Dlubal

Version
December 2018

Add-on Module

RF-CONCRETE Surfaces

Reinforced Concrete Design
According to ACI 318-14

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Module RF-CONCRETE Surfaces

Although reinforced concrete is as frequently used for plate structures as for frameworks, standards and technical literature provide rather little information on the design of two-dimensional structural components. Particularly the design of shell structures that are simultaneously subjected to moments and axial forces is rarely described in reference books. Since the finite element method allows for realistic modeling of two-direction objects, design assumptions and algorithms must be found to close this "regulatory gap" between member-oriented rules and computer-generated internal forces of plate structures.

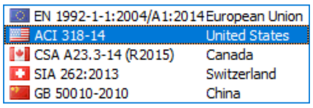
DLUBAL SOFTWARE meets this challenge with the add-on module RF-CONCRETE Surfaces. Based on the compatibility equations by BAUMANN from 1972 [1], a consistent design algorithm has been developed to dimension reinforcements with two or three directions of reinforcement. The module is more than just a tool to determine the statically required reinforcement. It also includes regulations concerning the allowable minimum and maximum reinforcement ratios for different types of structural components (2D plates, 3D shells, walls, deep beams) as they can be found in the form of design specifications defined in the standards.

When determining the reinforcing steel, **RF-CONCRETE Surfaces** checks whether the concrete's plate thickness, which stiffens the reinforcement mesh, is sufficient to meet all requirements arising from bending and shear loading.

In addition to the strength limit state design, the serviceability limit state design is possible as well. These designs include the limitation of the concrete compressive and the reinforcing steel stresses, the minimum reinforcement for the crack control, as well as the crack control by limiting rebar diameter and rebar spacing. For this purpose, analytical and nonlinear design check methods are available for selection.

If you also have a license for **RF-CONCRETE Deflect**, you can calculate the deformations with the influence of creep, shrinkage, and tension stiffening according to the analytical method.

With a license of **RF-CONCRETE NL**, you can consider the influence of creep and shrinkage in the determination of deformations, crack widths, and stresses according to the nonlinear method.



Design according to ACI 318-14

The design is possible according to the following standards:

- ACI 318-14
- CSA A23.3-14 (R2015)
- EN 1992-1-1:2004/A1:2014
- SIA 262:2013
- GB 50010-2010

All intermediate results for the design are comprehensively documented in the results tables and in the printout report. In line with the DLUBAL philosophy, this provides a special transparency and traceability of the design results.

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

Your DLUBAL SOFTWARE Team

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 Surfaces.

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

At the end of the manual, there is the index. However, if you do not find what you are looking for, you can check the Knowledge Base or FAQs at our website and look for a solution.

1.3 Opening RF-CONCRETE Surfaces

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

Menu

To start the add-on module from of the RFEM menu, select

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

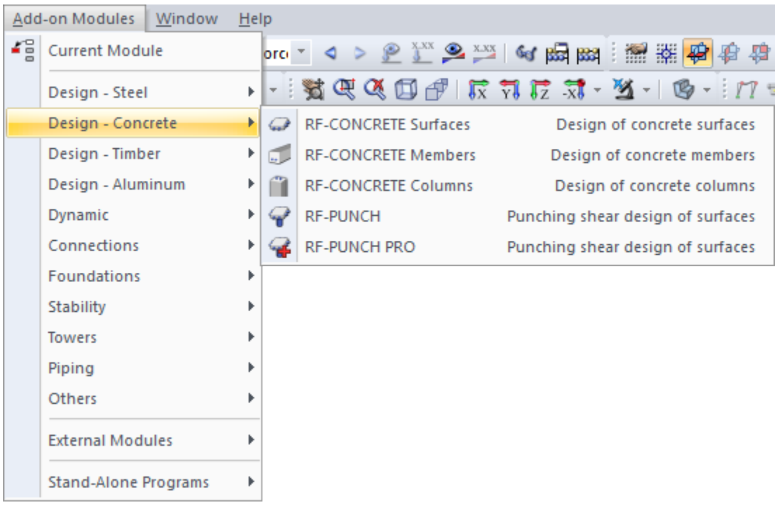


Figure 1.1: Menu *Add-on Modules → Design - Concrete → RF-CONCRETE Surfaces*

Navigator

To start die add-on module in the *Data* navigator, select

Add-on Modules → RF-CONCRETE Surfaces.

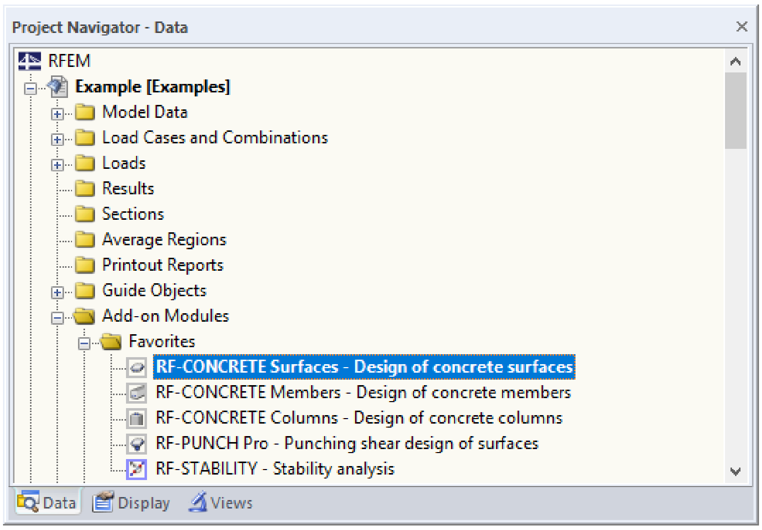
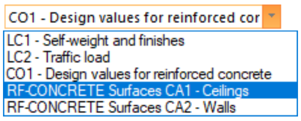


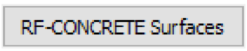
Figure 1.2: *Data* navigator item *Add-on Modules → RF-CONCRETE Surfaces*



Panel

If results from RF-CONCRETE Surfaces are already available in the RFEM model, you can also start the design module in the panel. Select the relevant RF-CONCRETE Surfaces design case in the load case list of the menu bar. Then use the toolbar button [Show Results] to display the reinforcements graphically.

Now you can click the [RF-CONCRETE Surfaces] button in the panel to open the add-on module.



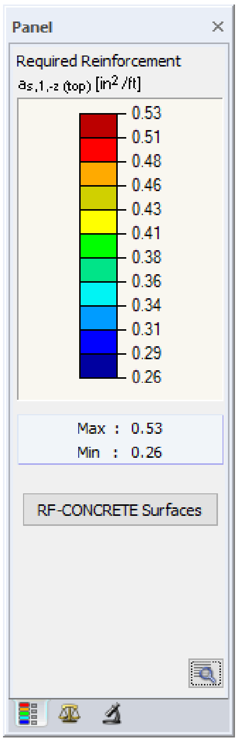


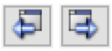
Figure 1.3: Panel button [RF-CONCRETE Surfaces]

2. Input Data

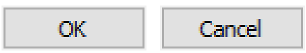
When you start the add-on module, a new window opens. In this module window, the navigator on the left lists all available windows. The drop-down list above the navigator contains the design cases (if created).

The design-relevant data is to be defined in several input windows. When you have opened RF-CONCRETE Surfaces for the first time, the following parameters are automatically available:

- Load cases, load and result combinations
- Materials
- Surfaces
- Internal forces (in background, if calculated)



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



To save the results, click [OK]. You exit RF-CONCRETE Surfaces and return to the main program. By clicking [Cancel], you exit the add-on module without saving the data.

2.1 General Data

In Window *1.1 General Data*, you specify the design standard and the actions. The tabs manage the load cases, load and result combinations for the strength limit state and serviceability limit state designs.

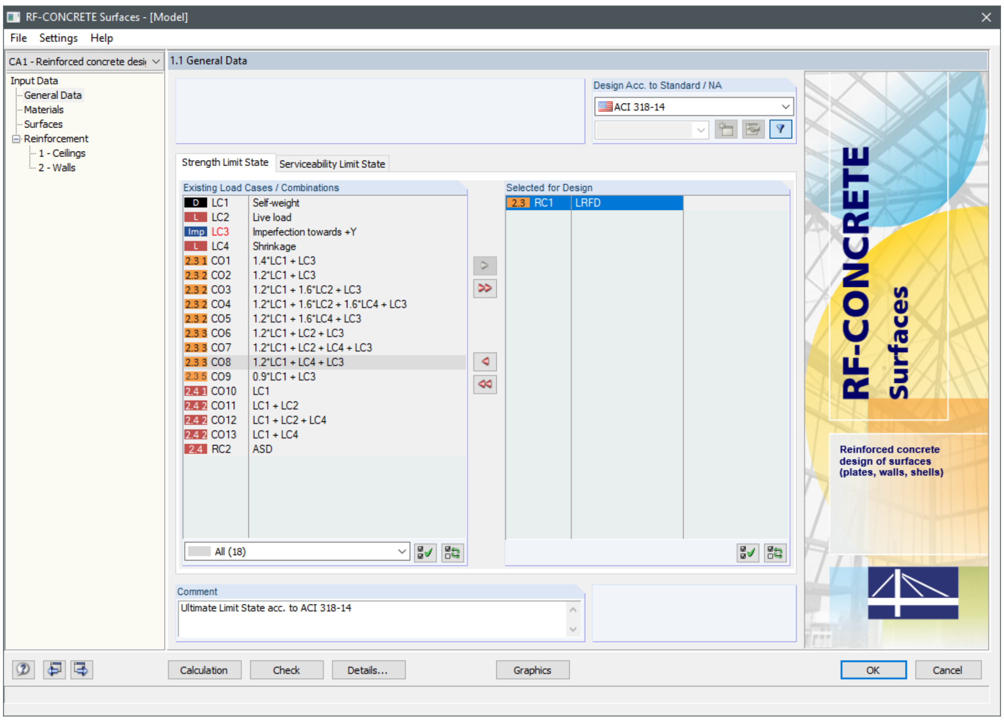


Figure 2.1: Window *1.1 General Data*

Design Acc. to Standard / NA

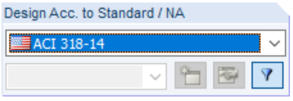
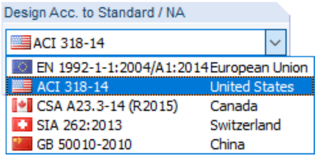


Figure 2.2: Standard and (if applicable) National Annex for reinforced concrete design



Specify the *Standard* which is to be applied for the strength limit state and serviceability limit state designs. The following standards for reinforced concrete design are available in the list:

- EN 1992-1-1:2004/A1:2014 European Union
- **ACI 318-14** **United States of America**
- CSA A23.3-14 (R2015) Canada
- SIA 262:2013 Switzerland
- GB 50010-2010 China

Comment

Figure 2.3: User-defined comment

In the text box, you can enter notes to describe the current design case, for example.

2.1.1 Strength Limit State

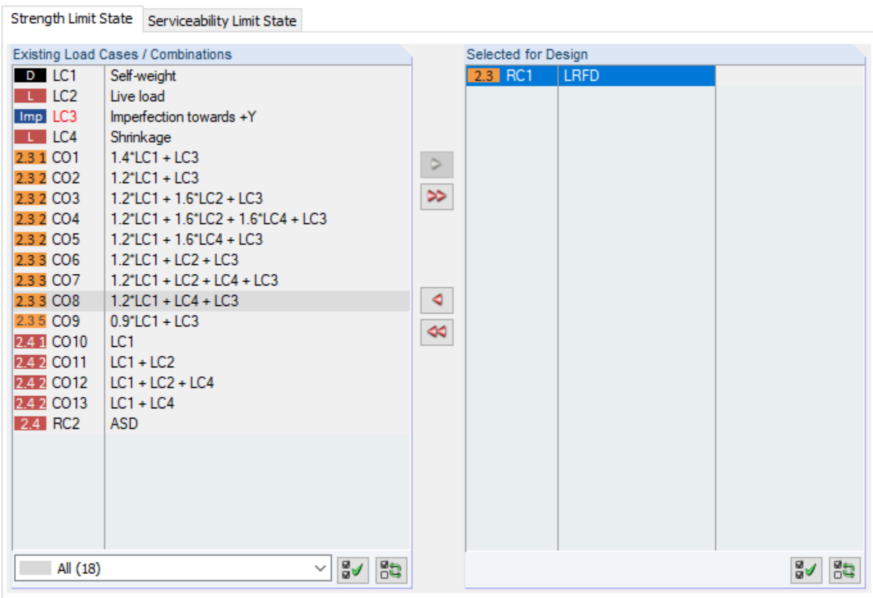


Figure 2.4: Window *1.1 General Data*, tab *Strength Limit State*

Existing Load Cases / Combinations

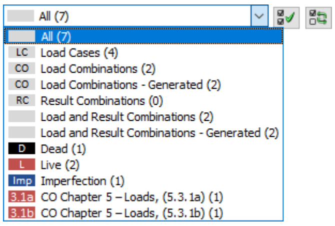
This column lists all load cases, load combinations, and result combinations created in RFEM.

Click [▶] to transfer the selected entries to the *Selected for Design* list on the right. You can also double-click the items to select them. To transfer the complete list to the right, click [▶▶].

To select several load cases at the same time, click them one by one while pressing [Ctrl]. This allows you to transfer more load cases at once.

If a load case is highlighted in red (e.g. *LC3* in Figure 2.4), it is not possible to calculate it: this load case has no load data or imperfections only. When you try to transfer it, a corresponding warning will be shown.

Below the list, several filter options are available. They make it easier to assign the entries sorted by load cases, combinations, or action categories. The buttons have the following functions:



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

Table 2.1: Buttons in tab *Strength Limit State*

Selected for Design

The column on the right lists all loads cases, load and result combinations selected for the design. To remove selected items from the list, click [◀] or double-click the item. To transfer the entire list to the right, click [◀◀].

The design of an enveloping max/min result combination will be faster than the one of all load cases and load combinations globally selected for design. In the design of a result combination, however, it is difficult to make out the influence of the included actions (see also Chapter 3.1, page 39).

2.1.2 Serviceability Limit State

The serviceability limit state design depends on the reinforcement results of the strength limit state design. Thus, it is not possible to perform the serviceability limit state design exclusively.

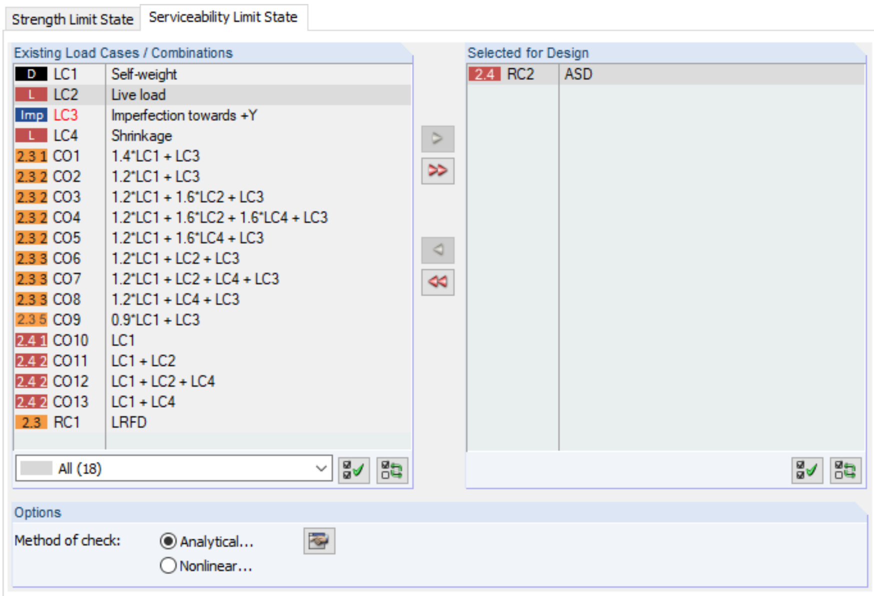


Figure 2.5: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases / Combinations

These two sections list all load cases, load and result combinations defined in RFEM.

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

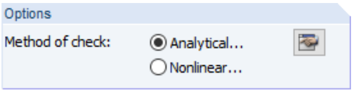
Selected for Design

Load cases, load and result combinations can be added or removed, as described in Chapter 2.1.1 above.

Options

With the *Method of check* options, you decide whether you want to carry out the serviceability limit state designs according to either the *Analytical* or the *Nonlinear* method.

2.1.2.1 Analytical Method



The *Analytical* analysis is preset. This method uses the equations given by the standards for reinforced concrete.

To open the dialog box to check and, if necessary, adjust the design parameters, click [Settings].

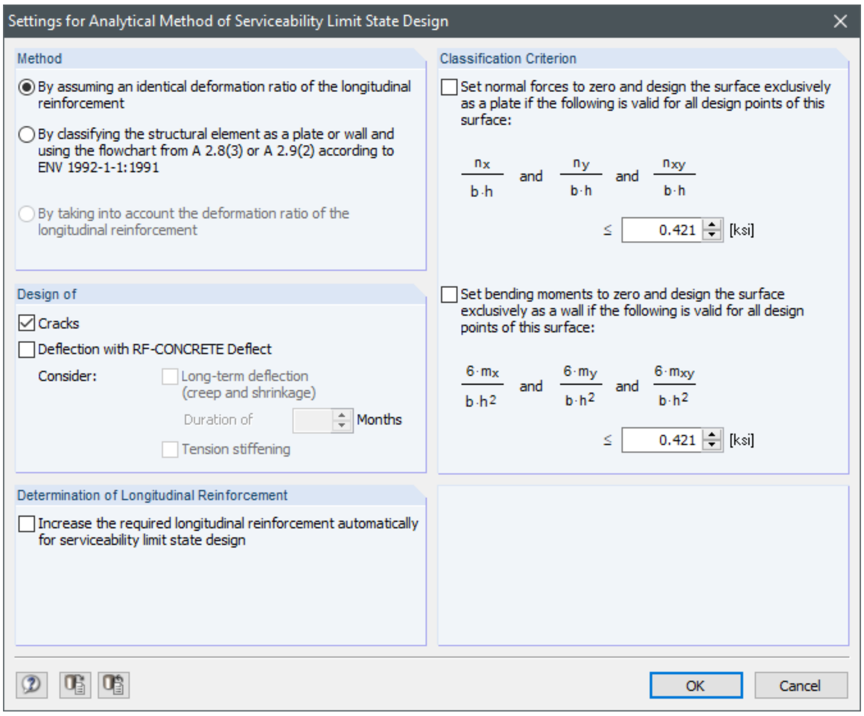


Figure 2.6: Dialog box *Settings for Analytical Method of Serviceability Limit State Design*

Method

In this section, you decide which deformation ratio of the directions of reinforcement is to be applied for the serviceability limit state design.

By assuming an identical deformation ratio of the longitudinal reinforcement, the program uses the same deformation ratio of the provided reinforcement. All rebars in the individual reinforcement directions are subjected to the same strain. This approach represents a fast and exact procedure. For it, the selected inclination of the concrete strut plays a significant role. This method is based on a purely geometrical division. It is applicable when the provided reinforcement corresponds to the required reinforcement.

The option *By classifying the structural element as a plate or wall* offers you a simplified solution that you can use for a non-rotated, orthogonal reinforcement mesh. The program checks for each design point if the tensile stresses from axial forces or bending moments do not exceed a certain stress. The limit value of the stress is to be defined in the *Classification Criterion* section. This criterion is used to classify whether the surface is to be designed as a plate (axial forces set to zero) or as a wall (moments set to zero). By neglecting minor internal force components, it is possible to use the flowchart shown in ENV 1992-1-1, Annex A 2.8 or 2.9. The design internal forces correspond to the values displayed in Table 4.17 of RFEM (see RFEM manual, Chapter 8.17).

If the program cannot satisfy the classification criterion for a design point of the surface, an error message appears during the calculation.

The third option *By taking into account the deformation ratio of the longitudinal reinforcement* is enabled for 2D model types only. This method considers the effective deformation ratios due to the selected reinforcement and takes them into account for the serviceability limit state design.

Design of

In this section, you can specify whether to analyze cracks and/or deflections in the design. You have to select at least one of those two check boxes.

If you select *Cracks*, the module checks the maximum reinforcement spacings, max s_l, and the crack widths, w_k. The settings for the individuals checks can be specified in Window *1.3 Surfaces* (see Chapter 2.3, page 17).

Furthermore, it is possible to calculate the *Deflection with RF-CONCRETE Deflect*, taking into account creep, shrinkage, and tension stiffening. You need a license of the **RF-CONCRETE Deflect** add-on module to use this option.

Determination of Longitudinal Reinforcement

By selecting the *Increase the required longitudinal reinforcement automatically* check box, you decide to dimension the longitudinal reinforcement in such a way that the serviceability limit state designs will be satisfied at any rate. If this check box is deactivated, the program uses only the specifications in the *Longitudinal Reinforcement* tab of Window 1.4 (see Chapter 2.4.3, page 27): basic reinforcement, required reinforcement from strength limit state design, or basic reinforcement with provided additional reinforcement.

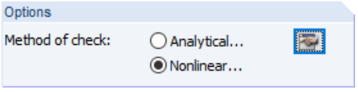
The design of the reinforcement for the serviceability limit state is determined by increasing the reinforcement iteratively. As initial value for the iterations to resist the given characteristic load, the program takes the required strength limit state reinforcement. The dimensioning will have no results if the rebar spacing, s_l, of the applied reinforcement reaches the rebar diameter, d_{sl}. In this case, the results windows will indicate that the respective point cannot be designed.

If the resulting crack width, $w_{k,res}$, is governing to satisfy the check of crack widths, the reinforcement is increased equally for each direction.

Classification Criterion

This section is only available for 3D model types. With the check boxes, you decide if minor *normal forces* and/or *moments* may be neglected in order to design surfaces in an idealized way as pure plates (selection of first check box) or walls (second check box). As limit value, the mean axial tensile strength, f_{ct}, is preset as 0.421 ksi of a concrete f'_c = 4000 psi for each option. It is assumed that the tensile strength of concrete compensates crack formation due to minor tensile stresses. That is the reason why they can be neglected.

If you have selected the classification of the surface as a plate or wall (second option in the *Method* dialog box section on the left), you have to activate at least one of the check boxes.



2.1.2.2 Nonlinear Method

To perform the SLS design according to the *Nonlinear* method, a license of the add-on module **RF-CONCRETE NL** is required. With it, a physical and a geometrical nonlinear analysis can be carried out.

The nonlinear design method is based on the interaction between model and action effects, which requires a clear distribution of internal forces. Therefore, it is only possible to analyze load cases and load combinations, no result combinations (RC). Result combinations provide two values for every FE node – one maximum and one minimum value.

The internal forces according to the nonlinear design method are generally determined by a second-order analysis.

To open the dialog box to check and, if necessary, adjust the design parameters, select the [Settings] button. The dialog box consists of two tabs: *Options* and *Material Properties*.

2.1.2.2.1 Options

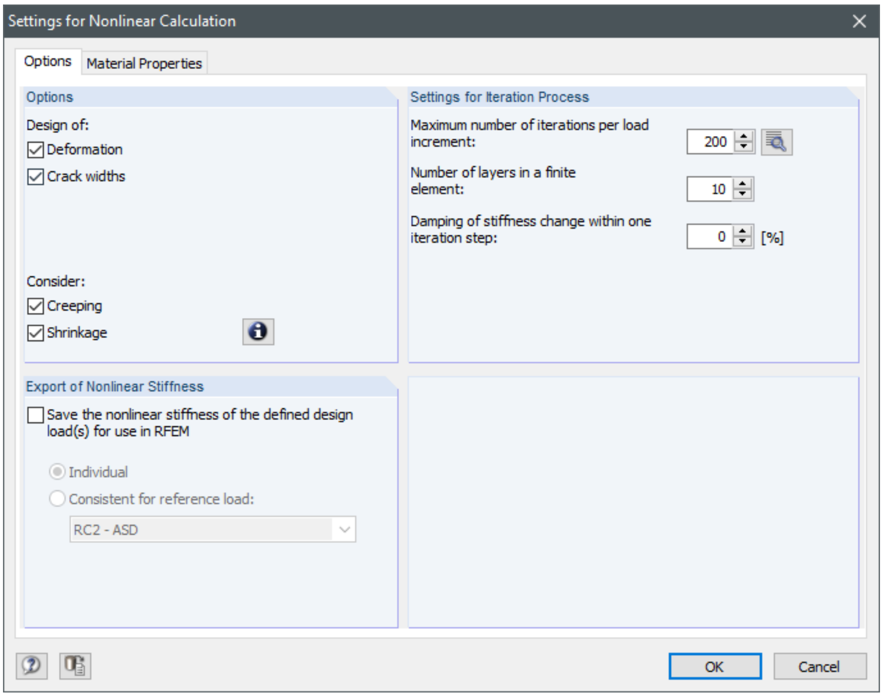


Figure 2.7: Dialog box *Settings for Nonlinear Calculation*, tab *Options*

Options

In this section, you select which types of serviceability limit state design are to be carried out: *Deformation* and/or *Crack widths*. You have to select at least one of the check boxes.

Furthermore, you can specify whether the influence of *Creeping* and/or *Shrinkage* is to be considered during the nonlinear analysis.

Detailed settings for the individual checks as well as for creeping and shrinkage can be made in Window *1.3 Surfaces* (see Chapter 2.3.2, page 19).

Export of Nonlinear Stiffness

With the option *Save the nonlinear stiffness of the defined design load(s) for use in RFEM*, you can transfer the resulting stiffnesses to RFEM and use it for further analyses.

The stiffnesses can by exported *Individually* for each designed load case. In the *Edit Load Cases and Combinations* dialog box of RFEM, you can then assign one of the stiffnesses determined by RF-CONCRETE Surfaces to the corresponding load case. RFEM allocates them automatically. If you select the *Consistent for reference load* option, you have to specify the governing load case in the drop-down list below. In RFEM, you can then assign the stiffness resulting from these loads to all defined load cases.

The consideration of nonlinear stiffnesses in RFEM is described in Chapter 7.3.1.3 of the RFEM manual.

Settings for Iteration Process

The settings in this section have an effect on the process of the nonlinear design method. For more information, see the general manual of RF-CONCRETE Surfaces.

When modifying the precision of iterations, take care that the *Maximum number of iterations per load increment* is higher than the step in the calculation process from which on the deformation criterion will be additionally considered. To open the *Calculation Parameters* dialog box of RFEM, click [Details]. In this dialog box, you can adjust the precision of the convergence criteria for the nonlinear calculation.

For the nonlinear analysis, the surface is divided into so-called *layers*. The recommended number of layers is 10.

You can manipulate the performance of the convergence behavior by the *Damping* option: The damping controls the magnitude of the stiffness change in subsequent iteration steps. With a damping of 50 %, for example, the maximum change of the stiffness between step 2 and 3 can be 50 % of the stiffness change between step 1 and step 2.

2.1.2.2.2 Material Properties

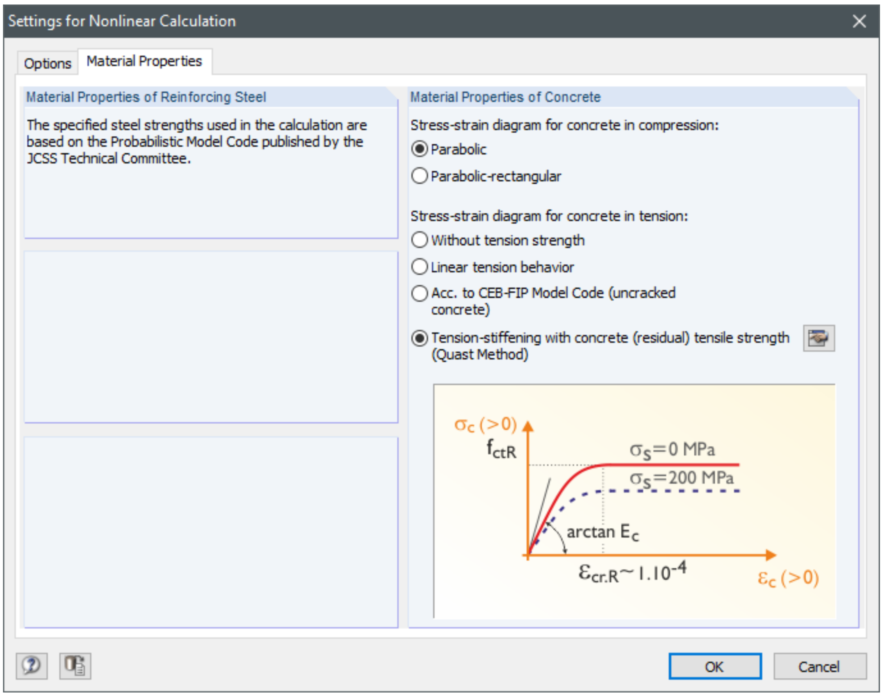


Figure 2.8: Dialog box *Settings for Nonlinear Calculation*, tab *Material Properties*

Material Properties of Reinforcing Steel

For ACI 318-14, no changes can be made in this dialog box section.

Material Properties of Concrete

In this section, you specify the stress-strain relationships of the *concrete in compression* and of the *concrete in tension*. The parabolic diagram is preset for compression. For tensile stresses, the tension stiffening approach (see below) is set as default.

For *Tension stiffening* (consideration of the stiffening effect of concrete in tension), you can specify the parameters in a separate dialog box where you define how the tensile strength of concrete between the cracks is to be applied. To open this dialog box, click [Edit].

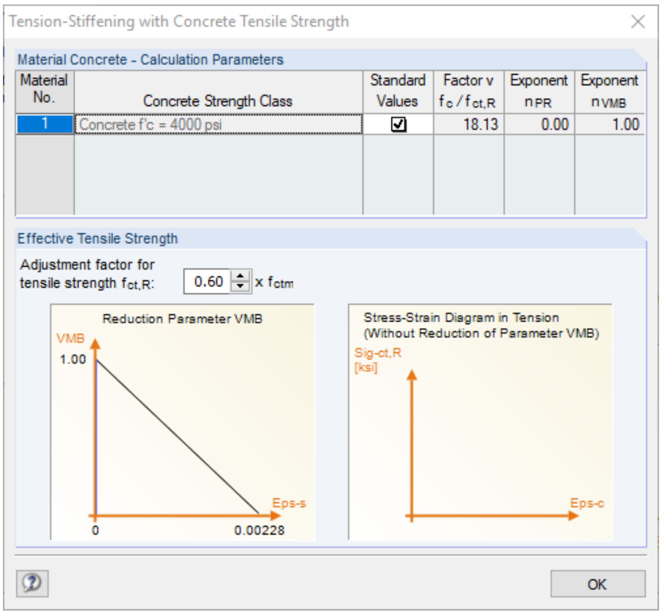


Figure 2.9: Dialog box *Tension Stiffening with Concrete Tensile Strength*

Modifications of the parameters will be immediately displayed in the diagrams below.

For more information on *tension stiffening*, see the general manual of RF-CONCRETE Surfaces.

2.2 Materials

Window *1.2 Materials* consists of two parts. The upper table lists the design-relevant concrete and steel grades. All materials of the 'Concrete' category that are used in RFEM for surfaces are preset. In the *Material Properties* section below, the properties of the current material (i.e. the row of the material selected in the upper table) are displayed.

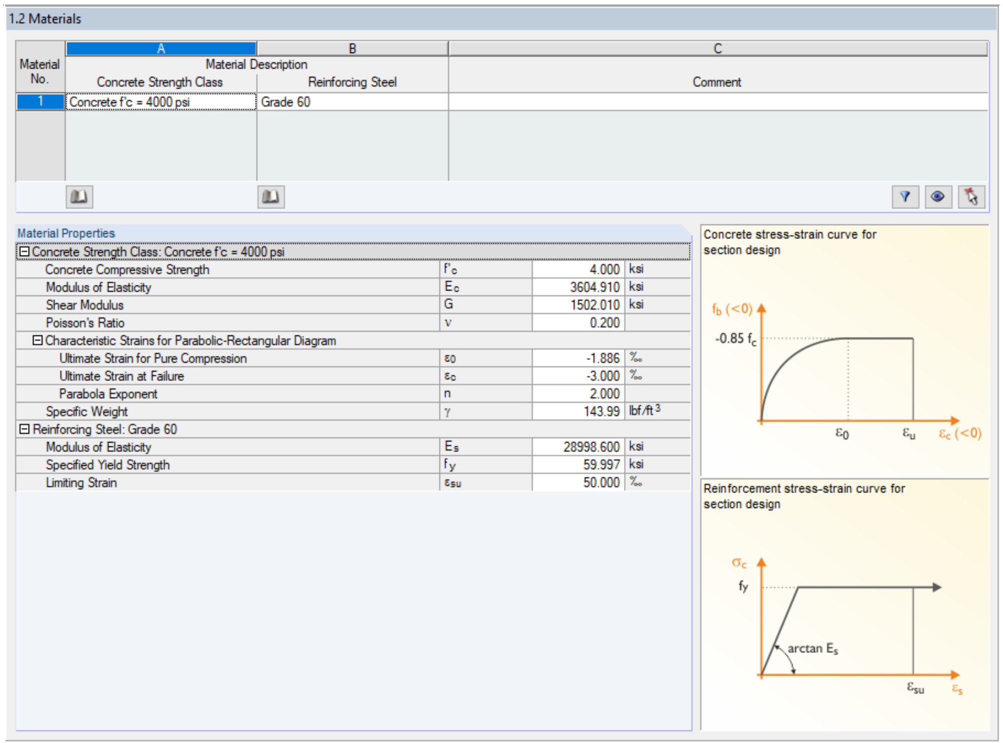


Figure 2.10: Window *1.2 Materials*

The table lists only materials selected for the design. Materials that are not allowed are highlighted in red; modified materials appear blue in color.

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

To adjust the units and decimal places of material properties and stresses, select **Units and Decimal Places** on the **Settings** menu.

Material Description

Concrete Strength Class

The concrete materials used in RFEM are preset; materials that are not relevant are hidden. It is always possible to modify the strength class: Click the material in column A to select the cell. Then, click [▼] or press [F7] to open the list of the strength classes.

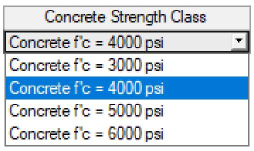


Figure 2.11: List of concrete strength classes

The list contains only strength classes complying with the design concept of ACI 318-14.

After the transfer, the program shows the updated design-relevant *Material Properties*.

As a matter of principle, the material properties cannot be edited in RF-CONCRETE Surfaces.

Reinforcing Steel

In this column, the program presets a steel grade that corresponds to the design concept of ACI 318-14.

Similarly to the concrete strength class, you can select some different reinforcing steel via the drop-down list. Select the material in column B to activate the cell. Then, click [▼] or press [F7] to open the list of reinforcing steels.

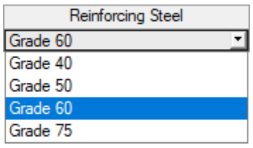


Figure 2.12: List of reinforcing steels

The list contains only steel grades that are relevant for the selected standard.

After the transfer, the program updates the *Material Properties*.

Material Library

The material library contains many materials. To access the corresponding material library, click the button shown on the left. The [Library] buttons are located below columns A and B. They open the libraries of the concrete strength classes or reinforcing steels, respectively.

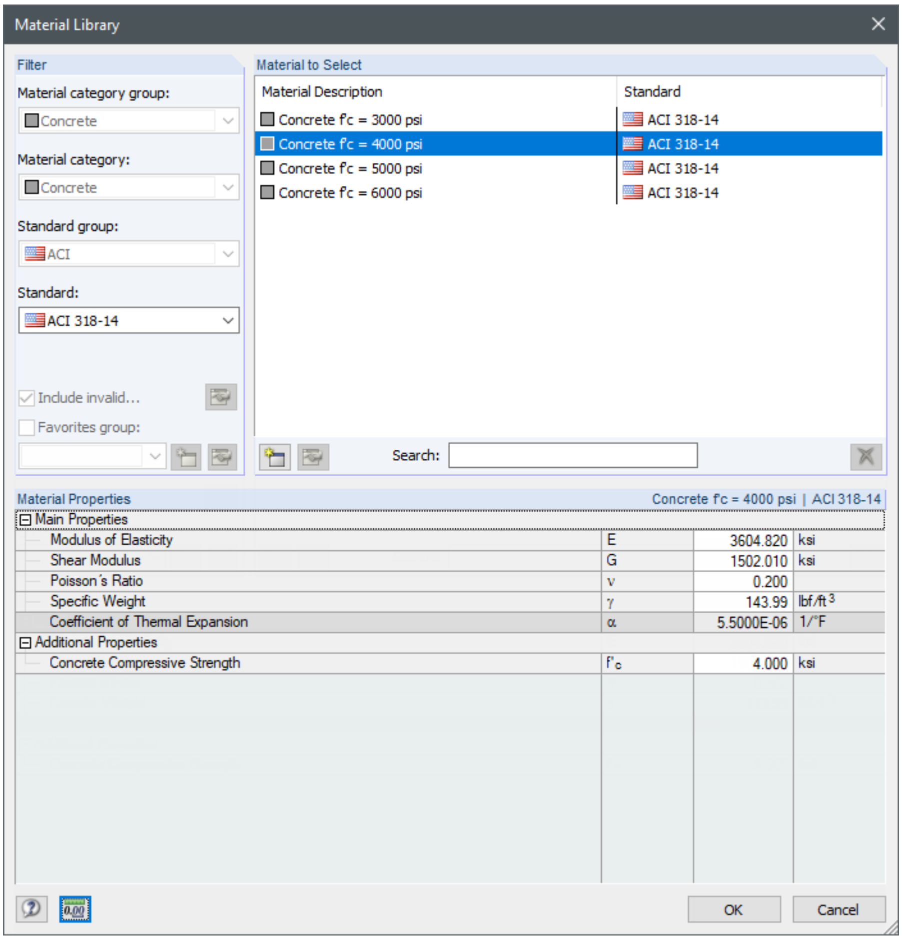


Figure 2.13: Dialog box *Material Library* showing *Concrete* materials

In the *Filter* section, the materials relevant for the Standard are preset, thus excluding all other categories or standards. You can select the appropriate concrete strength class or steel grade from the *Material to Select* list and check its *Material Properties* in the section below.

Click [OK] or press [↵] to transfer the selected material to Window *1.2 Materials*.

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

2.3 Surfaces

Window *1.3 Surfaces* lists the surfaces that are relevant for the design.

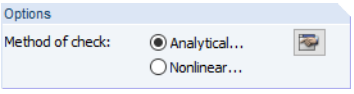
The makeup of the window depends on the settings in Window *1.1 General Data*: If you design only the strength limit state, the table lists only the surfaces with their thicknesses. If you have selected load cases for the serviceability limit state design as well (see Figure 2.5, page 9), more specific options will be available. They depend on the selected SLS design method.

The buttons below the table have the following functions:

Button	**Function**
	Shows only surfaces that are assigned to a reinforcement group in Window *1.4 Reinforcement* (see Chapter 2.4)
	Jumps to the RFEM work window to adjust the view
	Allows you to select a surface in the RFEM work window

Table 2.2: Buttons in Window *1.3 Surfaces*

2.3.1 Analytical Method



The general manual of RF-CONCRETE Surfaces gives you a description of the analytical method for the serviceability limit state design. If you use RF-CONCRETE Deflect, this window provides additional tabs and columns. They are described in Chapter 2.3.2 *Nonlinear Method*.

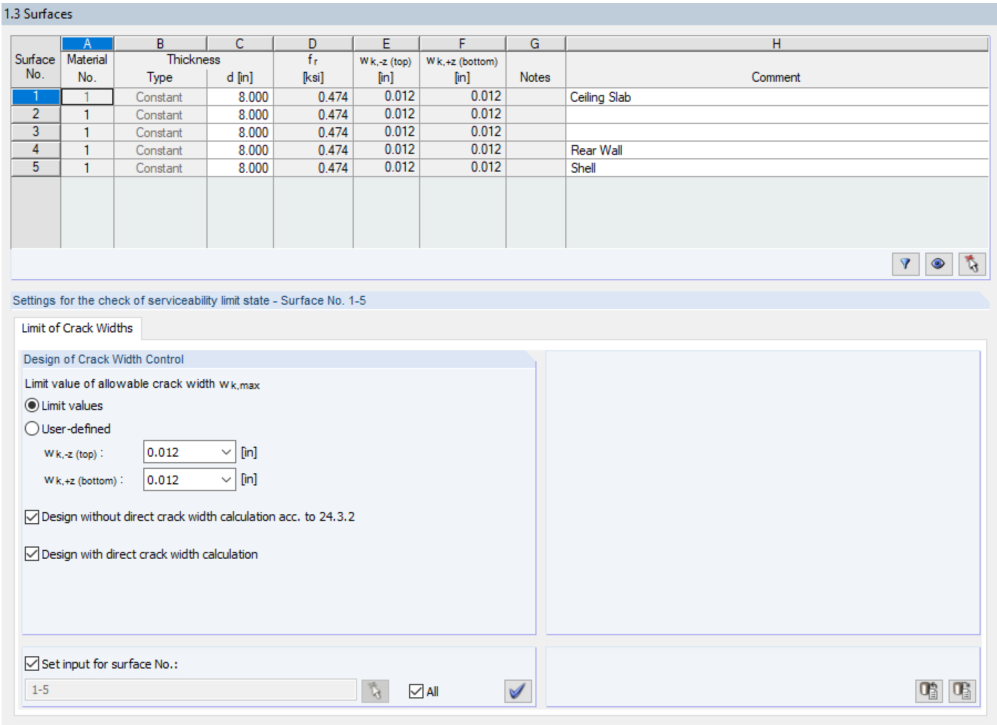


Figure 2.14: Window *1.3 Surfaces* with settings for analytical check method, tab *Limit of Crack Widths*

Material No.

For every surface, the table shows the material numbers which are managed in Window *1.2 Materials*.

Thickness

Type

The program can design constant and linearly variable thickness types as well as surfaces with orthotropic properties.

d

This column shows the surface thicknesses defined RFEM. The values can be changed for the design.

If the surface thicknesses are modified, the internal forces of RFEM are nevertheless used for the design. They result from the stiffnesses of the surface thicknesses defined in RFEM. In a statically indeterminate system, the surface thicknesses modified in RF-CONCRETE Surfaces must, therefore, be adjusted in RFEM as well. By that, the distribution of internal forces will be correctly considered in the design.

The headers of the other table columns depend on the settings in the tabs below. They again can be controlled in the *Settings* dialog box (see Figure 2.6, page 10) where you specify whether deflections and/or cracks are to be designed.

The values in the columns are based on the entries in the tabs below. Those specifications will be applied to all surfaces by default. It is possible, however, to assign the current specifications only to specific surfaces. Clear the selection in the *All* check box. Then, enter the numbers of the relevant surfaces or select them graphically in the RFEM work window after clicking [↖]. With [☑], you assign the current settings to the selected surfaces. Note that the assignment is applicable only for the active tab, for example *Stress Check*.

f_r

This column displays the respective value of the effective concrete tensile strength.

$w_{k,-z\,(top)}$ / $w_{k,+z\,(bottom)}$

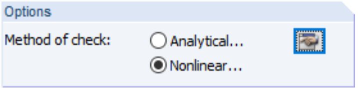
These parameters are allowable crack widths at the top and bottom sides of the surfaces (see general manual of RF-CONCRETE Surfaces). They can be defined in the *Limit of Crack Widths* tab (see Figure 2.14).

Notes

This column shows remarks in the form of footers that are described in detail in the status bar.

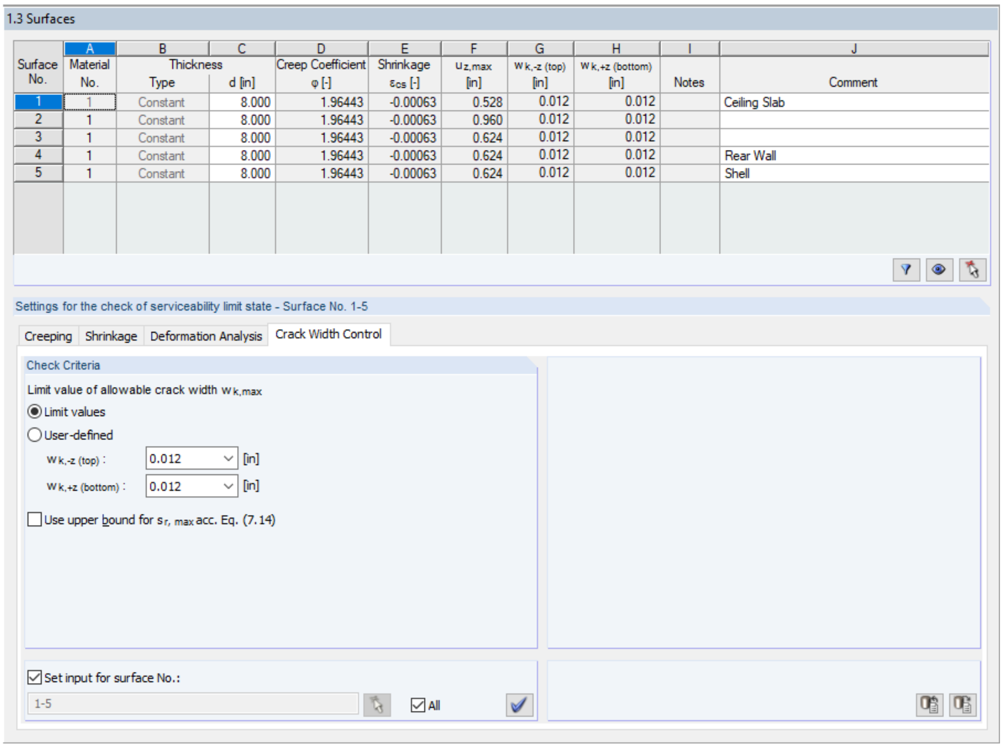
Comment

Use this text box to enter your comments.



2.3.2 Nonlinear Method

To design according to the *Nonlinear* method, you need a license of the **RF-CONCRETE NL** add-on module. This method for the serviceability limit state design is described in detail in the general manual of RF-CONCRETE Surfaces.



Surface No.	Material No.	Thickness Type	d [in]	Creep Coefficient φ [-]	Shrinkage εcs [-]	uz,max [in]	wk,-z (top) [in]	wk,+z (bottom) [in]	Notes	Comment
1	1	Constant	8.000	1.96443	-0.00063	0.528	0.012	0.012		Ceiling Slab
2	1	Constant	8.000	1.96443	-0.00063	0.960	0.012	0.012		
3	1	Constant	8.000	1.96443	-0.00063	0.624	0.012	0.012		
4	1	Constant	8.000	1.96443	-0.00063	0.624	0.012	0.012		Rear Wall
5	1	Constant	8.000	1.96443	-0.00063	0.624	0.012	0.012		Shell

Figure 2.15: Window *1.3 Surfaces* with settings for nonlinear method of check, tab *Crack Width Control*

The following columns are described in Chapter 2.3.1 *Analytical Method* above:

- Material
- Thickness
- $w_{k,-z}$ / $w_{k,+z}$

For orthotropic surfaces, no serviceability limit state design according to the nonlinear method is possible.

The values in the columns D through H are controlled in the tabs below. By default, the settings in these tabs will be applied to all surfaces. It is possible, however, to assign the current specifications only to specific surfaces. Clear the selection of the *All* check box. Then, enter the number of the relevant surfaces or use [↖] to select them graphically. With [☑], you assign the current settings to the selected surfaces. Note that the assignment is applicable only for the current tab (for example *Crackk Width Control*).

Creep Coefficient φ

The parameters for creep are defined in the *Creeping* tab. Based on these conditions, the program determines the creep coefficient φ. For the notional size of the member, h_0, the program applies the surface thickness, d.

The determination of the creep coefficient is described in Chapter 2.8.4.1 of the general manual of RF-CONCRETE Surfaces.

Shrinkage ε_{cs}

This column shows the shrinkage whose parameters can be defined in the *Shrinkage* tab. Based on the boundary conditions, the program determines the appropriate shrinkage ε_{cs}. For the notional size of the structural component, h_0, the program assumes the surface thickness, d, for the calculation.

The determination of shrinkage is described in Chapter 2.8.4.2 of the general manual of RF-CONCRETE Surfaces. If you do not want to apply any shrinkage strain to a surface, set zero for the user-defined shrinkage strain in the *Shrinkage* tab, and then apply it to the surface.

For pure plates that are defined as model type *2D - XY ($u_Z/\varphi_X/\varphi_Y$)*, it is not possible to consider shrinkage. There are only degrees of freedom for bending.

$u_{z,max}$

This value represents the maximum allowable deformation with respect to the serviceability limit state design. The design criteria are defined in the *Deformation Analysis* tab.

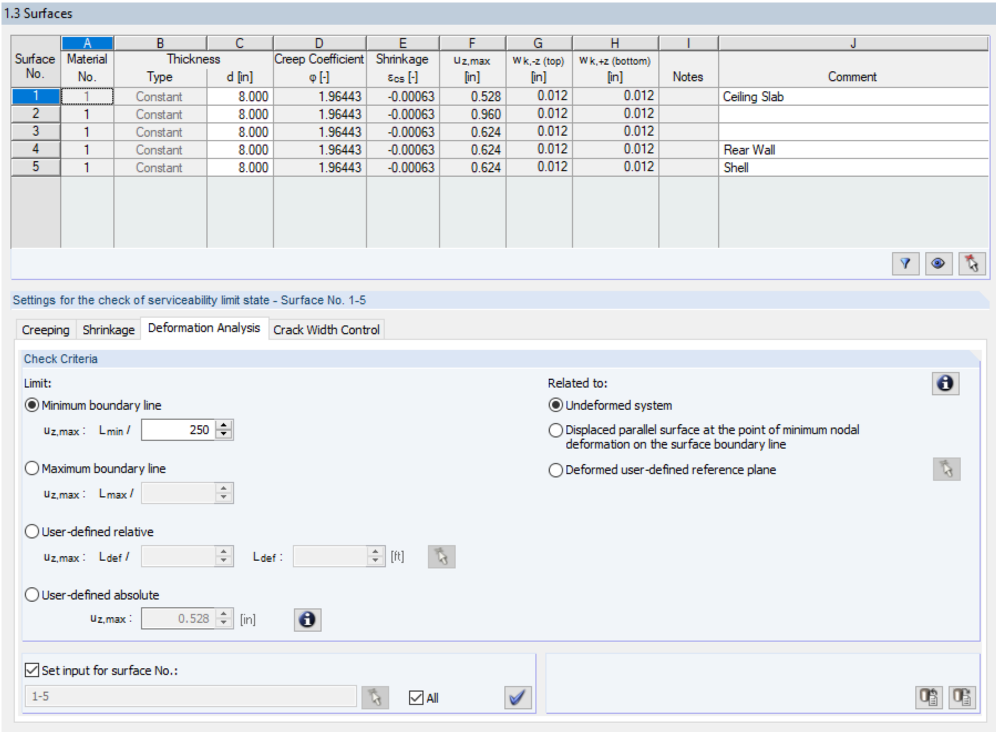


Figure 2.16: Window *1.3 Surfaces*, tab *Deformation Analysis*

Limit

To ensure the serviceability limit state according to [2] 24.2, the deflections must stay within the limit values given in [2] Table 24.2.2.

Member	Condition		Deflection to be considered	Deflection limitation
Flat roofs	Not supporting or attached to nonstructural elements likely to be damaged by large deflections		Immediate deflection due to maximum of L_r, *S*, and *R*	$\ell/180$[1]
Floors			Immediate deflection due to *L*	$\ell/360$
Roof or floors	Supporting or attached to nonstructural elements	Likely to be damaged by large deflections	That part of the total deflection occurring after attachment of nonstructural elements, which is the sum of the time-dependent deflection due to all sustained loads and the immediate deflection due to any additional live load[2]	$\ell/480$[3]
		Not likely to be damaged by large deflections		$\ell/240$[4]

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

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

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

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

Figure 2.17: [2] Table 24.2.2 – Maximum permissible calculated deflections

The options *Minimum border line*, *Maximum border line*, and *User-defined relative* determine which effective length l_{eff} is used. For the two *Border line* options, the program applies the shortest or greatest border line of the respective surface.

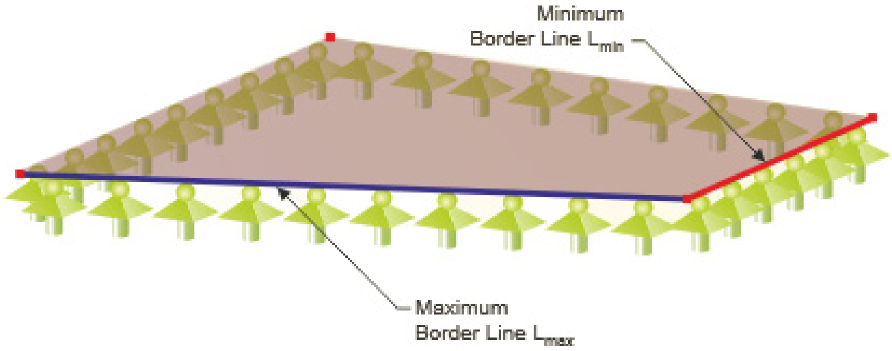


Figure 2.18: Maximum and minimum border line for determination of $u_{z,max}$

For the *User-defined relative* limit, you can enter the length directly or select it graphically between two points in the RFEM model by using [↖].

For all three options, you have to define a divisor by which the reference lengths are divided.

Alternatively, you can also specify the allowable maximum deformation $u_{z,max}$ as *User-defined absolute*.

Related to

The deformation design criterion uses the deflection of a surface – the vertical deformation relative to the shortest line connecting the points of support. The *Deformation Analysis* tab (Figure 2.16) offers three possibilities how to calculate the local deformation $u_{z,local}$ used in the design.

- *Undeformed system*: The deformation is related to the initial structure.
- *Displaced parallel surface*: This option is recommended for an elastic support of the surface. The deformation $u_{z,local}$ is related to a virtual reference surface that is displaced parallel to the undeformed system. The displacement vector of the reference surface is as long as the minimal nodal deformation within the surface.

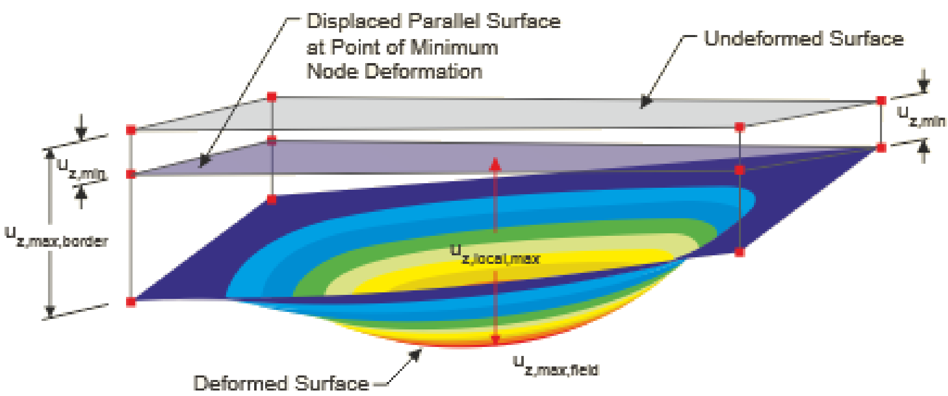


Figure 2.19: Displaced parallel surface (displacement vector: smallest nodal deformation $u_{z,min}$)

- *Deformed reference plane*: If the support deformations of a surface differ considerably from each other in size and degree, you can define an inclined reference plane for the deformation $u_{z,local}$ to be checked. You have to define this plane by three points of the undeformed system. The program determines the deformation of the three definition points, places the reference plane in those displaced points, and calculates the local deformation $u_{z,local}$.

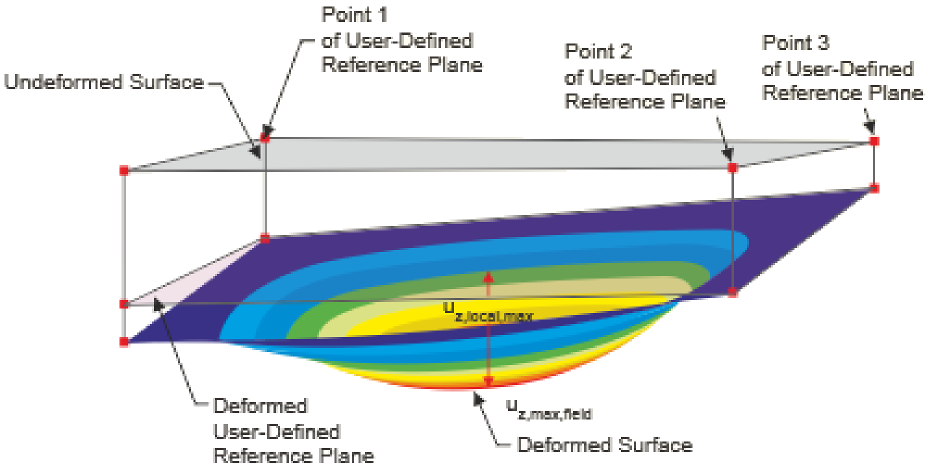


Figure 2.20: Displaced user-defined reference plane

2.4 Reinforcement

This window consists of five tabs where all reinforcement data is specified. As the individual surfaces often require different settings, define so-called "reinforcement groups" can be defined in every design case. Each reinforcement group contains the reinforcement parameters that are applied to particular surfaces.

Reinforcement Group

To create a new reinforcement group, click [New] in the *Reinforcement Group* section. The number is automatically assigned. A user-defined *Description* helps you to overlook all reinforcement groups of each design case.

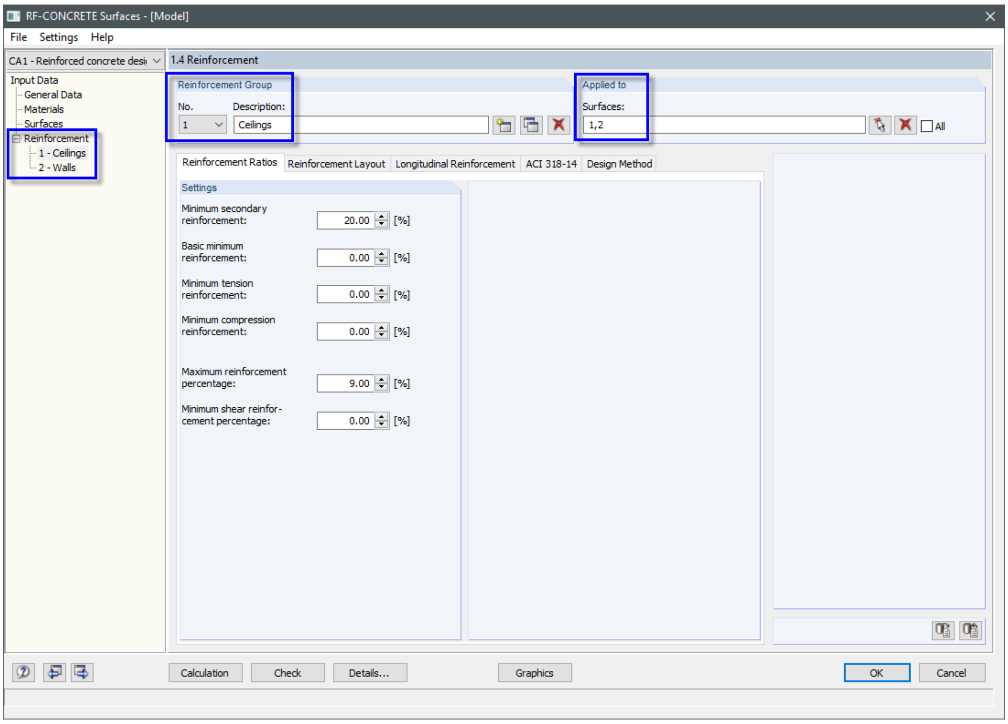
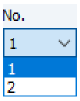


Figure 2.21: Window *1.4 Reinforcement* with two reinforcement groups



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

By using the [Delete] button, the currently selected reinforcement group is deleted from the design case without any further warning. Hence, surfaces contained in that reinforcement group will not be designed. If you want to design them, you have to reassign them to a new or existing reinforcement group.

Applied to Surfaces

In this section, you specify the surfaces to which the parameters of the current reinforcement group apply. By default, *All* surfaces are set. If this check box is selected, it is not possible to create any further reinforcement groups. The reason is that a surface cannot be designed according to different rules (this is only possible via "design cases", see item *New Case* on the *File* menu). Therefore, clear the *All* check box when you want to use several reinforcement groups.

In the box, enter the number of the surface(s) to which the reinforcement parameters of the tabs below apply. You can also select them graphically in the RFEM work window by using the [↖] function. Only surface numbers that have not yet been assigned to other reinforcement groups can be entered.

2.4.1 Reinforcement Ratios

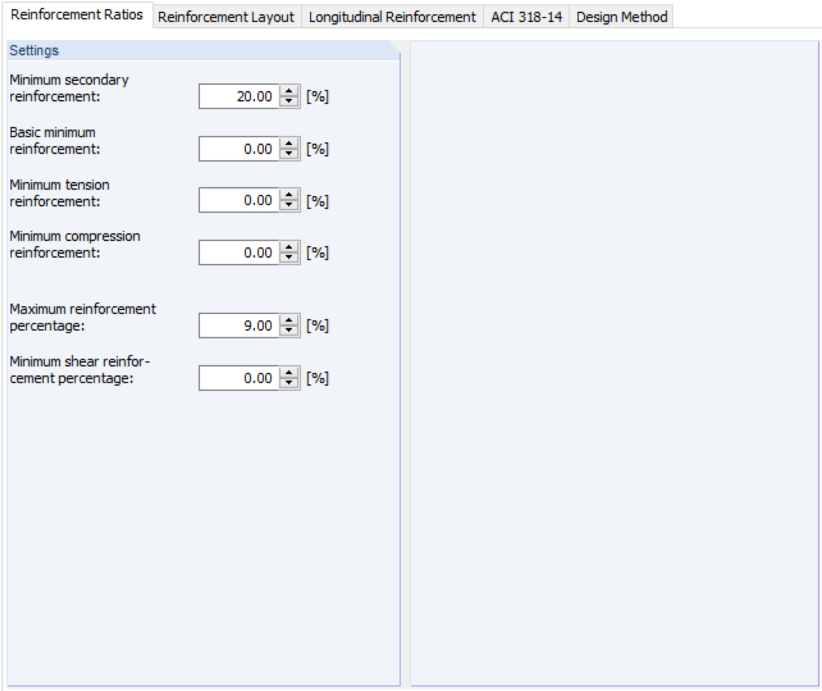


Figure 2.22: Window *1.4 Reinforcement*, tab *Reinforcement Ratios*

This tab controls the minimum and maximum reinforcements in percentages. The *Minimum secondary reinforcement* relates to the maximum longitudinal reinforcement that is applied. All other reinforcement ratios are related to the unit cross-sectional area of the concrete surface.

2.4.2 Reinforcement Layout

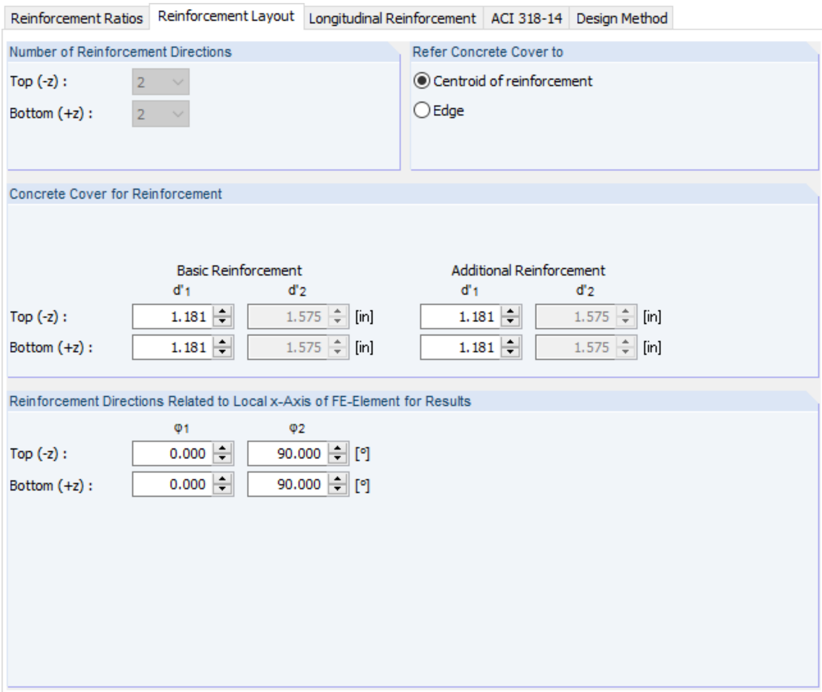
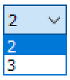


Figure 2.23: Window *1.4 Reinforcement*, tab *Reinforcement Layout*

This tab controls the geometric specifications of the reinforcement.



Number of Reinforcement Directions

The reinforcement mesh can be defined with two or three reinforcement directions for each surface side.

For serviceability limit state designs, only a reinforcement mesh with two directions is allowed.

The definition of the "top" and "bottom" sides of the surface is given in the *Concrete Cover for Reinforcement* section below.

Refer Concrete Cover to

The concrete covers that are defined in the *Concrete Cover for Reinforcement* section can be related to the *Centroid* or *Edge* distances of the reinforcement.

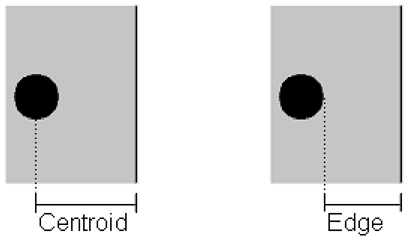


Figure 2.24: Reference of the concrete cover

If you select the *Edge* option, you have to specify the *Bar diameter D*.

Concrete Cover for Reinforcement

For both sides of the surface, you specify the concrete covers of the *Basic Reinforcement* and, if necessary, the *Additional Reinforcement*. The dimensions represent either the centroids, *d*, of the individual layers or the edge distances, c_c, of the reinforcement in direction φ_1. The reinforcement directions can be defined in the dialog box section below.

The "top" and "bottom" surface sides are defined as follows: The bottom surface is defined in direction of the positive local surface axis z, the top side in direction of the negative local axis z.

The RFEM graphic shows you the xyz-coordinate systems of the surfaces as soon as you move the pointer across a surface. You can also use the shortcut menu of a surface (right-click it) to switch the axes on and off.

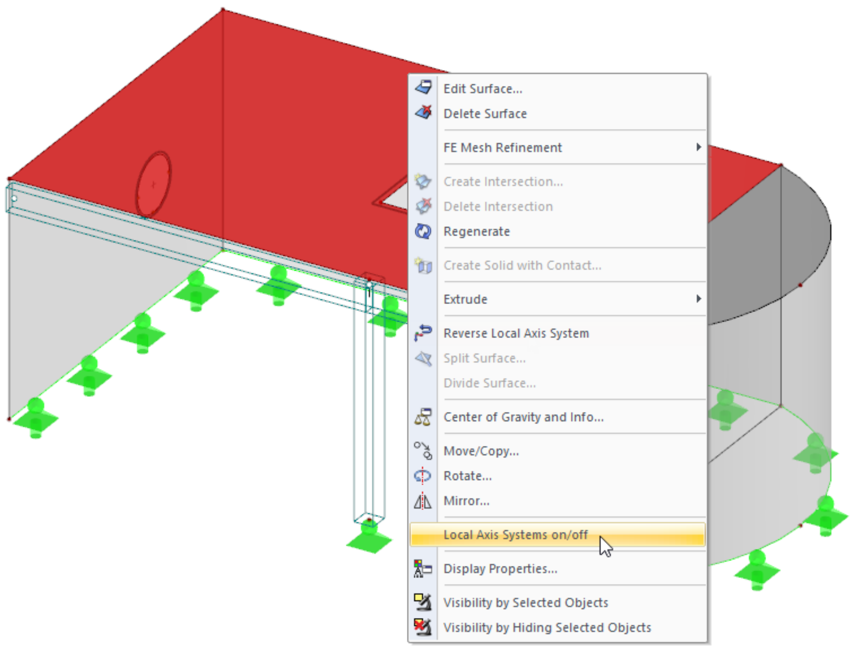
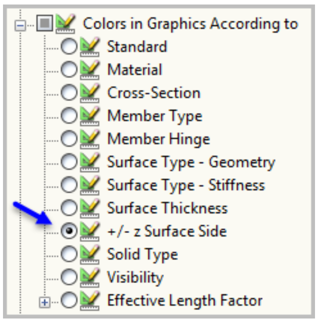


Figure 2.25: RFEM shortcut menu of a surface



To display the surface sides in different colors, select *Colors in Graphics According to* ***+/- z-Surface Side*** in the *Display* navigator (see figure on the left).

You can change the orientation of the local z-axis of a surface by using the *Reverse Local Axis System* option on the shortcut menu (see Figure 2.25). In this way, it is possible to unify, for example, the orientation of walls to assign the top and bottom reinforcement sides for vertical surfaces consistently.

The wall model types *2D - XZ ($u_X/u_Z/\varphi_Y$)* and *2D - XY ($u_X/u_Y/\varphi_Z$)* handle models whose surface planes are subjected to compression or tension exclusively. In those cases, it is not possible to create different reinforcement meshes for each surface side so that the input is limited to uniform concrete covers on both sides.

Reinforcement Directions Related to Local Axis x of FE-Element

The reinforcement directions φ are related to the local x-axes of the finite elements.

In the *Edit Surface* dialog box of RFEM, you can check and, if necessary, adjust the axis systems for the results of the surfaces.

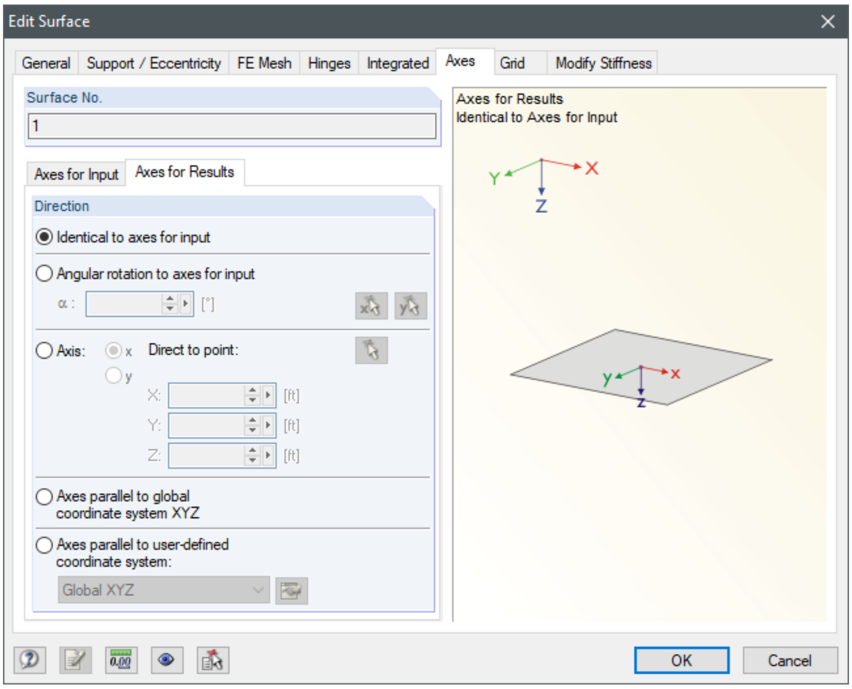


Figure 2.26: RFEM dialog box *Edit Surface*, tabs *Axes* and *Axes for Results*

For curved surfaces, it is recommended to check the axes of the finite elements graphically: In the *Display* navigator of RFEM, select the option *FE Mesh* → *On Surfaces* → *FE Axis Systems x,y,z* → *Indexes* (see Figure 8.41 in Chapter 8.15 of the RFEM manual).

The reinforcement directions are to be specified by means of the angle φ for each layer. Only positive angles are allowed. They represent the respective clockwise rotation of the reinforcement direction in relation to the corresponding x-axis.

For the wall model types *2D - XZ ($u_X/u_Z/\varphi_Y$)* and *2D - XY ($u_X/u_Y/\varphi_Z$)*, it is not possible to create different reinforcement meshes for each side of the surface. Thus, the input is limited to uniform reinforcement directions on both surface sides.

2.4.3 Longitudinal Reinforcement

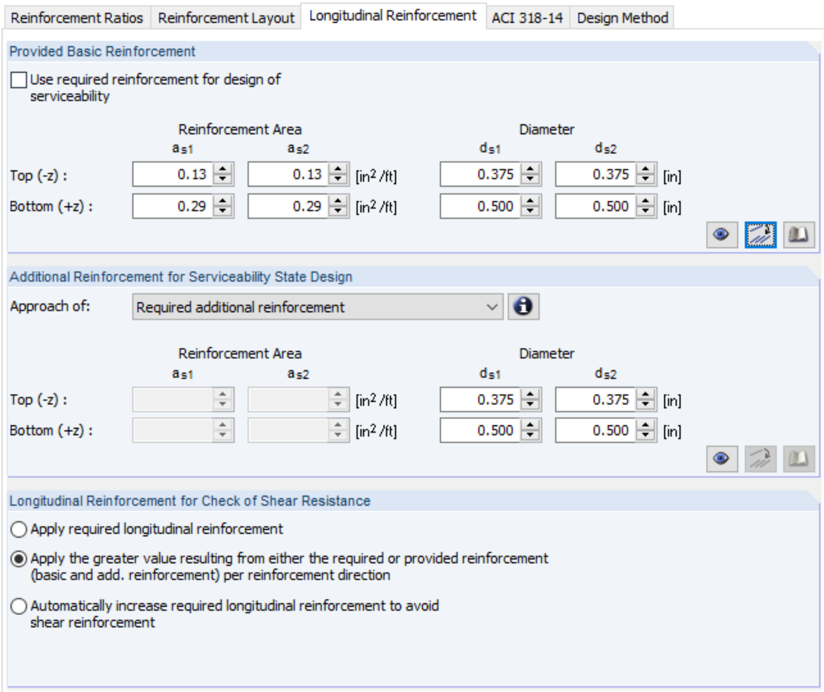


Figure 2.27: Window *1.4 Reinforcement*, tab *Longitudinal Reinforcement* (strength and serviceability limit state design)

The sections of the tab depend on the design selected in Window *1.1 General Data*: If you want to carry out the strength limit state design exclusively, no specific reinforcement settings are required. You only need to decide which longitudinal reinforcement is to be used for the *Check of Shear Resistance*. For the serviceability limit state design, however, you have to specify the reinforcement areas.

For more information on the reinforcement specifications concerning the SLS design, see the general manual of RF-CONCRETE Surfaces.

Provided Basic Reinforcement

For each surface side and each reinforcement direction, you can define a basic reinforcement that will be used for all surfaces of the reinforcement group. Enter the *Reinforcement Area* and the *Diameter* (required for the SLS design) in the corresponding boxes.

If the user-defined basic reinforcement exceeds the required one, no additional reinforcement is needed. High values of basic reinforcement are not customary, however. This it would be inefficient.

Entering reinforcement areas is facilitated by libraries for rebars and mesh reinforcements. To access them, use the two buttons shown on the left. They are described on the following page.

Rebars

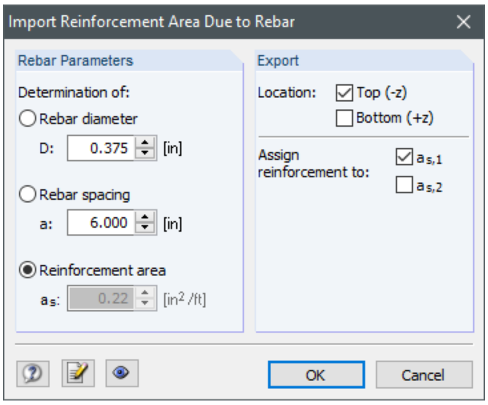


Figure 2.28: Dialog box *Import Reinforcement Area Due to Rebar*

The three options in the *Rebar Parameters* section are interactive. Normally, the program determines the reinforcement area from the rebar diameter and the rebar spacing.

In the *Export* section, you decide to which text boxes of the *Longitudinal Reinforcement* tab the calculated reinforcement areas are to be applied. The location and the reinforcement direction can be defined specifically (or generally by selecting all check boxes).

Mesh Reinforcement

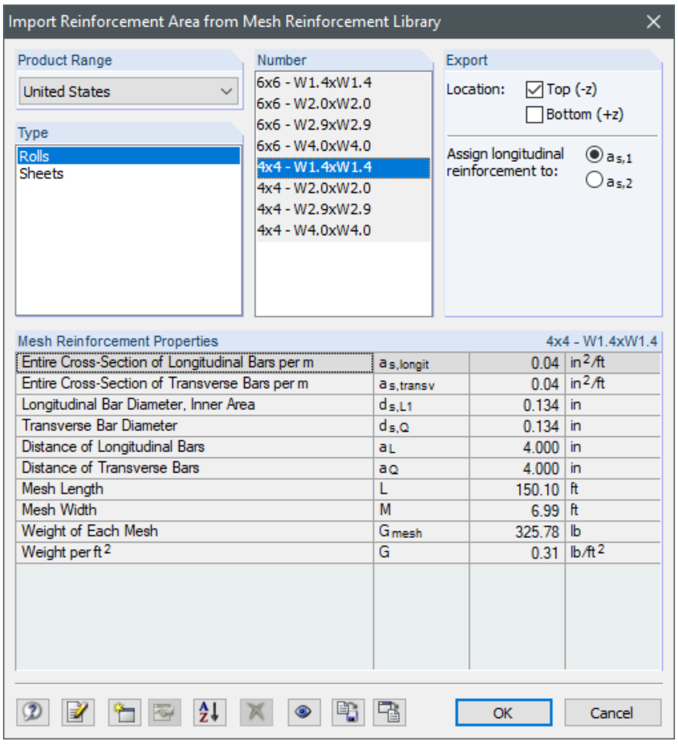
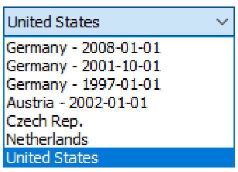


Figure 2.29: Dialog box *Import Reinforcement Area from Mesh Reinforcement Library*



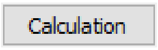
First, select the *Product Range* from the drop-down list shown on the left. Then, define the mesh *Type* and select the relevant *Number* in the section to the right. In the section below, you can check the *Mesh Reinforcement Properties*.

In the *Export* section, you decide to which text boxes of the *Longitudinal Reinforcement* tab the reinforcement areas are to be applied. The location and the reinforcement direction can be defined specifically.

Use required reinforcement for design of serviceability

The ideal approach to perform the SLS design would be the following:

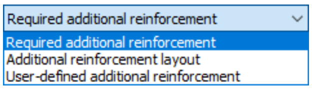
1. Determine the required reinforcement by using only the load specified in the *Strength Limit State* tab.
2. Create a reinforcement drawing including mesh reinforcements and rebars on the basis of the colored result diagrams.
3. If necessary, divide the surfaces based on the reinforcement drawing into smaller surfaces that have the same provided reinforcement area in each reinforcement direction.
4. Define the provided reinforcement area, rebar spacing, and diameter for each surface in RF-CONCRETE Surfaces.
5. Recalculate with the loads in the *Serviceability Limit State* tab.



That procedure is rather complex and contrary to the convention that you can determine the reinforcement and perform the SLS design by simply clicking [Calculation].

Therefore, you can select the option *Apply required longitudinal reinforcement* to quickly obtain a provided reinforcement for the individual surfaces: The program uses the required reinforcement from the strength limit state design as the reinforcement to be applied. If this option is selected, you only need to specify the rebar diameter.

Additional Reinforcement for Serviceability Limit State Design



In all areas where the statically required reinforcement exceeds the basic reinforcement, additional reinforcement is needed. Use the drop-down list to specify which additional reinforcement is applied for the serviceability limit state design.

If you select the *Required additional reinforcement* option, the actual $A_{s,req}$ distribution is applied as additional reinforcement in the SLS design.

The *Additional reinforcement layout* is determined as difference between the greater statically required reinforcement of all surfaces of the reinforcement group and the defined basic reinforcement:

$$a_{s,add} = \max a_{s,req} - a_{s,basic}$$

Equation 2.1

Click [Info] to open a dialog box illustrating the additional reinforcement (see Figure 2.30).

To dimension the additional reinforcement, you only need to specify the rebar diameter.

You can also specify a *User-defined additional reinforcement*. For this, the program offers libraries for the rebars and mesh reinforcements (see description in *Provided Basic Reinforcement* section above).

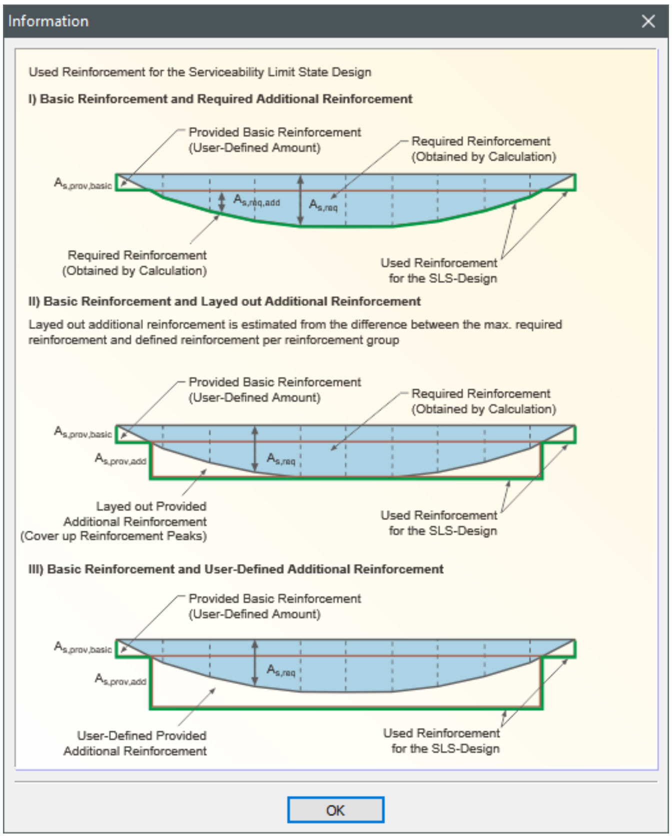


Figure 2.30: Options to apply additional reinforcement

Manual definition of reinforcement areas

As an alternative to the automatic geometric layout of the additional reinforcement used for the SLS design, you can define the areas manually. To activate this option, click the [Details] button and open the *Details* dialog box. Then, select the *Manual definition of the reinforcement areas* in the *Reinforcement* tab.

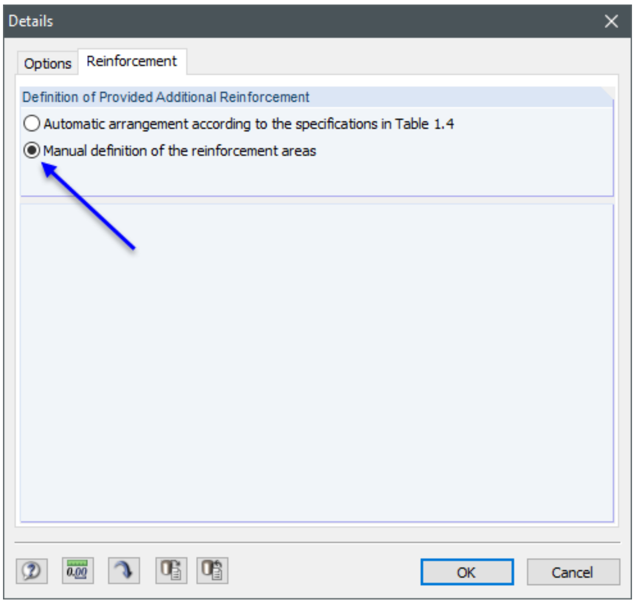


Figure 2.31: Activating manual definition of reinforcement areas in *Details* dialog box

In the *Longitudinal Reinforcement* tab, the dialog section *Provided Additional Reinforcement* is now shown (instead of *Additional Reinforcement for Serviceability State Design*).

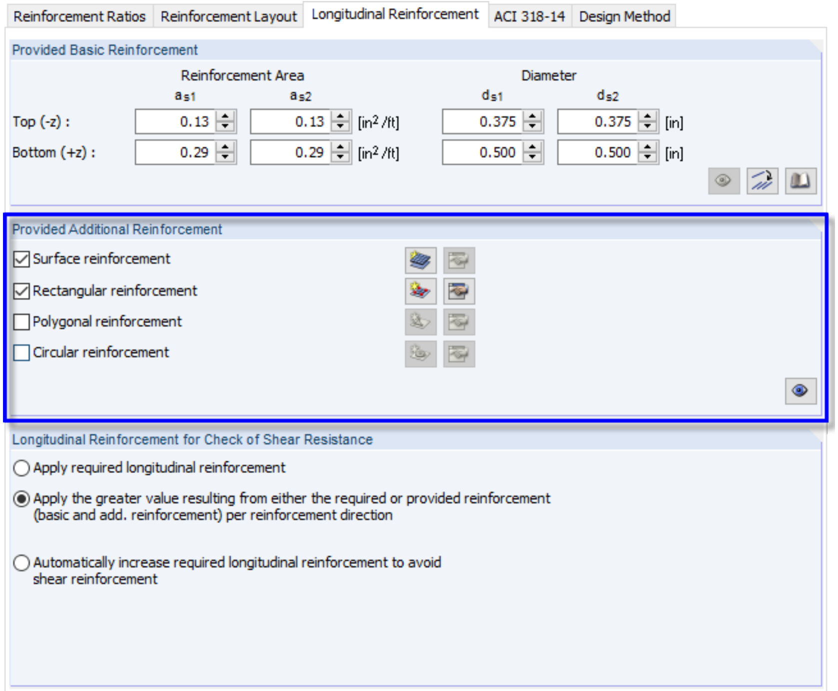


Figure 2.32: Window *1.4 Reinforcement*, tab *Longitudinal Reinforcement*

In the following, the functions are described for a rectangular reinforcement as an example. The explanations apply in the same way to surface, polygonal and circular reinforcements.

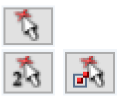
Click the [Apply free rectangular reinforcement] button to open the *New Rectangular Reinforcement* dialog box (see Figure 2.33). There you can define the properties as well as the position of the free reinforcement.

In the dialog section *On Surfaces No.*, you enter the surfaces to be used for the reinforcement. If the *All in RG* check box is selected, the new free reinforcement will be used for all surfaces of the current reinforcement group (RG).

The *Projection Plane* section determines on which plane the reinforcement is applied.

The *Type of Reinforcement* is either a mesh or a rebar reinforcement. You can select the mesh reinforcements in a library that you open with the [Library] button. For the rebar reinforcement, you can use the button seen on the left to determine the reinforcement area by means of rebar diameter, rebar spacing and reinforcement area.

The *Layout of Reinforcement* section controls the arrangement of the reinforcement. You have to specify the surface side as well as the direction of the reinforcement or the mesh main reinforcement. The concrete cover of the additional reinforcement is taken from the settings in the *Reinforcement Layout* tab; it cannot be changed here.



The *Reinforcement Position*, which means the region of the reinforcement, is defined by the coordinates of two points. Enter them directly, or select them with the [Pick] button in the work window. You can also draw a rectangular window either by selecting two corner points or using the rectangle's center point.

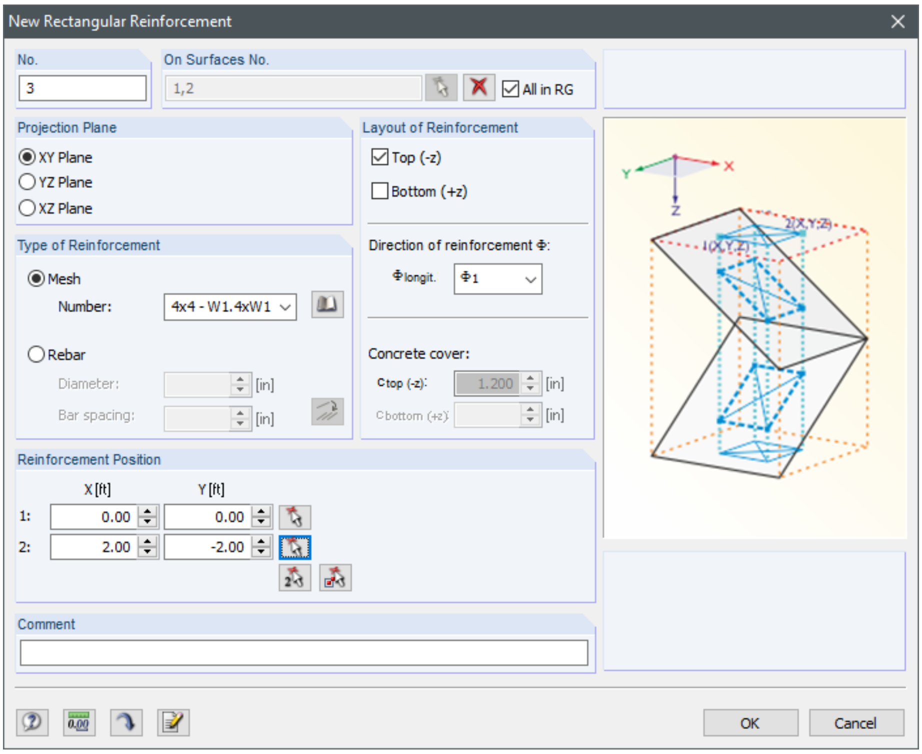
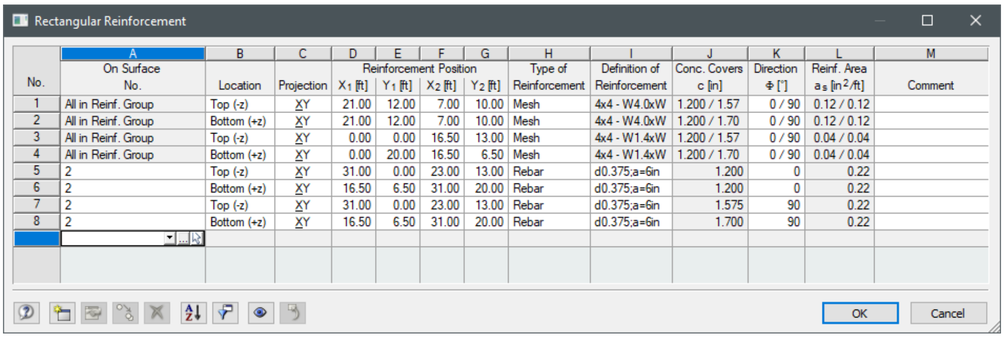


Figure 2.33: Dialog box *New Rectangular Reinforcement*

Note the following when you define the reinforcement position: The free reinforcement will be considered for a finite element when the rectangle includes the center of this element.

If two reinforcement areas lie one upon the other, the values in the respective elements will be added.

When you have defined the reinforcement, the [Edit] button is accessible in the *Provided Additional Reinforcement* section (see Figure 2.32). It opens a table where you can modify the reinforcement.


Rectangular Reinforcement

	A	B	C	D	E	F	G	H	I	J	K	L	M
No.	On Surface No.	Location	Projection	Reinforcement Position X1 [ft]	Y1 [ft]	X2 [ft]	Y2 [ft]	Type of Reinforcement	Definition of Reinforcement	Conc. Covers c [in]	Direction Φ [°]	Reinf. Area a_s [in²/ft]	Comment
1	All in Reinf. Group	Top (-z)	XY	21.00	12.00	7.00	10.00	Mesh	4x4 - W4.0xW	1.200 / 1.57	0 / 90	0.12 / 0.12	
2	All in Reinf. Group	Bottom (+z)	XY	21.00	12.00	7.00	10.00	Mesh	4x4 - W4.0xW	1.200 / 1.70	0 / 90	0.12 / 0.12	
3	All in Reinf. Group	Top (-z)	XY	0.00	0.00	16.50	13.00	Mesh	4x4 - W1.4xW	1.200 / 1.57	0 / 90	0.04 / 0.04	
4	All in Reinf. Group	Bottom (+z)	XY	0.00	20.00	16.50	6.50	Mesh	4x4 - W1.4xW	1.200 / 1.70	0 / 90	0.04 / 0.04	
5	2	Top (-z)	XY	31.00	0.00	23.00	13.00	Rebar	d0.375;a=6in	1.200	0	0.22	
6	2	Bottom (+z)	XY	16.50	6.50	31.00	20.00	Rebar	d0.375;a=6in	1.200	0	0.22	
7	2	Top (-z)	XY	31.00	0.00	23.00	13.00	Rebar	d0.375;a=6in	1.575	90	0.22	
8	2	Bottom (+z)	XY	16.50	6.50	31.00	20.00	Rebar	d0.375;a=6in	1.700	90	0.22	

Figure 2.34: Table *Rectangular Reinforcement*

The buttons below the table have the following functions:

Button	Function
	Creates a new free reinforcement area
	Allows for editing the selected reinforcement
	Moves or copies the selected reinforcement
	Deletes the selected reinforcement
	Sorts the table entries by location
	Opens the dialog box *Reinforcement Filter*
	Switches to RFEM work window for changing the view
	Turns on and off the synchronization in the graphic

Table 2.3: Buttons in Table *Rectangular Reinforcement*

Click the [Filter] button to open the dialog box shown in Figure 2.35. You can filter the table entries by *Surface* numbers, reinforcement *Location* and *Type of reinforcement*. By hiding particular properties, you can get a clear overview.

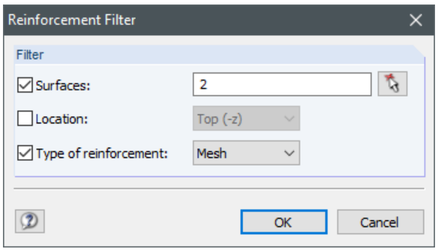
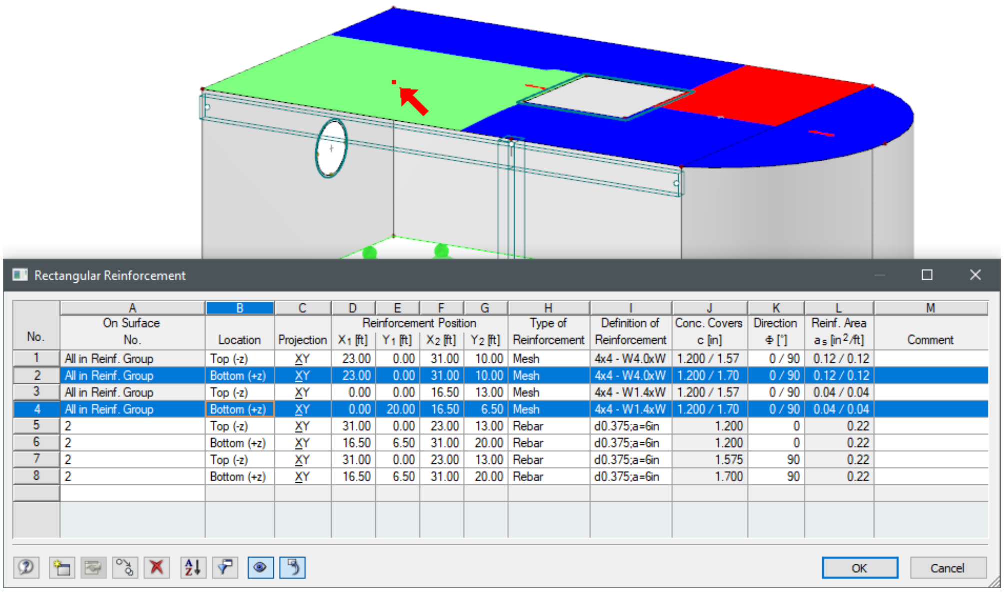


Figure 2.35: Dialog box *Reinforcement Filter*

If the [Synchronization] is enabled after the calculation, the RFEM graphic shows only those reinforcement areas that are selected in the table. This graphical representation is also available for several areas when the row numbers are selected while pressing the [Ctrl] key.



No.	A On Surface No.	B Location	C Projection	D Reinforcement Position X1 [ft]	E Y1 [ft]	F X2 [ft]	G Y2 [ft]	H Type of Reinforcement	I Definition of Reinforcement	J Conc. Covers c [in]	K Direction Φ [°]	L Reinf. Area a_s [in^2/ft]	M Comment
1	All in Reinf. Group	Top (-z)	XY	23.00	0.00	31.00	10.00	Mesh	4x4 - W4.0xW	1.200 / 1.57	0 / 90	0.12 / 0.12	
2	All in Reinf. Group	Bottom (+z)	XY	23.00	0.00	31.00	10.00	Mesh	4x4 - W4.0xW	1.200 / 1.70	0 / 90	0.12 / 0.12	
3	All in Reinf. Group	Top (-z)	XY	0.00	0.00	16.50	13.00	Mesh	4x4 - W1.4xW	1.200 / 1.57	0 / 90	0.04 / 0.04	
4	All in Reinf. Group	Bottom (+z)	XY	0.00	20.00	16.50	6.50	Mesh	4x4 - W1.4xW	1.200 / 1.70	0 / 90	0.04 / 0.04	
5	2	Top (-z)	XY	31.00	0.00	23.00	13.00	Rebar	d0.375;a=6in	1.200	0	0.22	
6	2	Bottom (+z)	XY	16.50	6.50	31.00	20.00	Rebar	d0.375;a=6in	1.200	0	0.22	
7	2	Top (-z)	XY	31.00	0.00	23.00	13.00	Rebar	d0.375;a=6in	1.575	90	0.22	
8	2	Bottom (+z)	XY	16.50	6.50	31.00	20.00	Rebar	d0.375;a=6in	1.700	90	0.22	

Figure 2.36: Synchronization with two reinforcement areas selected

After the calculation, the *Reinforcement Covering* item appears in the *Results* navigator. Use the two item options to evaluate how the required reinforcement is covered by the additional reinforcement.

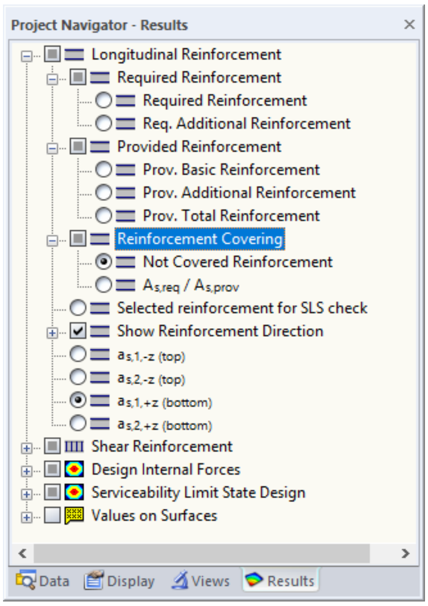


Figure 2.37: Selecting *Reinforcement Covering* in *Results* navigator

When the *Not Covered Reinforcement* option is set, only those areas are highlighted in the model graphic that still need reinforcement.

With the representation of *As,req / As,prov*, any still missing as well as already provided reinforcement is quantified by colored marks.

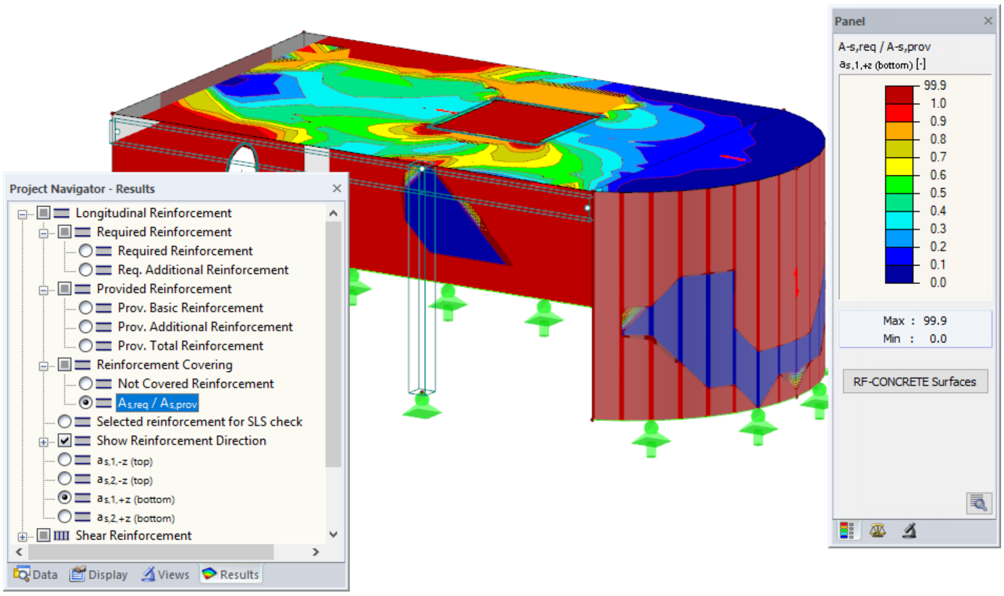


Figure 2.38: Ratios of required to provided reinforcement

Longitudinal Reinforcement for Check of Shear Resistance

The last section of the *Longitudinal Reinforcement* tab provides three options how to apply the longitudinal reinforcement for the shear check without shear reinforcement.

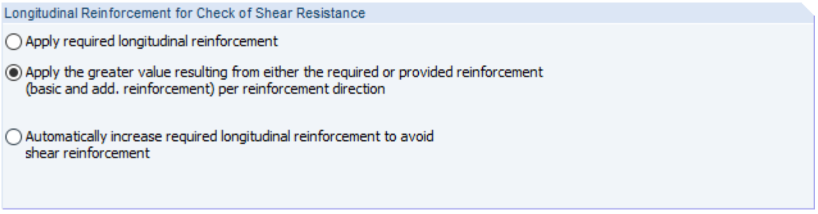


Figure 2.39: Section *Longitudinal Reinforcement for Check of Shear Resistance*

- *Apply required longitudinal reinforcement*
  The check of shear resistance is carried out with the transformed provided tension reinforcement in direction of the principal shear force (see general manual of RF-CONCRETE Surfaces).
- *Apply the greater value resulting from either the required or provided reinforcement*
  For the check of shear resistance, the program uses either the statically required or the user-defined longitudinal reinforcement (see general manual of RF-CONCRETE Surfaces).
- *Automatically increase required longitudinal reinforcement to avoid shear reinforcement*
  If the required longitudinal reinforcement is not sufficient for the shear force resistance, the longitudinal reinforcement will be increased in the main shear force direction until the shear check without shear reinforcement is satisfied (see general manual of RF-CONCRETE Surfaces).

2.4.4 Standard

The parameters of this tab depend on the Standard selected in Window *1.1 General Data*. In this tab, you specify the standard-specific reinforcement data. Here the settings are described for ACI 318-14 [2].

At the bottom right below the table, two buttons are available. Click [Default] to reset the initial values of the current standard. Use [Set as Default] to store the defined entries as new default settings.

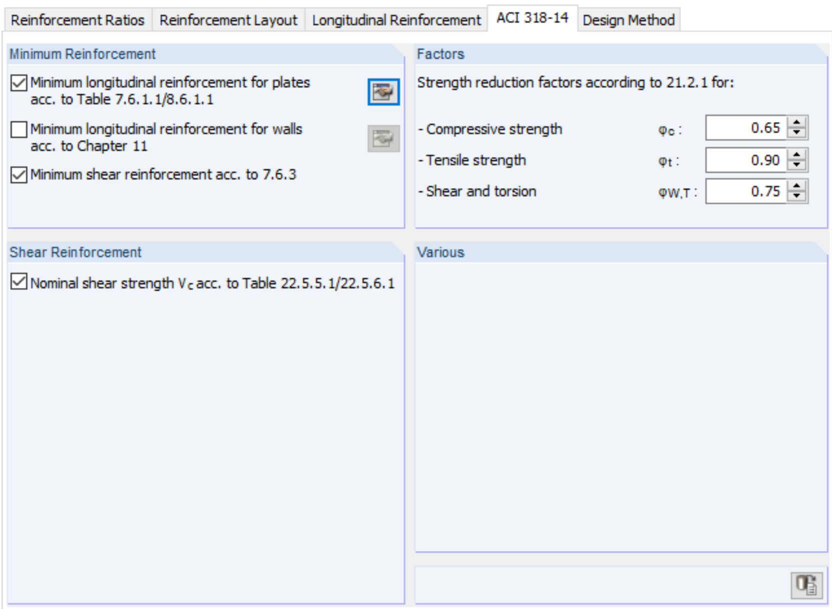


Figure 2.40: Window *1.4 Reinforcement*, tab *ACI 318-14*

Minimum Reinforcement

In this section, you decide which provisions of the standard regarding the minimum reinforcement are to be considered in the design.

For plates and walls, click [Settings] to set the direction of the minimum and main compression reinforcement.

Plates

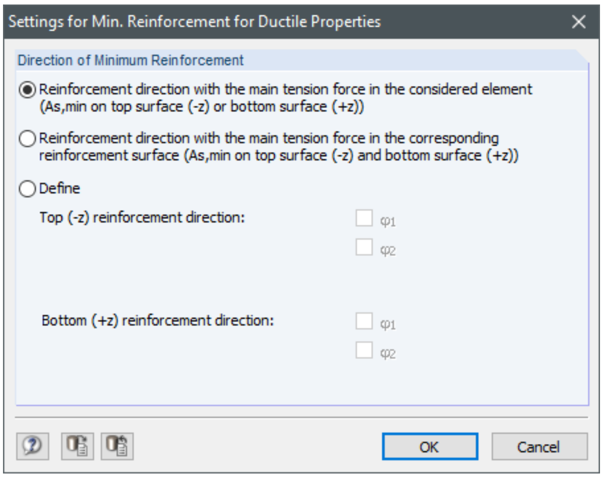


Figure 2.41: Dialog box *Settings for Min. Reinforcement for Ductile Properties*

According to ACI 318-14, 7.6.1.1and 8.6.1.1, the minimum reinforcement must be arranged as close as practicable to the face of the concrete in tension due to applied loads. The main direction of the span cannot be found automatically in the determination of the reinforcement by element. You can control the direction of the reinforcement in which you want to consider the minimum reinforcement, however, by setting one of the following options:

- *Reinforcement direction with main tension force in the considered element*
  The minimum reinforcement is considered only in the reinforcement direction featuring the greatest tension force of all reinforcement directions of both top surface (-z) and bottom surface (+z): The minimum reinforcement is placed only in one direction and on one side of the plate.
- *Reinforcement direction with main tension force in the corresponding reinforcement surface*
  For each reinforcement surface, the program searches for the reinforcement direction with the greatest tension force. Then, the minimum reinforcement is determined on each surface for these directions.
- *Define*
  The reinforcement direction in which you want to apply the minimum reinforcement can be specified manually.

Walls

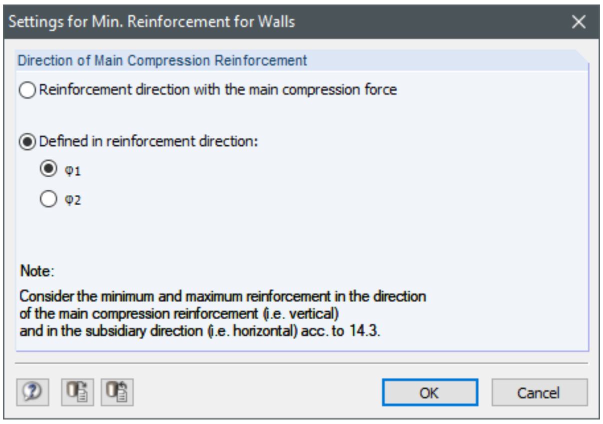


Figure 2.42: Dialog box *Settings for Min. Reinforcement for Walls*

You can specify the direction of the main compression reinforcement to determine the minimum longitudinal reinforcement for walls in the direction of the *main compression force* or *Defined*.

Shear Reinforcement

If the option for *Nominal shear strength* V_c *acc. to Table 22.5.5.1/22.5.6.1* is selected, the shear strength provided by the concrete is calculated with the detailed equations from these tables. Otherwise, the more conservative Equation (22.5.5.1) or Equation (22.5.6.1) is applied.

Factors

The three boxes control the *Strength reduction factors* for compressive strength, tensile strength, and shear and torsion in the design. The values for the different design situations are preset according to [2] Table 21.2.1.

2.4.5 Design Method

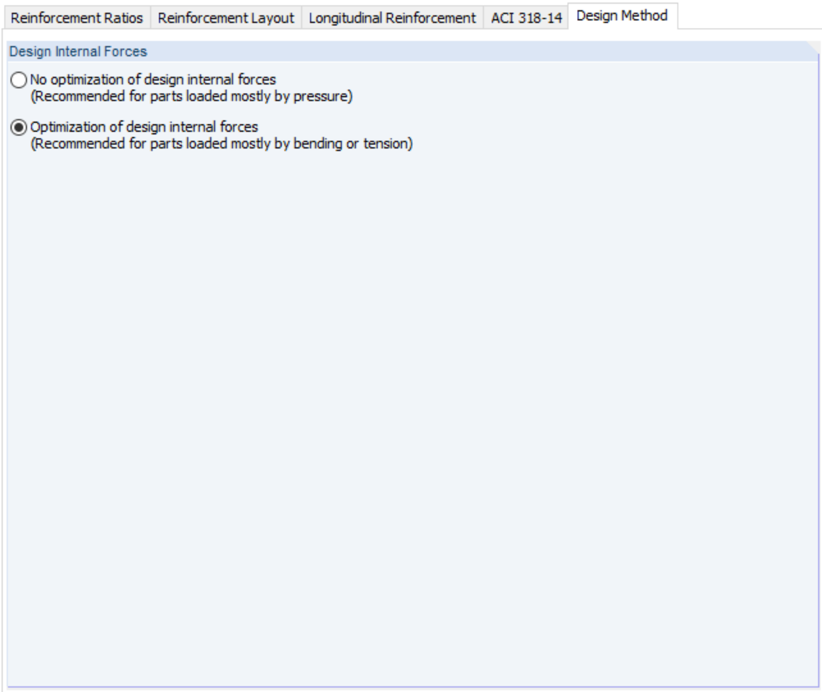


Figure 2.43: Window 1.4 *Reinforcement*, tab *Design Method*

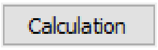
When the required reinforcement is determined, the principal internal forces are transformed into design forces (in direction of the reinforcement) and related concrete strut forces. The magnitudes of those design forces depend on the presumed angle of the concrete strut that braces the reinforcement mesh.

In the design situations "tension-tension" and "tension-compression", the design force may become negative in one reinforcement direction for a certain inclination of the concrete strut, that is, compressive forces would occur for the tension reinforcement. Due to the *Optimization of design internal forces*, the direction of the concrete strut is modified until the negative design force is zero.

During the optimization process of internal forces, the program determines the inclination angle of the concrete strut that produces the most favorable design result. The design moments are determined iteratively with adjusted inclination angles in order to find the smallest energy with the least required reinforcement. The optimization process is described in the general manual of RF-CONCRETE Surfaces.

The optimization for concrete components subjected to compression (such as walls) may result in non-designable elements due to failure of the compression strut. Therefore, the optimization is not recommended for the design situation "compression-compression".

3. Calculation



In RF-CONCRETE Surfaces, the [Calculation] is carried out with the internal forces of RFEM. If the RFEM results are not yet available, the internal forces will be calculated automatically.

3.1 Details

Click the [Details] button which is available in all windows to open the corresponding dialog box. It consists of two tabs with global settings for the design.

3.1.1 Options

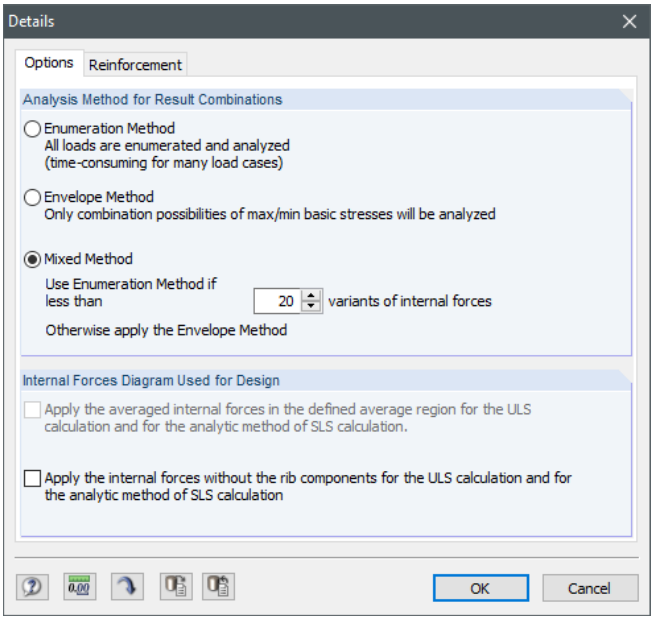


Figure 3.1: Dialog box *Details*, tab *Options*

Analysis Method for Result Combinations

This section controls how the design internal forces of the result combinations are included in the calculation. This specification also applies when there are several load cases or load combinations in the design case to be analyzed. The *Mixed Method* is preset: Before the design, the program analyzes whether the *Enumeration Method* or the *Envelope Method* needs less computation time.

Enumeration Method

Each load case and each load combination selected in Window *1.1 General Data* is designed individually. From the results, the reinforcement envelope is computed. For result combinations, 16 calculations are performed for the RFEM extreme values of the basic internal forces max/min m_x, max/min n_x, max/min m_y, max/min n_y, max/min m_{xy}, max/min n_{xy}, max/min v_x, and max/min v_y.

The *Enumeration Method* is very precise because every combination is calculated separately, and then the enveloping reinforcement is determined. However, the disadvantage of this method is that the number of the combinations to be analyzed increases exponentially with the number of load cases, as the program proceeds from row to row. If there are, for example, 50 selected load combinations, there will also be 50 reinforcement designs. On the other hand, all possible variants (constellations) are included in the designs.

Envelope Method

From the load cases, load and result combinations selected in Window *1.1 General Data*, the module computes an internal forces envelope. 16 extreme value variants are analyzed. The difference to the extreme values of result combinations in RFEM is the following: The module analyzes not only the variants of the extreme values that are based on the maximum basic internal forces, but also on their interaction (for example $m_x + m_{xy}$). With this envelope from 16 variants of extreme values, the determination of the reinforcement is started. Thus, 16 calculation runs are carried out to determine the reinforcement. Even if there is a great number of load cases, load or result combinations, the computing time will still be adequate.

Since the internal forces envelope is computed with 16 extreme values, it is possible that the most unfavorable variants are not considered – unlike in the *Enumeration Method* where the load cases are computed row by row. Combinations with load cases whose directions of action are orthogonal may be critical, however. In this case, a check according to the *Enumeration Method* is recommended.

Mixed Method

Before the design, the module analyzes how many designs are to be performed with the load cases, load and result combinations selected in Window *1.1 General Data* for each limit state. As mentioned in the description of the *Enumeration Method*, the module performs a separate design for each load case and each load combination. For one result combination, 16 calculations are required for the extreme value variants of the basic internal forces. If, for example, you select one result combination and five load combinations for the design, the program needs 16 + 5 = 21 calculation runs. The number is greater than the preset 20 variants of internal forces. Hence, the design is carried out with the *Envelope Method*.

In the box, you can specify the upper limit of the variants that are designed according to the precise *Enumeration Method*.

Thus, the *Mixed Method* is a compromise between precision of results and computation time.

Internal Forces Diagram Used for Design

Apply Averaged Internal Forces

By default, the design uses the continuous distribution of internal forces within surfaces from RFEM: RF-CONCRETE Surfaces transforms the moments and axial forces in the directions of the longitudinal reinforcement and then performs the checks.

If you select this check box, however, the design is carried out with the internal forces that are available in the average regions defined in RFEM. By means of the averaged results, you can reduce singularities and consider local redistribution effects in the model (see Figure 3.2).

The average regions are described in the RFEM manual, Chapter 9.7.3.

Apply Internal Forces Without Rib Components

In RFEM, you can model a T-beam by using a surface and an eccentrically connected member of the "rib" type. The internal forces of the T-beam from the surface component and member are determined by integration of the surface internal forces. They represent the "rib internal forces".

With this option, you can decide whether the surface internal forces assigned to the rib are to be included in the design of the surfaces. The design with the rib component is preset.

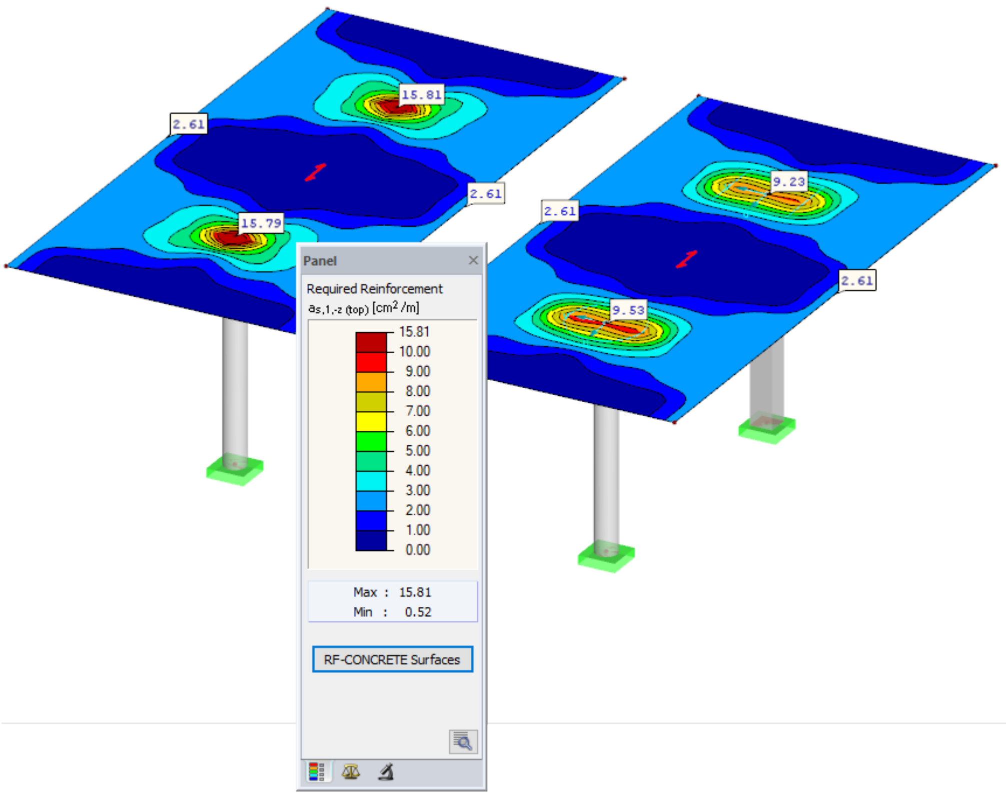


Figure 3.2: Top reinforcement for continuous distribution of internal forces (left) and average regions at columns (right)

3.1.2 Reinforcement

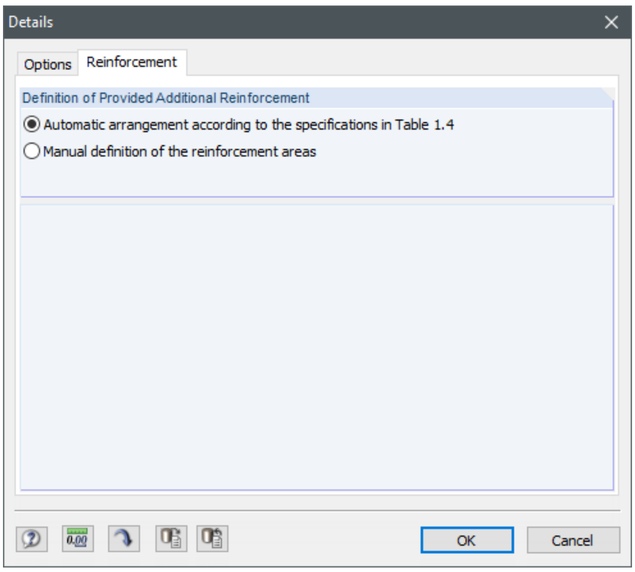


Figure 3.3: Dialog box *Details*, tab *Reinforcement*

Definition of Provided Additional Reinforcement

The *Automatic arrangement according to the specifications in Table 1.4* is preset for the additional reinforcement. This means that the rebars and mesh reinforcements will be arranged with the parameters of the *Longitudinal Reinforcement* tab to satisfy the serviceability limit state design.

As an alternative, a *Manual definition of the reinforcement areas* is possible. When this option is selected, the *Longitudinal Reinforcement* tab in Window 1.4 is accordingly adjusted (see Figure 2.32, page 31 with description below).

3.2 Check

Before you start the calculation, it is recommended to check if the input data is correct. The [Check] button is available in every input window of RF-CONCRETE Surfaces.

The program checks whether the data required for the design is complete and the references of the data sets are fine. If the program does not detect any error, it displays the following message:

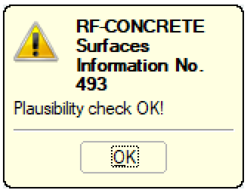
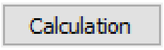


Figure 3.4: Plausibility check

3.3 Start Calculation



You can start the calculation out of each input window of the RF-CONCRETE Surfaces module by clicking [Calculation].

RF-CONCRETE Surfaces searches for the results of the load cases, load combinations, and result combinations that are to be designed. If they cannot be found, the program starts the RFEM calculation to determine the design-relevant internal forces.

You can also start the calculation out of the RFEM user interface: The *To Calculate* dialog box (menu *Calculate* → *To Calculate*) lists the design cases of the add-on modules like load cases or load combinations.

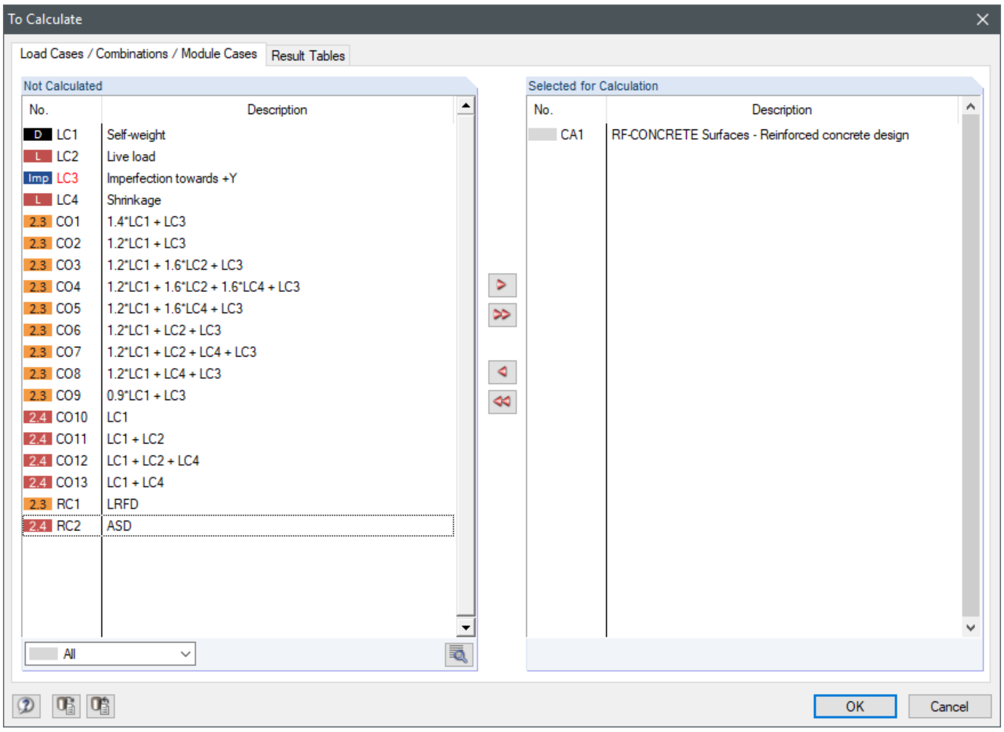
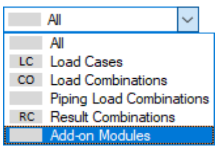


Figure 3.5: Dialog box *To Calculate*



If the design cases of RF-CONCRETE Surfaces are missing in the *Not Calculated* section, select *All* or *Add-on Modules* from the drop-down list below the section.

To transfer the selected RF-CONCRETE Surfaces cases to the dialog box section on the right, click [▶]. To start the calculation, click [OK].

Alternatively, you can use the drop-down list in the toolbar to calculate a design case: Select the RF-CONCRETE Surfaces case, and then click [Show Results].

Figure 3.6: Direct calculation of an RF-CONCRETE Surfaces design case in RFEM

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

4. Results

Window *2.1 Required Reinforcement Total* is displayed immediately after the calculation.

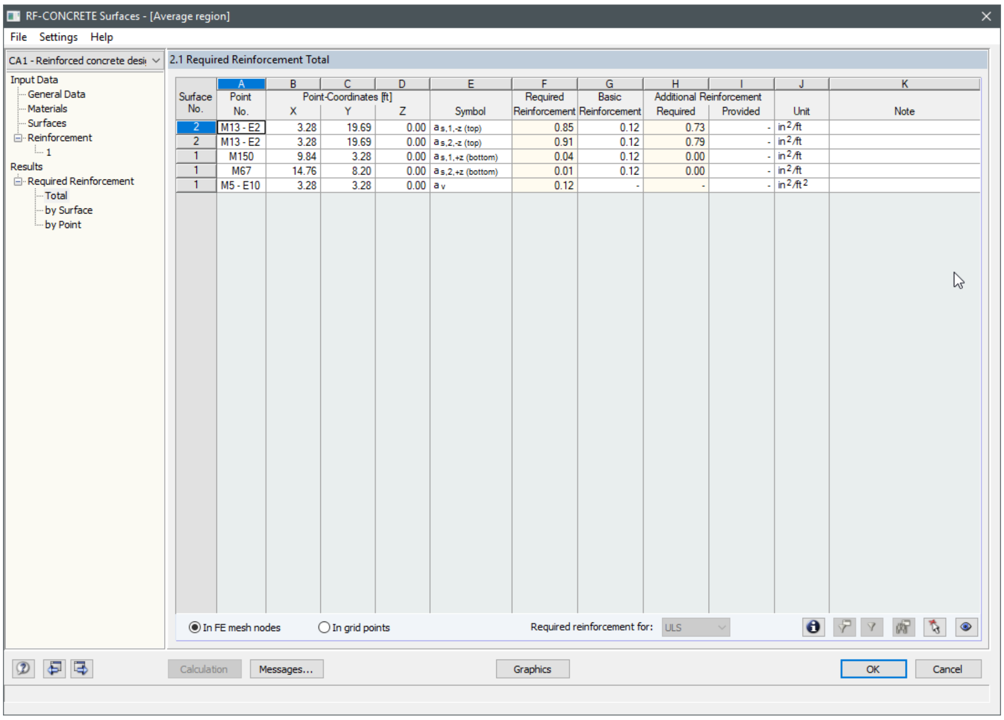
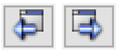


Figure 4.1: Results window

The strength limit state designs are listed in Window 2.1 through Window 2.3 by various criteria.

Window 3.1 through Window 3.3 contains the results of the serviceability limit state designs.



All windows can be selected directly by clicking the according entry in the navigator. Use the buttons shown on the left to go to the previous or next window. You can also use the function keys [F2] and [F3] to select the previous or next window.

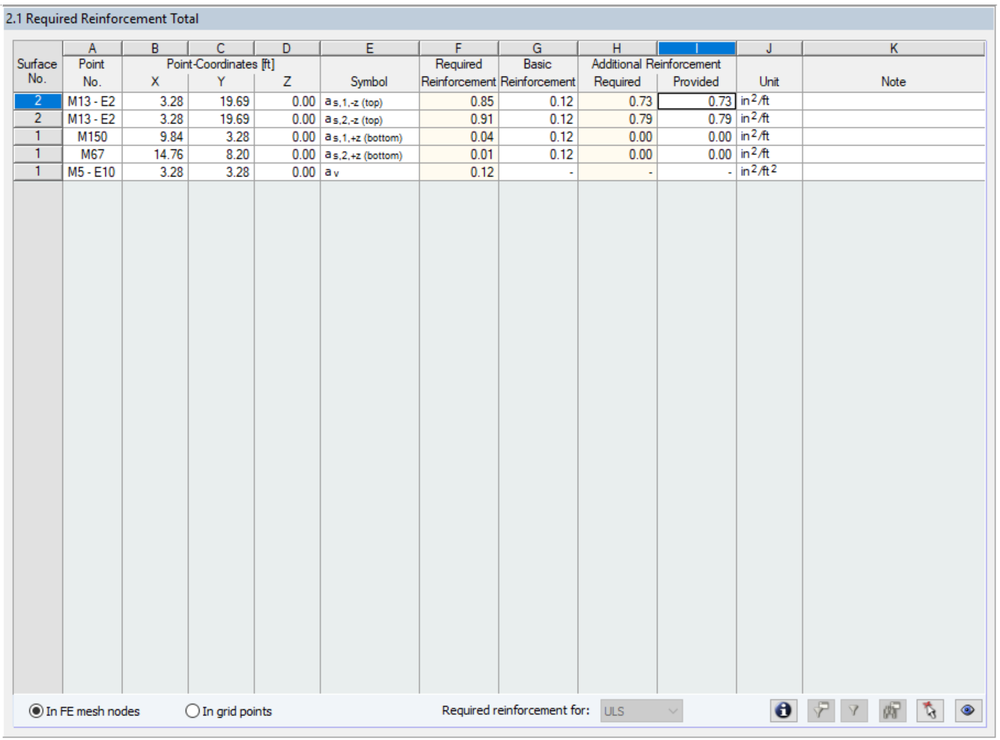
At the bottom of the windows, you find two option buttons. With these buttons, you decide whether to show the results data *In FE nodes* or *In grid points*. The results of the FE nodes are determined directly by the solver. The grid point results are determined by interpolations of the FE node results.

To save the results, click [OK]. Thus, you exit RF-CONCRETE Surfaces and return to the main program.

Chapter 4 *Results* presents the results windows in their proper order. Evaluating and checking results is described in Chapter 5 *Results Evaluation* on page 56.

4.1 Required Reinforcement Total

The table shows the maximum reinforcement areas of all analyzed surfaces. The areas are determined from the internal forces of the load cases, load and result combinations selected for the strength limit state design.



2.1 Required Reinforcement Total

Surface No.	A Point No.	B Point-Coordinates [ft] X	C Y	D Z	E Symbol	F Required Reinforcement	G Basic Reinforcement	H Additional Reinforcement Required	I Provided	J Unit	K Note
2	M13 - E2	3.28	19.69	0.00	$a_{s,1,-z}$ (top)	0.85	0.12	0.73	0.73	in²/ft	
2	M13 - E2	3.28	19.69	0.00	$a_{s,2,-z}$ (top)	0.91	0.12	0.79	0.79	in²/ft	
1	M150	9.84	3.28	0.00	$a_{s,1,+z}$ (bottom)	0.04	0.12	0.00	0.00	in²/ft	
1	M67	14.76	8.20	0.00	$a_{s,2,+z}$ (bottom)	0.01	0.12	0.00	0.00	in²/ft	
1	M5 - E10	3.28	3.28	0.00	a_v	0.12	-	-	-	in²/ft²	

Figure 4.2: Window *2.1 Required Reinforcement Total*

Surface No.

The column shows the numbers of surfaces containing the governing points.

Point No. / Grid Point

In these FE nodes or grid points, the greatest required reinforcement was determined for each position and direction. The type of each reinforcement is displayed in column E *Symbol*.

The FE mesh nodes, *M*, are generated automatically. By contrast, the grid points, *G*, can be controlled in RFEM where user-defined result grids are possible for surfaces. This function is described in Chapter 8.13 of the RFEM manual.

Point Coordinates X/Y/Z

The three columns show the coordinates of the governing FE nodes or grid points.

Symbol

Column E displays the reinforcement type. For the four (or six) longitudinal reinforcements, the module shows the directions (*1*, *2* and *3*, if available) as well as the surface sides (*top* and *bottom*).

The reinforcement directions are set in the *Reinforcement Layout* tab of Window *1.4 Reinforcement* (see Chapter 2.4.2, page 24).

Top and *bottom* surface

The top reinforcement is defined on the surface side in direction of the negative local surface axis z (-*z*). Accordingly, the top reinforcement is defined in direction of the positive z-axis (+*z*). Figure 2.25 on page 25 shows how to switch the axis systems of surfaces on or off.

The shear reinforcement is indicated as a_v.

Required Reinforcement

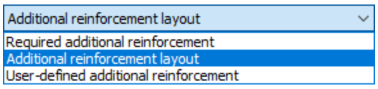
This column displays the reinforcement areas that are required to satisfy the strength limit state design.

Basic Reinforcement

This column shows the user-defined basic reinforcement defined in the *Longitudinal Reinforcement* tab of Window *1.4 Reinforcement* (see Chapter 2.4.3, page 27).

Additional Reinforcement

If you design the strength limit state exclusively, the column *Required* displays the difference between required reinforcement (column F) and provided basic reinforcement (column G).



If you additionally design the serviceability limit state, you see the reinforcement areas that are required by the specifications in the *Longitudinal Reinforcement* tab of Window *1.4 Reinforcement* (see Chapter 2.4.3, page 27) to satisfy the serviceability limit state designs. The *Provided* column shows the reinforcement that is available as additional reinforcement for the serviceability limit state design according to the specification in the *Longitudinal Reinforcement* tab of Window *1.4 Reinforcement*.

Note

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

The [Messages] button allows you to view all notes of the current design case. A dialog box appears showing the relevant messages.

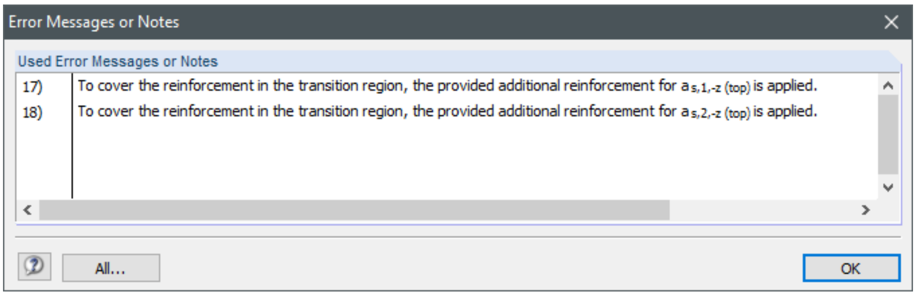


Figure 4.3: Dialog box *Error Messages or Notes*

The buttons available in Window 2.1 are described in Chapter 5 on page 56.

4.2 Required Reinforcement by Surface

2.2 Required Reinforcement by Surface

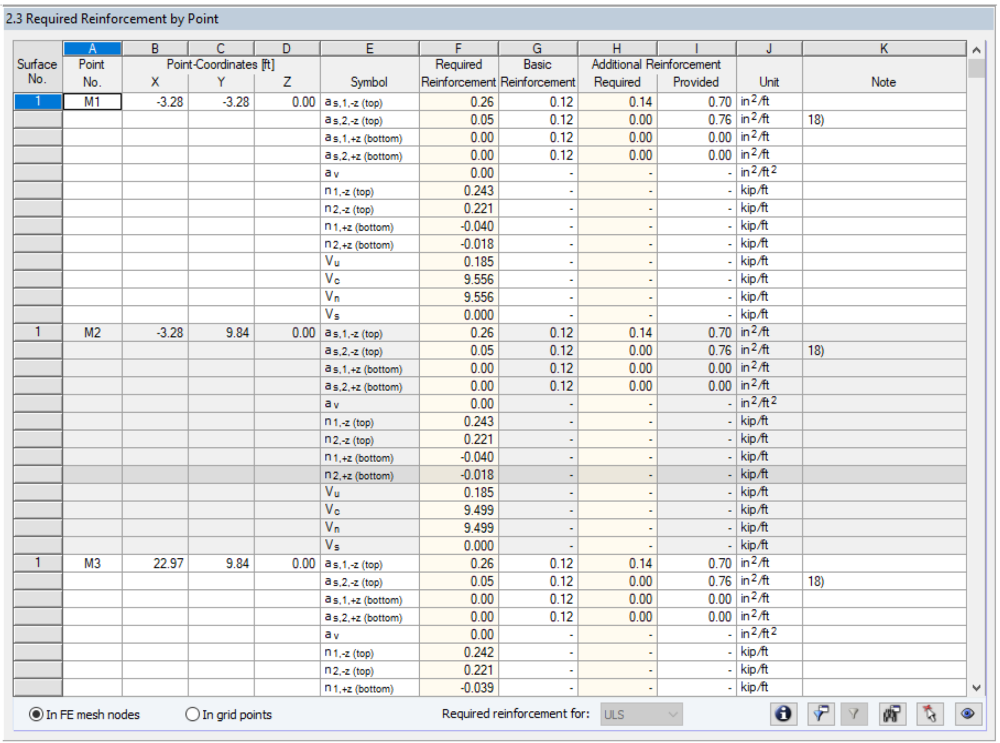
	A	B	C	D	E	F	G	H	I	J	K
Surface No.	Point No.	Point-Coordinates [ft] X	Y	Z	Symbol	Required Reinforcement	Basic Reinforcement	Additional Reinforcement Required	Provided	Unit	Note
1	M7 - E12	16.40	3.28	0.00	$a_{s,1,-z}$ (top)	0.52	0.12	0.40	0.70	in²/ft	
	M136 - E	18.04	3.28	0.00	$a_{s,2,-z}$ (top)	0.51	0.12	0.39	0.76	in²/ft	
	M150	9.84	3.28	0.00	$a_{s,1,+z}$ (bottom)	0.04	0.12	0.00	0.00	in²/ft	
	M67	14.76	8.20	0.00	$a_{s,2,+z}$ (bottom)	0.01	0.12	0.00	0.00	in²/ft	
	M5 - E10	3.28	3.28	0.00	a_v	0.12	-	-	-	in²/ft²	
2	M13 - E2	3.28	19.69	0.00	$a_{s,1,-z}$ (top)	0.82	0.12	0.70	0.70	in²/ft	
	M13 - E2	3.28	19.69	0.00	$a_{s,2,-z}$ (top)	0.88	0.12	0.76	0.76	in²/ft	
	M303	9.84	19.69	0.00	$a_{s,1,+z}$ (bottom)	0.04	0.12	0.00	0.00	in²/ft	
	M220	14.76	24.61	0.00	$a_{s,2,+z}$ (bottom)	0.01	0.12	0.00	0.00	in²/ft	
	M13 - E2	3.28	19.69	0.00	a_v	0.12	-	-	-	in²/ft²	

◉ In FE mesh nodes ○ In grid points Required reinforcement for: ULS

Figure 4.4: Window *2.2 Required Reinforcement by Surface*

This window shows the maximum reinforcement areas that are required for each of the designed surfaces. The columns are described in Chapter 4.1.

4.3 Required Reinforcement by Point



2.3 Required Reinforcement by Point

Surface No.	Point No.	Point-Coordinates [ft] X	Y	Z	Symbol	Required Reinforcement	Basic Reinforcement	Additional Reinforcement Required	Provided	Unit	Note
1	M1	-3.28	-3.28	0.00	$a_{s,1,-z\,(top)}$	0.26	0.12	0.14	0.70	in^2/ft	
					$a_{s,2,-z\,(top)}$	0.05	0.12	0.00	0.76	in^2/ft	18)
					$a_{s,1,+z\,(bottom)}$	0.00	0.12	0.00	0.00	in^2/ft	
					$a_{s,2,+z\,(bottom)}$	0.00	0.12	0.00	0.00	in^2/ft	
					a_v	0.00	-	-	-	in^2/ft^2	
					$n_{1,-z\,(top)}$	0.243	-	-	-	kip/ft	
					$n_{2,-z\,(top)}$	0.221	-	-	-	kip/ft	
					$n_{1,+z\,(bottom)}$	-0.040	-	-	-	kip/ft	
					$n_{2,+z\,(bottom)}$	-0.018	-	-	-	kip/ft	
					V_u	0.185	-	-	-	kip/ft	
					V_c	9.556	-	-	-	kip/ft	
					V_n	9.556	-	-	-	kip/ft	
					V_s	0.000	-	-	-	kip/ft	
1	M2	-3.28	9.84	0.00	$a_{s,1,-z\,(top)}$	0.26	0.12	0.14	0.70	in^2/ft	
					$a_{s,2,-z\,(top)}$	0.05	0.12	0.00	0.76	in^2/ft	18)
					$a_{s,1,+z\,(bottom)}$	0.00	0.12	0.00	0.00	in^2/ft	
					$a_{s,2,+z\,(bottom)}$	0.00	0.12	0.00	0.00	in^2/ft	
					a_v	0.00	-	-	-	in^2/ft^2	
					$n_{1,-z\,(top)}$	0.243	-	-	-	kip/ft	
					$n_{2,-z\,(top)}$	0.221	-	-	-	kip/ft	
					$n_{1,+z\,(bottom)}$	-0.040	-	-	-	kip/ft	
					$n_{2,+z\,(bottom)}$	-0.018	-	-	-	kip/ft	
					V_u	0.185	-	-	-	kip/ft	
					V_c	9.499	-	-	-	kip/ft	
					V_n	9.499	-	-	-	kip/ft	
					V_s	0.000	-	-	-	kip/ft	
1	M3	22.97	9.84	0.00	$a_{s,1,-z\,(top)}$	0.26	0.12	0.14	0.70	in^2/ft	
					$a_{s,2,-z\,(top)}$	0.05	0.12	0.00	0.76	in^2/ft	18)
					$a_{s,1,+z\,(bottom)}$	0.00	0.12	0.00	0.00	in^2/ft	
					$a_{s,2,+z\,(bottom)}$	0.00	0.12	0.00	0.00	in^2/ft	
					a_v	0.00	-	-	-	in^2/ft^2	
					$n_{1,-z\,(top)}$	0.242	-	-	-	kip/ft	
					$n_{2,-z\,(top)}$	0.221	-	-	-	kip/ft	
					$n_{1,+z\,(bottom)}$	-0.039	-	-	-	kip/ft	

Figure 4.5: Window *2.3 Required Reinforcement by Point*

This results window lists the maximum reinforcement areas for all FE nodes or grid points of each surface. The columns are described in Chapter 4.1.

In addition to the longitudinal and shear reinforcements, the table display design-relevant values of the actions and resistances. For ACI 318-14, these are the following:

Symbol	Meaning
$n_{1,-z\,(top)}$	Axial force or membrane force for design of reinforcement in first reinforcement direction on top side of the surface
$n_{2,-z\,(top)}$	Axial force or membrane force for design of reinforcement in second reinforcement direction on top side of the surface
$n_{1,+z\,(bottom)}$	Same as $n_{1,-z\,(top)}$, but on bottom side of the surface
$n_{2,+z\,(bottom)}$	Same as $n_{2,-z\,(top)}$, but on bottom side of the surface
$m_{1,-z\,(top)}$ $m_{2,-z\,(top)}$	Only for type of model *2D - XY* ($u_Z/\varphi_X/\varphi_Y$): moment for design of reinforcement in first or second reinforcement direction on top side of the surface
$m_{1,-z\,(bottom)}$ $m_{2,-z\,(bottom)}$	Same as $m_{1,-z\,(top)}$ / $m_{2,-z\,(top)}$, but on bottom side of the surface
V_u	Design value of applied shear force
V_c	Design shear resistance without shear reinforcement
V_n	Design shear resistance of concrete strut
V_s	Design shear resistance of shear reinforcement

Table 4.1: Output values in Window 2.3 for ACI 318-14

The search function, which you can start by clicking the button shown on the left, helps you to quickly find specific FE nodes or grid points (see Figure 5.7, page 61).

4.4 Serviceability Design Total

The upper table gives a summary of the governing serviceability limit state designs. Below, the *Intermediate Results* of the current FE node or grid point (i.e. the row selected in the upper table) including all design-relevant parameters are listed. You can expand or reduce entries by clicking [+] or [-].

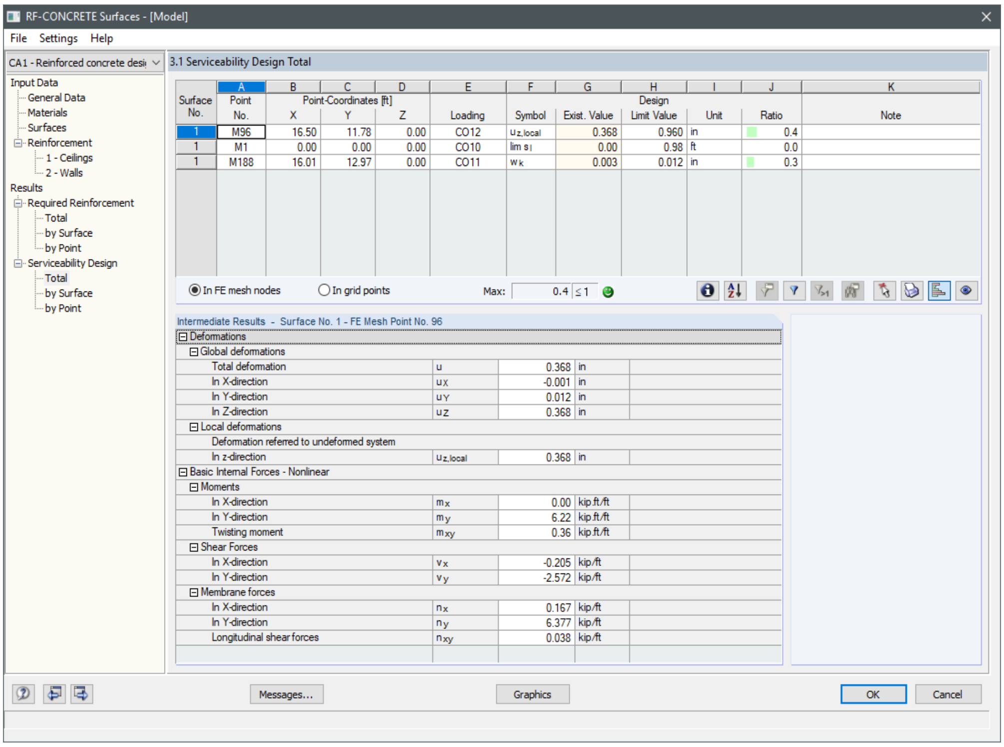


Figure 4.6: Window *3.1 Serviceability Design Total*

Figure 4.6 shows the results window of an analytical serviceability limit state check. Chapter 4.7 on page 53 describes the results window that appears when a nonlinear SLS analysis was carried out.

The method of check is defined in the *Serviceability Limit State* tab of Window *1.1General Data* (see Figure 2.5, page 9).

Surface No.

The column shows the numbers of surfaces containing the governing points.

Point No. / Grid Point

These FE nodes or grid points provide the maximum ratios for the required checks. The type of check is displayed in column F *Symbol*.

The FE mesh nodes, *M*, are generated automatically. The grid points, *G*, can be controlled in RFEM (see Chapter 8.13 of the RFEM manual).

Point Coordinates X/Y/Z

The three columns show the coordinates of the governing FE nodes or grid points.

Loading

Column E displays the load cases, load or result combinations whose internal forces produce the greatest ratios for the respective serviceability limit state design.

Symbol

Column F shows the type of the serviceability limit state design. If you have selected the analytical method, this column will show up to six types of check. Those types are described by an example in the general manual of RF-CONCRETE Surfaces.

The symbols stand for the following types of checks:

Symbol	Design SLS
$u_{z,local}$	Deformation in cracked state according to specifications in Window *1.3 Surfaces* This check requires the module extension RF-CONCRETE Deflect which enables the module to do an analytical deformation analysis. The stiffnesses are calculated independently from the stiffness modification for the strength limit state in RFEM.
lim s_l	Limitation of rebar spacing according to specifications in Window *1.3 Surfaces* (see Figure 2.14, page 17)
w_k	Limitation of crack width according to specifications in Window *1.3 Surfaces* (see Figure 2.14, page 17)

Table 4.2: Serviceability limit state designs according to analytical method

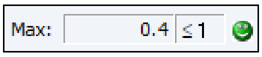
Existing Value

This column displays the values of all surfaces that are governing for the serviceability limit state designs.

Limit Value

The design limit values are determined from the standard specifications and the load situation. Their determination is described in the general manual of RF-CONCRETE Surfaces.

Ratio



Column J shows the ratios of the existing values (column G) and the limit values (column H). Ratios greater than 1 mean that the design criterion is not satisfied. The lengths of the colored scales illustrate the respective ratios graphically.

For the serviceability limit state designs, not all types of checks must be satisfied (see explanation in the general manual of RF-CONCRETE Surfaces.

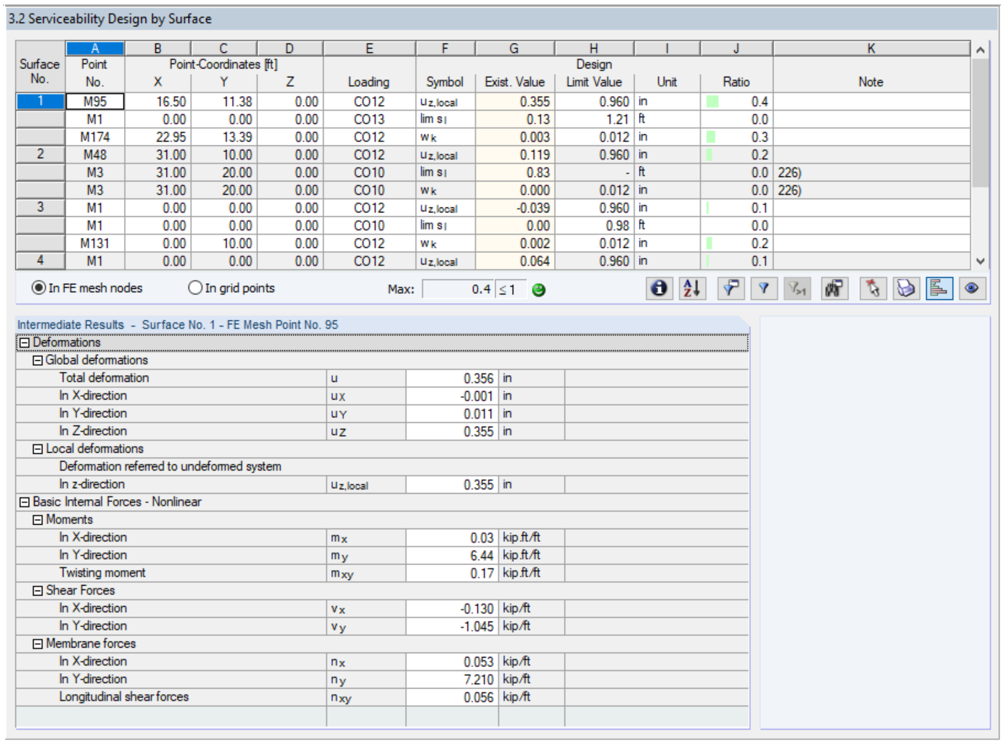
Note

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 current design case, use the [Messages] button shown on the left. A dialog box showing the relevant messages appears (see Figure 4.3, page 46).

The buttons available in Window 3.1 are described in Chapter 5 on page 56.

4.5 Serviceability Design by Surface



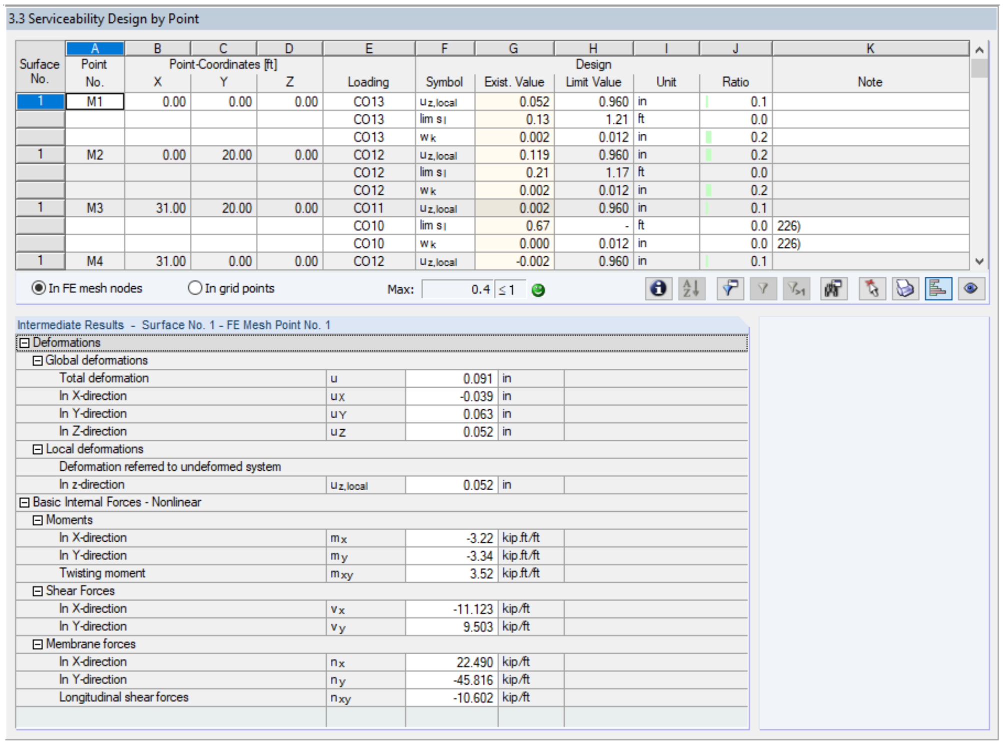
Surface No.	A Point No.	B Point-Coordinates [ft] X	C Y	D Z	E Loading	F Symbol	G Design Exist. Value	H Limit Value	I Unit	J Ratio	K Note
1	M95	16.50	11.38	0.00	CO12	$u_{z,local}$	0.355	0.960	in	0.4	
	M1	0.00	0.00	0.00	CO13	lim s_l	0.13	1.21	ft	0.0	
	M174	22.95	13.39	0.00	CO12	w_k	0.003	0.012	in	0.3	
2	M48	31.00	10.00	0.00	CO12	$u_{z,local}$	0.119	0.960	in	0.2	
	M3	31.00	20.00	0.00	CO10	lim s_l	0.83	-	ft	0.0	226)
	M3	31.00	20.00	0.00	CO10	w_k	0.000	0.012	in	0.0	226)
3	M1	0.00	0.00	0.00	CO12	$u_{z,local}$	-0.039	0.960	in	0.1	
	M1	0.00	0.00	0.00	CO10	lim s_l	0.00	0.98	ft	0.0	
	M131	0.00	10.00	0.00	CO12	w_k	0.002	0.012	in	0.2	
4	M1	0.00	0.00	0.00	CO12	$u_{z,local}$	0.064	0.960	in	0.1	

Deformations			
Global deformations			
Total deformation	u	0.356	in
In X-direction	u_X	-0.001	in
In Y-direction	u_Y	0.011	in
In Z-direction	u_Z	0.355	in
Local deformations			
Deformation referred to undeformed system			
In z-direction	$u_{z,local}$	0.355	in
Basic Internal Forces - Nonlinear			
Moments			
In X-direction	m_x	0.03	kip.ft/ft
In Y-direction	m_y	6.44	kip.ft/ft
Twisting moment	m_{xy}	0.17	kip.ft/ft
Shear Forces			
In X-direction	v_x	-0.130	kip/ft
In Y-direction	v_y	-1.045	kip/ft
Membrane forces			
In X-direction	n_x	0.053	kip/ft
In Y-direction	n_y	7.210	kip/ft
Longitudinal shear forces	n_{xy}	0.056	kip/ft

Figure 4.7: Window *3.2 Serviceability Design by Surface*

This window lists the maximum ratios of each designed surface resulting for the serviceability limit state designs. The columns are described in Chapter 4.4.

4.6 Serviceability Design by Point



3.3 Serviceability Design by Point

Surface No.	A Point No.	B Point-Coordinates [ft] X	C Y	D Z	E Loading	F Symbol	G Design Exist. Value	H Limit Value	I Unit	J Ratio	K Note
1	M1	0.00	0.00	0.00	CO13	$u_{z,local}$	0.052	0.960	in	0.1	
					CO13	lim s_l	0.13	1.21	ft	0.0	
					CO13	w_k	0.002	0.012	in	0.2	
1	M2	0.00	20.00	0.00	CO12	$u_{z,local}$	0.119	0.960	in	0.2	
					CO12	lim s_l	0.21	1.17	ft	0.0	
					CO12	w_k	0.002	0.012	in	0.2	
1	M3	31.00	20.00	0.00	CO11	$u_{z,local}$	0.002	0.960	in	0.1	
					CO10	lim s_l	0.67	-	ft	0.0	226)
					CO10	w_k	0.000	0.012	in	0.0	226)
1	M4	31.00	0.00	0.00	CO12	$u_{z,local}$	-0.002	0.960	in	0.1	

In FE mesh nodes | In grid points | Max: 0.4 ≤ 1

Intermediate Results - Surface No. 1 - FE Mesh Point No. 1

Deformations			
Global deformations			
Total deformation	u	0.091	in
In X-direction	u_X	-0.039	in
In Y-direction	u_Y	0.063	in
In Z-direction	u_Z	0.052	in
Local deformations			
Deformation referred to undeformed system			
In z-direction	$u_{z,local}$	0.052	in
Basic Internal Forces - Nonlinear			
Moments			
In X-direction	m_x	-3.22	kip.ft/ft
In Y-direction	m_y	-3.34	kip.ft/ft
Twisting moment	m_{xy}	3.52	kip.ft/ft
Shear Forces			
In X-direction	v_x	-11.123	kip/ft
In Y-direction	v_y	9.503	kip/ft
Membrane forces			
In X-direction	n_x	22.490	kip/ft
In Y-direction	n_y	-45.816	kip/ft
Longitudinal shear forces	n_{xy}	-10.602	kip/ft

Figure 4.8: Window *3.3Serviceability Design by Point*

This results window lists the maximum ratios for all FE nodes or grid points of each surface. The columns are described in Chapter 4.4.

The search function, which you can start by clicking the button shown on the left, helps you to quickly find specific FE nodes or grid points (see Figure 5.7, page 61).

4.7 Nonlinear Calculation Total

The upper table gives a summary of the governing serviceability limit state designs. Below, the *Intermediate Results* of the current FE nodes or grid point (i.e. the row selected in the upper table) including all design-relevant parameters are listed. You can expand or reduce entries by clicking [+] or [-].

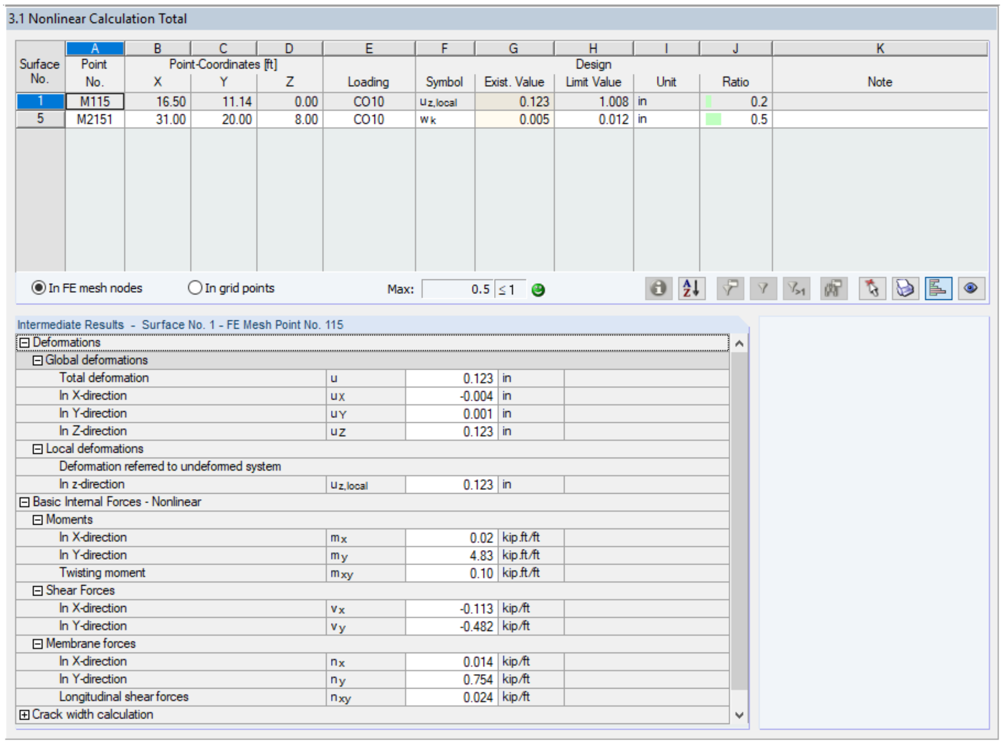


Figure 4.9: Window *3.1 Nonlinear Calculation Total*

Figure 4.9 shows the results window of a nonlinear serviceability limit state design. The method of check is specified in the *Serviceability Limit State* tab of Window *1.1 General Data* (see Figure 2.5, page 9).

The columns are described in Chapter 4.4, page 49.

The symbols stand for the following types of checks:

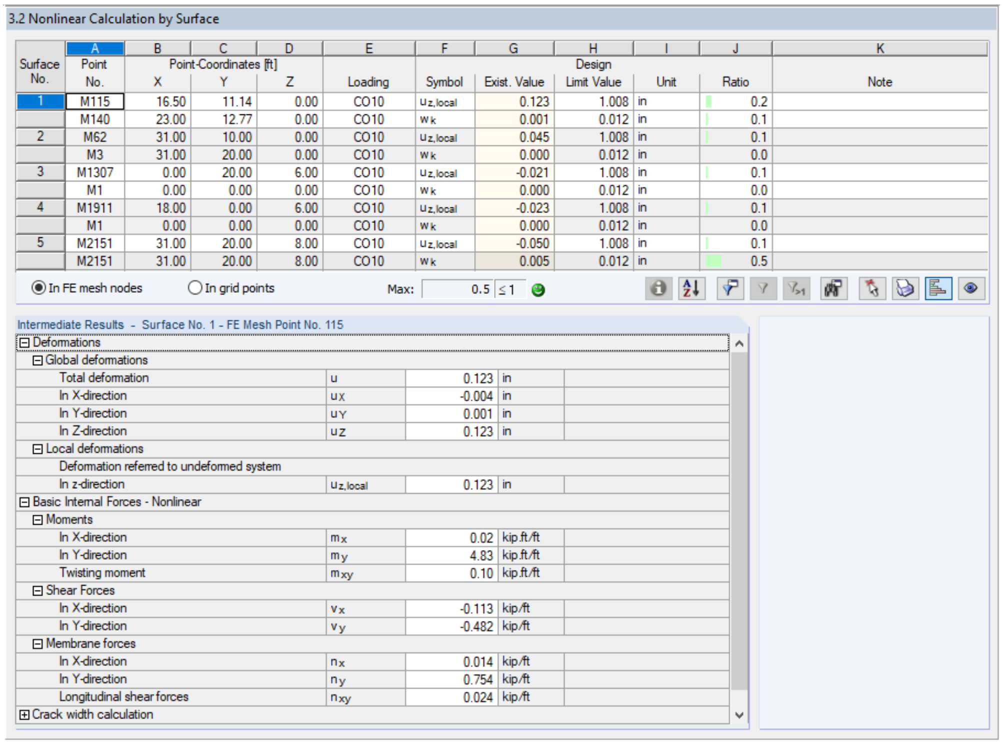
Symbol	**Design SLS**
$u_{z,local}$	Deformation in cracked state according to specifications in Window *1.3 Surfaces*
w_k	Limitation of crack width according to specifications in Window *1.3 Surfaces* (see Figure 2.14, page 17)

Table 4.3: Serviceability limit state designs according to nonlinear method

The values of the deformations, crack widths, and stresses represent the results in cracked sections (state II).

The crack widths, w_k, given in the *Intermediate Results* refer to the reinforcement directions. For example, the value for $w_{k,I,-z\,(top)}$ represents the crack width for the first direction of reinforcement on the top side of the surface; the crack runs perpendicularly to the reinforcement direction *1*.

4.8 Nonlinear Calculation by Surface



3.2 Nonlinear Calculation by Surface

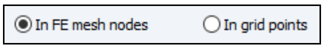
Surface No.	A Point No.	B Point-Coordinates [ft] X	C Y	D Z	E Loading	F Symbol	G Exist. Value	H Design Limit Value	I Unit	J Ratio	K Note
1	M115	16.50	11.14	0.00	CO10	$u_{z,local}$	0.123	1.008	in	0.2	
	M140	23.00	12.77	0.00	CO10	w_k	0.001	0.012	in	0.1	
2	M62	31.00	10.00	0.00	CO10	$u_{z,local}$	0.045	1.008	in	0.1	
	M3	31.00	20.00	0.00	CO10	w_k	0.000	0.012	in	0.0	
3	M1307	0.00	20.00	6.00	CO10	$u_{z,local}$	-0.021	1.008	in	0.1	
	M1	0.00	0.00	0.00	CO10	w_k	0.000	0.012	in	0.0	
4	M1911	18.00	0.00	6.00	CO10	$u_{z,local}$	-0.023	1.008	in	0.1	
	M1	0.00	0.00	0.00	CO10	w_k	0.000	0.012	in	0.0	
5	M2151	31.00	20.00	8.00	CO10	$u_{z,local}$	-0.050	1.008	in	0.1	
	M2151	31.00	20.00	8.00	CO10	w_k	0.005	0.012	in	0.5	

In FE mesh nodes   In grid points   Max: 0.5 ≤ 1

Intermediate Results - Surface No. 1 - FE Mesh Point No. 115

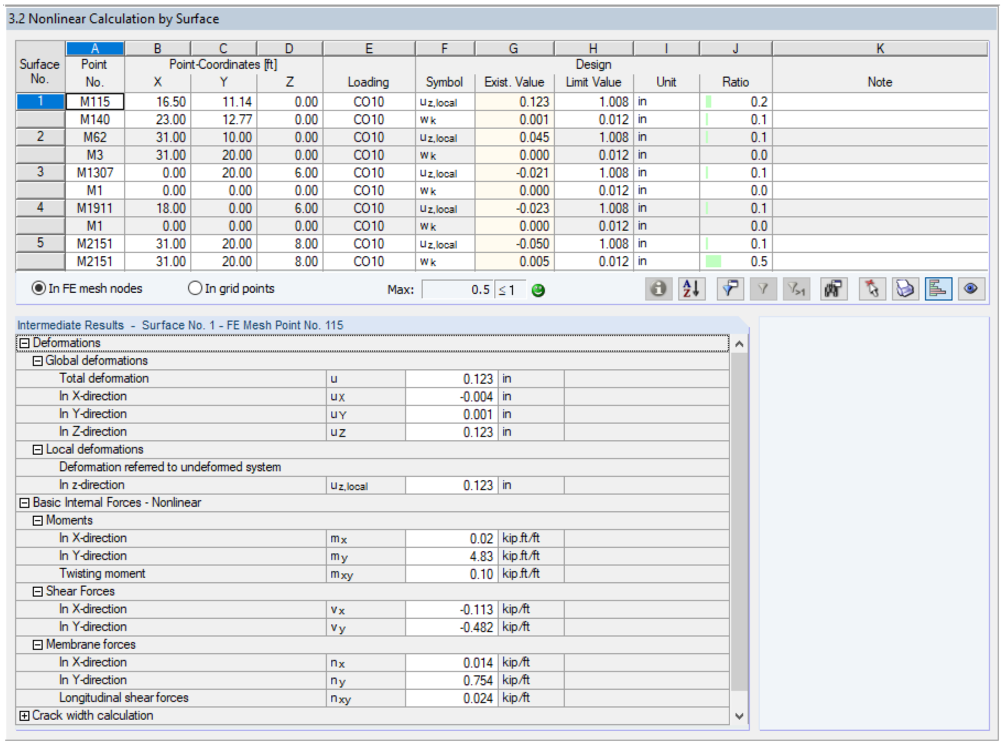
⊟ Deformations			
⊟ Global deformations			
Total deformation	u	0.123	in
In X-direction	u_X	-0.004	in
In Y-direction	u_Y	0.001	in
In Z-direction	u_Z	0.123	in
⊟ Local deformations			
Deformation referred to undeformed system			
In z-direction	$u_{z,local}$	0.123	in
⊟ Basic Internal Forces - Nonlinear			
⊟ Moments			
In X-direction	m_x	0.02	kip.ft/ft
In Y-direction	m_y	4.83	kip.ft/ft
Twisting moment	m_{xy}	0.10	kip.ft/ft
⊟ Shear Forces			
In X-direction	v_x	-0.113	kip/ft
In Y-direction	v_y	-0.482	kip/ft
⊟ Membrane forces			
In X-direction	n_x	0.014	kip/ft
In Y-direction	n_y	0.754	kip/ft
Longitudinal shear forces	n_{xy}	0.024	kip/ft
⊞ Crack width calculation			

Figure 4.10: Window *3.2 Nonlinear Calculation by Surface*



This window lists the maximum ratios of each designed surface that result in the serviceability limit state design. The columns are described in Chapters 4.4 and 4.7.

4.9 Nonlinear Calculation by Point



3.2 Nonlinear Calculation by Surface

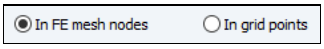
Surface No.	A Point No.	B Point-Coordinates [ft] X	C Y	D Z	E Loading	F Symbol	G Design Exist. Value	H Limit Value	I Unit	J Ratio	K Note
1	M115	16.50	11.14	0.00	CO10	$u_{z,local}$	0.123	1.008	in	0.2	
	M140	23.00	12.77	0.00	CO10	w_k	0.001	0.012	in	0.1	
2	M62	31.00	10.00	0.00	CO10	$u_{z,local}$	0.045	1.008	in	0.1	
	M3	31.00	20.00	0.00	CO10	w_k	0.000	0.012	in	0.0	
3	M1307	0.00	20.00	6.00	CO10	$u_{z,local}$	-0.021	1.008	in	0.1	
	M1	0.00	0.00	0.00	CO10	w_k	0.000	0.012	in	0.0	
4	M1911	18.00	0.00	6.00	CO10	$u_{z,local}$	-0.023	1.008	in	0.1	
	M1	0.00	0.00	0.00	CO10	w_k	0.000	0.012	in	0.0	
5	M2151	31.00	20.00	8.00	CO10	$u_{z,local}$	-0.050	1.008	in	0.1	
	M2151	31.00	20.00	8.00	CO10	w_k	0.005	0.012	in	0.5	

In FE mesh nodes   In grid points   Max: 0.5 ≤ 1

Intermediate Results - Surface No. 1 - FE Mesh Point No. 115

Deformations			
Global deformations			
Total deformation	u	0.123	in
In X-direction	u_X	-0.004	in
In Y-direction	u_Y	0.001	in
In Z-direction	u_Z	0.123	in
Local deformations			
Deformation referred to undeformed system			
In z-direction	$u_{z,local}$	0.123	in
Basic Internal Forces - Nonlinear			
Moments			
In X-direction	m_x	0.02	kip.ft/ft
In Y-direction	m_y	4.83	kip.ft/ft
Twisting moment	m_{xy}	0.10	kip.ft/ft
Shear Forces			
In X-direction	v_x	-0.113	kip/ft
In Y-direction	v_y	-0.482	kip/ft
Membrane forces			
In X-direction	n_x	0.014	kip/ft
In Y-direction	n_y	0.754	kip/ft
Longitudinal shear forces	n_{xy}	0.024	kip/ft
Crack width calculation			

Figure 4.11: Window *3.3 Nonlinear Calculation by Point*

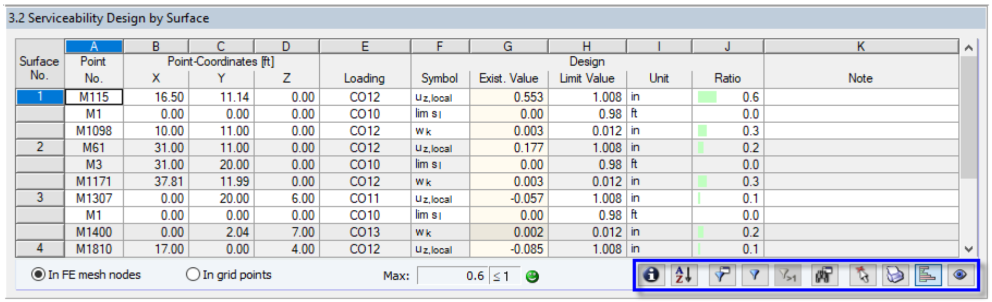


This results window lists the maximum ratios for all FE nodes or grid points of each surface. The columns are described in Chapters 4.4 and 4.7.

The search function, which you can start by clicking the button shown on the left, helps you to quickly find FE nodes or grid points (see Figure 5.7, page 61).

5. Results Evaluation

The design results can be evaluated in various ways. For this purpose, the buttons below the tables are very useful.


3.2 Serviceability Design by Surface

	A	B	C	D	E	F	G	H	I	J	K
Surface No.	Point No.	Point-Coordinates [ft] X	Y	Z	Loading	Symbol	Exist. Value	Design Limit Value	Unit	Ratio	Note
1	M115	16.50	11.14	0.00	CO12	$u_{z,local}$	0.553	1.008	in	0.6	
	M1	0.00	0.00	0.00	CO10	lim s_l	0.00	0.98	ft	0.0	
	M1098	10.00	11.00	0.00	CO12	w_k	0.003	0.012	in	0.3	
2	M61	31.00	11.00	0.00	CO12	$u_{z,local}$	0.177	1.008	in	0.2	
	M3	31.00	20.00	0.00	CO10	lim s_l	0.00	0.98	ft	0.0	
	M1171	37.81	11.99	0.00	CO12	w_k	0.003	0.012	in	0.3	
3	M1307	0.00	20.00	6.00	CO11	$u_{z,local}$	-0.057	1.008	in	0.1	
	M1	0.00	0.00	0.00	CO10	lim s_l	0.00	0.98	ft	0.0	
	M1400	0.00	2.04	7.00	CO13	w_k	0.002	0.012	in	0.2	
4	M1810	17.00	0.00	4.00	CO12	$u_{z,local}$	-0.085	1.008	in	0.1	

Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Design Details	Opens the *Design Details* dialog box → Chapter 5.1, page 57
	Sorts Results	Sorts the results by the maximum ratios (column J) or maximum values (column G) → Chapter 5.3, page 62
	Filter	Opens the *Filter Points* dialog box where you can select FE nodes or grid points by specific criteria → Chapter 5.3, page 62
	Only Designable Results	Hides all rows containing non-designable situations
	Exceeding	Shows only rows with ratios > 1 (design not fulfilled)
	Find	Opens the *Find FE Node / Grid Point* dialog box where you can look for a specific results row → Chapter 5.3, page 61
	Surface Selection	Allows you to select a surface in the graphics to show its results in the table
	Print	Includes the intermediate results of the current FE node or grid point in printout report
	Show Color Bars	Shows or hides the colored reference scales in the results windows
	View Mode	Jumps to the RFEM work window where you can adjust the view

Table 5.1: Buttons in results windows

5.1 Design Details

Click the [Info] button to check on the design details of the currently selected grid point, i.e. the table row in which the cursor is set. This button is available in all results windows.

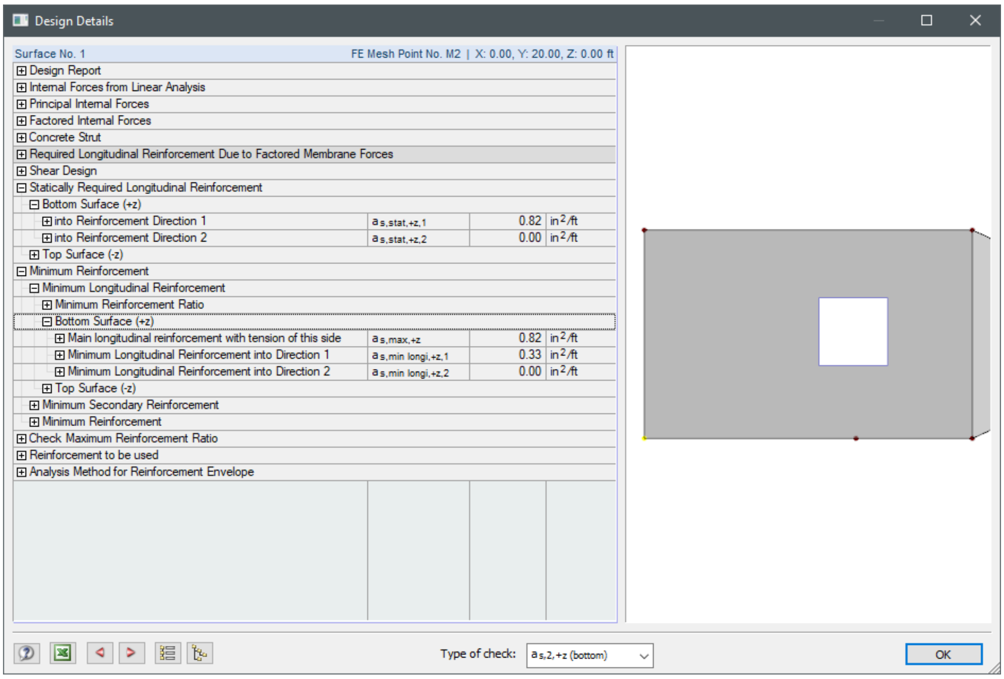


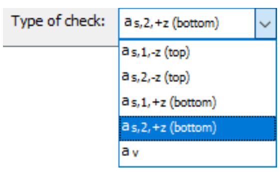
Figure 5.2: Dialog box *Design Details* for strength limit state design

The design details are listed in a tree structure. You can expand or reduce the entries by clicking [+] or [-], respectively. The buttons shown on the left [Close] or [Open] all subentries available in the directory tree.

The graphic in the right part shows the location of the selected point in the model.

For the strength limit state design, the following details are displayed:

- Design Report
- Internal Forces from Linear Analysis
- Principal Internal Forces
- Factored Internal Forces
- Concrete Strut
- Required Longitudinal Reinforcement Due to Factored Membrane Forces
- Shear Design
- Statically Required Longitudinal Reinforcement
- Minimum Reinforcement
- Check Maximum Reinforcement Ratios
- Reinforcement to be used
- Analysis Method for Reinforcement Envelope



The design details depend on the selected *Type of check*. Use this drop-down list at the bottom of the dialog box to select the results that you want to display.

In the serviceability limit state design, many detailed intermediate results are already shown in the bottom part of each window (see Figure 4.6, page 49). Click [Info] to view the list of the design details that are available for the current point. This possibility is only available for results calculated by the analytical method, however.

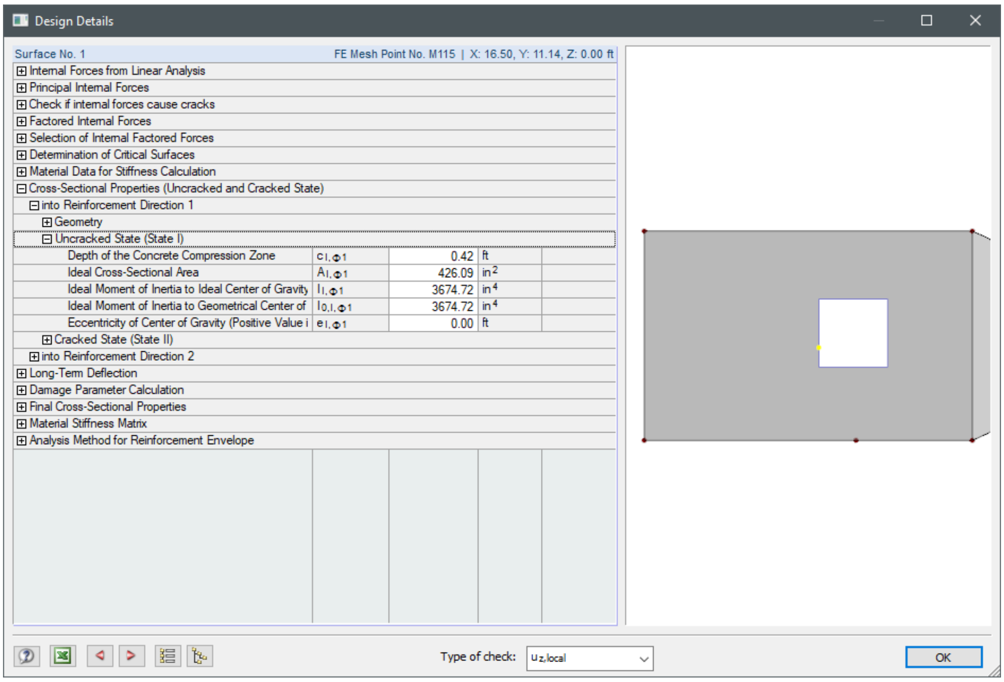
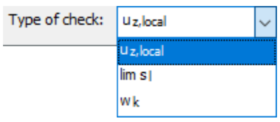


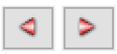
Figure 5.3: Dialog box *Design Details* for serviceability limit state design



A tree structure shows all design details relevant for each *Type of check*. Use this drop-down list at the bottom of the dialog box to select the results that you want to display.

Method of check	**Type of check**	
Analytical	$u_{z,local}$ lim s_l w_k	see Table 4.2, page 50

Table 5.2: *Type of check* for serviceability limit state designs



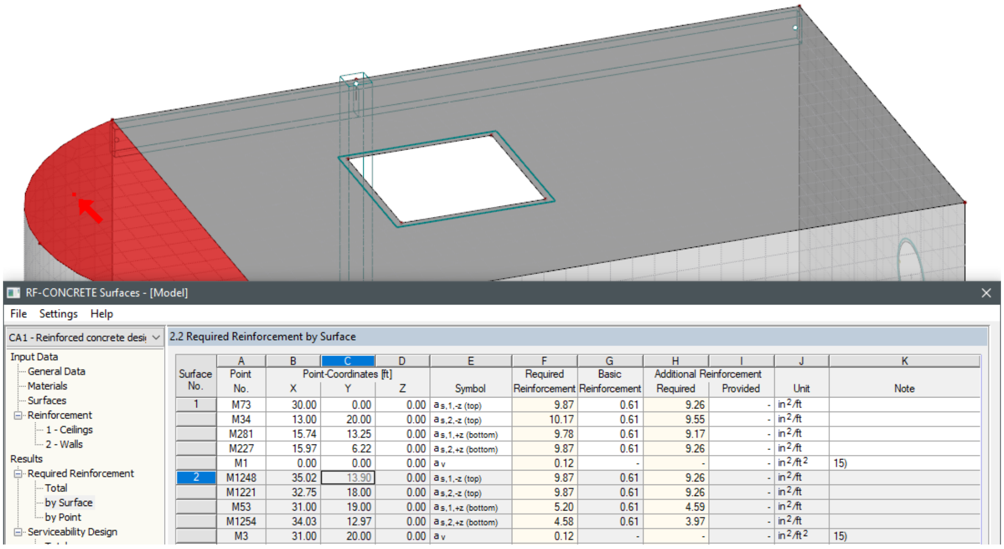
Click [◀] to go to the previous FE node or grid point. Click [▶] to select the next point.

5.2 Results on RFEM Model

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

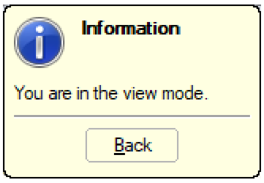
RFEM background graphic and view mode

The RFEM work window in the background helps you find the location of an FE node or grid point in the model. An arrow in the RFEM background graphic indicates the point selected in the results window of RF-CONCRETE Surfaces. The surface is highlighted in the selection color.



Surface No.	A Point No.	B Point-Coordinates [ft] X	C Y	D Z	E Symbol	F Required Reinforcement	G Basic Reinforcement	H Additional Reinforcement Required	I Provided	J Unit	K Note
1	M73	30.00	0.00	0.00	$a_{s,1,-z}$ (top)	9.87	0.61	9.26	-	in²/ft	
	M34	13.00	20.00	0.00	$a_{s,2,-z}$ (top)	10.17	0.61	9.55	-	in²/ft	
	M281	15.74	13.25	0.00	$a_{s,1,+z}$ (bottom)	9.78	0.61	9.17	-	in²/ft	
	M227	15.97	6.22	0.00	$a_{s,2,+z}$ (bottom)	9.87	0.61	9.26	-	in²/ft	
	M1	0.00	0.00	0.00	a_v	0.12	-	-	-	in²/ft²	15)
2	M1248	35.02	13.90	0.00	$a_{s,1,-z}$ (top)	9.87	0.61	9.26	-	in²/ft	
	M1221	32.75	18.00	0.00	$a_{s,2,-z}$ (top)	9.87	0.61	9.26	-	in²/ft	
	M53	31.00	19.00	0.00	$a_{s,1,+z}$ (bottom)	5.20	0.61	4.59	-	in²/ft	
	M1254	34.03	12.97	0.00	$a_{s,2,+z}$ (bottom)	4.58	0.61	3.97	-	in²/ft	
	M3	31.00	20.00	0.00	a_v	0.12	-	-	-	in²/ft²	15)

Figure 5.4: Highlighted surface and current FE node in RFEM model



If you cannot improve the view by moving the RF-CONCRETE Surfaces module window, click [Jump to graphic] to activate the *view mode*: Now, the module window is hidden so that you can adjust the view in the RFEM work window. The view mode provides only the functions of the *View* menu, for example zooming, moving, or rotating the display. The arrow remains visible while you the view.

To return to RF-CONCRETE Surfaces, click [Back].

RFEM work window

You can also graphically check the reinforcements and design ratios in the RFEM model. Click [Graphic] to exit the design module. The work window of RFEM now shows all design results, like the internal forces of a load case.

Results navigator

Reinforcement direction

The *Results* navigator is adjusted to the RF-CONCRETE Surfaces add-on module: It allows you to graphically show the results of the longitudinal reinforcements for each reinforcement direction and layer, of the shear reinforcement, the design internal forces or ratios, as well as the detailed results of the serviceability limit state designs (see Figure 5.5).

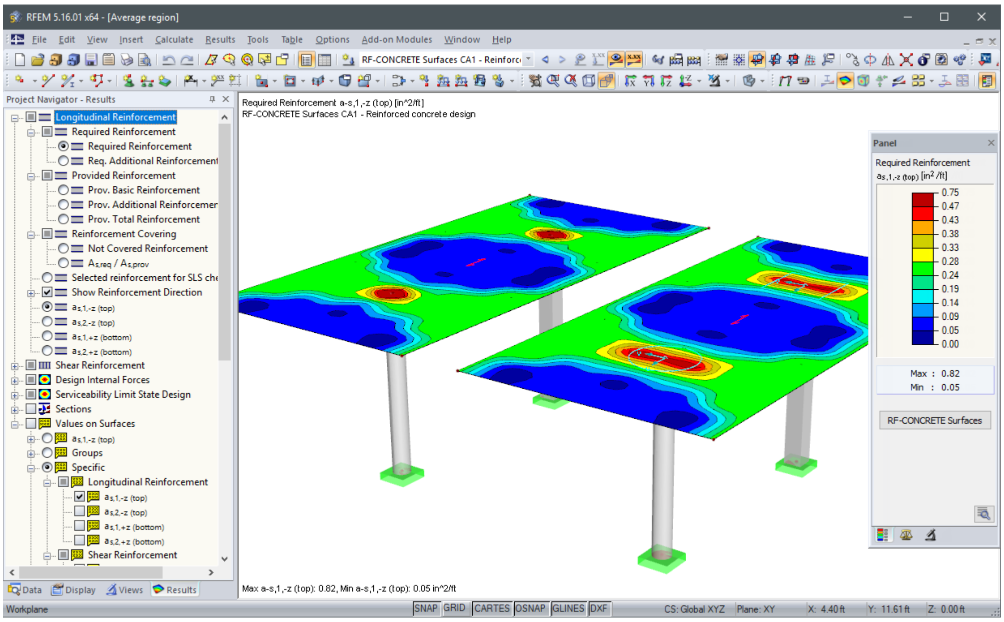
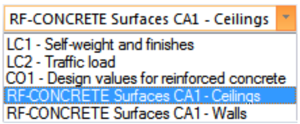


Figure 5.5: RFEM work window with *Results* navigator for RF-CONCRETE Surfaces

Similarly to the display of internal forces, the [Show Results] button shows or hides the display of the design results.

Since the RFEM tables are of no relevance for the evaluation of the design results, you may hide them.



You can select the design cases in the drop-down list in the RFEM toolbar.

Panel

You can use the panel with all control options to evaluate the design results. Its functions are described in the RFEM manual, Chapter 3.4.6. In the second tab, you can set the *Display Factors* for the reinforcements, internal forces, or ratios. The third tab of the panel allows you to display the results of selected surfaces (see the RFEM manual, Chapter 9.9.3).

Values on surfaces

You can use all options available in RFEM to display the result values of the reinforcements and ratios on the surfaces. This function is described in Chapter 9.4 of the RFEM manual. Figure 5.6 shows the group *Bottom (+z) reinforcement* which is required in addition to the user-defined basic reinforcement. Those values are to be applied in the reinforcement directions 1 and 2, respectively.

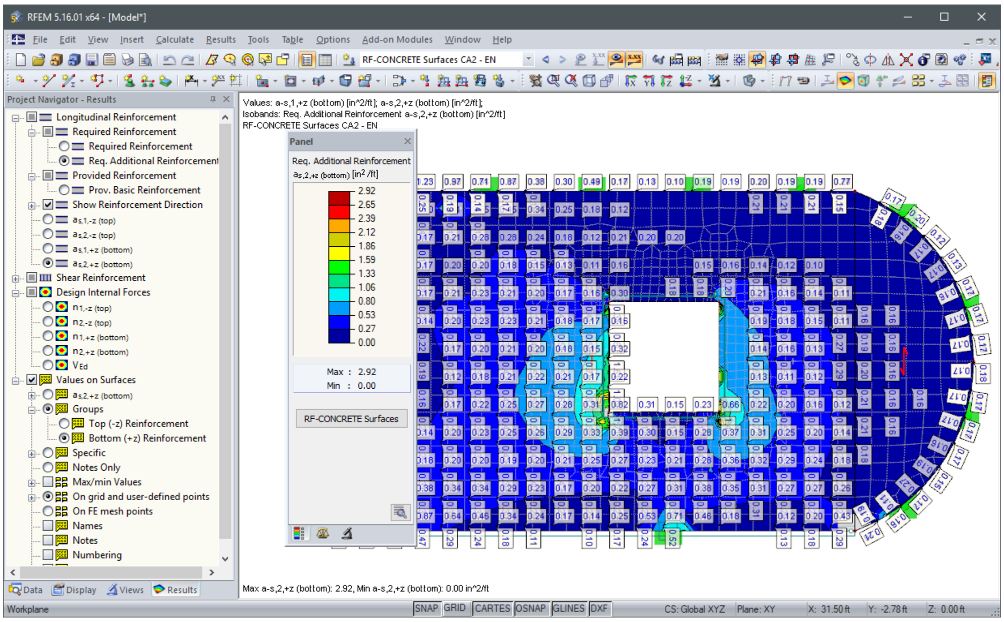


Figure 5.6: Group *Bottom (+z) reinforcement* for required additional reinforcement

The graphics of the design results can be transferred to the printout report which is described in the general manual of RF-CONCRETE Surfaces.

To return to the add-on module click [RF-CONCRETE Surfaces].

5.3 Filter for Results

The results windows of RF-CONCRETE Surfaces allow you to filter the results by various criteria. In addition, the module provides filter options to evaluate the design results graphically. These options are described in Chapter 9.9 of the RFEM manual.

For RF-CONCRETE Surfaces, you can also use the *Visibilities* (see RFEM manual, Chapter 9.9.1) to filter the surfaces for the evaluation.

Likewise, you can use the *Sections* in the RFEM model or create new ones (see RFEM manual, Chapter 9.6.1). This option allows you to evaluate the results specifically. For example, you can redistribute the reinforcement peaks resulting from singularities by using the smoothing function.

Find point

The results Windows 2.2 and 2.3 (reinforcement) as well as 3.2 and 3.3 (serviceability) provide a search function for FE nodes and grid points. Click the button shown on the left (see also Figure 5.1, page 56) to open the following dialog box.

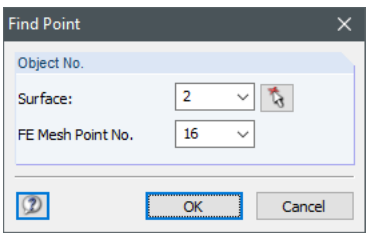


Figure 5.7: Dialog box *Find Point*

First, enter the number of the surface manually or use [↖] to select it graphically. Then, you can enter the number of the grid point or FE node, or select it in the list.

After clicking [OK], the current results window shows you the results row of this point.

Sort results

By default, Windows 3.1 and 3.2 show the results arranged by to the maximum design ratios. The sorting conforms to table column J.

You can also sort the results by existing values from column G. The greatest ratio of the deformation, for example, does not necessarily represent the maximum deformation because the limit values can be defined differently for each surface. Click the [Sort Results] button to switch between these two types of result arrangement.

Filter points

The button shown on the left is available in the results Windows 2.2 and 2.3 (reinforcement) as well as 3.2 and 3.3 (serviceability). By clicking it, you open the following dialog box.

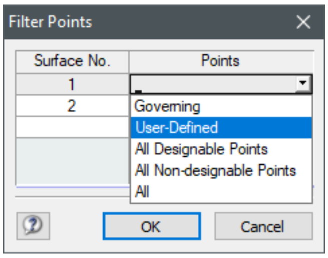


Figure 5.8: Dialog box *Filter Points*

In the *Surface No.* column, enter the number of the corresponding surface. Alternatively, you can select the surface graphically in the RFEM work window. To use this function, activate the cell and then click [...].

The *Points* column offers several criteria for filtering. In addition to *All Designable* and *All Non-designable Points*, you can select the *Governing* points. Those provide the maximum reinforcement areas or ratios for the respective type of check. You can also set the point numbers by *User-defined* entries.

Display only designable or non-designable results

The two buttons shown on the left allow you to display only designable results or only failed designs in the tables, respectively. Thus, you can, for example, hide failed designs due to singularities or analyze the causes for design problems.

Filtering results in work window

Graphics

The reinforcements and ratios can be used as filter criteria in the RFEM work window. Click [Graphics] to quit RF-CONCRETE Surfaces. To apply the filter function, the panel must be displayed. If the panel is not shown, select on the RFEM menu

View → Control Panel (Color Scale, Factors, Filter)

or use the toolbar button as seen to the left.

The panel is described in Chapter 3.4.6 of the RFEM manual. You can change the filter settings for the results in the first panel tab (color scale).

As illustrated in Figure 5.9, you can set the value scale of the panel in such a way that only reinforcements greater than 0.40 in²/ft are shown. The color ranges of the scale are set in levels of 0.30 in²/ft; the maximum value of 2.50 in²/ft eliminates effects due to singularities.

For the graphical display of the grid point or FE node values, the general control functions of RFEM are available. Those are described in Chapter 9.4 of the RFEM manual.

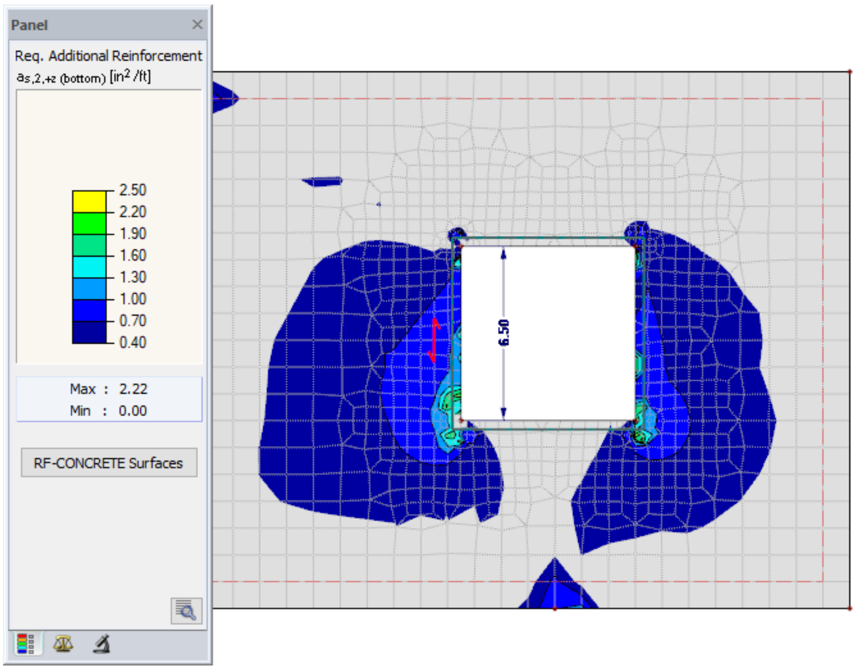


Figure 5.9: Filtering additional reinforcement with adjusted color spectrum

Chapter 5.4 describes how the value and color spectra can be adjusted to the diameters and spacings of the rebars.

Filtering surfaces in the work window

In the *Filter* tab of the control panel, you can enter the numbers of specific surfaces to display their result diagrams exclusively. This function is described in Chapter 9.9.3 of the RFEM manual.

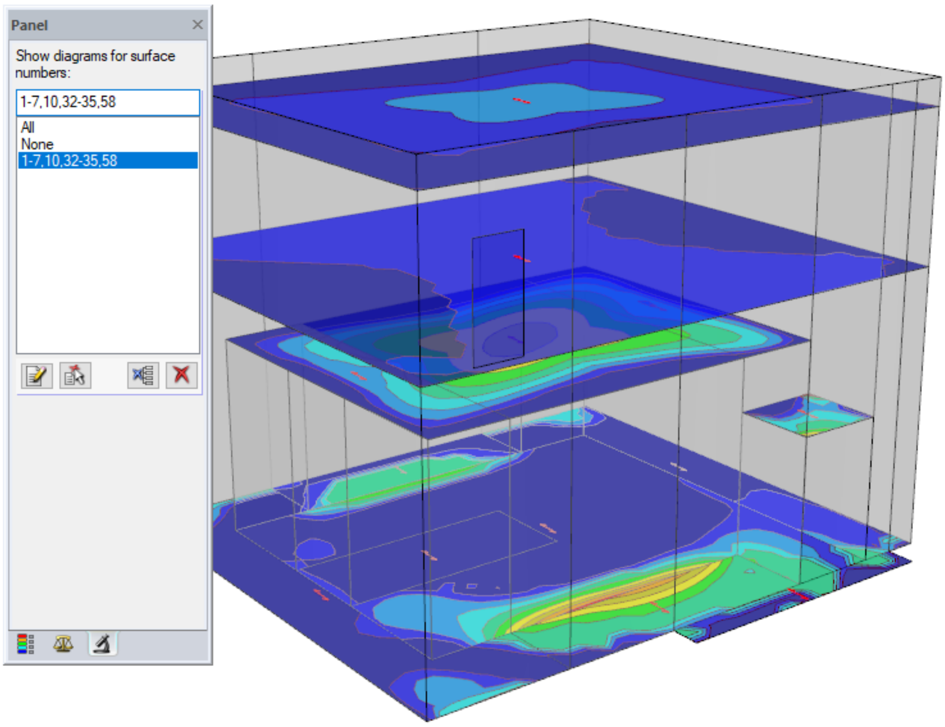


Figure 5.10: Surface filter for reinforcement of floor slabs and ceilings

In contrast to the visibility function, the entire model is displayed in the graphic. Figure 5.10 shows the reinforcement of all horizontal surfaces of a building. The remaining surfaces are shown in the model, but are displayed without reinforcement.

5.4 Configuration of Panel

The reinforcement results can be shown graphically as isobands or isolines. Twelve color zones are set by default for the value spectrum, covering the range between the minimum and maximum values. It is also possible to adjust the value spectrum with respect to the reinforcement layout in order to prepare the graphical results for a reinforcement drawing.

To adjust the panel, double click one of the colors. You can also use the [Options] button in the panel. The *Options* dialog box opens where you can click the [Edit] button to access another dialog box with options to modify the ranges of colors and values.

In the *Edit Isoline Value and Color Scales* dialog box, click [Edit] to open the *Edit Value Spectrum with Reinforcement Definition* dialog box.

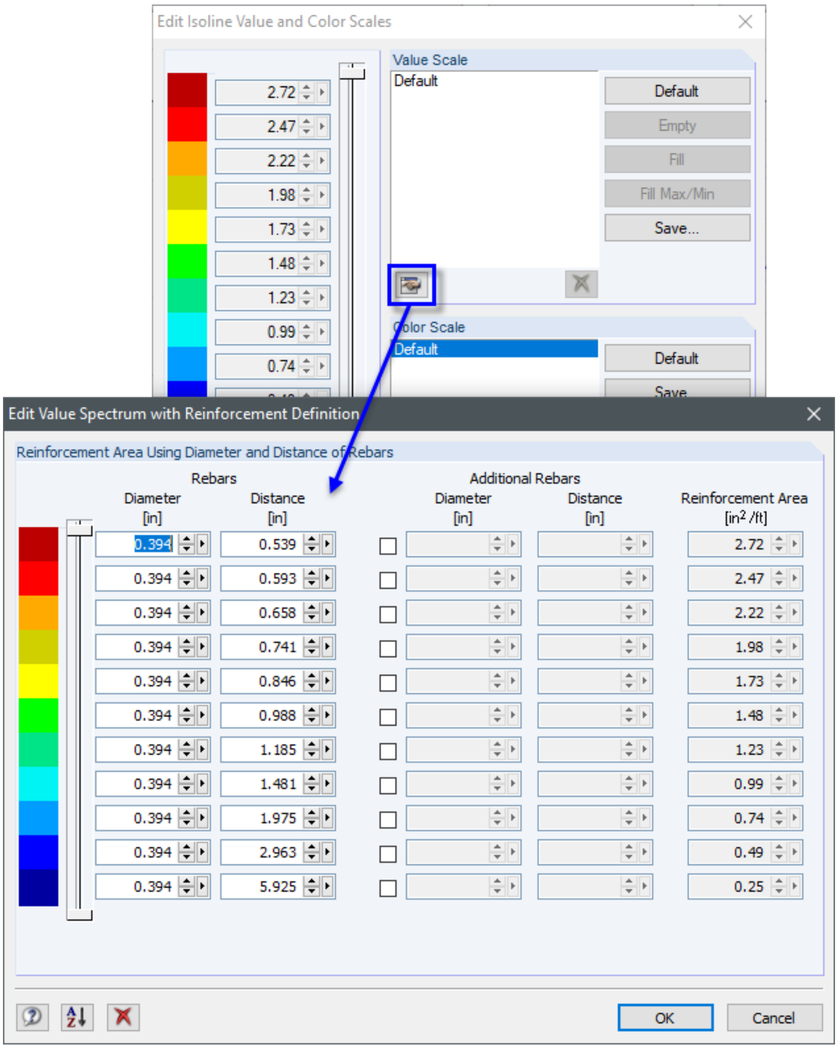


Figure 5.11: Dialog boxes *Edit Isoline Value and Color Scales* and *Edit Value Spectrum with Reinforcement Definition*

In this dialog box, the reinforcement area per foot is determined from the *Diameter* and the *Distance* of the rebars. In the *Additional Rebars* columns, you can assign further rebar diameters and distances (see Figure 5.12). Thus, you can set user-defined value spectra for the reinforcement and use them for a reinforcement drawing.

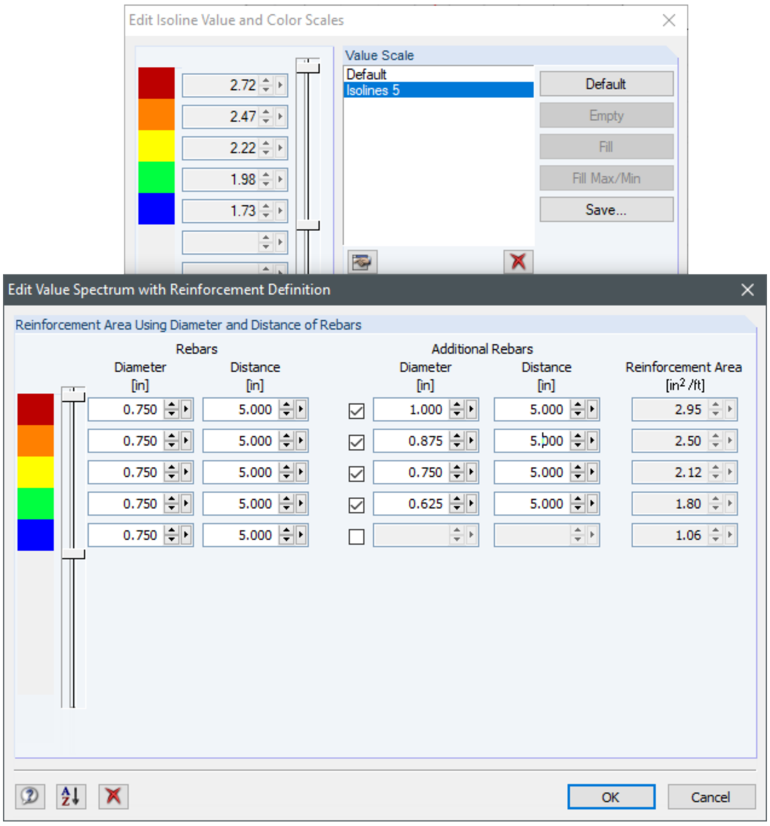


Figure 5.12: Dialog box *Edit Value Spectrum with Reinforcement Definitions* with rebar diameters and distances

Click [OK] to import the reinforcement areas resulting from the defined rebar diameters and rebar distances in the *Edit Isoline Value and Color Scales* dialog box.

In the panel, the diameters of the rebars with the according distances that are to be specified for the individual value ranges are shown.

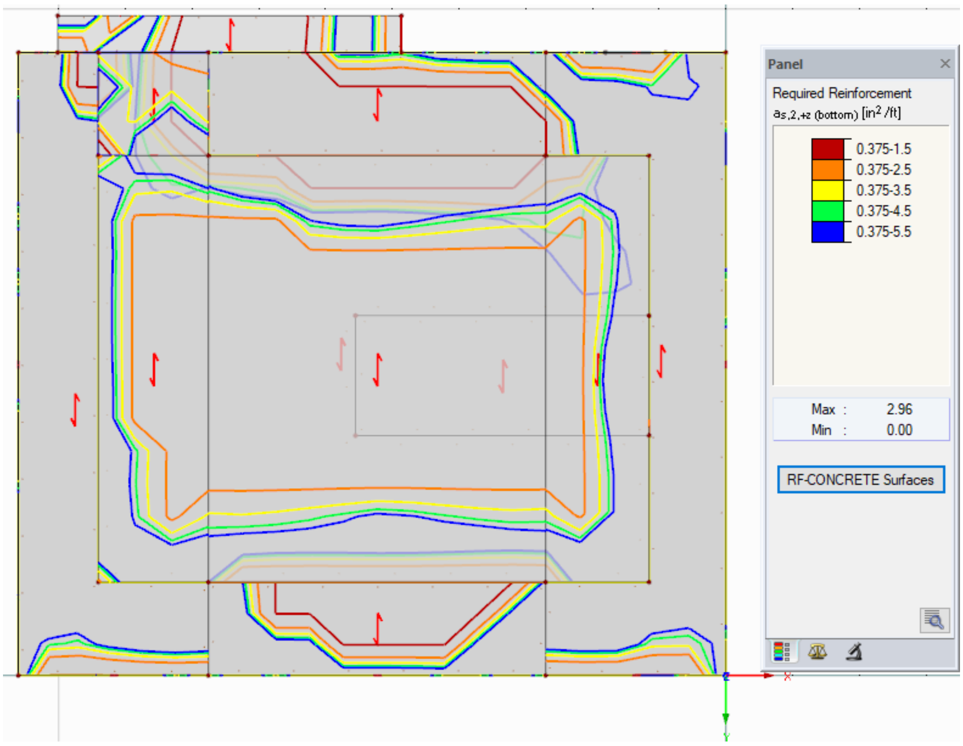


Figure 5.13: Graphic and panel with user-defined reinforcement zones

A Literature

[1] Deutscher Ausschuss für Stahlbeton, Heft 217: Tragwirkung orthogonaler Bewehrungsnetze beliebiger Richtung in Flächentragwerken aus Stahlbeton (by Theodor BAUMANN), Ernst & Sohn, Berlin 1972.

[2] ACI 318-14: Building Code Requirements for Structural Concrete (ACI 318-14) and Commentary, ACI Committee 318, September 2014

[3] EN 1992-1-1:2005 + AC:2010: Design of Concrete Structures – Part 1-1: General Rules and Rules for Buildings. 2005

[4] QUAST, Ulrich: Zur Mitwirkung des Betons in der Zugzone. Beton- und Stahlbetonbau, 1981, Heft 10, S. 247-250.

[5] QUAST, Ulrich: Zum nichtlinearen Berechnen im Stahlbeton- und Spannbetonbau. Beton- und Stahlbetonbau, 1994, Heft 9, S. 250-253, Heft 10, S. 280-284.

B Index