Version
April 2013

Add-on Module

RSBUCK

Buckling Lengths, Buckling Shapes, Critical Load Factors

Program Description

© Ing.-Software Dlubal GmbH
Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 9230-51
E-mail: info@dlubal.com
Web : www.dlubal.com

Contents

1. Introduction

1.1 Add-on Module RSBUCK

The RSTAB add-on module RSBUCK performs an eigenvalue analysis for frameworks to determine the critical load factors and buckling shapes. The critical load factor (critical buckling load factor of the entire system) allows you to evaluate the stability failure of the structural system. The corresponding buckling shape indicates the area that is prone to instability in the structural model.

With the add-on module RSBUCK, you can analyze several buckling shapes simultaneously. After the calculation, the program shows the relevant failure modes of the RSTAB model, sorted by the critical load factor. The corresponding buckling lengths and buckling loads are necessary for the further stability calculations which, as a rule, must be performed for structural parts subjected to compression.

In the graphical display of the buckling shapes, you can localize the areas in the model that are prone to instability. It is possible to derive structural measures in order to deal with the modes of failure. For these reasons, RSBUCK is a highly useful tool for analyzing systems susceptible to buckling, for example a slender beam or spatial trusses: The critical load factor allows you to evaluate whether or not the system is generally prone to instability (buckling and lateral-torsional buckling). Moreover, you can apply imperfections under consideration of buckling shapes, and then use them for the calculation in RSTAB.

RSBUCK provides the following special features:

- Determination of several buckling shapes in one calculation run
- Import of axial forces from RSTAB load cases or combinations
- Option to take into account favorable effects due to tension
- Optional consideration of stiffness modifications from RSTAB
- Powerful equation solver for eigenvalue analysis according to subspace iteration method with user-definable parameters
- Tabular output of critical load factors and corresponding buckling shapes
- Visualization of buckling shapes with animation option in RSTAB graphical user interface
- Print output in RSTAB printout report with automatic update in case of modifications
- Preparation of the buckling shapes for the add-on modules RSIMP, KAPPA, STEEL EC3, STEEL BS, STEEL AISC, STEEL CS, STEEL GB, STEEL IS, STEEL SIA, ALUMINIUM, and TIMBER Pro
- Data export to MS Excel, OpenOffice.org Calc, or as a CSV file

We hope you will enjoy working with RSBUCK.

Your team from Dlubal Engineering Software

1.2 RSBUCK Team

The following people were involved in the development of RSBUCK:

Program coordination

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

Programming

Dr.-Ing. Jaroslav Lain
Ing. Michal Balvon

Program design, dialog figures, and icons

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

Program supervision

Ing. Pavel Bartoš
Ing. František Knobloch

Localization, manual

Msc. Eliška Bartůňková
Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Ing. Jana Duníková
Dipl.-Ing. (FH) René Flori
Ing. Lara Freyer
Bc. Chelsea Jennings
Jan Jeřábek
Ing. Ladislav Kábrt
Ing. Aleksandra Kociołek
Mgr. Michaela Kryšková
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. Tingting Ling
Ing. Roberto Lombino
Eng.º Nilton Lopes
Mgr. Ing. Hana Macková
Ing. Téc. Ind. José Martínez
Ing. Petr Míchal
MA SKT Anton Mitleider
Dipl.-Ü. Gundel Pietzcker
Mgr. Petra Pokorná
Ing. Zoja Rendlová
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl
Ing. Marcin Wardyn

Technical support and quality management

M.Eng. Cosme Asseya
Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. Moritz Bertram
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Paul Kieloch
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
M.Sc. Dipl.-Ing. Frank Lobisch
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

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

The descriptions in this manual follow the sequence and structure of the module's input and results windows. The text of the manual shows the described **buttons** in square brackets, for example [Edit]. At the same time, they are pictured on the left. In addition to this, **expressions** used in dialog boxes, tables, and menus are set in *italics* to clarify the explanations.

Furthermore, you find an index at the end of the manual. However, if you still cannot find what you are looking for, please check our website **www.dlubal.com**, where you can go through our *FAQ* pages by selecting particular criteria.

1.4 Opening the Add-on Module RSBUCK

RSTAB provides the following options to start the add-on module RSBUCK.

Menu

To start the program from the RSTAB menu, click

Add-on Modules → Stability → RSBUCK.

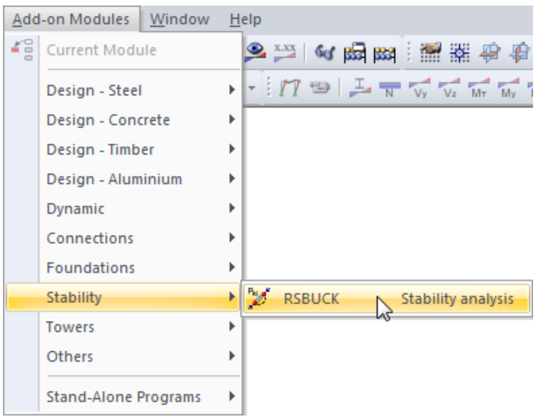


Figure 1.1: Menu: *Add-on Modules → Stability → RSBUCK*

Navigator

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

Add-on Modules → RSBUCK.

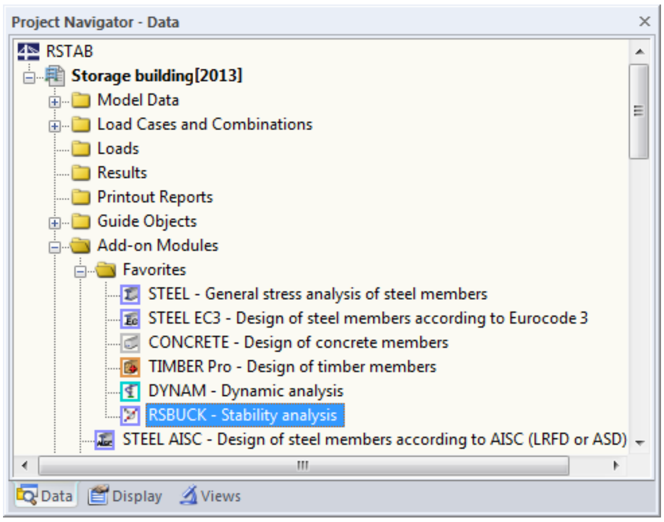
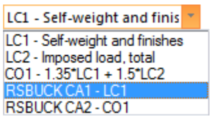
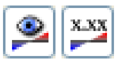


Figure 1.2: Data navigator: *Add-on Modules → RSBUCK*

Panel





If there are results available in RSBUCK, you can also start the add-on module by using the panel:

First, select the relevant RSBUCK case in the load case list, which is located in the menu bar. Then click [Show Results] to graphically display the buckling shape.

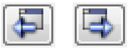
Now you can click the [RSBUCK] button to open the add-on module.

Figure 1.3: Panel button [RSBUCK]

2. Input Data

When you open the add-on module, a new window appears. The navigator on the left lists all available module windows. The pull-down list above the navigator contains the stability cases (see chapter 7.1, page 29).

When you open RSBUCK for the first time, the module automatically imports the created load cases and combinations.



To open a window, click the corresponding entry in the navigator. To go to the previous or next window, use the buttons shown on the left. You can also use the function keys [F2] (next) and [F3] (previous) to browse through the windows.

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

2.1 General Data

In window 1.1 *General Data*, you define the parameters for the stability analysis. In most cases, you can specify all settings in this one window.

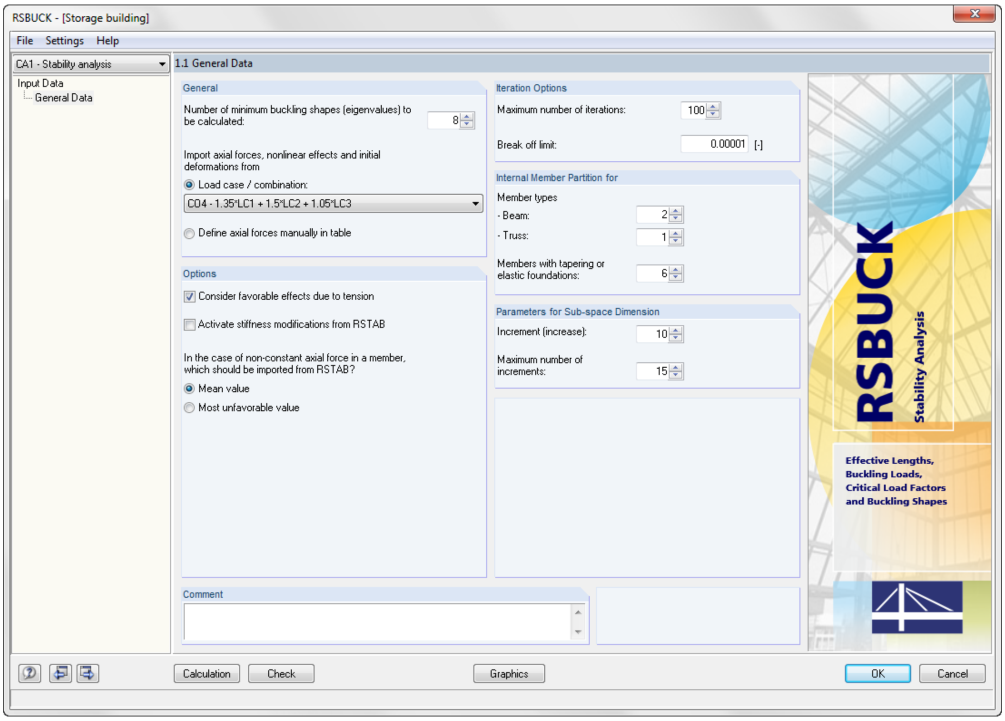


Figure 2.1: Window 1.1 *General Data*

General

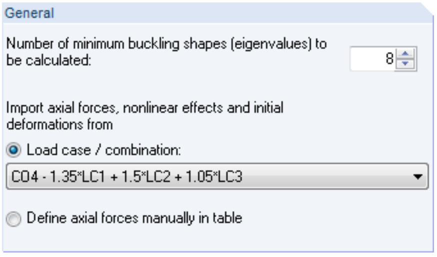


Figure 2.2: Number of buckling shapes and application of axial forces

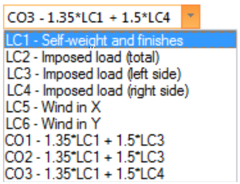
Number of buckling shapes

RSBUCK determines the most unfavorable buckling shapes of the model. With the number entered in the input field, you specify how many shapes will be determined. The upper limit is 10,000 of the lowest eigenvalues – provided that it is allowed by the number of possible degrees of freedom in the model as well as by the RAM capacity.

Generally, the theory of the calculation method does not allow for an exclusion of the low eigenvalues from the analysis in order to determine only the higher eigenvalues.

If negative critical load factors are displayed after the analysis, it is recommended to increase the values in the *Parameters for Subspace Dimension* accordingly (see description below). If the increments are too small, it is not possible to hide the negative eigenvalues to represent only the positive, realistic results.

Import axial forces, nonlinear effects and initial deformations



The drop-down list in this section contains all load cases, load combinations, result combinations, and super combinations that are available in the current model. Select an entry whose axial forces and, if necessary, stiffnesses you want to take into account for the determination of the buckling shape. This load case or this combination should be calculated in RSTAB according to the first-order analysis without stiffness reduction (material, cross-section, member).

If you select a load-case or a load combination, the program imports axial forces as well as the stiffness boundary conditions from RSTAB. Thus, you can represent nonlinear effects like, for example, failing members, supports and releases, or member nonlinearities in RSBUCK. In case of result and super combinations, however, you can only consider the axial forces. RSBUCK then uses the minimum value *min N* (greatest compressive force) of the two extreme values.

If results are not yet available for the selected load case or combination, they will be calculated automatically prior to the RSBUCK analysis.

Alternatively, you can *Define axial forces manually in table*. If you select this option, RSBUCK shows the additional input window 1.2 *Axial Forces* (see chapter 2.2, page 12). There, you can specify for each member the axial forces N that are relevant for the determination of the critical load factors.

The loads play an important role in the determination of buckling shapes and thus of buckling lengths because the buckling values depend not only on the structural model but also on the ratio of the axial forces and the total critical load N_{cr}. It is therefore recommended to define a load case with complete vertical loading (without wind) so that the majority of members are stressed by compression forces - after all, buckling lengths can be calculated only for members with compressive forces. Furthermore, the distribution of the loading in the entire system affects the determination of the critical factors.

Options

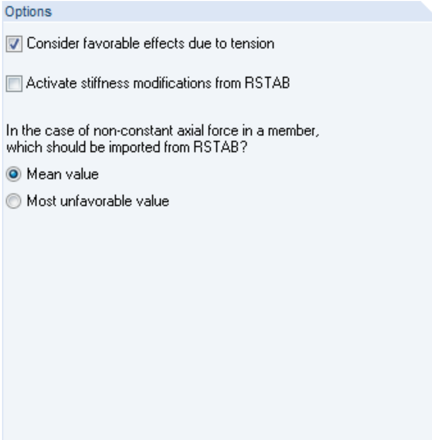


Figure 2.3: Options

Consider favorable effects due to tension

If the check box is selected, the axial tension forces acting in the model will also be taken into account in the determination of eigenvalues. Normally, those forces result in a stabilization of the model.

Activate stiffness modifications from RSTAB

With this check box, you decide whether the modification factors for the material, member, support, release, and cross-sectional stiffnesses are taken into account in the eigenvalue analysis. These factors are described in the according chapters of the RSTAB manual. If this check box is selected, the program applies all stiffness factors of the load case or load combination in the RSBUCK analysis, taking into account the failure criteria.

Thus, RSBUCK offers the control option to apply the stiffness modifications independent of the settings in RSTAB. It allows you to calculate, for example, axial forces of a load combination in RSTAB without reduction, and then to determine the critical loads in RSBUCK with the reduced material, member, and cross-section stiffnesses.

If, however, the stiffness modification for the load case or the load combination is activated in RSTAB, this check box is unavailable: This ensures that the eigenvalues correspond to the model assumptions.

If you determine the stability modes as a "characteristic" property of the model, you do not need to take into account the stiffness modifications.

In the case of non-constant axial force in a member, which should be imported from RSTAB?

The axial force distribution in a member is not always constant (for example, if a concentrated load is applied or if the automatic self-weight is considered in a column). The two option buttons in this section control whether RSBUCK uses the *Mean value* or the *Most unfavorable value* of the axial force for the analysis. The second option applies the maximum compression forces to the entire member. This may lead to lower eigenvalues.

Iteration Options

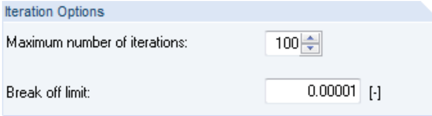


Figure 2.4: Iteration Options

The buckling shapes are determined by means of the eigenvalue of the entire system. The program uses an iterative equation solver for the determination. Generally, you have to specify two break-off limits for iterative calculation methods: An exact solution can be approached arbitrarily close without ever reaching it.

Maximum number of iterations indicates the iteration step after which the calculation will break off – no matter if the problem is converging and, therefore, a useful solution is found. There can never be a solution for divergence problems. The *Break off limit* defines the moment when an approximate solution can be considered as an exact result for a convergence problem.

Internal Member Partition

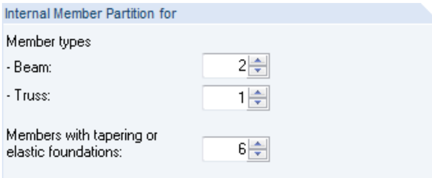


Figure 2.5: Internal member partition

In order to obtain a better approximate solution, you may have to refine the member partitions. When you enter a value higher than 1, the program will divide the member. You can specify partitions separately for the member types *Beam* and *Truss* as well as for *Members with tapering or elastic foundations*. A higher accuracy of representation is often necessary for tapered and foundation members.

If you specify *1* for the member partition of a spatially defined single-span beam, the program calculates, at most, the six lowest eigenvalues. If you enter the partition value *2* to define a simple member division, you can already determine the 12 lowest eigenvalues.

Parameters for Subspace Dimension

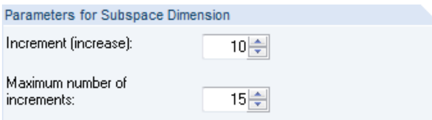


Figure 2.6: Parameters for subspace dimension

If RSBUCK, taking into account the specified number of iterations, does not find sufficient positive eigenvalues, the dimension of the subspace (that is, the number of eigenvalues) will be increased automatically per *Increment* and a new number of buckling shapes will be calculated. This procedure continues until either the number of required positive eigenvalues or the *Maximum number of increments* is reached.

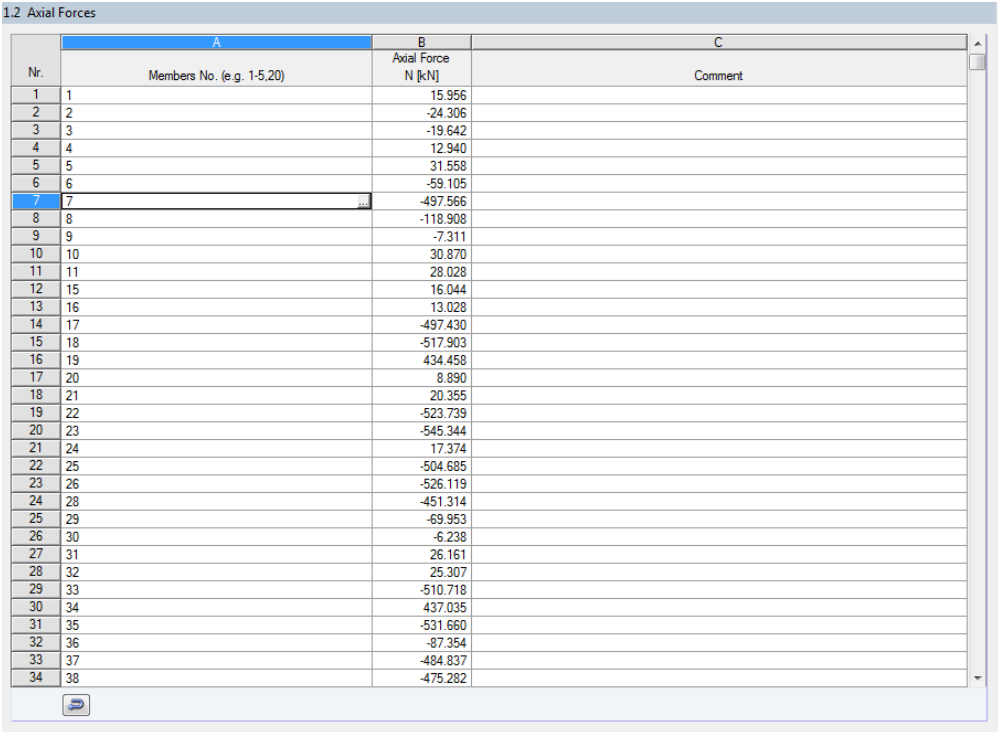
This method of calculation is converging against the minimum absolute values of the eigenvalues. Hence, it may happen that the program determines, due to tension forces, for example, only six negative eigenvalues whose absolute values represent the minima. Now, you can determine the lowest positive eigenvalue by increasing the subspace.

Comment

You can enter user-defined notes describing, for example, the current stability case.

2.2 Axial Forces

The program shows this window if you select a manual definition of axial forces in window 1.1 *General Data*. You can enter the member axial forces directly in this window.



1.2 Axial Forces

Nr.	A Members No. (e.g. 1-5,20)	B Axial Force N [kN]	C Comment
1	1	15.956	
2	2	-24.306	
3	3	-19.642	
4	4	12.940	
5	5	31.558	
6	6	-59.105	
7	7	-497.566	
8	8	-118.908	
9	9	-7.311	
10	10	30.870	
11	11	28.028	
12	15	16.044	
13	16	13.028	
14	17	-497.430	
15	18	-517.903	
16	19	434.458	
17	20	8.890	
18	21	20.355	
19	22	-523.739	
20	23	-545.344	
21	24	17.374	
22	25	-504.685	
23	26	-526.119	
24	28	-451.314	
25	29	-69.953	
26	30	-6.238	
27	31	26.161	
28	32	25.307	
29	33	-510.718	
30	34	437.035	
31	35	-531.660	
32	36	-87.354	
33	37	-484.837	
34	38	-475.282	

Figure 2.7: Window 1.2 *Axial Forces*

Members No.

In this column, you specify the numbers of the members to which you want to assign the axial forces entered in column B.

You can also select the members graphically. Click into the corresponding input field in column A to enable the selection function. Click the [...] button or press [F7] to jump to the RSTAB user interface. There, you can select the relevant members by clicking them one after the other.

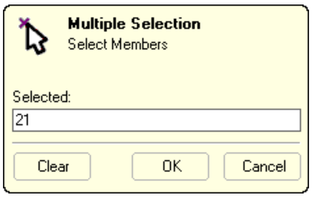


Figure 2.8: Graphical member selection in RSTAB

Axial Force

In this column, you enter the member axial forces that are to be taken into account in the eigenvalue analysis. Enter compression forces always with a negative sign.

The button [Import Axial Forces from RSTAB Load Case] below the table allows for the import of axial forces. In this way, it is possible to transfer automatically all members with axial forces of a load case, including corresponding internal forces.

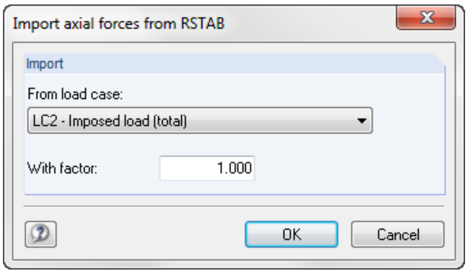


Figure 2.9: Dialog box *Import axial forces from RSTAB*

If necessary, you can apply a *factor* to the axial forces. After clicking [OK], you can also manually modify the forces in the table.

Comment

In this column, you can enter user-defined notes to describe, for example, the axial forces.

3. Calculation

3.1 Check

Before you start the calculation, it is recommended to quickly check if the input data is correct. To do this, click [Check].

If an inconsistency is detected, a warning appears showing the according information.

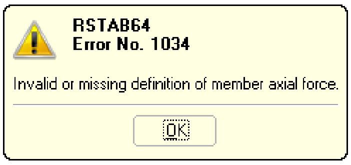


Figure 3.1: Result of check

3.2 Starting the Calculation

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

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

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

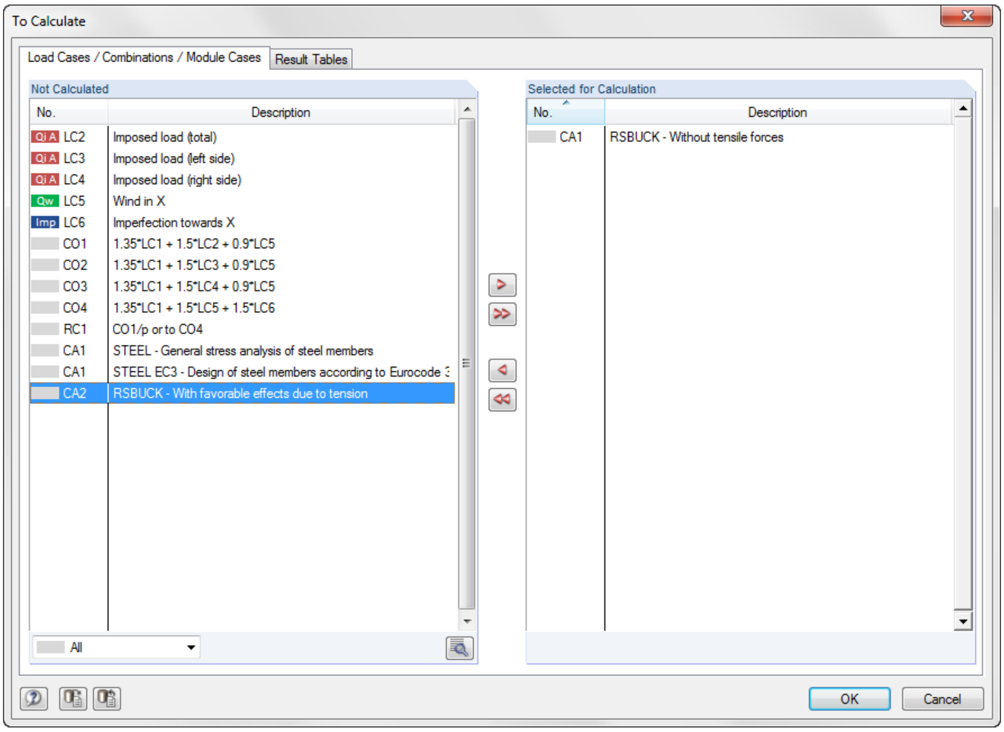
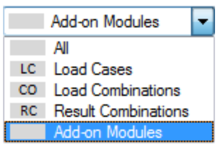


Figure 3.2: Dialog box *To Calculate*



If the RSBUCK 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 RSBUCK cases to the list on the right, use the [▶] button. Click [OK] to start the calculation.

To calculate a stability case directly, use the list in the RSTAB toolbar. Select the relevant RSBUCK case in the list, and then click [Show Results].

Figure 3.3: Direct calculation of an RSBUCK stability case in RSTAB

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

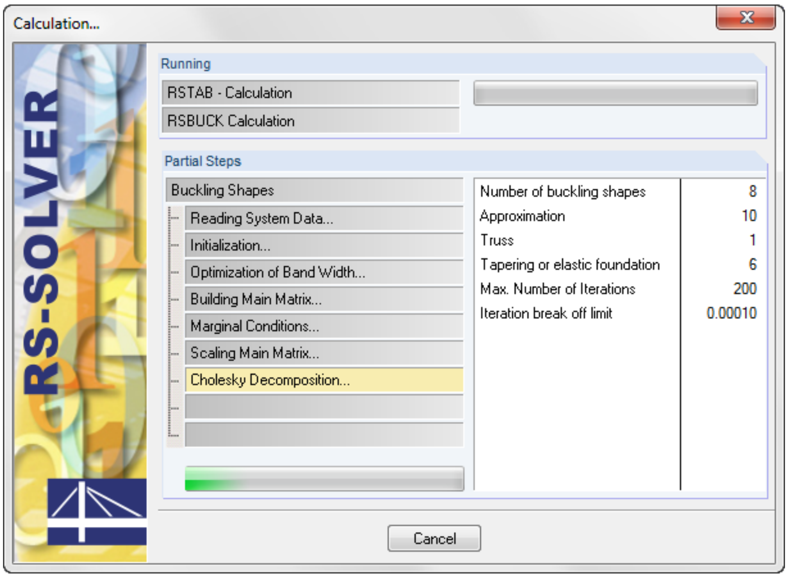
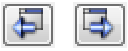


Figure 3.4: RSBUCK calculation

For the calculation according to the subspace method, the program runs the so-called *Cholesky Decomposition*, as shown in the figure above. It is used for solving the equation system during the iterative calculation in order to make new assumptions for eigenvalues and eigenmodes.

4. Results

Window 2.1 *Buckling Lengths and Loads* appears immediately after the calculation.



The tables of the windows 2.1 to 2.3 show the results. To select a window, click the respective entry in the navigator. To go to the previous or next window, use the buttons shown on the left. You can also use the function keys [F2] and [F3] to browse through the module windows.

To save the results, click [OK]. Thus you exit RSBUCK and return to the main program.

Chapter 4 *Results* describes the results windows in their proper order. Evaluating and checking results is described in chapter 5 *Results Evaluation*, page 21.

4.1 Buckling Lengths and Loads

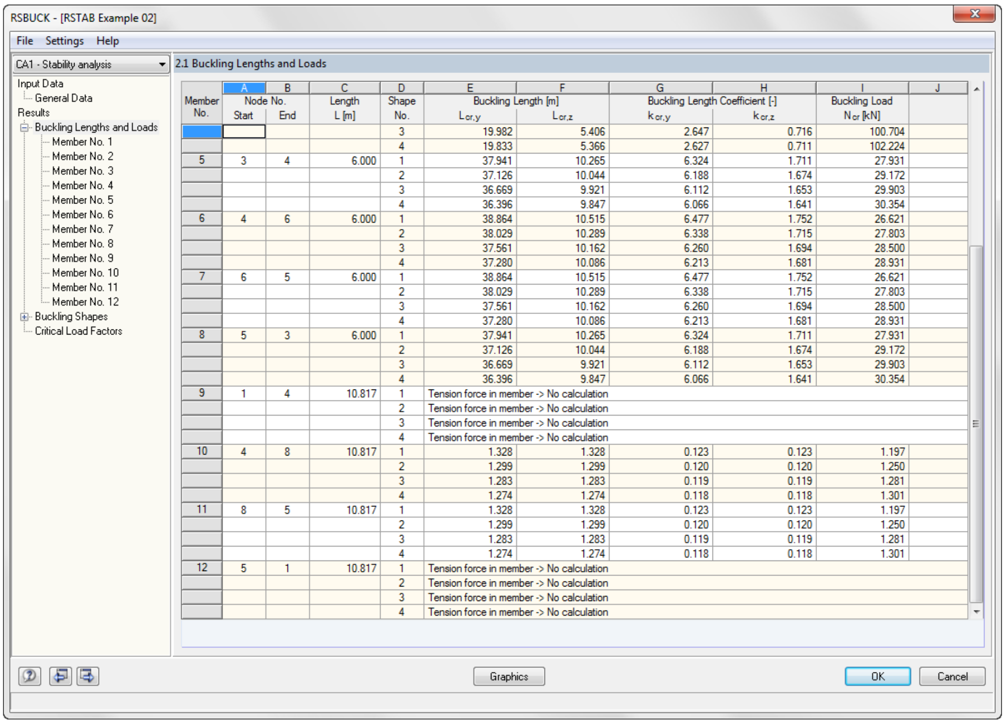
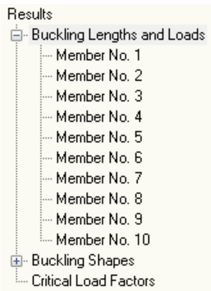


Figure 4.1: Window 2.1 *Buckling Lengths and Loads*



The results for buckling lengths and loads are listed by member. The results of a certain member can be shown quickly by using the navigator: Expand the list shown on the left, and then click the relevant *Member No.* RSBUCK then highlights the results of this member in the table.

Member No.

The results of the eigenvalue analysis are shown for all members of the model. Members with tension forces and failed members are marked accordingly.

Node No. Start / End

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

Length L

Column C indicates the geometric length of each member.

Shape No.

The results are listed by buckling shapes. The following chapter, 4.2, describes the eigenvalues with the according buckling shapes.

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

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

The buckling lengths L_{cr} are based on the member-specific buckling loads shown in column C. These loads are related to the respective critical load of the total model. Thus, the buckling lengths relate to the ratio of the member axial forces and the total critical load. For simple cases, we know the buckling lengths as the EULER buckling modes 1 to 4.

In some cases, the unfavorable buckling load of the system can be the critical load of an isolated member in the system, that is, a beam that is connected by a hinged support (see chapter 8.3, page 40). This can be seen in the graphic of the corresponding buckling shape, because a sinusoidal wave is available only on this member. This means that the structure shows a so-called local instability. The buckling lengths of all remaining members cannot be used for this failure case. Therefore, they must be taken from a "higher" buckling shape. The total model fails only there.

Buckling Length Coefficient $k_{cr,y}$ / $k_{cr,z}$

The buckling length coefficients relating to the local member axes y and z or u and v describe the ratio of buckling and member length.

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

Equation 4.1: Buckling length coefficient k_{cr}

Buckling Load N_{cr}

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

4.2 Buckling Shapes

2.2 Buckling Shapes

Member No.	A Node No.	B Shape No.	C u_X	D u_Y	E u_Z	F φ_X	G φ_Y	H φ_Z	I
			Scaled Buckling Shape [-]						
1	1	1	0.00000	0.00000	0.00000	-0.02763	-0.02763	0.15621	
		2	0.00000	0.00000	0.00000	-0.43894	-0.43894	-0.05137	
		3	0.00000	0.00000	0.00000	-0.00428	0.00428	0.00000	
		4	0.00000	0.00000	0.00000	0.51133	0.51133	0.20205	
2	3	1	0.01286	-0.01286	0.00000	-0.05980	-0.05980	0.13783	
		2	-0.00111	0.00111	0.00000	0.03490	0.03490	0.21939	
		3	0.01914	0.01914	0.01145	0.01592	-0.01592	0.00000	
		4	-0.01516	0.01516	0.00000	-0.02517	-0.02517	-0.10453	
3	2	1	0.00000	0.00000	0.00000	0.27587	-0.10216	0.03393	
		2	0.00000	0.00000	0.00000	-0.51260	0.52206	0.10107	
		3	0.00000	0.00000	0.00000	0.48395	-0.44341	-0.19627	
		4	0.00000	0.00000	0.00000	-0.15321	0.14674	0.07056	
4	4	1	0.01284	0.82326	0.23136	0.00230	0.00374	0.14235	
		2	-0.00120	0.04495	0.01317	0.04266	-0.04124	-0.21852	
		3	0.01904	0.16549	0.04195	-0.04571	0.03071	0.09053	
		4	-0.01497	-0.01564	-0.00023	0.00948	0.00222	-0.01722	
5	8	1	0.00000	0.00000	0.00000	0.34249	0.34249	-0.08695	
		2	0.00000	0.00000	0.00000	0.51985	0.51985	-0.09095	
		3	0.00000	0.00000	0.00000	0.03614	-0.03614	0.00000	
		4	0.00000	0.00000	0.00000	0.11983	0.11983	-0.04721	
6	6	1	-0.82336	0.82336	0.00000	-0.06045	-0.06045	0.14330	
		2	-0.04507	0.04508	0.00000	-0.04540	-0.04540	0.23205	
		3	0.16541	0.16541	-0.09451	0.00093	-0.00093	0.00000	
		4	0.01561	-0.01560	0.00000	-0.01040	-0.01040	0.02720	
7	7	1	0.00000	0.00000	0.00000	-0.10216	0.27587	0.03393	
		2	0.00000	0.00000	0.00000	0.52206	-0.51260	0.10107	
		3	0.00000	0.00000	0.00000	0.44341	-0.48395	0.19627	
		4	0.00000	0.00000	0.00000	0.14674	-0.15321	0.07056	
8	5	1	-0.82326	-0.01284	-0.23136	0.00374	0.00230	0.14235	
		2	-0.04495	0.00120	-0.01317	-0.04124	0.04266	-0.21852	
		3	0.16549	0.01904	0.04195	-0.03071	0.04571	-0.09053	
		4	0.01565	0.01497	0.00023	0.00222	0.00948	-0.01722	
9	1	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	

Figure 4.2: Window 2.2 *Buckling Shapes*

For each buckling shape, the table shows the displacements and rotations of the model's nodes.

Node No.

The buckling shapes are listed for the model objects defined in the RSTAB window 1.1 *Nodes*. Results in the member division points are not shown in the table.

Shape No.

The deformations are listed for each calculated eigenmode.

Scaled Buckling Shape u_X / u_Y / u_Z / φ_X / φ_Y / φ_Z

The displacements listed in the columns C to E refer to the axes of the global coordinate system and are scaled to the extreme value *1* of the total displacement.

The columns F to H list the node rotations related to the scaled displacements.

If the table shows only zero values for the scaled displacements, 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 buckling lengths and loads are not very conclusive.

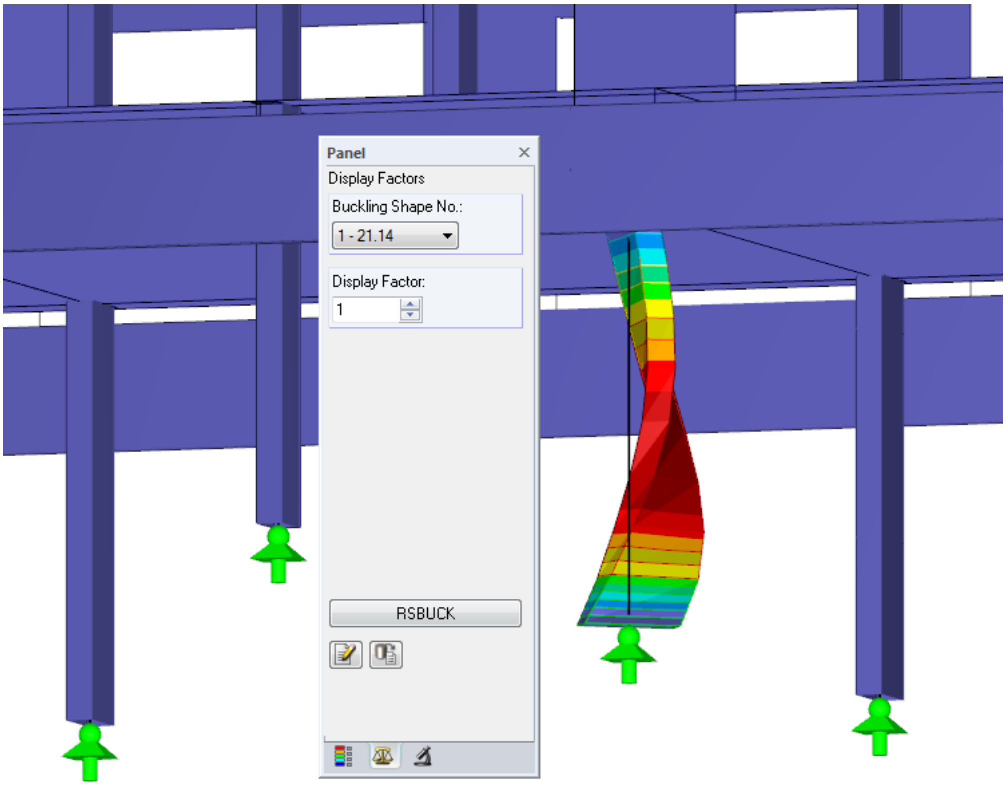


Figure 4.3: Torsion of a thin-walled rectangular column

4.3 Critical Load Factors

The final results window contains information concerning the critical load factors of the model.

2.3 Critical Load Factors

Shape No.	A Critical Load Factor f [-]	B Magnification Factor α [-]
1	0.832	0.000
2	1.038	27.599
3	1.042	24.943
4	1.226	5.428
5	1.288	4.467
6	1.487	3.052
7	1.633	2.580
8	1.795	2.257
9	1.953	2.049
10	2.000	2.000
11	2.159	1.863
12	2.234	1.810
13	2.274	1.785
14	2.321	1.757
15	2.375	1.727
16	2.456	1.687
17	2.592	1.628
18	2.784	1.561
19	2.871	1.534
20	3.011	1.497

Figure 4.4: Window 2.3 *Critical Load Factors*

Shape No.

For each eigenvalue, the table shows the critical load as well as the magnification factors. The result rows are sorted by buckling shape numbers.

Critical Load Factor f

The table shows the critical load factor for every 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 available axial forces multiplied with this factor results in the system's failure due to buckling.

According to DIN 18800 Part 2, a buckling load factor smaller than 10 requires 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:

$$\alpha = \frac{1}{1 - \frac{1}{f}}$$

Equation 4.2: Magnification factor

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

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

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

M^{II} Second-order moment

Equation 4.3: Relation of moments

Equation 4.3, however, applies only if the bending line due to the loading is similar to the buckling shape, and if the critical load factor f is greater than 1.00.

5. Results Evaluation

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

5.1 Results Windows

First, you should evaluate the buckling load factors displayed in window 2.3 *Critical Load Factors*.

A negative critical load factor indicates that no buckling failure could be detected because of the axial tension forces. That is: If the loading's direction of action (inverse signs) is reversed, a buckling failure would occur. To correct this, you can in some cases increase the number of buckling shapes to be determined.

Critical load factors less than 1.00 are an indicator of the system's instability!

2.3 Critical Load Factors

Shape No.	A Critical Load Factor f [-]	B Magnification Factor α [-]
1	0.832	0.000
2	1.038	27.599
3	1.042	24.943
4	1.226	5.428
5	1.288	4.467
6	1.487	3.052

Figure 5.1: Instable system

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

Window 2.1 shows the buckling length coefficients k_{cr} for each member. The coefficients differ depending on the buckling shape.

2.1 Buckling Lengths and Loads

Member No.	A Node No. Start	B End	C Length L [m]	D Shape No.	E Buckling Length [m] $L_{cr,y}$	F $L_{cr,z}$	G Buckling Length Coefficient [-] $k_{cr,y}$	H $k_{cr,z}$	I Buckling Load N_{cr} [kN]
1	1	3	7.550	1	27.899	7.548	3.695	1.000	51.657
				2	20.893	5.652	2.767	0.749	92.117
				3	20.761	5.617	2.750	0.744	93.285
				3	20.184	5.461	2.673	0.723	98.700

Figure 5.2: Buckling length coefficients k_{cr}

During the analysis, the program increases the axial forces iteratively until the critical load case occurs. From this critical load factor, RSBUCK determines the critical buckling load. In turn, the critical load allows conclusions regarding the buckling lengths and buckling length coefficients.

If you want to show, for example, the governing buckling length coefficient $k_{cr,y}$ for the deflection perpendicular to the "strong" member axis y, you normally have to calculate several buckling shapes. For square cross-sections, you obtain the same buckling lengths and buckling length coefficients in both axis directions.

The buckling length coefficients for continuous members cannot be determined directly with RSBUCK. It allows for the evaluation of the results of the individual members only. The member with the lowest buckling load N_{cr} can be considered as governing for the entire set of members. Then, the k_{cr} values can be determined from the buckling length of this member and the total length of the set of members.

5.2 Results on the RSTAB Model

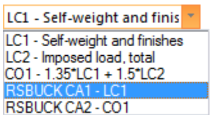
You can also use the RSTAB work window for the results evaluation.

RSTAB background graphic

The RSTAB work window is useful for finding the position of the member in the model: The member selected in the results window is marked in the background graphic with the selection color. To locate the member, move the RSBUCK window to the side.

RSTAB work window

The graphical evaluation of the different buckling shapes is a good way to evaluate the stability behavior of the model: Click [Graphic] to exit the RSBUCK module. Now, the buckling shapes are displayed graphically on the model in the work window of RSTAB like the deformations of a load case.



The current RSBUCK stability case is preset. The *Results* navigator controls which *Global Deformations* of the buckling shapes are displayed graphically.

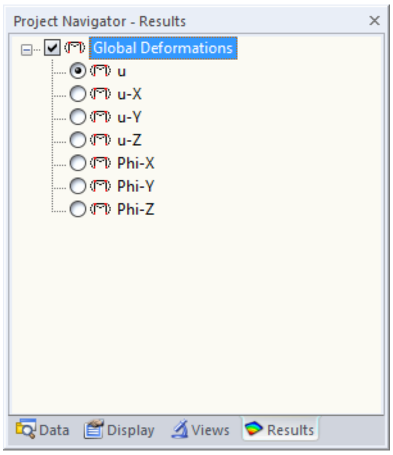


Figure 5.3: *Results* navigator for RSBUCK

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

Similar to the display of deformations, the [Show Results] button switches on or off the display of the buckling shapes. The [Show Result Values] to the right is of minor importance for RSBUCK.

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

The panel is adjusted to the RSBUCK module. The panel's default functions are described in detail in the RSTAB manual, chapter 3.4.6. The color scale tab in the panel appears only if the deformations are displayed with the option *Cross-Sections Colored* (see Figure 5.5).

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

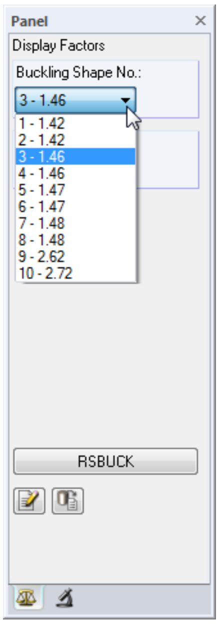
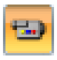


Figure 5.4: Selection of buckling shape in the *Factors*



If the members that are prone to buckling are difficult to find, increase the *Display Factors* of the deformation in the *Factors* tab of the panel. The animation of deformations is another helpful function. To activate it, click the button shown on the left.

You can control the results display in the *Display* navigator by selecting *Results*→ *Deformation* → *Members*. By default, buckling shapes are shown as single-colored *Lines*. The two remaining options can be used to illustrate the buckling behavior.

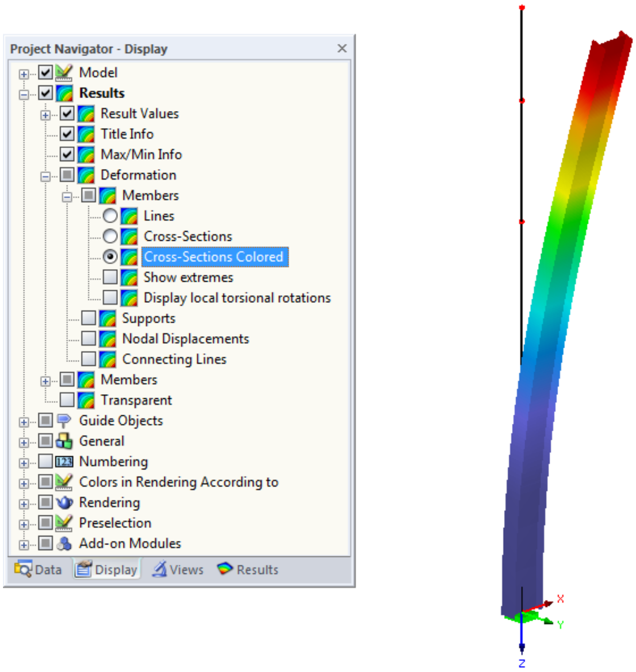


Figure 5.5: *Display* navigator: Results → Deformation → Members → Cross-Sections Colored

You can transfer the graphics of the buckling shapes to the printout report (see chapter 6.2, page 27).

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

5.3 Filter for Results

The RSBUCK results windows allow for a selection by various criteria. In addition to this, you can use the filter options described in chapter 9.7 of the RSTAB manual. The options allow you to graphically evaluate the results of the stability analysis.

In RSBUCK, you can also use the options of the *Visibilities* (see RSTAB manual, chapter 9.7.1) to filter the members for the evaluation.

Filtering designs

The scaled deformations can be used as filter criterion in the RSTAB work window. To go to the RSTAB window, click [Graphics]. The panel must also be available. If it is not available, you can switch it on by using the RSTAB menu

View → Control Panel (Color Scale, Factors, Filter)

or use the toolbar button shown on the left.

The panel is described in chapter 3.4.6 of the RSTAB manual. Use the first panel tab (color scale) to specify the filter settings. 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 5.5).

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

Filtering members

In the *Filter* tab of the control panel, you can specify the numbers of selected members to filter the display of the buckling shapes. The function is described in chapter 9.7.3 of the RSTAB manual. In contrast to the visibility function, the model is displayed, too.

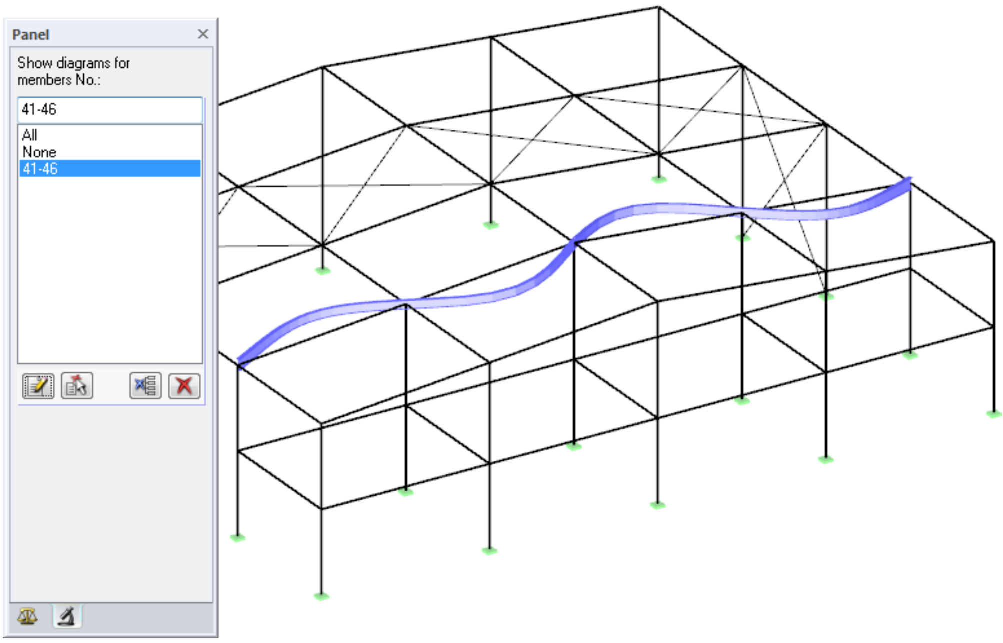


Figure 5.6: Member filter for buckling shape of a hall frame beam

5.4 Nonlinear RSTAB Calculation

The main program RSTAB also offers the possibility to calculate the critical load factor of a load case or load combination. To do this, open the *Edit Load Cases and Combinations* dialog box. In the *Calculation Parameters* subtab of the load cases or load combinations, select the *Calculate critical load factor* check box. The check box is only available for the second-order analysis.

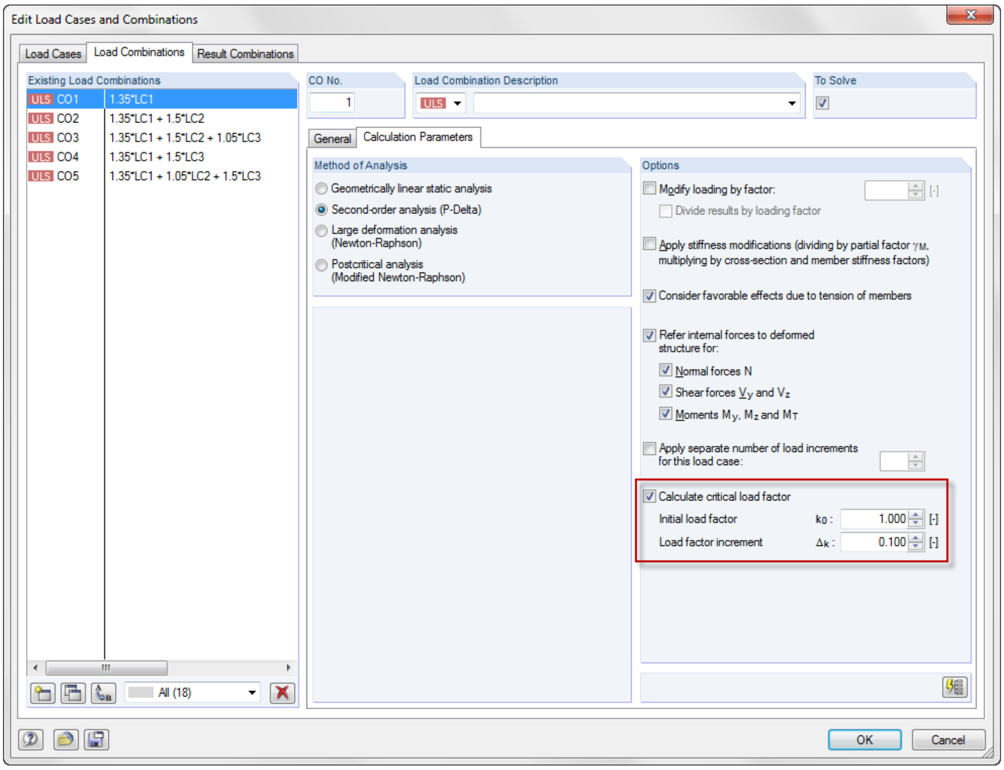
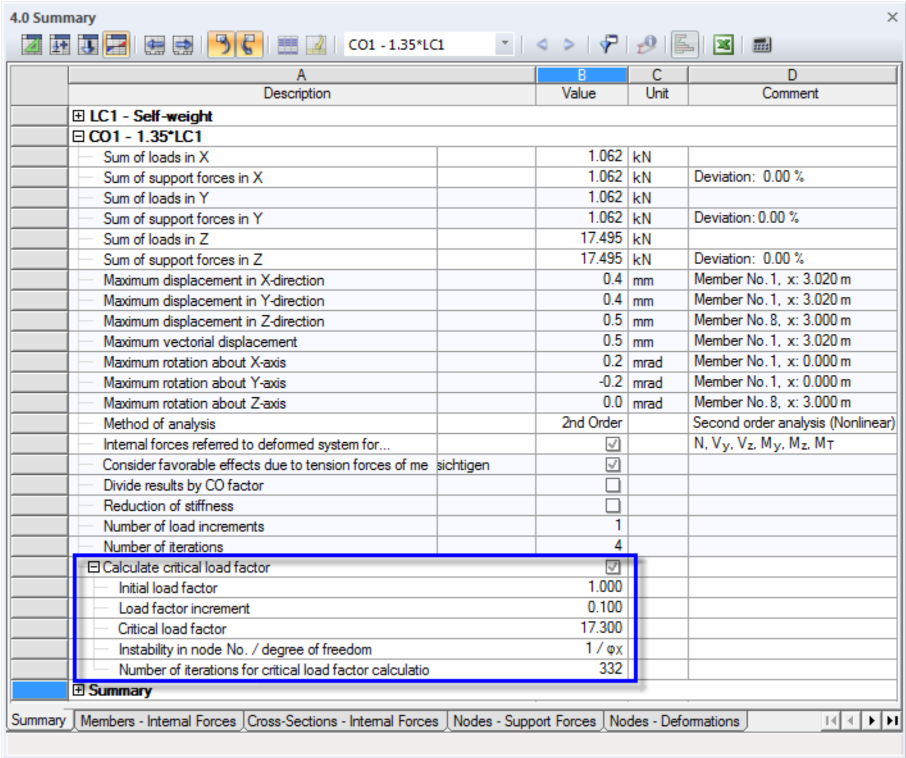


Figure 5.7: Dialog box *Edit Load Cases and Combinations*, tab *Calculation Parameters*

The *Initial load factor* k_0 should not be too high to ensure that the first eigenmode is also determined.

RSTAB determines the critical load factor according to a nonlinear calculation method. Instead of a linear eigenvalue analysis, the loading is gradually increased by the *Load factor increment* Δ_k. If a particular load increment is reached, the structural system becomes unstable. Thus, the critical load factor is found and shown in the RSTAB table 4.0 *Summary* together with the origin of instability and the number of iterations.



A Description	B Value	C Unit	D Comment
⊞ LC1 - Self-weight			
⊟ CO1 - 1.35*LC1			
Sum of loads in X	1.062	kN	
Sum of support forces in X	1.062	kN	Deviation: 0.00 %
Sum of loads in Y	1.062	kN	
Sum of support forces in Y	1.062	kN	Deviation: 0.00 %
Sum of loads in Z	17.495	kN	
Sum of support forces in Z	17.495	kN	Deviation: 0.00 %
Maximum displacement in X-direction	0.4	mm	Member No. 1, x: 3.020 m
Maximum displacement in Y-direction	0.4	mm	Member No. 1, x: 3.020 m
Maximum displacement in Z-direction	0.5	mm	Member No. 8, x: 3.000 m
Maximum vectorial displacement	0.5	mm	Member No. 1, x: 3.020 m
Maximum rotation about X-axis	0.2	mrad	Member No. 1, x: 0.000 m
Maximum rotation about Y-axis	-0.2	mrad	Member No. 1, x: 0.000 m
Maximum rotation about Z-axis	0.0	mrad	Member No. 8, x: 3.000 m
Method of analysis	2nd Order		Second order analysis (Nonlinear)
Internal forces referred to deformed system for...	☑		N, V_y, V_z, M_y, M_z, M_T
Consider favorable effects due to tension forces of me sichtigen	☑		
Divide results by CO factor	☐		
Reduction of stiffness	☐		
Number of load increments	1		
Number of iterations	4		
⊟ Calculate critical load factor	☑		
Initial load factor	1.000		
Load factor increment	0.100		
Critical load factor	17.300		
Instability in node No. / degree of freedom	1 / φx		
Number of iterations for critical load factor calculatio	332		
⊞ Summary			

Figure 5.8: RSTAB window 4.0 *Summery*

This procedure allows you to consider all nonlinear elements like failing members or supports. However, only the lowest eigenvalue is calculated. Furthermore, the computing time is often longer than the time needed for a linear analysis with RSBUCK.

Because of the different calculation methods, minor difference between the critical load factors from RSTAB and RSBUCK cannot be completely excluded. If the critical load factor according to the second-order analysis significantly differs from the RSBUCK calculation, you should check whether or not the calculation parameters

- favorable effect due to tension forces and
- stiffness modifications from RSTAB

have been taken into account in the same way in both cases.

6. Printout

6.1 Printout Report

The creation of printout reports is similar to the procedure in RSTAB. First, the program generates a printout report for the RSBUCK results to which you can add graphics and explanations. In the printout report, you select the data of the stability analysis that will finally appear in the printout.

The printout report is described in the RSTAB manual. Chapter 10.1.3.4 *Selecting Data of Add-on Modules* describes how you can prepare the input and output data of add-on modules for the printout.

6.2 Graphic Printout

In RSTAB, you can transfer every picture displayed in the work window to the printout report or send it directly to the printer. In this way, you can also prepare the buckling shapes that are shown on the RSTAB model for the printout.

The printing of graphics is described in chapter 10.2 of the RSTAB manual.

To print the current buckling shape, click

File → Print

or use the toolbar button shown on the left.

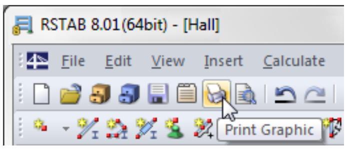


Figure 6.1: Button *Print Graphic* in the RSTAB toolbar

The following dialog box appears:

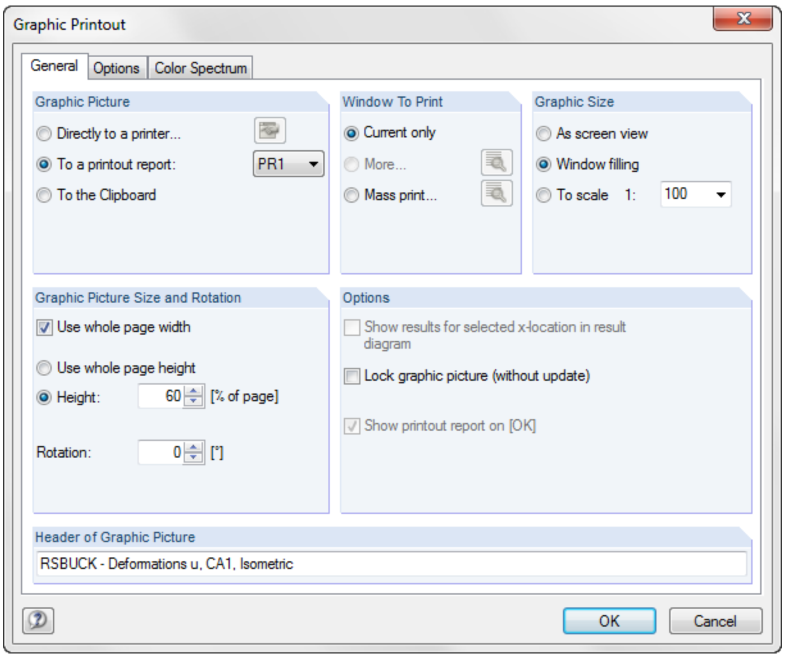
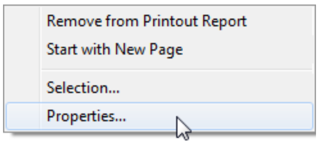


Figure 6.2: Dialog box *Graphic Printout*, tab *General*



This dialog box is described in chapter 10.2 of the RSTAB manual. The RSTAB manual also describes the two other tabs, *Options* and *Color Spectrum*.

You can move an RSBUCK graphic anywhere within the report by using the drag-and-drop function.

To adjust the graphic in the printout report subsequently, right-click the corresponding entry in the navigator of the report. By selecting the *Properties* option on the context menu, you reopen the *Graphic Printout* dialog box where you can make the necessary modifications.

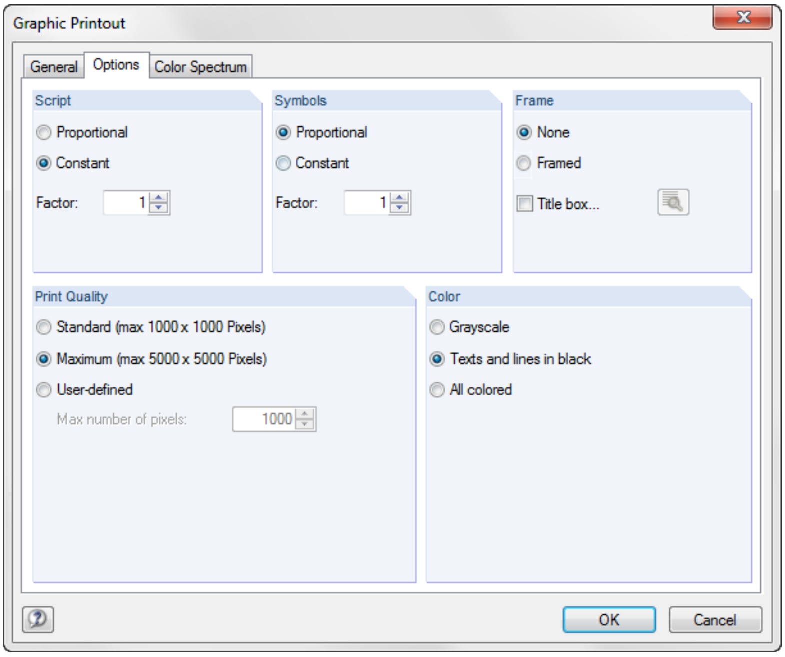


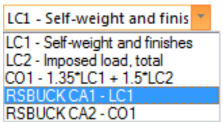
Figure 6.3: Dialog box *Graphic Printout*, tab *Options*

7. General Functions

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

7.1 RSBUCK Analysis Cases

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



To select the analysis cases of RSBUCK, you can also use the load case list in the RSTAB toolbar.

Creating a new analysis case

To create a new analysis case, use the RSBUCK menu, and select

File → New Case.

The following dialog box appears:

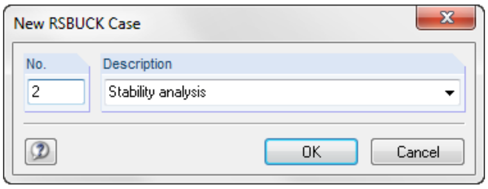


Figure 7.1: Dialog box *New RSBUCK Case*

Enter a *Number* (one which is not yet assigned) for the new RSBUCK analysis case. The *Description* helps you to select the right case in the load case list.

Click [OK] to display the RSBUCK 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

File → Rename Case.

The following dialog box appears:

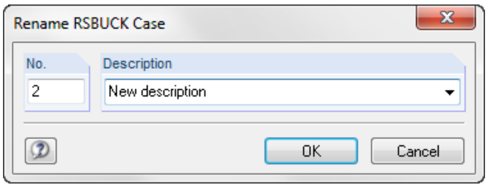


Figure 7.2: Dialog box *Rename RSBUCK Case*

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

File → Copy Case.

The following dialog box appears:

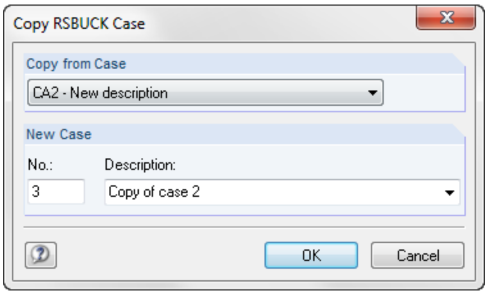


Figure 7.3: Dialog box *Copy RSBUCK Case*

Enter the *Number* and, if necessary, a *Description* of the new case.

Deleting an analysis case

To delete an analysis case, select

File → Delete Case.

The following dialog box appears:

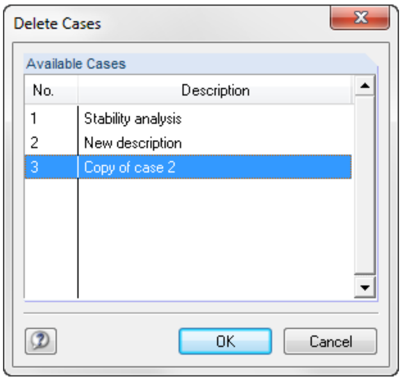


Figure 7.4: Dialog box *Delete Cases*

You can select the analysis case in the *Available Cases* list. By clicking [OK], you delete the selected case.

7.2 Units and Decimal Places

The units and decimal places for RSTAB and the add-on modules are managed together in one dialog box. In the add-on module RSBUCK, 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 RSTAB appears. RSBUCK is preset in the *Program / Module* list.

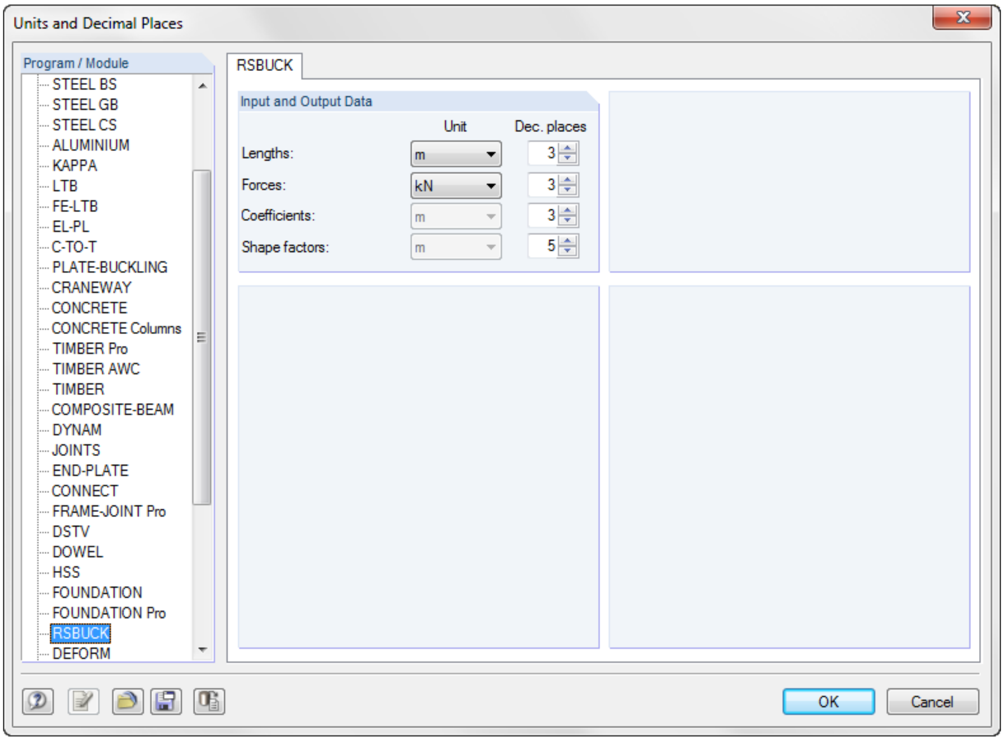


Figure 7.5: Dialog box *Units and Decimal Places*

You can save the settings as user profile to reuse them in other models. These functions are described in chapter 11.1.3 of the RSTAB manual.

7.3 Export of Results

The results of RSBUCK can also be used in other programs.

Clipboard

To copy cells selected in the results tables 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 will not be transferred.

Printout Report

The data of the RSBUCK add-on module can be printed into the global printout report (see chapter 6.1, page 27) to export them subsequently. Then, in the printout report, click

File → Export to RTF.

The function is described in detail in chapter 10.1.11 of the RSTAB manual.

Excel / OpenOffice

RSBUCK provides a function for the direct data export to MS Excel, OpenOffice.org Calc, or the CSV file format. To use this function, select

File → Export Tables.

The following export dialog box appears.

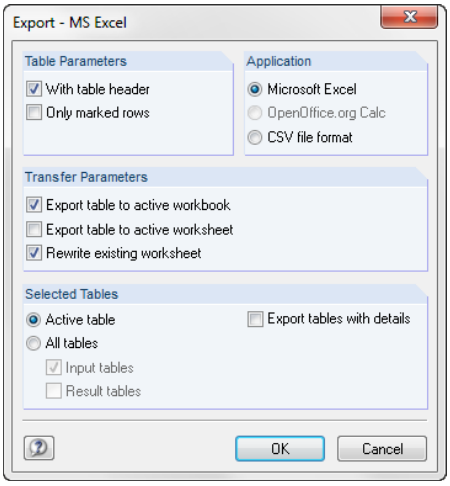


Figure 7.6: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking [OK]. Excel or OpenOffice are opened automatically, that is, you do not need to open them first.

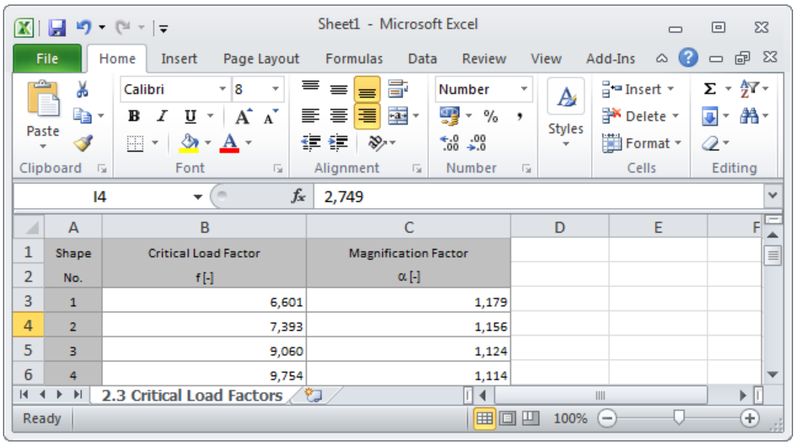


Figure 7.7: Results in *Excel*

RSIMP

If you want to use a stability mode in the RSIMP add-on module to generate equivalent imperfections or a pre-deformed initial model, you do not need to export the data. RSIMP allows you to select the relevant buckling shape *No.* as well as the RSBUCK *Case* directly in the corresponding lists.

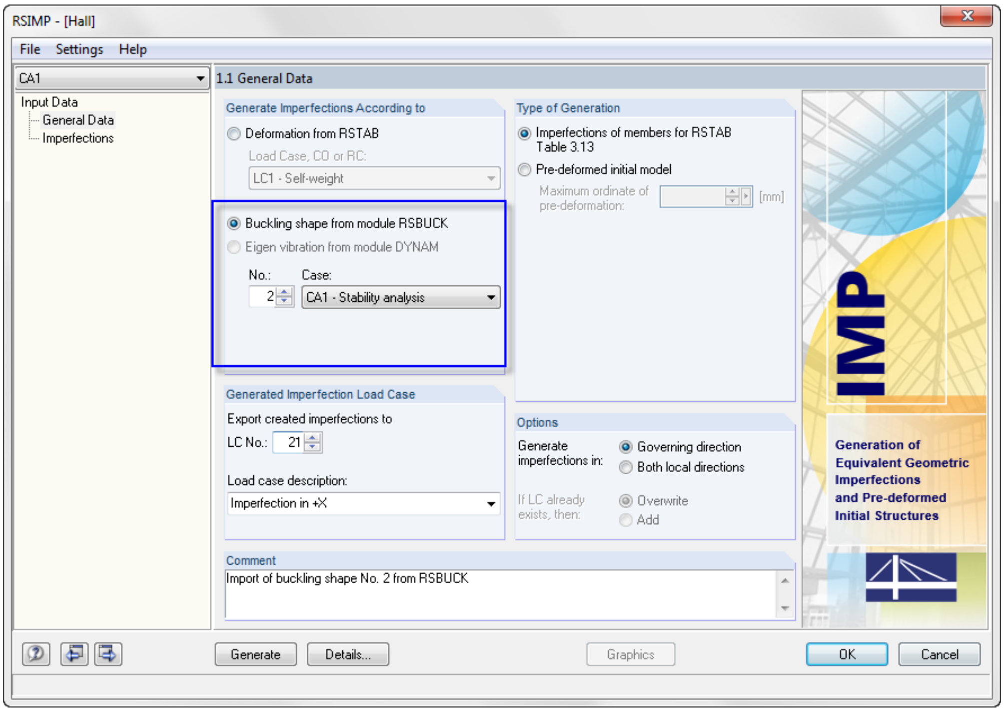


Figure 7.8: Importing buckling shape in RSIMP

STEEL EC3 / ALUMINIUM / KAPPA / TIMBER Pro

The modules STEEL EC3 / SIA / BS / AISC / CS / IS / GB, ALUMINIUM, KAPPA, and TIMBER Pro provide the option to use the buckling length coefficients from RSBUCK directly for the members that you want to analyze.

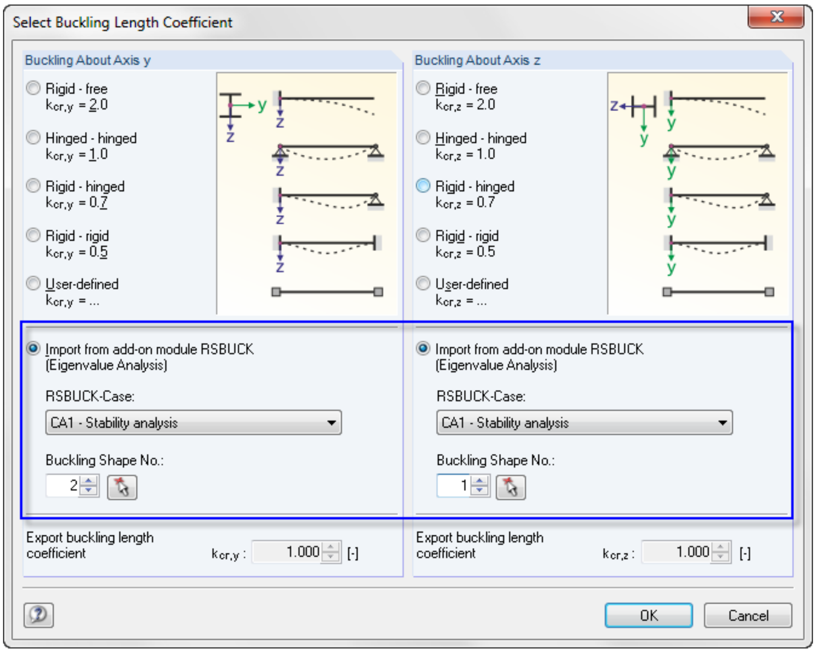


Figure 7.9: Selection of buckling length coefficients in the add-on module STEEL EC3

8. Examples

8.1 EULER Buckling Mode 1

Determine the critical load of a fixed column. Because of its type of loading and support conditions, the model corresponds to the EULER buckling mode 1.

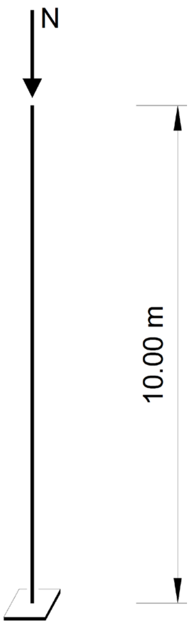


Figure 8.1: Model for EULER buckling mode 1

Analytical solution

The lowest critical load N_{cr} is determined according to the following equation:

$$N_{cr} = \frac{E \cdot I \cdot \pi^2}{L_{cr}^2}$$

Equation 8.1

The cross-section, defined as rolled cross-section HE-B 300, has the following second moments of area:

$I_y = 25{,}170 \text{ cm}^4$

$I_z = 8{,}560 \text{ cm}^4$

Steel S235 is used as material.

$E = 21{,}000 \text{ kN/cm}^2$

For a column that is fixed at one end and free at the other (EULER buckling mode 1), the buckling length coefficient $k_{cr} = 2$.

The buckling load for the deflection perpendicular to the z-axis is determined as follows:

$$N_{cr} = \frac{21{,}000 \cdot 8{,}560 \cdot \pi^2}{(2 \cdot 1{,}000)^2} = 443.54 \text{ kN}$$

Solution with RSTAB

We create the column as 3D model.

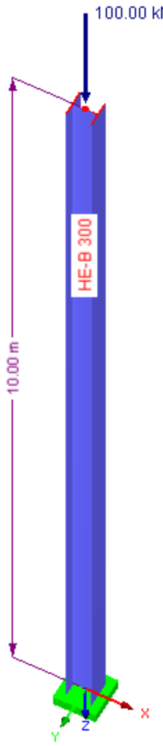


Figure 8.2: RSTAB model and loading

We clear the selection of the automatic self-weight in the general data of the load case. Next, we apply the concentrated loading of 100 kN to the top node of the column.

In the input window of RSBUCK, we enter the following data:

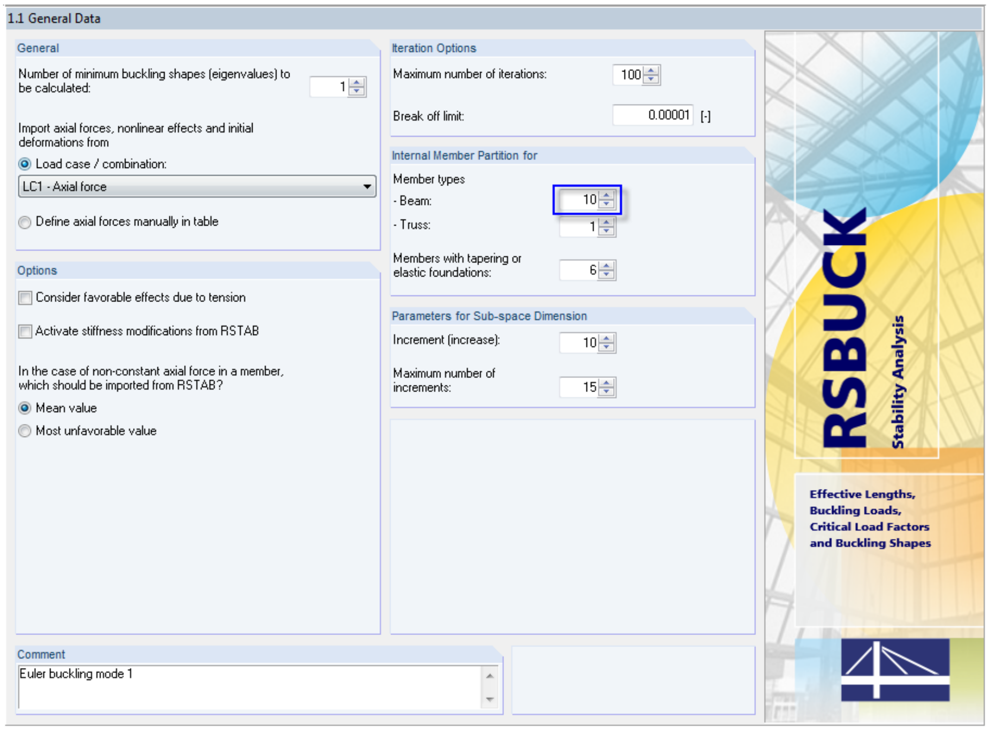


Figure 8.3: RSBUCK window 1.1 *General Data*

Calculation

After the [Calculation], RSBUCK shows a buckling load of 443.540 kN.

2.1 Buckling Lengths and Loads

	A	B	C	D	E	F	G	H	I
Member No.	Node No. Start	End	Length L [m]	Shape No.	Buckling Length [m] $L_{cr,y}$	$L_{cr,z}$	Buckling Length Coefficient [-] $k_{cr,y}$	$k_{cr,z}$	Buckling Load N_{cr} [kN]
1	1	2	10.000	1	34.295	20.000	3.430	2.000	443.540

Figure 8.4: RSBUCK window 2.1 *Buckling Lengths and Loads*

Since we have increased the member partition accordingly, the value completely corresponds to the analytical solution.

RSBUCK determines the following buckling shape:

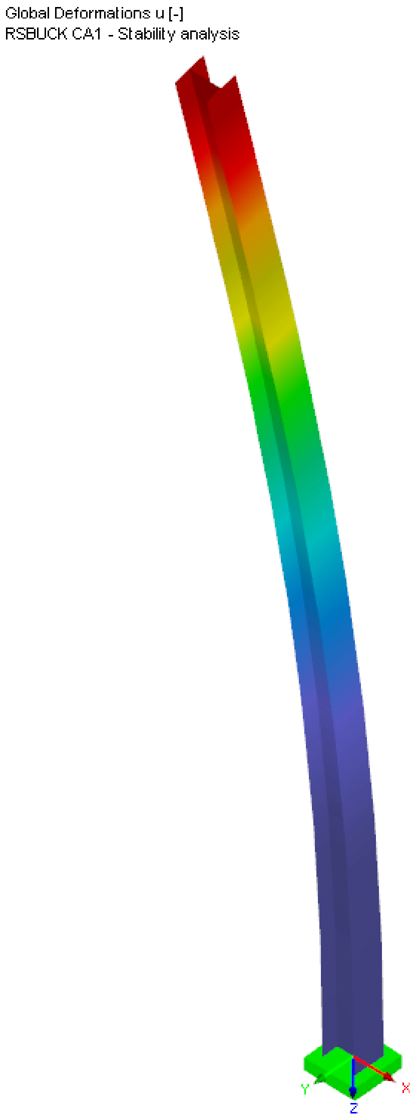


Figure 8.5: Buckling shape

8.2 Frame with K-Bracing

Based on the 2D model shown in the figure below, we want to determine the buckling length coefficients of the model's frame column. The example is taken from [1], page 395.

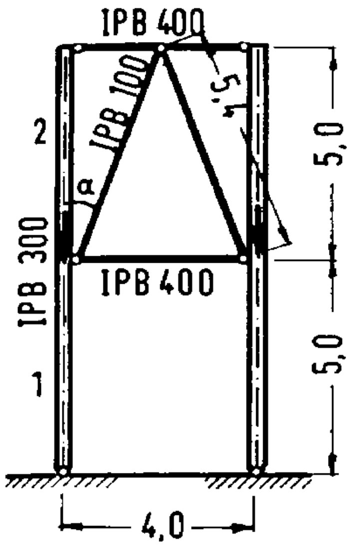


Figure 8.6: Steel frame model according to [1]

Analytical solution

The analytical solution is presented in [1], example 5.47. We assume a ratio of compression forces in the frame columns D2/D1 = 0.8, that is, the axial force in member 2 is 80 % of the axial force available in member 1.

For our plane system, the following buckling length coefficients are determined in [1]:

Member	Buckling length coefficient k_{cr}
1	2.73
2	3.07

Table 8.1: Buckling length coefficients according to [1], page 397

Solution with RSTAB

We create a 2D model and define the members with the according HE-B cross-sections. We define the frame columns as member type *Beam*, and the frame beams and diagonal members as member type *Truss (only N)*.

We assign hinged supports to both column footings.

To obtain the desired axial force distribution in the frame columns, we apply a concentrated load of 100 kN to the two topmost nodes, respectively. We apply further concentrated loads of 25 kN to the frame column centers at the connection nodes of the diagonals. The automatic self-weight is not taken into account.

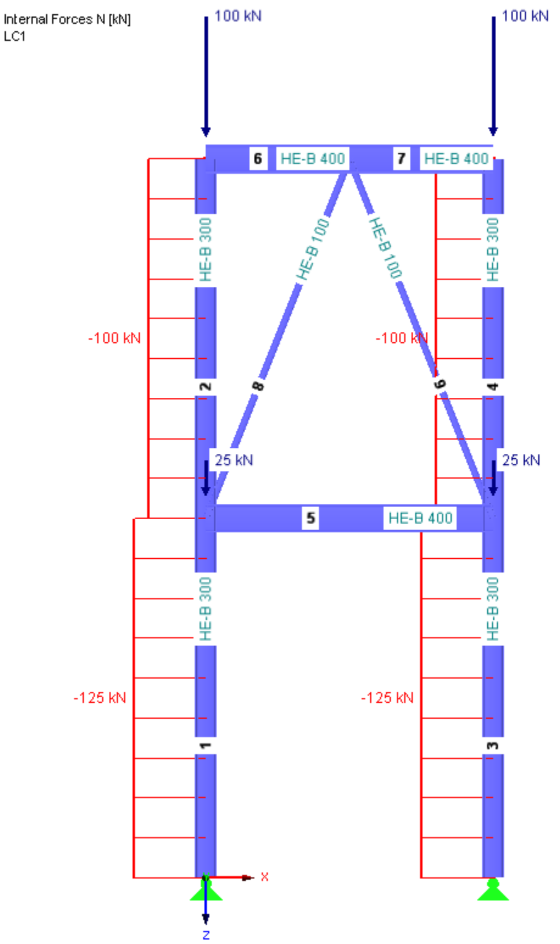


Figure 8.7: Distribution of axial force on the RSTAB model

We set the RSBUCK input windows as follows:

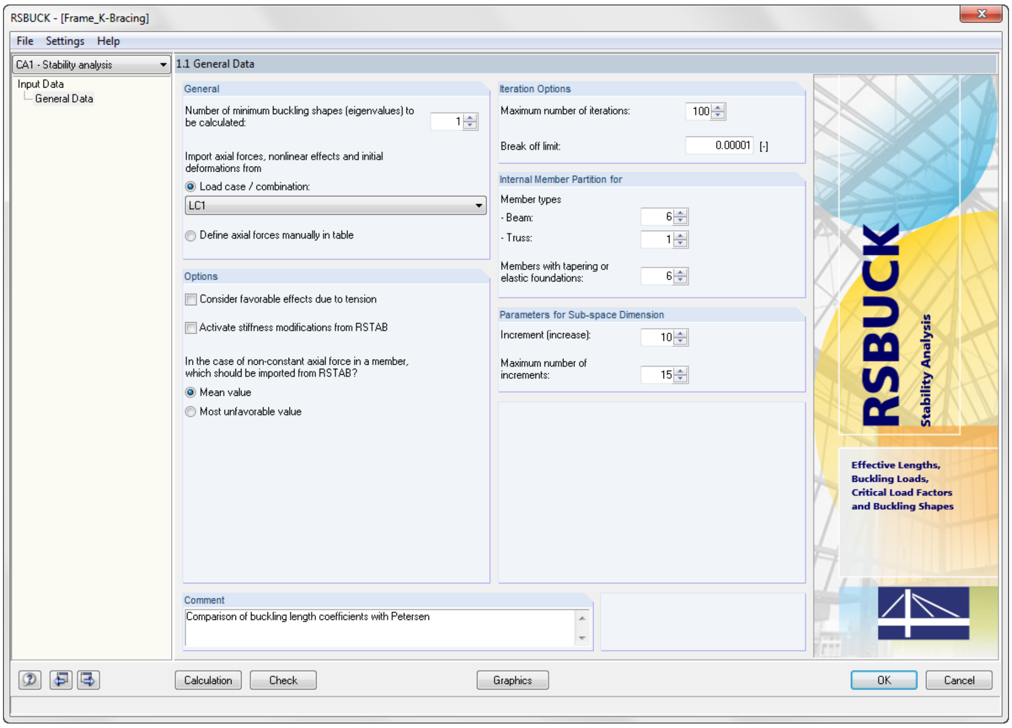


Figure 8.8: RSBUCK window 1.1 *General Data*

RSBUCK determines the following buckling length coefficients k_{cr}:

2.1 Buckling Lengths and Loads

	A	B	C	D	E	F	G	H	I
Member No.	Node No. Start	End	Length L [m]	Shape No.	Buckling Length [m] $L_{cr,y}$	$L_{cr,z}$	Buckling Length Coefficient [-] $k_{cr,y}$	$k_{cr,z}$	Buckling Load N_{cr} [kN]
1	1	2	5.000	1	13.711	0.000	2.742	0.000	2774.860
2	2	3	5.000	1	15.330	0.000	3.066	0.000	2219.890
3	4	5	5.000	1	13.711	0.000	2.742	0.000	2774.860
4	5	6	5.000	1	15.330	0.000	3.066	0.000	2219.890
5	2	5	4.000	1	0.000	0.000	0.000	0.000	0.000
6	3	7	2.000	1	Tension force in member -> No calculation				
7	7	6	2.000	1	0.000	0.000	0.000	0.000	0.000
8	2	7	5.385	1	0.000	0.000	0.000	0.000	0.000
9	5	7	5.385	1	Tension force in member -> No calculation				

Figure 8.9: RSBUCK window 2.1 *Buckling Lengths and Loads*

The results correspond very well to the analytical solution.

Graphics

The [Graphics] show the following buckling shape:

Global Deformations u [-]
RSBUCK CA1 - Stability analysis

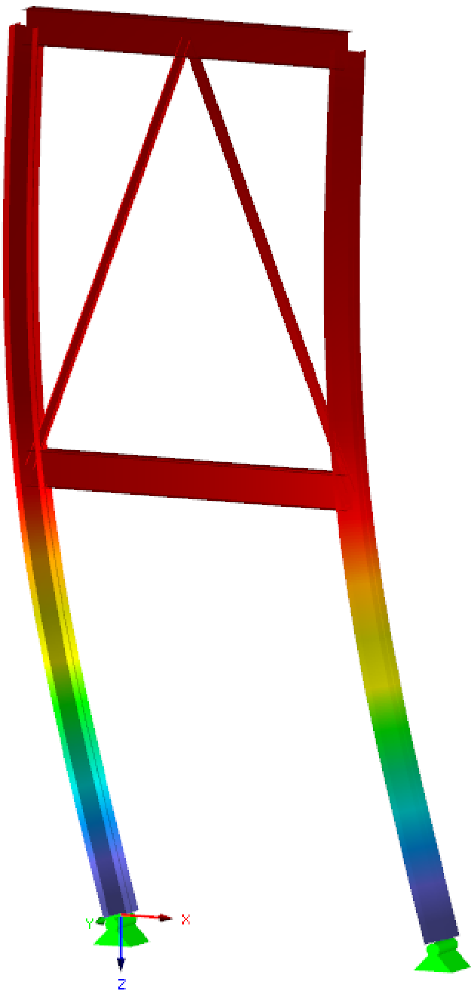


Figure 8.10: Buckling shape

8.3 Frame with Hinged-Hinged Column

We analyze the buckling behavior of a plane frame system according to Figure 8.11.

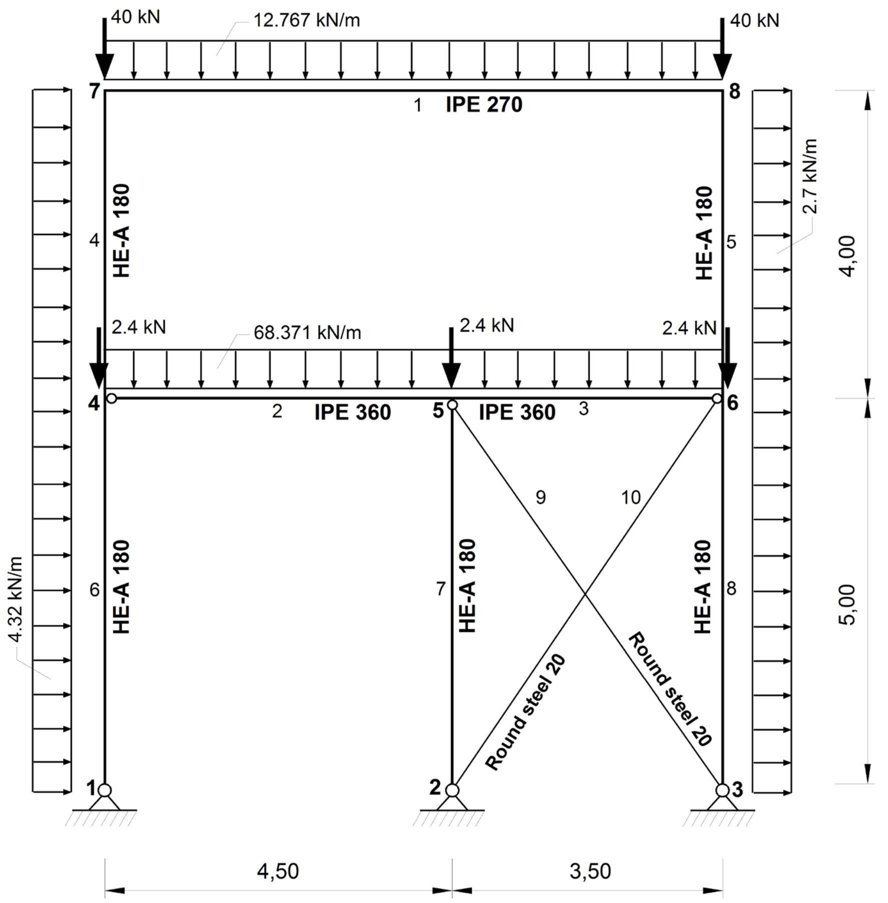


Figure 8.11: Frame system and loading

In this example, the lowest critical load factor (eigenvalue) is obtained on the isolated hinged-hinged member No. 7. This means: At first, member 7 buckles locally; the total model fails only in the second eigenvalue.

The two different buckling shapes shown in the figures on the following page provide information about the buckling behavior.

With an internal partition of **10** for beams, RSBUCK determines a critical load N_{cr} of 2080.94 kN for member 7.

2.1 Buckling Lengths and Loads

	A	B	C	D	E	F	G	H	I
Member No.	Node No. Start	End	Length L [m]	Shape No.	Buckling Length [m] $L_{cr,y}$	$L_{cr,z}$	Buckling Length Coefficient [-] $k_{cr,y}$	$k_{cr,z}$	Buckling Load N_{cr} [kN]
1	2	5	5.000	1	5.000	0.000	1.000	0.000	2080.940
2				2	4.254	0.000	0.851	0.000	2874.060

Figure 8.12: RSBUCK window 2.1 *Buckling Lengths and Loads*

A calculation for the EULER buckling mode 2 confirms the value:

$$N_{cr} = \frac{E \cdot I \cdot \pi^2}{{L_{cr}}^2} = \frac{21{,}000 \cdot 2{,}510 \cdot \pi^2}{500^2} = 2080.91 \text{kN}$$

Equation 8.2

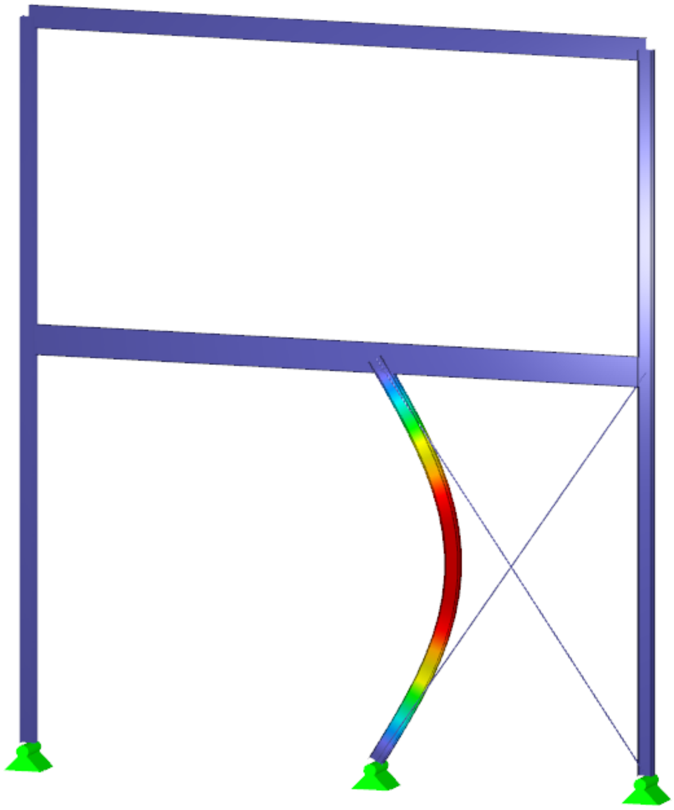


Figure 8.13: First buckling shape - local failure

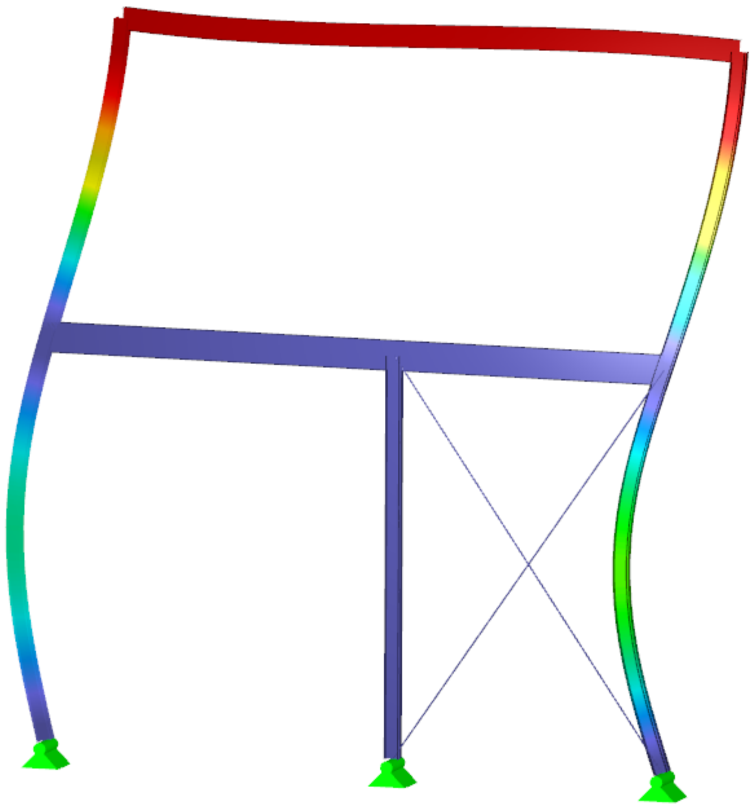


Figure 8.14: Second buckling shape - failure of entire system

A Literature

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

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

[3] HÜNERSEN, G.; FRITSCHE, E.: Stahlbau in Beispielen - Berechnungspraxis nach DIN 18800 Teil 1 bis Teil 3, Werner Verlag, Düsseldorf, 4. Auflage 1998

[4] RUBIN, H; SCHNEIDER, K.-J.: Baustatik - Theorie I. und II. Ordnung, Werner Verlag,Düsseldorf, 3. Auflage 1996

[5] OWCZARZAK, H; STRACKE, M.: Seminarunterlagen zum Dortmunder Praxisseminar -DIN 18800 und EC 3 vom 02.12.94

[6] WERKLE, H.: Finite Elemente in der Baustatik, Vieweg & Sohn Verlag, Wiesbaden,3. Auflage 2008

B Index