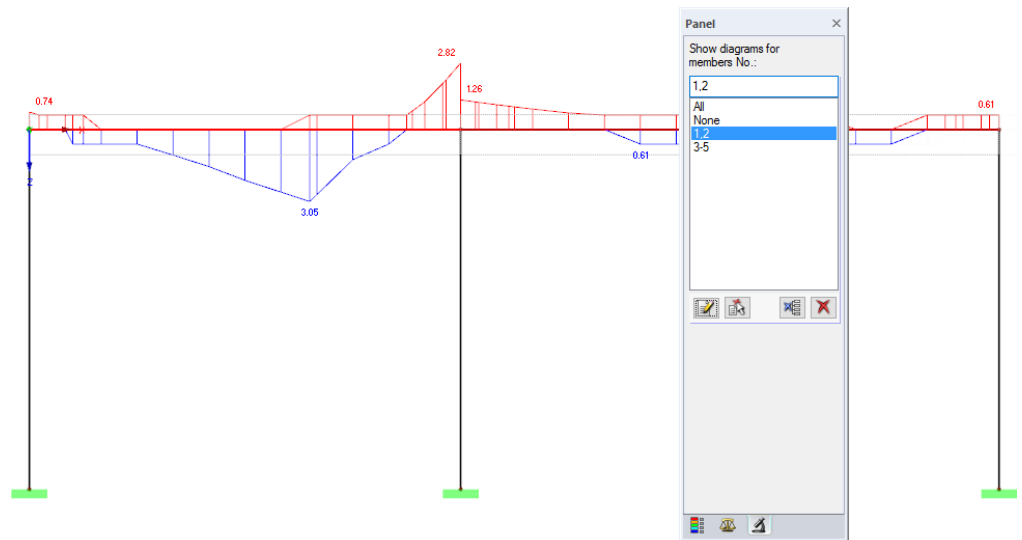


Figure 6.9: Filtering members in the panel

In contrast to the *Visibilities* function, the entire model is displayed in the graphics.

7. Printout

7.1 Printout Report

Creating printouts is similar to the printout procedure in RFEM. First, the program generates a printout report for the results of RF-CONCRETE Members. Then, graphics and descriptions can be added. In the printout report, you can select the types of results which you want to include for the reinforced concrete design.



When your model is rather 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 can be generated relatively quickly.

The printout report is described in detail in the RFEM manual. In particular, Chapter 10.1.3.5 *Selecting Data of Add-on Modules* provides information concerning the selection of input and output data in all add-on modules.

The Printout Report Selection dialog box provides various options to select input and output data. Thus, the reinforcements as well as the detailed results can be individually prepared for the documentation.

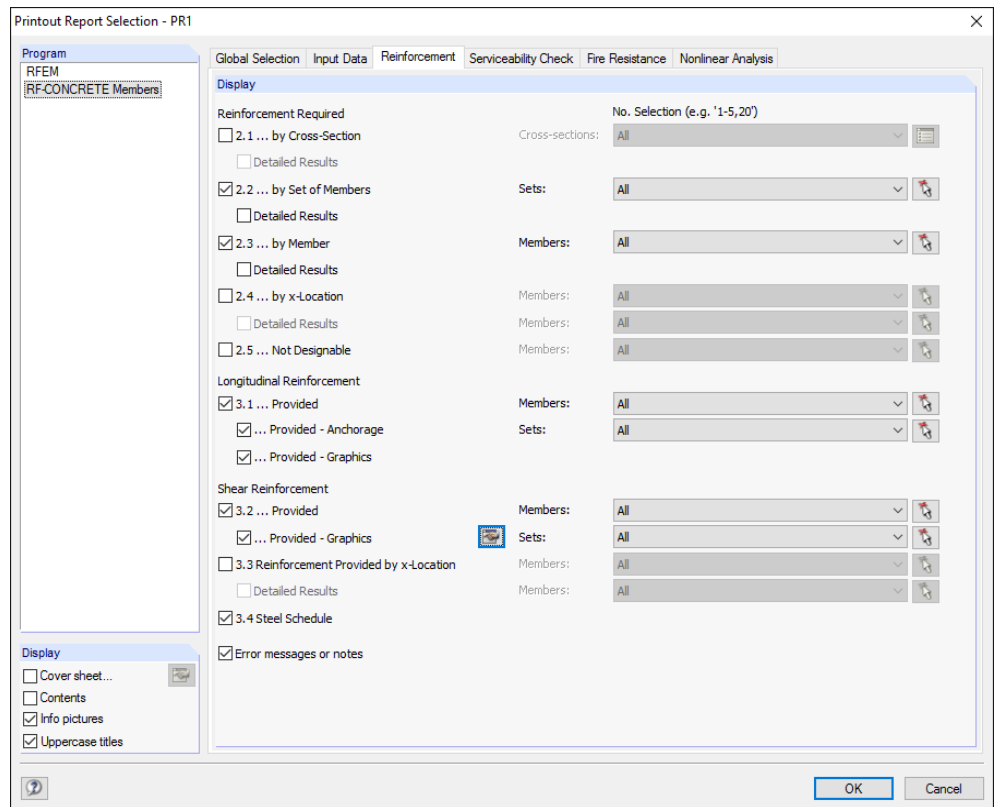


Figure 7.1: Printout report selection of results of RF-CONCRETE Members, tab *Reinforcement*

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.



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 incorporate the 3D rendering graphics and the member result diagrams in the printout report by using the [Print Graphic] 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.

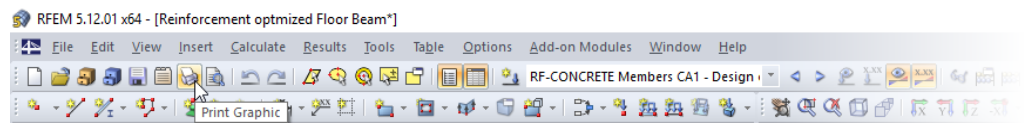


Figure 7.2: Button *Print* in the toolbar of the main window

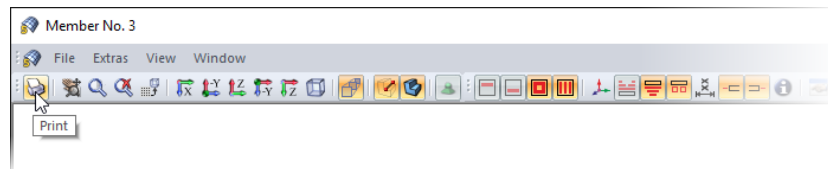


Figure 7.3: Button *Print* in the toolbar of the 3D rendering window

The following dialog box opens:

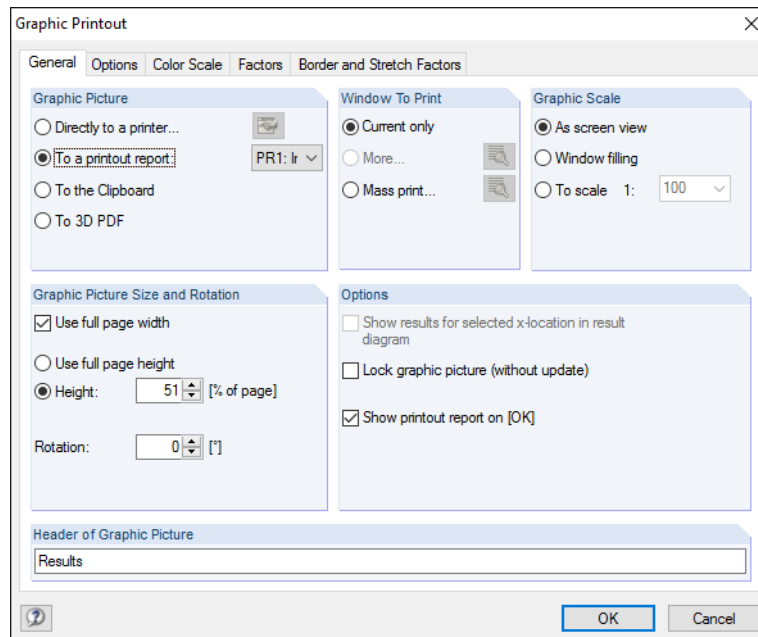


Figure 7.4: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in Chapter 10.2 of the RFEM manual.

To move a graphic within the report to another position, use the drag-and-drop function.

To modify a graphic subsequently in the printout report, right-click it in the report Navigator. The *Properties* option in the shortcut menu opens the *Graphic Printout* dialog box where you can adjust the settings.

Reinforcement Member No. 3, Perspective

- Remove from Printout Report
- Start with New Page
- Selection...
- Properties...

8. General Functions

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

8.1 Design Cases

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 groups of structural components, for example.

It is no problem to analyze the same member or set of member in different design cases.

In RFEM, you can set the design cases of RF-CONCRETE Members in the load case list of the toolbar.

Create new design case

To create a new design case,

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

The following dialog box appears.

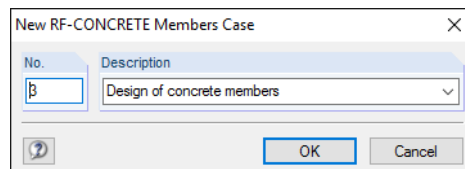


Figure 8.1: Dialog box *New RF-CONCRETE Members Case*

In this dialog box, enter a *No.* (which is not yet assigned) and a *Description* for the new design case. When you have clicked [OK], *Window 1.1 General Data* opens where you can enter the new design data.

Rename design case

To change the description of a design case,

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

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

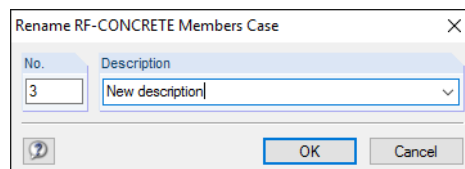
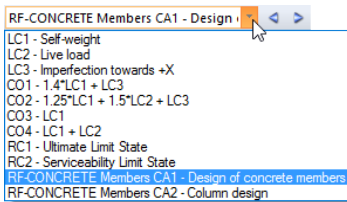


Figure 8.2: Dialog box *Rename RF-CONCRETE Members Case*

In this dialog box, you can specify a different *Description* as well as a different *No.* for the design case.



Copy design case

To copy the input data of the current design case,
select **Copy Case** on the **File** menu of the RF-CONCRETE Members add-on module.

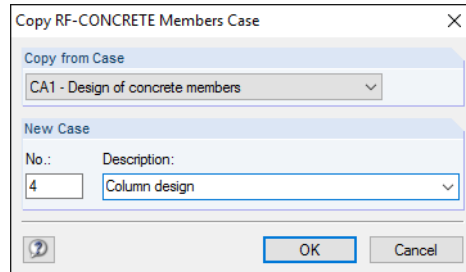


Figure 8.3: Dialog box *Copy RF-CONCRETE Members Case*

In the dialog box, you can specify the *No.* and, if necessary, the *Description* of the new case.

Delete design case

To delete a design case,
select **Delete Case** on the **File** menu of the RF-CONCRETE Members add-on module.

In the dialog box *Delete Cases*, you can select a design case in the *Available Cases* list.

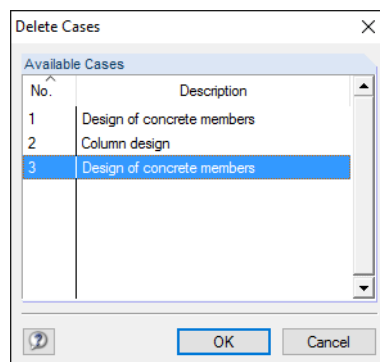


Figure 8.4: Dialog box *Delete Cases*

To delete the selected case, click [OK].

8.2 Cross-section Optimization

As mentioned in Chapter 3.3, RF-CONCRETE Members offers you the possibility to optimize cross-sections. Select the check box of the relevant cross-section in column C of Window 1.3 *Cross-Sections* (see Figure 3.6, page 22). Immediately afterwards, the following dialog box is opened where you can define the parameters.

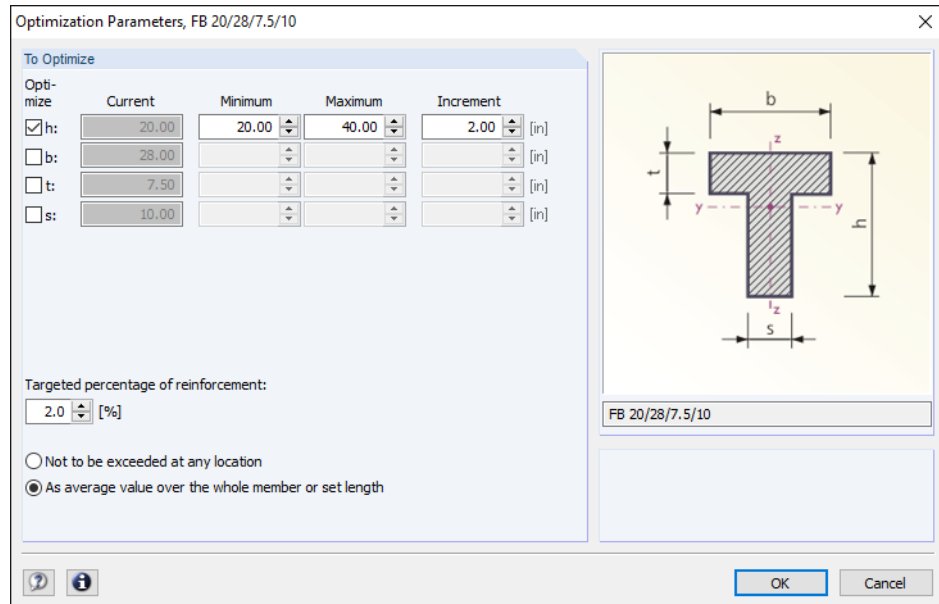


Figure 8.5: Dialog box *Optimization Parameters* of a T-beam

By selecting the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. The selected check boxes enable the *Minimum* and *Maximum* columns where you can define the upper and lower limits of each parameter for optimization. The *Increment* column controls the interval in which the dimensions of each parameter vary during the optimization.

The criterion for optimization is controlled by the specification that the *Targeted percentage of reinforcement* either is not exceeded at any location or is available as average value over the entire object. The reinforcement ratio can be defined individually.



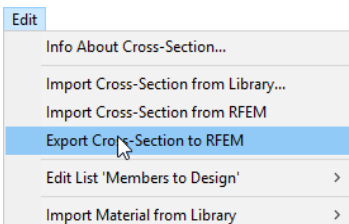
During the optimization, RF-CONCRETE Members finds out which dimensions of the selected parameter(s) should be used in order to still fulfill the design. Please note that the internal forces won't be recalculated automatically with the modified cross-sections (the internal forces may vary considerably because of the changed stiffnesses within the model). It is up to you to decide when to transfer the optimized cross-sections to RFEM for a new calculation. Therefore, it is recommended to recalculate the internal forces with the modified cross-sections after the first optimization, and then to optimize the sections once again.

You do not need to transfer the optimized cross-sections to RFEM manually: Got to Window 1.3 *Cross-Sections*, and then

select **Export Cross-Section to RFEM** on the **Edit** menu.

You can also use the shortcut menu of the table rows in Window 1.3 to export the modified cross-sections to RFEM.

Before the cross-sections are transferred, a query asks you whether the results of RFEM should be deleted. When you confirm this query and start the [Calculation] in RF-CONCRETE Members, the internal forces and areas of reinforcement are determined in one calculation run.



To reimport the original RFEM cross-section to RF-CONCRETE Members, use the menu function described above. Please note that this option is only available in Window 1.3 *Cross-Sections*, too.

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-sections 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 dialog box. In the module RF-CONCRETE Members, you can use the menu to change the units. To open the corresponding dialog box,

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

The following dialog box, which you already know from RFEM, opens. RF-CONCRETE Members is preset in the *Program / Module* list.

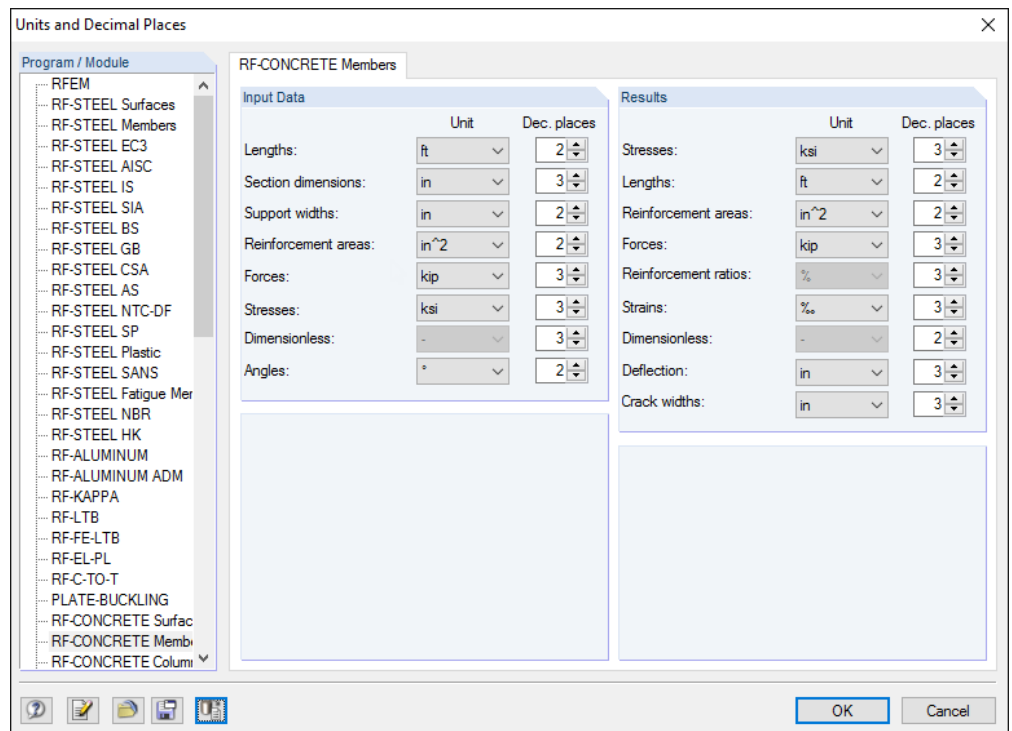


Figure 8.6: Dialog box *Units and Decimal Places*



The settings can be saved as a user profile so that they can be reused in other models. This function is described in Chapter 11.1.3 of the RFEM manual.

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 result windows of RF-CONCRETE Members to the clipboard, use the keys [Ctrl]+[C]. To insert them, 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 printout report (see Chapter 7.1, page 73) where they can be exported. In the printout report,

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

The function is described in Chapter 10.1.11 of the RFEM manual.

MS Excel

RF-CONCRETE Members provides a function for the direct data export to MS Excel 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 dialog box appears.

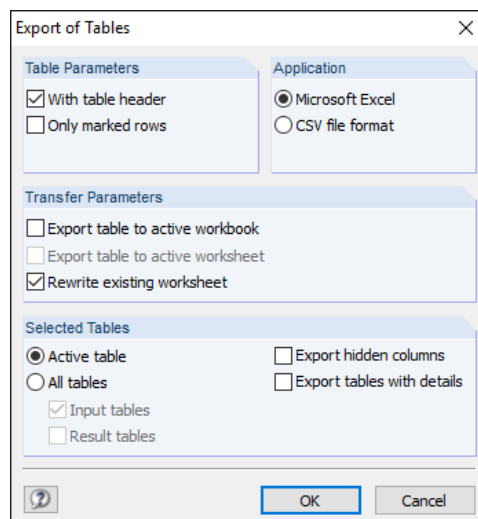


Figure 8.7: Dialog box *Export of Tables*

When you have selected the relevant parameters, start the export by clicking [OK].

Excel will be started automatically. It is not necessary to run the program in the background.

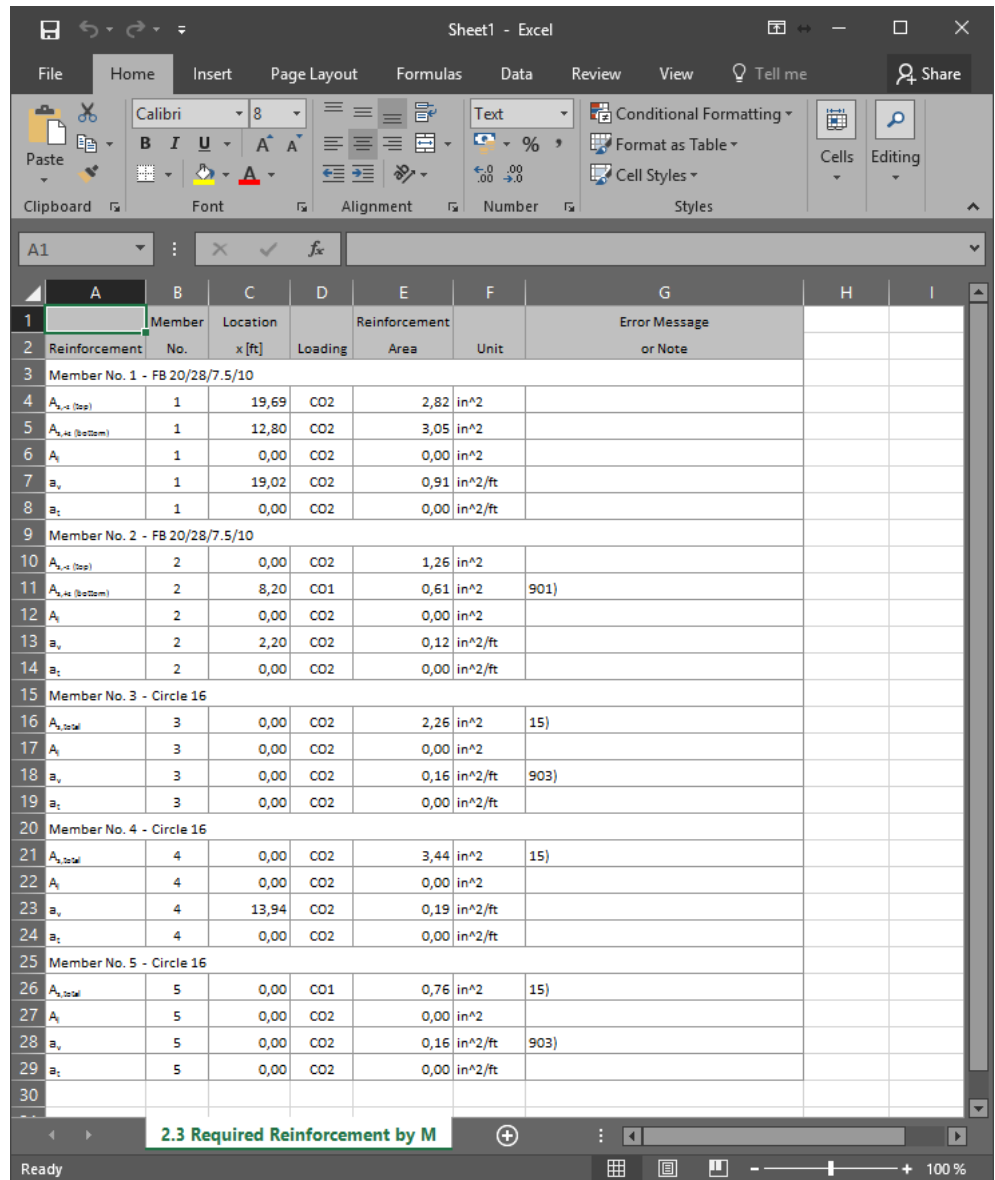


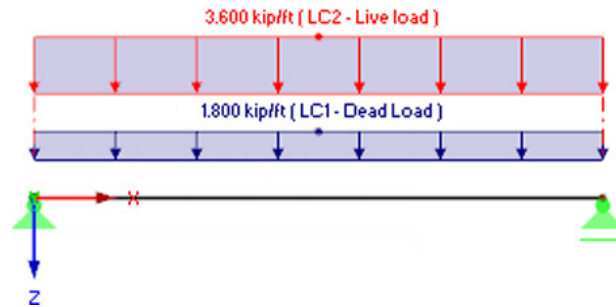
Figure 8.8: Result in MS Excel

9. Example

This chapter describes an example where the doubly reinforced beam is designed for strength limit state according to ACI 318-14.

9.1 Input Data

System



Cross-section

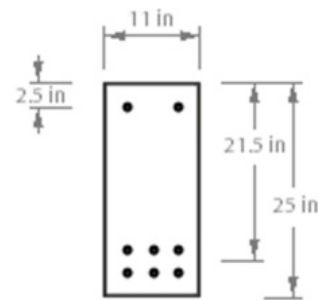


Figure 9.1: System, loads and cross-section

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

Loads

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

Factored internal forces

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

9.2 Flexural Strength

Check reinforcement ratio

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

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

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

Check yielding of compression reinforcement

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

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

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

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

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

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

$\epsilon'_s < \epsilon_y \Rightarrow$ Compression reinforcement does not yield, assumption is not valid.

Define rectangular concrete stress distribution

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

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

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

$$\phi \cdot M_n = \phi \left(A_s f_y - A'_s f'_s \right) \left(d - \frac{a}{2} \right) + A'_s f'_s (d-d') = 0.9 \left(\begin{array}{l} (4.71 \cdot 60 - 1.57 \cdot 47.81) \left(21.5 - \frac{4.44}{2} \right) + \\ + 1.57 \cdot 47.81 (21.5 - 2.5) \end{array} \right) =$$

$$= 4885 \text{ kips} \cdot \text{in} = 407.1 \text{ kips} \cdot \text{ft}$$

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

9.3 Shear Strength

Shear strength provided by concrete

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

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

$$\left(1.9 \lambda \sqrt{f'_c} + 2500 \rho_w \frac{V_u d}{M_{u,\text{support}}} \right) b_w d =$$

$$= (1.9 \cdot 1.0 \sqrt{5000} + 2500 \cdot 0.01992 \cdot 1) \cdot 11 \cdot 21.5 \frac{1}{1000} = 43.55 \text{ kips}$$

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

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

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

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

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

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

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

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

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

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 longitudinal and shear reinforcement designed by the add-on module RF-CONCRETE Members.

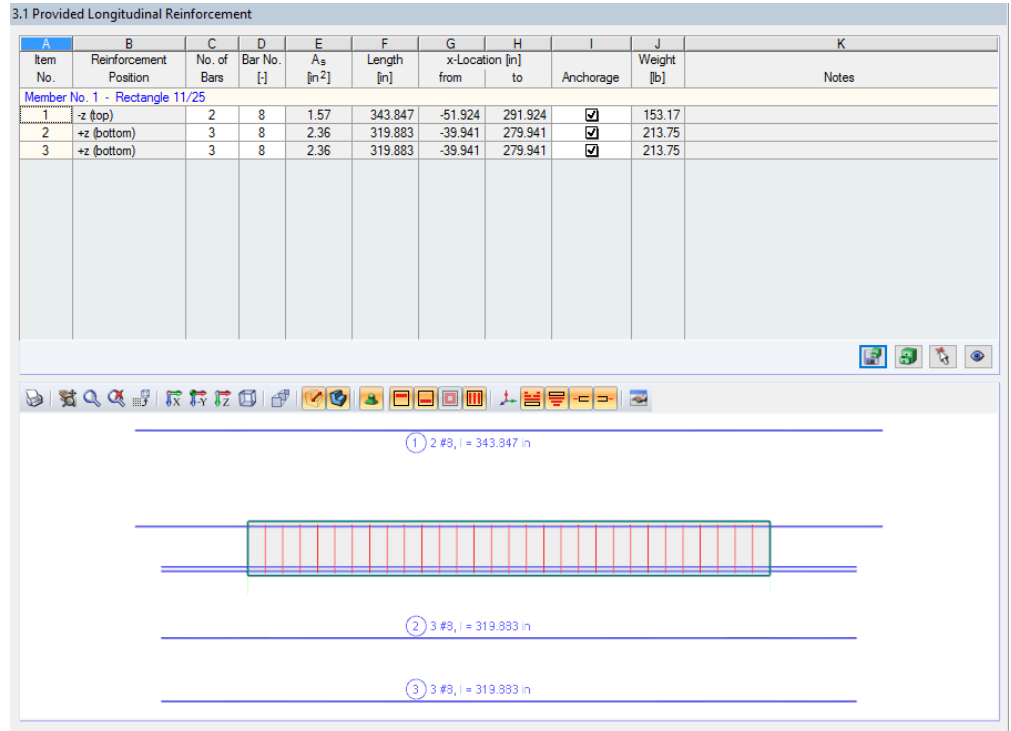


Figure 9.2: Window 3.1 Provided Longitudinal Reinforcement

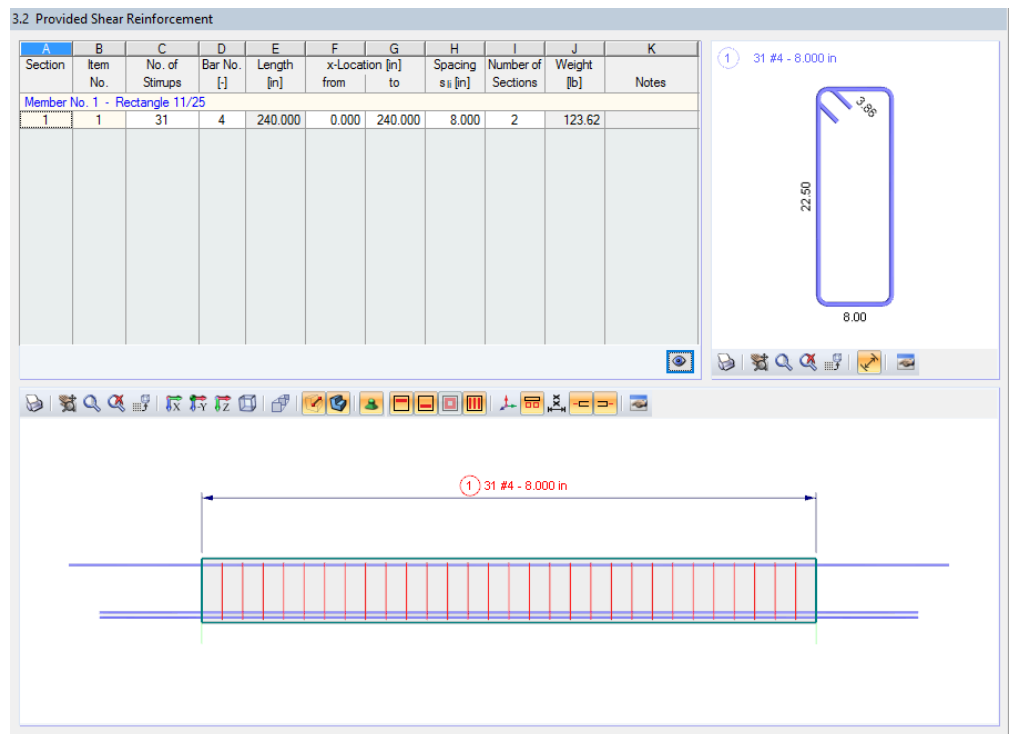


Figure 9.3: Window 3.2 Provided Shear Reinforcement

A Literature

- [1] ACI 318-14: Building Code Requirements for Structural Concrete (ACI 318-14) and Commentary, ACI Committee 318, September 2014
- [2] EN 1992-1-1:2005 + AC2010: Design of Concrete Structures – Part 1-1 : General Rules and Rules for Buildings. 2005

B Index

3	
3D Rendering.....	65, 74
A	
Anchorage.....	31, 52
Anchorage type.....	52, 59
Axial force.....	8
B	
Background graphic.....	67
Bar spacing.....	31
Basic reinforcement.....	31
Bearing capacity.....	68
Bend.....	52
Bending.....	8
Bending diameter.....	53, 59
Bond.....	52
Bottom reinforcement.....	45
Buttons.....	25, 66
C	
Calculation.....	42
Centroidal axis.....	34
Code.....	18, 36
Comment.....	18
Compression force.....	9
Compression reinforcement.....	9
Concrete cover.....	34
Concrete strength class.....	20
Corner.....	35
Crack width.....	12, 37
Cross-section.....	22, 29
Cross-section library.....	22
Cross-section optimization.....	77
CSV export.....	79
Curtailment.....	31
D	
Decimal places.....	20, 78
Deflection.....	13, 38
Deformation.....	4
Design.....	29
Design case.....	68, 75, 76
Detailed results.....	44, 71
Development length.....	14, 84
Diameter.....	30, 32
Direct support.....	26
Distribution.....	34
E	
Edge distance.....	34
Effective width.....	24
End support.....	27
Error message.....	46, 49
Excel.....	79
Export of cross-sections.....	77
Export of results.....	79
F	
Failed design.....	49
Filter.....	72
Filtering members.....	72
G	
General data.....	17
Gergely and Lutz equation.....	12
Graphic.....	68
H	
Hook.....	52
I	
Inclination.....	32, 51
Indirect support.....	26
Installation.....	4
Internal forces.....	77
Item.....	50, 55, 58, 64
Item member.....	50, 53, 64
L	
Leg.....	32
Length.....	52
Length of anchorage.....	31, 32
Load case.....	18, 19
Load combination.....	18, 19
Load near support.....	27
Location x.....	46, 48, 51, 52, 56, 57, 60, 63
Longitudinal reinforcement.....	30, 44, 45, 50, 58, 64, 66
M	
Material.....	20
Material library.....	21
Material properties.....	20
Maximum reinforcement ratio.....	36

Member	28, 47, 62	Rib	24, 25
Minimum reinforcement	35	S	
Minimum ticknesses	13	Safety	58
Moment redistribution	27	Scaling	69
Monolithic connection	26	Section	56
M-ratio δ	27	Selecting tables	17
N		Serviceability	11, 19, 50, 60, 68
Navigator	17	Serviceability limit state	11, 29, 37, 60
Nominal value	34	Set of members	25, 28, 38, 47, 62
Notes	53, 56, 61	Shear force	27
O		Shear force resistance	10
Open RF-CONCRETE Members	5	Shear reinforcement	32, 36, 44, 45, 55
Optimization	23, 77	Slab width	34
P		Standard	4
Panel	6	Start program	5
Percentage of reinforcement	36, 58, 77	Steel schedule	58
Plausibility check	42	Stirrup	32
Print	74	Stirrup parameter	32
Printing the graphic	74	Stirrup reinforcement	45, 58, 64, 66
Printout report	73, 79	Stirrup spacing	33, 56
Provided reinforcement	50	Stirrup zone	56
Q		Strength limit state	7, 8, 10, 18
Quit RF-CONCRETE Members	17, 44	Strength reduction factor	7, 37
R		Structural reinforcement	35
Rebar	51	Support	26
Rebar diameter	35, 55, 59	Support width	26
Rebar spacing	12	Surface	59
Reinforcement	28, 30, 32	Surrounding reinforcement	34
Reinforcement area	44, 46	T	
Reinforcement drawing	64	Taper	25, 38, 39, 47, 78
Reinforcement group	28	Template	29
Reinforcement layer	31	Tension force	9
Reinforcement layout	33, 34	Top reinforcement	45
Reinforcement position	51	Torsion	45
Reinforcement proposal	50, 53, 56, 64	Type of anchorage	31, 32
Reinforcing steel	21	U	
Required reinforcement	44	Units	20, 78
Result combination	18, 19, 27	V	
Result diagrams	71, 74	Visibility	72
Result values	68	Visualization	65, 68
Result windows	44	W	
RF-CONCRETE Members case	43, 75	Weight	53, 56, 59
RFEM work window	67	Window	17