Version
March 2009

Add-on Module

RF-STABILITY

Critical Load Factors
Buckling Modes

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

1. Introduction

1.1 Add-on Module RF-STABILITY

The add-on module RF-STABILITY of the main program RFEM is used for the calculation of critical load factors and corresponding buckling modes at member and surface structures. The results then form a basis for the stability design of members stressed by compression, that is usually carried out for a structure together with the general stress design.

The critical load factor of the whole structure determines the rate of system stability threat. The corresponding buckling mode predicates a region in a static model where the risk of stability violation exists. The RF-STABILITY module enables to design several buckling modes at the same time. Hence the user views the governing failure modes of a designed member or surface sorted according to the critical load factor.

The graphic display of buckling modes is useful to identify threatened regions in a structure and to make out possible structural arrangements targeted to prevent this failure. Hence the RF-STABILITY module presents a very useful tool for the design of structures endangered by buckling, e.g. slender girders and thin-walled shells. On the basis of the determined critical load factor it is firstly possible to design whether the structural stability is threatened (design of buckling, lateral buckling and plate buckling). Secondly, possible imperfections can be derived from the determined critical (lowest) buckling modes.

RF-STABILITY functional characteristics are:

- Determination of several buckling modes simultaneously in one calculation cycle.
- Automatic transfer of axial forces from a load case or load group entered in the main program RFEM.
- Possiblity to consider relieving actions of tension forces.
- Option to take into account actions of axial forces from a certain load case or load group in RFEM as the initial prestress.
- Effective equation methods for calculation of eigenvalues: the subspace iteration method for standard cases or the ICG (incomplete conjugate gradient) method that is applied in case of extensive structures with high RAM demands, at great bandwidth or in case of a large number of determined eigenmodes.
- Tabular display of the critical load factors and corresponding buckling modes.
- Graphic representation of a buckling mode using isosurfaces and isolines in the RFEM graphic interface.
- Integration into the RFEM printout report where all changes are updated automatically.
- Possibility to use the buckling modes in the add-on modules RF-IMP, RF-KAPPA and RF-TIMBER Pro.
- Direct data export to MS Excel.

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

Your DLUBAL ENGINEERING SOFTWARE team.

1.2 RF-STABILITY Team

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

Program Coordinators

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

Programmers

Doc. Dr.-Ing. Ivan Němec
Dr.-Ing. Radoslav Rusina
Ing. Jiří Doležal
Ing. Michal Balvon
Ing. Rostislav Totzauer
Ing. Petr Horák
Dr. Zbyněk Vlk
Ing. Roman Svoboda

Design of Program, Dialogs and Icons

Dipl.-Ing. Georg Dlubal
Ing. Jan Miléř

Testing

Ing. Martin Vasek
Ing. Václav Rek

Manuals, Documentation and Translations

Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) Robert Vogl
Ing. Dmitry Bystrov
Ing. Ladislav Kábrt
Mgr. Petra Pokorná
Mgr. Michaela Kryšková

Technical Support and Final Control

Dipl.-Ing. (FH) André Bergholz
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
M.Sc. Dipl.-Ing. Frank Lobisch
Dipl.-Ing. (FH) Alexander Meierhofer
Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Walter Rustler
M. Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ing. (FH) Anke Voggenreiter

1.3 Note to 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-STABILITY.

During the description of RF-STABILITY, we use the sequence and structure of the input and output masks. We feature the described **icons** (buttons) in square brackets, e.g. [Details]. The buttons are simultaneously displayed on the left margin. The **names** of dialogs, masks and particular menus are marked in *italics* in the text so that they can be easily found in the program.

We also put into this manual the index for fast finding of some terms. If you should not find the requested ones, please use the search function on our website **www.dlubal.com** to browse the *FAQ* list.

1.4 Starting RF-STABILITY

In RFEM, it is possible to initialize the add-on module RF-STABILITY in several ways.

Main Menu

You can call up RF-STABILITY by the command from the main menu of the RFEM program

Additional Modules → Stability → RF-STABILITY

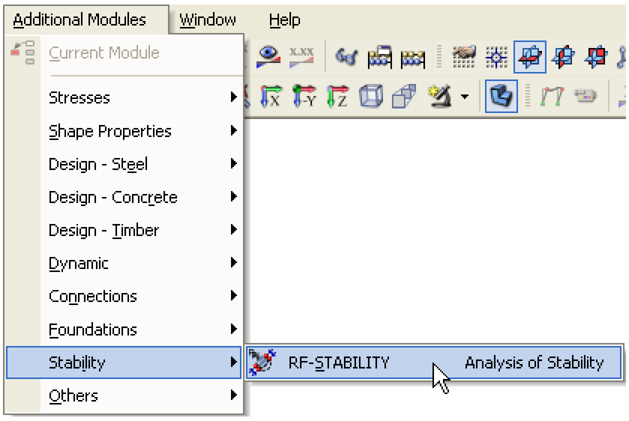


Figure 1.1: Main menu of RFEM: *Additional Modules → Stability → RF-STABILITY*

Navigator

Further, it is possible to start RF-STABILITY from the *Data* navigator by clicking the item

Additional Modules → RF-STABILITY.

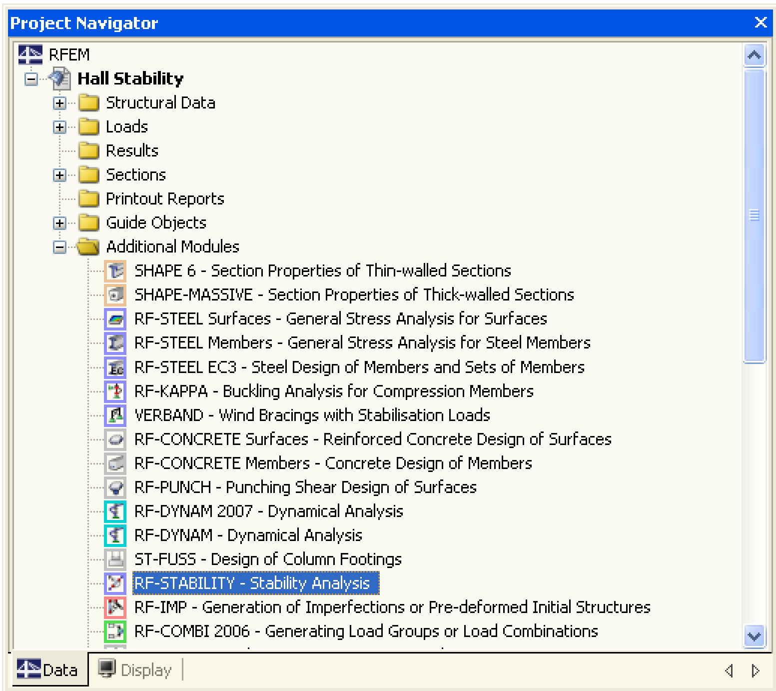
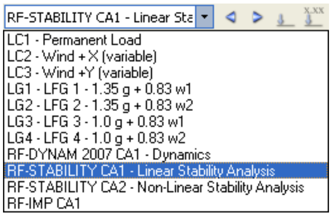


Figure 1.2: *Data* navigator : *Additional Modules → RF-STABILITY*



Panel

If results from RF-STABILITY are already available in a certain RFEM case, you can set the given case from this module in the list of load cases. The buckling modes can be displayed on the model in the graphics using the [Results on/off] button.

The [RF-STABILITY] button to access RF-STABILITY is now available in the panel.

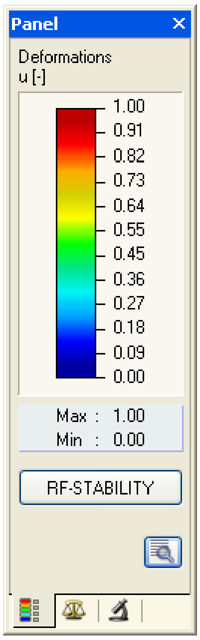


Figure 1.3: Panel: [RF-STABILITY] button

2. Input Data

All data to define stability cases is entered in a single input mask.

After initialization of the RF-STABILITY module a new window is displayed. In its left part you can see the navigator for access to all currently accessible input and output masks. The roll-out list of all possibly entered stability cases is located above the navigator (see Chapter 7.1, page 26).

If RF-STABILITY is started in a given case in RFEM for the first time, then already created load cases and load groups are automatically loaded in the add-on module.

You save entered data by the [OK] button and close the RF-STABILITY module, while by the [Cancel] button you terminate the module without data saving.

2.1 General Data

In the mask 1.1 *General Data* all the parameters for stability design are set.

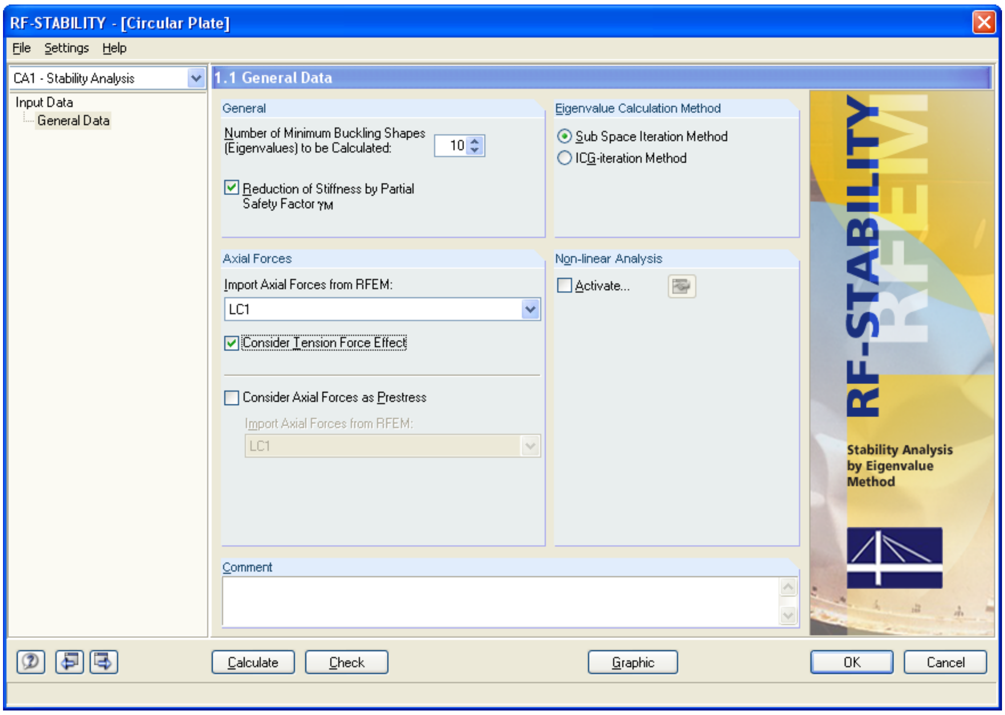


Figure 2.1: Mask 1.1 *General Data*

General

Number of Eigenvalues

RF-STABILITY calculates the least favourable structural stability modes. Their number is determined in this input field. At present, it is possible to calculate the 999 lowest eigenvalues of a structure.

The calculation theory generally does not permit to omit lower buckling shapes during the analysis and simultaneously calculate higher eigenvalues.

If negative critical load factors make the design result, the number of determined stability shapes should be increased adequately. In case that the entered number is too small, it is not possible to discard negative eigenvalues and to display only positive, realistic results. If you want to avoid negative critical load factors, you can alternatively use the method of conjugate gradients for analysis (later in this manual).

Stiffness Reduction

Using this check box you can determine whether the safety factors of used materials will be considered during the eigenvalue analysis. You can obtain the corresponding stiffness reduction by these factors. If you want to calculate eigenvalues as "characteristic" structural feature, it is not necessary to consider γ_M values of relevant materials.

Eigenvalue Calculation Method

You can use two calculation methods for the stability design.

Sub Space Iteration Method

During the subspace iteration method the stiffness matrix is saved in the computer RAM memory. When it is full, data are saved on the hard disk. Hence the computer is markedly slowed down. Therefore this method is not suitable for extensive structures.

An advantage of this method is that all eigenvalues are calculated in one step. The bandwidth of stiffness matrix has considerable influence on calculation duration.

This method should be applied in case of small and medium size structures, at which a large number of eigenvalues is necessary to calculate. At the same time you must realize that also negative critical load factors can form the calculation result.

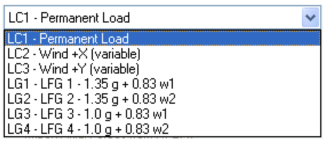
ICG-iteration Method

The method of conjugate gradients ICG (*Incomplete Conjugate Gradient*) does not impose high requirements on RAM. Only stiffness matrix coefficients that are not equal to zero are saved.

A disadvantage is that eigenvalues are calculated in sequence. The bandwidth has no influence on calculation duration.

This method should be applied in case of extensive structures, at which a smaller number of eigenvalues is determined. Negative critical load factors are not displayed after the calculation by this method, because the iteration proceeds until the prescribed number of positive eigenvalues is reached.

Axial Forces



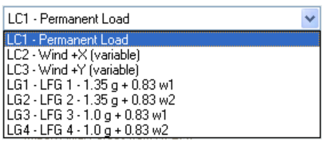
You can select a load case or load group whose axial forces should be considered during the calculation of the eigenmode. This load case or load group should be calculated according to linear first order theory.

If you do not have available results for a given load case or load group, these are calculated automatically before the proper stability design.

Consider Tension Force Effect

If you tick this box, also axial tension forces acting in a structure are considered during the calculation. Tension forces generally stabilize a structure.

Consider Axial Forces as Prestress



If you tick this box, you can select a load case or load group whose axial forces should be used as initial deformation. This option makes it for instance possible to consider the stabilizing influence of another load case (i.e. different from the load case entered earlier in the *Axial Forces* section) during the eigenvalue calculation.

Non-linear Analysis

If the stability design should run on the basis of the non-linear analysis, it is necessary to tick this box. By doing this, the button [Edit Calculation Parameters...] becomes accessible and you can open the following dialog:

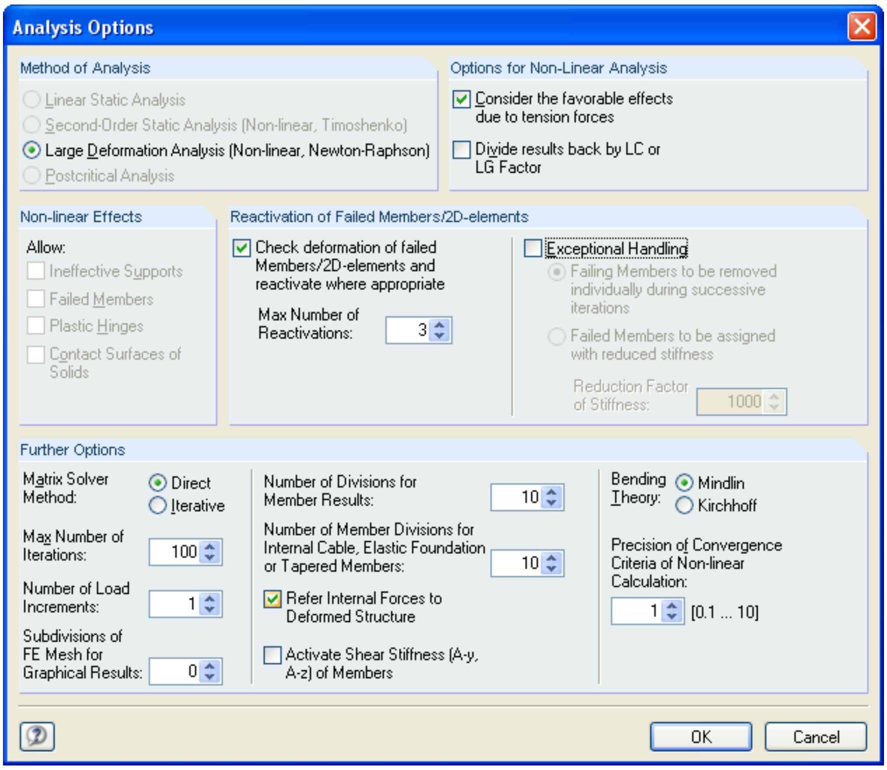


Figure 2.2: The *Analysis Options* dialog

The large deformation analysis according to NEWTON-RAPHSON is set by default and you can not change this option. Other parameters in this dialog are described in Chapter 8.3 *Calculation Parameters* in the manual to the main program RFEM on page 258.

In this method, the load is gradually increased instead of the linear eigenvalue analysis. The structure looses stability at a certain load. The critical load factor is determined in this way. In the instant of stability loss the linear analysis of eigenvalues is carried out on the structure to determine the eigenmode.

An advantage of this procedure is that you can exactly consider all non-linear elements (failed members or foundation etc.) during the calculation of the critical load factor. However, a disadvantage of this method is that you can always calculate exactly only the lowest eigenvalue. Moreover, the calculation lasts longer than for a linear analysis.

Comment

In this entry field you can write down your own remark, e.g. a detailed description of a current RF-STABILITY case. This comment is also shown in the printout report.

3. Calculation

Run the calulation by clicking the [Calculate] button. The stability design is carried out and axial forces entered in the mask 1.1 are taken into account.

3.1 Check

Before starting the calculation we recommend to quickly check the plausibility of all input data. You start the control using the [Check] button.

In case the program does not find any inconsistency, the relevant notification is displayed.

Figure 3.1: The result of plausibility check

3.2 Calculation Start

Start the calculation using the [Calculate] button.

RF-STABILITY firstly finds axial forces that should be considered. If there are no results for a given load case or load group available, the calculation of the corresponding axial forces is run automatically, based on the calculation parameters entered in RFEM.

The calculation of RF-STABILITY results can be also started from the user interface of the main program RFEM. Cases entered in add-on modules are displayed in the dialog *To Calculate* similar to load cases or load groups. You open this dialog in RFEM by the command from the main menu

Calculate → To Calculate....

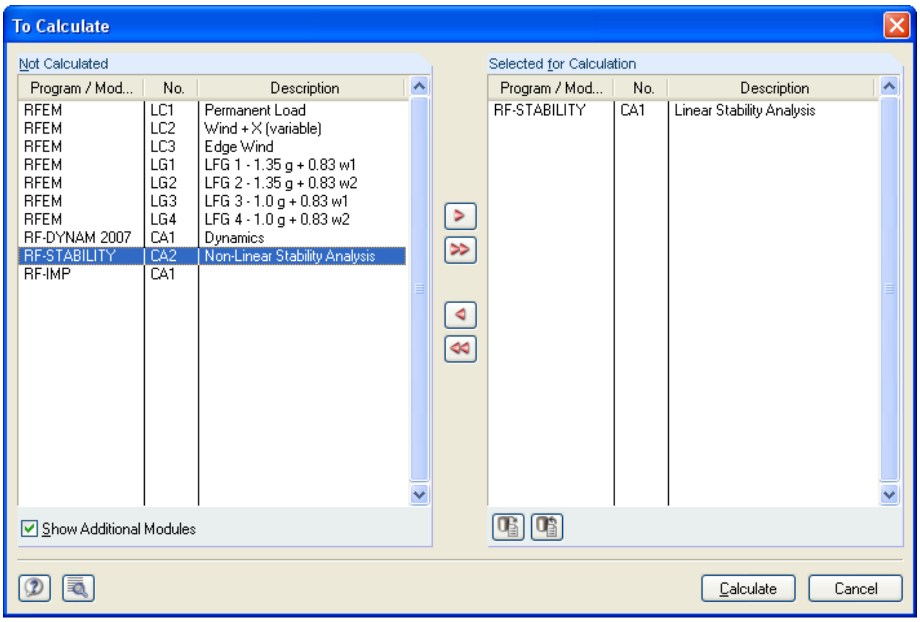


Figure 3.2: Dialog *To Calculate*

If the RF-STABILITY cases are missing in the list *Not Calculated*, it is necessary to tick the box *Show Additional Modules*.

You transfer selected stability cases to the list on the right by the [▶] button. The calculation is then started using the [Calculate] button.

The calculation of a certain RF-STABILITY case can be also run directly from the toolbar. You set the required stability case in the list and then click the button [Results on/off].

Figure 3.3: Direct calculation of a RF-STABILITY case in RFEM

The dialog where you can see the progress of the stability design is displayed consequently.

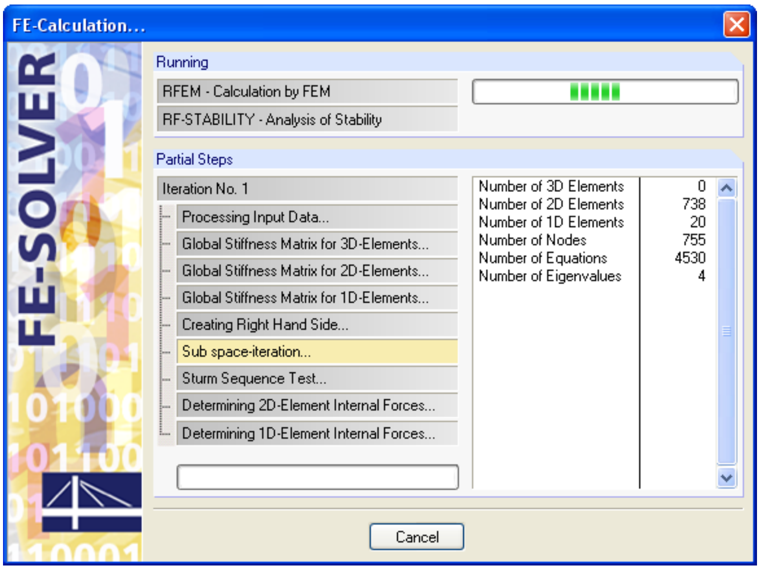
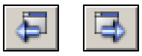


Figure 3.4: Calculation in RF-STABILITY

During the calculation via the subspace iteration method, the so-called *Sturm Sequence Test* proceeds, as you can see in the preceding picture. This test verifies whether some eigenvalue was not omitted in a certain interval. The diagonal matrix from the Gauss decomposition is used, where the number of negative diagonal matrix elements corresponds to the number of eigenvalues within the relevant interval. The Sturm test is thus carried out for the given limits of the interval. Then the difference is determined.

4. Results



The mask 2.1 *Critical Load Factors* is displayed immediately after the calculation is over. In the result masks 2.1 to 2.5 the summary of particular analyses is shown together with comments. All masks are accessible from the RF-STABILITY navigator. To browse among separate masks you can also use the buttons displayed on the left or functional keys [F2] and [F3].

You save the results by the [OK] button and close the RF-STABILITY module.

We describe in this chapter the individual masks gradually in their sequence. The check and interpretation of results is treated in Chapter 5 *Evaluation of Results* on page 20.

4.1 Critical Load Factors

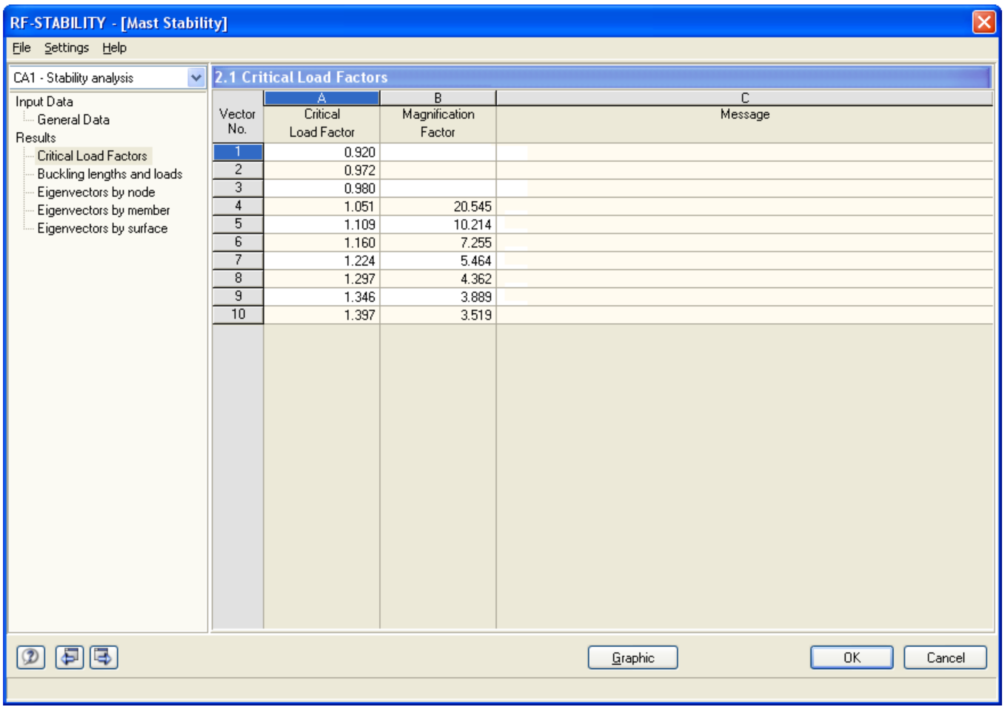


Figure 4.1: Mask 2.1 *Critical Load Factors*

The first result mask informs about critical load factors of a given structure. Values are ranked according to eigenvalues.

Vector No.

Critical load factors and magnification factors are displayed for each eigenvalue. Results are ranked upwardly according to the stability shape number.

Critical Load Factor

The critical load factor η_{Ki} is shown in the mask for every eigenvalue. A factor lower than 1.00 implies structural instability. Any factor greater than 1.00 means that the load due to entered axial forces multiplied by this factor leads to failure of the structure due to buckling or plate buckling.

According to DIN 18800, part 2, critical load factors lower than 10 require the calculation after non-linear second order theory.

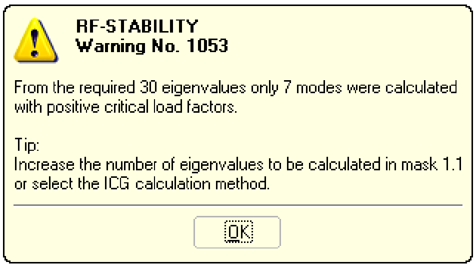


Figure 4.2: Warning after the calculation in case of negative eigenvalues

If the calculation proceeded via the subspace iteration method, negative critical load factors may be issued. Hence for the given eigenvalues no stability problem occurs due to tension forces and it is not possible to evaluate the plate buckling or buckling behavior.

Magnification Factor

The magnification factor α is calculated using the following equation:

$$\alpha = \frac{1}{1 - \frac{1}{\eta_{Ki}}}$$

Equation 4.1: Magnification factor

This magnification factor expresses the relation between the moments according to linear first order and non-linear second order theory.

$$M^{II} = \alpha \cdot M^{I}$$

where M^{I} moment according to first order theory, but equivalent load for deformation is considered

M^{II} moment according to second order theory

Equation 4.2: Moment ratio

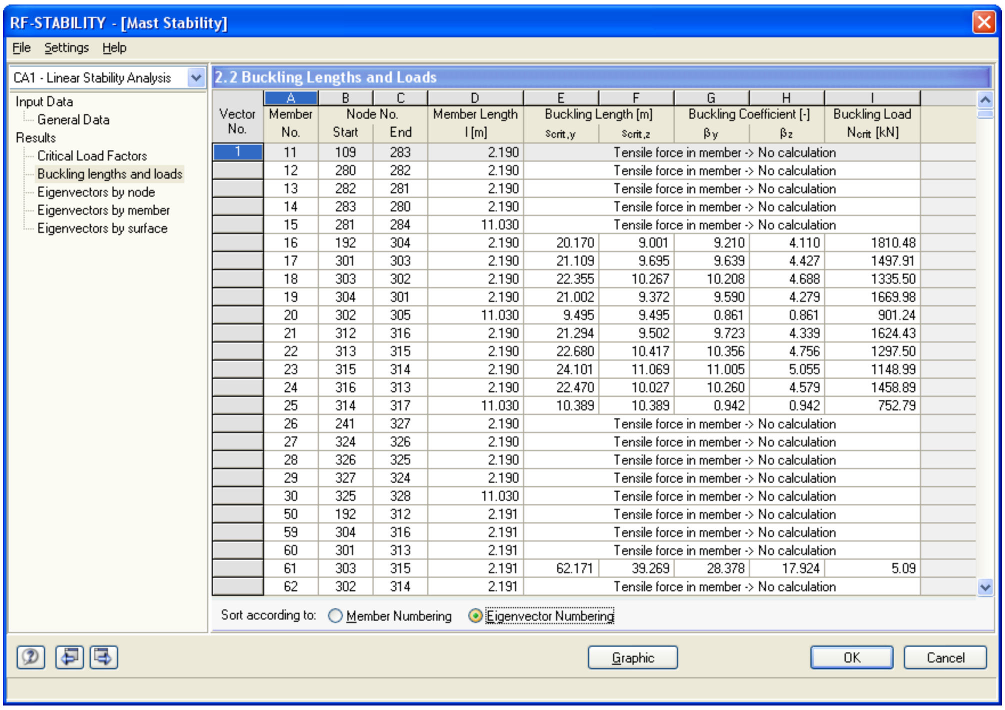
This equation is valid only in case that the bending line during load application is getting near the buckling shape and that η_{Ki} is greater than 1.00.

Message

Under certain circumstances, a message appears in column C that a negative critical load factor was calculated. This means: If the entered loads act in the opposite direction (opposite sign), failure due to buckling or plate buckling would occur.

This problem can be eliminated by increasing the number of determined eigenvalues or by applying the conjugate gradient method.

4.2 Buckling Lengths and Loads



Vector No.	A Member No.	B Node No. Start	C Node No. End	D Member Length l [m]	E Buckling Length [m] $s_{crit,y}$	F Buckling Length [m] $s_{crit,z}$	G Buckling Coefficient [-] β_y	H Buckling Coefficient [-] β_z	I Buckling Load N_{crit} [kN]
1	11	109	283	2.190	Tensile force in member -> No calculation				
	12	280	282	2.190	Tensile force in member -> No calculation				
	13	282	281	2.190	Tensile force in member -> No calculation				
	14	283	280	2.190	Tensile force in member -> No calculation				
	15	281	284	11.030	Tensile force in member -> No calculation				
	16	192	304	2.190	20.170	9.001	9.210	4.110	1810.48
	17	301	303	2.190	21.109	9.695	9.639	4.427	1497.91
	18	303	302	2.190	22.355	10.267	10.208	4.688	1335.50
	19	304	301	2.190	21.002	9.372	9.590	4.279	1669.98
	20	302	305	11.030	9.495	9.495	0.861	0.861	901.24
	21	312	316	2.190	21.294	9.502	9.723	4.339	1624.43
	22	313	315	2.190	22.680	10.417	10.356	4.756	1297.50
	23	315	314	2.190	24.101	11.069	11.005	5.055	1148.99
	24	316	313	2.190	22.470	10.027	10.260	4.579	1458.89
	25	314	317	11.030	10.389	10.389	0.942	0.942	752.79
	26	241	327	2.190	Tensile force in member -> No calculation				
	27	324	326	2.190	Tensile force in member -> No calculation				
	28	326	325	2.190	Tensile force in member -> No calculation				
	29	327	324	2.190	Tensile force in member -> No calculation				
	30	325	328	11.030	Tensile force in member -> No calculation				
	50	192	312	2.191	Tensile force in member -> No calculation				
	59	304	316	2.191	Tensile force in member -> No calculation				
	60	301	313	2.191	Tensile force in member -> No calculation				
	61	303	315	2.191	62.171	39.269	28.378	17.924	5.09
	62	302	314	2.191	Tensile force in member -> No calculation				

Figure 4.3: Mask 2.2 *Buckling Lengths and Loads*

This result mask is displayed if the structure contains members. Buckling lengths and critical loads can be listed according to *Member Numbering* or *Eigenvector Numbering.*

Member No.

The results of buckling design are displayed for all members in the structure.

Node No. - Start / End

Every member is defined by its start and end node. Their numbers are shown in two corresponding columns.

Member Length l

The geometric length of every member is displayed in this column for checking purposes.

Buckling Length $s_{crit,y}$ / $s_{crit,z}$

The buckling lengths s_{crit} are related to the critical forces N_{crit} of each member as stated in the last table column. These forces also depend on the corresponding critical load of the entire model. In simple cases, the buckling lengths represent the EULER cases 1 to 4. Therefore the buckling lengths are related to the ratio of axial forces in the member and the total critical load.

The buckling length $s_{crit,y}$ is related to the buckling perpendicularly to the "principal" member axis y, whereas $s_{crit,z}$ is related to the buckling perpendicularly to "secondary" member axis z.

Buckling Coefficient β_y / β_z

The buckling length coefficients express the ratio between the buckling length and member length with respect to the local member axes y and z.

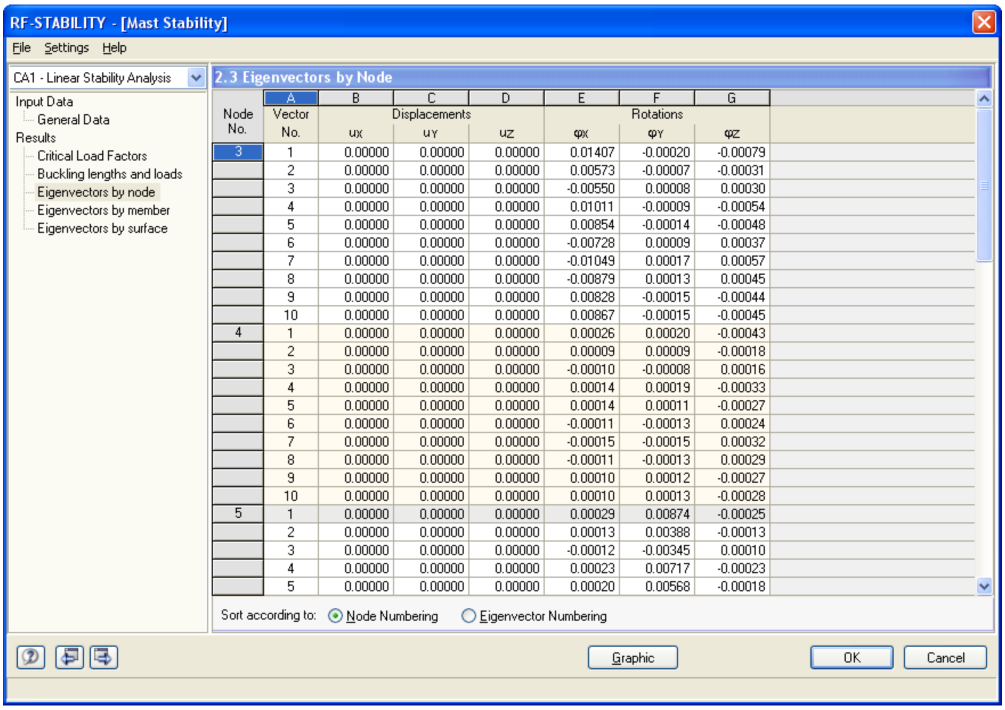
$$\beta = \frac{s_K}{l}$$

Equation 4.3: Buckling lenght coefficient β

Buckling Load N_{crit}

In this column the critical axial force N_{crit} is shown that was calculated with respect to the corresponding eigenmode. It means that all critical loads and corresponding buckling lengths must be always considered in context of the given critical load of the whole model.

4.3 Eigenvectors by Node



Node No.	A Vector No.	B Displacements ux	C uY	D uZ	E Rotations φX	F φY	G φZ
3	1	0.00000	0.00000	0.00000	0.01407	-0.00020	-0.00079
	2	0.00000	0.00000	0.00000	0.00573	-0.00007	-0.00031
	3	0.00000	0.00000	0.00000	-0.00550	0.00008	0.00030
	4	0.00000	0.00000	0.00000	0.01011	-0.00009	-0.00054
	5	0.00000	0.00000	0.00000	0.00854	-0.00014	-0.00048
	6	0.00000	0.00000	0.00000	-0.00728	0.00009	0.00037
	7	0.00000	0.00000	0.00000	-0.01049	0.00017	0.00057
	8	0.00000	0.00000	0.00000	-0.00879	0.00013	0.00045
	9	0.00000	0.00000	0.00000	0.00828	-0.00015	-0.00044
	10	0.00000	0.00000	0.00000	0.00867	-0.00015	-0.00045
4	1	0.00000	0.00000	0.00000	0.00026	0.00020	-0.00043
	2	0.00000	0.00000	0.00000	0.00009	0.00009	-0.00018
	3	0.00000	0.00000	0.00000	-0.00010	-0.00008	0.00016
	4	0.00000	0.00000	0.00000	0.00014	0.00019	-0.00033
	5	0.00000	0.00000	0.00000	0.00014	0.00011	-0.00027
	6	0.00000	0.00000	0.00000	-0.00011	-0.00013	0.00024
	7	0.00000	0.00000	0.00000	-0.00015	-0.00015	0.00032
	8	0.00000	0.00000	0.00000	-0.00011	-0.00013	0.00029
	9	0.00000	0.00000	0.00000	0.00010	0.00012	-0.00027
	10	0.00000	0.00000	0.00000	0.00010	0.00013	-0.00028
5	1	0.00000	0.00000	0.00000	0.00029	0.00874	-0.00025
	2	0.00000	0.00000	0.00000	0.00013	0.00388	-0.00013
	3	0.00000	0.00000	0.00000	-0.00012	-0.00345	0.00010
	4	0.00000	0.00000	0.00000	0.00023	0.00717	-0.00023
	5	0.00000	0.00000	0.00000	0.00020	0.00568	-0.00018

Figure 4.4: Mask 2.3 *Eigenvectors by Node*

The displacements and rotations of structural nodes are shown in this mask for every buckling mode.

Node No.

The buckling shapes are displayed in the list for structural objects defined in the mask 1.1 *Nodes* in the main program RFEM. Therefore neither the results in FE nodes nor in member division points are shown in the table.

Vector No.

The deformations are shown for every calculated eigenvector .

Displacements u_X / u_Y / u_Z

The columns B to D contain lists of displacements that are related to the axes of the global coordinate system. They are normed for every direction to the maximum *1*.

Rotations φ_X / φ_Y / φ_Z

In the columns E to G the corresponding nodal rotations of the normed displacements are displayed.

The tabular data can be sorted according to *Node Numbering* or *Eigenvector Numbering*.

If only zero values are displayed in the column with normed displacements of the member structure, it generally refers to considerable torsion in members themselves (cp. the following picture). Because this phenomenon has no influence on the displacement of member end nodes, neither displayed buckling lengths nor critical forces possess a great significance for given members.

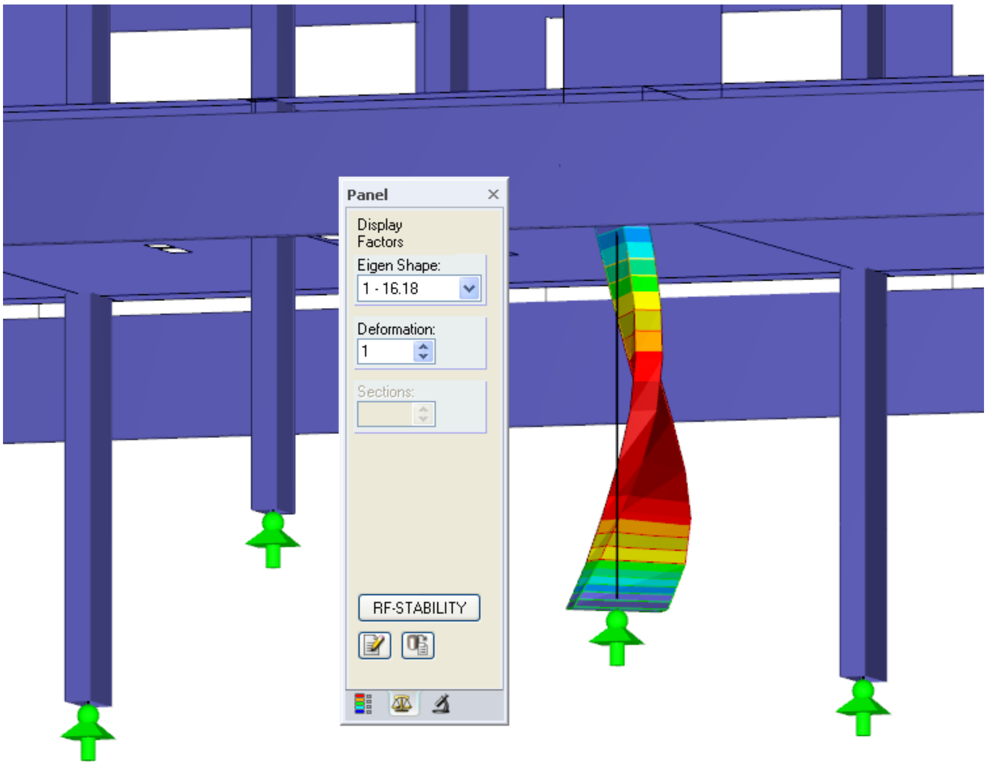
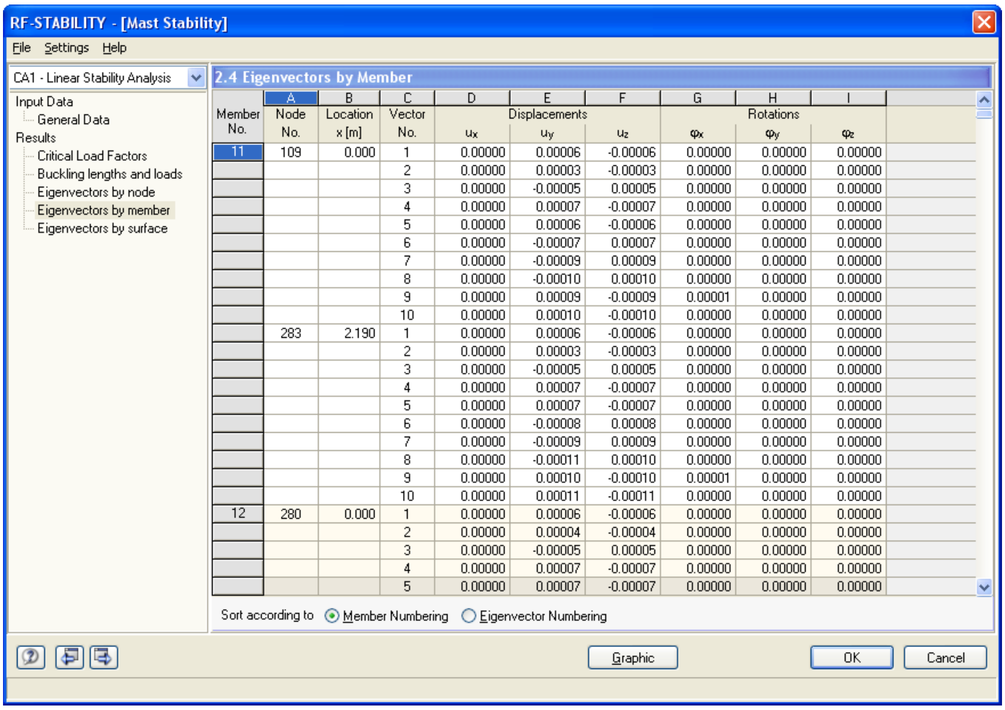


Figure 4.5: Torsion of the thin-walled column with a rectangular cross-section

4.4 Eigenvectors by Member



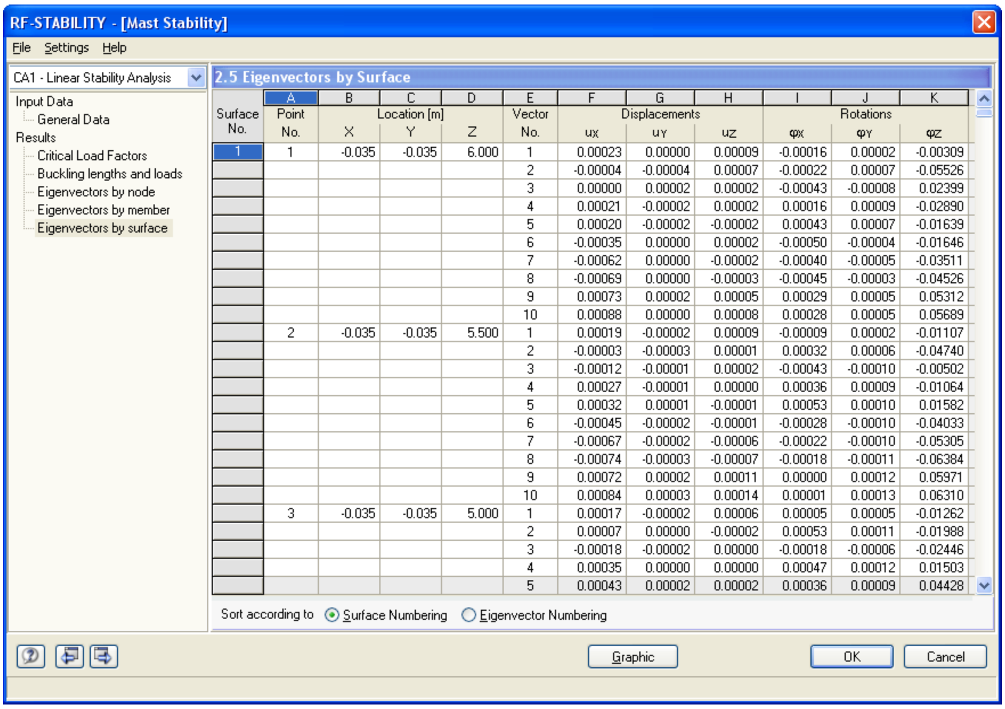
	A	B	C	D	E	F	G	H	I
Member No.	Node No.	Location x [m]	Vector No.	Displacements u_x	u_y	u_z	Rotations φ_x	φ_y	φ_z
11	109	0.000	1	0.00000	0.00006	-0.00006	0.00000	0.00000	0.00000
			2	0.00000	0.00003	-0.00003	0.00000	0.00000	0.00000
			3	0.00000	-0.00005	0.00005	0.00000	0.00000	0.00000
			4	0.00000	0.00007	-0.00007	0.00000	0.00000	0.00000
			5	0.00000	0.00006	-0.00006	0.00000	0.00000	0.00000
			6	0.00000	-0.00007	0.00007	0.00000	0.00000	0.00000
			7	0.00000	-0.00009	0.00009	0.00000	0.00000	0.00000
			8	0.00000	-0.00010	0.00010	0.00000	0.00000	0.00000
			9	0.00000	0.00009	-0.00009	0.00001	0.00000	0.00000
			10	0.00000	0.00010	-0.00010	0.00000	0.00000	0.00000
	283	2.190	1	0.00000	0.00006	-0.00006	0.00000	0.00000	0.00000
			2	0.00000	0.00003	-0.00003	0.00000	0.00000	0.00000
			3	0.00000	-0.00005	0.00005	0.00000	0.00000	0.00000
			4	0.00000	0.00007	-0.00007	0.00000	0.00000	0.00000
			5	0.00000	0.00007	-0.00007	0.00000	0.00000	0.00000
			6	0.00000	-0.00008	0.00008	0.00000	0.00000	0.00000
			7	0.00000	-0.00009	0.00009	0.00000	0.00000	0.00000
			8	0.00000	-0.00011	0.00010	0.00000	0.00000	0.00000
			9	0.00000	0.00010	-0.00010	0.00001	0.00000	0.00000
			10	0.00000	0.00011	-0.00011	0.00000	0.00000	0.00000
12	280	0.000	1	0.00000	0.00006	-0.00006	0.00000	0.00000	0.00000
			2	0.00000	0.00004	-0.00004	0.00000	0.00000	0.00000
			3	0.00000	-0.00005	0.00005	0.00000	0.00000	0.00000
			4	0.00000	0.00007	-0.00007	0.00000	0.00000	0.00000
			5	0.00000	0.00007	-0.00007	0.00000	0.00000	0.00000

Figure 4.6: Mask 2.4 *Eigenvectors by Member*

This result mask is displayed if there are members in the structural model.

The individual columns are explained in Chapter 4.3 on page 16. Additionally, the *Location x* on the member is displayed where the corresponding start or end node is situated.

4.5 Eigenvectors by Surface



Surface No.	A Point No.	B Location [m] X	C Y	D Z	E Vector No.	F Displacements ux	G uY	H uZ	I Rotations φX	J φY	K φZ
1	1	-0.035	-0.035	6.000	1	0.00023	0.00000	0.00009	-0.00016	0.00002	-0.00309
					2	-0.00004	-0.00004	0.00007	-0.00022	0.00007	-0.05526
					3	0.00000	0.00002	0.00002	-0.00043	-0.00008	0.02399
					4	0.00021	-0.00002	0.00002	0.00016	0.00009	-0.02890
					5	0.00020	-0.00002	-0.00002	0.00043	0.00007	-0.01639
					6	-0.00035	0.00000	0.00002	-0.00050	-0.00004	-0.01646
					7	-0.00062	0.00000	-0.00002	-0.00040	-0.00005	-0.03511
					8	-0.00069	0.00000	-0.00003	-0.00045	-0.00003	-0.04526
					9	0.00073	0.00002	0.00005	0.00029	0.00005	0.05312
					10	0.00088	0.00000	0.00008	0.00028	0.00005	0.05689
	2	-0.035	-0.035	5.500	1	0.00019	-0.00002	0.00009	-0.00009	0.00002	-0.01107
					2	-0.00003	-0.00003	0.00001	0.00032	0.00006	-0.04740
					3	-0.00012	-0.00001	0.00002	-0.00043	-0.00010	-0.00502
					4	0.00027	-0.00001	0.00000	0.00036	0.00009	-0.01064
					5	0.00032	0.00001	-0.00001	0.00053	0.00010	0.01582
					6	-0.00045	-0.00002	-0.00001	-0.00028	-0.00010	-0.04033
					7	-0.00067	-0.00002	-0.00006	-0.00022	-0.00010	-0.05305
					8	-0.00074	-0.00003	-0.00007	-0.00018	-0.00011	-0.06384
					9	0.00072	0.00002	0.00011	0.00000	0.00012	0.05971
					10	0.00084	0.00003	0.00014	0.00001	0.00013	0.06310
	3	-0.035	-0.035	5.000	1	0.00017	-0.00002	0.00006	0.00005	0.00005	-0.01262
					2	0.00007	0.00000	-0.00002	0.00053	0.00011	-0.01988
					3	-0.00018	-0.00002	0.00000	-0.00018	-0.00006	-0.02446
					4	0.00035	0.00000	0.00000	0.00047	0.00012	0.01503
					5	0.00043	0.00002	0.00002	0.00036	0.00009	0.04428

Figure 4.7: Mask 2.5 *Eigenvectors by Surface*

This mask is displayed only in case that the structure contains surfaces. The eigenmodes in the table can be sorted according to *Surface Numbering* or *Eigenvector Numbering*.

Point No.

The data are displayed for every surface grid point. Settings can be executed in the *Grid* register in the *Edit Surface* dialog for every surface individually. The grid with spacing of 50 cm among the points is set by default for the results.

Location X / Y / Z

In the columns B to D the location of grid points in the global coordinate system is defined.

Displacements u_X / u_Y / u_Z

In the columns F to H is shown the list of displacements that are related to the axes of the global coordinate system and normed for every direction to the maximum *1*.

Rotations φ_X / φ_Y / φ_Z

In the columns I to K the corresponding nodal rotations of the normed displacements are displayed.

5. Evaluating Results

As soon as the design ends, several possibilities for evaluation of results are offered. For graphic interpretation you have the RFEM work window available.

5.1 Result Masks

At first after the calculation is displayed the mask 2.1 *Critical Load Factors*: negative critical load factors indicate that due to tension axial forces it was impossible to determine structural failure via buckling of plate buckling. It means that in case of the same load action in the opposite direction (opposite sign) failure via buckling or plate buckling would occur. A high number of negative critical load factors implies that the structure is mainly stressed by tension.

2.1 Critical Load Factors

Vector No.	A: Critical Load Factor	B: Magnification Factor	C: Message
1	-2.083		Critical factor is negative.
2	-2.083		Critical factor is negative.
3	-2.004		Critical factor is negative.
4	-2.004		Critical factor is negative.
5	-1.984		Critical factor is negative.
6	-1.984		Critical factor is negative.
7	-1.980		Critical factor is negative.
8	-1.980		Critical factor is negative.
9	-1.937		Critical factor is negative.
10	-1.937		Critical factor is negative.

Figure 5.1: Negative critical load factors

This problem can be solved by increasing the number of determined eigenvalues or by applying the conjugate gradient method.

Any critical load factor lower than 1.00 means that the model is instable.

2.1 Critical Load Factors

Vector No.	A: Critical Load Factor	B: Magnification Factor
1	0.920	
2	0.972	
3	0.980	
4	1.051	20.545
5	1.109	10.214
6	1.160	7.255

Figure 5.2: Instable model

Only a positive critical load factor greater than 1.00 allows for the following conclusion: If you multiply the load due to axial forces by this factor, failure of the stable structure due to buckling or plate buckling occurs.

For members, different buckling length coefficients β are displayed for particular buckling modes in the mask 2.2.

2.2 Buckling Lengths and Loads

Member No.	A: Node No. Start	B: Node No. End	C: Length l [m]	D: Vector No.	E: Buckling Length [m] $s_{K,y}$	F: Buckling Length [m] $s_{K,z}$	G: Buckling Coefficient [-] β_y	H: Buckling Coefficient [-] β_z	I: Buckling Load N_{crit} [kN]
3	12	3	3.500	1	20.450	5.497	5.843	1.571	414.315
				2	20.450	5.497	5.843	1.571	414.315
				3	10.059	2.704	2.874	0.772	1712.520
				4	9.363	2.517	2.675	0.719	1976.480

Figure 5.3: Buckling length coefficients for members

During the analysis, axial forces are iteratively increased till the critical load case is reached. From the given critical load factor the critical force is then calculated, which makes it possible to determine retroactively the buckling lenghts and buckling length coefficients.

If you want to find out the governing buckling length coefficient β_y for buckling vertically to the "strong" member axis, it is usually necessary to calculate several buckling shapes. Only for square cross-sections the buckling lengths and buckling length coefficients are identical.

It is not possible to directly calculate the buckling length coefficients for sets of members in RF-STABILITY. You can analyze only results for individual members. The member at which the minimum critical force N_{cr} is displayed can be considered as governing in a member set. The values β can be then calculated from the buckling length of this member and from the total length of the member set.

5.2 Graphic Display of Results

The stability behavior of a structure can be well observed on a graphic display of different buckling shapes. The work window of the main program RFEM is used for the graphic evaluation of the eigenvalues. After clicking the [Graphic] button the RF-STABILITY module is closed. The buckling shapes are then graphically displayed in the RFEM work window similar to the deformations of a load case.

The current RF-STABILITY case is set by default. In the left part of the screen you see the modified navigator *Results*.

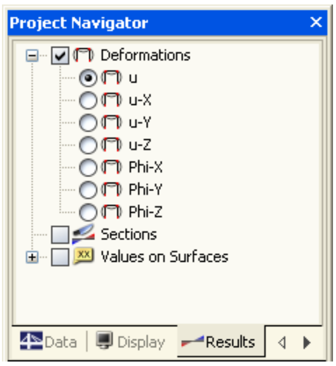


Figure 5.4: *Results* navigator for RF-STABILITY

The *Deformations* of stability modes can be displayed as results. You can also view a particular component of the total displacement or rotation.

The options *Sections* and *Values on Surfaces* can be also used for the graphic evaluation of stability shapes. These functions are described in detail in the RFEM manual, Chapter 10.6 and 10.4 on page 319 and following.

As in case of the deformations for an RFEM load case you can activate or deactivate the display of stability modes using the [Results on/off] button. Using the button on the right [Show Result Values] you can set the display of result values in the picture.

Regarding the fact that RFEM tables are not important for the evaluation of RF-STABILITY results, you can deactivate them using the button shown on the left.

Both navigator and control panel are adapted to the RF-STABILITY module. The panel standard functions are described in detail in the RFEM manual, Chapter 4.4.6 on page 77 and following.

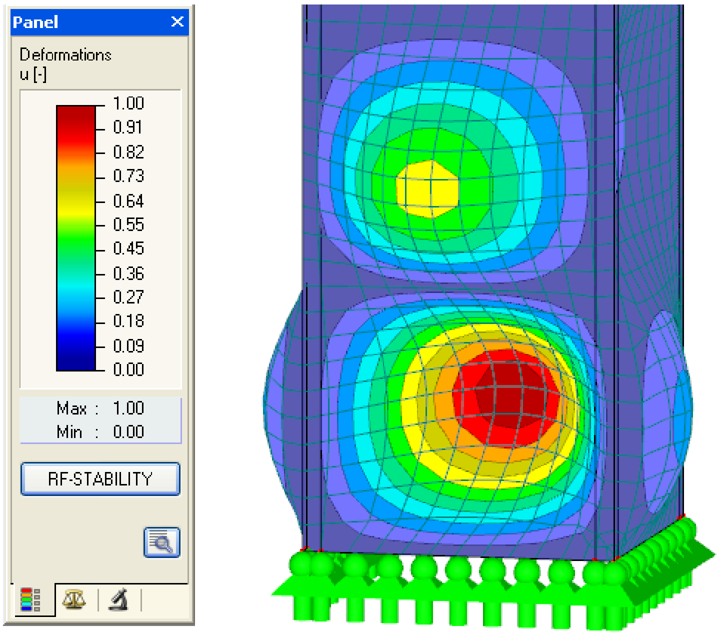


Figure 5.5: RF-STABILITY panel

In the second register of the control panel you can select particular eigenmodes.

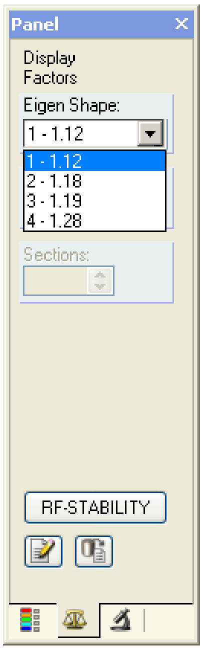


Figure 5.6: Selection of eigenshapes in the panel

At more complex structures it is often impossible to make out at first glance the buckling members or surfaces. You discern them better if you increase the *Deformation* factor in the second register of the control panel. Also can be helpful the animation of deformations which is activated by clicking the button shown on the left.

The display of results on the member can be set in the *Display* navigator in the item *Results* → *Deformation* → *Members.* The buckling shapes are displayed as *Lines* by default. Two remaining options are used to view behavior of the stucture at buckling.

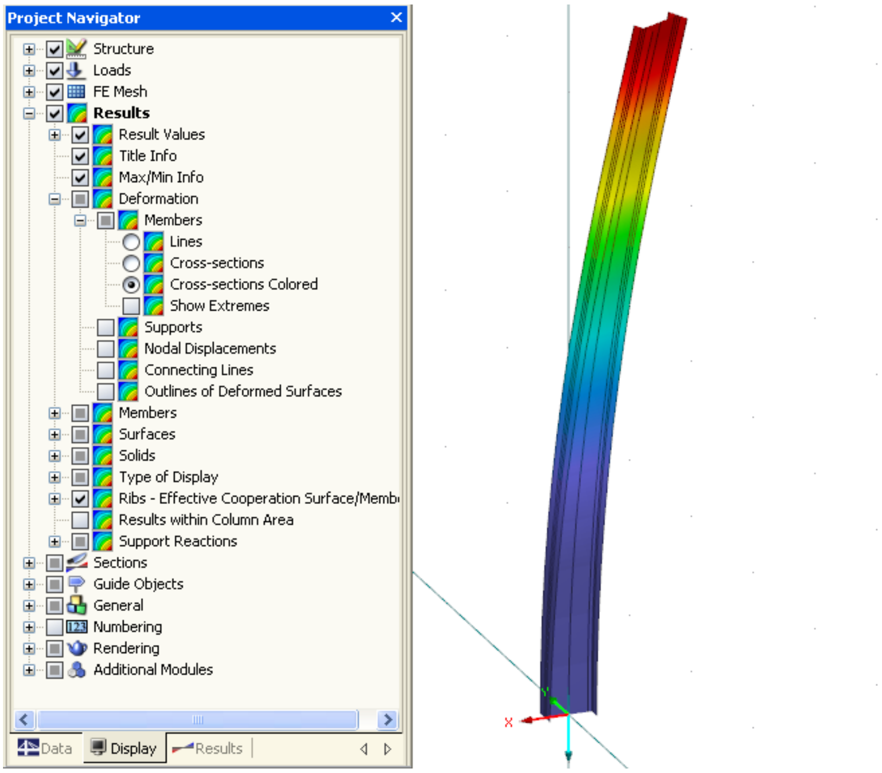
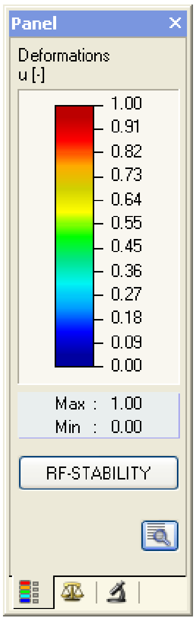


Figure 5.7: *Display* navigator: Results → Deformation → Cross-sections Colored



All graphic displays of results from RF-STABILITY can be integrated in the printout report, just like the RFEM images (see Chapter 6.2, page 25).

You can again open the RF-STABILITY module by clicking the [RF-STABILITY] button in the control panel.

5.3 Filtering Results

Besides the RF-STABILITY tables, that enable to select results according to certain criteria due to their structure, you can use for graphic evaluation of results the filtering functions, which are described in the manual of the main program RFEM.

Firstly, you can use already defined partial views (cp. the RFEM manual, Chapter 10.8.1, page 333) that suitably group certain objects. Also existing or new sections can be used to evaluate particular results (cp. the RFEM manual, Chapter 10.6, page 326).

Secondly, you can set standardized deformations as a criterion for filtering of results on the RFEM workspace. The setting of filters that you can perform in the *Color Spectrum* register is described in Chapter 4.4.6 of the RFEM manual on page 77.

In case of the colored display of results you can for example set in the panel to view only standardized deformations greater than 0.55. Hence you can more easily distinguish in an extensive structural model the surfaces and members endangered by stability loss.

Filtering of Surfaces and Members

In the *Filters* register of the control panel you can enter the numbers of members or surfaces whose deformations you wish to display in a graphic window. This function is described in detail in the RFEM manual, Chapter 4.4.6 on page 80.

Contrary to the partial view function, the entire structure is displayed here.

6. Output

6.1 Printout Report

Just like in the RFEM case, at first for RF-STABILITY data the printout report is created, where you can insert graphic displays or your own comments. In the print preview you can also choose which results of stability analyses are to be printed.

In case of extensive structures we recommend to create a few smaller printout reports instead of one large protocol. If you create a separate protocol only for RF-STABILITY data, the printout report can be processed relatively quickly.

The printout report is described in detail in the RFEM manual. Particularly important is Chapter 11.1.3.4 *Selection of Add-on Modules Data* on page 350 that deals with selection of input and output data in add-on modules.

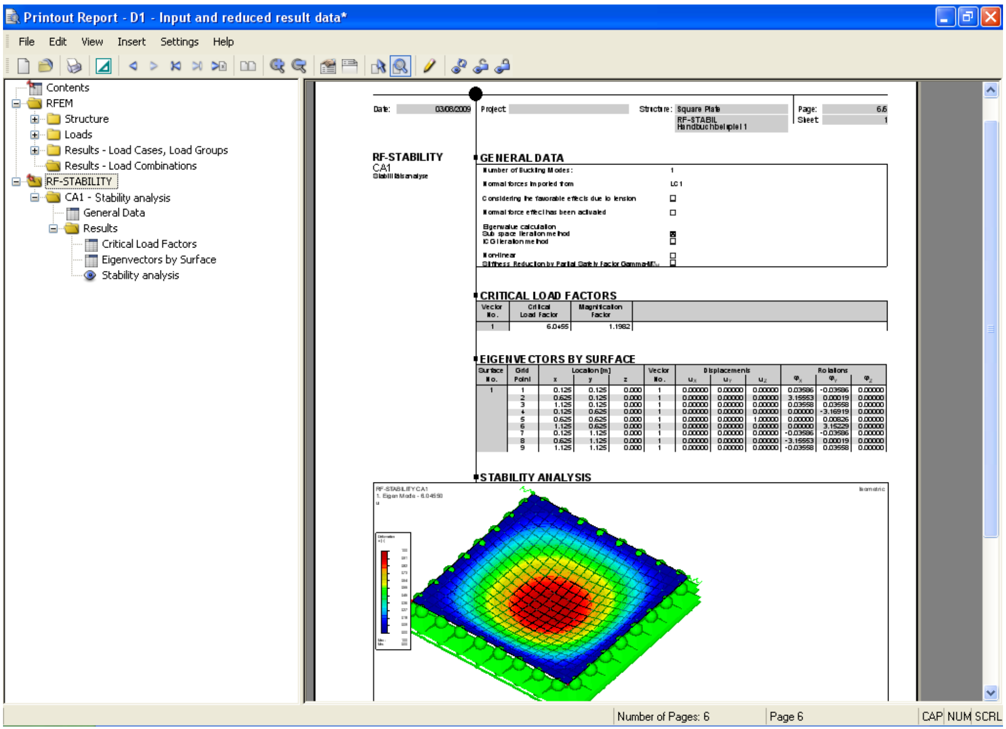


Figure 6.1: Print preview of the printout report

6.2 Printing RF-STABILITY Graphics

Graphic displays of performed analyses can be integrated in the printout report or sent directly to the printer. The printing of graphics is described in detail in Chapter 11.2 of the RFEM manual.

Every picture that is displayed in the RFEM graphic window can be inserted to the printout report. Also the sections or result diagrams on members can be transferred to the protocol by clicking the [Print] button in the respective window.

The current graphic display from RF-STABILITY in the RFEM work window can be printed by the command from the main menu

File → Print ...

or by clicking the corresponding button in the toolbar.

Figure 6.2: *Print* button in the main window toolbar

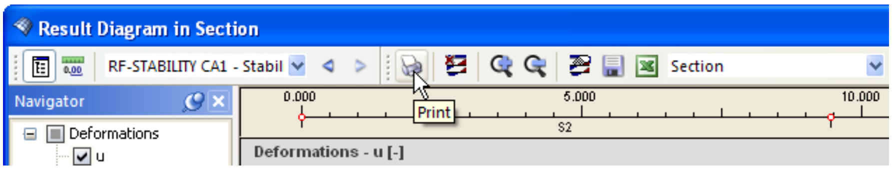


Figure 6.3: *Print* button in the toolbar in the window with result diagrams

The following dialog opens:

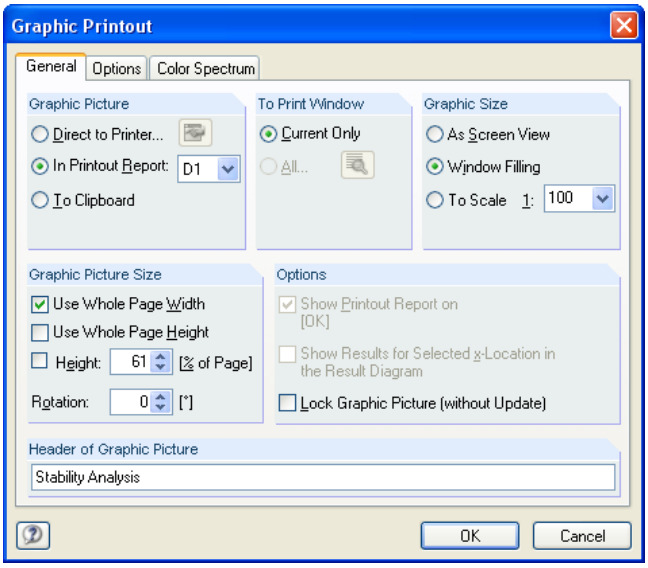
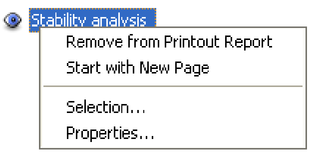


Figure 6.4: *Graphic Printout* dialog, the *General* register

This dialog is described in detail in Chapter 11.2 on page 368 in the RFEM manual. The other two registers *Options* and *Color Spectrum* are explained there as well.



The graphic display from RF-STABILITY in the printout report can be moved to a different position using the Drag&Drop function. Inserted pictures can be modified subsequently: you right-click the corresponding item in the protocol navigator and select *Properties*. The *Graphic Printout* dialog is again displayed where you can set possible changes.

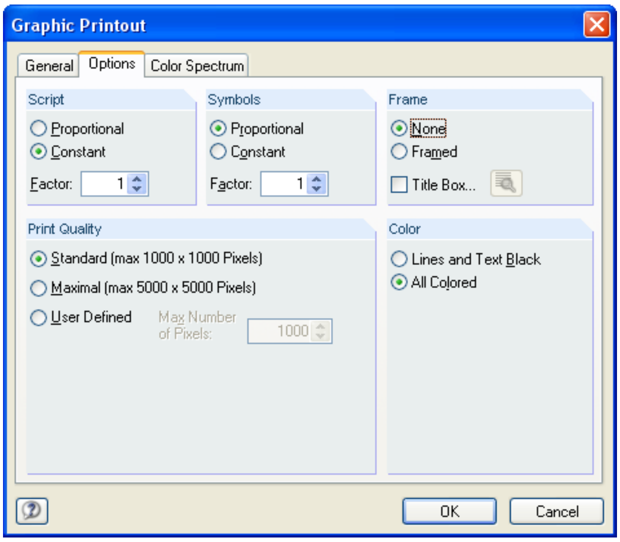


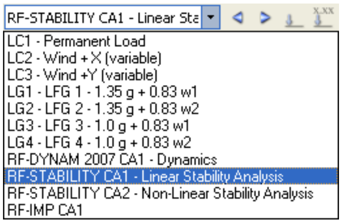
Figure 6.5: The *Graphic Printout* dialog, the *Options* register

7. General Functions

In this chapter are described commonly used functions from the main menu and also options for export of design results.

7.1 Cases in RF-STABILITY

You can design the structure in different stability cases. For example, you can analyze influence of axial forces from various load cases or load groups as initial prestress or without prestress.



The cases created in RF-STABILITY are included in the list of load cases and load groups in the toolbar in the RFEM work window.

Create New RF-STABILITY Case

A new stability case can be created by the command from the RF-STABILITY main menu

File → New Case ...

The following dialog opens:

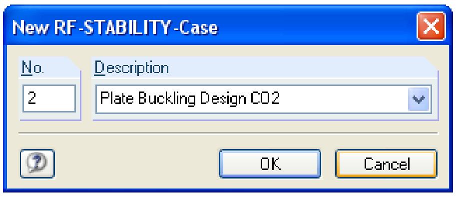


Figure 7.1: *New RF-STABILITY Case* dialog

In this dialog you need to fill in the *No.* (not used so far) and *Description* of a new case. After closing the dialog by clicking the [OK] button, the RF-STABILITY mask 1.1 *General Data* is displayed where you define new data for the stability analysis.

Rename RF-STABILITY Case

The description of the stability case can be altered by the command from the RF-STABILITY main menu

File → Rename Case ...

The dialog *Rename RF-STABILITY Case* opens.

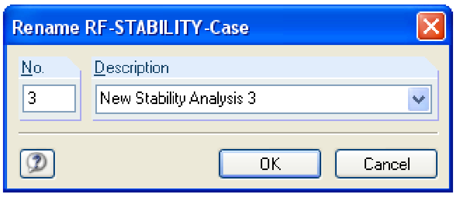


Figure 7.2: *Rename RF-STABILITY Case* dialog

Copy RF-STABILITY Case

The input data of the current stability case can be copied by the command from the RF-STABILITY main menu

File → Copy Case ...

The dialog *Copy RF-STABILITY Case* opens where you can specify the number and description of the new case to which the selected case is copied.

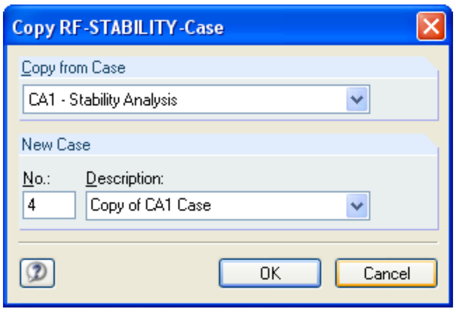


Figure 7.3: *Copy RF-STABILITY Case* dialog

Delete RF-STABILITY Case

You can delete stability cases by the command from the RF-STABILITY main menu

File → Delete Case ...

In the *Delete Cases* dialog you select a certain case from the *Available Cases* list. This case is deleted after clicking the [OK] button.

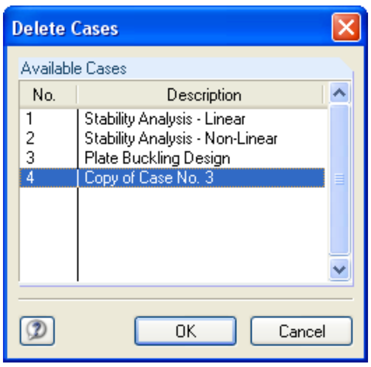


Figure 7.4: The *Delete Cases* dialog

7.2 Units and Decimal Places

The units and decimal places are centrally set for RFEM and all its add-on modules. In RF-STABILITY, you open the dialog for setting the units by the command from the main menu

Settings → Units and Decimal Places ...

The familiar RFEM dialog opens. The RF-STABILITY module is set by default here.

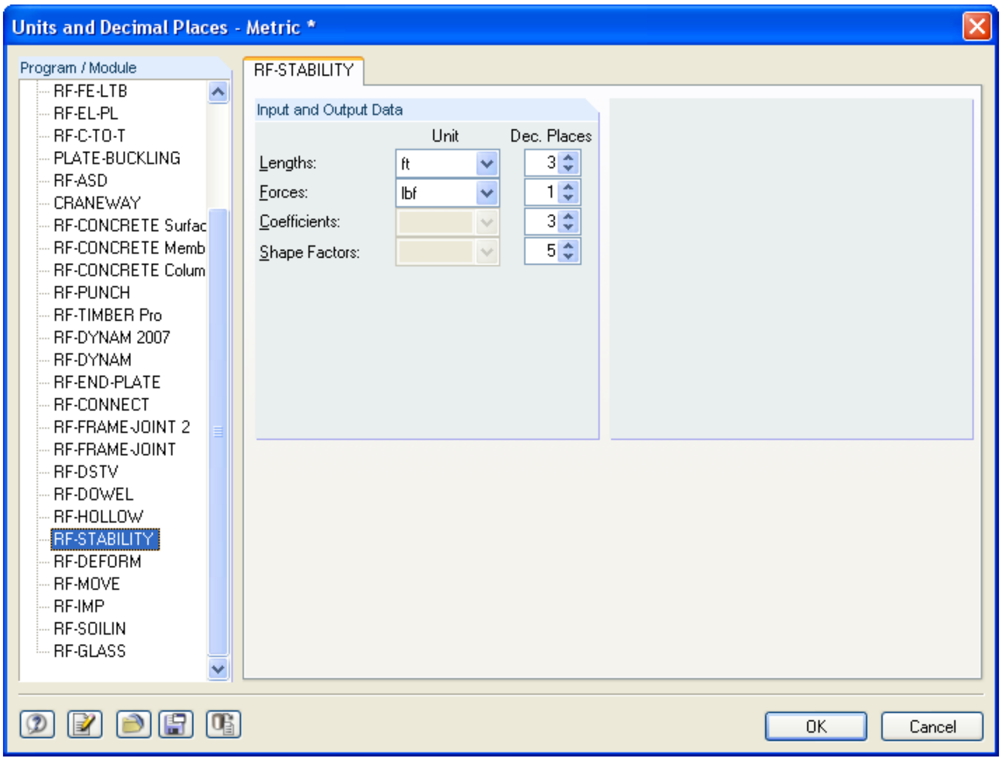
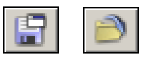


Figure 7.5: Dialog *Units and Decimal Places*



The settings can be saved as a user profile and applied in different structures as well. You find the description of this function in Chapter 12.6.2 of the RFEM manual on page 460.

7.3 Exporting Results

The results of the stability design can be exported to other programs using different ways.

Clipboard

Selected rows in the RF-STABILITY result mask can be copied to the clipboard using the buttons [Ctrl]+[C] and then transferred for example to some text processor using [Ctrl]+[V]. The column headings in the mask are not exported.

Printout Report

RF-STABILITY data can be sent to the printout report (cp. Chapter 6.1, page 24) and then be exported by the command from the main menu

File → Export to RTF File or BauText ...

This function is described in Chapter 11.1.11 on page 363 of the RFEM manual.

Excel

RF-STABILITY includes the direct data export to MS Excel. You call up this function from the main menu

File → Export to MS Excel …

The following dialog for data export opens:

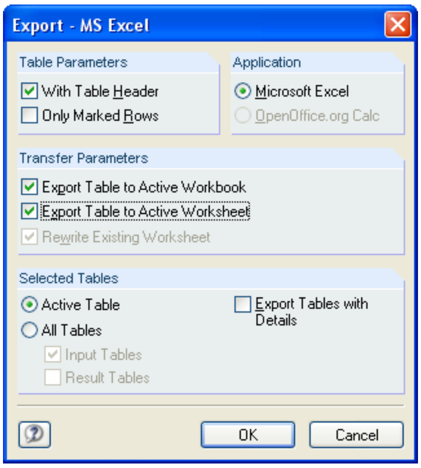


Figure 7.6: *Export - MS Excel* dialog

As soon as you have selected the required parameters, you can start the export by clicking the [OK] button. It is not necessary to run Excel in the background as it is automatically started before the export.

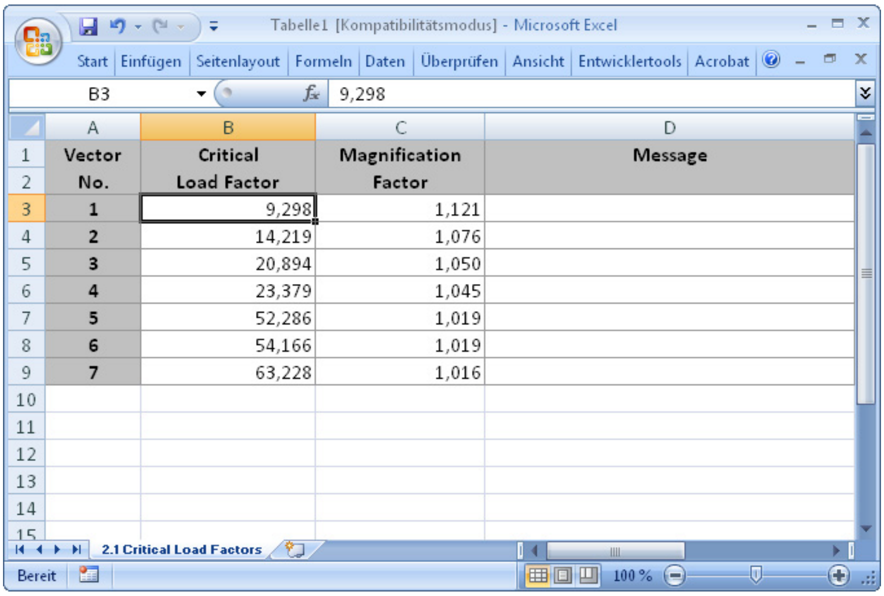


Figure 7.7: Results in Excel

RF-IMP

If a certain stability shape is to be used in the add-on module RF-IMP to generate equivalent imperfections or an equivalent structrue with initial deformation, it is not necessary to export it. The number of the required eigenvalue and the particular RF-STABILITY case can be directly selected in the relevant RF-IMP list.

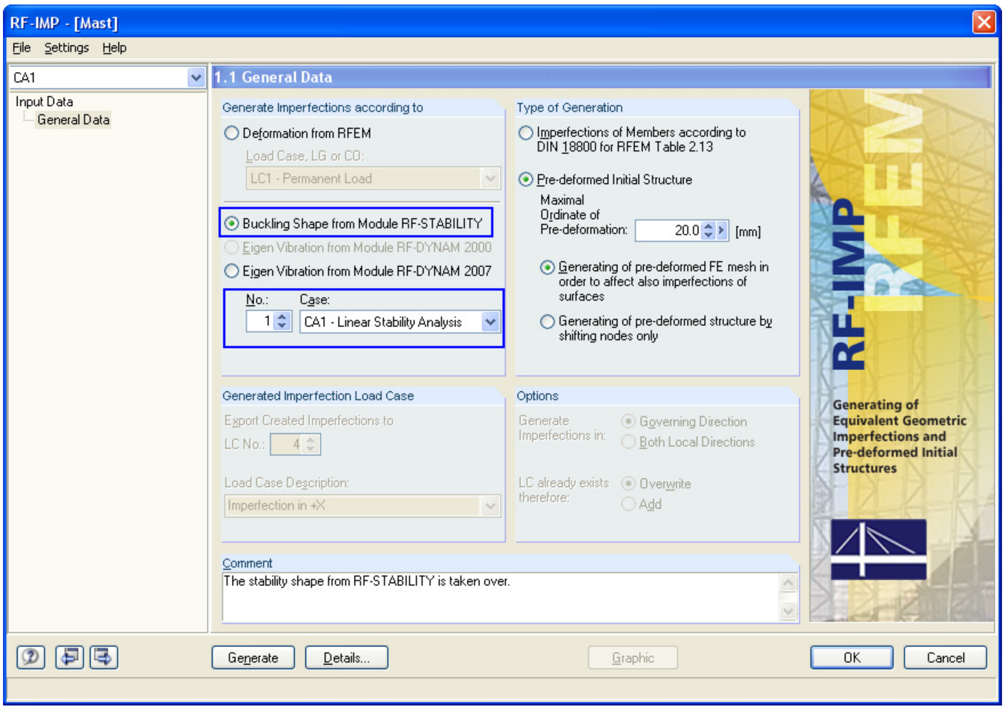


Figure 7.8: Selection of the stability shape and case from RF-STABILITY in the RF-IMP module

RF-KAPPA / RF-TIMBER Pro

In the add-on modules RF-KAPPA and RF-TIMBER Pro you can directly use buckling length coefficients from RF-STABILITY for designed members.

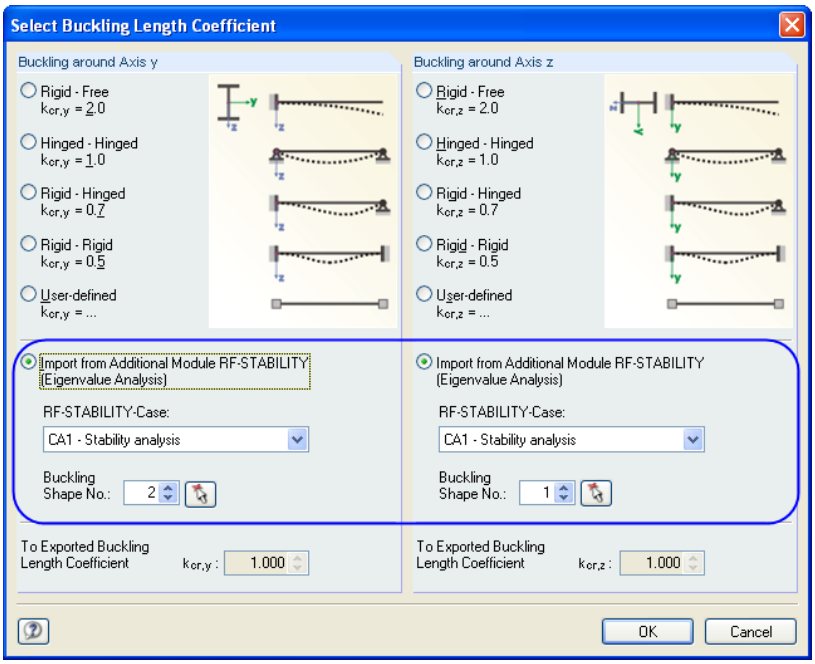


Figure 7.9: Selection of buckling length coefficients in RF-KAPPA

8. Examples

8.1 Square Plate

In this example the critical load due to buckling of a square plate with the side length 1 m is calculated.

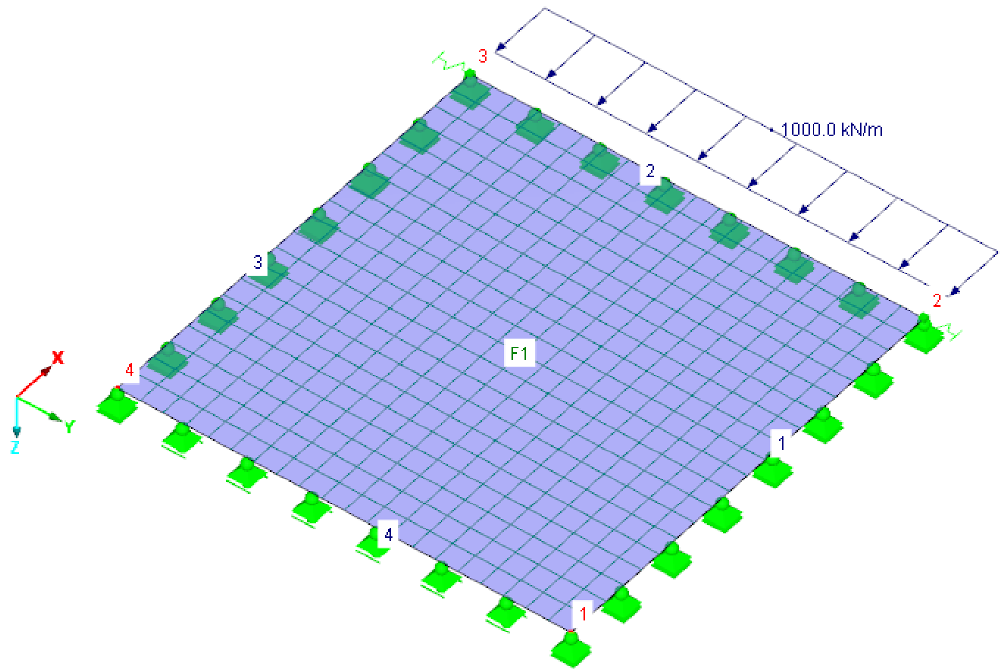


Figure 8.1: FE model and load

Analytical Solution

This surface has hinged supports on all edges. Analytically, you can obtain N_{cr} by the following formula:

$$N_{cr} = k \cdot \frac{\pi^2 \cdot E \cdot h^3}{12 \cdot (1 - \mu^2) \cdot a^2}$$

Equation 8.1

The plate is square-shaped and the length of its sides is 100 cm, hence:

$k = 4$

$a = 100$ cm

The plate is made of steel:

$E = 21000$ kN/cm^2

$\mu = 0.3$

The surface has the thickness of 2 cm:

$h = 2$ cm

Thus the critical load due to buckling is calculated as follows:

$$N_{cr} = 4 \cdot \frac{\pi^2 \cdot 21000 \cdot 2^3}{12 \cdot (1 - 0.3^2) \cdot 100^2} = 60.736 \frac{kN}{cm} = 6073.6 \frac{kN}{m}$$

RFEM Solution

The following foundation conditions are defined for the plate in RFEM:

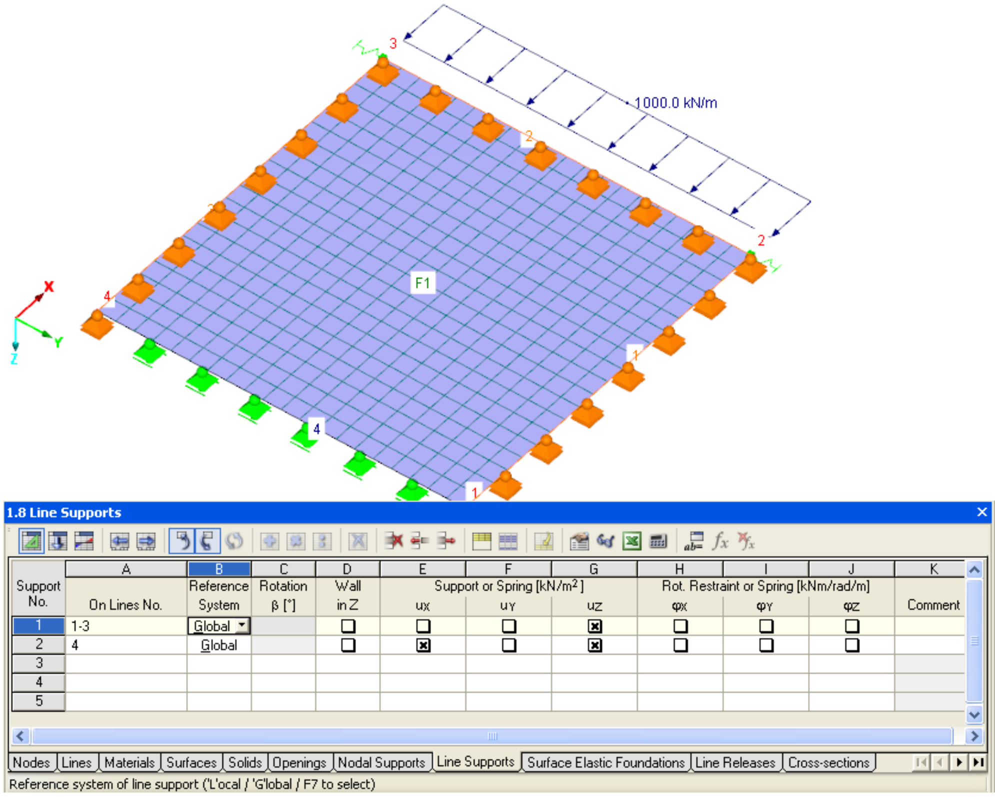


Figure 8.2: Line Supports

In addition, the point supports with low spring stiffness were defined in the nodes 2 and 3 to support the structure in the Y direction.

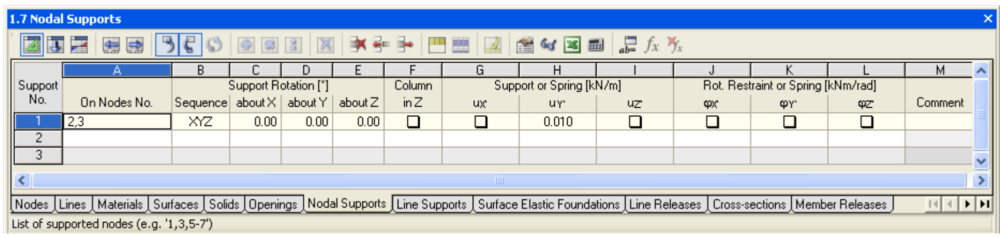


Figure 8.3: Nodal Supports

The distance among FE mesh points is 5 cm.

The square plate is loaded by the force 1000 kN/m, as you can see in figure 8.1.

You fill in the input mask of RF-STABILITY in this way:

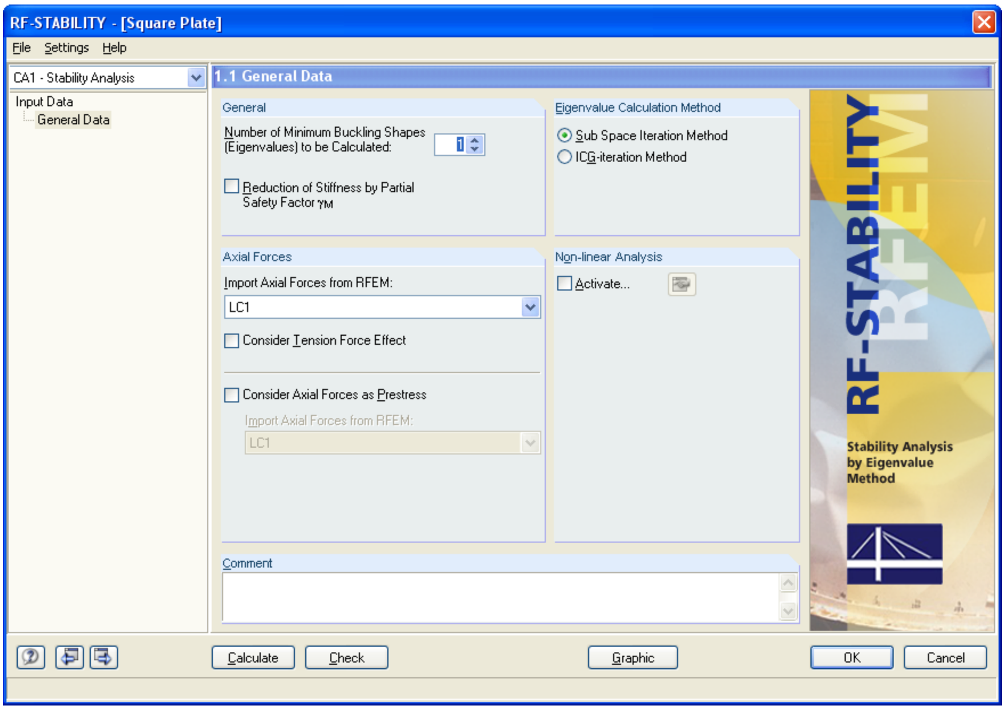


Figure 8.4: RF-STABILITY mask 1.1 *General Data*

The result of the RF-STABILITY calculation is the critical load factor 6.0455.

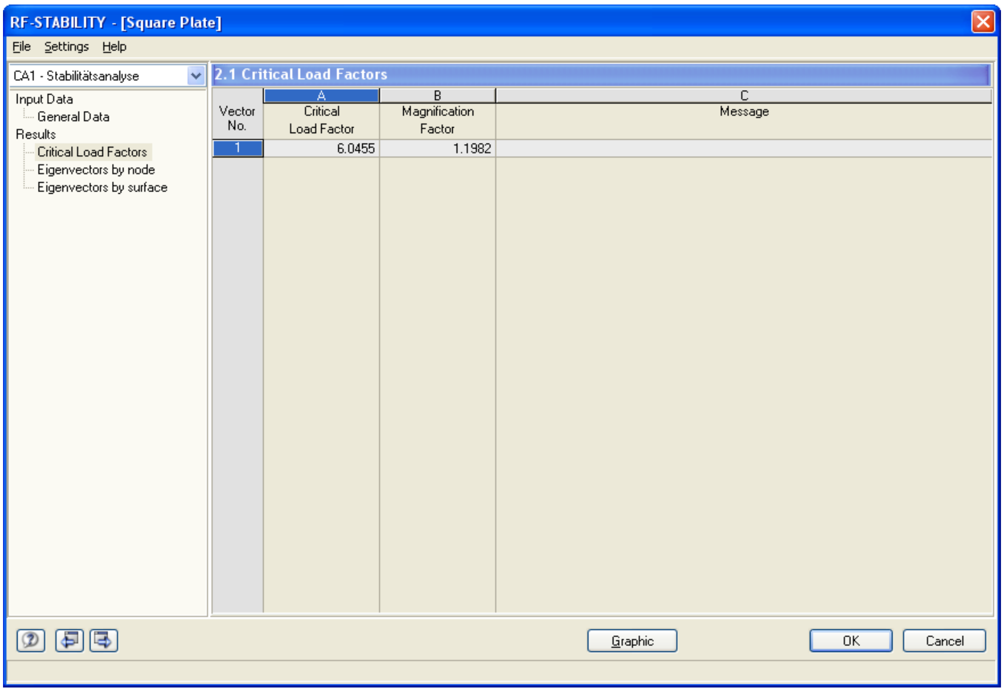


Figure 8.5: RF-STABILITY mask 2.1 *Critical Load Factors*

The critical force is obtained from the critical load factor and the applied load:

$$N_{cr} = 6.0455 \cdot 1000 \text{ kN/m} = 6045.5 \text{ kN/m}$$

The deviation from the analytical solution thus makes approximately 0.5 %.

RF-STABILITY calculates the following buckling shape:

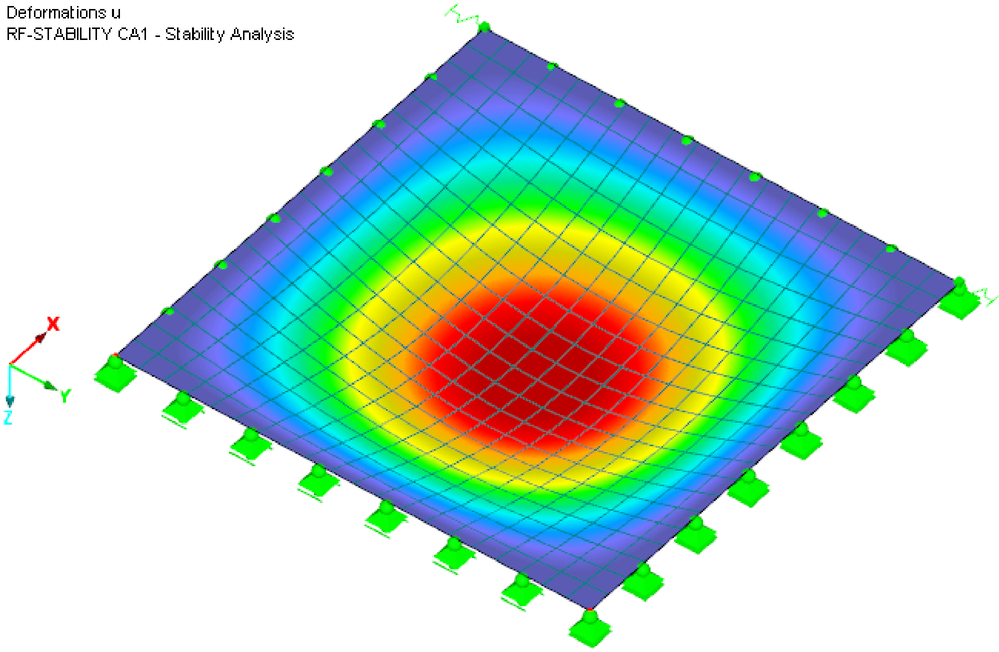


Figure 8.6: RF-STABILITY - buckling shape

8.2 Circular Plate

The critical load due to buckling of a circular plate with the diameter 3 m is calculated.

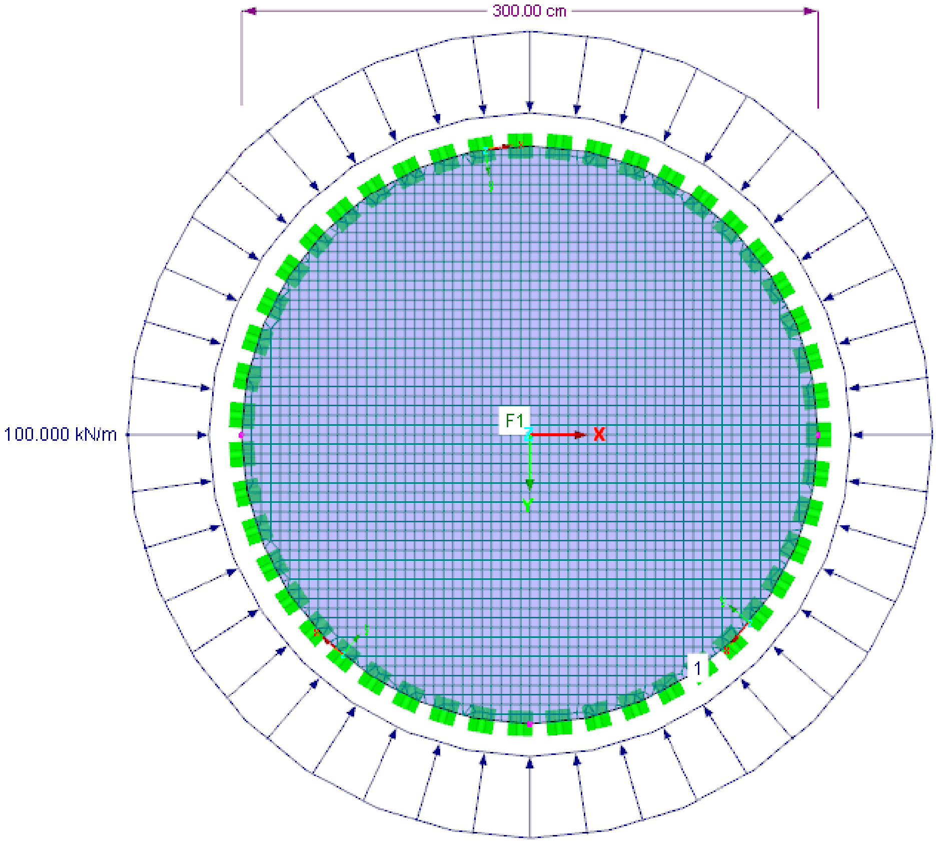


Figure 8.7: FE model and load

Analytical Solution

The surface has hinged supports around its circumference. Analytically, the critical force N_{cr} can be obtained via the following formula (according to [3], page 559):

$$N_{cr} = 14.68 \cdot \frac{E \cdot h^3}{12 \cdot (1 - \mu^2) \cdot a^2}$$

Equation 8.2

The plate has the radius 150 cm, hence:

a = 150 cm

The plate is made of steel:

E = 21000 kN/cm^2

μ = 0.3

The plate has the thickness of 2 cm:

h = 2 cm

For the critical load due to buckling then holds:

$$N_{cr} = 14.68 \cdot \frac{21000 \cdot 2^3}{12 \cdot (1 - 0.3^2) \cdot 150^2} = 10.038 \, \frac{kN}{cm} = 1003.8 \, \frac{kN}{m}$$

RFEM Solution

In RFEM, the line support is locally defined for the plate.

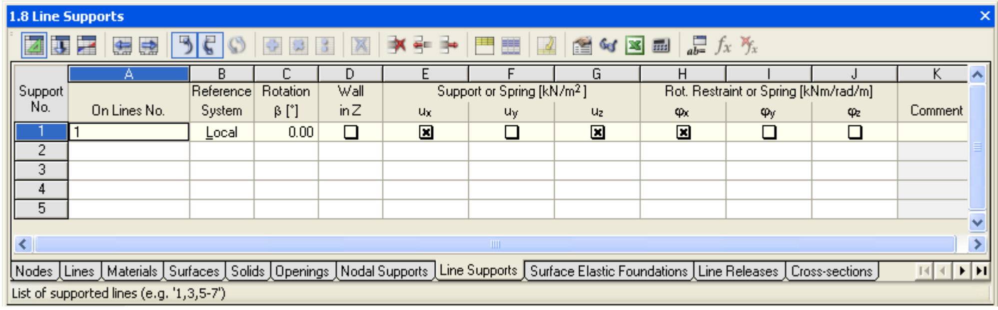


Figure 8.8: Line supports

This prevents the rotation of the plate edge. In addition, the plate is supported in the vertical direction. The support in the line x-direction restrains the plate rotation around the global axis Z.

For the FE mesh, a global target length of 5 cm was set.

The line load of 100 kN/m acts on the plate as you can see in figure 8.7.

You fill in the input mask of RF-STABILITY in this way:

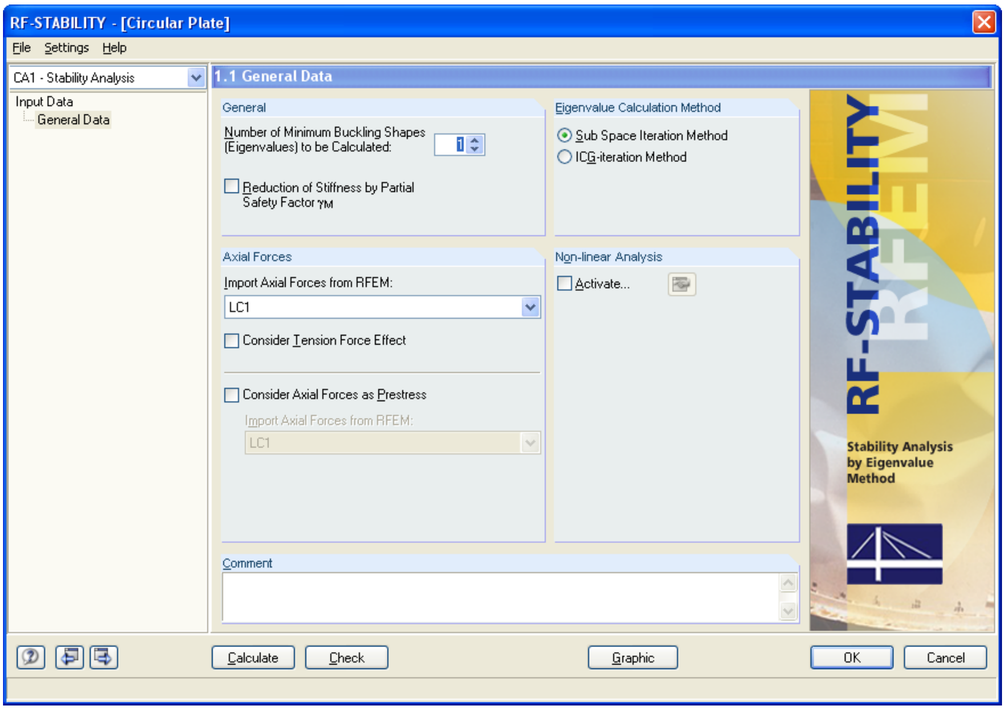


Figure 8.9: RF-STABILITY mask 1.1 *General Data*

The result of the RF-STABILITY calculation is the critical load factor 10.054.

2.1 Critical Load Factors

Vector No.	A Critical Load Factor	B Magnification Factor
1	10.054	1.110

Figure 8.10: RF-STABILITY mask 2.1 *Critical Load Factors*

The critical force is obtained from the critical load factor and applied load:

$$N_{cr} = 10.054 \cdot 100 \text{ kN/m} = 1005.4 \text{ kN/m}$$

The deviation from the analytical solution thus makes approximately 0.2 %.

RF-STABILITY calculates the following buckling shape:

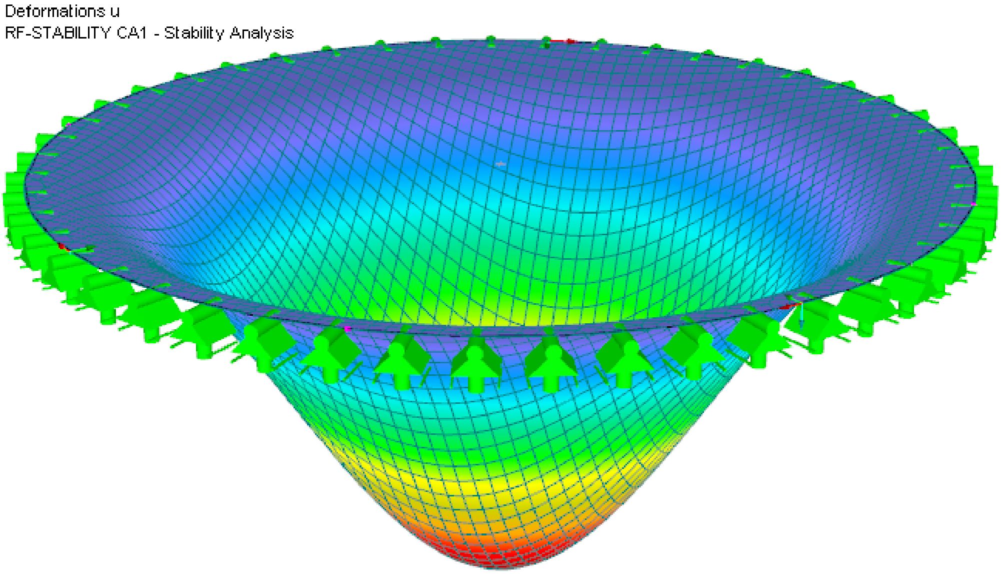


Figure 8.11: RF-STABILITY buckling shape

8.3 Tapered Cantilever

This example was adapted from [4]. In this article the failure shapes and allowable loads of cantilevered tapered T-girders are analyzed. The calculation via the finite element method is applied here.

This case represents the girder 1 of table 4.

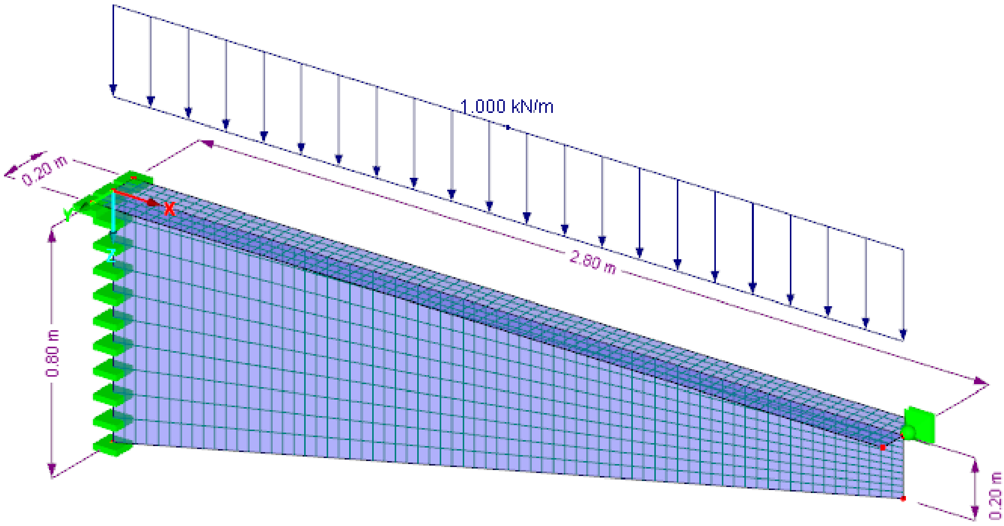


Figure 8.12: FE model and load

The girder has the following dimensions:

Length l	2800	mm
Web height h_{w0}	800	mm
Web height h_{wl}	200	mm
Flange width b	200	mm
Web thickness t_w	10	mm
Flange thickness t_f	20	mm

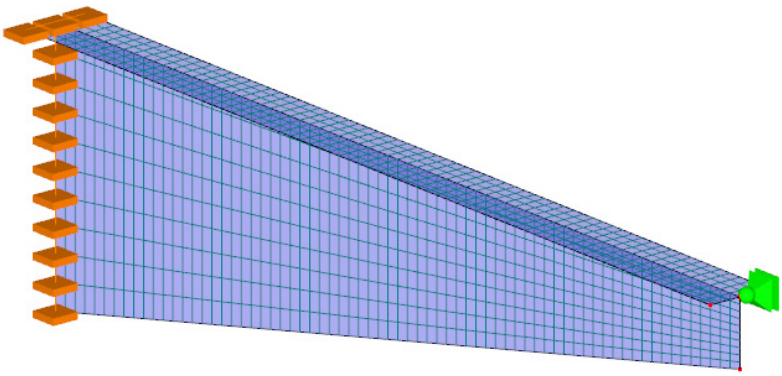
Tabulka 8.1: Geometrical data of the girder

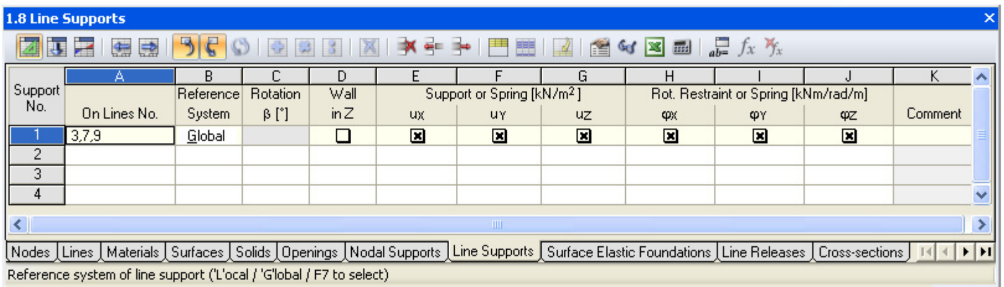
The line load 1.0 kN/m acts on the tapered girder flange.

The elastic critical force $\mathbf{q_{cr}}$ **= 43.6 kN/m** as the calculation result is stated in [4].

RFEM Solution

The following support conditions are defined for the restrained end of the girder:





1.8 Line Supports

Support No.	A On Lines No.	B Reference System	C Rotation β [°]	D Wall in Z	E Support or Spring [kN/m²] uX	F uY	G uZ	H Rot. Restraint or Spring [kNm/rad/m] φX	I φY	J φZ	K Comment
1	3,7,9	Global		☐	☒	☒	☒	☒	☒	☒	
2											
3											
4											

Figure 8.13: Girder restraint

In addition, the beam tip has a lateral support.

The required distance among the FE mesh points is 4 cm.

The line load 1 kN/m acts on the girder, as you can see in figure 8.12.

You fill in the RF-STABILITY input mask in this way:

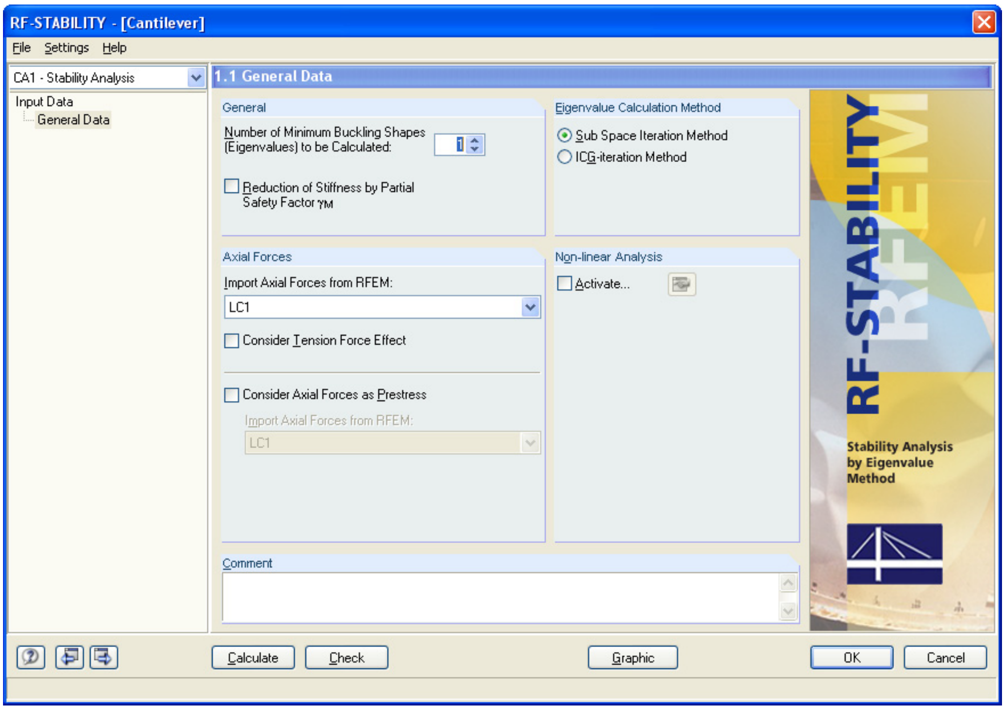


Figure 8.14: RF-STABILITY mask 1.1 *General Data*

The result of the RF-STABILITY calculation is the critical load factor 41.711.

2.1 Critical Load Factors

	A	B
Vector No.	Critical Load Factor	Magnification Factor
1	41.711	1.025

Figure 8.15: RF-STABILITY mask 2.1 *Critical Load Factors*

The critical force is obtained from the critical load factor and applied load:

$$q_{cr} = 41.711 \cdot 1.0 = 41.7 \text{ kN/m}$$

The deviation from the solution stated in [4] thus makes approximately 4 %.

RF-STABILITY calculates the following web buckling as the governing failure mode:

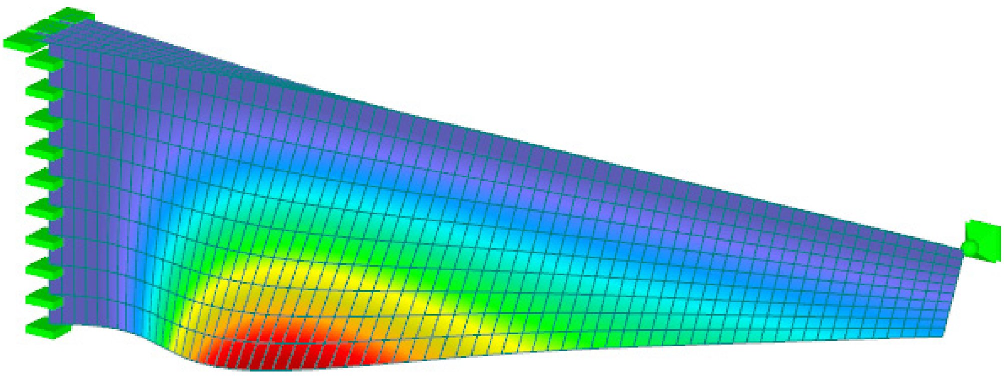


Figure 8.16: RF-STABILITY buckling shape

This result completely matches the result in the article [4].

A Literature

[1] PETERSEN, Chr.: Statik und Stabilität der Baukonstruktionen, Verlag Friedrich Vieweg und Sohn, Braunschweig/Wiesbaden, 2nd edition 1982

[2] PETERSEN, Chr.: Stahlbau, Verlag Friedrich Vieweg und Sohn, Braunschweig/Wiesbaden, 1988

[3] BAREŠ, R.: Tabulky pro výpočet desek a stěn, SNTL – Nakladatelství technické literatury, Prague 1989

[4] FISCHER, M; SMIDA, M: Dimensionierung und Nachweis von gevouteten Kragträgern mit T-förmigen Querschitt, in Stahlbau 70. Jahrgang (2001) Heft 12, S. 927-938, Ernst & Sohn, Berlin

B Index