Ing. Software
Dlubal

Version
January 2010

Add-on Module

RF-SOILIN

Soil-Structure Interaction Analysis

Program Description

All rights, including those of the translation, are reserved.

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

© Ing.-Software Dlubal

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

	Contents	Page
1.	**Introduction**	**4**
1.1	Add-on Module RF-SOILIN	4
1.2	RF-SOILIN Team	5
1.3	System Requirements	5
1.4	Installation	5
2.	**Theoretical Principles**	**6**
2.1	Soil-Structure Interaction	6
3.	**Using RF-SOILIN**	**8**
3.1	Starting RF-SOILIN	8
3.2	Tables	8
3.3	Input Tables	9
3.3.1	General Data	9
3.3.2	Soils, Soil Samples and Soil Layers	12
3.4	Results Tables	14
3.4.1	Stresses and Settlements	14
3.4.2	Elastic Foundation Coefficients	16
3.5	Main Menus	18
3.5.1	File	18
3.5.2	Settings	18
3.5.3	Help	18
4.	**Results**	**19**
4.1	Screen Display	19
4.2	Printing	20
A:	**Literature**	**21**

1. Introduction

1.1 Add-on Module RF-SOILIN

In civil engineering practice, the WINKLER's foundation model is still used at present. This model assumes a linear relation between load and soil settlement, the modulus of foundation C is considered constant. However, the reality is different. Therefore, this calculation method has been increasingly criticized recently. More and more challenging structures require a more accurate soil-structure interaction analysis because the foundation has an indispensable influence on the internal forces in a structure itself.

Hence, we implemented an improved foundation model with more parameters to the RFEM program. With this model, you can perform very realistic soil settlement analyses. The problem is how to determine the exact values of the elastic foundation coefficients $C_{1,z}$, $C_{2,x}$ and C_{2y} for RFEM.

The add-on module RF-SOILIN assists you to calculate these parameters. On the basis of loads and data taken from a soil expertise (stiffness modulus, or modulus of elasticity and Poisson's ratio, weight density, layer thickness), the foundation parameters are calculated by means of a non-linear method for every finite element. These parameters depend on the load and influence the structure's behavior. The analysis runs through an iteration process. Those results represent the realistic settlement values and, thus, the more accurate internal forces in a structure.

We wish you much success and delight when working with our module RF-SOILIN.

Your DLUBAL ENGINEERING SOFTWARE team.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

1.2 RF-SOILIN Team

The following people participated in the development of the RF-SOILIN module:

Program Coordinators:

- Dipl.-Ing. Georg Dlubal
- Ing. Pavel Bartoš
- Ing. Jirí Buček
- Dipl.-Ing. Frank Faulstich
- Dipl.-Ing. Rafael Ceglarek

Programmers:

- David Schweiner
- Ing. Zdenek Kosáček
- Ing. Radoslav Rusina
- Doc. Ing. Ivan Nemec, CSc.

Testing:

- Ing. Martin Vasek
- Karel Kolár
- Dipl.-Ing. Frank Faulstich

Manuals and Support:

- Dipl.-Ing. Frank Faulstich
- Mgr. Michaela Kryšková
- Dipl.-Ing. (FH) Robert Vogl

1.3 System Requirements

The system requirements do not differ from the general requirements for working with the basic program RFEM. Please see the RFEM manual [1], page 11, for more details.

1.4 Installation

RF-SOILIN is not a stand-alone program; it represents an add-on module integrated to the main program RFEM. Therefore, you have to start the common installation of the RFEM program after having purchased this module. During the installation, you must use a new authorization file. The modules with purchased licenses will be available depending on the authorization file of the disk or e-mail.

The installation process is described in detail in the RFEM manual [1], on pages 11-13.

2. Theoretical Principles

The soil contact stress under foundation bodies depends on plasticity of the foundation bodies and of the soil. We distinguish the limit cases *'soft load bundle'* and *'rigid plate'*. The soft load bundle clings to the induced subsidence basin. The soil contact stress remains constant. In case of a rigid plate, the stress peaks at the margins.

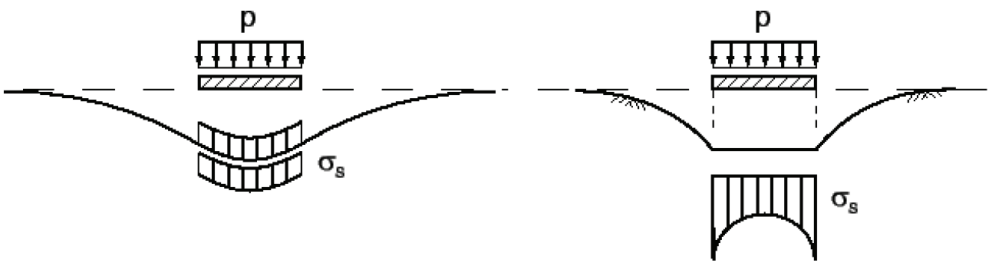


Figure 2.1: Soil contact stress depends on rigidity of foundation bodies

The foundation parameters as a soil characteristic also depend on the soil contact stresses which occur under the foundation plate. Thus, the rigidity of foundations and of the entire construction has a decisive influence on the distribution of the foundation parameters. On the other hand, the foundation parameters influence the internal forces within the structure.

2.1 Soil-Structure Interaction

The distribution of the foundation parameters under the foundation plate is necessary for the calculation of the soil contact stresses. Simultaneously, the distribution depends on the soil contact stresses. On the basis of the complex soil-structure interaction analysis, it is not possible to determine the foundation parameters by using a simple calculation procedure.

In RF-SOILIN, the foundation parameters are calculated iteratively. For the first iteration step, it is necessary to use the default initial values of the foundation parameters. With these initial values, the finite element analysis can be executed for FE-models in RFEM. The distribution of the soil contact stresses represents the available results in this case.

The soil contact pressure from the first iteration step enters the analysis in RF-SOILIN as an initial value. The soil settlement can be calculated for every finite element, together with the stiffness moduli of the defined soil layers. From the soil settlement and soil contact stresses, the foundation parameters are determined. For the next iteration step, the new foundation parameters are applied. A new finite element analysis is carried out which again produces a different distribution of the soil contact stresses. To check on the convergence behavior, this new distribution is compared with the previous one. As long as the deviation exceeds the limit defined as convergence criterion, the new contact stress distribution reenters the analysis and the new foundation parameters are calculated in RF-SOILIN again.

If the deviation in the distribution of the soil contact stresses is below the limit in two following iterations, the analysis terminates. These foundation parameters of the last iteration step represent the RF-SOILIN results.

In the following picture, you can see the process of analysis in RF-SOILIN.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

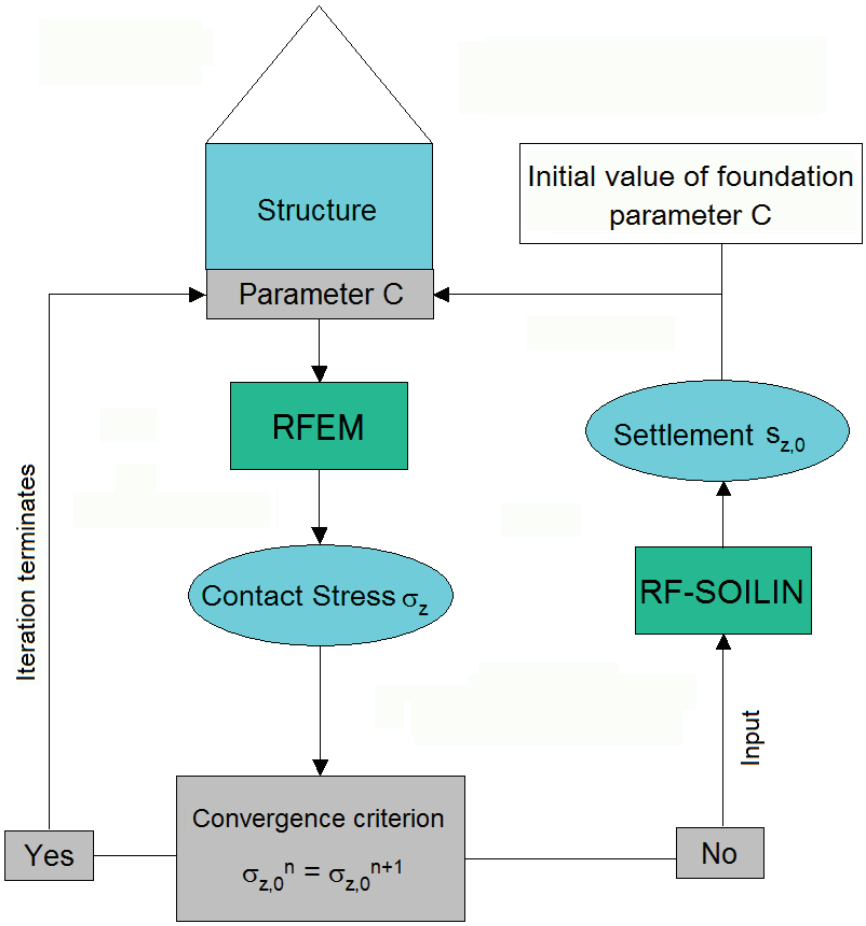


Figure 2.2: Process of analysis in RF-SOILIN

The soil settlements represent the decisive intermediate value for the calculation of the foundation parameters. For the stress distribution due to an additional load, RF-SOILIN idealizes the soil as a homogenous half-space with a linear-elastic, isotropic material (BOUSSINESQ model).

The stress is integrated by sublayers, and together with the corresponding stiffness modulus, the soil settlements are calculated.

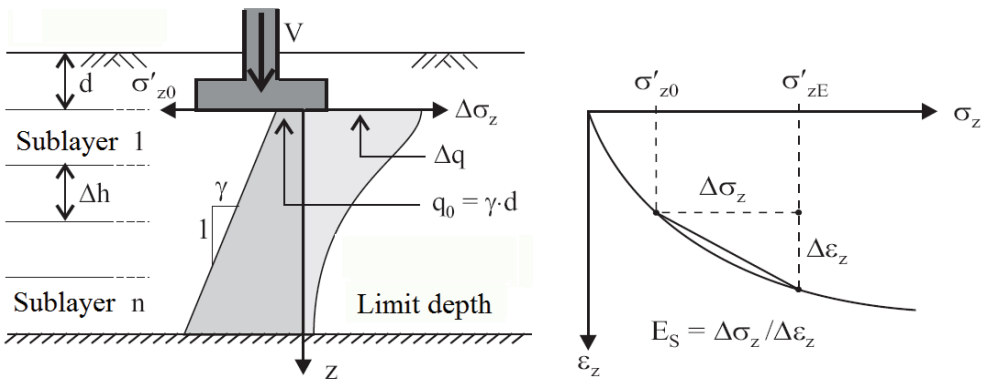


Figure 2.3: Stress distribution in the elastic half-space down to the limit depth

By using the soil contact stress and soil settlements, the foundation parameters are calculated.

3. Using RF-SOILIN

3.1 Starting RF-SOILIN

The RF-SOILIN module can be started by using the command from the RFEM main menu

Additional Modules → Others → RF-SOILIN

or from the *Data* navigator by clicking the item

Additional Modules → RF-SOILIN.

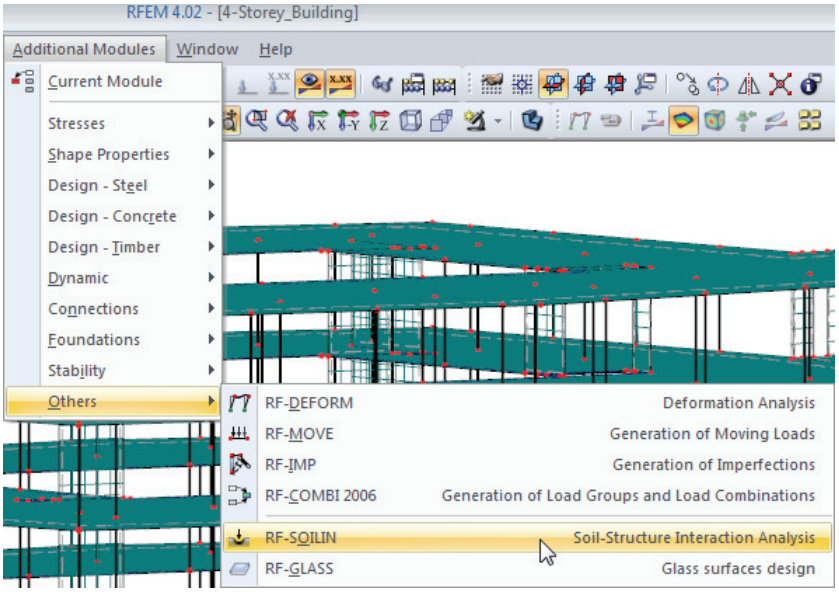


Figure 3.1: Starting the RF-SOILIN module from the main menu *Additional Modules*

3.2 Tables

Different module windows are used to enter input data and also later to display numerical results on a screen.

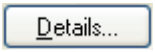
On the left side of the RF-SOILIN window, a navigator with the list of all currently available tables is shown.

You can open the tables either by clicking the relevant navigator item in RF-SOILIN or you can browse them by using the keys [F2] and [F3] or the buttons [⇐] and [⇒].

By using the [Calculation] button, you run the analysis after having defined all input data.

By the [OK] button, you save both input data and results before you exit the RF-SOILIN module, while by the button [Cancel] you exit the module without saving data.

By the [?] button or the functional key [F1], you start the *Help* function.



By the [Details…] button, you open a dialog box in which you can set *Grid for Result Values* and the edge distance of *Subsidence Basin* from the foundation surface.

The buttons [Calculation] and [Details…] are only available in the input tables.

By the [Graphic] button, you return to the RFEM work window.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

3.3 Input Tables

In the input tables, all data and parameters are entered that are necessary for the analysis.

3.3.1 General Data

After starting RF-SOILIN, the table 1.1 *General Data* opens automatically in the module window.

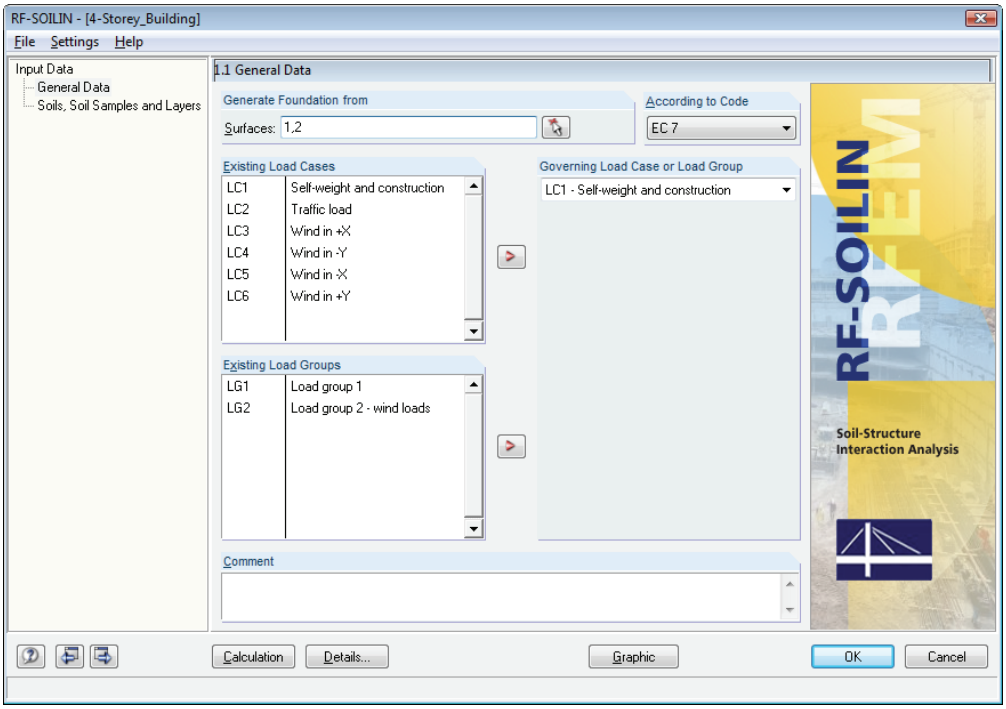


Figure 3.2: Table 1.1 *General Data*

In the section *Generate Foundation from*, you enter the numbers of the *Surfaces* for which you wish to calculate the soil-structure interaction. The surface numbers can either be written directly into this box or be selected graphically by using the button [Pick Surfaces] in the work window of RFEM.

You can also define the surfaces already when defining the elastic foundation in RFEM (see the following picture).

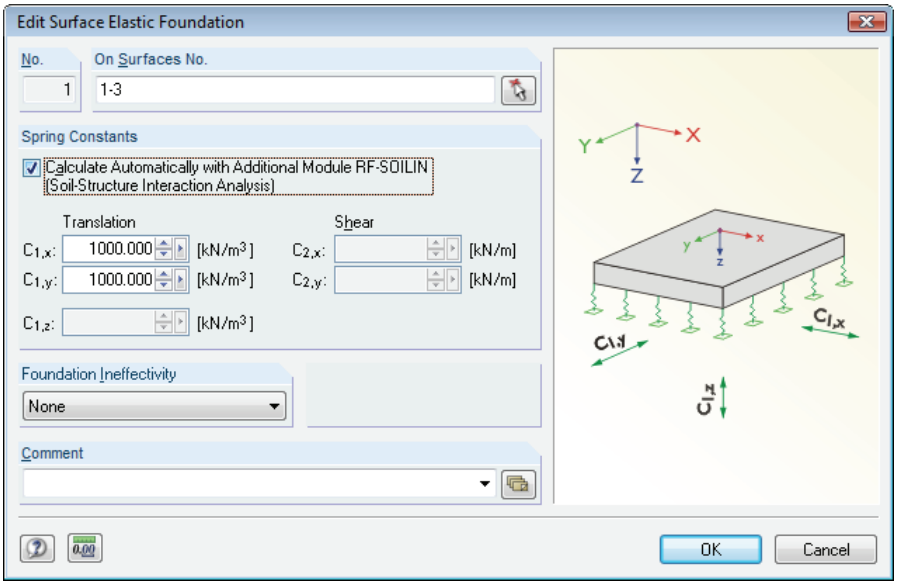


Figure 3.3: Dialog box *Edit Surface Elastic Foundation* in RFEM

If you select the check box *Calculate automatically with additional module RF-SOILIN (Soil-structure interaction analysis)* in the dialog box *Edit Surface Elastic Foundation* in RFEM, all these surface numbers will automatically be set in the first table of RF-SOILIN.

You can carry out the analysis according to three different codes:

- EC 7 (European) [9]
- DIN 4019 (German) [10]
- CSN 73 1001 (Czech) [11]

In table 1.1, you select the appropriate standard in the dialog section *According to Code*.

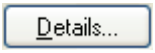
The selection of loads in RF-SOILIN proceeds in a different way than in other modules, for example in RF-CONCRETE Surfaces. There, you can choose several load cases, load groups or load combinations from which the governing internal forces are determined for the design.

In RF-SOILIN, however, the stiffness for every single finite element is calculated during the soil-structure interaction analysis. Hence, there is an interaction with the structural data. The stiffness values of the foundation springs depend on the load. This means that the calculation is only possible for one specific load situation. Therefore, you can select only one load case or one load group in table 1.1. Load combinations have always two values at every location, the minimum and maximum, so you can not choose any load combination.

For the same reason, you can not define several design cases in RF-SOILIN, which is usually possible in other add-on modules.

Regarding the code rules, usually several different combinations of load cases have to be created. RF-SOILIN does not restrict you here: From different superpositions of load cases, you have to choose only one that will be used to calculate the foundation moduli. Generally, this will be the load group (!) with permanent and quasi-permanent service loads. For the steel design or reinforced concrete design, you can use the combinations for the ultimate limit states as usual.

You can select the governing load case or load group from the list *Existing Load Cases* or from *Existing Load Groups* by using the button [>]. You can also set the relevant load directly in the list *Governing Load Case or Load Group*.



By using the button [Details...], you can set a number of parameters for the analysis.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

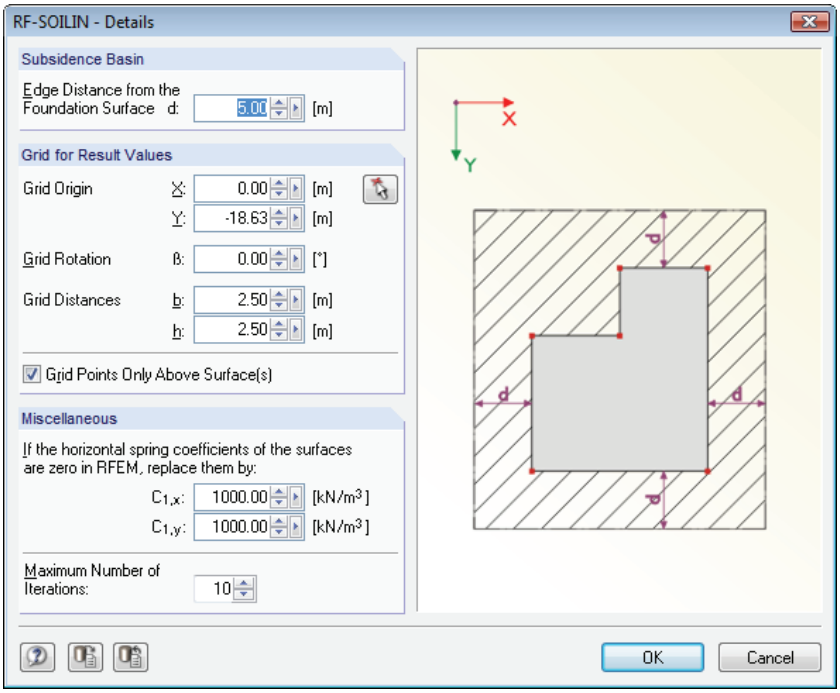


Figure 3.4: Dialog box *RF-SOILIN – Details*

Subsidence Basin

For the analysis, it is necessary to consider a certain area around the actual structure.

This area is marked by a dotted line in the following picture.

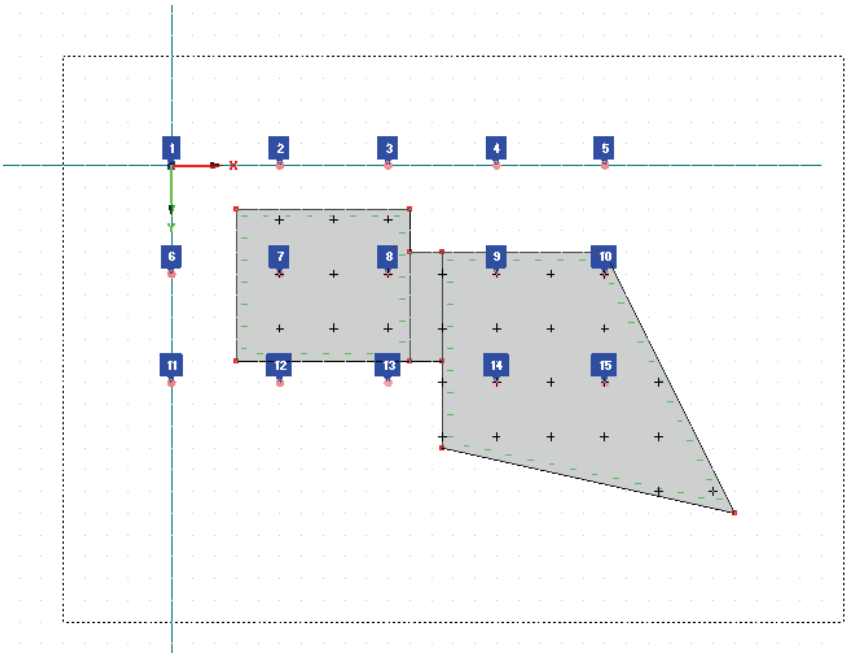


Figure 3.5: Area of the subsidence basin

For example, when you enter an *Edge Distance* of 5.00 m, this minimum distance will be kept from the structural edges and soil samples.

Grid for Result Values

In this section, you define the grid points for which the resulting stresses and settlement values are to be displayed.

Miscellaneous

If there haven't been defined any horizontal springs for the surface elastic foundation in RFEM and if no other horizontal supports exist, the structure is instable for the analysis. In order to avoid this, horizontal springs can be defined in the dialog section *Miscellaneous*.

The calculation of the foundation moduli is carried out iteratively. If no equilibrium can be reached and the *Maximum number of iterations* has been reached, the analysis is interrupted and a message appears whether the results are nevertheless to be shown.

3.3.2 Soils, Soil Samples and Soil Layers

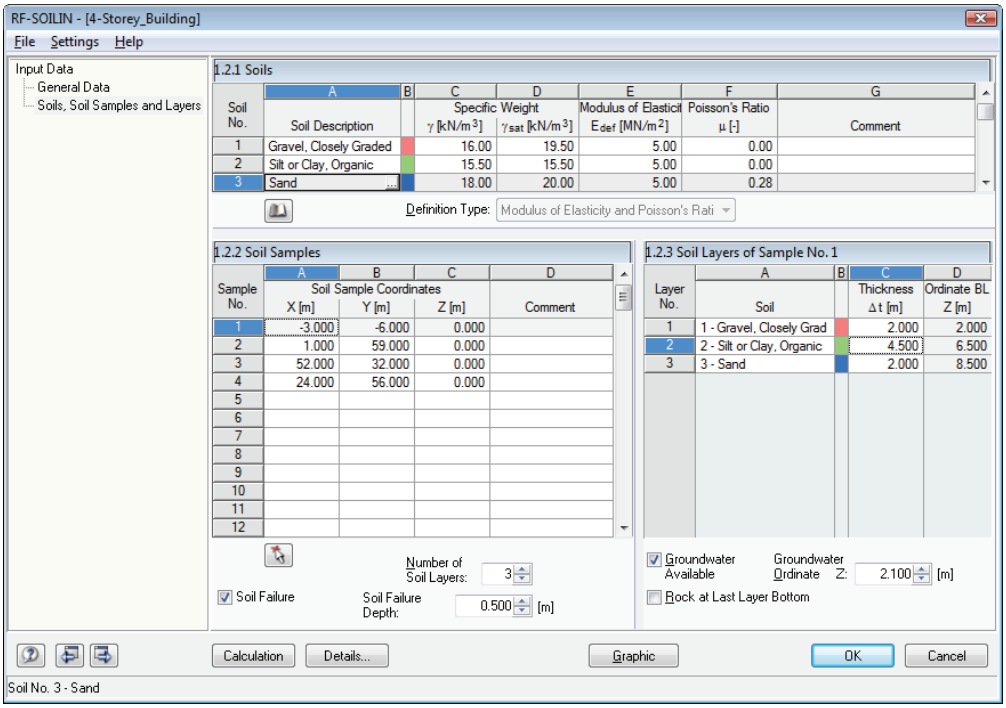


Figure 3.6: Table 1.2 *Soils, Soil Samples and Soil Layers*

In table 1.2, you define the soil parameters, the soil sample coordinates in a global system of coordinates, and the soil layers.

Soils

At first, you have to enter the geomechanical properties for all existing soils in table 1.2.1. Each line corresponds to one soil category. When you enter input data in the following tables, the soil numbers refer to the line numbers of table 1.2.1.

Regardless of the code and the definition type, you have to fill in the following values:

- γ weight density
- γ_{sat} weight density of water-saturated soil

Every type of soil is marked by a specific color in column B. The same colors are also shown in the table 1.2.3 *Soil Layers* below.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

The remaining input options depend on the code that has been selected in the previous table 1.1 *General Data*.

EC 7

If the analysis should proceed according to Eurocode 7, you have to enter the modulus of elasticity E_{def} and the Poisson's ratio μ in table 1.2.1.

DIN 4019

You can choose from two options:

- Definition by using the *Stiffness modulus* E_s
- Definition by using the *Modulus of elasticity* and *Poisson's ratio*

If you select the first option, you have to enter only the stiffness modulus E_s in table 1.2.1. For the second option, the modulus of elasticity E_{def} and Poisson's ratio μ. are to be entered.

The following relation exists between E_s and E_{def}:

$$E_s = \frac{E_{def} \cdot (1-\mu)}{(1+\mu)\cdot(1-2\mu)}$$

Formula 3.1

CSN 731001

Similar to the analysis according to EC 7, you can only enter the modulus of elasticity E_{def} and the Poisson's ratio μ. Additionally, you also have to define the coefficient *m* which represents the corrective coefficient for additional loads.

Soil Samples

In table 1.2.2, you specify the coordinates of soil samples. The function *Pick position of the sample* facilitates this input. The coordinates of a soil sample location can be picked directly in the RFEM work window and are then transferred to the current table line.

Below table 1.2.2, you can enter the *Number of soil layers*. You can take into account potential *Soil Failure* by selecting the check box below the table. Then, you specify the *Soil Failure Depth* in the input field next to the check box.

Soil Layers of Sample

Before you define soil layers, it is important to place the cursor in the appropriate line of table 1.2.2 *Soil Samples*. Then you can enter the exact order of layers for the soil sample that is currently selected.

The number of lines in table 1.2.3 corresponds to the *Number of soil layers* defined in table 1.2.2 *Soil Samples*. In areas between the locations of the samples, the program interpolates from the layers of each sample. Thus, all samples must have the same numbers of layers. Therefore, the highest number of layers is set for all samples automatically. The layers that are not contained in a certain soil sample can be defined with the thickness of 0.0 m.

In the column *Soil*, you can select any entry from the list of defined soil materials or enter the soil number directly.

The *Thickness* of each layer is defined in column C. The value of the *Ordinate BL Z* can be used to check the input.

There are two check boxes located beneath the table. If you select the check box *Groundwater available*, you have to specify the level of subsoil water in the next field.

You can define *Rock at last layer bottom* by using the second check box. This disruption of the elastic, isotropic half-space has an influence on the stress distribution due to the loads with increasing depth.

3.4 Results Tables

3.4.1 Stresses and Settlements

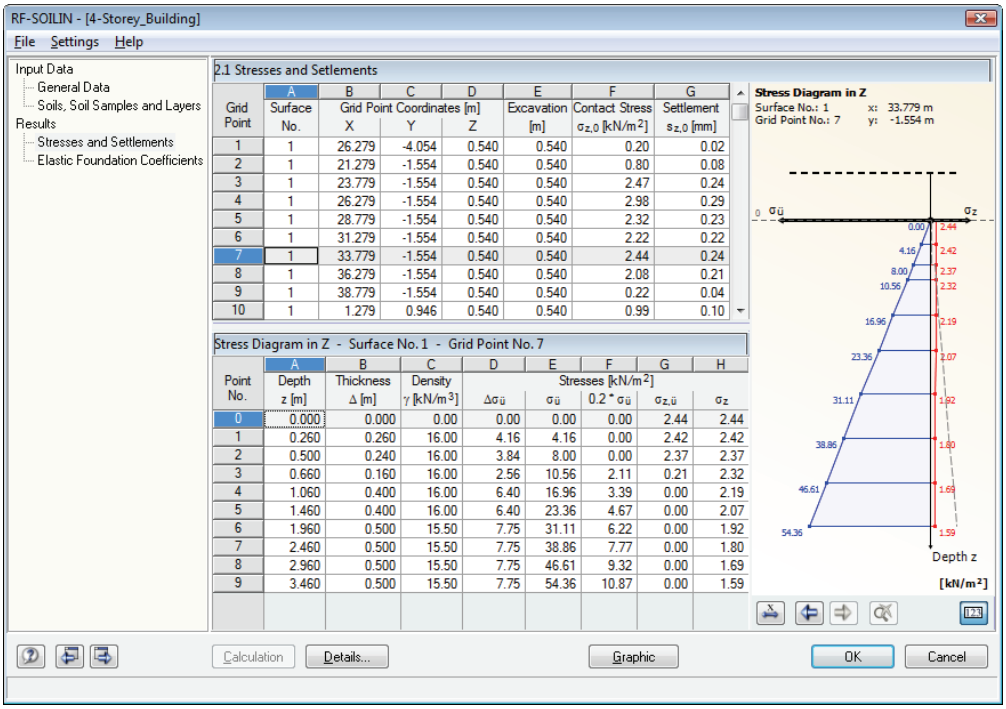


Figure 3.7: Table 2.1 *Stresses and Settlements*

Table 2.1 is divided into two parts. In the upper part, the stresses and settlements are shown which refer to the terrain surface. In the lower part, all stresses are shown in the different soil layers. They are related to the grid point that is selected in the upper table.

In the upper table, you can find the coordinates of the individual grid points (on the terrain surface) in the global coordinate system and the associated surface numbers. For every grid point, the *Excavation* depth, the *Contact stress* $\sigma_{z,0}$, and the *Settlement* $s_{z,0}$ is displayed.

In the lower table, the detailed stresses are shown depending on the selected code. The different parameters are described in the following tables.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

EC 7

In the following table, you can find the values related to EC 7:

Value	Description
z	Soil depth
Δd	Layer thickness
γ	Weight density of soil
$\Delta\sigma_{Or}$	Change of geostatic stress from soil self-weight without buoyancy: $\Delta\sigma_{Or} = \gamma \cdot \Delta d$ with buoyancy: $\Delta\sigma_{Or} = \gamma' \cdot \Delta d = (\gamma_{sat} - \gamma)\Delta d$
σ_{Or}	Geostatic stress from soil self-weight $\sigma_{Or,n} = \sigma_{Or,n-1} + \Delta\sigma_{Or}$ n: layer number
$0.2*\sigma_{Or}$	20%-value of geostatic stress from soil self-weight for determination of limit depth
$\sigma_{z,Or}$	Effective stress $\sigma_{z,Or} = \sigma_z - (0{,}2 \cdot \sigma_{Or})$

Table 3.1: Results for EC 7

DIN 4019

The following table includes the values related to the German standard DIN 4019:

Value	Description
z	Soil depth
Δd	Layer thickness
γ	Weight density of soil
$\Delta\sigma_{ü}$	Change of geostatic stress from soil self-weight without buoyancy: $\Delta\sigma_{ü} = \gamma \cdot \Delta d$ with buoyancy: $\Delta\sigma_{ü} = \gamma' \cdot \Delta d = (\gamma_{sat} - \gamma)\Delta d$
$\sigma_{ü}$	Geostatic stress from soil self-weight $\sigma_{ü,n} = \sigma_{ü,n-1} + \Delta\sigma_{ü}$ n: layer number
$0.2*\sigma_{ü}$	20%-value of geostatic stress from soil self-weight for determination of limit depth
σ_z	Stress from structure load

Table 3.2: Results for DIN 4019

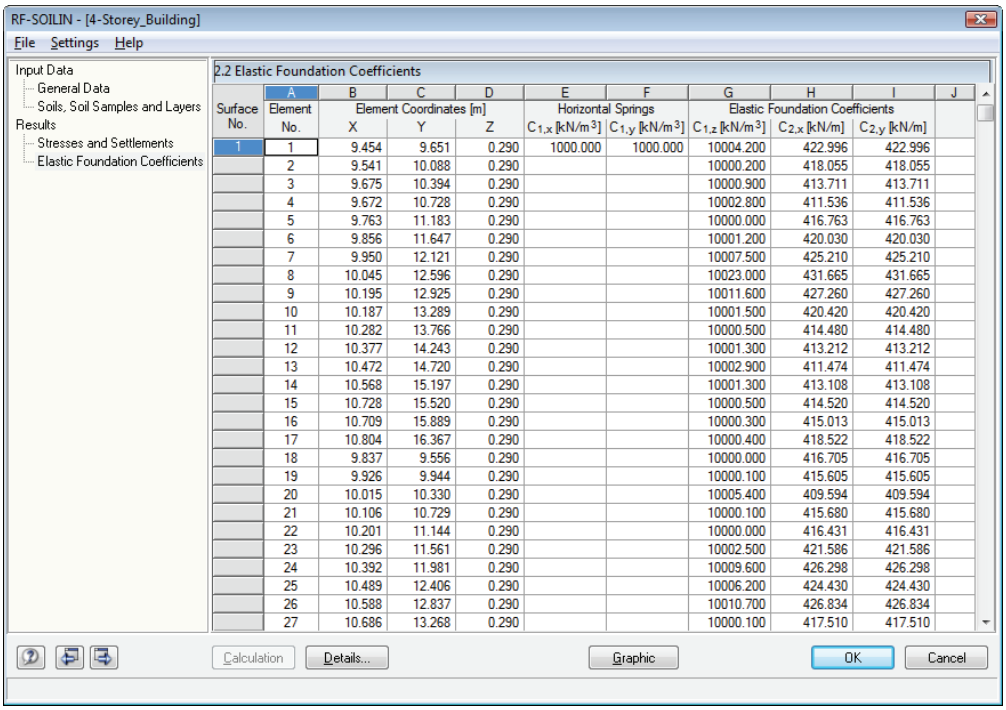
CSN 731001

In the following table, you can find the values related to the Czech code CSN 731001:

Value	Description
z	Soil depth
Δd	Layer thickness
γ	Weight density of soil
$\Delta\sigma_{Or}$	Change of geostatic stress from soil self-weight without buoyancy: $\Delta\sigma_{Or} = \gamma \cdot \Delta d$ with buoyancy: $\Delta\sigma_{Or} = \gamma' \cdot \Delta d = (\gamma_{sat} - \gamma)\Delta d$
σ_{Or}	Geostatic stress from soil self-weight $\sigma_{Or,n} = \sigma_{Or,n-1} + \Delta\sigma_{Or}$ n: layer number
$m^*\sigma_{Or}$	Stress for determination of limit depth
$\sigma_{z,Or}$	Effective stress $\sigma_{z,Or} = \sigma_z - (m \cdot \sigma_{Or})$

Table 3.3: Results for CSN 731001

3.4.2 Elastic Foundation Coefficients



A	B	C	D	E	F	G	H	I	J
Surface No.	Element No.	Element Coordinates [m] X	Y	Z	Horizontal Springs $C_{1,x}$ [kN/m³]	$C_{1,y}$ [kN/m³]	Elastic Foundation Coefficients $C_{1,z}$ [kN/m³]	$C_{2,x}$ [kN/m]	$C_{2,y}$ [kN/m]
1	1	9.454	9.651	0.290	1000.000	1000.000	10004.200	422.996	422.996
	2	9.541	10.088	0.290			10000.200	418.055	418.055
	3	9.675	10.394	0.290			10000.900	413.711	413.711
	4	9.672	10.728	0.290			10002.800	411.536	411.536
	5	9.763	11.183	0.290			10000.000	416.763	416.763
	6	9.856	11.647	0.290			10001.200	420.030	420.030
	7	9.950	12.121	0.290			10007.500	425.210	425.210
	8	10.045	12.596	0.290			10023.000	431.665	431.665
	9	10.195	12.925	0.290			10011.600	427.260	427.260
	10	10.187	13.289	0.290			10001.500	420.420	420.420
	11	10.282	13.766	0.290			10000.500	414.480	414.480
	12	10.377	14.243	0.290			10001.300	413.212	413.212
	13	10.472	14.720	0.290			10002.900	411.474	411.474
	14	10.568	15.197	0.290			10001.300	413.108	413.108
	15	10.728	15.520	0.290			10000.500	414.520	414.520
	16	10.709	15.889	0.290			10000.300	415.013	415.013
	17	10.804	16.367	0.290			10000.400	418.522	418.522
	18	9.837	9.556	0.290			10000.000	416.705	416.705
	19	9.926	9.944	0.290			10000.100	415.605	415.605
	20	10.015	10.330	0.290			10005.400	409.594	409.594
	21	10.106	10.729	0.290			10000.100	415.680	415.680
	22	10.201	11.144	0.290			10000.000	416.431	416.431
	23	10.296	11.561	0.290			10002.500	421.586	421.586
	24	10.392	11.981	0.290			10009.600	426.298	426.298
	25	10.489	12.406	0.290			10006.200	424.430	424.430
	26	10.588	12.837	0.290			10010.700	426.834	426.834
	27	10.686	13.268	0.290			10000.100	417.510	417.510

Figure 3.8: Table 2.2 *Elastic Foundation Coefficients*

In this table, the values of the elastic foundation coefficients are displayed for every finite element. Those are sorted by surface numbers and finite element numbers.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

Subsequent to the element number and the element coordinates X, Y, Z, the initial values of the *Horizontal springs* $C_{1,x}$ and $C_{1,y}$ are shown for every surface. If the spring constants have been defined in the RFEM dialog box *Edit Surface Elastic Foundation*, those values are taken over to RF-SOILIN.

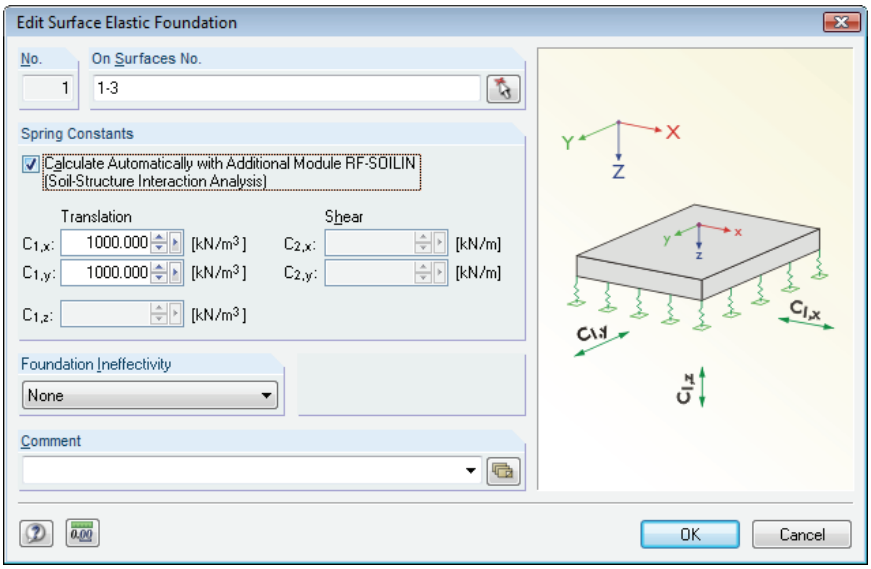


Figure 3.9: Dialog box *Edit Surface Elastic Foundation* in RFEM

If the spring constants are equal to zero in this dialog box, the value that has been set in the section *Miscellaneous* of the dialog box *RF-SOILIN - Details* is used.

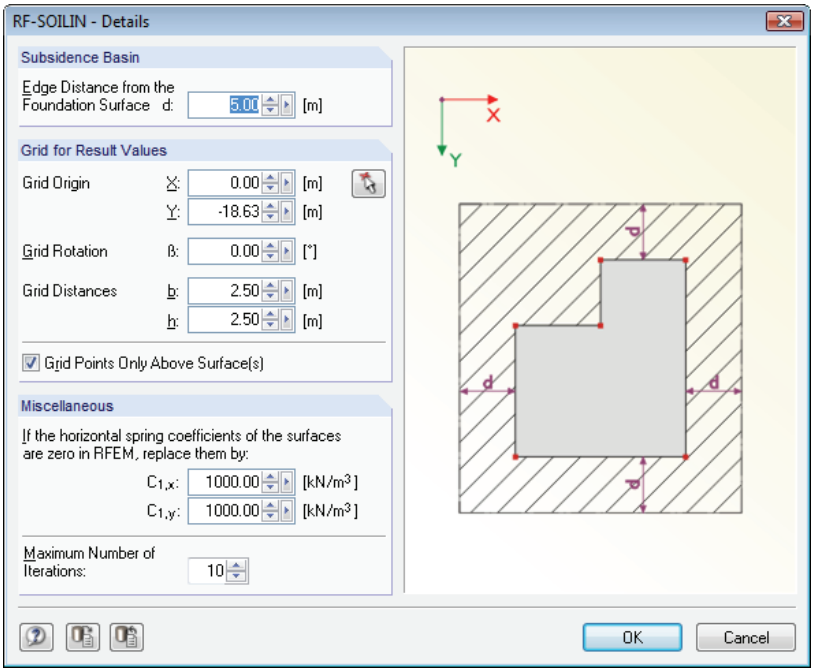


Figure 3.10: Dialog box *RF-SOILIN - Details*

Next to the horizontal spring constants, the *Elastic foundation coefficients* $C_{1,z}$, $C_{2,x}$, and $C_{2,y}$ calculated by RF-SOILIN are listed. The foundation model of RFEM and the significance of these three coefficients is described in detail in the RFEM manual [1], Chapter 5.9.

These elastic foundation coefficients represent the most important output of the module. They can be used for a realistic soil settlement analysis. These coefficients are always considered as constant for a every finite element.

3.5 Main Menus

The main menus contain all necessary commands for working with RF-SOILIN. You open the main menu by clicking its name or by pressing the [Alt] key together with the key whose letter is underlined in the menu name. For example, you press [Alt+F] to open the main menu *File*.

The commands contained in the main menus can also be called up by pressing their underlined letter on the keyboard.

3.5.1 File

Because the elastic foundation coefficients calculated in RF-SOILIN are an essential part of the structural input, you cannot define several design cases in RF-SOILIN (as it is possible in all other modules).

Delete Case

By using this command, you can delete the case calculated in RF-SOILIN.

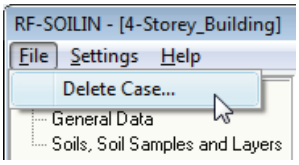


Figure 3.11: Deleting case in RF-SOILIN

When you click the command *Delete Case*, a message appears, warning you that all results are going to be deleted by deleting this case.

Figure 3.12: Warning in RF-SOILIN

3.5.2 Settings

Units and Decimal Places

By using this command, you can change the units and decimal places used in the module.

3.5.3 Help

This main menu calls up the online *Help*. The RF-SOILIN manual opens in the PDF format.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

4. Results

4.1 Screen Display

After the analysis, you can switch to the graphical display of results by using the [Graphic] button.

The following project navigator opens where you can set the options for the graphical display of results:

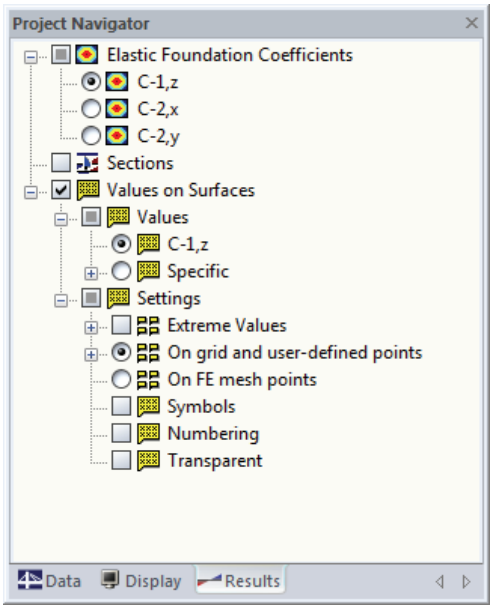


Figure 4.1: Project navigator for RF-SOILIN results

Using this navigator, the results can be evaluated graphically. You can find more information in Chapter 10 of the RFEM manual [1].

4.2 Printing

If you want to the print numerical results from RF-SOILIN, you have to return to the main program RFEM at first and open the printout report there. The functions to edit and layout the printout report are described in detail in the RFEM manual [1], Chapter 11.1.

In the dialog box *Printout Report Selection* in RFEM, more tabs become available when you click the item RF-SOILIN in the list *Program / Modules* on the left. In the section *Display* of the *Main selection* tab, you can choose whether you want to include *Input data* and/or *Results* in the printout report.

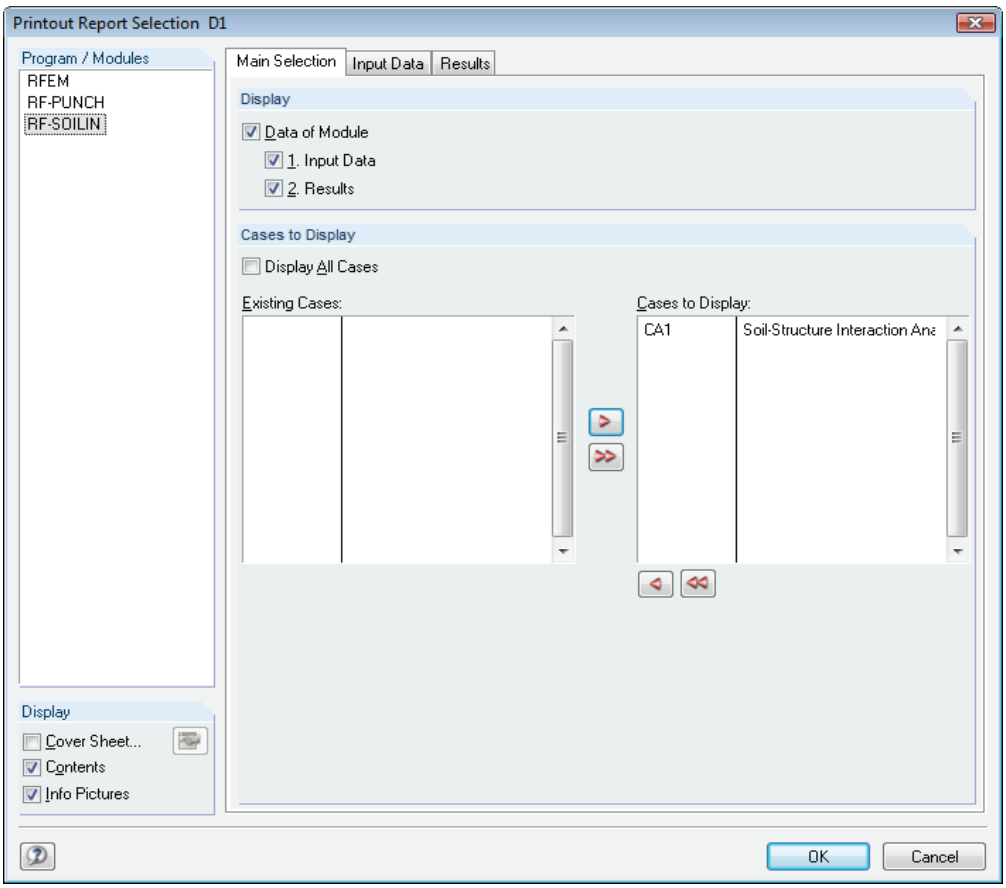


Figure 4.2: Dialog box *Printout Report Selection*

The selection of data from add-on modules is described in Chapter 11.1.3.4 of the RFEM manual [1].

All graphical results in RFEM work windows can either be printed directly by using the [Print] button or integrated to the printout report.

Program RF-SOILIN © 2010 by Ing.-Software Dlubal

A: Literature

[1] RFEM 4 manual, Dlubal Engineering Software, October 2009, (English).

[2] KOLÁŘ, V. et al.: Výpočet plošných a prostorových konstrukcí metodou konečných prvků, SNTL Prague, 1972 (Czech).

[3] KOLÁR, V. et al.: Berechnung von Flächen- und Raumtragwerken nach der Methode der finiten Elemente, Springer-Verlag Wien, New York, 1975, (German).

[4] KOLÁR, V., NEMEC, I.: Modeling of Soil-Structure Interaction, Elsevier Science Publishers, Amsterdam, co-published with Academica Prague, 1989, second revised edition, (English).

[5] PASTERNAK, P.L.: Osnovy novogo metoda rasčjota fundamentov na uprugom osnovanii pri pomosči dvuch koefficientov posteli, Gos. Isd. Stroj. i Arch., Moscow, 1954, (Russian).

[6] KOLÁŘ, V. Et al.: Kurs pro projektanty základových konstrukcí a zemních těles, p. 146 and following, Technical publishing house, Ostrava, 1983, (Czech).

[7] KOLÁR, V. - NEMEC, I.: Contact Stress and Settlement in the Structure-Soil Interface. Paper by Czech Academy of Sciences, No. 16., Academia Prague 1991, 160 pages, (English).

[8] KOLÁR, NEMEC, KANICKÝ: FEM – Principy a praxe (Czech).

[9] EN 1997 Geotechnical Design (Eurocode 7) (English).

[10] DIN 4019 Baugrund; Setzungsberechnungen. Published in 1979 and 1981 (German).

[11] CSN 73 1001 Zakládání staveb. Základová půda pod plošnými základy. Published in 1987 (Czech).