Version
August 2013

Add-on Module

RF-FE-LTB

Lateral-Torsional Second-Order Analysis of Members (FEM)

Program Description

Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
E-Mail: Info@dlubal.com
Web: www.dlubal.com

Contents

	Contents	Page
1.	**Introduction**	**5**
1.1	Add-on Module RF-FE-LTB	5
1.2	RF-FE-LTB Team	6
1.3	Using the Manual	7
1.4	Open the RF-FE-LTB Module	7
2.	**Theoretical Background**	**9**
2.1	Preliminary Notes	9
2.1.1	General	9
2.1.2	Basis of the Calculation Method	11
2.1.3	Determination of the Initial Deformation	12
2.1.4	Calculation According to Second-Order Analysis	12
2.2	Definitions	13
2.2.1	Coordinates and Displacements	13
2.2.2	Internal Forces	14
2.2.3	Single Springs and Continuous Springs	15
2.2.4	Loads	17
2.2.5	Boundary Conditions	19
2.3	Stress Calculation	19
2.4	Determination of Restrained Axis of Rotation	22
2.5	Determination of Spring Stiffnesses	25
2.5.1	Rotational Springs	25
2.5.2	Translational Springs	27
2.5.3	Warp Springs	29
2.6	Analyses According to DIN 18800	32
2.6.1	Equivalent Member Method	33
2.6.1.1	Equivalent Member Analysis	34
2.6.1.2	Ultimate Limit State Analysis for Spatially Imperfect Individual Member	36
2.6.2	Determination of Initial Deformations	37
2.6.3	Second-Order Ultimate Limit State Analysis	38
2.6.4	Limit Loads F_T or F_G	39
3.	**Input Data**	**40**
3.1	General Data	40
3.2	Materials	43
3.3	Cross-Sections	45
3.4	Nodal Supports	49
3.5	Elastic Member Foundations	55
3.6	Member End Springs	63
3.7	Member End Releases	67
3.8	Load	69
3.8.1	Nodal Loads	69
3.8.2	Member Loads	71
3.8.3	Imperfections	74
4.	**Calculation**	**77**
4.1	Detail Settings	77
4.2	Start Calculation	79
5.	**Results**	**80**
5.1	Stresses by Cross-Section	81
5.2	Stresses by Set of Members	83
5.3	Stresses by x-Location	83
5.4	Stresses by Stress Point	84
5.5	Internal Forces	85
5.6	Deformations	86
5.7	Support Reaction Forces	87
5.8	Critical Load Factors	88
6.	**Results Evaluation**	**90**
6.1	Results on Cross-Section	91
6.2	Results on RFEM Model	93
6.3	Result Diagrams	97
6.4	Filter for Results	98
7.	**Printout**	**100**
7.1	Printout Report	100
7.2	Graphic Printout	100
8.	**General Functions**	**102**
8.1	Design Cases	102
8.2	Units and Decimal Places	104
8.3	Data Exchange	105
8.3.1	Material Export to RFEM	105
8.3.2	Export Cross-Section to RFEM	105
8.3.3	Export Results	105
9.	**Worked Examples**	**107**
9.1	Beam Under Concentrated Load	107

Contents

	Contents	Page
9.1.1	Bending Without Rotational Restraint	107
9.1.2	Bending with Rotational Restraint	108
9.2	Beam Under Uniform Loading	109
9.3	Cantilevered Beam with Warping Restraint	110
9.4	Cantilevered Beam Under Uniform Load	111
9.4.1	Crane Runway Beam with Free End	111
9.4.2	Cantilever End with Lateral Restraint	112
9.5	Beam Under Uniform Loading	112
9.6	Continuous Beam Under Two Nodal Loads	113
9.7	Continuous Beam Under Uniform Loads	113
9.8	Three-Span Beam with Uniform Loadings	115
9.9	Beam on Foundation with Axial Force	117
9.10	Roof Beam in Office Building	118
9.11	Beam with Eccentric Uniform Load	122
A	**Literature**	**123**
B	**Index**	**124**

1. Introduction

1.1 Add-on Module RF-FE-LTB

The module RF-FE-LTB is not a stand-alone program but is integrated in the RFEM environment as an add-on module. Thus, the model-specific input and loading data of all members is automatically available in the post-processing program. Conversely, the results from RF-FE-LTB can be graphically evaluated in the RFEM workspace and incorporated in the printout report.

RF-FE-LTB carries out the check against lateral buckling and lateral-torsional buckling according to the finite elements method. The analysis is carried out on the entire structural system for notionally singled out sets of members. For this, the program determines the internal forces, deformations, and stresses of spatially stressed structural systems according to the second-order analysis. Furthermore, RF-FE-LTB determines for a given load combination the stability load or the maximum resistance load while observing the normal, shear, and equivalent stresses.

Separate design cases allow for a flexible analysis of lateral buckling and lateral-torsional buckling behavior.

According to DIN 18800, you can carry out the analysis according to various methods. RF-FE-LTB provides the following approaches:

- Calculation of the critical loads on a perfect system. This yields the
  - Elastic flexural buckling load $N_{cr,z}$ about the z-axis (out-of-plane),
  - Elastic torsional buckling load $N_{cr,\vartheta}$ or
  - Elastic critical moment for lateral-torsional buckling M_{cr} about the y-axis.

  With these ideal values, you can carry out the stability analysis according to DIN 18800, Part 2 for I-sections according to the equivalent member method (e. g. with RF-LTB).
- Calculation of the ultimate capacity F_T before a loss of stability occurs not exceeding the predefined elastic limit stress ($F_T \leq F_G$) or determination of the elastic limit load F_G, under which the elastic limit stress is reached ($F_G \leq F_T$). The calculations are carried out on an imperfect system.
- Stress analysis with the internal forces calculated under γ times the loads according to second-order analysis (imperfect system with loads F_d)

The calculation is carried out according to the lateral-torsional second-order analysis with the following possibilities:

- Analysis on entire system to take into account, for example, the restraining effects for structural components susceptible to LTB in a way that is appropriate for the system.
- Determination of imperfections by an eigenvalue analysis prior to the calculation and application of the scaled eigenvector as system initial deformation
- Consideration of the influence of bracings and other supporting structural components by placing eccentric nodal spring as well as idealization of warping restraints by means of according single springs
- Consideration of the elastic rotational restraint by trapezoidal sheeting (corrugated sheeting) of the roof membrane and/or bracing shear stiffnesses in the form of distributed springs and rotational springs acting in both axes directions of the cross-section
- Realization of a possibly existing restrained axis of rotation by specifying appropriate boundary conditions

We hope you will enjoy working with RF-FE-LTB.

Your DLUBAL Team

1.2 RF-FE-LTB Team

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

Program coordination

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

Programming

Prof. Dr.-Ing. Peter Wriggers
Dipl.-Ing. Georg Dlubal
Mgr. Petr Oulehle
Dis. Jiří Šmerák
Lukáš Tůma

Cross-section and material database

Ing. Ph.D. Jan Rybín
Mgr. Petr Oulehle
Ing. Jiří Kubíček

Program design, dialog figures, icons

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

Program supervision

Ing. Martin Vasek
Ing. František Knobloch
M.Eng. Dipl.-Ing. (FH) Walter Rustler

Localization, manual

Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Ing. Jana Duníková
Dipl.-Ing. (FH) René Flori
Ing. Lara Freyer
Alessandra Grosso
Bc. Chelsea Jennings
Jan Jeřábek
Ing. Ladislav Kábrt
Ing. Aleksandra Kociołek
Ing. Roberto Lombino
Eng.º Nilton Lopes
Mgr. Ing. Hana Macková
Ing. Téc. Ind. José Martínez
MA Trans. Anton Mitleider
Dipl.-Ü. Gundel Pietzcker
Mgr. Petra Pokorná
Ing. Michaela Prokopová
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl
Ing. Marcin Wardyn

Technical support, 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) Wieland Götzler
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
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) 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 RFEM. The present manual focuses on special features of the add-on module RF-FE-LTB.

The description of the module follows the sequence and structure of the module's input and results window. The text of the manual shows the described **buttons** in square brackets, for example [View mode]. At the same time, they are shown on the left margin. In addition, **expressions** that are used in dialog boxes, tables, and menus are set in *italics* to clarify the explanations.

The index at the end of the manual helps you find specific terms and subjects. 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 Open the RF-FE-LTB Module

RFEM provides you the following ways to open the RF-FE-LTB add-on module.

Menu

You can open the add-on module by using the RFEM menu:

Add-on Modules → **Design - Steel** → **RF-FE-LTB**.

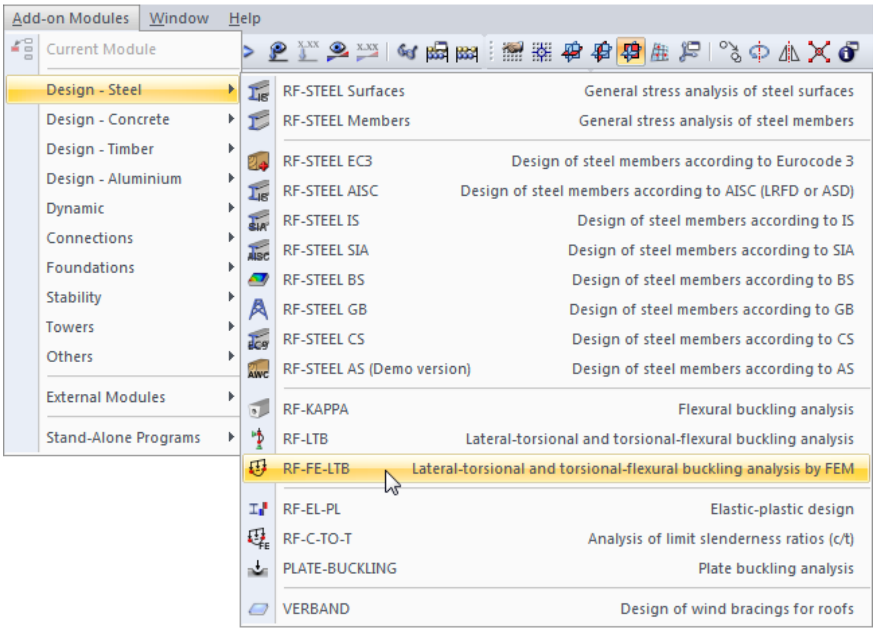


Figure 1.1: Menu: *Add-on Module → Design - Steel → RF-FE-LTB*

Navigator

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

Add-on Modules → RF-FE-LTB.

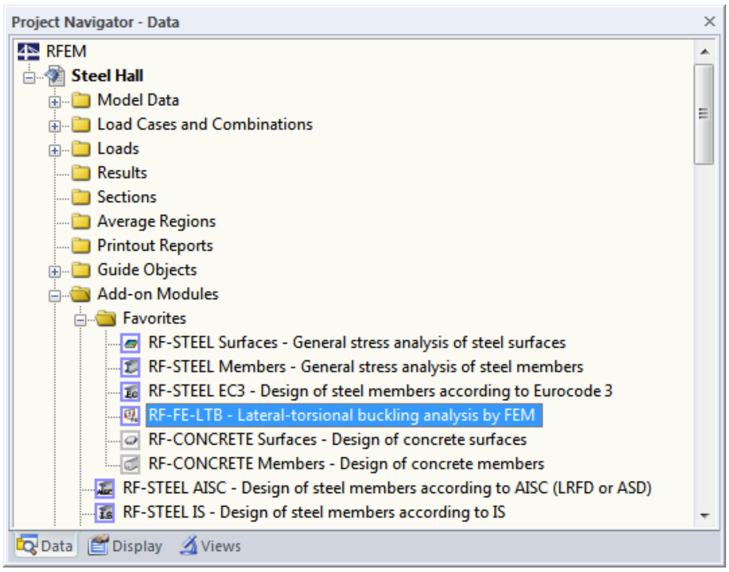
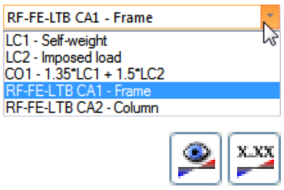


Figure 1.2: Data navigator: *Add-on Modules → RF-FE-LTB*

Panel



If results from RF-FE-LTB are already available in the RFEM model, you can also open the add-on module in the panel:

Select the relevant RF-FE-LTB design case in the load case list located in the menu bar. Then, click [Results on/off] to graphically display the design criterion on the members.

On the panel, you can click [RF-FE-LTB] to return to the module.

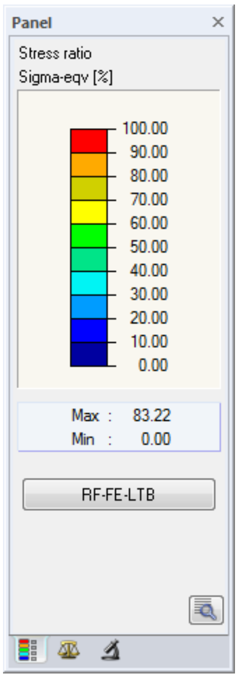


Figure 1.3: Panel button [RF-FE-LTB]

2. Theoretical Background

This chapter provides the theoretical background that is important for working with RF-FE-LTB. Basically, we introduce the theoretical approaches described in the literature. However, this introductory chapter cannot replace a textbook.

2.1 Preliminary Notes

2.1.1 General

Lateral-torsional buckling represents a stability case in which a primary flexural deformation is superimposed with a lateral displacement including torsion. Lateral-torsional buckling and flexural-torsional buckling are closely related terms. The difference is that lateral-torsion buckling is commonly associated with a stress from eccentric compression force, whereas flexural-torsional buckling is induced by bending. Furthermore, there is the case of compression bending. In all cases, the position of the line of action of the loads applied to a member has a considerable influence on the magnitude of the stability load.

All mentioned stability problems can be analyzed in RF-FE-LTB. You can use different methods to calculate the lateral-torsional buckling of beams. Here are some of them:

- Equivalent member method according to DIN 18800, Part 1 and 2 (program RF-LTB [10])
- Calculation of eigenvalues (M_{cr}, N_{cr}) for continuous members or any frameworks subjected to three-dimensional stress (program RF-FE-LTB)
- Limit load or stability calculation of frameworks subjected to three-dimensional stress according to second-order analysis on an imperfect system (program RF-FE-LTB)
- Limit load or stability calculation of frameworks subjected to three-dimensional stress according to a geometrically exact theory on the imperfect system

The equivalent member method is sufficiently accurate for many practical construction purposes. This method is described and verified in DIN 18800, Part 1 [7] and 2 [8] and many other publications. The method is implemented, for example, in the add-on module RF-LTB [10], where the lateral-torsional buckling analysis is carried out for members with monosymmetrical or doubly symmetrical I-section subject to uniaxial or biaxial bending and constant axial force.

The equivalent member method according to DIN 18800 is limited in its application to particular cross-sections (see above). In addition to this, you have to define boundary conditions for the equivalent member, which is not easy for general framework systems and therefore can only be estimated. For a more precise calculation, the framework subjected to three-dimensional stresses is to be computed by second-order analysis. Usually, this has to do with the calculation of the elastic stability load of a single-span or multi-span beam of a frame.

The add-on module RF-FE-LTB is based on the finite element method and can be used to calculate the stability loads of members. Here, the elastic material behavior is assumed for a geometric nonlinear behavior. The following basic assumptions apply for the warping torsional theory:

1. Shape-constant cross-sections to exclude local instabilities
2. Bernoulli bending
3. Moderate displacements or rotations that are small overall compared to the system dimensions

The calculations are three-dimensional according to the second-order analysis for flexural-torsional buckling, where the individual member elements are regarded as straight.

In the analysis, initial deformations can be taken as scaled eigenvectors of the system. Furthermore, it is also possible to calculate eccentrically acting loads (for example load on top or bottom flange).

Depending on the geometric shape of the structural system, the actions, and the initial deformations (imperfections), different maximum failure and/or ultimate limit states can occur. Figure 2.1 presents the basic structural responses.

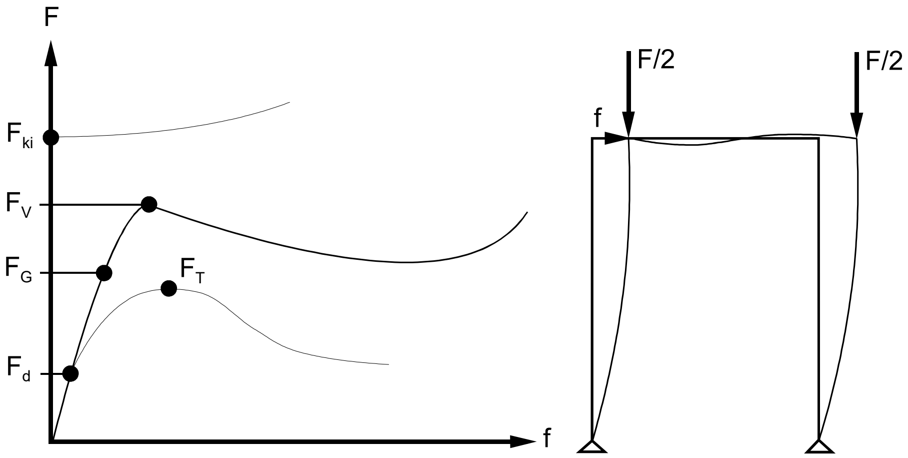


Figure 2.1: Structural responses

RF-FE-LTB gives the following results, as appropriate (see also Figure 2.1):

1. Critical load (bifurcation load) F_{cr}
   - Elastic critical moment for lateral-torsional buckling $M_{cr,y}$
   - Elastic flexural buckling load $N_{cr,z}$
   - Elastic torsional buckling load $N_{cr,\vartheta}$

   The program calculates always the smallest critical load of the system without considering initial deformations. The elastic critical loads are required in the application of the equivalent member method (see chapter 2.6.1, page 33).

2. Ultimate capacity F_T due to loss of stability (limit load) observing the elastic limit stress on the imperfect system

   The limit load F_T is determined assuming a pure elastic material behavior with limitation by an elastic limit stress to be defined.

3. Elastic limit load F_G on imperfect system

   This is a load that can be resisted by the system without that in any cross-section part the normal stress, the shear stress, or the equivalent stress (according to VON MISES) is greater than the respective limit stress. The calculation is to be carried out only in case of specified initial deformations.

4. Possible limit load F_v due to loss of stability for specification of initial deformations without observing the elastic limit stresses

5. Verification of the limit stresses under design loads F_d on the imperfect system according to second-order analysis

Based on the second-order analysis, RF-FE-LTB can thus automatically find the critical, snap-through, or elastic limit loads. These loads are determined iteratively.

To determine the limit load F_T or the elastic limit load F_G (see Figure 2.1), an initial deformation is to be applied to the system. In RF-FE-LTB, it is automatically generated from the first, lowest eigenmode (eigenvector), as this buckling mode corresponds to the lowest stability load. The scaling of this eigenmode is carried out according to DIN 18800 Part 2; however, it can also be user-defined (see chapter 2.6.2, page 37 and chapter 3.8.3, page 75).

Section types

RF-FE-LTB carries out analyses for all rolled sections, monosymmetrical and doubly symmetrical I-sections, channels, T-sections, L-sections, rectangular sections, C-sections, hollow- and annular sections, as well as built-up cross-sections. Here, the stresses are determined according to the second-order analysis at the governing cross-section points. The determination of the elastic limit load F_G (see point 3 above) or the limit stress analysis (see point 5 above) is based on these stress calculations.

Moreover, you can also analyze any cross-sections (for example SHAPE-THIN sections). They are directly imported from RFEM. Then, the stress analyses are run in the RF-FE-LTB module.

Springs can be defined in RF-FE-LTB as single springs or continuous springs with an arbitrary point of application in the cross-section. As a rule, this is required when the stiffening effect of the roof cladding (for example trapezoidal sheeting) is to be taken into account.

Concentrated loads and line loads can act at arbitrary locations in the cross-section.

2.1.2 Basis of the Calculation Method

The theoretical basis of the program RF-FE-LTB is very extensive and cannot be discussed here in detail. The according background can be found, for example, in PETERSEN [2] or in RAMM, HOFMANN [11].

Usually, there are no analytical solutions for flexural-torsional problems that include nonlinear deformational dependencies. Therefore, the method of finite elements (FEM) is used in order to determine approximate solutions for the differential equations given in [2] or [11]. The precision of the solution is dependent on the selected number of finite elements (see chapter 9.2).

For the FE discretization, elements with two nodes are used. Cubic Hermite polynomials are applied within the elements for displacements in y- or z-direction and for the torsion about the x-axis. The longitudinal displacement in x-direction is described by an application of a linear polynomial. These applications solve the homogeneous differential equation of the corresponding linear analysis precisely, but are approximations for the second-order theory. The practical application of the method showed that usually eight elements per span of a beam are sufficient to calculate deformations with deviations of less than 5% from the convergent solution. A solution is called convergent if there are no more changes in the solution in case of a doubled number of elements (see example 9.2, page 109).

Thus, we obtain a total of seven degrees of freedom per element node: u_x, v_y, w_z, φ_x, φ_y, φ_z, φ'_x. Where u_x is the longitudinal displacement in the member direction; v_y or w_z is the displacement in y- or z-direction, respectively; φ_x, φ_y, φ_z is the torsion about the x-, y-, or z-axis, respectively; and φ'_x is the warping.

2.1.3 Determination of the Initial Deformation

The initial deformation is computed by solving the eigenvalue problem:

$$(K - \lambda \cdot I) \cdot \Phi = 0$$

Equation 2.1: Eigenvalue analysis

In Equation 2.1, the stiffness matrix *K* is a function of the axial forces and moments of the basic load state. *I* is the unit matrix.

By solving the eigenvalue problems by an iterative method, we obtain the eigenvector Φ belonging to the lowest eigenvalue; this eigenvector then determines the shape of the initial deformation. The scaling of the initial deformation is according to DIN 18800 Part 2.

2.1.4 Calculation According to Second-Order Analysis

For the second-order analysis, the following preconditions and assumptions are made:

- The cross-sections are thin-walled and constant within each portion.
- The individual member elements are regarded as straight.
- The shape of the cross-section shall remain unchanged in the deformation of the member. Thus, we exclude local instabilities, which possibly are also to be prevented by the stiffening of cross-sections.
- For bending, the BERNOULLI hypothesis applies stating that the cross-section remains plane.
- The displacements and torsions are small compared to the dimensions of the system.

The internal forces computed by second-order analysis are relative to the displaced and rotated coordinate system and therefore do not need to be transformed for the stress analysis.

The analysis for a flexural-torsional problem can be carried out in different ways in RF-FE-LTB. The ways are the following:

1. Determination of the critical load factor on the undeformed system
2. Determination of the critical load factor on the deformed system
3. Analysis of the stresses under design load
4. Calculation of the maximum resisting load not exceeding the stresses

Basically, the calculation is iterative, with the stiffness matrix *K* changing due to the already computed internal forces and deformations. For the analyses according to the points 2, 3, and 4 described above, the eigenmodes are determined with the internal forces from the first step and considered according to chapter 2.6.2 before the actual iterative calculation.

The determination of the critical load factor according to point 1 or 2 gives the stability load of the analyzed structural system. In a numeric calculation, this load is characterized by the fact that either the determinant of the matrix *K* becomes zero, or that very high displacements occur for very small load increases in the calculation. In both cases, the module RF-FE-LTB determines that the corresponding state of equilibrium is no longer stable.

In the actual calculation, the program first computes the internal forces, deformations, and stresses for the load increments specified by the user (see chapter 2.3 *Stress Calculation*). At this point, there are two possibilities:

a) The load increments defined by the user are stable states of equilibrium. In this case, RF-FE-LTB automatically increases the load exceeding the defined maximum load until an instability occurs. The according value is then precisely determined by means of a nested iteration.

b) The load increments defined by the user cannot be reached all. In this case, RF-FE-LTB nests the load increment of the instability, starting from the last stable load increment.

Thus, the critical load factor is known that belongs to the critical load F_{cr} (lateral-torsional buckling load). For point 1 (see above), we thus obtain the lateral-torsional buckling load belonging to the undeformed system. The maximum bending moment M_y thus corresponds to the ideal lateral-torsional buckling moment. For point 2, we obtain the possible limit load F_V due to loss of stability. In both cases, the program does not check whether or not the limit stresses are observed. The stresses can be found in the results windows, where exceeded limits are highlighted.

The calculation of the system with initial deformation (see point 2 above) can also be carried out in a way that the limit stresses defined by the user are observed. In this case, RF-FE-LTB carries out the steps a) and b) and checks whether the limit stresses are observed.

Finally, it is also possible to carry out the second-order analysis on the system susceptible to lateral-torsional buckling under the design load (see point 3 above). If RF-FE-LTB can find an equilibrium in this case, then the design is directly verified if all limit stresses are observed.

For examples of the presented cases, see chapter 9.

2.2 Definitions

2.2.1 Coordinates and Displacements

Figure 2.2 shows the cross-section coordinates and the positive displacement values.

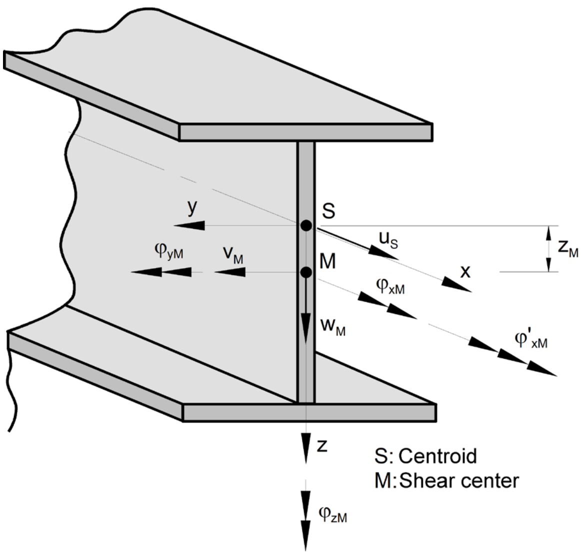


Figure 2.2: Cross-section coordinates and displacement values

The longitudinal displacement u_S is related to the centroid *S*. In contrast, the displacements v_M and w_M as well as the rotations φ_{xM}, φ_{yM}, φ_{zM}, and the warping φ'_{xM} are relative to the shear center *M*. The displacements v, w, and u of an arbitrary cross-section point can be expressed with the linearization common in the second-order analysis by means of the displacement values of the shear center.

$$v = v_M - (y - y_M)(1 - \cos\varphi_{xM}) - (z - z_M)\sin\varphi_{xM} \approx v_M - (z - z_M)\sin\varphi_{xM}$$

$$w = w_M - (z - z_M)(1 - \cos\varphi_{xM}) + (y - y_M)\sin\varphi_{xM} \approx w_M + (y - y_M)\sin\varphi_{xM}$$

Equation 2.2: Displacement values

The displacement u of a point results from the translation of the cross-section in the x-direction, the rotation about the y- and z-axis and from the warping due to torsion:

$$u = u_S - w'_M\, z - v'_M\, y - \varphi'_{xM}\, \omega_o$$

with ω_0 Unit warping

Equation 2.3: Displacement values

2.2.2 Internal Forces

Figure 2.3 shows the used definitions of internal forces.

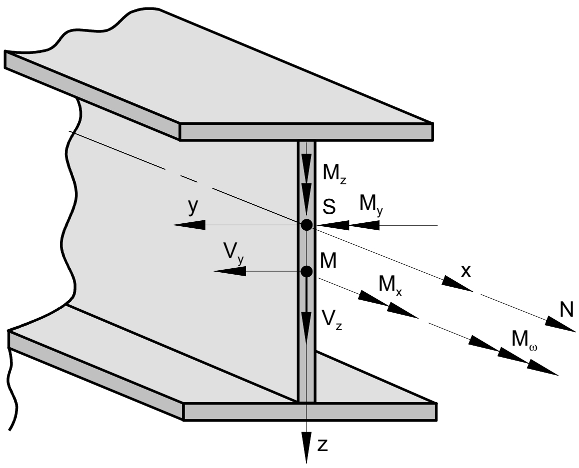


Figure 2.3: Definitions on internal forces at the positive cut face

The shear forces V_z and V_y as well as the torsional moment M_x and the warping moment M_ω are relative to the shear center *M*; the bending moments M_y and M_z as the axial force N are relative to the centroid *S*.

The internal forces are always related to the principal axes of the cross-section. For unsymmetric cross-sections, we therefore assume the shear forces V_v and V_u and the bending moments as M_u and M_v.

2.2.3 Single Springs and Continuous Springs

Elastic supports can be modeled by considering centrically or eccentrically arranged single springs or/and continuous springs (element springs).

Figure 2.4 shows the **centric single springs** on node K. These springs are related to the global coordinate system (CSYS).

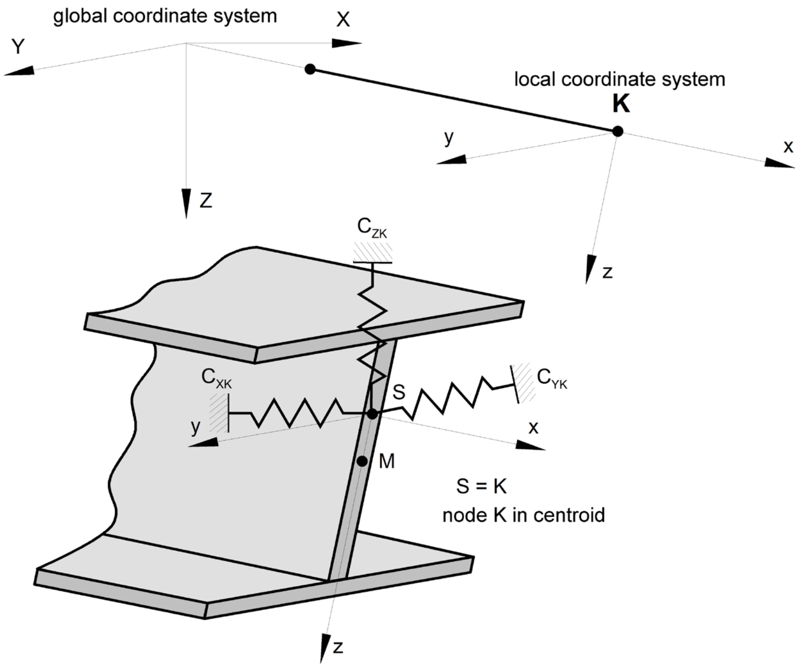


Figure 2.4: Centric nodal springs

The spring constants in Figure 2.4 mean:

C_{XK}	Nodal spring constant in global X-direction in [kN/cm]
C_{YK}	Nodal spring constant in global Y-direction in [kN/cm]
C_{ZK}	Nodal spring constant in global Z-direction in [kN/cm]
$C_{\varphi XK}$	Nodal rotational spring constant about global X-axis in [kNcm]
$C_{\varphi YK}$	Nodal rotational spring constant about global Y-axis in [kNcm]
$C_{\varphi ZK}$	Nodal rotational spring constant about global Z-axis in [kNcm]
$C_{\omega K}$	Warp spring constant in [kNcm^3]

The **eccentric nodal springs** at node K are relative to the local coordinate system:

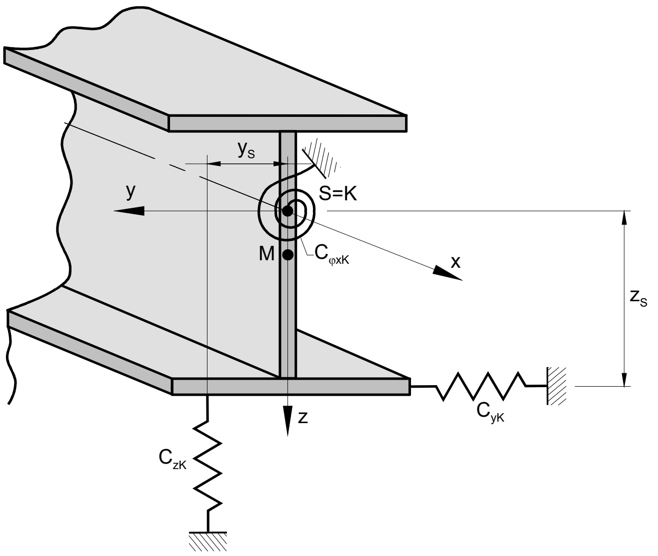


Figure 2.5: Eccentric nodal springs

x,y,z	Local coordinate system
C_{yK}	Nodal spring constant in local y-direction in [kN/cm]
C_{zK}	Nodal spring constant in local z-direction in [kN/cm]
$C_{\varphi xK}$	Rotational spring constant about local x-axis in [kNcm]
$C_{\omega K}$	Warp spring constant in [kNcm^3] relative to the local x-axis (not shown in figure)
y_S	Distance of spring C_{zK} from centroid S
z_S	Distance of spring C_{yK} from centroid S

The **continuous springs** (element springs) are defined in Figure 2.6. These subgrade moduli are relative to the local coordinate system and constant along the member. In the program, they are related to the shear center *M* and recalculated.

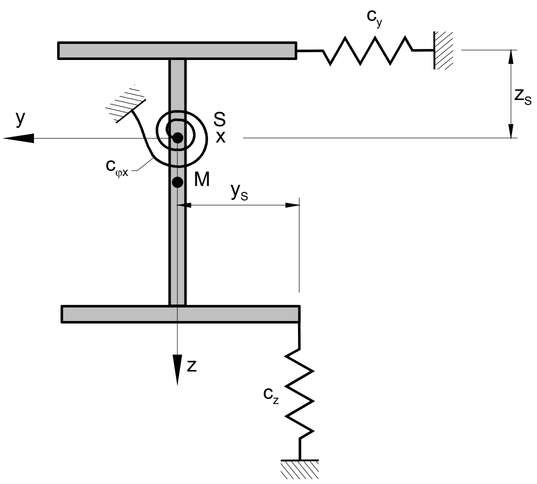


Figure 2.6: Continuous springs

c_y	Nodal spring constant in local y-direction in [kN/cm]
c_z	Nodal spring constant in local z-direction in [kN/cm]
$c_{\varphi x}$	Rotational spring constant about local x-axis in [kNcm]
y_S	Distance of spring c_z from centroid S
z_S	Distance of spring c_y from centroid S

2.2.4 Loads

Figure 2.7 shows concentrated loads defined as **centric nodal loads**.

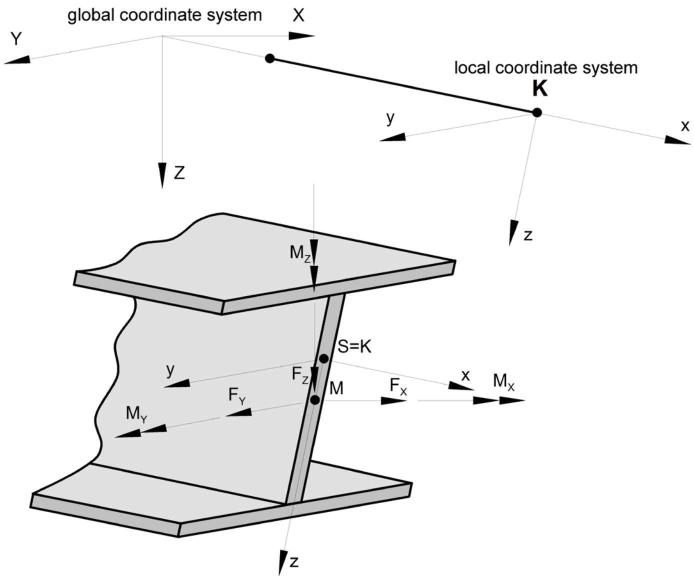


Figure 2.7: Centric concentrated loads

F_X	Concentrated load in global X-direction relative to M
F_Y	Concentrated load in global Y-direction relative to M
F_Z	Concentrated load in global Z-direction relative to M
M_X	Concentrated moment about global X-axis, relative to M
M_Y	Concentrated moment about global Y-axis, relative to M
M_Z	Concentrated moment about global Z-axis, relative to M

If the concentrated loads act centrically in the node *K* and point in the direction of the local coordinates, we have the simple option to specify these loads locally as eccentric loads (Figure 2.8) by setting the according coordinates to zero.

Eccentric concentrated loads at node *K* are to be related to the local coordinate system.

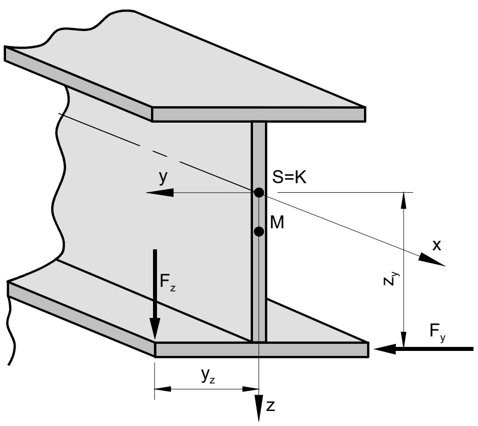


Figure 2.8: Eccentric concentrated loads

F_y	Concentrated load in local y-direction
F_z	Concentrated load in local z-direction
z_y	Distance of load F_y from centroid in z-direction
y_z	Distance of load F_z from centroid in y-direction

The following figure shows the definitions of the **line loads.**

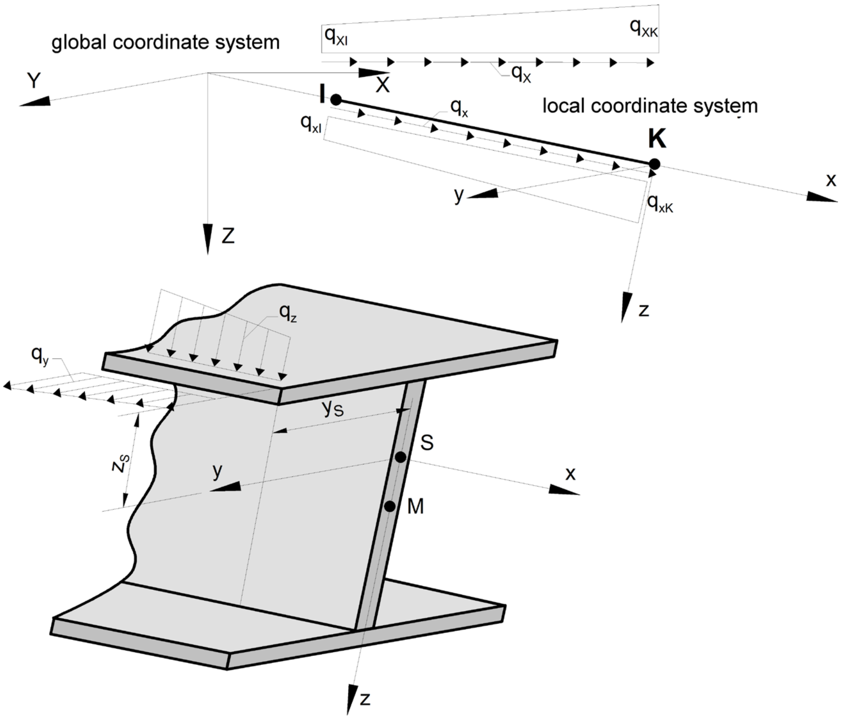


Figure 2.9: Line loads

q_x / q_X	Line load in local x- or global X-direction
q_y / q_Y	Line load in local y- or global Y-direction
q_z / q_Z	Line load in local z- or global Z-direction
y_S	Local y-coordinate (relative to S) of the line load q_z
z_S	Local z-coordinate (relative to S) of the line loads q_y
m_x / m_X	Local or global uniform torsional moment relative to S

The global line loads are automatically assumed to be acting in the shear center *M*, the local line loads are to be specified relative to the centroid *S*.

The line loads can be specified globally as well as locally. Eccentric line loads can only be defined as locally related. For the specification, the loads are relative to the centroid *S*. Within the program, they are converted to the shear center *M*.

The loads always relate to the main axes of the cross-section. For unsymmetric cross-sections, the concentrated loads F_u and F_v and the line loads q_v and q_u are to be assumed.

2.2.5 Boundary Conditions

The following figure shows the components of the displacement, torsion, and warping for specifying the boundary conditions.

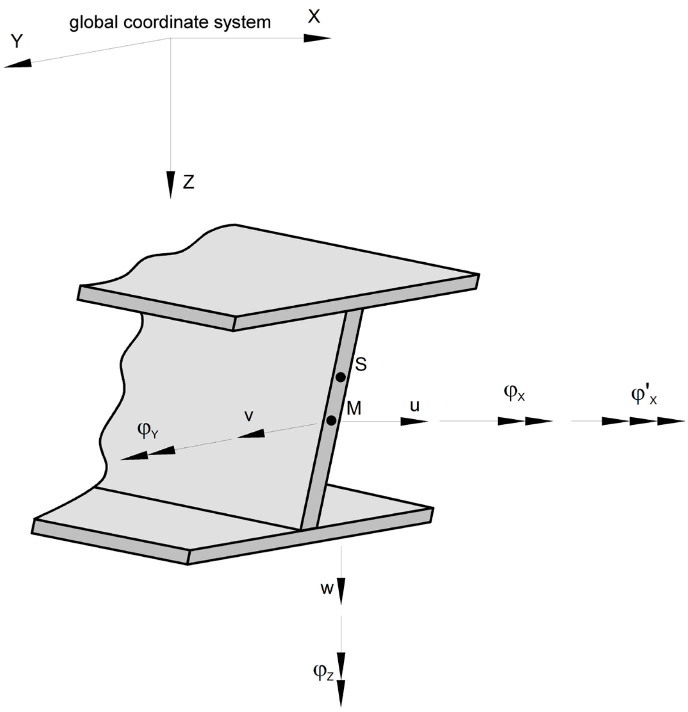


Figure 2.10: Boundary conditions

The restraints of the structural system by the support reactions (boundary conditions) must be specified in the global direction, that is, they are related to the global axes X, Y, and Z. Thus, the individual translational and rotational components are set to zero or set free by specifying reference numbers.

2.3 Stress Calculation

RF-FE-LTB calculates the normal, shear, and VON MISES equivalent stresses at the governing stress points ***i*** of the cross-section. All rolled, built-up, and parametric thin-walled cross-sections of the library are permitted.

In the following equations, the stress points of the cross-section are indicated by the coordinates (y_i, z_i). All stresses are determined from internal forces that are calculated by second-order analysis taking into account the partial safety factors for the actions.

The governing point for the determination of the stresses are dependent on the cross-sectional shape. In Figure 6.4 on page 92, they are shown for an exemplary cross-section. In the cross-section graphic, you can identify the stress point numbers listed in the output tables.

The bending moment $\mathbf{M_y}$ is positive if tensile stresses occur on the positive member side (in direction of the z-axis). $\mathbf{M_z}$ is positive if compressive stresses occur on the positive member side (in direction of the y-axis). The sign definition for torsional moments, axial forces and shear forces conforms to the usual conventions: These internal forces are positive if they act in a positive direction.

With the consideration of the warping torsion, for the **normal stresses σ_x** not only components from axial force and bending, but also from the warping torsional moment occur. We obtain the following normal stress in a point *i* of the cross-section:

$$\sigma_{x,i} = \frac{N}{A} + \frac{M_y}{S_y(y_i, z_i)} - \frac{M_z}{S_z(y_i, z_i)} - \frac{M_\omega}{I_\omega} \omega_M(y_i, z_i)$$

Equation 2.4: Normal stress σ_x

The symbols mean:

Symbol	Description
N	Axial force
M_y	Bending moment about y-axis
M_z	Bending moment about z-axis
M_ω	Warping torsional moment
A	Cross-sectional area
$S_y(y_i,z_i)$	Section modulus about y-axis for point (y_i,z_i)
$S_z(y_i,z_i)$	Section modulus about z-axis for point (y_i,z_i)
I_ω	Warping constant
ω_M	Main warping at point (y_i,z_i)

Table 2.1: Parameter for normal stresses σ_x

The **shear stresses** are composed of shear force and torsional parts. The primary shear stresses τ_p in a point *i* of the cross-section are determined as follows:

$$\tau_{pi} = \frac{V_y \cdot Q_z(y_i, z_i)}{I_z \cdot t(y_i, z_i)} + \frac{V_z \cdot Q_y(y_i, z_i)}{I_y \cdot s(y_i, z_i)} + \left| \frac{M_{x,p}}{W_T(y_i, z_i)} \right|$$

Equation 2.5: Primary shear stresses τ_p

The symbols mean:

Symbol	Description
V_y	Shear force in direction of the y-axis
V_z	Shear force in direction of the z-axis
$M_{x,p}$	Primary torsional moment
I_y	Second moment of area relative to y-axis
I_z	Second moment of area relative to z-axis
$Q_y(y_i,z_i)$	Statical moment relative to y-axis for point (y_i,z_i)
$Q_z(y_i,z_i)$	Statical moment relative to z-axis for point (y_i,z_i)
$t(y_i,z_i)$	Thickness of the governing cross-section parts in point (y_i,z_i)
$s(y_i,z_i)$	Thickness of the governing cross-section parts in point (y_i,z_i)
$W_T(y_i,z_i)$	Torsional section modulus for point (y_i,z_i)

Table 2.2: Parameters for primary shear stresses τ_p

Furthermore, it is possible to calculate the secondary shear stress τ_s due to the secondary torsional moment $M_{x,s}$.

$$\tau_{si} = \frac{M_{x,s} \cdot A_\omega(y_i, z_i)}{I_\omega \cdot t(y_i, z_i)}$$

Equation 2.6: Secondary shear stress τ_s

The symbols mean:

Symbol	Description
$M_{x,s}$	Secondary torsional moment
$A_\omega(y_i,z_i)$	Warping area in point (y_i,z_i)
I_ω	Warping constant
$t(y_i,z_i)$	Thickness of the governing cross-section parts in point (y_i,z_i)

Table 2.3: Parameter for secondary shear stresses τ_s

The calculation of the secondary shear stresses is possible for rolled cross-sections, monosymmetrical and doubly symmetrical I-sections, and box sections.

In RF-FE-LTB, it is up to you to decide whether or not the secondary shear stresses are to be taken into account in the stress calculation. If they are to be included in the stress calculation, they are directly added to the primary shear stresses.

The **Equivalent stress σ_{eqv}** according to VON MISES is determined from the normal and shear stress as follows:

$$\sigma_{eqvi} = \sqrt{\sigma_x^2 + 3\,\tau_{p,si}^2}$$

Equation 2.7: Equivalent stress σ_{eqv}

In the standard case, it is assumed for the calculation of the equivalent stresses that the secondary shear stresses may be neglected. However, if they are considered (see above), the sum from primary and secondary shear stress is taken for $\tau_{p,s}$. The shear stresses due to primary torsional moment according to Equation 2.5 are always considered in Equation 2.7.

The normal, shear, and equivalent stresses are calculated for all points in the cross-section that can be governing for the three-dimensional stress resulting in lateral-torsional buckling. In the output, the location where the maximum value occurs for every type of stress (normal, shear, and equivalent stress) is shown.

In the limit load calculation, the limit load F_G is calculated for which at no location in the cross-section the allowable values for the stresses due to γ times the actions are exceeded. To this end, it is necessary to determine the maximum stress in the cross-section. Thus, the following conditions are to be satisfied:

$$\max_i(\sigma_{xi}) \le \frac{f_{y,k}}{\gamma_M}; \quad \max_i(\tau_{p,si}) \le \frac{f_{y,k}}{\gamma_M * \sqrt{3}}; \quad \max_i(\sigma_{eqvi}) \le \frac{f_{y,k}}{\gamma_M}$$

Equation 2.8: Conditions for the limit load F_G

If the design is carried out according to DIN 18800 Part 2, element (121) and Part 1, element (749), then the normal or equivalent stresses may exceed these limit values by 10 % (see chapter 2.6.1).

2.4 Determination of Restrained Axis of Rotation

In practical constructions, there is often a constructively caused lateral-torsional problem with restrained axis of rotation (translational restraint) at a distance z_D of the centroid. This restrained rotational axis is implemented as continuous or discrete translational springs in y-direction. For the spring stiffnesses, values of the magnitude 10^8 through 10^{10} for c_y are to be taken to suppress the displacements in the restrained axis of rotation.

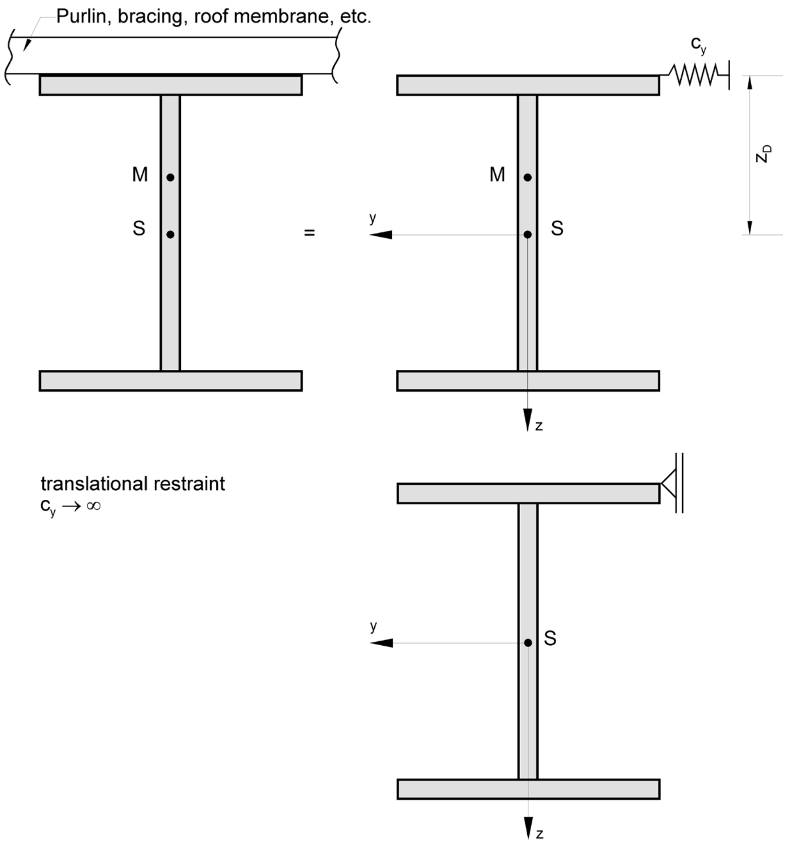


Figure 2.11: Torsional restraint

The translational restraint may be taken according to DIN 18800 Part 2 [8] in the design of a sufficient lateral restraint of deformation. To achieve a sufficient restraint, you can, for example, use masonry permanently connecting to the compression flange. If trapezoidal sheets according to DIN 18807 [13] are connected to the girder and the condition

$$\text{prov } S \geq \text{req } S$$

Equation 2.9: Condition for shear panel stiffness

where

$$\text{req } S = S_a = \left(E I_\omega \frac{\pi^2}{L^2} + G I_T + E I_z \frac{\pi^2}{4L^2} h_p^2 \right) \frac{70}{h_p^2}$$

Equation 2.10: Required shear panel stiffness for attachment in every groove

for an attachment in every groove (rib) is satisfied, then the point of connection must be seen as rigidly fixed in the plane of the trapezoidal sheeting.

S_a refers to the portion of the shear strength of the trapezoidal sheeting for the analyzed beam according to DIN 18800 Part 1 [7] in case of attachment in every rib. Here, L is the span width of the beam to be braced and h_p is the depth of the beam (provided that it is an I-section).

If the trapezoidal sheeting is fastened only in each <u>second</u> rib, then the following applies:

$$\text{req } S = S_b = 5 \cdot S_a$$

Where S_a see Equation 2.10

Equation 2.11: Required shear panel strength for fixation in every second groove

Equation 2.10 and Equation 2.11 for the determination of the lateral restraint of a beam (translational restraint) can, in case of an according type of the points of connection, used for other sheeting than trapezoidal sheeting, see note on DIN 18800 Part 2, element (308).

The ideal shear modulus of a trapezoidal sheet is given as:

$$G_s = \frac{10^4}{K_1 + 100 \frac{K_2}{L_s}} \quad \left[\frac{kN}{m}\right]$$

where K_1 Shear panel value according to approval in [m/kN]

K_2 Shear panel value according to approval in [m²/kN]

L_S Shear panel length in [cm], see Figure 2.12

Equation 2.12: Shear modulus of trapezoidal sheeting

Thus, it follows for the shear stiffness part of the beam to be braced (for example frame beam in Figure 2.12):

$$S_T = \frac{a}{100} G_s \quad [kN]$$

where a Distance of the beam to be braced (frame beam) in [cm]

Equation 2.13: Shear stiffness of trapezoidal sheeting

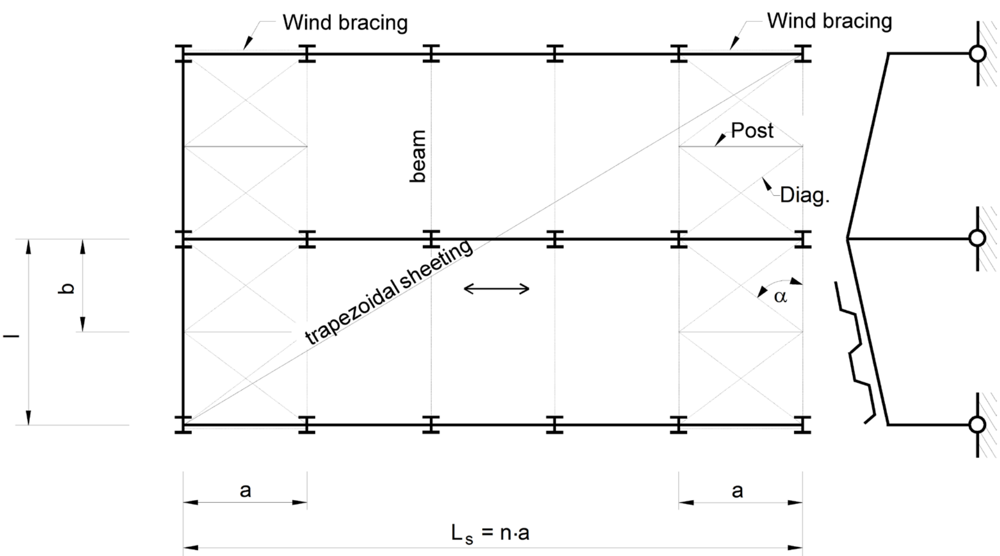


Figure 2.12: Frame beam with trapezoidal sheeting and bracings

The shear strength of the wind bracings and stabilizing bracings can also be considered. For the ideal shear stiffness of <u>one</u> bracing with connections without slip, we obtain (see [8] and [1]):

$$S_V = \frac{1}{\dfrac{1}{E \cdot A_D \cdot \sin^2\alpha \cdot \cos\alpha} + \dfrac{1}{E \cdot A_P \cdot \cot\alpha}}$$

where	S_V	Shear stiffness of bracing in [kN]
	A_D	Area of the diagonal in [cm]
	A_P	Area of posts in [cm]
	α	Angle between diagonal and frame beam flange

Equation 2.14: Shear stiffness of bracing

In the equation above, only the tension diagonals of the cross bracing are considered. If different posts or diagonals are intended, the minimum cross-section areas are to be taken for A_P or A_D.

Equation 2.14 can be converted as follows.

$$S_V = \frac{a^2\, b\, E}{\dfrac{\left(\sqrt{a^2+b^2}\right)^3}{A_D} + \dfrac{a^3}{A_P}}$$

Equation 2.15: Shear stiffness of bracing

Thus, it is possible to approximately calculate the shear stiffness for <u>one</u> frame beam or girder (only from bracings).

$$S_R = m \frac{a}{L_s} S_V$$

where m Number of stiffening bracings in roof plane

Equation 2.16: Shear modulus of trapezoidal sheeting

If the shear stiffnesses from trapezoidal sheeting and bracing are applied <u>at the same time</u>, we obtain from Equation 2.10, Equation 2.14, Equation 2.15, and Equation 2.13:

- Connection in <u>every</u> groove:

  $$\text{prov } S = S_T + S_R$$

  Equation 2.17: Shear panel stiffness

- Connection in every <u>second</u> groove

  $$\text{prov } S = \frac{1}{5} S_T + S_R$$

  Equation 2.18: Shear panel stiffness

The analysis is then run according to Equation 2.9.

In RF-FE-LTB, a continuous translational restraint is to be accordingly idealized by continuous lateral translational springs c_y with a high stiffness, for example 10^6 (kN/cm)/cm. An example for this can be found in chapter 9.9, page 117.

Laterally adjacent members that are fixed in the longitudinal direction (for example single purlins resting on the frame beam) can be idealized by discrete single springs c_y in those points, for example as follows:

$$c_y = \frac{E \cdot A}{L}$$

where	L	Length of purlin up to the support point
	E	Modulus of elasticity
	A	Cross-sectional area of purlins

Equation 2.19: Translational spring by single support

If the check of a translational restraint according to DIN 18800 Part 2 is not fulfilled, it is possible to determine a continuous translational spring by means of the determined ideal shear stiffness ***prov S*** (see chapter 2.5.2).

2.5 Determination of Spring Stiffnesses

2.5.1 Rotational Springs

The calculation of the provided rotational restraint coefficient is based on the model of several springs placed one behind the other (see [8], [14]).

$$\frac{1}{\text{prov } c_{\vartheta,k}} = \frac{1}{c_{\vartheta M,k}} + \frac{1}{c_{\vartheta A,k}} + \frac{1}{c_{\vartheta P,k}}$$

Equation 2.20: Effective rotational restraint

For simplification, Equation 2.20 in DIN 18800 is expressed with the characteristic values. The symbols of this equation are described in the following.

Rotational restraint $c_{\vartheta M,k}$ from supporting structural component

$$c_{\vartheta M,k} = \frac{E \cdot I_a}{a} \cdot k$$

Equation 2.21: Rotational restraint from supporting structural component

The value $c_{\vartheta M,k}$ represents the theoretical rotational restraint from bending stiffness I_a of the supporting structural component *a* assuming a rigid connection. Furthermore, the following is valid for Equation 2.21:

I_a	Moment of inertia of the supporting structural component in [cm⁴/cm]		
A	Span of the supporting structural component in [cm]		
K	Coefficient:	k = 2	for single-span and double-span beams, end-span beams
		k = 4	for continuous beam with three or more spans

For a noncontinuous rotational restraint (for example by purlins), the moment of inertia I_a of the supporting structural component is converted to a continuous support according to $\mathbf{I_a = I / e}$, where *e* is the distance of the supporting single-span beams (for example purlins).

If the supported beam can rotate only in one direction, the $c_{\vartheta M,k}$ value may be multiplied by factor 3.0 according to Equation 2.21. This is the case if, for example, a supported beam is part of the structure of an inclined roof.

Rotational restraint $c_{\vartheta A,k}$ from deformation of the connection

$c_{\vartheta A,k}$ represents the rotational restraint from deformation of the connection. For connections of single girders by means of bolts without slip (alternatively left and right from web of the cross-section to be stiffened), we can assume a rigid connection as approximation, that is, $c_{\vartheta A,k}$ is equal to infinity and is discarded in Equation 2.20.

For rotationally elastic support by means of trapezoidal sheeting, we obtain:

$$c_{\vartheta A,k} = \bar{c}_{\vartheta A,k} \left(\frac{b_1}{10} \right)^2 \quad \text{for} \ \frac{b_1}{10} \leq 1.25$$

$$c_{\vartheta A,k} = 1.25\, \bar{c}_{\vartheta A,k} \left(\frac{b_1}{10} \right) \ \text{for} \ 1.25 < \frac{b_1}{10} \leq 2.0$$

Width b_1 Width of the top flange of the supported beam in [cm]

Equation 2.22: Rotational restraint from deformation of the connection

The characteristic value for the joint stiffness $\bar{c}_{\vartheta A,k}$ of steel trapezoidal sheeting profiles is taken from Table 7 of DIN 18 800 Part 2. This table is included in the program.

If ratio $b_1/10 > 2.0$, the ratio in the equation above is limited to 2.0 in order to be on the safe side. According to OSTERRIEDER [8] (note in paragraph 4), is also possible to specify values for $\bar{c}_{\vartheta A,k}$ that are greater than those given by Table 7. This option is also available in the program.

If the coefficient $\bar{c}_{\vartheta A,k}$ is determined according to Table 7 and the trapezoidal sheeting profiles show thicknesses *t* greater than 0.75 mm, we obtain greater joint stiffnesses. Approximately, the table value may be increased by a factor as follows [15]:

$$\left(\frac{t_{prov}}{0{,}75} \right)^2 \qquad t_{prov} \text{ in [mm]}$$

Equation 2.23: Increase factor for thicknesses > 0.75

Rotational restraint $c_{\vartheta P,k}$ from cross-section deformation

$c_{\vartheta P,k}$ is the torsional restraint due to deformation of the supported beam section. It is obtained as follows [15].

$$c_{\vartheta P,k} = \frac{E}{4 \cdot (1-\mu^2)} \cdot \frac{1}{\frac{h_m}{s^3} + 0.5 \cdot \frac{b_1}{t_1^3}}$$

Where b_1, t_1 Width or thickness of the top flange of the supported beam in [cm]

S Web thickness of the supported beam in [cm]

h_m Distance of the flange's centerline in [cm]

μ Poisson's ratio of steel with the fixed value $\mu = 0.3$

Equation 2.24: Rotational restraint from distortional buckling

The determination of $c_{\vartheta P,k}$ according to Equation 2.24 in [15] necessarily requires that in the case of a noncontinuous rotational restraint, the concentrated loads from the supported structural component (transferred to the supported beam) reach only a maximum of 50 % of the limit loads for constructions without stiffeners (see for example [7]). Further values can be found in the manual RF-LTB [10] and in the commentary [15], page 169.

The continuous rotational springs according to Equation 2.20 ($c_{\vartheta,k} = c_{\vartheta,x}$) can be used as discrete rotational spring (single spring) if the continuous spring is multiplied by the corresponding influence width.

2.5.2 Translational Springs

For the frequent case of a beam span (for example frame beam, platform beam, or floor beam) stabilized by one or several bracings, the translational springs can be determined c_y according to PETERSEN [2] as follows:

$$c_y = \text{prov}\,S \cdot \frac{\pi^2}{L^2} \qquad \left[\frac{\text{kN/m}}{\text{m}}\right]$$

where prov S Part of the shear stiffness of a beam according to Equation 2.17 or Equation 2.18

L Length of bracing

Equation 2.25: Translational spring constant

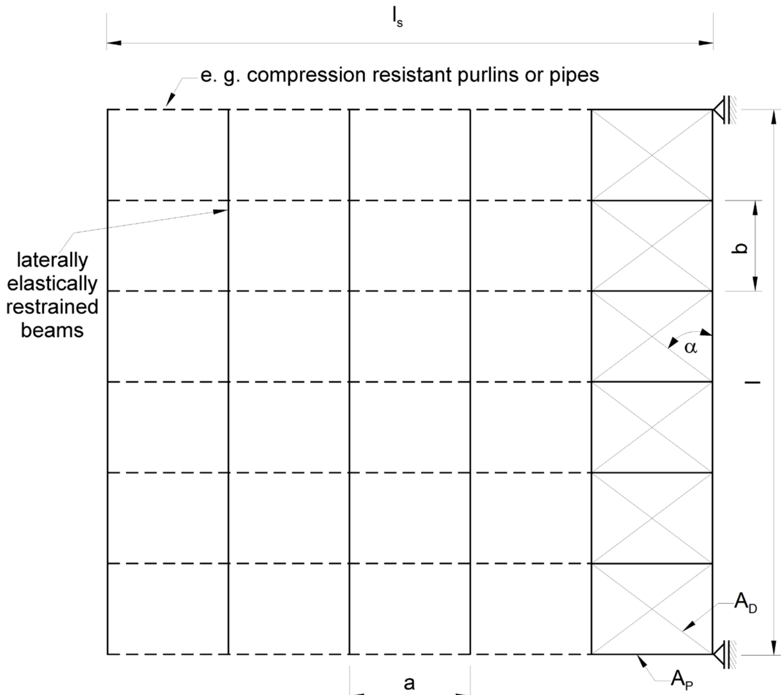


Figure 2.13: Frame beam with trapezoidal sheeting and bracings

The bracing should have a regular structure, because the equation for c_y is derived by a "uniform smearing" of the bracing over the length L.

The shear stiffnesses from bracing and trapezoidal sheeting may be added only if the beams that are to be restrained by laterally connected sections resistant to compression and the trapezoidal sheets above are connected to the bracing:

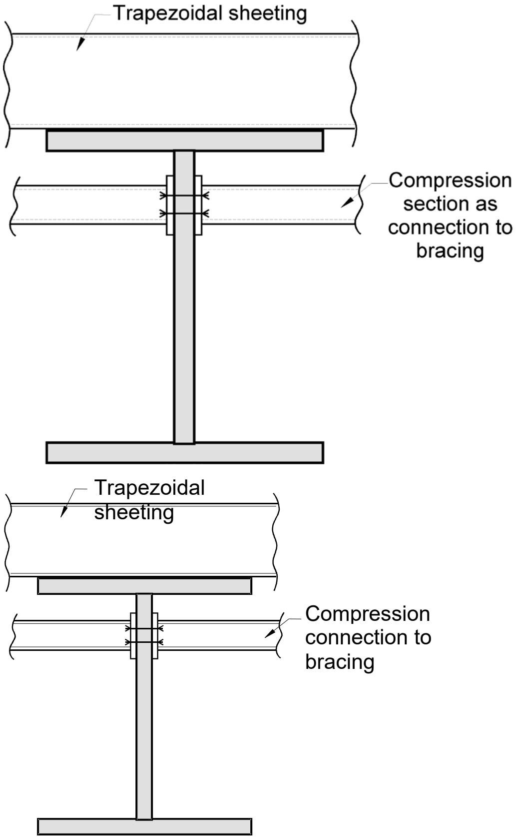


Figure 2.14: Stiffening by trapezoidal sheeting and compression sections

Thus, the trapezoidal sheeting and the bracing act as parallely placed springs that can be added up to a global spring. If the beams are connected only by supporting purlins with each other (which, conversely, are possibly supported by trapezoidal sheeting), then only the part S_R (see Equation 2.16) from the bracing is taken for *prov S*.

An example for the determination of a lateral translational spring c_x is given in PETERSEN [2] chapter 7.17.3. In [2] and in the Stahlbau Handbuch [17], chapter 3.2, you can find further information for the determination of equivalent stiffnesses.

2.5.3 Warp Springs

The restraint of the warping increases the torsional stiffness of a girder with a thin-walled open cross-section. This increase can be considered by discrete warp springs C_ω.

Warping restrained by an end plate [4], [10]

In this case, the warp spring is given according to Equation 2.29 as follows:

$$C_\omega = \frac{1}{3} \cdot G \cdot b \cdot h \cdot t^3$$

Equation 2.26: Warp spring by end plate

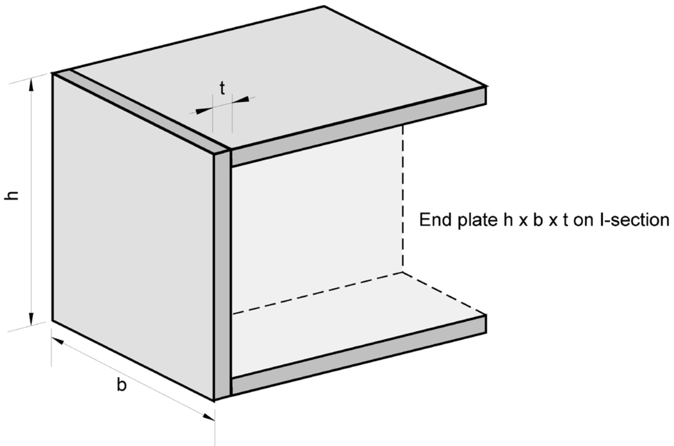


Figure 2.15: Warp spring from end plate

Warping restraint by a cantilevered portion [2]

The warp spring due to a cantilevered portion is determined acc. to the following equation:

$$C_\omega = G \cdot I_T \cdot \frac{1}{\lambda} \cdot \tanh(\lambda \cdot L_k)$$

where $\lambda = \sqrt{\frac{G \cdot I_T}{E \cdot I_\omega}}$

L_k Length of cantilevered portion

Equation 2.27: Warp spring from cantilevered portion

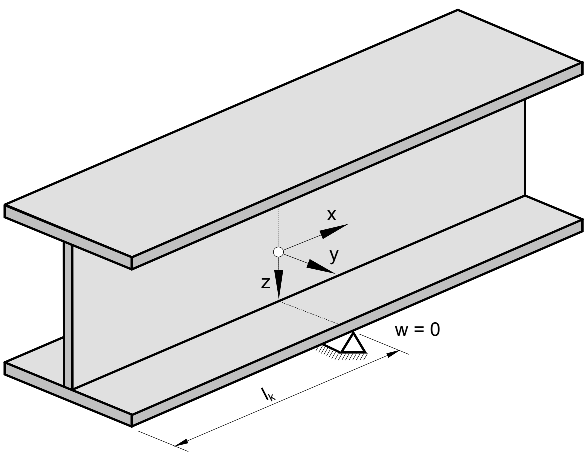


Figure 2.16: Warp spring from cantilevered portion

Warping restraint by means of a diaphragm resistant to torsion

The warp springs from the end plates or cantilevered portions results only in a relatively small restraint. The intended installation of cross-stiffenings resistant to torsion in the form of welded channel sections or angles is more effective[2]. This results in a closed box section about the z-axis (vertical axis)

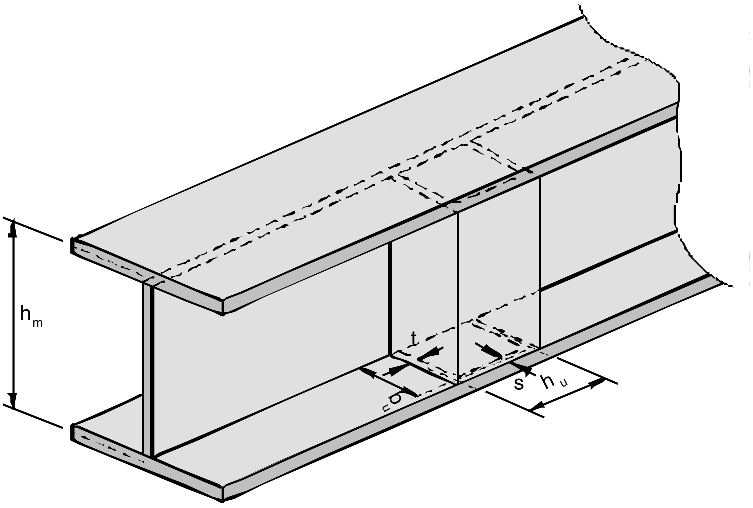


Figure 2.17: Warp spring from cross-stiffening

$$C_\omega = G \cdot h \cdot \frac{4 \cdot A_m^2}{\sum \frac{L_i}{t_i}} = G \cdot h \cdot \frac{4\left(b_u \cdot t_u\right)^2}{2\left(\frac{b_u}{t} + \frac{h_u}{s}\right)}$$

where A_m Area enclosed by the centerline

$\sum \frac{L_i}{t_i}$ Sum of the side lengths divided by the respective plate thickness

Equation 2.28: Warp spring from cross-stiffening

Warping restraint by a column connection

The warp spring C_ω for the frame beam can be calculated according to the basic expression:

$$C_\omega = \frac{1}{3} \cdot G \cdot b \cdot h_m \cdot t^3$$

where G Shear modulus

I_T Torsional moment of inertia for

- closed sections: $I_{T,Bredt} = \frac{4\,A_m^2}{\sum \frac{L_i}{t_i}}$

  A_m area of the section enclosed by the centerline

- open sections: $I_{T,St.Ven.} = \sum \frac{1}{3} L_i \cdot t_i^3 \left[1 - 0.63 \frac{t_i}{L_i} + 0.052 \left(\frac{t_i}{L_i} \right)^5 \right]$

  The expression in the brackets is a correction factor that takes into account the solidness of individual rectangular components (length L_i, thickness t_i). For thin-walled sections, this factor can be taken as 1.0.

h_m Distance of the flange centerlines

Equation 2.29: Warp spring from column joint

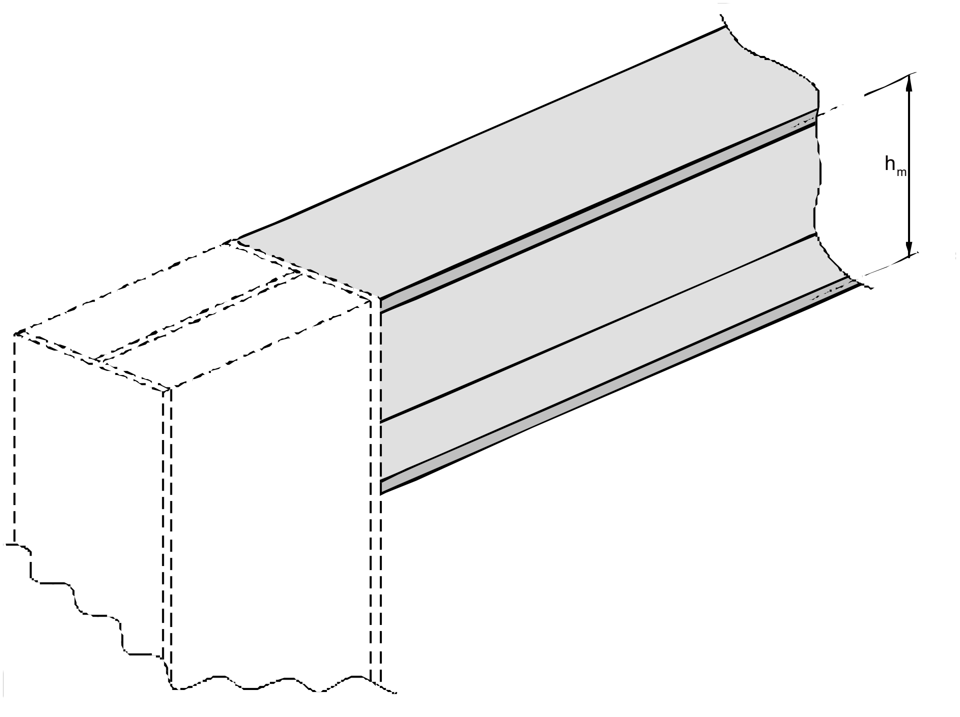


Figure 2.18: Warp spring from column connection

2.6 Analyses According to DIN 18800

The following chart shows the possible analyses that are supported by RF-FE-LTB and compliant with DIN 18800.

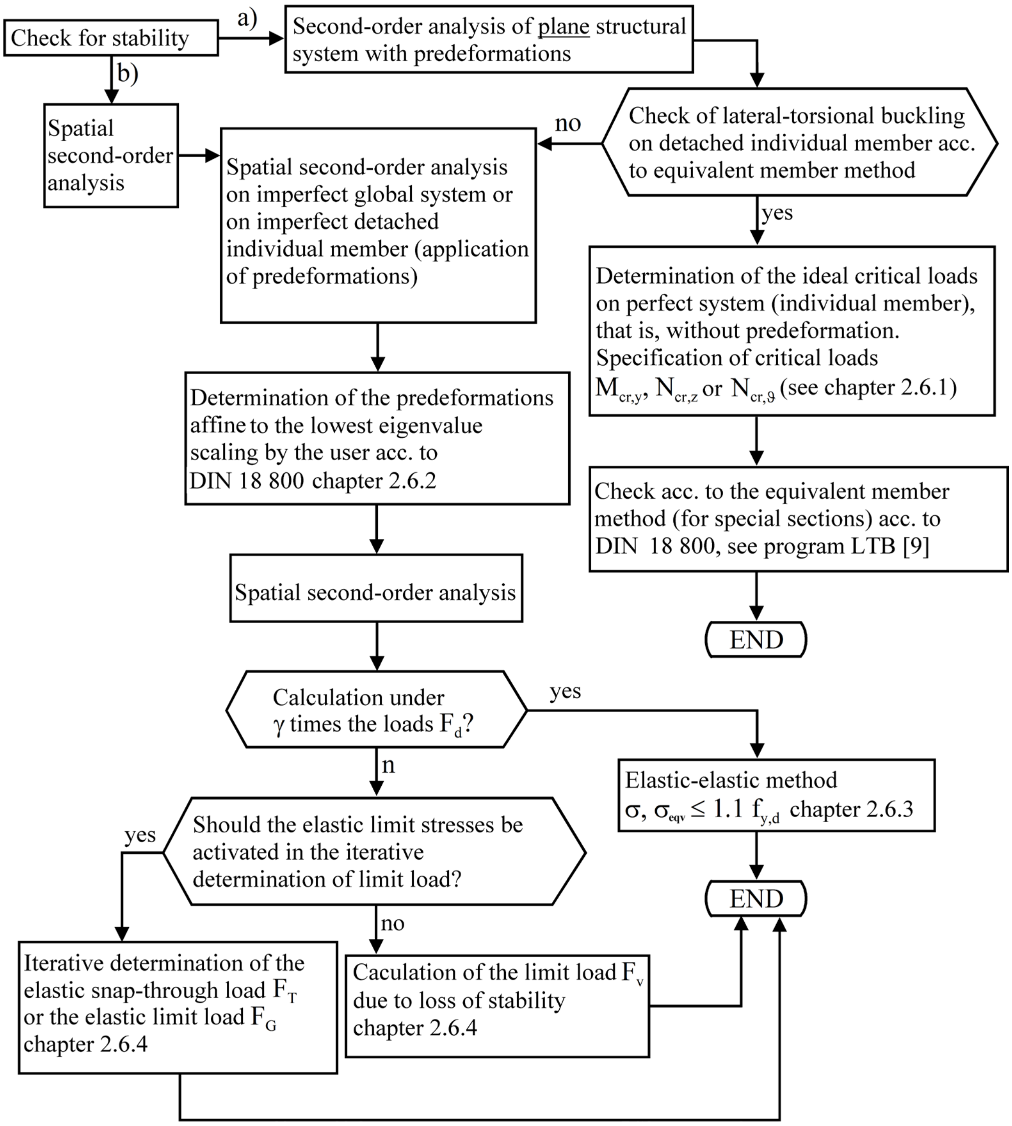


Figure 2.19: Analyses according to DIN 18800 with RF-FE-LTB

a) Plane second-order analysis

b) Spatial second-order Analysis

The calculation of the critical load factor on the global system according to the second-order analysis or the check of the elastic limit stresses under (γ times) design loads according to second-order analysis of the global structural system should always be preferred to the equivalent member method, as the real boundary or transition conditions are considered on the global structural system.

The ideal critical loads whose values are included in the equivalent member method can also be determined in RF-FE-LTB for the global system (that is, more precise than by means of analytical expressions that consider the boundary and transition conditions only approximately), but the critical loads can be qualified precisely (see example in chapter 2.6.1).

2.6.1 Equivalent Member Method

For simplification according to DIN 18800 Part 2, the analyses for lateral buckling and lateral-torsional buckling are carried out separately. As a rule, the analysis of the **lateral buckling** (see the following figure) is carried out in the plane of the structural system by calculating the plane structure according to second-order analysis as stress analysis under the design loads and by applying the initial deformations.

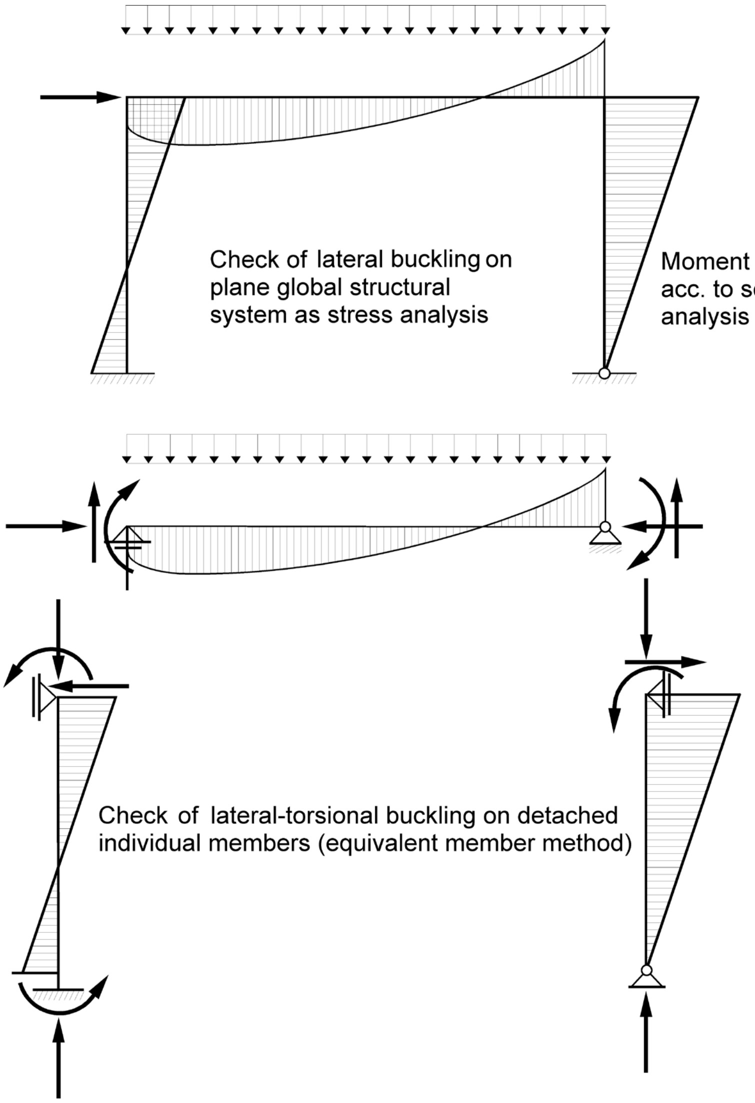


Figure 2.20: Analysis of a structure: flexural buckling in the plane and lateral-torsional buckling for an individual member

The analysis of the **lateral-torsional buckling** is performed on a notionally singled out individual member with the following boundary conditions and loads:

- **Loads**

  The individual member is loaded by the design loads and at the member ends by the internal forces determined on the global structural system.

- **Geometrical boundary conditions and elastic supports**

  The kinematic conditions that you can see when cutting out the individual member from the global system are to be specified as boundary conditions, especially those regarding the deflection perpendicular to the plane of the structural system and the torsional restraints. Elastic supports by adjacent structural components can be considered by means of equivalent springs according to chapter 2.5.

For an individual member defined in such a way, there are two possibilities for the lateral-torsional analysis (see also Figure 2.19):

2.6.1.1 Equivalent Member Analysis

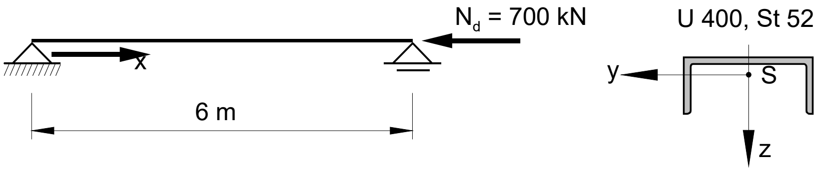
The simplified analysis according to DIN 18800 Part 2, elements (306), (307), (311), (320), and (323) is essentially limited to doubly symmetrical or monosymmetrical I-sections (see element (311), note 1). In addition to this, this verification format can be used only for specific loads that are based on the following ideal critical values:

$M_{cr,y}$ Elastic critical moment for lateral-torsional buckling according to elasticity theory for the sole action of moments M_y without axial force

$N_{cr,z}$ Axial force under the smallest critical buckling load according to elastic theory (the smallest load from buckling about the z-axis or lateral-torsional buckling about this axis, or torsional buckling, see for example manual for program RF-LTB [9].

In RF-FE-LTB, you can calculate these ideal critical loads also on the undeformed individual member (perfect system). The program always gives the smallest critical buckling load. In the equivalent member method, however, the values $M_{cr,y}$ and $N_{cr,z}$ are used. Hence, it is necessary to check whether the critical buckling load calculated by the program corresponds to these values (see also example 9.9 on page 117, where $N_{cr,y}$ is smaller than $N_{cr,\vartheta}$).

Example



Left support:	$u = v = w = \varphi_x = 0$
Right support:	$v = w = \varphi_x = 0$
Centric compression force:	$N_d = 700$ kN

Figure 2.21: Channel section with centric compression force

For lateral buckling, the following critical loads result:

$$N_{cr,z} = \frac{E \cdot I_z \cdot \pi^2}{L^2} = \frac{21{,}000 \cdot 20{,}350\, \pi^2}{800^2} = 6{,}590.3\,\text{kN}$$

$$N_{cr,y} = \frac{E \cdot I_y \cdot \pi^2}{L^2} = \frac{21{,}000 \cdot 846\, \pi^2}{800^2} = 274.0\,\text{kN}$$

Equation 2.30: Critical loads

The elastic torsional buckling load $N_{cr,\vartheta}$ according to [10] is determined according to the following equation.

$$N_{cr,\vartheta} = \frac{E \cdot I_z \cdot \pi^2}{\lambda_{eqv}^2 \cdot i_z^2}$$

Equation 2.31: Elastic torsional buckling load

To solve Equation 2.31, the equivalent slenderness λ_{eqv} is necessary.

$$\lambda_{eqv}^2 = \left(\frac{L}{i_z}\right)^2 \cdot \frac{c^2 + i_M^2}{2 \cdot c^2} \cdot \left(1 + \sqrt{\frac{4 \cdot c^2 \cdot i_p^2}{\left(c^2 + i_M^2\right)^2}}\right)$$

$$= \left(\frac{800}{14.9}\right)^2 \cdot \frac{10.54^2 + 16.04^2}{2 \cdot 10.54^2} \cdot \left(1 + \sqrt{\frac{4 \cdot 10.54^2 \cdot 15.21^2}{\left(10.54^2 + 16.04^2\right)^2}}\right) = 8939.5$$

$$\Rightarrow \lambda_{eqv} = 94.5$$

Equation 2.32: Equivalent slenderness

The slenderness values in Equation 2.32 are determined as follows:

$$i_p = \sqrt{i_y^2 + i_z^2} = \sqrt{14.9^2 + 3.04^2} = 15.21\,\text{cm}$$

$$z_M = -5.11\,\text{cm}$$

$$i_M = \sqrt{i_y^2 + z_M^2} = \sqrt{15.21^2 + (-5.11)^2} = 16.04\,\text{cm}$$

Equation 2.33: Slenderness values

c is the so-called twisting radius.

$$c^2 = \frac{I_\omega}{I_z} + \frac{L^2}{\pi^2} \cdot \frac{G \cdot I_T}{E \cdot I_z} = \frac{221{,}000}{20{,}350} + \frac{800^2}{\pi^2} \cdot \frac{8{,}100 \cdot 81.6}{21{,}000 \cdot 20350} = 111.15\,\text{cm}^2$$

$$\Rightarrow c = 10.54\,\text{cm}$$

Equation 2.34: Twisting radius

Thus, it is possible to compute the elastic torsional buckling load according to Equation 2.31.

$$N_{cr,\vartheta} = \frac{21{,}000 \cdot 20{,}350 \cdot \pi^2}{94.5^2 \cdot 14.9^2} = 2{,}125.2\,\text{kN}$$

Equation 2.35: Elastic torsional buckling load

$N_{cr,y}$ is the lateral buckling load about the y-axis in the plane of the structural system, $N_{cr,z}$ is the lateral buckling load for buckling about the z-axis (that is, deflection in the direction of the y-axis). $N_{cr,\vartheta}$ represents the torsional buckling load: the cross-section in y-direction is displaced and at the same time rotated about the longitudinal x-axis. Since $N_{cr,\vartheta}$ is smaller than $N_{cr,z}$, this value is governing for the check of the equivalent member according to element (306) and (304).

Buckling curve c $\rightarrow$ $\alpha = 0.49$

$$\bar{\lambda}_k = \frac{\lambda_k}{\lambda_a} = \frac{94.5}{92.9} = 1.017$$

$$k = 0.5 \cdot \left[1 + 0.49 \cdot (1.017 - 0.2) + 1.017^2\right] = 1.217$$

$$\kappa_z = \frac{1}{1.217 + \sqrt{1.217^2 - 1.017^2}} = 0.530$$

Equation 2.36: Reduction factor

$$N_{pl,d} = \frac{36}{1.1} \cdot 91.5 = 2994.5\,kN$$

Equation 2.37: Plastic axial force

Check:

$$\frac{N_d}{\kappa_z \cdot N_{pl,d}} = \frac{700}{0.53 \cdot 2994.5} = 0.44 \le 1.0$$

Equation 2.38: Plastic axial force

Thus, the check against lateral-torsional buckling would be fulfilled, while the check for lateral buckling about the y-axis would fail because N_d = 700 kN > $N_{cr,y}$ = 274 kN.

The program RF-FE-LTB gives the smallest critical load factor $N_{cr,y}$ = 274 kN. To determine the governing critical load for deflection in y-direction, you might have to apply a mid-span support in z-direction ($w_{x=l/2} = 0$). The resulting values are shown in the following table:

	N_{cr}	**analytical**
RF-FE-LTB $w_{l/2} \neq 0$	273.8 kN	$N_{cr,y}$ = 274.0 kN
RF-FE-LTB $w_{l/2} = 0$	2451.0 kN	$N_{cr,\vartheta}$ = 2125.2 kN

Table 2.4: Critical buckling loads

The ideal values $M_{cr,y}$ (without axial force!) and $N_{cr,z}$ or $N_{cr,\vartheta}$ (under the sole action of a centric axial force) must be determined separately with RF-FE-LTB, that is, in two calculation runs at the respective perfect system.

This example shows the problem of the equivalent member method. A better method is the second possibility for the individual member presented in the following chapter.

2.6.1.2 Ultimate Limit State Analysis for Spatially Imperfect Individual Member

As an alternative design method, you can use the calculation of the spatially imperfect individual member according to second-order analysis of elasticity according to element (121) together with element (201) in RF-FE-LTB.

In case of a stable equilibrium, the maximum equivalent stress must be smaller than the limit stress $f_{y,d}$. In small portions, the equivalent stress may exceed the limit stress $f_{y,d}$ by 10% (see chapter 2.6.3, page 38). The imperfections are to be applied in compliance with DIN 18800 Part 2 (see the following chapter 2.6.2). Thus, the procedure corresponds to the elastic-elastic method.

2.6.2 Determination of Initial Deformations

According to DIN 18800 Part 2 for the second-order analysis for considering geometrical and structural imperfections, geometrical equivalent imperfections are to be specified. For unbraced systems, these are usually **initial sway imperfections** caused by angles of bar rotation. For braced systems, these are **initial bow imperfections** (initial camber) in the form of sinusoidal or parabolic half-waves.

The shape of the initial deformation should be taken affine to the lowest buckling or lateral-torsional buckling eigenmode, see element (202). According to the commentary to DIN 18800 [15], it is sufficient to select the initial deformation in such a way that a sufficiently great component of the lowest eigenmode is included. Thus, the program aims at ensuring that the load-deformation-curve tends towards the first eigenvalue.

To this end, RF-FE-LTB calculates the eigenmode belonging to the smallest eigenvalue (preliminary eigenvalue analysis) and chooses it as deformation mode (imperfection shape). The deformation modes in the direction of the main axes y and z are analyzed and the deflection direction belonging to the smallest eigenvalue is chosen (initial deformation in y-direction vv, in z-direction wv). Taking into account these initial deformations, bending moments result on the uniformly loaded beam about both cross-section axes as well as torsional moments.

Next, the imperfection is considered by a user-defined scaling of the eigenmode. For this purpose, the following menu options are available in the program (see chapter 3.8.3, page 75):

- Direct numerical specification of the maximum camber rise of the initial deformation by a graphical representation of the eigenvector and the location of the maximum displacement.
- Calculation of the *camber rise* (initial bow imperfection) according to element (204), Table 3 with the user-defined governing buckling curve and the reference length; furthermore, the calculation of the *initial sway imperfection* according to element (205) considering the reduction factors r_1 and r_2. In determining the initial sway imperfection, the user specifies the required data (like reference length and number *n* of the mutually independent causes for the initial sway imperfections of members).

The initial deformations to be taken according to [3] chapter 2.2 and 2.3 may be reduced under certain conditions. In RF-FE-LTB, you have the following options:

Reduction 1) according to element (201)

For the elastic-elastic method, camber rises of the bow imperfections or the initial sway imperfections φ_0, which depend on the buckling curve, may be reduced by the factor $^2/_3$.

Reduction 2) according to element (202)

In the lateral-torsional buckling analysis, the amplitudes of the initial bow imperfections out of the principal stress plane may be reduced by another 50 %.

The reduction option 2) is not without its problems, as this reduction may be carried out only if the initial deformation mode for the lateral-torsional buckling belongs to the smallest eigenmode. This effect is illustrated in the following Figure 2.22.

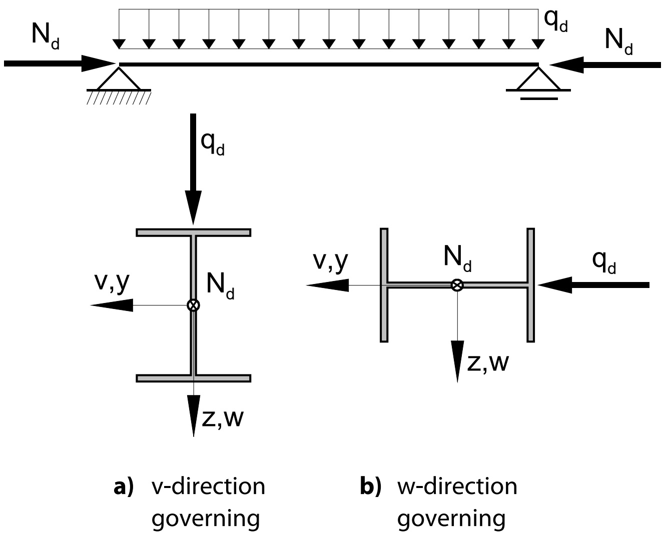


Figure 2.22: Governing initial deformation for the reduction according to DIN 18800 Part 2, element (202).

RF-FE-LTB analyzes both principal axes directions y and z. This is because depending on the spatial arrangement of the member and the load constellation, either the w- or the v-direction can be governing for the lateral-torsional buckling.

In case a), v corresponds to the direction of deflection for the lateral-torsional buckling. If the lowest eigenmode belongs to this direction, a reduction by 50 % is allowed.

In case b), the displacement mode w can be imagined as the initial deformation mode in the direction of v according to DIN 18800 Part 2. Thus, a reduction by 50 % may be carried out only if the eigenmode of the lowest eigenvalue (which is to be scaled by the user) runs in the w-direction.

To learn more about the determination of the initial deformation, see chapter 3.8.3.

2.6.3 Second-Order Ultimate Limit State Analysis

RF-FE-LTB determines the internal forces according to the second-order analysis taking into account spatial initial deformations (see chapter 2.6.2).

For the elastic-elastic method: according to DIN 18800 Part 2, element (121), it is necessary to verify that under the design actions (γ_F times the loads) the following conditions are met

$$\max \sigma_x \le f_{y,d}\,; \quad \max \tau \le \frac{1}{3} f_{y,d}$$

$$\max \sigma_v \le f_{y,d} = \frac{f_{y,k}}{\gamma_M}$$

Equation 2.39: Design conditions for stresses

According to DIN 18800 Part 1, element (749), the equivalent stress may exceed the limit stress σ_{eqv} by 10 % in <u>isolated</u> points.

$$\max \sigma_{eqv} \le 1.1\, f_{y,d}$$

Equation 2.40: Allowable overstressing

For members subjected to axial force and bending, an "isolated point" can be assumed if the following is valid at the same time:

$$\left|\frac{N}{A}+\frac{M_y}{I_y}z\right| \le 0.8\, f_{y,d} \quad \text{and}$$

$$\left|\frac{N}{A}+\frac{M_z}{I_z}y\right| \le 0.8\, f_{y,d}$$

Equation 2.41: Allowance for locally limited plastification

As a rule, the maximum stress occurs on a cross-section edge at which the shear stresses from the shear forces become zero. Thus, the analyses are reduced to the analysis of the normal stresses.

2.6.4 Limit Loads F_T or F_G

RF-FE-LTB also offers the option to carry out the ultimate limit state analysis by comparing the limit loads (ultimate capacity) with the design loads F_d. Only the following analysis is of practical importance:

$$F_T \ge F_d \quad \text{or} \quad F_G \ge F_d$$

Equation 2.42: Design conditions for ultimate loads

The program determines the load level $\mathbf{F_T}$ or $\mathbf{F_G}$ by an iterative load increase (see Figure 2.1, page 10).

F_T Ultimate capacity due to loss of stability (limit load) on the imperfect system not exceeding the elastic limit stress

F_G Elastic Limit Load on imperfect system (all shear, normal, and equivalent stresses are less or equal to the respective elastic limit stress)

RF-FE-LTB additionally allows you to calculate the limit load $\mathbf{F_V}$ due to initial deformations without observing the elastic limit stress. This ultimate load, however, is only of theoretical interest.

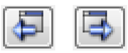
3. Input Data

After you start the add-on module, a new window appears. The navigator on the left shows the available module windows. Above the navigator, you find a drop-down list with the design cases (see chapter 8.1, page 102).

You must define the design-relevant data in several input windows. When you open RF-FE-LTB for the first time, the following RFEM data is imported:

- Sets of members
- Load cases and load combinations
- Materials
- Cross-sections

The sets of members are considered as notionally singled out from the model, that is, they are not coupled to the members in the RFEM model. Changes of cross-sections and the geometry are automatically compared with RF-FE-LTB. Imperfections from RFEM are not imported.



To go to a particular module window, click the according entry in the navigator. To go to the previous or next module window, use the buttons shown on the left. Alternatively, you can also press the function keys [F2] (forward) and [F3] (backward) to browse through the module windows.

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

3.1 General Data

In the 1.1 *General Data* window, you specify the sets of members and actions for the design.

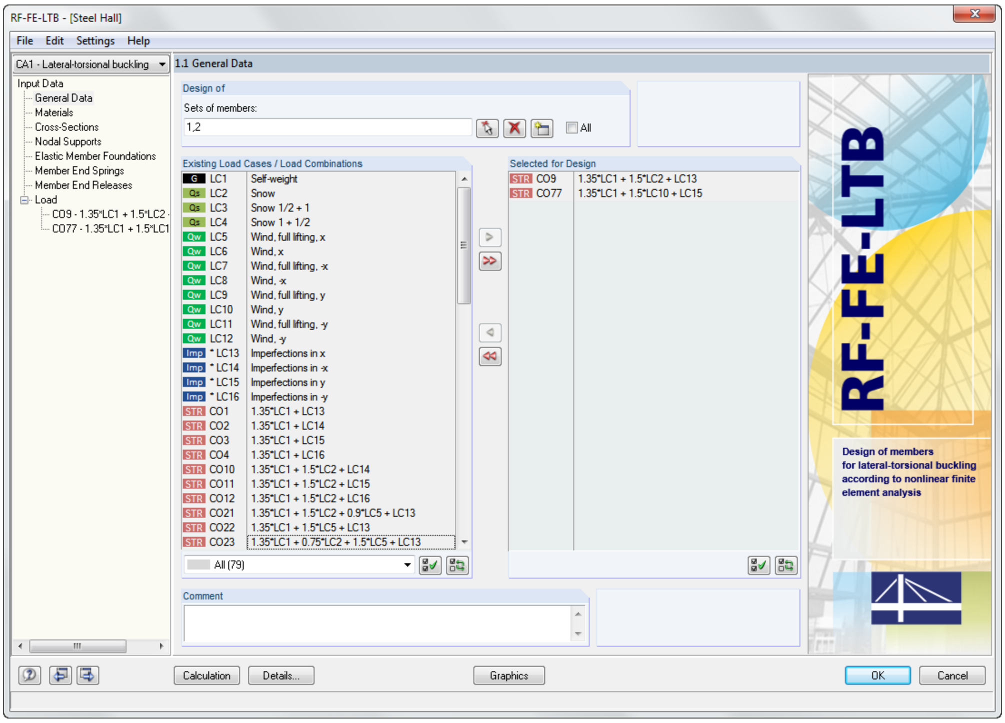


Figure 3.1: Window 1.1 *General Data*

Design of

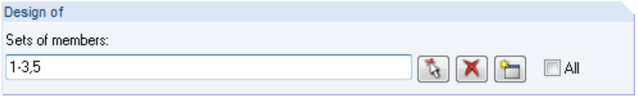


Figure 3.2: Analysis of sets of members

Only *sets of members* of the 'continuous members' type can be analyzed. If sets of members are already defined in RFEM, it is possible to enter their numbers in the list or to select them graphically in the RFEM work window upon clicking [↖]. By selecting the *All* check box, you can select all sets of members available in the model for the design.

The design is only possible for the set of members type 'continuous members' (with connected members that do not branch out). A set of members of the type 'group of members' leads to an error message before the calculation.

Figure 3.3: Error message when designing a group of members

To define a new set of members, click [New]. The dialog box known from RFEM appears where you specify the parameters of the set of members.

During the design of the set of members, the members are analyzed as cut out from the system. Here, you must consider the boundary conditions of a built-up column or a complete frame as a whole. This is done in the other input windows of RF-FE-LTB.

Existing Load Cases and Combinations

This section lists all load cases and load combinations created in RFEM.

To transfer the selected entries to the *To Design* list on the right, click [▶]. Alternatively, you can double-click the relevant entry. To transfer the entire list to the right, click [▶▶].

To make a multiple selection, select the load cases while pressing [Ctrl], as common in Windows. Thus, you can transfer several load cases at once.

If a load case is marked by an asterisk (*), for example LC 13 in Figure 3.1, it is not possible to design it: This is a load case without load data or an imperfection load case. When you try to transfer such a load case, an according warning appears.

Result combinations are not available for selection, because unambiguous internal forces must be available for the analysis. Result combinations, however, have two values for each location: maximum and minimum.

Below the list, you find a drop-down list with filter options. They can help you to assign the entries sorted by load cases, load combinations, or action categories. The buttons have the following functions:

[Select all]	Select all load cases in the list.
[Reverse]	Reverse the selection of the load cases.

Table 3.1: Buttons in window 1.1 *General Data*

To Design

The section on the right contains the load cases and load combinations selected for the design. To remove selected items from the list, click [◀] or double-click them. To empty the entire list, that is, transfer it to the right, click [◀◀].

By transferring items to the *To Design* list, the loads of the load cases and load combinations are automatically entered in the load module windows 2.1 through 2.3. You can edit the load parameters individually and, if necessary, extend them (see chapter 3.8, page 69). In the navigator of RF-FE-LTB, the selected load cases and load combinations are listed under the entry *Load*.

Comment

Figure 3.4: User-defined comment

This input field is available for user-defined comments, for example the current analysis case.

3.2 Materials

This window is subdivided into two parts. The upper part shows all materials specified in RFEM. The *Material Properties* section shows the properties of the current material, that is, the material whose row is selected in the section above.

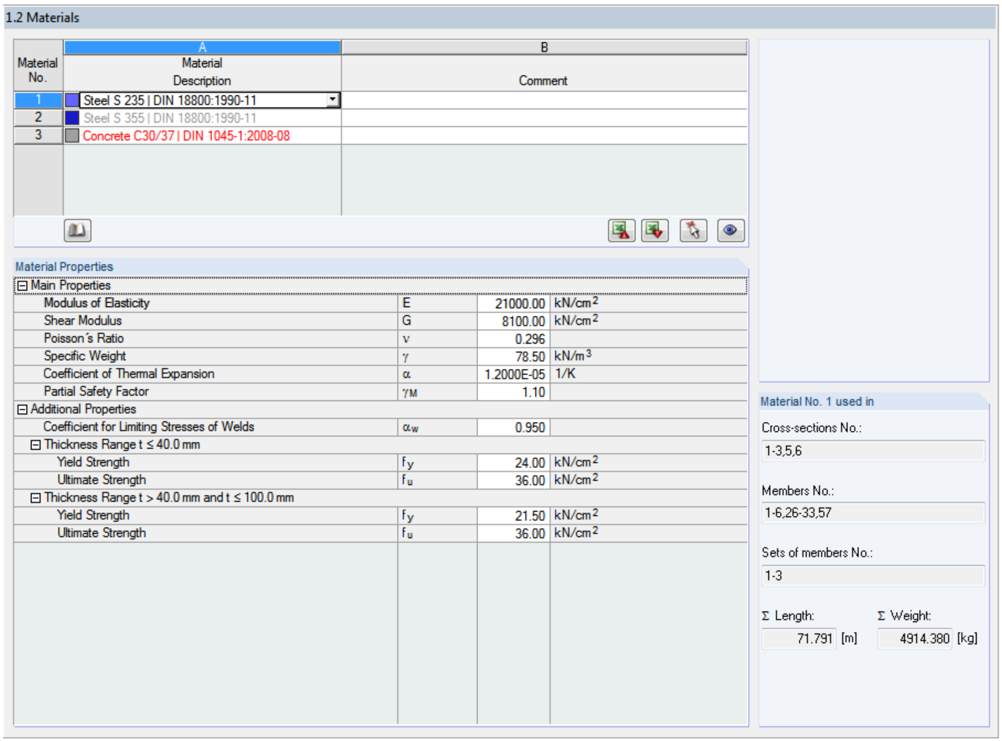


Figure 3.5: Window 1.2 *Materials*

Materials not used in the design appear dimmed. Invalid materials are highlighted in read, changed materials in blue.

Chapter 4.3 of the RFEM manual describes the material properties that are used for the determination of internal forces (*Main Properties*). The global material library also shows the properties of the materials required for the design. These values are preset (*Additional Properties*).

To adjust the units and decimal places of the properties and stresses, select **Settings** → **Units and Decimal Places** (see chapter 8.2, page 104).

Material Description

The materials defined in RFEM are preset, but you can always change them. Click the material in column A to activate this field. Then, click [▼] or press [F7] to open the material list.

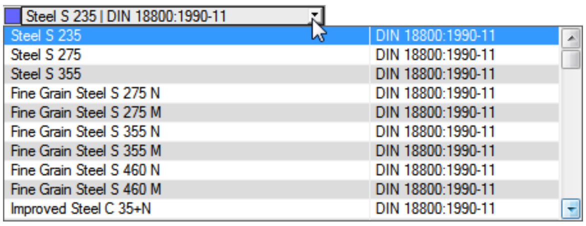


Figure 3.6: List of materials

In compliance with the design concept of the Standard [8], you can select only the materials of the "Steel" category.

After the transfer of the material, the design-relevant *Material Properties* are updated.

If you change the Material Description manually and the new entry is listed in the material library, RF-FE-LTB also imports the material properties.

The material properties cannot be edited in RF-FE-LTB on principle.

Material Library

The library contains a large number of predefined materials. To open this database, click

Edit → Material Library

or use the button shown on the left.

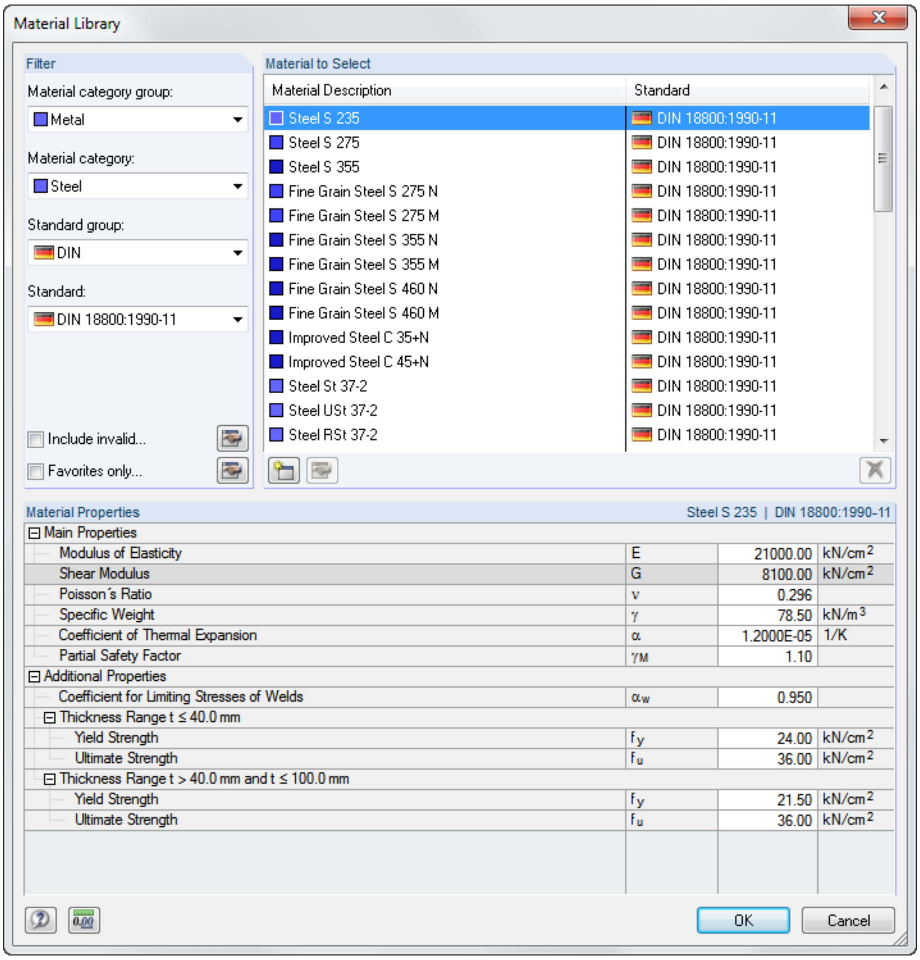


Figure 3.7: Dialog box *Material Library*

In the *Filter* section, the material category *Steel* is predefined. You can select the desired material grade in the *Material to Select* list. In the table below, you can check the various properties of the selected material.

You can also select materials of the category *Cast Iron*, *Aluminum*, *Stainless Steel*, etc., even though these materials are not covered by the design concept of the DIN 18800. RF-FE-LTB can perform stability analyses on the basis of an FEM analysis that are not subject to the limitations of the equivalent member method.

OK

To transfer the selected material to window 1.2 of RF-FE-LTB, click [OK] or [↵].

Chapter 4.3 of the RFEM manual describes how to add, filter, or sort the materials.

3.3 Cross-Sections

This window manages the cross-sections used for the design.

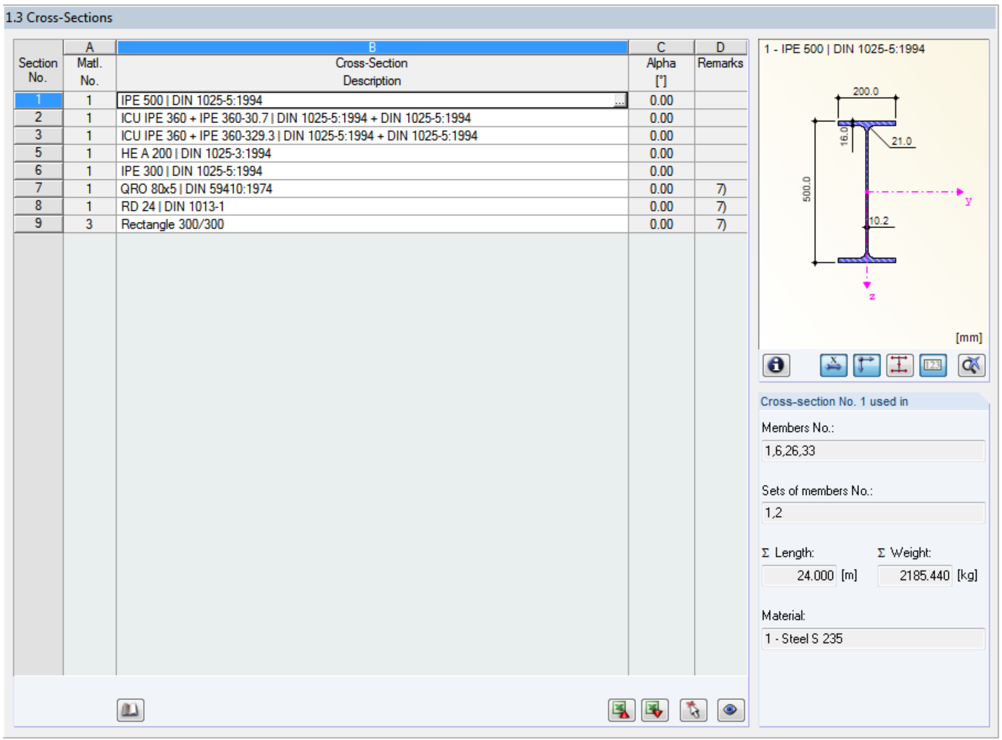


Figure 3.8: Window 1.3 *Cross-Section*

Cross-Section Description

The cross-sections defined in RFEM are preset together with their according material numbers.

To change a cross-section, click the entry in column B, thus activating the field. To open the cross-section table (series) of the currently selected input field, click [Library] or [...] in the input field, or press [F7] (see the following figure).

In this dialog box, you can select another cross-section or a different cross-section table. If you want to use a completely different cross-section category, click [Back to Cross-Section Library] to access the general cross-section library.

Chapter 4.13 of the RFEM manual describes how to select cross-sections in the library.

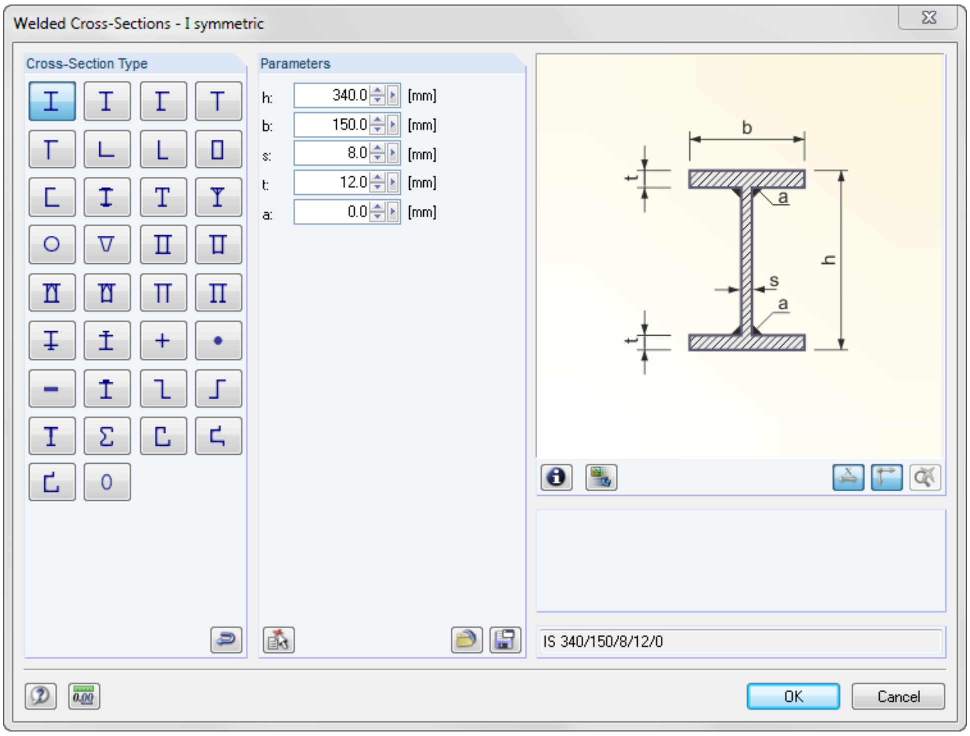
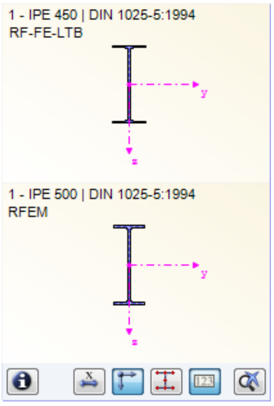


Figure 3.9: IS-section table in the cross-section library



You can also enter the new cross-section description directly in the input field. If the entry is stored in the database, RF-FE-LTB imports the RF-FE-LTB cross-section properties.

A modified cross-section is highlighted in blue.

If different cross-sections are available in RF-FE-LTB and in RFEM, the graphic on the right of the module window shows both cross-sections.

The lateral-torsional buckling analysis according to DIN 18 800 Part 2, element (323) covers all monosymmetrical and doubly symmetrical I-shaped cross-sections. Moreover, RF-FE-LTB can analyze all cross-sections in the library except for the parametric solid cross-section and SHAPE-MASSIVE cross-sections (see also chapter 2.1.1, page 11). The design also covers the analysis of SHAPE-THIN sections.

Alpha

For control, this column shows the rotation angle of principal axes α for each section.

Remarks

In this column, you may see notes in the form of footers which are described in more detail at the bottom of the cross-section list.

Member with tapered cross-section

For tapered members with different cross-sections at the start and end of the member, both cross-section numbers are shown in two rows according to the definition in RFEM.

RF-FE-LTB also designs tapered members if there is the same number of stress points for the start and end cross-section. The normal stresses are computed from the second moments of area and the centroid distances of the stress points. If there is a different number of stress points for the cross-section at the start and the end of the tapered member, the intermediate values cannot be interpolated. The calculation is therefore impossible in RFEM or RF-FE-LTB.

You can also graphically check the stress-points of the cross-section including the numbering: Select the cross-section in window 1.3 and click [Info]. The dialog shown in Figure 3.10 appears.

To carry out a successful design of a taper, you therefore have to achieve an equal amount of stress points. To do this, you can, for example, model the cross-section at the end of the tapered member as a copy of the cross-section at the start and only modify the geometric parameters. It might also be necessary to model both cross-sections as parameterized ('welded') sections. For tapered members, the parameterized tapered [IVU]-*I-Section Plus Lower Flange* is particularly well suited.

Info About Cross-Section

In the *Info About Cross-Section* dialog box, you can view the cross-section properties, stress points, and c/t cross-section parts.

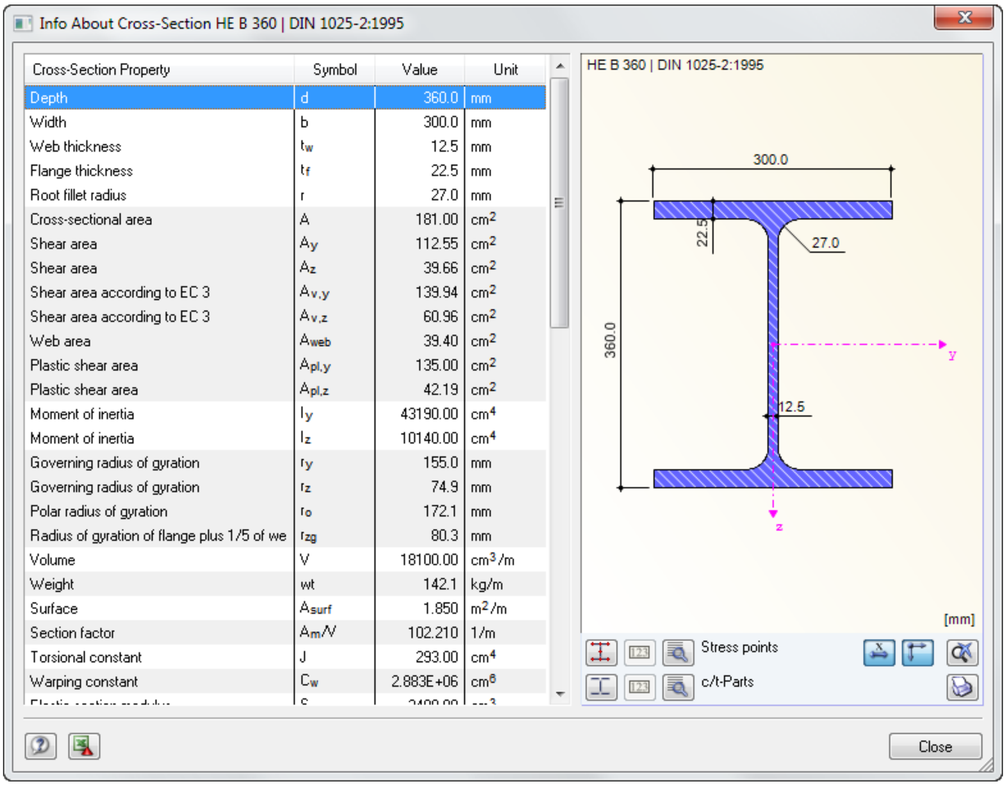


Figure 3.10: Dialog box *Info About Cross-Section*

The current cross-section is shown in the right part of the dialog box.

The buttons below the graphic have the following functions:

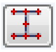
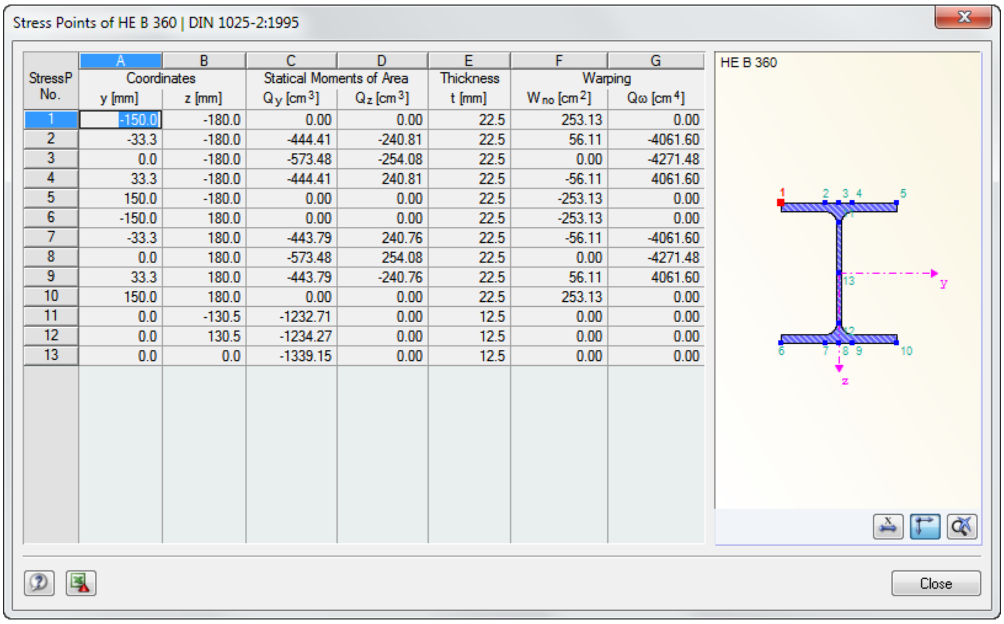
Button	Function
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the stress points or c/t-parts
	Displays the stress points or c/t-parts (see Figure 3.11)
	Displays or hides the dimensioning of the cross-section
	Displays or hides the principal axes of the cross-section
	Returns to the full view of the cross-section

Table 3.2: Buttons of the cross-section graphic

To see specific information about stress points (stress point distances, statical moments, warping ordinates, etc.), and c/t-parts, click [Details].



StressP No.	A Coordinates y [mm]	B z [mm]	C Statical Moments of Area Q_y [cm³]	D Q_z [cm³]	E Thickness t [mm]	F Warping W_{no} [cm²]	G Q_ω [cm⁴]
1	-150.0	-180.0	0.00	0.00	22.5	253.13	0.00
2	-33.3	-180.0	-444.41	-240.81	22.5	56.11	-4061.60
3	0.0	-180.0	-573.48	-254.08	22.5	0.00	-4271.48
4	33.3	-180.0	-444.41	240.81	22.5	-56.11	4061.60
5	150.0	-180.0	0.00	0.00	22.5	-253.13	0.00
6	-150.0	180.0	0.00	0.00	22.5	-253.13	0.00
7	-33.3	180.0	-443.79	240.76	22.5	-56.11	-4061.60
8	0.0	180.0	-573.48	254.08	22.5	0.00	-4271.48
9	33.3	180.0	-443.79	-240.76	22.5	56.11	4061.60
10	150.0	180.0	0.00	0.00	22.5	253.13	0.00
11	0.0	-130.5	-1232.71	0.00	12.5	0.00	0.00
12	0.0	130.5	-1234.27	0.00	12.5	0.00	0.00
13	0.0	0.0	-1339.15	0.00	12.5	0.00	0.00

Figure 3.11: Dialog box *Stress Points of HE B 360*

3.4 Nodal Supports

In window 1.4, you specify the support conditions on the member nodes of the set of members notionally singled out from the system. The nodal supports defined in RFEM are preset; if necessary, they can be adjusted.

You can define additional supports, for example, to represent the lateral restraint by an eaves purlin in the spatial model of RFEM. If this support is missing in the model of the notionally singled out set of members, instabilities become possible.

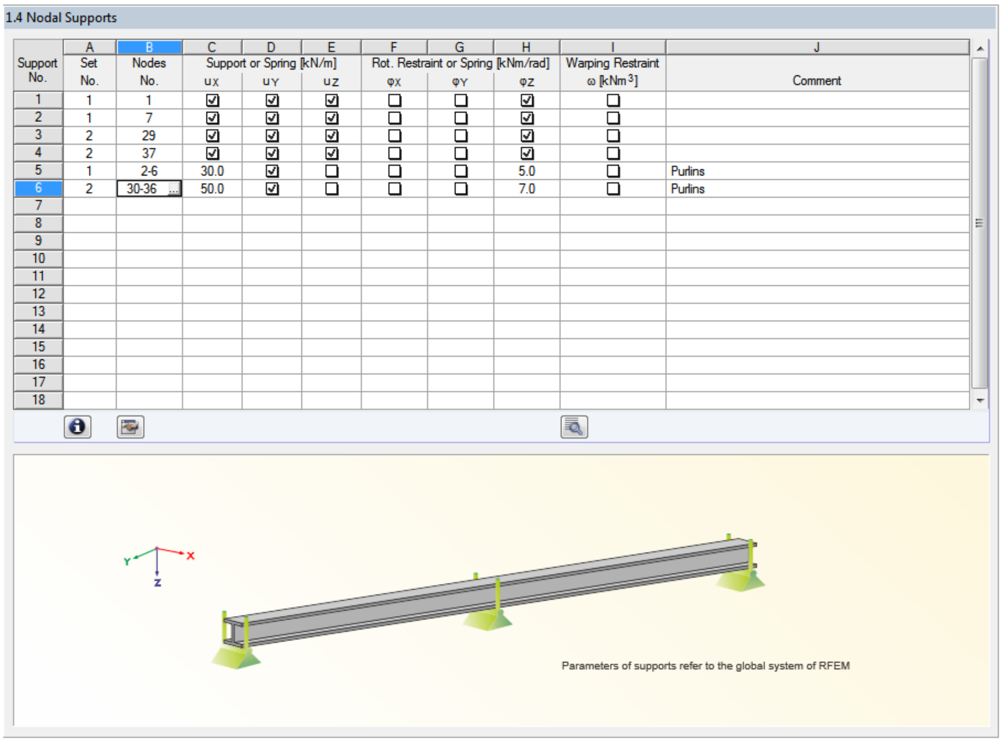
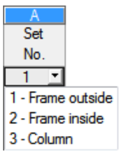


Figure 3.12: Window 1.4 *Nodal Supports*

The supports and springs are centrically related to the global coordinate system of RFEM (see chapter 2.2.3, page 15).

The stabilizing effect of objects connected along the set of members can be considered in the following window 1.5 *Elastic Member Foundations* (see chapter 3.5, page 55). There, the translational and rotational spring constants of purlins, bracings, and trapezoidal sheeting can be represented with eccentricities and as continuous supports.

Set No.



In this column, you specify the set of members for which the support conditions apply.

To insert an additional nodal support, place the cursor in a free cell of this column. Next, enter the number of the set of members or select it in the list. Then, you can specify the supported nodes in the *Nodes* column.

Nodes No.

You can specify the nodes with support properties individually or as list. Alternatively, you can also click [...] (see Figure 3.12) to graphically select them in the RFEM work window.

Support / restraint or spring

In the columns C to H, you must enter the support conditions of the selected nodes. To activate or deactivate degrees of freedom of the supports or restraints, select the respective check boxes. Alternatively, you can manually enter the constants of the translational or rotational springs.

To adjust the support conditions, you can also click [Edit] at the end of column B. The following dialog box appears.

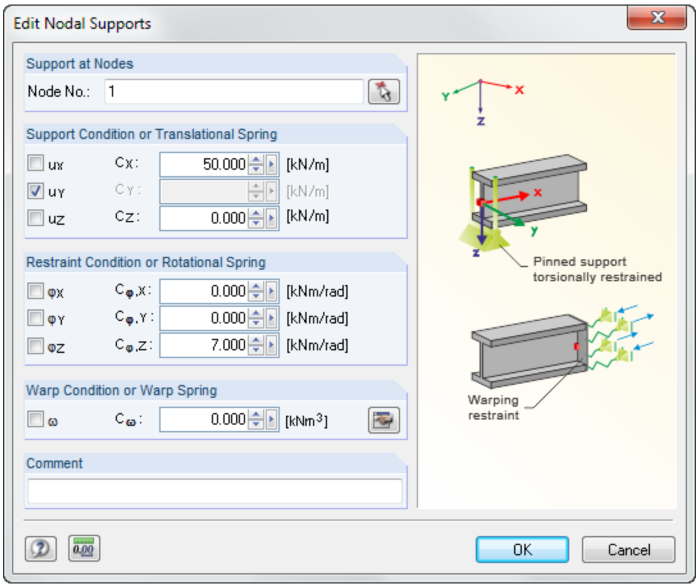


Figure 3.13: Dialog box *Edit Nodal Supports*

Warping Restraint

Column I manages the warping parameters of the supported nodes. You can specify a full or no warping restraint and enter the constants of warp springs manually.

The restraint of the warping increases the torsional stiffness of a beam with an open thin-walled cross-section. For the theoretical descriptions to determine warp springs, see chapter 2.5.3, page 29.

By clicking [Edit] at the bottom of column I, you can easily determine the warping constants resulting from the geometrical conditions. RF-FE-LTB calculates the warp springs for the following types of stiffeners:

- End plate (→ chapter 2.5.3, page 29)
- Channel section (→ chapter 2.5.3, page 30)
- Angle (→ chapter 2.5.3, page 30)
- Connected column (→ chapter 2.5.3, page 31)
- Cantilevered portion (→ chapter 2.5.3, page 29)

Warp spring from end plate

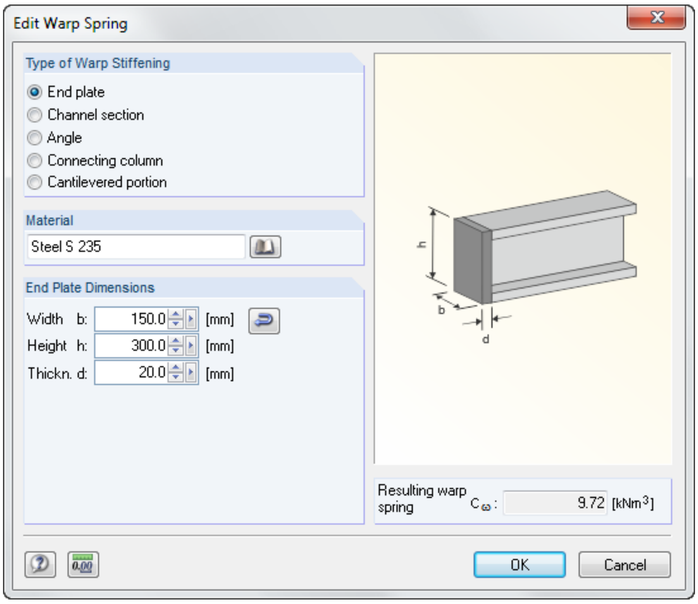


Figure 3.14: Dialog box *Edit Warp Spring*, type *End plate*

You can change the predefined *Material* of the end plate by using the material library.

Furthermore, you can manually enter the geometry of the *End Plate Dimensions*. To access the cross-section connected to the nodal support to import its width and depth to the dialog box, click [Import dimensions of cross-section].

The *Resulting warp spring* is shown in the lower-right corner of the dialog box.

Warp spring from channel section

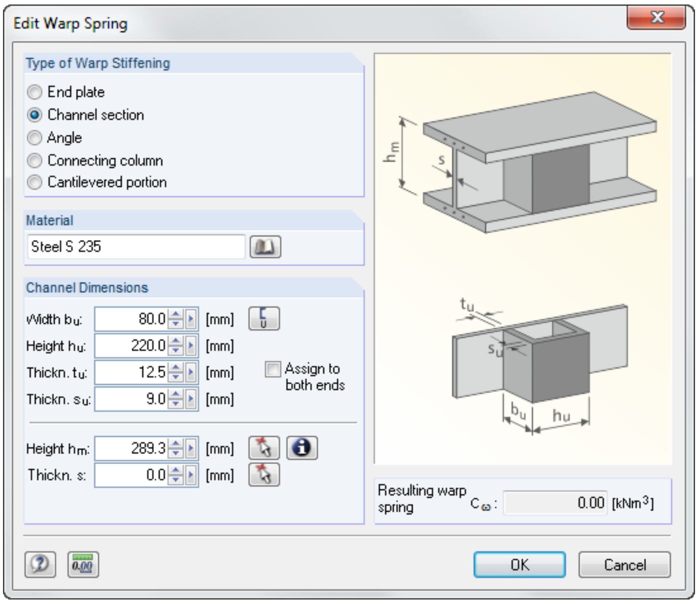


Figure 3.15: Dialog box *Edit Warp Spring*, type *Channel section*

You can change the predefined *Material* of the channel section using the material library.

Moreover, you can manually enter the *Channel Dimensions*. To access the cross-section library of all channel sections to import the relevant geometric parameters in the dialog box, click [Select channel from the library and import the dimensions].

To select the distance of the flange centerlines h_m and the web thickness s of the member cross-section graphically on the cross-section, use [↖].

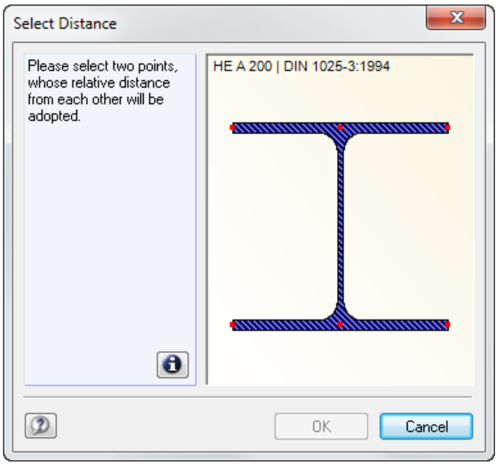


Figure 3.16: Dialog box *Select Distance*

The *Resulting warp spring* is shown in the lower-right corner of the dialog box.

Warp spring from angle

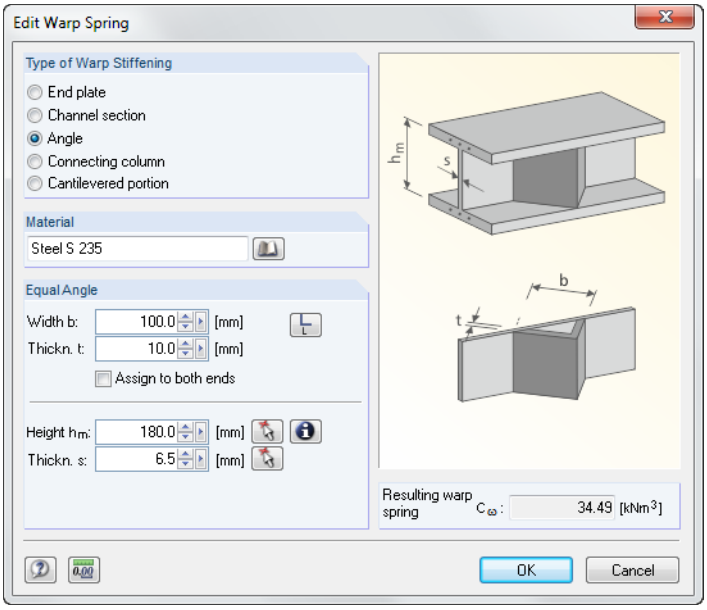


Figure 3.17: Dialog box *Edit Warp Spring*, type *Angle*

The preset *Material* of the angle can be changed in the material library.

You can enter the dimensions of the *Equal Angle* manually. To access the cross-section library of all angle sections in order to import the relevant geometric parameters in the dialog box, click [Select angle from library and import the dimensions].

You can graphically select the distance of the flange centerlines h_m and the web thickness s of the member cross-section upon clicking [↖].

The *Resulting warp spring* is shown in the lower-right corner of the dialog box.

Warp spring from connecting column

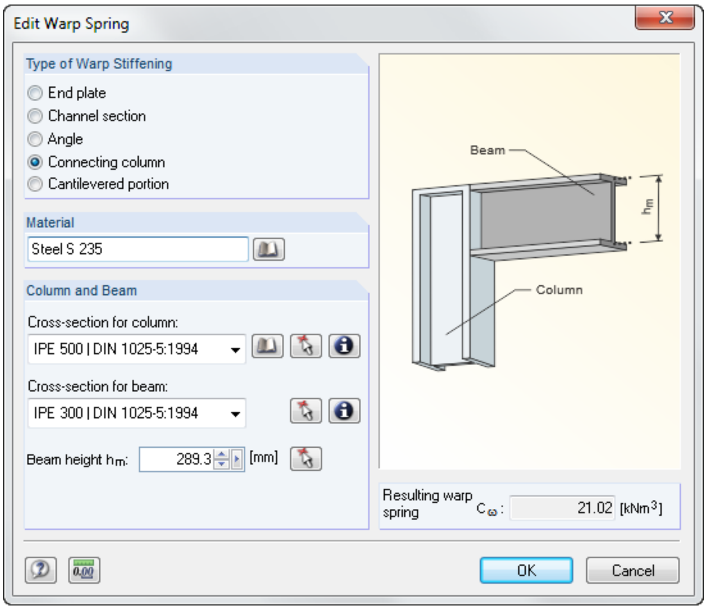


Figure 3.18: Dialog box *Edit Warp Spring*, type *Connecting column*

To change the predefined *Material* of the column, use the material library.

You can select the cross-section of the *column* in the list. Alternatively, you can select it graphically in the RFEM work window upon clicking [↖]. To select a new cross-section for the column, click [Library]. The cross-section of the frame beam is automatically preset, but if necessary you can change it upon clicking [↖].

To graphically select the distance of the beam's flange centerlines h_m on the cross-section, click [↖] (see Figure 3.16).

The *Resulting warp spring* is shown in the lower-right corner of the dialog box.

Warp spring from cantilevered portion

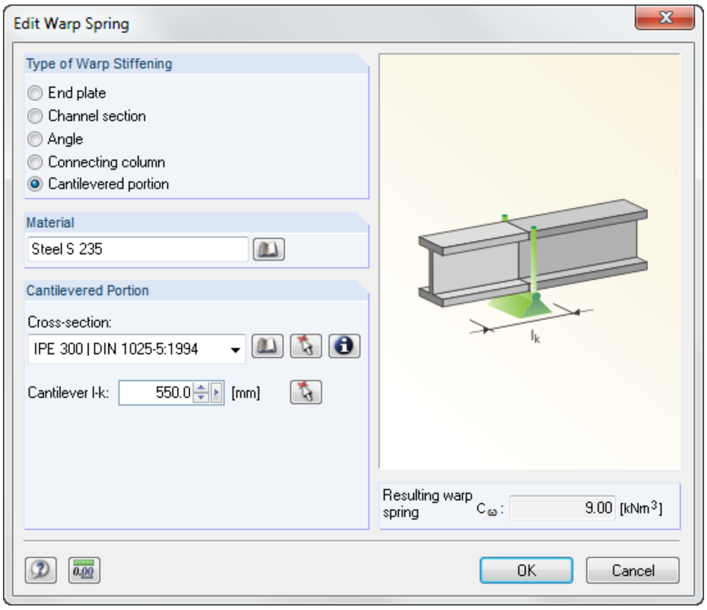


Figure 3.19: Dialog box *Edit Warp Spring*, type *Cantilevered portion*

To change the predefined *Material* of the beam, use the material library.

The cantilevered *Cross-section* is predefined as the cross-section of the beam, but you can also change it graphically in the RFEM work window upon clicking [↖]. To access the cross-section database and select a new cross-section, click [Library].

You can enter the cantilever length l_k directly or specify it by clicking two nodes in the RFEM model upon clicking [↖].

The *Resulting warp spring* is shown in the lower-right corner of the dialog box.

Comment

In the last column of window 1.4, you can enter your own comments for every set of members to describe, for example, the selected boundary conditions.

3.5 Elastic Member Foundations

If there are continuous supports for sets of members, for example, due to trapezoidal sheeting, you can define them in window 1.5. The trapezoidal sheets supporting the beams result in a rotational restraint of the frame beams and also act as shear panel (see chapter 2.4, page 22). In addition to this, you can consider the stabilizing effect of purlins and bracings in this window.

The determination of the spring constants is described in the chapters 2.5.1 and 2.5.2, page 25f.

If trapezoidal sheeting is used as shear panel, it is absolutely necessary to satisfy the conditions stated in the standards, see [13].

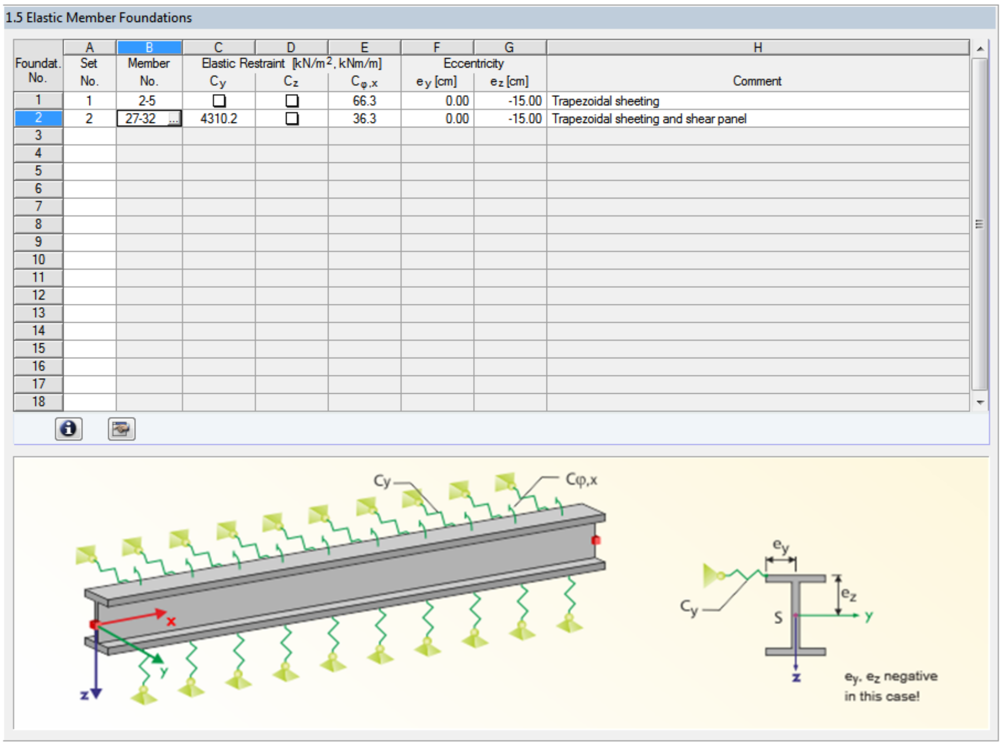
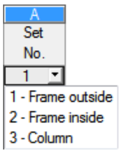


Figure 3.20: Window 1.5 *Elastic Member Foundations*

The springs are relative to the local coordinate system. They act constantly along the member (see chapter 2.2.3, page 16).

Set No.



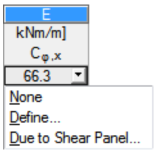
In this column, you specify the set of members for which the support conditions apply.

To define a member foundation, place the cursor in a free cell of the column. Then, enter the number of the set of members or select it in the list. Next, define the member with the elastic foundation in the *Member* column.

Member No.

You can enter the members with elastic foundation individually or as list. Alternatively, you can click [...] (see Figure 3.20) to select them graphically in the RFEM work window.

Elastic foundation C_y / C_z / $C_{\varphi,x}$



In the columns C through E, you specify the translational and rotational spring constants of the selected members. By clicking in a cell, a list becomes available that allows for an automatic determination of the spring constants. Alternatively, you can enter the constants of the translational or rotational springs manually.

To determine the spring constants by the program, click [Edit] below column B. The following dialog box appears.

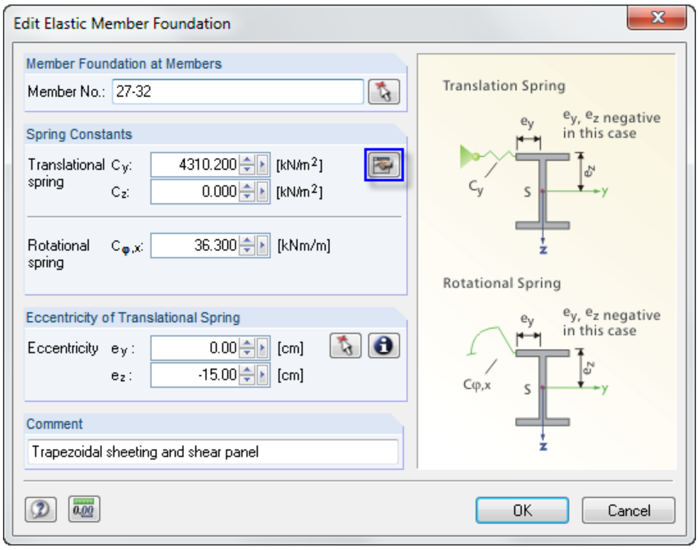


Figure 3.21: Dialog box *Edit Elastic Member Foundation*

The button [Edit] in the *Spring Constants* section (marked in Figure 3.21) allows accessing the program-internal determination of the translational and rotational spring constants for different types of continuous supports:

- Shear panel (→ chapter 2.4, page 23)
- Bracing (→ chapter 2.4, page 24)
- Trapezoidal sheeting (→ chapter 2.4, page 23)
- Purlins (→ chapter 2.4, page 25)
- Section (→ chapter 2.5.1, page 26)

After clicking the button, the *Calculate Translational and Rotational Springs from Shear Panel* appears (see Figure 3.22). The dialog box is subdivided into several tabs that are described in the following pages.

The options and check boxes selected in the *Active* section control which tabs below are available for further specifications. You must specify, whether there is only a lateral restraint, only a rotational restraint, or if both actions should be considered simultaneously.

The *Lateral Restraint* can be due to bracing, trapezoidal sheeting, or to both influences. As a rule, the lateral restraint acts in the direction of the member axis *y*. However, you can also change it to the member axis z in the *Determined Translational Spring* section.

The *Rotational Restraint* of the frame beams is either due to a trapezoidal sheet or by purlins, depending on whether the roof sheeting is spanned without purlins between the beams or is placed on purlins.

In the *Determined Translational Spring* and *Determined Rotational Restraint* sections, the constants of the springs are shown, which the program calculates from the specifications.

Shear Panel

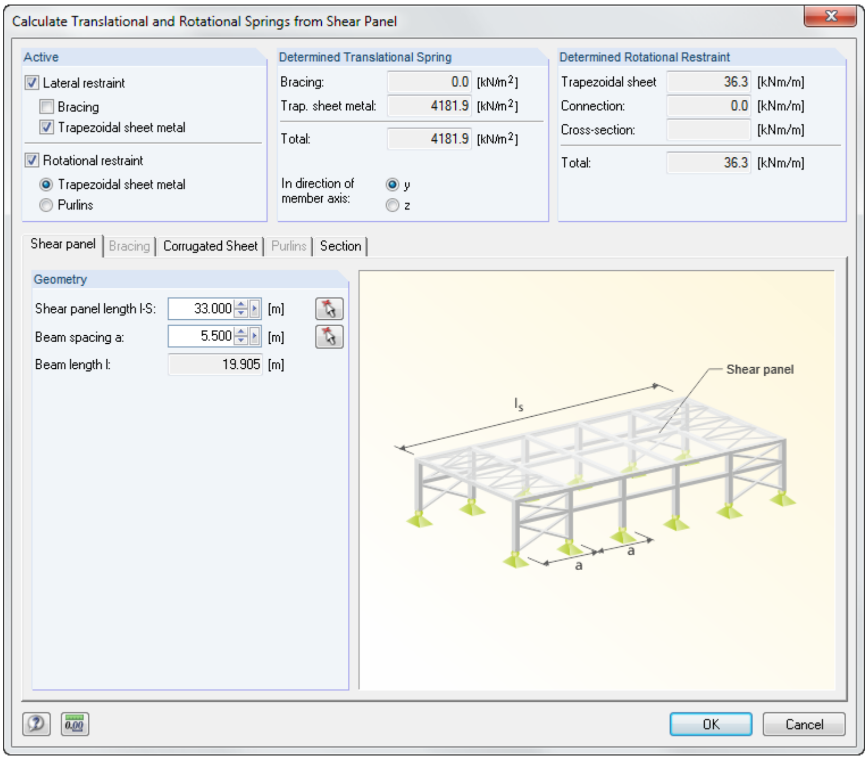


Figure 3.22: Dialog box *Calculate Translational and Rotational Springs from Shear Panel*, tab *Shear panel*

You must specify the length of the shear panel l_S and the distance *a* of the beams to be stabilized. You can also specify the values graphically in the RFEM work window by clicking the two nodes after you click [↖].

Bracing

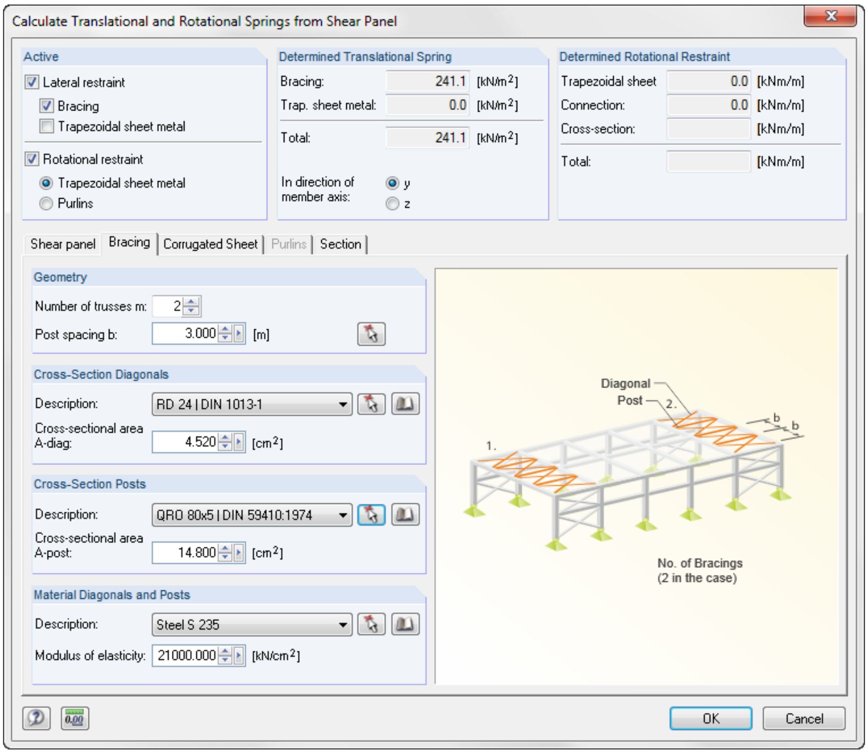


Figure 3.23: Dialog box *Determine Translational and Rotational Spring from Shear Panel*, tab *Bracing*

In the *Geometry* section, you enter the Number of trusses *m* stabilizing the roof plane. You can enter the distance *b* of the bracing posts directly. Alternatively, you can click [↖], and then specify them directly in the RFEM work window by clicking two nodes.

You can enter the cross-sectional area of the *Diagonals* and *Posts* or determine them from the cross-section properties. You can specify the cross-sections by using the lists or graphically by clicking [↖]. To select a new cross-section, click [Library].

You can change the preset *Material* of the diagonals and posts using the list or the [Library] button. Furthermore, you can also directly specify the modulus of elasticity.

Corrugated Sheet

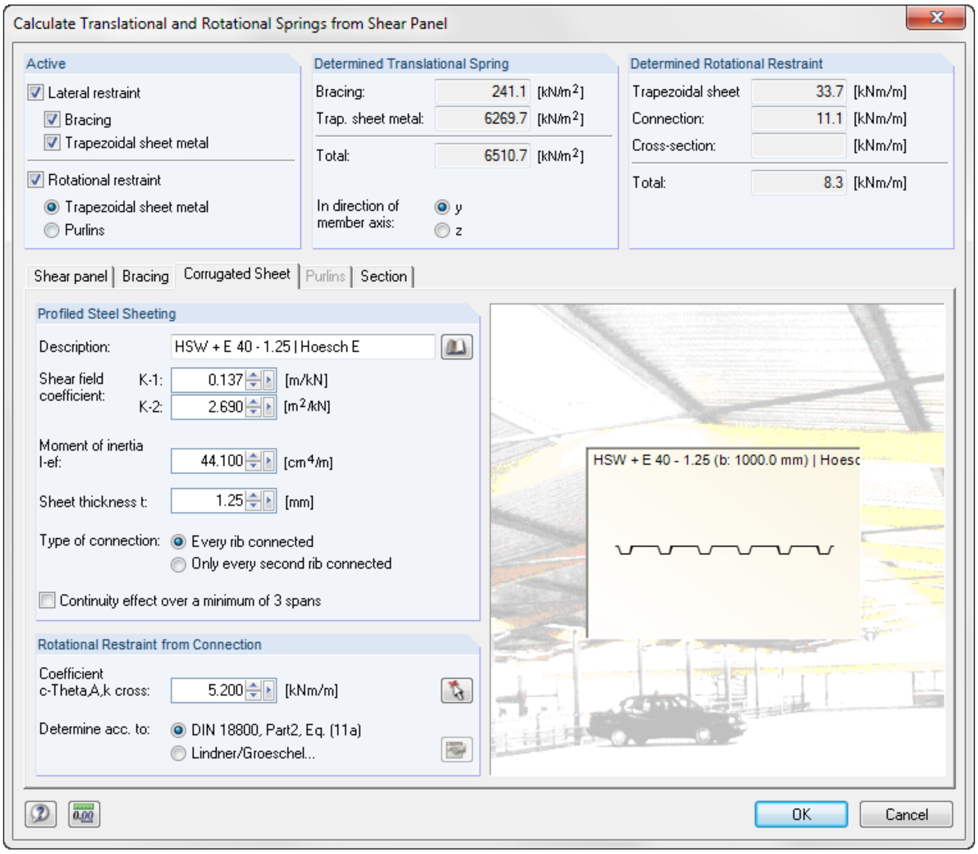


Figure 3.24: Dialog box *Calculate Translational and Rotational Springs from Shear Panel*, tab *Corrugated Sheet*

The *Corrugated Sheet* (trapezoidal sheet) can be selected from a [Library] of common corrugated sheets (see Figure 3.25). The shear field coefficients K_1 and K_2, the moment of inertia I_{ef}, and the sheet thickness *t* are automatically filled with the values from the library. If necessary, you can manually adjust them.

With the *Type of connection* options, you decide whether the trapezoidal sheeting is connected in every rib or in every second rib (groove). The specification has an influence on the rotational restraint (see chapter 2.4, page 24).

If the trapezoidal sheeting is placed over a minimum of three spans on the frame beams, you can activate the *Continuity effect* by selecting the according check box.

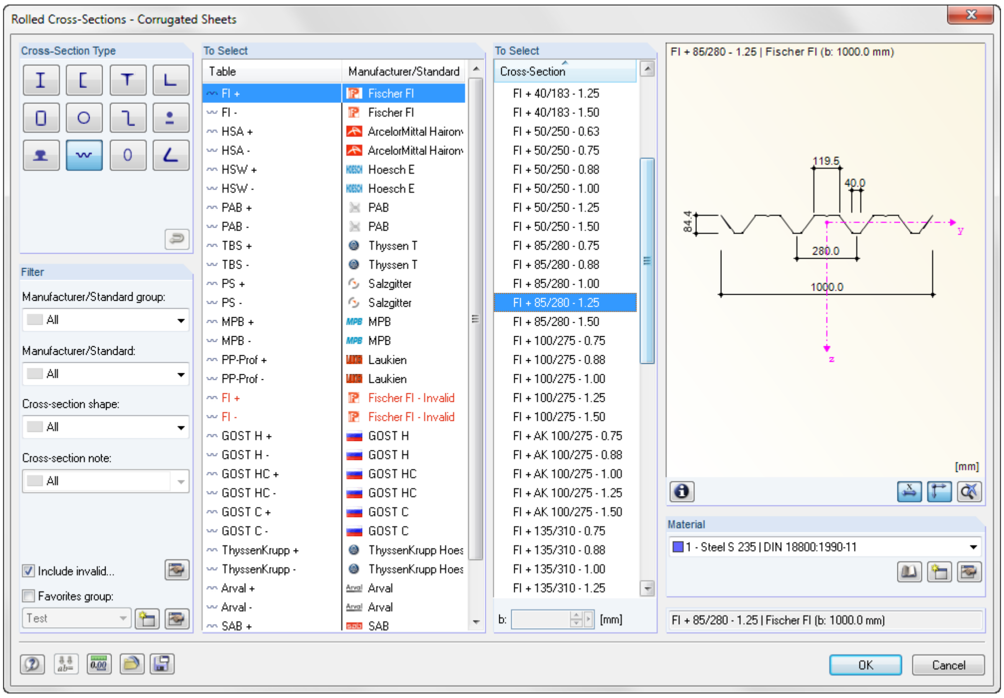


Figure 3.25: Library *Corrugated Sheets*

In the *Rotational Restraint from Connection* section, you can specify the coefficient for the rotational restraint $\bar{c}_{\vartheta A,k}$ (see chapter2.5.1, page 26). By clicking [↖], you open Table 7 of DIN 18800 Part 2. Depending on the boundary conditions, you can import the appropriate coefficient into the initial dialog box by clicking in the according row, and then clicking [OK].

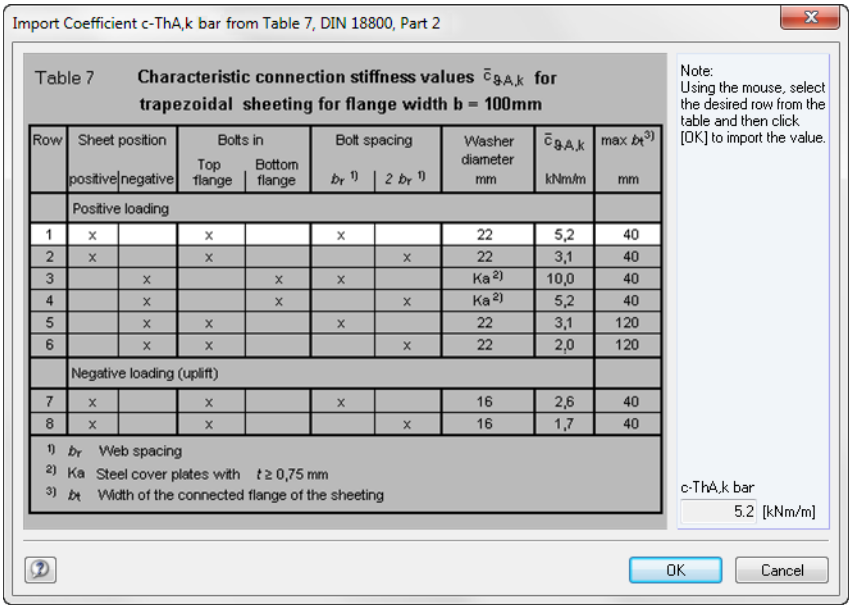


Table 7 **Characteristic connection stiffness values $\bar{c}_{\vartheta A,k}$ for trapezoidal sheeting for flange width b = 100mm**

Row	Sheet position positive	Sheet position negative	Bolts in Top flange	Bolts in Bottom flange	Bolt spacing b_r 1)	Bolt spacing 2 b_r 1)	Washer diameter mm	$\bar{c}_{\vartheta A,k}$ kNm/m	max b_t 3) mm
	Positive loading								
1	x		x		x		22	5,2	40
2	x		x			x	22	3,1	40
3		x		x	x		Ka 2)	10,0	40
4		x		x		x	Ka 2)	5,2	40
5		x	x		x		22	3,1	120
6		x	x			x	22	2,0	120
	Negative loading (uplift)								
7	x		x		x		16	2,6	40
8	x		x			x	16	1,7	40

1) b_r Web spacing
2) Ka Steel cover plates with $t \geq 0{,}75$ mm
3) b_t Width of the connected flange of the sheeting

Figure 3.26: Dialog box *Coefficient c-ThA,k bar from Table 7, DIN 18800 Part 2*

The coefficient for the rotational restraint $\bar{c}_{\vartheta A,k}$ can also be determined according to the (more favorable) method by LINDNER / GROESCHEL [18]. Further specifications are required for this. You can enter them in a dialog box, which you open by clicking [Edit] in the lower part of the *Corrugated Sheet* tab.

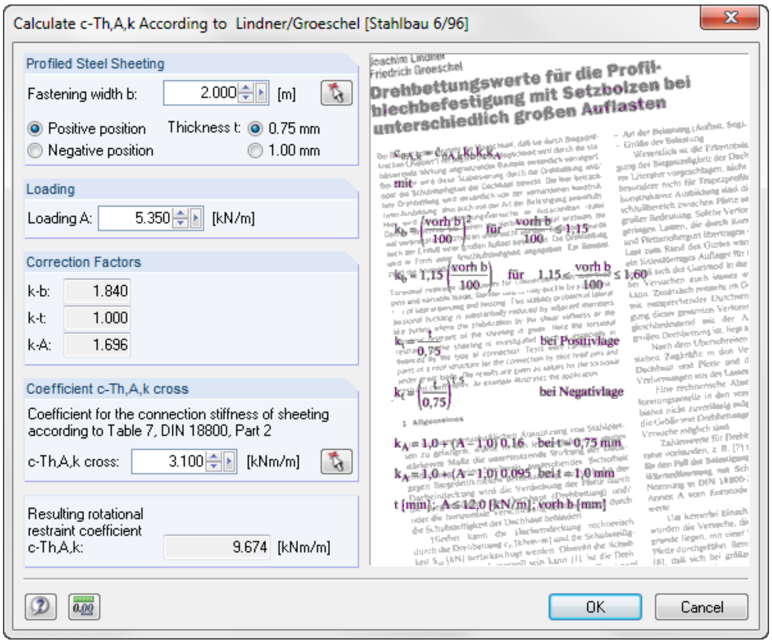


Figure 3.27: Dialog box *Calculate c-ThA,k According to Lindner/Groeschel*

In the *Profiled Steel Sheeting* section, you specify the fastening width *b* of the trapezoidal sheeting on the beam. In addition, the position of the trapezoidal sheeting, its thickness *t*, as well as the Loading *A* is to be specified as design value of the support force of the trapezoidal sheeting. From these specifications, the program determines the *Correction Factors* k_b, k_t, and k_A.

The *coefficient* $\bar{c}_{\vartheta A,k}$ of the rotational restraint can be directly selected from Table 7 of DIN 18800 Part 2 (see Figure 3.26). Below in the dialog box, you can see the computed rotational spring.

Purlins

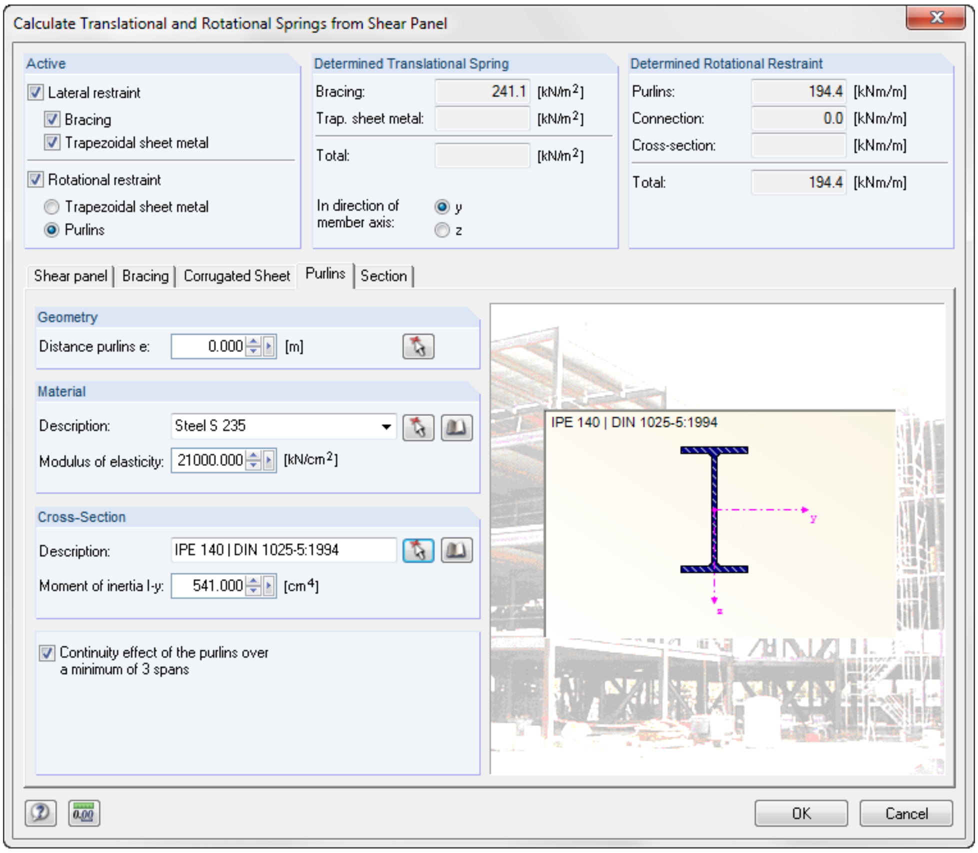


Figure 3.28: Dialog box *Calculate Translational and Rotational Springs from Shear Panel*, tab *Purlins*

In the *Geometry* section, you specify the distance of the purlins *e*. You can also specify the distance graphically in the RFEM work window by selecting two nodes after clicking [↖].

The predefined *Material* of the purlins can be changed by means of the list or the material library. The modulus of elasticity is imported to the input field and can by adjusted, if necessary. Upon clicking [↖], you can determine a member to import its material.

The *Cross-Section* of the purlins can be specified by clicking [↖], and then clicking a member in the RFEM work window. To select a different cross-section for the purlin, click [Library]. Its moment of inertia I_y is imported to the input field below and can be modified, if necessary.

If the purlins cover a minimum of three spans, you can activate the *Continuity effect* by selecting the according check box.

Section

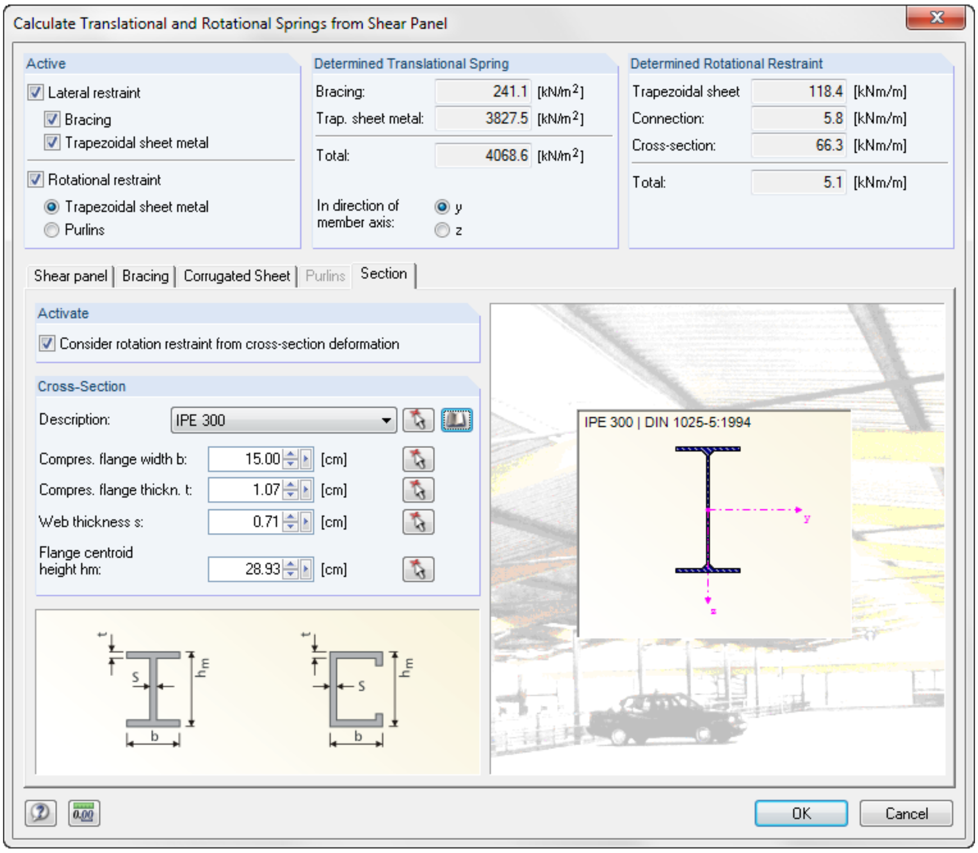


Figure 3.29: Dialog box *Calculate Translational and Rotational Spring from Shear Panel*, tab *Section*

You can change the preset *Section* of the member with the elastic foundation by using the list or graphically in the RFEM work window upon clicking [↖]. Alternatively, you can select a different section in the [Library].

The compression flange width b_1, compression flange thickness t_1, web thickness *s*, and flange centroid height h_m are shown according to the cross-section selection. By clicking [↖], you open the cross-section graphic in which you can specify the relevant cross-section parts or distances by clicking them (see Figure 3.16, page 52).

Eccentricity e_y /e_z

In the columns F and G of window 1.5 (see Figure 3.20, page 55), you can specify the eccentricities of the elastic member foundations. They refer to the local member axes y and z (or u and v for unsymmetric cross-sections). The distances of the springs from the cross-section centroid can also be adjusted in the *Edit Elastic Member Foundation* dialog box (see Figure 3.21, page 56).

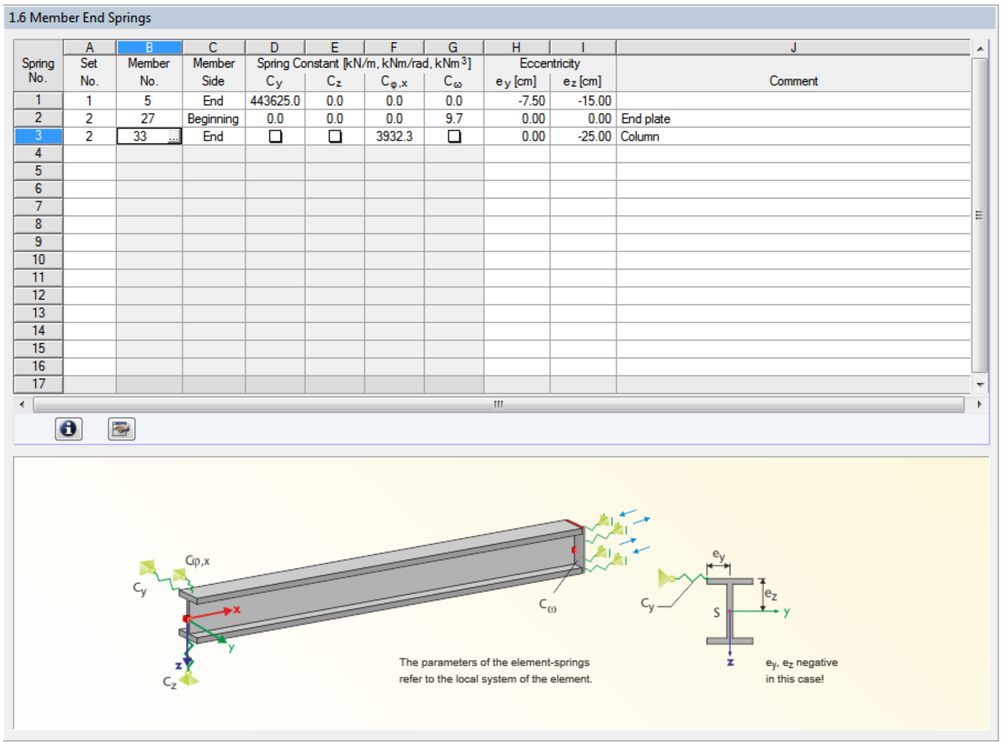
Upon clicking [...] in the table row or [↖] in the dialog box, you can specify the eccentricities in the cross-section graphic by clicking the relevant stress points (see Figure 3.16, page 52).

Comment

In the last column of this window, you can specify user-defined comments for each set of members, for example, the selected boundary conditions.

3.6 Member End Springs

In window 1.6, you can limit the degrees of freedom at the nodes of selected members by means of translational, rotational, and warp springs. In the set of members model, you can thus represent, for example, the warping restraint of a frame beam by means of an end plate connection.



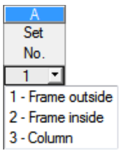
Spring No.	A Set No.	B Member No.	C Member Side	D Spring Constant [kN/m, kNm/rad, kNm³] C_y	E C_z	F $C_{\varphi,x}$	G C_ω	H Eccentricity e_y [cm]	I e_z [cm]	J Comment
1	1	5	End	443625.0	0.0	0.0	0.0	-7.50	-15.00	
2	2	27	Beginning	0.0	0.0	0.0	9.7	0.00	0.00	End plate
3	2	33	End	☐	☐	3932.3	☐	0.00	-25.00	Column
4										
5										
6										
7										
8										
9										
10										
11										
12										
13										
14										
15										
16										
17										

Figure 3.30: Window 1.6 *Member End Releases*

The boundary conditions defined here are to be seen in their interaction with the nodal support parameters of window 1.4. Therefore, you have to take care during the input that no double releases or such like are defined.

In contrast to the nodal supports of window 1.4, you can also define eccentricities for the single springs.

Set No.



Define for which set of members the springs parameters should apply.

To define a member end spring, place the cursor in a free cell of this row. Then, you can enter the number of the set of members or select it in the list. In the *Member* column, specify the members with springs at their ends.

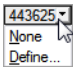
Member No.

You can enter the members with end springs individually as list or select them graphically in the RFEM work window upon clicking [...] (see Figure 3.30)

Member Side

After you click in a cell, the [▼] button appears. By clicking it, you can access the list shown on the left. There you can select at which member end the release spring is located.

Spring constant C_y / C_z / $C_{\varphi,x}$ / C_ω



In the columns D through G, you specify the translational, rotational, and warping constants for the selected member sides (see chapter 2.5, page 25). The spring constants relate to the local member axes y, z, and x (or u and v for unsymmetric cross-sections).

To manually enter the spring constants, click in the cells. Alternatively, you can click [▼] to open the list and *Define* them in a dialog box.

You can determine the member end springs also in the program upon clicking the [Edit] button located below column B. The following dialog box appears.

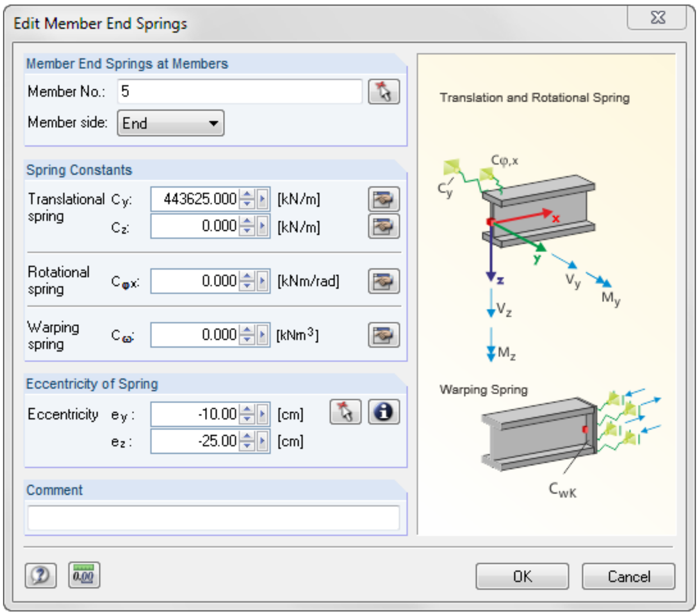


Figure 3.31: Dialog box *Edit Member End Springs*

Translational spring

By clicking [Edit] in the *Spring Constants* section (Figure 3.31), you open a dialog box where you can consider the support of the member end by a connecting component. From the geometric parameters, the program determines the constant of the translational spring at the member end in the local y- or z-direction of the member.

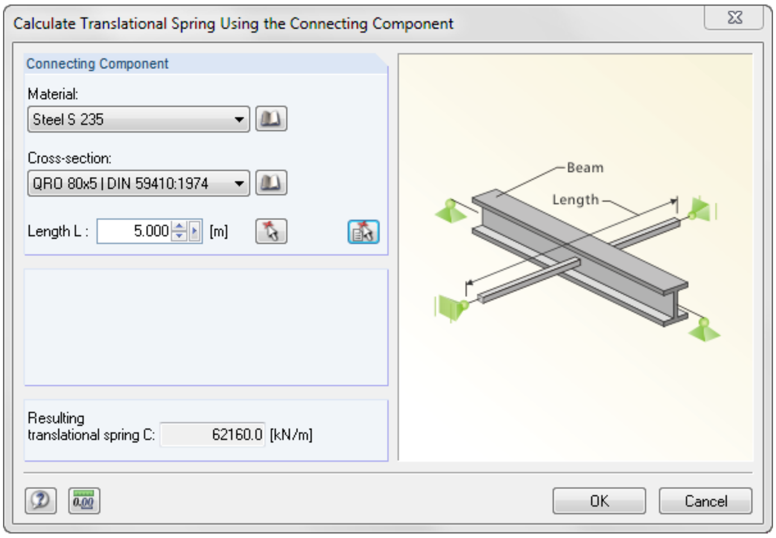


Figure 3.32: Dialog box *Calculate Translational Spring Using the Connecting Component*

To select the *Material* and the *Cross-section* of the connecting component, use the drop-down list or the material and section [Library]. To specify the *Length L* of the component [↖], you can either enter it or select it in the RFEM work window by clicking two nodes.

To transfer the material, cross-section, and length of a graphically selected member, click [Import Member Properties].

Rotational Spring

By clicking [Edit] in the *Spring Constant* section (Figure 3.31), you open another dialog box where you can consider the elastic restraint at the end of the member resulting from a connected column. From the geometric parameters, the program determines the constant of the rotational spring at the member end acting about the local x-direction of the member.

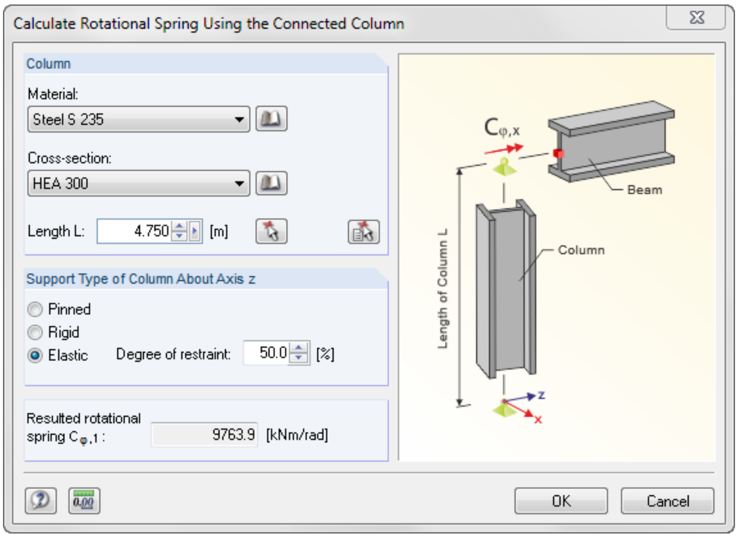


Figure 3.33: Dialog box *Calculate Rotational Spring Using the Connected Column*

To select the *Material* and the *Cross-Section* of the connected column, you can use the drop-down list or the material and section [Library]. To specify the *Length L* of the column, you can either enter it or use [↖] to select it in the RFEM work window by clicking two nodes.

To transfer the material, cross-section, and length of a graphically selected column, click [Import Member Properties].

The *Support Type of Column About Axis z* also influences the rotational spring $C_{\varphi,1}$ of the beam (see dialog graphic). In addition to a pinned and rigid support, the degree of restraint can be freely defined between 0 % (pinned) and 100 % (rigid).

Warp spring

The restraint of warping increases the torsional stiffness of the beam. The [Edit] button in the *Spring Constants* section (Figure 3.31) allows you to open a dialog box where you can consider the warp spring by warp stiffening. From the geometric parameters, the program determines the constant of the warp spring active on the member end.

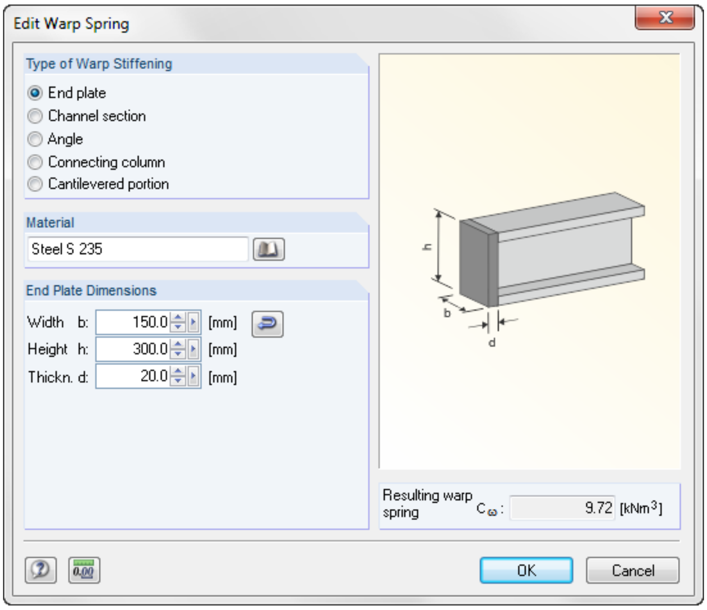


Figure 3.34: Dialog box *Edit Warp Spring*

The theoretical background on determining the warp springs is provided in chapter 2.5.3, page 29f. The warp spring can be defined by means of an end plate, a channel section, an angle, a connected column, or a cantilevered portion.

The *Edit Warp Spring* dialog box is described in chapter 3.4, page 51f.

Eccentricity e_y /e_z

In the columns H and I of window 1.6 (see Figure 3.30, page 63), you can specify the eccentricities of the springs. They refer to the local member axes y and z (or u and v for unsymmetric cross-sections) and are relevant for the spring constants C_y and C_z. The distances of the springs from the cross-section's centroid can also be defined in the *Edit Member End Springs* dialog box (see Figure 3.31, page 64).

To specify the eccentricities in the cross-section graphic by clicking the relevant stress point, click […] in the table cell or [↖] in the dialog box (see Figure 3.16, page 52).

Comment

In the last column of the window, you can enter user-defined comments for each spring, for example, to describe the spring parameters.

3.7 Member End Releases

In window 1.7, you can define the member end releases for individual members in the set of members; this is done independently of RFEM. The releases defined in RFEM are preset for the members of the set of members.

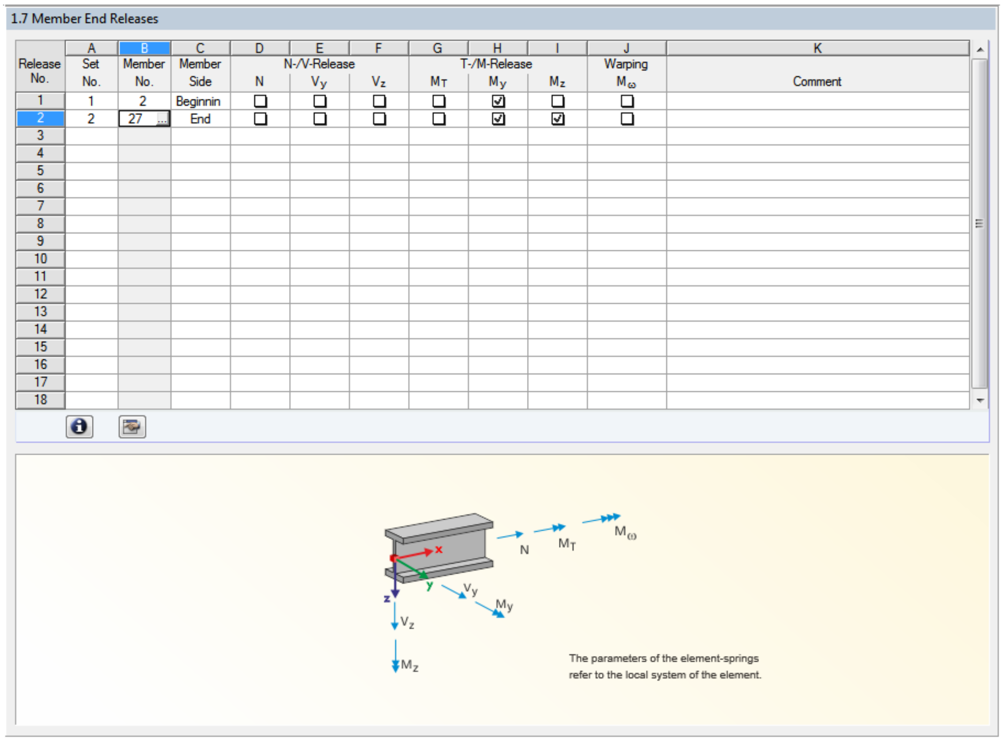
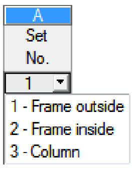


Figure 3.35: Window 1.7 *Member End Releases*

Set No.



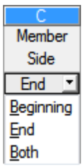
You must specify for which set of members the release conditions should apply.

To define a member end release, place the cursor in a free cell of this column. Then, you can enter the number of the set of members or select it in the list. In the *Members* column, specify the members with the releases at their ends.

Member No.

The members with end releases can be entered individually or graphically in the RFEM work window upon clicking [...] (see Figure 3.35)

Member Side



After you click in a cell of this column, the [▼] button appears that you can use to open the list on the left. In the list, you can select, at which member end the release exists.

N-/V-Release

In the columns D, E, and F, you can specify for the selected member sides the release parameters controlling the transfer of axial and shear forces. The internal forces are relative to the local member axis system xyz.

To control the degrees of freedom, use the relevant check boxes: If you select a check box, the respective internal force is <u>not</u> transferred. Spring constants are not permitted.

T-/M-Release

In the columns G, H, and I, you can specify for the selected member sides the release parameters controlling the torsional and bending moments. These internal forces are relative to the local member axes.

To activate or deactivate the DOF, use the check boxes: If you select a check box, the moment is not transferred. Spring constants are not permitted.

Warping

The column J controls whether the warping torsional moment can be transferred at the selected member sides. If you select the check box, there is a release; the moment is not transferred.

To adjust the release parameters, click [Edit]. The following dialog box appears.

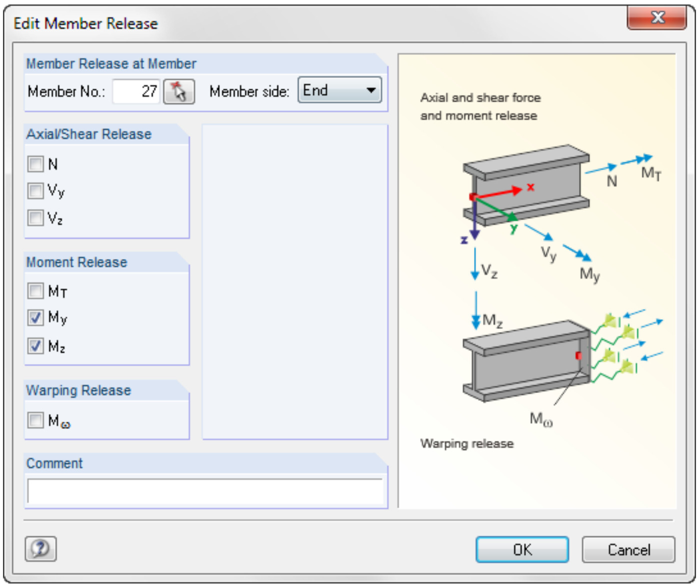


Figure 3.36: Dialog box *Edit Member End Release*

Comment

In the last column of the window, you can enter user-defined comments for each member end release, for example, to describe the release parameters.

3.8 Load

There are three windows to enter the load. To select them, use the tabs at the bottom of the main window:

- 2.1 Nodal Loads
- 2.2 Member Loads
- 2.3 Imperfections

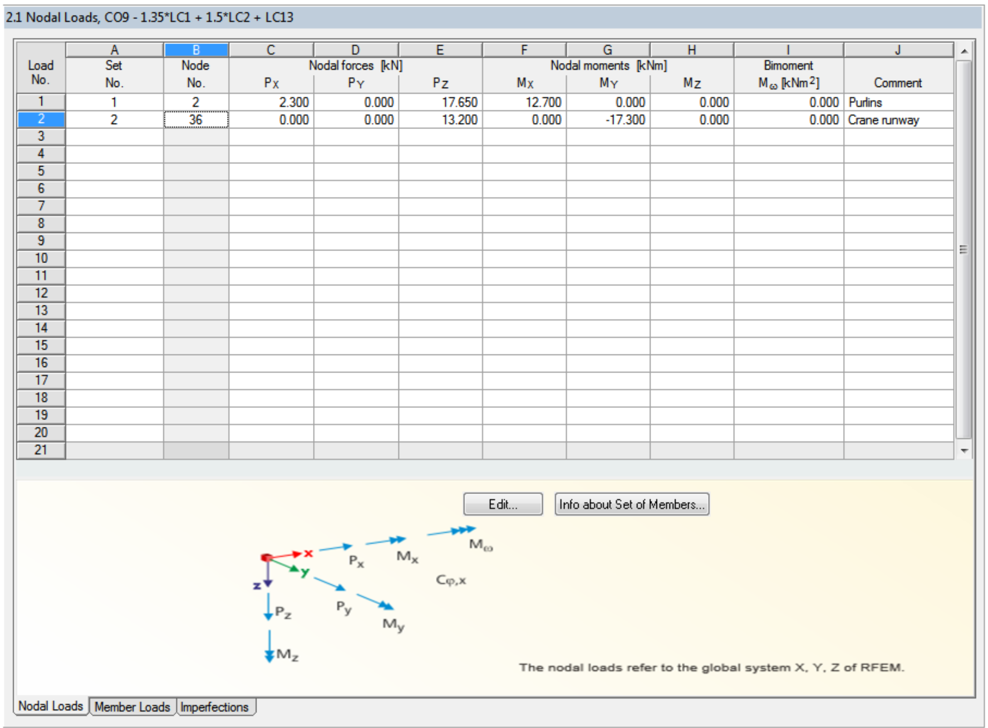
In the navigator on the left under the *Load* entry, you can see all load cases and load combinations selected in window 1.1 *General Data* for design. There, you must first select the entry whose load data you want to define.

If you defined nodal and member loads in RFEM for the members contained in the set of members, they are preset in the window 2.1 *Nodal Loads* and 2.2 *Member Loads*. You can change the loads, if necessary. Imperfections, however, are not imported: You must define them in the 2.3 *Imperfections* window depending on the eigenvalue.

Caution! If structural components that do not belong to the set of members introduce loads to the set of members like, for example hall frame with craneway consoles or 3D halls with purlin roofs, the loads are not automatically imported from RFEM. It is absolutely necessary that you add these loads so that the model of the notionally singled out set of members can be created correctly! As a rule, you can define the introduced loads as additional nodal loads. For eccentrically acting additional loads, however, we recommend the 2.2 *Member Loads* window.

3.8.1 Nodal Loads

You enter the loads for the load case or load combination that you selected on the left in the navigator.

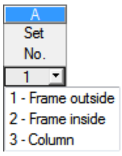

2.1 Nodal Loads, CO9 - 1.35*LC1 + 1.5*LC2 + LC13

	A	B	C	D	E	F	G	H	I	J
Load No.	Set No.	Node No.	Nodal forces [kN] P_X	P_Y	P_Z	Nodal moments [kNm] M_X	M_Y	M_Z	Bimoment M_ω [kNm²]	Comment
1	1	2	2.300	0.000	17.650	12.700	0.000	0.000	0.000	Purlins
2	2	36	0.000	0.000	13.200	0.000	-17.300	0.000	0.000	Crane runway
3										
4										
5										
6										
7										
8										
9										
10										
11										
12										
13										
14										
15										
16										
17										
18										
19										
20										
21										

Figure 3.37: Window 2.1 *Nodal Loads*

Set No.

In this column, you specify for which set of member the nodal loads are effective. All nodal loads that you defined in RFEM for the nodes in the set of members are preset.



To insert an additional nodal load, place the cursor in a free cell of this column. Then, you can enter the number of the set of members or select it in the list. In the *Nodes* column, you specify the loaded nodes.

Node No.

You can specify the nodes subject to loads either individually or as list. Alternatively, click [...] (see Figure 3.37) to select them graphically in the RFEM work window.

Nodal Forces P_X / P_Y / P_Z

In the columns C to E, you specify the forces that act on the selected nodes. Thus, you can consider the internal forces introduced in the set of members. These internal forces are transmitted as axial and shear forces by connecting components (for example craneway console, purlins, or posts): Internal forces of members or surfaces that are not part of the set of members are not automatically imported from RFEM! You must define these additional loads manually.

The nodal forces in this window are relative to the global XYZ coordinate system. Thus, it can be necessary to transform the local RFEM internal forces of members (xyz coordinate system).

Nodal Moments M_X / M_Y / M_Z

In the columns F through H, you can consider torsional and bending moments that are transmitted at the selected nodes to the set of members. The moments are also relative to the global XYZ axis system.

Bimoment M_ω

In this column, you can enter additionally acting nodal warping torsional moments. RFEM calculates no internal forces due to warping torsion.

You can also adjust the current nodal load by clicking [Edit] below the list of nodal loads. The following dialog box appears.

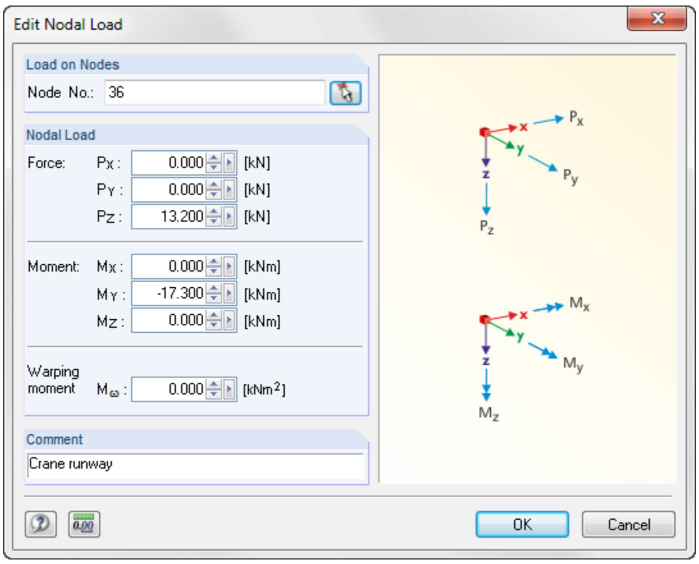


Figure 3.38: Dialog box *Edit Nodal Load*

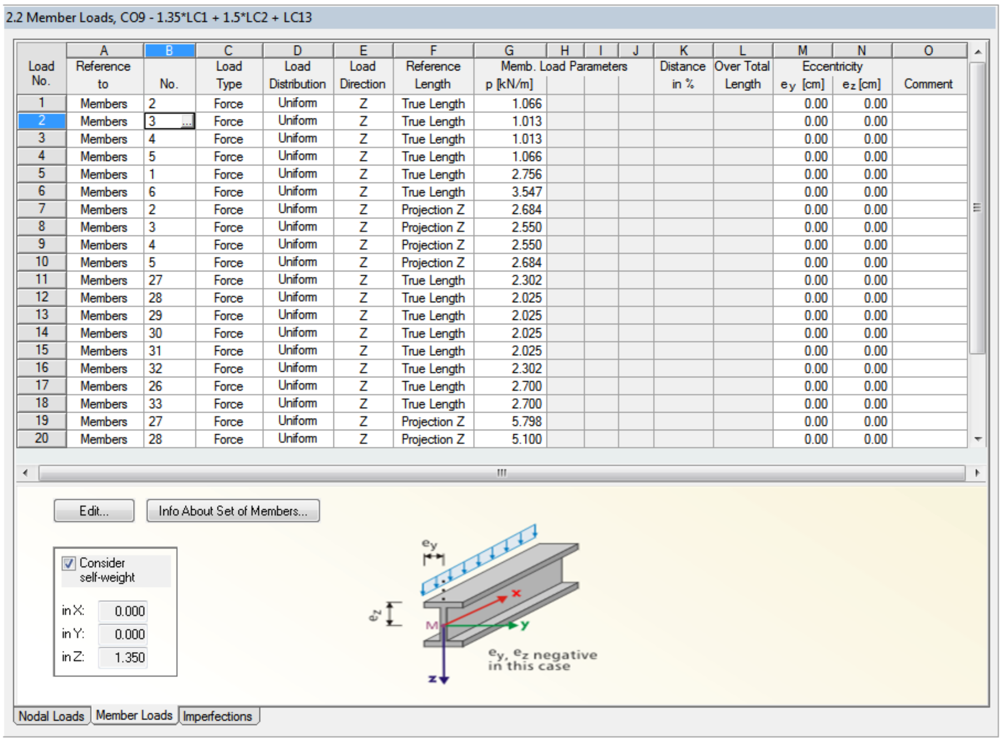
Comment

In the last column of this window, you can write your own comments, for example to describe the additionally acting nodal load.

3.8.2 Member Loads

In window 2.2, all member loads are preset that were defined in RFEM for the members contained in the set of members. If loads transmitted eccentrically to the continuous members are considered, they can be taken into account as additional individual forces or moments.

You specify the loads for the load case or load combination that you selected in the navigator on the left.

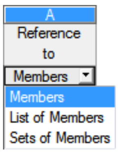

2.2 Member Loads, CO9 - 1.35*LC1 + 1.5*LC2 + LC13

Load No.	A Reference to	B No.	C Load Type	D Load Distribution	E Load Direction	F Reference Length	G Memb. Load Parameters p [kN/m]	H	I	J	K Distance in %	L Over Total Length	M Eccentricity e_y [cm]	N e_z [cm]	O Comment
1	Members	2	Force	Uniform	Z	True Length	1.066						0.00	0.00	
2	Members	3	Force	Uniform	Z	True Length	1.013						0.00	0.00	
3	Members	4	Force	Uniform	Z	True Length	1.013						0.00	0.00	
4	Members	5	Force	Uniform	Z	True Length	1.066						0.00	0.00	
5	Members	1	Force	Uniform	Z	True Length	2.756						0.00	0.00	
6	Members	6	Force	Uniform	Z	True Length	3.547						0.00	0.00	
7	Members	2	Force	Uniform	Z	Projection Z	2.684						0.00	0.00	
8	Members	3	Force	Uniform	Z	Projection Z	2.550						0.00	0.00	
9	Members	4	Force	Uniform	Z	Projection Z	2.550						0.00	0.00	
10	Members	5	Force	Uniform	Z	Projection Z	2.684						0.00	0.00	
11	Members	27	Force	Uniform	Z	True Length	2.302						0.00	0.00	
12	Members	28	Force	Uniform	Z	True Length	2.025						0.00	0.00	
13	Members	29	Force	Uniform	Z	True Length	2.025						0.00	0.00	
14	Members	30	Force	Uniform	Z	True Length	2.025						0.00	0.00	
15	Members	31	Force	Uniform	Z	True Length	2.025						0.00	0.00	
16	Members	32	Force	Uniform	Z	True Length	2.302						0.00	0.00	
17	Members	26	Force	Uniform	Z	True Length	2.700						0.00	0.00	
18	Members	33	Force	Uniform	Z	True Length	2.700						0.00	0.00	
19	Members	27	Force	Uniform	Z	Projection Z	5.798						0.00	0.00	
20	Members	28	Force	Uniform	Z	Projection Z	5.100						0.00	0.00	

Figure 3.39: Window 2.2 *Member Loads*

For loads and load combinations whose automatic self-weight are deactivated in RFEM, the *Consider self-weight* check box is available. It controls whether or not the members contained in the set of members are also applied in RF-FE-LTB. If you select the check box, the self-weight factor defined in RFEM is entered taking into account the load case factor.

Reference to



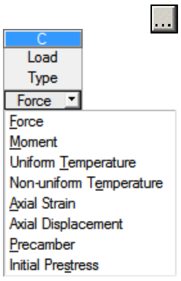
In the list of this cell, you specify whether the load acts on individual members, a list of members, or the entire set of members. The effect of the reference possibilities is described in chapter 6.2 of the RFEM manual.

To insert an additional member load, you have to place the cursor in a free cell of the column and specify the reference of the load. In the next column, you can specify the *Numbers* of the loaded objects.

Members / List of Members / Sets of Members No.

To specify the members or sets of members subjected to loads, you can enter them individually or as list. You can also select them graphically in the RFEM work window upon clicking [...] (see Figure 3.39)

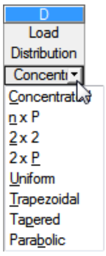
Load Type



The list contains all types of member loads of RFEM (see chapter 6.2 of the RFEM manual).

The current version of RF-FE-LTB supports only **Forces** and **Moments**.

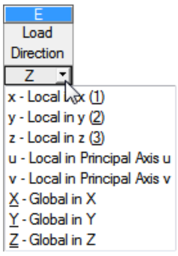
Load Distribution



For forces, the list shown on the left presents the available load distributions (except for the parabolic distribution). They are described in chapter 6.2 of the RFEM manual.

Moments can also be considered in RF-FE-LTB if they are defined as concentrated single or multiple loads. Distributed moments are not supported in the current version.

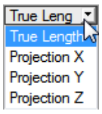
Load Direction



The force or the concentrated moment can act in the direction of the global axes X, Y, and Z or of the local member axes x, y, and z (or u and v for unsymmetrical cross-sections). The load reference axes are described in chapter 6.2 of the RFEM manual.

For the analysis in RF-FE-LTB, it is unimportant whether a load is defined locally or, equivalently, as global.

Reference Length



The load entry can be relative to a total, true member or set of members length, or to the projection of the member or set of members in one direction of the global coordinate system.

Parameters of member load P / M / p / p_1 / p_2 / n / A / B

Columns G through J define the load magnitudes for *P*, *M*, or *p*, and possible additional parameters. The input fields are enabled and accordingly named depending on the previously activated entries.

The parameter *n* describes the number of concentrated loads; the parameters *A* and *B* describe the distances of the load from the start of the member / set of members.

Distance in %

If you select the check box in column K, the distances of concentrated or trapezoidal loads can be defined relative to the length of the member or set of members.

Over Total Length

You can select the check box in column L only in case of trapezoidal loads. It has the effect that the linearly variable load is distributed from the start to the end of the member or set of members. Then, the columns I and J become available.

Eccentricity e_y /e_z

In the columns M and N, you can define eccentricities for the point of application of the load. The eccentricities refer to the local member axes y and z.

The program applies global line loads in the shear center and local line loads in the centroid (see Figure 2.3 and Figure 2.9). Vertical loads often act in the neutral axis of the cross-section. For monosymmetrical cross-sections like channel sections, you therefore have to consider that the shear center and centroid have a different position. By means of the eccentricity e_y in column M, you can consider an intended torsion.

Moreover, the load often does not act at the height of the shear center but at the top of the cross-section. This eccentricity can be defined in column N. For the load application at the top flange, please take care of the fact that you must specify a **negative** value for e_z.

To specify the eccentricities in the cross-section graphic, click [...] in the table cell or [↖] in the dialog box, and then click the relevant stress point (see Figure 3.16, page 52).

To edit the parameters of the current member load, click [Edit] below the table. The following dialog box appears.

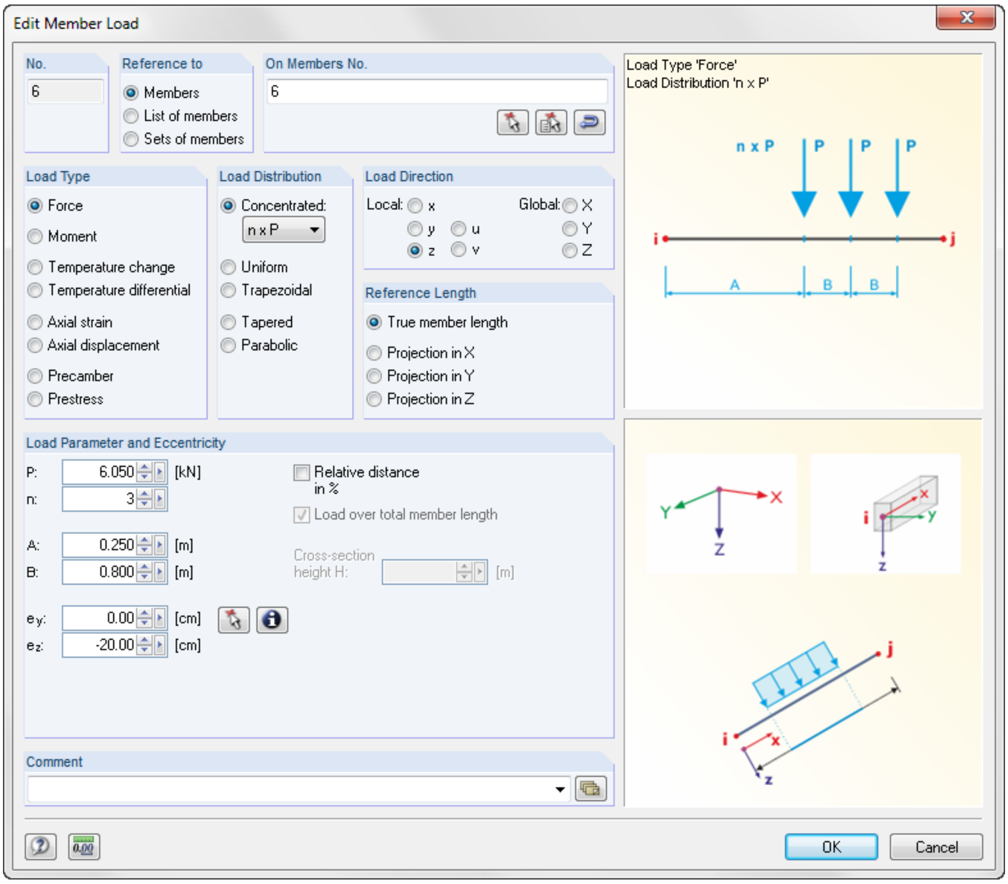


Figure 3.40: Dialog box *Edit Member Load*

Comment

In the last column of this window, you can write your own comments, for example, to describe an additional eccentric member load.

3.8.3 Imperfections

RF-FE-LTB does not use equivalent loads but runs its own calculation of the eigenvectors. Therefore, you must separately define the imperfections in window 2.3. The geometric equivalent loads of the RFEM imperfection load cases are not considered.

According to DIN 18800 Part 2 [8] and EN 1993-1-1 [9], the imperfections are to be applied according to the mode shape that belongs to the lowest or governing buckling eigenvalue. In window 2.3, you can specify the relevant eigenvectors and value *s* (magnitude of imperfection).

You enter the imperfections for the load case or load combination that you selected in the navigator on the left.

For more details on imperfections, see chapter 2.6.2 on page 37.

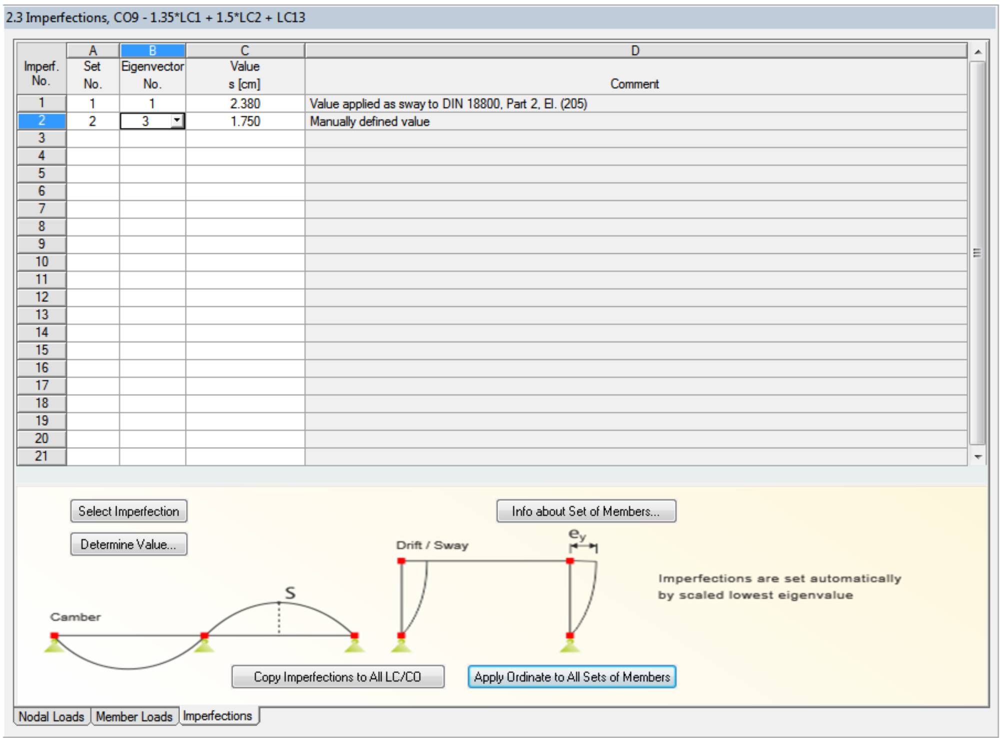
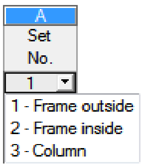


Figure 3.41: Window 2.3 *Imperfections*



Set No.

In this column, you can specify for which set of members the imperfections should apply.

To define a new imperfection, place the cursor in a free cell of this column. Then, you can enter the number of the set of members or select it in the list.

Eigenvector No.

In column B, you can directly enter or select form the list the number of the governing eigenvector. The first eigenvector, which is often governing, is preset.

A higher eigenvector can be governing for the lateral-torsional buckling. Hence, it is necessary to analyze various eigenvectors. By clicking [Select Imperfection], you can check the eigenvector graphically in the RFEM work window (see Figure 3.42) and also import it from there in window 2.3. Here, RF-FE-LTB runs an eigenvalue analysis prior to the actual calculation.

The number of the eigenvectors shown in the list is managed in the *Details* dialog box (see chapter 4.1, page 77), which you open by clicking [Details]. The predefined number of imperfection shapes is 10. The *Details* dialog box allows also you to calculate the eigenvectors without considering rotational and shear panel restraint.

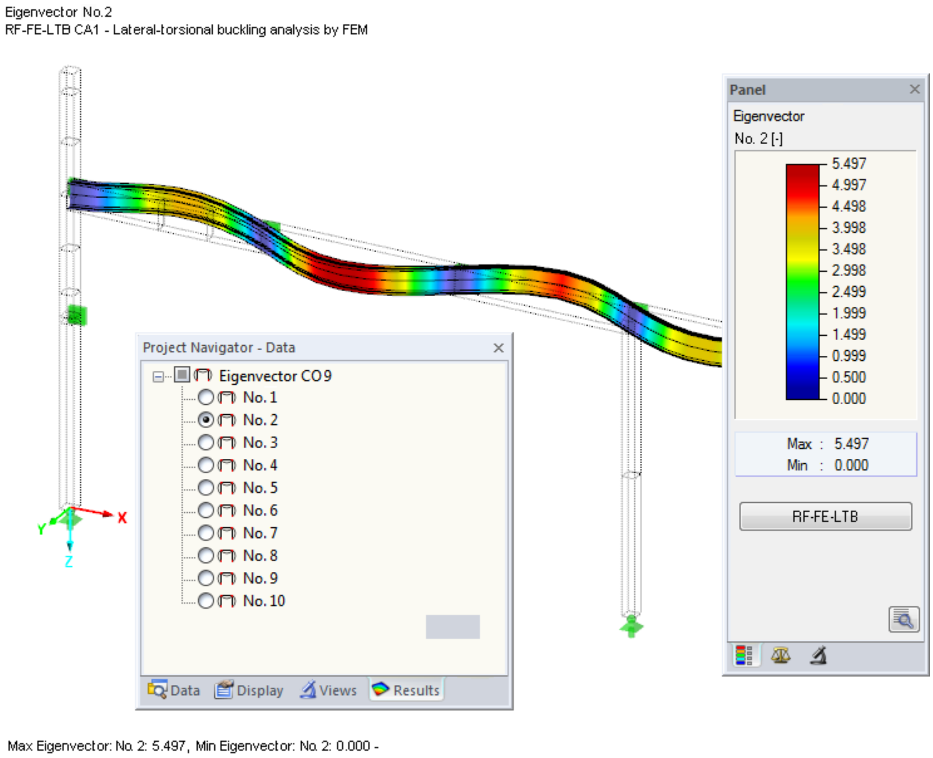
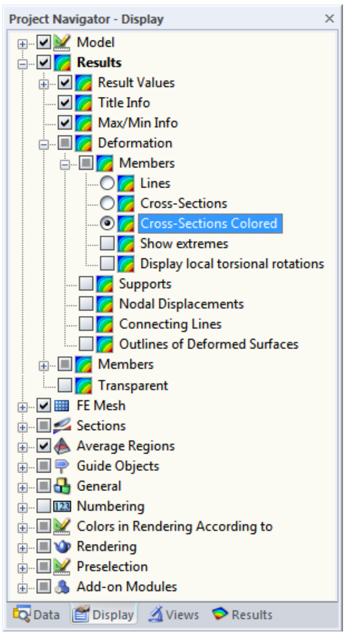


Figure 3.42: Graphical control of the eigenvectors in RFEM work window



View navigator:
Rendering of eigenvector

Value s

In column C, the magnitude of imperfection is to be specified in [cm]. To determine this reference value from the specifications of the geometry and the standards in a dialog box, click [Determine Value].

You can determine the magnitude of imperfection by means of the initial sway or initial camber. The specifications in the section *Calculate the Magnitude of Imperfection by* influence the view of the dialog box.

Initial sway

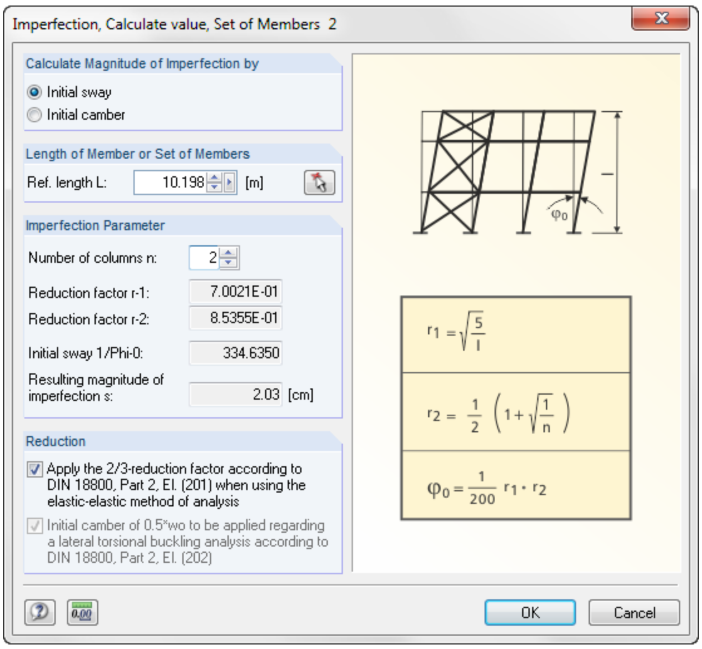


Figure 3.43: Dialog box *Imperfection, Calculate value* by *Initial sway*

This method for the determination of the magnitude of imperfection *s* is recommended for sway systems. To specify the *Reference length L* of the governing member or set of members, you can enter it directly or, after clicking [↖], select it in the RFEM work window by clicking two nodes. For a frame, the reference length is usually the length of the column. With the member length, the reduction factor r_1 and the magnitude of imperfection (value *s*) are determined.

The *Number of columns n* is used for the calculation of the reduction factor r_2. Notice that according to DIN 18800 Part 2, element (205), only those columns may be considered that show at least 25 % of the axial force of the most loaded column.

In the *Elastic-elastic method*, the imperfections may be reduced according to DIN 18800 Part 2, element (201), to $^2/_3$ of the values.

Initial camber

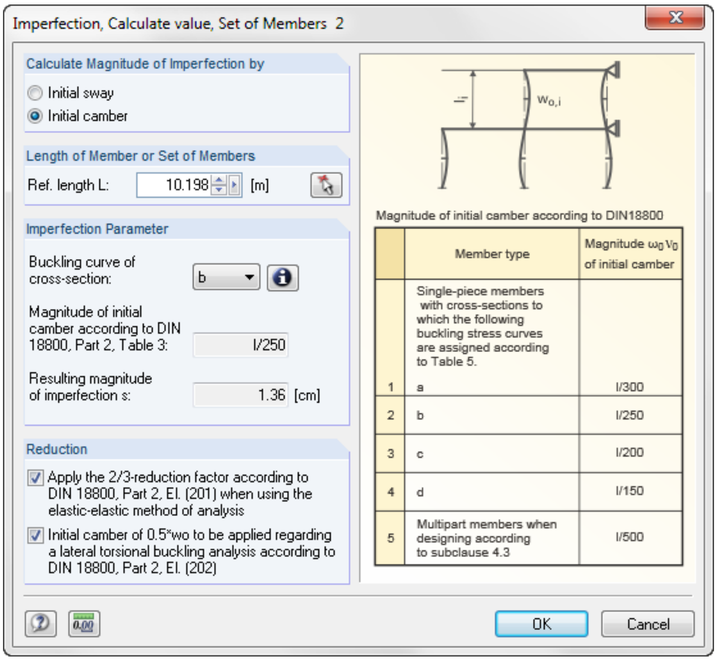
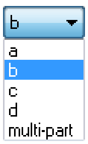


Figure 3.44: Dialog box *Imperfection, Calculate value* by *Initial camber*

This method for determining the magnitude of imperfection is recommended for non-sway systems. To specify the *Reference length L* of the governing member or set of members, you can enter it directly or, after clicking [↖], select it in the RFEM work window by clicking two nodes.



The buckling curve of the cross-section is to be specified according to DIN 18800 Part 2, Table 5. For the sections from the library, the buckling curve BC_z is preset; you can change it by using the list, if necessary.

In the *Elastic-elastic method* according to DIN 18800 Part 2, element (201), it is possible to reduce the magnitude of imperfection to $^2/_3$. In addition, you can reduce the initial camber to $0.5 \cdot w_0$ according to DIN 18800 Part 2, element (202) (see also chapter 2.6.2, page 37).

Below in window 2.3 (see Figure 3.41, page 74), you can use the buttons [Copy Imperfections to All LC/CO] and [Apply Ordinate to All Sets of Members]. Thus, you can copy the current imperfections for all load constellations to be designed or assign the ordinate (magnitude of imperfection) of the current row to all sets of members.

Comment

The final column allows you to enter your own comments to describe the selected eigenvector or determine the magnitude of imperfection.

4. Calculation

4.1 Detail Settings

Before the calculation, check the design details. You can open the corresponding dialog box in every window of RF-FE-LTB by clicking [Details].

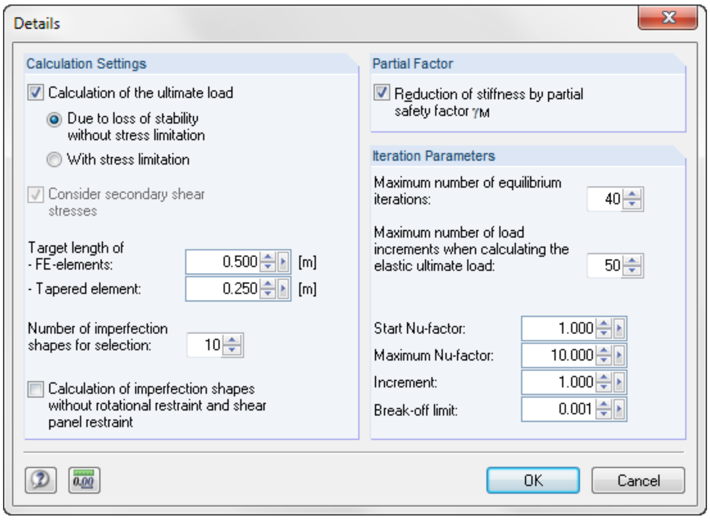


Figure 4.1: Dialog box *Details*

Calculation Settings

In RF-FE-LTB, you can run the *Calculation of the ultimate capacity*: For this analysis of the structural system, you analyze whether the design loads F_d are less than the ultimate capacity F_G or the ultimate capacity F_T of the structural system (see chapter 2.6.4, page 39).

The ultimate capacity (limit load) is determined by an iterative load increase (see Figure 2.1, page 10). For this, you can select from two options which are described in chapter 2.1.4, page 12f:

- Ultimate capacity due to loss of stability (snap-through load) on the imperfect system without stress limitation
- Ultimate capacity due to loss of stability on the imperfect system observing the elastic limit stress (all normal, shear, and equivalent stresses are smaller or equal to the respective elastic limit stress)

Optionally, you can *Consider secondary shear stresses* in the calculation.

It is recommended to adjust the preset *Lengths of the FE elements* to the dimensions of the sets of members. For short sets of members, the FE lengths should be reduced accordingly. For tapered elements, you can use a separate refinement option to consider the cross-section changes by an according discretization.

The *Number of imperfection shapes* has an influence on the eigenvalues in the 2.3 *Imperfections* window (see figure on the left margin of page 74).

If the *imperfection shapes are determined without rotational and shear panel restraint*, then the eigenvectors, which were computed in window 2.3 on [Select Imperfection], represent "pure" eigenvectors: They are obtained without considering the different coefficients for translational and rotational restraints due to stabilizations.

Partial Factor

If the check box in the section is selected, the stiffnesses E · I or E · A are divided by the material partial safety factor γ_M. This coefficient can be specified separately for each material in RFEM.

Iteration Parameters

The critical load factor is determined iteratively. You can influence the calculation procedure and the convergence behavior by means of the input fields. As a rule, it is not necessary to change the preset values.

Starting with the *Start Nu-Factor* ν ("nu"), the loading is continuously increased according to the defined *Increment* until the system becomes unstable. The *Maximum Nu-factor* (Maximum load factor ν) is limited to 10, since, according to DIN 18800 Part 1, element (728), a second-order stability analysis is no longer necessary beyond this value.

4.2 Start Calculation

You can start the calculation from every input window of the module RF-FE-LTB by clicking [Calculation].

RF-FE-LTB runs an independent analysis on the notionally singled out model. Therefore, it is not important whether the internal forces of the load cases and load combinations to be designed were calculated in RFEM.

You can also start the calculation in the RFEM user interface: The *To Calculate* dialog box (menu *Calculation* → *To Calculate*) lists the design cases of the add-on modules like load cases or load combinations.

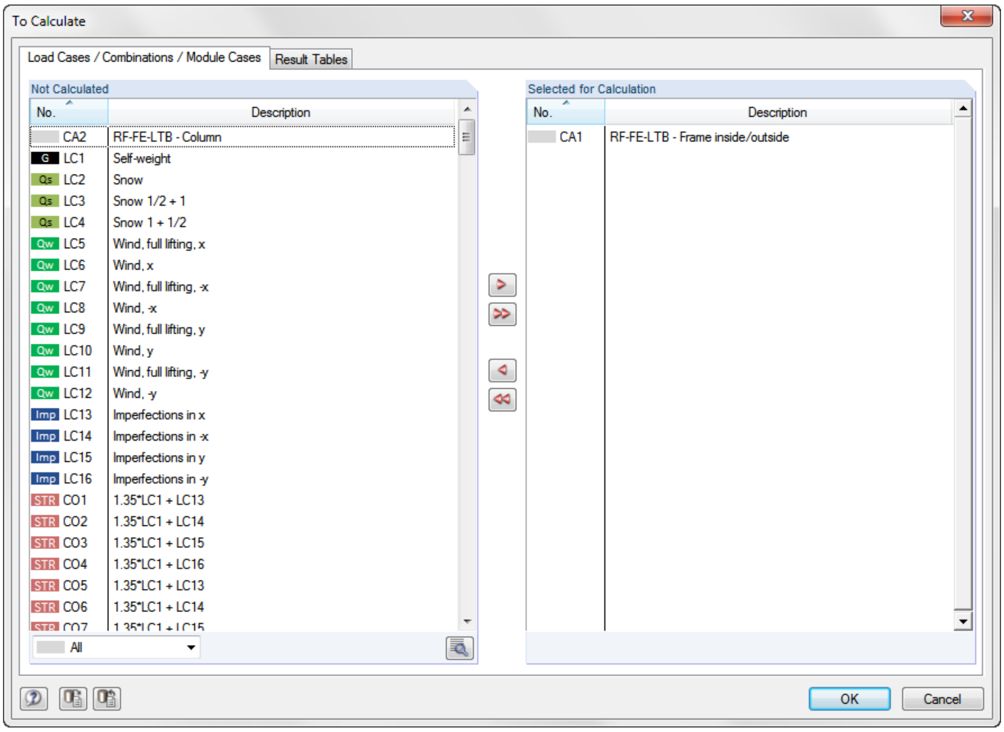
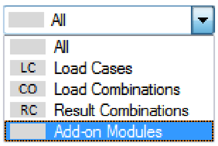


Figure 4.2: Dialog box *To Calculate*



If RF-FE-LTB cases are missing in the *Not Calculated* list, use the drop-down list at the left bottom of the dialog box to change the display to *All* or *Add-on Modules*.

To transfer the selected RF-FE-LTB cases to the list on the right, click [▶]. To start the calculation, click [OK].

You can also directly calculate a design case by using the list in the toolbar: Select the relevant RF-FE-LTB case, and then click [Show Results].

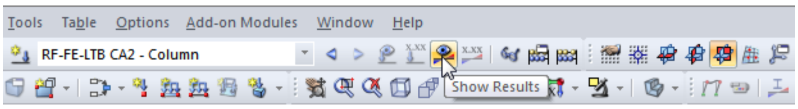


Figure 4.3: Direct calculation of a RF-FE-LTB design case in RFEM

You can then follow the calculation in a solver window.

If more and more memory is used and the calculation takes a long time, no convergence can be reached during the iteration: The notionally singled out model of set of members is instable. After some time, the *Calculation Errors and Notes* (see Figure 5.10, page 89) appears. Often, you can resolve the problem by adjusting the support conditions.

5. Results

After the successful calculation, the 3.1 *Stresses by Cross-Section* window appears. If only window 3.8 *Critical Load Factors* appears, this means that the model is instable because the loading is too large: The critical load factor is less than 1.

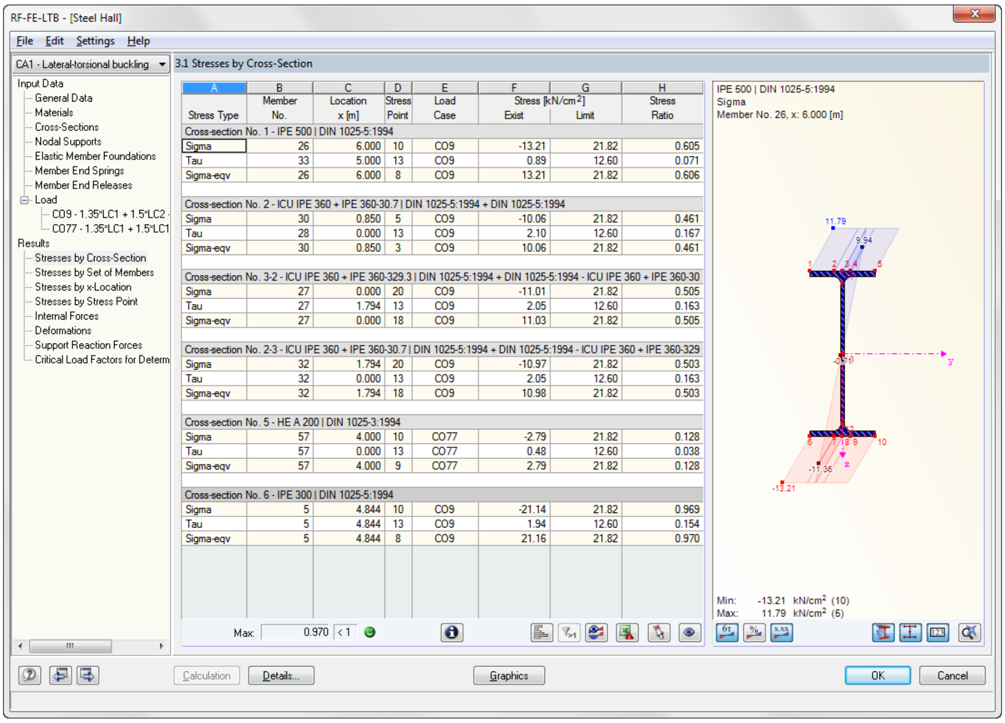


Figure 5.1: Results window with stresses in table and cross-section graphic

The stresses are sorted in the results windows 3.1 through 3.4 by various criteria.

Windows 3.5 and 3.6 list the internal forces and deformations of the sets of members. Window 3.7 provides information on the support forces.

The final window, 3.8, shows the critical load factors. In this window, you should check if all critical load factors are greater than or equal to 1: Only then is the stability of the system ensured!

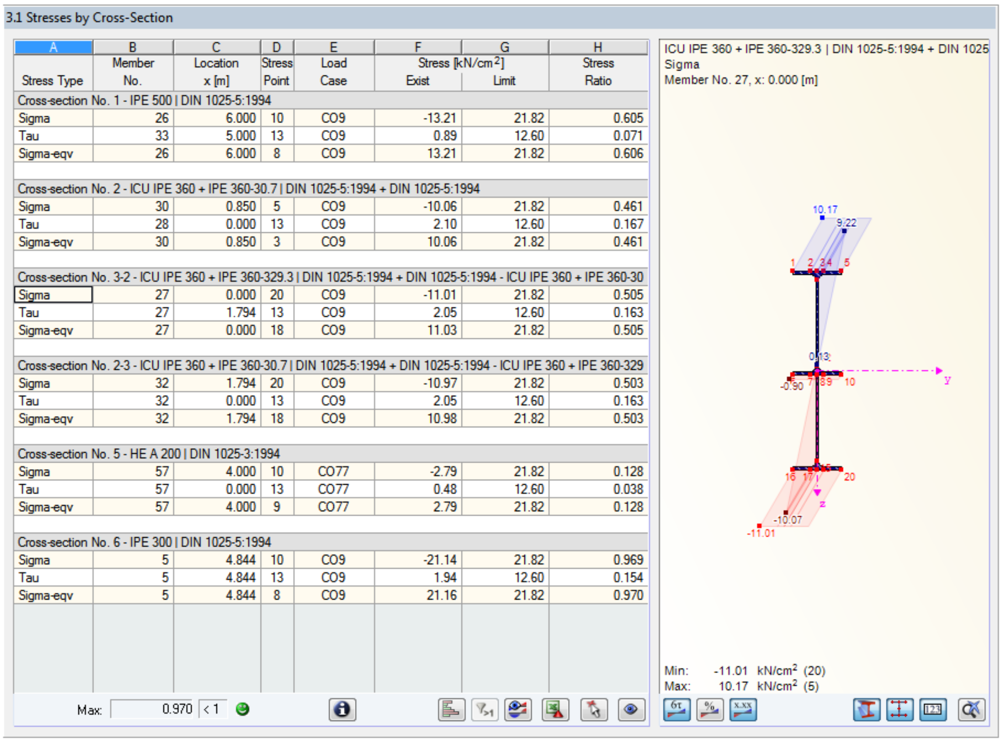
To go directly to a module window, click the according entry in the navigator. To go to the previous or next module window, use the buttons shown on the left. You can also browse through the windows using the function keys [F2] and [F3].

To save the results, click [OK]. Thus, you exit RF-FE-LTB and return to the main program.

Chapter 5 *Results* presents the results windows one by one. For details on evaluating and checking the results, see chapter 6 *Results Evaluation*, page 90f.

5.1 Stresses by Cross-Section

This window shows for all analyzed cross-sections the maximum stress ratios that result from the loadings of the governing load cases and load combinations.



A Stress Type	B Member No.	C Location x [m]	D Stress Point	E Load Case	F Stress [kN/cm²] Exist	G Stress [kN/cm²] Limit	H Stress Ratio
Cross-section No. 1 - IPE 500 | DIN 1025-5:1994							
Sigma	26	6.000	10	CO9	-13.21	21.82	0.605
Tau	33	5.000	13	CO9	0.89	12.60	0.071
Sigma-eqv	26	6.000	8	CO9	13.21	21.82	0.606
Cross-section No. 2 - ICU IPE 360 + IPE 360-30.7 | DIN 1025-5:1994 + DIN 1025-5:1994							
Sigma	30	0.850	5	CO9	-10.06	21.82	0.461
Tau	28	0.000	13	CO9	2.10	12.60	0.167
Sigma-eqv	30	0.850	3	CO9	10.06	21.82	0.461
Cross-section No. 3-2 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN 1025-5:1994 - ICU IPE 360 + IPE 360-30							
Sigma	27	0.000	20	CO9	-11.01	21.82	0.505
Tau	27	1.794	13	CO9	2.05	12.60	0.163
Sigma-eqv	27	0.000	18	CO9	11.03	21.82	0.505
Cross-section No. 2-3 - ICU IPE 360 + IPE 360-30.7 | DIN 1025-5:1994 + DIN 1025-5:1994 - ICU IPE 360 + IPE 360-329							
Sigma	32	1.794	20	CO9	-10.97	21.82	0.503
Tau	32	0.000	13	CO9	2.05	12.60	0.163
Sigma-eqv	32	1.794	18	CO9	10.98	21.82	0.503
Cross-section No. 5 - HE A 200 | DIN 1025-3:1994							
Sigma	57	4.000	10	CO77	-2.79	21.82	0.128
Tau	57	0.000	13	CO77	0.48	12.60	0.038
Sigma-eqv	57	4.000	9	CO77	2.79	21.82	0.128
Cross-section No. 6 - IPE 300 | DIN 1025-5:1994							
Sigma	5	4.844	10	CO9	-21.14	21.82	0.969
Tau	5	4.844	13	CO9	1.94	12.60	0.154
Sigma-eqv	5	4.844	8	CO9	21.16	21.82	0.970

Figure 5.2: Window 3.1 *Stresses by Cross-Section*

The list is sorted by cross-section. For a taper, the table shows both cross-section descriptions.

Stress Type

RF-FE-LTB analyzes the following stress types:

- Normal stresses σ
- Shear stresses τ
- Equivalent stresses σ_{eqv}

The normal stress ***Sigma*** is determined from the stress components of the axial force N, bending moments M_y and M_z, as well as the warping torsional moment M_ω (see Equation 2.4, page 20).

The shear stress ***Tau*** is determined from the shear forces V_y and V_z and the torsional moment M_T (see Equation 2.5, page 20).

The equivalent stress ***Sigma$_{eqv}$*** is determined from the parts of normal stress σ and shear stress τ (see Equation 2.7, page 21).

Member No.

This column shows the respective number of the member with the highest stress ratio.

Location x

At this x-location of the member, the maximum ratio occurs. For the tabular and graphical output, the following member locations *x* are used:

- Start and end node
- FE division points

This reflects the FE length that is specified in the *Details* dialog box (see chapter 4.1, page 77). For the calculation, the members are divided into finite elements acc. to the specification.

The member and FE divisions in RFEM are of no relevance for RF-FE-LTB.

Stress Point

The design is carried out on so-called stress points of the cross-section. The locations are defined by centroidal distances, statical moments, and thicknesses of the cross-section parts, which allow for a design according to Equation 2.4 and Equation 2.5.

All standard cross-sections of the library as well as the SHAPE-THIN and SHAPE-MASSIVE cross-sections have stress points at the design-relevant locations of the cross-section. For user-defined cross-sections, you must specify the parameters of the stress points manually to enable a design in STEEL.

In the cross-section graphic on the right, the stress points are shown with their numbers. The current stress point (that is, the stress point of the currently selected row) is highlighted in red.

To control the properties of the stress points, click [Info] (see chapter 6.1, page 92).

The normal and shear stresses are determined at each single stress point. For the equivalent stresses, it is therefore necessary to consider the stress components at which the same stress points exist. Therefore, it is usually not correct to superimpose the maximum stresses σ and τ shown in window 2.1: Usually, they occur at different stress points! You can view and evaluate specific results in window 3.4 *Stresses by Stress Point* (see chapter 5.4, page 84).

Load Case

This column shows the number of the load cases and load combinations whose loads result in maximum utilization ratios.

Stress Exist

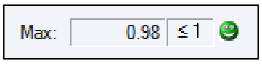
Column F shows the extreme values of the existing stresses determined according to Equation 2.4, Equation 2.5, and Equation 2.7 (see pages 20 and 21).

Stress Limit

This column shows the limit stresses of window 1.2 (see chapter 3.2, page 43). In particular, this includes the following limit stresses:

- Limit normal stress σ as the allowable stress for the stress due to shear force, bending, and warping
- Limit shear stress τ as allowable shear stress due to shear force and torsion
- Limit equivalent stress σ_{eqv} as the allowable equivalent stress for the simultaneous action of normal and shear stresses

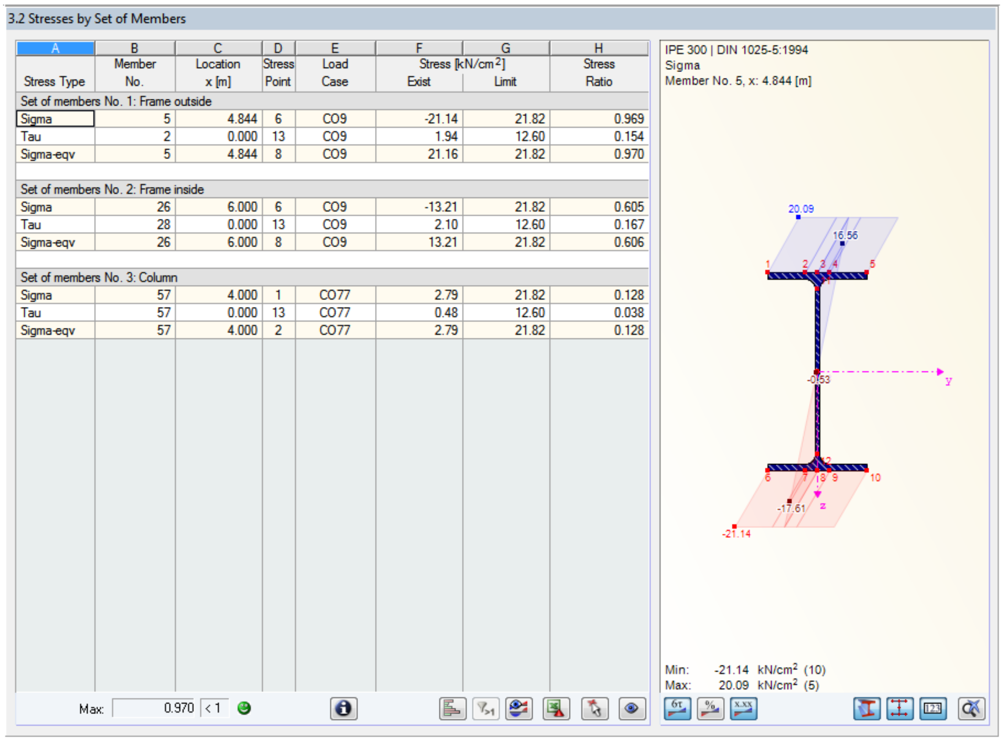
Stress Ratio



The last column shows the quotient from existing and limit stress. If the limit stress is not exceeded, then the ratio is less than or equal to 1 and the stress check is satisfied.

The length of the colored scale presents the respective ratio in graphical form.

5.2 Stresses by Set of Members

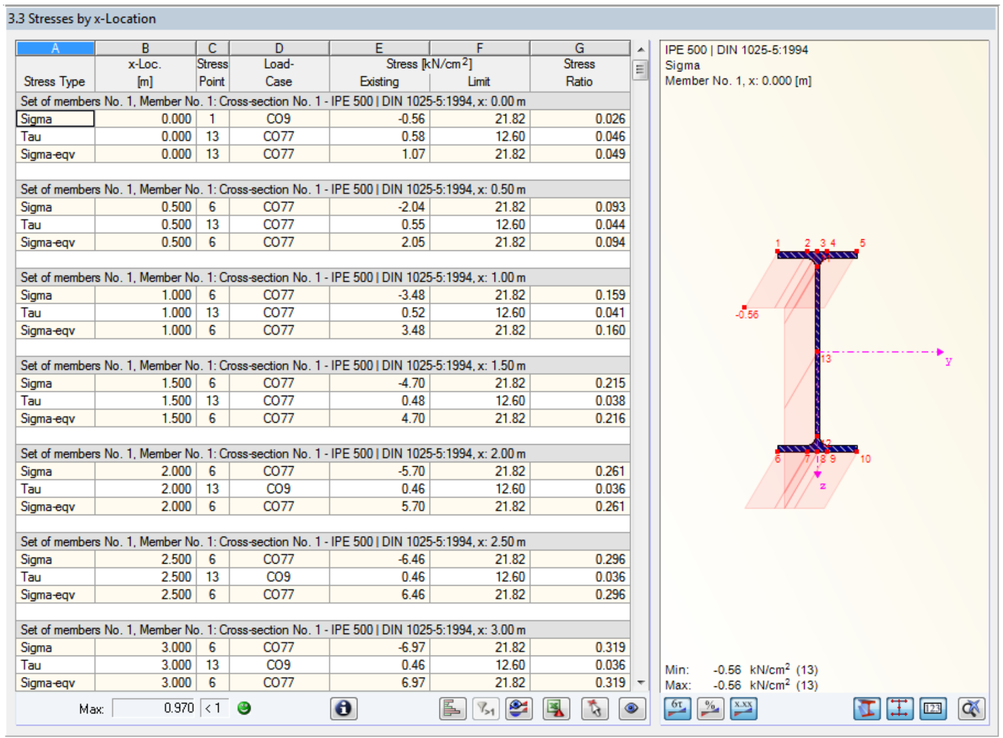


Stress Type	Member No.	Location x [m]	Stress Point	Load Case	Stress [kN/cm²] Exist	Limit	Stress Ratio
Set of members No. 1: Frame outside							
Sigma	5	4.844	6	CO9	-21.14	21.82	0.969
Tau	2	0.000	13	CO9	1.94	12.60	0.154
Sigma-eqv	5	4.844	8	CO9	21.16	21.82	0.970
Set of members No. 2: Frame inside							
Sigma	26	6.000	6	CO9	-13.21	21.82	0.605
Tau	28	0.000	13	CO9	2.10	12.60	0.167
Sigma-eqv	26	6.000	8	CO9	13.21	21.82	0.606
Set of members No. 3: Column							
Sigma	57	4.000	1	CO77	2.79	21.82	0.128
Tau	57	0.000	13	CO77	0.48	12.60	0.038
Sigma-eqv	57	4.000	2	CO77	2.79	21.82	0.128

Figure 5.3: Window 3.2 *Stresses by Set of Members*

This window lists the maximum stresses sorted by set of members.

5.3 Stresses by x-Location

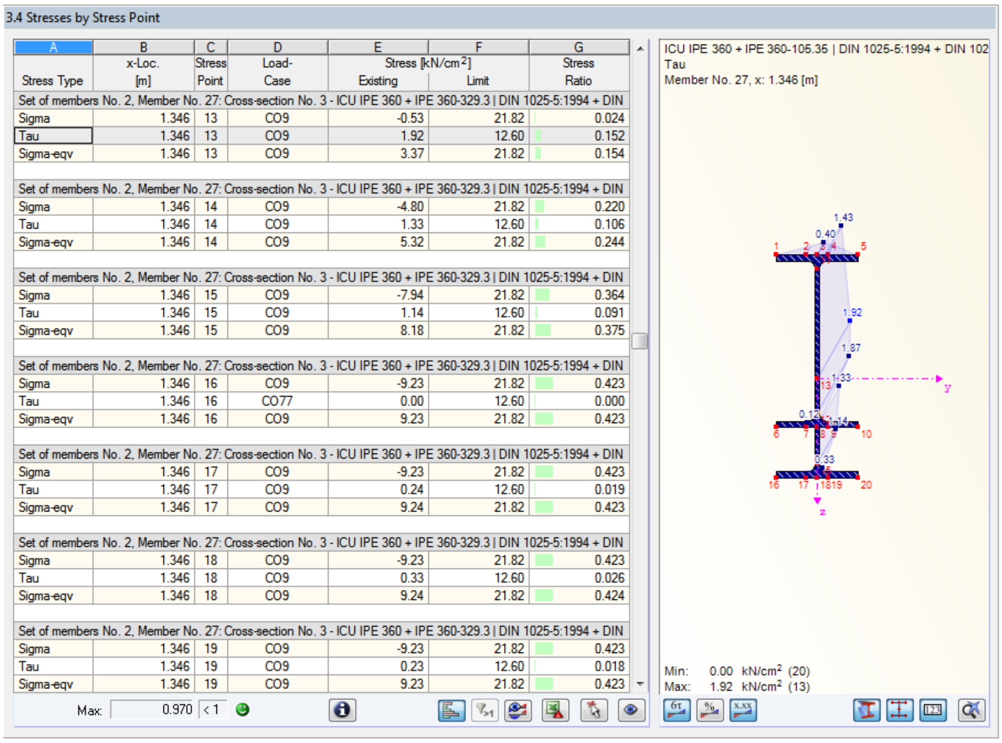


Stress Type	x-Loc. [m]	Stress Point	Load-Case	Stress [kN/cm²] Existing	Limit	Stress Ratio
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 0.00 m						
Sigma	0.000	1	CO9	-0.56	21.82	0.026
Tau	0.000	13	CO77	0.58	12.60	0.046
Sigma-eqv	0.000	13	CO77	1.07	21.82	0.049
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 0.50 m						
Sigma	0.500	6	CO77	-2.04	21.82	0.093
Tau	0.500	13	CO77	0.55	12.60	0.044
Sigma-eqv	0.500	6	CO77	2.05	21.82	0.094
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 1.00 m						
Sigma	1.000	6	CO77	-3.48	21.82	0.159
Tau	1.000	13	CO77	0.52	12.60	0.041
Sigma-eqv	1.000	6	CO77	3.48	21.82	0.160
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 1.50 m						
Sigma	1.500	6	CO77	-4.70	21.82	0.215
Tau	1.500	13	CO77	0.48	12.60	0.038
Sigma-eqv	1.500	6	CO77	4.70	21.82	0.216
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 2.00 m						
Sigma	2.000	6	CO77	-5.70	21.82	0.261
Tau	2.000	13	CO9	0.46	12.60	0.036
Sigma-eqv	2.000	6	CO77	5.70	21.82	0.261
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 2.50 m						
Sigma	2.500	6	CO77	-6.46	21.82	0.296
Tau	2.500	13	CO9	0.46	12.60	0.036
Sigma-eqv	2.500	6	CO77	6.46	21.82	0.296
Set of members No. 1, Member No. 1: Cross-section No. 1 - IPE 500 | DIN 1025-5:1994, x: 3.00 m						
Sigma	3.000	6	CO77	-6.97	21.82	0.319
Tau	3.000	13	CO9	0.46	12.60	0.036
Sigma-eqv	3.000	6	CO77	6.97	21.82	0.319

Figure 5.4: Window 3.3 *Stresses by x-Location*

This window lists the stresses that occur at the FE division points acc. to the specifications in the *Details* dialog box (see page 77). The individual columns are described in chapter 5.1.

5.4 Stresses by Stress Point



A	B	C	D	E	F	G
Stress Type	x-Loc. [m]	Stress Point	Load-Case	Stress [kN/cm²] Existing	Limit	Stress Ratio
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	13	CO9	-0.53	21.82	0.024
Tau	1.346	13	CO9	1.92	12.60	0.152
Sigma-eqv	1.346	13	CO9	3.37	21.82	0.154
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	14	CO9	-4.80	21.82	0.220
Tau	1.346	14	CO9	1.33	12.60	0.106
Sigma-eqv	1.346	14	CO9	5.32	21.82	0.244
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	15	CO9	-7.94	21.82	0.364
Tau	1.346	15	CO9	1.14	12.60	0.091
Sigma-eqv	1.346	15	CO9	8.18	21.82	0.375
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	16	CO9	-9.23	21.82	0.423
Tau	1.346	16	CO77	0.00	12.60	0.000
Sigma-eqv	1.346	16	CO9	9.23	21.82	0.423
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	17	CO9	-9.23	21.82	0.423
Tau	1.346	17	CO9	0.24	12.60	0.019
Sigma-eqv	1.346	17	CO9	9.24	21.82	0.423
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	18	CO9	-9.23	21.82	0.423
Tau	1.346	18	CO9	0.33	12.60	0.026
Sigma-eqv	1.346	18	CO9	9.24	21.82	0.424
Set of members No. 2, Member No. 27: Cross-section No. 3 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN						
Sigma	1.346	19	CO9	-9.23	21.82	0.423
Tau	1.346	19	CO9	0.23	12.60	0.018
Sigma-eqv	1.346	19	CO9	9.23	21.82	0.423

Figure 5.5: Window 3.4*Stresses by Stress Point*

The list is sorted for each member by *x-Location* and *Stress Point*. The individual columns of this window are described in chapter 5.1.

5.5 Internal Forces

3.5 Internal Forces

A	B	C	D	E	F	G	H	I	J	K
x-Loc. [m]	Load-Case	Forces [kN] N	V-y	V-z	Moments [kNm] M-T	M-y	M-z	Moments [kNm2, kNm] M-Om	M-Tpri	M-Tsec
Set of members No. 1, Member No. 1										
0.000	CO9	-65.40	0.26	-20.54	0.00	0.00	0.00	0.00	0.00	0.00
0.000	CO77	-44.12	5.25	-21.97	-0.14	0.00	0.00	0.00	-0.08	-0.06
0.500	CO9	-63.41	0.25	-20.54	0.00	-10.40	0.00	0.00	0.00	0.00
0.500 l	CO77	-42.15	4.45	-20.80	-0.14	-10.87	-2.25	-0.03	-0.08	-0.06
0.500 r	CO77	-42.15	4.45	-20.80	-0.14	-10.87	-2.25	-0.03	-0.08	-0.06
1.000	CO9	-61.42	0.24	-20.54	0.00	-20.79	0.00	0.00	0.00	0.00
1.000 l	CO77	-40.17	3.65	-19.61	-0.13	-21.15	-4.10	-0.06	-0.07	-0.06
1.000 r	CO77	-40.17	3.65	-19.61	-0.13	-21.15	-4.10	-0.06	-0.07	-0.06
1.500	CO9	-59.43	0.23	-20.54	0.00	-31.18	0.00	0.00	0.00	0.00
1.500 l	CO77	-38.21	2.84	-18.42	-0.11	-30.84	-5.56	-0.09	-0.06	-0.05
1.500 r	CO77	-38.21	2.84	-18.42	-0.11	-30.84		-0.09	-0.06	-0.05
2.000	CO9	-57.43	0.21	-20.54	0.00	-41.56	0.00	0.00	0.00	0.00
2.000 l	CO77	-36.24	2.03	-17.23	-0.09	-39.93	-6.61	-0.12	-0.05	-0.04
2.000 r	CO77	-36.24	2.03	-17.23	-0.09	-39.93	-6.61	-0.12	-0.05	-0.04
2.500	CO9	-55.44	0.19	-20.54	0.00	-51.93	0.00	0.00	0.00	0.00
2.500 l	CO77	-34.27	1.21	-16.04	-0.05	-48.42	-7.26	-0.13	-0.03	-0.02
2.500 r	CO77	-34.27	1.21	-16.04	-0.05	-48.42	-7.26	-0.13	-0.03	-0.02
3.000	CO9	-53.45	0.16	-20.54	0.00	-62.28	0.00	0.00	0.00	0.00
3.000 l	CO77	-32.30	0.39	-14.85	-0.01	-56.32	-7.49	-0.14	-0.01	0.00
3.000 r	CO77	-32.30	0.39	-14.85	-0.01	-56.32	-7.49	-0.14	-0.01	0.00
3.500	CO9	-51.46	0.14	-20.54	0.00	-72.63	0.00	0.00	0.00	0.00
3.500 l	CO77	-30.33	-0.43	-13.66	0.04	-63.63	-7.30	-0.13	0.01	0.04
3.500 r	CO77	-30.33	-0.43	-13.66	0.04	-63.63	-7.30	-0.13	0.01	0.04
4.000	CO9	-49.46	0.11	-20.54	0.00	-82.96	0.00	0.00	0.00	0.00
4.000 l	CO77	-28.36	-1.25	-12.48	0.10	-70.34	-6.70	-0.10	0.02	0.08
4.000 r	CO77	-28.36	-1.25	-12.48	0.10	-70.34	-6.70	-0.10	0.02	0.08
4.500	CO9	-47.47	0.08	-20.54	0.00	-93.28	0.00	0.00	0.00	0.00
4.500 l	CO77	-26.38	-2.07	-11.31	0.15	-76.46	-5.69	-0.05	0.04	0.12
4.500 r	CO77	-26.38	-2.07	-11.31	0.15	-76.46	-5.69	-0.05	0.04	0.12
5.000	CO9	-45.48	0.06	-20.54	0.00	-103.60	0.00	0.00	0.00	0.00
5.000 l	CO77	-24.39	-2.89	-10.14	0.20	-81.99	-4.26	0.02	0.04	0.16
5.000 r	CO77	-24.39	-2.89	-10.14	0.20	-81.99	-4.26	0.02	0.04	0.16
5.500	CO9	-43.48	0.03	-20.54	0.00	-113.90	0.00	0.00	0.00	0.00

Figure 5.6: Window 3.5 *Internal Forces*

For each member, this window shows the internal forces that are available for all analyzed load cases and load combinations at the FE division points.

x-Loc.

The following member *x*-locations are used for the tabular and graphical output:

- Start and end node
- FE division points

In case of discontinuities in the internal force diagram, the cut faces are indicated by *l* (left) or *r* (right).

Load Case

This column shows the numbers of the analyzed load cases and combinations.

Forces / Moments

The individual symbols of the internal forces have the following meanings:

N	Axial force
V_y	Shear force in direction of the local member axis y (or u)
V_z	Shear force in direction of the local member axis z (or v)
M_T	Torsional moment
M_y	Bending moment about the local member axis y (or u)
M_z	Bending moment about the local member axis z (or v)
M_ω	Warping torsional moment
M_{Tpri}	Primary torsional moment (St. Venant torsion)
M_{Tsec}	Secondary torsional moment (warping torsion)

Table 5.1: Internal forces

It is recommended to compare these internal forces with the internal force diagrams that are available in RFEM for the respective load cases and combinations. Thus, you can check whether the boundary conditions of the sets of members notionally singled out from the system are considered correctly (nodal support, loads, imperfections, etc.).

5.6 Deformations

3.6 Deformations

A x-Loc. [m]	B Load- Case	C Displacements [mm] u-X	D u-Y	E u-Z	F Rotations [mrad] Phi-X	G Phi-Y	H Phi-Z	I Warping Om [1/mm]
Set of members No. 1, Member No. 1								
0.000	CO9	0.00	0.00	0.00	0.000	4.042	0.000	0.00
0.000	CO77	0.00	0.00	0.00	3.654	12.070	0.000	0.00
0.500	CO9	-2.02	0.00	0.01	0.000	4.014	0.000	0.00
0.500	CO77	-6.03	1.80	0.01	3.513	12.040	0.596	0.00
1.000	CO9	-4.00	0.00	0.03	0.000	3.929	0.000	0.00
1.000	CO77	-12.03	3.47	0.02	3.120	11.950	1.160	0.00
1.500	CO9	-5.94	0.00	0.04	0.000	3.788	0.000	0.00
1.500	CO77	-17.97	4.89	0.03	2.525	11.810	1.661	0.00
2.000	CO9	-7.78	0.00	0.06	0.000	3.590	0.000	0.00
2.000	CO77	-23.83	5.97	0.04	1.775	11.620	2.068	0.00
2.500	CO9	-9.52	0.00	0.07	0.000	3.336	0.000	0.00
2.500	CO77	-29.58	6.65	0.04	0.922	11.380	2.355	0.00
3.000	CO9	-11.11	0.00	0.08	0.000	3.026	0.000	0.00
3.000	CO77	-35.20	6.88	0.05	0.015	11.090	2.504	0.00
3.500	CO9	-12.53	0.00	0.09	0.000	2.659	0.000	0.00
3.500	CO77	-40.67	6.66	0.06	-0.894	10.770	2.509	0.00
4.000	CO9	-13.76	0.00	0.10	0.000	2.237	0.000	0.00
4.000	CO77	-45.96	6.00	0.07	-1.756	10.400	2.381	0.00
4.500	CO9	-14.76	0.00	0.11	0.000	1.758	0.000	0.00
4.500	CO77	-51.07	4.92	0.07	-2.518	10.000	2.148	0.00
5.000	CO9	-15.51	0.00	0.13	0.000	1.223	0.000	0.00
5.000	CO77	-55.96	3.50	0.08	-3.131	9.573	1.862	0.00
5.500	CO9	-15.97	0.00	0.14	0.000	0.632	0.000	0.00
5.500	CO77	-60.63	1.83	0.08	-3.545	9.113	1.595	0.00
6.000	CO9	-16.13	0.00	0.14	0.000	-0.014	0.000	0.00
6.000	CO77	-65.07	0.00	0.09	-3.712	8.629	1.444	0.00
Set of members No. 1, Member No. 2								
0.000	CO9	-16.13	0.00	0.15	0.000	-0.015	0.000	0.00
0.000	CO77	-65.50	0.18	-2.07	-3.710	8.629	1.444	0.00
0.484	CO9	-15.98	0.00	0.94	0.000	-3.235	0.000	0.00
0.484	CO77	-66.21	0.45	-5.56	-3.825	6.134	1.233	0.00
0.969	CO9	-15.56	0.00	3.15	0.000	-5.944	0.000	0.00
0.969	CO77	-66.69	0.62	-7.94	-3.995	3.897	1.061	0.00

Figure 5.7: Window 3.6 *Deformations*

For each member, the deformations are shown that exist for all designed load cases and load combinations at the FE division points. They are relative to the centroid of the cross-section; RF-FE-LTB does not determine any local cross-section deformations.

x-Loc.

The output is sorted by x-location (start and end nodes, FE division points).

Load Case

This column shows the numbers of the analyzed load cases and combinations.

Displacements / Rotations / Warping

The symbols of the deformations have the following meanings:

u_X	Displacement in direction of the global X-axis
u_Y	Displacement in the direction of the global Y-axis
u_Z	Displacement in the direction of the global Z-axis
φ_X	Rotation about the global X-axis
φ_Y	Rotation about the global Y-axis
φ_Z	Rotation about the global Z-axis
ω	Warping

Table 5.2: Deformations

5.7 Support Reaction Forces

3.7 Support Reaction Forces

A	B	C	D	E	F	G	H	I
Node No.	Load Case	Forces [kN] P-X	P-Y	P-Z	Moments [kNm, kNm²] M-X	M-Y	M-Z	M-Om
Set of members No. 1								
1 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	-0.04
1	CO9	-20.59	0.69	65.41	0.00	0.00	0.39	0.00
1 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	0.01
1	CO77	-21.80	4.32	44.26	0.00	0.00	-0.07	0.00
2 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	-0.01
2	CO9	0.00	-0.14	0.00	0.00	0.00	0.00	0.00
2 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	CO77	0.00	5.85	0.00	0.00	0.00	0.00	0.00
3 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	-0.21
3	CO9	0.00	-1.48	0.00	0.00	0.00	0.00	0.00
3 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	-0.11
3	CO77	0.00	-1.36	0.00	0.00	0.00	0.00	0.00
4 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	1.15
4	CO9	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	-0.22
4	CO77	0.00	0.93	0.00	0.00	0.00	0.00	0.00
5 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	-0.21
5	CO9	0.00	1.48	0.00	0.00	0.00	0.00	0.00
5 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	0.11
5	CO77	0.00	-0.53	0.00	0.00	0.00	0.00	0.00
6 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	-0.01
6	CO9	0.00	0.14	0.00	0.00	0.00	0.00	0.00
6 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	0.00
6	CO77	0.00	5.00	0.00	0.00	0.00	0.00	0.00
7 (F)	CO9	0.00	0.00	0.00	0.00	0.00	0.00	-0.05
7	CO9	20.59	-0.69	70.15	0.00	0.00	0.39	0.00
7 (F)	CO77	0.00	0.00	0.00	0.00	0.00	0.00	0.00
7	CO77	-6.04	4.67	40.29	0.00	0.00	0.00	0.00
Set of members No. 2								
29	CO9	-43.47	0.00	108.30	0.00	0.00	0.00	0.00
29	CO77	-29.24	0.00	57.59	0.00	0.00	0.00	0.00
30	CO9	0.00	0.00	0.00	0.00	0.00	0.00	0.00
30	CO77	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Figure 5.8: Window 3.7 *Support Reaction Forces*

Window 3.7 shows the support forces at the individual nodal supports. These are the forces and moments that are introduced to the supports. Thus the signs of the values are not the reaction forces from the supports.

Nodes No.

This column lists the numbers of all nodes to which you assigned support properties in the 1.4 *Nodal Loads* window (see chapter 3.4, page 49).

Nodal supports with spring properties are indicated by an (F).

Load Case

This column shows the numbers of the analyzed load cases and combinations.

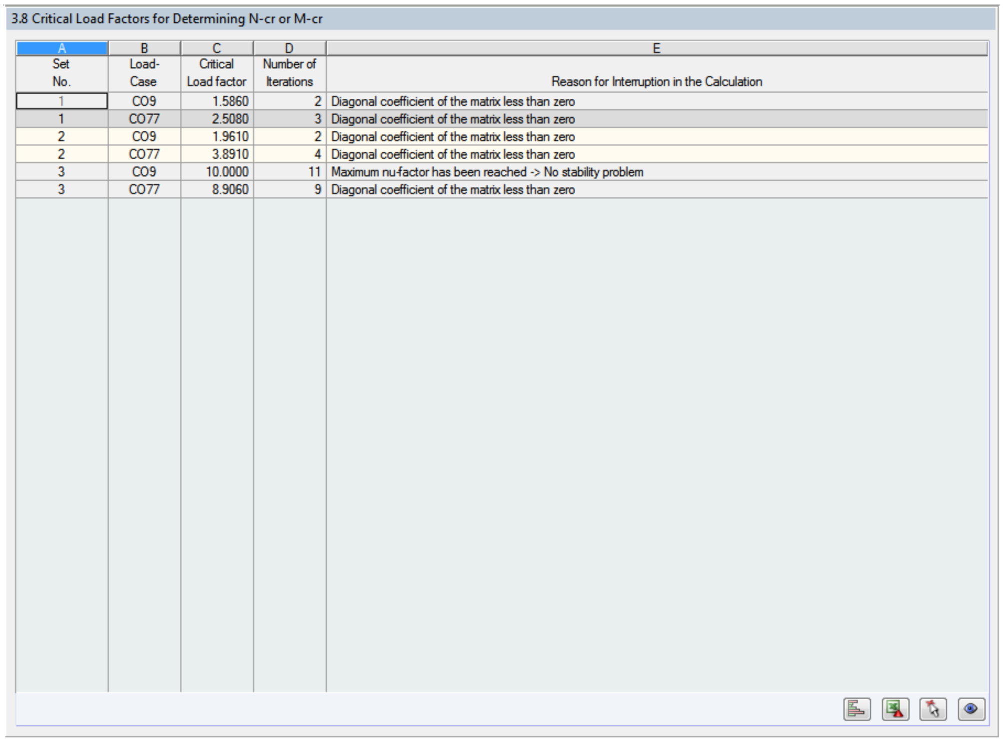
Forces / Moments

The symbols of the support forces have the following meanings:

P_X	Support force in the direction of the global X-axis
P_Y	Support force in the direction of the global Y-axis
P_Z	Support force in the direction of the global Z-axis
M_X	Support moment about the global X-axis
M_Y	Support moment about the global Y-axis
M_Z	Support moment about the global Z-axis
M_ω	Warping torsional moment at support

Table 5.3: Support forces and moments

5.8 Critical Load Factors


3.8 Critical Load Factors for Determining N-cr or M-cr

A Set No.	B Load-Case	C Critical Load factor	D Number of Iterations	E Reason for Interruption in the Calculation
1	CO9	1.5860	2	Diagonal coefficient of the matrix less than zero
1	CO77	2.5080	3	Diagonal coefficient of the matrix less than zero
2	CO9	1.9610	2	Diagonal coefficient of the matrix less than zero
2	CO77	3.8910	4	Diagonal coefficient of the matrix less than zero
3	CO9	10.0000	11	Maximum nu-factor has been reached -> No stability problem
3	CO77	8.9060	9	Diagonal coefficient of the matrix less than zero

Figure 5.9: Window 3.8 *Critical load factors for determining* N_{cr} *or* M_{cr}

The last results window allows you to evaluate the stability behavior of the sets of members: It shows the critical load factors that are available for all analyzed load cases and load combinations.

Set No.

The critical load factors are shown sorted by set of members.

Load Case

This column shows the numbers of the load cases and load combinations of those loads that result in the respective load factors.

Critical Load Factor

A critical load factor of for example *1.5860* (see Figure 5.9) means that this load combination has to be increased by the factor 1.5860 for the system to become instable (that is, the diagonal coefficient of the matrix is smaller than zero). An elastic behavior of the material is assumed.

If the critical load factor is smaller than 1, this means that the system has already become instable before the design load is reached. After the calculation, it is therefore also important to check if, in addition to the stress ratio (see window 3.1), all critical load factors are greater than or equal to 1.

If RF-FE-LTB determines a critical load factor of 0 during the iterations, a convergence can be reached: This means, there is a general instability and the *Calculation Errors and Notes* window appears (see Figure 5.10). In this case, the definitions of supports and releases should be checked. The system notionally singled out from the RFEM model is mostly likely to be kinematic.

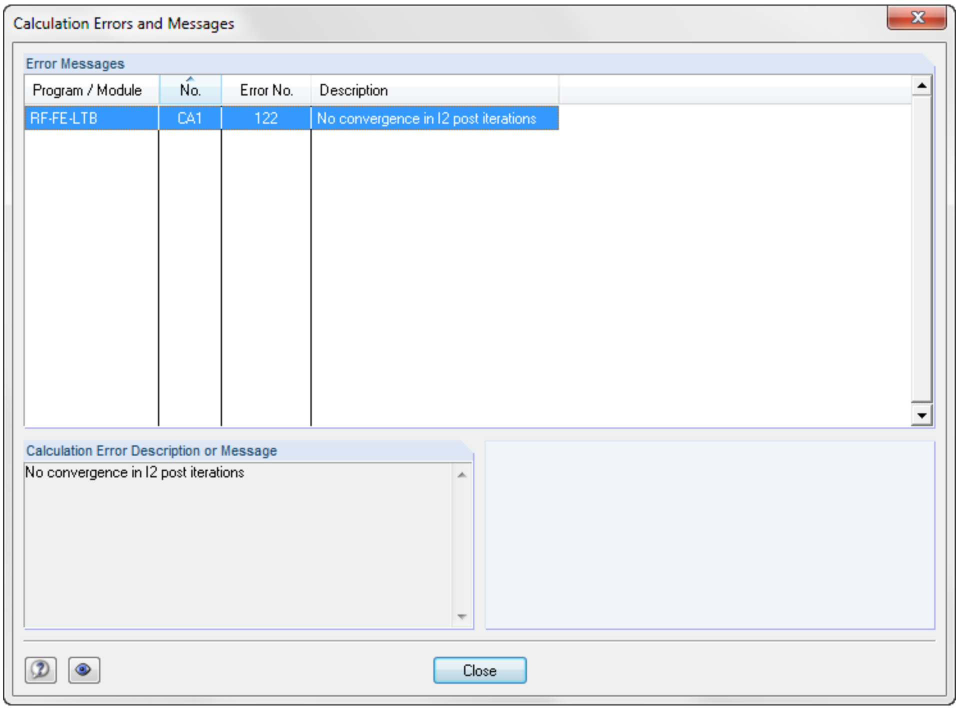


Figure 5.10: Window *Calculation Errors and Notes*

Number of Iterations

Thus column shows how many iterations it takes to reach the instability.

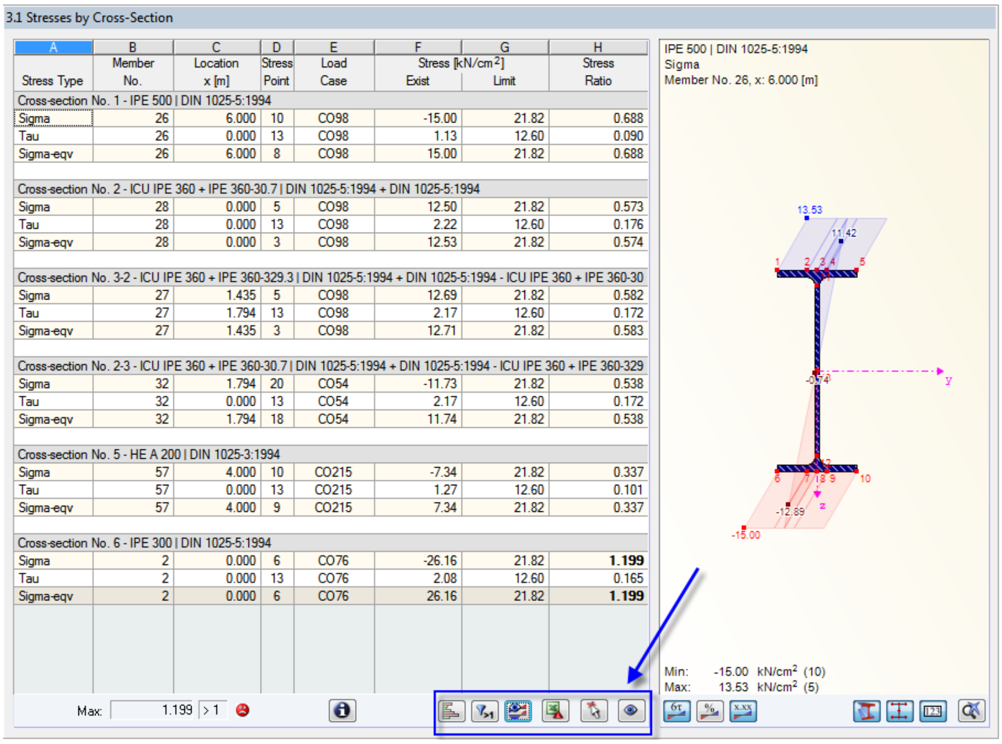
Reason for Interruption in the Calculation

The comments provide information about the stability behavior of individual sets of members.

In the *Details* dialog box, you can specify the iteration parameters (see Figure 4.1, page 77).

6. Results Evaluation

The design results can be evaluated in different ways. For this, the buttons at the end of the table are also useful.



3.1 Stresses by Cross-Section

Stress Type	Member No.	Location x [m]	Stress Point	Load Case	Stress [kN/cm²] Exist	Limit	Stress Ratio
Cross-section No. 1 - IPE 500 | DIN 1025-5:1994							
Sigma	26	6.000	10	CO98	-15.00	21.82	0.688
Tau	26	0.000	13	CO98	1.13	12.60	0.090
Sigma-eqv	26	6.000	8	CO98	15.00	21.82	0.688
Cross-section No. 2 - ICU IPE 360 + IPE 360-30.7 | DIN 1025-5:1994 + DIN 1025-5:1994							
Sigma	28	0.000	5	CO98	12.50	21.82	0.573
Tau	28	0.000	13	CO98	2.22	12.60	0.176
Sigma-eqv	28	0.000	3	CO98	12.53	21.82	0.574
Cross-section No. 3-2 - ICU IPE 360 + IPE 360-329.3 | DIN 1025-5:1994 + DIN 1025-5:1994 - ICU IPE 360 + IPE 360-30							
Sigma	27	1.435	5	CO98	12.69	21.82	0.582
Tau	27	1.794	13	CO98	2.17	12.60	0.172
Sigma-eqv	27	1.435	3	CO98	12.71	21.82	0.583
Cross-section No. 2-3 - ICU IPE 360 + IPE 360-30.7 | DIN 1025-5:1994 + DIN 1025-5:1994 - ICU IPE 360 + IPE 360-329							
Sigma	32	1.794	20	CO54	-11.73	21.82	0.538
Tau	32	0.000	13	CO54	2.17	12.60	0.172
Sigma-eqv	32	1.794	18	CO54	11.74	21.82	0.538
Cross-section No. 5 - HE A 200 | DIN 1025-3:1994							
Sigma	57	4.000	10	CO215	-7.34	21.82	0.337
Tau	57	0.000	13	CO215	1.27	12.60	0.101
Sigma-eqv	57	4.000	9	CO215	7.34	21.82	0.337
Cross-section No. 6 - IPE 300 | DIN 1025-5:1994							
Sigma	2	0.000	6	CO76	-26.16	21.82	**1.199**
Tau	2	0.000	13	CO76	2.08	12.60	0.165
Sigma-eqv	2	0.000	6	CO76	26.16	21.82	**1.199**

Figure 6.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Relation scale	Shows or hides the colored relation scales in the results window
	Exceeding	Shows only rows where the ratio is greater than 1, that is, the check is not fulfilled
	Result diagrams	Opens the *Result Diagram on Member* dialog box → chapter 6.3, page 97
	Excel export	Export the table to MS Excel / OpenOffice → chapter 8.3.3, page 106
	Member selection	Allows you to select a member graphically to show its results in the table
	View mode	Allows you to switch to the RFEM work window in order to change the view

Table 6.1: Buttons in the results windows

6.1 Results on Cross-Section

The tabular results are illustrated by a dynamic stress graphic. This graphic shows the stress diagram on the cross-section, which is present at the current x-location for the selected stress type. If you select a different x-location or type of stress by clicking it, the display is updated too. The governing stress point is highlighted in red.

In the graphic, you can show stresses as well as the stress ratios.

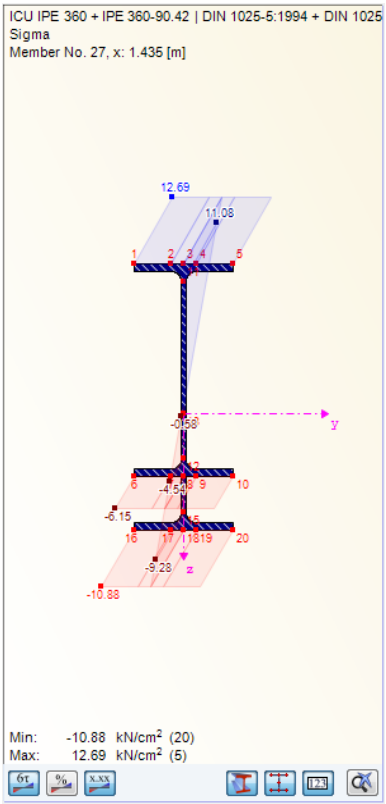


Figure 6.2: Distribution of the normal stresses on the cross-section

The buttons below the graphic have the following functions:

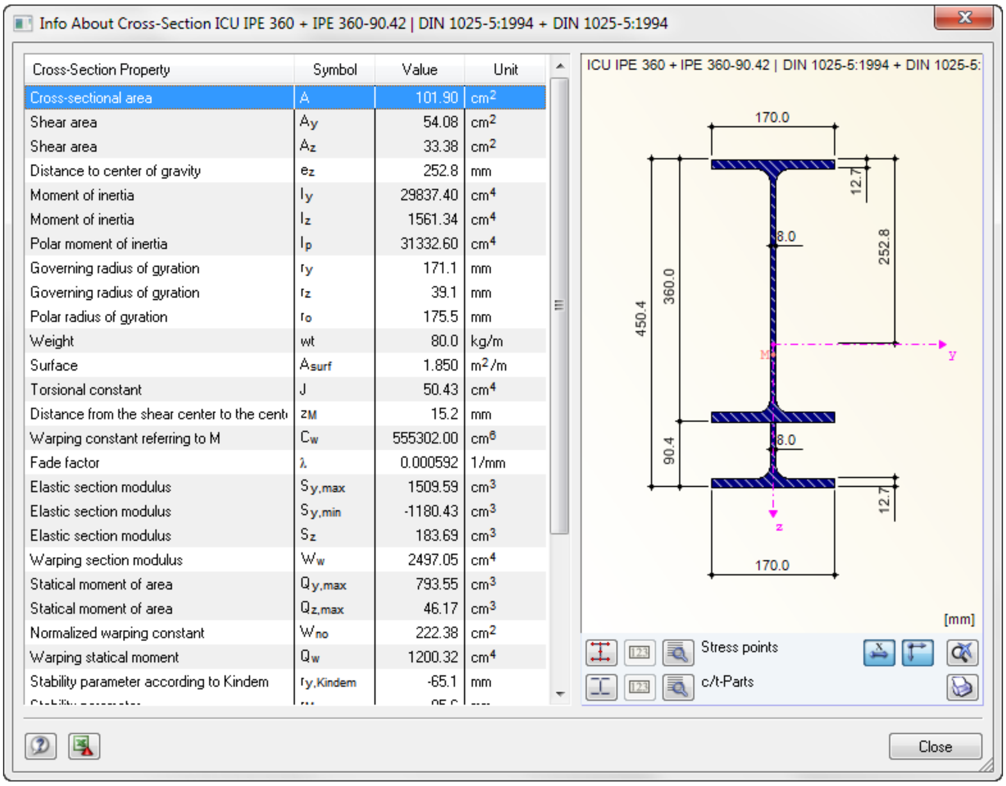
Button	Description	Function
	Stress diagram	Shows or hides the display of stresses
	Stress ratio	Shows or hides the ratios
	Values	Shows or hides the results values
	Cross-section outlines	Shows or hides the cross-section outlines
	Stress points	Shows or hides the stress points
	Numbering	Shows or hides the numbers of the stress points
	Show all graphic	Restores the total view of the results graphic

Table 6.2: Graphic button in the windows 3.1 through 3.4.

You can zoom in or out using the mouse wheel. To shift the stress graphic, you can use the drag-and-drop function. To return to the global view, click [Show all graphic].

Cross-section properties and stress points

To view the properties of the cross-sections and stress points, click [Info]. The *Info About Cross-Section* dialog box with the cross-section properties appears.

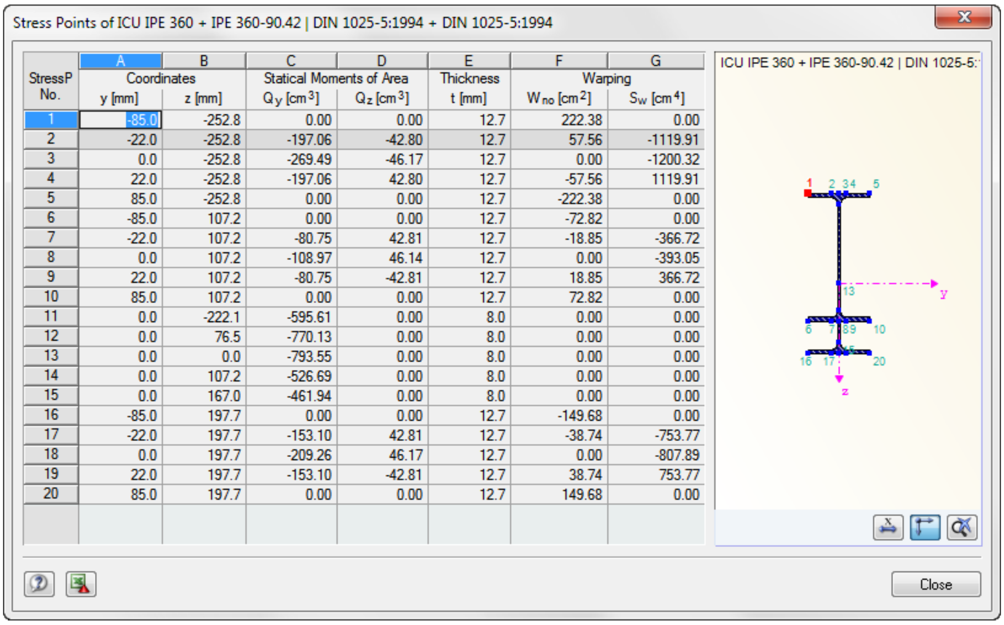


Info About Cross-Section ICU IPE 360 + IPE 360-90.42 | DIN 1025-5:1994 + DIN 1025-5:1994

Cross-Section Property	Symbol	Value	Unit
Cross-sectional area	A	101.90	cm²
Shear area	A_y	54.08	cm²
Shear area	A_z	33.38	cm²
Distance to center of gravity	e_z	252.8	mm
Moment of inertia	I_y	29837.40	cm⁴
Moment of inertia	I_z	1561.34	cm⁴
Polar moment of inertia	I_p	31332.60	cm⁴
Governing radius of gyration	r_y	171.1	mm
Governing radius of gyration	r_z	39.1	mm
Polar radius of gyration	r_o	175.5	mm
Weight	wt	80.0	kg/m
Surface	A_{surf}	1.850	m²/m
Torsional constant	J	50.43	cm⁴
Distance from the shear center to the cent	z_M	15.2	mm
Warping constant referring to M	C_w	555302.00	cm⁶
Fade factor	λ	0.000592	1/mm
Elastic section modulus	$S_{y,max}$	1509.59	cm³
Elastic section modulus	$S_{y,min}$	-1180.43	cm³
Elastic section modulus	S_z	183.69	cm³
Warping section modulus	W_w	2497.05	cm⁴
Statical moment of area	$Q_{y,max}$	793.55	cm³
Statical moment of area	$Q_{z,max}$	46.17	cm³
Normalized warping constant	W_{no}	222.38	cm²
Warping statical moment	Q_w	1200.32	cm⁴
Stability parameter according to Kindem	$r_{y,Kindem}$	-65.1	mm

Figure 6.3: Dialog box *Info About Cross-Section*

To view information on the stress points, click [Details].



Stress Points of ICU IPE 360 + IPE 360-90.42 | DIN 1025-5:1994 + DIN 1025-5:1994

StressP No.	A: Coordinates y [mm]	B: Coordinates z [mm]	C: Statical Moments of Area Q_y [cm³]	D: Q_z [cm³]	E: Thickness t [mm]	F: Warping W_{no} [cm²]	G: S_w [cm⁴]
1	-85.0	-252.8	0.00	0.00	12.7	222.38	0.00
2	-22.0	-252.8	-197.06	-42.80	12.7	57.56	-1119.91
3	0.0	-252.8	-269.49	-46.17	12.7	0.00	-1200.32
4	22.0	-252.8	-197.06	42.80	12.7	-57.56	1119.91
5	85.0	-252.8	0.00	0.00	12.7	-222.38	0.00
6	-85.0	107.2	0.00	0.00	12.7	-72.82	0.00
7	-22.0	107.2	-80.75	42.81	12.7	-18.85	-366.72
8	0.0	107.2	-108.97	46.14	12.7	0.00	-393.05
9	22.0	107.2	-80.75	-42.81	12.7	18.85	366.72
10	85.0	107.2	0.00	0.00	12.7	72.82	0.00
11	0.0	-222.1	-595.61	0.00	8.0	0.00	0.00
12	0.0	76.5	-770.13	0.00	8.0	0.00	0.00
13	0.0	0.0	-793.55	0.00	8.0	0.00	0.00
14	0.0	107.2	-526.69	0.00	8.0	0.00	0.00
15	0.0	167.0	-461.94	0.00	8.0	0.00	0.00
16	-85.0	197.7	0.00	0.00	12.7	-149.68	0.00
17	-22.0	197.7	-153.10	42.81	12.7	-38.74	-753.77
18	0.0	197.7	-209.26	46.17	12.7	0.00	-807.89
19	22.0	197.7	-153.10	-42.81	12.7	38.74	753.77
20	85.0	197.7	0.00	0.00	12.7	149.68	0.00

Figure 6.4: Dialog box *Stress Points*

The columns *Coordinates* show the centroidal distances y and z of the stress points. The columns *Statical Moments of Area* show the statical moments of area Q_y and Q_z (or Q_u and Q_v for unsymmetric cross-sections). The *Thickness* t of the structural component at the stress point is necessary for the determination of shear stresses.

The columns *Warping* provide information on the warping ordinates W_{no} and the warping statical moment S_w. For closed cross-sections, the last column shows the core area A* of the cell that is necessary to determine the Bredt torsional shear stresses.

6.2 Results on RFEM Model

For the evaluation, you can also use the RFEM work window.

RFEM background graphic and view mode

The RFEM work window in the background helps you find the position of a member in the model. In the background graphic, the member selected in the results window of RF-FE-LTB is highlighted by color. An arrow also indicates the x-location of the member selected in the table row.

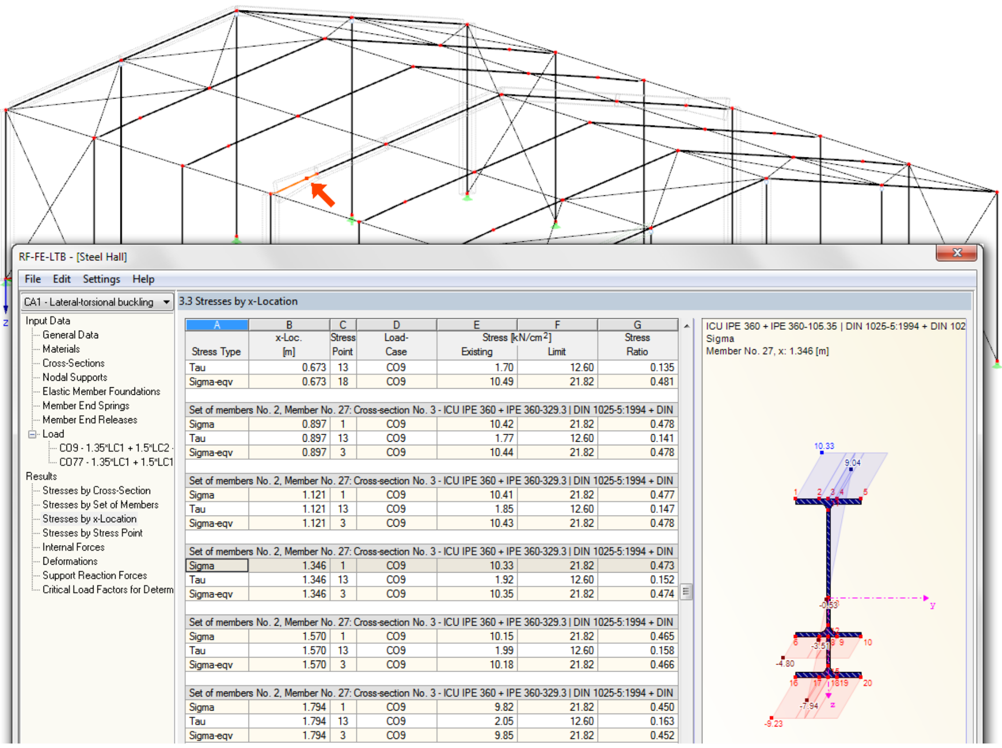
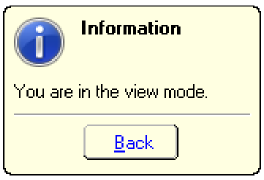


Figure 6.5: Indication of the member and the current *x-loc.* in the RFEM model



If it is not possible to improve the view by moving the RF-FE-LTB window, click [Jump to graphics to change view] to start the *View Mode*: Now, the module window is hidden so that you can adjust the view in the RFEM work window. In the view mode, you can use the functions of the *View* menu, for example, zoom, move, or rotate the view. The arrow remains visible.

To return to RF-FE-LTB, click [Back].

RFEM work window

The stresses, ratios, deformations, internal forces, and eigenvectors can also be evaluated graphically on the RFEM model: Click [Graphics] to quit RF-FE-LTB. The work window of RFEM now shows the results like the internal forces or deformations of a load case.

The *Results* navigator is adjusted to the checks in the RF-FE-LTB module. It provides various stress types, stress ratios, internal forces, and eigenvectors for selection.

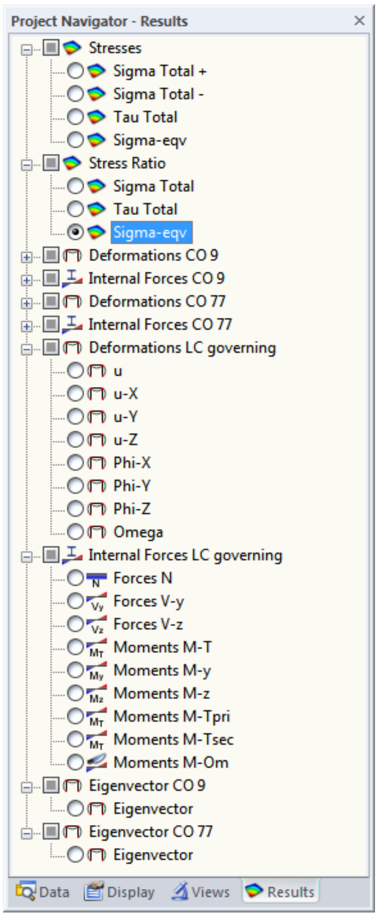
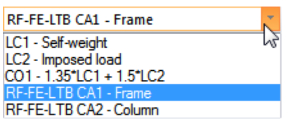


Figure 6.6: *Results* navigator for RF-FE-LTB

The normal stresses are shown separately by tensile (+) and compression stresses (-). Graphically, you can also evaluate the internal forces and deformations for each analyzed load case or each load combination.

Similarly to the display of internal forces, the [Show results] button displays or hides the design results. The [Show result values] button to the right controls the display of the result values.

Since the RFEM tables serve no function for the evaluation of the design results, you can hide them.



To set the design cases in RFEM, select them in the list located in the RFEM menu bar.

You can control the results display in the *View* navigator under the entry *Results→ Members*. By default, it shows the stresses, ratios, and internal forces as *Two-colored* and the deformations and eigenvectors as *Lines*.

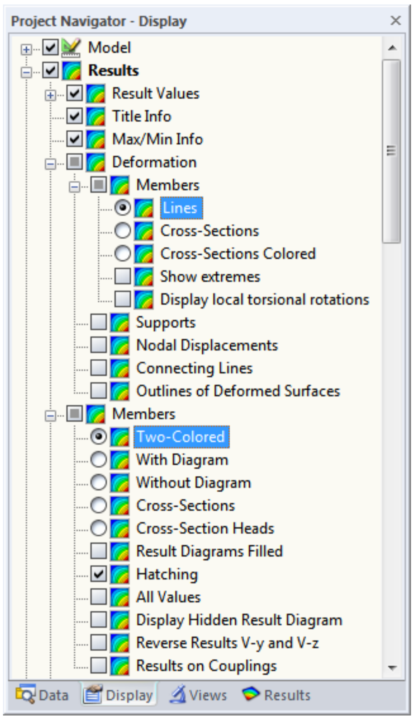


Figure 6.7: *Display* navigator: Results → Deformation / Members

In case of a multi-colored display (Options *Cross-Sections Colored* or *With/Without Diagram*) the color panel with the common control possibilities is available. These functions are described in chapter 3.4.6 of the RFEM manual.

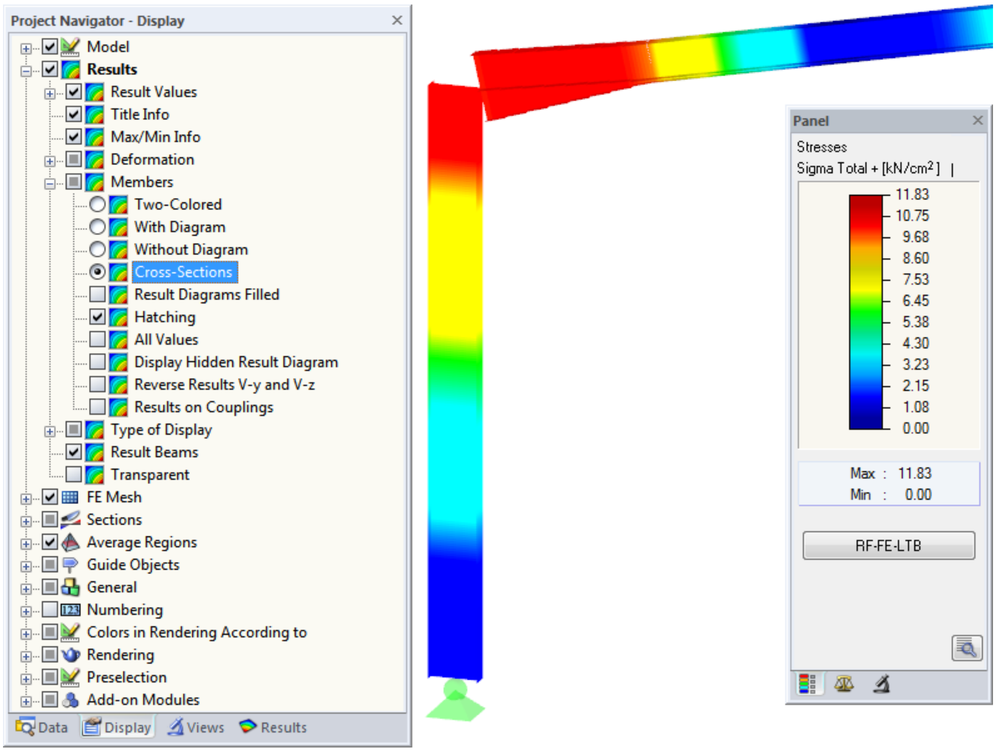


Figure 6.8: Normal stresses with display option *Cross-Sections*

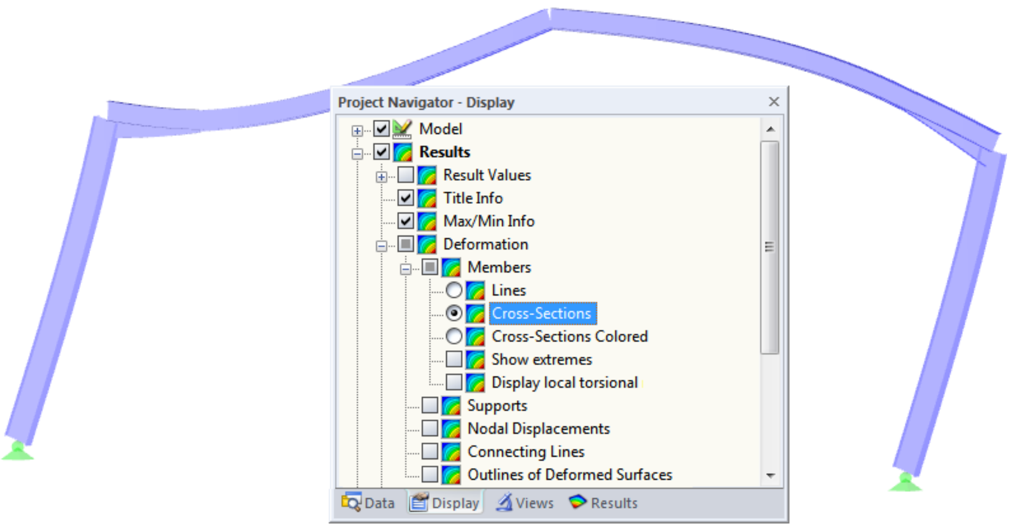


Figure 6.9: Eigenvector with display option *Cross-Sections*

You can export the graphics of the stresses, ratios, internal forces, and eigenvectors to the printout report (see chapter 7.2, page 100).

The eigenvectors and deformations are relative to the centroid of the cross-section. The program does not determine local cross-section deformations.

You can compare the internal forces of RF-FE-LTB with the diagrams of the internal forces that are available in RFEM for the respective load cases and combinations. Thus, you can check whether the boundary conditions of the sets of members notionally singled out from the system have been considered correctly (nodal supports, loads, imperfections, etc.)

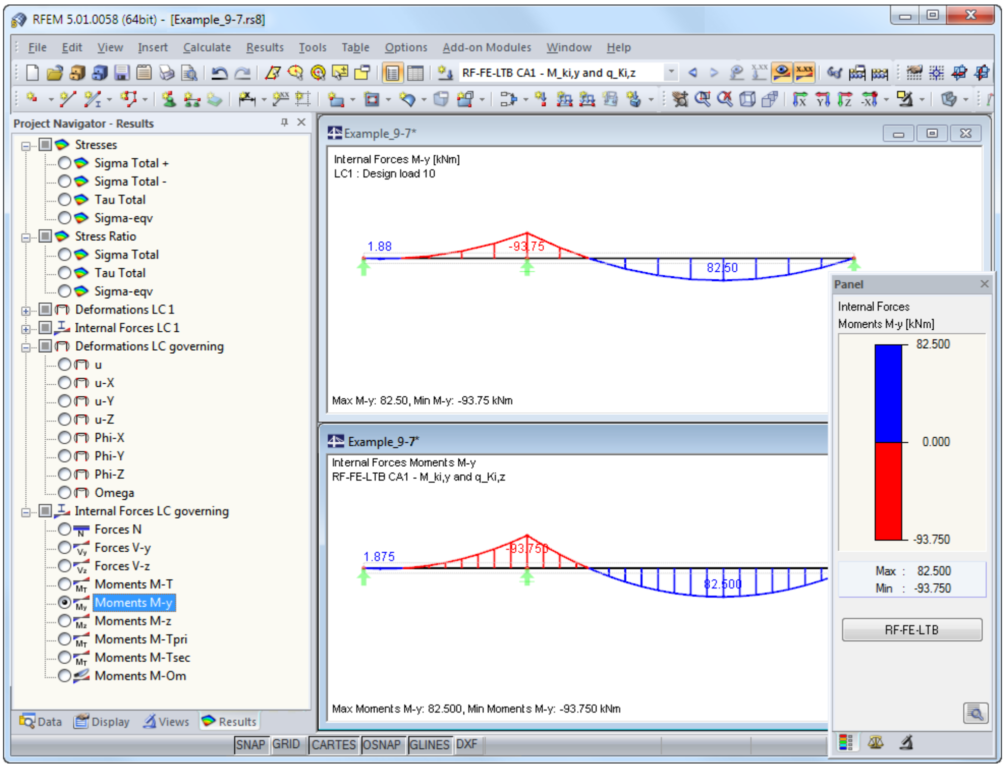


Figure 6.10: Moments M_y in RFEM (above) and RF-FE-LTB (below)

RF-FE-LTB

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

6.3 Result Diagrams

The result diagrams of a set of members can be evaluated graphically in the results diagram, too.

To do this, select the member or set of members in the RF-FE-LTB results window by clicking in the table row of the member. Then, open the *Result Diagram on Member* dialog box by clicking the button on the left. It is located at the bottom of the table (see Figure 6.1, page 90).

In the RFEM graphic, you can access the result diagrams by selecting

Results → Result Diagrams for Selected Members

or click the according button in the RFEM toolbar.

A window opens, graphically showing the diagram of the results on the member or set of members.

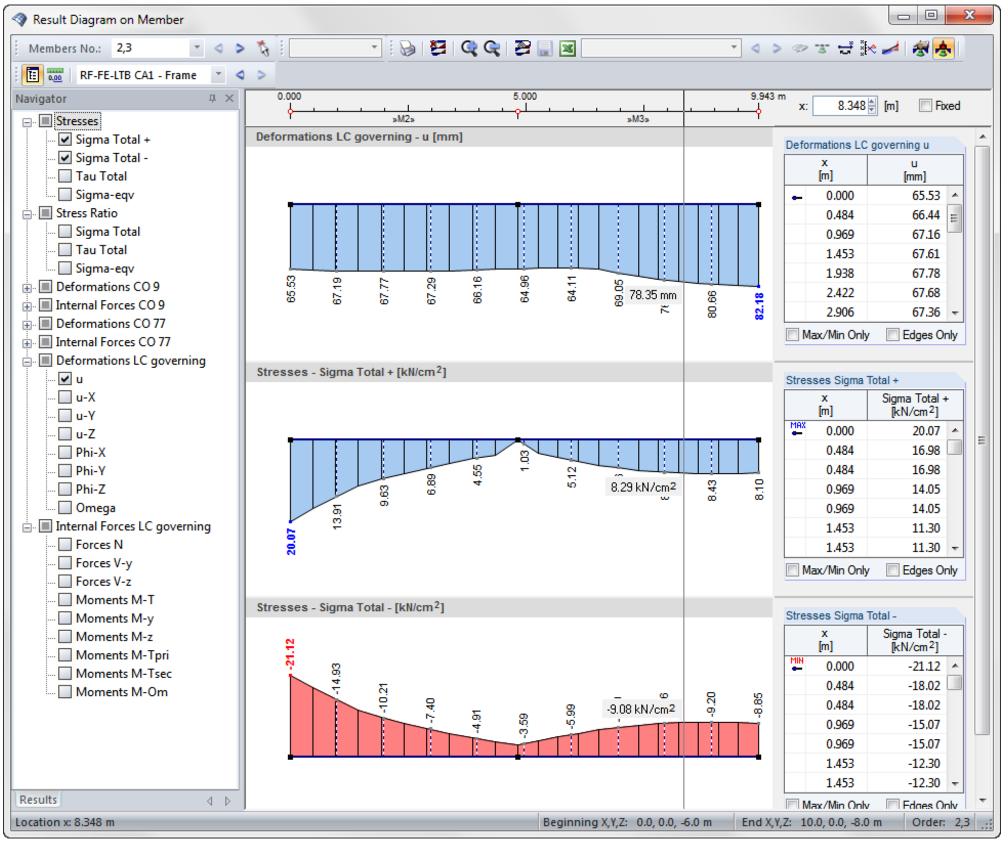
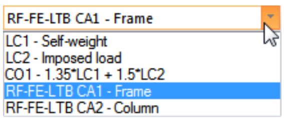


Figure 6.11: Dialog box *Result Diagram on Member*



In the navigator, you can select the stresses, ratios, deformations, internal forces, or eigenvectors that should be displayed in the results diagram. To switch between the design cases of RF-FE-LTB, use the list in the toolbar.

The *Result Diagram on Member* dialog box is described in chapter 9.5 of the RFEM manual.

6.4 Filter for Results

The RF-FE-LTB results windows allow you a selection by various criteria. In addition, you can use the filter options (described in chapter 9.9 of the RFEM manual) in order to evaluate the design results graphically.

In RF-FE-LTB, you can use the possibilities of the *Visibilities* (see RFEM manual, chapter 9.9.1) to filter the sets of members and members for the evaluation.

Filtering Checks

You can also easily use the stresses and ratios as filter criteria in the RFEM work window, which you can access by clicking [Graphics]. To do this, the panel must be shown. If it is not active, you can switch it on by using the RFEM menu

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

or click the according button in the toolbar.

The panel is described in chapter 3.4.6 of the RFEM manual. To specify the filter settings for the results, use the first panel tab (color scale). This tab is not available for the two-colored display. Hence, you must set in the *Display* navigator the display option *With/Without Diagram* or *Cross-Sections*.

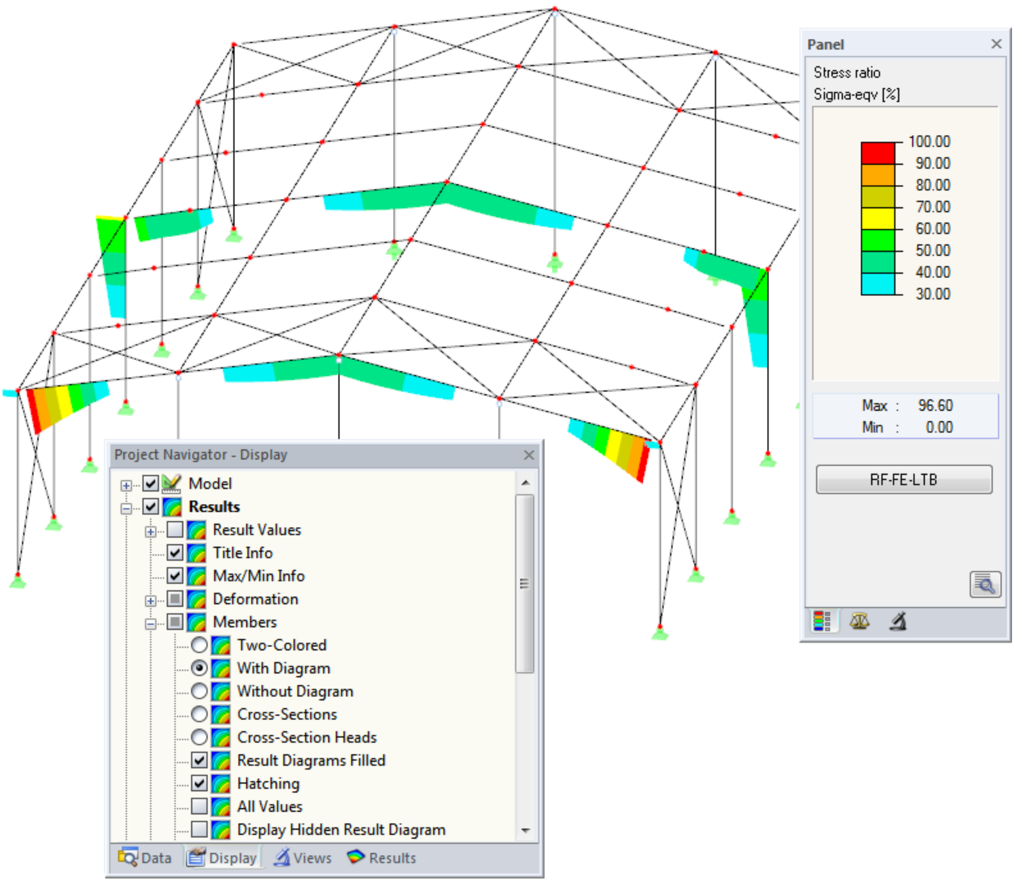


Figure 6.12: Filtering the utilization ratios with adjusted color scale

As the figure above shows, you can set the value scale of the panel in such a way that only utilization ratios greater than 30% are shown in the color interval between cyan and red.

To show all utilization distributions that are not covered by the value scale, select the *Display Hidden Result Diagram* check box in the *Display* navigator (*Results*→ *Members*). These distributions are represented by dotted lines.

Filtering members

In the *Filter* tab of the control panel, you can specify the numbers of selected members to filter the results. This function is described in chapter 9.9.3 of the RFEM manual.

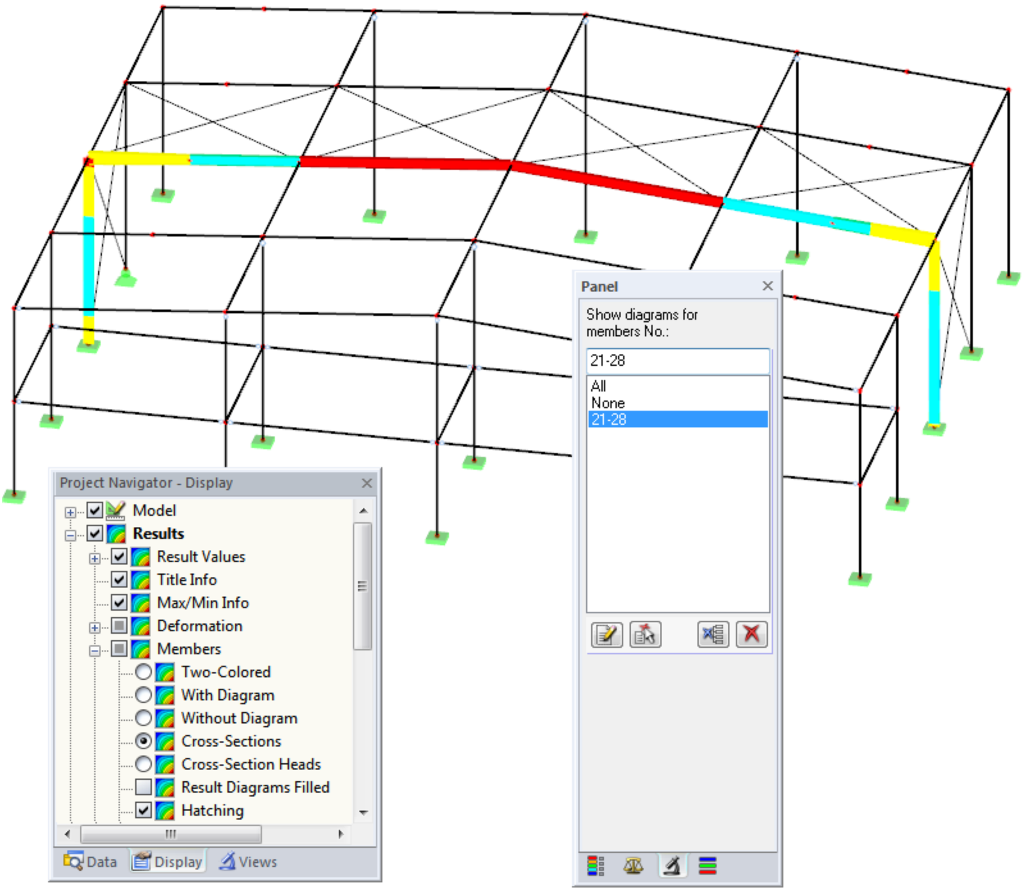


Figure 6.13: Member filter for utilization ratios of a frame

In contrast to the visibility function, the complete model is also shown. The figure above shows the utilizations of a set of members in a frame. The remaining members are shown in the model but without the utilization ratios.

7. Printout

7.1 Printout Report

For the data of the module RF-FE-LTB, a printout report is generated to which you can add graphics and descriptions, just like in RFEM. The selection in the printout report controls which data of the design module will finally appear in the printout.

The printout report is described in the RFEM manual. Chapter 10.1.3.4 *Selecting Data of Add-on Modules* explains how to prepare the output data of add-on modules for the printout.

For complicated systems with many design cases, splitting the data into several printout reports contributes to a clear overview.

7.2 Graphic Printout

You can import every picture shown in the work window to the printout report or send it directly to the printer. Thus, you can prepare the utilizations shown on the RFEM model for the printout.

Printing graphics is described in chapter 10.2 of the RFEM manual.

Analyses on the RFEM model

To print the currently displayed graphic of the stress ratios, use the menu

File → Print

or click the according button in the toolbar.

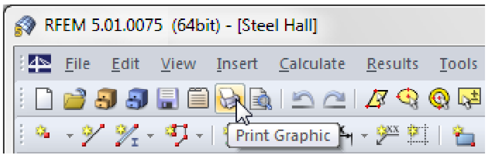


Figure 7.1: Button *Print* in RFEM toolbar

Result Diagrams

In the *Result Diagram on Member* dialog box, you can also add the graphic of stresses and stress ratios to the report by clicking [Print]. Moreover, you can print it directly.

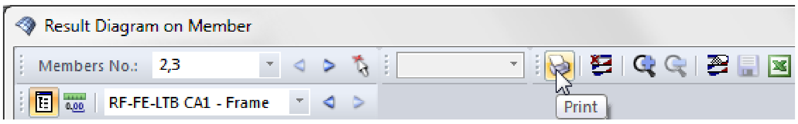


Figure 7.2: Button *Print* in the *Result Diagram on Member*

The following dialog box appears.

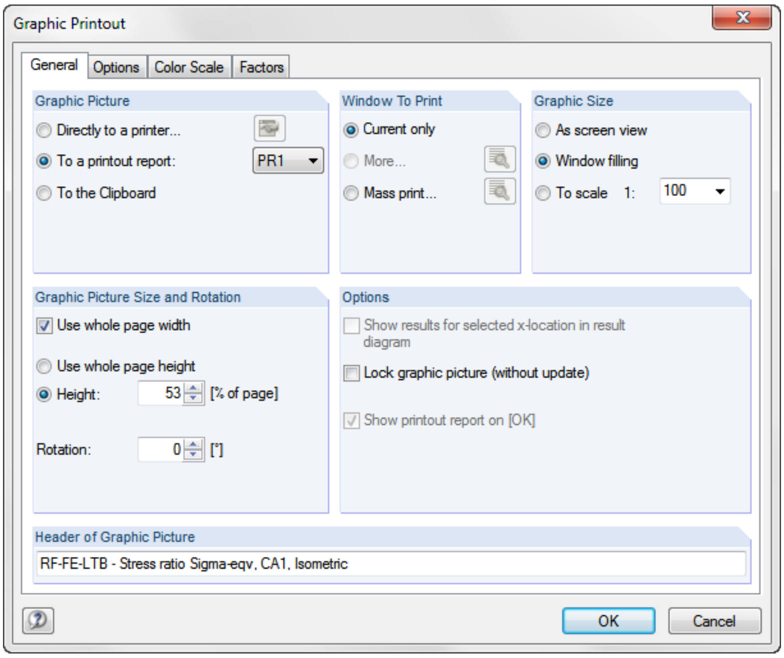
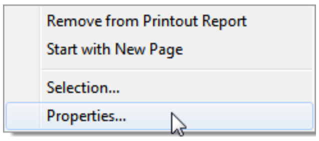


Figure 7.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in chapter 10.2 of the RFEM manual. There, you can also find information on the tabs *Options* and *Color Scale*.

To move the graphic to another place in the printout report, use the drag-and-drop function.



To adjust a graphic in the printout report subsequently, right-click the according entry in the navigator of the report. The *Properties* command in the context menu reopens the *Graphic Printout* dialog box where you can adjust the settings.

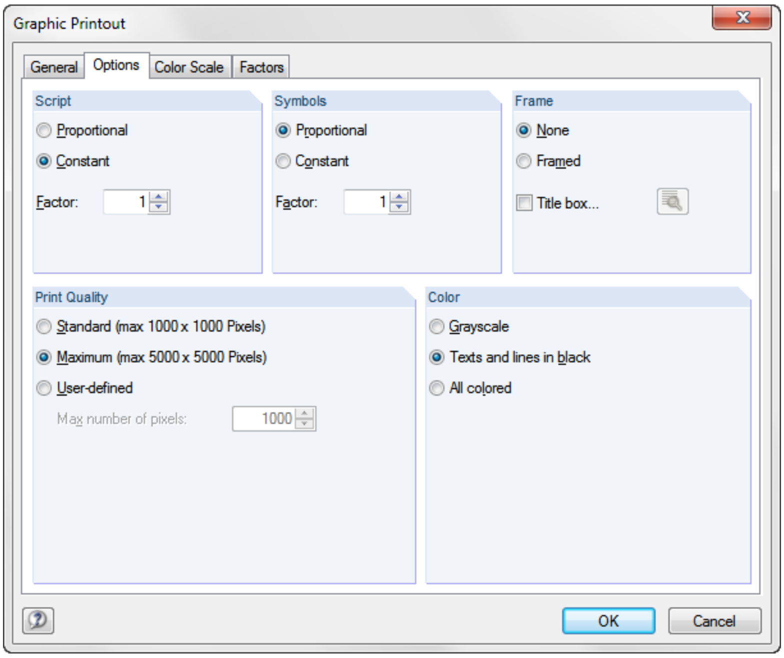


Figure 7.4: Dialog box *Graphic Printout*, tab *Options*

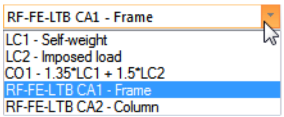
8. General Functions

This chapter describes useful menu functions and presents export options for the designs.

8.1 Design Cases

Design cases allow you to combine groups of structural components or sets of members with certain design specifications (for example modified materials, partial safety factors, or optimization).

It is no problem to analyze a set of members in different design cases.



In RFEM, you can also set the design cases of RF-FE-LTB using the load case list in the toolbar.

Create new design case

To create a new design case, select from the RF-FE-LTB menu

File → New Case.

The following dialog box appears.

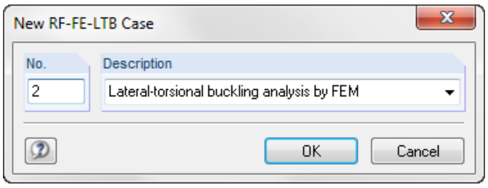


Figure 8.1: Dialog box *New RF-FE-LTB Case*

In this dialog box, you must enter a *No.* (one that is still free) for the new design case. The *Description* facilitates the selection in the load case list.

After you click [OK], the 1.1 *General Data* window of RF-FE-LTB appears where you enter the design data.

Rename design case

To change the name of the design case, select on the RF-FE-LTB menu

File → Rename Case.

The following dialog box appears.

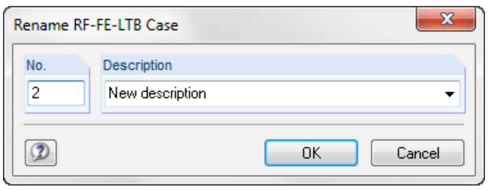


Figure 8.2: Dialog box *Rename RF-FE-LTB Case*

Here, you can specify a new *Description* as well as a different *No.* for the design case.

Copy design case

To copy the input data of the current design case, select on the RF-FE-LTB menu

File → Copy Case.

The following dialog box appears.

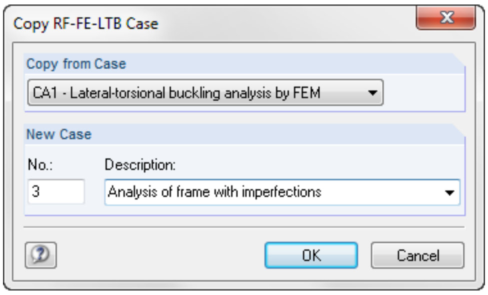


Figure 8.3: Dialog box *Copy RF-FE-LTB Case*

In this dialog box, you can specify the *Number* and, if necessary, a *Description* for the new case.

Delete design case

To delete a design case, select on the RF-FE-LTB menu

File → Delete Case.

The following dialog box appears.

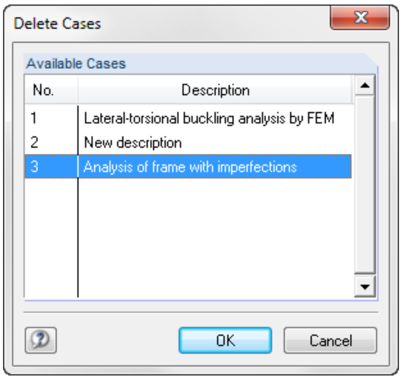


Figure 8.4: Dialog box *Delete Case*

You can select the design case from the *Available Cases* list. To delete the case, click [OK].

8.2 Units and Decimal Places

The units and decimal places are commonly managed for RFEM and for the add-on modules. In RF-FE-LTB, you can open the dialog box for the adjustment of units by selecting on the menu

Settings → **Units and Decimal Places**.

The dialog box known from RFEM appears. In the *Program / Module* section, RF-FE-LTB is preset.

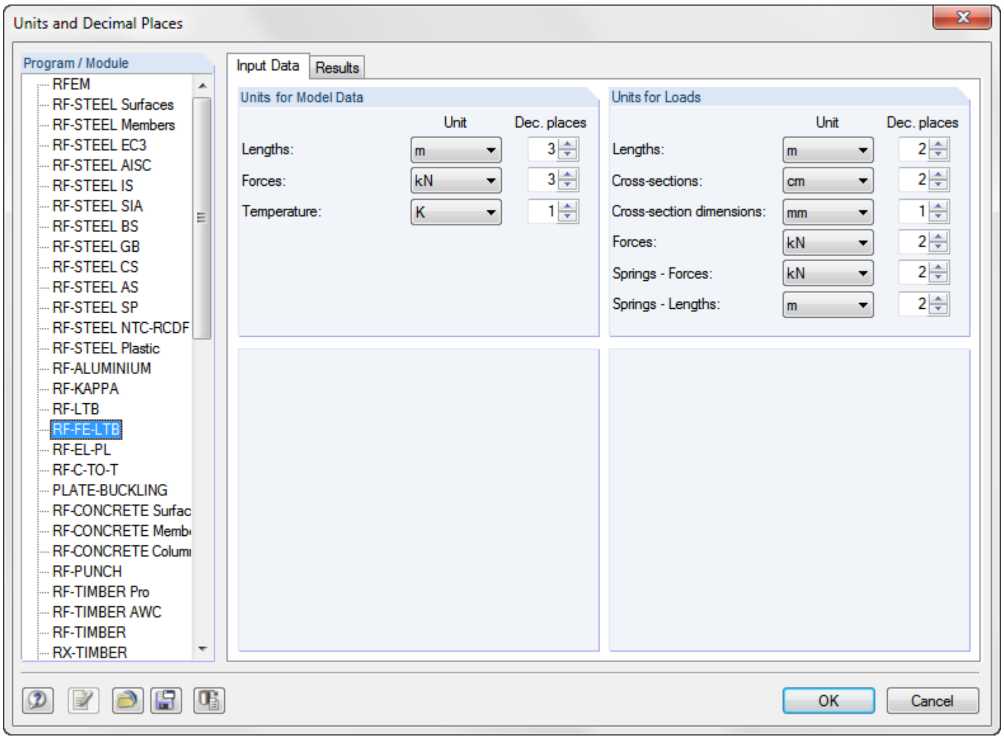


Figure 8.5: Dialog box *Units and Decimal Places*

For RF-FE-LTB, there are two tabs so that the specifications for the *Input Data* and the *Results* are done separately.

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

8.3 Data Exchange

8.3.1 Material Export to RFEM

If you adjust the materials for the design in RF-FE-LTB, you can export the modified materials to RFEM. Go to window 1.2 *Materials* and select on the menu

Edit → **Export All Materials to RFEM**.

Alternatively, you can open the context menu in window 1.2 to export the materials to RFEM.

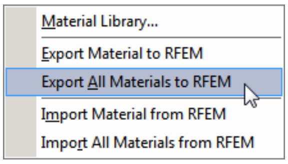


Figure 8.6: Context menu of window 1.2 *Materials*

Before the material is exported, a query appears as to whether the results of RFEM should be deleted.

If the changed materials have not been exported to RFEM yet, you can use the options shown in Figure 8.6 to reload the original materials in RF-FE-LTB. Please notice that this option is available only in the 1.2 *Materials* window.

8.3.2 Export Cross-Section to RFEM

If you adjust the cross-sections for the check in RFEM, you can also export the changed cross-sections to RFEM: Go to the 1.3 *Cross-Sections* window, and then select on the menu

Edit → **Export All Cross-Sections to RFEM**.

Alternatively, use the context menu in window 1.3 to export the cross-sections to RFEM.

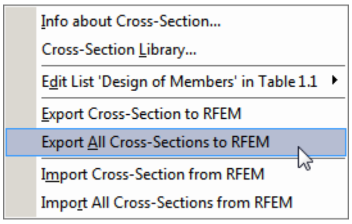


Figure 8.7: Context menu of window 1.3 *Cross-Sections*

Before the material is exported, a query appears as to whether the results of RFEM should be deleted.

If the changed cross-sections have not been exported to RFEM yet, you can reload the original cross-section in RF-FE-LTB by using the options shown in Figure 8.7. Note that this possibility is available only in the 1.3 *Cross-Sections* window.

8.3.3 Export Results

You can use the results of RF-FE-LTB in other programs, too.

Clipboard

To copy cells of the results windows to the Clipboard, press [Ctrl]+[C]. To paste them, for example in a word processor, press [Ctrl]+[V]. The headings of the table columns are not exported.

Printout Report

The data from RF-FE-LTB can be printed to the printout report (see chapter 7.1, page 100). You can export the data from the report by using the menu

File → Export in RTF.

This function is also described in chapter 10.1.11 of the RFEM manual.

Excel / OpenOffice

RF-FE-LTB allows for the direct data export to MS Excel, OpenOffice.org Calc, or the CSV format. To start this function, select on the menu

File → Export Tables.

The following export dialog box appears.

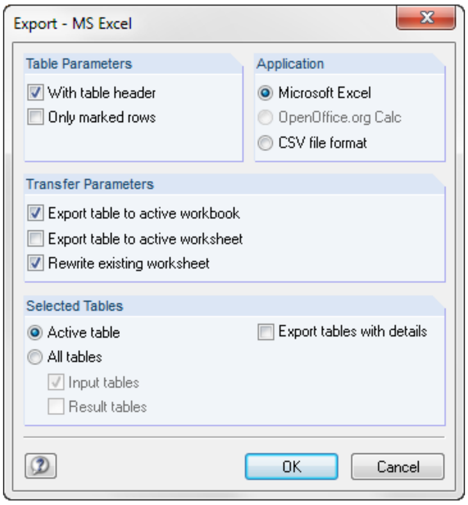
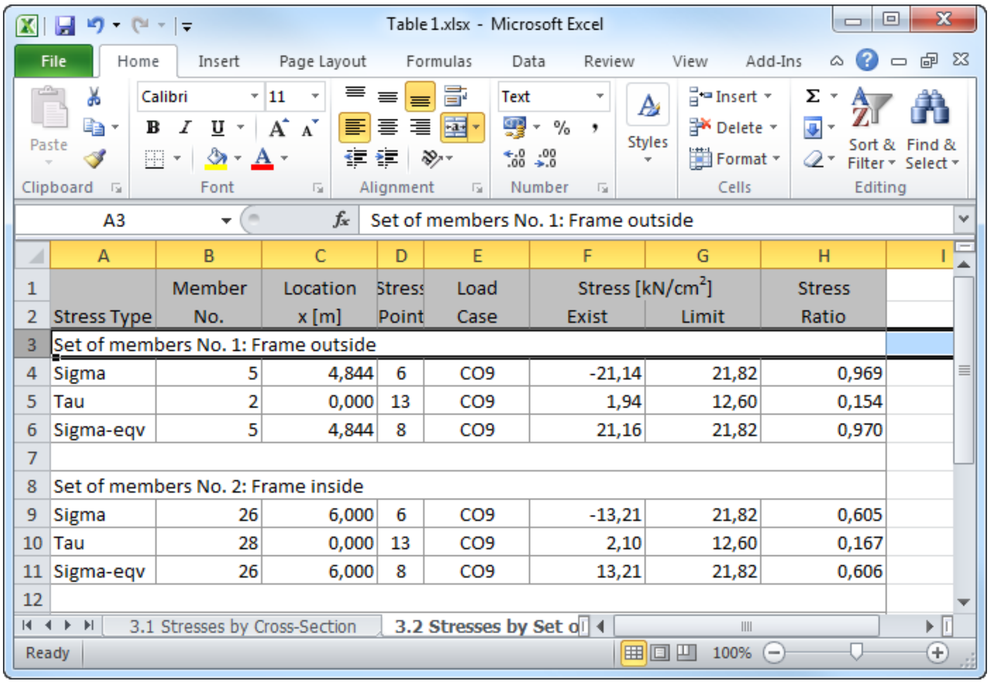


Figure 8.8: Dialog box *Export - MS Excel*

Once the selection is complete, start the export by clicking [OK]. Excel or OpenOffice is started automatically, that is, you do not have to open them first.



Stress Type	Member No.	Location x [m]	Stress Point	Load Case	Stress [kN/cm²] Exist	Limit	Stress Ratio
Set of members No. 1: Frame outside							
Sigma	5	4,844	6	CO9	-21,14	21,82	0,969
Tau	2	0,000	13	CO9	1,94	12,60	0,154
Sigma-eqv	5	4,844	8	CO9	21,16	21,82	0,970
Set of members No. 2: Frame inside							
Sigma	26	6,000	6	CO9	-13,21	21,82	0,605
Tau	28	0,000	13	CO9	2,10	12,60	0,167
Sigma-eqv	26	6,000	8	CO9	13,21	21,82	0,606

Figure 8.9: Exported results in *Excel*

9. Worked Examples

In this chapter, we present some examples from literature and compare them to the results in RF-FE-LTB.

For these examples, there are usually no analytical solutions. The approaches used in the literature are either numerical or based on the RITZ or GALERKIN method with one or multi-elemental approaches, which in turn represent approximate solutions describing the lateral-torsional buckling by means of differential equations. Thus, it cannot be expected that the results from RF-FE-LTB are completely identical to results in the literature.

9.1 Beam Under Concentrated Load

System

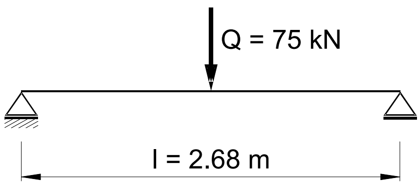


Figure 9.1: System and loading

Cross-section values and material: IPE 200, steel S 235

The beam has pinned supports at both edges but there is no warping restraint. Thus, the following boundary conditions exist:

Left support: $u = v = w = \varphi_x = 0$

Right support: $v = w = \varphi_x = 0$

Two cases are analyzed:

First case: pure bending without rotational restraint

Second case: pure bending with rotational restraint

9.1.1 Bending Without Rotational Restraint

$N = 0$, $c_\vartheta = 0$

The elastic critical moment for lateral-torsional buckling according to ROIK, CARL, LINDNER [3] is:

$$M_{cr,y} = \zeta \frac{\pi^2 E I_z}{L^2} \sqrt{\frac{I_\omega + 0.039 L^2 J}{I_z}}$$

For $\zeta = 1.35$, we obtain after inserting the according values: **M_{cr} = 83.9 kNm**

RF-FE-LTB gives the following load factors ν and critical moments $M_{cr,y}$:

Initial deformation (L/400)	**Direction of displacement**	**ν**	**$M_{cr,y}$** [kNm]
none		1.68	50.25*1.68 = **84.42**
Eigenvectors	v	1.63	50.21*1.63 = **81.84**

The elastic critical moment for lateral-torsional buckling should always be calculated without initial deformations, because it belongs to the critical load of the system. If we apply an initial deformation, we obtain a smaller value of M_{cr} than in case of a calculation without consideration of the imperfection.

The value of $M_{cr,y}$ = 84.42 kNm should be used in the calculation according to the equivalent member method according to DIN 18800 Part 2: In window 1.4 of the add-on module **RF-LTB**, you can define M_{cr} manually.

Einstellungen für Stab Nr. 1		
Cross-section		1 - IPE 200 | DIN 1025-5:1994
Support Type		Pinned support
Shear panel		☐
Rotational restraint		☐
Load application point		In centroid
⊟ **Method of determining M-cr**		Assign LC/CO individually
⊟ LC1 - Moment for lateral-torsional bucklin		Define M-Cr
Critical buckling moment M-Cr		84.420 kNm
Beam type		Rolled beam
Comment		

Figure 9.2: RF-LTB window 1.4 with manual possibility of defining M_{cr}

9.1.2 Bending with Rotational Restraint

N = 0, c_ϑ = 50 kNm/m

With the equation for the ideal torsional constant

$$J_{id} = J + c_\vartheta \frac{L^2}{\pi^2 G}$$

we obtain $\mathbf{M_{cr,y}}$ **= 191.1 kNm**.

Taking into account the rotational restraint, RF-FE-LTB determines the following load factors ν and elastic critical moments $M_{cr,y}$:

Initial deformation (L/400)	**Direction of displacement**	**ν**	$\mathbf{M_{cr,y}}$ [kNm]
none		3.67	50.25*3.67 = **184.42**
Eigenvectors	v	3.60	50.25*3.60 = **180.90**

9.2 Beam Under Uniform Loading

System

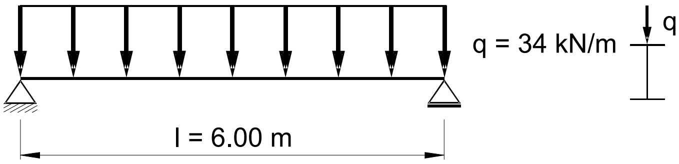


Figure 9.3: System and loading

This example is taken from the SCHNEIDER Bautabellen (18th edition, page 8.46) [5].

Cross-section values and material: IPE 400, steel S 235

The system has pinned supports at both end but no warping restraint. Thus, we have the following boundary conditions:

Left support: $u = v = w = \varphi_x = 0$
Right support: $v = w = \varphi_x = 0$

The uniform load q acts on the upper flange of the beam.

As expected, RF-FE-LTB yields different stability loads with and without initial deformations. These are summarized in the following table; a convergence study was carried out to determine the precision of the calculation depending on the amount of elements. For the calculation including the initial deformation based on the eigenvectors, an initial bow imperfection of L/400 is applied.

Elements	**ν**	$M_{cr,y}$ [kNm]	ν_E	q_{max} [kN/m]
4	1.31	153.0*1.31 = 200.4	1.27	0.91*34 = 30.94
8	1.28	153.0*1.28 = 195.8	1.24	0.89*34 = 30.26
16	1.27	153.0*1.27 = 194.3	1.23	0.89*34 = 30.26
32	1.26	153.0*1.26 = 192.8	1.23	0.89*34 = 30.26
64	1.26	153.0*1.26 = 192.8	1.23	0.89*34 = 30.26

In the table, **ν** denotes the load factor without considering the initial deformation and ν_E denotes the load factor of the imperfect system based on the eigenvectors. From the results, we can see that all types of calculation (with/without initial deformation and in case of stress limitation) reach a very good precision with 8 elements. The calculation with 32 elements represents the converging solution, as the doubling of the number of element results in no change.

In the SCHNEIDER Bautabellen [5], the value is given as $\mathbf{M_{cr,y} = 169.2}$ **kNm**.

The limitation of the elastic stress to $f_{y,k} = 24$ kN/cm^2 results in RF-FE-LTB in the maximum resisting load of $q_{max} = 0.89 \cdot 34 = 30.26$ kN/m. This load is less than the value given in [5]. The reason for this is that the plastic reserve of the cross-section is not taken into account in the determination of the elastic limit load F_G. With the program RF-LTB, however, you can consider this.

9.3 Cantilevered Beam with Warping Restraint

In this example according to SCHNEIDER Bautabellen (18th edition, page 8.20) [5], the warping torsional moment M_ω is to be calculated.

System

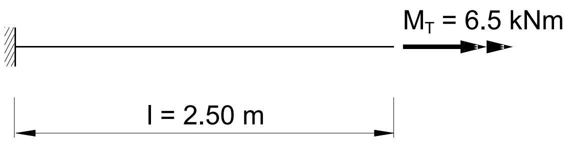


Figure 9.4: System and loading

Cross-section properties and material: HEB 240, steel S 235

The cross-section is restrained to warping at the location of fixity. Thus, on the left end of the beam, the following boundary conditions exist:

$u = v = w = \varphi_x = \varphi_y = \varphi_z = \omega = 0$

The calculation according to [5] gives the **warping torsional moment** M_ω = -70,494 kNcm².

The analysis with RF-FE-LTB gives the value M_ω = -70,400 kNcm².

The according **Rotation** ϑ is given in [5] as 6.3°.

RF-FE-LTB determines the value $\varphi_x = 0.1214 \cdot 180/\pi = 6.96°$.

The comparison of the stresses in [5] and RF-FE-LTB gives the following result:

Stress	**SCHNEIDER [5]** [kN/cm²]	**RF-FE-LTB** [kN/cm²]
σ_ω	±19.4	±19.35
$\tau_{Mx,p}$	8.5	8.73
$\tau_{Mx,s}$	1.07	1.07

9.4 Cantilevered Beam Under Uniform Load

In this example according to PETERSEN [2], page 732, the lateral-torsional load and the according critical moment for lateral-torsional buckling are calculated.

System

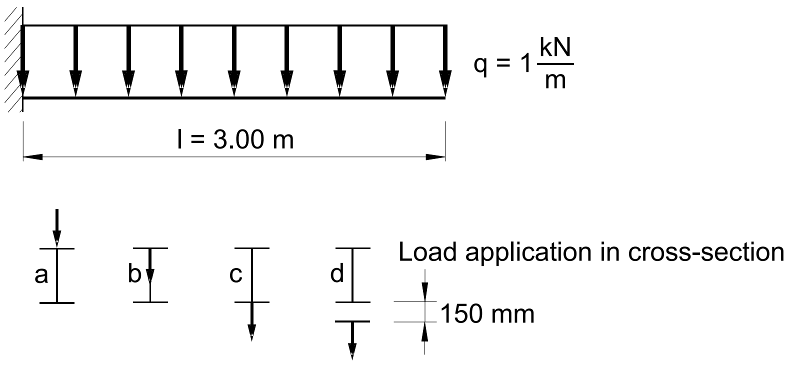


Figure 9.5: System and loading

Cross-section properties and material: IPE 200, steel S 235

9.4.1 Crane Runway Beam with Free End

The cross-section is restrained to warping at the point of fixity. Thus, on the left end of the beam, the following boundary conditions exist:

$u = v = w = \varphi_x = \varphi_y = \varphi_z = \omega = 0$

Load effect	ν_4	ν_{E4}	ν_8	ν_{E8}	ν_{64}	ν_{E64}	$\nu_{PETERSEN}$
Top (a)	20.27	20.19	20.52	20.44	20.56	20.56	19.06
Centroid (b)	36.81	36.78	37.78	37.62	38.06	36.06	38.13
Below (c)	52.75	52.75	54.02	53.75	54.25	54.06	57.19
Suspended (d)	66.56	66.25	67.75	67.25	68.25	67.62	76.25

The values $\boldsymbol{\nu}$ mean: ν_n is the result from RF-FE-LTB without imperfections for *n* elements; ν_{En} is the result with initial deformation based on the scaled eigenvectors. The scaling is carried out with the value 0.75 (displacement perpendicular to the drawing plane at the end of the cantilever).

Here, the analysis with eight elements also comes close to the converged solution.

9.4.2 Cantilever End with Lateral Restraint

The cross-section is warping-restrained and restrained by a bracing on the right side. Furthermore, torsion is restrained at the end of the cantilever. Thus, the following boundary conditions exist:

Left support: $u = v = w = \varphi_x = \varphi_y = \varphi_z = \omega = 0$
Right support: $v = \varphi_x = 0$

Load effect	$\nu_{RF\text{-}FE\text{-}LTB}$	$\nu_{PETERSEN}$
Top (a)	68.75	67.10
Centroid (b)	90.56	91.51
Below (c)	114.50	115.9
Suspended (d)	146.20	149.5

In RF-FE-LTB, eight elements give already a very good approximation for the values for the load factor ν that in PETERSEN [2] result from nomograms for flexural-torsional buckling.

9.5 Beam Under Uniform Loading

For the following example according to PETERSEN [2], page 731, the critical load factor ν is calculated.

System

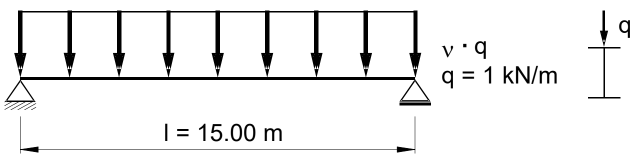


Figure 9.6: System and loading

Cross-section values and material: HEB 800, steel S 235

The system has fork supports at both ends but no warping restraint. Hence, we have the following boundary conditions:

Left support: $u = v = w = \varphi_x = 0$
Right support: $v = w = \varphi_x = 0$

The effect of the load *q* is analyzed for the top flange, the centroid, and the bottom flange.

RF-FE-LTB gives the following load factors ν or ν_E for an initial deformation of L/400.

Load effect	ν	ν_E	$\nu_{PETERSEN}$
Top flange	37.38	36.31	39.50
Centroid	46.38	44.88	48.61
Bottom flange	57.53	55.56	57.73

9.6 Continuous Beam Under Two Nodal Loads

In Lindner [6] and Dickel et al. [4], we can find calculations for the following two-span beam. The two concentrated loads of F = 37.5 kN act in the span center in the centroid of the cross-section, respectively.

System

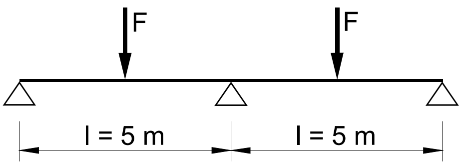


Figure 9.7: System and loading

Cross-section properties and material: IPE 200, steel S 235

All supports of the beam are pinned supports. Thus, the following boundary conditions exist:

Left support: $u = v = w = \varphi_x = 0$
Middle support: $v = w = \varphi_x = 0$
Right support: $v = w = \varphi_x = 0$

The calculation is carried out with eight elements. The following values result for the elastic critical moment for lateral-torsional buckling $M_{cr,y}$ in [kNm] over the middle support.

$M_{cr,y\ Lindner}$	$M_{cr,y\ Dickel}$	$M_{cr,y}$	$M_{cr,y\ E}$
51.24	51.06	1.46*35.16 = 51.33	1.38*35.14 = 48.85

For the determination of $M_{cr,y\ E}$, an initial deformation of L/400 was applied.

The results of RF-FE-LTB correspond very well to those of [4] and [6].

9.7 Continuous Beam Under Uniform Loads

In Dickel et al. [4], the elastic critical moments for lateral-torsional buckling of a two-span beam are determined. The uniform load of the magnitude q = 10 kN/m acts on the top flange at a distance of 14.5 cm from the centroid of the cross-section.

System

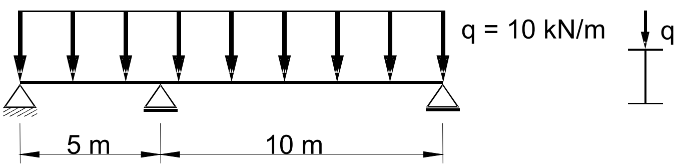


Figure 9.8: System and loading

Cross-section values and material: HEA 300, steel S 235

All supports of the beam are pinned supports. Thus, the following boundary conditions exist:

Left support: $u = v = w = \varphi_x = 0$
Middle support: $v = w = \varphi_x = 0$
Right support: $v = w = \varphi_x = 0$

For the calculation in RF-FE-LTB, we specify an FE length of 50 cm. At the middle support, we obtain the following values for the elastic critical lateral-torsional moment $M_{cr,y}$ and the corresponding uniform load $q_{cr,z}$.

	DICKEL et al. [4]	RF-FE-LTB	PETERSEN [2]	ROIK et al. [3]
$\mathbf{q_{cr,z}}$ [kN/m]	42.84	42.85	35.86	37.08
$\mathbf{M_{cr,y}}$ [kNm]	401.6	401.7	336.2	347.6

The results of RF-FE-LTB correspond very well to those of [4].

In further analyses, the maximum equivalent stresses for various values of uniform loads are calculated on the predeformed system. RF-FE-LTB determines the maximum uniformly distributed load for which the stress σ_{eqv} = 24 kN/cm² is observed.

q [kN/m]	**max σ_{eqv}** [kN/cm²]
10	8.75
20	18.76
24.3	24.0

An analytical equivalent calculation with the expressions given in [10] yields for the critical span moment ($r_y = z_M = 0$):

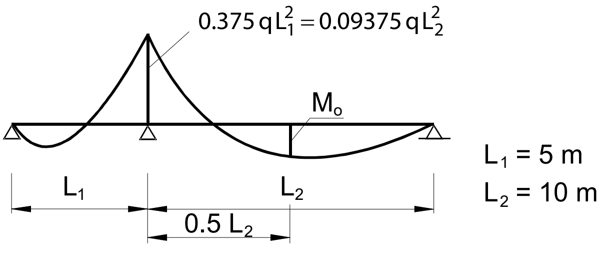
$$M_{cr,y} = \zeta \frac{\pi^2 EI_z}{L^2} \left(\sqrt{0.25 z_p^2 + c^2} + 0.5 z_p \right) = 28965 \, \text{kNcm} = 289.65 \, \text{kNm}$$

where ζ = 1.08 (see below) and z_p = -14.5 cm and

$$c^2 = \frac{I_\omega}{I_z} + \frac{L^2 GJ}{\pi^2 EI_z} = \frac{1{,}200{,}000}{6310} + \frac{1000^2 \cdot 8100 \cdot 85.2}{\pi^2 \cdot 21{,}000 \cdot 6310} = 717.9 \, \text{cm}^2$$

The value of ζ according to [3] represents the correction factor. The solution $M_{cr,y}$ of a member with fork supports at both ends and subjected to equal end moments is to be multiplied with this correction factor.

It follows according to [3]:



$$M_o = q\frac{L_2^2}{8} - \frac{0.09375}{2} qL_2^2 = 0.078125\,qL_2^2$$

M_o was used in [3] to calculate ν_{cr}, that is, M_{cr}; thus, the elastic critical moment for lateral-torsional buckling given in [3] is relative to the center of span 2!

$$\chi = \frac{E I_\omega}{L^2 GJ} = \frac{21{,}000 \cdot 1{,}200 \cdot 10^3}{(1{,}000)^2 \cdot 8{,}100 \cdot 85.2} = 0.037$$

Table 5.23 [3]

$$\overline{M} = -\frac{q L_2^2}{8} = \psi M_{support} = \psi \cdot (-0.09375) q \cdot L_2^2$$
$$\Rightarrow \psi = 0.09375 \cdot 8 = 0.75 \quad \Rightarrow \zeta \cong 1.08$$

Thus, the ideal column moment as the maximum moment calculated by RF-FE-LTB as well as DICKEL [4] is:

$$M_{cr,y\,span} = 0.078125\, q_{ki} \cdot 10^2 = 289.65$$
$$\Rightarrow q_{cr} = 37.08\,kN/m$$
$$M_{cr,support} = 0.09375 \cdot 37.08 \cdot 10^2 = 347.58\,kNm$$

According to Petersen [2], Table 7.23, we obtain for the end span of a continuous beam with a degree of restraint of 0.09375/0.125 · 100 = 75 % the following ideal values:

$$\mu = \frac{E I_\omega}{L^2 GJ} = 0.0365, \quad \chi = +\left(\frac{z_p}{L}\right)^2 \frac{E I_z}{G J} = 0.04$$
$$\Rightarrow \gamma_{cr} \cong 37.5 \quad \Rightarrow q_{cr} = \frac{\gamma_{cr}}{L^3} \sqrt{E I_z\, GJ} = 35.86 \frac{kN}{m}$$
$$\Rightarrow M_{cr}\,(support) = 0.09375 \cdot 35.86 \cdot 10^2 = 336.2\,kNm$$

The ideal values determined according to [3] or [4] are smaller than the values determined here, since the restraining effects due to warping restraint of the first span are not considered in the analytical formulas.

9.8 Three-Span Beam with Uniform Loadings

In PETERSEN [1], page 405 and DICKEL et al. [4], the elastic critical moments for lateral-torsional buckling of a continuous beam are determined.

1st case: system with uniform loads

The line load of the magnitude q = 30.5 kN/m is constant over the entire beam. It acts on the top flange at a distance of 18 cm from the centroid of the cross-section.

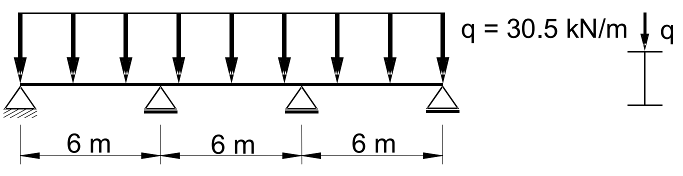


Figure 9.9: System and loading

Cross-section properties and material: IPE 360, steel S 235

All supports of the beam are pinned supports. Thus, the following boundary conditions exist:

Left support: $u = v = w = \varphi_x = 0$

Middle supports: $v = w = \varphi_x = 0$

Right support: $v = w = \varphi_x = 0$

The analysis in RF-FE-LTB is run with 18 and with 72 elements. At the inner supports, we obtain the following values for the elastic critical moment for lateral-torsional buckling $M_{cr,y}$ and the according uniformly distributed load $q_{cr,z}$:

	PETERSEN [1]	DICKEL [4]	RF-FE-LTB 18 elements	RF-FE-LTB 72 elements
$\mathbf{q_{cr,z}}$ [kN/m]	45.32	48.8	1.66*30.5 = 50.6	1.63*30.5 = 49.7
$\mathbf{M_{cr,y}}$ [kNm]	163.2	175.7	1.66*109.8 = 182.3	1.63*109.8 = 179.0

The results from RF-FE-LTB correspond very well to those of [1] and [4].

For an initial deformation of L/400 = 600/400 = 1.5 cm, we obtain for this line load the maximum equivalent stress σ_{eqv} = 16.9 kN/cm².

For a stress limitation to σ_{eqv} = 24 kN/cm², we obtain the maximum resistant load q_{max} = 36.2 kN/m.

2nd case: system with different loads

In the second case, the center span is not loaded by q = 30.5 kN/m but by a reduced line load of g = 5.5 kN/m.

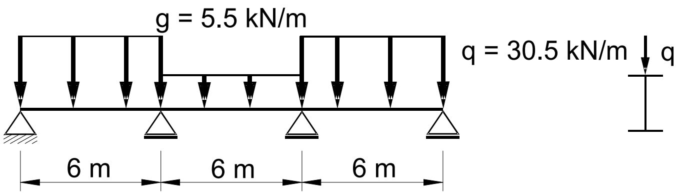


Figure 9.10: System and loading

The elastic critical moments for lateral-torsional buckling under this loading are governing not for columns but for the outer spans.

	PETERSEN [1]	DICKEL et al. [4]	RF-FE-LTB
$\mathbf{M_{cr,y}}$ [kNm]	137.8	162.24	1.57 * 104.8 = 164.5

The result in RF-FE-LTB corresponds well to the value in [4]. The elastic critical moment for lateral-torsional buckling given in [1] is on the safe side, because there the warping restraints over the supports and the restraint effects due to warping restraint of the less loaded adjacent span are neglected.

For an initial deformation of L/400 = 600/400 = 1.5 cm, we obtain for this line load the maximum equivalent stress σ_{eqv} = 21.2 kN/cm².

In case of a stress limitation to σ_{eqv} = 24 kN/cm², the maximum resistant loads are $q_{max} = 1.06 \cdot 30.5$ = 32.3 kN/m (outer spans) and $q_{max} = 1.06 \cdot 5.5$ = 5.8 kN/m (inner spans).

9.9 Beam on Foundation with Axial Force

A beam with pinned supports is restrained at the top flange and in addition has an elastic torsional restraint. The analytical solution can be taken from the publication of WITTEMANN [12], where the elastic lateral-torsional buckling load of a compressive member with elastic torsional restraint and translational restraint is determined. For this, we consider the following system.

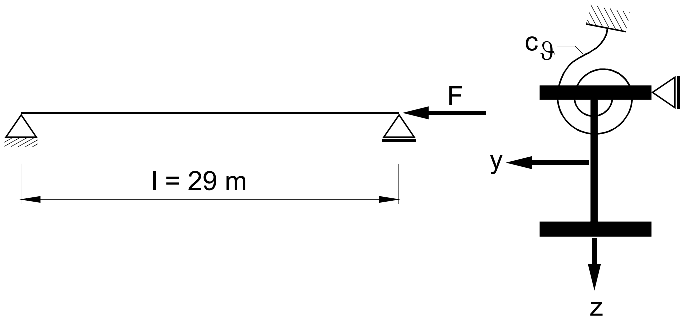


Figure 9.11: System and loading

Cross-section values and material: IPE 400, steel S 235

The beam has pinned supports on both ends but no warping restraint. Thus, the following boundary conditions exist:

Left support: $u = v = w = \varphi_x = 0$

Right support: $v = w = \varphi_x = 0$

At the top flange, the beam is laterally restrained by a stiffening. In RF-FE-LTB, this lateral restraint is modeled by a very stiff spring. Furthermore, there is a rotational restraint of $c_\vartheta = 9.936$ kNm/m.

At first, the calculation in RF-FE-LTB (without initial deformation) gives the value

$$N_{cr,y} = 100 \cdot 5.7 = 570.0\,\text{kN}\,.$$

This result, however, represents the buckling load about the main axis:

$$N_{cr,y} = \frac{21{,}000 \cdot 23{,}130 \cdot \pi^2}{2{,}900^2} = 570.0\,\text{kN}$$

By means of a vertical support in the midspan, that is, w*(L/2) = 0, a significantly lower critical load is determined, since RF-FE-LTB now uses the higher eigenvectors:

$$N_{cr,\vartheta} = 100 \cdot 2.27 = 227\,\text{kN}$$

Thus, we can see how the modeling of members on elastic foundation with translational restraint influences the elastic critical load for lateral-torsional buckling, especially for large span lengths.

9.10 Roof Beam in Office Building

The roof beams of an office building are arranged at a distance of 5.0 m. They are stiffened by a connecting Thyssen T135.1 - 0.75 trapezoidal sheet and a connected cross-bracing.

The shear panel length of the trapezoidal sheet is L_S = 20 m. The cross-bracing consists of only one panel with crossed diagonals and two posts. Hence, the inclination angle of the diagonal is α = arctan (5.0/7.75) = 32.8°.

System

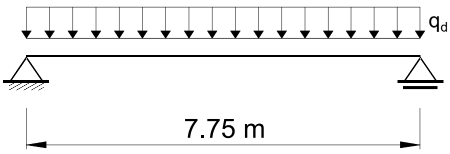


Figure 9.12: System and loading

Cross-section properties and material: IPE 270, steel S 235

Because of the shear connections by end plates to the laterally restrained columns (by means of vertical bracing), we can assume pinned supports at both ends of the beam. Thus, the following support conditions exist:

Left support: $u = v = w = \varphi_x = 0$

Right support: $v = w = \varphi_x = 0$

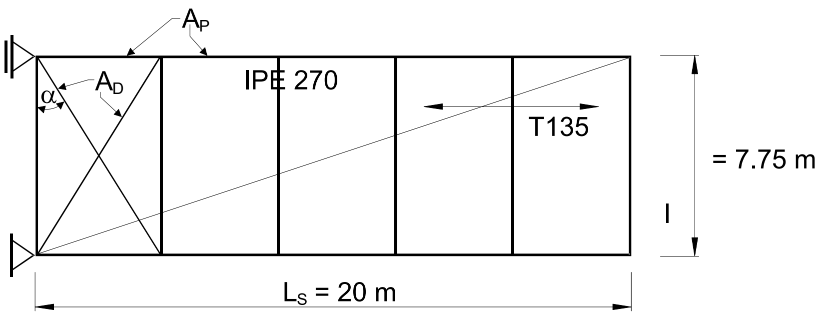


Figure 9.13: Beam in roof plane

The trapezoidal sheet provides a lateral elastic restraint and a rotational elastic spring to the roof beam at the height of the top flange.

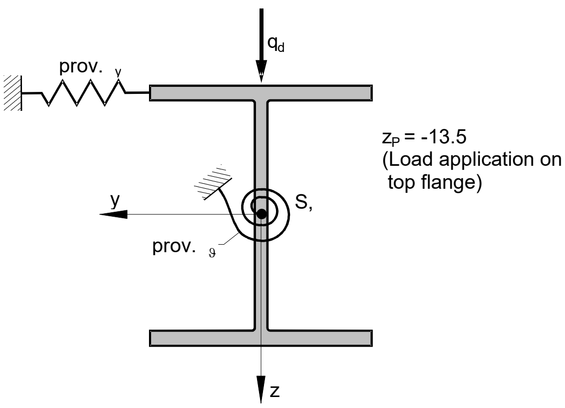


Figure 9.14: Support of the beam and effect of the load

The design load from self-weight and snow is:

$q_d = 1.35\ g + 1.5\ p = 11.79$ kN/m

Determination of the horizontal spring

The calculation of the spring constants for the shear stiffness from roof bracing is described in chapter 2.5.2 on page 27 as well as in [2] and [10].

Continuous lateral translational spring:

$$c_y = \text{prov } S_a \frac{\pi^2}{L^2} = 1{,}158 \frac{\text{kN}}{\text{m}^2}$$

Since the trapezoidal sheeting is fixed only in each second groove, S_T may be applied only with 1/5 of the shear stiffness.

$$\text{prov } S_a = \frac{1}{5} S_T + S_R$$

Bracing

$$S_R = m \cdot \frac{a}{L_s} \cdot S_V = 3{,}347 \text{ kN}$$

where $m = 1$ one bracing

$a = 5.0$ m

$L_s = 20.0$ m

$$S_V = \frac{a^2 \cdot b \cdot E}{\frac{\left(\sqrt{a^2 + b^2}\right)^3}{A_D} + \frac{a^3}{A_P}} = 13{,}389 \text{ kN}$$

$a = 500$ cm

$b = 750$ cm

$E = 21{,}000$ kN/cm²

$A_D = 2.63$ cm² area of diagonal

$A_P = 8.00$ cm² area of post

Trapezoidal sheeting

$$S_T = \frac{a}{100} \cdot G_S = 16{,}250 \text{ kN}$$

where $G_S = \frac{10^4}{k_1 + 100 \cdot \frac{k_2}{L_s}} = 3{,}252$ kN/m ideal shear modulus

$k_1 = 0.275$ m/kN

$k_2 = 56$ m²/kN

$L_s = 20$ m

The spring c_y is laterally applied at the top flange.

Determination of the rotational spring

The calculation of the spring constants from the connecting trapezoidal sheeting is described in chapter 2.5.1 on page 25 as well as in [2], [8] and [10].

$$\frac{1}{\text{prov}\, c_{\vartheta,k}} = \frac{1}{c_{\vartheta M,k}} + \frac{1}{c_{\vartheta A,k}} + \frac{1}{c_{\vartheta P,k}} = \frac{1}{499} + \frac{1}{5.23} + \frac{1}{59.6} = 0.20999$$

$$\Rightarrow \text{prov}\, c_{\vartheta,k} = 4.76 \frac{\text{kNcm}}{\text{cm}}$$

Restraining structural component

The rotational restraint from the bending stiffness $I_a = 297\ \text{cm}^4/\text{m}$ of the trapezoidal sheeting ($k = 4$ for continuous beams with three spans) is obtained as follows:

$$c_{\vartheta M,k} = \frac{EI_a}{a} \cdot k = \frac{21{,}000 \cdot 2.97}{500} \cdot 4 = 499 \frac{\text{kNcm}}{\text{cm}}$$

Deformation of cross-section

The rotational restraint from the section deformation of the supported beam is obtained as follows:

$$c_{\vartheta P,k} = \frac{E}{4 \cdot (1-\mu^2)} \cdot \frac{1}{\frac{h_m}{s^3} + 0.5 \cdot \frac{b_1}{t_1^3}} = \frac{21{,}000}{4 \cdot (1-0.3^2)} \cdot \frac{1}{\frac{27-1.02}{0.66^3} + 0.5 \cdot \frac{13.5}{1.02^3}} = 59.6 \frac{\text{kNcm}}{\text{cm}}$$

where b_1 Flange width of the compression flange

Connection deformation

The rotational restraint from the deformation of the connection is determined for $1.25 \leq b_1 / 10 \leq 2.0$ as follows:

$$c_{\vartheta A,k} = \bar{c}_{\vartheta,Ak} \cdot 1.25 \cdot \frac{b_1}{10} = 3.1 \cdot 1.25 \cdot \frac{13.5}{10} = 5.23 \frac{\text{kNcm}}{\text{cm}}$$

where $\bar{c}_{\vartheta A,k}$ Characteristic values for connecting stiffness acc. to Table 7 [8]

Selection of the eigenvector

The elastically supported member is calculated on the imperfect system according to second-order analysis. Here, an initial deformation in direction y is applied with the following camber rise of the parabola (see chapter 2.6.2, page 37):

$$v_0 = 0.5 \cdot \frac{2}{3} \cdot \frac{L}{250} = \frac{L}{750} = 1.03\ \text{cm}$$

This value is obtained according to Table 3 [8] for the buckling curve *b* and the reductions according to element (201) and (202) [8].

In this example, the eigenvectors are compared in order to find the governing eigenvalue (see the following figure).

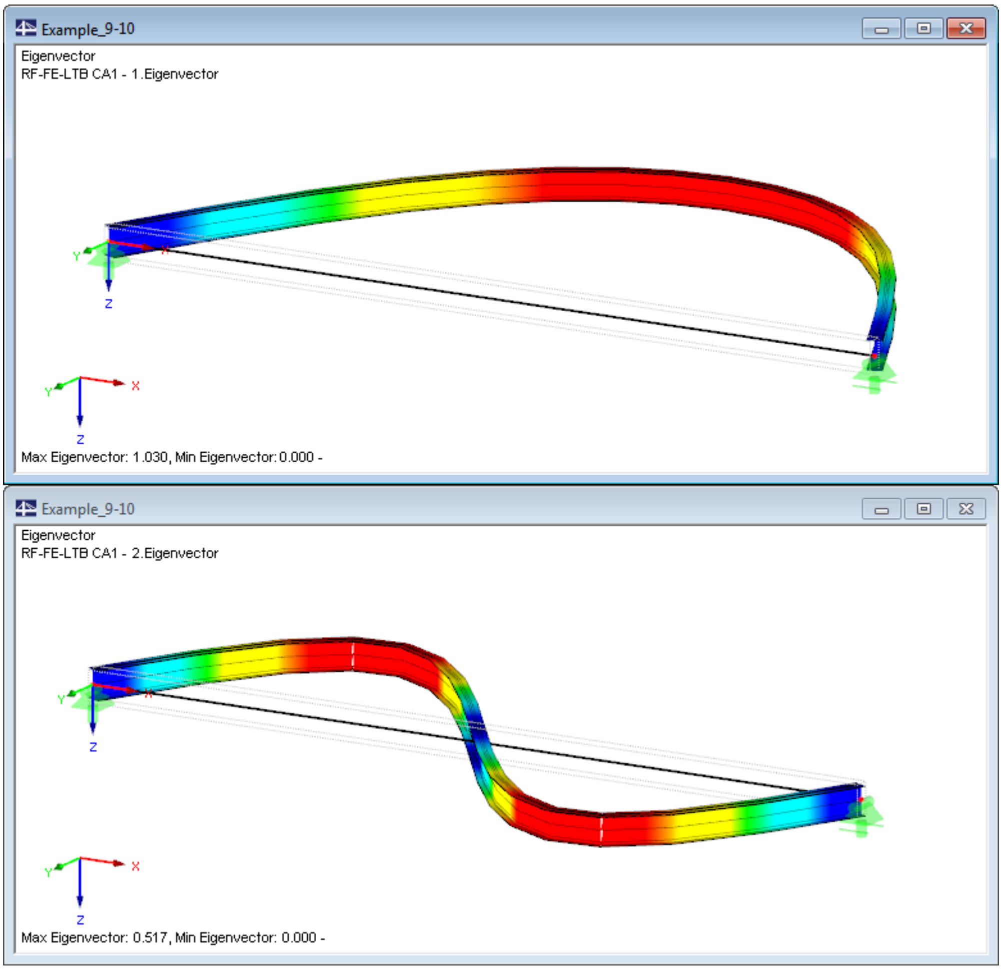


Figure 9.15: First and second eigenvector

The graphic of eigenvectors makes it clear that the first eigenvalue is governing.

Determination of the magnitudes of imperfection

Different values *s* are relevant for the two eigenvectors.

Eigenvector No. 1:

$$v_0 = 0.5 \cdot \frac{2}{3} \cdot \frac{L}{250} = \frac{775}{750} = 1.03\,\text{cm}$$

Eigenvector No. 2:

$$v_0 = 0.5 \cdot \frac{2}{3} \cdot \frac{L}{250} = \frac{0.5 \cdot 775}{750} = 0.517\,\text{cm}$$

For the second eigenvector, only the length of one half-wave is taken as reference size.

The following table shows the equivalent stresses that result from the different eigenvectors and magnitudes of imperfection.

	Eigenvector No. 1	Eigenvector No. 2
σ_{eqv} [kN/cm^2]	21.18	20.71

9.11 Beam with Eccentric Uniform Load

We want to analyze a steel beam subjected to biaxial bending. The loads act 5 cm above the top flange.

In PETERSEN [2], Table 7.29, the normal stress σ_x in the shear point (1) is determined analytically.

System

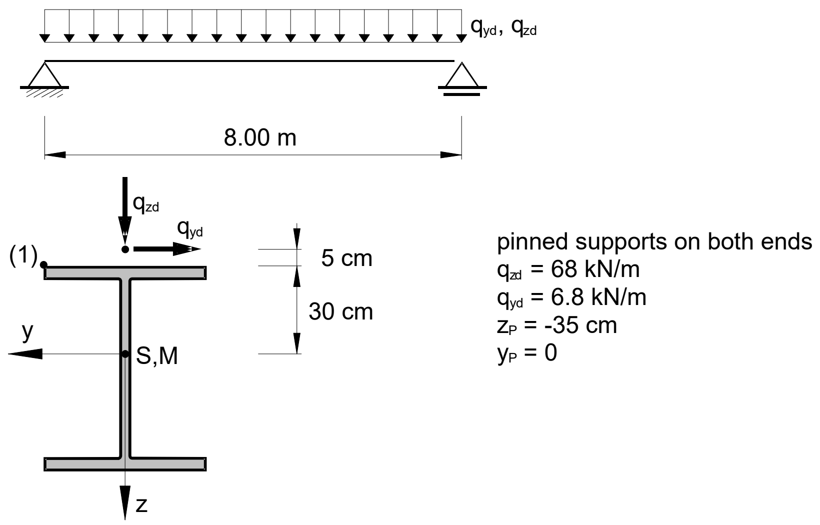


Figure 9.16: System and loading

Cross-section properties and material: HEB 600, steel S 235

As imperfection, we take an initial deformation in the direction of the member axis y with the following magnitude of imperfection (see Table 3 in [8]):

$$v_0 = 0.5 \cdot \frac{2}{3} \cdot \frac{L}{250} = 1.07\,\text{cm}$$

The calculation in RF-FE-LTB uses 16 elements.

We obtain the following stresses and rotational movements:

	PETERSEN [2]	RF-FE-LTB
σ_x in (1) [kN/cm²]	22.55	24.15
Rotation φ_x in L/2 [rad]	$3.645 \cdot 10^{-2}$	$4.360 \cdot 10^{-2}$

A Literature

[1] PETERSEN, C.: Stahlbau, Vieweg und Sohn, Braunschweig/Wiesbaden, 3. Auflage 1993

[2] PETERSEN, C.: Statik und Stabilität der Baukonstruktionen, Vieweg und Sohn, Braunschweig/Wiesbaden, 2. Auflage 1982

[3] ROIK, K./ CARL, J./ LINDNER, J.: Biegetorsionsprobleme gerader dünnwandiger Stäbe, Ernst und Sohn, Berlin/München/Düsseldorf, 1971

[4] DICKEL, T./ KLEMENS, H.-P./ ROTHERT, H.: Ideale Biegedrillknickmomente, Vieweg und Sohn, Braunschweig/Wiesbaden 1991

[5] SCHNEIDER Bautabellen, 18. Auflage, Werner Verlag, 2008

[6] LINDNER, J.: Biegedrillknicken in Theorie, Versuch und Praxis. Berichte aus der Forschung und Entwicklung (DASt Heft 9/1980), Stahlbau-Verlags-GmbH, Köln 1980

[7] DIN 18 800 Teil 1: Stahlbauten - Bemessung und Konstruktion, 1990

[8] DIN 18 800 Teil 2: Stahlbauten - Stabilitätsfälle, Knicken von Stäben und Stabwerken, 1990

[9] EN 1993-1-1: Design of steel structures – Part 1-1: General rules and rules for buildings, 2005

[10] Handbuch RF-BGDK (Software für Biegedrillknicksicherheitsnachweis nach DIN 18800, Teil 1 und 2), DLUBAL SOFTWARE GmbH, Tiefenbach 2011

[11] RAMM, E./ HOFMANN, T. J.: Stabtragwerke. Der Ingenieurbau, Baustatik/Baudynamik, Ernst und Sohn, Berlin 1996

[12] WITTEMANN, K.: Ideale Biegedrillknicklast für einen drehgebetteten Druckstab mit gebundener Achse. Stahlbau 61 (1992), 125-126

[13] DIN 18807: Stahltrapezprofile, 1987

[14] LINDNER, J.: Stabilisierung von Biegeträgern durch Drehbettung - eine Klarstellung. Stahlbau 56 (1987), 365-373

[15] LINDNER, J./ SCHEER, J./ SCHMIDT, H.: Stahlbauten - Erläuterungen zu DIN 18 800 Teil 1 bis 4, Beuth, 2. Auflage 1994

[16] OSTERRIEDER, P./ VOIGT, M./ SAAL, H.: Zur Neuregelung des Biegedrillknicknachweises nach EDIN 1880 Teil 2 (Ausgabe März 1988). Stahlbau 58 (1989), 341-347

[17] Stahlbau Handbuch, Band 1, Stahlbau-Verlags-GmbH, Köln 1982

[18] LINDNER, J./ GROESCHEL, F.: Drehbettungswerte für die Profilblechbefestigung mit Setzbolzen. Stahlbau 65 (1996), 218-224

Dlubal

B Index

A

Alpha ... 46
Aluminum ... 44
Angle ... 52
Axial force ... 85

B

Background graphic ... 93
Bending moment ... 85
Bimoment M_{ω} ... 70
Boundary conditions ... 19, 96
Bracing ... 24, 27, 57
Browsing through windows ... 40
Buckling curve ... 76
Buckling eigenvalue ... 74
Buckling load N_{cr} ... 34
Buttons ... 90

C

Calculation error ... 79, 88, 89
Cantilevered portion ... 29, 54
Channel section ... 51
Check ... 80
Clipboard ... 105
Color scale ... 98
Column ... 31, 53, 65, 76
Comment ... 42, 54, 62, 73
Concentrated load ... 17
Connection deformation ... 26, 59
Continuity effect ... 58, 61
Continuous members ... 41
Control panel ... 98
Convergence ... 11, 78, 79, 88
Coordinates of stress points ... 93
Correction factors ... 60
Corrugated sheet ... 58
Craneway console ... 69
Critical load ... 10, 13, 34
Critical load factor ... 32, 80, 88
Cross-section ... 11, 19, 45
Cross-section deformation ... 26, 62
Cross-section info ... 47
Cross-section library ... 45
Cross-section stresses ... 91
Cross-section table ... 46
Cross-stiffening ... 30

D

Decimal places ... 43, 104
Deformation ... 86
Design case ... 94, 102, 103
Design of ... 41
Detail settings ... 77
Diagonal ... 58
DIN 18800 ... 32
Displacements ... 86
Display navigator ... 95, 98

E

Eccentricity ... 62, 63, 66, 72
Effective lengths ... 105
Eigenvalue analysis ... 37, 77
Eigenvector ... 37, 74
Elastic foundation ... 55
Elastic-elastic ... 38, 76
End plate ... 29, 51
Equivalent member method ... 9, 33
Equivalent stress σ_{eqv} ... 21, 38, 81, 82
Excel ... 106
Exit RF-FE-LTB ... 40
Export ... 105
Export cross-section ... 105
Export material ... 105

F

FE division length ... 77, 82, 85
Filter ... 98
Filtering members ... 99
Finite elements ... 11
Flexural buckling load N_{cr} ... 10
Flexural torsion ... 11
Flexural-torsional buckling ... 9

G

General data ... 40
Graphic printout ... 100
Graphics ... 94
Groove ... 22, 24
Group of members ... 41

H

Hidden result diagram 98

I

Imperfection 37, 69, 74, 77
Increment 78
Initial camber 37, 76
Initial deformation 10, 11, 37
Initial sway 37, 75
Instability 88
Installation 7
Internal forces 85
Introduced load 71
Introduced loads 69
Iteration parameter 78
Iterations 89

L

Lastfall 71
Lateral load 72
Lateral restraint 56
Lateral-torsional buckling 9
Limit load F_G 10, 21, 39, 77
Limit load F_T 10, 39, 77
Limit stress 82
Lindner/Groeschel 59
Line load 18
List of members 71
Load 69
Load case 41, 69, 82, 85
Load combination 41, 69, 71, 85
Load direction 72
Load distribution 72
Load increment 77
Load type 71
Location x 82, 85
Loss of stability 77
LTB 9

M

Material 43, 105
Material description 43
Material library 44
Material properties 43
Member end release 67
Member end spring 63
Member foundation 55
Member load 71
Member loads 72
Member side 63, 67
Module windows 40
Moment for lateral-torsional buckling M_{cr} 10, 34

N

Navigator 40
Nodal forces 70
Nodal loads 69
Nodal moments 70
Nodal support 49
Noncontinuous rotational restraint 25
Normal stress σ 81
Normal stresses σ 20

O

OpenOffice 106

P

Panel 8, 95, 98
Partial safety factor 78
Post 58
Print 100
Printout report 100, 101
Purlin 25, 61

R

Reaction forces 87
Reduction factor 76
Reference length 72, 76
Relation scale 90
Release 67, 68
Remark 46
Rendering 98
Restrained axis of rotation 22
Restraint 50
Result combination 41
Result diagram 97, 100
Result values 94
Results display 95
Results evaluation 90
Results windows 80
RFEM 86, 96
RFEM graphic 100
RFEM work window 93
Rotational restraint 25, 26, 56
Rotational restraint coefficient 59

Rotational spring 25, 65

Rotations 86

S

Second-order analysis 9, 12, 32

Self-weight 71

Set of members 41, 71, 83, 88

SHAPE-MASSIVE 82

SHAPE-THIN 82

SHAPE-THIN sections 46

Shear center 18, 72

Shear force 85

Shear panel 23, 57

Shear stress τ 20, 81, 93

Shear stresses τ 21

Sign 19

Sings of support forces 87

Spring 11, 15, 49, 55, 64

Stability 88

Stability problem 9

Stainless steel 44

Start calculation 79

Start nu-factor ν 78

Start program 7

Start RF-FE-LTB 7

Statical moment 93

Stress 77, 81

Stress analysis 33, 82

Stress diagram 91

Stress graphic 91

Stress point 19, 48, 72, 82, 84, 91, 92

Stress ratio 82, 91

Stress type 81

Stresses 19, 82

Support 50

Supporting component 25

T

Taper 46, 77, 81

Thickness of structural component 93

Torsion 72

Torsional buckling load $N_{cr,\vartheta}$ 10

Torsional moment 85

Translational restraint 24

Translational spring 27, 64

Trapezoidal sheet 60

Trapezoidal sheeting 22, 24, 27

Twisting radius 35

Type of connection 58

U

Units 43, 104

User profile 104

User-defined cross-section 82

V

Value s 74, 75

View mode 90, 93

Visibilities 98

W

Warp spring C_ω 29, 31, 50, 66

Warping 68, 86, 93

Warping restraint 50

Warping torsion 20, 85

Warping torsional moment 68, 85, 87

Warping torsional theory 9

With/without diagram 98

X

x-location 82, 83, 85