Engineering Software
Dlubal

Version
May 2010

Add-on Module

RF-STAGES

FEA Calculations considering Different Stages

Program Description

Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 1770
E-mail: info@dlubal.com
Web: www.dlubal.com

Contents

	Contents	Page
1.	**Introduction**	**4**
1.1	Add-on Module RF-STAGES	4
1.2	RF-STAGES Team	5
1.3	Using the Manual	6
1.4	Open the Add-on Module	6
2.	**Basics**	**8**
2.1	Calculation without Stage Analysis	8
2.2	Calculation with Stage Analysis	9
3.	**Input Data**	**10**
3.1	General Data	10
3.2	Members	11
3.3	Nodal Supports	12
3.4	Loads	13
3.5	Temporary Loads	15
3.6	Results in the Add-on Module	16
3.7	Export of Results	16
4.	**Printout**	**17**
4.1	Printout Report	17
4.2	Print RF-STAGES Graphics	19
5.	**General Functions**	**20**
6.	**Calculation Example**	**21**
6.1	Input	21
6.2	Results	25
A	**Literature**	**27**
B	**Index**	**28**

1. Introduction

1.1 Add-on Module RF-STAGES

RF-STAGES is not a stand-alone program but an add-on module that is integrated in the graphical user interface of the main program RFEM.

RF-STAGES allows for analyzing the structure in different structural stages. First, enter the complete structure in RFEM. Then, define the individual structural stages in the add-on module. Finally, assign the relevant members, columns and loads to the stages. When you have entered all required data in the tables, you can calculate the stages.

All results can be displayed in the module. It is also possible to export the results. You can even create a superposition of the results and export it. Using other add-on modules, you can design the structure by taking into account the exported results and thus considering the construction progress.

The following functions provide for a smooth workflow in RF-STAGES:

- Easy structure modeling in the module
- Adding, removing and modifying members and supports
- Permanent as well as temporary loads
- Adjusting loads by means of wind and snow load generator
- Taking into account non-linear effects
- Export of single structural stages as well as superposition of all stages
- Clearly arranged results representation in the printout report

We hope you will enjoy working with RF-STAGES.

Your team from DLUBAL ENGINEERING SOFTWARE

Engineering Software
Dlubal

1.2 RF-STAGES Team

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

Program coordination

Dipl.-Ing. Georg Dlubal
Ing. Pavol Červeňák

Programming

Petr Zámorský
Ing. Jiří Buček

Program design, dialog figures and icons

Dipl.-Ing. Georg Dlubal
Petr Zámorský

Program supervision

Ing. Tomáš Ferencz
Wieland Götzler

Manual, help system and translation

Wieland Götzler
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ü Gundel Pietzcker

Technical support and quality management

Dipl.-Ing. Frank Faulstich
M. Eng. Dipl.-Ing. (FH) Walter Rustler
Wieland Götzler

1.3 Using the Manual

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

The descriptions in this manual follow the sequence of the module's input and results tables as well as their structure. The manual text 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.

The index at the end of the manual helps you to find specific terms.

1.4 Open the Add-on Module

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

Menu

To start the program in the menu bar,

point to **Others** on the **Additional Modules** menu, and then select **RF-STAGES**.

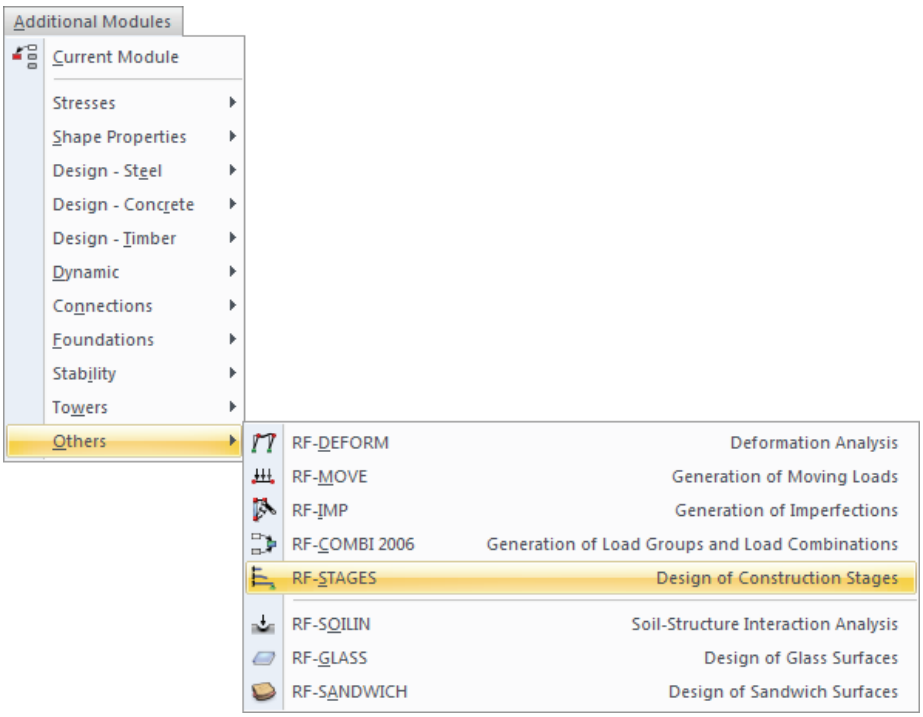


Figure 1.1: Menu: *Additional Modules → Others → RF-STAGES*

Navigator

To start RF-STAGES in the *Data* navigator,

select **RF-STAGES** in the **Additional Modules** folder.

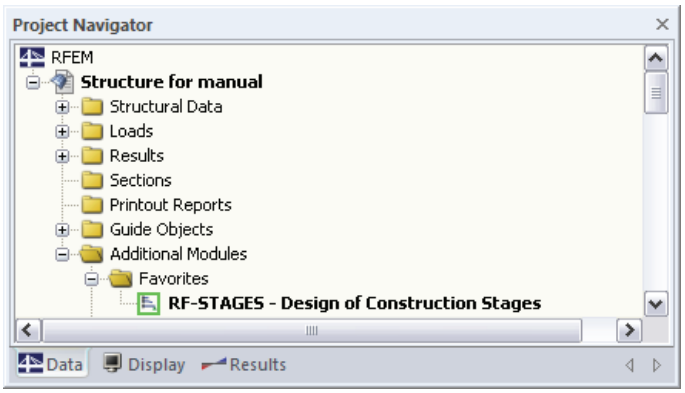


Figure 1.2: Data navigator: *Additional Modules* → *RF-STAGES*

Panel

In case RF-STAGES results are already available in the RFEM structure, you can set the corresponding RF-STAGES case in the load case list of the RFEM toolbar. By using the button [Results on/off], you can display the results for the relevant structural components.

When the results display is activated, the panel appears showing the button [RF-STAGES] which you can use to open the add-on module.

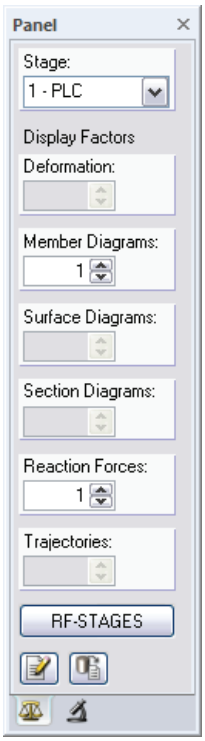


Figure 1.3: Panel button [RF-STAGES]

2. Basics

This chapter describes the basic calculation steps in RF-STAGES by presenting an example.

2.1 Calculation without Stage Analysis

RFEM 4 allows for the creation of complex 3D structures simulating structural conditions as realistic as possible. However, when you enter data in RFEM 4, you cannot take into account the construction progress. Thus, the results could be falsified considerably.

The following example describes a section of a high-rise building that has 9 stories with a ceiling height of 4 m for each room. The inner core which is represented by the elevator shaft is designed more solid and stiff than the outer columns. As a result, the outer columns are more compressed than the inner supports. This results in a displacement that is increasing with the rising height of the construction. The ceilings run the risk to "stick" to the inner supports. As a consequence, the supporting moment is increasing while the moment of span is decreasing.

However, such conditions do not reflect the real structural behavior of the present structure because the columns' deformations are compensated during the construction progress. Moreover, as a design result, the supporting components of the ceiling are undersized in the top floor spans and oversized at the inner supports. Therefore, it is necessary to develop a more accurate and realistic model.

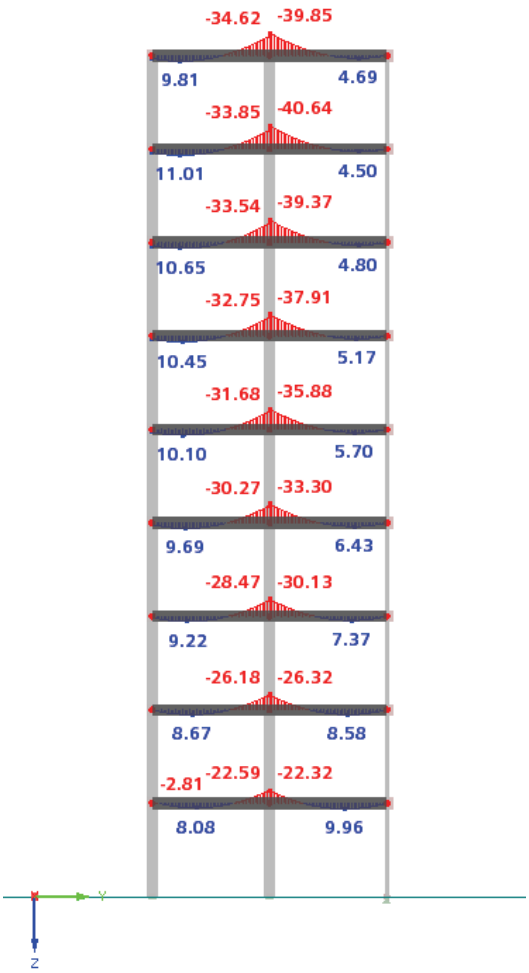


Figure 2.1: Moments M_y in load case self-weight without stage analysis

2.2 Calculation with Stage Analysis

By using the add-on module RF-STAGES, you are now able to calculate different construction stages in order to represent the real structural behavior of a building more accurately. The calculation is carried out according to the large deformation analysis (Newton-Raphson) and follows the procedure described below:

1. Calculation of deformations and internal forces of a stage in the non-deformed state
2. Application of the subsequent (non-deformed) stage to the previous, deformed stage and new calculation

Due to this algorithm, the deformations are compensated during the construction progress. In the results you can see that the supporting moment is increasing until half of the building's height has been completed. The reason is the following: At the inner supports, the floor beams still "stick" but this time chronologically shifted. Thus, when taking into account the different structural stages, the maximum moment in this model won't occur in the top section of the building, as described in chapter 2.1, but in the middle of the structure. The location where the maximum moment occurs is especially significant for the design. This is the reason why it is so important to analyze the structure in RF-STAGES.

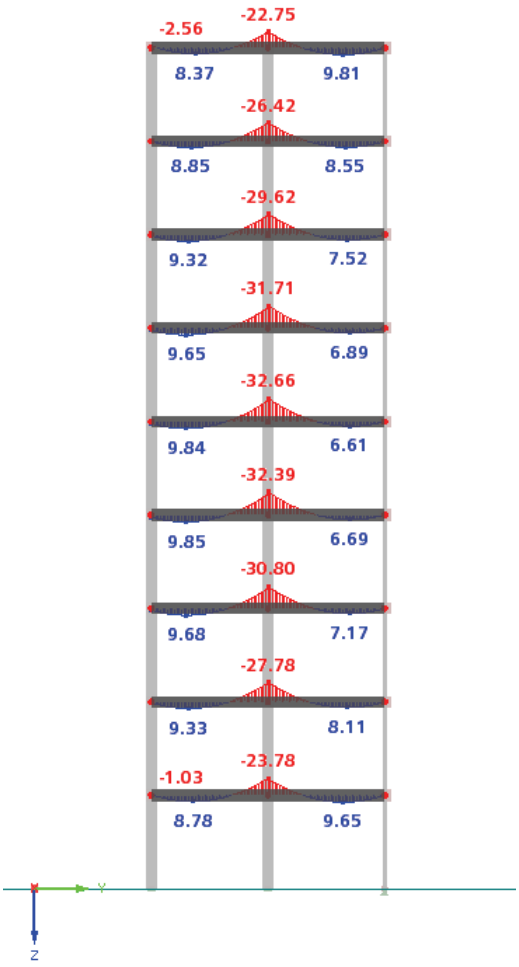
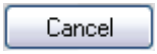


Figure 2.2: Moments M_y in load case self-weight with stage analysis

3. Input Data



Remark: Before you start working in RF-STAGES, you must create the structure as well as the corresponding load cases and groups by considering the sequence of construction!

To select a table, click the corresponding entry in the RF-STAGES 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. To access the Help, use the [Help] button.

3.1 General Data

In table 1.1 *General Data*, you define how many stages you want to analyze. In addition, it is possible to assign a description or a comment to the corresponding stage.

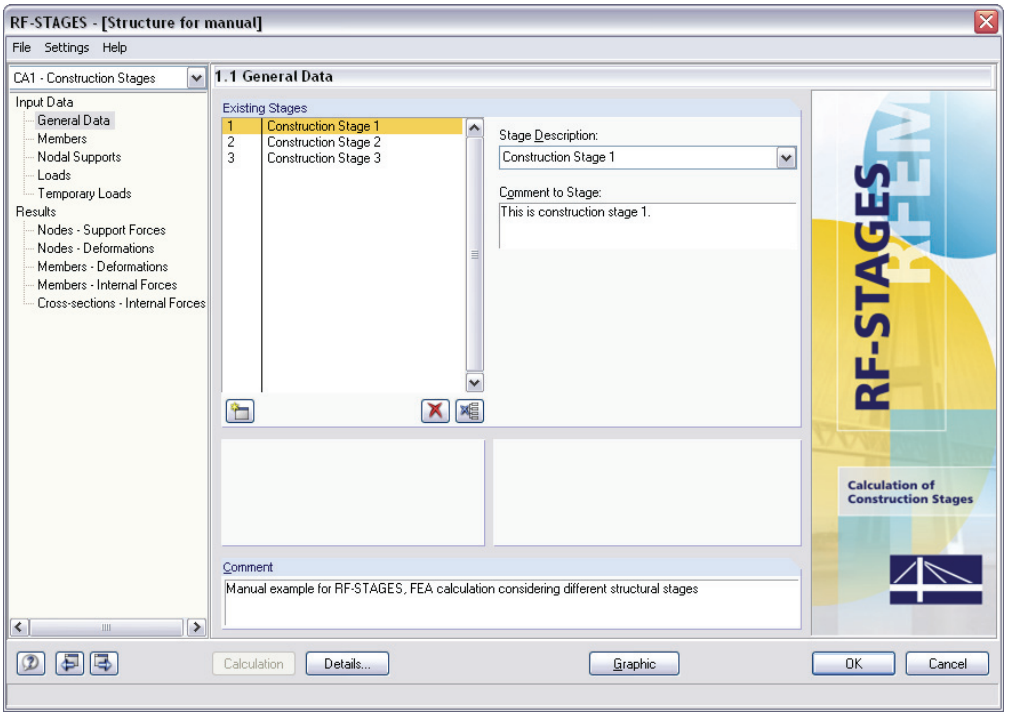


Figure 3.1: Table 1.1 *General Data*

The buttons in the table section *Existing Stages* are reserved for the following functions:

Button	Function
	Add a new stage
	Delete the selected stage
	Delete all stages

Table 3.1: Buttons

3.2 Members

In table 1.2 *Members*, you assign available members to the stages. In addition, you can modify and remove members of previous stages. Select the members graphically by using the [Pick] function or define them directly by entering the relevant member number in the input field *List of Members*.

When a member has already been added to a stage, it is not possible to add it to one of the subsequent stages. Available members from previous stages are not displayed in the *List of Members*, but it is possible to select them for modification or removal.

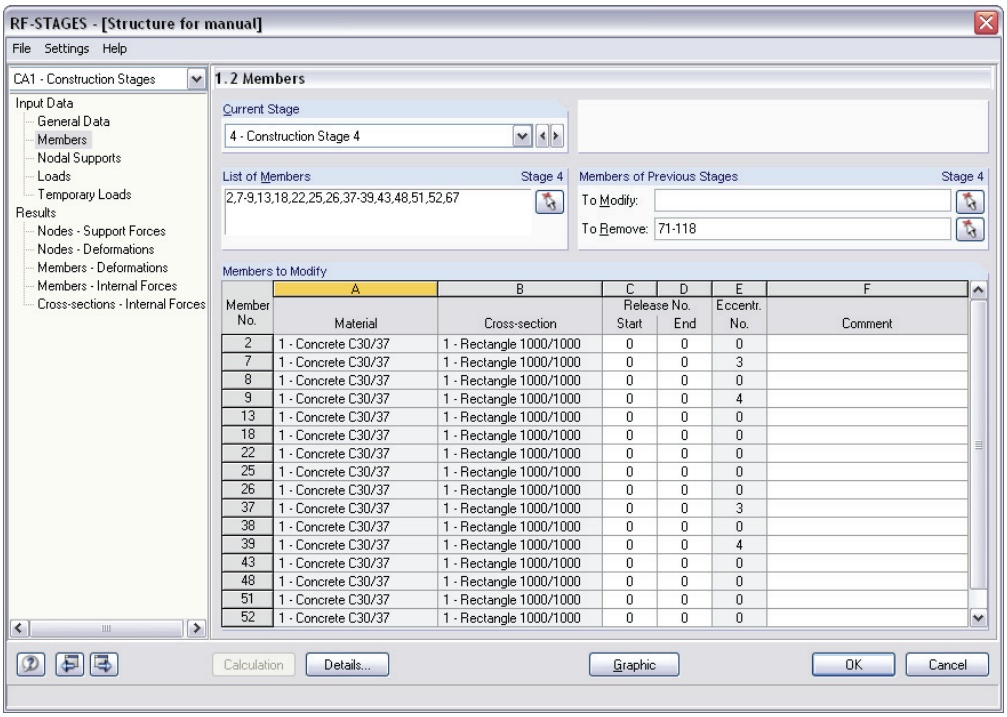


Figure 3.2: Table 1.2 *Members*

Add a member

To add members graphically, use the [Pick] button. To add them numerically, enter the corresponding member number into the *List of Members*. The added members will appear as entries in the list.

Remove a member

If you want to remove a member from a previous stage, select the member by using the [Pick] button or enter the member number directly into the input field *To Remove*.

Modify a member

To modify a member, select the member in the graphic by using the [Pick] button or enter the corresponding member number into the input field *To Modify*. The selected member will appear in the list *Members to Modify* where you can modify its member releases.

3.3 Nodal Supports

In table 1.3 *Nodal Supports*, you assign available nodal supports to the individual stages. It is possible to select nodal supports of previous stages to modify or remove them. To enter nodal supports graphically, use the [Pick] button. You can also define them directly by entering the nodal support number in the relevant list.

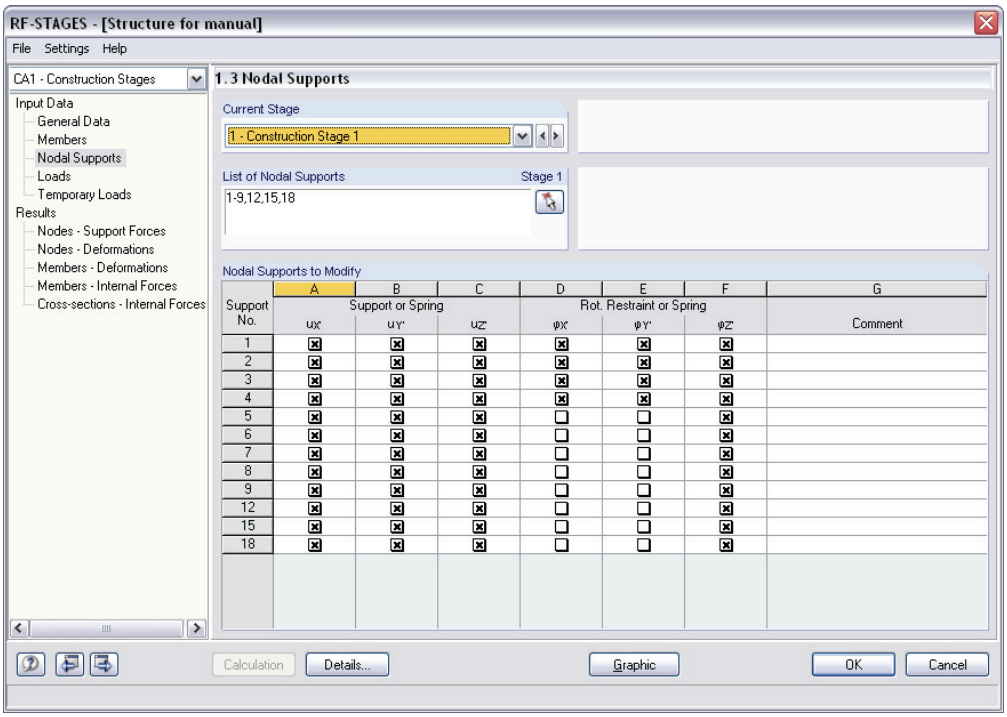


Figure 3.3: Table 1.3 *Nodal Supports*

Add a nodal support

To add nodal supports graphically, use the [Pick] button. To add them numerically, enter the corresponding support number into the *List of Nodal Supports*. The added supports will appear as entries in the list *Nodal Supports to Modify*.

Remove a nodal support

If you want to remove a nodal support from a previous stage, select the support by using the [Pick] button or enter the corresponding support number directly into the input field *To Remove*.

Modify a nodal support

To modify a nodal support, select the nodal support in the graphic by using the [Pick] button or enter the corresponding support number into the input field *To Modify*. The selected nodal support will appear additionally in the list *Nodal Supports to Modify* where you can modify its support conditions.

3.4 Loads

In table 1.4 *Loads*, you assign available load cases and groups to the table sections *Permanent Loads* or *Temporary Loads*. Once a permanent load is added to a stage, it will appear in all subsequent stages as well. In contrast, temporary loads affect only the stage to which they have been added. Each stage must have at least one permanent load case or group, whereas a temporary load case or group is not mandatory. A load case or group can be assigned only once. It is not possible to assign the same load case or group to several stages.

A defined load in RF-STAGES only applies to the members and nodes that have been added to the selected or to a previous stage. If a defined load of a stage affects nodes or members that are not available in such a stage, the load won't have any effect on the corresponding structural elements. This also applies if the relevant nodes or members are added to a subsequent stage later. Thus, it is necessary to adjust the loads for the individual stages.

Self-weight of structure

You can easily assign the structure's self-weight by ticking the check box *Assign Self-weight to All Stages*. By using this function, the program checks if the self-weight of the defined members is available as permanent load in each stage. If no self-weight is available, a permanent load case containing the self-weight will be created by the module. When the function is activated, the factor displayed in the *Details* dialog box will be decisive for the calculation (see Figure 3.5). This also means that an already defined self-weight will be replaced by this factor.

Details...

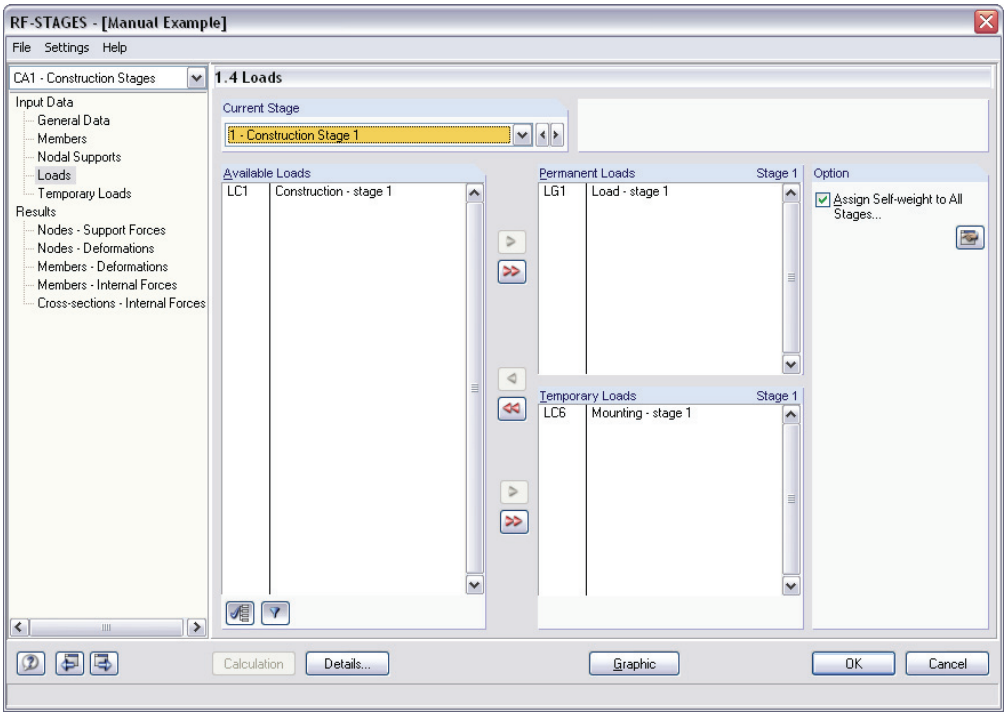
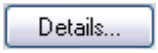


Figure 3.4: Table 1.4 *Loads*

Calculation

When you have assigned all data properly, use the [Calculation] button to start the calculation.

Details



In the *Details* dialog box, it is possible to specify the calculation settings. For example, you can define the factor by which the self-weight is assigned automatically to the different stages. You can also reduce the stiffness by the factor γ_M or divide the results back by a LC factor.

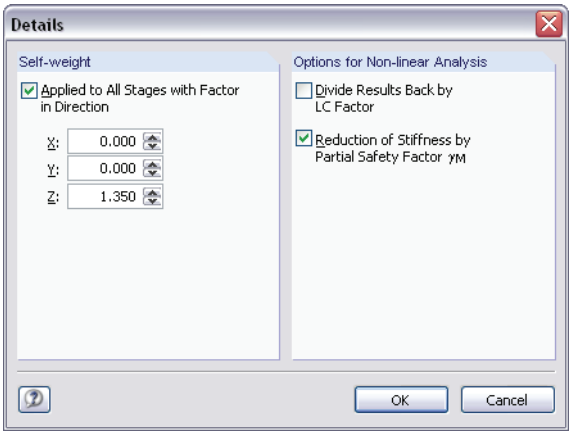


Figure 3.5: Dialog box *Details*

3.5 Temporary Loads

In case you have defined some load cases or groups as *Temporary Loads* in table 1.4, you can use table 1.5 *Temporary Loads* to combine them according to defined rules.

There is no difference between load combinations in RF-STAGES and RFEM. Therefore, the same principles are applied: It is recommended to do without the use of load combinations if non-linearities exist (tension members, failing supports etc.). The results of the individual load combinations are determined on the structure that is deformed by the permanent loads of the respective stage according to the linear static analysis. For the data export to RFEM, RF-STAGES will add the results of the temporary loads to the results of the permanent loads. This means that a load combination must be created even if only one load case or group was defined as temporary load.

By means of such load combinations it is possible to determine the governing internal forces for different structures from the temporary loads on the structure, for example crane positions.

When you have completed the creation of load combinations, start the calculation by clicking the [Calculation] button.

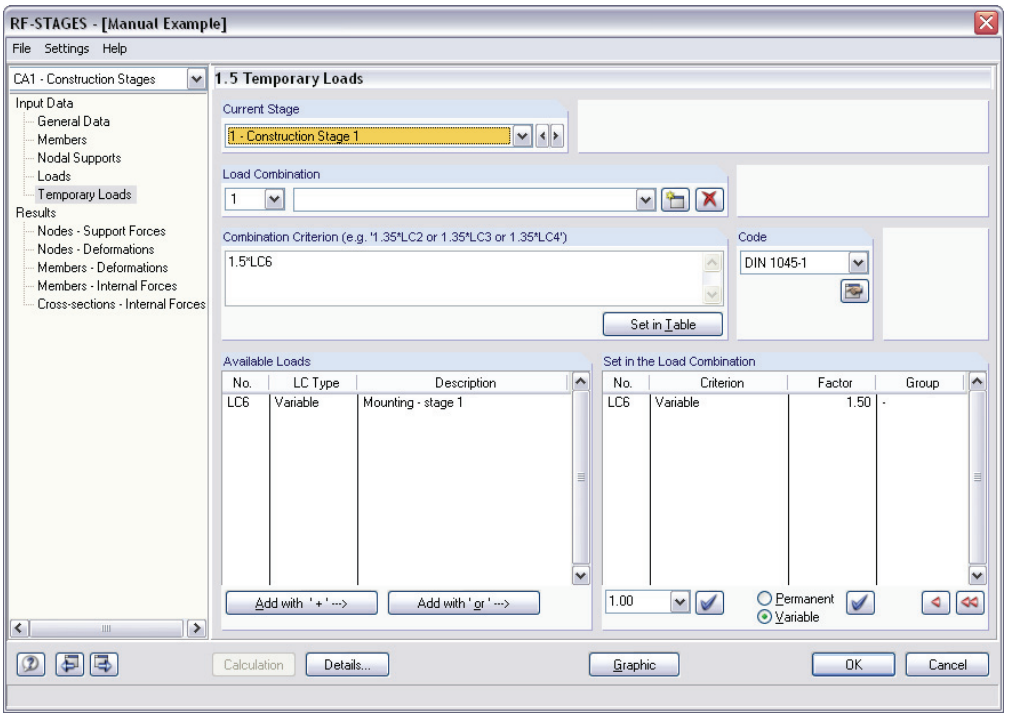


Figure 3.6: Table 1.5 *Temporary Loads*

3.6 Results in the Add-on Module

When the calculation has been completed successfully, the module shows the results. Use the corresponding buttons to page easily through the different stages. In the results, you may also select the envelope and display the respective maximum and minimum values of all stages.

Finally, you can export the results to [MS Excel]. The results can also be displayed on the structural model in the RFEM work window by using the [Graphic] button while the add-on module remains open. In this way, you can find particular results locations in the structure easily. It is also possible to select a node by using the [Pick] button in order to display its results in the results table. The colored reference scale can be deactivated.

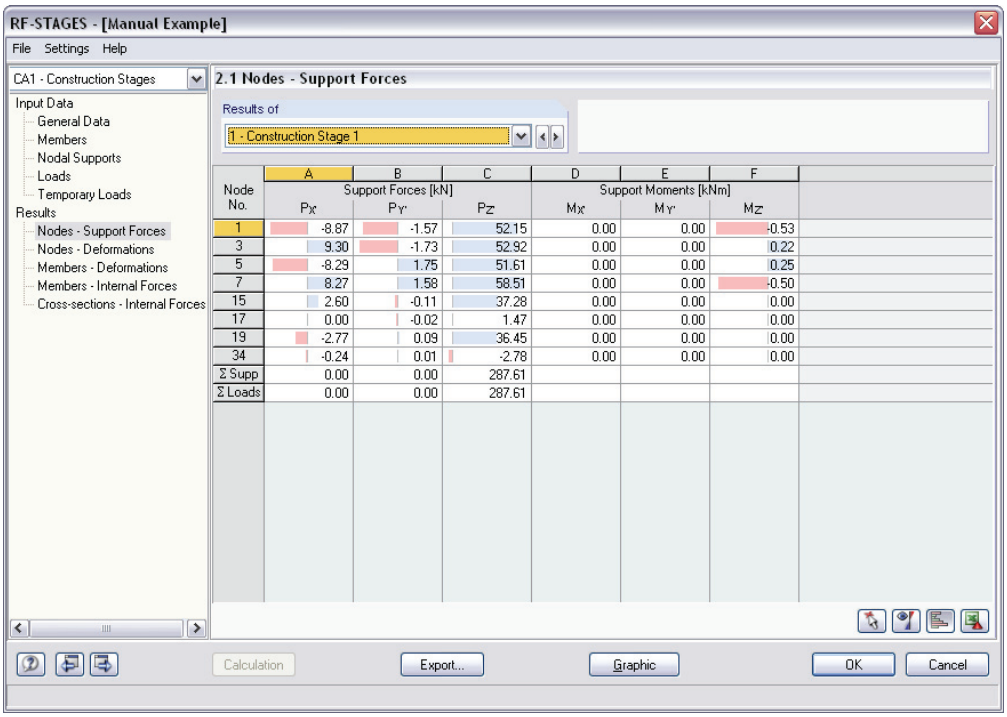
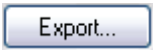


Figure 3.7: *Results* in the module

3.7 Export of Results



To export the results as load combinations, use the [Export] button. In this way, they can be used in other add-on modules, for example when you want to design a structure. By selecting the exported results in the load case list, you can directly display the results in the graphical representation of the structure.

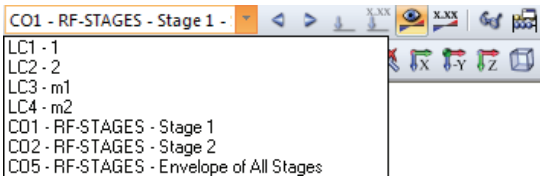


Figure 3.8: Exported results

4. Printout

4.1 Printout Report

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

For complex structural systems, it is recommended to create several short reports instead of one comprehensive printout because they can be generated more quickly. Furthermore, the computing capacity can be reduced in this way.

The printout report is described in detail in chapter 11 of the RFEM manual. For more detailed information about the selection of module data, see chapter 11.1.3.4.

Selection options

In the *Global Selection* tab of the *Printout Report Selection* dialog box, you can decide if input data and results are included in the printout report. Furthermore, it is possible to select the individual stages for which you want to show the results.

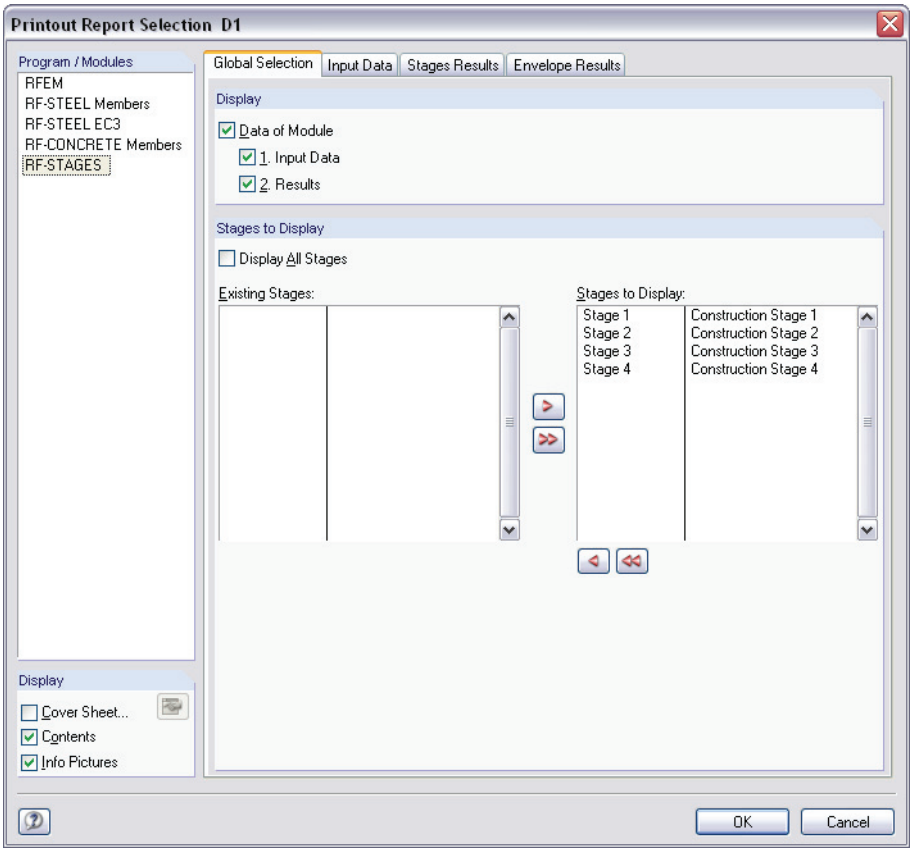


Figure 4.1: Dialog box *Printout Report Selection*, tab *Global Selection*

In the second tab *Input Data*, you determine the input data (general data, members and nodal supports) that you want to include in the individual stages.

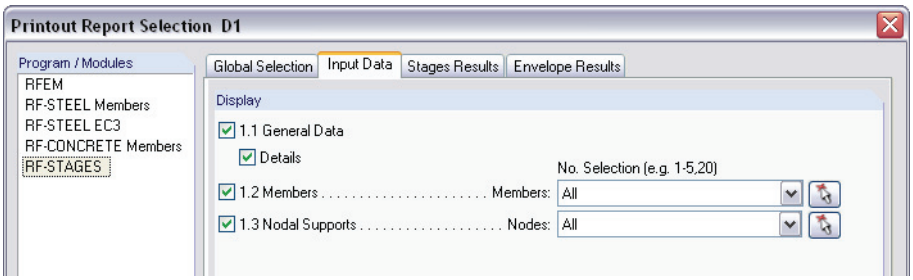


Figure 4.2: Dialog box *Printout Report Selection*, tab *Input Data*

In the two final tabs, you select the *Results* that you want to show in the individual stages. In addition, you can open the *To Display* windows for members and cross-sections to select internal forces and deformations individually.

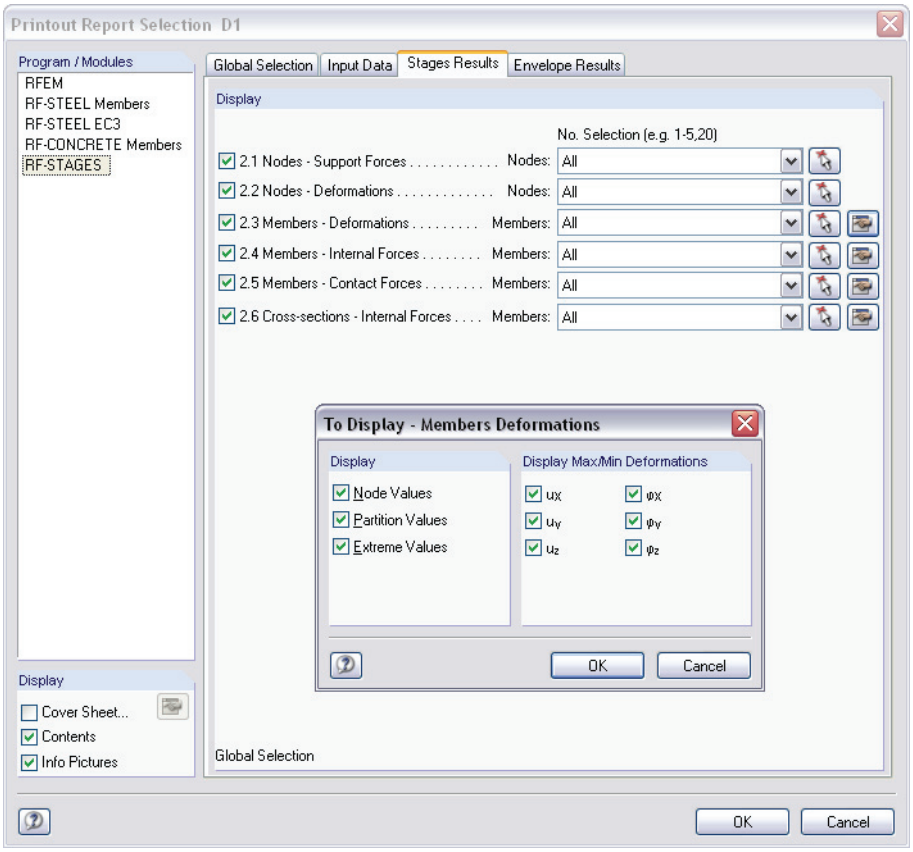


Figure 4.3: Dialog box *Printout Report Selection*, tab *Stages Results*

4.2 Print RF-STAGES Graphics

Results graphics can either be integrated into the printout report or directly sent to the printer. For more information, see chapter 11.2 in the RFEM manual.

Every picture that is displayed in the graphic window of the main program RFEM can be included into the printout report. In the same way, you can integrate result diagrams of members into the report by using the [Print] button.

To print the current RF-STAGES graphic in the RFEM window,

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

or use the toolbar button shown on the left.

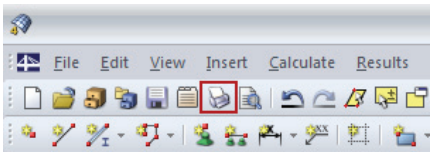


Figure 4.4: Button *Print* in the toolbar of the main window

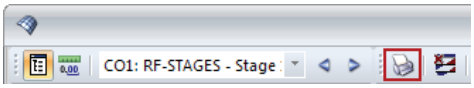


Figure 4.5: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog box opens:

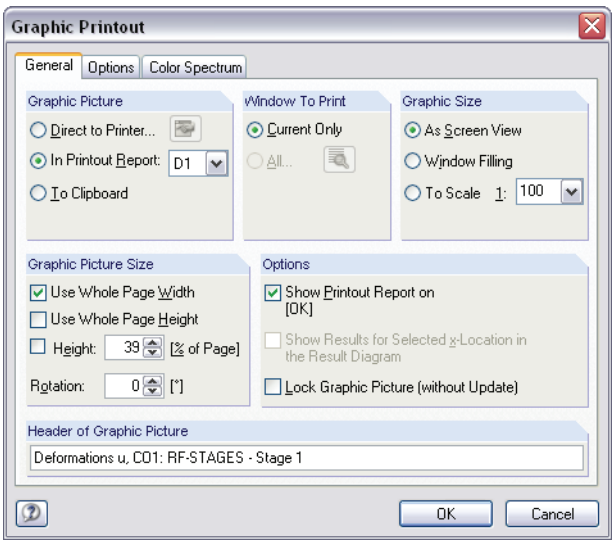


Figure 4.6: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in chapter 11.2.1 of the RFEM manual. The RFEM manual also describes the *Options* and *Color Spectrum* tab.

A graphic from RF-STAGES, that has been integrated in the printout report, can be moved anywhere within the report by using the drag-and-drop function. In addition, it is possible to edit the integrated graphics. Right-click the corresponding graphic in the navigator of the printout report and select *Properties* on the context menu. The dialog box *Graphic Printout* appears again where you can adjust the data appropriately.

5. General Functions

Rename a RF-STAGES case

To change the description of a case subsequently,

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

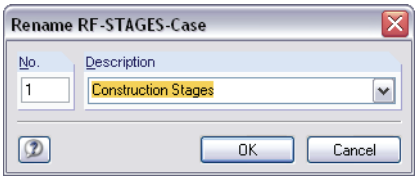


Figure 5.1: Dialog box *Rename RF-STAGES-Case*

Delete a RF-STAGES case

To delete a case,

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

Figure 5.2: Dialog box *Delete Cases*

6. Calculation Example

6.1 Input

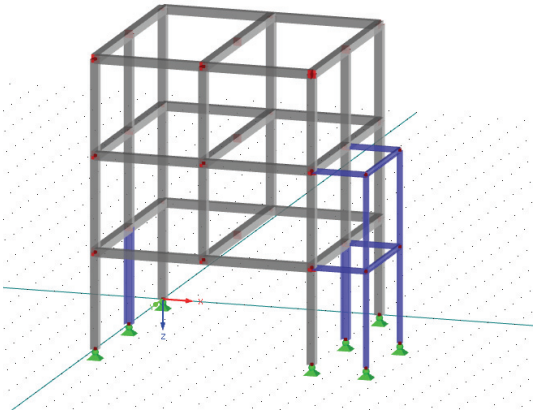


Figure 6.1: Reinforced concrete example

Before you can start to specify the structure in the module, it must be created in RFEM first, including load cases and load groups. The corresponding input data can be found in the folder *Examples*.

Assignment of model data in the module

Table 1.1 *General Data*

When you have opened the structure in RFEM, use the *Data* navigator to start the module. You can also start the program by pointing to *Others* on the *Additional Modules* menu and selecting *RF-STAGES*. Table 1.1 *General Data* appears.

In our example, the RF-STAGES case contains three stages as defined in the following table.

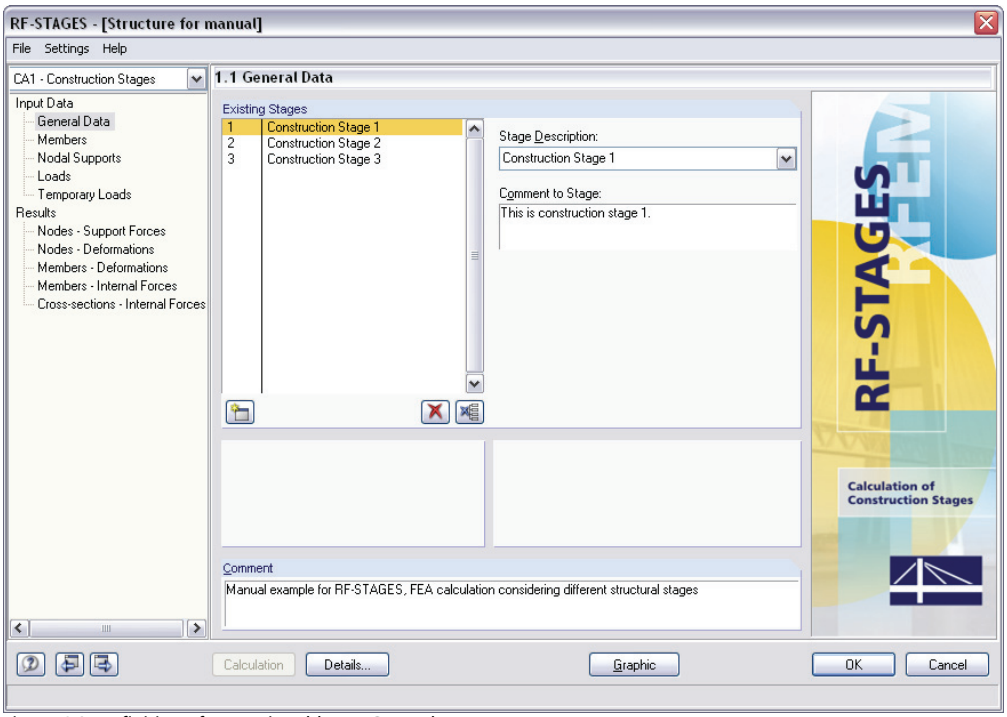


Figure 6.2: Definition of stages in table 1.1 *General Data*

Table 1.2 *Members*

In the subsequent table 1.2 *Members*, assign the available members to the individual stages. For stage 1, enter the members 1-12, 46, 48-53, 59, 60. To check the input, look at the members highlighted in red in the structure.

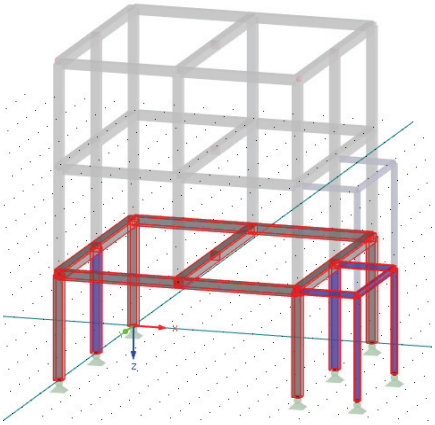


Figure 6.3: Members of stage 1

In stage 2, remove the two construction supports HEB-300 (members 46 and 48) and add the members 13-24, 37, 42-44, 54-58, 61, 62. Again, you can check the input graphically.

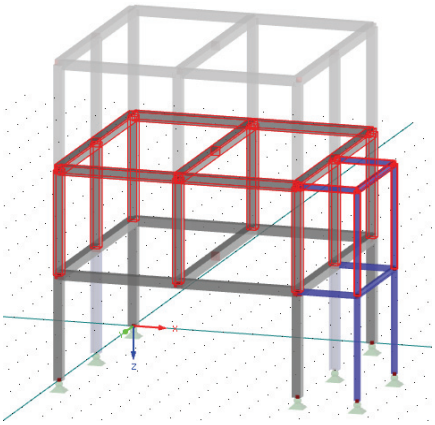


Figure 6.4: Members of stage 2

In stage 3, remove the members of the scaffold (IPE 200, members 49-58).

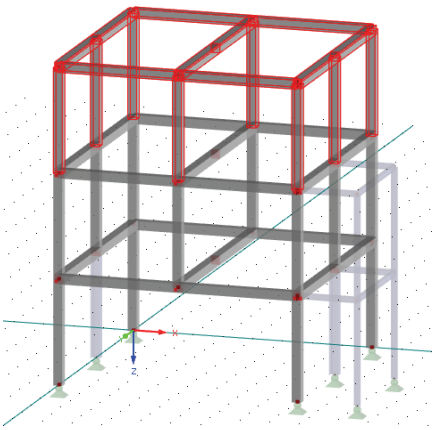


Figure 6.5: Members of stage 3

Now, the input of members is complete and you can change to the next table.

Table 1.3 *Nodal Supports*

Add all nodal supports to stage 1. The supports will be highlighted in red in the structure in the background. Due to the removal of the construction support members and the scaffold, you do not need to remove nodal supports in the subsequent stages. Thus, the input for nodal supports is completed.

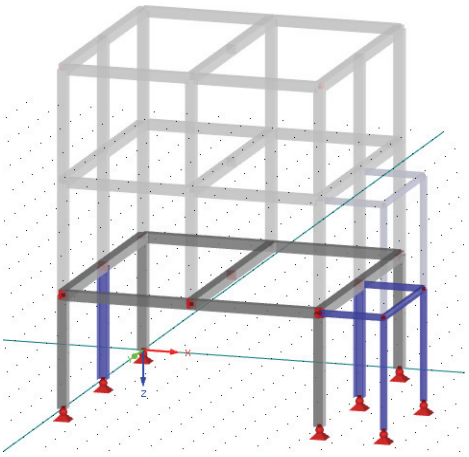


Figure 6.6: Nodal supports of stage 1

Table 1.4 *Loads*

In table 1.4 *Loads*, assign the load cases or groups to the individual stages by classifying them as *Permanent* or *Temporary Loads*. A temporary load only occurs in one stage, whereas a permanent load is taken into account in each stage. To activate the self-weight of the structure in all stages, tick the check box *Assign Self-weight to All Stages*.

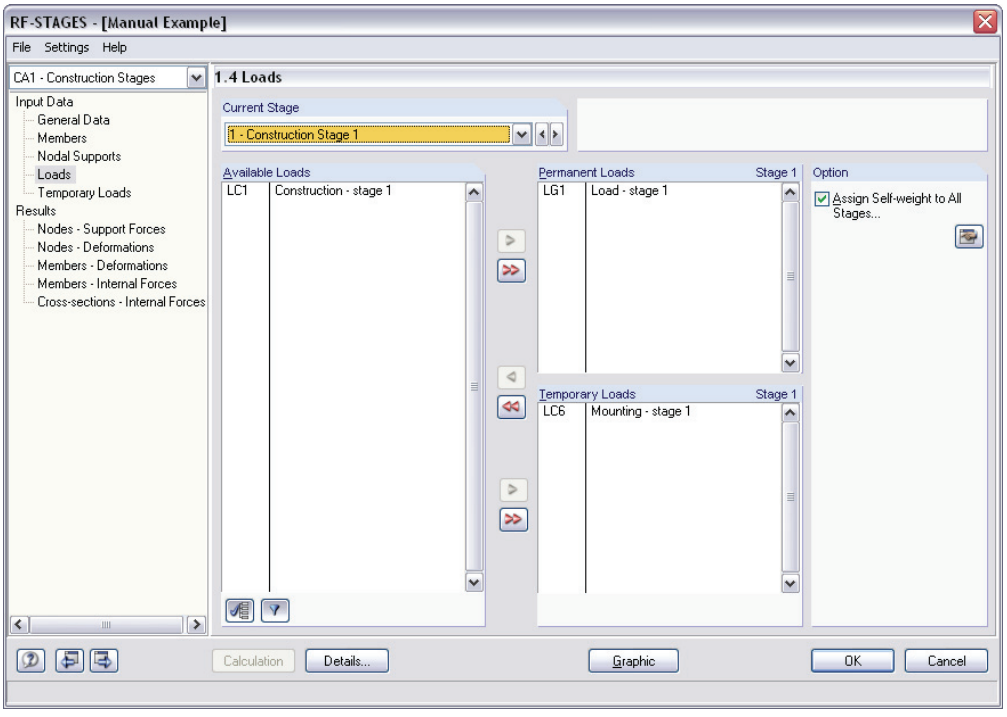


Figure 6.7: Assignment of loads in table 1.4 *Loads*

In stage 1 and 2, construction loads are affecting the scaffold and the floor beams. In addition, the framework is already in use while the construction is still in progress (starting with stage 2). Therefore, a *live load* has been created for stage 2 and 3 respectively.

Details...

The two live loads, together with the permanent loads (construction), are combined as load groups including partial safety factors. The load groups are assigned to the respective stage. By ticking the check box *Assign Self-weight to All Stages* the load case self-weight is assigned to each stage with the factor defined in the *Details* dialog box.

When you have assigned the load cases and groups, you can switch to the final input table 1.5.

Table 1.5 *Temporary Loads*

Table 1.5 *Temporary Loads* is used to define superpositions of temporary load cases and groups. The loads are superimposed as load combinations according to the specified scheme. If non-linearities exist, for example in case of tension members or failure of support, it is recommended to define no superposition because it would lead to erroneous results. In such a case, it is better to do without temporary load cases or groups.

As non-linearities do not occur in the example structure, superpositions can be created. The internal forces from the temporary loads are added to the permanent loads. The program creates a load combination for stage 1 and for stage 2 where the respective temporary load case is added with the factor 1.5.

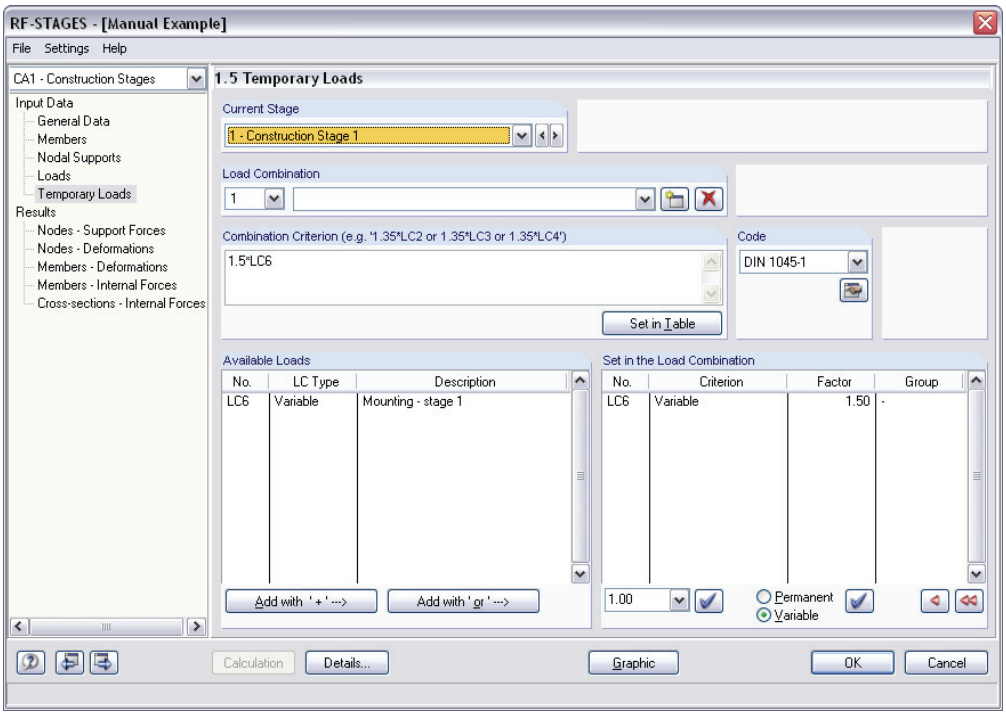


Figure 6.8: Table 1.5 *Temporary Loads*, creating combinations

Calculation

When you have defined all required load combinations, start the calculation by clicking the [Calculation] button.

6.2 Results

Subsequent to the calculation, the module displays the results of the permanent and temporary loads.

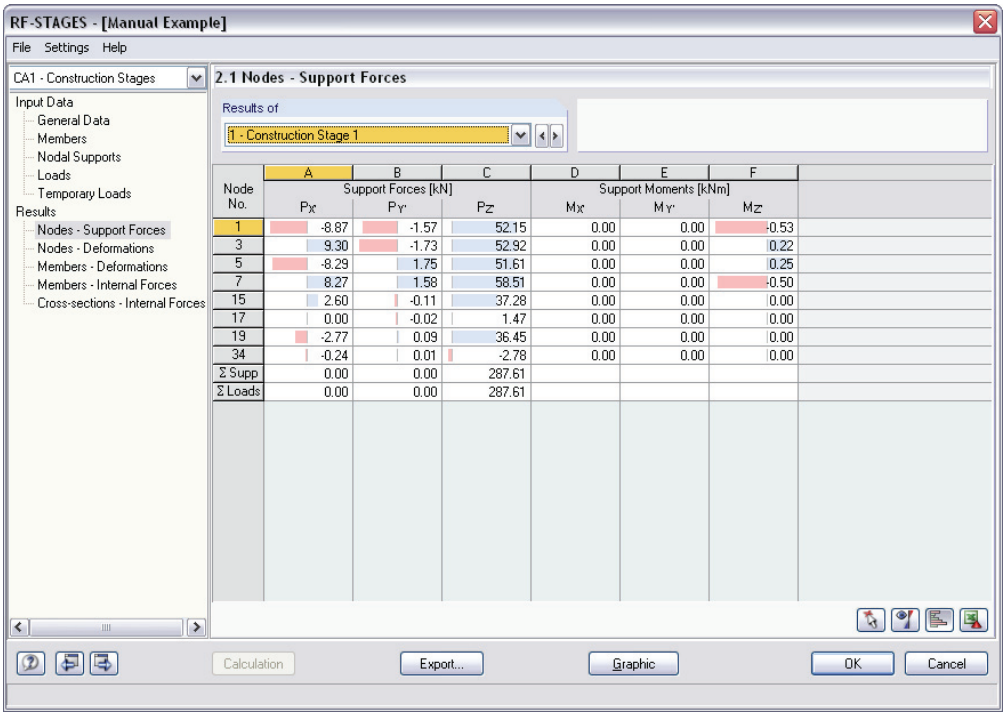


Figure 6.9: Results evaluation in the module

Graphic

RF-STAGES

The results can also be displayed directly on the structure in RFEM by clicking the [Graphic] button. To set specific results, use the panel. To return to the module, click the panel button [RF-STAGES].

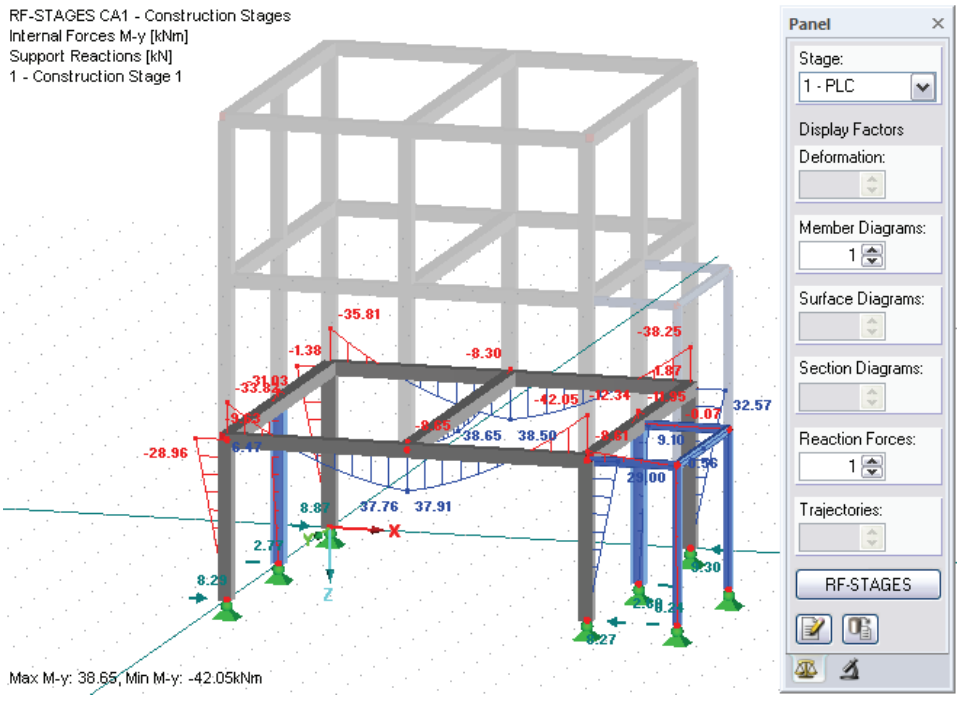


Figure 6.10: Results display on the RFEM structure

Export

Export...

If you want to use the results in further design modules, for example RF-CONCRETE Members or RF-STEEL Members, you have to export the RF-STAGES results first. Use the module's [Export] function that is available in all results tables.

The results from RF-STAGES will be exported as load combinations according to the combination scheme defined in table 1.5. Then, they can be used in other modules.

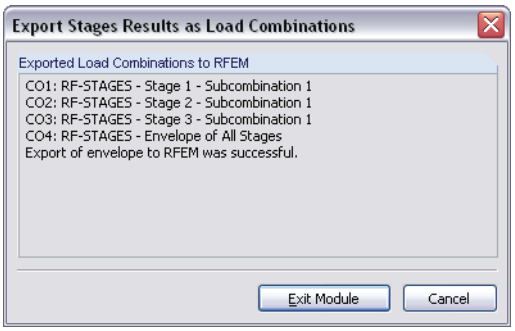


Figure 6.11: Export of results

Design with exported results

Now, by taking into account the exported results, you can design the structure concerning its reinforced concrete and steel parts. Select the RF-STAGES load combination that is relevant for the design and define the corresponding design specifications. In the present example, several cases have already been created for the design in some additional design modules.

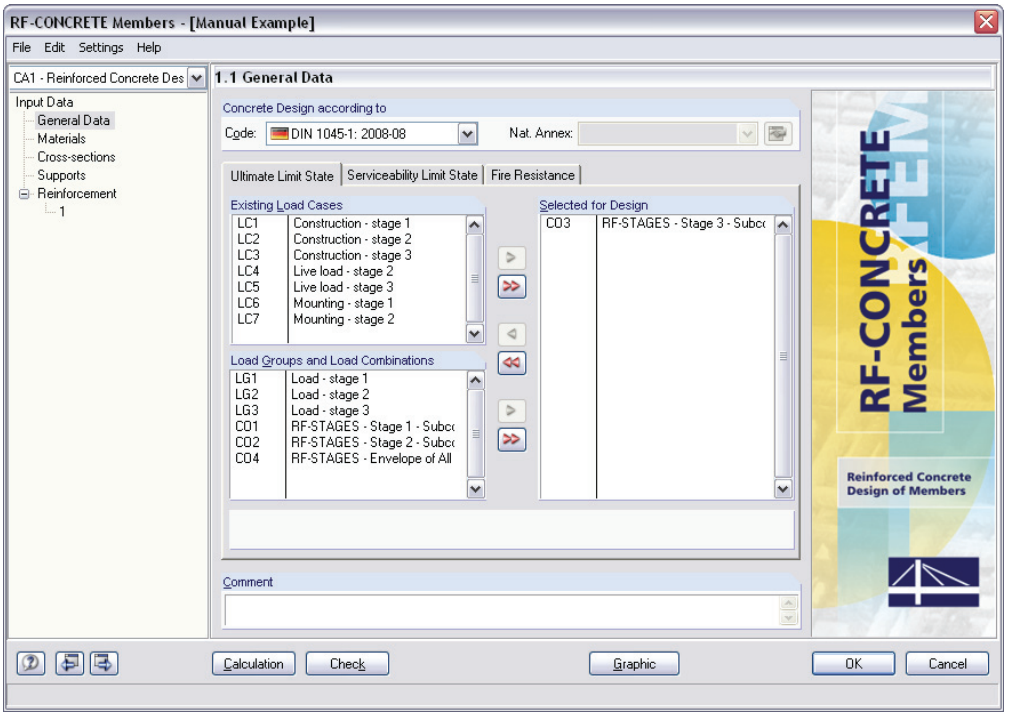


Figure 6.12: Design of RF-STAGES CO with RF-CONCRETE Members

A Literature

[1] BISCHOFF, M., BLETZINGER, K.-U.: Statik am Gesamtmodell – Möglichkeiten und Ansprüche, Fachtagung Baustatik-Baupraxis 10, Karlsruhe 2008

[2] BARTH C., RUSTLER, W.: Baustatik-Praxis mit FEM

B Index

A

Add members 11

Add nodal supports 12

B

Basics 8

C

Calculation 9

Calculation example 21

D

Design 26

Details 14

E

Export results 16

F

Functions 4

G

General data 10

I

Installation 6

L

Load 13

Load case self-weight 13

Loads 23

M

Members 11

Modify members 11

Modify nodal supports 12

N

Nodal supports 12

O

Open the program 6

P

Panel 7

Permanent loads 13

Print RF-STAGES graphic 19

Printout 17

Printout report 17

Q

Quit RF-STAGES 10

R

Remove members 11

Remove nodal supports 12

Results 15

Results in module 16

RF-STAGES case 20

S

Selecting tables 10

Selection options for printout 17

Stage analysis 8

Start program using menu 6

Start program using navigator 7

Start program using panel 7

T

Temporary loads 15, 24