

**Version**  
**March 2010**

**Add-on Module**

# **RF-IMP**

**Generation of Equivalent Geometric  
Imperfections and Pre-Deformed  
Initial Structures**

## **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 1770

E-mail: [0Hinfo@dlubal.com](mailto:0Hinfo@dlubal.com)

Web: [www.dlubal.com](http://www.dlubal.com)



# Contents

	Contents	Page
1.1	Add-on Module RF-IMP	4
1.2	RF-IMP Team	5
1.3	Using the Manual	5
1.4	Starting RF-IMP Add-on Module	6
2.1	General Data	9
2.2	Imperfections	12
3.1	Starting Generation	16
3.2	Generated Imperfections	19
3.2.1	Equivalent imperfections	19
3.2.2	Pre-deformed initial structure	20
3.3	Exporting Imperfections	22
3.4	Example: Pre-Deformed Initial Structure	23
4.1	Cases of Generation with RF-IMP	25
4.2	Units and Decimal Places	27
4.3	Exporting Data	27
A	Literature	29
B	Index	30

# 1. Introduction

## 1.1 Add-on Module RF-IMP

Many standards consider imperfections in analysis of second order effects. For this purpose, you can select between two options: You can define equivalent loads, which is provided for example in the standards DIN 18800 Part 2, Section 2, EN 1992-1-1:2004-12 or EN 1993-1-1:2005-07, or you can create a pre-deformed initial structure by moving FE nodes according to the standard provision. Add-on module RF-IMP includes both options.

Initial deformations should be determined in unfavorable direction of action. For simple frameworks, you are able to define imperfections manually. However, this procedure is not sufficient for more complicated structures, such as with surface and volume elements. The rules of equivalent load calculation exist for members only. RFEM allows you to generate imperfections in the form of equivalent structure by using the deformation mode or the buckling mode of the structure.

You can quickly, and consequently economically, examine different cases by generating imperfections in RF-IMP. After that, the procedure in the most unfavorable direction is assured according to standards. Implementation of buckling stress curves of the profiles according to EN 1992-1-1:2004-12, EN 1993-1-1:2005-07 or DIN 18800 is very comfortable while defining equivalent member structure.

If you enter a displacement and an initial curvature, RF-IMP generates imperfections in critical directions and transfers them to RFEM. Then you can find imperfections in a separate load case and join them to a load group. By using the pre-deformed initial structure, you implement shifted FE nodes into parameters of the load group. The advantage is that you can determine different pre-deformed initial structures in one model while keeping the structural input data.

You can export tables directly to MS Excel or OpenOffice.org Calc for further manipulation or archiving.

As other add-on modules, RF-IMP is fully integrated into RFEM. This add-on module represents a product part of RFEM, not only by its visual appearance. You can comfortably generate all equivalent imperfections by using results from RFEM (deformations under load cases, load groups and load combinations), RF-STABILITY (buckling shapes) and RF-DYNAM (eigenmodes).

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

Your DLUBAL ENGINEERING SOFTWARE team

## 1.2 RF-IMP Team

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

### Program coordination

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

### Programming

DiS. Jiří Šmerák  
Ing. Roman Svoboda

### Program design, dialog figures and icons

Dipl.-Ing. Georg Dlubal  
MgA. Robert Kolouch

Ing. Jan Milěř

### Program supervision

Ing. František Knobloch  
Ctirad Martinec

### Manual, help system and translation

Dipl.-Ing. (FH) Robert Vogl	Ing. Ladislav Kábrt
Ing. Dmitry Bystrov	Mgr. Petra Pokorná
Jan Jeřábek	Dipl.-Ing. Veronika Erlebachová
Ing. Jakub Harazín	Ing. Jana Sejkotová

### Technical support and quality management

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

## 1.3 Using the Manual

All general topics such as installation, user interface, evaluation of results and output are described in detail in the RFEM manual; hence, we omit them in this manual. On the contrary, we focus concentration on typical features of the add-on module RF-IMP.

During the description of RF-IMP, we use the sequence and structure of the input and output dialogs. The text of the manual shows the described **buttons** in square brackets, for example [Details]. At the same time, they are pictured on the left. Additionally, the used **expressions** for dialog boxes, tables and menus are set in *italics* to clarify the explanations.

We also put into this manual the index for fast finding of some terms. If you do not find the requested ones, please check our website [www.dlubal.cz](http://www.dlubal.cz) where you can go through our *FAQ pages* by selecting particular criteria.

## 1.4 Starting RF-IMP Add-on Module

You can choose one of the following options in RFEM to start the add-on module RF-IMP.

### Main menu

Start the module by the command from the RFEM main menu

**Additional Modules → Others → RF-IMP.**

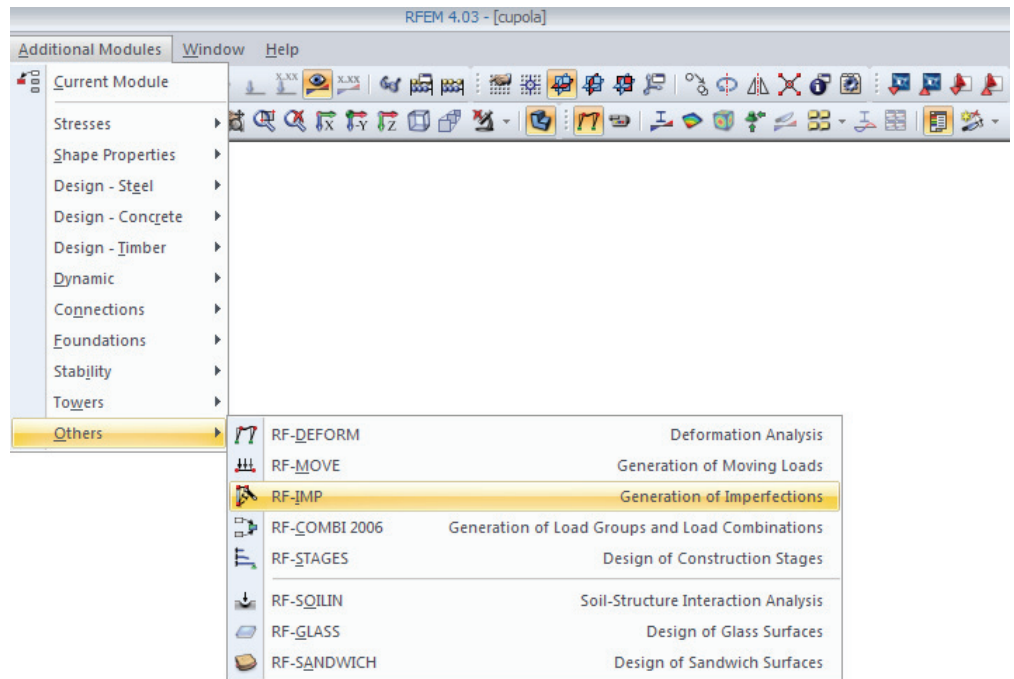


Figure 1.1: Main menu: *Additional modules* → *Others* → *RF-IMP*

## Navigator

You can also start the module RF-IMP in the *Data* navigator by selecting the folder

**Additional Modules → RF-IMP.**

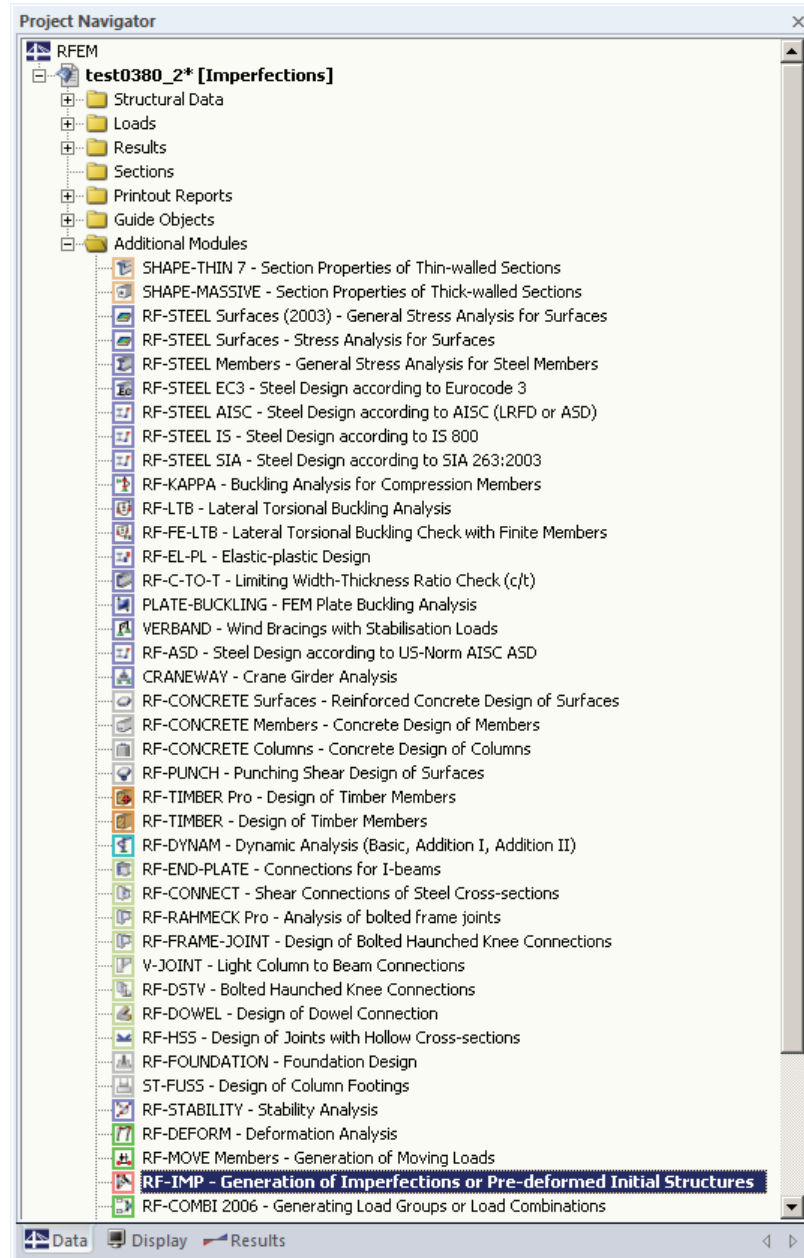


Figure 1.2: Data navigator: *Additional Modules* → *RF-IMP*

## 2. Input Data

Entering the input data for generation takes place in tables. You can use helpful function [Pick] for graphical selection of members or sets of members.

After starting RF-IMP, the navigator, which appears in a new window on the left, organizes both dialogs. You can find a list with possibly existing generation cases above the navigator (see chapter 4.1, page 25).

While starting RF-IMP for the first time in the RFEM program, the add-on module automatically loads following input data:

- Members and sets of members
- Load cases and load combinations



Control the tables either by clicking their names in the RF-IMP navigator or by using buttons displayed on the left margin. You can also list the dialogs forward and back by pressing the function keys [F2] and [F3].

Click [OK] to save the input data and to close the add-on module RF-IMP or click [Cancel] to close the add-on module without saving the data.

## 2.1 General Data

Determine the input for generation of imperfections and the type of generation in the table 1.1 *General Data*.

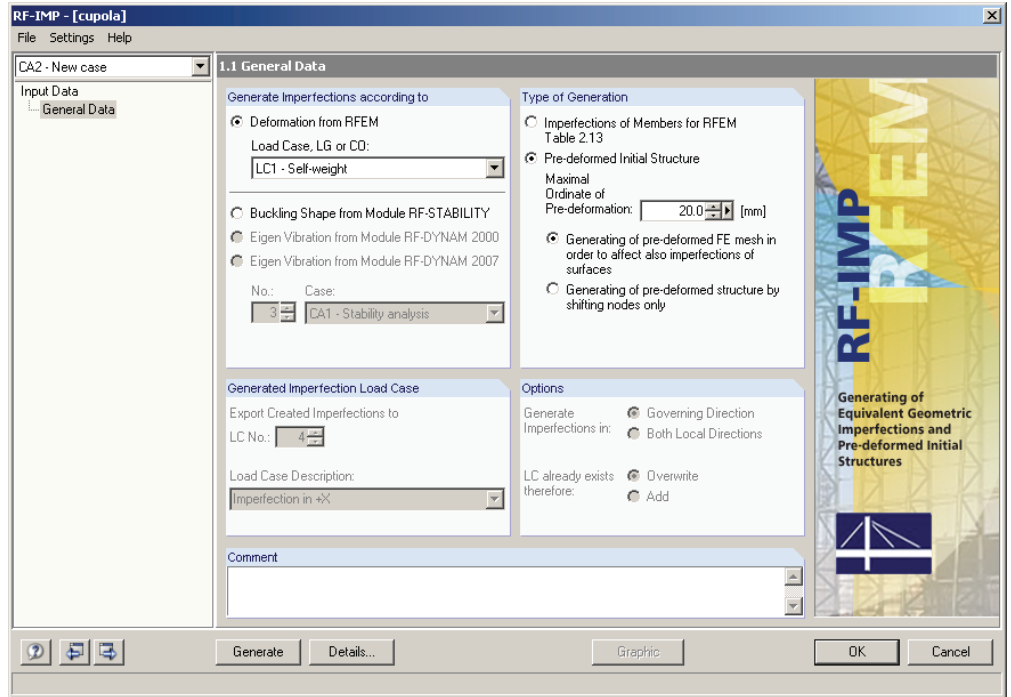


Figure 2.1: Table 1.1 *General Data*

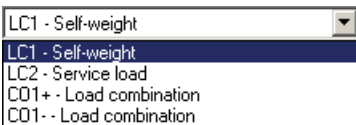
### Generating imperfections according to

At first, select which results you want to use for generation of imperfections, resp. of pre-deformed initial structure. You can choose one of three options.

#### Deformation from RFEM

Determine the load action in the list *Load Case, LG or CO*.

Take into account a special feature while selecting a load combination. There are two different results for each of x-values – maximal and minimal superposition of actions. Therefore, you can select one of two options in the list of load combinations: CO+ (maximum) and CO- (minimum) as shown in the picture on the left.

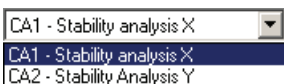


According to the standard DIN 18800 Part 2, el. (202), equivalent geometric imperfections are related to the lowest buckling eigenmode. In principle, you should create a particular load case of imperfections for each load in the critical direction with the equivalent deformation mode. Generation of cases in RF-IMP (see chapter 4.1, page 25) provides you a possibility to create suitable imperfections based on different deformations.

#### Buckling shape from module RF-STABILITY

You can determine imperfections – according to the standard DIN 18800 Part 2, el. (202) – by fitting them, as precisely as possible, to the lowest stability mode. However, you need the licensed RF-STABILITY add-on module to have an access to stability modes.

Enter the number of a relevant stability mode in the list *No.* and select the resulting case from RF-STABILITY in the list *Case*.





### Eigen vibration from module RF-DYNAM

You can base imperfections, resp. pre-deformed initial structure, on an eigenmode from the add-on module for dynamic analysis. According to the license that you keep at disposal, you can select the *RF-DYNAM 2000* or the *RF-DYNAM 2007* add-on module.

Specify determining case from dynamic analysis in the list *Case*. In the list *No.*, enter the number of an eigenvibration, which you want to use for the generation of imperfections.

If the add-on module RF-IMP cannot determine any imperfections because of missing results from RFEM or from modules RF-STABILITY or RF-DYNAM, it automatically calculates relevant deformations, stability modes or eigen vibrations during the following analysis.

### Type of Generation

In principal, you can create imperfections in RF-IMP in two different ways. Select specific options in this section to activate or to deactivate them.

#### Equivalent imperfections of members

According to the standard DIN 18800, it creates a load case with equivalent imperfections of members or set of members. You can further use the load case in load groups.

If you select this option for generation, additional data is required in the table sections *Generated Imperfection Load Case* and *Options* and in the table 1.2 *Imperfections*.

#### Pre-deformed initial structure

This option of generation transforms by scaling the standard deformations from RFEM, RF-STABILITY or RF-DYNAM to the *maximal ordinate of pre-deformation*, which you determine. There are two different options for pre-deformed initial structures, which are suitable especially for big shells:

- **Generating of pre-deformed FE mesh**

For each FE node, RF-IMP determines a deformation which is located between zero and a maximal ordinate of pre-deformation. Before the second or the third order analysis, the FE mesh is shaped according to the relevant RF-IMP case. For this purpose, you should select the required RF-IMP case in a load group in RFEM. Structural data from RFEM will stay retained.

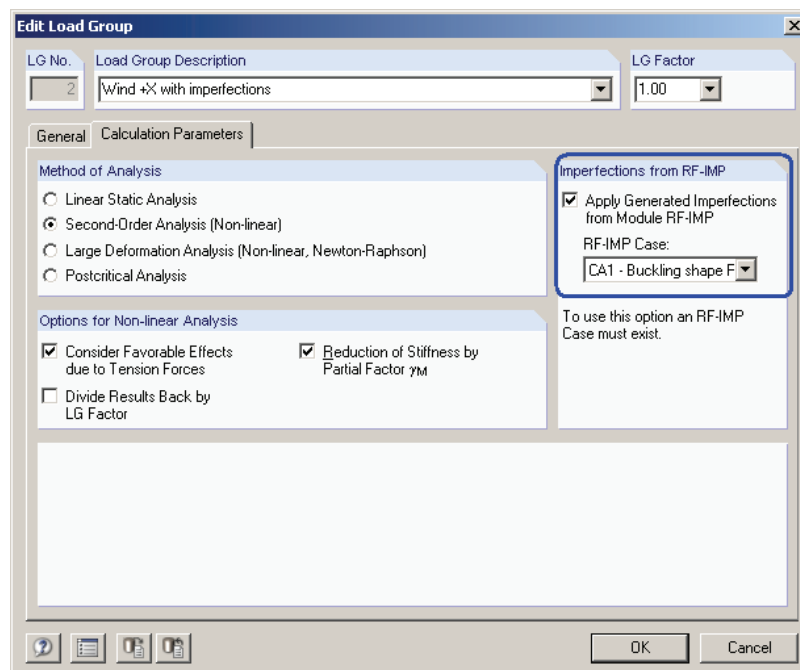


Figure 2.2: RFEM-Dialog box *Edit Load Group*, tab *Calculation Parameters*

- **Generating of pre-deformed structure by shifting nodes only**

Contrary to the method described above, this option implicates changes of structural data. It moves coordinates of nodes in the RFEM-table 1.1 according to the determined deformation. Therefore, it is not necessary to apply the input data to load groups.



This option of generation can consider one mode of imperfection only, because the nodes can be moved one time only.

### Generated imperfection load case

This section is active only with selected type of generation *Imperfections of members for RFEM Table 2.13*.

After generation, RF-IMP exports imperfections to the load case specified in the *LC No.* box. The first available number is set by default. You can directly enter *Load case description* or select it from the predefined list.

After generation, a control question appears before final transferring data to RFEM.

### Options

This section is also active only with selected type of generation *Imperfections of members for RFEM Table 2.13*.

You can generate imperfections in a *Governing direction* or in *Both local directions*. If you select the first option (default setting), RF-IMP determines imperfections in one direction only, either in the direction of the local y-coordinate or z-coordinate of the member. Resulting deformation, resp. deflection of an eigenmode, will be stated according to the tolerance (compare chapter 3.1, page 16). This predefined tolerance is also considered in the case of generation in both local directions.

By selecting the number of an existing load case in the section *Generated imperfection load case*, you activate two additional option buttons. You can select if RF-IMP *Overwrites* the existing load case or if RF-IMP *Adds* generated equivalent imperfections to the existing load case.

### Comment

Use this box for user-defined remarks, for example to describe and interpret defined parameters for actual case of generation in RF-IMP.

Details...

Click [Details] to control the tolerance settings for RF-IMP. The dialog box *RF-IMP - Details*, which is described later (chapter 3.1, page 16), appears.

## 2.2 Imperfections

The second table of this add-on module is available only in the case of selected *Imperfections of members for RFEM Table 2.13* in the previous table 1.1.

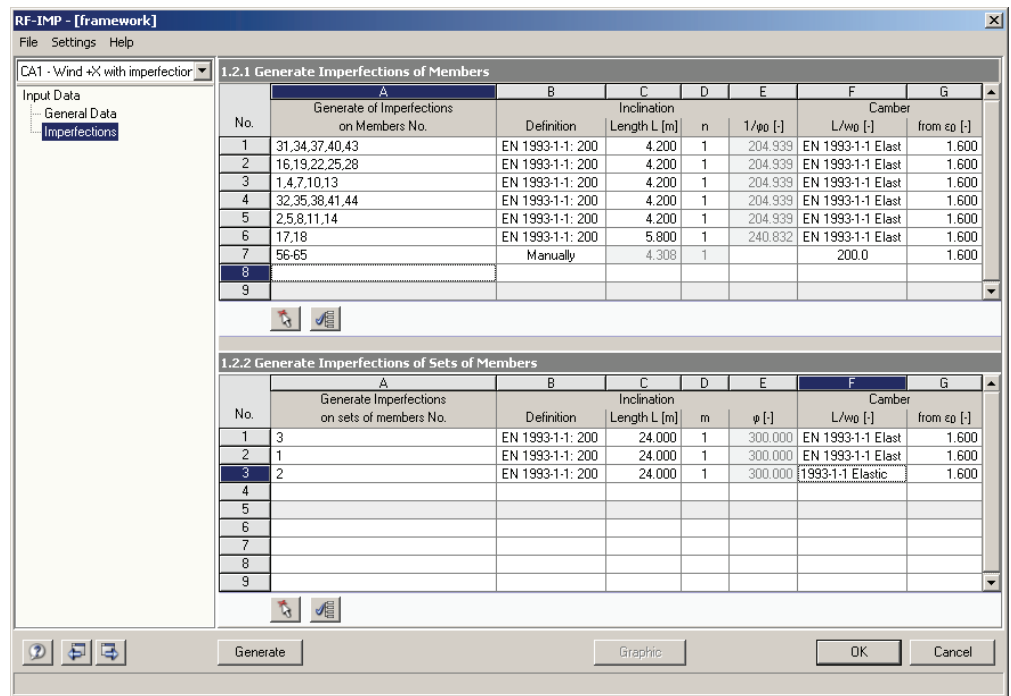
The table 1.2 *Imperfections* is divided into two parts: upper part 1.2.1 *Generate Imperfections of Members* consider relevant members for generation, lower part 1.2.2 *Generate Imperfections of Sets of Members* takes into account the continuous members (groups of members are usually unsuitable for imperfections).

Both parts of the table provide identical style of entering data. The only difference is that all input in the table 1.2.1 refers to particular members and the parameters of the table 1.2.2 refer to sets of members. Because both parts are identical except this difference, they are described together later in the text.

You can edit the input by standard key combinations, for example deleting an actual line by [CTRL+Y] (compare the RFEM program manual, chapter 4.4.8).

Edit units and decimal places of lengths and loads in the main menu

**Options** → **Units and Decimal Places** (compare the chapter 4.2, page 27).



1.2.1 Generate Imperfections of Members							
No.	A Generate Imperfections on Members No.	B Definition	C Inclination		E 1/ρ <sub>0</sub> [-]	F Camber	
			Length L [m]	n		L/w <sub>0</sub> [-]	from e <sub>0</sub> [-]
1	31,34,37,40,43	EN 1993-1-1: 200	4.200	1	204.939	EN 1993-1-1 Elast	1.600
2	16,19,22,25,28	EN 1993-1-1: 200	4.200	1	204.939	EN 1993-1-1 Elast	1.600
3	1,4,7,10,13	EN 1993-1-1: 200	4.200	1	204.939	EN 1993-1-1 Elast	1.600
4	32,35,38,41,44	EN 1993-1-1: 200	4.200	1	204.939	EN 1993-1-1 Elast	1.600
5	2,5,8,11,14	EN 1993-1-1: 200	4.200	1	204.939	EN 1993-1-1 Elast	1.600
6	17,18	EN 1993-1-1: 200	5.800	1	240.832	EN 1993-1-1 Elast	1.600
7	56-65	Manually	4.308	1		200.0	1.600
8							
9							

1.2.2 Generate Imperfections of Sets of Members							
No.	A Generate Imperfections on sets of members No.	B Definition	C Inclination		E ψ [-]	F Camber	
			Length L [m]	m		L/w <sub>0</sub> [-]	from e <sub>0</sub> [-]
1	3	EN 1993-1-1: 200	24.000	1	300.000	EN 1993-1-1 Elast	1.600
2	1	EN 1993-1-1: 200	24.000	1	300.000	EN 1993-1-1 Elast	1.600
3	2	EN 1993-1-1: 200	24.000	1	300.000	EN 1993-1-1 Elastic	1.600
4							
5							
6							
7							
8							
9							

Figure 2.3: Table 1.2 Imperfections

### Generation imperfections of members or sets of members

Determine member or set of member numbers in the column A.



Please note, that you should have defined sets of members in the RFEM program to be able to select them in the RF-IMP add-on module. The RF-IMP add-on module does not allow you to create sets of members.



Select members or sets of members graphically in the RFEM window after clicking the [Pick Members/Sets of Members] button. By clicking the cell in the table, you can alternatively enable the button [...], which also runs the graphical selection.

By picking members or sets of members in the RFEM workspace, you can transfer their numbers to the dialog box *Multiple Selection*. This dialog box allows you to return to RF-IMP and to import the corresponding numbers into the add-on module.



Select all the members or set of members for generation by clicking the [Import All Members] button. The members or set of members with identical parameters will be gathered in one line.

It is not necessary to numerate the members in sets of members continuously. In addition, the orientation of members can also differ. However, the only requirement is that the rotation of all the members in a set is the same.

### Inclination

Specify parameters to define inclination in the columns from B through E. You can select one of the following options of definition:

- DIN 18800 E-E (elastic-elastic method of analysis)
- DIN 18800 E-P (elastic-plastic method of analysis)
- EN 1992-1-1: 2004-12 (Eurocode 2)
- EN 1993-1-1: 2005-07 (Eurocode 3)
- DIN 1045-1: 2001-07
- DIN 1052: 2004-08
- Manually

#### Definition option

In the column B, select one of the options of definition described earlier in this chapter. Clicking the cell in the column B enables the [▼] button; use this button to select suitable definition from the list. You can also display the list of definitions by pressing the [F7] function key.

RF-IMP automatically loads the values of an inclination  $1/\varphi_0$  to the column E for both options of definition according to the standards.

#### Length L

*Length L* refers to the theoretical length of an imperfect member. This length is considered in the Equation 2.1 (see later in the text).

In the column C, the member lengths are preset, but you can edit them where appropriate. Enable the [...] button by clicking the cell in the column C. Use it for a graphical selection of the length in the RFEM workspace.

#### n

*n* refers to a number of self-existent imperfections of members or sets of members that should be considered. For example, see Figure 5 in the standard DIN 18800 Part 2, Paragraph 2.3. Generally, you can set the *n* value as a number of frame columns in a floor in view of a frame plane.

#### $1/\varphi_0$

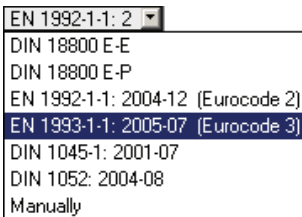
If you select the option of definition *Manually* in the column B, directly enter the value of an inclination. Both columns *Length L* and *n* are inactive for this option.

If you select one of the two options according to the standard DIN 18800, the automatic calculation of an inclination of one-part members usually proceeds according to the equation:

$$\varphi_0 = \frac{1}{200} \cdot r_1 \cdot r_2$$

Equation 2.1: Inclination  $\varphi_0$  according to DIN 18800 Part 2, Equation (1)

This equation is based on the **elastic-plastic** method of analysis. Both reduction factors are determined as follows.



$$r_1 = \sqrt{\frac{5}{l}}$$

where  $l$ : the theoretical length of a member or set of members

Equation 2.2: Reduction factor  $r_1$

Equation 2.2 is valid only for the lengths  $l \geq 5.00$  m. For  $l < 5.00$  m, the value for  $r_1$  is 1.

$$r_2 = \frac{1}{2} \cdot \left( 1 + \sqrt{\frac{1}{n}} \right)$$

where  $n$ : the number of the self-existent inclinations

Equation 2.3: Reduction factor  $r_2$

If you select the **elastic-elastic** method of analysis, the calculation proceeds the same way. However, according to the standard DIN 18800 Part 2, el. (201), the inclination can be reduced by  $\frac{2}{3}$  of its basis value (then  $\varphi_0 = \frac{1}{300}$ ), because the stress design is based on the elastic-elastic method, for example as in the add-on module RF-STEEL. It takes into account the fact that the plastic reserve of the section is not fully used.

In RF-IMP, the reduction is preset by default, because the elastic-elastic method of analysis represents a usual calculation method in programs from Dlubal company.

### Precamber

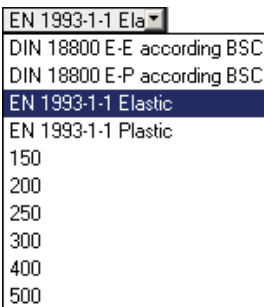
You can input parameters for precamber in the columns F and G. Select one of the following options of definition:

- DIN 18800 E-E according to BSC (buckling stress curve)
- DIN 18800 E-P according to BSC
- EN 1993-1-1 elastic
- EN 1993-1-1 plastic
- Selection of a precamber  $L/w_0$

#### $L/w_0$

In the column F, you can select one of the options of definition, as listed above in the text. Enable the [▼] button by clicking the cell in the column F and use this button to select suitable definition from the list. You can also display the list of definitions by pressing the [F7] function key.

The list contains usual values of a precamber according to the standards DIN 18800 and EN 1993-1-1, including two options *EN 1993-1-1 Elastic* and *EN 1993-1-1 Plastic*. The calculation of precamber according to the buckling stress curves is described in the standard EN 1993-1-1, chapter 5.3. For this option, you can find the values of a precamber for one-part members in the standard EN 1993-1-1, chapter 5.3, Table 5.1.



BSC	The value of a precamber $w_0$	
	Elastic analysis	Plastic analysis
$a_0$	$L/350$	$L/300$
a	$L/300$	$L/250$
b	$L/250$	$L/200$
c	$L/200$	$L/150$
d	$L/150$	$L/100$

Table 2.1: The value of a precamber according to EN 1993-1-1, chapter 5.3, Table 5.1

An assignment of the buckling stress curves from a through d to the section types proceeds according to the standard EN 1993-1-1, chapter 6.3, Table 6.2 (see following Figure 2.4):

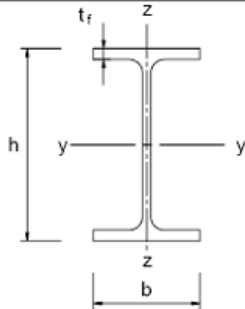
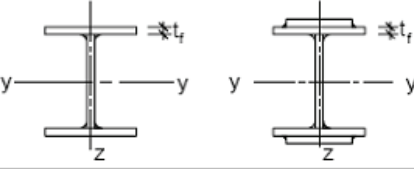

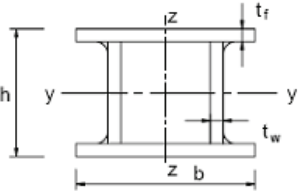
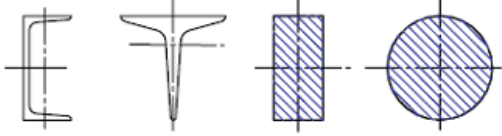
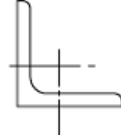
Cross section	Limits	Buckling about axis	Buckling curve	
			S 235 S 275 S 355 S 420	S 460
Rolled sections 	$h/b > 1,2$	y-y z-z	$t_f \leq 40 \text{ mm}$	a a <sub>0</sub>
			$40 \text{ mm} < t_f \leq 100$	b c
	$h/b \leq 1,2$	y-y z-z	$t_f \leq 100 \text{ mm}$	b c
			$t_f > 100 \text{ mm}$	d c
Welded I-sections 	$t_f \leq 40 \text{ mm}$	y-y z-z	b c	
	$t_f > 40 \text{ mm}$	y-y z-z	c d	
Hollow sections 	hot finished	any	a	
	cold formed	any	c	
Welded box sections 	generally (except as below)	any	b	
	thick welds: $a > 0,5t_f$ $b/t_f < 30$ $h/t_w < 30$	any	c	
U-, T- and solid sections 		any	c	
L-sections 		any	b	

Figure 2.4: Selection of buckling curve for a cross-section according to EN 1993-1-1, chapter 6.3, Table 6.2

**From  $\epsilon_0$**



In the last column of the table, enter the value of a parameter  $\epsilon_0$  which states the initial value for consideration of the precamber together with the inclination. According to the standard DIN 18800 Part 2, el. (207), the value of  $\epsilon_0 = 1.6$  is set by default as it is the condition of a relevant precamber. However, you can define parameters or select them by clicking the [▼] button in the cell.

The program RFEM calculates these parameters for different members or sets of members by the second order theory and then it compares them with the initial values. If it is necessary, it also takes into account a precamber in following processes.

## 3. Generating

This chapter describes the creation of imperfections by generating equivalent loads or a pre-deformed initial structure and the following transferring of the results to the RFEM program.

### 3.1 Starting Generation

Details...

Before generation, we recommend you to check the settings of tolerances in RF-IMP. Click the [Details] button in the table 1.1 *General Data*. The following dialog box appears:

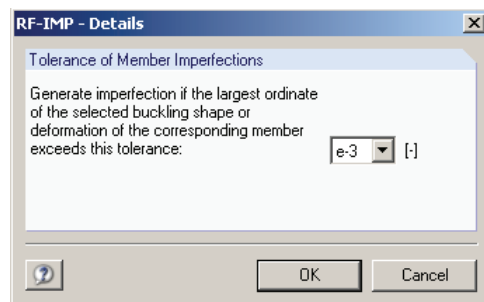


Figure 3.1: Dialog box *RF-IMP - Details*

In the *Tolerance of Member Imperfections* section, set the ordinate of the selected deformation mode or buckling shape that induces imperfections. Therefore, small deflections, which are less than the defined value, are not considered in the deformation mode or the buckling shape. No imperfections are generated for the members and the sets of members with deflections less than the defined limit.

Set the value of the tolerance in a list that appears after clicking the [▼] button. You can select a value from  $e-7$  through  $e-9$ . In the list,  $e-7$  represents rounding off to one decimal place and selection of  $e-9$  takes into consideration nine decimal places.

RF-IMP examines normed shifts of nodes in the deformation mode or buckling shape. Maximal values refer to the normed value of 1. For example, if the maximal deformation of your structure is 10 cm, which is referring to the normed value of 1, then the normed value of 0.008 represents the deflection of 0.08 cm. If you set the tolerance to  $e-3$  (it means the rounding off to three decimal places: to 0.005) RF-IMP generates an equivalent imperfection for the member, because the normed value is greater than the limit. If the tolerance is set to  $e-2$ , imperfections of the member are not taken into account, because its normed value of deformation is less than the limit with two decimal places 0.05.

In case RF-IMP cannot generate any imperfections of members because of the tolerance settings, the corresponding warning appears after clicking the [Generate] button.

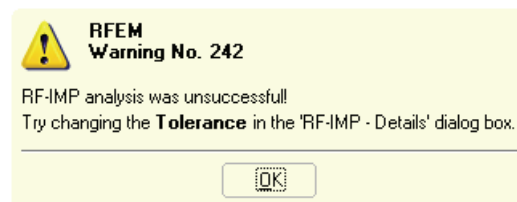


Figure 3.2: Warning because of a too low value of a tolerance

Increase the tolerance by taking into account more decimal places, as described above.

This dialog box does not influence the generation of pre-deformed initial structures.

Generate

In the both RF-IMP tables, the button [Generate] is available for data generation of a new load case or of a pre-deformed initial structure for RFEM.

You can also start the generation in RF-IMP from the RFEM workspace. Find RF-IMP listed as an add-on module in the dialog box *To Calculate*, analogously to a load case or a load group. Start this dialog box from the RFEM main menu

Calculate → To Calculate.

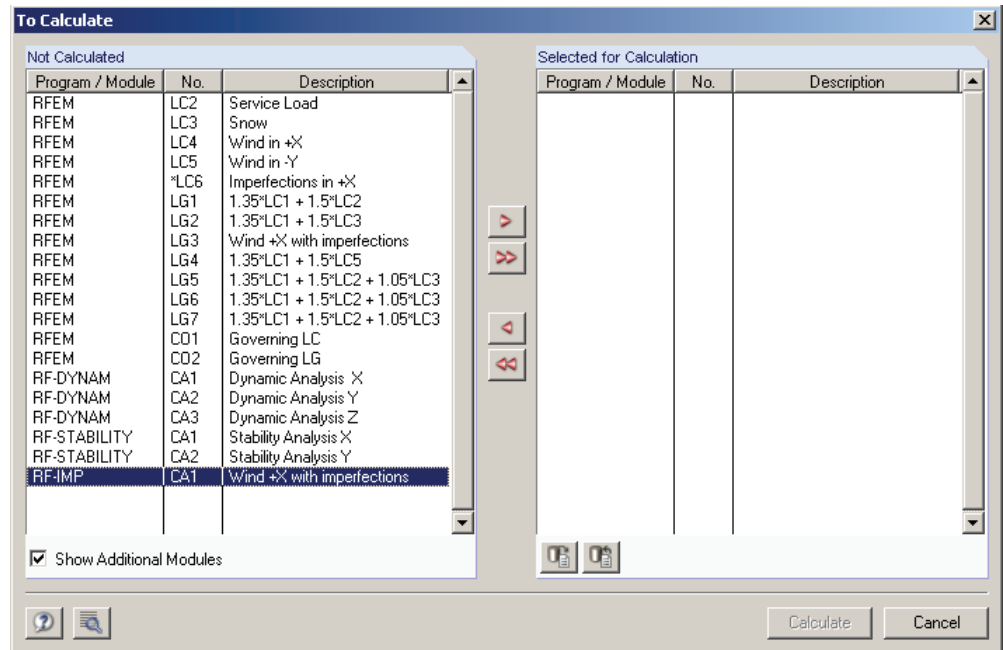


Figure 3.3: Dialog box *To Calculate*

If RF-IMP is missing in the list of *Not Calculated* cases, select the check box *Show Additional Modules*.

Add selected cases from RF-IMP to the list on the right by the [▶] button and start the calculation by clicking the [Calculate] button.

You can also calculate a selected case of generation imperfections directly from the list in the toolbar. Select the required case from RF-IMP and click the button [Results on/off].

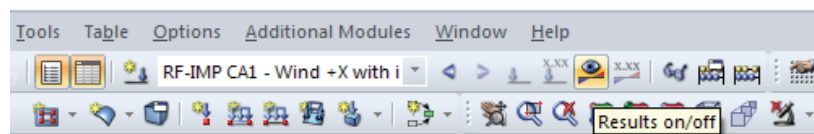


Figure 3.4: Direct approach to the calculation of a case of generation from RF-IMP in the RFEM program

If you create the pre-deformed initial structure with the option *Generating of pre-deformed structure by shifting nodes only* based on already calculated *Deformation from RFEM*, a question about deleting the results appears.

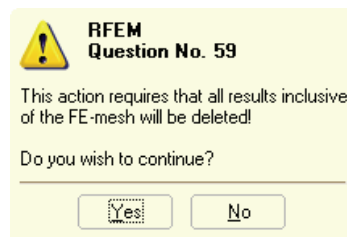
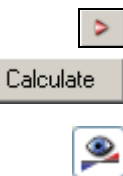


Figure 3.5: Question about deleting the results



If you agree to continue, RF-IMP changes the original structure. Deformations which are already calculated are wrong in this case.

Depending on the structural data, the generation of imperfections or pre-deformed initial structure takes a very short time. After a successful generation, the table 2.1 *Pre-deformed Initial Structure or Imperfections* of members, sets of members or surfaces with FE mesh points appears (see Figure 3.2 and Figure 3.4). If you create a pre-deformed initial structure by shifting nodes, correspondent information appears.

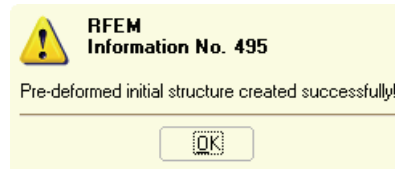


Figure 3.1: Information after successful generation of a pre-deformed initial structure

The generation of imperfections or a pre-deformed initial structure is already finished. If you have defined more cases in RF-IMP, you must proceed a generation for each RF-IMP case separately.



If you calculate all cases at once by clicking the command

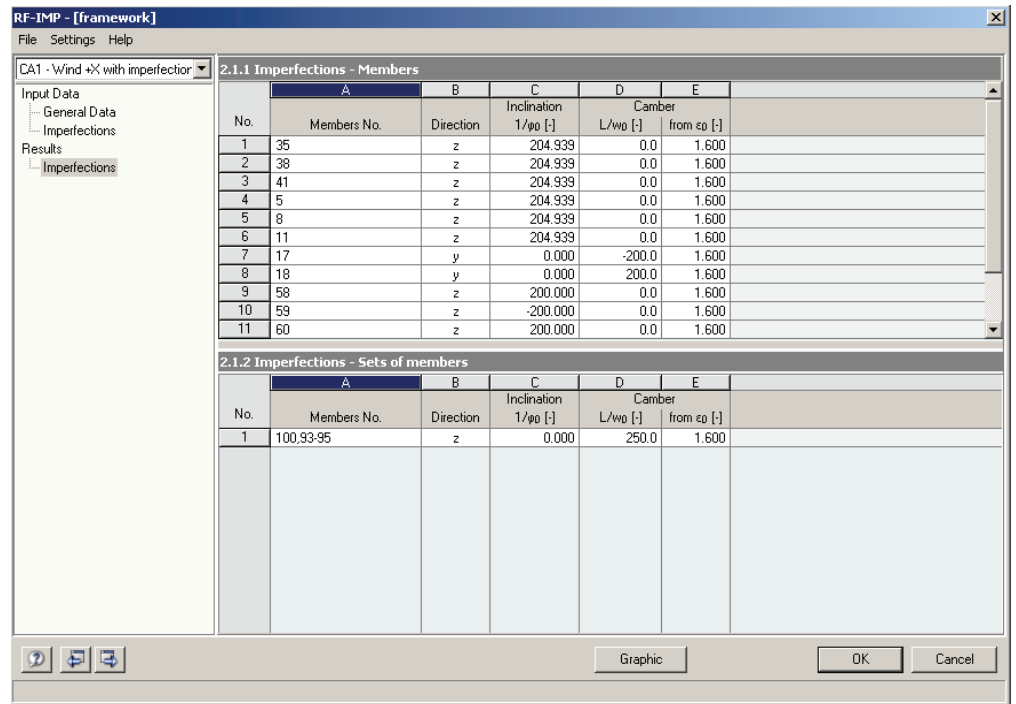
**Calculate** → **Calculate All**,

RFEM considers all its load cases and all load cases from add-on modules. Defined cases from RF-IMP with the type of generation *Imperfections of members for RFEM Table 2.13* are also calculated. However, RFEM does not automatically import the generated imperfections as a load case. Export them manually from the add-on module RF-IMP. Cases in RF-IMP with the type of generation *Pre-deformed initial structure* are omitted in global calculation. If they were calculated, all other results would be deleted and the calculation would be rejected.

## 3.2 Generated Imperfections

The results output of generation differs for *imperfections* and for a *pre-deformed initial structure*. The type of this table depends on the type of generation imperfections.

### 3.2.1 Equivalent imperfections



The screenshot shows the 'RF-IMP - [framework]' window with the 'Imperfections' section selected in the left sidebar. The main area displays two tables:

**2.1.1 Imperfections - Members**

No.	A		C	D		E
	Members No.	B		Inclination 1/ρ₀ [°]	Camber L/w₀ [-] from ε₀ [-]	
1	35	z	204.939	0.0	1.600	
2	38	z	204.939	0.0	1.600	
3	41	z	204.939	0.0	1.600	
4	5	z	204.939	0.0	1.600	
5	8	z	204.939	0.0	1.600	
6	11	z	204.939	0.0	1.600	
7	17	y	0.000	-200.0	1.600	
8	18	y	0.000	200.0	1.600	
9	58	z	200.000	0.0	1.600	
10	59	z	-200.000	0.0	1.600	
11	60	z	200.000	0.0	1.600	

**2.1.2 Imperfections - Sets of members**

No.	A		C	D		E
	Members No.	B		Inclination 1/ρ₀ [°]	Camber L/w₀ [-] from ε₀ [-]	
1	100,93-95	z	0.000	250.0	1.600	

Figure 3.2: Table 2.1 Imperfections

After the generation, the two-part table 2.1 *Imperfections* appears. In both table sections, you find particular members or sets of members with generated imperfections. A *direction* of an action of inclination and a *precamber* are defined in a local coordinate system in the column B.

Tension members and tension sets of members are not considered in generation of imperfections.

You cannot edit the generated imperfections in this table.

Click the [Graphic] button to display generated imperfections in the RFEM workspace. You can control now, if imperfections are generated for all the members and sets of members that are endangered by buckling.

Graphic

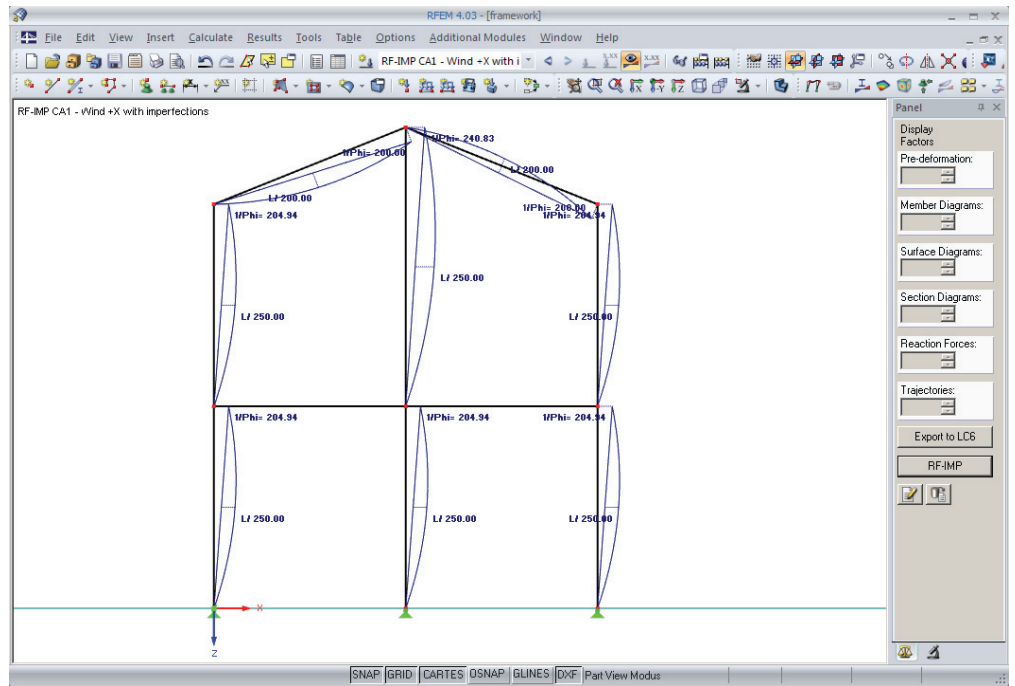


Figure 3.3: Graphic with generated equivalent imperfections

RF-IMP

Click the [RF-IMP] button to return to the RF-IMP module, where you can eventually edit the input data or start a new generation process.

### 3.2.2 Pre-deformed initial structure

#### Generating of pre-deformed FE mesh

2.1.1 Pre-deformed Initial Structure - Surfaces							
Surface No.	FE-Mesh Point	FE-Mesh Point Coordinates [m]			Pre-deformations [mm]		
		X	Y	Z	v <sub>X</sub>	v <sub>Y</sub>	v <sub>Z</sub>
1	16	0.000	0.000	-4.200	19.9	0.0	-0.1
	17	4.000	0.000	-4.200	19.9	0.0	0.0
	18	8.000	0.000	-4.200	19.9	0.0	0.1
	19	0.000	6.000	-4.199	19.9	0.0	-0.1
	20	4.000	6.000	-4.199	19.9	0.0	0.0
	21	8.000	6.000	-4.199	19.9	0.0	0.1
	22	0.000	12.000	-4.199	19.9	0.0	-0.1
	23	4.000	12.000	-4.199	19.9	0.0	0.0
	24	8.000	12.000	-4.199	19.9	0.0	0.1
	25	0.000	18.000	-4.199	19.9	0.0	-0.1
	26	4.000	18.000	-4.199	19.9	0.0	0.0

2.1.2 Pre-deformed Initial Structure - Members				
Member No.	Location x [m]	Pre-deformations [mm]		
		v <sub>X</sub>	v <sub>Y</sub>	v <sub>Z</sub>
1	0.000	0.0	0.0	0.0
	4.200	19.9	0.0	-0.1
2	0.000	19.9	0.0	-0.1
	4.200	30.5	0.0	-0.1
4	0.000	0.0	0.0	0.0
	4.199	19.9	0.0	-0.1
5	0.000	19.9	0.0	-0.1
	4.200	33.1	0.0	-0.1
7	0.000	0.0	0.0	0.0
	4.199	19.9	0.0	-0.1
8	0.000	19.9	0.0	-0.1

Figure 3.4: Table 2.1 Pre-deformed Initial Structure

For the type of generation *Generating of pre-deformed FE mesh*, the two-part table 2.1 *Pre-deformed initial structure* appears. In both tables, you can find the original locations and generated pre-deformations of FE mesh points for particular *surfaces* and *members*. Displayed *pre-deformations* are defined in the global coordinate system XYZ.

Find a short example of creation of a pre-deformed initial structure starting from the page 23, chapter 3.4.

### Generating of pre-deformed structure by shifting nodes

Before *generation of pre-deformed structure by shifting nodes*, information that you need to convert plane surfaces into quadrangle surfaces appears. This is necessary for fitting the boundary lines of surfaces to a deformation shape or a buckling mode.

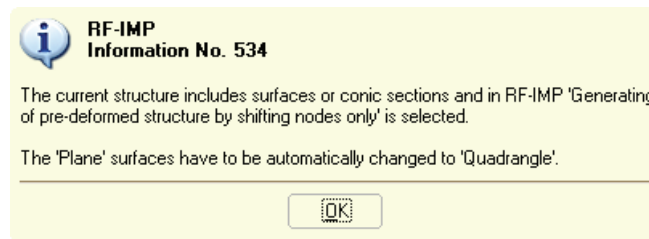
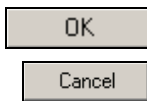


Figure 3.5: Information about an existence of plane surfaces or conic sections

After generation, the information that is displayed in the Figure 3.1 appears. It means that the deformation of an original structural system just proceeded. The generation is already finished.

In a pre-deformed initial structure, the coordinates of nodes are modified. Close the RF-IMP module by clicking the [OK] or the [Cancel] button to control the modified coordinates, for example in the RFEM table 1.1 *Nodes*. Take into account that it is not possible to change the center of members or surfaces, only the coordinates of boundary nodes are shifted.



### 3.3 Exporting Imperfections

This chapter describes exporting of generated *imperfections* as a load case to the RFEM program. In contrast, RF-IMP creates a *pre-deformed initial structure* automatically for RFEM and provides it to load groups (compare Figure 2.2, page 10) or it modifies coordinates of nodes in the RFEM table 1.1 *Nodes*.

You can export generated equivalent imperfections by clicking [OK] in any of the RF-IMP tables. The following information appears.

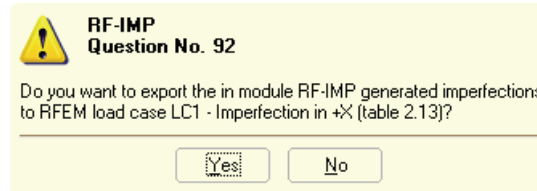
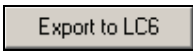


Figure 3.6: Question about data exportation of imperfections as a load case



As displayed in the Figure 3.3, you can also transfer the data in the RFEM graphical workspace. Click the button [Export to LC...] in the panel and the same question, as described above, appears.

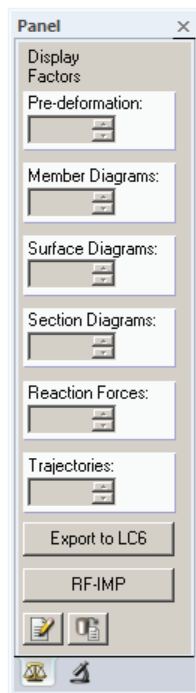


Figure 3.7: Panel with the *Export* button

After exporting imperfections, you will find the transferred data in the RFEM table 2.13 *Imperfections* where you can edit or complete it eventually.

### 3.4 Example: Pre-Deformed Initial Structure

In this example, you can follow a generation of imperfections as a pre-deformed initial structure of a cupola structure that is modeled in the RFEM program. You should determine the initial deformation of FE mesh using a buckling mode to take into account buckling of surfaces.

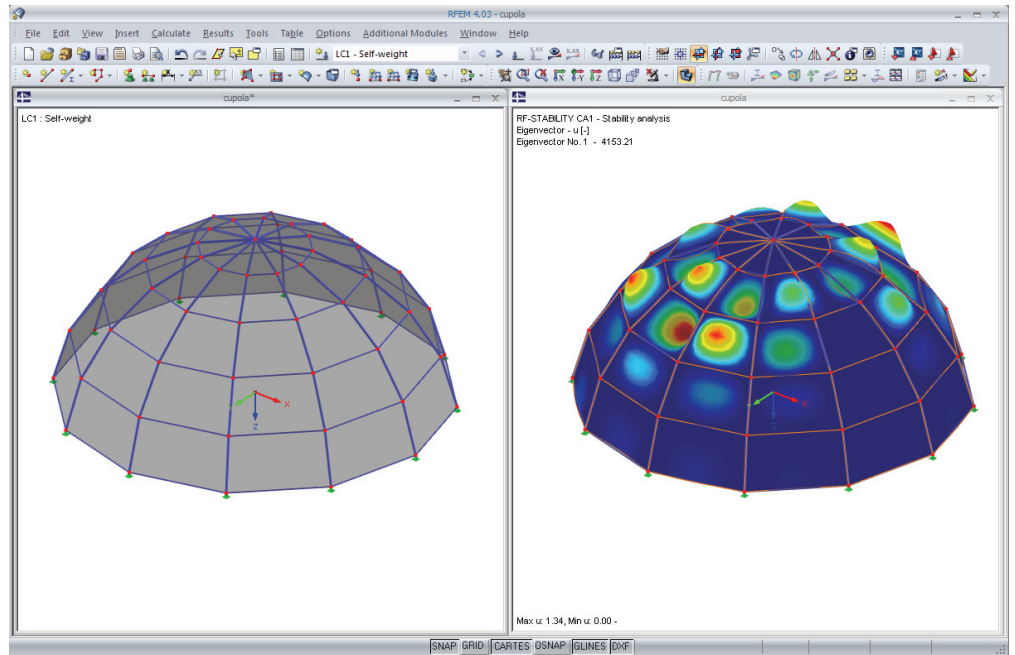


Figure 3.8: Model of a cupola and its buckling mode

In the module RF-STABILITY, stability modes are based on a self-weight of structure.

Open the RF-IMP table 1.1 *General Data* and enter the following input.

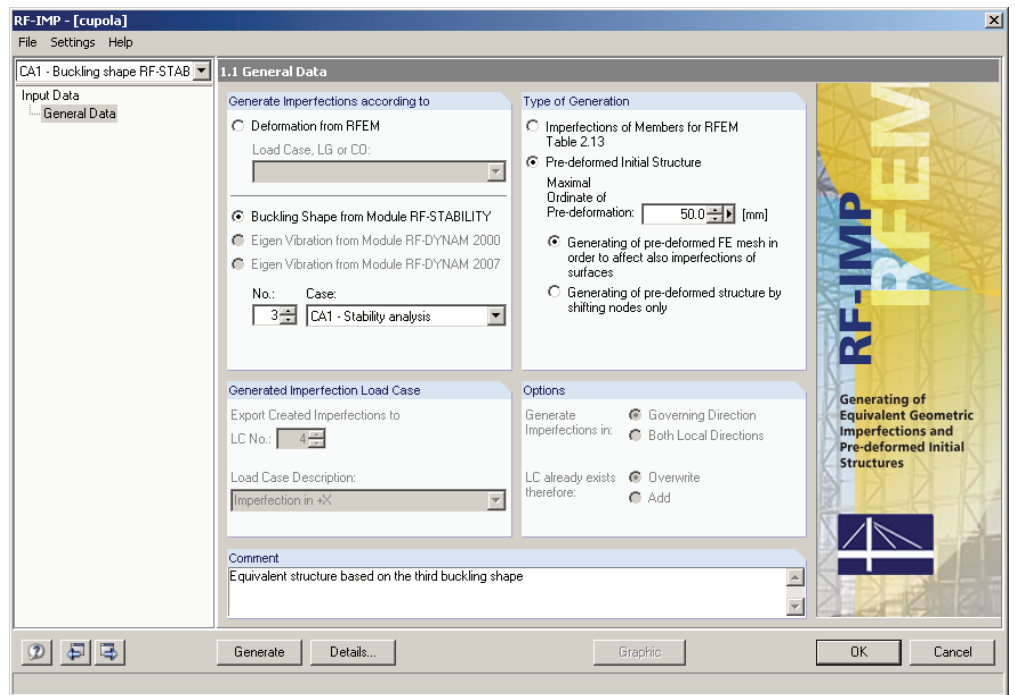


Figure 3.9: RF-IMP table 1.1 *General Data*

The **third buckling shape** is set in the *No.* box. Enter the value of the *Maximal ordinate of pre-deformation* of **50 mm**, then the normed values of deformation will be summed with the coordinates of FE nodes. Input is now complete.

Generate

Click the [Generate] button to display modified FE nodes of *surfaces* in the table.

**2.1.1 Pre-deformed Initial Structure - Surfaces**

Surface No.	FE-Mesh Point	FE-Mesh Point Coordinates [m]			Pre-deformations [mm]		
		X	Y	Z	v <sub>X</sub>	v <sub>Y</sub>	v <sub>Z</sub>
1740		0.920	-3.434	-7.778	3.0	-11.1	-25.0
1741		1.104	-3.385	-7.778	2.8	-10.6	-23.8
1742		1.288	-3.335	-7.778	2.4	-9.1	-20.4
1743		1.472	-3.286	-7.778	1.7	-6.6	-14.9
1744		1.656	-3.237	-7.778	0.9	-3.5	-7.9
1745		1.840	-3.187	-7.778	0.0	0.0	0.0
1746		0.206	-4.057	-7.581	1.2	-4.7	-10.4
1747		0.411	-4.002	-7.581	2.4	-8.8	-19.5
1748		0.617	-3.947	-7.581	3.2	-12.1	-26.8
1749		0.822	-3.892	-7.581	3.8	-14.2	-31.5
1750		1.028	-3.836	-7.581	4.0	-14.9	-33.1

**2.1.2 Pre-deformed Initial Structure - Members**

Member No.	Location x [m]	Pre-deformations [mm]		
		v <sub>X</sub>	v <sub>Y</sub>	v <sub>Z</sub>
67	0.000	0.0	0.0	0.0
	0.512	-0.9	-0.3	-0.2
	1.025	-2.8	-0.8	-0.8
	1.537	-4.5	-1.2	-1.2
	2.049	-5.7	-1.5	-1.6
	2.562	-6.1	-1.6	-1.7
	3.074	-5.7	-1.5	-1.6
	3.587	-4.6	-1.2	-1.2
	4.099	-2.8	-0.7	-0.8
	4.611	-0.9	-0.2	-0.2
	5.124	0.0	0.0	0.0

Figure 3.10: RF-IMP table 2.1 Pre-deformed Initial Structure

Graphic

Click the [Graphic] button for a visual control of imperfections.

Create a new load group and select *Apply generated imperfections* in the *New Load Group* dialog box, *Calculation Parameters* tab.

**New Load Group**

LG No.: 1 | Load Group Description: Design internal forces | LG Factor: 1.00

General | **Calculation Parameters**

Method of Analysis:

- Linear Static Analysis
- Second-Order Analysis (Non-linear)
- Large Deformation Analysis (Non-linear, Newton-Raphson)
- Postcritical Analysis

Options for Non-linear Analysis:

- Consider Favorable Effects due to Tension Forces
- Reduction of Stiffness by Partial Factor  $\gamma_M$
- Divide Results Back by LG Factor

Imperfections from RF-IMP:

- Apply Generated Imperfections from Module RF-IMP
- RF-IMP Case: CA1 - Buckling shape F

To use this option an RF-IMP Case must exist.

Figure 3.11: Dialog box *New Load Group*, the tab *Calculation Parameters*

## 4. General Functions

This chapter finally presents other menu options and options for exporting cases of generation.

### 4.1 Cases of Generation with RF-IMP

You can group the parameters or members and sets of members in separated cases of generation. This way, you can apply the generation of particular cases in RF-IMP on certain members, loads or buckling shapes. It is possible, that critical buckling shape differs in some part of the structure. Consider separately all the possibilities in dependence on your technical background.

Be careful not to rewrite existing cases and their parameters for generation by entering a case number that is already used, while you are defining different cases of generation.

#### Creating new cases in RF-IMP

Create a new case of generation by clicking the command in the RF-IMP menu

**File** → **New Case**.

The following dialog box appears.

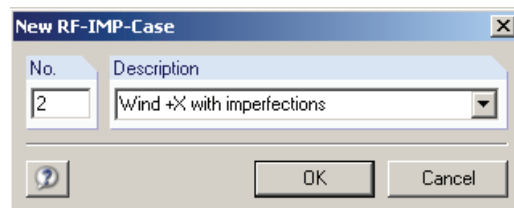


Figure 4.1: Dialog box *New RF-IMP-Case*

Enter an unassigned number to the *No.* box and type a *Description* of a new case of generation. By pressing [OK], display the table 1.1 *General Data* in RF-IMP for definition of new parameters.

#### Renaming RF-IMP cases

Edit the name of a case of generation by clicking the command in the RF-IMP menu

**File** → **Rename Case**.

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

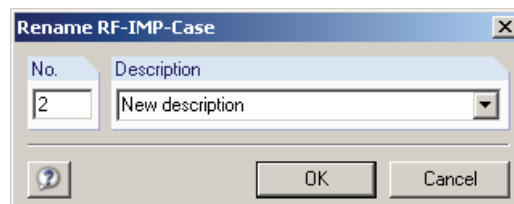


Figure 4.2: Dialog box *Rename RF-IMP-Case*

## Copying RF-IMP cases

You can copy input data of an actual case of generation by clicking the command in the RF-IMP menu

**File** → **Copy Case**.

The dialog box *Copy RF-IMP-Case* appears. Specify the *No.* and the *Description* of a new case.

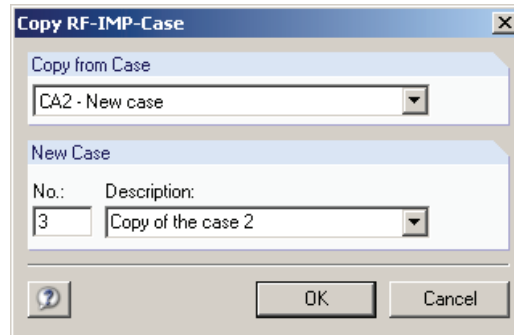


Figure 4.3: Dialog box *Copy RF-IMP-Case*

## Deleting RF-IMP cases

You can delete a case of generation by clicking the command in the RF-IMP menu

**File** → **Delete Case**.

Select the case of generation in the list of *Available Cases* in the dialog box *Delete Cases* and press [OK] to delete it.

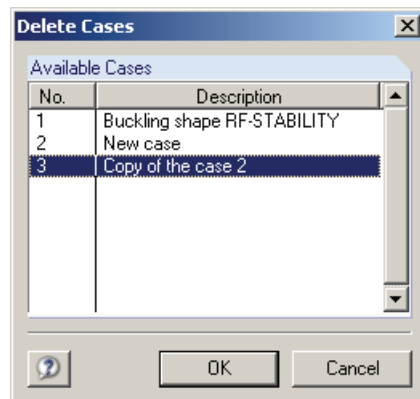


Figure 4.4: Dialog box *Delete Cases*

## 4.2 Units and Decimal Places

Manage units and decimal places in one global dialog box for RFEM and all add-on modules. Display this dialog box to control the units in RF-IMP by clicking the command

**Settings** → **Units and Decimal Places**.

The dialog box, already known from RFEM, appears, and the module RF-IMP is already selected.

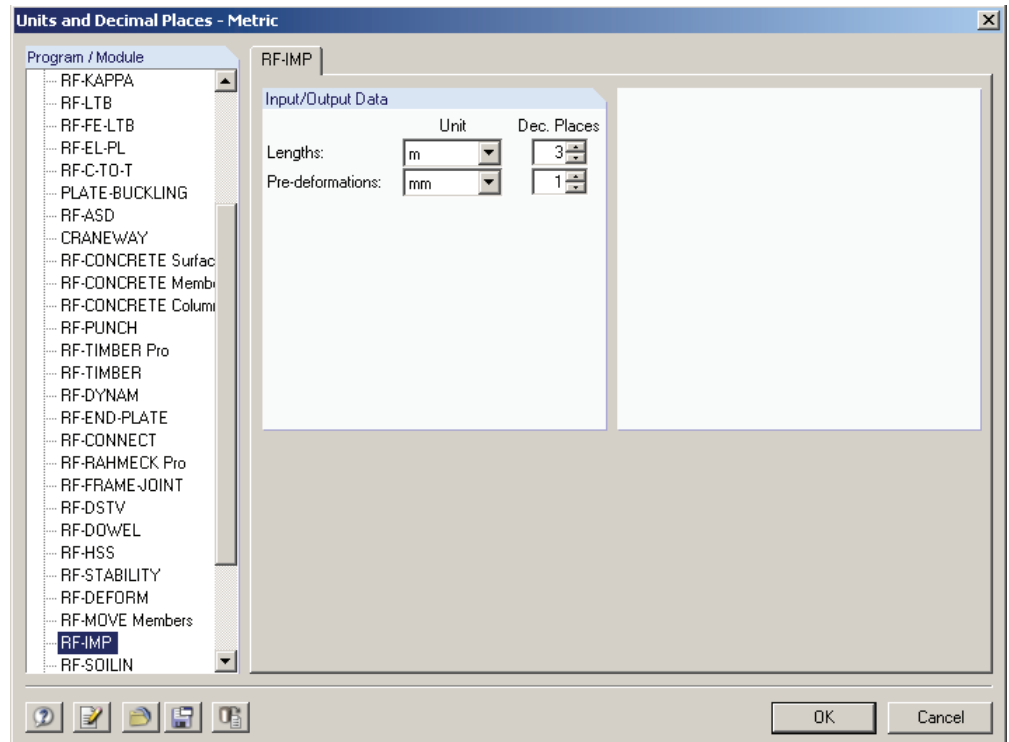


Figure 4.5: Dialog box *Units and Decimal Places*



You can save the parameters as your profile and reuse it for other structures. Find further description of these functions in the RFEM manual, chapter 12.6.2.

## 4.3 Exporting Data

Data from generated equivalent imperfections is exported directly to the main RFEM program by creation of a new load case of *imperfections*. Eventually, it is possible to edit or complete the load case. For detailed description of transferring equivalent loads to RFEM see chapter 3.3, page 22. With some restrictions, you can also provide data directly from RF-IMP to other programs.

### Clipboard

You can copy selected cells from the tables 1.2 *Imperfections* and 2.1 *Pre-deformed Initial Structure or Imperfections* by pressing the key combination [CTRL+C] and insert it by pressing [CTRL+V], for example into one of the text editors. Column headings are not taken into account.

### Printout report

You cannot directly add data from RF-IMP in the printout report. However, you can directly export imperfections as created RFEM load cases from the printout report by the command

**File → Export to RTF File or BauText.**

Find a description of this function in the RFEM manual, chapter 11.1.11.

### MS Excel / OpenOffice

You can directly export data from RF-IMP to MS Excel or OpenOffice.org Calc. Use this function by clicking the command

**File → Export Tables.**

The following dialog for exporting appears.

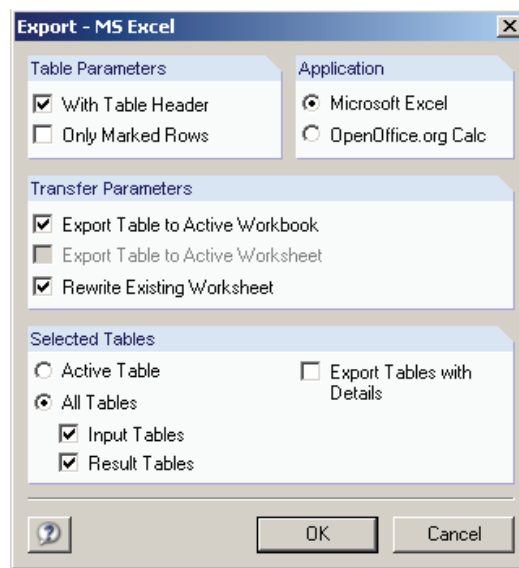


Figure 4.6: Dialog box *Export - MS Excel*

In accordance with a conception, only the tables 1.2 *Imperfections* and 2.1 *Pre-deformed Initial Structure or Imperfections* are considered for exporting. Press [OK] to start exporting data. MS Excel starts automatically, its running in the background is not necessary.

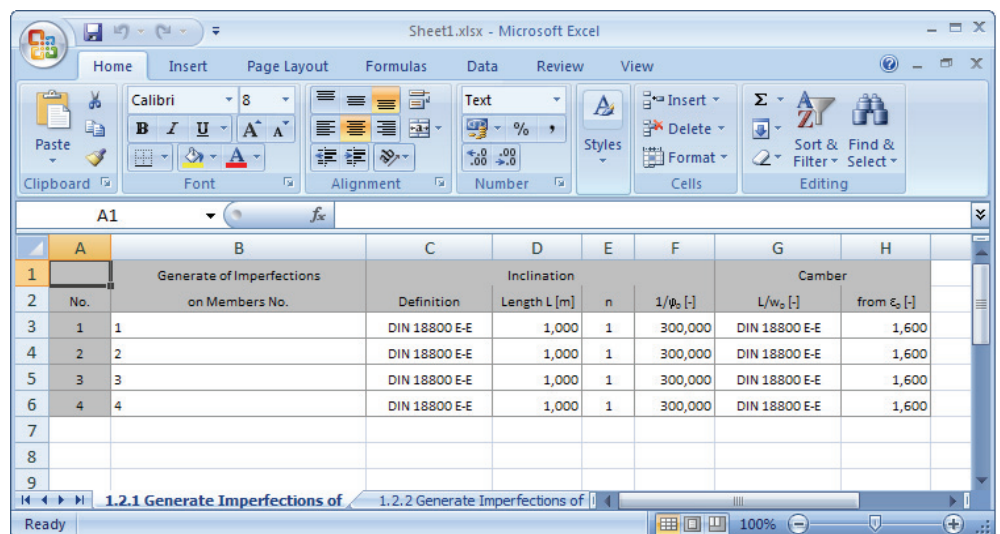


Figure 4.7: Result of exporting data to MS Excel

# A Literature

- [1] DIN 18800 (11.90) Teil 1: Stahlbauten - Bemessung und Konstruktion, Beuth Verlag, Berlin/Wien/Zürich, 1992
- [2] DIN 18800 (11.90) Teil 2: Stahlbauten - Stabilitätsfälle, Knicken von Stäben und Stabwerken, Beuth Verlag, Berlin/Wien/Zürich, 1992
- [3] EN 1993-1-1 (Eurocode 3): Design of steel structures - Part 1-1: General rules and rules for buildings, 2005

# B Index

<b>A</b>		
Add a load case .....	11	
<b>B</b>		
BSC .....	14	
Buckling shape.....	24	
Buckling stress curve.....	14, 15	
<b>C</b>		
Case of generation .....	25, 26	
Closing RF-IMP.....	8	
Comment.....	11	
<b>D</b>		
Decimal places .....	12, 27	
Deformation .....	9	
Details.....	11, 16	
<b>E</b>		
Eigen vibration .....	10	
Elastic-elastic .....	13, 14	
Elastic-plastic .....	13, 14	
Epsilon $\varepsilon_0$ .....	15	
Equivalent imperfections .....	10, 19	
Excel .....	28	
Exporting .....	22, 27	
<b>F</b>		
FE mesh points .....	21	
<b>G</b>		
Governing direction.....	11	
Graphic.....	19, 22	
<b>I</b>		
Imperfection load case .....	11	
Imperfections.....	12, 19	
Initial structure .....	10	
<b>L</b>		
Length .....	13	
Load case LC.....	11	
Load combination CO.....	9	
Load group LG .....	10	
<b>M</b>		
Manually .....	13	
Members .....	12, 19	
MS Excel.....	28	
<b>N</b>		
<i>n</i> causes .....	13	
Navigator .....	8	
<b>O</b>		
OpenOffice.....	28	
Options of definition.....	14	
Ordinate of deformation .....	10	
Overwrite a load case.....	11	
<b>P</b>		
Parameter $\varepsilon_0$ .....	15	
Precamber .....	14	
Pre-deformed FE mesh .....	10, 20	
Pre-deformed initial structures.....	16	
Pre-deformed structure by shifting nodes... 11, 21		
Profile, user .....	27	
<b>R</b>		
Reduction factor.....	13	
RF-DYNAM .....	10	
RF-IMP case .....	10, 18, 25	
RF-STABILITY.....	9, 23	
Rotation of members .....	13	
<b>S</b>		
Sets of members .....	12, 19	
Starting generation .....	16	
<b>T</b>		
Tolerance.....	16	
<b>U</b>		
Units.....	12, 27	
User profile.....	27	