Ing. Software
Dlubal

Version
May 2011

Add-on Module

RF-DEFORM

Deformation Analysis for Members and Sets of Members

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Add-on Module RF-DEFORM

The program system of the DLUBAL company already offers you powerful add-on modules to perform the stress design. The serviceability limit state design, however, can be decisive as well in structural analysis. Therefore, we provide the add-on module RF-DEFORM, a program used to analyze the maximum allowable deformations of members and continuous members.

RF-DEFORM is completely integrated in the main program RFEM. Thus, the add-on module is not only an integral part of the FEA program in its appearance. The results of the RF-DEFORM calculation can be included in the RFEM printout report. In this way, you can present the entire analysis very easily in a consistent form.

We hope you will enjoy working with RF-DEFORM.

Your team from ING.-SOFTWARE DLUBAL

1.2 RF-DEFORM Team

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

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem

Programming

Bc. Ondřej Šašinka
Dipl.-Ing. Georg Dlubal
Mgr. Petr Oulehle
Ing. Roman Svoboda
Dis. Jiří Šmerák
Lukáš Tůma

Program design and icons

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

Program supervision

Ing. Tomáš Ferencz
Dipl.-Ing. (FH) Matthias Entenmann
Ing. Ctirad Martinec

Manual, help system and translation

Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ü. Gundel Pietzcker

Technical support and quality management

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

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-DEFORM 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 text of the manual shows the described **buttons** in square brackets, for example [Pick]. 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.

At the end of the manual, you find the index. 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-DEFORM

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

Menu

To start the program in the menu bar,

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

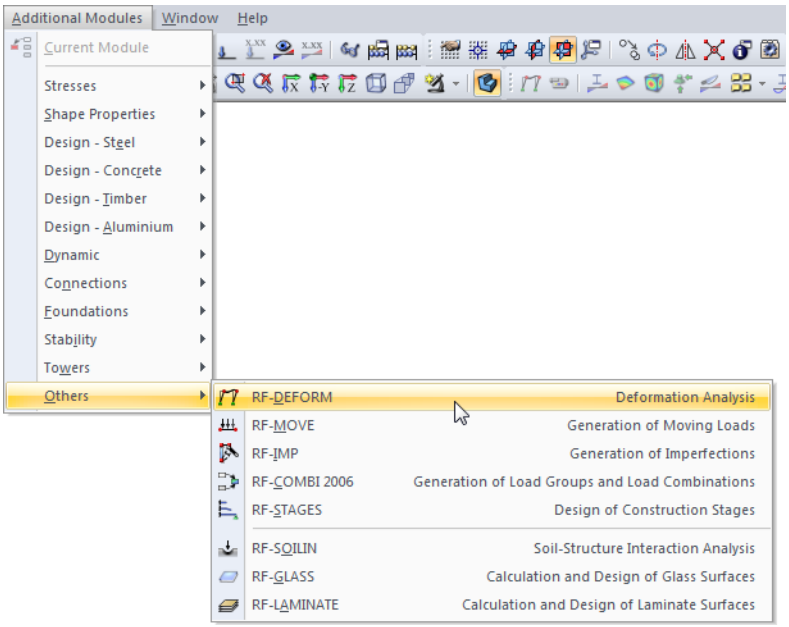


Figure 1.1: Menu *Additional Modules* → *Others* → *RF-DEFORM*

Navigator

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

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

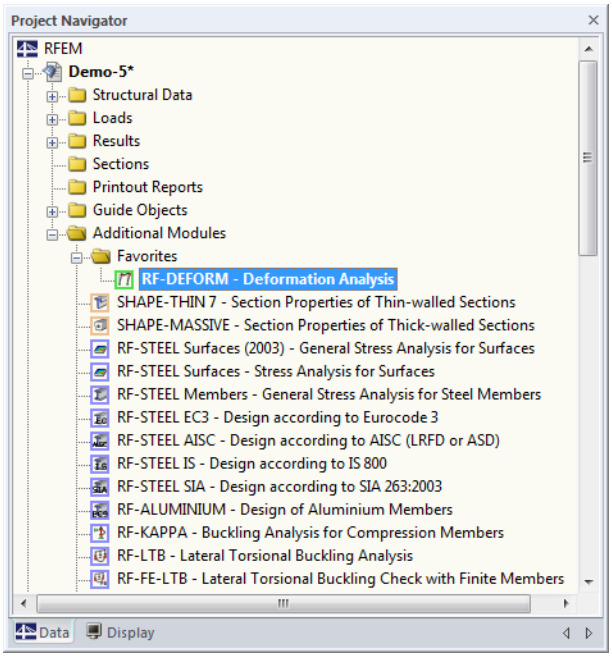


Figure 1.2: Data navigator: *Additional Modules* → *RF-DEFORM*

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 members and sets of members that you want to design.

When you have started the add-on module, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently. The pull-down list above the navigator contains the design cases that are already available (see chapter 6.1, page 15).

If you open RF-DEFORM in an RFEM structure for the first time, the module will import the already created load cases, groups and combinations automatically.

To select a table, click the corresponding entry in the RF-DEFORM 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 add-on module but without saving the data.

2.1 General Data

In table 1.1 *General Data*, you select the actions that you want to design.

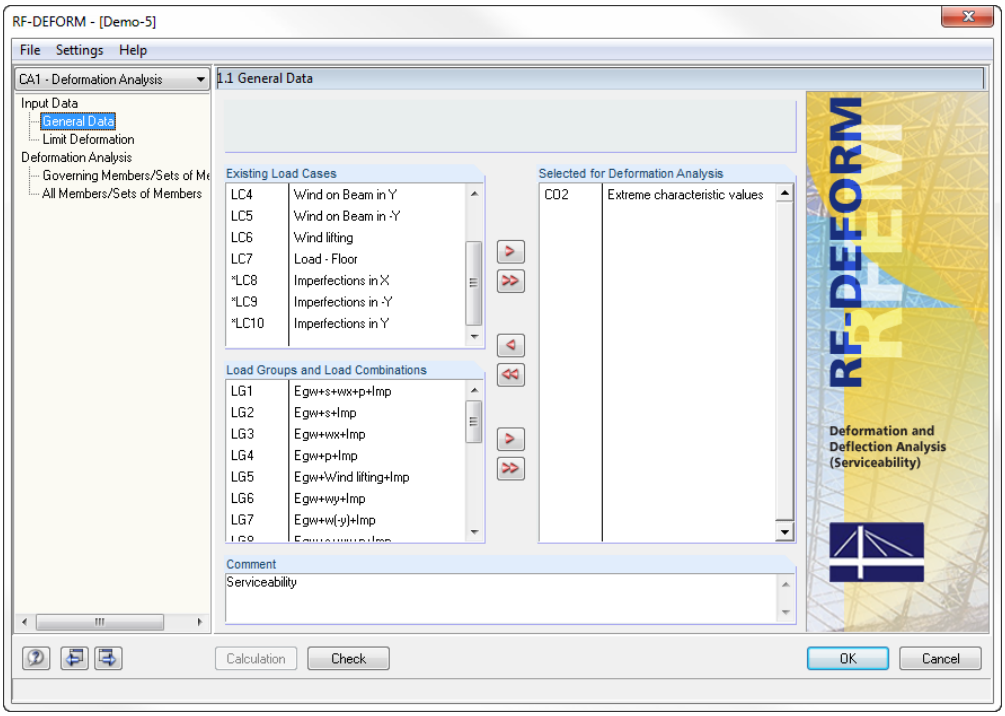


Figure 2.1: Table 1.1 *General Data*

Existing Load Cases / Load Groups and Load Combinations

These two table 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 actions or combinations to the list *Selected for Deformation Analysis* on the right. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].

Load cases that are marked by an asterisk (*), like load cases 8 to 10 in the figure above, cannot be designed. This may be the case when no loads are defined or when the load case contains only imperfections.

Selected for Deformation Analysis

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. To empty the complete list, use the button [◀◀].

Comment

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

2.2 Limit Deformation

In this table which is subdivided into two parts you define the 1D objects that you want to design. The upper table section defines the limit deformations of *Members*, the lower table section the deformations of *Sets of Members*. Except for this difference, both tables offer you the same input options. Therefore, they are described together in the following.

To modify the units and decimal places of lengths and deformations, select **Units and Decimal Places** in the module's **Settings** menu (see chapter Figure 6.5, page 17).

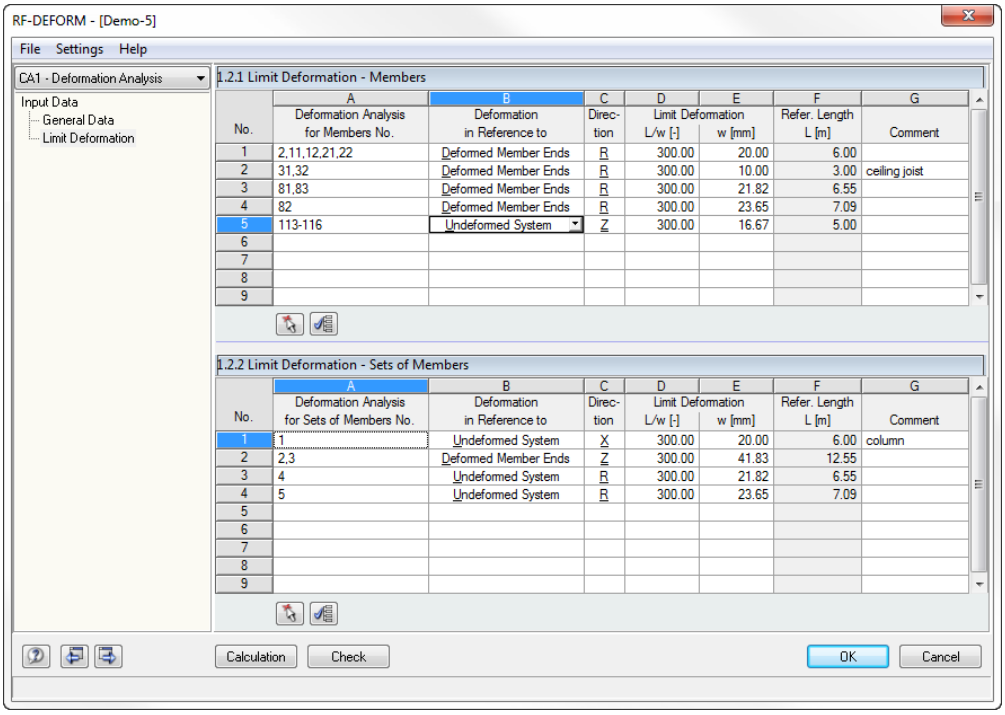


Figure 2.2: Table 1.2 *Limit Deformation*

Deformation Analysis for Members / Sets of Members No.

In this table column, you specify the numbers of the members or sets of members that are relevant for the design. Enter the members and sets of members directly into the table or use the [Pick] button to select them graphically in the RFEM work window. When you select the objects graphically, RF-DEFORM summarizes all objects with identical lengths row by row (column F).

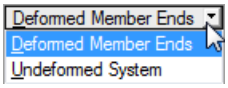
Use the button [Set All Members] or [Set All Sets of Members] to select all objects for the design and to list them all in table column A.

When you select sets of members, please note the following:

- The sets of members must have already been defined in RFEM.
- Only sets of members of the type *Continuous Members* can be designed. It is not possible to analyze groups of members.

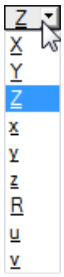
Deformation in Reference to

This table column determines if the results of the deformation analysis are set in relation to the *Undeformed System* or an imaginary connection line between member start and member end in the deformed system, thus to the *Deformed Member Ends*.



It is possible to specify the reference by using the list: Place the pointer in the corresponding field of the table column, and then click the button [▼] or use the function key [F7]. The list shown on the left opens, offering you both options.

Direction



In table column C, you define the direction of the deformation that you want to design. Again, you can use the list of the active table row to select the direction.

Symbol	Direction
X	Global axis X
Y	Global axis Y
Z	Global axis Z
x	Local member axis x (longitudinal axis)
y	Local member axis y ("strong" axis)
z	Local member axis z ("weak" axis)
R	Resultant force of the deformation
u	Strong local member axis u of unsymmetrical section
v	Weak local member axis v of unsymmetrical section

Table 2.1: Direction of deformation

Limit Deformation L/w and w

In table column D, you enter the value of the allowable limit deformation *L/w* directly. The ratio of length and deformation is preset with 300. If you modify the value, the deformation *w* indicated in table column E will be updated automatically when you confirm the table cell.

If a table row contains members or continuous members that are different in length, RF-DEFORM cannot display the limit deformation *w* as a result of *L/w* by a single value. In this case, the table field *w* remains empty.

However, it is possible to enter the allowable deformation *w* directly in column E. Then, RF-DEFORM will determine the ratio *L/w* automatically using this specification.

Refer. Length L

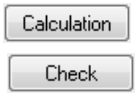
The lengths to which the limit deformation refers are preset, based on the lengths of the members and sets of members defined in RFEM. The values indicated in table column F cannot be modified.

In case a table row contains members or sets of members of different lengths, RF-DEFORM displays the maximum and the minimum length in this column.

Comment

You can describe each table row by a comment for a clear management of data.

3. Calculation



To start the calculation, click the [Calculation] button that is available in both input tables.

Before you start the calculation, it is recommended to [Check] the input data by using the button shown on the left.

Figure 3.1: Check result

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

You can start the calculation of RF-DEFORM results also out of the RFEM user interface. All 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.

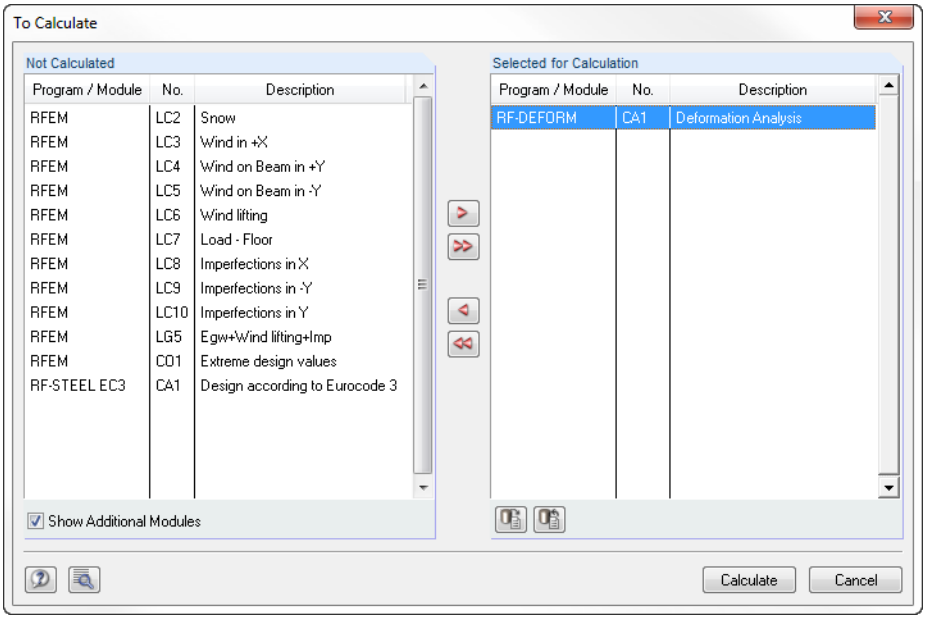
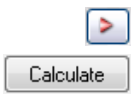


Figure 3.2: RFEM dialog box *To Calculate*

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



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

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

4. Results

Immediately after the calculation, table 2.1 *Deformation Analysis - Governing Members / Sets of Members* appears, which is also subdivided into two parts. The individual designs are listed in the two results tables 2.1 and 2.2 that can be selected by means of the RF-DEFORM navigator. 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-DEFORM.

4.1 Deformation Analysis - Governing Members/Sets of Members

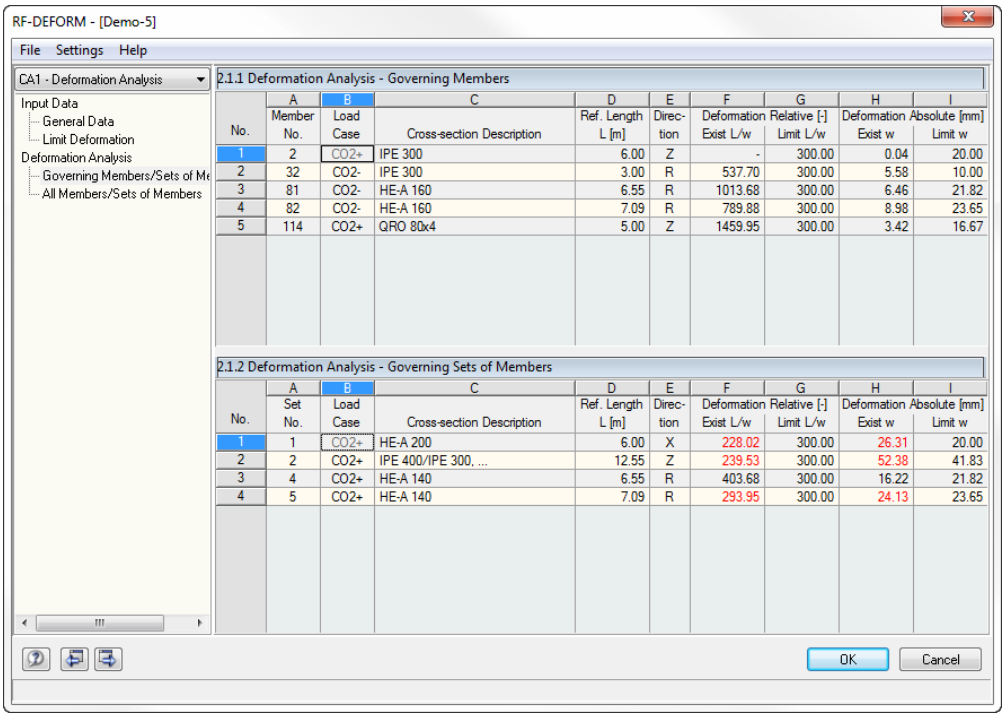


Figure 4.1: Table 2.1 *Deformation Analysis - Governing Members/Sets of Members*

This two-part table shows the maximum deformation ratios resulting from the governing loads. The results are sorted by members and sets of members.

Member / Set No.

For each input row of table 1.2, RF-DEFORM displays the number of the member or set of members that has the smallest ratio of *Exist L/w* in relation to *Limit L/w*.

Load Case

Table column B shows the action whose loads produce the most unfavorable deformation ratio.

Cross-section Description

The cross-section descriptions of the governing members or sets of members are indicated so that it is easier to check the corresponding data.

Ref. Length L

This column shows you the values from table column F of table 1.2.

Direction

The direction of the deformation to be designed is specified in table column C of table 1.2. The settings are displayed again so that you can check the entries.

Deformation Relative L/w

Table columns F and G show respectively the ratios *Exist L/w* and *Limit L/w*. The value of the allowed limit deformation *L/w* is determined in column D of table 1.2.

If *Exist L/w* < *Limit L/w*, the design fails and the corresponding values are highlighted in red.

Deformation Absolute w

The two final table columns inform you about the absolute values of the existing and the allowable deformations. In this way, you can compare the maximum deformations *Exist w* directly with the limit deformations *Limit w*.

4.2 Deformation Analysis - All Members/Sets of Members

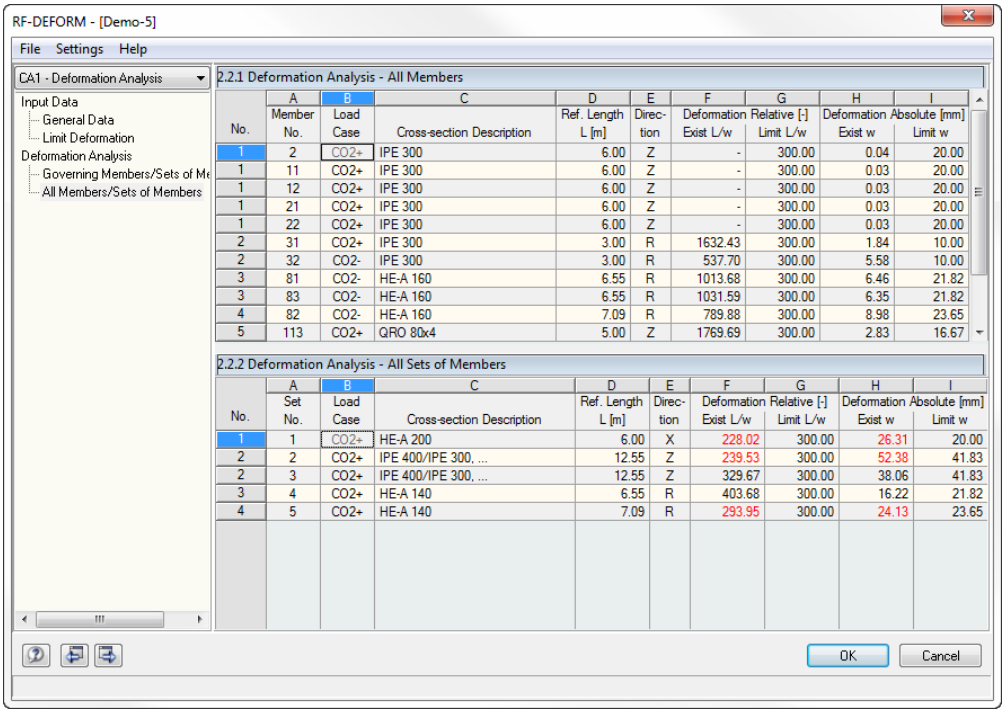


Figure 4.2: Table 2.2 *Deformation Analysis - All Members/Sets of Members*

This table is also subdivided into two parts listing the results for each individual member and set of members that has been selected in table 1.2 for the deformation analysis.

Details on the table columns can be found in the previous chapter 4.1.

5. Printout

The creation of printouts is similar to the procedure in RFEM. First, the program generates a printout report for the RF-DEFORM results. Graphics and descriptions can be added. In addition, you can use the print preview to specify the results of the deformation analysis 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 report for RF-DEFORM, the 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.

All general selection options are available in order to print the design cases as well as input and results data of RF-DEFORM.

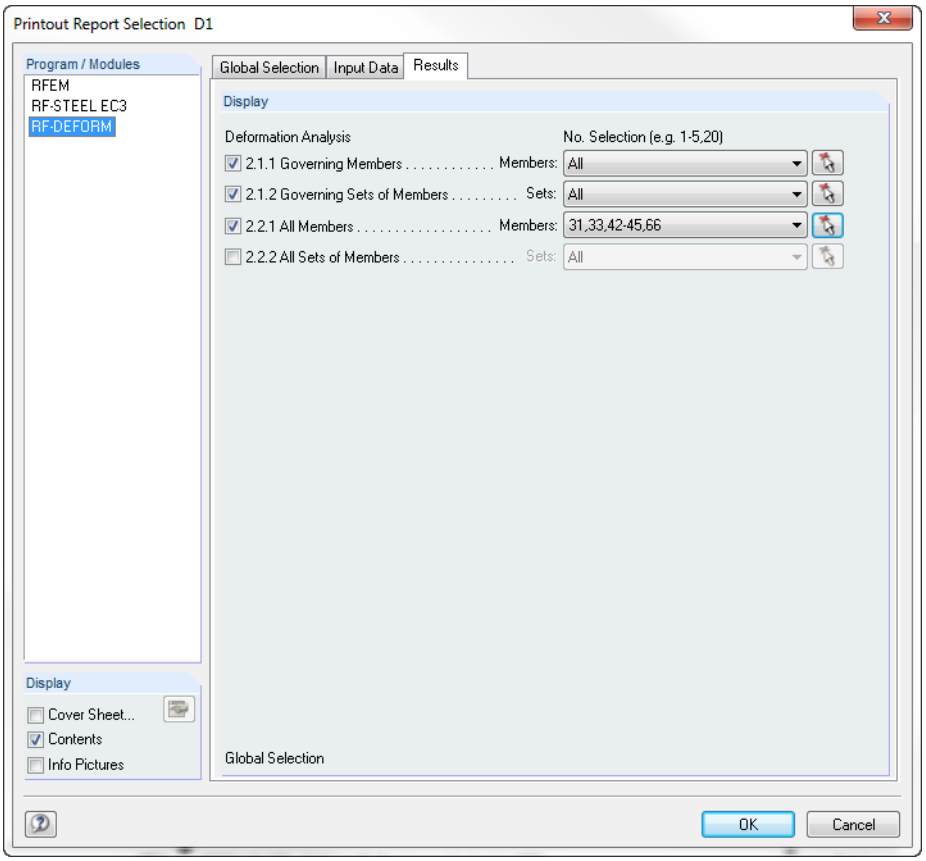


Figure 5.1: Printout report selection of RF-DEFORM results

6. General Functions

This chapter describes some menu functions as well as export options for the results of the deformation analysis.

6.1 RF-DEFORM Design Cases

Members and sets of members can be arranged in groups for separate design cases. In this way, you can for example define particular specifications for the limit deformations to apply them to structural component groups.

Create a new RF-DEFORM case

To create a new design case,

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

The following dialog box appears.

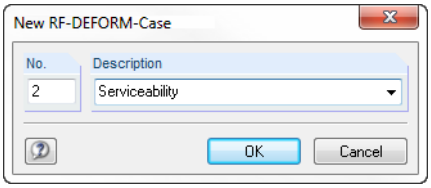


Figure 6.1: Dialog box *New RF-DEFORM-Case*

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 data.

Rename an RF-DEFORM case

To change the description of a design case subsequently,

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

The dialog box *Rename RF-DEFORM-Case* appears.

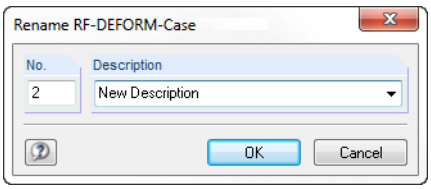


Figure 6.2: Dialog box *Rename RF-DEFORM-Case*

Copy an RF-DEFORM case

To copy the input data of the current design case,

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

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

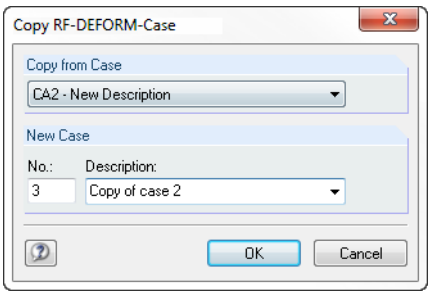


Figure 6.3: Dialog box *Copy RF-DEFORM-Case*

Delete an RF-DEFORM case

To delete a design case,

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

In the dialog box *Delete Cases*, you can select a design case in the *Available Cases* list to delete it by clicking [OK].

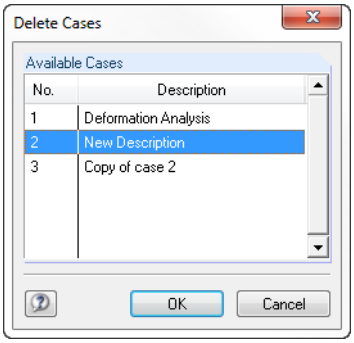


Figure 6.4: Dialog box *Delete Cases*

6.2 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 add-on module RF-DEFORM, 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 following dialog box opens, which you already know from RFEM. The add-on module RF-DEFORM is preset.

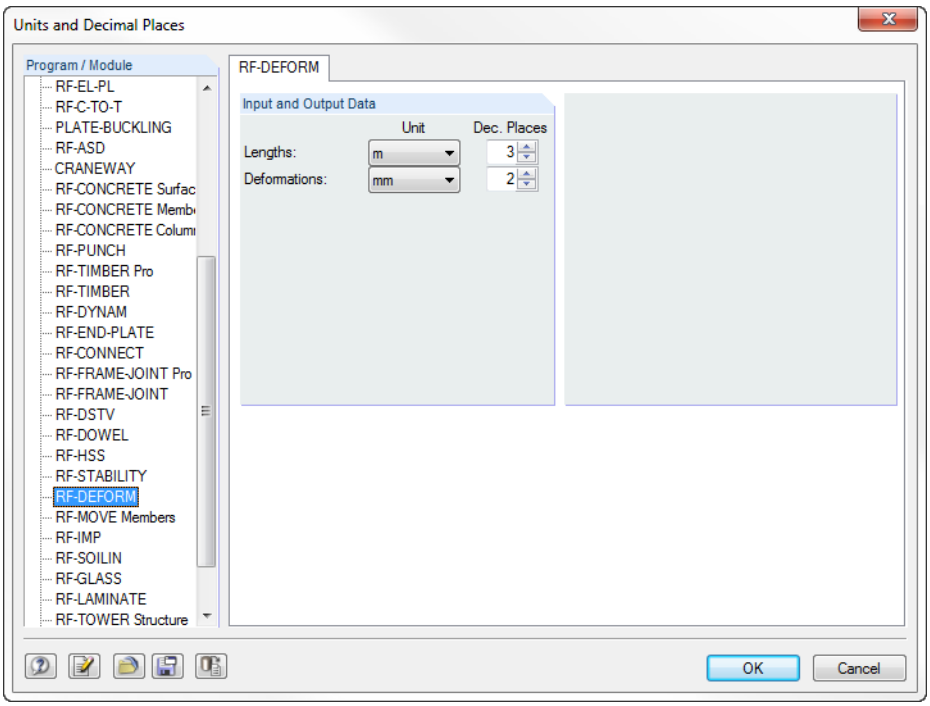


Figure 6.5: Dialog box *Units and Decimal Places*

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

6.3 Export of Results

The results of the deformation analysis can be provided for other programs. Several options are available.

Clipboard

To copy cells selected in the RF-DEFORM results tables 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 RF-DEFORM data can be printed into the global printout report (cf. chapter 5, page 14) 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-DEFORM 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-DEFORM add-on module.

The following export dialog box appears.

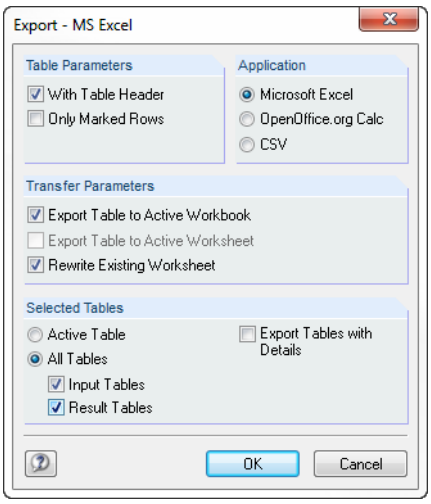
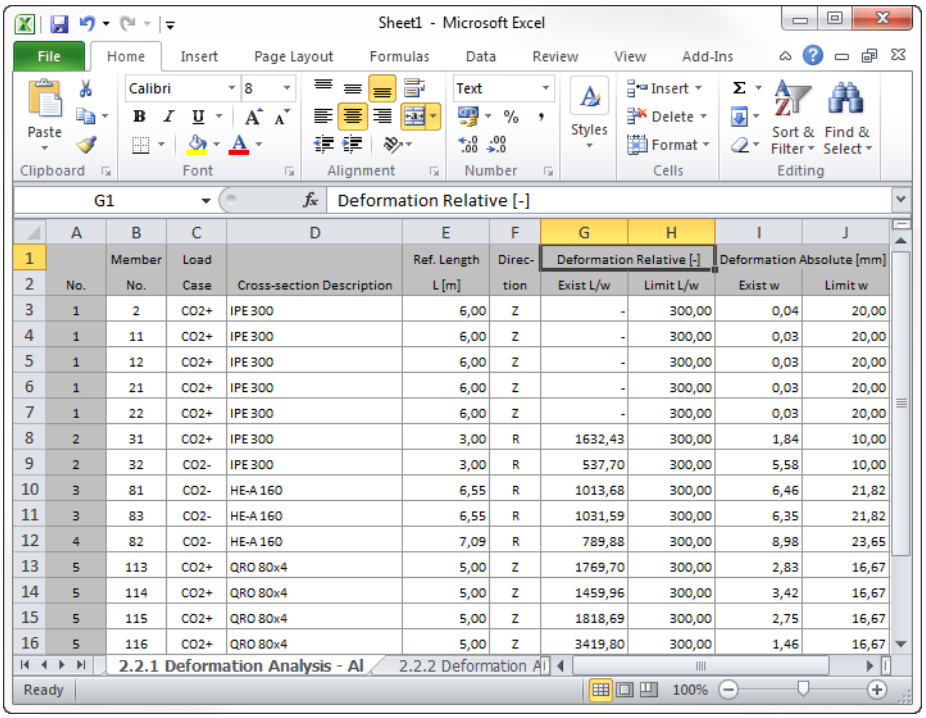


Figure 6.6: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking [OK]. Excel or 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
1		Member	Load		Ref. Length	Direc-	Deformation Relative [-]		Deformation Absolute [mm]	
2	No.	No.	Case	Cross-section Description	L [m]	tion	Exist L/w	Limit L/w	Exist w	Limit w
3	1	2	CO2+	IPE 300	6,00	Z	-	300,00	0,04	20,00
4	1	11	CO2+	IPE 300	6,00	Z	-	300,00	0,03	20,00
5	1	12	CO2+	IPE 300	6,00	Z	-	300,00	0,03	20,00
6	1	21	CO2+	IPE 300	6,00	Z	-	300,00	0,03	20,00
7	1	22	CO2+	IPE 300	6,00	Z	-	300,00	0,03	20,00
8	2	31	CO2+	IPE 300	3,00	R	1632,43	300,00	1,84	10,00
9	2	32	CO2-	IPE 300	3,00	R	537,70	300,00	5,58	10,00
10	3	81	CO2-	HE-A 160	6,55	R	1013,68	300,00	6,46	21,82
11	3	83	CO2-	HE-A 160	6,55	R	1031,59	300,00	6,35	21,82
12	4	82	CO2-	HE-A 160	7,09	R	789,88	300,00	8,98	23,65
13	5	113	CO2+	QRO 80x4	5,00	Z	1769,70	300,00	2,83	16,67
14	5	114	CO2+	QRO 80x4	5,00	Z	1459,96	300,00	3,42	16,67
15	5	115	CO2+	QRO 80x4	5,00	Z	1818,69	300,00	2,75	16,67
16	5	116	CO2+	QRO 80x4	5,00	Z	3419,80	300,00	1,46	16,67

Figure 6.7: Result in Excel

A Index