Version
December 2010

Program

RF-CONCRETE Surfaces

Reinforced Concrete Design

Program Description

All rights, including those of translations, are reserved.

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

© Dlubal Engineering Software
Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Design with RF-CONCRETE Surfaces

Although reinforced concrete is as frequently used for plate structures as for frameworks, standards and technical literature provide rather little information about the design of two-dimensional structural components. In particular, shell structures which are characterized by a simultaneous loading due to moment and axial force are hardly described in reference books. As the finite element method enables a realistic model representation of two-dimensional objects, design theories and algorithms must be found to close this "instructive gap" between member-oriented rules and computer-generated internal forces of plate structures.

DLUBAL ENGINEERING SOFTWARE meets this special challenge with the add-on module RF-CONCRETE Surfaces. Based on the compatibility equations by THEODOR BAUMANN from 1972, a consistent design algorithm has been developed to dimension two- and three-layer reinforcements. To increase the efficiency of this module as a tool used to determine the structurally required reinforcement, RF-CONCRETE Surfaces includes instructions concerning the allowable maximum and minimum reinforcement ratios for different structural component types (2D plates, 3D shells, walls, wall-like beams), as they can be found in the form of design specifications defined in the standards.

In addition to the dimensioning of reinforcing steel, the program always checks if the concrete's plate thickness, that stiffens the reinforcement mesh, is sufficient to meet the requirements arising from bending and shear loading.

In addition to the ultimate limit state design, the serviceability limit state design is possible, too. These designs include the limitation of the concrete compressive and the reinforcing steel stresses, the minimum reinforcement for the limitation of crack widths as well as the limitation of the crack width by limiting rebar diameter and rebar spacing. Here, analytical and non-linear design methods are available for selection.

In the non-linear design (available as additional module), it is possible to take into account the influence of creep and shrinkage for the analysis of crack formation in the deformed system.

The design can be carried out according to the following standards:

- DIN 1045 (1988-07)
- DIN 1045-1 (2001-07) and (2008-08)
- DIN V ENV 1992-1-1 (1992-06)
- ÖNORM B 4700 (2001-06)
- EN 1992-1-1
- CSN EN 1992-1-1
- ÖNORM B 1992-1-1
- STN EN 1992-1-1
- UNI EN 1992-1-1
- NEN EN 1992-1-1
- BS EN 1992-1-1
- NF EN 1992-1-1
- PN EN 1992-1-1

The seamless documentation of intermediate results, in line with the DLUBAL philosophy, provides a special transparency and traceability of design results.

Your team from DLUBAL ENGINEERING SOFTWARE

1.2 RF-CONCRETE Team

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

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Peter Konrad, B.I.S.M
Ing. Jan Fráňa
Ing. Ph.D. Pavel Marek
Dipl.-Ing. (FH) Alexander Meierhofer
Dipl.-Ing. (FH) Younes El Frem

Programming

Ing. Michal Balvon
Jaroslav Bartoš
Ing. Ladislav Ivančo
Dipl.-Ing. (FH) Peter Konrad, B.I.S.M
Ing. Alexandr Průcha
Ing. Lukáš Weis

Program design, dialog figures and icons

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch
Zdeněk Ballák
Ing. Jan Miléř

Program supervision

Ing. Jan Fráňa
Dipl.-Ing. (FH) Peter Konrad, B.I.S.M
M.Sc. Dipl.-Ing. Frank Lobisch
Ing. Ph.D. Pavel Marek
Dipl.-Ing. (FH) Alexander Meierhofer
Ing. Bohdan Šmid

Manual, help system and translation

Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ing. (FH) Peter Konrad, B.I.S.M
Ing. Dmitry Bystrov
Jan Jeřábek
Ing. Ladislav Kábrt
Ing. Petr Míchal
Dipl.-Ü. Gundel Pietzcker
Mgr. Petra Pokorná

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (BA) Sandy Baumgärtel
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Andreas Hörold
Dipl.-Ing. (FH) Bastian Kuhn
M.Sc. Dipl.-Ing. Frank Lobisch
Dipl.-Ing. (FH) Alexander Meierhofer
M. Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ing. (FH) Andreas Wopperer

1.3 Using the Manual

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

The description of the module follows the sequence of the module's input and results tables as well as its structure. The theoretical background of the design methods used in this module are described in detail in the German manual RF-BETON Flächen. The final manual section describes the various program functions for evaluating and documenting the design results.

The text of the manual shows the described **buttons** in square brackets, for example [New]. At the same time, they are pictured on the left. In addition, **expressions** used in dialog boxes, tables and menus are set in *italics* to clarify the explanations.

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

1.4 Open the Add-on Module RF-CONCRETE Surfaces

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

Menu

To start the program in the menu bar,

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

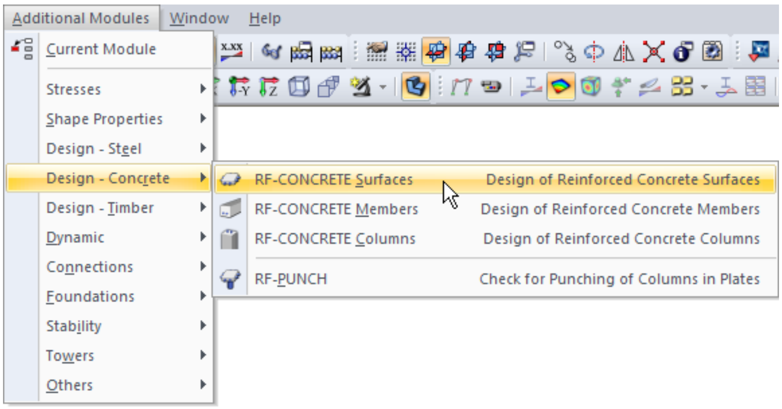


Figure 1.1: Menu *Additional Modules* → *Design - Concrete* → *RF-CONCRETE Surfaces*

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

Navigator

To start the design module in the *Data* navigator,

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

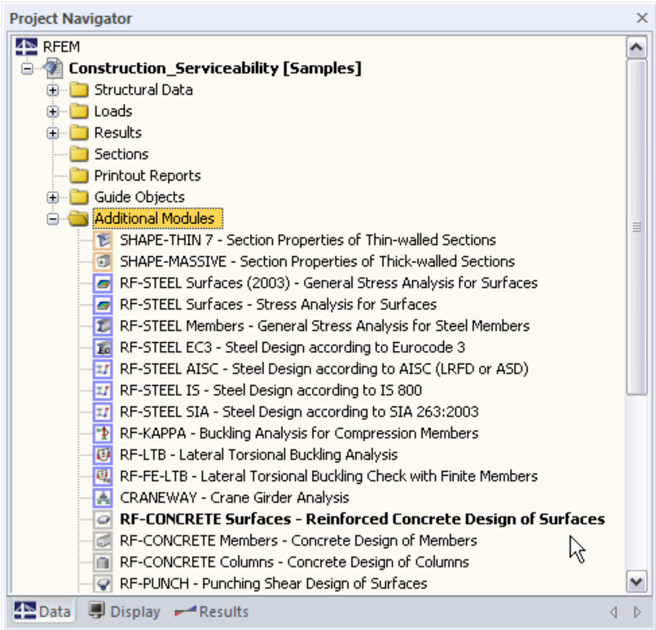
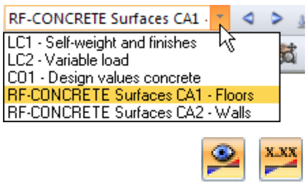


Figure 1.2: Data navigator: *Additional Modules* → *RF-CONCRETE Surfaces*

Panel



In case design results are already available in the RFEM structure, you can set the corresponding RF-CONCRETE Surfaces case in the load case list of the RFEM toolbar. By using the button [Results on/off], you can display the reinforcements or internal forces in the graphic.

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

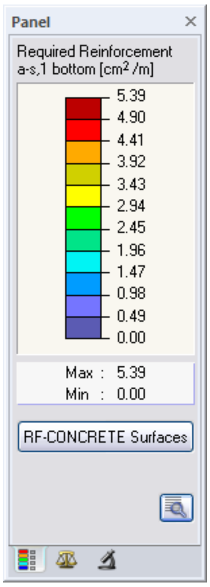


Figure 1.3: Panel button [RF-CONCRETE Surfaces]

2. Input Data

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

When you have started the module, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently. The pull-down list above the navigator contains the design cases that are already available (see chapter 7.1, page 60).

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

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


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

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

2.1 Type of Structure

The *Type of Structure*, that you define when you create a new structure, has a considerable influence on how the structural components will be stressed.

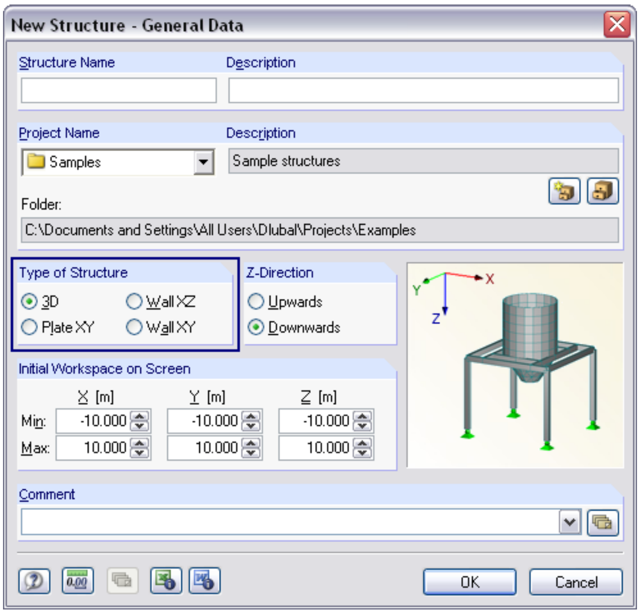


Figure 2.1: Dialog box *New Structure - General Data*, dialog section *Type of Structure*

When you select the structure type *Plate XY*, the structural component will be affected only by bending. The internal forces to be designed will be exclusively represented by moments whose vectors lie in the component's plane.

When you select *Wall XZ* or *Wall XY*, the structural component will be affected only by compression or by tension in the component's plane. The internal forces used for the design will be exclusively represented by axial forces whose vectors lie in the plane of the structural component.

When you set the spatial *3D* structure type, both loadings (moments and axial forces) will be combined. Therefore, the structural component may be affected by tension/compression and bending simultaneously. Thus, the internal forces to be designed will be represented by axial forces as well as by moments whose vectors lie in the component's plane.

2.2 General Data

In table 1.1 *General Data*, two tabs are available. Use these tabs to determine the loading for the *Ultimate Limit State* and the *Serviceability Limit State* designs.

2.2.1 Ultimate Limit State

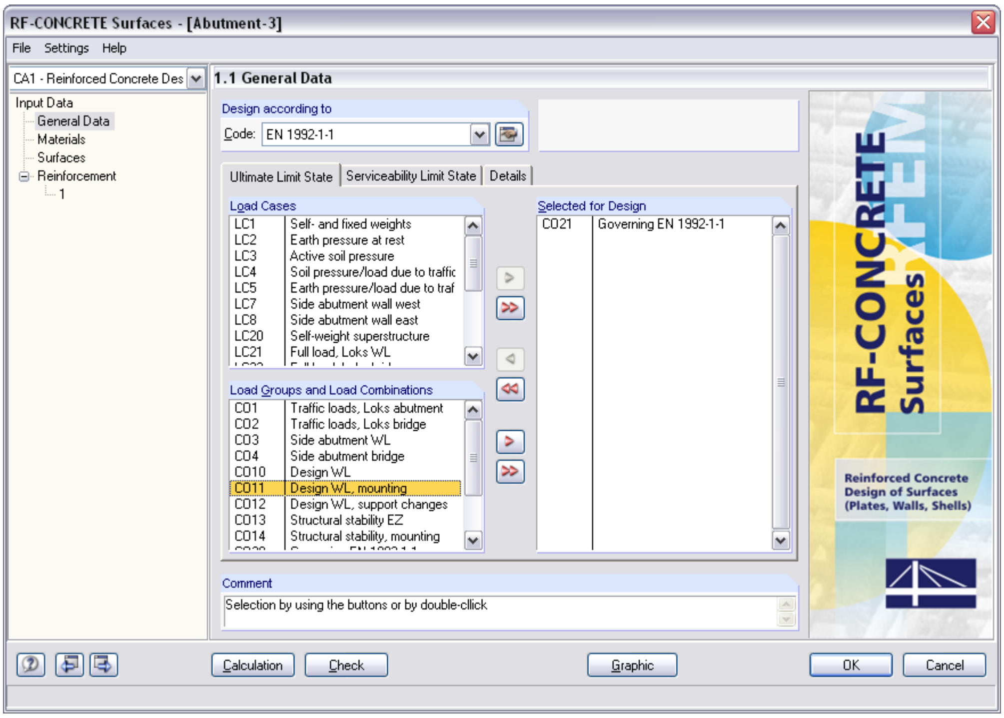


Figure 2.2: Table 1.1 *General Data*, tab *Ultimate Limit State*

Concrete Design According to Code / National Annex

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

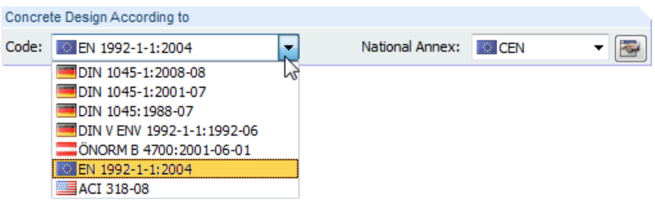


Figure 2.3: Selection of design standard

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

Figure 2.4: Selection of national annex

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

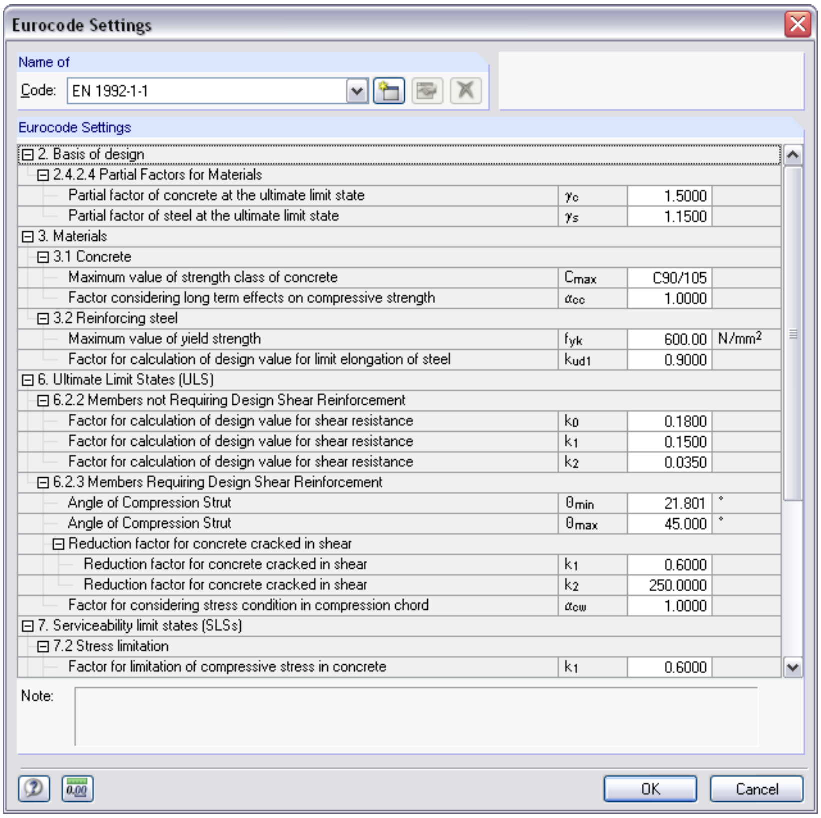


Figure 2.5: Dialog box *Eurocode Settings*

You can modify partial and reduction factors, angles of compression struts etc. to consider particular national user specifications.

Click the [New] button to create a copy of the original code. This copy can be modified appropriately and saved as a new Eurocode setting.

Load Cases / Load Groups and Load Combinations

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

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

Load cases that are marked by an asterisk (*) cannot be designed. This may be the case when no loads are defined or when the load case contains only imperfections.

When you design load combinations, please note that in this case, in contrast to the design of load cases or load groups, several sets of internal forces instead of a single one may exist on the respective design location. Depending on the defined type of structure, up to 16 sets of internal forces may be available. The analysis of all maxima and minima including corresponding internal forces affects the calculation speed accordingly.

Selected for Design

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

Comment

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

2.2.2 Serviceability Limit State

To access the tab *Serviceability Limit State*, you must select at least one action in the *Ultimate Limit State* tab. The defined standard applies to both tabs.

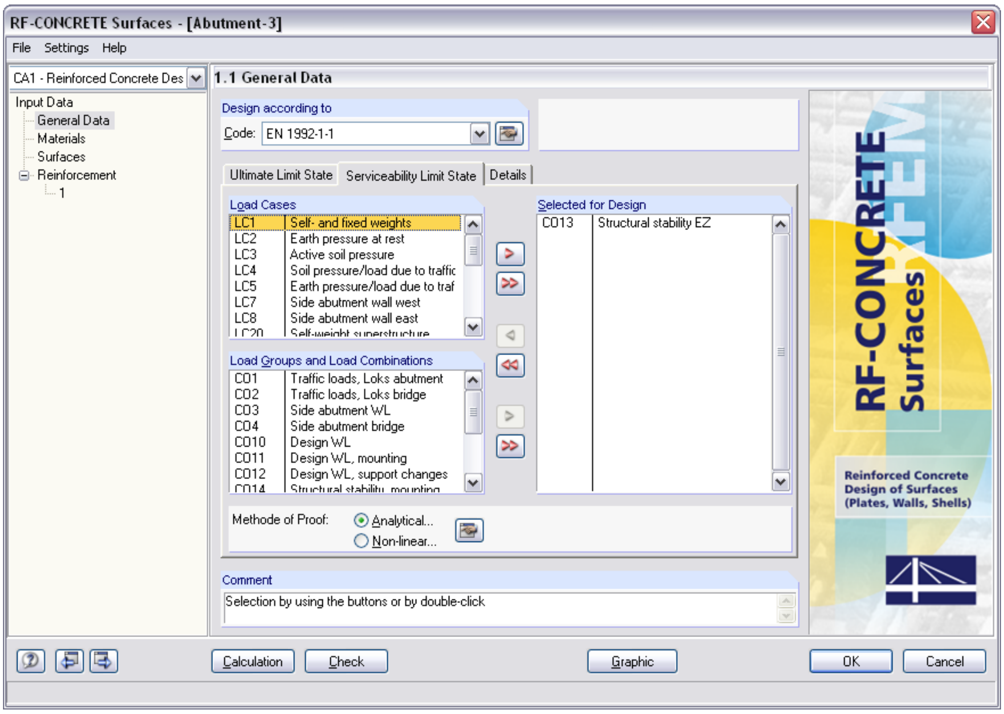


Figure 2.6: Table 1.1 *General Data*, tab *Serviceability Limit State*

Load Cases / Load Groups and Load Combinations


The load cases, groups and combinations (only for analytical design method!) that you want to design can be selected in the same way as described in the previous chapter 2.2.1.

Generally, the actions that are relevant for the serviceability limit state (SLS) are different from the ones considered for the ultimate limit state. The appropriate constellations must be defined already when superimposing load cases in RFEM.

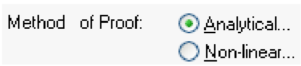
Selected for Design


The column on the right lists the loads selected for the design. As already described in the previous chapter, you can use the buttons [◀] or [◀◀] to remove selected or even all load cases, groups or combinations from the list.

Method of Proof

With the options in the dialog section below the tabs, you decide whether you want to carry out the serviceability limit state designs according to the analytical or the non-linear method.

2.2.2.1 Analytical Method



The serviceability limit state designs are carried out by default according to the *Analytical* method. This method uses the equations that are valid for the individual standards (for example DIN 1045-1, section 11). You find a detailed method description together with an example in the theoretical part of the German manual RF-BETON Flächen.

Use the [Edit Settings] button to open a dialog box where you can specify the settings for the calculation of crack widths.

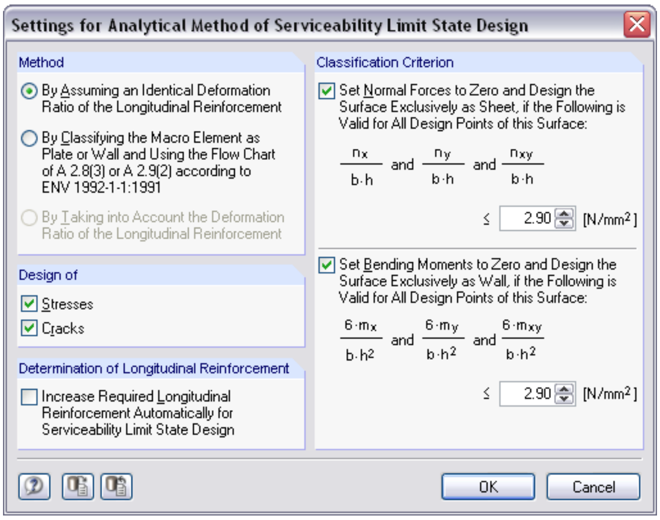


Figure 2.7: Dialog box *Settings for Analytical Method of Serviceability Limit State Design*

In the dialog section ***Method***, you decide which deformation ratio of the reinforcement set is applied for the serviceability limit state design. *By Assuming an Identical Deformation Ratio of the Longitudinal Reinforcement*, the program assumes a deformation ratio of 1.0 for the inserted reinforcement. This means that the rebars in the individual reinforcement directions are stressed in the same way. As a complete solution, this approach represents a quick and accurate design method. However, the choice of an appropriate inclination for the concrete strut plays a decisive role. Furthermore, this method represents a pure geometric distribution that is described in detail in the theoretical part of the German manual RF-BETON Flächen. This approach is applicable when the existing reinforcement corresponds more or less to the required reinforcement.

The option *By Classifying the Macro Element as 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 due to axial forces or bending moments exceed a certain stress. The corresponding limit value is defined in the dialog section *Classification Criterion*. This criterion is used to classify the relevant surface as plate (axial forces are set to zero) or wall (moments are set to zero). By neglecting minor internal force components, it is possible to use the flow chart shown in ENV 1992-1-1, annex A 2.8 or 2.9. The design internal forces are corresponding to the values displayed in the RFEM table 3.12 (cf. RFEM manual, chapter 9.12, page 285).

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

The option *By Taking into Account the Deformation Ratio of the Longitudinal Reinforcement* is enabled only for the 2D structure types "plate" and "wall" (cf. Figure 2.1, page 8). This method considers the effective deformation ratios due to the selected reinforcement and takes them into account for the serviceability limit state design.

The dialog section ***Classification Criterion*** is only available for 3D structure types. Use these settings to allow the neglect of minor *Normal Forces* and/or *Bending Moments* in order to design surfaces, in an idealized way, as pure plates or walls. For both options, the mean axial tensile strength f_{ctm} with 2.9 N/mm^2 of concrete C30/37 is preset as limit value. It is assumed that the tensile strength of concrete compensates a crack formation due to minor tensile stresses. This is the reason why they can be neglected.

If you have selected the simplified surface classification as plate or wall in the *Method* dialog section, you have to tick at least one of the two check boxes.

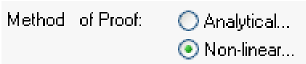
In the dialog section ***Design of***, you can determine if the program analyzes stresses and/or cracks for the design. You have to tick at least one of the two check boxes.

When you select *Stresses*, the program designs the concrete compressive stresses σ_c and the steel stresses σ_s. When you select *Cracks*, the module designs the minimum reinforcements $a_{s,min}$, the limit diameters lim d_s, the maximum crack spacings max s_l as well as the crack widths w_k. The settings for the individual designs are defined in table 1.3 *Surfaces* (see chapter 2.4, page 19).

In the dialog section ***Determination of Longitudinal Reinforcement***, you decide if the longitudinal reinforcement is dimensioned so that the serviceability limit state designs will be fulfilled. If the check box is not ticked, the program uses the settings specified in table 1.4 *Reinforcement*, tab *Longitudinal Reinforcement* (see chapter 2.5.3, page 30): application of basic reinforcement, required reinforcement or basic reinforcement with additional reinforcement.

The dimensioning of the reinforcement for the serviceability limit state design is determined by increasing the reinforcement iteratively. As initial value for the iterations to absorb the given characteristic load, the program uses the required ULS 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 tables will mark the respective point as non-designable.

2.2.2.2 Non-linear Method



To perform the *Non-linear* design method, you need a license for the add-on module **RF-CONCRETE NL**. For more theoretical details on this design method, see the German manual RF-BETON Flächen.

The non-linear design method excludes load combinations. The interaction between structure and load requires a clear distribution of internal forces. In a load combination, however, maximum and minimum values are available for each FE node.

Use the [Edit Settings] button in the tab *Serviceability Limit State* of table 1.1 (see Figure 2.6, page 11) to open a dialog box where you can define the conditions for the non-linear serviceability limit state design.

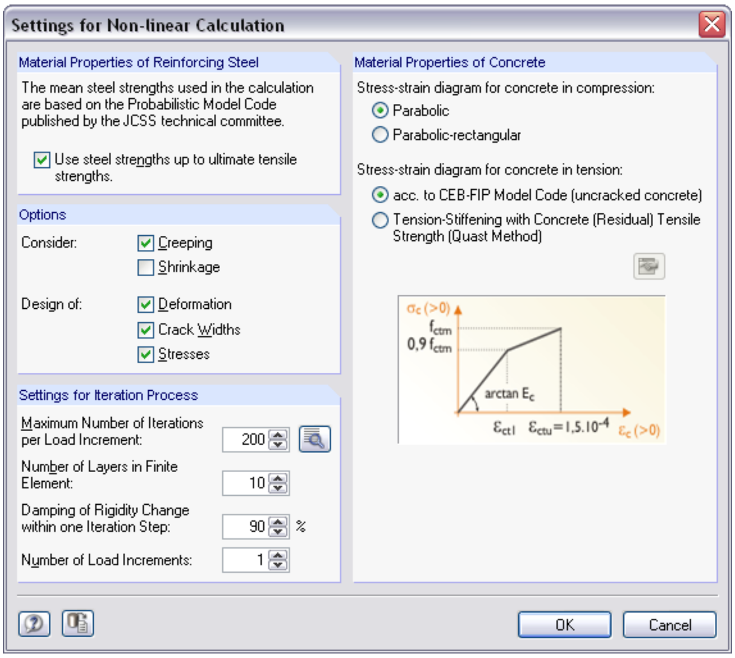


Figure 2.8: Dialog box *Settings for Non-linear Calculation*

In the dialog section ***Material Properties of Reinforcing Steel***, you decide if the calculation in the plastic range of the reinforcing steel's stress-strain diagram will be carried out with a rising or a horizontal graph.

The settings in the dialog section ***Options*** determine if the influence of *Creeping* and *Shrinkage* is considered in the calculation. In addition, you can decide which serviceability limit state designs (deformation, crack widths, stresses) are performed. You have to tick at least one of the three check boxes.

The settings for creeping and shrinkage and all individual designs are defined in table 1.3 *Surfaces* (see chapter 2.4.2, page 22).

In the dialog section ***Settings for Iteration Process***, you define all settings that affect the process of the non-linear design method. For more theoretical information on this method, see the German manual RF-BETON Flächen. When modifying the accuracy of iterations, take care that the *Maximum Number of Iterations* is higher than the step in the calculation process from which the deformation criterion will be taken into account additionally. Use the [Details] button to open the RFEM dialog box *Calculation Parameters* where you can adjust the precision of the convergence criteria for the non-linear calculation.

In the non-linear calculation, the program divides the surface into several *Layers*. The recommended number of layers is 10.

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

The dialog box ***Material Properties of Concrete*** (cf. Figure 2.8) determines whether a parabolic or a parabolic-rectangular graph is applied to the stress-strain diagram of the concrete *in compression*. Furthermore, you can define the concrete's stress-strain diagram *in tension*. In accordance with the CEB-FIB model code, the presetting assumes a non-cracked concrete.

When you select the concrete's stiffening effects in tension (*Tension-Stiffening*) for the stress-strain diagram, you can specify the settings for the application of the tensile strength of concrete between the cracks in a separate dialog box. To open the dialog box, use the button shown on the left.

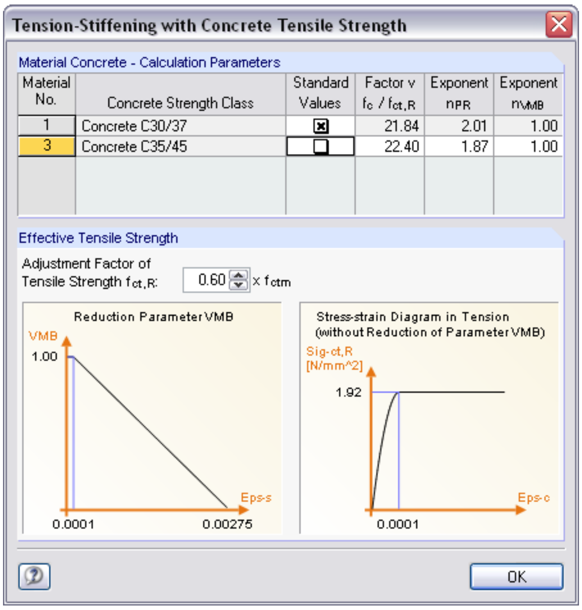


Figure 2.9: Dialog box *Tension-Stiffening*

More information about tension-stiffening can be found in the German manual RF-BETON Flächen.

The two graphics in the dialog box represent dynamically the modifications of parameters.

2.2.3 Details

If you perform the design according to EN 1992-1-1 or one of the national EN application documents, the *Details* tab is displayed additionally. This tab is unnecessary for serviceability limit state designs according to DIN 1045-1 because the factor k_t is generally defined with 0.4 in equation (136), section 11.2.4 (2).

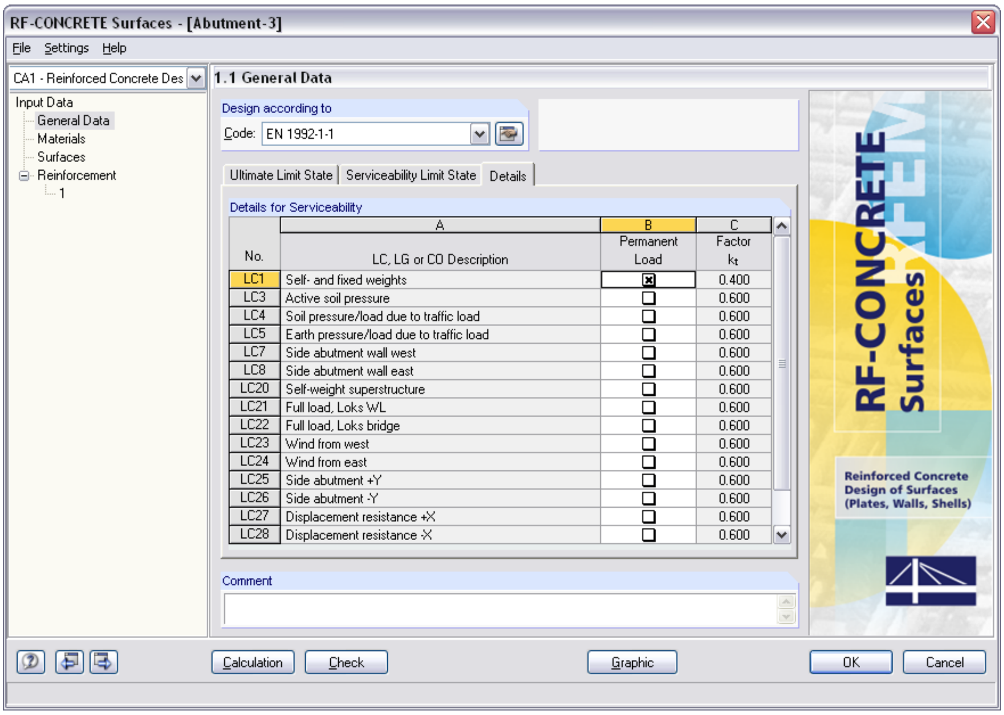


Figure 2.10: Table 1.1 *General Data*, tab *Details*

During the crack width analyses, the program calculates the differences of the mean strains of concrete and reinforcing steel. According to equation (7.9) of EN 1992-1-1, section 7.3.4 (2), the load duration factor k_t must be specified for these differences.

LC, LG or CO Description

Column A lists all load cases, groups and combinations that have been selected for design in the *Serviceability Limit State* tab. The load cases contained in these load groups and combinations appear in this list, too.

Permanent Load

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

Factor k_t

The load duration factor k_t is used to determine the loading period. For a long-term load action, the factor is 0.4. For a short-term action, the factor is 0.6.

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

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

Equation 2.1

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

2.3 Materials

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

The material properties required for the determination of internal forces in RFEM are described in detail in chapter 5.3 of the RFEM manual. The design relevant material properties are stored in the global material library and preset automatically.

To adjust the units and decimal places of material properties and stresses, select **Units and Decimal Places** on the module's **Settings** menu (see Figure 7.5, page 62).

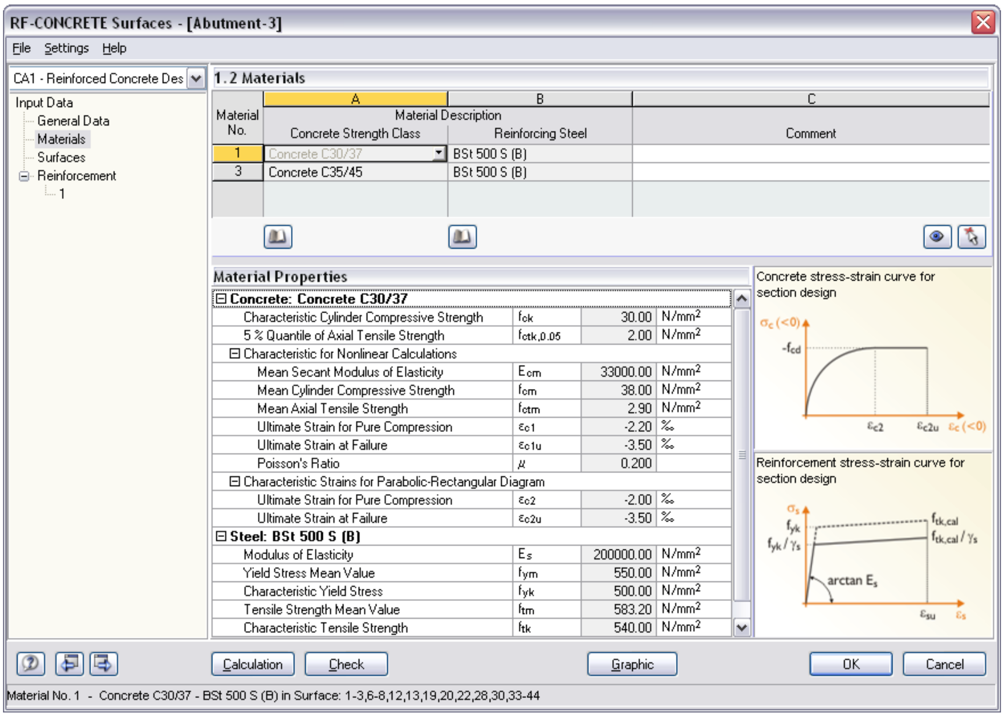
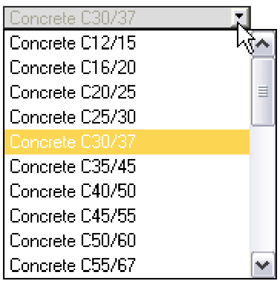


Figure 2.11: Table 1.2 *Materials*

Material Description

Concrete Strength Class



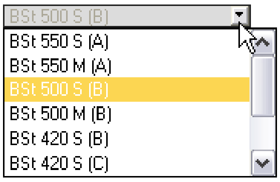
The concrete materials defined in RFEM are already preset; different materials are hidden. When the *Material Description* corresponds to an entry of the material library, RF-CONCRETE Surfaces will import the appropriate material properties.

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

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

Reinforcing Steel

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



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

The import of materials from the library is described below.

Material Library

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

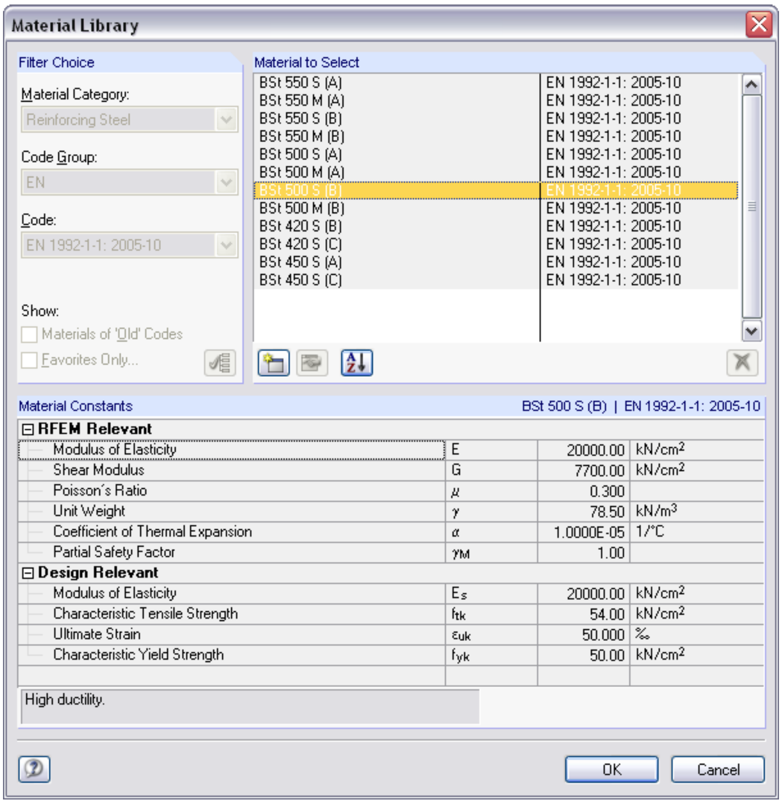


Figure 2.12: Dialog box *Material Library*

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

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

Chapter 5.3 in the RFEM manual describes in detail how materials can be added or rearranged. By using the [Create New Material] button, you can create a new concrete or reinforcing steel with user-defined material properties and store it for later use.

2.4 Surfaces

In this table, the surfaces that you want to design are listed. In addition, you can define the settings for the serviceability limit state design, provided that you have selected at least one action in the *Serviceability Limit State* tab of table 1.1 *General Data* (cf. Figure 2.6, page 11).

The table's appearance depends on the design method that has been specified for the serviceability limit state designs (SLS) in table 1.1. When you design the ultimate limit state exclusively, only the thicknesses of the respective surfaces are listed.

2.4.1 Analytical Method for SLS

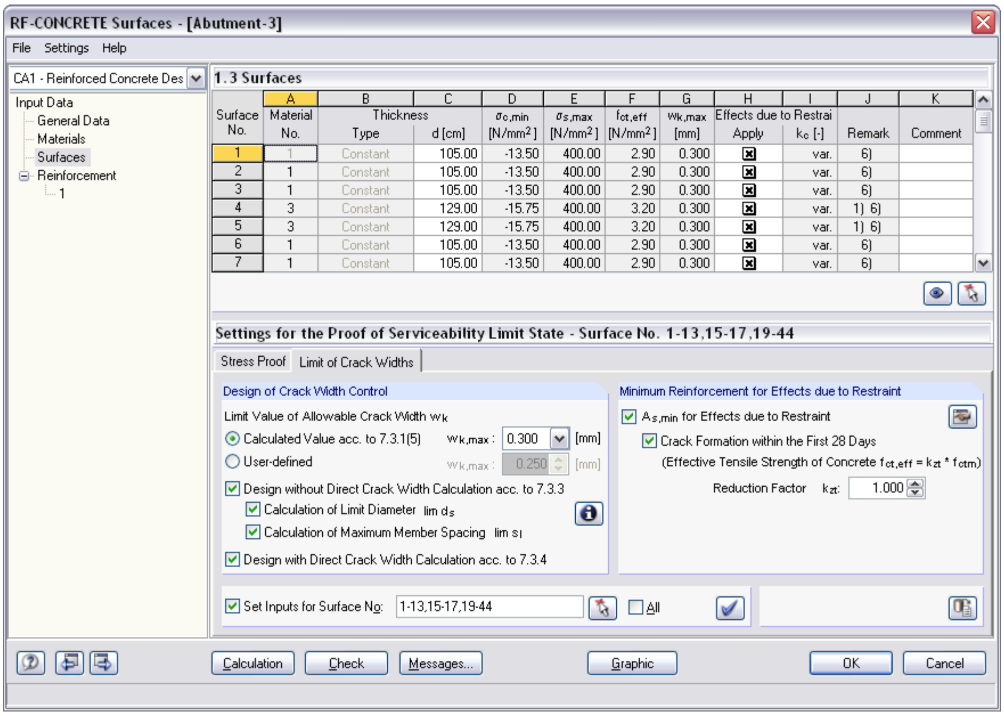


Figure 2.13: Table 1.3 *Surfaces* with settings of the analytical design method for the serviceability limit state

Material No.

For each surface, the table shows the material numbers that are managed in table 1.2 *Materials*. The assigned materials cannot be changed in this table.

Thickness

Type

As additional information, the table displays the thickness types that have been assigned in RFEM. The program is able to design constant and linearly variable thicknesses as well as orthotropic surfaces (without non-linear method for SLS).

d

This column shows the thicknesses that have been imported from RFEM. It is always possible to modify the preset values for the design.

Please note that the design will be carried out with the internal forces determined in RFEM which refer to the stiffnesses of the surface thicknesses defined in the main program. In case of a statically indeterminate structure, the surface thicknesses modified in RF-CONCRETE Surfaces must be adjusted in RFEM, too, in order to ensure a correct design.

The number of the subsequent table columns depends on the tabs displayed in the lower table part. The tabs are determined by the settings in the *Settings* dialog box that you can access in the *Serviceability Limit State* tab of table 1.1 *General Data* (see Figure 2.7, page 12). In this dialog box, you can decide whether the program designs stresses and/or cracks.

The values cannot be entered directly into the columns, but are determined by the tab settings in the lower table part. Please note that, by default, the tab specifications apply exclusively to the surface selected in the upper table part. It is possible, however, to assign the current specifications to another surface, or to all surfaces or only some of them. In this case, tick the check box *Set Inputs for Surface No* and clear the *All* check box to the right (cf. Figure 2.13). Then you can enter the numbers of the relevant surfaces manually. You can also select them graphically in the RFEM work window. If you want to apply the settings to all surfaces, tick the *All* check box. Finally, click the button [☑] to assign the specifications to the selected surfaces. The assignment is only applicable for the current tab (for example *Stress Proof*).

$\sigma_{c,min}$

This column shows the respective value of the minimum concrete stress for the limitation of the concrete pressure stresses. According to DIN 1045-1, the following applies to

- quasi-permanent action combination, if serviceability, ultimate limit state or durability are considerably affected by creeping:
  $\sigma_c \leq 0.45 \cdot f_{ck}$
- rare action combination in exposure classes XD1 to 3, XF1 to 4, XS1 to 3:
  $\sigma_c \leq 0.60 \cdot f_{ck}$

The parameters are defined in the *Stress Proof* tab (see Figure 2.15).

$\sigma_{s,max}$

This value represents the maximum reinforcing steel stress for the limitation of the reinforcement's tensile stresses. According to DIN 1045-1, the following applies to

- rare action combination:
  $\sigma_s \leq 0.80 \cdot f_{yk}$
- exclusive effects due to restraint:
  $\sigma_s \leq 1.00 \cdot f_{yk}$

The parameters are defined in the *Stress Proof* tab (see Figure 2.15).

$f_{ct,eff}$

This column displays the respective value of the effective tensile strength of the concrete. This value is required to control the rebar diameter.

The parameters are defined in the tab *Limit of Crack Widths* (see Figure 2.13). In this tab, you can also specify detail settings concerning the minimum reinforcement for effects due to restraint by using the [Edit] button.

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

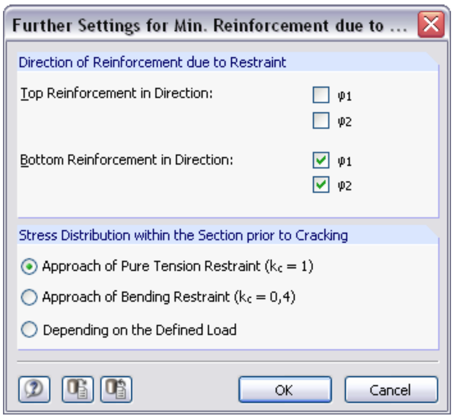


Figure 2.14: Dialog box *Further Settings for Min. Reinforcement due to Restraint*

$w_{k,max}$

This value represents the maximum allowable crack width.

The parameters are defined in the tab *Limit of Crack Widths* (see Figure 2.13).

Remark

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

Comment

This input field can be used to enter user-defined comments.

2.4.2 Non-linear Method for SLS

To design according to the non-linear method, a license of the add-on module **RF-CONCRETE NL** is required.

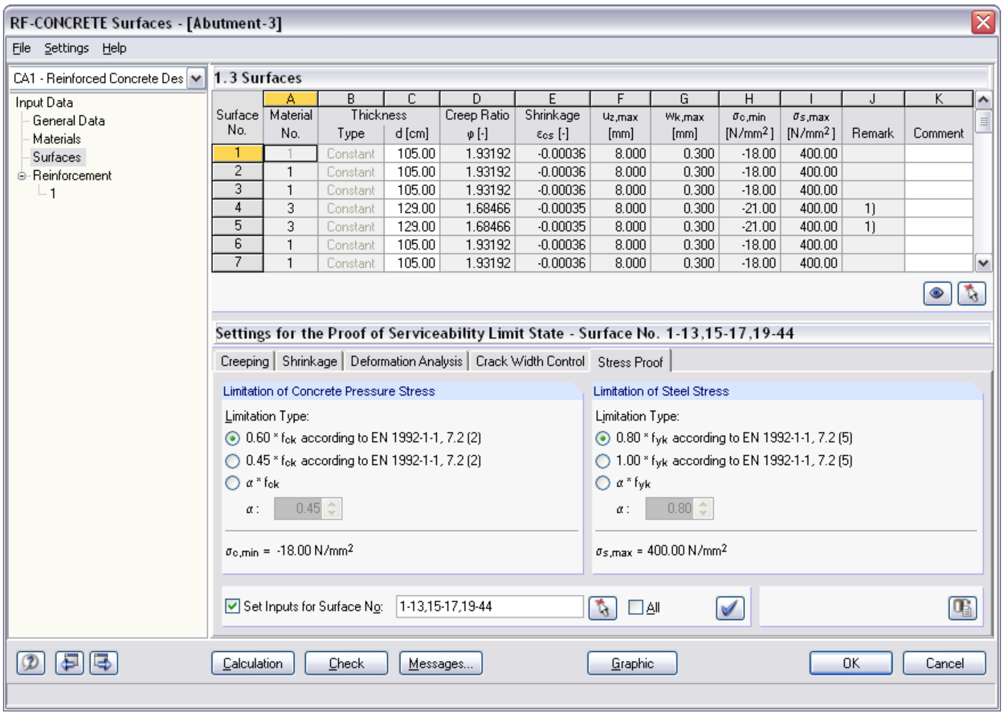


Figure 2.15: Table 1.3 *Surfaces* with settings of the non-linear design method for the serviceability limit state

The table columns *Material* and *Thickness* are similar to the ones in the analytical design method. They are described in the previous chapter 2.4.1 on page 19.

The values of the subsequent columns cannot be entered directly, but are determined by the tab settings in the lower table part. Please note that, by default, the tab specifications apply exclusively to the surface selected in the upper table part. It is possible, however, to assign the current specifications to another surface, or to all surfaces or only some of them. In this case, tick the check box *Set Inputs for Surface No* and clear the *All* check box to the right. Then you can enter the numbers of the relevant surfaces manually. You can also select them graphically in the RFEM work window. If you want to apply the settings to all surfaces, tick the *All* check box. Finally, click the button [☑] to assign the specifications to the selected surfaces. The assignment is only applicable for the current tab.

Creep Ratio φ

The parameters for creeping are defined in the *Creeping* tab. Based on these conditions, RF-CONCRETE NL determines the creep ratio. For the effective thickness of the structural component h_0, the program assumes the surface thickness d for the calculation.

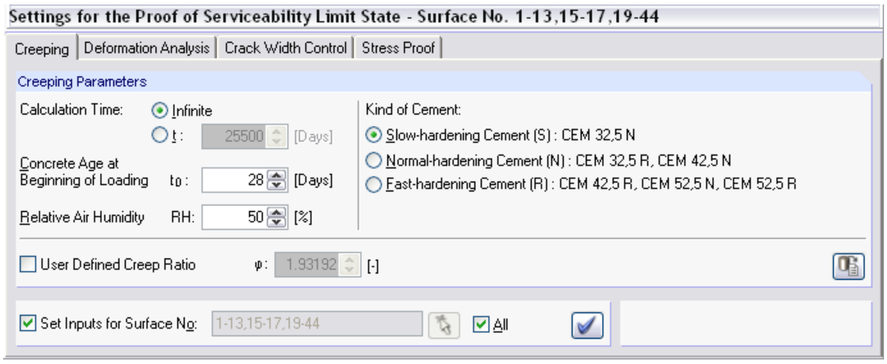


Figure 2.16: Dialog section *Settings for the Proof of Serviceability Limit State,* tab *Creeping*

The determination of the creep ratio φ is described in detail in the German manual RF-BETON Flächen.

Shrinkage ε_{cs} (in preparation)

This column displays the respective shrinkage. The corresponding parameters are defined in the *Shrinkage* tab. Based on these conditions, the program determines the appropriate shrinkage automatically. For the effective thickness of the structural component h_0, the program assumes the surface thickness d for the calculation.

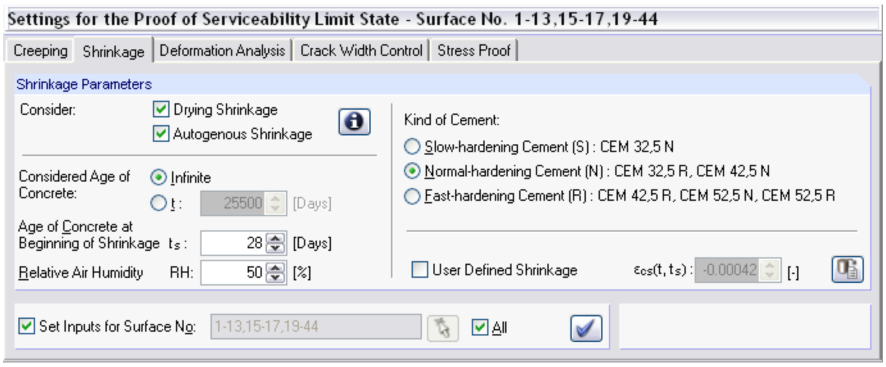


Figure 2.17: Dialog section *Settings for the Proof of Serviceability Limit State,* tab *Shrinkage*

The determination of the shrinkage ε_{cs} is described in detail in the German manual RF-BETON Flächen. If you do not want to apply any strain due to shrinkage for particular surfaces, set 0 ‰ for the user-defined shrinkage in the *Shrinkage* tab.

Please note that you cannot consider any shrinkage for the structure type *Plate XY* (cf. Figure 2.1, page 8) because this type provides degrees of freedom only for bending.

$u_{z,max}$

This value represents the maximum allowable deformation that must be observed for the serviceability limit state design. The corresponding parameters are defined in the *Deformation Analysis* tab.

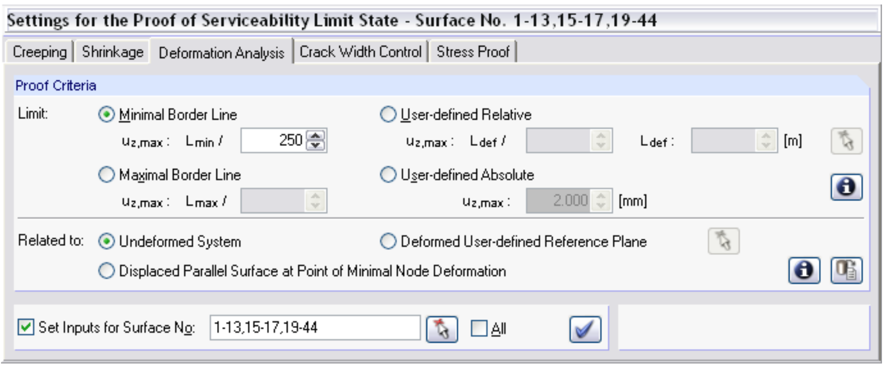


Figure 2.18: Table 1.3 *Surfaces*, tab *Deformation Analysis*

To ensure the serviceability limit state for common structures, for example according to DIN 1045-1, the deflection due to the quasi-permanent action combination must not exceed the following limit values:

- common case $\quad u_{z,max} = \frac{l_{eff}}{250}$
- structural components for which excessive deformations can result in subsequent damages $\quad u_{z,max} = \frac{l_{eff}}{500}$

The selection fields *Minimal Border Line*, *Maximal Border Line* and *User-defined Relative* determine the length that is used for l_{eff}. For the two border line options, the program applies the shortest or longest border line of the respective surface (see Figure 2.19). For the user-defined relative specification, you can enter the length directly or select it graphically between two points in the RFEM structure. For all three options, you must define a divisor by which the defined lengths will be divided.

When you select *User-defined Absolute*, you can specify the allowable maximum deformation $u_{z,max}$ directly.

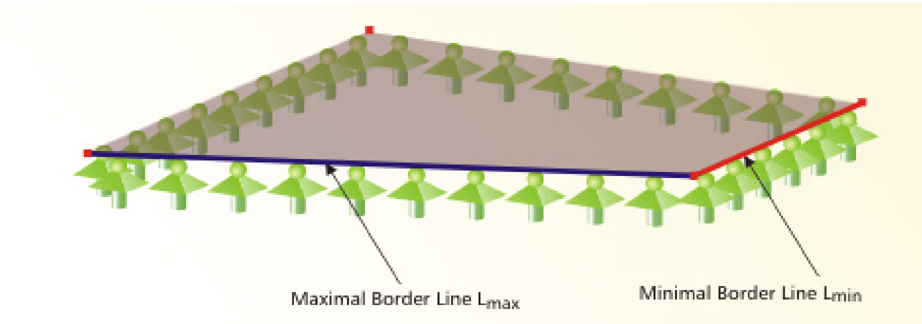


Figure 2.19: Border lines for the determination of $u_{z,max}$

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

The deformation design criterion takes into account the sag of a surface, that means the "vertical deformation relative to the shortest line connecting the points of support". Therefore, three options are available in the lower part of the *Deformation Analysis* tab (Figure 2.18). You can decide on how the locally applied deformation $u_{z,local}$ should be calculated for the deformation analysis.

- *Undeformed System*: The deformation is related to the initial structure.
- *Displaced Parallel Surface*: In case of a semi-rigid support of the surface, select this option to consider the existing deformation. The deformation $u_{z,local}$ refers to a virtual reference surface that is displaced parallel to the undeformed system. The displacement vector of this reference surface is as long as the minimal nodal deformation within the surface.

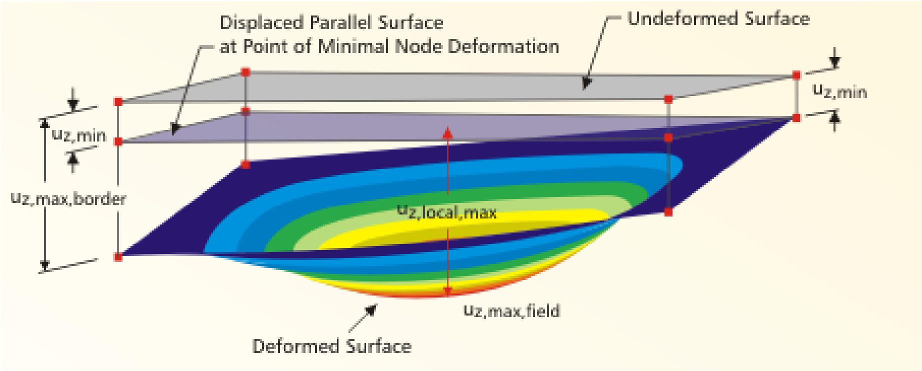


Figure 2.20: Displaced parallel surface at position of minimal deformed node

- *Deformed Reference Plane*: In case the support deformations of a surface differ considerably from each other in size and degree, you can define a slanted reference plane to which the deformation $u_{z,local}$ is related. This plane is defined by three points of the undeformed system. Then, RF-CONCRETE NL calculates the deformation of the three definition points and places the reference plane in the displaced points in order to calculate the local deformation $u_{z,local}$.

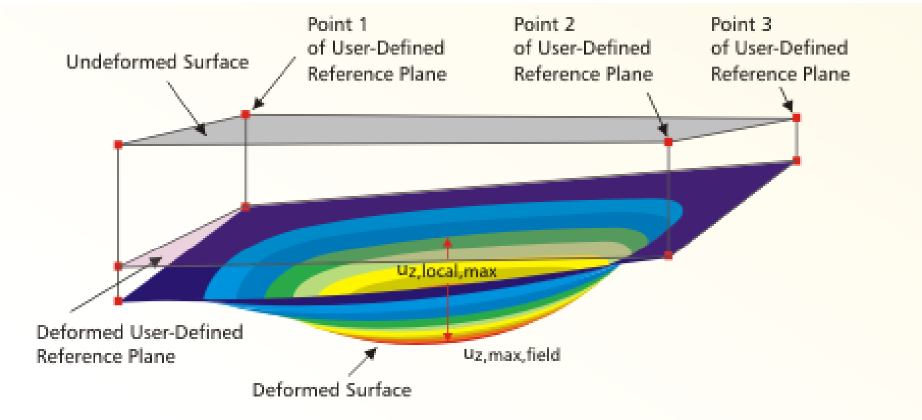


Figure 2.21: Displaced user-defined reference plane

$w_{k,max}$

This value represents the maximum allowable crack width.

The parameters are defined in the tab *Crack Width Control* (cf. Figure 2.13).

$\sigma_{c,min}$

High concrete pressure and reinforcing steel stresses in the serviceability limit state affect the serviceability and durability adversely. This table column shows the respective value of the minimum concrete stress for the limitation of the concrete pressure stresses.

The parameters are defined in the *Stress Proof* tab (see Figure 2.15).

$\sigma_{s,max}$

This value represents the maximum reinforcing steel stress for the limitation of the reinforcement's tensile stresses.

The parameters are defined in the *Stress Proof* tab (see Figure 2.15).

2.5 Reinforcement

This table consists of five tabs where all reinforcement data is specified. As the individual surfaces often require different settings, you can create several reinforcement groups for each RF-CONCRETE Surfaces case. The reinforcement definitions can be entered surface by surface.

Reinforcement groups

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

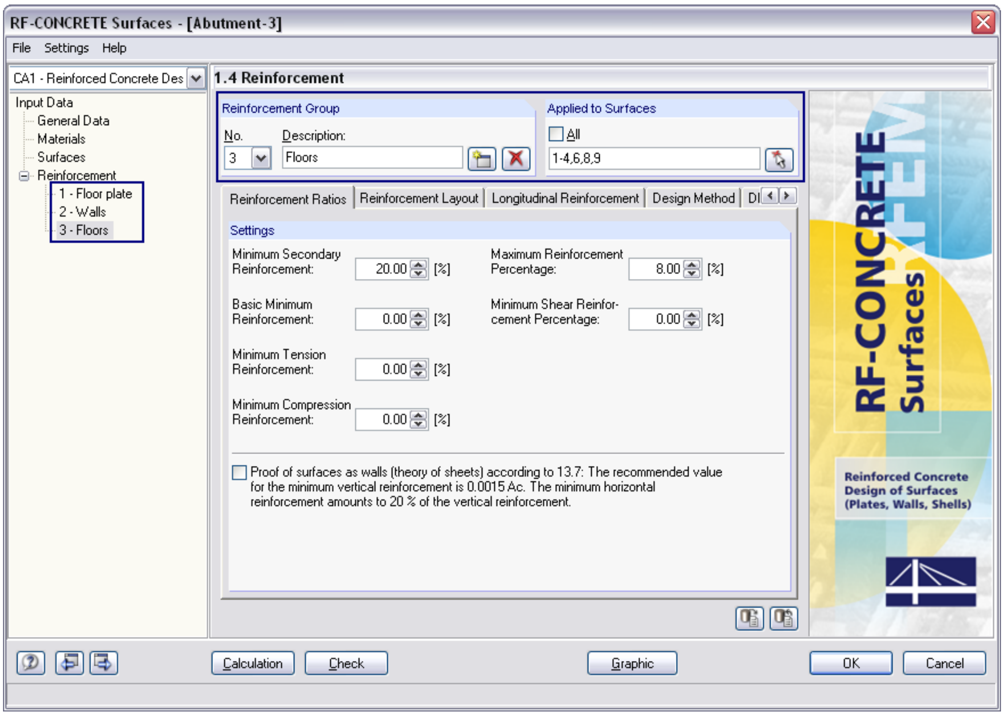
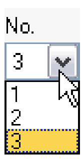


Figure 2.22: Table 1.4 *Reinforcement* with three reinforcement groups



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

By using the [Delete] button, the currently selected reinforcement group is deleted from the RF-CONCRETE Surfaces case without any additional warning. Surfaces that were contained in such a reinforcement group won't be designed. To design those surfaces, you must reassign them to a new or an existing reinforcement group.

In the dialog section *Applied to Surfaces*, you determine the surfaces to which the current reinforcement group is applied. *All* surfaces are preset. If the check box is ticked, no further reinforcement group can be created because surfaces cannot be assigned to different reinforcement specifications. Therefore, clear the *All* check box if you want to use several reinforcement groups.

In the input field below, you can enter the numbers of the relevant surfaces manually. You can also use the [Pick] button to select them graphically in the RFEM work window. Furthermore, the button [Create New Reinforcement Group] is enabled. Only surface numbers that have not yet been assigned to other reinforcement groups are allowed to be entered in the input field.

2.5.1 Reinforcement Ratios

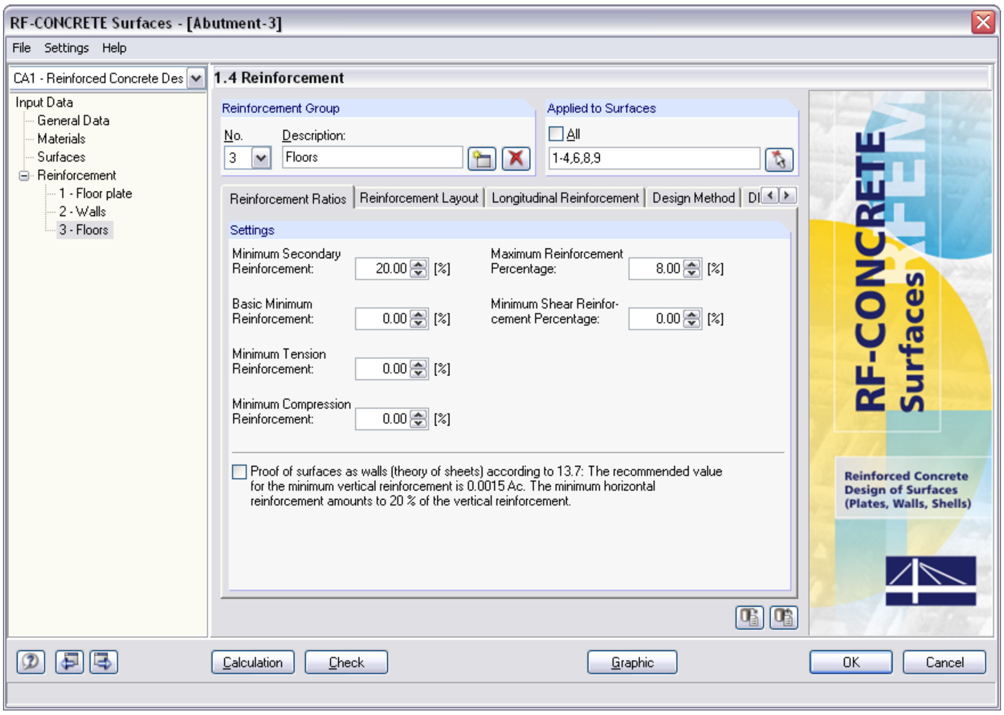


Figure 2.23: Table 1.4 *Reinforcement*, tab *Reinforcement Ratios*

This tab defines the minimum and maximum reinforcements in percentage. The *Minimum Secondary Reinforcement* is related to the maximum longitudinal reinforcement that is to be applied. All further specifications are set in relation to the cross-section area of a surface stripe with the width of one meter. An exception represents the *Minimum Compression Reinforcement* according to DIN 1045-88 that is related to the statically required cross-section.

Find examples for the minimum and maximum reinforcements in the German manual RF-BETON Flächen.

To perform wall-specific designs, for example according to DIN 1045-1, section 13.7, or EN 1992-1-1, section 9.6, select the option *Proof of surfaces as walls*. Depending on the standard selected in table 1.1 *General Data*, the program takes into account criteria of the minimum reinforcements for walls.

2.5.2 Reinforcement Layout

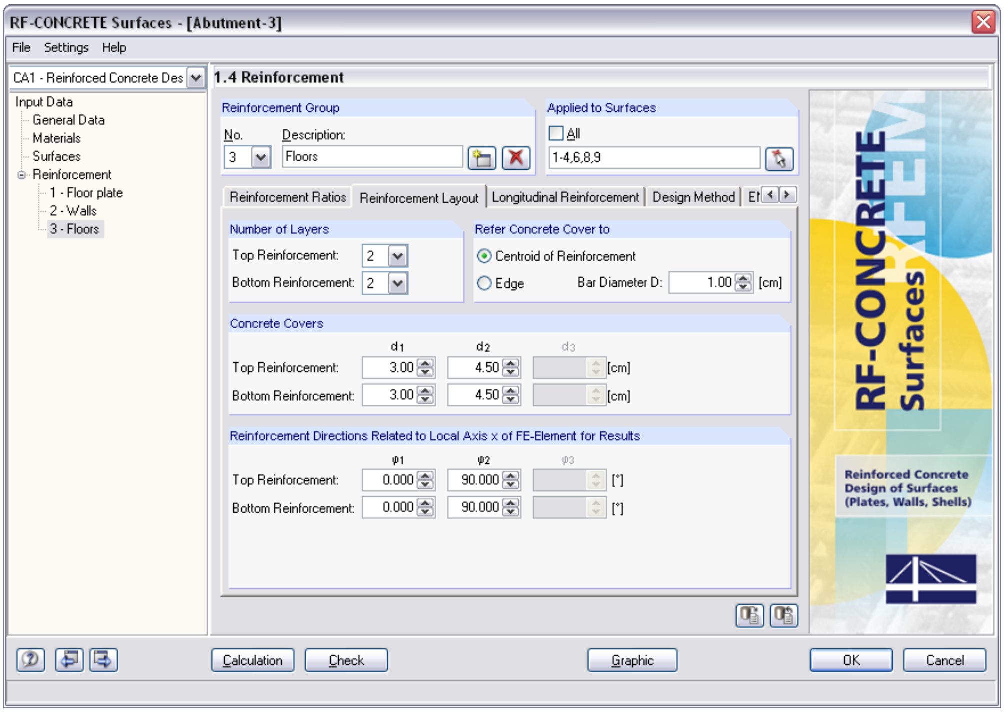
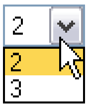


Figure 2.24: Table 1.4 *Reinforcement*, tab *Reinforcement Layout*

This tab determines the geometric reinforcement specifications of the reinforcement group.

Number of Layers



The reinforcement mesh can be applied in the form of two or three reinforcement layers for each surface side. For serviceability limit state designs, only a two-layer reinforcement mesh is allowed.

The definition of the "top" and "bottom" side of the surface can be found in the description of the *concrete covers* below.

Refer Concrete Cover to

The concrete covers defined in the following can be related to the reinforcement's *Centroid* or *Edge* distance .

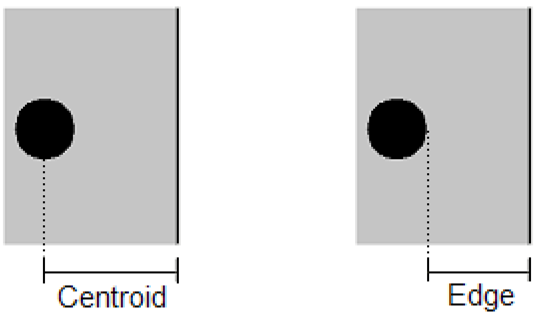


Figure 2.25: Relations of concrete cover

When you design only the ultimate limit state and you select the second option, you must specify the *Bar Diameter D*. In the dialog section below, you define the edge distance c (that means the nominal value of the concrete cover c_{nom}).

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

Concrete Covers

In these input fields, you specify the concrete covers for both surface sides. The dimensions represent either the centroids of the individual layers or the reinforcements' edge distances c_{nom} in direction φ_1. The reinforcement directions are defined in the dialog section below.

Top and *Bottom* layers

The "top" and "bottom" surface side is defined as follows: The bottom side is defined in direction of the positive local surface axis z, the top side is defined accordingly in direction of the negative local axis z. To display the surface axes, use the *Display* navigator in RFEM and select the entries *Structure* → *Surfaces* → *Surface Axis Systems x,y,z*. You can also use the context menu of the corresponding surface.

If you want to reverse the local z-axis of a surface, right-click the corresponding surface in the RFEM work window to open its context menu. Then select *Reverse Local Axis System*. In this way, it is possible to unify, for example, the orientations of walls and to assign the top and bottom reinforcement sides for surfaces in vertical position appropriately.

If the structure in RFEM was created with the structure type *Wall* (cf. Figure 2.1, page 8), it is not possible to generate different reinforcement meshes for both surface sides. In this case, the module allows only for input options concerning uniform concrete covers so that they can be applied synchronously to both surface sides.

Reinforcement Directions

The reinforcement directions φ refer respectively to the local x-axis of the finite elements in the surfaces' results axis system. The angle φ must be entered with a positive value. It represents the clockwise rotation of the reinforcement direction in relation to the corresponding x-axis.

The results axis system belongs to the surface's properties which can be edited and adjusted, if necessary, in the RFEM dialog box *Edit Surface*.

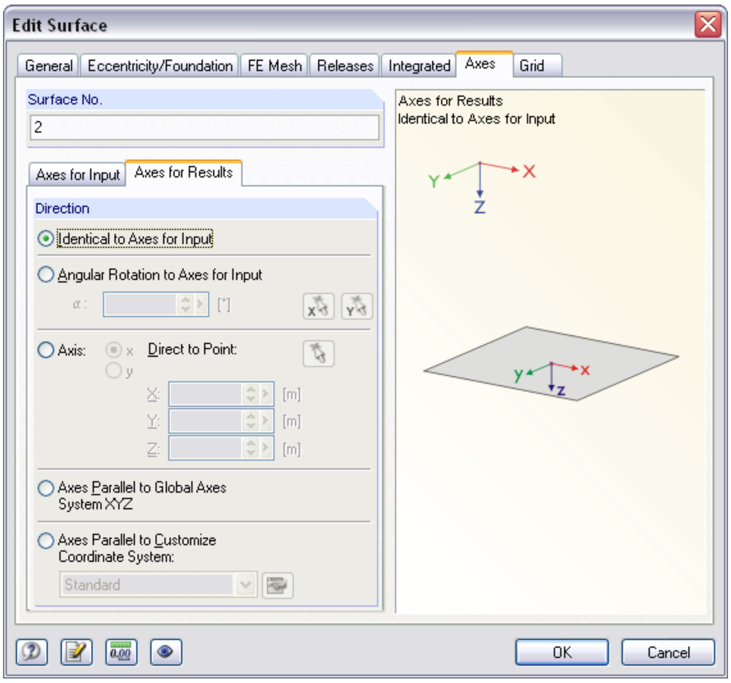


Figure 2.26: RFEM dialog box *Edit Surface*, tab *Axes*

The axes of the finite elements can be checked graphically in the RFEM work window by selecting the following entries in the *Display* navigator: *FE Mesh* → *On Surfaces* → *FE Axis Systems x,y,z* or *Numbering* → *FE Mesh* → *FE Axis Systems x,y,z* (cf. Figure 9.30 on page 281 in the RFEM manual).

If the structure in RFEM was created with the structure type *Wall* (cf. Figure 2.1, page 8), it is not possible to generate different reinforcement meshes for both surface sides. In this case, the module allows only for input options concerning uniform directions of the reinforcement sets so that they can be applied synchronously to both surface sides.

2.5.3 Longitudinal Reinforcement

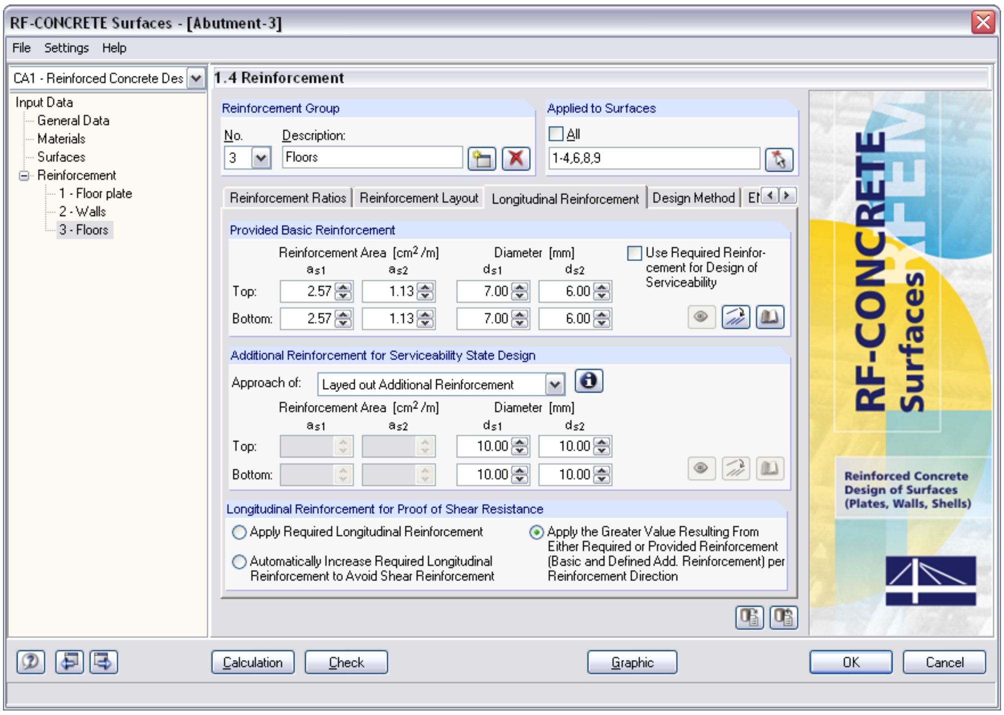


Figure 2.27: Table 1.4 *Reinforcement*, tab *Longitudinal Reinforcement* for ultimate and serviceability limit state designs

The tab's appearance depends on the designs that you want to perform: For the serviceability limit state designs (SLS), you must specify reinforcement areas. If you want to carry out the ultimate limit state design exclusively, no specific reinforcement settings are required. You only need to decide which longitudinal reinforcement you want to use for the shear design.

For more information on the reinforcement specifications for the SLS design, see the German manual RF-BETON Flächen.

Provided Basic Reinforcement

For each surface side and each reinforcement direction, you can define a basic reinforcement that will be inserted respectively in all surfaces of the reinforcement group. In the corresponding input fields, enter the *Reinforcement Area* and the *Diameter* that is relevant for the serviceability limit state design.

If the user-defined basic reinforcement exceeds the maximum required reinforcement, no additional reinforcement is needed. However, big basic reinforcements are usually not applied to surfaces because this would not be efficient.

Entering reinforcement areas is facilitated by libraries that are available for rebars and mesh reinforcements. To access these libraries, use the two buttons shown on the left.

In the libraries (cf. Figure 2.28 and Figure 2.29), you can select rebars or reinforcing steel meshes. The reinforcement areas defined or stored in the libraries can be transferred to the input fields of the dialog section.

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

Libraries

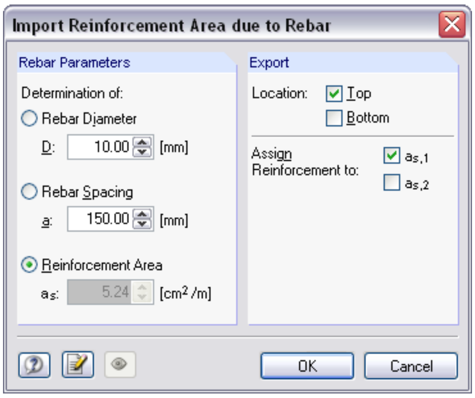


Figure 2.28: Dialog box *Import Reinforcement Area due to Rebar*

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

In the dialog section *Export*, you decide in which input fields of the initial dialog box the determined reinforcement areas will be imported. The location as well as the reinforcement direction can be defined specifically (or generally by ticking all check boxes).

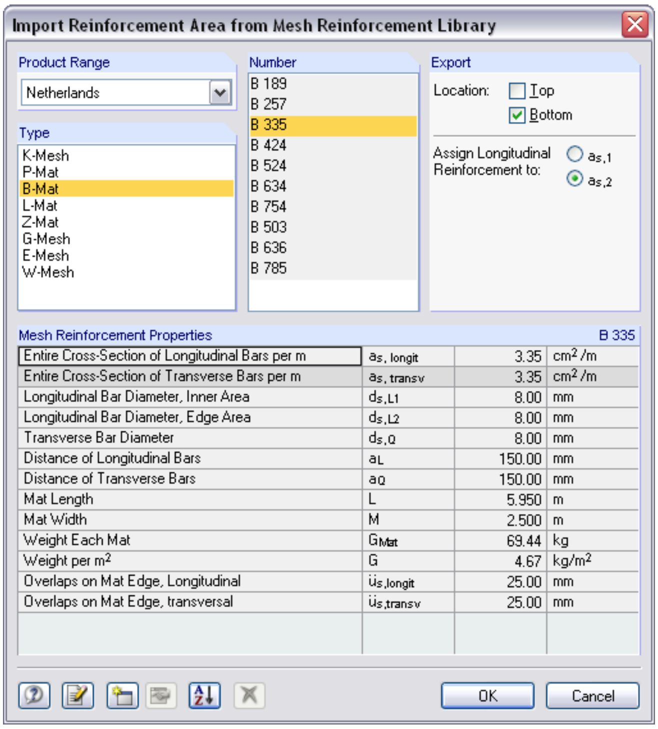
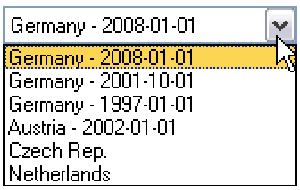


Figure 2.29: Dialog box *Import Reinforcement Area from Mesh Reinforcement Library*



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

In the dialog section *Export*, you decide in which input fields of the initial dialog box the determined reinforcement areas will be imported. The location as well as the reinforcement direction can be defined specifically (or generally by ticking all check boxes).

Use Required Reinforcement for Design of Serviceability

The ideal way to perform serviceability limit state designs would be the following:

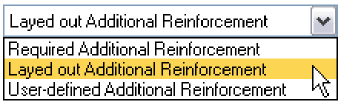
1. Determine the required reinforcement by using only the load specified in the *Ultimate Limit State* tab
2. Create a reinforcement drawing by considering the colored result diagram including mesh reinforcements and rebars
3. If necessary, divide the surfaces in RFEM on the basis of the reinforcement drawing into smaller surfaces that have the same existing reinforcement area in each reinforcement direction
4. Define the existing reinforcement area, rebar spacing and diameter for each surface in the add-on module RF-CONCRETE Surfaces
5. Calculate once again with the loads of the *Serviceability Limit State* tab

This procedure seems to be time-consuming and, in some way, is contrary to the program's convention indicating that you can determine the reinforcement and perform the serviceability limit state designs by simply using the [Calculation] button.

Therefore, you can tick the check box *Use Required Reinforcement* to quickly get a provided reinforcement for the individual surfaces: The program uses the required reinforcement from the ultimate limit state design as the reinforcement that is to be applied. When the check box is ticked, you only need to specify the rebar diameter.

Additional Reinforcement

This dialog section is only available for the serviceability limit state design.



In the areas where the statically required reinforcement exceeds the defined basic reinforcement, an additional reinforcement is needed. Use the selection list in this dialog section to define the additional reinforcement that you want to apply for the serviceability limit state design.

If you select *Required Additional Reinforcement*, the program assumes the effective $A_{s,req}$ distribution as the additional reinforcement to be applied.

If you select *Layed out Additional Reinforcement*, the program determines the additional reinforcement as the difference from the maximum statically required reinforcement and the defined basic reinforcement.

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

Equation 2.2

The additional reinforcement is dimensioned as shown in the graphical representation below.

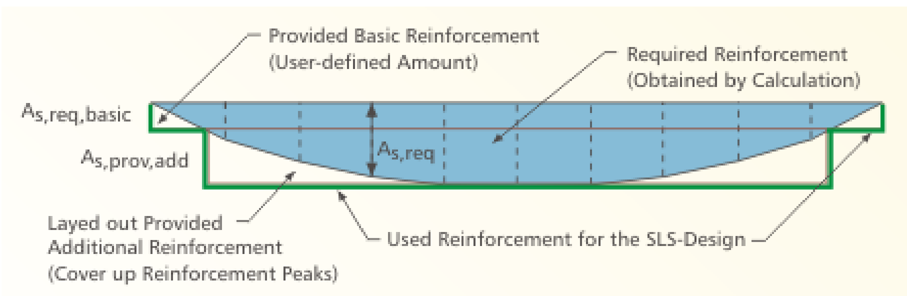


Figure 2.30: Dimensioning of additional reinforcement

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

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

You can also enter a *User-defined Additional Reinforcement*. If you chose this setting, the two library buttons shown on the left will be enabled so that rebars and mesh reinforcements are available for selection.

The [Info] button opens a dialog box where all three options described above are visualized.

Longitudinal Reinforcement for Proof of Shear Resistance

In the last dialog section, three options are available to define the applied longitudinal reinforcement for the shear design without shear reinforcement.

- *Apply required longitudinal reinforcement*

  The design of shear force resistance is carried out with the transformed existing tension reinforcement in direction of the principal shear force.

- *Automatically increase required longitudinal reinforcement to avoid shear reinforcement*

  In case the required longitudinal reinforcement is not sufficient for the shear force resistance, the longitudinal reinforcement will be increased in the principal shear force direction until the shear design without shear reinforcement is fulfilled.

- *Apply the greater value resulting from either required or defined provided reinforcement*

  For the design of shear force resistance, the program uses either the statically required or the user-defined longitudinal reinforcement.

2.5.4 Design Method

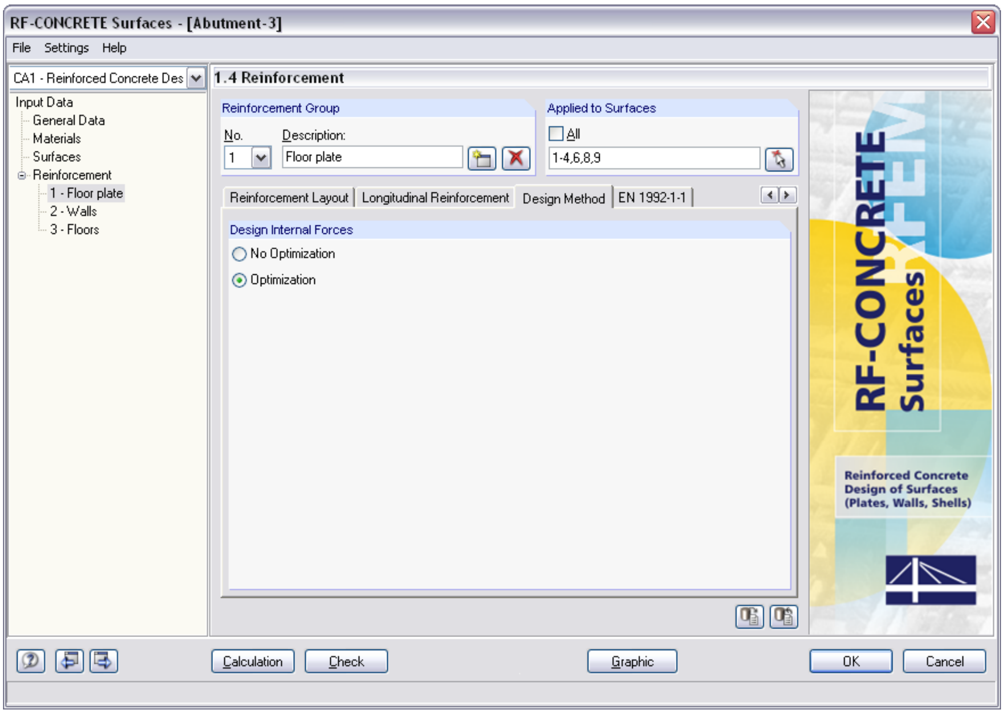


Figure 2.31: Table 1.4 *Reinforcement*, tab *Design Method*

When determining the required reinforcement, the principal internal forces are transformed into design forces in direction of the reinforcement and in a related concrete strut force. The sizes of these design forces depend on the presumed angle of the concrete compression strut that stiffens the reinforcement mesh.

In the load situations "tension-tension" and "tension-compression", the design force may become negative in one reinforcement direction for specific angles of compression struts. That means that compressive forces would occur for the tension reinforcement. Due to the optimization of design forces, the direction of the concrete compression 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 energetic minimum with the least required reinforcement. The optimization process including an example is described in detail in the German manual RF-BETON Flächen.

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 possibly not adequate for the load situation "compress-compression".

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

2.5.5 Standard

The final table tab is reserved for the standard which you have selected in table 1.1 *General Data*. In this tab, the standard-specific reinforcement data is defined. In the following, specifications for DIN 1045-1 are described.

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.

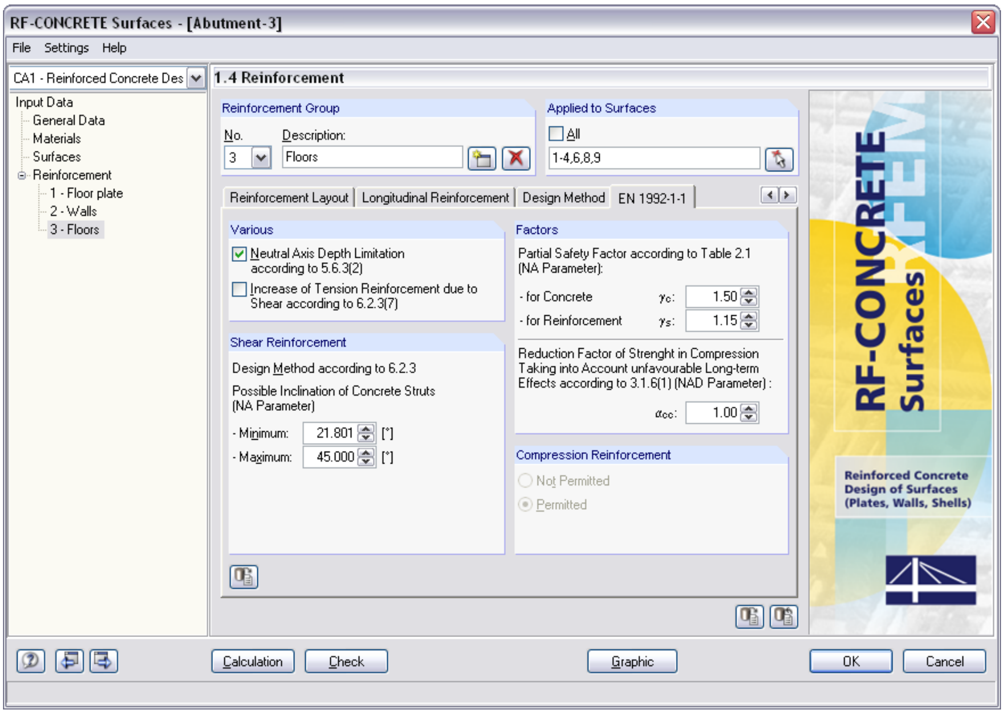


Figure 2.32: Table 1.4 *Reinforcement*, tab *DIN 1045-1*

Various

In this dialog section, the depth of the neutral axis can be limited according to DIN 1045-1, section 8.2 (3). In this case, the maximum ratio is $x_d / d = 0.45$ for concrete up to strength class C50/60 and $x_d / d = 0.35$ for concrete from strength class C55/67.

Factors

In the first two input fields, the *Partial Safety Factors* for concrete γ_c and for reinforcing steel γ_s are defined. The values according to DIN 1045-1, table 2, are preset.

The value that is recommended in DIN 1045-1, section 9.1.6 (2), is preset as reduction factor α that is used to consider long-term effects on the concrete compressive strength.

Shear Reinforcement

Use the two input fields in this dialog section to define the allowable range of the concrete struts' inclination. In case the entered angles are outside the allowed limits mentioned in the standards, an error message appears.

3. Calculation

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

The reinforced concrete design is carried out by using the internal forces that have been determined in RFEM. If the RFEM results are not yet available, the program starts the calculation of the internal forces automatically.

3.1 Check

Before you start the calculation, it is recommended to check the input data of the add-on module RF-CONCRETE Surfaces for correctness. The [Check] button is available in each input table of the module and is used to start the check.

The program checks if the data required for the design is complete and if the references of the data sets are alright. In case input errors are detected, you can access the corresponding table directly to make corrections.

If the plausibility check has been successful, the following message appears.

Figure 3.1: Plausibility check of input data

3.2 Start Calculation

You can start the calculation out of each of the four input tables by clicking the [Calculation] button.

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

It is also possible to start the design in the RFEM user interface. The add-on modules are listed in the dialog box *To Calculate* like load cases or load groups. To open the dialog box in RFEM,

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

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

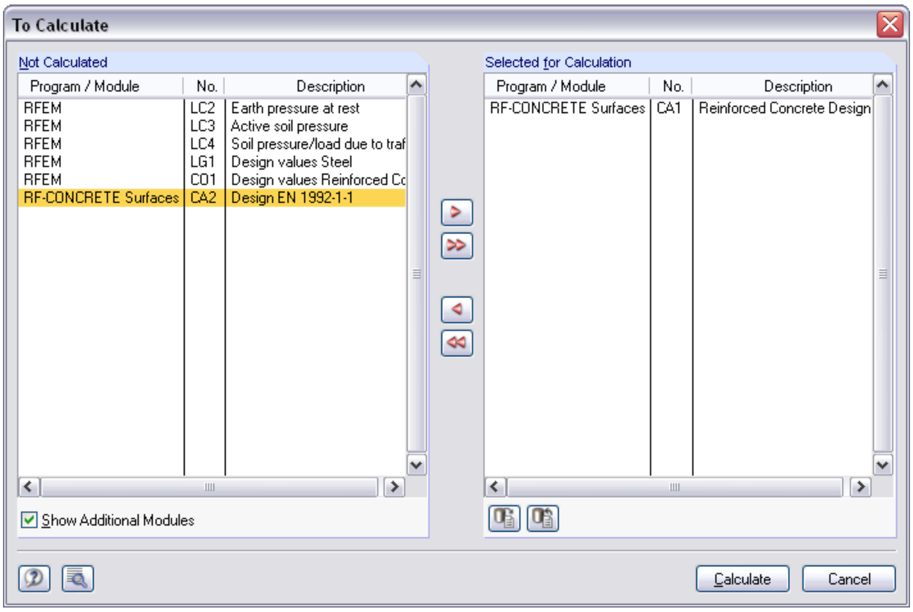


Figure 3.2: Dialog box *To Calculate*

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

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

You can also use the list in the RFEM toolbar to calculate the RF-CONCRETE Surfaces cases directly: Select the relevant design case and click the button [Results on/off].

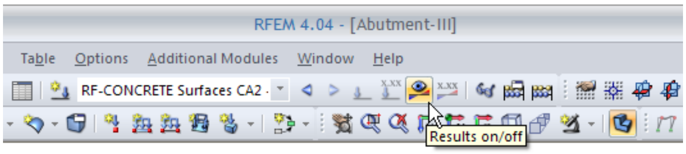


Figure 3.3: Direct calculation of an RF-CONCRETE Surfaces design case in RFEM

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

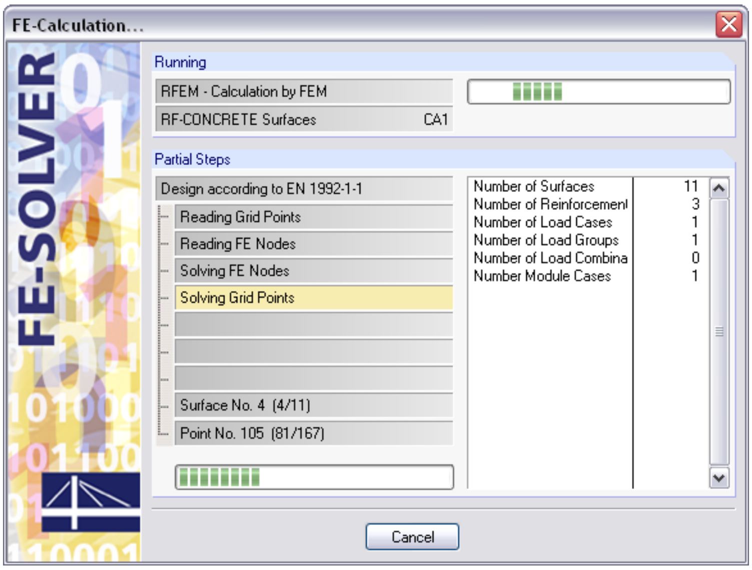


Figure 3.4: RF-CONCRETE Surfaces calculation

4. Results

Table 2.1 *Required Reinforcement Total* is displayed immediately after the calculation. Results tables 2.1 to 2.3 list the results of the ultimate limit state design including descriptions. Tables 3.1 to 3.3 are reserved for the results of serviceability limit state designs. Use the RF-CONCRETE Surfaces navigator to access the relevant results table. You can also use the two buttons shown on the left or the function keys [F2] and [F3] to select the previous or subsequent table.

Click [OK] to save the results and quit the add-on module RF-CONCRETE Surfaces.

In the following, the different results tables are described in sequence. Evaluating and checking results is described in detail in chapter 5 *Results Evaluation* on page 51.

By selecting one of the options below the results tables you can decide whether the results data is displayed *In FE-Nodes* or *In Grid Points* in the individual tables. The results of the FE points are determined directly by the analysis core. The grid point results are determined by interpolation of the FE point results.

The results tables 3.1 to 3.3 are subdivided into two parts (cf. Figure 4.5). The upper part represents a design overview in tables. The lower part displays the intermediate results of the current FE or grid point (that means the entry selected above) including all design relevant parameters. The different chapters of the tree structure in the lower table part can be opened by clicking [+] and closed by clicking [-].

4.1 Required Reinforcement Total

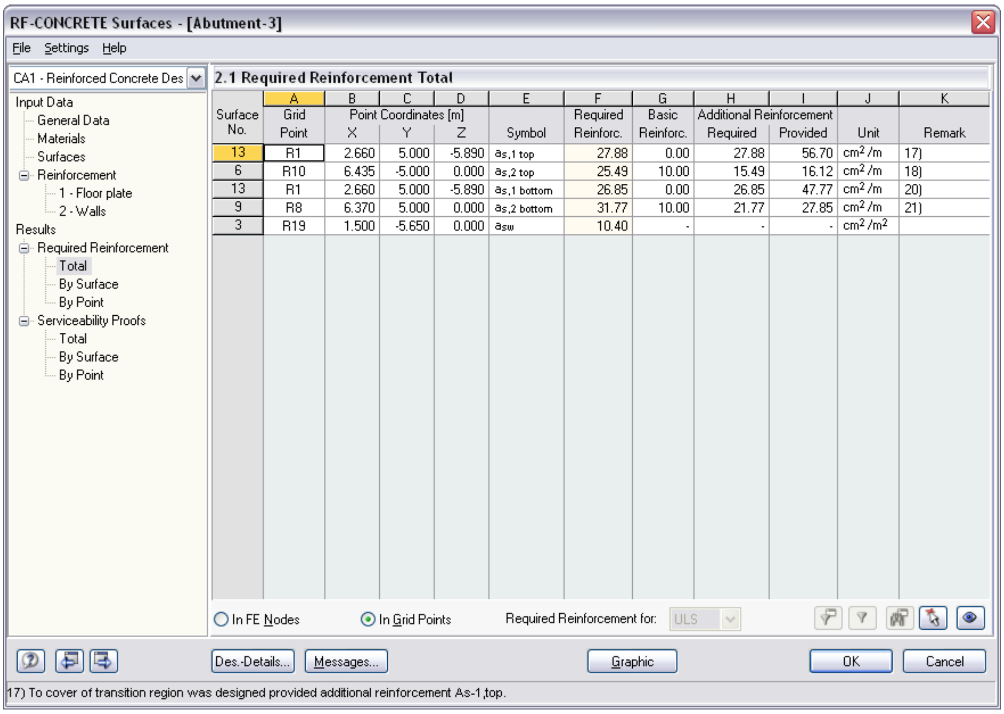


Figure 4.1: Table 2.1 *Required Reinforcement Total*

The table shows the maximum reinforcement results of all surfaces provided for the design. The output data is determined from the internal forces of the load cases, load groups and load combinations selected for the ultimate limit state design.

Point No.

This table column displays the numbers of the FE or grid points where the maximum required reinforcement for each position and direction has been determined. The reinforcement type is displayed in column E *Symbol*.

The FE mesh points are generated automatically. The grid points represent a property of the surface that can be set in RFEM. For each surface you can create user-defined results grids. For detailed information on grid points, see the RFEM manual, chapter 9.9, page 277.

Point-Coordinates

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

Symbol

Column E displays the reinforcement type. For the four (or six) longitudinal reinforcements, the module shows respectively direction (*1*, *2* and *3*, where applicable) and surface side (*top* and *bottom*).

The reinforcement directions are defined in the *Reinforcement Layout* tab of table 1.4 *Reinforcement* (cf. chapter 2.5.2, page 28).

Top and *Bottom* layers

The bottom reinforcement layer is defined on the surface side in direction of the positive local surface axis z. Accordingly, the top reinforcement layer is defined in direction of the negative z-axis.

The shear reinforcement is indicated as a_{sw}.

Required Reinforcement

This column displays the reinforcement areas that are required for the ultimate limit state design.

Basic Reinforcement

This column contains the user-defined basic reinforcement that has been specified in the *Longitudinal Reinforcement* tab in table 1.4 *Reinforcement* (cf. chapter 2.5.3, page 30).

Additional Reinforcement

When you design the ultimate limit state exclusively, column H displays the difference between required reinforcement (column F) and existing basic reinforcement (column G).

When you design the serviceability additionally, you see in table column I the reinforcement areas that are required by the specifications in the *Longitudinal Reinforcement* tab of table 1.4 *Reinforcement* (cf. chapter 2.5.3, page 30) to fulfill the serviceability limit state designs.

Remark

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

Messages...

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

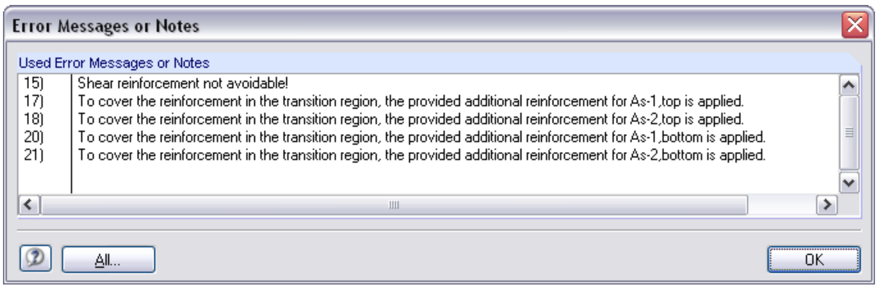


Figure 4.2: Dialog box *Error Messages or Notes*

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

Button	Description	Function
	Filter Points	FE and grid points can be filtered by surfaces and according to certain criteria. → chapter 5.3, page 55
	Only Designable Results	Table rows with non-designable elements are hidden.
	Find Point	You can search for table rows with results of single FE or grid points by specifying surface or number. → chapter 5.3, page 57
	Pick Surface	You can click a surface in the RFEM window to display its results in the table.
	View mode	You can jump to the RFEM work window to set another view.

Table 4.1: Buttons of results tables 2.1 to 2.3

Des.-Details...

When you display the design results *In Grid Points* (see Figure 4.1), the button [Des.-Details] is enabled. Use this button to check the design details of each grid point. The display refers to the selected grid point, that means the table row where the pointer is currently placed.

The design details are displayed only for the results of load cases and load groups. The dialog box *Design Details* is described in chapter 5.1 on page 51.

4.2 Required Reinforcement By Surface

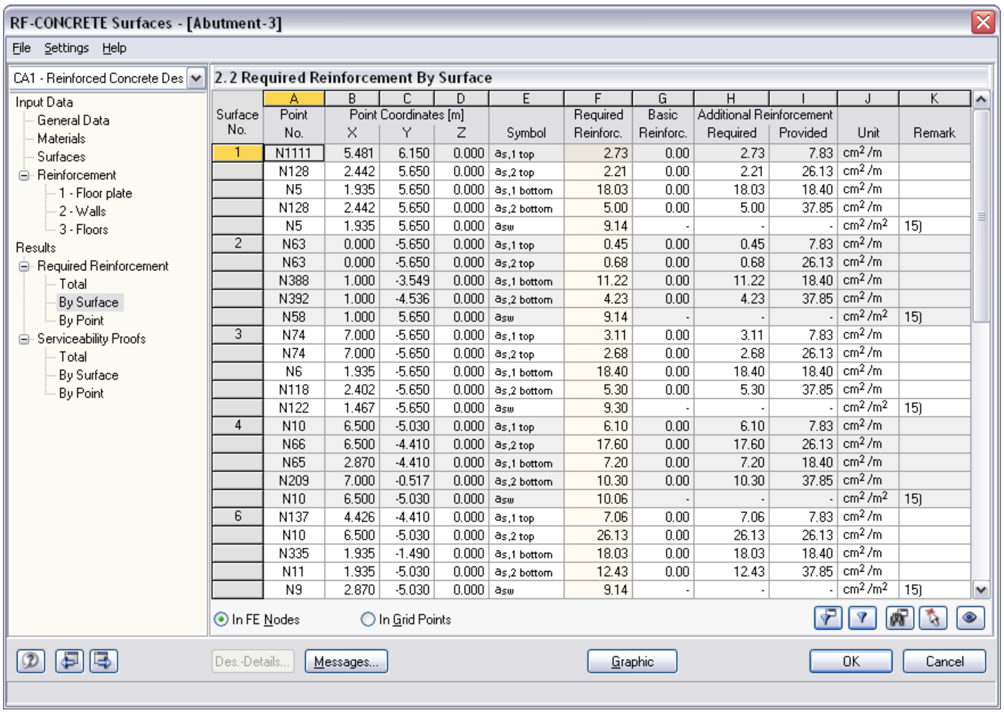


Figure 4.3: Table 2.2 *Required Reinforcement By Surface*

This table shows the maximum reinforcement areas that are required for each of the designed surfaces. Details on the table columns can be found in the previous chapter 4.1.

4.3 Required Reinforcement By Point

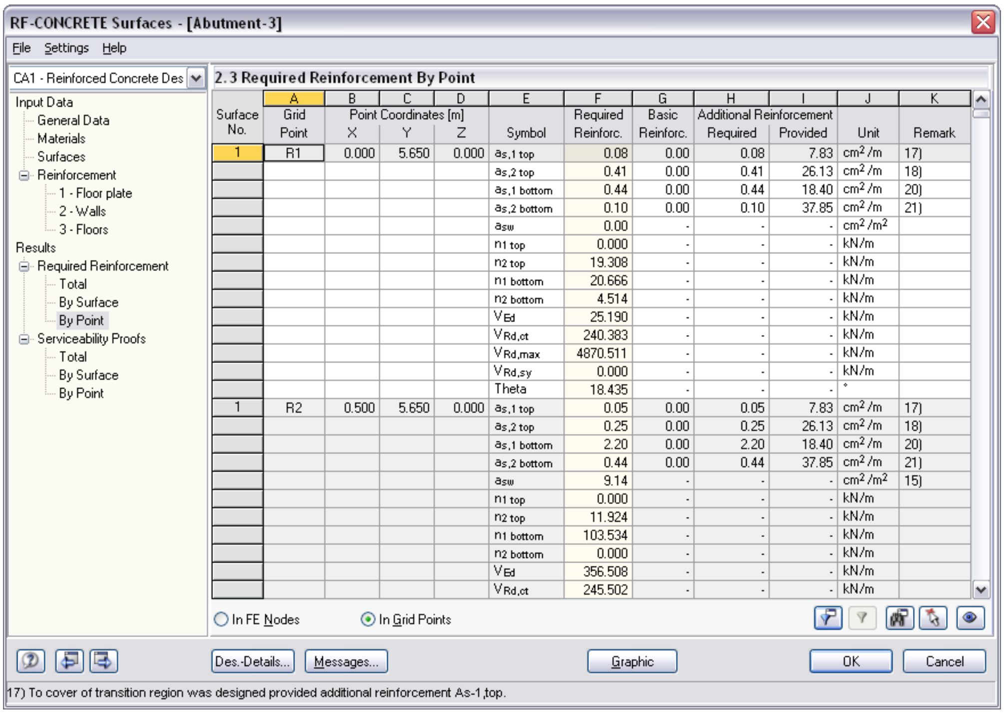


Figure 4.4: Table 2.3 *Required Reinforcement By Point*

This table lists the maximum reinforcement areas for all FE or grid points of each surface. The different columns are described in detail in chapter 4.1 on page 39.

In addition to the table rows showing the various reinforcement types, the table displays significant values that are important for the determination of the reinforcement. For the standard DIN 1045-1, these are the following:

Symbol	Meaning
$n_{1\ top}$	Axial force or membrane force for design of reinforcement in first reinforcement direction on top side of the surface
$n_{2\ top}$	Axial force or membrane force for design of reinforcement in second reinforcement direction on top side of the surface
$n_{1\ bottom}$	Same as $n_{1\ top}$, but on bottom side of the surface
$n_{2\ bottom}$	Same as $n_{2\ top}$, but on bottom side of the surface
$m_{1\ top}$ / $m_{2\ top}$	Only for structure type *Plate XY*: Moment for design of reinforcement in first or second reinforcement direction on top side of the surface
$m_{1\ bottom}$ / $m_{2\ bottom}$	Same as $m_{1\ top}$ / $m_{2\ top}$, but on bottom side of the surface
V_{Ed}	Design value of applied shear force
$V_{Rd,ct}$	Design shear resistance without shear reinforcement
$V_{Rd,max}$	Design shear resistance of concrete compression strut
$V_{Rd,sy}$	Design shear resistance of shear reinforcement
Theta	Inclination angle of concrete compression strut θ

Table 4.2: Output values in table 2.3 for DIN 1045-1

The search function that you can use by clicking the button shown on the left helps you to find specific FE or grid points quickly (cf. Figure 5.8, page 57).

4.4 Serviceability Total

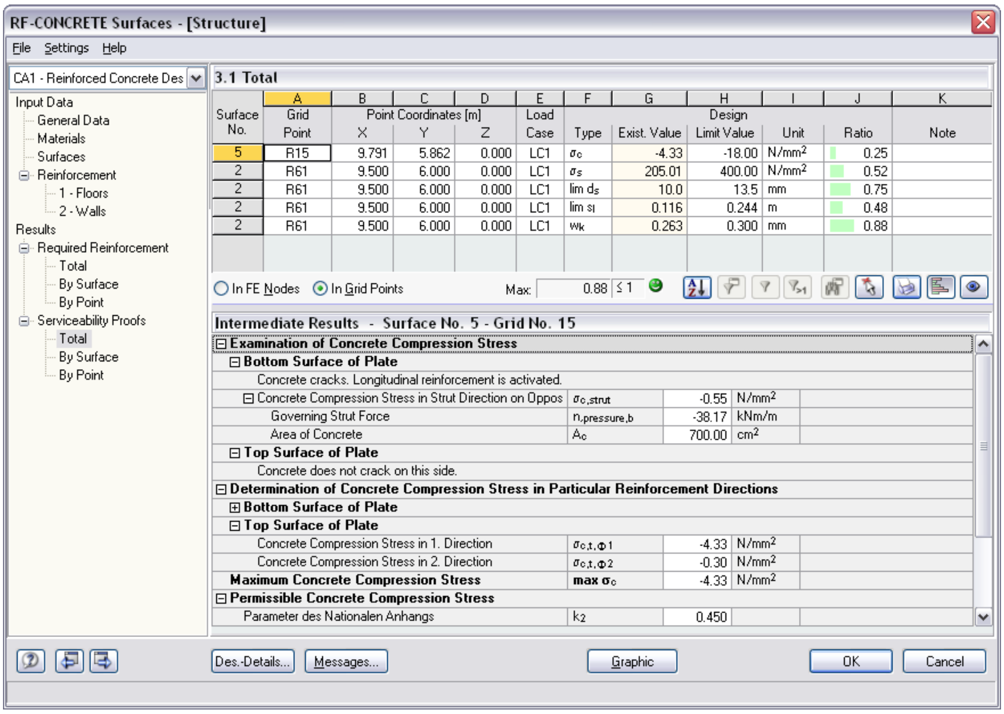


Figure 4.5: Table 3.1 *Total*

This table shows the governing results of the various serviceability limit state designs of all surfaces provided for the design. It is subdivided into two parts: The upper part represents a design overview in tables. The lower part displays the *Intermediate Results* of the FE or grid point selected above.

Figure 4.5 shows the results table of an analytical serviceability limit state design. The design method is defined in the *Serviceability Limit State* tab of table 1.1 *General Data* (cf. Figure 2.6, page 11). Chapter 4.7 on page 47 describes the results tables that appear when a non-linear SLS analysis was carried out.

Point No.

This table column displays the numbers of the FE or grid points for which the program has determined the maximum ratios for each required design. The design type is displayed in column F *Type*.

The FE mesh points are generated automatically. The grid points represent a property of the surface that can be set in RFEM. For each surface you can create user-defined results grids. For detailed information on grid points, see the RFEM manual, chapter 9.9, page 277.

Point-Coordinates

These three columns contain the coordinates of the governing FE mesh or grid points.

Load Case

Table column E displays the load case, load group or load combination whose internal forces produce the maximum ratio for the respective serviceability limit state design.

Type

Column F shows the type of the serviceability limit state design. If the analytical method was used, the column lists up to six design types. These types are described in detail including example in the German manual RF-BETON Flächen.

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

The individual design types have the following meanings:

Type	Design SLS
σ_c	Limitation of concrete compressive stress according to specifications in table 1.3 *Surfaces* (see Figure 2.15, page 22)
σ_s	Limitation of reinforcing steel stress according to specifications in table 1.3 *Surfaces* (see Figure 2.15, page 22)
$a_{s,min}$	Minimum reinforcement for the limitation of crack widths according to specifications in table 1.3 *Surfaces* (see Figure 2.13, page 19 and Figure 2.14, page 21)
lim d_s	Limitation of rebar diameter according to specifications in table 1.4 *Reinforcement* (see Figure 2.27, page 30)
lim s_l	Limitation of rebar spacing according to specifications in table 1.4 *Reinforcement* (see Figure 2.27, page 30)
w_k	Limitation of crack widths according to specifications in table 1.3 *Surfaces* (see Figure 2.13, page 19)

Table 4.3: Serviceability limit state designs according to analytical method

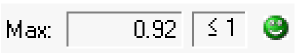
This Existing Value

This table column displays the total extreme values of all surfaces that are decisive for the respective serviceability limit state designs.

Limit Value

The limit values result from the standard specifications and the current load situation. The determination of the limit values is described in detail in the German manual RF-BETON Flächen.

Ratio



This column represents the design quotients from the existing (column G) and the limit value (column H). In this way, you can find out immediately whether the design criterion of *1* has been fulfilled or exceeded.

In the background of this table column, colored scales are displayed whose lengths reflect the ratio of the area. A green scale means that the design is fulfilled. A red scale indicates an exceeding. To display or hide the scales in the table, use the button shown on left.

Note

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

To display all messages of the currently selected design case, use the [Messages] button shown on the left. A dialog box with relevant information appears (cf. Figure 4.2, page 39).

The buttons in the upper part of the table are reserved for the following functions:

Button	Description	Function
	Sorting of Results	The results can be arranged according to maximum ratios (column J) or values (column G). ➙ chapter 5.3, page 56
	Filter Points	FE and grid points can be filtered by surfaces and according to certain criteria. ➙ chapter 5.3, page 55
	Only Designable Results	Table rows with non-designable elements are hidden.
	Show Ratios with Ratio > 1	The table shows only rows with ratios > 1.00.
	Find Point	You can search for table rows with results of single FE or grid points by specifying surface or number. ➙ chapter 5.3, page 57
	Pick Surface	You can click a surface in the RFEM window to display its results in the table.
	Print results	The intermediate results of the current FE or grid point are printed in the printout report.
	Show Color Bars	The colored reference scales can be displayed or hidden.
	View mode	You can jump to the RFEM work window to set another view.

Table 4.4: Buttons of results tables 3.1 to 3.3

When you display the design results *In Grid Points* (see Figure 4.1), the button [Des.-Details] is enabled. Use this button to check the design details of each grid point. The display refers to the selected grid point, that means the table row where the pointer is currently placed.

4.5 Serviceability By Surface

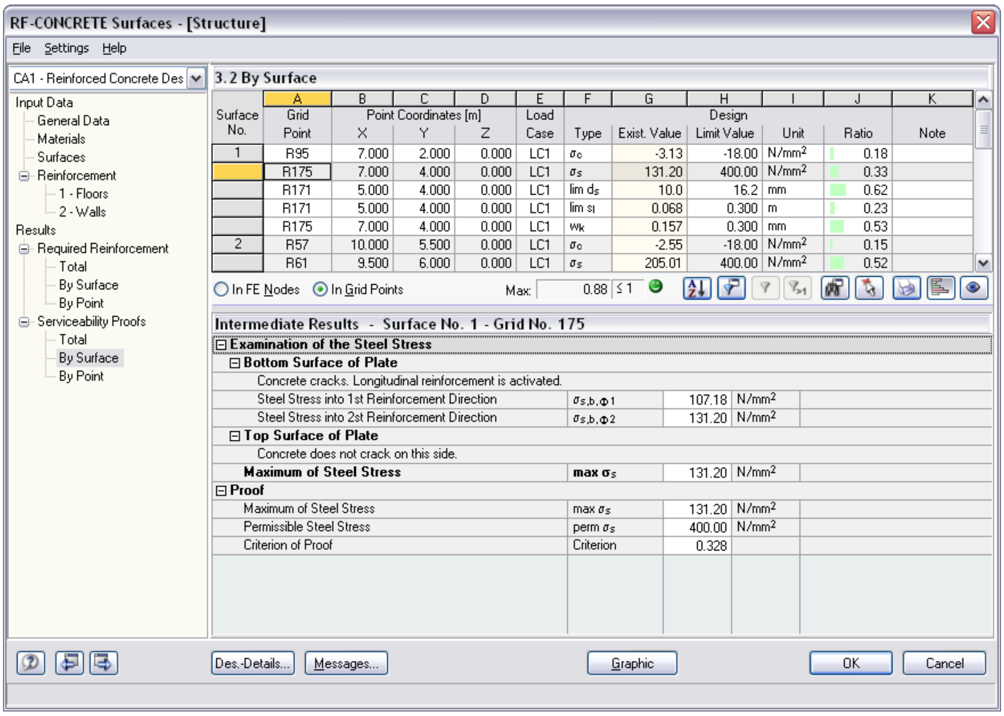


Figure 4.6: Table 3.2 *By Surface*

This results table displays the maximum ratios for the various serviceability limit state designs that are decisive for each of the designed surfaces.

Details on the table columns can be found in the previous chapter 4.4.

4.6 Serviceability By Point

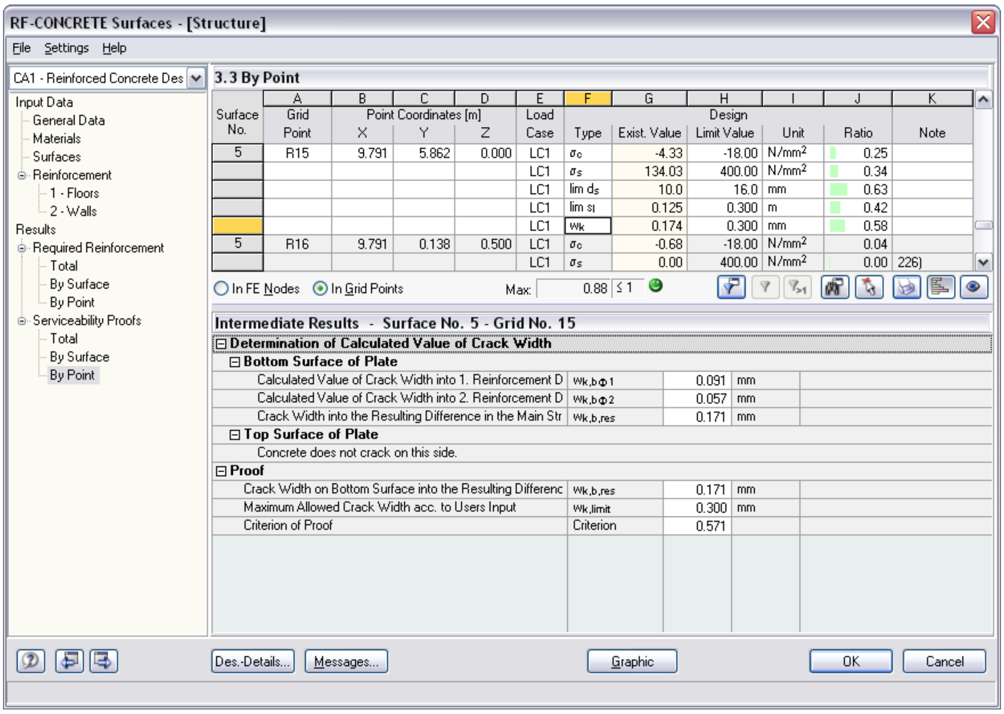


Figure 4.7: Table 3.3 *By Point*

This table lists the maximum ratios for all FE or grid points of a surface. The different columns are described in detail in chapter 4.4 on page 43.

In addition to the table rows showing the various serviceability limit state designs, the table displays significant values that are considered in the analyses.

The search function that you can use by clicking the button shown on the left helps you to find specific FE or grid points quickly (cf. Figure 5.8, page 57).

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

4.7 Nonlinear Calculation Total

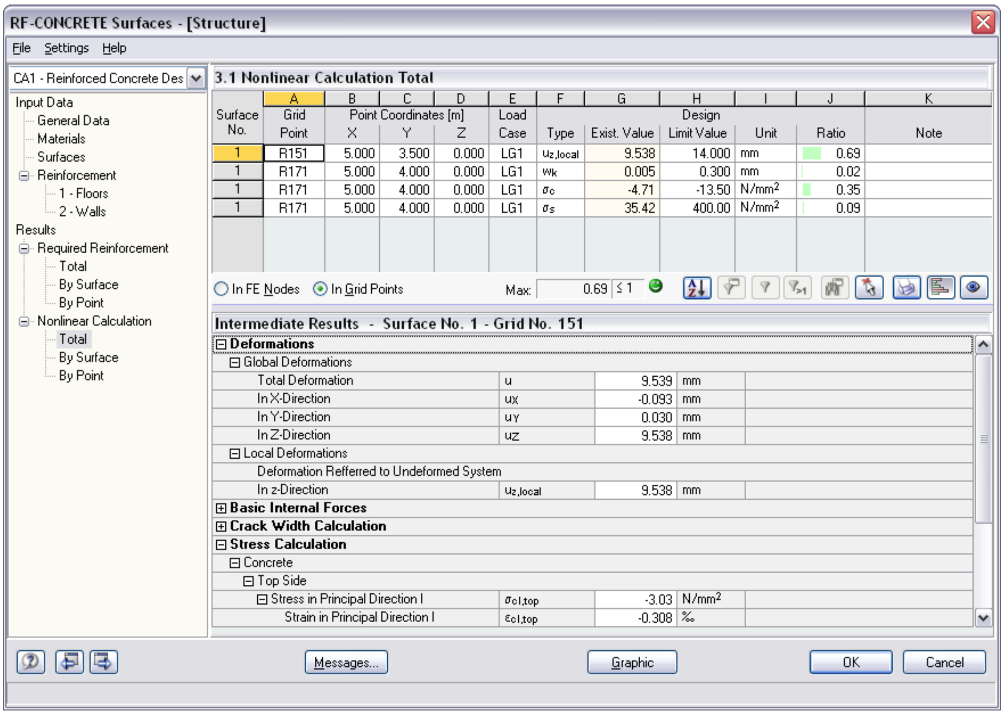


Figure 4.8: Table 3.1 *Nonlinear Calculation Total*

This table shows the governing results of the various serviceability limit state designs of all surfaces provided for the design. It is subdivided into two parts: The upper part represents a design overview in tables. The lower part displays the *Intermediate Results* of the FE or grid point selected above.

Figure 4.8 shows the results table of a non-linear serviceability limit state design. The design method is defined in the *Serviceability Limit State* tab of table 1.1 *General Data* (cf. Figure 2.6, page 11). Chapter 4.4 on page 42 describes the results tables that appear when an analytical SLS analysis was carried out.

Grid Point

This table column displays the numbers of the FE or grid points for which the program has determined the maximum ratios for each required design. The design type is displayed in column F *Type*.

The FE mesh points are generated automatically. The grid points represent a property of the surface that can be set in RFEM. For each surface you can create user-defined results grids. For detailed information on grid points, see the RFEM manual, chapter 9.9, page 277.

Point Coordinates

These three columns contain the coordinates of the governing FE mesh or grid points.

Load Case

Table column E displays the number of the load case or load group whose internal forces produce the maximum ratio for the respective serviceability limit state design.

Type

Column F shows the type of the serviceability limit state design. If the non-linear method was used, the column lists up to four design types. The types are described in detail in the German manual RF-BETON Flächen.

The individual design types have the following meanings:

Type	Design SLS
$u_{z,local}$	Deformation in cracking sections according to specifications in table 1.3 *Surfaces* (see Figure 2.18, page 24)
w_k	Limitation of crack widths according to specifications in table 1.3 *Surfaces* (see Figure 2.13, page 19)
σ_c	Limitation of concrete compressive stress according to specifications in table 1.3 *Surfaces* (see Figure 2.15, page 22)
σ_s	Limitation of reinforcing steel stress according to specifications in table 1.3 *Surfaces* (see Figure 2.15, page 22)

Table 4.5: Serviceability limit state designs according to non-linear method

Existing Value

This table column displays the total extreme values of all surfaces that are decisive for the respective serviceability limit state designs. The values of the deformations, crack widths and stresses represent the results in cracked sections (state II).

The crack widths w_k listed among the intermediate results in the lower part of the table refer to the reinforcement directions 1 and 2. For example, the value for $w_{k1,top}$ represents the crack width of the first reinforcement direction on the top side of the surface (that means the crack runs perpendicular to the first reinforcement direction).

Limit Value

The limit values result from the standard specifications and the current load situation. The determination of limit values is described in detail in chapter 2.4.2 on page 24 as well as in the German manual RF-BETON Flächen.

Ratio

This column represents the design quotients from the existing value (column G) and the limit value (column H). In this way, you can find out immediately whether the design criterion of *1* was fulfilled or exceeded.

In the background of this table column, colored scales are displayed whose lengths reflect the various design ratios. A green scale means that the design is fulfilled. A red scale indicates an exceeding. To display or hide the scales in the table, use the button shown on left.

Note

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

To display all messages of the currently selected design case, use the [Messages] button shown on the left. A dialog box with relevant information appears (cf. Figure 4.2, page 39).

The buttons in the upper part of the table are described in Table 4.4 on page 44.

4.8 Nonlinear Calculation By Surface

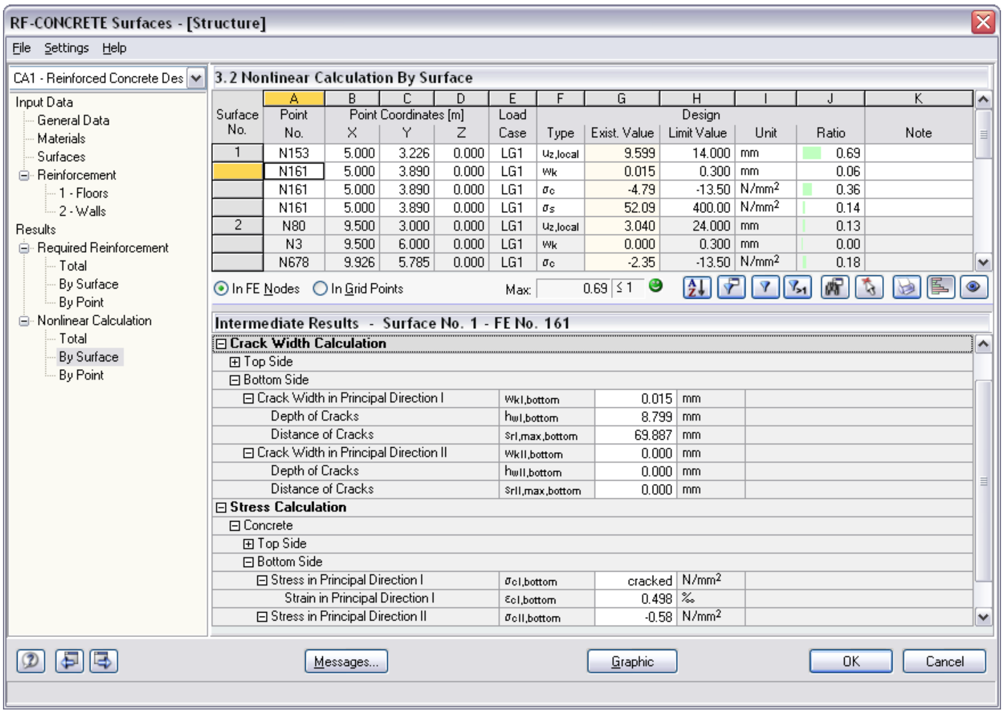


Figure 4.9: Table 3.2 *Nonlinear Calculation By Surface*

This results table displays the maximum ratios for the various serviceability limit state designs that are decisive for each of the designed surfaces. Details on the table columns can be found in the previous chapter 4.7.

4.9 Nonlinear Calculation By Point

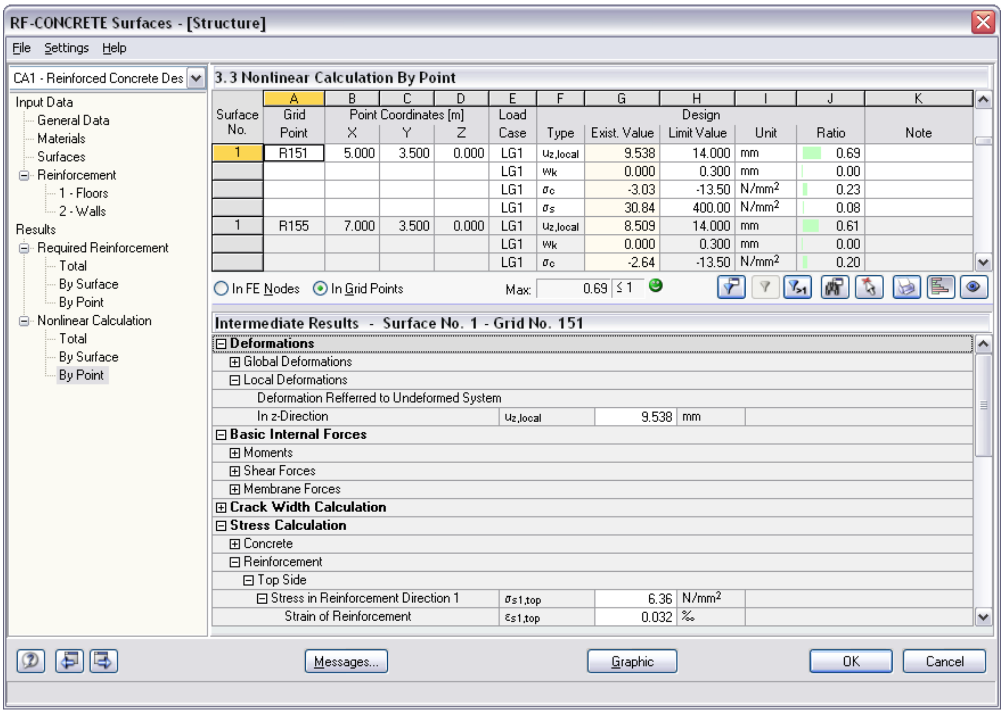


Figure 4.10: Table 3.3 *Nonlinear Calculation By Point*

This table lists the maximum ratios for each single FE or grid point of a surface. The different columns are described in detail in chapter 4.7 on page 48.

The search function that you can use by clicking the button shown on the left helps you to find specific FE or grid points quickly (cf. Figure 5.8, page 57).

5. Results Evaluation

When the design is completed, several options are available for results evaluation. The design details, which can be displayed in a separate window, help you to evaluate the data. Furthermore, it is possible to evaluate the results graphically in the RFEM work window.

5.1 Design Details

When you select to display the results *In Grid Points* (see Figure 4.1) in the results tables 2.1 to 2.3 for the ultimate limit state designs, the button [Des.-Details] is enabled. You can use this button to look at the design details of the selected grid point, that means the table row where the pointer is currently placed.

The details of the ultimate limit state design are available only if the governing internal forces result from one single load case or one single load group. The design of several load cases, load groups or of a load combination does not allow for an unambiguous assignment.

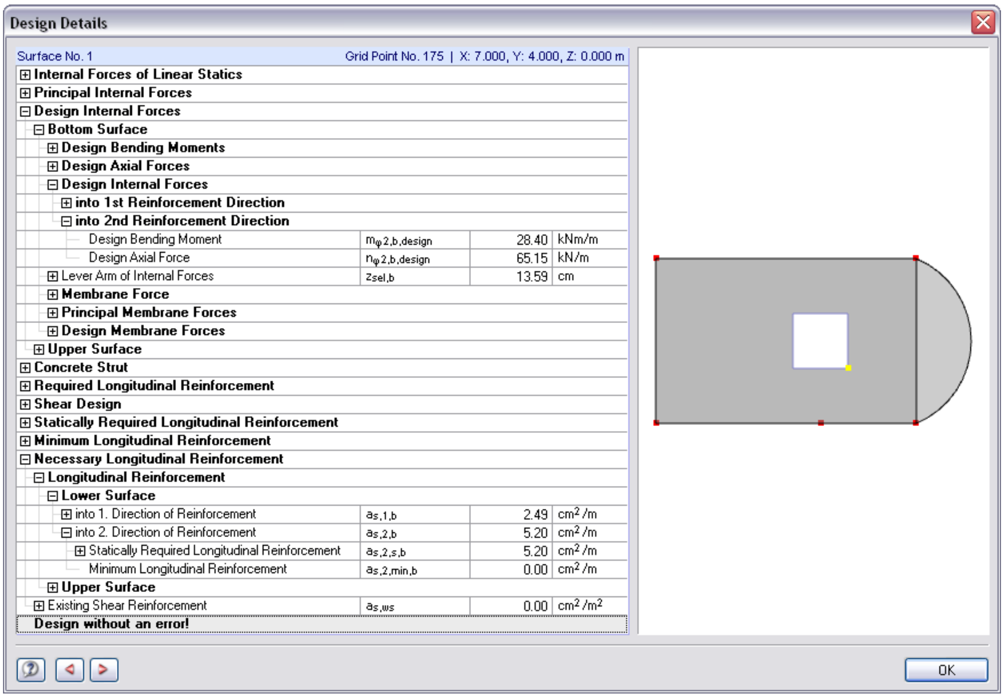


Figure 5.1: Dialog box *Design Details* for ultimate limit state design

All relevant data is listed in a tree structure on the left. To open a chapter of the tree, click [+]. To close a chapter, click [-]. On the right, the dialog box shows the location of the grid point in the structure graphically.

The following design details are displayed:

- Internal forces of linear statics
- Principal internal forces
- Design internal forces
- Concrete strut
- Required longitudinal reinforcement
- Shear design
- Statically required longitudinal reinforcement
- Minimum longitudinal reinforcement
- Necessary longitudinal reinforcement

For the serviceability limit state designs, the program shows significant intermediate results directly in the tables 3.1 to 3.3 that are subdivided into two parts (cf. Figure 4.5, page 42). In addition, you can check all design details for each grid and FE point for the analytical SLS design. In contrast to the results of the ultimate limit state designs, the program enables now access to the design details of several load cases as well as of load combinations provided for the design.

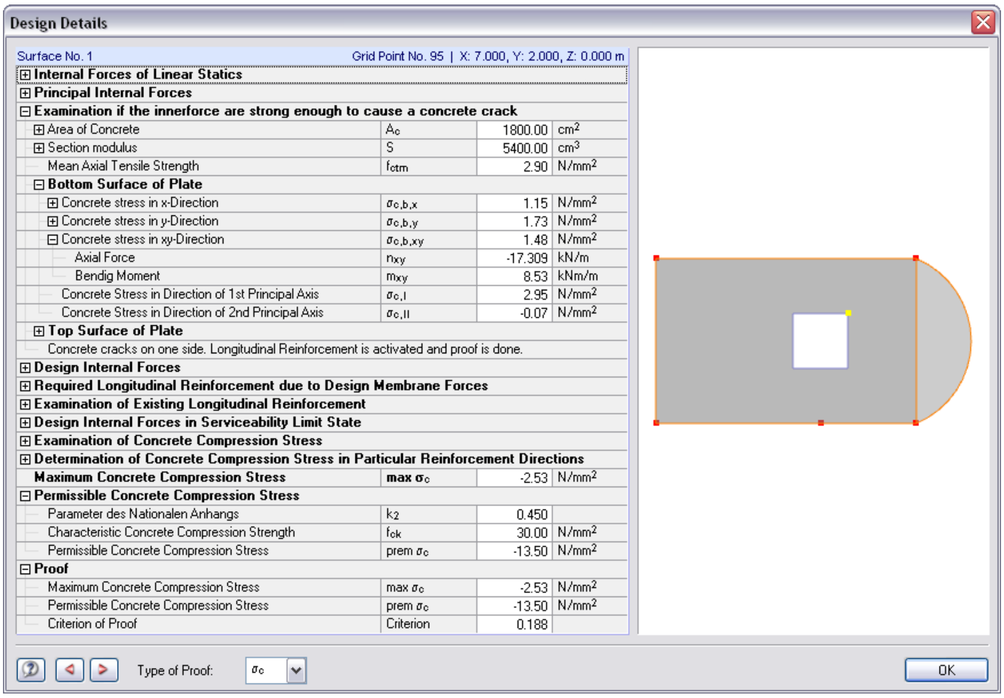
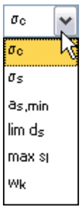


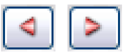
Figure 5.2: Dialog box *Design Details* for serviceability limit state design



The tree structure displays the relevant result rows for each design type. Use the list *Type of Proof* at the bottom of the dialog box shown above to select the design details that you want to display.

Design method	Type of design	
analytical	σ_c σ_s $a_{s,min}$ lim d_s lim s_l w_k	see Table 4.3, page 43

Table 5.1: *Types of Proof* for serviceability limit state designs



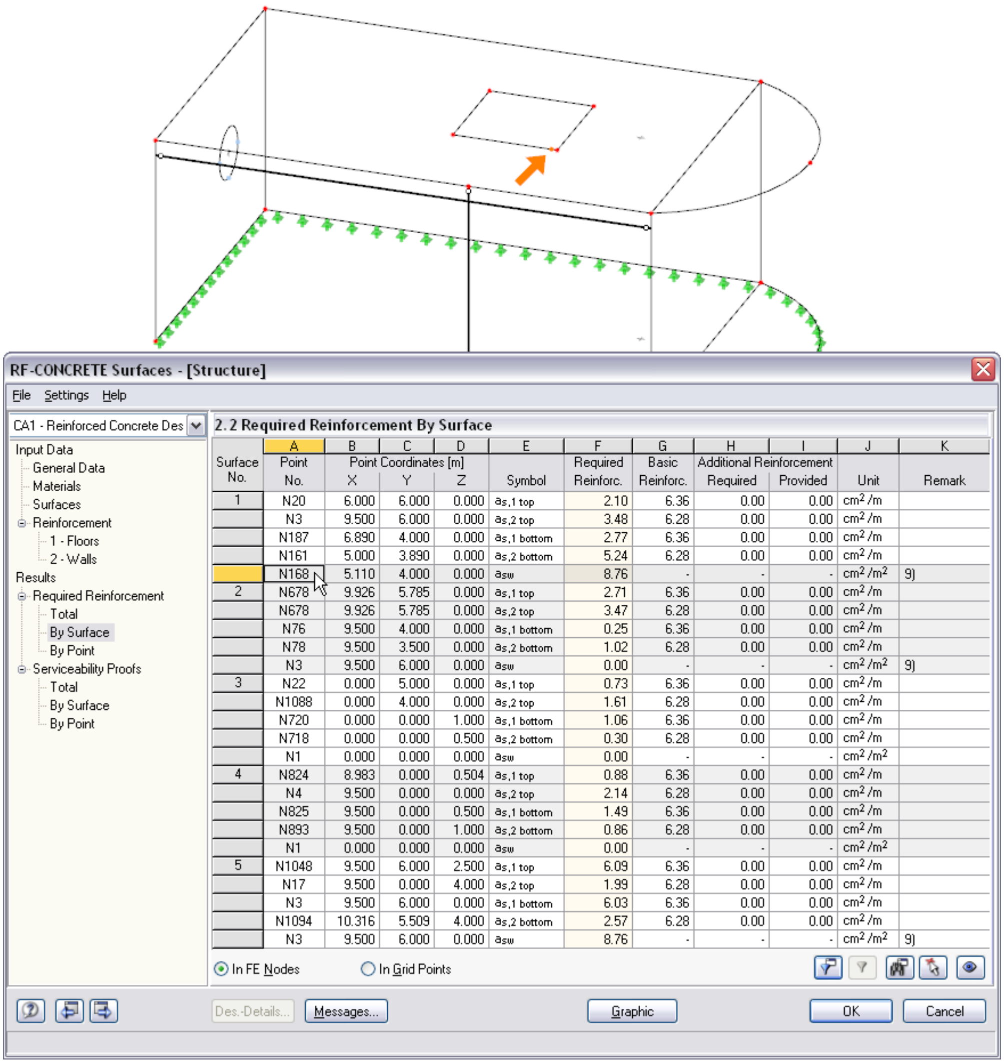
Use the button [◀] to jump to the previous FE or grid point. Click the button [▶] to select the subsequent point.

5.2 Results in the RFEM Model

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

RFEM background graphic

Use the RFEM graphic in the background to check the location of a particular FE or grid point in the model. An arrow in the RFEM background graphic indicates the point that you have selected in the results table.



Surface No.	A: Point No.	B: Point Coordinates [m] X	C: Y	D: Z	E: Symbol	F: Required Reinforc.	G: Basic Reinforc.	H: Additional Reinforcement Required	I: Additional Reinforcement Provided	J: Unit	K: Remark
1	N20	6.000	6.000	0.000	$a_{s,1\ top}$	2.10	6.36	0.00	0.00	cm^2/m	
	N3	9.500	6.000	0.000	$a_{s,2\ top}$	3.48	6.28	0.00	0.00	cm^2/m	
	N187	6.890	4.000	0.000	$a_{s,1\ bottom}$	2.77	6.36	0.00	0.00	cm^2/m	
	N161	5.000	3.890	0.000	$a_{s,2\ bottom}$	5.24	6.28	0.00	0.00	cm^2/m	
	N168	5.110	4.000	0.000	a_{sw}	8.76	-	-	-	cm^2/m^2	9)
2	N678	9.926	5.785	0.000	$a_{s,1\ top}$	2.71	6.36	0.00	0.00	cm^2/m	
	N678	9.926	5.785	0.000	$a_{s,2\ top}$	3.47	6.28	0.00	0.00	cm^2/m	
	N76	9.500	4.000	0.000	$a_{s,1\ bottom}$	0.25	6.36	0.00	0.00	cm^2/m	
	N78	9.500	3.500	0.000	$a_{s,2\ bottom}$	1.02	6.28	0.00	0.00	cm^2/m	
	N3	9.500	6.000	0.000	a_{sw}	0.00	-	-	-	cm^2/m^2	9)
3	N22	0.000	5.000	0.000	$a_{s,1\ top}$	0.73	6.36	0.00	0.00	cm^2/m	
	N1088	0.000	4.000	0.000	$a_{s,2\ top}$	1.61	6.28	0.00	0.00	cm^2/m	
	N720	0.000	0.000	1.000	$a_{s,1\ bottom}$	1.06	6.36	0.00	0.00	cm^2/m	
	N718	0.000	0.000	0.500	$a_{s,2\ bottom}$	0.30	6.28	0.00	0.00	cm^2/m	
	N1	0.000	0.000	0.000	a_{sw}	0.00	-	-	-	cm^2/m^2	
4	N824	8.983	0.000	0.504	$a_{s,1\ top}$	0.88	6.36	0.00	0.00	cm^2/m	
	N4	9.500	0.000	0.000	$a_{s,2\ top}$	2.14	6.28	0.00	0.00	cm^2/m	
	N825	9.500	0.000	0.500	$a_{s,1\ bottom}$	1.49	6.36	0.00	0.00	cm^2/m	
	N893	9.500	0.000	1.000	$a_{s,2\ bottom}$	0.86	6.28	0.00	0.00	cm^2/m	
	N1	0.000	0.000	0.000	a_{sw}	0.00	-	-	-	cm^2/m^2	
5	N1048	9.500	6.000	2.500	$a_{s,1\ top}$	6.09	6.36	0.00	0.00	cm^2/m	
	N17	9.500	0.000	4.000	$a_{s,2\ top}$	1.99	6.28	0.00	0.00	cm^2/m	
	N3	9.500	6.000	0.000	$a_{s,1\ bottom}$	6.03	6.36	0.00	0.00	cm^2/m	
	N1094	10.316	5.509	4.000	$a_{s,2\ bottom}$	2.57	6.28	0.00	0.00	cm^2/m	
	N3	9.500	6.000	0.000	a_{sw}	8.76	-	-	-	cm^2/m^2	9)

Figure 5.3: Localization of the current FE point in the RFEM model

If you move the RF-CONCRETE Surfaces window to another place in the display and you still cannot see the graphic clearly, use the button [Jump to Graphics] to activate the *view mode*: The RF-CONCRETE Surfaces window will be hidden so that you can modify the display in the RFEM work window appropriately. The view mode provides only the functions of the *View* menu, for example zooming, moving or rotating the display.

RFEM work window

It is also possible to visualize the design results and ratios directly in the structural model. Use the [Graphic] button to quit the ad-on module RF-CONCRETE Surfaces and access the RFEM work window. The window shows all design results and design criteria graphically.

Reinforcement direction

The *Results* navigator in RFEM is adjusted to the results from the add-on module RF-CONCRETE Surfaces. It provides the results of the longitudinal reinforcements for each reinforcement direction and layer, the results of the shear reinforcement, the design internal forces as well as various ratios and detail results of the serviceability limit state designs.

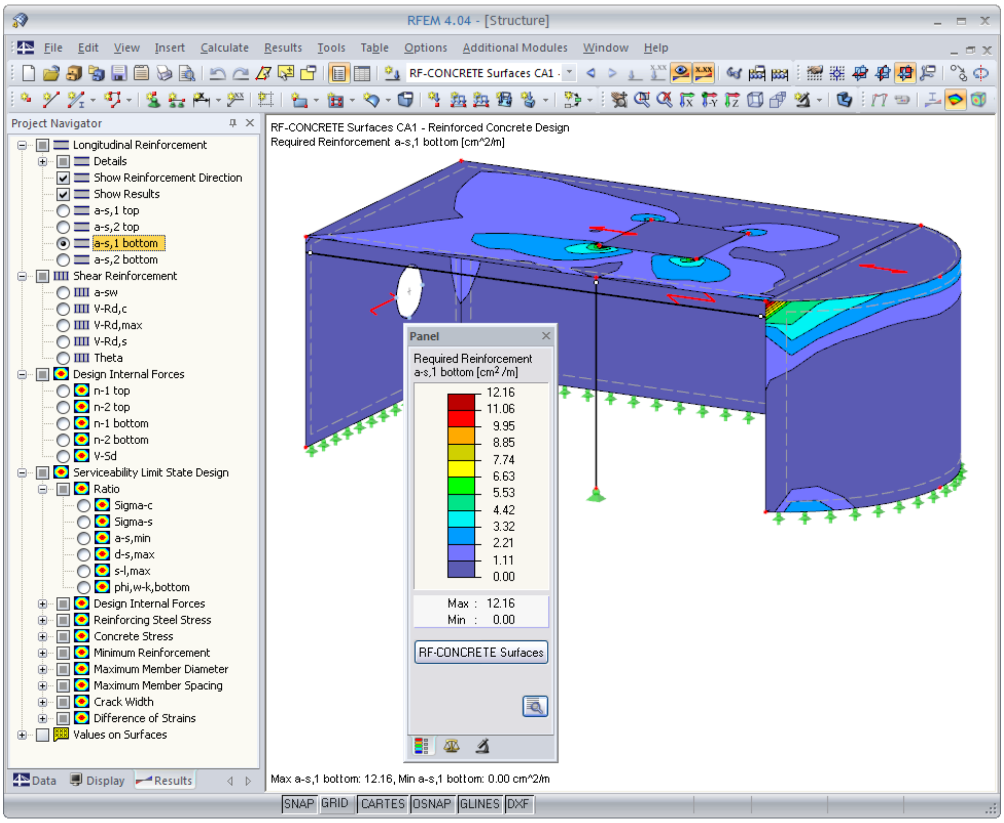
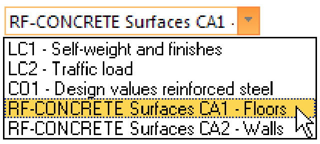


Figure 5.4: *Results* navigator of RF-CONCRETE Surfaces

To switch the display for the design results on and off, use the toolbar button [Results on/off] shown on the left. To display the result values in the graphics, use the toolbar button [Show Result Values] to the right.

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



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

The panel providing the usual control options helps you to evaluate the design results. The panel functions are described in detail in the RFEM manual, chapter 4.4.6, page 77. Use the *Factors* tab to scale the surface diagrams of reinforcements, internal forces or ratios. In the *Filter* tab, it is possible to select the results of particular surfaces specifically.

For the display and evaluation of numerical design results, you can use all options that are available in RFEM. The corresponding functions are described in detail in the RFEM manual, chapter 10.4, page 306. The following figure, for example, shows the bottom reinforcement that must be inserted additionally to the specified basic reinforcement as a *Group*. The values are applied in direction of the reinforcement.

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

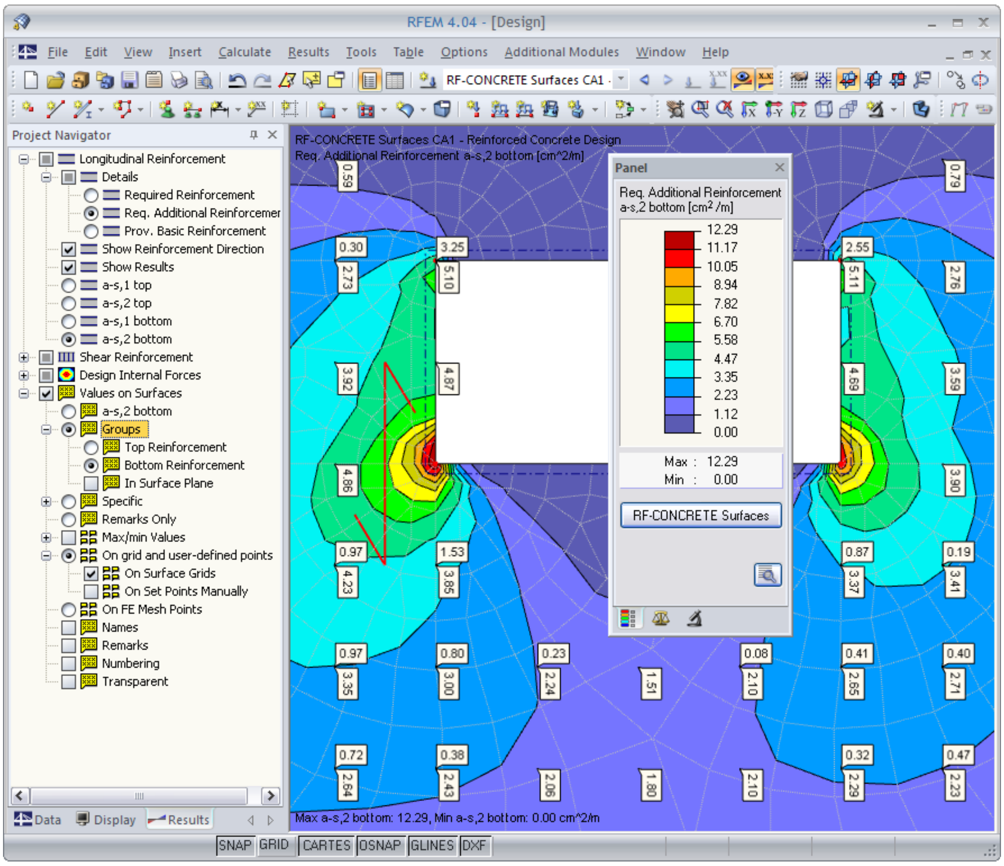


Figure 5.5: Group *Bottom Reinforcement*, representation of additional reinforcement

The graphics like RFEM graphics can be transferred to the printout report (see chapter 6.2, page 59).

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

5.3 Filter for Results

Due to their arrangement, the results tables of the add-on module RF-CONCRETE Surfaces already provide a selection of the total and surfaces maxima. In addition, results tables 2.3 and 3.3 displaying the design results point by point offer filter options for a specific selection.

Filter Points

In the results tables 2.2 and 2.3 as well as 3.2 and 3.3, the button shown on the left is available. Use this button to open the dialog box *Filter Points*.

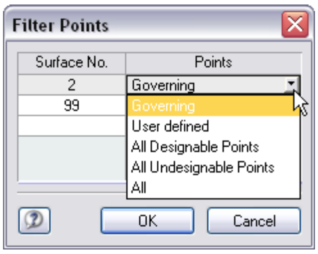
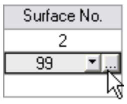
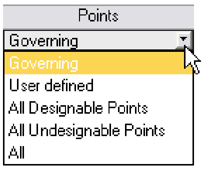


Figure 5.6: Dialog box *Filter Points*



In the column *Surface No.*, enter the number of the corresponding surface. It is also possible to select the surface graphically in the RFEM work window. Both functions are available as soon as you click into the input field.



The *Points* column offers different criteria to filter the results. In addition to *All Designable* and *All Undesignable Points*, you can select the *Governing* points. These points provide the maximum reinforcement areas or ratios for the respective ultimate or serviceability limit state designs.

If you select the option *User defined*, you can enter the numbers of the points directly. Click *All* to reset the complete display.

Display only designable or non-designable results

By means of the two buttons shown on the left, all designable results or failed designs can be displayed exclusively. Use these buttons to hide, for example, failed designs due to singularities or analyze the reasons for design problems.

Sorting of Results

By default, table 3.1 and 3.2 show the results arranged according to the maximum design ratios. The sorting conforms to table column J. To control the maximum values in column G, it is possible to sort the results according to the existing values. The maximum ratio of the deformation, for example, does not necessarily represent the maximum deformation as the limit values can be defined differently surface by surface. Use the [Sorting] button to switch between these two types of result arrangement.

Partial View

In addition to the functions in the results tables, you can use all filter options described in the RFEM manual to evaluate the design results graphically. You can take advantage of already existing partial views (cf. RFEM manual, chapter 10.9, page 321) used to group objects appropriately. If necessary, you can create a new partial view for the results of RF-CONCRETE Surfaces.

Section

In the same way, you can use sections in the RFEM model. It is also possible to define new sections to evaluate the results according to your needs (cf. RFEM manual, chapter 10.6, page 313).

Results panel

In the RFEM work window, you can also use the reinforcement areas and ratios as filter criteria. To apply this filter function, the panel must be displayed. If the panel is not active,

select **Control Panel (Colour Scale, Factors, Filter)** on the **View** menu

or use the button in the RFEM toolbar shown on the left.

The panel is described in the RFEM manual, chapter 4.4.6, page 77. The filter settings for the design results can be specified in the first panel tab *Color Spectrum*. The selection of surfaces is defined in the *Filter* tab.

Use the panel, for example, to define that reinforcement areas are displayed only for a specific range of values. In Figure 5.7, the lower limit value is 2.00 cm^2/m. Moreover, the reduced color spectrum has been adjusted so that one color range covers exactly 1.00 cm^2/m. To limit singularities, the upper value is defined by 9.00 cm^2/m.

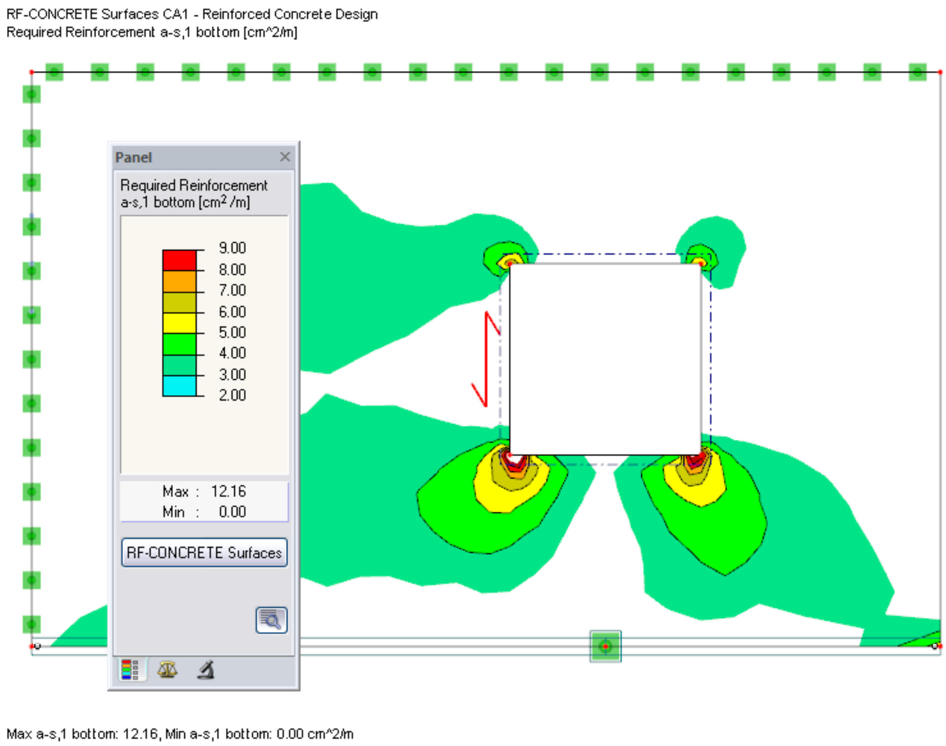


Figure 5.7: Filtering required reinforcement with adjusted color spectrum

FE node and grid values

In the results tables 2.2 and 2.3 (reinforcement) as well as 3.2 and 3.3 (serviceability), a search function for FE nodes and grid points is available. To use this function, click the button shown on the left. The following dialog box opens.

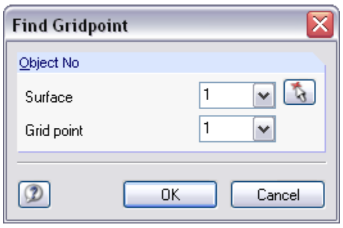


Figure 5.8: Dialog box *Find Gridpoint*

First, enter the number of the surface or use the [Pick] button to select the surface graphically. Then, enter the number of the relevant gird point or FE node or select the the point from the list.

For the graphical display of the grid point or FE node values, the same control functions as used in RFEM are available. The corresponding functions are described in the RFEM manual, chapter 10.4, page 306.

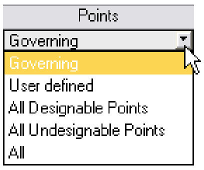
6. Printout

6.1 Printout Report

The creation of printouts is similar to the procedure in RFEM. First, the program generates a printout report for the RF-CONCRETE Surfaces results. Graphics and descriptions can be added. In addition, you can use the print preview to determine the design results that will finally appear in the printout.

When your structure is quite extensive, it is advisable to split the data into several small reports. If you create a separate printout report, for example only for the results of the RF-CONCRETE Surfaces module, this printout report will be generated relatively quickly.

The printout report is described in detail in the RFEM manual. In particular, chapter 11.1.3.4 *Selecting Data of Add-on Modules* on page 338 provides information concerning the selection of input and output data in add-on modules.



A special selection option is available for the intermediate results of serviceability limit state designs. The *Points* column offers different criteria to filter the results. In addition to *All Designable* and *All Undesignable Points*, you can select the *Governing* points. These points provide the maximum ratios for the respective serviceability limit state designs.

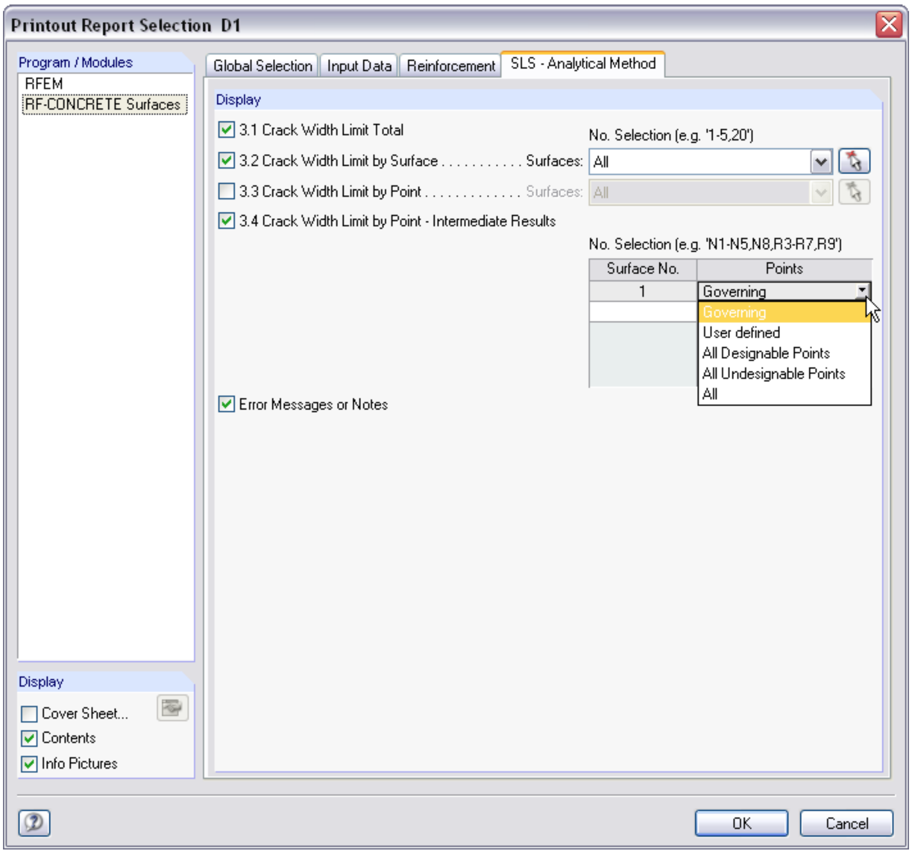


Figure 6.1: Dialog box *Printout Report Selection*, tab *SLS - Analytical Method*

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

6.2 Print RF-CONCRETE Surfaces Graphics

The design graphics can be either integrated in the printout report or sent directly to the printer. Printing graphics is described in detail in the RFEM manual, chapter 11.2, page 354.

Each picture that is displayed in the RFEM work window can be imported into the printout report. Even result diagrams of sections can be integrated in the printout report by using the [Print] button.

To print the current RF-CONCRETE Surfaces graphic in the RFEM window,

select **Print** on the **File** menu

or use the toolbar button shown on the left.

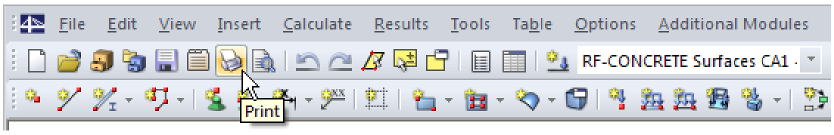


Figure 6.2: Button *Print* in the toolbar of the main window

The following dialog box opens:

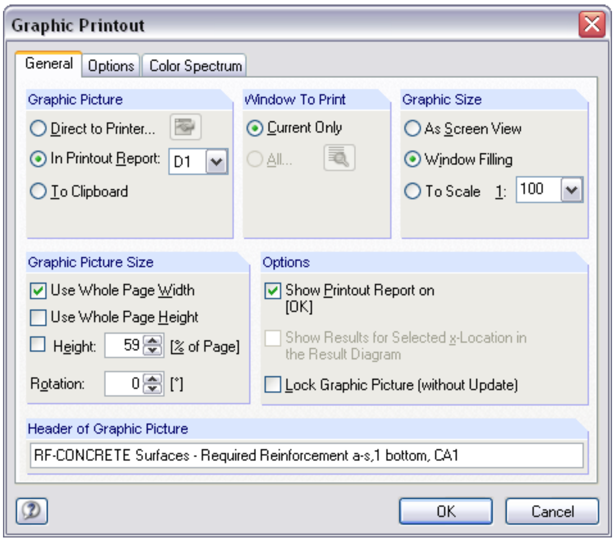
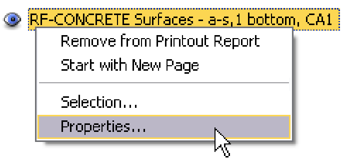


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

Every graphic from RF-CONCRETE Surfaces, that has been integrated in the printout report, can be moved anywhere within the report by using the drag-and-drop function.



Furthermore, it is possible to adjust imported graphics subsequently: Right-click the relevant entry in the navigator of the printout report and select *Properties* in the context menu. The dialog box *Graphic Printout* appears again where you can adjust the data appropriately.

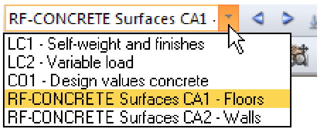
7. General Functions

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

7.1 Design Cases in RF-CONCRETE Surfaces

You can group surfaces in separate design cases. In this way, it is possible to summarize for example surface groups or to specify particular types of design (standard, reinforcement, non-linear analysis etc.).

It is important that the surfaces in a reinforcement group of a design case are assigned without ambiguity (cf. chapter 2.5, page 27). In contrast, it is no problem to analyze one and the same surface in different design cases.



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

Create a new RF-CONCRETE Surfaces case

To create a new design case,

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

The following dialog box appears.

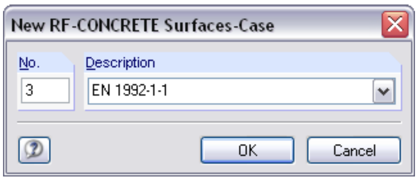


Figure 7.1: Dialog box *New RF-CONCRETE Surfaces-Case*

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

Rename a RF-CONCRETE Surfaces case

To change the description of a design case subsequently,

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

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

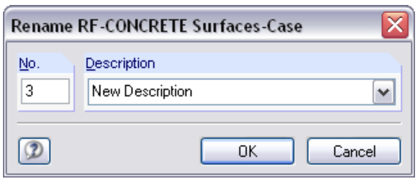


Figure 7.2: Dialog box *Rename RF-CONCRETE Surfaces-Case*

Program RF-CONCRETE Surfaces © 2010 Dlubal Engineering Software

Copy a RF-CONCRETE Surfaces case

To copy the input data of the current design case,

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

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

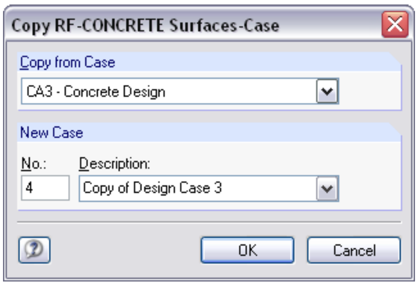


Figure 7.3: Dialog box *Copy RF-CONCRETE Surfaces-Case*

Delete a RF-CONCRETE Surfaces case

To delete a design case,

select **Delete Case** on the **File** menu in the RF-CONCRETE Surfaces add-on module.

In the dialog box *Delete Cases*, select the relevant RF-CONCRETE Surfaces design case in the *Available Cases* list to delete it by clicking [OK].

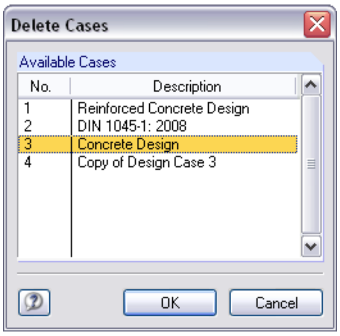


Figure 7.4: Dialog box *Delete Cases*

7.2 Units and Decimal Places

The units and decimal places for RFEM and all add-on modules are managed in one global dialog box. In the RF-CONCRETE Surfaces add-on module, you can use the menu to define the units. To open the corresponding dialog box,

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

The program opens the following dialog box that you already know from RFEM. The add-on module RF-CONCRETE Surfaces is already preset.

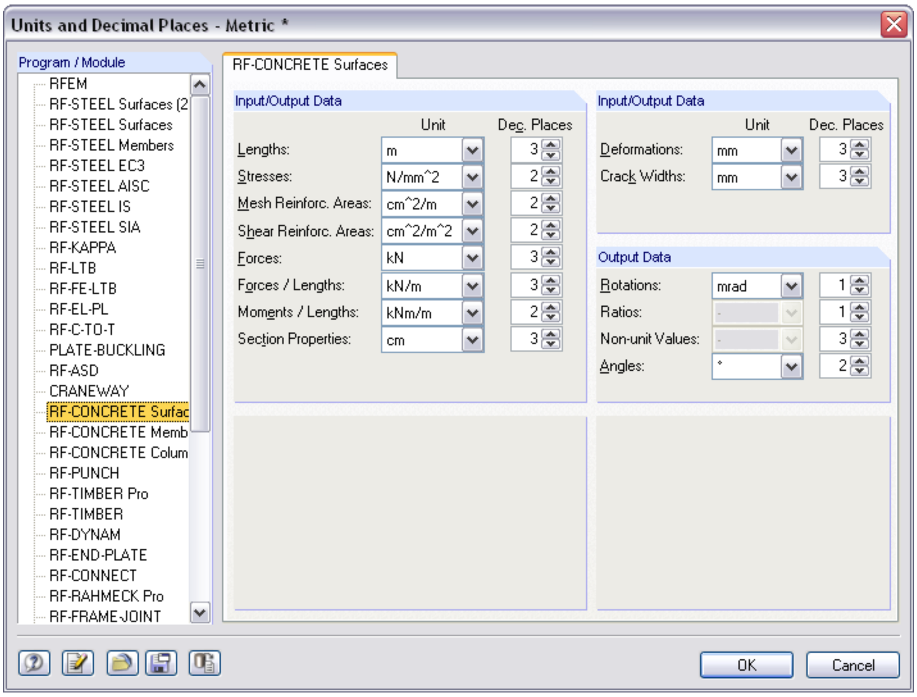


Figure 7.5: Dialog box *Units and Decimal Places*

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

7.3 Export of Results

The design results can be provided for other programs in several ways.

Clipboard

To copy cells selected in the results tables of RF-CONCRETE Surfaces to the clipboard, use the keyboard keys [Ctrl]+[C]. To insert the cells, for example in a word processing program, press [Ctrl]+[V]. The headers of the table columns won't be transferred.

Printout report

The data of the RF-CONCRETE Surfaces module can be printed into the global printout report (cf. chapter 6.1, page 58) to export them subsequently. In the printout report,

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

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

Excel / OpenOffice

RF-CONCRETE Surfaces provides a function for the direct data export to MS Excel and OpenOffice.org Calc. To open the corresponding dialog box,

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

The following export dialog box appears.

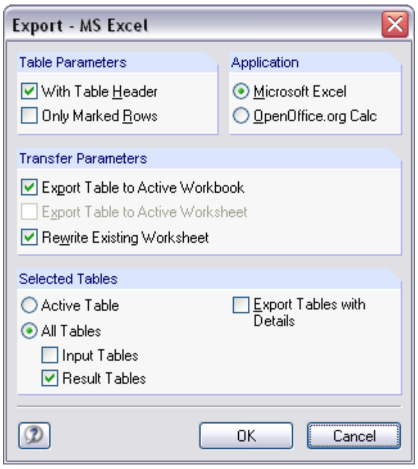
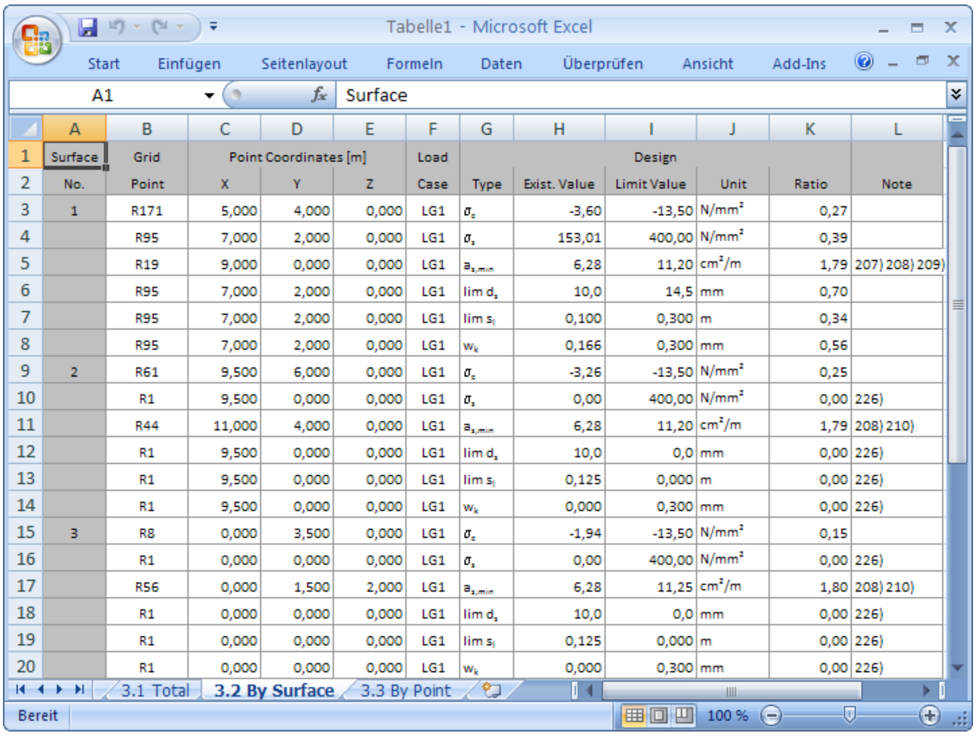


Figure 7.6: Dialog box *Export - MS Excel*

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



	A	B	C	D	E	F	G	H	I	J	K	L
1	Surface	Grid	Point Coordinates [m]			Load	Design					
2	No.	Point	X	Y	Z	Case	Type	Exist. Value	Limit Value	Unit	Ratio	Note
3	1	R171	5,000	4,000	0,000	LG1	σ_c	-3,60	-13,50	N/mm²	0,27	
4		R95	7,000	2,000	0,000	LG1	σ_s	153,01	400,00	N/mm²	0,39	
5		R19	9,000	0,000	0,000	LG1	$a_{s,min}$	6,28	11,20	cm²/m	1,79	207) 208) 209)
6		R95	7,000	2,000	0,000	LG1	lim d_s	10,0	14,5	mm	0,70	
7		R95	7,000	2,000	0,000	LG1	lim s_l	0,100	0,300	m	0,34	
8		R95	7,000	2,000	0,000	LG1	w_k	0,166	0,300	mm	0,56	
9	2	R61	9,500	6,000	0,000	LG1	σ_c	-3,26	-13,50	N/mm²	0,25	
10		R1	9,500	0,000	0,000	LG1	σ_s	0,00	400,00	N/mm²	0,00	226)
11		R44	11,000	4,000	0,000	LG1	$a_{s,min}$	6,28	11,20	cm²/m	1,79	208) 210)
12		R1	9,500	0,000	0,000	LG1	lim d_s	10,0	0,0	mm	0,00	226)
13		R1	9,500	0,000	0,000	LG1	lim s_l	0,125	0,000	m	0,00	226)
14		R1	9,500	0,000	0,000	LG1	w_k	0,000	0,300	mm	0,00	226)
15	3	R8	0,000	3,500	0,000	LG1	σ_c	-1,94	-13,50	N/mm²	0,15	
16		R1	0,000	0,000	0,000	LG1	σ_s	0,00	400,00	N/mm²	0,00	226)
17		R56	0,000	1,500	2,000	LG1	$a_{s,min}$	6,28	11,25	cm²/m	1,80	208) 210)
18		R1	0,000	0,000	0,000	LG1	lim d_s	10,0	0,0	mm	0,00	226)
19		R1	0,000	0,000	0,000	LG1	lim s_l	0,125	0,000	m	0,00	226)
20		R1	0,000	0,000	0,000	LG1	w_k	0,000	0,300	mm	0,00	226)

Figure 7.7: Result in *Excel*

CAD programs

The reinforcement areas determined in RF-CONCRETE Surfaces can also be used in CAD programs. RFEM provides interfaces to the following programs:

- Nemetschek (FEM format for Allplan *.asf)
- Glaser (format *.fem)
- Strakon (format *.cfe)

To use this export function,

select **Export** on the **File** menu in RFEM.

The *Export* dialog box appears where you can select the relevant interface. The dialog box is described in detail in the RFEM manual, chapter 13.5.

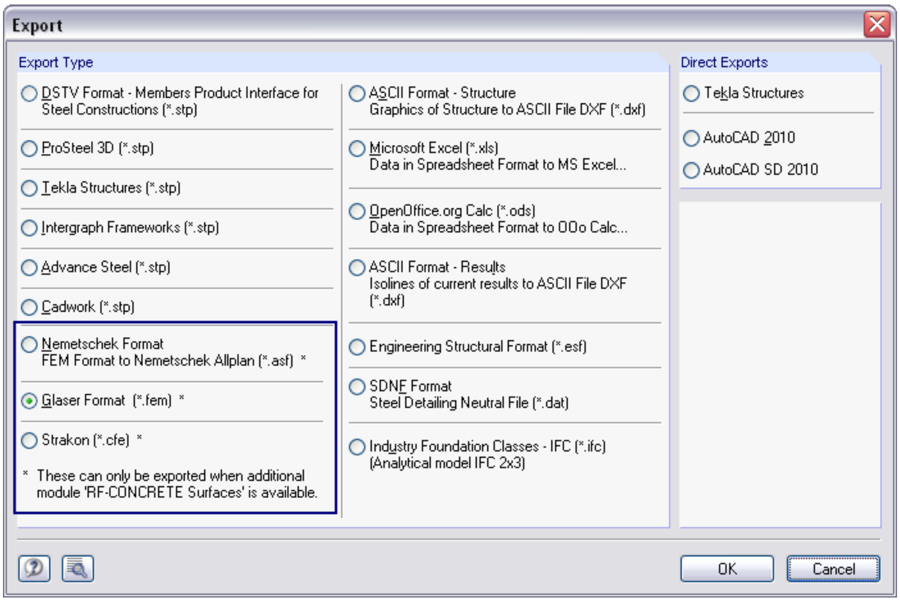


Figure 7.8: RFEM dialog box *Export*

Depending on the interface, specific options are available for the export of reinforcements. To access the corresponding options, use the [Details] button in the *Export* dialog box. Another dialog box opens where you can define specific settings for the interface.

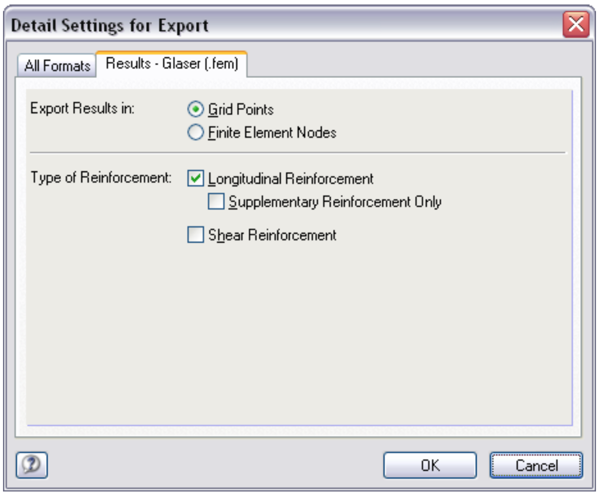


Figure 7.9: Dialog box *Detail Settings for Export*, results for *Glaser*

A Literature

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

[2] DIN 1045: Beton- und Stahlbetonbau. Juli 1988.

[3] DIN 1045-1: Tragwerke aus Beton, Stahlbeton und Spannbeton - Teil 1: Bemessung und Konstruktion. Juni 2001.

[4] DIN V ENV 1992-1-1 (Eurocode 2): Planung von Stahlbeton- und Spannbetontragwerken - Teil 1: Grundlagen und Anwendungsregeln für den Hochbau. Juni 1992.

[5] REYMENDT Jörg: DIN 1045 neu, Anwendung und Beispiele. Papenberg Verlag, Frankfurt 2001.

[6] Deutscher Beton-Verein e.V.: Beispiele zur Bemessung von Betontragwerken nach EC2. Bauverlag, Wiesbaden/Berlin 1994.

[7] AVAK, Ralf.: Stahlbetonbau in Beispielen, DIN 1045 und Europäische Normung, Teil 2: Konstruktion-Platten-Treppen-Fundamente. Werner Verlag, Düsseldorf 1992.

[8] AVAK, Ralf: Stahlbetonbau in Beispielen, DIN 1045 und Europäische Normung, Teil 2: Bemessung von Flächentragwerken, Konstruktionspläne für Stahlbetonbauteile, 2. Auflage. Werner Verlag, Düsseldorf 2002.

[9] SCHNEIDER, Klaus-Jürgen: Bautabellen für Ingenieure mit Berechnungshinweisen und Beispielen, 15. Auflage. Werner Verlag, Düsseldorf 2002.

[10] PFEIFFER, Uwe: Die nichtlineare Berechnung ebener Rahmen aus Stahl- oder Spannbeton mit Berücksichtigung der durch das Aufreißen bedingten Achsendehnung. Cuviller Verlag, Göttingen 2004.

[11] LANG, Christian, MEISWINKEL, Rüdiger, WITTEK, Udo: Bemessung von Stahlbetonplatten mit dem nichtlinearen Verfahren nach DIN 1045-1. Beton- und Stahlbetonbau 95, 2000, Heft 5, S. 270-278.

[12] SCHLAICH/SCHÄFER: Konstruieren im Stahlbetonbau. Betonkalender 1993 Teil II. Verlag Ernst & Sohn, Berlin 1993.

[13] MEISWINKEL, Rüdiger: Nichtlineare Nachweisverfahren von Stahlbeton-Flächentragwerken. Beton- und Stahlbetonbau 96, 2000, Heft 1, S. 27-34.

[14] RAHM, Heiko: Modellierung und Berechnung von Alterungsprozessen bei Stahlbeton-Flächentragwerken. Universität Kaiserslautern 2002.

[15] KUPFER, Herbert, HILSDORF, Hubert K., RÜSCH, Hubert: Behavior of concrete under biaxial stresses, ACI Journal, 1969.

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

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

[18] SCHNEIDER, Klaus-Jürgen: Bautabellen für Ingenieure mit Berechnungshinweisen und Beispielen, 13. Auflage. Werner Verlag, Düsseldorf 1998.

B Index