Dlubal

Version
October 2017

Program

RF-CONCRETE Members CSA

Reinforced Concrete Design According to CSA A23.3-14

Program Description

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

1. Introduction

1.1 Add-on Module RF-CONCRETE Members

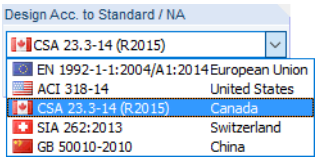
The add-on module RF-CONCRETE Members for reinforced concrete design is completely integrated in the RFEM user interface. Thus, a continuous analysis process is guaranteed for the design of framework elements consisting of reinforced concrete.

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

RF-CONCRETE Members analyzes the ultimate and the serviceability limit state. The analysis for cracks and deflections are performed by calculating a minimum reinforcement for crack width control and direct evaluation of deformations.

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:



Design according to CSA 23.3-14

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

The required reinforcement that is determined by the program also includes a reinforcement proposal. It takes into account all user specifications concerning the rebars in the longitudinal and stirrup reinforcement. The layout of this reinforcement can be adjusted anytime. All designs related to the modifications will be updated automatically.

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

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

Your team from DLUBAL SOFTWARE GMBH

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 with respect to the Canadian Standard.

The descriptions follow the order of the input and result windows of the add-on module, as well as their structure. The text shows the described **buttons** in square brackets, for example [Graphic]. They are also shown on the left. In addition, **expressions** used in dialog boxes, windows 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 at our website that can answer your question.

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 **Add-on Modules** menu, and then select **RF-CONCRETE Members**.

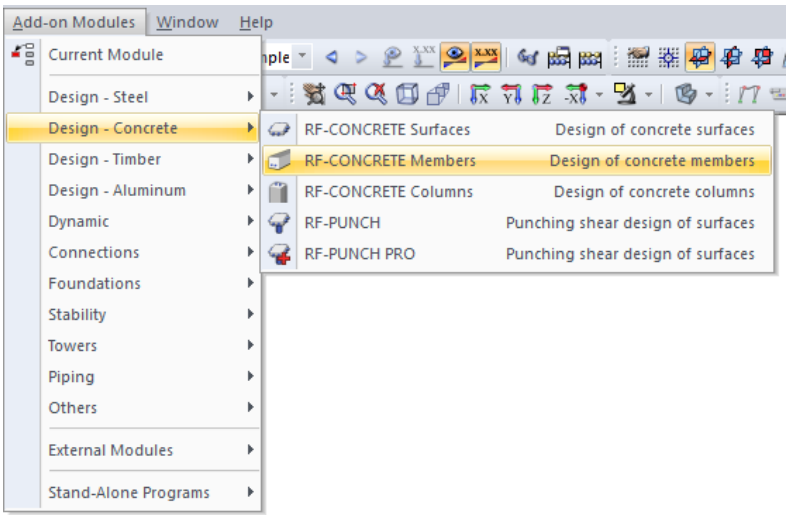


Figure 1.1: Menu *Add-on Modules* → *Design - Concrete* → *RF-CONCRETE Members*

Navigator

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

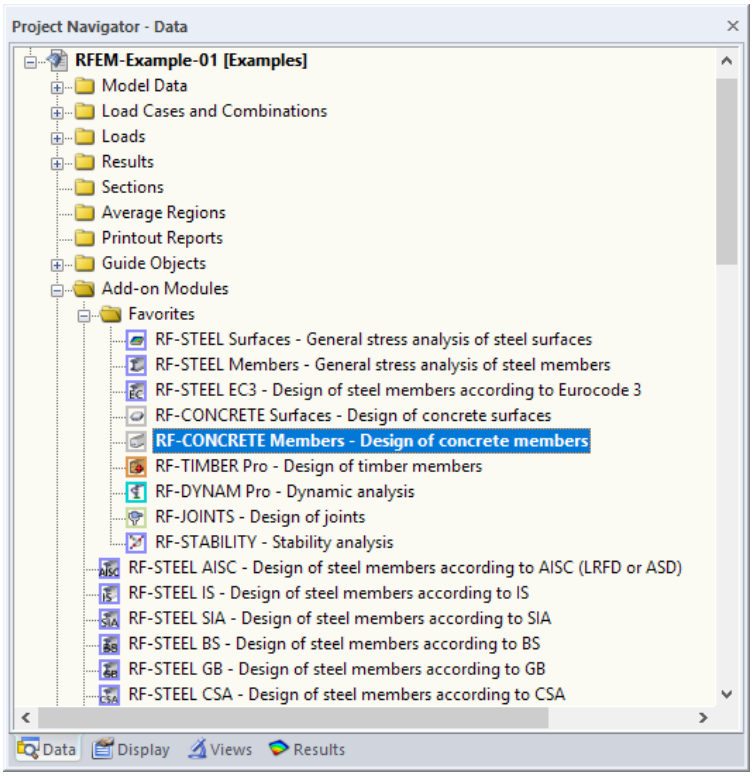
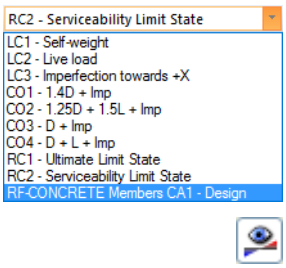
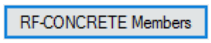


Figure 1.2: *Data* navigator: *Add-on Modules* → *RF-CONCRETE Members*





Panel

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

When the results display is activated, the panel appears. It shows the [RF-CONCRETE Members] button which you can use to access the design module.

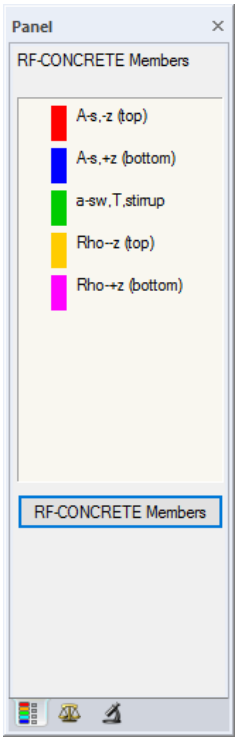


Figure 1.3: Panel button [RF-CONCRETE Members]

2. Input Data

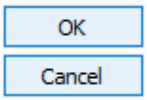
All data required for the definition of design cases is entered in windows. The [Select] function allows for a graphical selection of the objects that you want to design.

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 7.1, page 62).

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.

2.1 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 ultimate limit state and the serviceability limit state by using the respective tabs.

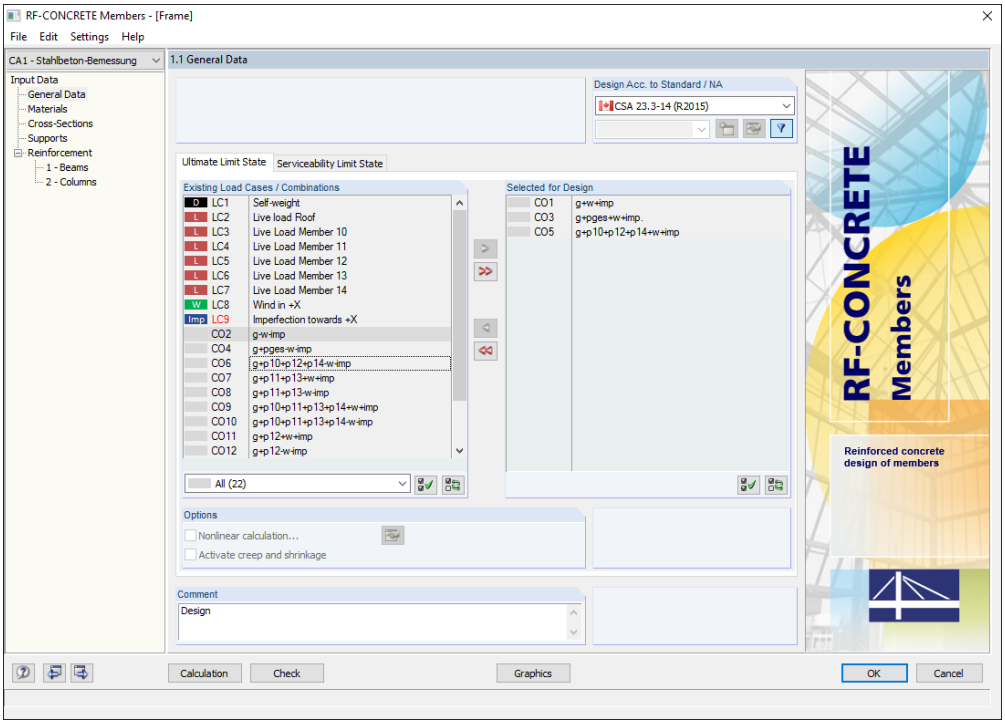


Figure 2.1: WIndow *1.1 General Data*, tab *Ultimate Limit State*

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.

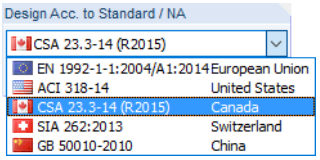


Figure 2.2: Selection of CSA design standard

2.1.1 Ultimate Limit State

The first tab of Window *1.1 General Data* is shown in Figure 2.1 on page 7.

Existing Load Cases / Combinations

This dialog 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 LC9 in Figure 2.1, 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 items 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.

2.1.2 Serviceability Limit State

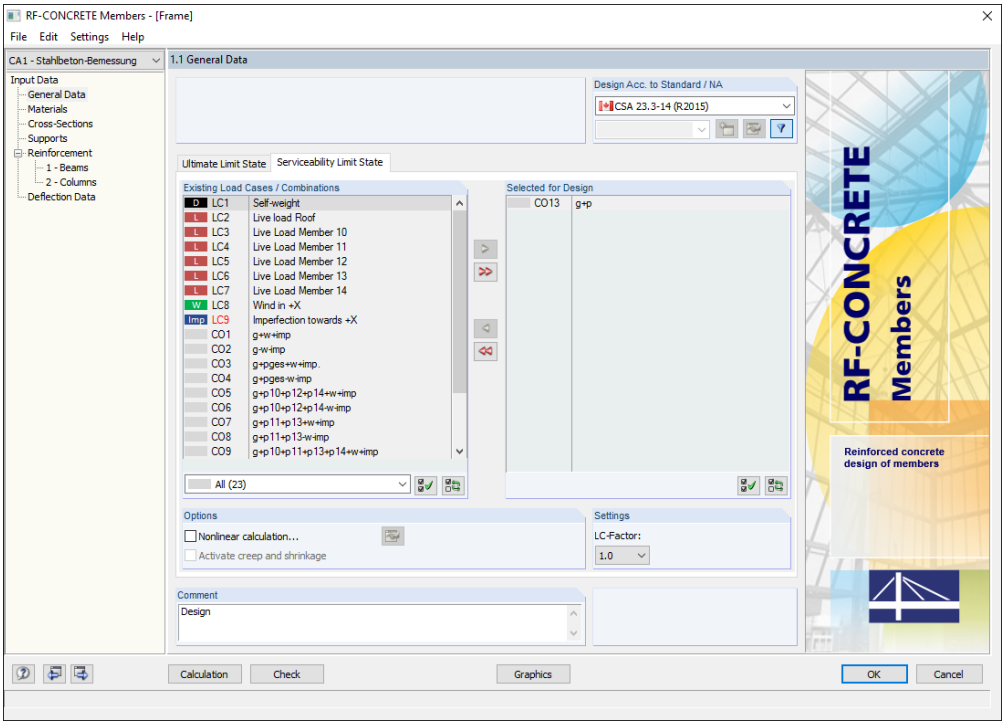


Figure 2.3: WIndow *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases / Combinations

This dialog section list 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.

2.2 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 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 in 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 7.6, page 65).

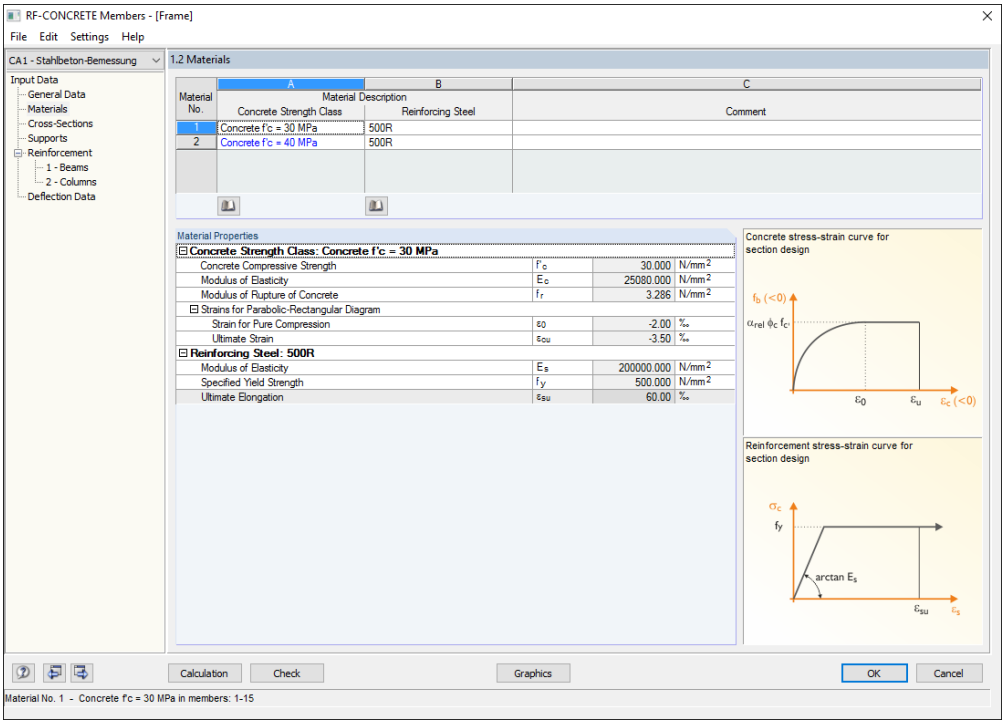
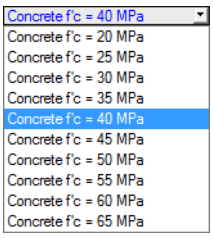


Figure 2.4: Window *1.2 Materials*

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.

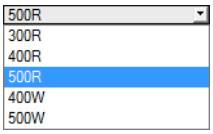
It is possible to select a different material by using the list: Place the pointer in a table row of column A, and then click the button [▼] or use the function key [F7]. The list shown on the left opens. After the selection of the new class, the material properties are updated.

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 different 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. After the selection of the new grade, the material properties are 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 columns A and B.

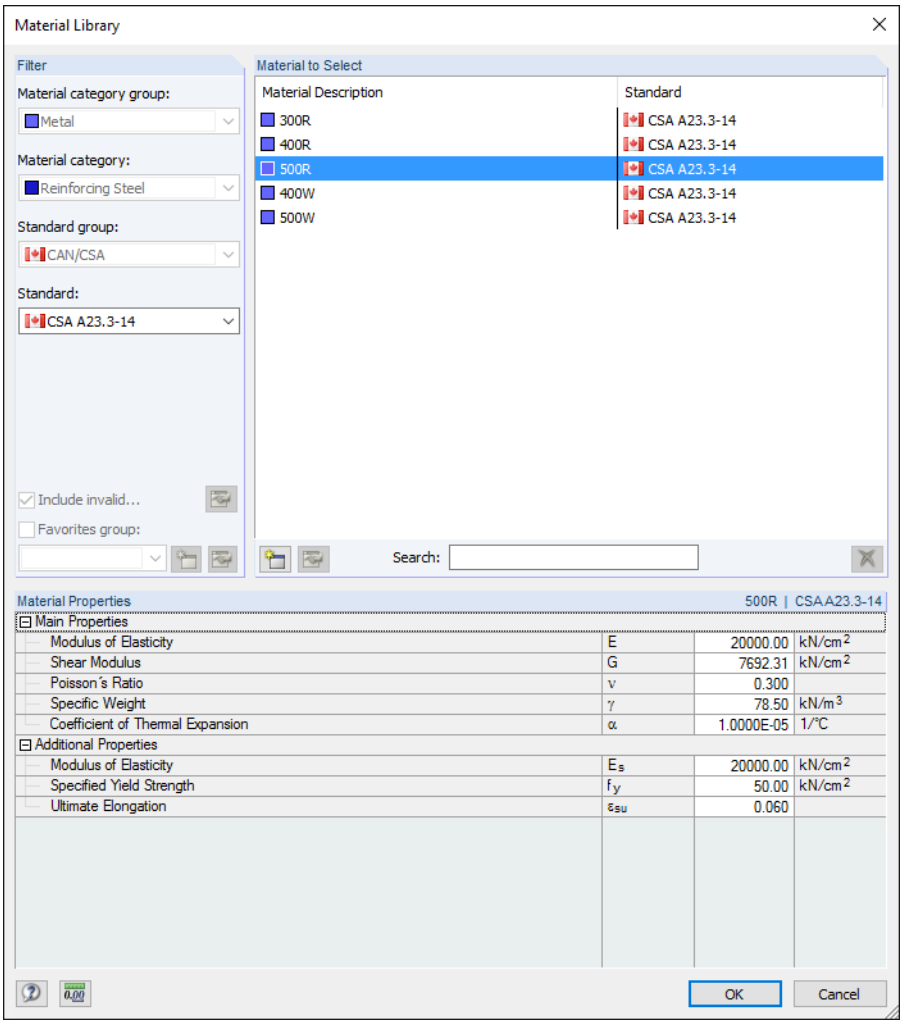


Figure 2.5: Dialog box *Material Library*

The materials 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.

2.3 Cross-Sections

This window lists the design relevant cross-sections.

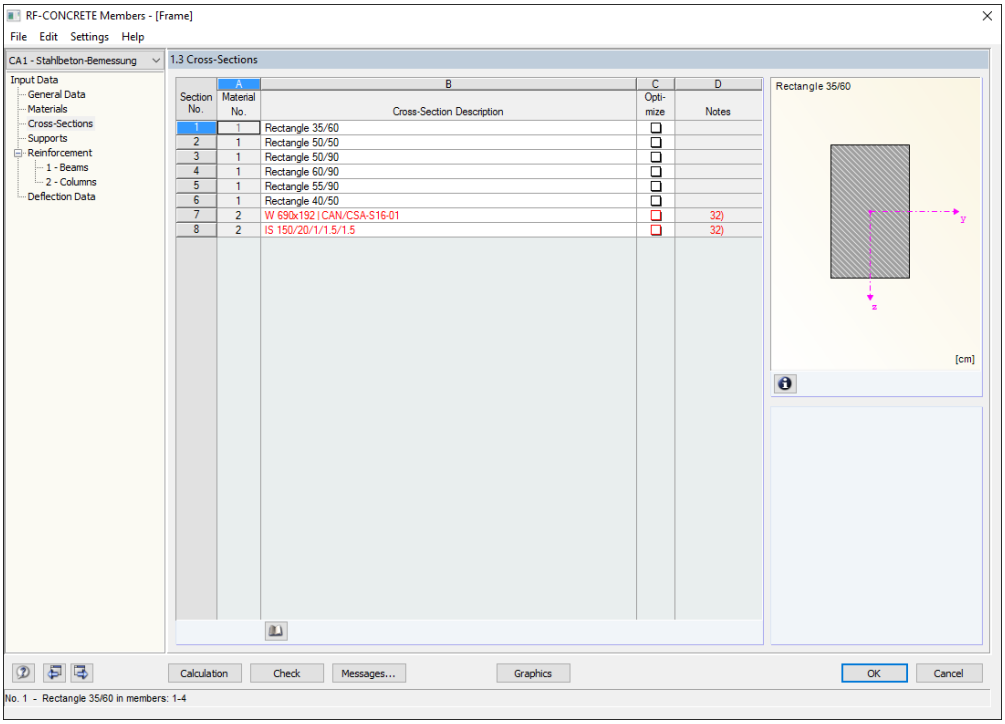


Figure 2.6: Window *1.3 Cross-Sections*

Cross-Section Description

When you open this window, all cross-sections used in RFEM are preset with the assigned material numbers.

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

To change a cross-section, enter its 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 2.7):

- 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.

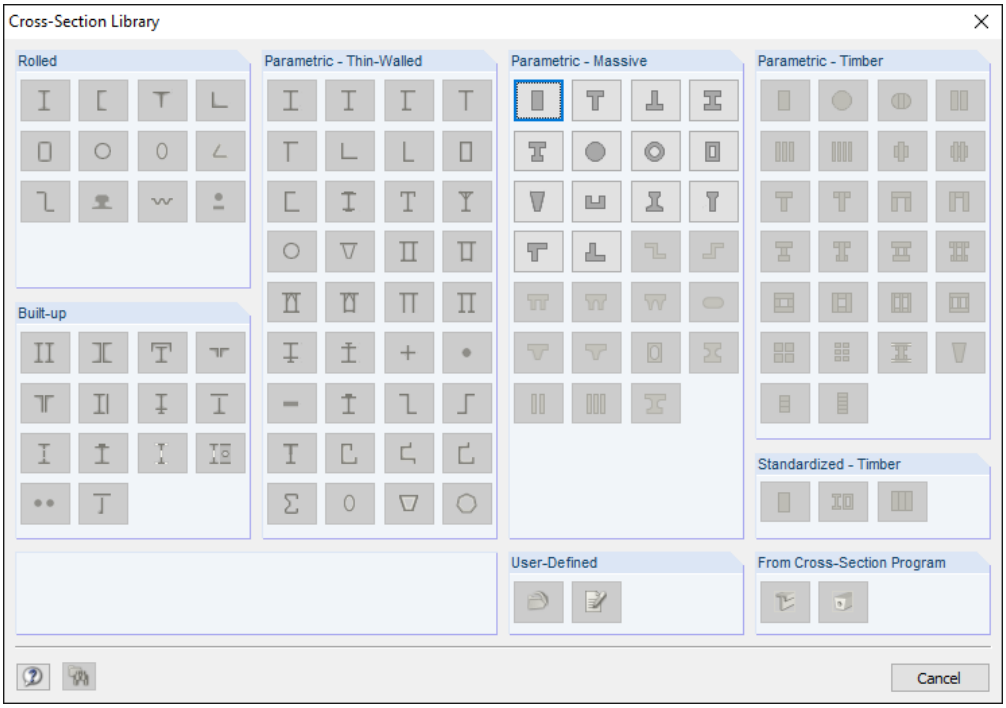
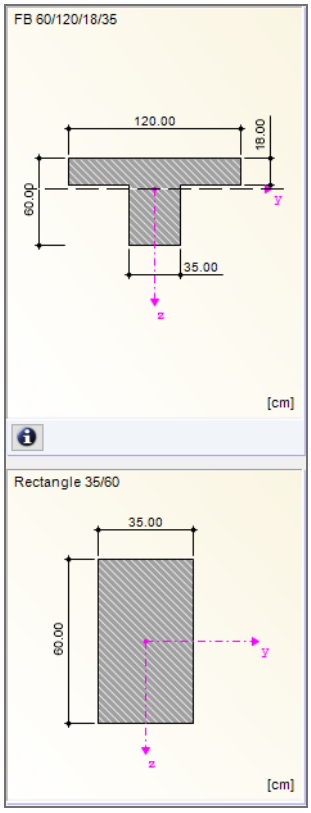


Figure 2.7: Cross-section Library



If the cross-sections in RF-CONCRETE Members are different from the ones used in RFEM, both cross-sections are displayed in the 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 7.5, page 64).

To optimize a specific cross-section, select its check box in column C. Recommendations on the optimization can be found in Chapter 7.2 on page 64.

Notes

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

2.4 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.

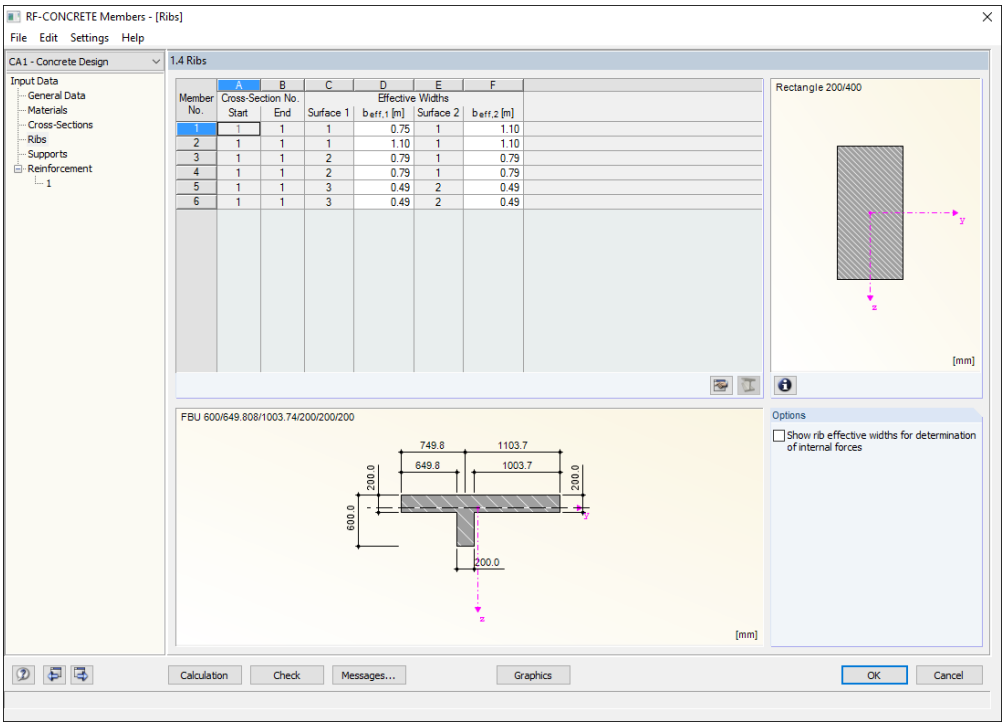


Figure 2.8: Window *1.4 Ribs*

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.

These columns show the numbers of the members that have been defined as member type *Rib* in RFEM.

Cross-Section No. Start/End

Column A and B give the cross-section numbers (see Chapter 2.3). 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 2.9).

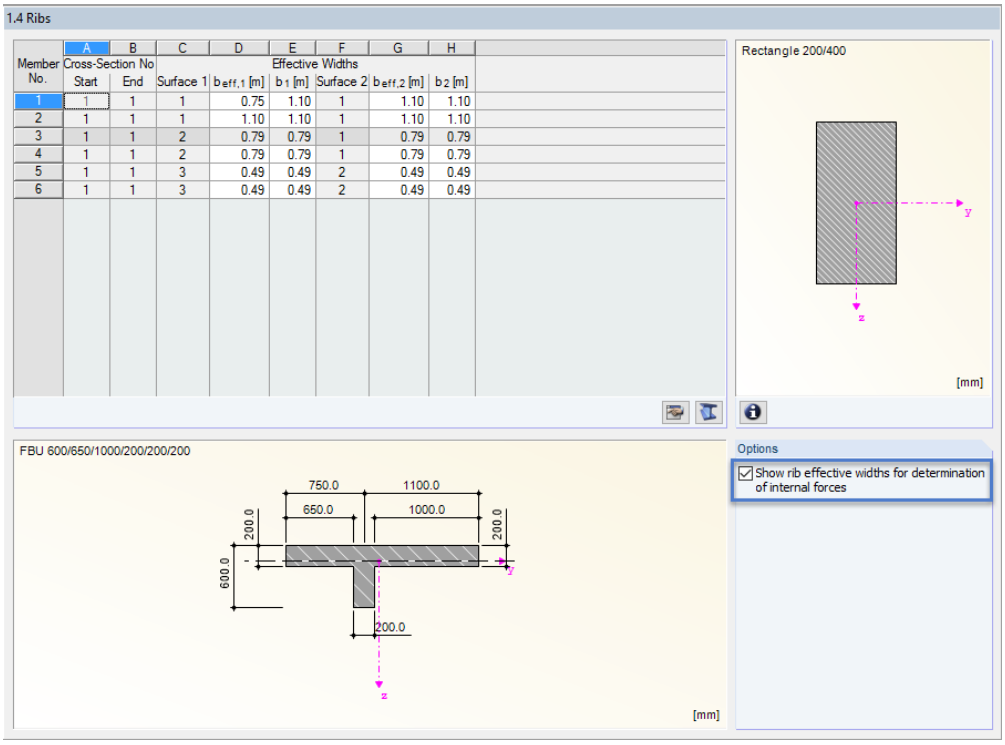


Figure 2.9: Window *1.4 Ribs*

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 *Solid Cross-Sections - Unsymmetric Floor Beam* dialog box with parameters of equivalent cross-section
	Edit Rib	Opens *Edit Rib* 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 2.1: 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).

2.5 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. not columns!

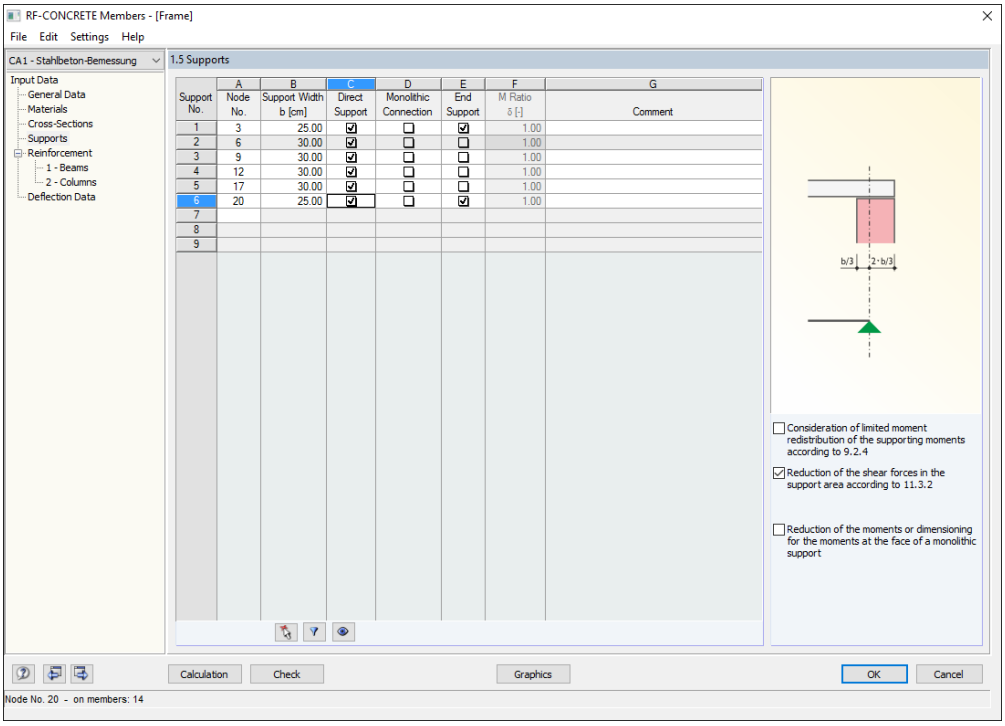


Figure 2.10: Window *1.5 Supports*

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.

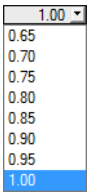
Monolithic Connection

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



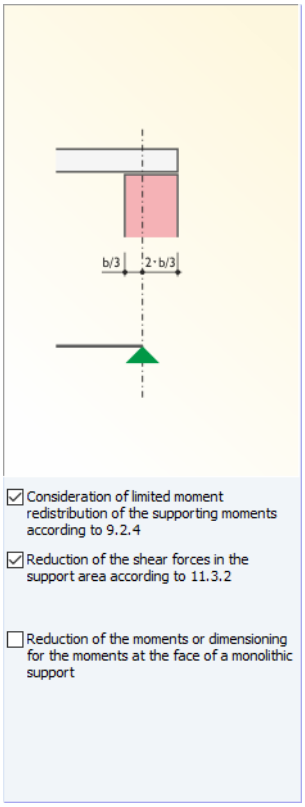
For continuous structural components, you can define the ratio δ for the redistributed moment and the elastically determined initial moment. Column F is only available if you have selected the option *Consideration of a limited moment redistribution of the supporting moments according to 9.2.4* below the graphic on the right.

The δ-values can be determined according to CSA A23.3-14, 9.2.4.

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 three 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 CSA A23.3-14, 9.2.4.

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

Reduction of shear forces in support area

For direct supports, the design value of the shear force can be reduced (cf. CSA A23.3-14, 11.3.2).

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 Standard for a "uniformly distributed load" by means of envelopes is not verifiable in detail.

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

Optionally, the program reduces the moment when all of the following requirements are met:

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

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

2.6 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 for 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 of the design case.

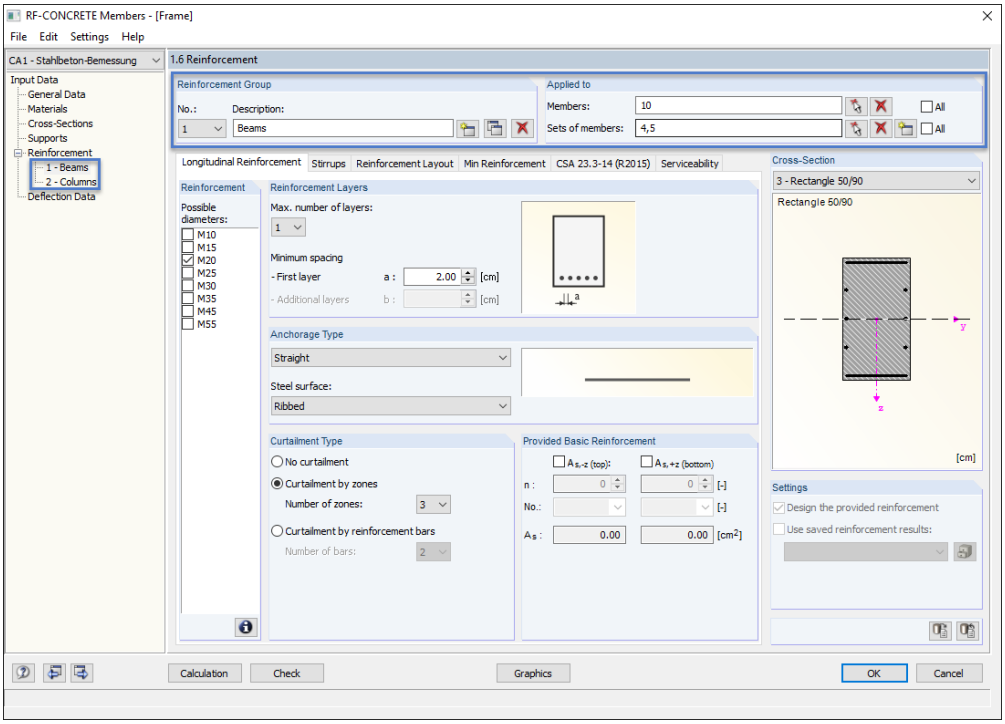
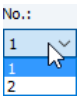


Figure 2.11: Window *1.6 Reinforcement* with two reinforcement groups



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

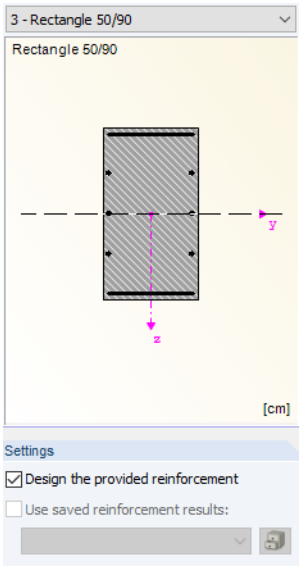
By using the [Delete] button, the currently selected reinforcement group is deleted from the RF-CONCRETE Members case without any additional warning. Members and sets of members that were contained in such a reinforcement group will not be designed. You must reassign them to a new or an existing reinforcement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

In the dialog section *Applied to*, you specify 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 affects 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.

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.

The individual tabs of Window 1.6 are described in the following chapters.

2.6.1 Longitudinal Reinforcement

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

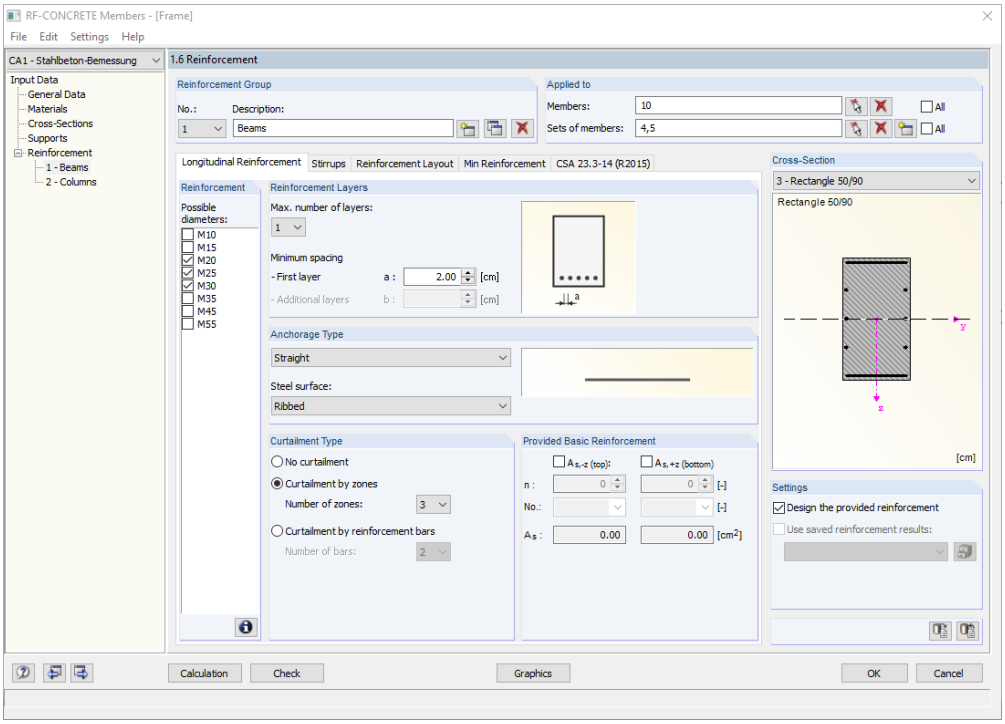


Figure 2.12: Window *1.6 Reinforcement*, tab *Longitudinal Reinforcement*

Reinforcement

The list of *Possible diameters* includes the nominal diameters for rebars according to CSA G30.18. 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 2.13).

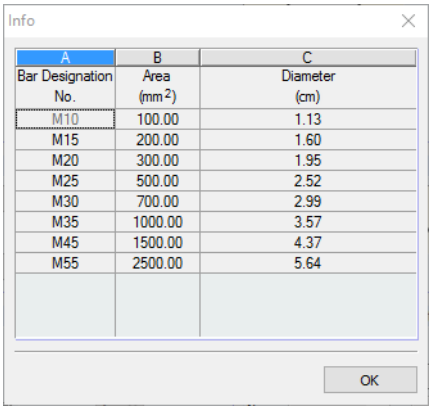
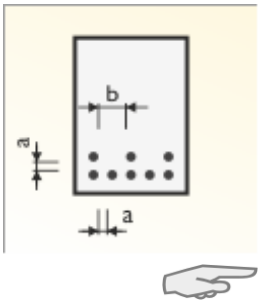


Figure 2.13: *Info* box of rebar properties

Reinforcement Layers

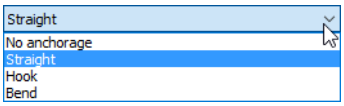


For the reinforcement proposal, the program also takes into account multi-layer arrangements of rebars. Use the list to specify the *Max. Number of Layers*. It is possible to define up to ten reinforcement layers. The boxes below allow for entering specifications concerning the *Minimum Spacing a* for 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, i.e. modified settings are immediately displayed.

The anchorage type affects the required anchorage length.

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.

2.6.2 Stirrups

This tab manages the specifications concerning the shear reinforcement.

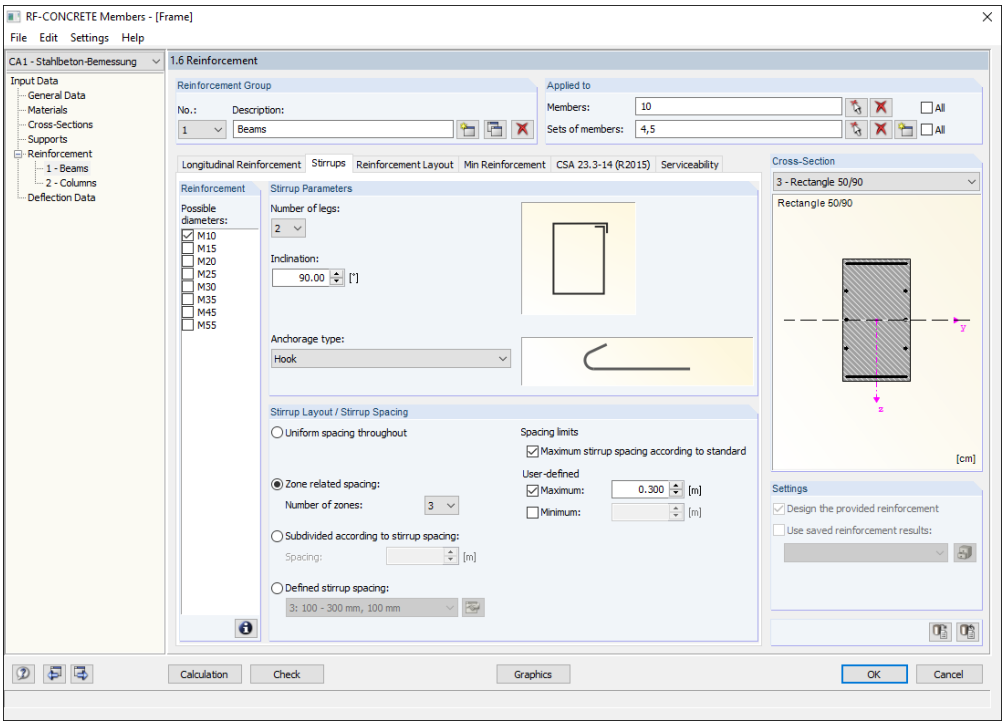


Figure 2.14: Window *1.6 Reinforcement*, tab *Stirrups*

Reinforcement

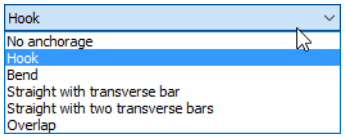
The list of *Possible diameters* includes the nominal diameters for rebars according to CSA G30.18. A multiple selection is no problem for the design.

Use the [Info] button to view the areas and diameters associated with the bar designation numbers (see Figure 2.13).

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 standard specifications for nonprestressed members: CSA A23.3-14, 11.2.4 only allows angles between 45° and 90°.



The *Anchorage type* list offers several possibilities for stirrup anchorages which affect the anchorage lengths. The graphic to the right is dynamic, i.e. the modified settings are displayed immediately.

Stirrup Layout / Stirrup Spacing

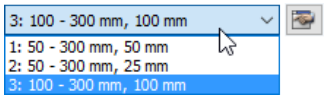
This dialog section is accessible when a reinforcement proposal is created.

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.

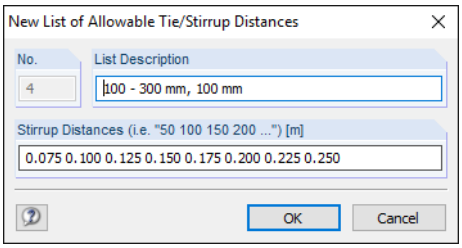


Figure 2.15: Dialog box *New List of Allowable Tie/Stirrup Distances*

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 4.2.2, page 43).

2.6.3 Reinforcement Layout

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

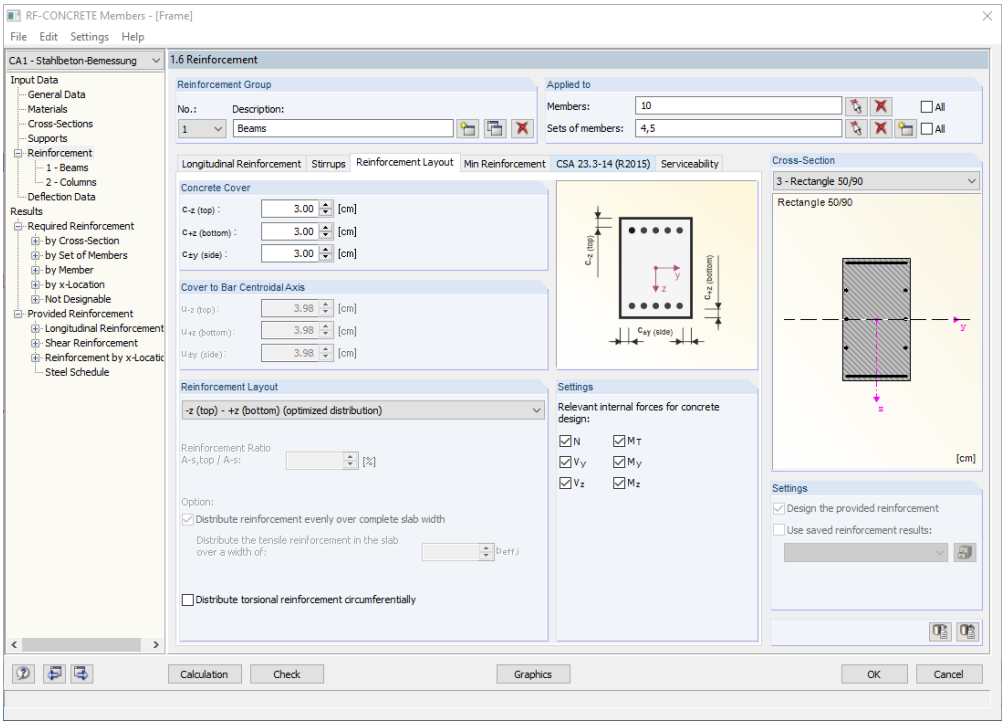


Figure 2.16: Window *1.6 Reinforcement*, tab *Reinforcement Layout*

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, however, these values refer to the dimensions of the centroidal axes *u* of the rebars. Figure 2.17 explains the difference.

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

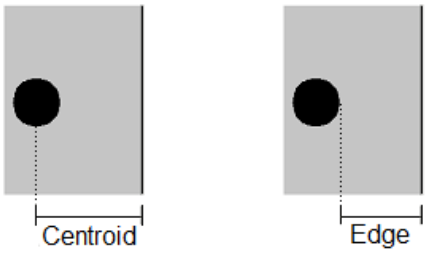
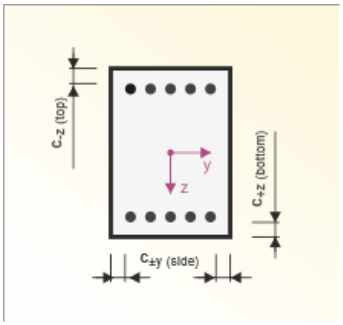


Figure 2.17: Relations of concrete cover

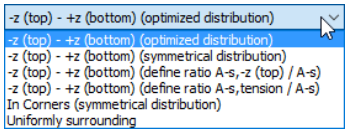


In the $c_{-z\,(top)}$ box, enter the concrete cover of the longitudinal reinforcement at the top side of the member. The $c_{+z\,(bottom)}$ box accordingly defines the cover of the longitudinal reinforcement at the bottom. Those values represent the values c_c of the clear concrete cover. On the basis 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

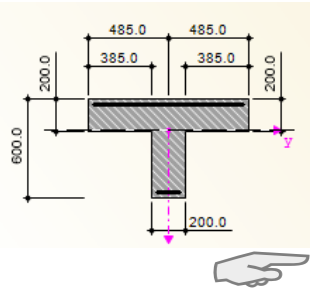


The list contains several options how to lay out the reinforcement in the cross-section:

- -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 graphics 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.

2.6.4 Min Reinforcement

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

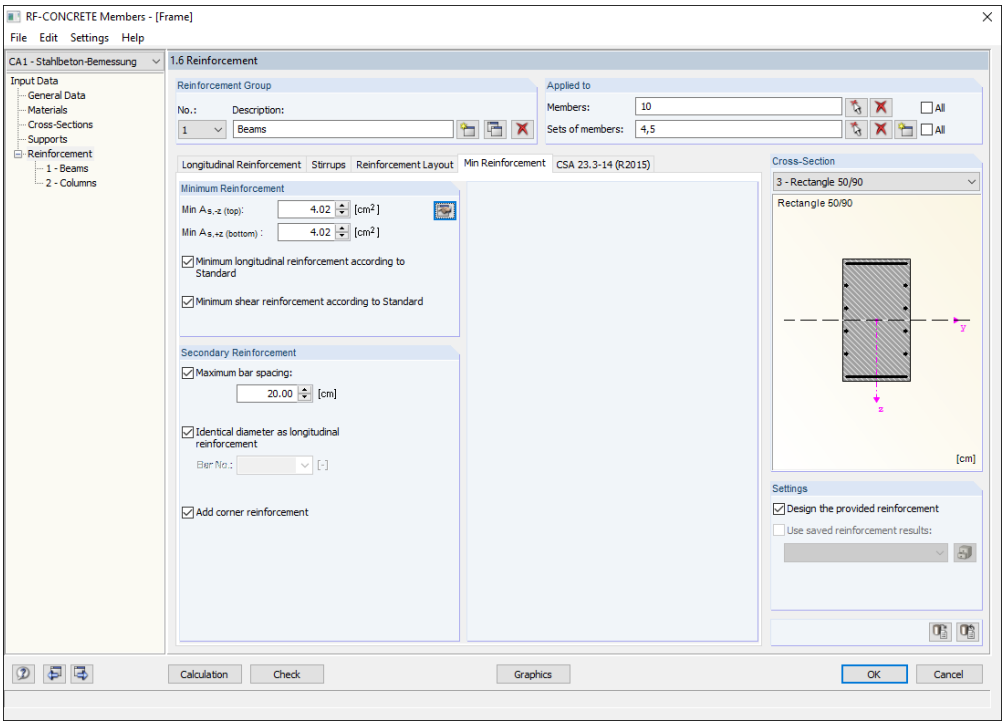


Figure 2.18: Window *1.6 Reinforcement*, tab *Min Reinforcement*

Minimum Reinforcement

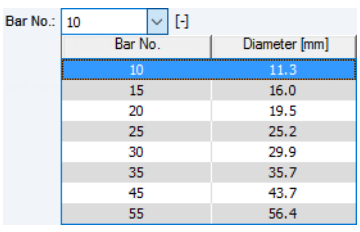
You can define a basic minimum reinforcement in the two text boxes. Enter the steel areas for *Min* $A_{s,-z\,(top)}$ and *Min* $A_{s,+z\,(bottom)}$. Via the [Edit] button, you can define the cross-sectional areas in a separate dialog box from the number of rebars and the rebar diameters.

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 Standard.

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.

2.6.5 CSA 23.3-14(R2015)

The fifth tab manages the settings of the Standard that has been selected in Window *1.1 General Data* (see Figure 2.2, page 8). It contains the standard-specific settings concerning the reinforcement. In the following, the specifications of CSA A23.3-14 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 Standard.

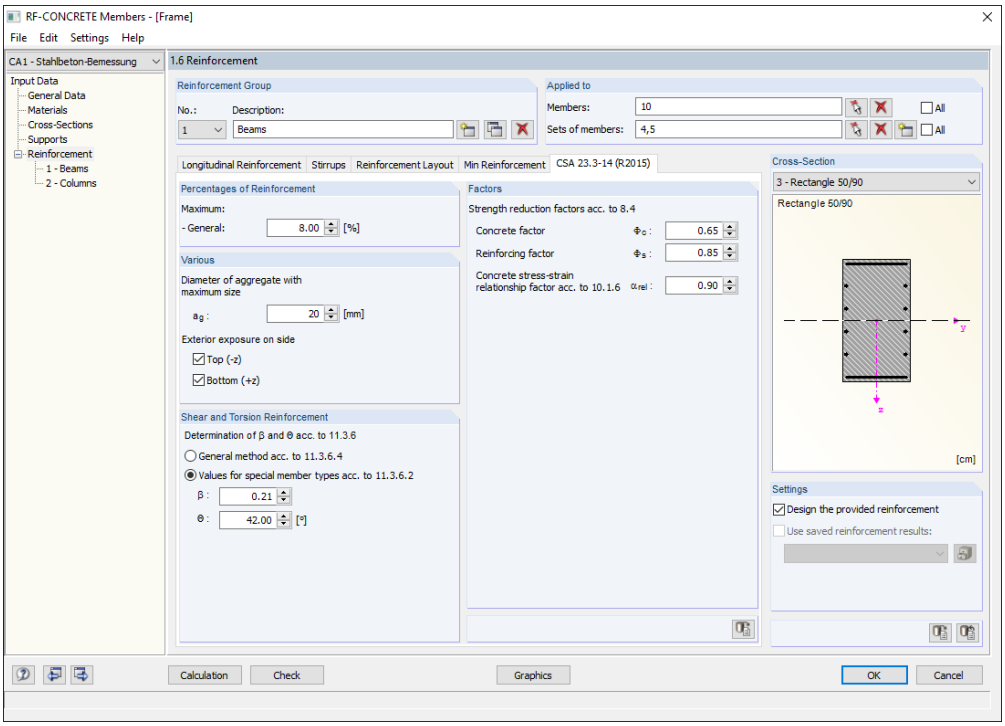


Figure 2.19: Window *1.6 Reinforcement*, tab *CSA A23.3-14 (R2015)*

Percentages of Reinforcement

The entry in this text box defines the general maximum reinforcement ratio for beams.

Various

The *Diameter of aggregate* influences a number of values, such as spacing of reinforcement bars, the determination of β for shear resistance, and concrete cover. Enter the maximum size in the text box.

The check of the *Exterior exposure* affects the reinforcement ratio of the skin reinforcement for beams with an overall depth exceeding 750 mm and the quantity z according to Equation 10.6 (CSA A23.3-14). The value z shall not exceed 30 kN/mm for interior and 25 kN/mm for exterior exposure corresponding to a crack width of 0.4 mm and 0.33 mm respectively.

Shear and Torsion Reinforcement

With the option *General method acc. to 11.3.6.4*, the program computes the shear strength provided by the concrete and the reinforcement using these equations to determine *β* and *θ*:

$$\beta = \frac{0.40}{(1+1500\varepsilon_x)} \cdot \frac{1300}{(1000+s_{ze})}$$ CSA Code, eq. (11.11)

$$\theta = 29 + 7000\varepsilon_x$$ CSA Code, eq. (11.12)

using the longitudinal strain at mid-depth of the cross-section computed from

$$\varepsilon_x = \frac{M_f/d_v + V_f - V_p + 0.5N_f - A_p f_{po}}{2(E_s A_s + E_c A_c)}$$

CSA Code, eq. (11.13)

Otherwise the values for special member types according to 11.3.6.2 or values using the simplified method in CSA A23.3-14, 11.3.6.3 can be used for the calculation of the shear strength.

Factors

The three boxes define the *Strength reduction factors* of the concrete compressive strength, ϕ_c, of the tensile strength of steel, ϕ_s, and of the concrete stress-strain relationship factor, α_{rel}.

The values according to CSA A23.3-14, 8.4 and 10.1.6 are preset.

2.6.6 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*.

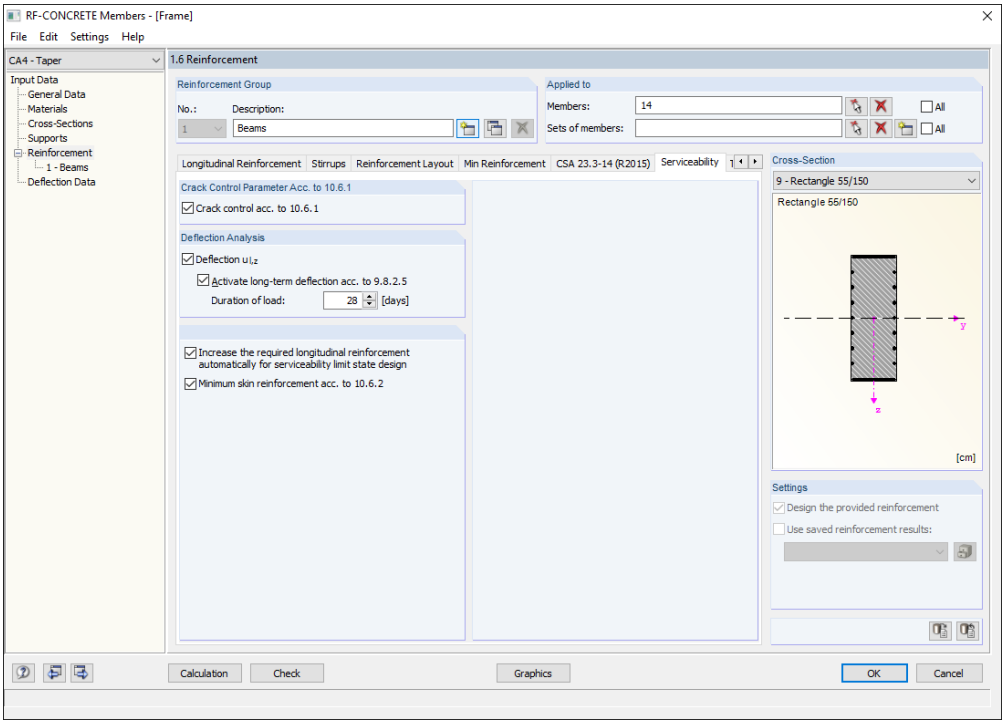


Figure 2.20: Window *1.6 Reinforcement*, tab *Serviceability*

Crack Control Parameter Acc. to 10.6.1

This option defines whether the crack control parameter according to CSA A23.3-14, 10.6.1 has to be taken into account. The check box is selected by default.

The crack control parameter *z* is intended to provide a distribution of flexural reinforcement providing reasonable control of flexural cracking at specified loads. Limiting *z* to 30 kN/mm for interior or 25 kN/mm for exterior exposure corresponds to limiting crack widths to about 0.4 mm and 0.33 mm.

Deflection Analysis

This dialog section can be used for the consideration of 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 CSA A23.3-14, 9.8.2.5 using a time dependent factor, ζ_s, with which the immediate deflections have to be multiplied.

Additional options

These options include the automatic increase of reinforcement for the serviceability limit state design and the application of a minimum skin reinforcement acc. to 10.6.2 for members exceeding a height of 750 mm.

2.6.7 Tapered

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

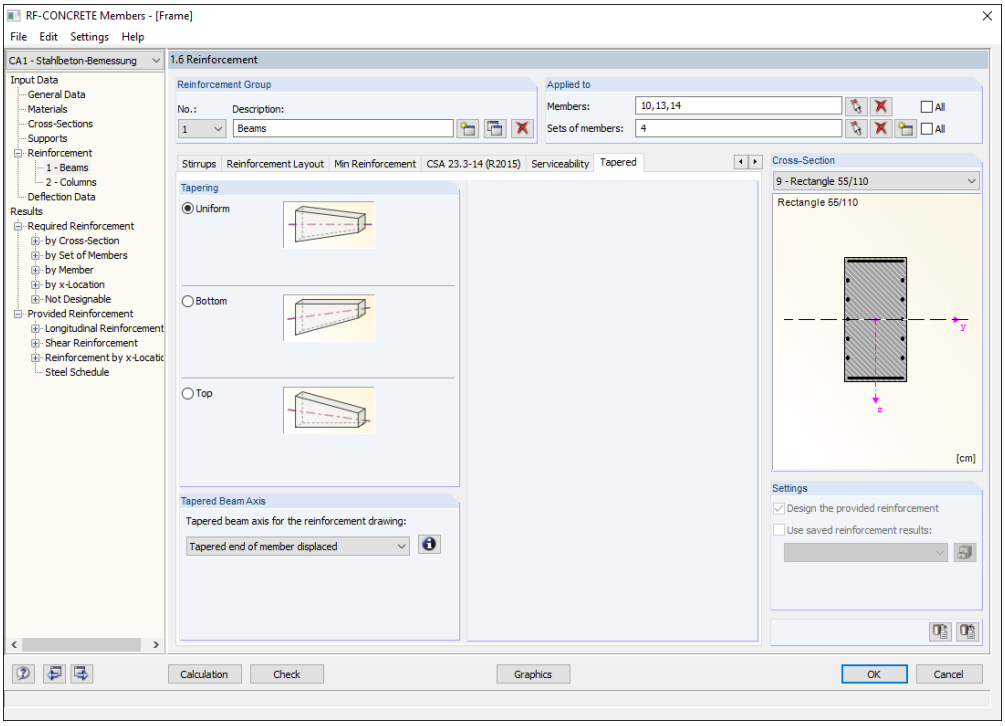


Figure 2.21: Window *1.6 Reinforcement*, tab *Tapered*

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 the intermediate values, and RFEM displays an error message before the calculation is started.

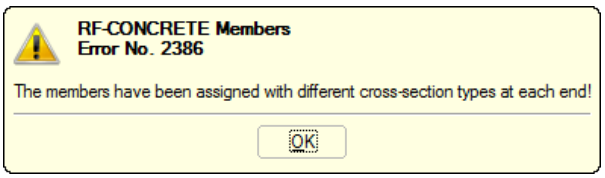
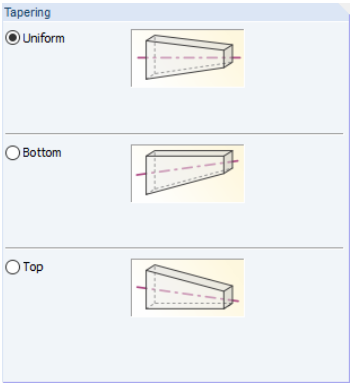


Figure 2.22: Error message for incompatible tapered cross-sections

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.

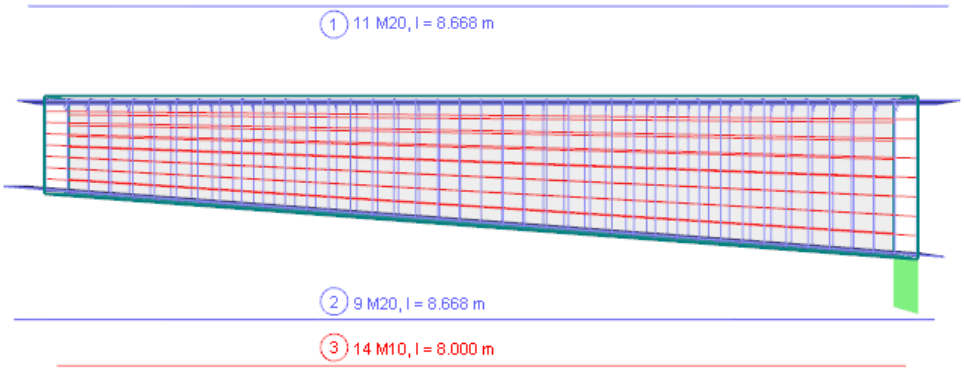
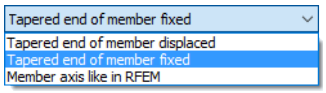


Figure 2.23: Taper with inclined bottom side

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 centrically 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 been already done in RFEM by a member eccentricity.

Additional explanations are available via the [Info] button.

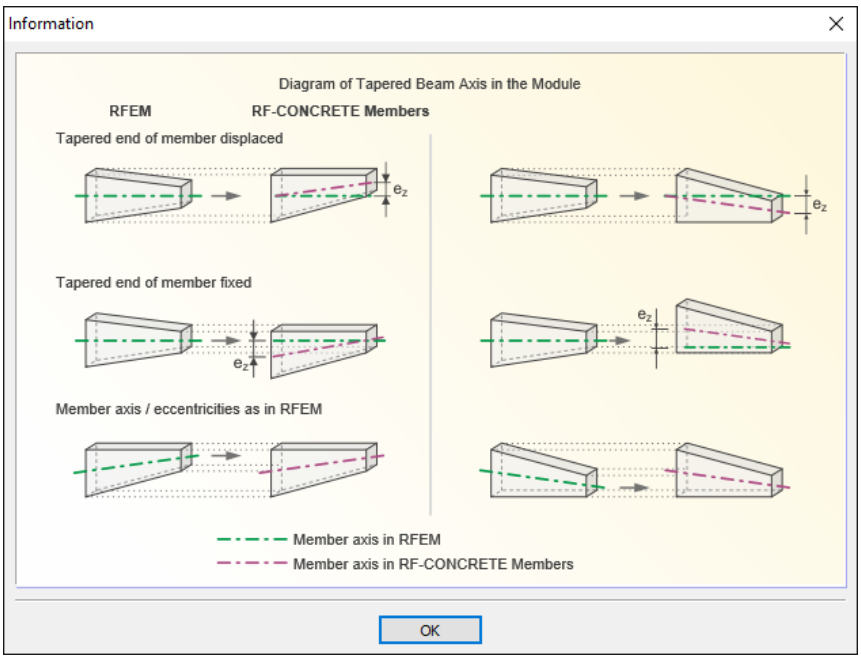


Figure 2.24: Dialog box *Information* for further explanation on Tapered Beam Axis

2.7 Deflection Data

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

1.7 Deflection Data

No.	A Reference to	B Set of members No.	C Reference Length Element	D L [m]	E Precamber w_0 [cm]	F Limit value L / [-]	G $u_{z,max}$ [cm]	H Comment
1	Member	1	Member	6.000	1.00	240	2.50	
2	Member	2	Distance between support	4.400	0.00	240	1.83	
3	Set of members	4	User-defined	16.000	0.00	240	6.67	
4	Set of members	5	Set of member	24.000	0.00	240	10.00	
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 2.25: Window *1.7 Deflection Data*

The design criterion for the deflection, $u_{l,z}$, 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 2.5, page 16).

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 [m]	E Precamber w_0 [cm]	F Limit value L / [-]	G $u_{z,max}$ [cm]
1	Set of members	6	Distance between supports	var.	0.00	240	var.
2							
3							
4							
5							

Nodes 22 - 23: L_1= 4.569
Nodes 23 - 24: L_2= 5.431
Nodes 24 - 25: L_3= 7.511

Figure 2.26: 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.

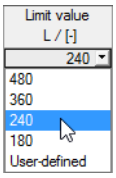
The shape of the precamber can be determined as follows:

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

where

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

Limit value / $u_{z,max}$



The relative limit of the deflection has to be specified in column F. Default is the limit of L/240 as specified in CSA A23.3-14, Table 9.3 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).

3. Calculation

3.1 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 3.1: Result of plausibility check

3.2 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.

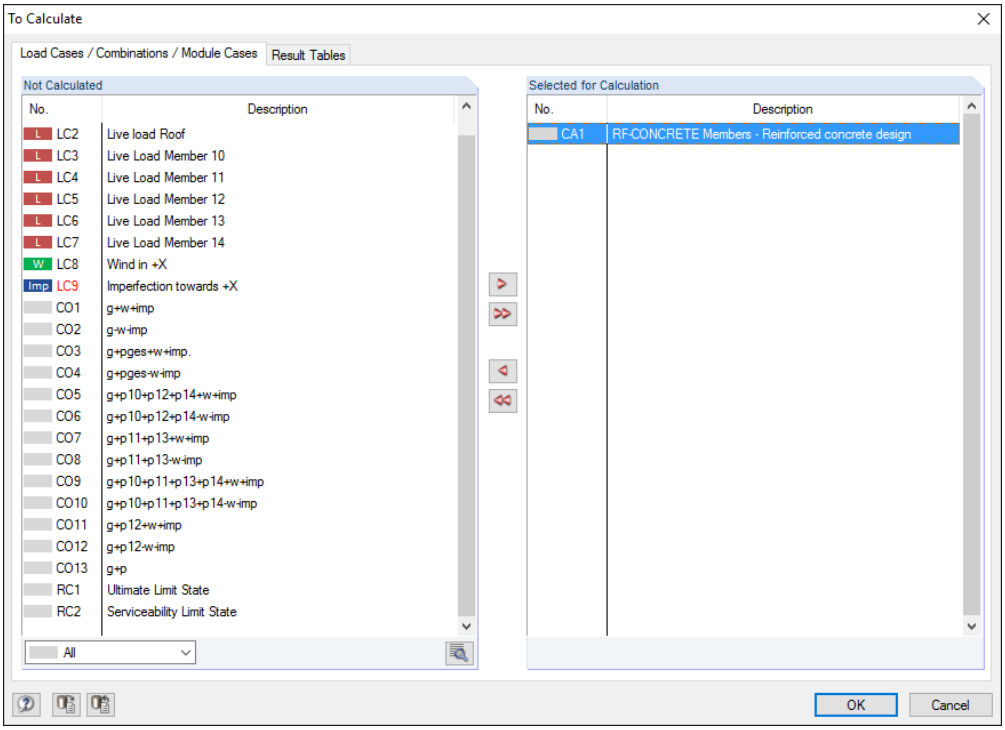


Figure 3.2: Dialog box *To Calculate*

To transfer the selected RF-CONCRETE Members case(s) 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].

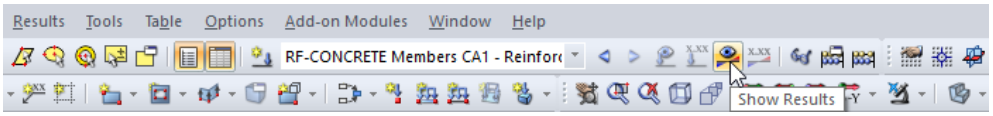


Figure 3.3: Calculating RF-CONCRETE Members design case in RFEM

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

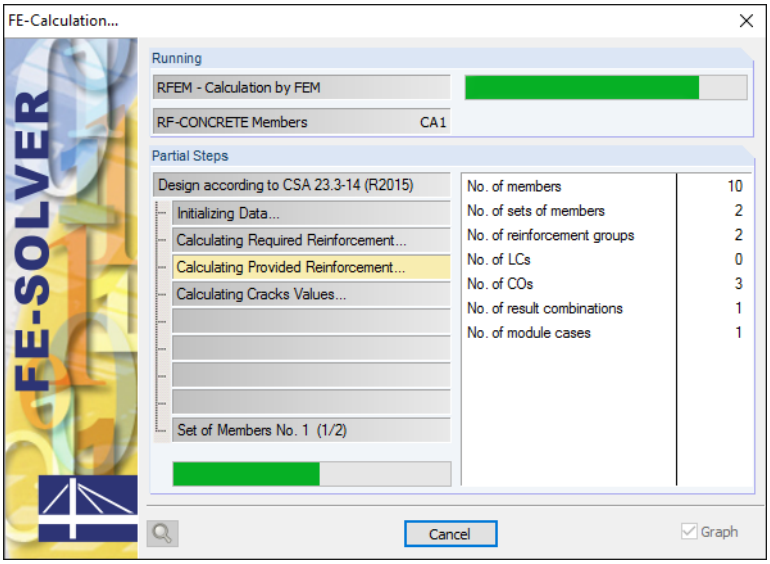


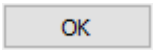
Figure 3.4: Design with RF-CONCRETE Members

4. Results

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

The reinforcement areas required for the ultimate limit state design are listed in the result windows 2.1 to 2.4. If the program created a reinforcement proposal, the provided reinforcement including steel schedule is displayed in the result windows 3.1 to 3.4. 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 explained in Chapter 5 *Results Evaluation*, page 52.

4.1 Required Reinforcement

4.1.1 Required Reinforcement by Cross-Section

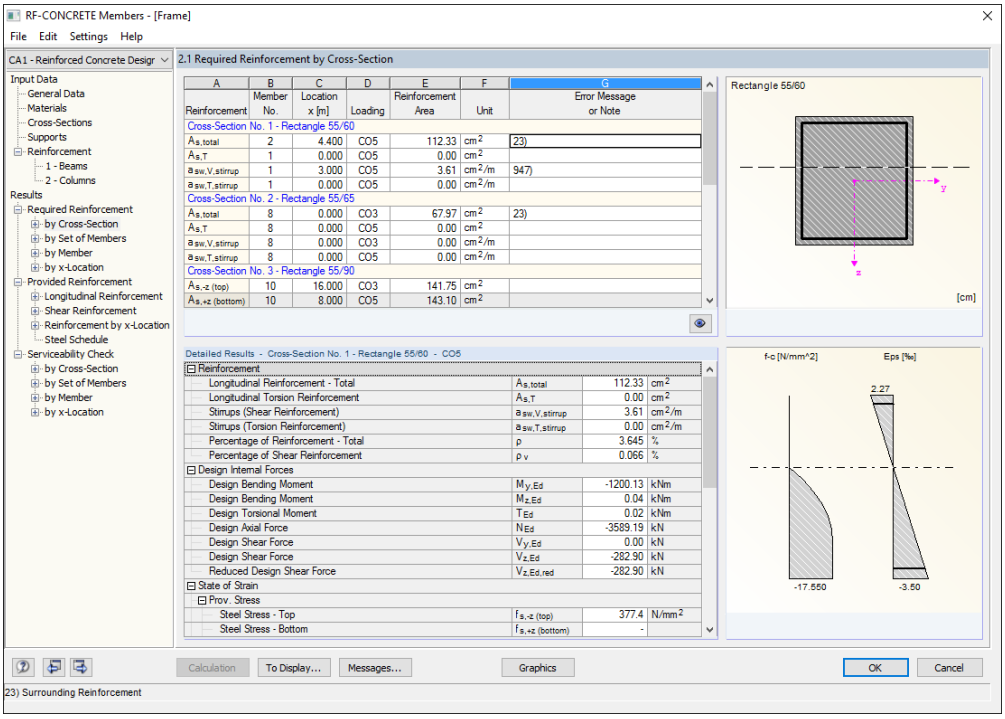


Figure 4.1: Window *2.1 Required Reinforcement by Cross-Section*

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 4.2).

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

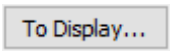
The following types of longitudinal and stirrup reinforcement are preset:

Reinforcement	Explanation
$A_{s,top}$	Reinforcement area of required top longitudinal reinforcement due to bending with or without axial force or axial force only
$A_{s,bottom}$	Reinforcement area of required bottom longitudinal reinforcement due to bending with or without axial force or axial force only
$A_{s,total}$	Reinforcement area of required total longitudinal reinforcement due to bending with or without axial force or axial force only if surrounding reinforcement or reinforcement in corners have been selected
$A_{s,T}$	Reinforcement area of required longitudinal torsion reinforcement, if applicable
$a_{sw,V,stirrup}$	Area of required shear reinforcement for absorption of shear force, referring to standard length of 1 m
$a_{sw,T,stirrup}$	Area of required stirrup reinforcement for absorption of torsional moment, referring to standard length of 1 m

Table 4.1: Longitudinal and stirrup reinforcement

Top and *Bottom* layer

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 in which you can specify the types of reinforcement are to be displayed. These settings also define the types of results appearing in the printout report.

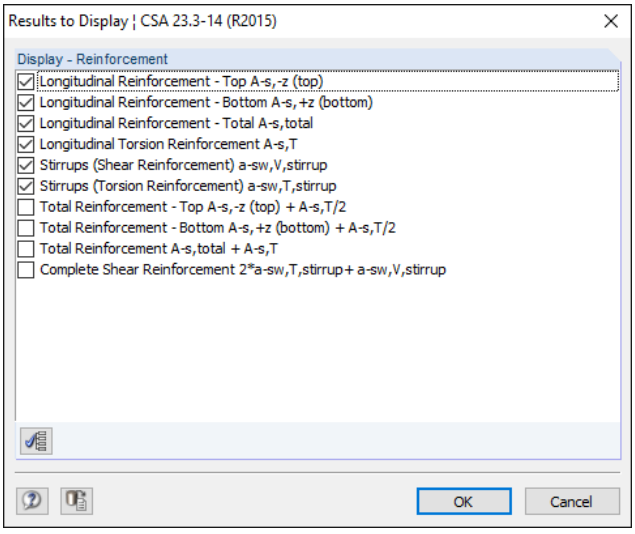


Figure 4.2: Dialog box *Results to Display*

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 ULS 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 7.6 on page 65 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.

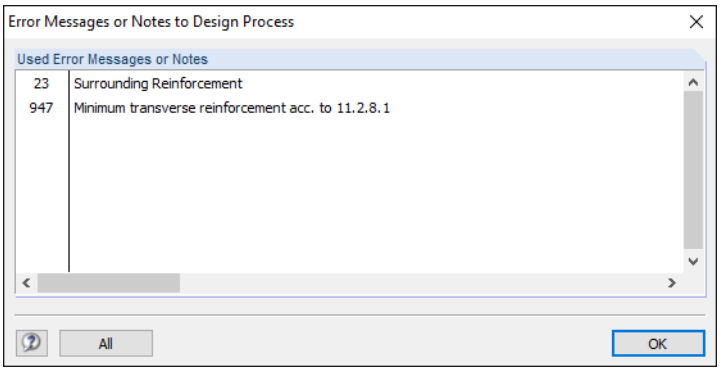


Figure 4.3: Dialog box *Error Messages or Notes to Design Process*

4.1.2 Required Reinforcement by Set of Members

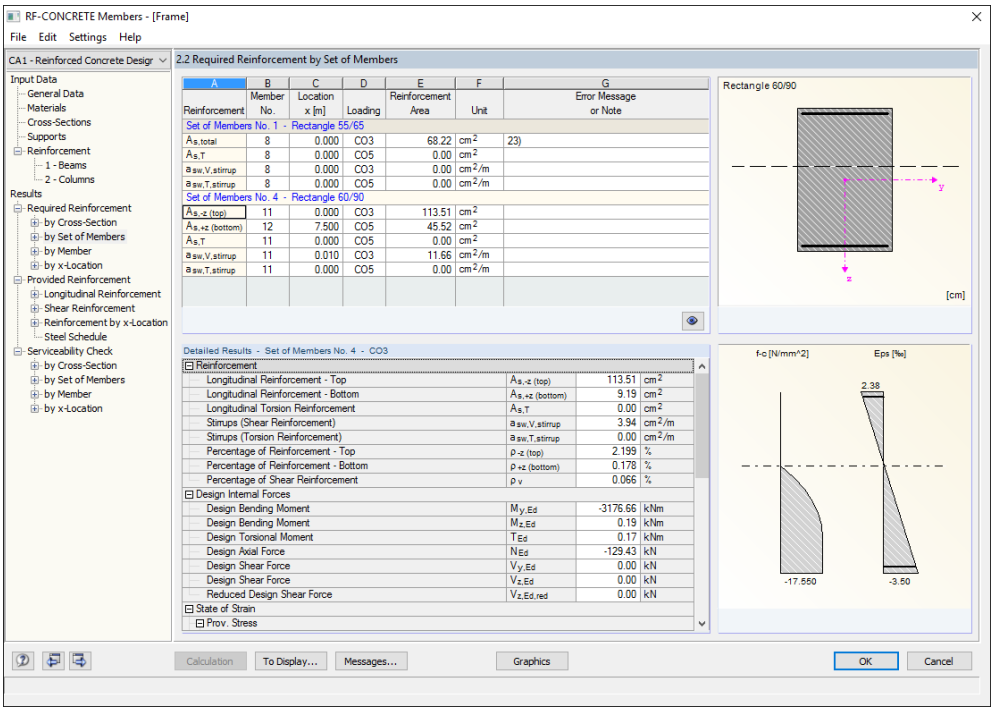


Figure 4.4: Window *2.2 Required Reinforcement by Set of Members*

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 4.1.1.

4.1.3 Required Reinforcement by Member

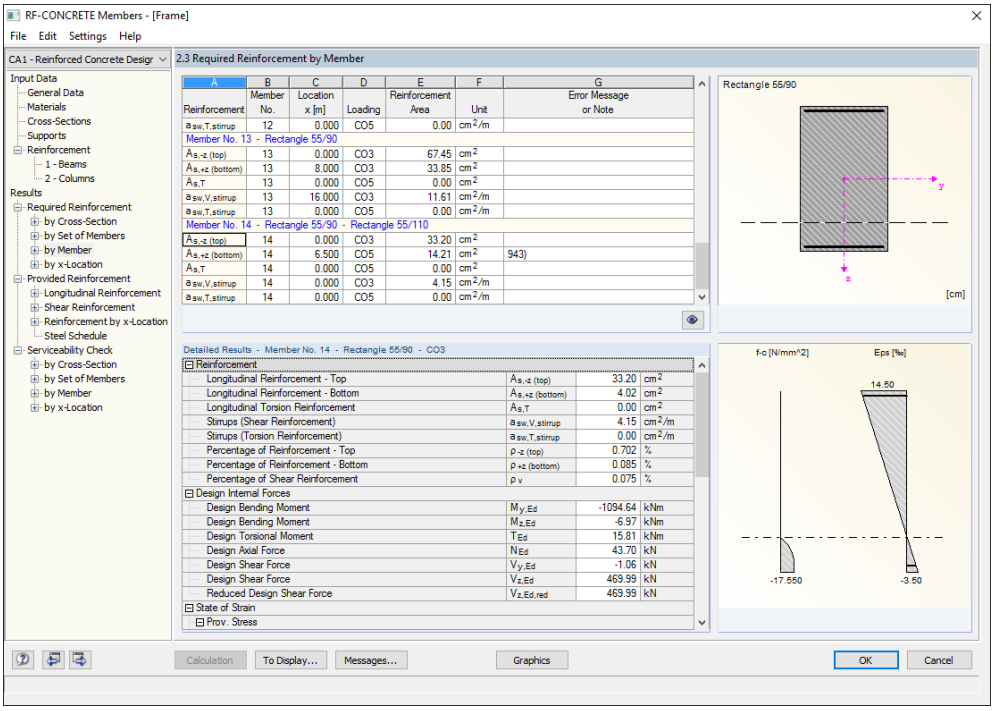


Figure 4.5: Window *2.3 Required Reinforcement by Member*

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.

4.1.4 Required Reinforcement by x-Location

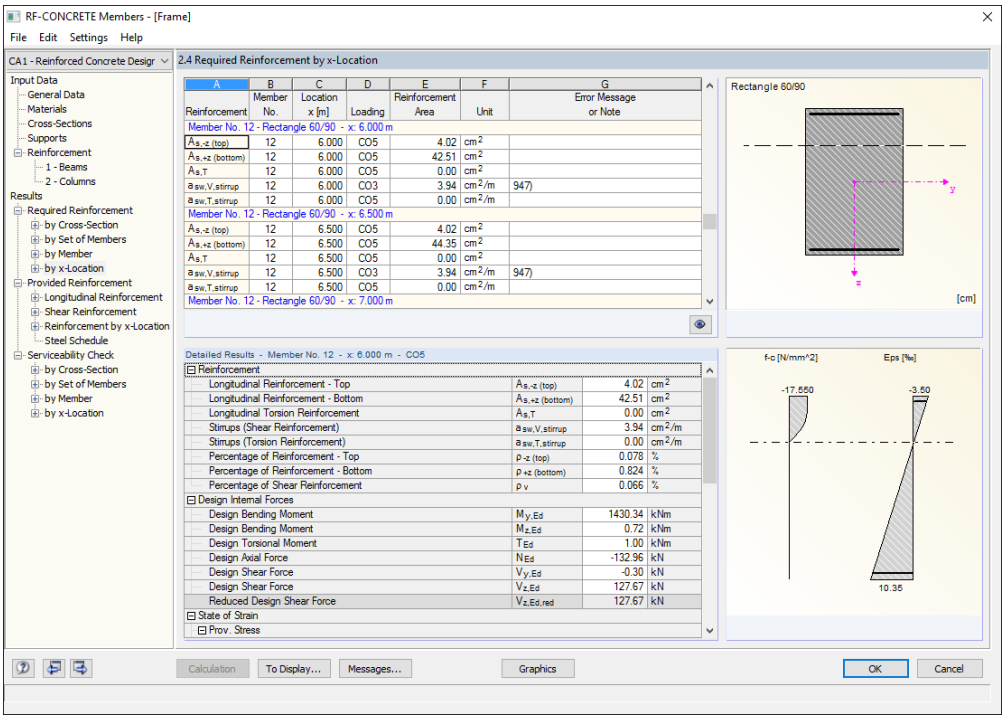


Figure 4.6: Window *2.4 Required Reinforcement by x-Location*

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

- Start and end 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

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 shear reinforcement with the related details of a particular member location (designed location).

The different columns are described in Chapter 4.1.1.

4.1.5 Required Reinforcement Not Designable

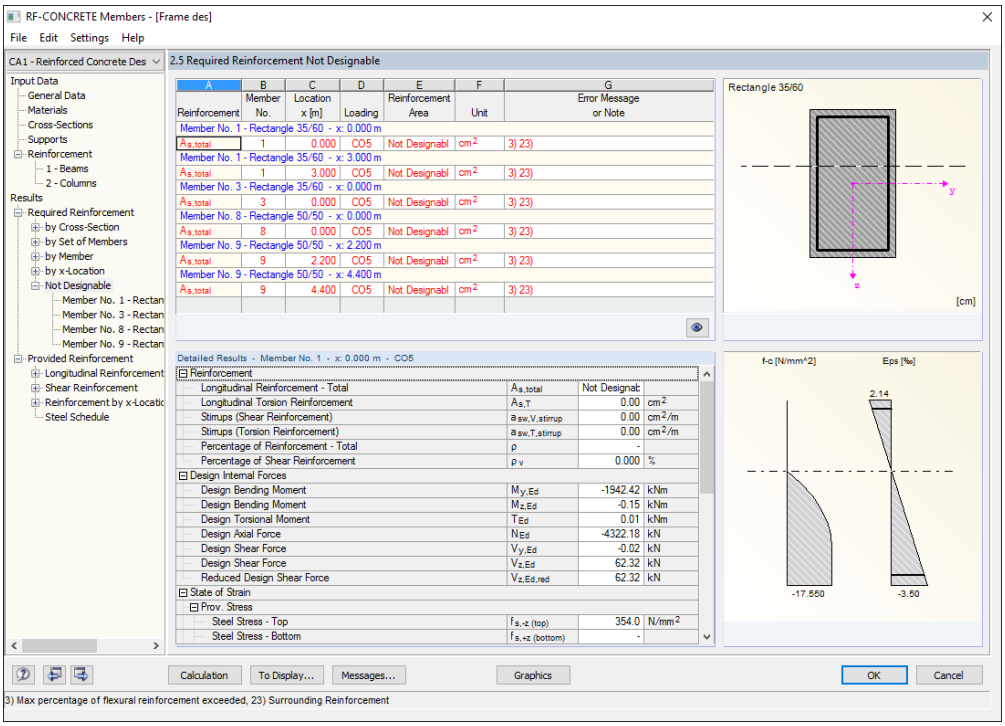


Figure 4.7: Window *2.5 Required Reinforcement Not Designable*

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.

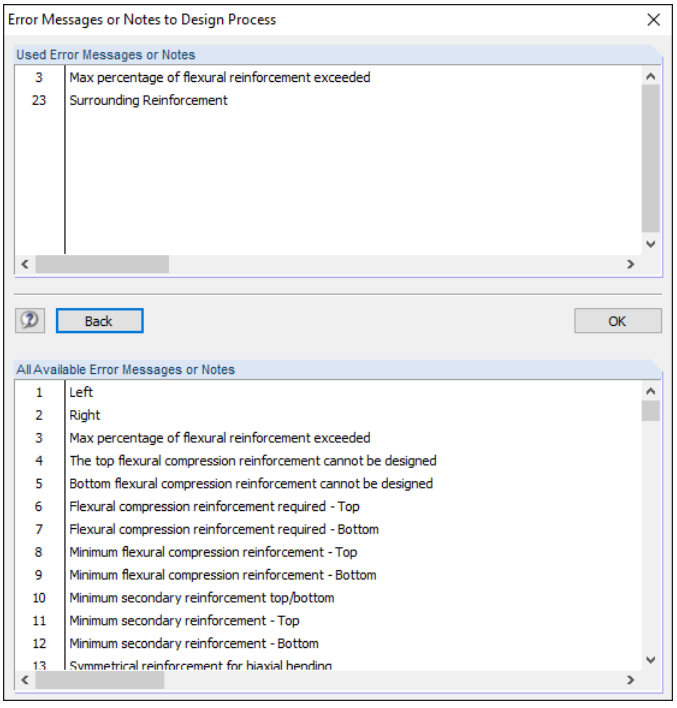
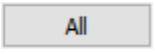


Figure 4.8: Dialog box *Error Messages or Notes to Design Process*



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

4.2 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 19), and if no design problems have been detected (see Chapter 4.1.5, page 38). The serviceability limit state design also requires 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.

4.2.1 Provided Longitudinal Reinforcement

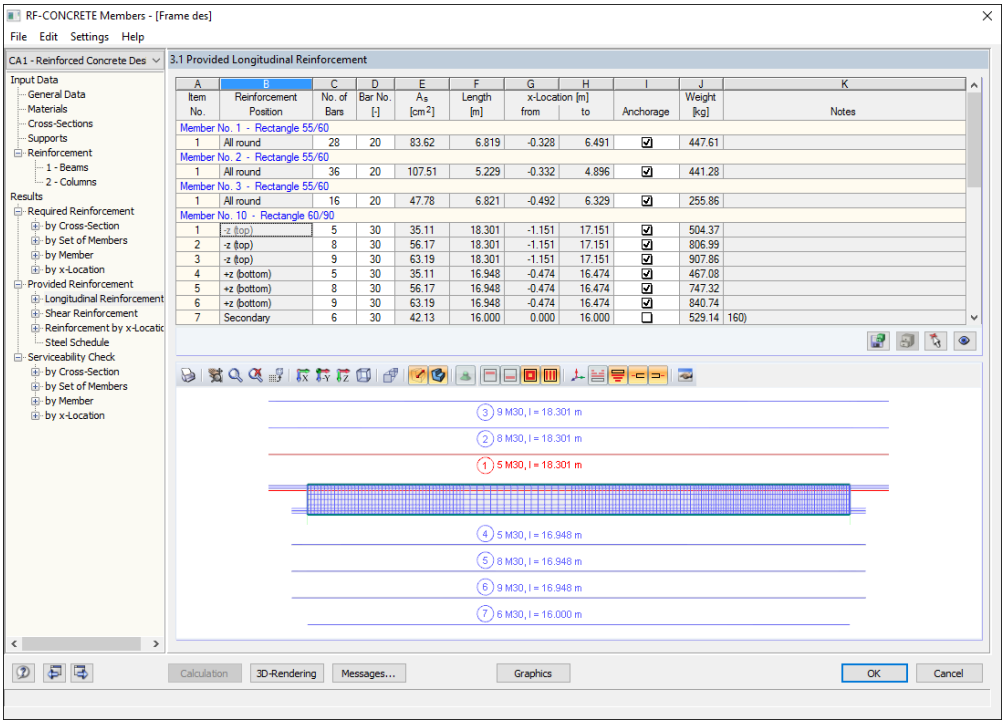


Figure 4.9: Window *3.1 Provided Longitudinal Reinforcement*

The results of 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 from CSA A23.3-14.

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*.

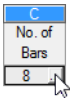
Reinforcement Position

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

- Basic -z (top)
- Basic +z (bottom)
- -z (top)
- +z (bottom)
- Corners
- All round
- Secondary

RF-CONCRETE Members considers the specifications set in Window *1.6 Reinforcement*, tab *Reinforcement Layout* (see Chapter 2.6.3, page 23) 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.

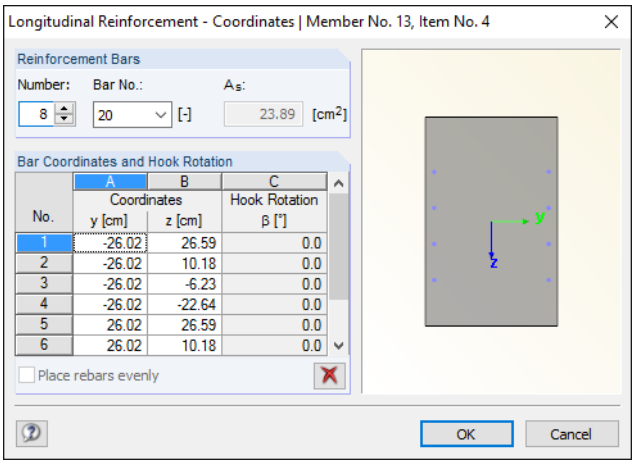
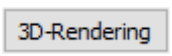


Figure 4.10: Dialog box *Longitudinal Reinforcement - Coordinates*

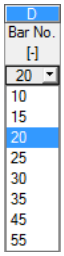
The *Number* of *Reinforcement Bars* can be set manually by using the spin buttons or by directly entering a number. In the section below, you can adjust the position of the rebars in the individual input rows. To delete a row selected in the lower section, click the [Delete] button.

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 β = 90° results in a downward rotation (i.e. in direction z) for the top reinforcement. The angle β = 270° rotates the anchorage end of the bottom reinforcement upwards. For the 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 designation number for the current item number.

A_s

The total reinforcement area of each item is listed in column E. It is derived from the number of bars and their diameters.

Length

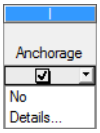
For each item, the column displays the total length of a representative rebar. The entry, composed by the required member length and the lengths of anchorage at both member ends, cannot be edited in this table.

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 4.12, page 42).

Anchorage



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

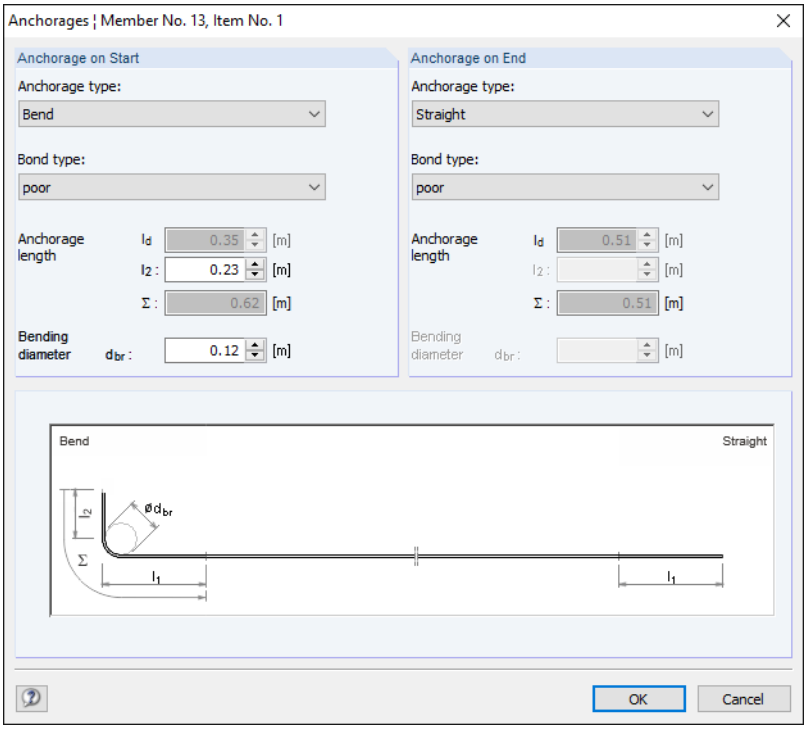
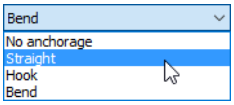


Figure 4.11: Dialog box *Anchorages*

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 2.6.1 on page 20. 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 the reinforcement location factor k_1 according to CSA A23.3-14, 12.2.4 (a).

For control reasons, the program displays the design value of the *Anchorage length* l_d. This length, which consists of the anchorage and additional lengths, is determined by Equation 12.1 according to CSA A23.3-14, 12.2.2. It cannot be modified.

Length l_2 is the length of anchorage for hooks and bends, in accordance with CSA A23.3-14, 12.5 and Clause 6.6.2 of Annex A. For 90° hooks, for example, it should be at least 12 d_b.

The required *Bending Diameter* d_{br} is specified according to CSA A23.3-14, Annex A 6.6.2.3 and Table 16. 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 footer 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 4.3, page 35).

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.

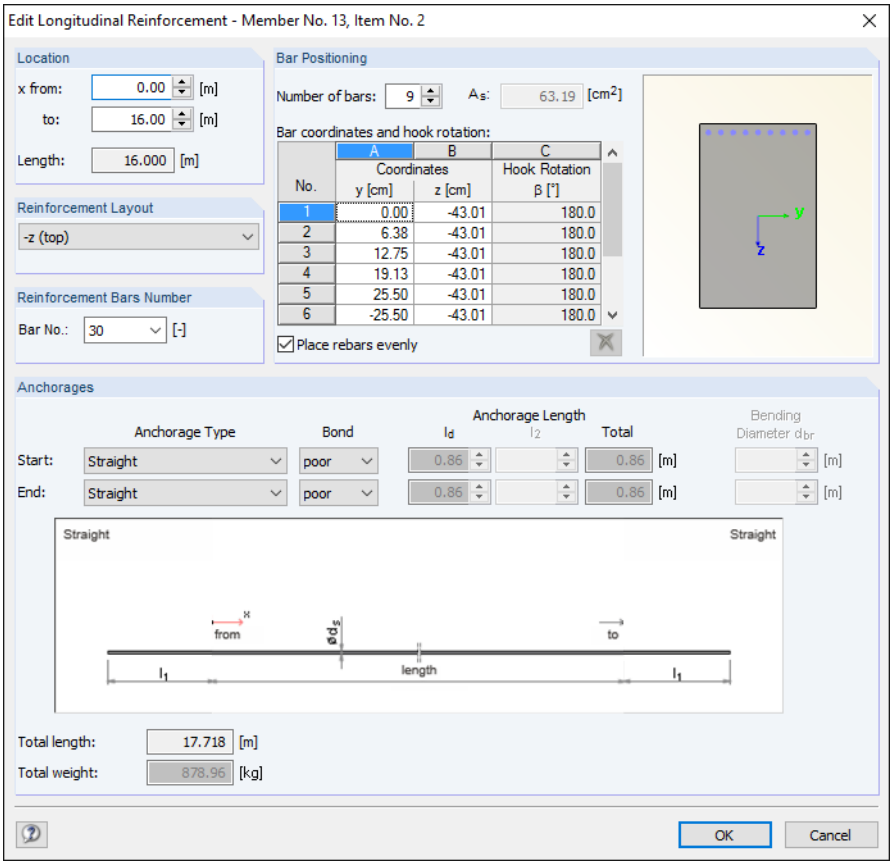


Figure 4.12: Dialog box *Edit Longitudinal Reinforcement*

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 program automatically recalculates the design, using the values of the new provided reinforcement.

4.2.2 Provided Shear Reinforcement

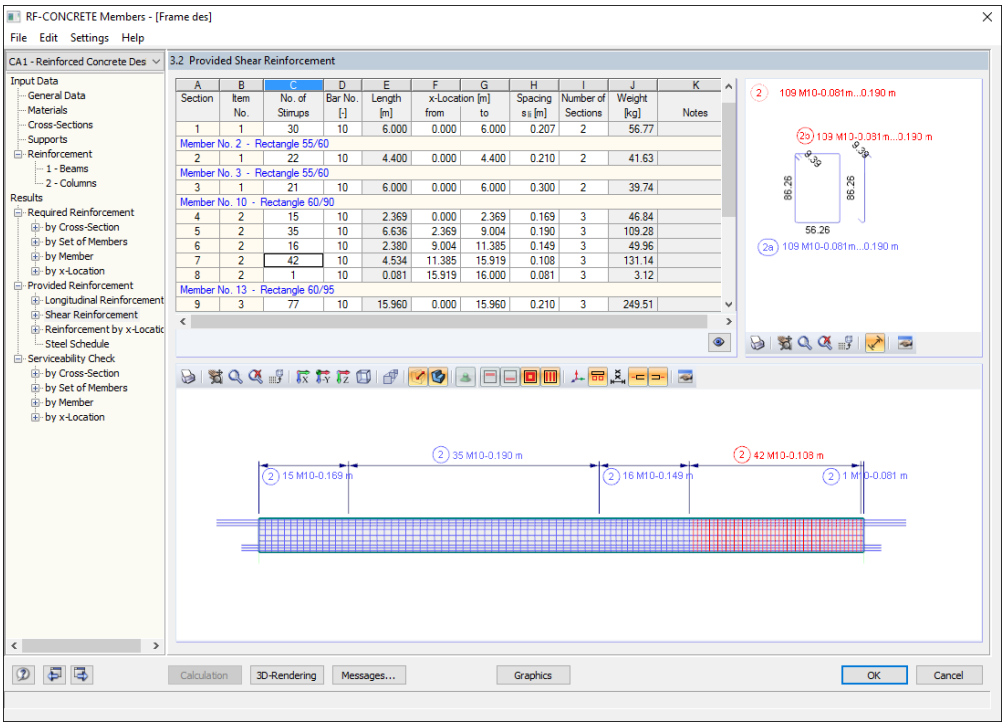


Figure 4.13: Window *3.2 Provided Shear Reinforcement*

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 CSA A23.3-14, 11.3.8.1, the maximum spacing of stirrups shall neither exceed $0.7d_v$ nor 600 mm.

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, spacing).

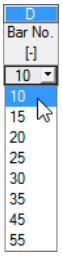
The items of all members and sets of members are summarized in Window *3.4 Steel Schedule*.

No. of Stirrups

When the program determines the shear reinforcement, it takes into account the user-defined specifications entered in Window *1.6 Reinforcement*, tab *Stirrups* (see Chapter 2.6.2, page 21).

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 *Stirrups*. Use the list to change the bar number 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 4.14, page 44).

x-Location from/to

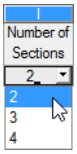
These values represent the start and end positions of the stirrup zones. They refer to the start node of the member given in RFEM (x = 0). The entries in both columns can be edited so that it is possible to shift the zone limits by modifying the values.

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.

Spacing s_{li}

The proposed stirrup spacing considers the specifications defined in Window *1.6 Reinforcement*, tab *Stirrups* (see Chapter 2.6.2, page 21). 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 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 *Stirrups* (see Chapter 2.6.2, page 21). 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 4.3, page 35).

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.

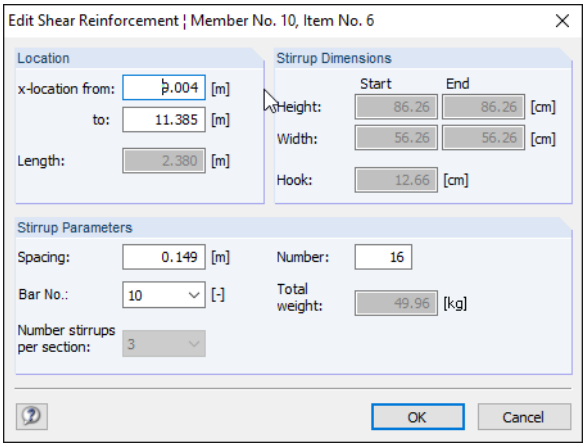


Figure 4.14: Dialog box *Edit Shear Reinforcement*

This dialog box summarizes the reinforcement parameters described above. Here you can control or adjust, if necessary, the *Location*, *Stirrup Dimensions* and *Stirrup Parameters*.

When you have modified data, the program automatically recalculates the design, using the values of the new provided stirrup reinforcement.

4.2.3 Provided Reinforcement by x-Location

This window contains information about the ULS 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.

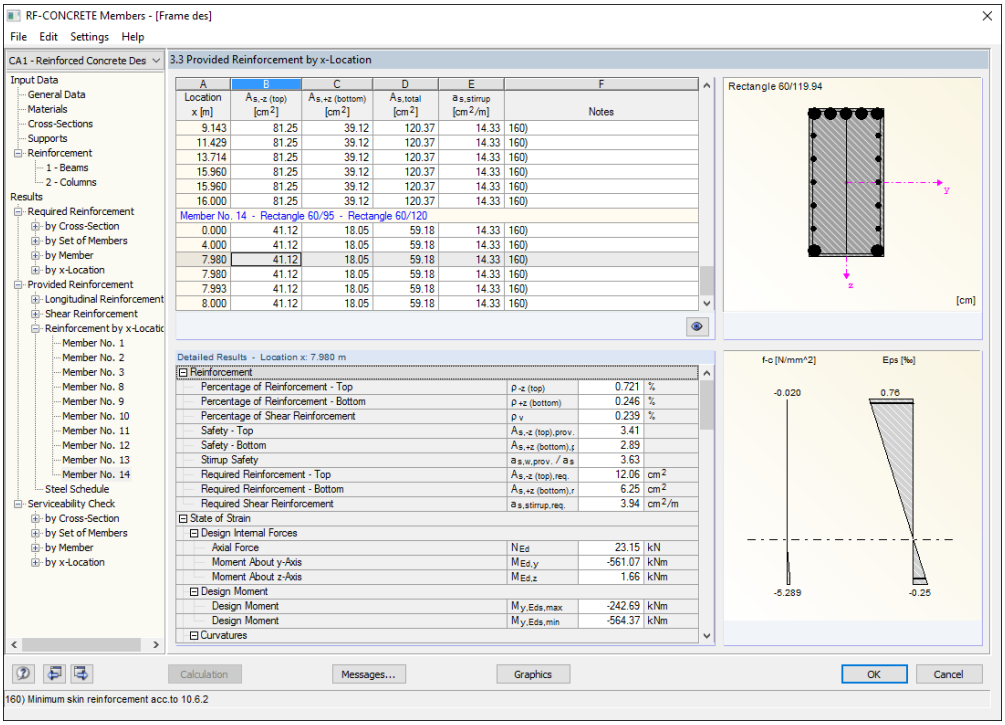


Figure 4.15: Window *3.3 Provided Reinforcement by x-Location*

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 are listed by x-location for each member:

- 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 the 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.

$a_{s,stirrup}$

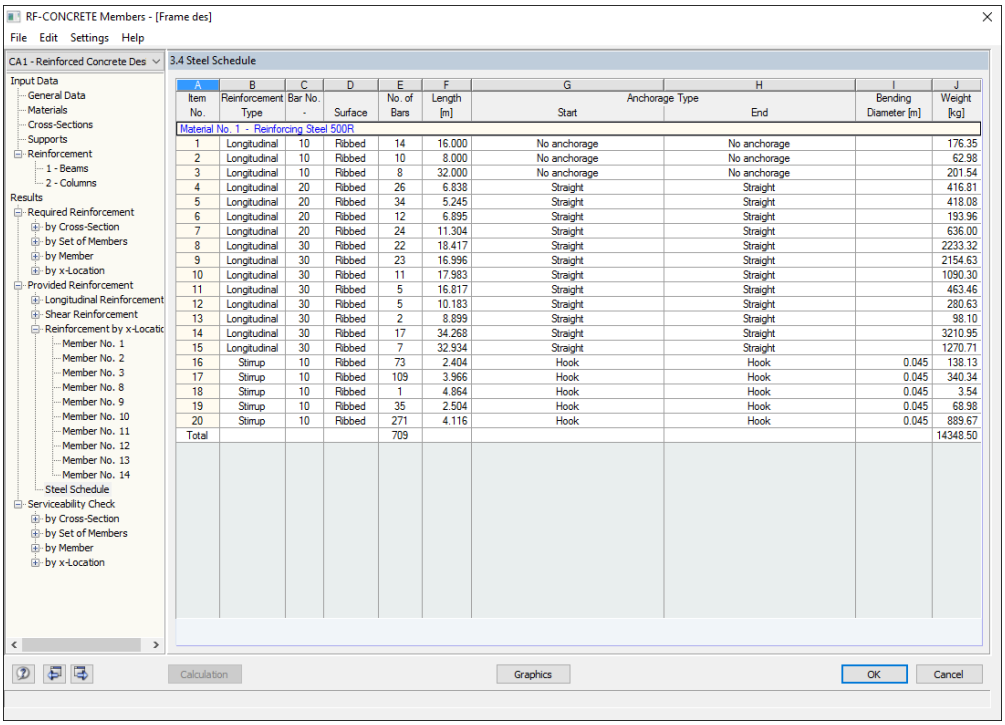
This column shows the areas of the provided shear reinforcement per unit length.

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.

4.2.4 Steel Schedule

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



A	B	C	D	E	F	G	H	I	J
Item No.	Reinforcement Type	Bar No. -	Surface	No. of Bars	Length [m]	Anchorage Type Start	Anchorage Type End	Bending Diameter [m]	Weight [kg]
Material No. 1 - Reinforcing Steel 500R									
1	Longitudinal	10	Ribbed	14	16.000	No anchorage	No anchorage		176.35
2	Longitudinal	10	Ribbed	10	8.000	No anchorage	No anchorage		62.98
3	Longitudinal	10	Ribbed	8	32.000	No anchorage	No anchorage		201.54
4	Longitudinal	20	Ribbed	26	6.838	Straight	Straight		416.81
5	Longitudinal	20	Ribbed	34	5.245	Straight	Straight		418.08
6	Longitudinal	20	Ribbed	12	6.895	Straight	Straight		193.96
7	Longitudinal	20	Ribbed	24	11.304	Straight	Straight		636.00
8	Longitudinal	30	Ribbed	22	18.417	Straight	Straight		2233.32
9	Longitudinal	30	Ribbed	23	16.996	Straight	Straight		2154.63
10	Longitudinal	30	Ribbed	11	17.983	Straight	Straight		1090.30
11	Longitudinal	30	Ribbed	5	16.817	Straight	Straight		463.46
12	Longitudinal	30	Ribbed	5	10.183	Straight	Straight		280.63
13	Longitudinal	30	Ribbed	2	8.899	Straight	Straight		98.10
14	Longitudinal	30	Ribbed	17	34.268	Straight	Straight		3210.95
15	Longitudinal	30	Ribbed	7	32.934	Straight	Straight		1270.71
16	Stirrup	10	Ribbed	73	2.404	Hook	Hook	0.045	138.13
17	Stirrup	10	Ribbed	109	3.966	Hook	Hook	0.045	340.34
18	Stirrup	10	Ribbed	1	4.864	Hook	Hook	0.045	3.54
19	Stirrup	10	Ribbed	35	2.504	Hook	Hook	0.045	68.98
20	Stirrup	10	Ribbed	271	4.116	Hook	Hook	0.045	889.67
Total				709					14348.50

Figure 4.16: 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 Window 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 *Plain*.

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.

4.3 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 2.1.2, page 9) and no design problems were detected by the program (see Chapter 4.1.5, page 38 and Chapter 4.2.3, page 45).

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

4.3.1 Serviceability Check by Cross-Section

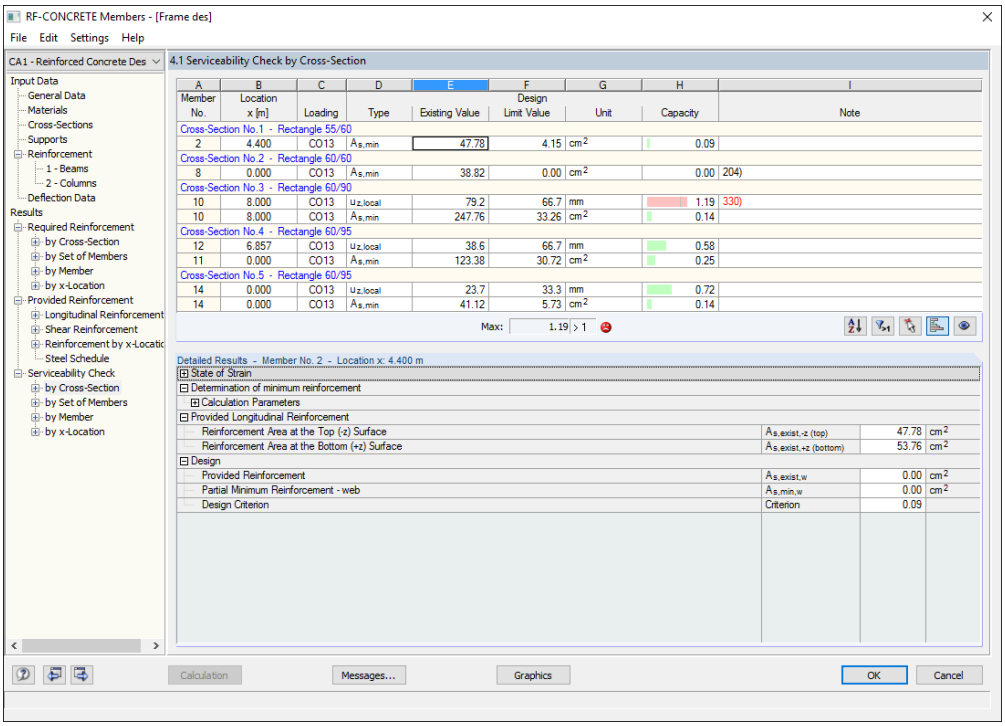


Figure 4.17: Window *4.1 Serviceability Check by Cross-Section*

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 2.6.6, page 26), 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 SLS design. They depend on the specifications made in Window *1.6 Reinforcement*, tab *Serviceability*.

$u_{z,local}$

This is the absolute value of the calculated deformation in direction of the local member axis z.

The allowable relative deformations are administered in Window *1.7 Deflection Data* (see Chapter 2.7, page 29).

$A_{s,min}$

This value represents the minimum reinforcement for crack control according to CSA A23.3-14, 10.6.1. It is compared to the provided reinforcement.

Existing Value

This column 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 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.

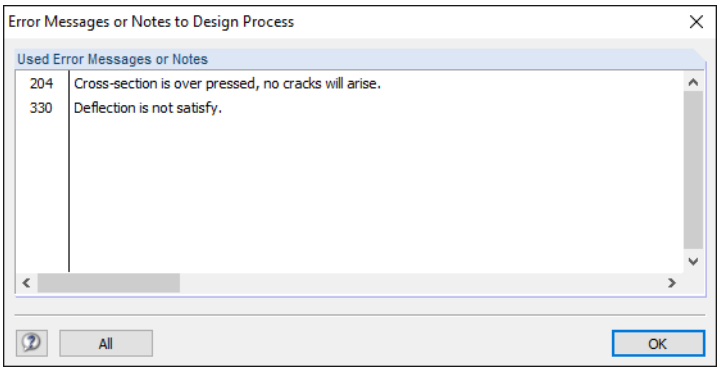


Figure 4.18: Dialog box *Error Messages or Notes to Design Process*

4.3.2 Serviceability Check by Set of Members

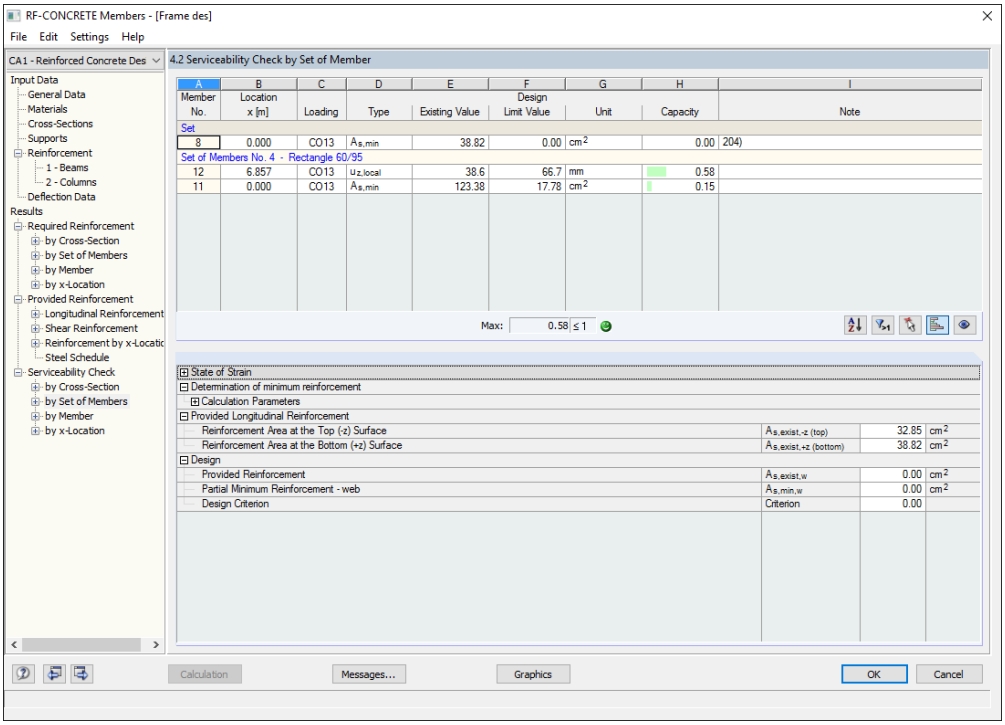


Figure 4.19: Window *4.2 Serviceability Check by Set of Members*

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

Details on the individual columns can be found in Chapter 4.3.1.

4.3.3 Serviceability Check by Member

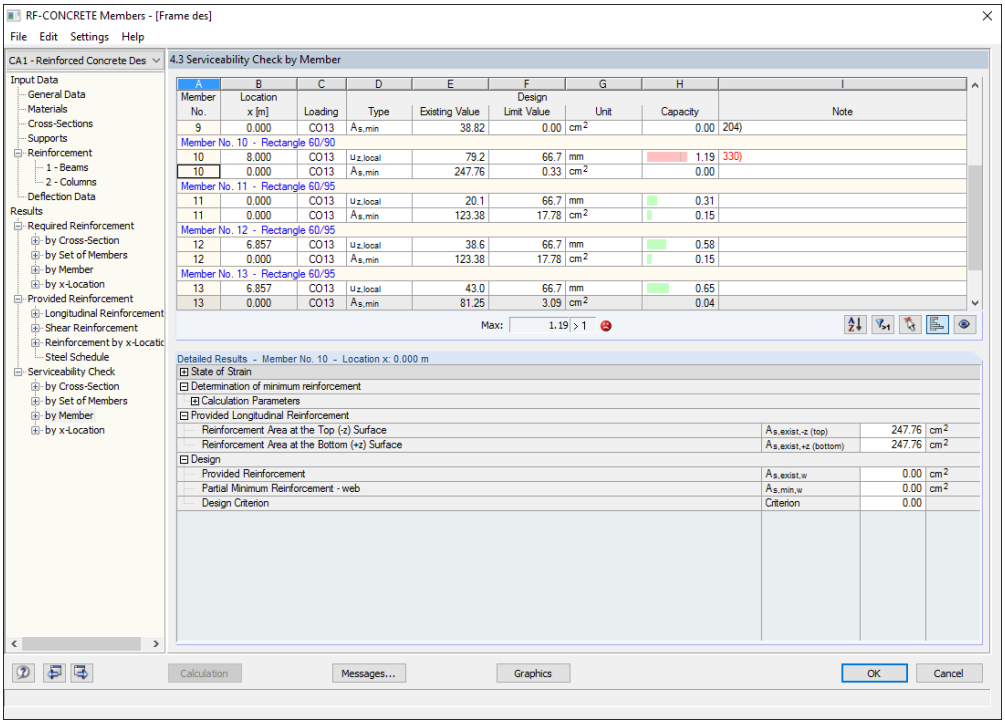


Figure 4.20: Window *4.3 Serviceability Check by Member*

This result window shows the output for the serviceability design sorted by member. The table columns correspond to the columns of Window 4.1. They are described in Chapter 4.3.1.

4.3.4 Serviceability Check by x-Location

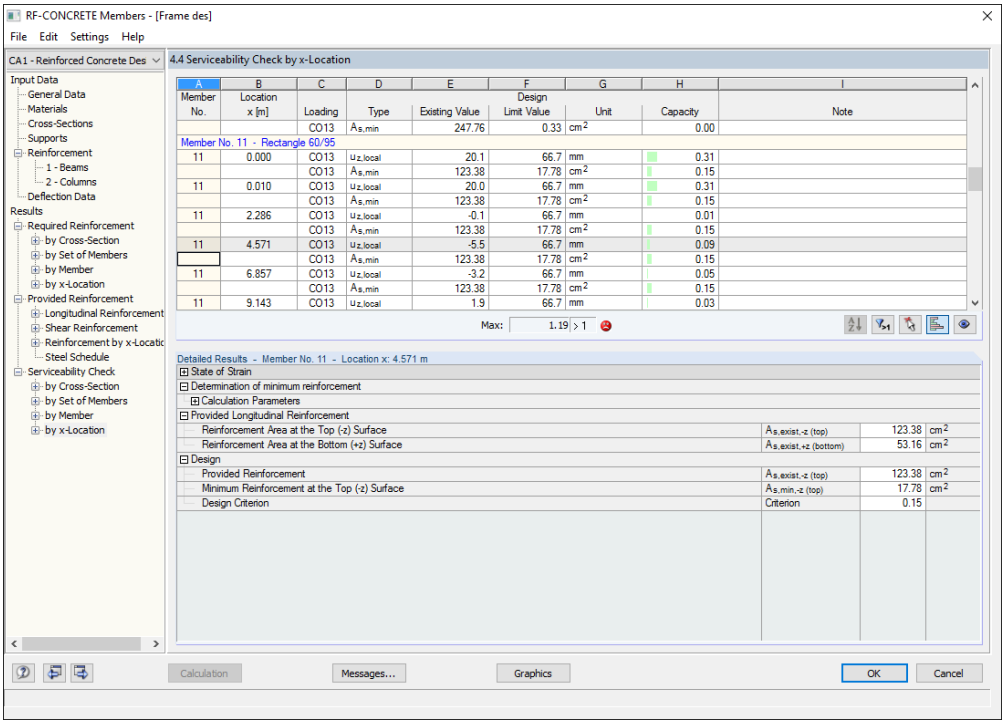


Figure 4.21: Window *4.4 Serviceability Check by x-Location*

This window lists the SLS design results (see Chapter 4.3.1) for all locations x of the designed members.

5. Results Evaluation

The results of the design can be evaluated and adjusted in different ways. The result windows are described in detail in Chapter 4. This chapter describes the graphical evaluation.

5.1 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 4.9, page 39 and Figure 4.13, page 43).

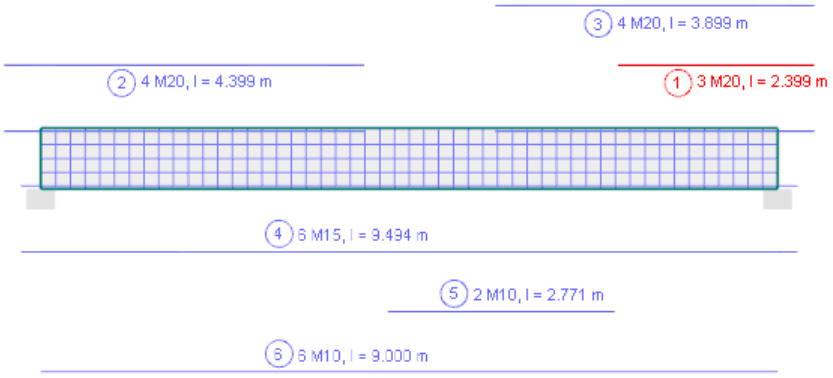


Figure 5.1: Reinforcement drawing in Window *3.1 Provided Longitudinal Reinforcement*

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 4.12 on page 42 or in Figure 4.14 on page 44 appears. In this dialog box, you can check the individual parameters of the selected longitudinal or stirrup reinforcement and adjust them, if necessary.

5.2 3D Rendering of Reinforcement

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).

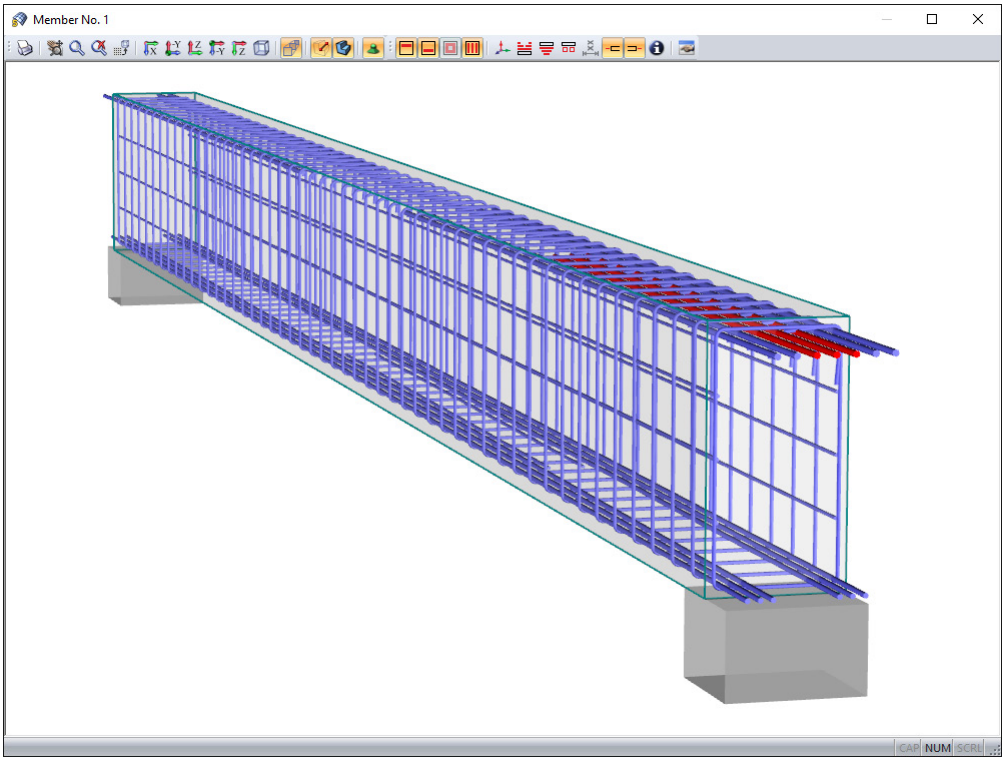
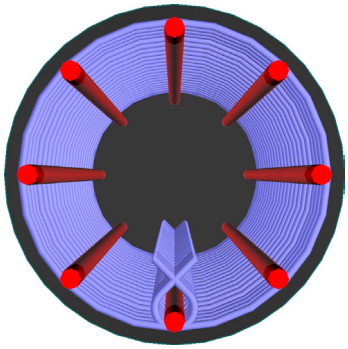


Figure 5.2: 3D rendering of provided longitudinal and stirrup reinforcement

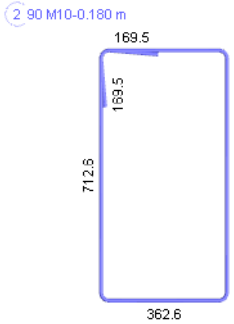
By means of the graphic, you can check the selected reinforcement close to reality.

To set the graphic display appropriately, use the *View* menu or the corresponding buttons (see Table 5.1). 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 be sent to the printer, the printout report or the clipboard.



Reinforcement of column in perspective view



The buttons in the toolbar have the following functions:

Button	Description	Function
	Print	Opens the *Graphic Printout* dialog box (see Figure 6.4, page 61) 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 Window *1.5 Supports*
	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
	Stirrup 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 4.12, page 42 and Figure 4.14, page 44)

Table 5.1: Buttons for 3D rendering

5.3 Results on RFEM Model

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

5.3.1 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.

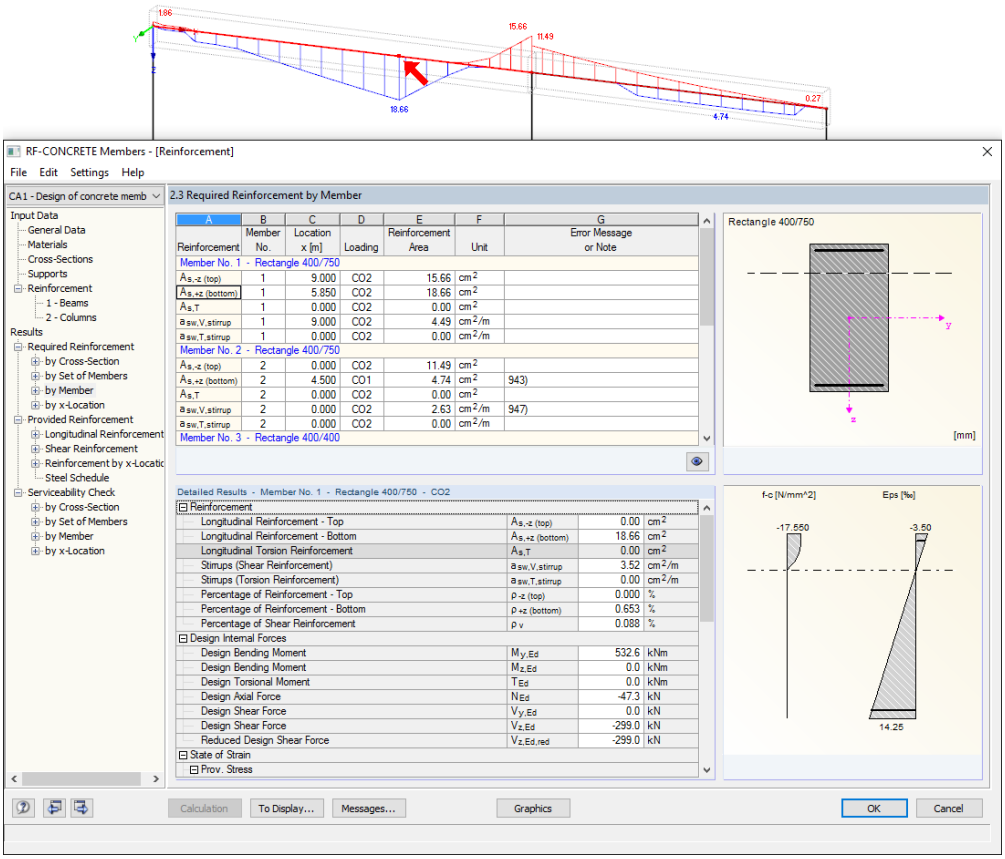
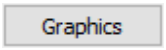
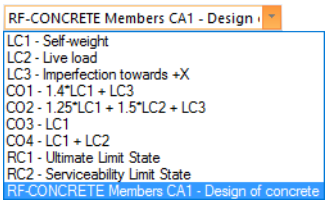


Figure 5.3: Indication of member and current *Location x* on RFEM model

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.

5.3.2 RFEM Work Window





You can check the reinforcements 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 ultimate and the serviceability limit state designs including all detailed results.

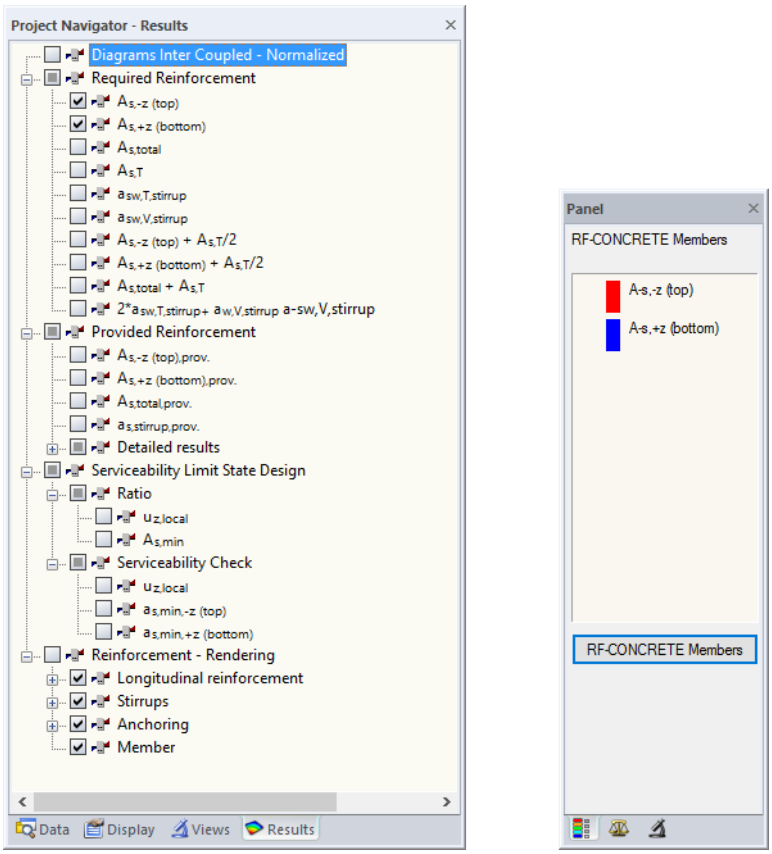


Figure 5.4: *Results* navigator and panel for RF-CONCRETE Members

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/support-and-learning/support/knowledge-base/000715

Due to the 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.

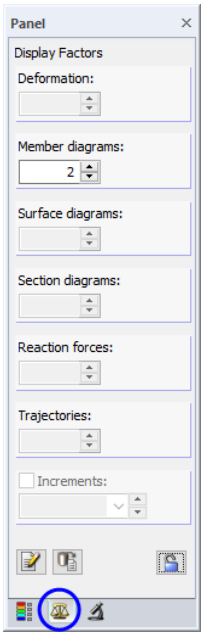


Figure 5.5: Panel tab *Display Factors*

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

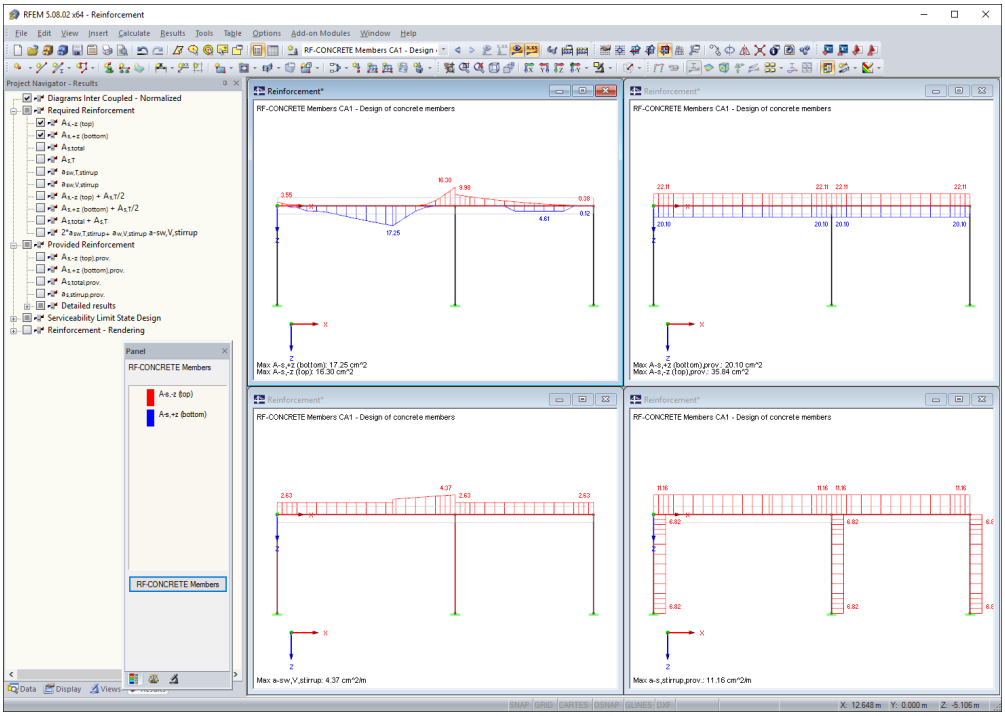


Figure 5.6: Graphical output of required and provided reinforcement

With the *Reinforcement - Rendering* option in the navigator, you can display the rebars and stirrups in the RFEM work window.

All results graphics can be transferred to the global printout report (see Chapter 6.2, page 61).

To return to the add-on module, click the [RF-CONCRETE Members] button in the panel.

5.4 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 as seen to the left.

A window opens showing the distribution of the reinforcement areas and detailed results on the selected member or set of members.

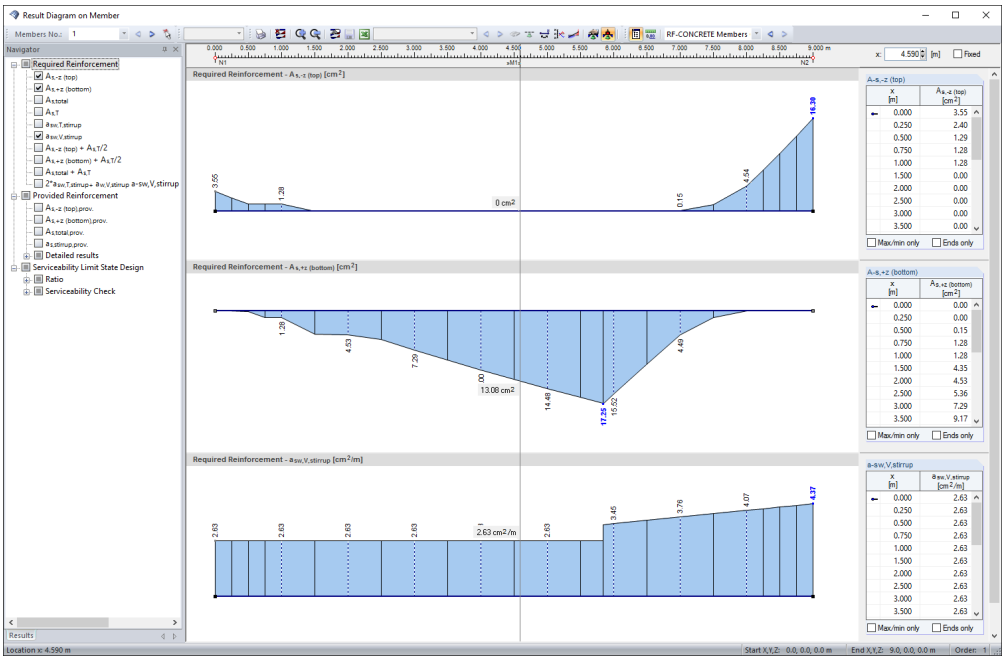


Figure 5.7: Dialog box *Result Diagram on Member*

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.

5.5 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.

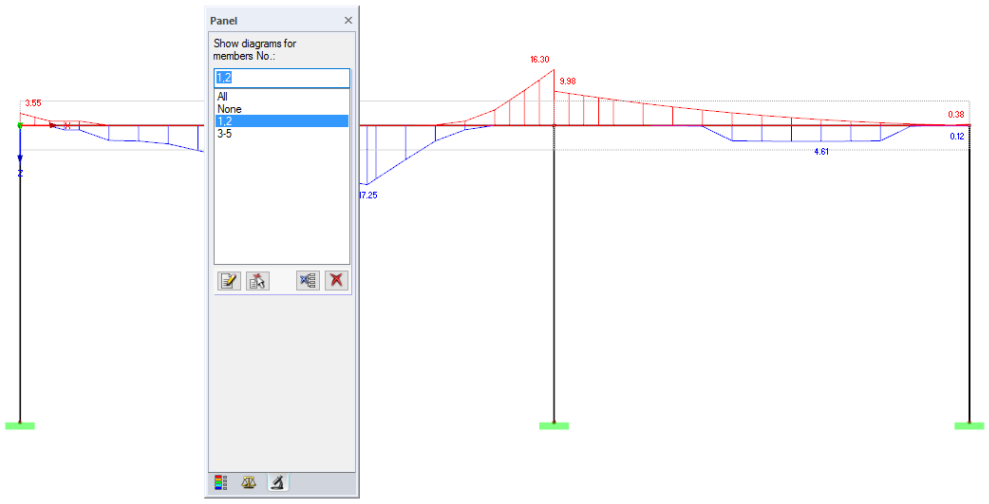


Figure 5.8: Filtering members in the panel

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

6. Printout

6.1 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.

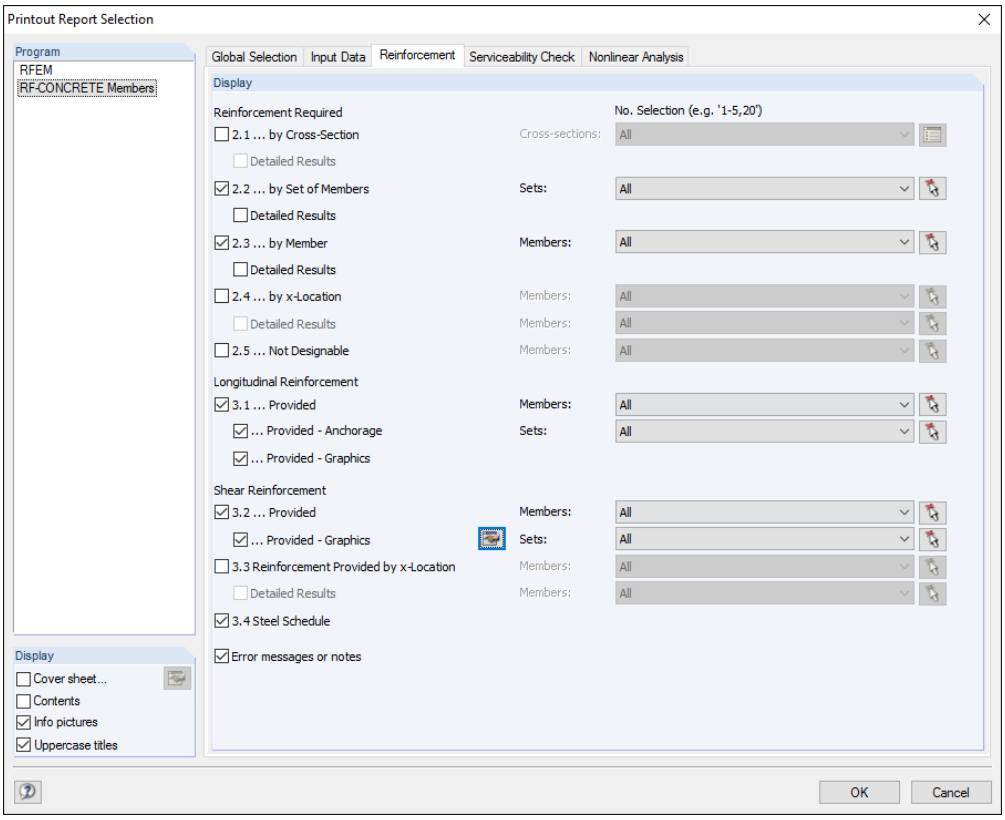


Figure 6.1: Printout report selection of results for RF-CONCRETE Members, tab *Reinforcement*

6.2 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.

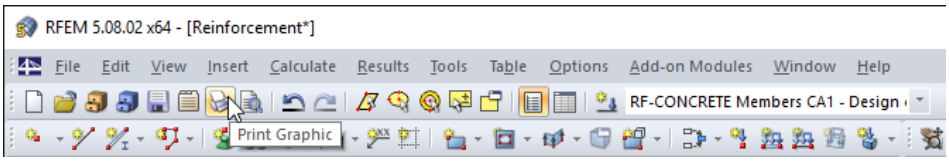


Figure 6.2: Button *Print* in the toolbar of the main window

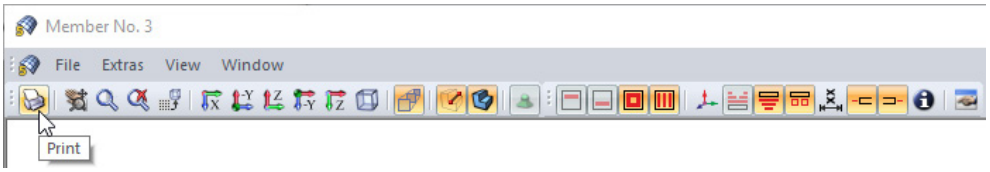


Figure 6.3: Button *Print* in the toolbar of the 3D rendering window

The following dialog box opens:

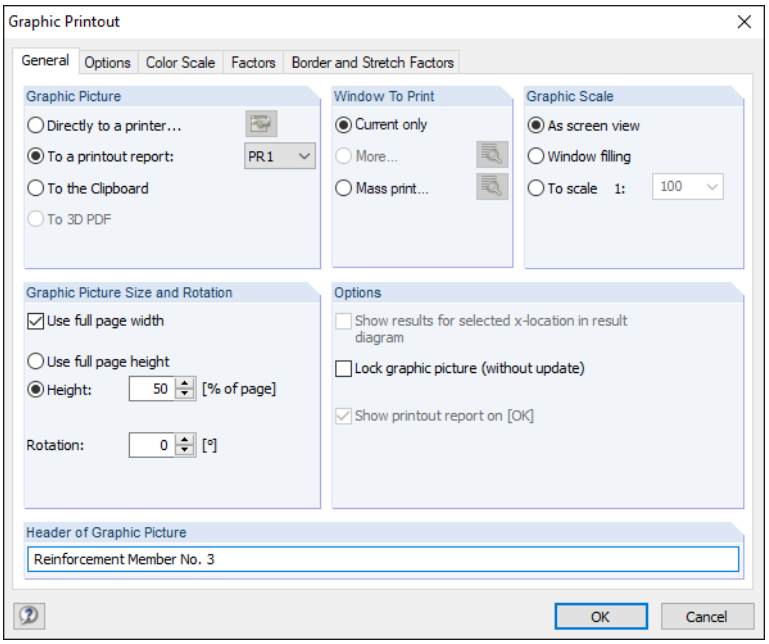
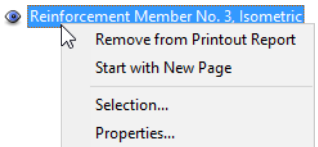


Figure 6.4: Dialog box *Graphic Printout*, tab *General*

This dialog box is described 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.

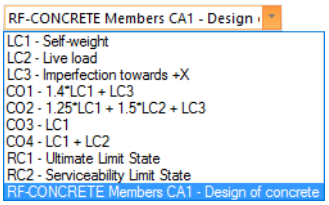
7. General Functions

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

7.1 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 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.

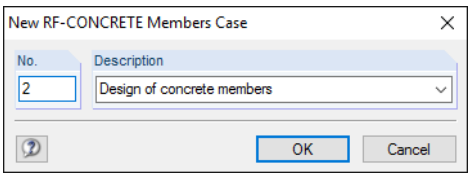


Figure 7.1: Dialog box *New RF-CONCRETE Members Case*

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.

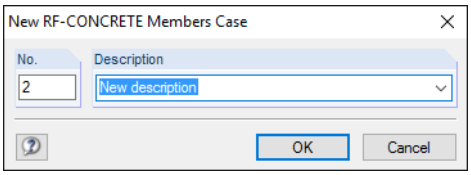


Figure 7.2: Dialog box *Rename RF-CONCRETE Members-Case*

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.

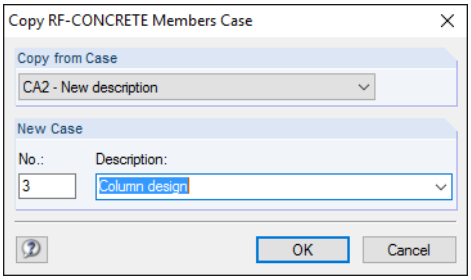


Figure 7.3: Dialog box *Copy RF-CONCRETE Members-Case*

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.

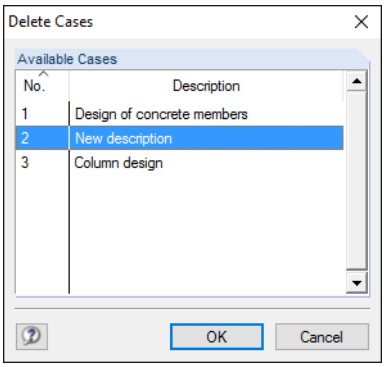


Figure 7.4: Dialog box *Delete Cases*

To delete the selected case, click [OK].

7.2 Cross-section Optimization

As mentioned in Chapter 2.3, 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 2.6, page 12). Immediately afterwards, the following dialog box is opened where you can define the parameters.

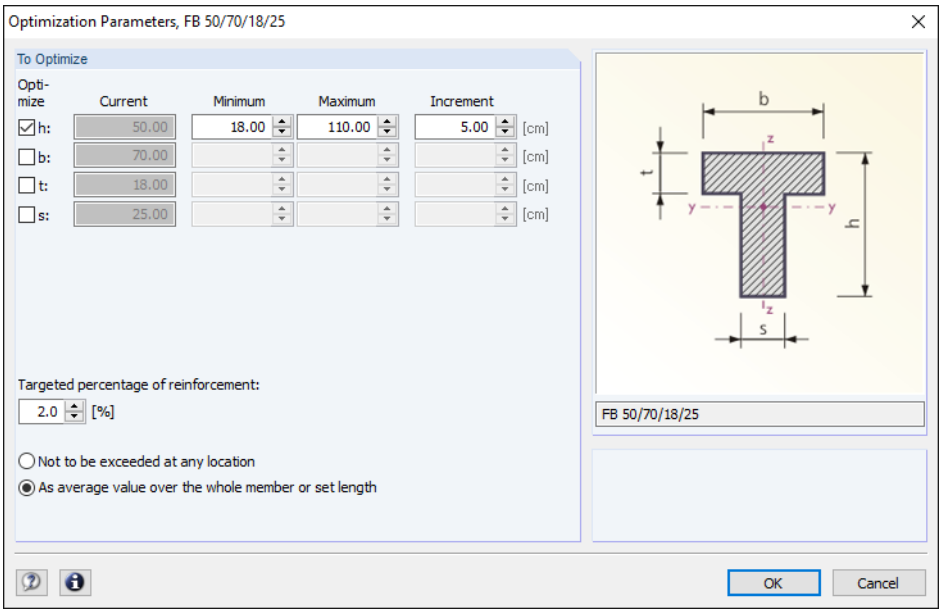


Figure 7.5: Dialog box *Optimization Parameters* of a floor beam

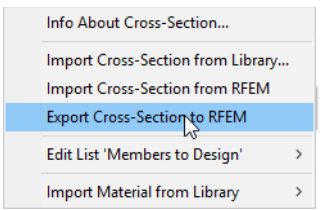
By selecting the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. The selected check boxes enable the *Minimum* and *Maximum* columns where you can define the upper and lower limits of each parameter for optimization. The *Increment* column controls the interval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

7.3 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.

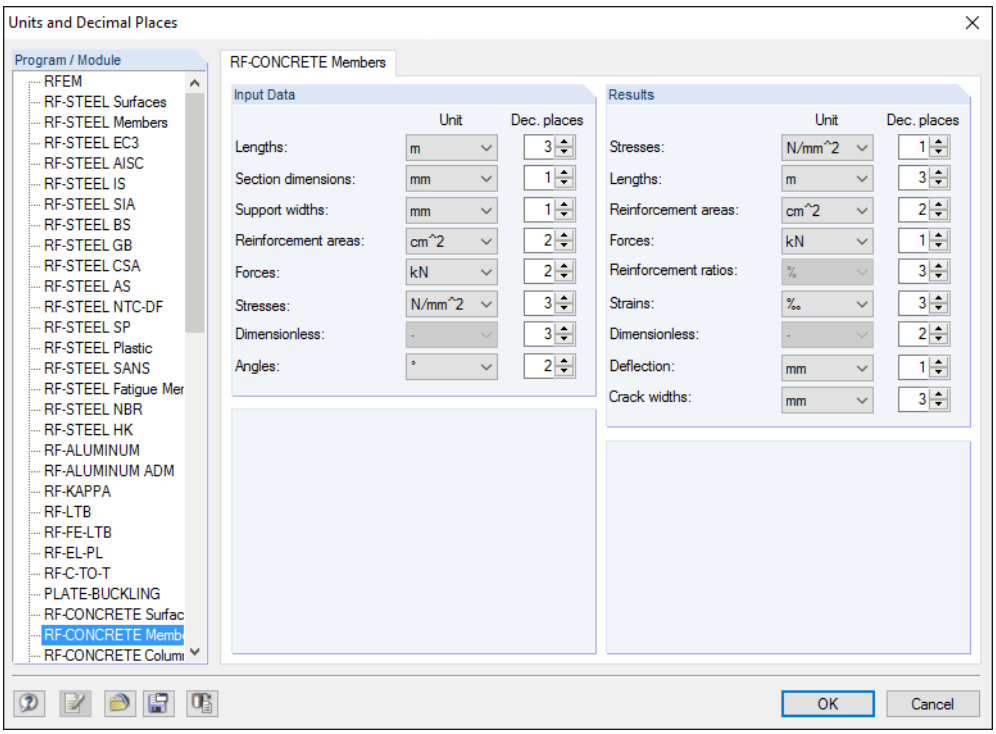
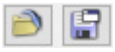


Figure 7.6: Dialog box *Units and Decimal Places*



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.

7.4 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 6.1, page 60) 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.

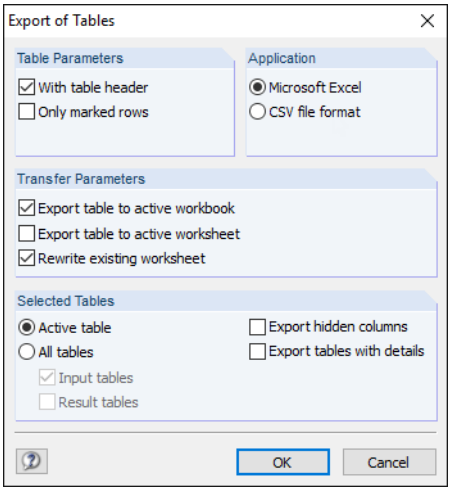
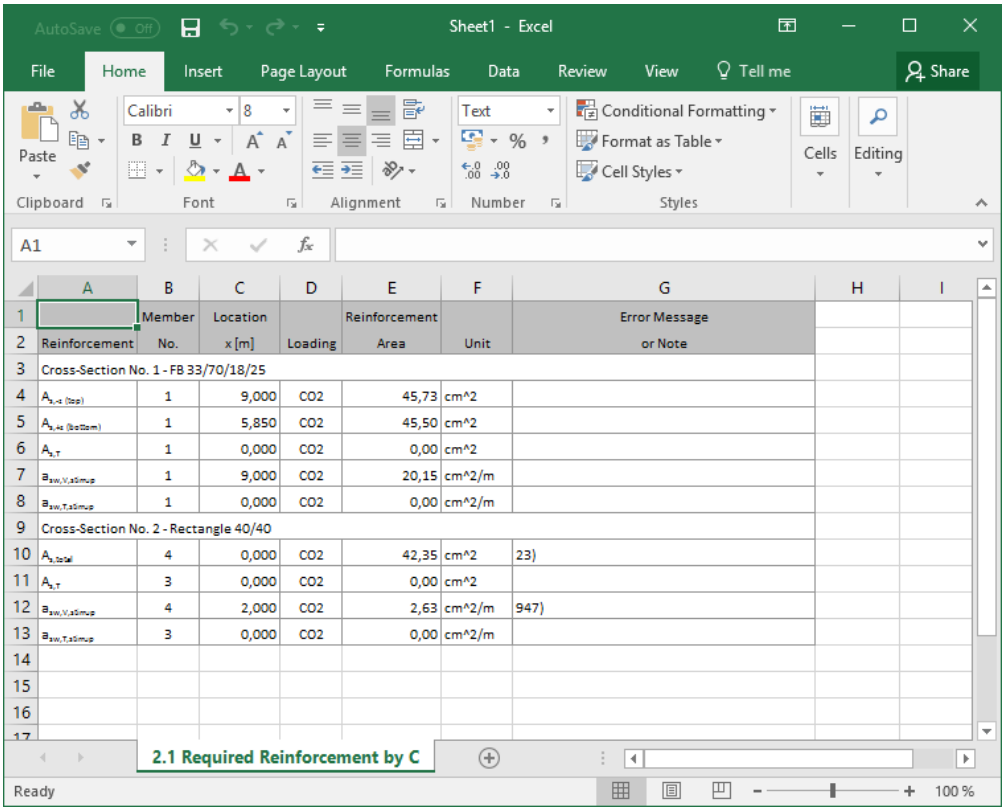


Figure 7.7: Dialog box *Export of Tables*

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.



	A	B	C	D	E	F	G	H	I
1		Member	Location		Reinforcement		Error Message		
2	Reinforcement	No.	x [m]	Loading	Area	Unit	or Note		
3	Cross-Section No. 1 - FB 33/70/18/25								
4	$A_{s,-t\ (top)}$	1	9,000	CO2	45,73	cm^2			
5	$A_{s,+c\ (bottom)}$	1	5,850	CO2	45,50	cm^2			
6	$A_{s,T}$	1	0,000	CO2	0,00	cm^2			
7	$a_{sw,V,stirrup}$	1	9,000	CO2	20,15	cm^2/m			
8	$a_{sw,T,stirrup}$	1	0,000	CO2	0,00	cm^2/m			
9	Cross-Section No. 2 - Rectangle 40/40								
10	$A_{s,total}$	4	0,000	CO2	42,35	cm^2	23)		
11	$A_{s,T}$	3	0,000	CO2	0,00	cm^2			
12	$a_{sw,V,stirrup}$	4	2,000	CO2	2,63	cm^2/m	947)		
13	$a_{sw,T,stirrup}$	3	0,000	CO2	0,00	cm^2/m			
14									
15									
16									

Figure 7.8: Result in MS Excel

A Literature

Standards and other materials

[1] CSA Group standard A23.3-14 – Design of concrete structures, Ontario, 2014

[2] CAC Concrete Design Handbook, 4th Edition, 2016

B Index