Version
May 2014

Add-on Module

RF-STABILITY

Critical Load Factors, Effective Lengths,
Eigenvectors

Program Description

Am Zellweg 2
D-93464 Tiefenbach
Germany

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
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 performs eigenvalue analyses for member and surface models to determine critical load factors and eigenvectors (buckling modes). The critical load factor (critical buckling load factor of the global system) allows you to evaluate the stability behavior of the structural system. The corresponding eigenvector indicates the region in the model that is prone to buckling. This analysis also considers solid elements.

RF-STABILITY allows you to analyze several eigenvectors at once. After the calculation, the governing eigenvectors of the RFEM model are listed by the critical load factor. The corresponding effective lengths and critical loads are required for further stability analyses that need to be carried out for members in compression.

In the graphical representation of the eigenvectors, you can localize the regions prone to buckling and take constructive measures regarding the eigenvectors, if necessary. This makes RF-STABILITY a very useful tool for the analysis of structures prone to buckling, for example, slender beams and thin-walled shells. Based on the critical load factor, you can evaluate whether the model is generally prone to buckling (flexural buckling, flexural-torsional buckling as well as lateral-torsional buckling and plate buckling for 2D objects). Furthermore, you can apply imperfections under consideration of eigenvectors and use them for the analysis in RFEM.

Features of RF-STABILITY:

- Determination of several eigenvectors in one calculation run
- Import of axial forces from RFEM load cases or combinations
- Option to take into account favorable effects due to tension
- Option to take into account stiffness modifications from RFEM
- Eigenvalue analysis under consideration of user-defined load increments
- Determination of eigenvectors allowing you to resolve problems of instability
- Powerful solver using different eigenvalue methods with user-defined parameters
- Option to normalize eigenvectors
- Tabular display of the critical load factors and corresponding eigenvectors
- Visualization of eigenvectors including option for animation in the RFEM graphic window
- Printout in RFEM report including automatic update for changes
- Preparation of eigenvectors for the add-on modules RF-IMP, RF-KAPPA, RF-STEEL EC3, RF-STEEL AISC/AS/BS/CS/GB/IS/NTC-DF/SANS/SIA/SP, RF-ALUMINIUM, and RF-TIMBER Pro
- Data export to MS Excel, OpenOffice Calc, or the CSV file format

We hope you enjoy working with RF-STABILITY.

Your team from DLUBAL SOFTWARE, INC.

1.2 Using the Manual

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

The sequence and structure of the manual follows the input and results windows of the module. In the text, the **buttons** are given in square brackets, for example [Edit]. At the same time, they are pictured on the left. The **Expressions** that appear in dialog boxes, windows, and menus are set in *italics* making the connection between the explanations in the manual and the program clearer.

At the end of the manual, you find the index. If you still cannot find what you are looking for, check our website https://www.dlubal.com/en, where you can go through the *FAQ* pages and use various filter criteria.

1.3 Opening the RF-STABILITY Module

In RFEM, you have the following possibilities to start the add-on module RF-STABILITY.

Menu

To open the add-on module, you can select on the RFEM menu

Add-on Modules → Stability → RF-STABILITY

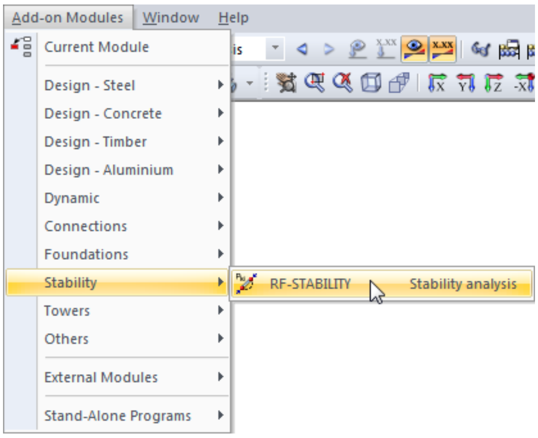
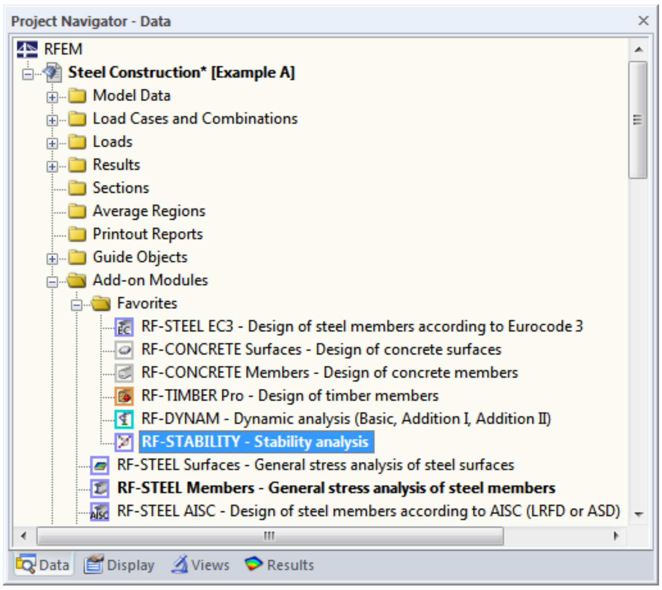


Figure 1.1: Menu: *Add-on Modules → Stability → RF-STABILITY*

Navigator

Alternatively, you can open the add-on module in the *Data* navigator by clicking

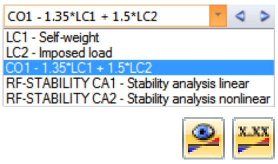
Add-on Modules → **RF-STABILITY**.



Project Navigator - Data
RFEM
Steel Construction* [Example A]
Model Data
Load Cases and Combinations
Loads
Results
Sections
Average Regions
Printout Reports
Guide Objects
Add-on Modules
Favorites
RF-STEEL EC3 - Design of steel members according to Eurocode 3
RF-CONCRETE Surfaces - Design of concrete surfaces
RF-CONCRETE Members - Design of concrete members
RF-TIMBER Pro - Design of timber members
RF-DYNAM - Dynamic analysis (Basic, Addition I, Addition II)
RF-STABILITY - Stability analysis
RF-STEEL Surfaces - General stress analysis of steel surfaces
RF-STEEL Members - General stress analysis of steel members
RF-STEEL AISC - Design of steel members according to AISC (LRFD or ASD)

Figure 1.2: Data navigator: *Add-on Modules* → *RF-STABILITY*

Panel



CO1 - 1.35*LC1 + 1.5*LC2
LC1 - Self-weight
LC2 - Imposed load
CO1 - 1.35*LC1 + 1.5*LC2
RF-STABILITY CA1 - Stability analysis linear
RF-STABILITY CA2 - Stability analysis nonlinear

If results from RF-STABILITY are already available in the RFEM model, you can also open the design module from the panel:

First, set the relevant RF-STABILITY case in the load case list of the RFEM toolbar. Then, click the [Show Results] button to graphically display the eigenvector.

In the panel, you can click [RF-STABILITY] to open the module.

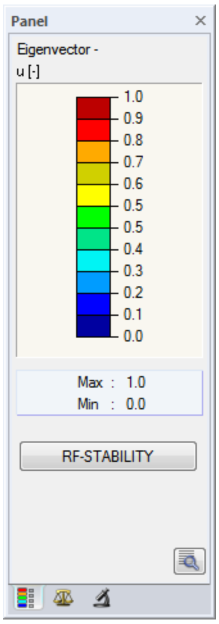


Figure 1.3: Panel button [RF-STABILITY]

2 Input Data

When you start the add-on module, a new window opens. In this window, a navigator is displayed on the left that manages the available input and output windows. The drop-down list above the navigator contains the analysis cases (see Chapter 7.1, page 28).

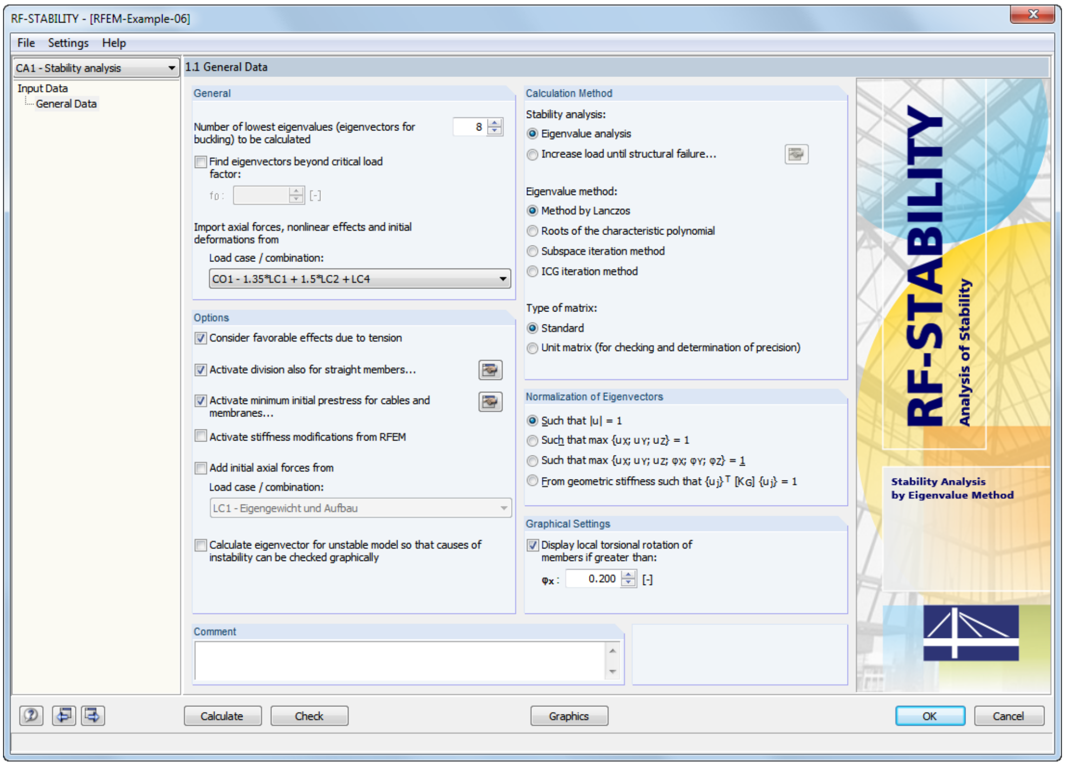
When you open RF-STABILITY for the first time, the created load cases and combinations are imported automatically.

To select a module window, click the corresponding entry in the navigator. To go to the previous or next input window, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] window.

To save the results, click [OK]. Thus, you exit RF-STABILITY and return to the main program. To exit the module without saving the new data, click [Cancel].

2.1 General Data

In the 1.1 *General Data* window, you select the members, sets of members, and actions that you want to analyze. It is the only input window of the module.



General

Number of lowest eigenvalues (eigenvectors for buckling) to be calculated: 8

☐ Find eigenvectors beyond critical load factor:

f0: [-]

Import axial forces, nonlinear effects and initial deformations from

Load case / combination: CO1 - 1.35*LC1 + 1.5*LC2 + LC4

Options

☑ Consider favorable effects due to tension

☑ Activate division also for straight members...

☑ Activate minimum initial prestress for cables and membranes...

☐ Activate stiffness modifications from RFEM

☐ Add initial axial forces from

Load case / combination: LC1 - Eigengewicht und Aufbau

☐ Calculate eigenvector for unstable model so that causes of instability can be checked graphically

Comment

Calculation Method

Stability analysis:

● Eigenvalue analysis

○ Increase load until structural failure...

Eigenvalue method:

● Method by Lanczos

○ Roots of the characteristic polynomial

○ Subspace iteration method

○ ICG iteration method

Type of matrix:

● Standard

○ Unit matrix (for checking and determination of precision)

Normalization of Eigenvectors

● Such that $|u| = 1$

○ Such that $\max\{u_X; u_Y; u_Z\} = 1$

○ Such that $\max\{u_X; u_Y; u_Z; \varphi_X; \varphi_Y; \varphi_Z\} = 1$

○ From geometric stiffness such that $\{u_j\}^T [K_G] \{u_j\} = 1$

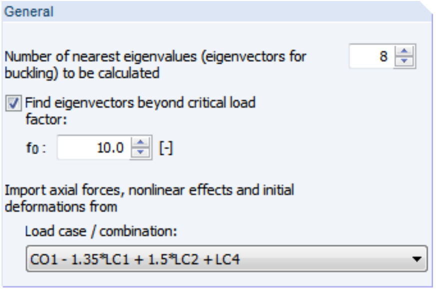
Graphical Settings

☑ Display local torsional rotation of members if greater than:

φ_x: 0.200 [-]

Figure 2.1: Window 1.1: *General Data*

General



General

Number of nearest eigenvalues (eigenvectors for buckling) to be calculated: 8

☑ Find eigenvectors beyond critical load factor:

f0: 10.0 [-]

Import axial forces, nonlinear effects and initial deformations from

Load case / combination: CO1 - 1.35*LC1 + 1.5*LC2 + LC4

Figure 2.2: Definition of eigenvectors and axial forces

Number of smallest (or next) eigenvalues

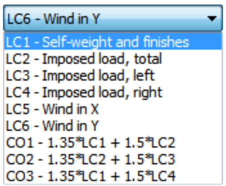
RF-STABILITY determines the most unfavorable eigenvalues of the model, whose number you specify in this text box. The upper limit is 1,000 eigenvalues (and thus eigenvectors) – that is, if the number of degrees of freedom and the RAM allow it.

If negative critical load factors are shown after running the analysis, you should increase the number of the eigenvectors to be determined. For a small number, it is not possible to hide the negative eigenvalues in order to show only positive, realistic results. Alternatively, you can use the ICG (incomplete conjugate gradient) method (see below) to exclude the negative critical load factors.

Find eigenvectors beyond critical load factor

By using this function, RF-STABILITY can show you only those eigenvectors that exceed a certain eigenvalue. To do this, select the corresponding check box, and then specify the minimum load increment f_0 in the text box.

Import axial forces, nonlinear actions, and initial deformations from load case / combination


LC6 - Wind in Y
LC1 - Self-weight and finishes
LC2 - Imposed load, total
LC3 - Imposed load, left
LC4 - Imposed load, right
LC5 - Wind in X
LC6 - Wind in Y
CO1 - 1.35*LC1 + 1.5*LC2
CO2 - 1.35*LC2 + 1.5*LC3
CO3 - 1.35*LC1 + 1.5*LC4

The list contains all load cases and load combinations of the model. Select an entry. The axial forces and, if necessary, the stiffnesses of these actions are taken into account when determining the eigenvector. This load case or this combination should be calculated in RFEM by linear static analysis without stiffness reductions (material, cross-section, member, surface).

Result combinations are not available in the list, because due to the max/min extreme values they don't show unambiguous distributions of axial forces.

In addition to the axial forces, the stiffness conditions are also imported from RFEM. This means: The module imports the structural system after the load case or the load combination is calculated in RFEM. This system is the starting point of the eigenvalue analysis. In this way, you can approximately consider nonlinear effects as, for example, failing members, supports, and releases, or member nonlinearities in the linear eigenvalue analysis in RF-STABILITY.

If no results exist for the load case or load combination, the internal forces are automatically determined before the RF-STABILITY analysis.

When calculating the eigenvector of a beam structure and the effective lengths, the loading is of decisive importance, because the buckling values depend not only on the structural model, but also on the ratio of the axial forces to the total critical buckling load N_{cr}. Therefore it is recommended to specify a load case with a full vertical loading (without wind) so that most of the members are subjected to compressive forces.

Options

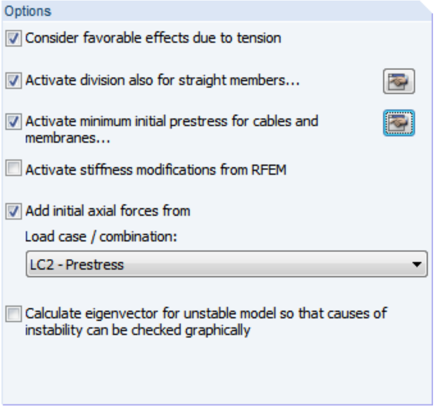

Options
☑ Consider favorable effects due to tension
☑ Activate division also for straight members...
☑ Activate minimum initial prestress for cables and membranes...
☐ Activate stiffness modifications from RFEM
☑ Add initial axial forces from
Load case / combination:
LC2 - Prestress
☐ Calculate eigenvector for unstable model so that causes of instability can be checked graphically

Figure 2.3: Options

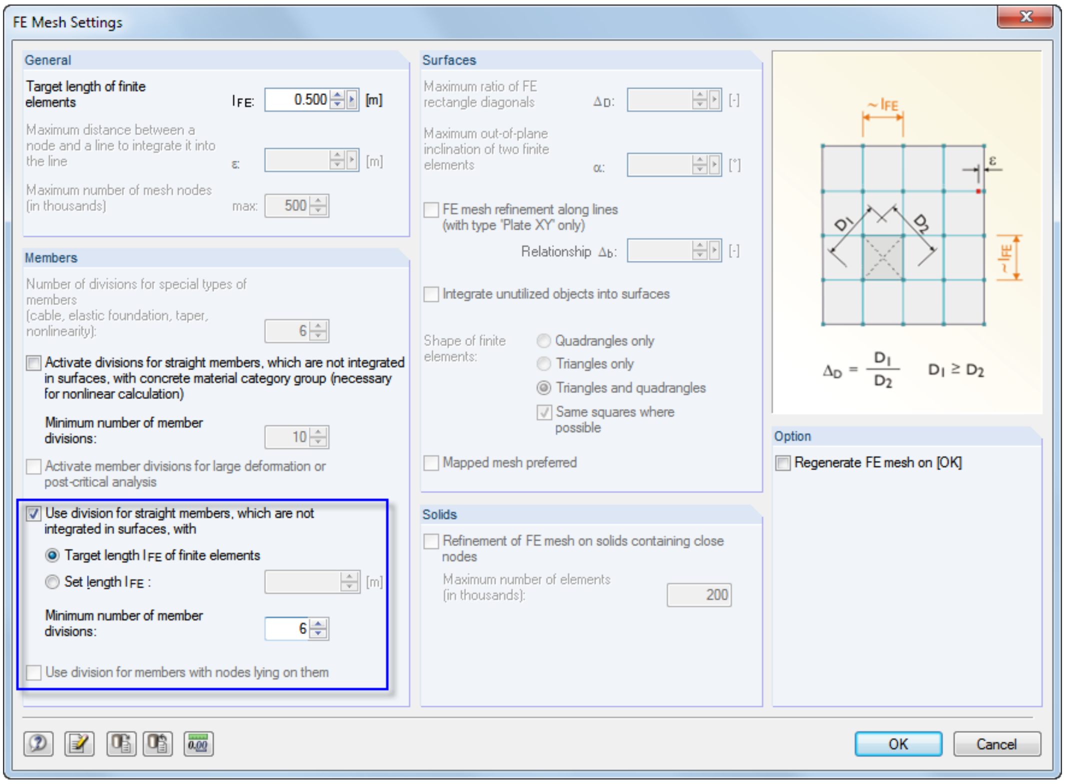
Consider favorable effects due to tension

If the check box is selected, the tensile forces in the model are taken into account during the determination of the eigenvalue. Usually, they stabilize the model.

Activate division also for straight members

For better approximation solutions, it can be necessary to refine the member division, in particular for tapers and members with elastic foundations.

To adjust the member division, click [Edit]. The *FE Mesh Settings* dialog box from RFEM appears.


FE Mesh Settings

General
Target length of finite elements l_{FE}: 0.500 [m]
Maximum distance between a node and a line to integrate it into the line ε: [m]
Maximum number of mesh nodes (in thousands) max: 500

Members
Number of divisions for special types of members (cable, elastic foundation, taper, nonlinearity): 6
☐ Activate divisions for straight members, which are not integrated in surfaces, with concrete material category group (necessary for nonlinear calculation)
Minimum number of member divisions: 10
☐ Activate member divisions for large deformation or post-critical analysis
☑ Use division for straight members, which are not integrated in surfaces, with
● Target length l_{FE} of finite elements
○ Set length l_{FE} : [m]
Minimum number of member divisions: 6
☐ Use division for members with nodes lying on them

Surfaces
Maximum ratio of FE rectangle diagonals Δ_D: [-]
Maximum out-of-plane inclination of two finite elements α: [°]
☐ FE mesh refinement along lines (with type 'Plate XY' only)
Relationship Δ_b: [-]
☐ Integrate unutilized objects into surfaces
Shape of finite elements:
○ Quadrangles only
○ Triangles only
● Triangles and quadrangles
☑ Same squares where possible
☐ Mapped mesh preferred

Solids
☐ Refinement of FE mesh on solids containing close nodes
Maximum number of elements (in thousands): 200

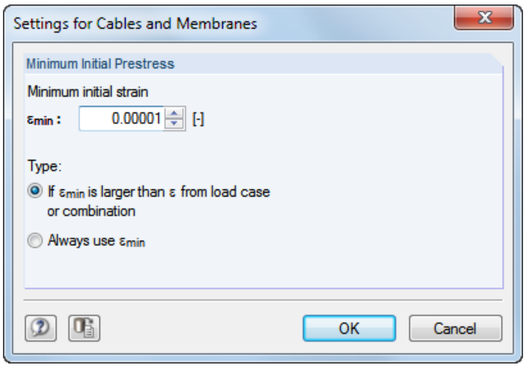
Option
☐ Regenerate FE mesh on [OK]

Figure 2.4: Dialog box *FE Mesh Settings*

Activate minimum initial prestress for cables and membranes

By selecting this check box, cable members and membrane surfaces get a very small prestress that prevents the failure of these elements before the start of the calculation. In this way, you can also stabilize the model.

To specify the magnitude and type of the prestress, click [Edit].



Settings for Cables and Membranes

Minimum Initial Prestress

Minimum initial strain

ε_{min} : 0.00001 [-]

Type:

● If ε_{min} is larger than ε from load case or combination

○ Always use ε_{min}

Figure 2.5: Dialog box *Settings for Cables and Membranes*

Activate stiffness modifications from RFEM

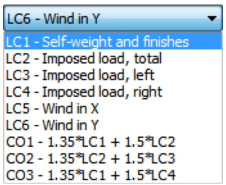
This check box controls whether the adjustment factors defined in RFEM for the material, member, support, release, and cross-section stiffnesses are taken into account during the eigenvalue analysis. These factors are described in the corresponding chapters of the RFEM manual. If you select this check box, all stiffness factors of the load case or the load combination are used under consideration of the failure criteria during the calculation in RF-STABILITY.

This control option in RF-STABILITY allows you to apply the stiffness modifications independent of the settings in RFEM. In this way, you can, for example, calculate the axial forces of a load combination in RFEM without reduction and then determine the critical loads in RF-STABILITY with the reduced material, member, and cross-section stiffnesses.

If, however, the stiffness change for the load case or the load combination is active, this check box is unavailable: In this way, the program ensures that the eigenvalues correspond to the model assumptions.

If the eigenvalues are determined as "characteristic" properties of the model, the stiffness changes need not be considered.

Add initial axial forces from load case / combination



LC6 - Wind in Y
LC1 - Self-weight and finishes
LC2 - Imposed load, total
LC3 - Imposed load, left
LC4 - Imposed load, right
LC5 - Wind in X
LC6 - Wind in Y
CO1 - 1.35*LC1 + 1.5*LC2
CO2 - 1.35*LC2 + 1.5*LC3
CO3 - 1.35*LC1 + 1.5*LC4

After selecting this check box, you can choose a load case or load combination whose axial forces you want to use for an initial deformation. You can use this option to, for example, take into account the stabilizing effect of another load case (that is, a different one from the load case selected in the *General* section above) for the eigenvalue analysis.

Result combinations are also missing in this list, because they don't show unambiguous axial force distributions.

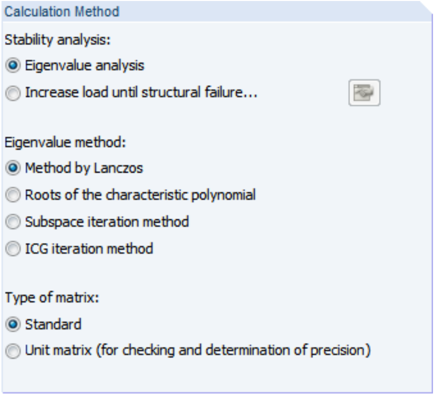
Calculate eigenvector of unstable model

This function allows you to find modeling errors in your structural system. For models determined to be unstable, the program runs a load-independent eigenvalue analysis. In the graphical evaluation of the results, you can easily find, for example, unconnected members or members with too many open degrees of freedom.

However, that function cannot detect calculation terminations due to overloading.

For an example of that function, see this DLUBAL blog: https://www.dlubal.com/blog/7907

Calculation Method



Calculation Method

Stability analysis:
- ● Eigenvalue analysis
- ○ Increase load until structural failure...

Eigenvalue method:
- ● Method by Lanczos
- ○ Roots of the characteristic polynomial
- ○ Subspace iteration method
- ○ ICG iteration method

Type of matrix:
- ● Standard
- ○ Unit matrix (for checking and determination of precision)

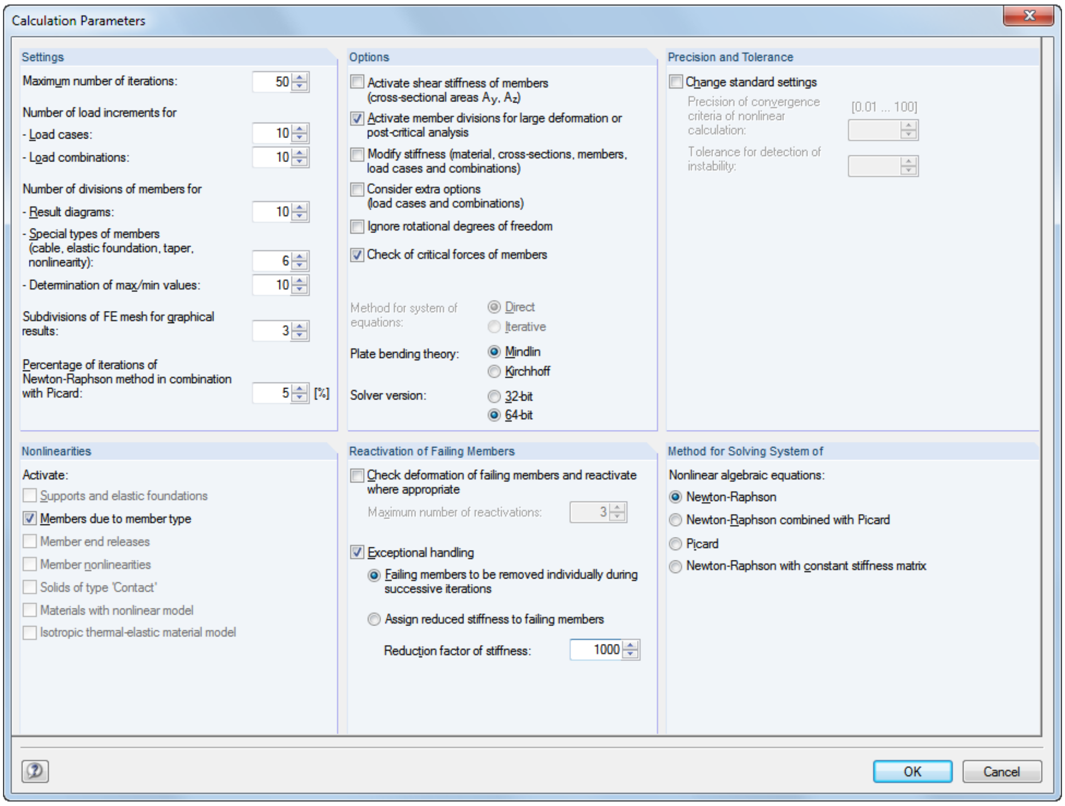
Figure 2.6: Method of Calculation

Stability analysis

When using the default setting *Eigenvalue analysis*, the eigenvectors are determined by the purely linear method. The properties of nonlinearly acting elements, for example, tension members or supports with failure criteria, are not taken into account. The stiffnesses of such elements are applied in the matrix as permanently acting.

The *Load increased to structural failure* option allows you to take into account all nonlinearities in the eigenvalue analysis. During the incremental increase of the load, failure criteria and nonlinear effects of members, supports, releases, materials, etc. are taken into account. The calculation is run iteratively and, therefore, needs more time. Due to the underlying principle, this procedure allows you to reliably determine only the lowest eigenvalue.

To specify the iteration parameters, click [Edit].



Calculation Parameters

Settings
- Maximum number of iterations: 50
- Number of load increments for
  - Load cases: 10
  - Load combinations: 10
- Number of divisions of members for
  - Result diagrams: 10
  - Special types of members (cable, elastic foundation, taper, nonlinearity): 6
  - Determination of max/min values: 10
- Subdivisions of FE mesh for graphical results: 3
- Percentage of iterations of Newton-Raphson method in combination with Picard: 5 [%]

Options
- ☐ Activate shear stiffness of members (cross-sectional areas A_y, A_z)
- ☑ Activate member divisions for large deformation or post-critical analysis
- ☐ Modify stiffness (material, cross-sections, members, load cases and combinations)
- ☐ Consider extra options (load cases and combinations)
- ☐ Ignore rotational degrees of freedom
- ☑ Check of critical forces of members
- Method for system of equations: ● Direct ○ Iterative
- Plate bending theory: ● Mindlin ○ Kirchhoff
- Solver version: ○ 32-bit ● 64-bit

Precision and Tolerance
- ☐ Change standard settings
- Precision of convergence criteria of nonlinear calculation: [0.01 ... 100]
- Tolerance for detection of instability:

Nonlinearities

Activate:
- ☐ Supports and elastic foundations
- ☑ Members due to member type
- ☐ Member end releases
- ☐ Member nonlinearities
- ☐ Solids of type 'Contact'
- ☐ Materials with nonlinear model
- ☐ Isotropic thermal-elastic material model

Reactivation of Failing Members
- ☐ Check deformation of failing members and reactivate where appropriate
  - Maximum number of reactivations: 3
- ☑ Exceptional handling
  - ● Failing members to be removed individually during successive iterations
  - ○ Assign reduced stiffness to failing members
  - Reduction factor of stiffness: 1000

Method for Solving System of

Nonlinear algebraic equations:
- ● Newton-Raphson
- ○ Newton-Raphson combined with Picard
- ○ Picard
- ○ Newton-Raphson with constant stiffness matrix

Figure 2.7: Dialog box *Calculation Parameters*

The *Calculation Parameters* dialog box is described in the RFEM manual, Chapter 7.3.3. The calculation is run automatically according to the direct solver method.

If, as shown in Figure 2.7, you specify 10 load increments, the program applies 10 %, then 20 %, then 30 %, …100 %, 110 % …480 %, etc. of the load: The load is increased in 10 % steps until the system is determined to be unstable. Thus the critical load factor is found.

The duration of calculation can be reduced by the following settings of the *Calculation Parameters* (suggestions in brackets):

- Maximum number of iteration shouldn't be too great (50)
- Number of load increments shouldn't be too fine (5)
- Reactivation of failing members should be deactivated

Eigenvalue method

There are different calculation methods available in RF-STABILITY:

- **Method by Lanczos**

  The eigenvalues are determined according to the direct method. In most cases, this algorithm allows you to reach a quick convergence. Because the Lanczos method is faster than the subspace method (see below), it is the default option in RF-STABILITY.

  For further information, see https://en.wikipedia.org/wiki/Lanczos_algorithm

- **Roots of the characteristic polynomial**

  This method is also based on a direct calculation method. For larger structural systems, this method can be faster than the Lanczos method. The main advantage of this method is the precision of higher eigenvalues.

  For more information, see https:/en.wikipedia.org/wiki/Characteristic_polynomial

- **Subspace iteration method**

  In this method, all eigenvalues are determined in one step. The spectrum of the stiffness matrix has a great influence on the duration of calculation in this method.

  The stiffness matrix is stored in the operating memory. As soon as this memory is insufficient, Windows stores files on the hard drive. This considerably slows down the computer. The subspace method is therefore not suitable for complex systems.

  In the subspace method, negative critical load factors cannot be excluded.

  For more information, see https:/en.wikipedia.org/wiki/Krylov_subspace

- **ICG iteration method**

  The ICG (*Incomplete Conjugate Gradient*) iteration method needs little operating memory. Only those coefficients of the stiffness matrix not equal to zero are saved. A disadvantage is that the eigenvalues are determined one after the other. The spectrum, however, has no influence on the calculation duration.

  The ICG method is suited to analyze very large systems with few eigenvalues. This method also does not yield any negative critical load factors. The procedure continues until the number of specified positive eigenvalues is reached.

  The ICG method sometimes doesn't find all of the lowest eigenvalues.

  For more information, see https:/en.wikipedia.org/wiki/Conjugate_gradient_method

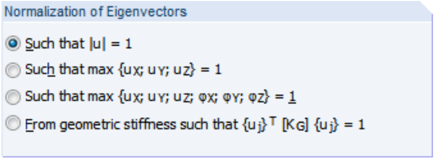
Type of matrix

With the default option *Standard*, RF-STABILITY uses the geometric stiffness matrix of the linear equation system.

The *Unit matrix* is a quadratic matrix with the value one on the main diagonal. This “mathematic” approach should be used only for numerical and kinematic analyses. Loads and axial forces are of no importance when solving the general eigenvalue problem.

The matrix types for eigenvalue solutions are described in more detail in [1], Chapter 7.

Normalization of Eigenvectors



Normalization of Eigenvectors

● Such that |u| = 1

○ Such that max {uX; uY; uZ} = 1

○ Such that max {uX; uY; uZ; φX; φY; φZ} = 1

○ From geometric stiffness such that {uj}T [KG] {uj} = 1

Figure 2.8: Normalization of the Eigenvectors

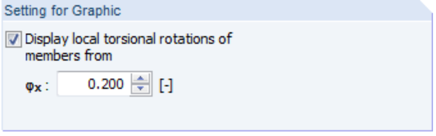
The eigenvectors can be normalized (“norm one”) differently:

- Maximum displacement $|\mathrm{u}| = 1$
- Maximum global displacement $(\text{max from } \mathrm{u_X}, \mathrm{u_Y}, \mathrm{u_Z}) = 1$
- Maximum global deformation $(\text{max from } \mathrm{u_X}, \mathrm{u_Y}, \mathrm{u_Z}, \varphi_\mathrm{X}, \varphi_\mathrm{Y}, \varphi_\mathrm{Z}) = 1$
- Maximum value of the product $\left\{\mathrm{u_j}\right\}^\mathrm{T} \cdot [\mathrm{K_G}] \cdot \left\{\mathrm{u_j}\right\} = 1$ where:

  $\left\{\mathrm{u_j}\right\}^\mathrm{T}$: Transposed eigenvector
  $[\mathrm{K_G}]$: Geometric stiffness
  $\left\{\mathrm{u_j}\right\}$: Eigenvector

Depending on the specification, the greater displacement or deformation represents the reference value one, to which the other results are scaled.

Setting for Graphic

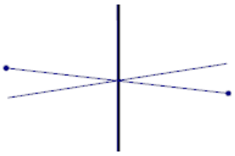


Setting for Graphic

☑ Display local torsional rotations of members from

φx : 0.200 [-]

Figure 2.9: Setting for Graphic



Use this check box to display the *torsional rotation of members* in the results graphic of the eigenvectors (see figure on the left).

By default, only torsional rotations φ_x with normalized values greater than 0.2 are displayed. This ensures a clear representation in the graphic.

Comment

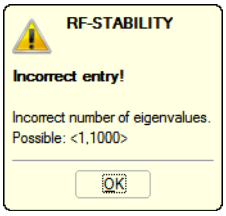
In this text box, you can type user-defined notes describing, for example, the current analysis case.

3 Calculation

3.1 Check

Before starting the calculation, check if the input data is correct. To do this, click [Check] in the bottom part of the module.

If the module finds an inconsistency, a warning appears showing the corresponding information.

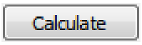

RF-STABILITY

Incorrect entry!

Incorrect number of eigenvalues.
Possible: <1,1000>

Figure 3.1: Result of check

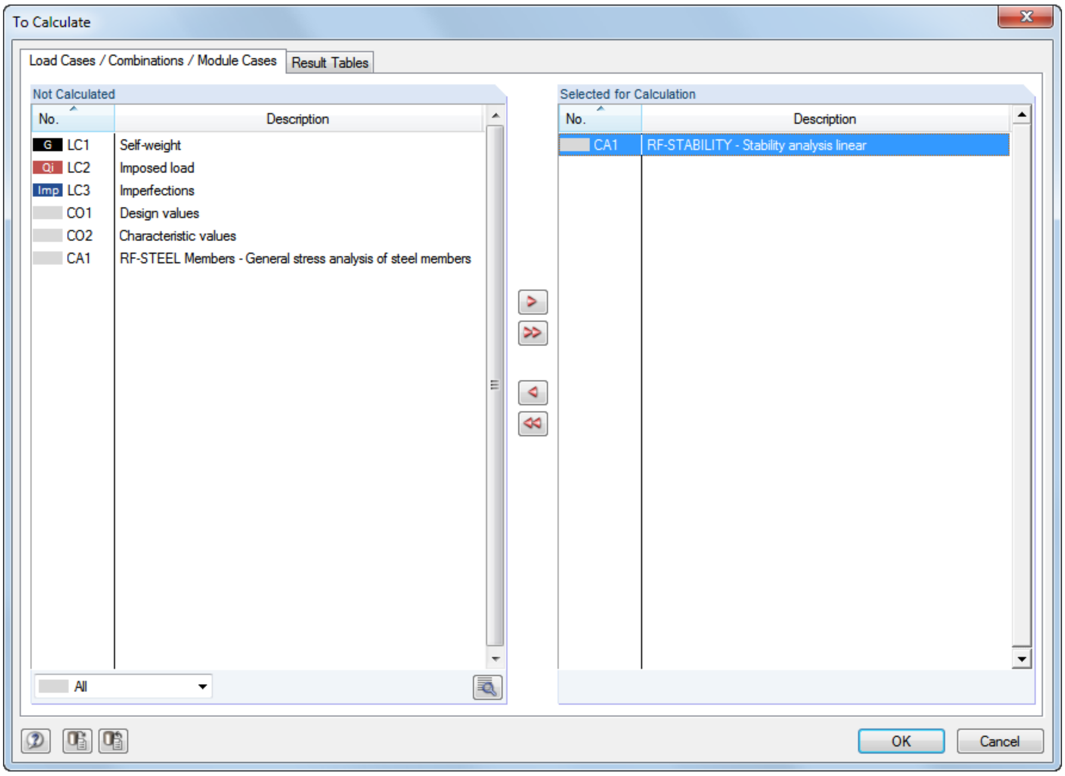
3.2 Start Calculation



To start the calculation, click the [Calculation] button.

RF-STABILITY searches for the axial forces to be taken into account in the stability analysis. If no results are available for the load case or load combination, the RFEM calculation starts to determine the internal forces.

You can also start the calculation in the RFEM user interface. The *To Calculate* dialog box (menu ***Calculate*** → ***To Calculate***) lists the add-on module cases as well as load cases and load combinations.



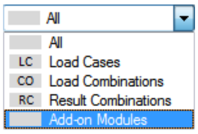
Not Calculated

No.	Description
G LC1	Self-weight
Qi LC2	Imposed load
Imp LC3	Imperfections
CO1	Design values
CO2	Characteristic values
CA1	RF-STEEL Members - General stress analysis of steel members

Selected for Calculation

No.	Description
CA1	RF-STABILITY - Stability analysis linear

Figure 3.2: Dialog box *To Calculate*



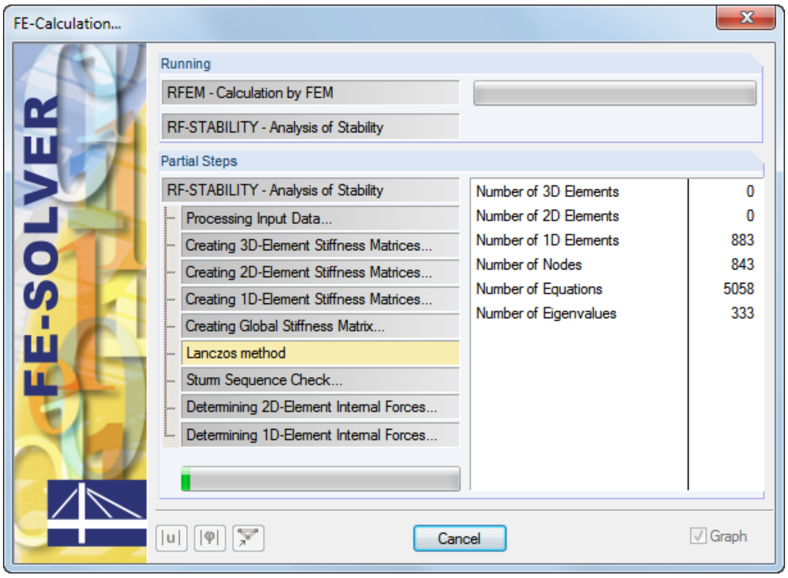
If the RF-STABILITY cases are missing in the *Not Calculated* list, select *All* or *Add-on Modules* in the drop-down list at the end of the section.

To transfer the selected RF-STABILITY cases to the list on the right, use the ▶ button. Click [OK] to start the calculation.

To calculate an analysis case directly, use the list in the RFEM toolbar. Select the relevant RF-STABILITY case in the list, and then click [Show Results].

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

You then can observe the stability analysis process in a separate dialog box.



Running

RFEM - Calculation by FEM

RF-STABILITY - Analysis of Stability

Partial Steps

RF-STABILITY - Analysis of Stability
- Processing Input Data...
- Creating 3D-Element Stiffness Matrices...
- Creating 2D-Element Stiffness Matrices...
- Creating 1D-Element Stiffness Matrices...
- Creating Global Stiffness Matrix...
- Lanczos method
- Sturm Sequence Check...
- Determining 2D-Element Internal Forces...
- Determining 1D-Element Internal Forces...

Number of 3D Elements	0
Number of 2D Elements	0
Number of 1D Elements	883
Number of Nodes	843
Number of Equations	5058
Number of Eigenvalues	333

Figure 3.4: RF-STABILITY calculation

In the eigenvalue analysis according to one of the direct calculation methods, the so-called *Sturm Sequence Check* is run. It checks whether all eigenvalues are found in a certain interval. The diagonal matrix from the Gauss decomposition is used, whose number of negative diagonal elements corresponds to the number of eigenvalues below the corresponding interval boundary. The Sturm check is carried out for the given interval limits. From this, the difference is determined.

4 Results

The results are shown in the tables of the results windows 2.1 through 2.5.

Immediately after the calculation, the 2.1 *Design by Load Case* window appears. The Windows 2.2 through 2.5 show the effective lengths and critical loads as well as the eigenvectors sorted by certain criteria.

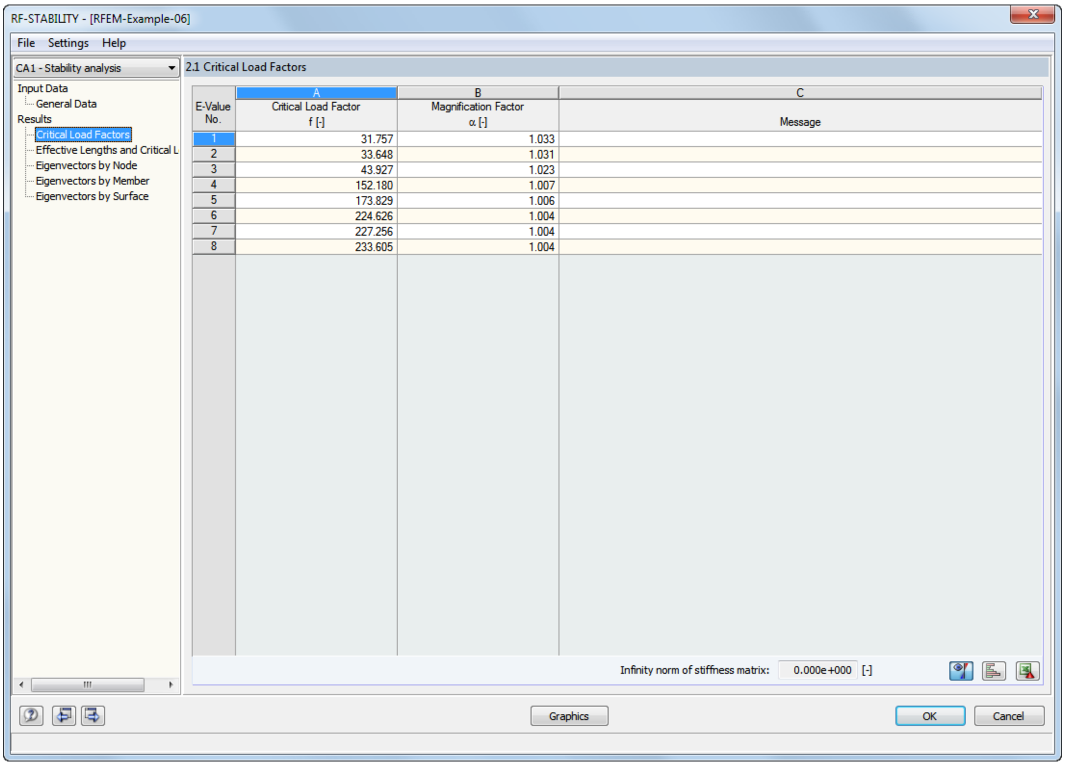
To go to a window, click the corresponding entry in the navigator. To go to the previous or next input window, use the buttons shown on the left. Alternatively, you can use the function keys to go the next [F2] or previous [F3] window.

Click [OK] to save the results. Thus you exit RF-STABILITY and return to the main program.

Chapter 4 *Results* describes the different results windows one by one. Evaluating and checking results is described in Chapter 5 on page 21.

4.1 Critical Load Factors

The first results window provides information on the critical load factors of the model.



2.1 Critical Load Factors

E-Value No.	A Critical Load Factor f [-]	B Magnification Factor α [-]	C Message
1	31.757	1.033	
2	33.648	1.031	
3	43.927	1.023	
4	152.180	1.007	
5	173.829	1.006	
6	224.626	1.004	
7	227.256	1.004	
8	233.605	1.004	

Infinity norm of stiffness matrix: 0.000e+000 [-]

Figure 4.1: Window 2.1 *Critical Load Factors*

E-Value No.

For each eigenvalue, the table shows the critical load and magnification factors. The result rows are sorted by numbers of the eigenvectors.

Critical Load Factor f

The table shows the critical load factor for each eigenvalue. If the factor is less than 1.00, this means that the structural system is unstable. A factor higher than 1.00 means that the loading from the given axial forces multiplied with this factor results in system failure due to buckling.

According to [2], clause 5.2.1(3), critical load factors less than 10 for elastic calculations require a second-order analysis.

If the table output contains a negative critical load factor, no stability failure occurs because of the tension forces. No conclusion about the expected buckling behavior is possible.

Magnification Factor α

The magnification factor is determined as follows:

$$\alpha = \frac{1}{1 - 1/f} \tag{4.1}$$

The magnification factor describes the relation between the moments according to the geometrically linear static and the second-order analysis.

$$M^{II} = \alpha \cdot M^{I} \tag{4.2}$$

where

M^{I} : Moment according to geometrically linear static analysis, but taking into account the equivalent loading for the deformation

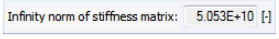
M^{II} : Second-order moment

Equation 4.2, however, applies only if the bending line due to the loading of the eigenvector is similar, and if the critical load factor f is greater than 1.00.

Message

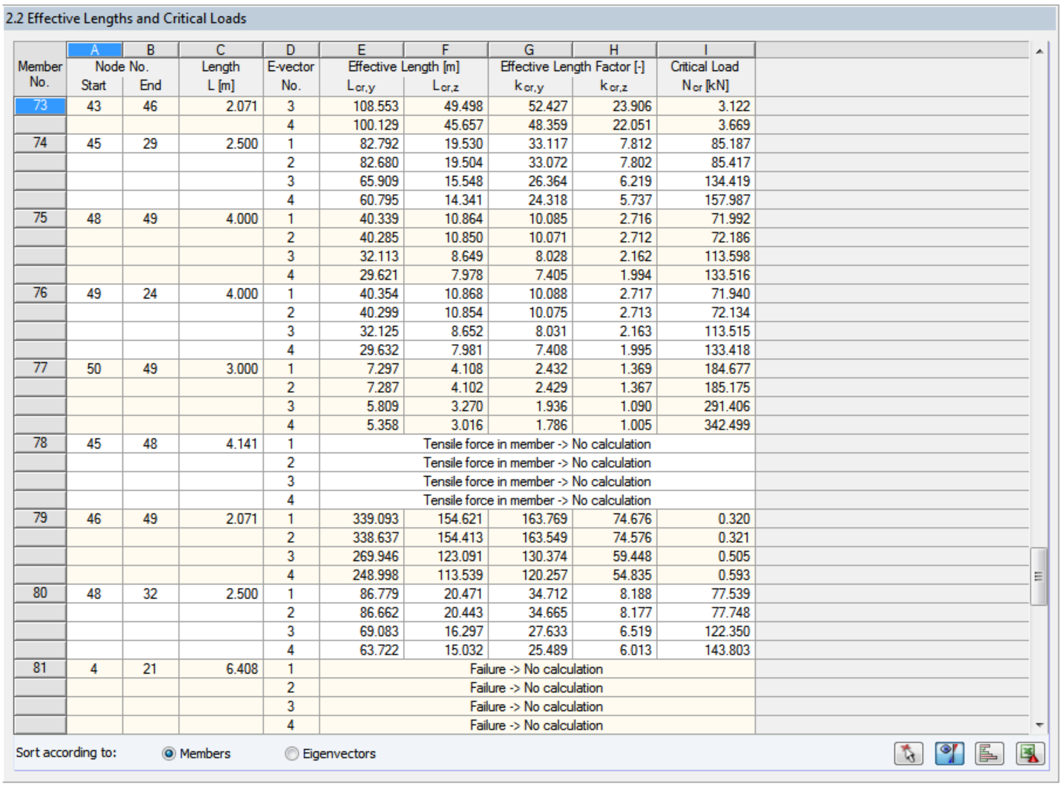
In column C, a message about a negative critical load factor might appear. This means a reversed direction of action of the given loading (inverse signs) would result in buckling failure.

You can avoid negative critical load factors by increasing the eigenvectors to be calculated or by using the ICG iteration method.


Infinity norm of stiffness matrix: 5.053E+10 [-]

At the lower end of the table, the *Infinity norm of stiffness matrix* is given. With this mathematical norm, you can estimate the magnitude of the highest eigenvalue. This value is important for, e.g., the precision of the solution (see [1], Chapter 3, 2.2.3 and Chapter 7).

4.2 Effective Lengths and Critical Loads

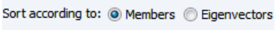


2.2 Effective Lengths and Critical Loads

	A	B	C	D	E	F	G	H	I
Member No.	Node No.		Length	E-vector	Effective Length [m]		Effective Length Factor [-]		Critical Load
	Start	End	L [m]	No.	$L_{cr,y}$	$L_{cr,z}$	$k_{cr,y}$	$k_{cr,z}$	N_{cr} [kN]
73	43	46	2.071	3	108.553	49.498	52.427	23.906	3.122
				4	100.129	45.657	48.359	22.051	3.669
74	45	29	2.500	1	82.792	19.530	33.117	7.812	85.187
				2	82.680	19.504	33.072	7.802	85.417
				3	65.909	15.548	26.364	6.219	134.419
				4	60.795	14.341	24.318	5.737	157.987
75	48	49	4.000	1	40.339	10.864	10.085	2.716	71.992
				2	40.285	10.850	10.071	2.712	72.186
				3	32.113	8.649	8.028	2.162	113.598
				4	29.621	7.978	7.405	1.994	133.516
76	49	24	4.000	1	40.354	10.868	10.088	2.717	71.940
				2	40.299	10.854	10.075	2.713	72.134
				3	32.125	8.652	8.031	2.163	113.515
				4	29.632	7.981	7.408	1.995	133.418
77	50	49	3.000	1	7.297	4.108	2.432	1.369	184.677
				2	7.287	4.102	2.429	1.367	185.175
				3	5.809	3.270	1.936	1.090	291.406
				4	5.358	3.016	1.786	1.005	342.499
78	45	48	4.141	1	Tensile force in member -> No calculation				
				2	Tensile force in member -> No calculation				
				3	Tensile force in member -> No calculation				
				4	Tensile force in member -> No calculation				
79	46	49	2.071	1	339.093	154.621	163.769	74.676	0.320
				2	338.637	154.413	163.549	74.576	0.321
				3	269.946	123.091	130.374	59.448	0.505
				4	248.998	113.539	120.257	54.835	0.593
80	48	32	2.500	1	86.779	20.471	34.712	8.188	77.539
				2	86.662	20.443	34.665	8.177	77.748
				3	69.083	16.297	27.633	6.519	122.350
				4	63.722	15.032	25.489	6.013	143.803
81	4	21	6.408	1	Failure -> No calculation				
				2	Failure -> No calculation				
				3	Failure -> No calculation				
				4	Failure -> No calculation				

Sort according to: ● Members ○ Eigenvectors

Figure 4.2: Window 2.2 *Effective Lengths and Critical Loads*



Sort according to: ● Members ○ Eigenvectors

This results window is displayed if the model contains members. You can sort the effective lengths and critical loads according to *Members* or *Eigenvectors*.

Member No.

The results of the stability analysis are shown for all members of the model. Members subject to tension forces and failed members are indicated accordingly.

Node No. Start / End

Every member is defined by a start and end node. The numbers are shown in these two columns A and B, respectively.

Length L

This column shows the geometric length of each member.

E-vector No.

If the table is sorted by members, this column shows you the numbers of eigenvectors.

Effective Length $L_{cr,y}/L_{cr,z}$

The effective length $L_{cr,y}$ (or $L_{cr,u}$) describes the buckling behavior perpendicular to the major member y axis (or u for unsymmetric cross-sections). Accordingly, $L_{cr,z}$ or $L_{cr,v}$ describe the deflection perpendicular to the minor member z axis (or v).

The effective lengths L_{cr} are based on the member-specific critical loads shown in column I. These loads are relative to the critical load of the total model. This means that the effective lengths need to be evaluated in the context of the relation of the member axial forces to the total critical load. For simple cases, the effective lengths are known as the EULER buckling modes 1 through 4.

In some cases, the most unfavorable critical load of the system can be the critical load of an isolated, that is, an individual member that is connected by a hinge (see Chapter 8.3, page 38). This can be seen in the graphic of the corresponding eigenvectors, because a sinusoidal wave is available only on this member. In this case, the structure shows a so-called local instability. The effective lengths of all other members cannot be used for this buckling mode. Therefore, they must be taken from a "higher" eigenvector. The total model fails only there.

Effective Length Factor $k_{cr,y}/k_{cr,z}$

The effective length factors relating to the local member axes y and z (or u and v) describe the ratio of effective and member length.

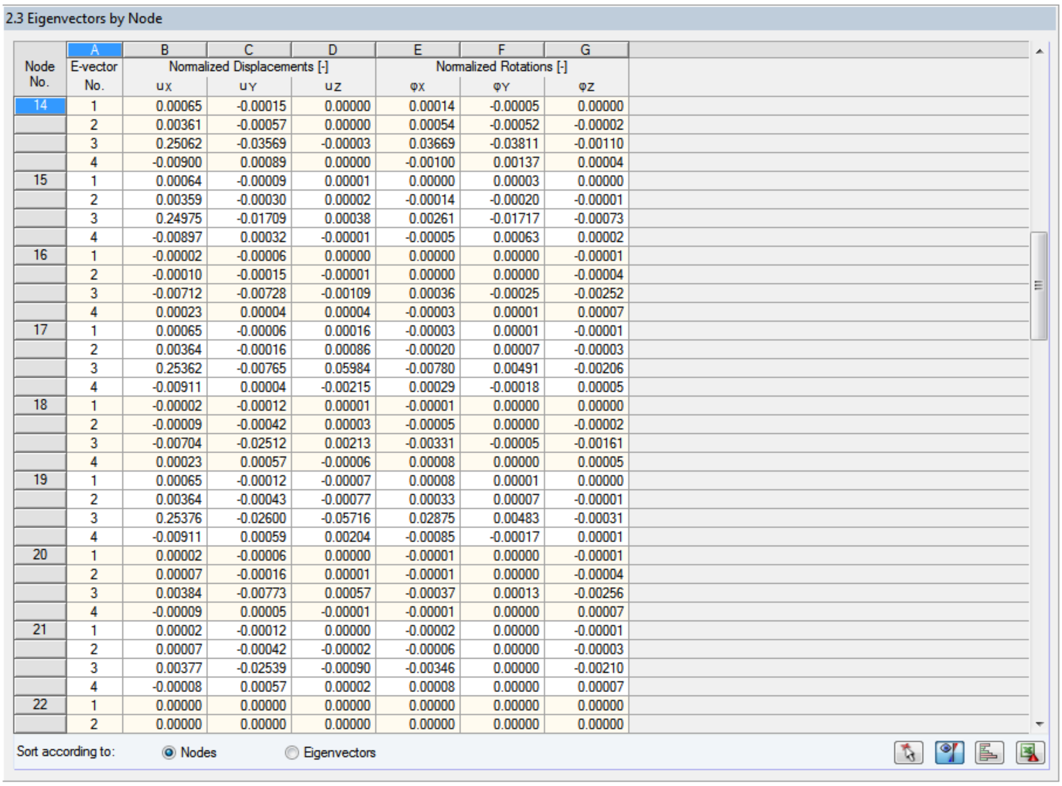
Effective length factor k_{cr}:

$$k_{cr} = \frac{L_{cr}}{L} \tag{4.3}$$

Critical Load N_{cr}

This table column shows for each member the critical axial force N_{cr} that was determined relative to the respective eigenvector. Therefore, the individual critical loads and the corresponding effective lengths must always be seen in the context of the corresponding critical load of the total system.

4.3 Eigenvectors by Node

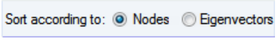


2.3 Eigenvectors by Node

	A	B	C	D	E	F	G
Node No.	E-vector No.	Normalized Displacements [-]			Normalized Rotations [-]		
		uX	uY	uZ	φX	φY	φZ
14	1	0.00065	-0.00015	0.00000	0.00014	-0.00005	0.00000
	2	0.00361	-0.00057	0.00000	0.00054	-0.00052	-0.00002
	3	0.25062	-0.03569	-0.00003	0.03669	-0.03811	-0.00110
	4	-0.00900	0.00089	0.00000	-0.00100	0.00137	0.00004
15	1	0.00064	-0.00009	0.00001	0.00000	0.00003	0.00000
	2	0.00359	-0.00030	0.00002	-0.00014	-0.00020	-0.00001
	3	0.24975	-0.01709	0.00038	0.00261	-0.01717	-0.00073
	4	-0.00897	0.00032	-0.00001	-0.00005	0.00063	0.00002
16	1	-0.00002	-0.00006	0.00000	0.00000	0.00000	-0.00001
	2	-0.00010	-0.00015	-0.00001	0.00000	0.00000	-0.00004
	3	-0.00712	-0.00728	-0.00109	0.00036	-0.00025	-0.00252
	4	0.00023	0.00004	0.00004	-0.00003	0.00001	0.00007
17	1	0.00065	-0.00006	0.00016	-0.00003	0.00001	-0.00001
	2	0.00364	-0.00016	0.00086	-0.00020	0.00007	-0.00003
	3	0.25362	-0.00765	0.05984	-0.00780	0.00491	-0.00206
	4	-0.00911	0.00004	-0.00215	0.00029	-0.00018	0.00005
18	1	-0.00002	-0.00012	0.00001	-0.00001	0.00000	0.00000
	2	-0.00009	-0.00042	0.00003	-0.00005	0.00000	-0.00002
	3	-0.00704	-0.02512	0.00213	-0.00331	-0.00005	-0.00161
	4	0.00023	0.00057	-0.00006	0.00008	0.00000	0.00005
19	1	0.00065	-0.00012	-0.00007	0.00008	0.00001	0.00000
	2	0.00364	-0.00043	-0.00077	0.00033	0.00007	-0.00001
	3	0.25376	-0.02600	-0.05716	0.02875	0.00483	-0.00031
	4	-0.00911	0.00059	0.00204	-0.00085	-0.00017	0.00001
20	1	0.00002	-0.00006	0.00000	-0.00001	0.00000	-0.00001
	2	0.00007	-0.00016	0.00001	-0.00001	0.00000	-0.00004
	3	0.00384	-0.00773	0.00057	-0.00037	0.00013	-0.00256
	4	-0.00009	0.00005	-0.00001	-0.00001	0.00000	0.00007
21	1	0.00002	-0.00012	0.00000	-0.00002	0.00000	-0.00001
	2	0.00007	-0.00042	-0.00002	-0.00006	0.00000	-0.00003
	3	0.00377	-0.02539	-0.00090	-0.00346	0.00000	-0.00210
	4	-0.00008	0.00057	0.00002	0.00008	0.00000	0.00007
22	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
	2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000

Sort according to: ● Nodes ○ Eigenvectors

Figure 4.3: Window 2.3 *Eigenvectors by Node*



Sort according to: ● Nodes ○ Eigenvectors

For each eigenvector, the table shows the normalized displacements and rotations of the nodes in the model. You can sort the window according to *Nodes* or *Eigenvectors*.

Node No.

The eigenvectors are listed for the objects defined in the RFEM Table 1.1 *Nodes*. Results in the FE nodes or in the member division points are not shown in the table.

E-vector No.

The deformations are listed for each calculated eigenvector.

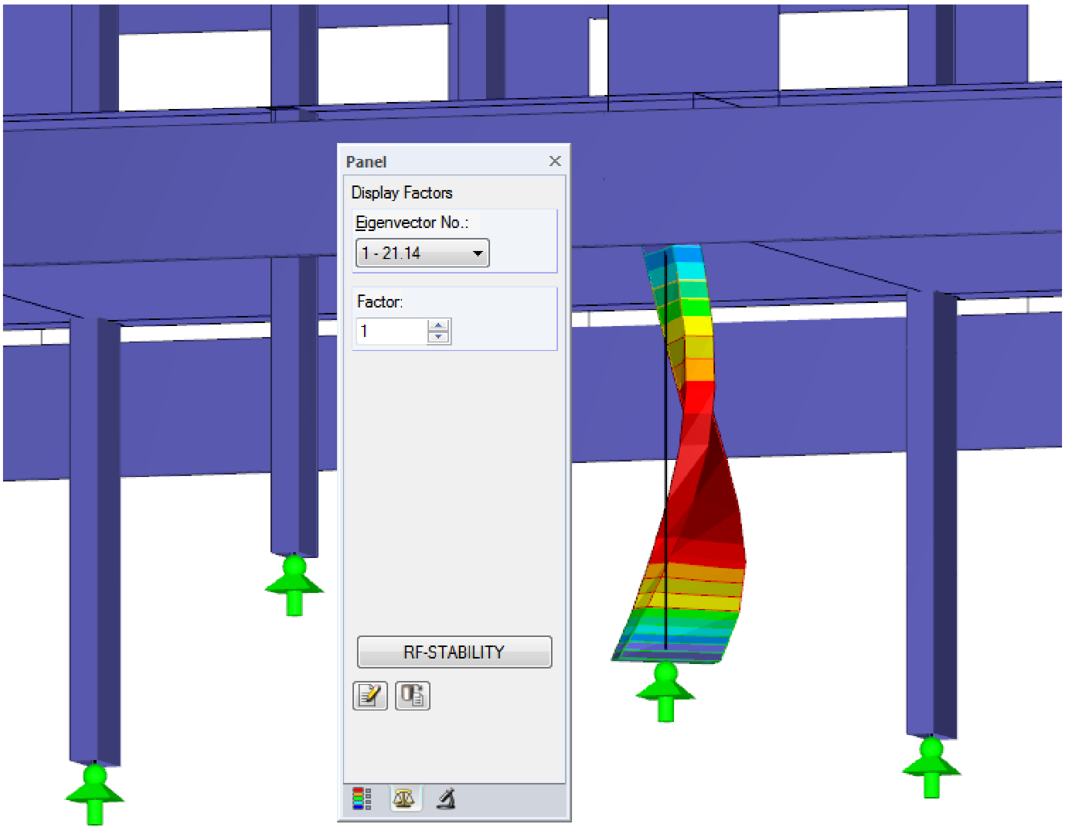
Normalized displacements u_X / u_Y / u_Z

The displacements listed in columns B through D are relative to the axes of the global coordinate system. They are normalized to the extreme value 1 of the total displacement or the maximum given in Window 1.1 *General Data* (see Figure 2.8, page 11).

Normalized rotations φ_X / φ_Y / φ_Z

The columns E through G list the values of the normalized nodal rotations.

If the table shows only zero values for the normalized displacements of a member model, the reason is often to be found in large torsions within the members (see the figure below). These effects do not influence the displacements of the member ends. Therefore, the displayed effective lengths and critical loads are not very conclusive for these members.



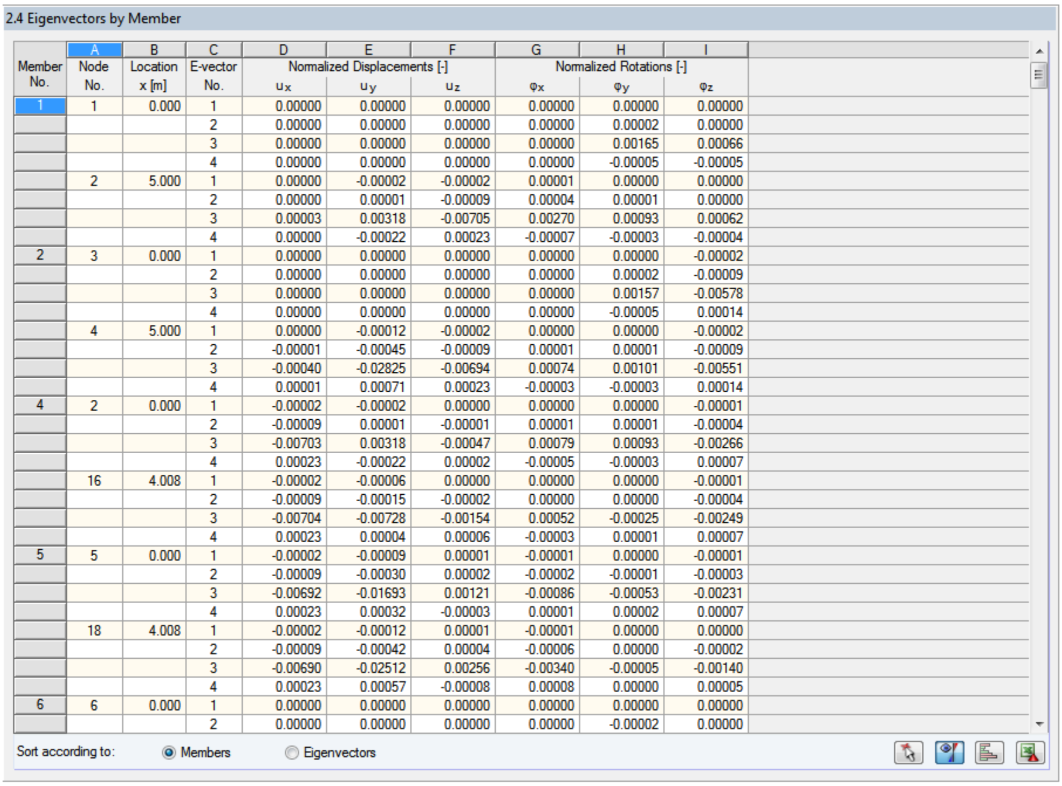
Eigenvector No.: 1 - 21.14

Factor: 1

Figure 4.4: Torsion of a thin-walled rectangular column

4.4 Eigenvectors by Member

This results window is shown only if the model contains members.



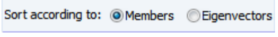
2.4 Eigenvectors by Member

	A	B	C	D	E	F	G	H	I
Member No.	Node No.	Location x [m]	E-vector No.	Normalized Displacements [-]			Normalized Rotations [-]		
				u_x	u_y	u_z	φ_x	φ_y	φ_z
1	1	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
			2	0.00000	0.00000	0.00000	0.00000	0.00002	0.00000
			3	0.00000	0.00000	0.00000	0.00000	0.00165	0.00066
			4	0.00000	0.00000	0.00000	0.00000	-0.00005	-0.00005
	2	5.000	1	0.00000	-0.00002	-0.00002	0.00001	0.00000	0.00000
			2	0.00000	0.00001	-0.00009	0.00004	0.00001	0.00000
			3	0.00003	0.00318	-0.00705	0.00270	0.00093	0.00062
			4	0.00000	-0.00022	0.00023	-0.00007	-0.00003	-0.00004
2	3	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	-0.00002
			2	0.00000	0.00000	0.00000	0.00000	0.00002	-0.00009
			3	0.00000	0.00000	0.00000	0.00000	0.00157	-0.00578
			4	0.00000	0.00000	0.00000	0.00000	-0.00005	0.00014
	4	5.000	1	0.00000	-0.00012	-0.00002	0.00000	0.00000	-0.00002
			2	-0.00001	-0.00045	-0.00009	0.00001	0.00001	-0.00009
			3	-0.00040	-0.02825	-0.00694	0.00074	0.00101	-0.00551
			4	0.00001	0.00071	0.00023	-0.00003	-0.00003	0.00014
4	2	0.000	1	-0.00002	-0.00002	0.00000	0.00000	0.00000	-0.00001
			2	-0.00009	0.00001	-0.00001	0.00001	0.00001	-0.00004
			3	-0.00703	0.00318	-0.00047	0.00079	0.00093	-0.00266
			4	0.00023	-0.00022	0.00002	-0.00005	-0.00003	0.00007
	16	4.008	1	-0.00002	-0.00006	0.00000	0.00000	0.00000	-0.00001
			2	-0.00009	-0.00015	-0.00002	0.00000	0.00000	-0.00004
			3	-0.00704	-0.00728	-0.00154	0.00052	-0.00025	-0.00249
			4	0.00023	0.00004	0.00006	-0.00003	0.00001	0.00007
5	5	0.000	1	-0.00002	-0.00009	0.00001	-0.00001	0.00000	-0.00001
			2	-0.00009	-0.00030	0.00002	-0.00002	-0.00001	-0.00003
			3	-0.00692	-0.01693	0.00121	-0.00086	-0.00053	-0.00231
			4	0.00023	0.00032	-0.00003	0.00001	0.00002	0.00007
	18	4.008	1	-0.00002	-0.00012	0.00001	-0.00001	0.00000	0.00000
			2	-0.00009	-0.00042	0.00004	-0.00006	0.00000	-0.00002
			3	-0.00690	-0.02512	0.00256	-0.00340	-0.00005	-0.00140
			4	0.00023	0.00057	-0.00008	0.00008	0.00000	0.00005
6	6	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
			2	0.00000	0.00000	0.00000	0.00000	-0.00002	0.00000

Sort according to: ● Members ○ Eigenvectors

Figure 4.5: Window 2.4 *Eigenvectors by Member*

The columns are described in detail in Chapter 4.3. In addition, the *Location* x is given for the start and end node of the corresponding member.

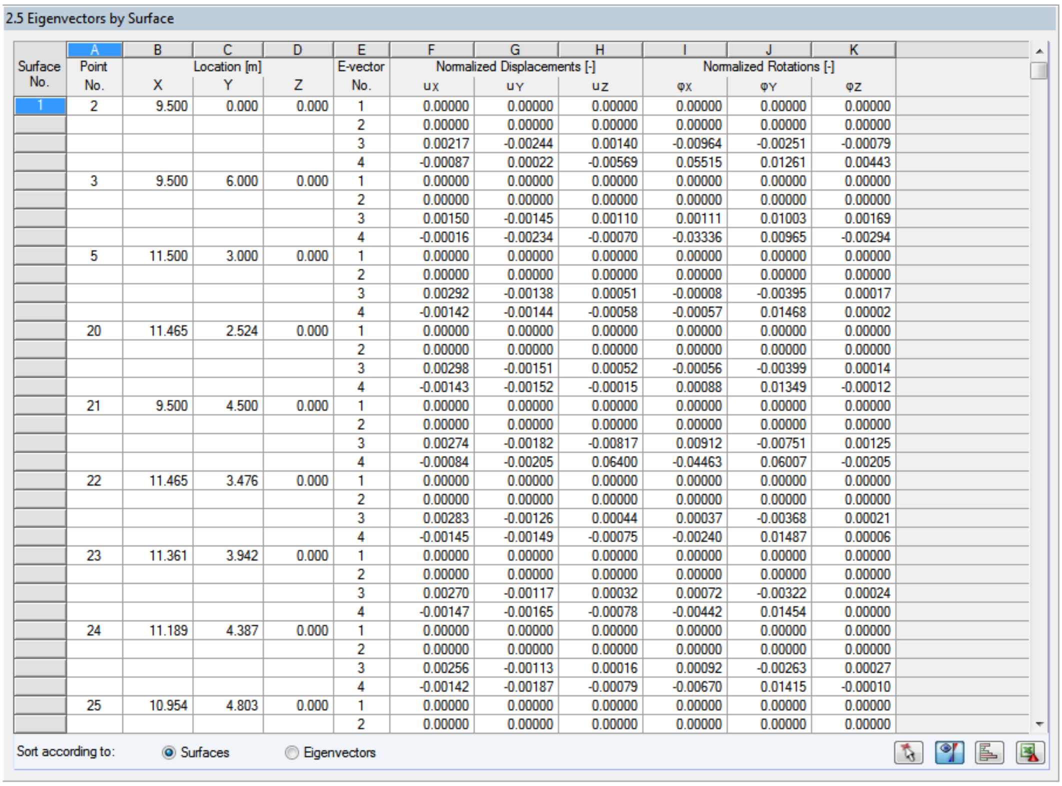


Sort according to: ● Members ○ Eigenvectors

You can sort this window according to *Members* or *Eigenvectors*.

4.5 Eigenvector by Surface

This Window is shown only if the model contains surfaces.

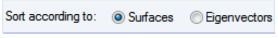


2.5 Eigenvectors by Surface

Surface No.	A Point No.	B Location [m] X	C Y	D Z	E E-vector No.	F Normalized Displacements [-] uX	G uY	H uZ	I Normalized Rotations [-] φX	J φY	K φZ
1	2	9.500	0.000	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00217	-0.00244	0.00140	-0.00964	-0.00251	-0.00079
					4	-0.00087	0.00022	-0.00569	0.05515	0.01261	0.00443
	3	9.500	6.000	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00150	-0.00145	0.00110	0.00111	0.01003	0.00169
					4	-0.00016	-0.00234	-0.00070	-0.03336	0.00965	-0.00294
	5	11.500	3.000	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00292	-0.00138	0.00051	-0.00008	-0.00395	0.00017
					4	-0.00142	-0.00144	-0.00058	-0.00057	0.01468	0.00002
	20	11.465	2.524	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00298	-0.00151	0.00052	-0.00056	-0.00399	0.00014
					4	-0.00143	-0.00152	-0.00015	0.00088	0.01349	-0.00012
	21	9.500	4.500	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00274	-0.00182	-0.00817	0.00912	-0.00751	0.00125
					4	-0.00084	-0.00205	0.06400	-0.04463	0.06007	-0.00205
	22	11.465	3.476	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00283	-0.00126	0.00044	0.00037	-0.00368	0.00021
					4	-0.00145	-0.00149	-0.00075	-0.00240	0.01487	0.00006
	23	11.361	3.942	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00270	-0.00117	0.00032	0.00072	-0.00322	0.00024
					4	-0.00147	-0.00165	-0.00078	-0.00442	0.01454	0.00000
	24	11.189	4.387	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					3	0.00256	-0.00113	0.00016	0.00092	-0.00263	0.00027
					4	-0.00142	-0.00187	-0.00079	-0.00670	0.01415	-0.00010
	25	10.954	4.803	0.000	1	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000
					2	0.00000	0.00000	0.00000	0.00000	0.00000	0.00000

Sort according to: ● Surfaces ○ Eigenvectors

Figure 4.6: Window 2.5 *Eigenvectors by Surface*



Sort according to: ● Surfaces ○ Eigenvectors

You can sort the list according to *Surfaces* or *Eigenvectors*.

Point No.

The results are shown for each grid point of the surface.

You can control the grid points in RFEM, because user-defined result grids are possible for surfaces. This function is described in Chapter 8.12 of the RFEM manual. By default, the result grid has a mesh size of 50 cm.

Location X / Y / Z

The columns B through D (or C through E) indicate the location of the grid points in the global coordinate system.

Normalized Displacements u_X / u_Y / u_Z

The displacements given in the columns F through H refer to the axes of the global coordinate system. They are normalized to the maximum specified in the 1.1 *General Data* Window (see Figure 2.8, page 11).

Normalized Rotations φ_X / φ_Y / φ_Z

The column I through K show the values of the normalized nodal rotations.

5 Evaluation of Results

You can evaluate the eigenvalue analysis in various ways. To evaluate the results graphically, use the RFEM work window.

5.1 Results Windows

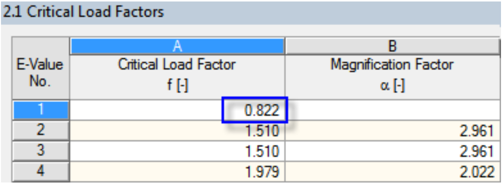
First you should check the *Critical Load Factors* given in Window 2.1.

Negative Critical Load Factor

If a negative critical load factor is shown, this means that no buckling failure due to tension forces could be found. That is, if the load were to act in the opposite direction (inverse signs), the model would buckle. Negative critical load factors can often be avoided by increasing the eigenvectors to be determined or using the ICG iteration method.

Critical load factor < 1

Critical load factors less than 1.00 indicate that the system is unstable.



2.1 Critical Load Factors

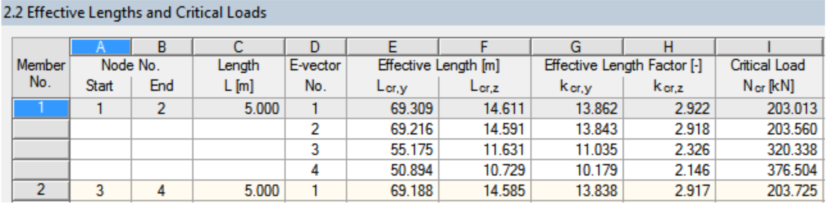
E-Value No.	A Critical Load Factor f [-]	B Magnification Factor α [-]
1	0.822	
2	1.510	2.961
3	1.510	2.961
4	1.979	2.022

Figure 5.1: Unstable system

Only a positive critical load factor that is higher than 1.00 permits to say that the loading due to the specified axial forces multiplied with this factor results in the buckling of the stable structural system.

Effective Lengths

For members, in Window 2.2 the effective length factors k_{cr} are shown for each eigenvector. They describe the buckling behavior perpendicular to the respective axes.



2.2 Effective Lengths and Critical Loads

Member No.	A Node No. Start	B End	C Length L [m]	D E-vector No.	E Effective Length [m] $L_{cr,y}$	F $L_{cr,z}$	G Effective Length Factor [-] $k_{cr,y}$	H $k_{cr,z}$	I Critical Load N_{cr} [kN]
1	1	2	5.000	1	69.309	14.611	13.862	2.922	203.013
				2	69.216	14.591	13.843	2.918	203.560
				3	55.175	11.631	11.035	2.326	320.338
				4	50.894	10.729	10.179	2.146	376.504
2	3	4	5.000	1	69.188	14.585	13.838	2.917	203.725

Figure 5.2: Effective length factors k_{cr} of members

During the analysis, the program increases the axial forces iteratively until the critical load case occurs. From this critical load factor, RF-STABILITY determines the critical load. In turn, the critical load allows the determination of the effective lengths and effective length factors.

If you want to show, for example, the governing effective length factor $k_{cr,y}$ for the deflection perpendicular to the major member y-axis, you normally have to calculate several eigenvectors. For square sections, you obtain the same effective lengths and effective length factors in both axis directions.

The effective length factors for continuous members cannot be determined directly by RF-STABILITY. It is only possible to evaluate the results of the individual members. The member with the lowest critical load N_{cr} can be considered as governing for the entire set of members. Then, the

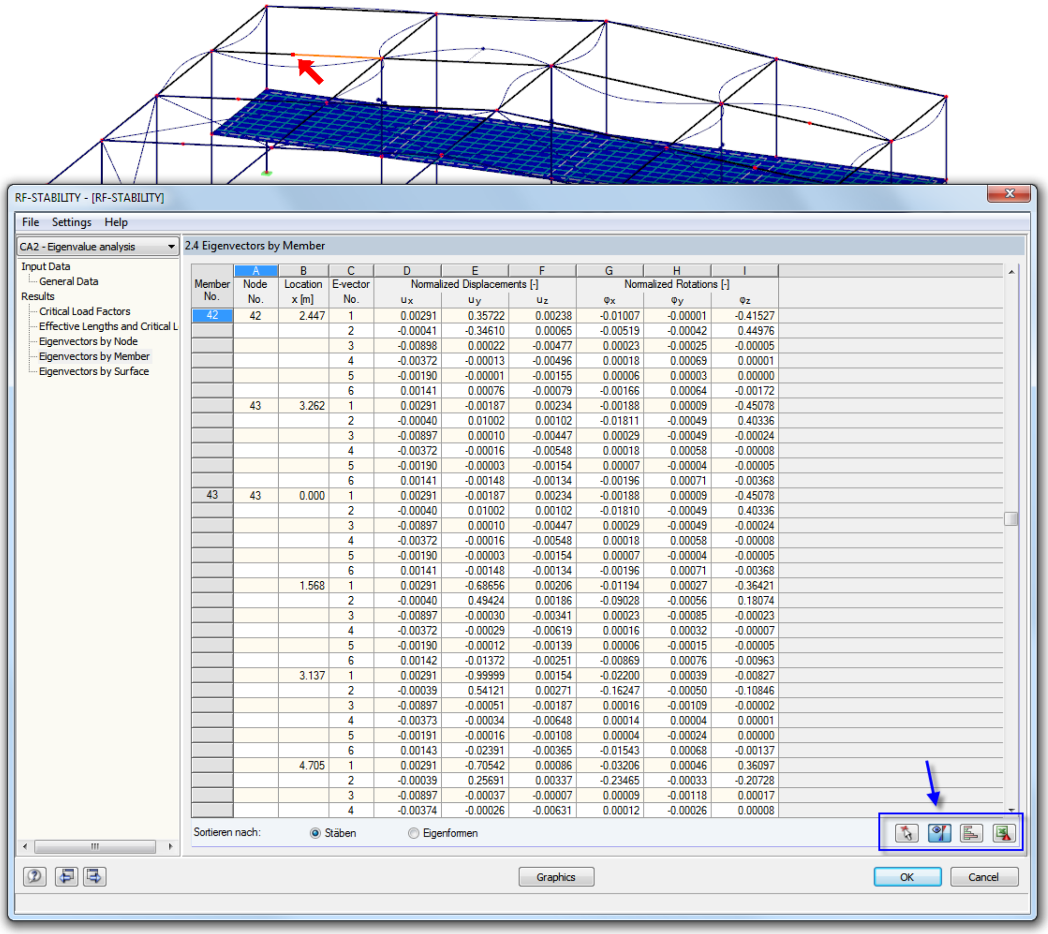
k_{cr} values can be determined from the effective length of this member and the total length of the set of members.

5.2 Results on the RFEM Model

You can also evaluate the results in the RFEM work window.

RFEM background graphic

The RFEM work window in the background is useful for finding the position of a surface or member in the model. The member selected in the results window of RF-STABILITY is marked in the background graphic with the selection color. Furthermore, an arrow indicates the member x-location selected in the currently chosen table row. To locate the member, move the RF-STABILITY window.



2.4 Eigenvectors by Member

Member No.	A Node No.	B Location x [m]	C E-vector No.	D Normalized Displacements [-] ux	E uy	F uz	G Normalized Rotations [-] φx	H φy	I φz
42	42	2.447	1	0.00291	0.35722	0.00238	-0.01007	-0.00001	-0.41527
			2	-0.00041	-0.34610	0.00065	-0.00519	-0.00042	0.44976
			3	-0.00898	0.00022	-0.00477	0.00023	-0.00025	-0.00005
			4	-0.00372	-0.00013	-0.00496	0.00018	0.00069	0.00001
			5	-0.00190	-0.00001	-0.00155	0.00006	0.00003	0.00000
			6	0.00141	0.00076	-0.00079	-0.00166	0.00064	-0.00172
	43	3.262	1	0.00291	-0.00187	0.00234	-0.00188	0.00009	-0.45078
			2	-0.00040	0.01002	0.00102	-0.01811	-0.00049	0.40336
			3	-0.00897	0.00010	-0.00447	0.00029	-0.00049	-0.00024
			4	-0.00372	-0.00016	-0.00548	0.00018	0.00058	-0.00008
			5	-0.00190	-0.00003	-0.00154	0.00007	-0.00004	-0.00005
			6	0.00141	-0.00148	-0.00134	-0.00196	0.00071	-0.00368
43	43	0.000	1	0.00291	-0.00187	0.00234	-0.00188	0.00009	-0.45078
			2	-0.00040	0.01002	0.00102	-0.01810	-0.00049	0.40336
			3	-0.00897	0.00010	-0.00447	0.00029	-0.00049	-0.00024
			4	-0.00372	-0.00016	-0.00548	0.00018	0.00058	-0.00008
			5	-0.00190	-0.00003	-0.00154	0.00007	-0.00004	-0.00005
			6	0.00141	-0.00148	-0.00134	-0.00196	0.00071	-0.00368
		1.568	1	0.00291	-0.68656	0.00206	-0.01194	0.00027	-0.36421
			2	-0.00040	0.49424	0.00186	-0.09028	-0.00056	0.18074
			3	-0.00897	-0.00030	-0.00341	0.00023	-0.00085	-0.00023
			4	-0.00372	-0.00029	-0.00619	0.00016	0.00032	-0.00007
			5	-0.00190	-0.00012	-0.00139	0.00006	-0.00015	-0.00005
			6	0.00142	-0.01372	-0.00251	-0.00869	0.00076	-0.00963
		3.137	1	0.00291	-0.99999	0.00154	-0.02200	0.00039	-0.00827
			2	-0.00039	0.54121	0.00271	-0.16247	-0.00050	-0.10846
			3	-0.00897	-0.00051	-0.00187	0.00016	-0.00109	-0.00002
			4	-0.00373	-0.00034	-0.00648	0.00014	0.00004	0.00001
			5	-0.00191	-0.00016	-0.00108	0.00004	-0.00024	0.00000
			6	0.00143	-0.02391	-0.00365	-0.01543	0.00068	-0.00137
		4.705	1	0.00291	-0.70542	0.00086	-0.03206	0.00046	0.36097
			2	-0.00039	0.25691	0.00337	-0.23465	-0.00033	-0.20728
			3	-0.00897	-0.00037	-0.00007	0.00009	-0.00118	0.00017
			4	-0.00374	-0.00026	-0.00631	0.00012	-0.00026	0.00008

Sortieren nach: ● Stäben ○ Eigenformen

Figure 5.3: Indication of the member in the current eigenvector of the RFEM model

If you cannot improve the display by moving the RF-STABILITY window, click to select the object graphically. The window is now hidden so that you can select the surface or member by clicking it. If necessary, you can also change the view. You can use the functions of the *View* menu, for example zooming, moving, or rotating the display. The indication arrow remains visible.

The buttons in the results windows have the following functions:

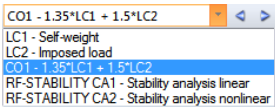
Button	Description	Function
	Select objects in RFEM graphic	Allows you to graphically select an object (member, node, surface) to display its results in the table
	Graphic of the current eigenvector	Shows or hides the eigenvector of the current table row in the RFEM graphic
	Color bars	Shows or hides the color reference scales in the results windows
	Excel export	Exports the table to MS Excel / OpenOffice → Chapter 7.3, page 31

Table 5.1: Buttons in the results windows 2.1 through 2.5

RFEM work window

Graphics

The graphical evaluation of the individual eigenvectors is a good way to evaluate the stability behavior of the model: First, click [Graphic] to exit the RF-STABILITY module. Now, the eigenvectors are displayed graphically on the model in the work window of RFEM similar to the deformations of a load case.



The current RF-STABILITY case is preset. The *Results* navigator controls which displacements or rotations are graphically displayed as an *Eigenvector*.

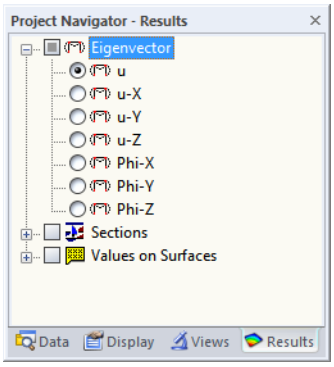


Figure 5.4: *Results* navigator for RF-STABILITY

In addition to the total vector u, you can specifically display the displacement and rotation components in each of the global directions.

The options *Sections* and *Values on Surfaces* can also be used for the graphical evaluation of eigenvectors. These functions are described in Chapter 9.6 and 9.4 of the RFEM manual.

Similar to the display of deformations, the [Show Results] button switches on or off the display of the eigenvectors. To show or hide the result values in the graphic, click the [Show Values] button to the right.

You may deactivate the RFEM tables, because they are of no relevance for the evaluation of the stability analysis.

The panel is adjusted to RF-STABILITY. Chapter 3.4.6 of the RFEM manual describes the standard functions in detail.

In the *Factors* tab of the panel, you can select the eigenvectors.

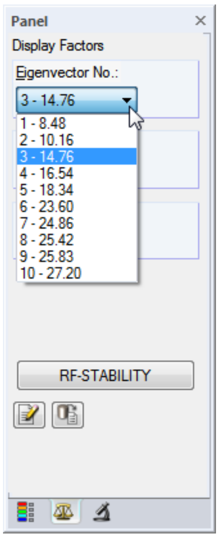


Figure 5.5: Selection of eigenvector in the *Factors*

If the members or surfaces that are prone to buckling are difficult to find, increase the *Factor* of the deformation in the *Factors* tab of the panel. The animation of deformations is also a useful function. To activate it, click the button shown on the left.

You can control the results display of the members in the *Display* navigator by selecting ***Results*** → ***Deformation*** → ***Members***. By default, the eigenvectors are shown as *Lines*. The two other options can also be used to represent the buckling behavior.

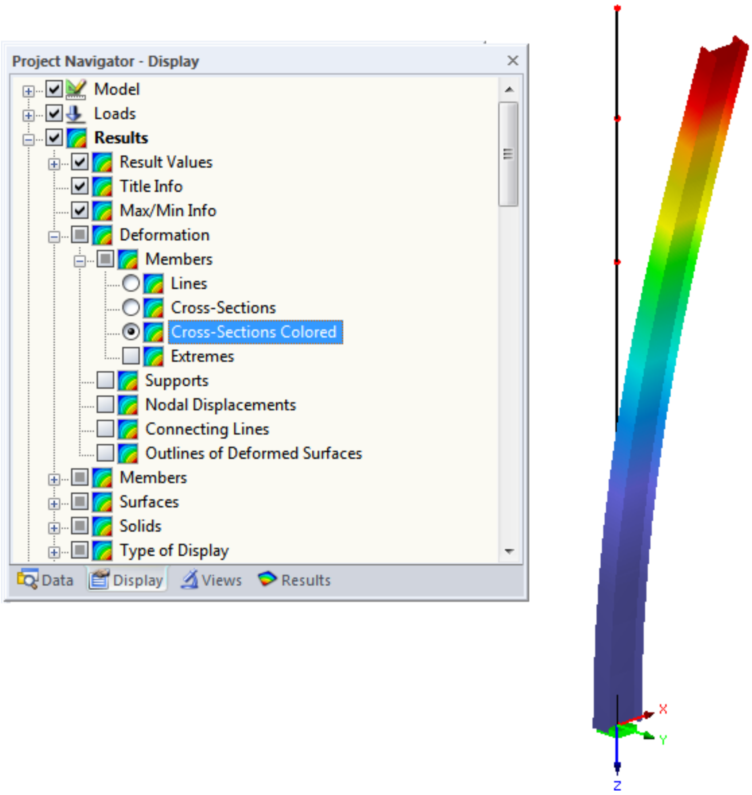


Figure 5.6: *Display* navigator: Results → Deformation → Members → Cross-Sections Colored

You can transfer the graphics of the eigenvectors to the printout report (see Chapter 6.2, page 26).

To return to the add-on module, click [RF-STABILITY] in the panel.

5.3 Filter for Results

The results windows in RF-STABILITY allow for a selection by various criteria. In addition to this, you can use the filter options described in the RFEM manual, Chapter 9.9. The options allow you to graphically evaluate the results of the stability analysis.

In RF-STABILITY, you can also use the options of the *Visibilities* (see RFEM manual, Chapter 9.9.1) to filter the members in order to evaluate them.

Filtering eigenvectors

Graphics

The normalized deformations can be used as filter criterion in the RFEM work window. To go to the RFEM window, click [Graphics]. To apply this filter function, the panel must be displayed. If it is not, select in the RFEM menu

View → Control Panel

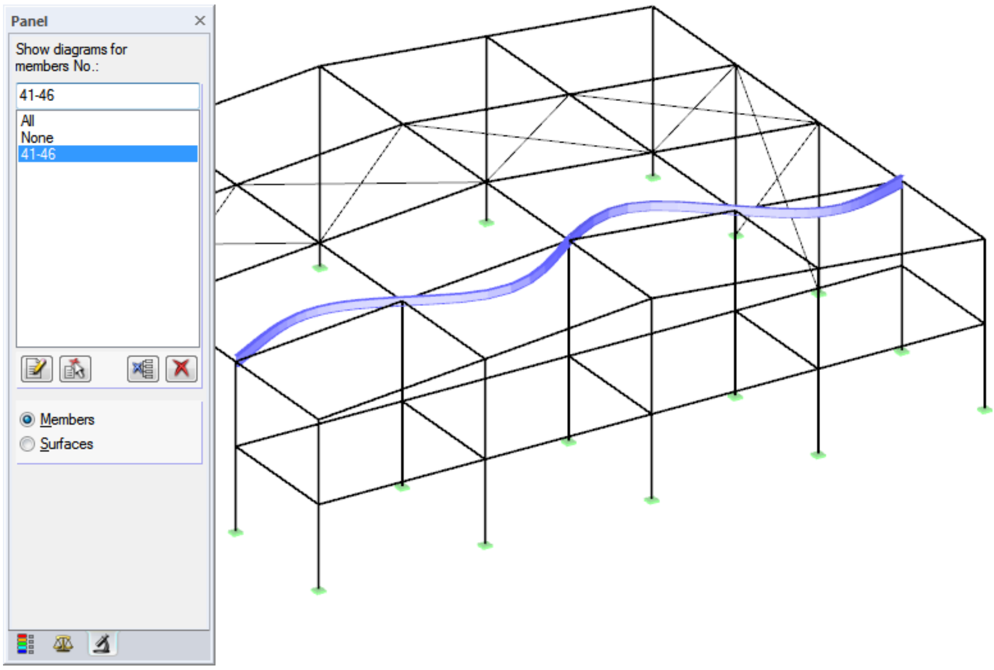
or use the toolbar button shown on the left.

The panel is described in the RFEM manual, Chapter 3.4.6. You define the filter settings for the results in the first tab of the panel (Color Spectrum). This tab is not available for the line or cross-section display. Therefore, you have to select the *Cross-Sections Colored* option in the *Display* navigator (see Figure 5.6).

In the panel you can specify, for example, that only normalized deformations greater than 0.55 are shown. This option makes it easier to locate the areas prone to buckling.

Filtering members and surfaces

In the *Filter* tab of the control panel, you can specify the numbers of selected members or surfaces to filter the display of the eigenvectors. This function is described in the RFEM manual, Chapter 9.9.3. In contrast to the visibility function, the model is displayed, too.



Show diagrams for members No.: 41-46

● Members
○ Surfaces

Figure 5.7: Member filter for eigenvectors of a frame beam

6 Printout

6.1 Printout Report

Like in RFEM, the program generates a printout report for the RF-STABILITY results, to which you can add graphics and descriptions. In the printout report, you can select which data of the stability analysis will finally appear in the printout.

The printout report is described in the RFEM manual. In particular, Chapter 10.1.3.4 *Selecting Data of Add-on Modules* describes how to select input and output data from add-on modules for the printout report.

6.2 Graphic Printout

In RFEM, you can add every picture displayed in the work window to the printout report or send it directly to a printer. In this way, you can also prepare the eigenvectors displayed on the RFEM model for the printout.

Printing graphics is described in the RFEM manual, Chapter 10.2.

To print the current eigenvector, click

File → Print

or use the toolbar button shown on the left.

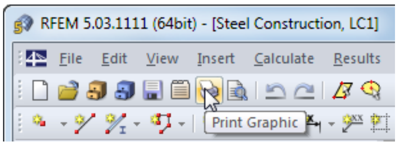


Figure 6.1: Button [Print Graphic] in RFEM toolbar

The result distributions of sections or members can also be transferred to the report or printed directly by using the [Print] button.

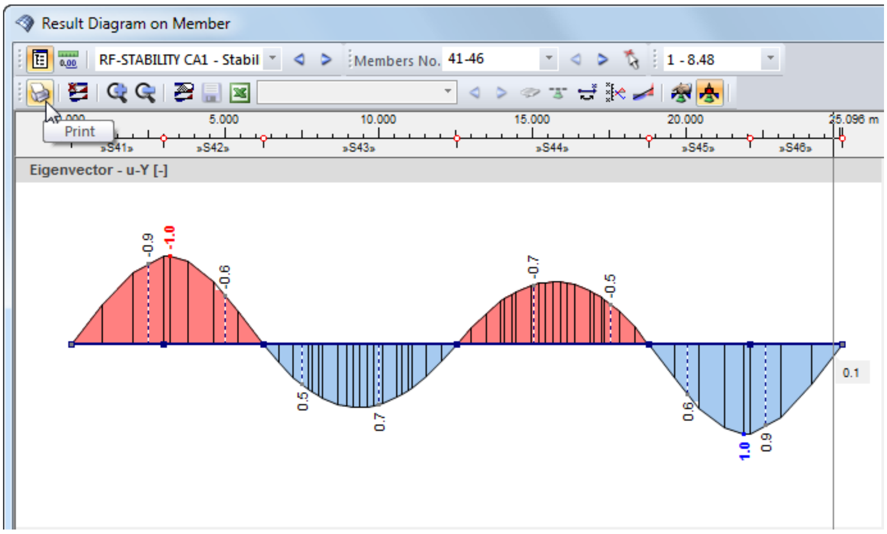
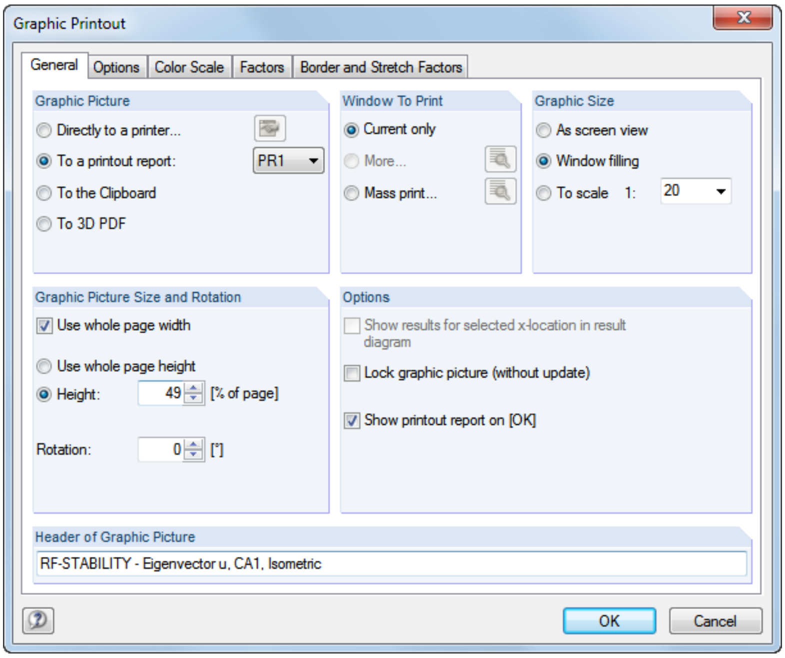


Figure 6.2: Button *Print* in the toolbar of the results diagram window

The *Printout Report* dialog box appears (see the following figure).



Graphic Printout

General | Options | Color Scale | Factors | Border and Stretch Factors

Graphic Picture
- ○ Directly to a printer...
- ● To a printout report: PR1
- ○ To the Clipboard
- ○ To 3D PDF

Window To Print
- ● Current only
- ○ More...
- ○ Mass print...

Graphic Size
- ○ As screen view
- ● Window filling
- ○ To scale 1: 20

Graphic Picture Size and Rotation
- ☑ Use whole page width
- ○ Use whole page height
- ● Height: 49 [% of page]
- Rotation: 0 [°]

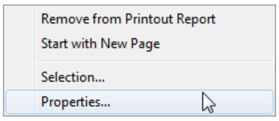
Options
- ☐ Show results for selected x-location in result diagram
- ☐ Lock graphic picture (without update)
- ☑ Show printout report on [OK]

Header of Graphic Picture: RF-STABILITY - Eigenvector u, CA1, Isometric

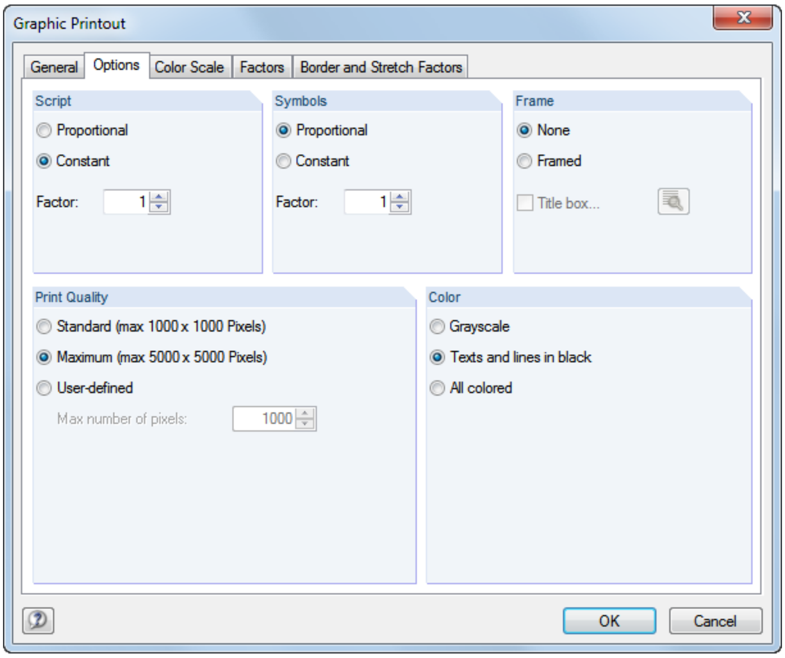
Figure 6.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in the RFEM manual, Chapter 10.2. The manual also describes the *Options* and *Color Scale* tab.

To move a graphic within the printout report, use the drag-and-drop function.



To adjust a graphic in the printout report after the fact, right-click the corresponding entry in the navigator of the printout report. The *Properties* option in the context menu return you to the *Graphic Printout* dialog box, offering various options for adjustment.



Graphic Printout

General | Options | Color Scale | Factors | Border and Stretch Factors

Script
- ○ Proportional
- ● Constant
- Factor: 1

Symbols
- ● Proportional
- ○ Constant
- Factor: 1

Frame
- ● None
- ○ Framed
- ☐ Title box...

Print Quality
- ○ Standard (max 1000 x 1000 Pixels)
- ● Maximum (max 5000 x 5000 Pixels)
- ○ User-defined
- Max number of pixels: 1000

Color
- ○ Grayscale
- ● Texts and lines in black
- ○ All colored

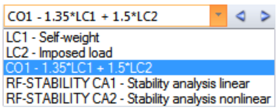
Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

7 General Functions

This chapter describes useful menu functions and export options for the results of the stability analysis.

7.1 RF-STABILITY Analysis Cases

Analysis cases (or RF-STABILITY cases) allow you to evaluate different aspects of stability. For example, you can analyze the influence of the axial force from different load cases or load combinations with or without the consideration of tensile forces.


CO1 - 1.35*LC1 + 1.5*LC2
LC1 - Self-weight
LC2 - Imposed load
CO1 - 1.35*LC1 + 1.5*LC2
RF-STABILITY CA1 - Stability analysis linear
RF-STABILITY CA2 - Stability analysis nonlinear

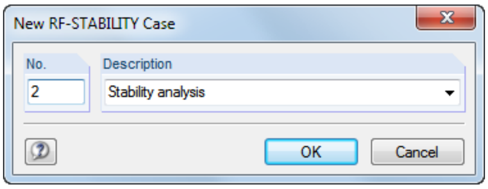
To select the analysis cases of RF-STABILITY, you can also use the load case list in the RFEM toolbar.

Creating a new analysis case

To create a new analysis case, select on the RF-STABILITY menu

File → **New Case**.

The following dialog box appears:



No. 2
Description: Stability analysis

Figure 7.1: Dialog box *New RF-STABILITY Case*

Enter a *Number* (one that is still available) for the new RF-STABILITY case. Define an appropriate *Description* to make the selection in the load case list easier later on.

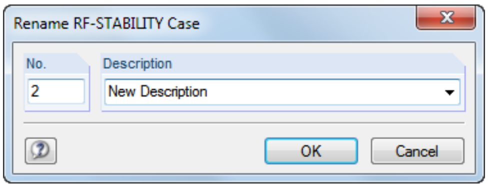
Click [OK] to display the RF-STABILITY Window 1.1 *General Data*, where you can enter the new calculation parameters.

Renaming an analysis case

To change the description of an analysis case, select on the RF-STABILITY menu

File → **Rename Case**.

The following dialog box appears:



No. 2
Description: New Description

Figure 7.2: Dialog box *Rename RF-STABILITY Case*

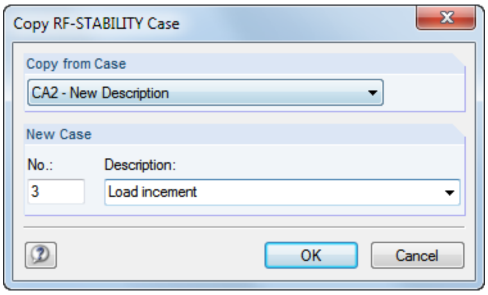
In this dialog box, you can specify a different *Description* as well as a different *Number* for the analysis case.

Copying an analysis case

To copy the input data of the current analysis case, select on the RF-STABILITY menu

File → **Copy Case**.

The following dialog box appears:



Copy from Case: CA2 - New Description

No.: 3

Description: Load incement

Figure 7.3: Dialog box *Copy RF-STABILITY Case*

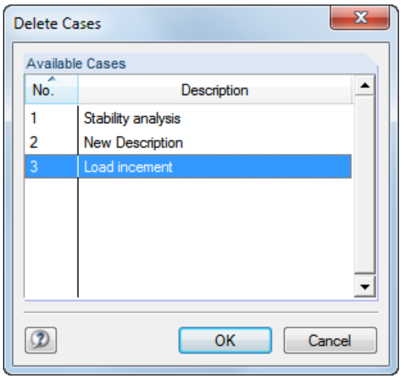
Define the *No.* and, if necessary, a *Description* for the new case.

Deleting an analysis case

To delete an analysis case, select on the RF-STABILITY menu

File → **Delete Case**.

The following dialog box appears:



No.	Description
1	Stability analysis
2	New Description
3	Load incement

Figure 7.4: Dialog box *Delete Cases*

You can select the analysis case in the *Available Cases* list. To delete the selected case, click [OK].

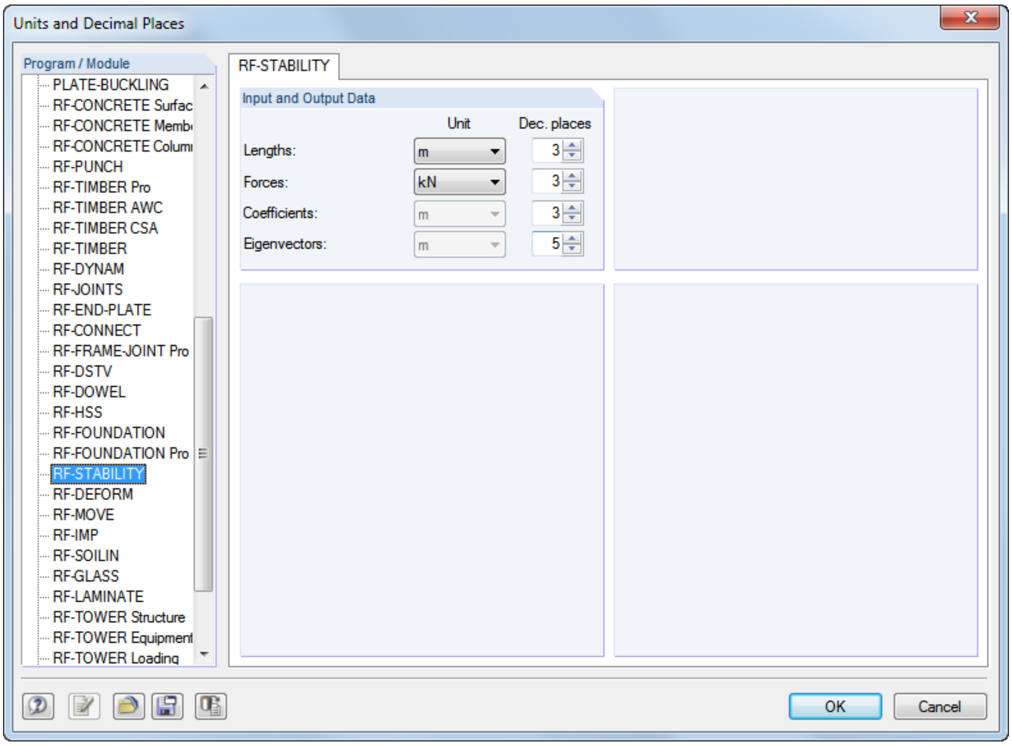
7.2 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed in one common dialog box. In the add-on module RF-STABILITY, you can use the menu to define the units.

To open the corresponding dialog box, select

Settings → **Units and Decimal Places**.

The dialog box known from RFEM appears. RF-STABILITY is preset in the list *Program / Module*.



	Unit	Dec. places
Lengths:	m	3
Forces:	kN	3
Coefficients:	m	3
Eigenvectors:	m	5

Figure 7.5: Dialog box *Units and Decimal Places*

To reuse the settings in other models, save them as a user-profile. These functions are described in the RFEM manual, Chapter 11.1.3.

7.3 Exporting Results

The results of RF-STABILITY can also be used in other programs.

Clipboard

To copy cells selected in the results windows to the Clipboard, press the keys [Ctrl]+[C]. To insert the cells, for example in a word processor, press [Ctrl]+[V]. The headers of the table columns are not transferred.

Printout report

You can print the data of the RF-STABILITY add-on module into the global printout report (see Chapter 6.1, page 26) to export them subsequently. Then, in the printout report, click

File → Export in RTF.

This function is described in the RFEM manual, Chapter 10.1.11.

Excel / OpenOffice

RF-STABILITY provides a function for the direct data export to MS Excel, OpenOffice Calc, or the CSV file format. To open the corresponding dialog box, select on the menu of the module

File → Export Tables.

The following dialog box appears.

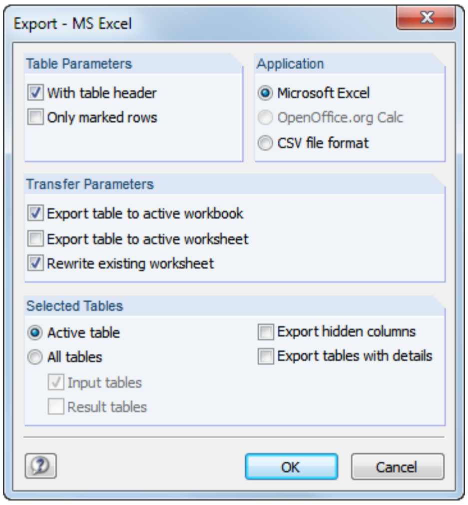


Table Parameters
☑ With table header
☐ Only marked rows

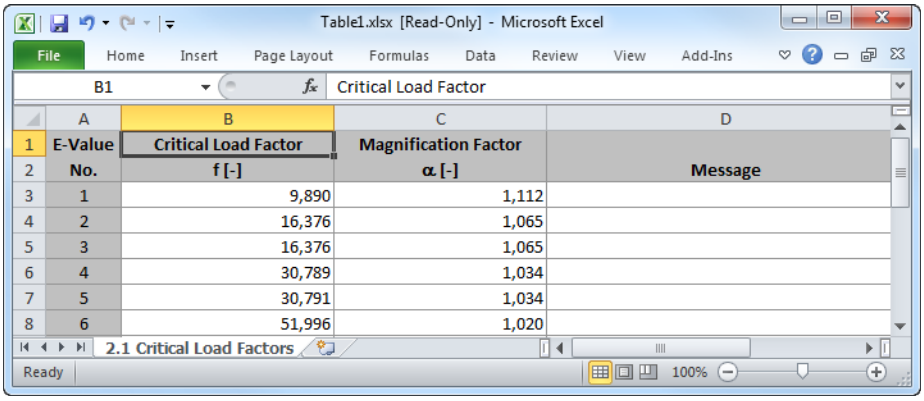
Application
● Microsoft Excel
○ OpenOffice.org Calc
○ CSV file format

Transfer Parameters
☑ Export table to active workbook
☐ Export table to active worksheet
☑ Rewrite existing worksheet

Selected Tables
● Active table
○ All tables
☑ Input tables
☐ Result tables
☐ Export hidden columns
☐ Export tables with details

Figure 7.6: Dialog box *Export - MS Excel*

Having selected the relevant options, you can start the export by clicking [OK]. Excel or OpenOffice are started automatically; that is, you do not need to open the programs beforehand.

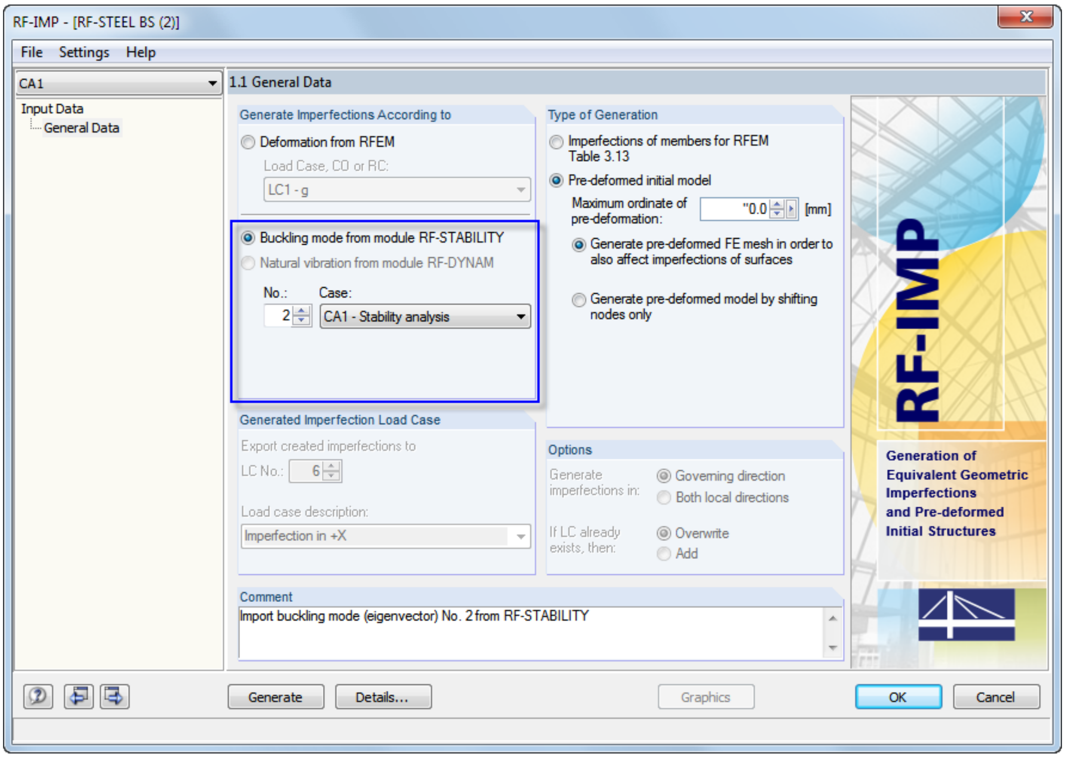


	A	B	C	D
1	E-Value	Critical Load Factor	Magnification Factor	
2	No.	f [-]	α [-]	Message
3	1	9,890	1,112	
4	2	16,376	1,065	
5	3	16,376	1,065	
6	4	30,789	1,034	
7	5	30,791	1,034	
8	6	51,996	1,020	

Figure 7.7: Results in *Excel*

RF-IMP

If you want to use a buckling mode (eigenvector) from RF-STABILITY in the add-on module RF-IMP to generate equivalent imperfections or a pre-deformed initial model, you do not need to export the data. RF-IMP allows you to select the relevant eigenvector *No.* as well as the RF-STABILITY *Case* directly in the corresponding lists.



RF-IMP - [RF-STEEL BS (2)]

CA1

Input Data
General Data

1.1 General Data

Generate Imperfections According to
○ Deformation from RFEM
Load Case, CO or RC: LC1 - g
● Buckling mode from module RF-STABILITY
○ Natural vibration from module RF-DYNAM
No.: 2 Case: CA1 - Stability analysis

Type of Generation
○ Imperfections of members for RFEM Table 3.13
● Pre-deformed initial model
Maximum ordinate of pre-deformation: "0.0 [mm]
● Generate pre-deformed FE mesh in order to also affect imperfections of surfaces
○ Generate pre-deformed model by shifting nodes only

Generated Imperfection Load Case
Export created imperfections to
LC No.: 6
Load case description: Imperfection in +X

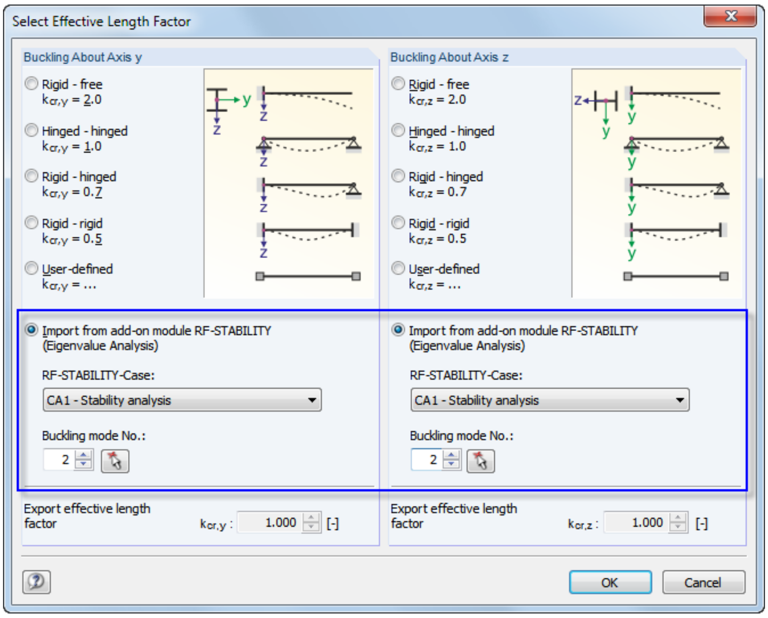
Options
Generate imperfections in: ● Governing direction ○ Both local directions
If LC already exists, then: ● Overwrite ○ Add

Comment
Import buckling mode (eigenvector) No. 2 from RF-STABILITY

Figure 7.8: Importing the eigenvector in the module RF-IMP

RF-STEEL EC3 / RF-ALUMINIUM / RF-KAPPA / RF-TIMBER Pro

The modules RF-STEEL ISC/AS/BS/CS/EC3/GB/IS/NTC-DF/SANS/SIA/SP, RF-ALUMINIUM, RF-KAPPA, and RF-TIMBER Pro provide the option to use the effective length factors from RF-STABILITY directly for the members that you want to analyze.



Select Effective Length Factor

Buckling About Axis y
○ Rigid - free $k_{cr,y} = 2.0$
○ Hinged - hinged $k_{cr,y} = 1.0$
○ Rigid - hinged $k_{cr,y} = 0.7$
○ Rigid - rigid $k_{cr,y} = 0.5$
○ User-defined $k_{cr,y} = \dots$
● Import from add-on module RF-STABILITY (Eigenvalue Analysis)
RF-STABILITY-Case: CA1 - Stability analysis
Buckling mode No.: 2
Export effective length factor $k_{cr,y}$: 1.000 [-]

Buckling About Axis z
○ Rigid - free $k_{cr,z} = 2.0$
○ Hinged - hinged $k_{cr,z} = 1.0$
○ Rigid - hinged $k_{cr,z} = 0.7$
○ Rigid - rigid $k_{cr,z} = 0.5$
○ User-defined $k_{cr,z} = \dots$
● Import from add-on module RF-STABILITY (Eigenvalue Analysis)
RF-STABILITY-Case: CA1 - Stability analysis
Buckling mode No.: 2
Export effective length factor $k_{cr,z}$: 1.000 [-]

Figure 7.9: Selection of effective length factors in the add-on module RF-STEEL EC3

8 Worked Examples

8.1 Rectangular Plate

Determine the critical buckling load of a square plate with a length of 1 m.

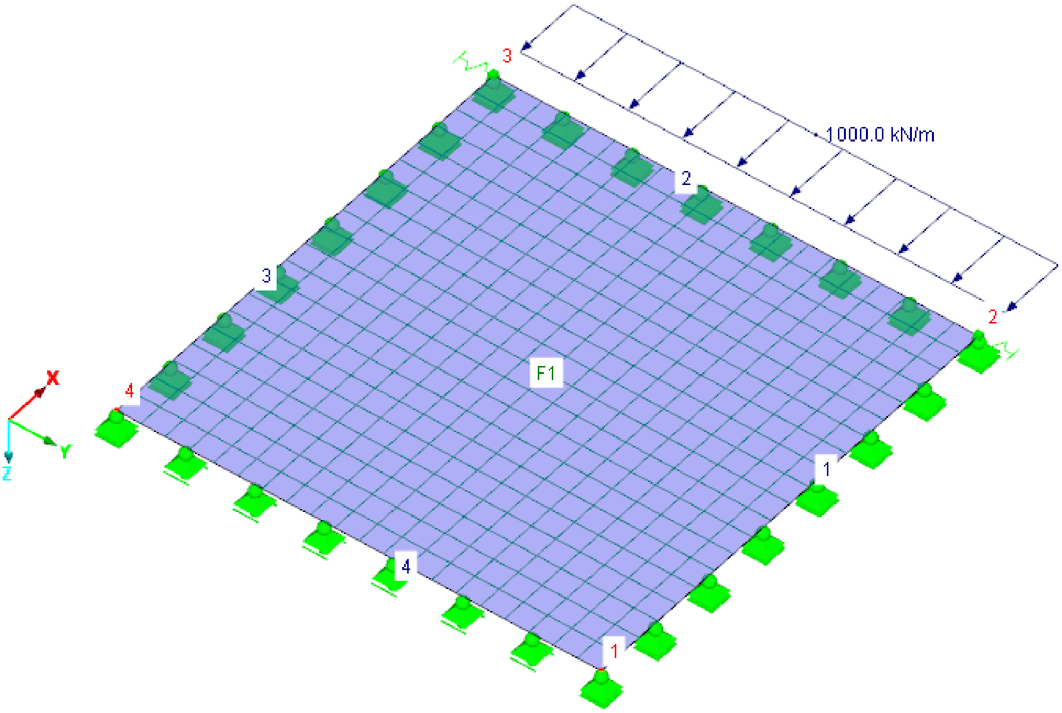


Figure 8.1: FE-model and loading

Analytical solution

This surface is simply supported on all edges. The critical load N_{cr} can be analytically determined according to the following equation:

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

For the square plate with a length of 100 cm, you obtain:

$k = 4$

$a = 100$ cm

Steel S 235 is used as material.

$E = 21.000\ \text{kN}/\text{cm}^2$

$\nu = 0.30$

The surface has a thickness of 2 cm:

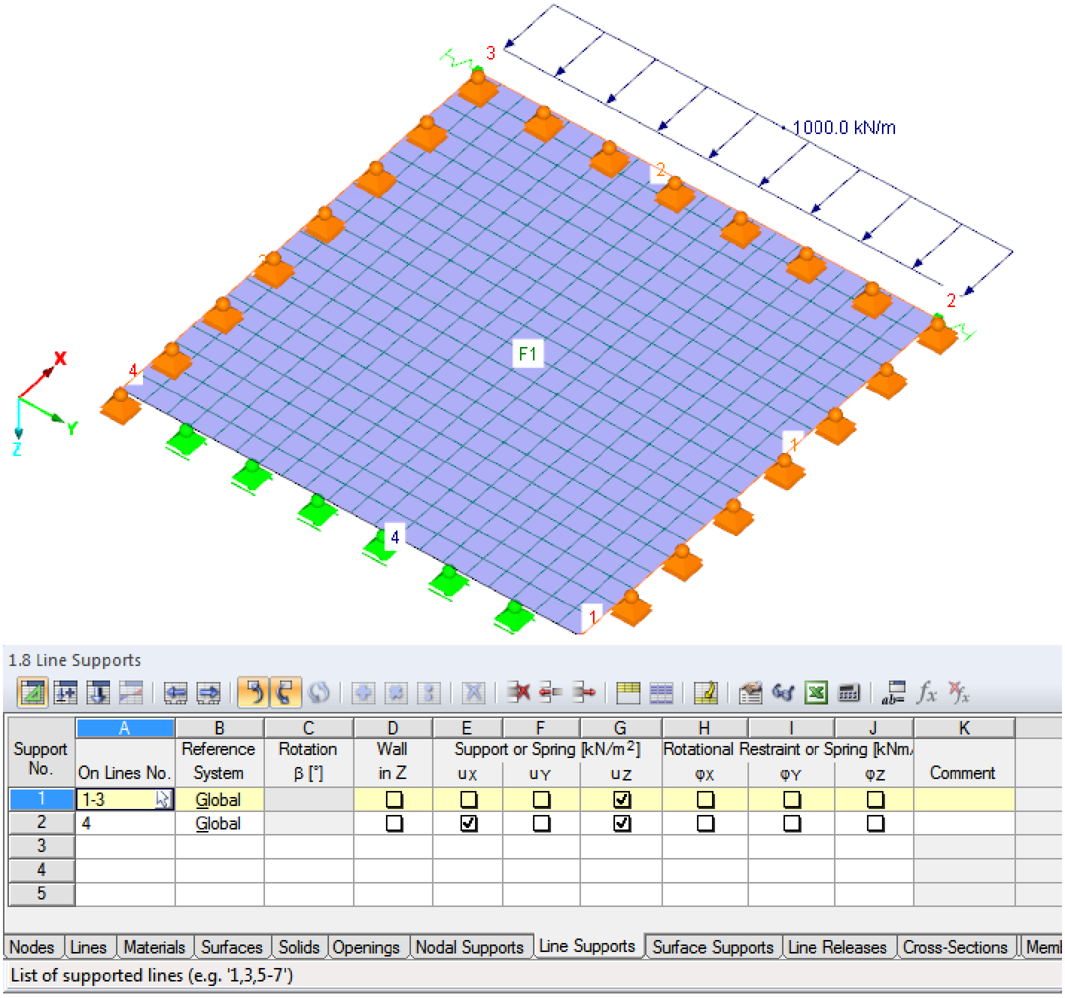
$h = 2$ cm

For the critical load, you obtain:

$$N_{cr} = 4 \cdot \frac{\pi^2 \cdot 21{,}000 \cdot 2^3}{12 \cdot (1 - 0.30^2) \cdot 100^2} = 60.736\ \text{kN}/\text{cm} = 6{,}037.6\ \text{kN}/\text{m}$$

Solution with RFEM

The plate is defined with the following support conditions in RFEM.

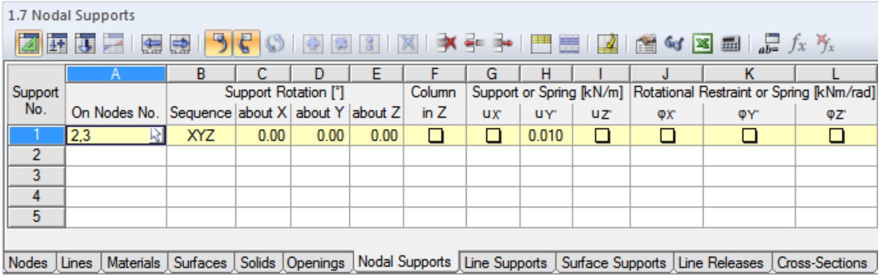


Support No.	A On Lines No.	B Reference System	C Rotation β [°]	D Wall in Z	E Support or Spring [kN/m2] uX	F uY	G uZ	H Rotational Restraint or Spring [kNm/ φX	I φY	J φZ	K Comment
1	1-3	Global		☐	☐	☐	☑	☐	☐	☐	
2	4	Global		☐	☑	☐	☑	☐	☐	☐	
3											
4											
5											

List of supported lines (e.g. '1,3,5-7')

Figure 8.2: Line support

In addition, nodal supports with a small spring stiffness are defined for nodes 2 and 3, thus supporting the system in Y-direction.



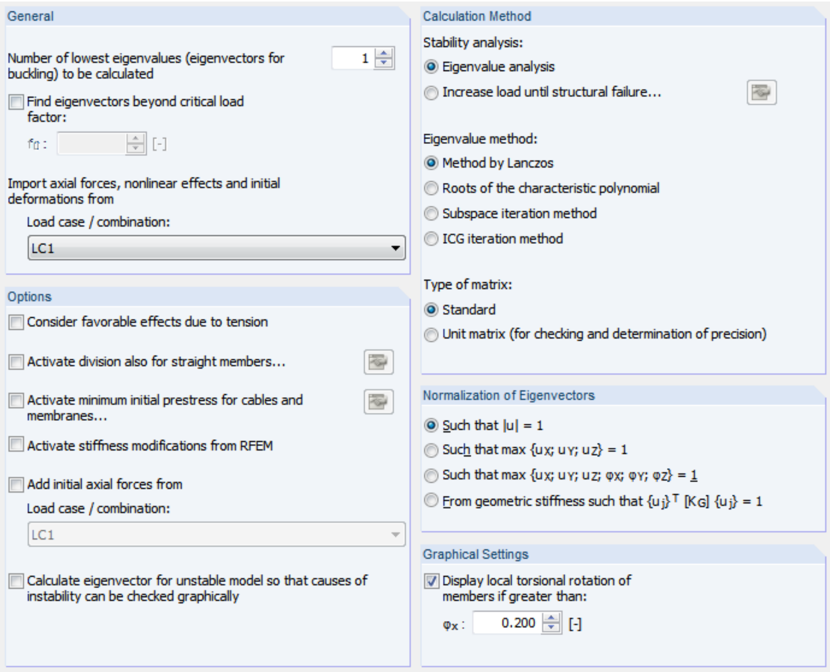
Support No.	A On Nodes No.	B Support Rotation [°] Sequence	C about X	D about Y	E about Z	F Column in Z	G Support or Spring [kN/m] uX'	H uY'	I uZ'	J Rotational Restraint or Spring [kNm/rad] φX'	K φY'	L φZ'
1	2,3	XYZ	0.00	0.00	0.00	☐	☐	0.010	☐	☐	☐	☐
2												
3												
4												
5												

Figure 8.3: Nodal support

The FE mesh has a size of 5 cm.

A uniform line load of 1,000 kN/m is applied as shown in Figure 8.1.

Specify the following in RF-STABILITY:



General

Number of lowest eigenvalues (eigenvectors for buckling) to be calculated: 1

☐ Find eigenvectors beyond critical load factor:

f_{cr} : [-]

Import axial forces, nonlinear effects and initial deformations from

Load case / combination: LC1

Options

☐ Consider favorable effects due to tension

☐ Activate division also for straight members...

☐ Activate minimum initial prestress for cables and membranes...

☐ Activate stiffness modifications from RFEM

☐ Add initial axial forces from

Load case / combination: LC1

☐ Calculate eigenvector for unstable model so that causes of instability can be checked graphically

Calculation Method

Stability analysis:

● Eigenvalue analysis

○ Increase load until structural failure...

Eigenvalue method:

● Method by Lanczos

○ Roots of the characteristic polynomial

○ Subspace iteration method

○ ICG iteration method

Type of matrix:

● Standard

○ Unit matrix (for checking and determination of precision)

Normalization of Eigenvectors

● Such that |u| = 1

○ Such that max $\{u_X; u_Y; u_Z\}$ = 1

○ Such that max $\{u_X; u_Y; u_Z; \varphi_X; \varphi_Y; \varphi_Z\}$ = 1

○ From geometric stiffness such that $\{u_j\}^T [K_G] \{u_j\} = 1$

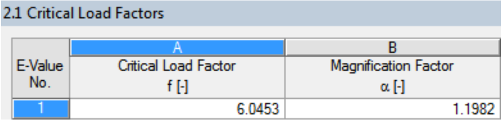
Graphical Settings

☑ Display local torsional rotation of members if greater than:

φ_x : 0.200 [-]

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

As a result, RF-STABILITY yields a critical load factor of **6.0453**.



2.1 Critical Load Factors

E-Value No.	A Critical Load Factor f [-]	B Magnification Factor α [-]
1	6.0453	1.1982

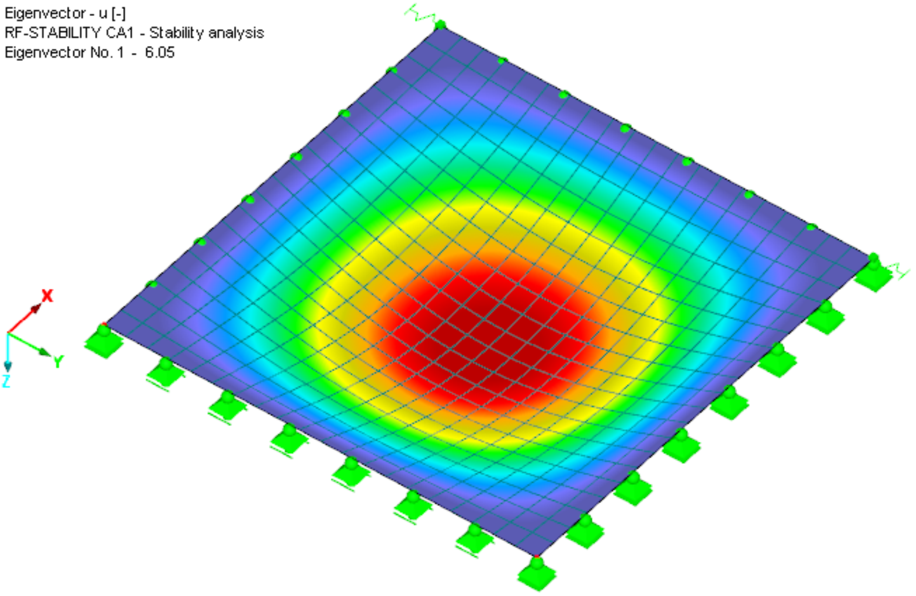
Figure 8.5: Critical load factor

From the critical load factor and the applied loading, you obtain:

$N_{cr} = 6.0453 \cdot 1{,}000 \text{ kN/m} = 6{,}045.3 \text{ kN/m}$

Thus, the difference to the analytical solution is about 0.5 %.

RF-STABILITY determines the following eigenvector:



Eigenvector - u [-]
RF-STABILITY CA1 - Stability analysis
Eigenvector No. 1 - 6.05

Figure 8.6: RF-STABILITY eigenvector

8.2 Circular Plate

Determine the critical load of a circular plate with a diameter of 3 m.

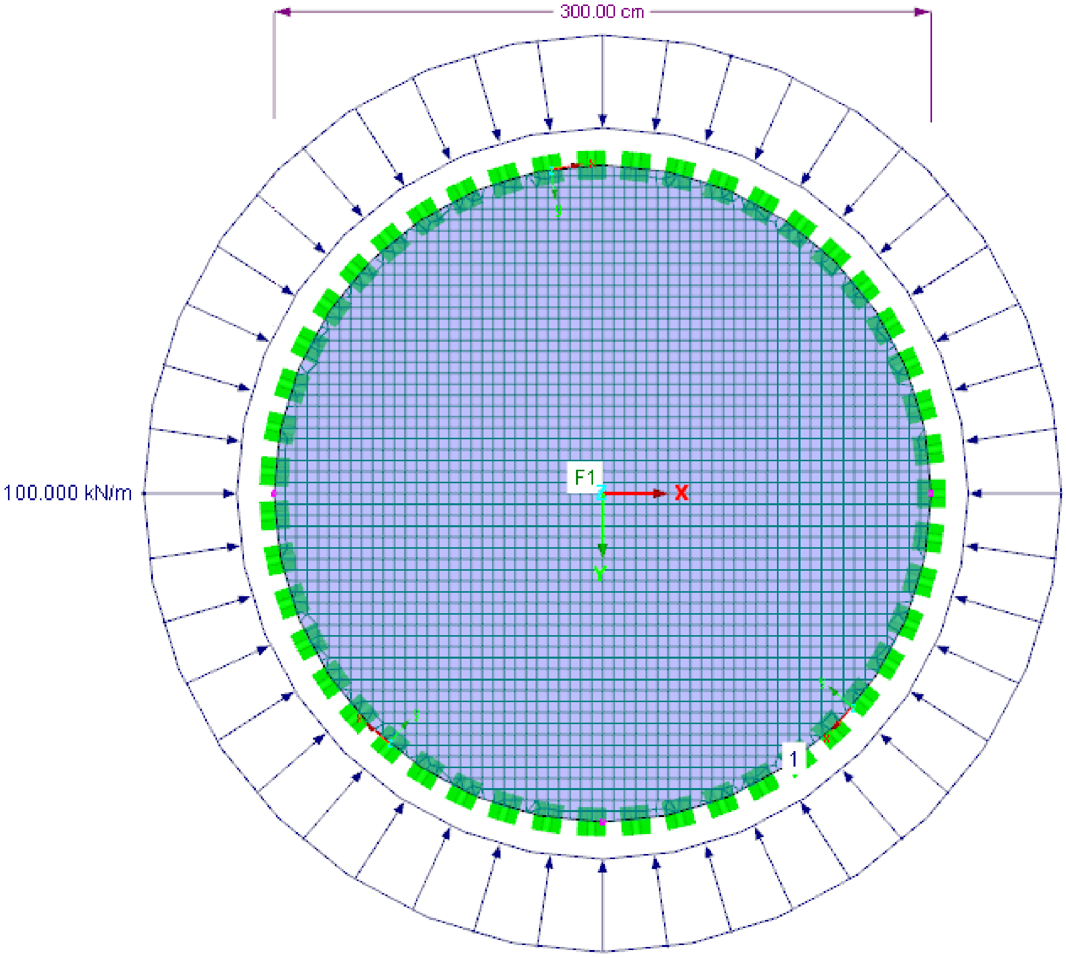


Figure 8.7: FE-model and loading

Analytical solution

The surface is simply supported on all edges. The critical load N_{cr} can be determined by using the following expression (see [3], p. 559):

$$N_{cr} = 14.86 \cdot \frac{E \cdot h^3}{12 \cdot (1 - \nu^2) \cdot a^2}$$

The plate has a radius of 150 cm, therefore:

$a = 150$ cm

Steel S235 is used as material.

$E = 21{,}000\ \mathrm{kN/cm^2} \nu = 0.30$

The surface has a thickness of 2 cm.

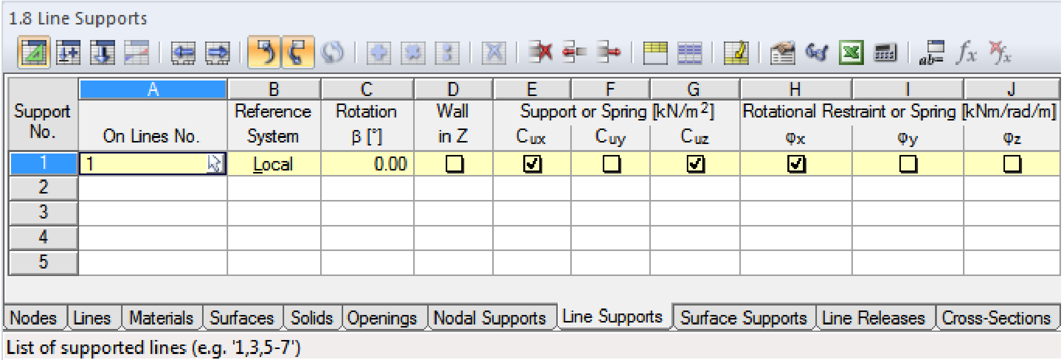
$h = 2$ cm

For the critical load, you obtain:

$$N_{cr} = 14.68 \cdot \frac{21{,}000 \cdot 2^3}{12 \cdot (1 - 0.30^2 \cdot 150^2} = 10.038\ \mathrm{kN/cm} = 1{,}003.8\ \mathrm{kN/m}$$

Solution with RFEM

The plate in RFEM has a locally defined line support.



1.8 Line Supports

	A	B	C	D	E	F	G	H	I	J
Support No.	On Lines No.	Reference System	Rotation β [°]	Wall in Z	Support or Spring [kN/m²]			Rotational Restraint or Spring [kNm/rad/m]		
					C_{ux}	C_{uy}	C_{uz}	φ_x	φ_y	φ_z
1	1	Local	0.00	☐	☑	☐	☑	☑	☐	☐
2										
3										
4										
5										

List of supported lines (e.g. '1,3,5-7')

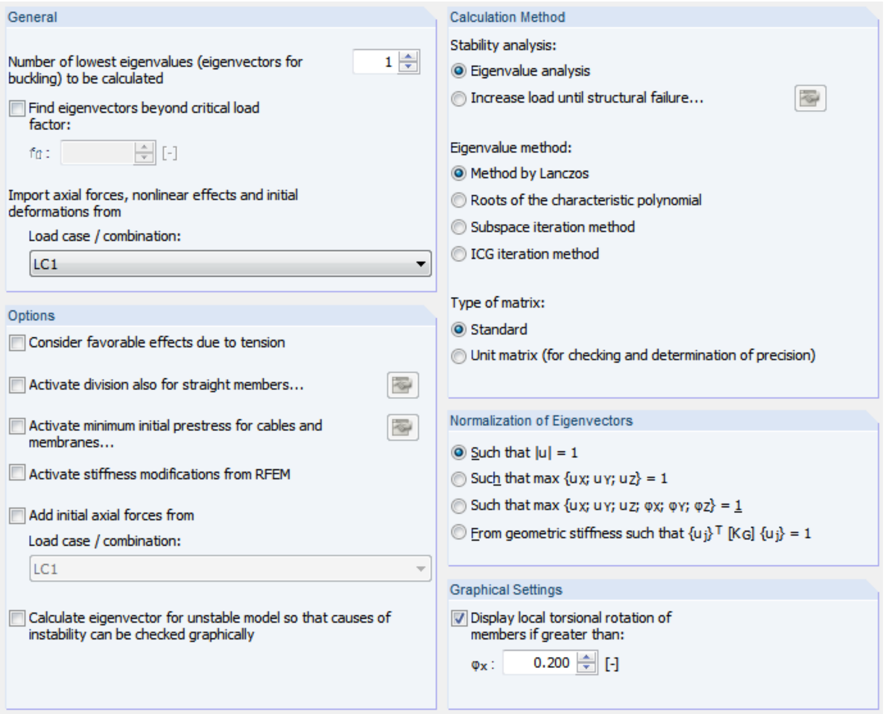
Figure 8.8: Line support

The rotation at the edge is thus restrained. Furthermore, the plate is supported in the perpendicular direction. The support in the line direction x prevents the torsion of the plate about the global Z-axis.

For the FE mesh, define an element length of 5 cm.

As loading, apply a line load of 100 kN/m corresponding to Figure 8.7.

Specify the following in RF-STABILITY:



General

Number of lowest eigenvalues (eigenvectors for buckling) to be calculated: 1

☐ Find eigenvectors beyond critical load factor:

f_{cr}: [-]

Import axial forces, nonlinear effects and initial deformations from

Load case / combination: LC1

Options

☐ Consider favorable effects due to tension

☐ Activate division also for straight members...

☐ Activate minimum initial prestress for cables and membranes...

☐ Activate stiffness modifications from RFEM

☐ Add initial axial forces from

Load case / combination: LC1

☐ Calculate eigenvector for unstable model so that causes of instability can be checked graphically

Calculation Method

Stability analysis:

● Eigenvalue analysis

○ Increase load until structural failure...

Eigenvalue method:

● Method by Lanczos

○ Roots of the characteristic polynomial

○ Subspace iteration method

○ ICG iteration method

Type of matrix:

● Standard

○ Unit matrix (for checking and determination of precision)

Normalization of Eigenvectors

● Such that |u| = 1

○ Such that max {ux; uY; uZ} = 1

○ Such that max {ux; uY; uZ; φX; φY; φZ} = 1

○ From geometric stiffness such that $\{u_j\}^T$ [KG] $\{u_j\}$ = 1

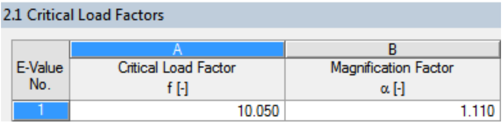
Graphical Settings

☑ Display local torsional rotation of members if greater than:

φ_x: 0.200 [-]

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

As result, RF-STABILITY yields a critical load factor of **10.050**.



2.1 Critical Load Factors

	A	B
E-Value No.	Critical Load Factor f [-]	Magnification Factor α [-]
1	10.050	1.110

Figure 8.10: Critical load factor

From the critical load factor and the applied loading, you obtain:

$$N_{cr} = 10.050 \cdot 100\ \mathrm{kN/m} = 1{,}005.0\ \mathrm{kN/m}$$

The difference to the analytical solution is thus about 0.1 %.

RF-STABILITY determines the following eigenvector:

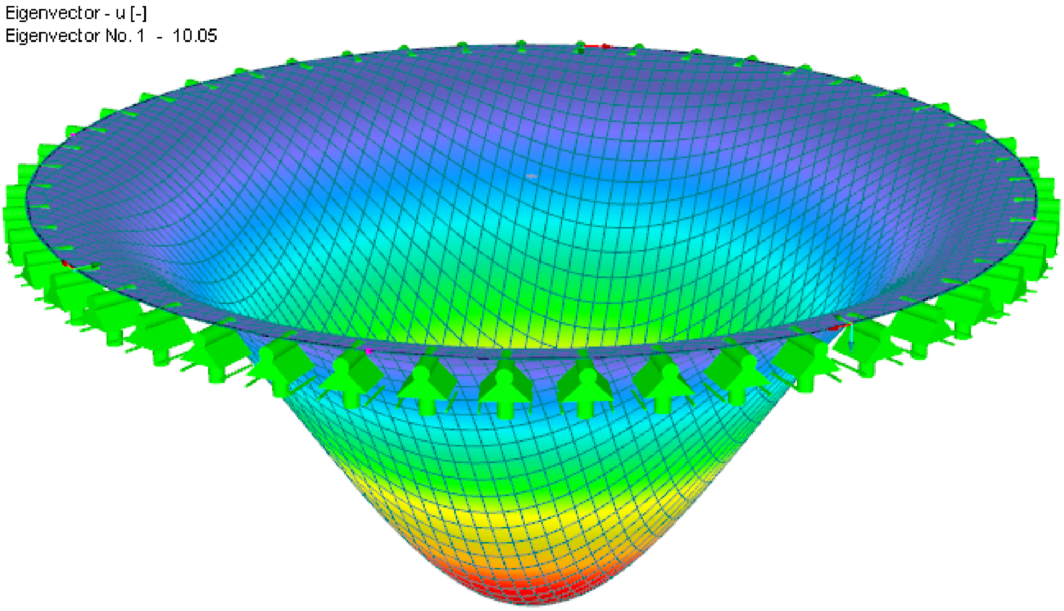


Figure 8.11: RF-STABILITY eigenvector

8.3 Tapered Cantilever

This example is from [4]. The paper looks at the buckling modes and the loadings of tapered T-cantilevers by using experiments and FEM calculations.

In this example, beam 1 from Table 4 is used.

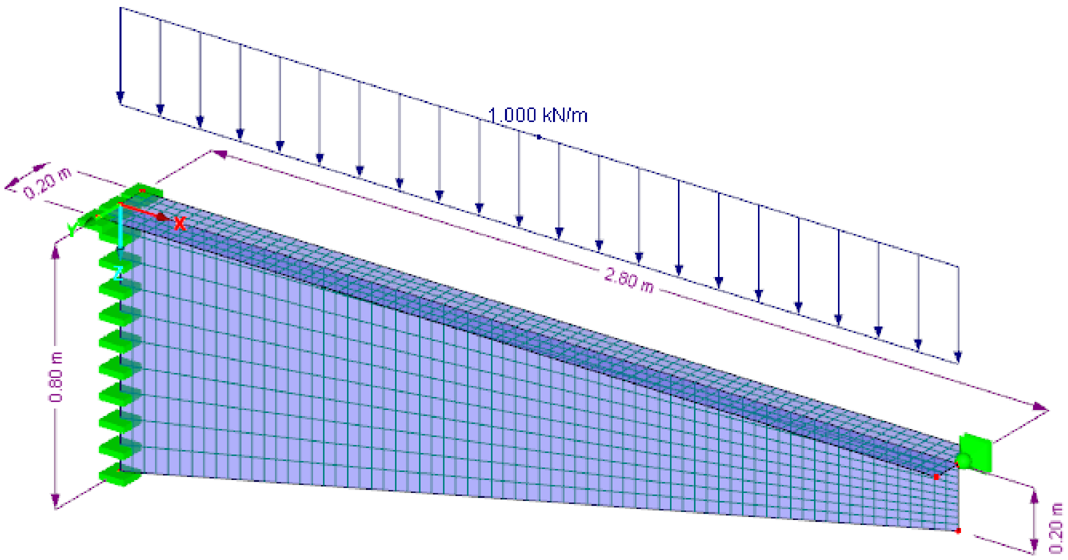


Figure 8.12: FE-model and loading

The beam has the following dimensions:

Length l	2,800	mm
Web depth h_{w0}	800	mm
Web depth h_{wl}	200	mm
Flange width b	200	mm
Web thickness t_w	10	mm
Thickness of flange t_f	20	mm

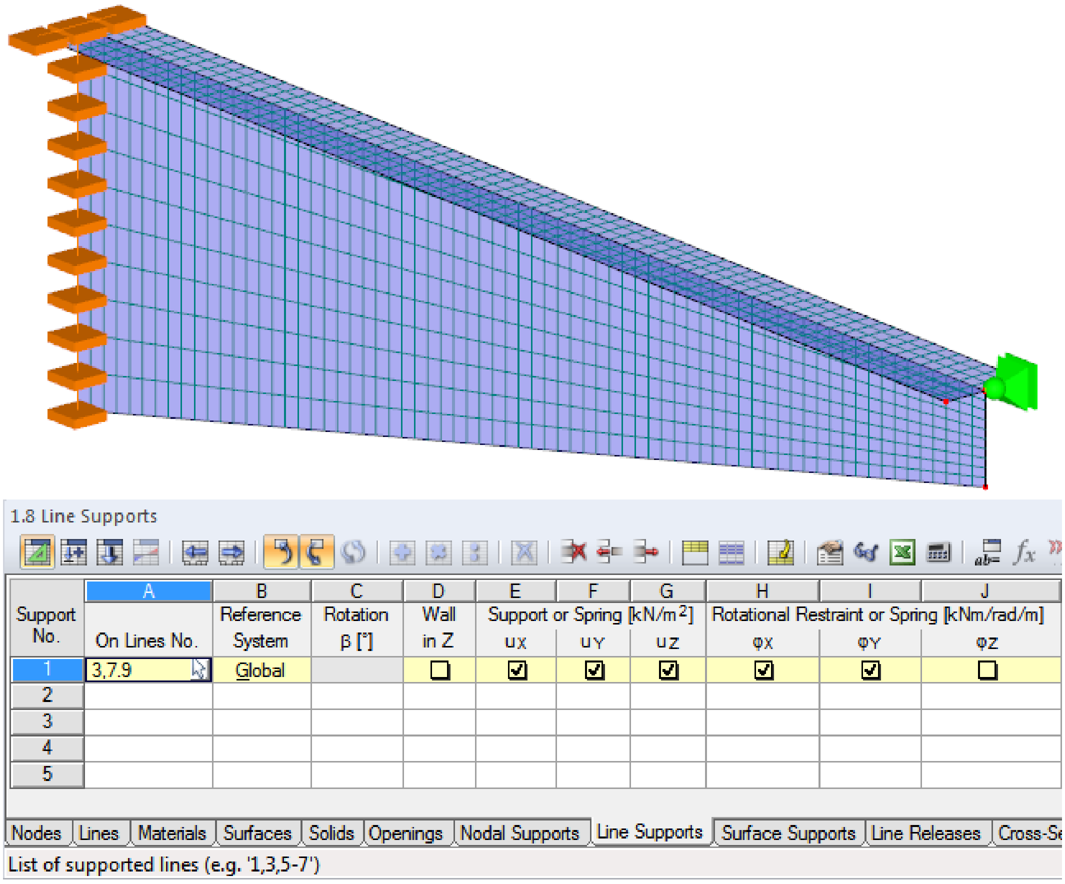
Table 8.1: Beam geometry

The tapered beam has a line load of 1.0 kN/m at the flange.

In [4], the ideal critical load is determined as $q_{cr} = 43.6\ \mathrm{kN/m}$.

Solution with RFEM

For the beam, specify the following support conditions at the location of restraint:



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 Rotational Restraint or Spring [kNm/rad/m] φX	I φY	J φZ
1	3,7,9	Global		☐	☑	☑	☑	☑	☑	☐
2										
3										
4										
5										

List of supported lines (e.g. '1,3,5-7')

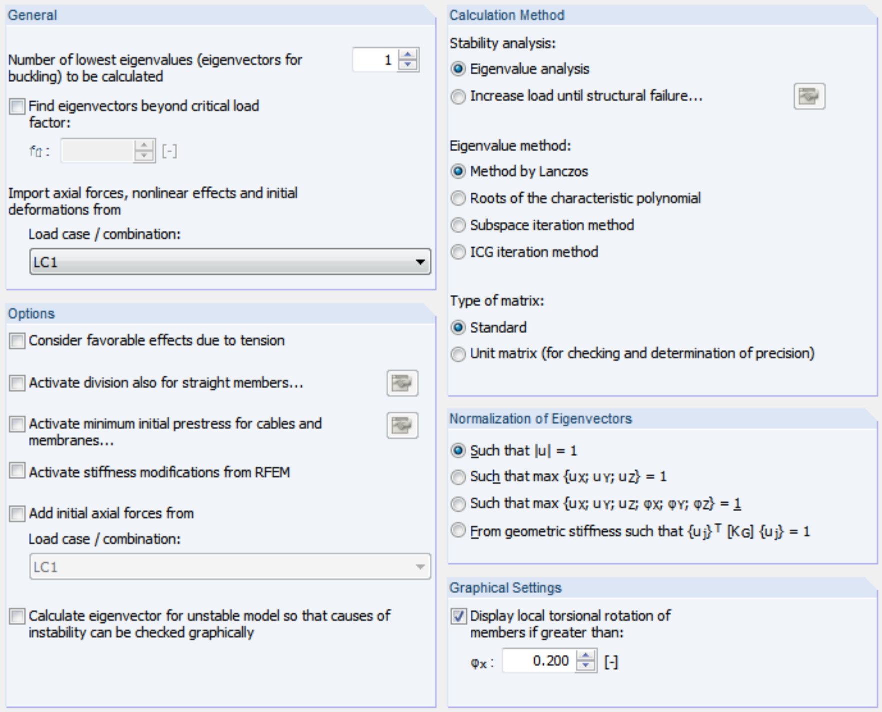
Figure 8.13: Restraint of beam

In addition, the end of the beam is laterally restrained.

The FE mesh has a target length of the finite elements of 4 cm.

As loading, a line load of 1 kN/m is applied corresponding to Figure 8.12.

Specify the following in RF-STABILITY:



General

Number of lowest eigenvalues (eigenvectors for buckling) to be calculated: 1

☐ Find eigenvectors beyond critical load factor:

f_Ω: [-]

Import axial forces, nonlinear effects and initial deformations from

Load case / combination: LC1

Options

☐ Consider favorable effects due to tension

☐ Activate division also for straight members...

☐ Activate minimum initial prestress for cables and membranes...

☐ Activate stiffness modifications from RFEM

☐ Add initial axial forces from

Load case / combination: LC1

☐ Calculate eigenvector for unstable model so that causes of instability can be checked graphically

Calculation Method

Stability analysis:

● Eigenvalue analysis

○ Increase load until structural failure...

Eigenvalue method:

● Method by Lanczos

○ Roots of the characteristic polynomial

○ Subspace iteration method

○ ICG iteration method

Type of matrix:

● Standard

○ Unit matrix (for checking and determination of precision)

Normalization of Eigenvectors

● Such that $|u| = 1$

○ Such that max $\{u_X; u_Y; u_Z\} = 1$

○ Such that max $\{u_X; u_Y; u_Z; \varphi_X; \varphi_Y; \varphi_Z\} = 1$

○ From geometric stiffness such that $\{u_j\}^T [K_G] \{u_j\} = 1$

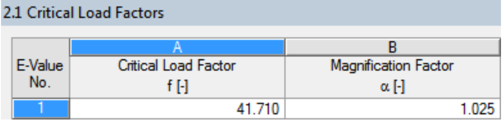
Graphical Settings

☑ Display local torsional rotation of members if greater than:

φ_x: 0.200 [-]

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

As result, RF-STABILITY gives a critical buckling factor of 41.710.



2.1 Critical Load Factors

	A	B
E-Value No.	Critical Load Factor f [-]	Magnification Factor α [-]
1	41.710	1.025

Figure 8.15: Critical load factor

From the critical buckling factor and the applied loading, you obtain:

$$q_{cr} = 41.710 \cdot 1.0 = 41.7 \text{ kN/m}$$

The difference to the result in [4] is about 4 %.

As governing eigenvector, RF-STABILITY yields the web plate-buckling:

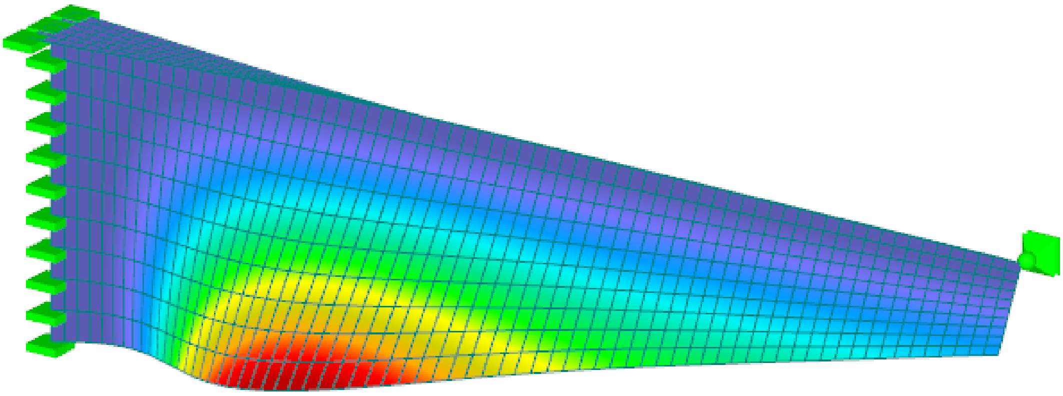


Figure 8.16: RF-STABILITY eigenvector

This result is corresponding to the buckling mode in [4].

Literature

[1] Christian Barth and Walter Rustler. *Finite Elemente in der Baustatik-Praxis*. Beuth, Berlin, 2. edition, 2013.

[2] *EN 1993-1-1: Bemessung und Konstruktion von Stahlbauten Teil 1-1: Allgemeine Bemessungsregeln und Regeln für den Hochbau*. Beuth Verlag GmbH, Berlin, 2005.

[3] Richard Bareš. *Tabulky pro výpočet desek a stěn*. SNTL - Nakladatelství technické literatury, Praha, 3. edition, 1989a.

[4] Manfred Fischer and M. Smida. Dimensionierung und Nachweis von gevouteten Kragträgern mit T-förmigem Querschnitt. *Stahlbau*, 70, 2001.

[5] Christian Petersen. *Statik und Stabilität der Baukonstruktionen*. Vieweg & Sohn, Wiesbaden, 2. edition, 1982.

[6] Christian Petersen. *Stahlbau*. Vieweg & Sohn, Wiesbaden, 3. edition, 1993.

[7] *Finite Elemente in der Baustatik*. Vieweg & Sohn, Wiesbaden, 3. edition, 2008.

Index