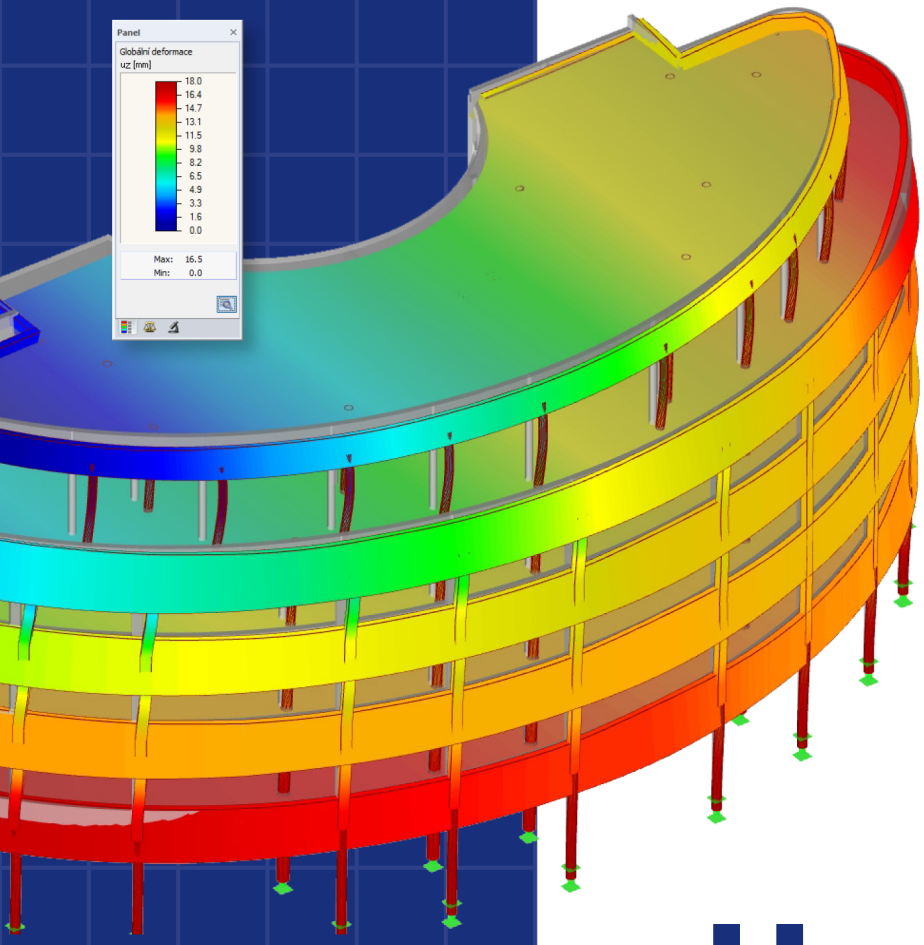
# RF-/LIMITS

## Comparison of Results with Defined Limit States




# User Manual

Version

**April 2018**

Dlubal

**Dlubal Software**

# Short Overview




**Dlubal Software GmbH**
Am Zellweg 2
93464 Tiefenbach
Germany

Telephone: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com

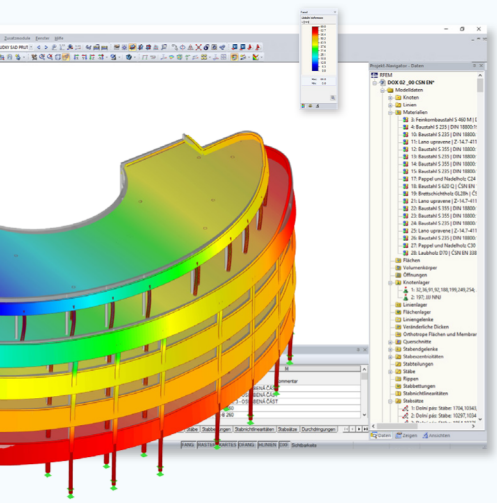
**Dlubal Software, Inc.**
The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Phone: +1 267 702-2815
E-mail: info@dlubal.com




## Using the Manual

The program description is organized in chapters which follow the order and structure of the input and result tables. The chapters present the individual tables column by column. They help to better understand the functioning of the add-on module. General functions are described in the manuals of the main program RFEM or RSTAB.



### *Hint*

The text of the manual shows the described buttons in square brackets, for example [OK]. In addition, they are pictured on the left. Expressions appearing in dialog boxes, tables, and menus are set in *italics* to clarify the explanation. You can also use the search function for the Knowledge Base and FAQs to find a solution in the posts about add-on modules.

### *Topicality*

The high quality standards placed on the software are guaranteed by a continuous development of the program versions. This may result in differences between program description and the current software version you are using. Thank you for your understanding that no claims can be derived from the figures and descriptions. We always try to adapt the documentation to the current state of the software.

# Table of Contents

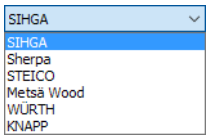
# 1 Introduction

## 1.1 RF-/LIMITS Add-on Module

The add-on modules RF-LIMITS (for RFEM) and LIMITS (for RSTAB) provide active structural engineers a comprehensive overview about the utilization of connecting elements or whether internal forces and deformations are met in the entire model.

In the following, the add-on modules of both main programs are described in one manual and are referred to as **RF-/LIMITS**.

Basically, RF/-LIMITS does not depend on any standard. However, you have the possibility to automatically include the specifications of the timber standards EN 1995 [1] and DIN 1052 [2] for the designs.




An extensive database offering connecting elements makes it easier to start working with the add-on module. It is also possible to extend the database by any other element. Especially for the design of connecting elements used in timber construction (for example joist hangers) the load influence factors $k_{mod}$ describing long-term effects can be determined conforming to standards.

RF-/LIMITS, like all other add-on modules, is integrated in the main programs RFEM and RSTAB: With the option to access the results of RFEM/RSTAB you can analyze modifications of the model or loading quickly in the add-on module. The results of the limit check can be documented as well in the printout report of RFEM and RSTAB.

Short description:

- Checking internal forces on member ends, members and surfaces (RFEM only)
- Checking deformations of nodes
- Checking forces and moments of nodal supports
- Check of cross-section dimensions
- User-defined design areas are taken into account
- Interaction design for axial force, shear force and torsional moment
- Database with DSTV connections according to EN 1993-1-8 [3]

We hope you will enjoy working with RF-/LIMITS.

Your Dlubal Software team

## 1.2 Using the Manual

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

The description of the program follows the sequence and structure of the module's input and results windows. The text of the manual shows the described **buttons** in square brackets, for example [Preset all members]. The buttons are also shown on the left margin. **Expressions** appearing in dialog boxes, tables, and menus are set in *italics* to clarify the explanation.

As usual, a full-text search is possible in the PDF manual with [Ctrl] + [F]. If you do not find what you are looking for, you can also use the search function for the Knowledge Base on our website to find a solution among the articles on the RF-/LIMITS product.

## 1.3 Open the Add-on Module RF-/LIMITS

RFEM and RSTAB provide the following options to start the add-on module RF-/LIMITS.

### Menu

To start the program on the RFEM or RSTAB menu bar, select

**Add-on Modules ⟶ Connections ⟶ RF-/LIMITS.**

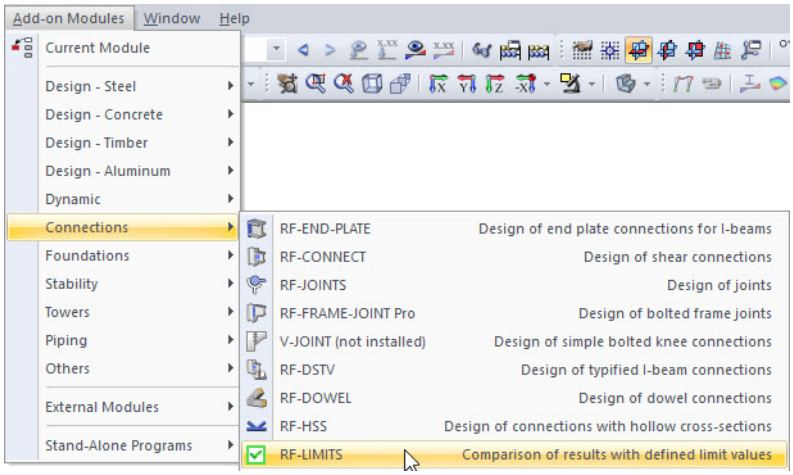



**Figure 1.1** Menu *Add-on Modules ⟶ Connections ⟶ RF-/LIMITS*

## Navigator

To start RF-/LIMITS in the *Data* navigator, select

**Add-on Modules ⟶ RF-/LIMITS.**

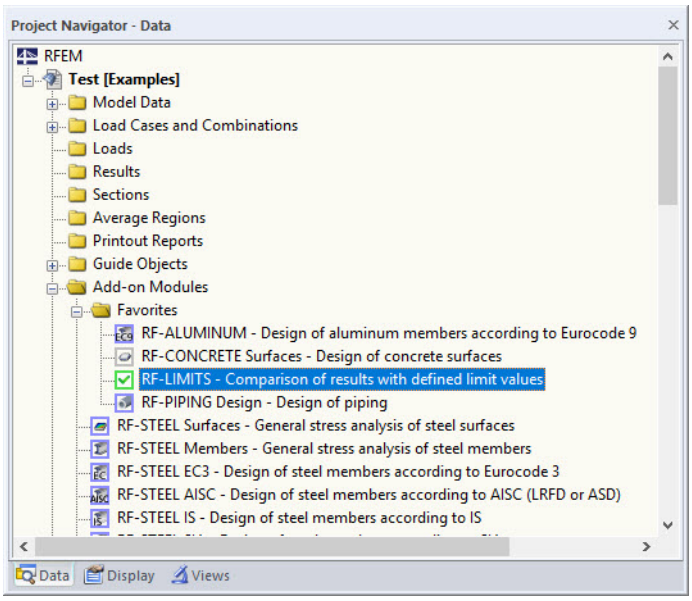



**Figure 1.2** *Data navigator: Add-on Modules ⟶ RF-/LIMITS*

# 2 Input Data

When you start the add-on module, a new window opens. In this window, a navigator is displayed on the left, managing the available module windows. The drop-down list above the navigator contains the design cases.

The data that is relevant for the limit values must be defined in two input windows. If you set a timber standard for the analysis, a third input window for the definition of the load duration class is automatically added.

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

To save the entered data, click [OK]. You will exit RF-/LIMITS and return to the main program. Click [Cancel] to exit the add-on module without saving the data.

## Units and Decimal Places

The units and decimal places for RFEM and RSTAB and the add-on modules are managed together. To access the dialog box for adjusting the units in RF-/LIMITS, use the menu and click

**Settings → Units and Decimal Places.**

The program opens the following dialog box that you already know from RFEM and RSTAB. RF-/LIMITS is preset in the *Program / Module* list.

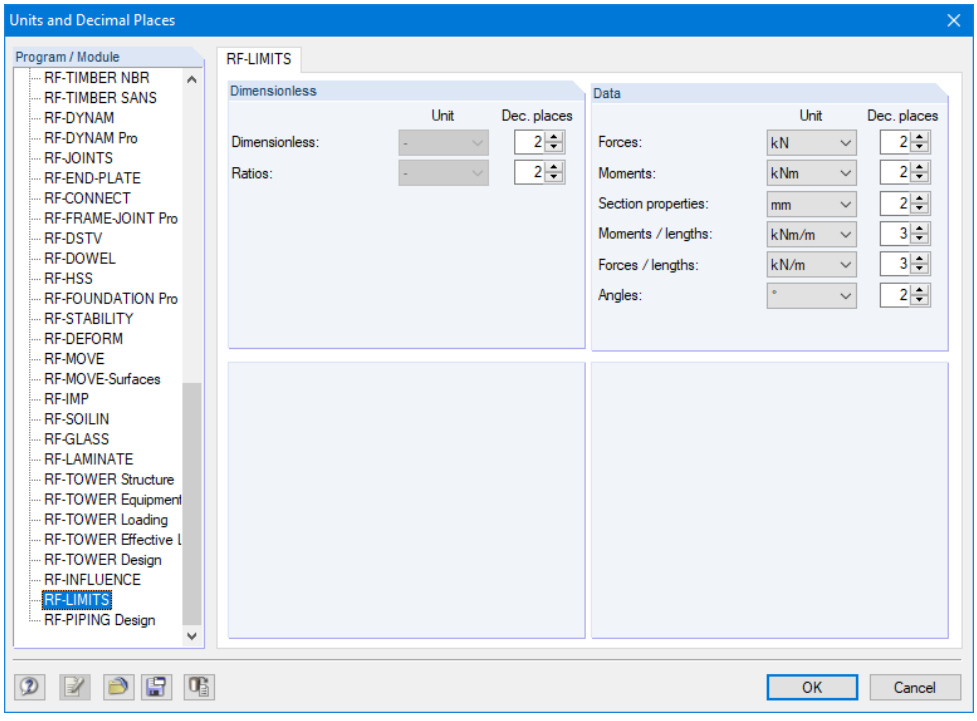
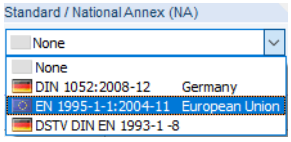



**Figure 2.1** Dialog Box *Units and Decimal Places*

The modified settings can be saved as user profile and reused in other models. The functions are described in chapter 11.1.3 of the RFEM or RSTAB manual.

## 2.1

# General Data




In module window *1.1 General Data*, select the load cases and combinations whose results you want to compare with the limit states. Then, you decide if you want to perform the limit check according to a standard. The European standard EN 1995-1-1 [1], the German standard DIN 1052:2008 [2] as well as DSTV DIN EN 1993-1-8 [3] including different national application documents are available.

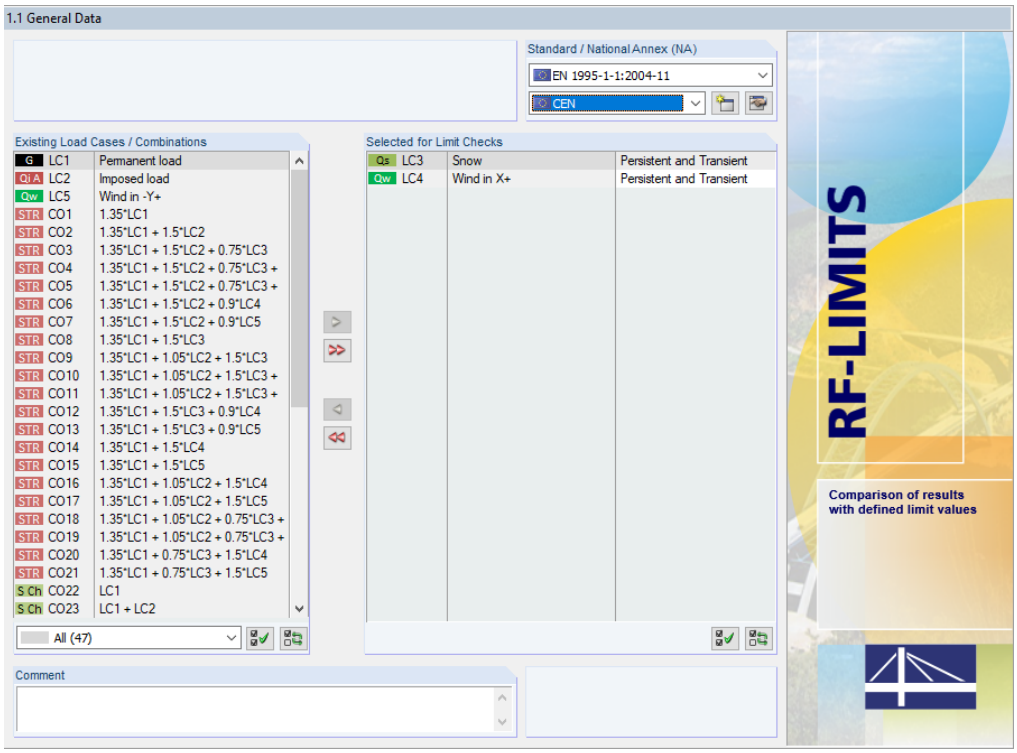



**Figure 2.2** Module window *1.1 General Data*

## Standard / National Annex


**Figure 2.3** Selection of standard and National Annex

The partial safety factor as well as the modification factor for the chosen material are defined automatically with the selection of the national application documents of the European countries.

Use the [Edit] button to open a dialog box where you can check and, if necessary, adjust the parameters of the selected standard or National Annex (see Image 2.4 ).

Nat. Annex...

You can also use the [Nat. Annex] button to open the *Material Factors* dialog box.

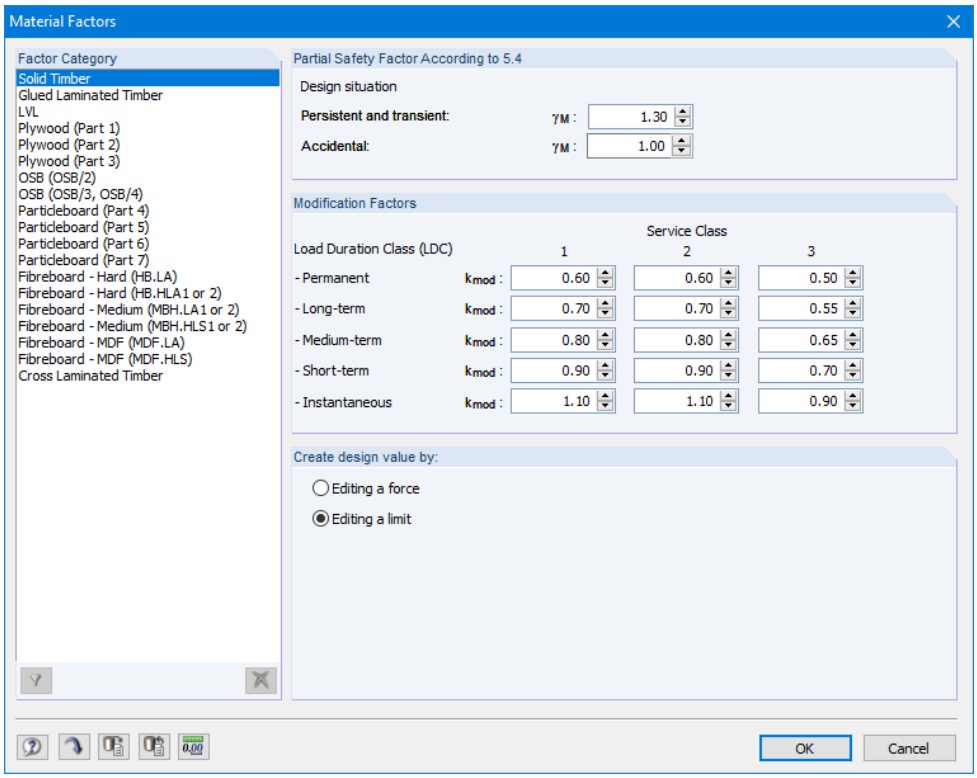



**Figure 2.4** Dialog box *Material Factors*

If *None* is set, you can only activate a user-defined partial safety factor $\gamma_M$ in this dialog box.

## Factor Category

This dialog section offers various timber materials for selection. The entries in the list differ according to the selected standard and annex.

Use the button [Include unassigned] to also display categories which are not used.

## Partial Safety Factor / Modification Factors

The two dialog sections show the partial safety and modification factors of the service classes which apply to the selected material and the corresponding design situations.

For example, there is no service class 3 possible for oriented strand boards (OSB): The column is grayed out in this case.

## Create Design Value

The strengths that are contrasted with the stresses are reduced by default as follows, depending on the class (LDC) defined in module window *1.3 Load Duration Class*:

$$f_d = \frac{f_k \cdot k_{mod}}{\gamma_M}$$

**Equation 2.1** Creation of design value by editing a limit (reduction of strength)

Use the options to decide if the strengths are raised to a design level (*Editing a limit*, see Equation 2.1) or if the actions are increased. If the *design value is increased by editing a force*, Equation 2.1 is inverted:

$$f_d = \frac{F_k \cdot \gamma_M}{k_{mod}}$$

**Equation 2.2** Creation of design value by editing a force (increase of action)

If you want to define the factors and coefficients by user-defined specifications, you have to create a standard first by clicking the button. Then, you can set the parameters individually for that standard.

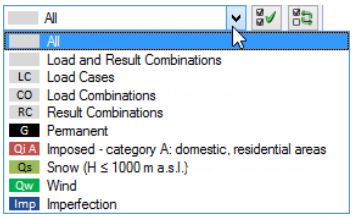
## Existing Load Cases and Combinations

This column lists all load cases, load combinations, and result combinations that have been created in RFEM or RSTAB.

To transfer selected entries to the [Selected for Design] list on the right, click the button. Alternatively, you can double-click the entries. To transfer the entire list to the right, use the button.

As common for Windows applications, selecting several load cases is possible by clicking them one by one while holding down the [Ctrl] key. Thus, you can transfer several load cases at the same time.

If a load case's number is marked in red, you cannot design it: It indicates a load case without load data, or a load case that contains imperfections. A warning appears if you try to transfer it.




Below the list, several filter options are available. They help you assign the entries sorted by load case, load combination, or action category. The buttons have the following functions:

| | |
|---|---|
| | Selects all load cases in the list |
| | Inverts selection of load cases |

**Table 2.1** Buttons in module window *1.1 General Data*

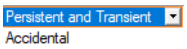
## Selected for Limit Checks

The column on the right lists the load cases as well as the load and result combinations that have been selected for the design. To remove selected items from the list, click or double-click the entries. To empty the entire list, click .

You can assign the load cases as well as the load and result combinations to the following design situations:

- *Persistent and Transient*
- *Accidental*

This classification controls the partial factors $\gamma_M$ that are included in the determination of the resistances (see Equation 2.1).




To change the design situation, use the list which you can access by clicking the button at the end of the text box.

For a multiple selection, press the [Ctrl] key and click the corresponding entries. Thus, you can change several entries at once.

# 2.2 Limit Parameters

In module window 1.2, select the objects (member ends, members, nodes, etc.) and the limit values of the action effects.

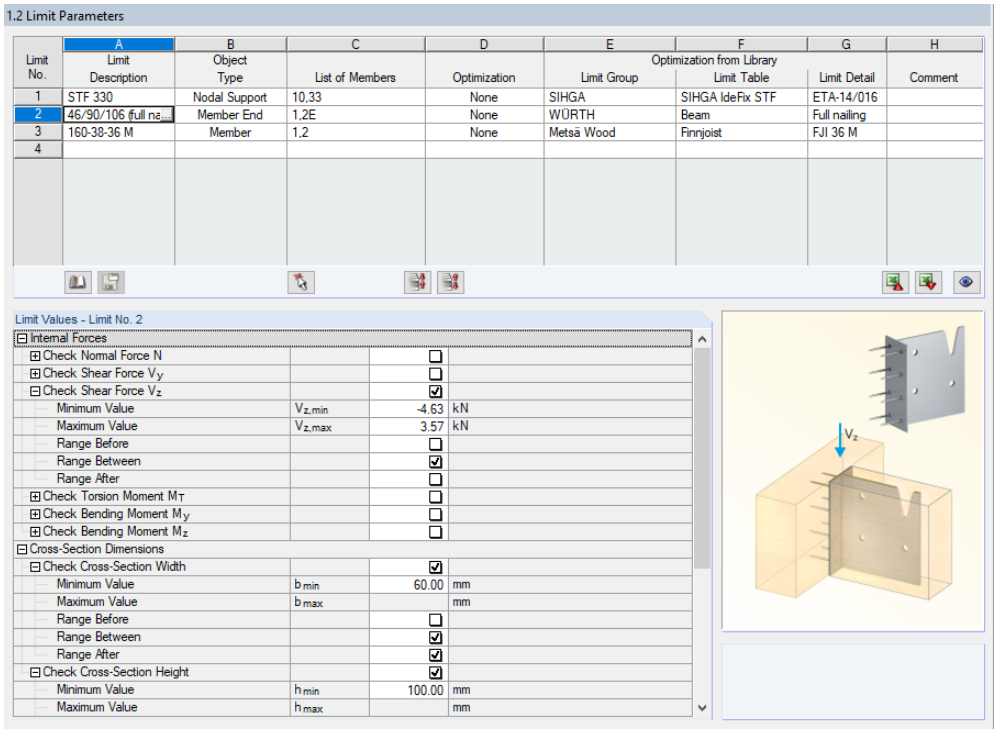



**Figure 2.5** Module window *1.2 Limit Parameters*

## Limit Description

In the first table column, enter the descriptions of the limit checks. To access the *Limit Library* use the [Library] button and ... that appear when you click into a table cell.

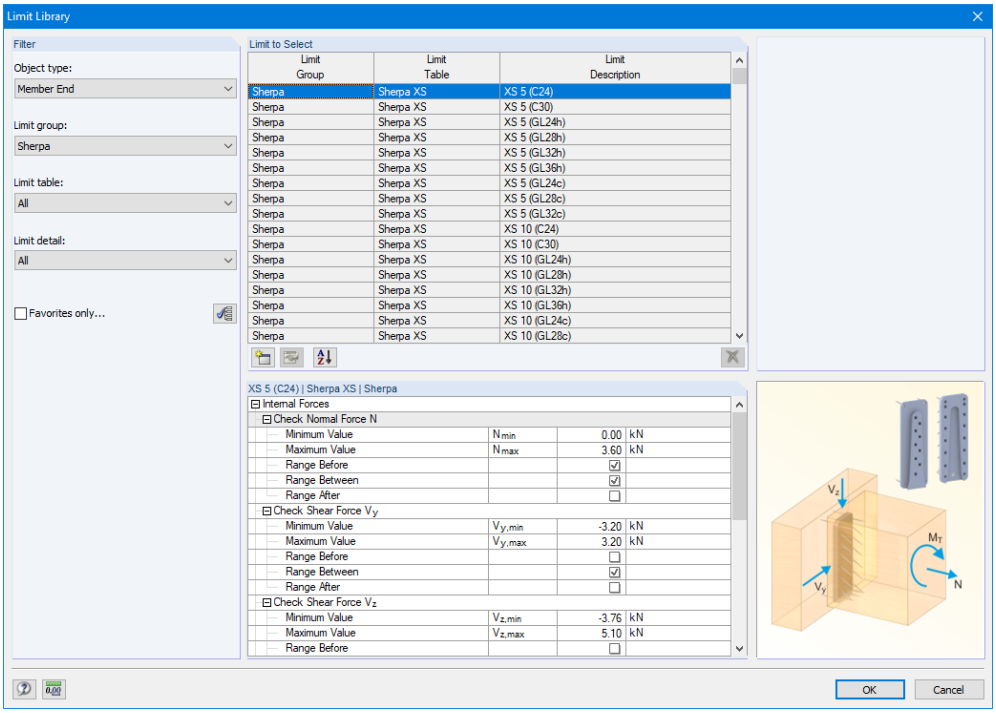
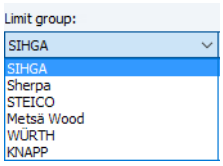
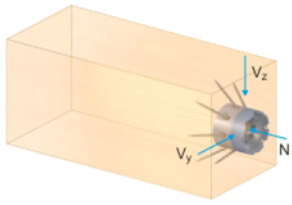
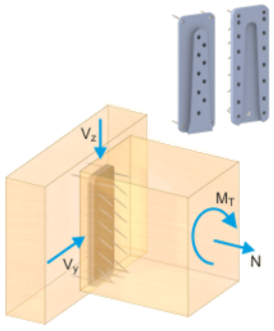



**Figure 2.6** Dialog box *Limit Library*




At the moment, the library contains connections of the following companies: Sihga, Sherpa, Steico, Metsä Wood, Würth, Knapp and M-connect. Furthermore, the library contains connections which are authorized by DSTV (German Steel Engineering Association).

| **Limit group** | **Connection** |
|---|---|
| Sihga |  IdeFix group |
| Sherpa |  |

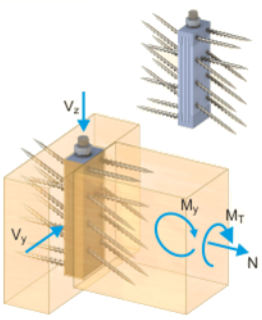
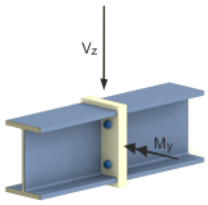
| | |
|---|---|
| Knapp |  |
| DSTV |  <br>IH group |

**Table 2.2** Connections of the library (selection)

With the button in the library you can define the relevant limit values for a new "limit element" (see Image 2.10 ).

Furthermore, it is possible to define a new *Limit Group* and *Limit Table* so that limit zones can be filtered quickly later.

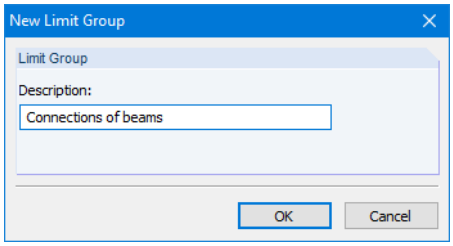



**Figure 2.7** Dialog box *New Limit Group*

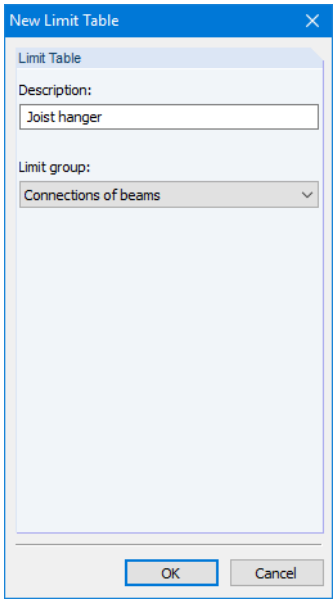



**Figure 2.8** Dialog box *New Limit Table*.

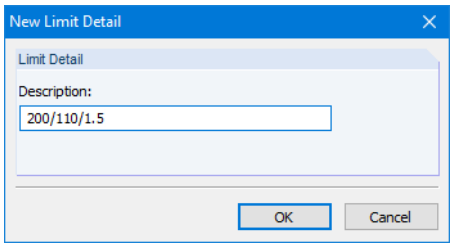


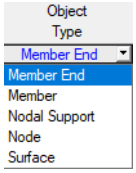

**Figure 2.9** Dialog box *Limit Detail*

In the dialog box, you have also the possibility to add a picture to describe the limit value. Many manufacturers of connecting elements or CAD software offer graphics on their web pages.

It is always possible to adjust the limit groups or tables by using the button.

Frequently used limit elements can be marked as *Favorite* to find them quickly.

## Object Type




In table column B, you can select the object type that is relevant for the limit. If the object does not fit to the corresponding limit, the type will be shown in blue.

### Member End

For the first object type of the list you can define limit values (min/max) for the *Internal Forces* N, $V_y$, $V_z$, $M_y$, $M_z$, $M_T$ and the *Cross-Section Dimensions* (width and height).

| | | | |
|---|---|---|---|
| ⊟ Internal Forces | | | |
| ⊞ Check Normal Force N | | ☐ | |
| ⊞ Check Shear Force $V_y$ | | ☐ | |
| ⊟ Check Shear Force $V_z$ | | ☑ | |
| Minimum Value | $V_{z,min}$ | -4.63 | kN |
| Maximum Value | $V_{z,max}$ | 3.57 | kN |
| Range Before | | ☐ | |
| Range Between | | ☑ | |
| Range After | | ☐ | |
| ⊞ Check Torsion Moment $M_T$ | | ☐ | |
| ⊞ Check Bending Moment $M_y$ | | ☐ | |
| ⊞ Check Bending Moment $M_z$ | | ☐ | |
| ⊟ Cross-Section Dimensions | | | |
| ⊟ Check Cross-Section Width | | ☑ | |
| Minimum Value | $b_{min}$ | 60.00 | mm |
| Maximum Value | $b_{max}$ | | mm |
| Range Before | | ☐ | |
| Range Between | | ☑ | |
| Range After | | ☑ | |
| ⊟ Check Cross-Section Height | | ☑ | |
| Minimum Value | $h_{min}$ | 100.00 | mm |
| Maximum Value | $h_{max}$ | | mm |

**Figure 2.10** Dialog box *Limit Values* for object type *Member End* (part of module window)

With the check boxes for *Range Before*, *Range Between* and *Range After* you decide if the zone is relevant

- **before** the defined minimum value (*Range Before*)

For limit axial forces of e.g. $N_{min}$ = and $N_{max}$ = the program checks if the member axial force is smaller than 5 kN.

- **between** both defined values (*Range Between*)

The zone between both limit values is verified in the check. In our example above, the program checks if the shear forces are available in the zone between ± 10 kN.

- **after** the defined maximum value (*Range After*)

The check is done for the zone above the defined maximum value. In our axial force example above, the program checks if the tension force is greater than 10 kN.

when checking the internal forces, deformations or support forces.

Interaction

In addition, an interaction design is possible for the simultaneous action of the internal forces. This allows, amongst other things, to determine the influence of a biaxial bending. The internal forces are superimposed as follows:

$$\text{Int} = (\eta N)^2 + (\eta V_y)^2 + (\eta V_z)^2 + \eta M_T$$

**Equation 2.3**

The ratio from torsion is thus not squared.

## Member

For this object type you can define the same limits like for the member ends. In contrast to the ends of a member, the limit area is checked on the entire member.

## Nodal Support

The support force is checked in X-, Y- and Z-direction. A definition of the limit moments $M_X$, $M_Y$ and $M_Z$ is additionally possible. The ranges must be defined in the same way.

### Node

For the *Node* object type you can check the rotations about the X-, Y- and Z-axes in addition to the displacements in X, Y and Z.

The following Knowledge Base Article describes how limits are defined for horizontal displacements of a building: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000833

### Surface (Only RF-LIMITS)

All internal forces in surfaces ($m_x$, $m_y$, $m_{xy}$, $v_x$, $v_y$, $n_x$, $n_y$, $n_{xy}$ etc.) can be analyzed. The zones are defined in the same way.

## List of Elements

In table column C, specify the numbers of the nodes, members, member ends, supports and surfaces that you want to check.

It is also possible to select the objects graphically in the RSTAB or RFEM work window by using the button.

## Service Class

This column is only displayed if an analysis according to EN 1995-1-1 [1] or DIN 1052 [2] in module window *1.1 General Data* has been defined. The service classes 1 to 3 are available for selection.

## Factor Category

Also this table column is displayed only for EN 1995-1-1 or DIN 1052. The categories of the materials are described in Chapter 2.1.

## Optimization

This table column offers the option to optimize the selected limits. The optimization can be performed in accordance with the defined *Limit Group*, *Limit Table* or *Limit Detail*.

| Optimization | Optimization from Library | | |
|---|---|---|---|
| | Limit Group | Limit Table | Limit Detail |
| From Limit Table | Sherpa | Sherpa S | C24 |
| None | | | |
| From Limit Group | | | |
| From Limit Table | | | |
| From Limit Detail | | | |

**Figure 2.11** Optimization options

## Optimization from Library

### Limit Group

When an optimization is set, you can define the corresponding *Limit Group* in table column G. Use the button to select the group.

### Limit Table

Table column H manages the tables that are relevant for the optimization.

### Limit Detail

In this column, the governing category details for the optimization are displayed.

These options simplify the selection – particularly for the numerous DSTV connection types.

### Comment

It is possible to enter a note in the last column which explains the limit parameter. It is also displayed in the printout report.

The data of module window *1.2 Limit Parameters* can be exchanged with MS Excel by using the import and export functions. The buttons have the following meanings:

| Button | Description | Function |
|---|---|---|
|  | Excel export | Exports content of the module window to MS Excel |
|  | Excel import | Imports content of a table from MS Excel |
|  | Export to library | Saves a new defined limit group |
|  | View mode | Switches to RFEM or RSTAB work window without quitting RF/-LIMITS |

Table 2.3 Buttons in module window *1.2 Limit Parameters*

## 2.3 Load Duration Class

This input window appears only if an analysis according to EN 1995-1-1 [2] or DIN 1052 [2] has been set in module window *1.1 General Data*. You define the load duration of the loads in order to determine the climatic conditions for the limit checks.

1.3 Load Duration Class

| LC/CO/RC | A<br>LC/CO/<br>RC Description | B<br>LC Type | C<br>Load Duration Class | D<br>Comments |
|---|---|---|---|---|
| LC1 | Permanent load | Permanent | Permanent | |
| LC4 | Wind in X+ | Wind | Short-term | |
| CO35 | LC1 + LC4 | - | | |

Permanent
Long-term
Medium-term
Short-term
Instantaneous

**Figure 2.12** Module window *1.3 Load Duration Class*

## LC/CO/RC

The table column lists all actions that have been selected for the limit checks in module window *1.1 General Data*. In case of combinations, included load cases are also displayed.

## LC/CO/RC description

The load case descriptions make the classification easier.

## LC Type

This table column shows the action types of the load cases as they were defined in RFEM or RSTAB during their creation. They form the basis for the settings in the next table column.

## Load Duration LDC

Load Duration Class
Permanent
Permanent
Long-term
Medium-term
Short-term
Instantaneous

The limit checks require the assignment of loads and their superpositions to particular load duration classes. For load cases you can change the load duration with the list shown on the left: Click into a table cell in column C. The button will be enabled. In case of unambiguous load and result combinations, RF-/LIMITS carries out the classification automatically taking intoaccount the respective governing action.

The load duration class LDC is required for the determination of the modification factor $k_{mod}$. The factor is also considered for the material stiffness.

If an automatic combination of the actions was set in RFEM or RSTAB, the load duration classes are automatically taken into account according to the specifications in RFEM or RSTAB. Thus, a redefinition in RF-/LIMITS is not necessary. However, it is possible to adjust this classification by selecting another setting.

# 3 Results

Module window *2.1 Limit Checks by Loading* appears immediately after the calculation.

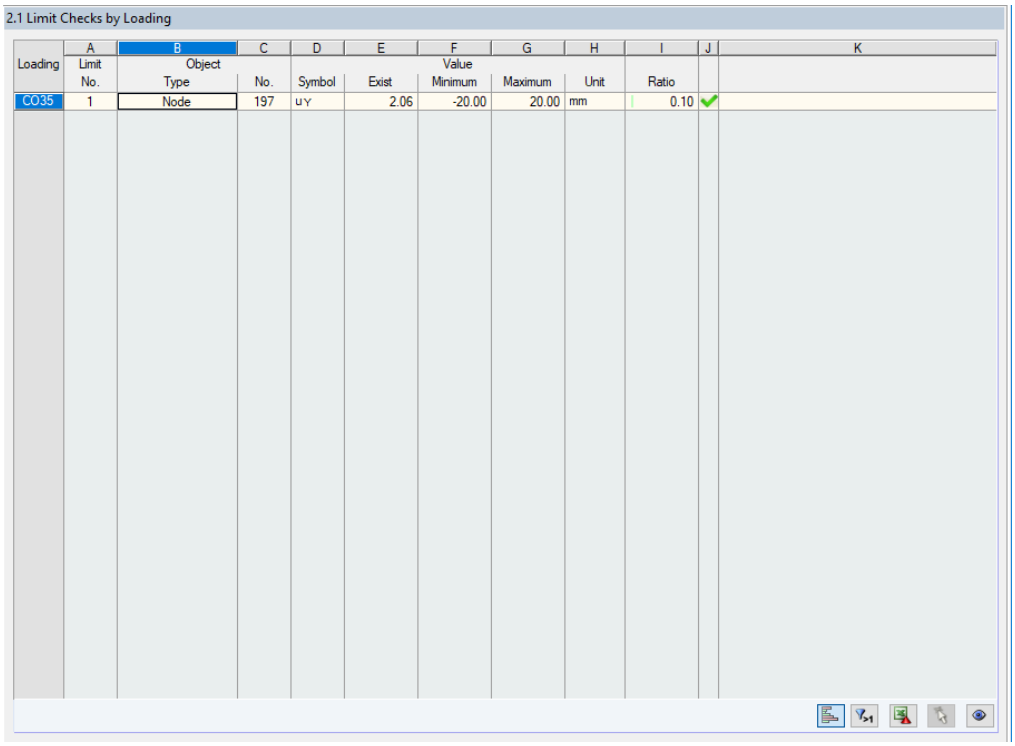



**Figure 3.1** Module window *2.1 Limit Checks by Loading*

The results of the limit value check are sorted by various criteria in the result windows 2.1 to 2.6.

Every window can be selected by clicking the corresponding entry in the navigator. To go to the previous or next module window, use the buttons shown on the left. You can also use the function keys [F2] and [F3] to go through the windows.

The results can be evaluated in different ways. The buttons below the table may help you. They have the following functions:

| Button | Description | Function |
|---|---|---|
| | Relation scales | Shows or hides the colored scales in the table column *Ratio* |
| | Ratios > 1 | Shows only rows with ratios greater than 1 (design not fulfilled) |
| | Export to MS Excel | Exports contents of the table to MS Excel |
| | Member selection | Allows for the graphical selection of a member to display its results in the table |

| | | |
|---|---|---|
|  | View mode | Jumps to the RFEM or RSTAB work window to change the view |

Table 3.1 Buttons in result windows

## 3.1 Limit Checks by Loading

This result window shows the maximum ratio for each of the selected objects.

The governing object is displayed in table columns B and C. The available internal force or deformation of the object is shown in column E. The indicated values are contrasted with the limits registered in the limit library.

As far as it is possible for the limit, table column I shows a corresponding ratio.

2.1 Limit Checks by Loading

| Loading No. | A Limit No. | B Object Type | C No. | D Symbol | E Exist | F Value Minimum | G Maximum | H Unit | I Ratio | J | K |
|---|---|---|---|---|---|---|---|---|---|---|---|
| LC1 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO1 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| LC2 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO2 | 1 | Member End | 21 | $V_z$ | -25.17 | -23.15 | 23.15 | kN | 1.087 | ✗ | |
| LC3 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO3 | 2 | Nodal Support | 15 | Pz | 81.12 | -70.42 | 70.42 | kN | 1.152 | ✗ | |
| LC4 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO4 | 2 | Nodal Support | 15 | Pz | 82.71 | -70.42 | 70.42 | kN | 1.175 | ✗ | |
| CO5 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO6 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO7 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO8 | 1 | Member End | 5 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ | |
| CO9 | 2 | Nodal Support | 15 | Pz | 92.86 | -70.42 | 70.42 | kN | 1.319 | ✗ | |
| CO10 | 2 | Nodal Support | 15 | Pz | 96.04 | -70.42 | 70.42 | kN | 1.364 | ✗ | |

Figure 3.2 Module window 2.1 *Limit Checks by Loading*

Table column E always shows as the *Exist* value the first possible value that was defined by you. The other values are shown in the subsequent result windows.

## 3.2 Limit Checks by Set of Members

This results window shows the ratios available in the sets of members.

2.2 Limit Checks by Set of Members

| Set\No. | A Limit No. | B Object Type | C Member No. | D Loading No. | E Value Symbol | F Value Exist | G Value Minimum | H Value Maximum | I Value Unit | J Ratio | K | L |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 3 | 1 | Member End - End | 21 | CO2 | $V_z$ | -25.17 | -23.15 | 23.15 | kN | 1.087 | ✗ | |
| 4 | 1 | Member End - End | 24 | LC1 | b | 180.00 | 100.00 | 100.00 | mm | 0.556 | ✓ | |

**Figure 3.3** Module window 2.2 *Limit Checks by Set of Members*

## 3.3 Limit Checks by Cross-Section

The results of the limit value checks performed for all selected cross-sections are shown in this results window. The criterion for members stored in the database serves as a limit.

2.3 Limit Checks by Cross-Section

| Section No. | A Cross-Section Description | B Limit No. | C Object Type | D Object No. | E Loading No. | F Value Symbol | G Value Exist | H Value Minimum | I Value Maximum | J Value Unit | K Ratio | L |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Rectangle 180/400 | 1 | Member End - End | 3 | LC1 | b | 180.00 | 100.00 | 100.00 | mm | 0.556 | ✓ |
| 2 | Rectangle 120/220 | 1 | Member End - Start | 4 | LC1 | h | 220.00 | 220.00 | 220.00 | mm | 1.000 | ✓ |
| 5 | Rectangle 140/440 | 1 | Member End - End | 21 | CO2 | $V_z$ | -25.17 | -23.15 | 23.15 | kN | 1.087 | ✗ |

**Figure 3.4** Module window 2.3 *Limit Checks by Cross-Section*

## 3.4 Limit Checks by Surface (Only RF-LIMITS)

Module window 2.4 is displayed when at least one surface was selected for the design. The limiting criteria are described in detail in Chapter 2.2.

2.4 Limit Checks by Surface

| No. | A Limit No. | B Object Type | C Surface No. | D Loading No. | E Symbol | F Value Exist | G Value Minimum | H Value Maximum | I Unit | J Ratio | K | L |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Custom material 4 | | | | | | | | | | | |
| 2 | 4 | Surface | 1 | LC1 | $m_{x,min}$ | -0.179 | -2.308 | 2.308 | kNm/m | 0.078 | ✓ | |
| 3 | 4 | Surface | 1 | LC1 | $m_{x,max}$ | 0.720 | -2.308 | 2.308 | kNm/m | 0.312 | ✓ | |
| 4 | 4 | Surface | 2 | LC1 | $m_{x,min}$ | -0.128 | -2.308 | 2.308 | kNm/m | 0.055 | ✓ | |
| 5 | 4 | Surface | 2 | LC1 | $m_{x,max}$ | 0.432 | -2.308 | 2.308 | kNm/m | 0.187 | ✓ | |
| 6 | 4 | Surface | 3 | LC1 | $m_{x,min}$ | -0.087 | -2.308 | 2.308 | kNm/m | 0.038 | ✓ | |
| 7 | 4 | Surface | 3 | LC1 | $m_{x,max}$ | 0.635 | -2.308 | 2.308 | kNm/m | 0.275 | ✓ | |
| 8 | 4 | Surface | 1 | CO1 | $m_{x,min}$ | -0.242 | -2.308 | 2.308 | kNm/m | 0.105 | ✓ | |
| 9 | 4 | Surface | 1 | CO1 | $m_{x,max}$ | 0.973 | -2.308 | 2.308 | kNm/m | 0.421 | ✓ | |
| 10 | 4 | Surface | 2 | CO1 | $m_{x,min}$ | -0.173 | -2.308 | 2.308 | kNm/m | 0.075 | ✓ | |
| 11 | 4 | Surface | 2 | CO1 | $m_{x,max}$ | 0.583 | -2.308 | 2.308 | kNm/m | 0.253 | ✓ | |
| 12 | 4 | Surface | 3 | CO1 | $m_{x,min}$ | -0.117 | -2.308 | 2.308 | kNm/m | 0.051 | ✓ | |
| 13 | 4 | Surface | 3 | CO1 | $m_{x,max}$ | 0.858 | -2.308 | 2.308 | kNm/m | 0.372 | ✓ | |
| 14 | 4 | Surface | 1 | LC2 | $m_{x,min}$ | -0.767 | -2.692 | 2.692 | kNm/m | 0.285 | ✓ | |
| 15 | 4 | Surface | 1 | LC2 | $m_{x,max}$ | 4.319 | -2.692 | 2.692 | kNm/m | 1.604 | ✗ | |
| 16 | 4 | Surface | 2 | LC2 | $m_{x,min}$ | -0.569 | -2.692 | 2.692 | kNm/m | 0.211 | ✓ | |
| 17 | 4 | Surface | 2 | LC2 | $m_{x,max}$ | 2.409 | -2.692 | 2.692 | kNm/m | 0.895 | ✓ | |
| 18 | 4 | Surface | 3 | LC2 | $m_{x,min}$ | -0.304 | -2.692 | 2.692 | kNm/m | 0.113 | ✓ | |
| 19 | 4 | Surface | 3 | LC2 | $m_{x,max}$ | 3.839 | -2.692 | 2.692 | kNm/m | 1.426 | ✗ | |
| 20 | 4 | Surface | 1 | CO2 | $m_{x,min}$ | -1.385 | -2.692 | 2.692 | kNm/m | 0.514 | ✓ | |
| 21 | 4 | Surface | 1 | CO2 | $m_{x,max}$ | 7.452 | -2.692 | 2.692 | kNm/m | 2.768 | ✗ | |
| 22 | 4 | Surface | 2 | CO2 | $m_{x,min}$ | -1.022 | -2.692 | 2.692 | kNm/m | 0.380 | ✓ | |
| 23 | 4 | Surface | 2 | CO2 | $m_{x,max}$ | 4.197 | -2.692 | 2.692 | kNm/m | 1.559 | ✗ | |
| 24 | 4 | Surface | 3 | CO2 | $m_{x,min}$ | -0.574 | -2.692 | 2.692 | kNm/m | 0.213 | ✓ | |
| 25 | 4 | Surface | 3 | CO2 | $m_{x,max}$ | 6.616 | -2.692 | 2.692 | kNm/m | 2.457 | ✗ | |
| 26 | 4 | Surface | 1 | LC3 | $m_{x,min}$ | -1.226 | -3.077 | 3.077 | kNm/m | 0.399 | ✓ | |
| 27 | 4 | Surface | 1 | LC3 | $m_{x,max}$ | 1.398 | -3.077 | 3.077 | kNm/m | 0.454 | ✓ | |
| 28 | 4 | Surface | 2 | LC3 | $m_{x,min}$ | -0.306 | -3.077 | 3.077 | kNm/m | 0.099 | ✓ | |
| 29 | 4 | Surface | 2 | LC3 | $m_{x,max}$ | 1.383 | -3.077 | 3.077 | kNm/m | 0.450 | ✓ | |
| 30 | 4 | Surface | 3 | LC3 | $m_{x,min}$ | -0.505 | -3.077 | 3.077 | kNm/m | 0.164 | ✓ | |
| 31 | 4 | Surface | 3 | LC3 | $m_{x,max}$ | 2.200 | -3.077 | 3.077 | kNm/m | 0.715 | ✓ | |
| 32 | 4 | Surface | 1 | CO3 | $m_{x,min}$ | -2.262 | -3.077 | 3.077 | kNm/m | 0.735 | ✓ | |
| 33 | 4 | Surface | 1 | CO3 | $m_{x,max}$ | 7.195 | -3.077 | 3.077 | kNm/m | 2.338 | ✗ | |
| 34 | 4 | Surface | 2 | CO3 | $m_{x,min}$ | -1.244 | -3.077 | 3.077 | kNm/m | 0.404 | ✓ | |

**Figure 3.5** Module window 2.4 *Limit Checks by Surface*

## 3.5 Limit Checks by Member

2.5 Limit Checks by Member

| No. | A Limit No. | B Object Type | C Member No. | D Loading No. | E Symbol | F Value Exist | G Value Minimum | H Value Maximum | I Unit | J Ratio | K | L |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 176 | 1 | Member End - End | 21 | CO1 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 177 | 1 | Member End - End | 21 | CO1 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 178 | 1 | Member End - End | 21 | LC2 | $V_y$ | 0.00 | -12.22 | 12.22 | kN | 0.000 | ✓ | |
| 179 | 1 | Member End - End | 21 | LC2 | $V_z$ | -14.27 | -23.15 | 23.15 | kN | 0.616 | ✓ | |
| 180 | 1 | Member End - End | 21 | LC2 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 181 | 1 | Member End - End | 21 | LC2 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 182 | 1 | Member End - End | 21 | CO2 | $V_y$ | 0.00 | -12.22 | 12.22 | kN | 0.000 | ✓ | |
| 183 | 1 | Member End - End | 21 | CO2 | $V_z$ | -25.17 | -23.15 | 23.15 | kN | 1.087 | ✗ | |
| 184 | 1 | Member End - End | 21 | CO2 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 185 | 1 | Member End - End | 21 | CO2 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 186 | 1 | Member End - End | 21 | LC3 | $V_y$ | 0.00 | -13.97 | 13.97 | kN | 0.000 | ✓ | |
| 187 | 1 | Member End - End | 21 | LC3 | $V_z$ | -3.97 | -26.46 | 26.46 | kN | 0.150 | ✓ | |
| 188 | 1 | Member End - End | 21 | LC3 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 189 | 1 | Member End - End | 21 | LC3 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 190 | 1 | Member End - End | 21 | CO3 | $V_y$ | 0.00 | -13.97 | 13.97 | kN | 0.000 | ✓ | |
| 191 | 1 | Member End - End | 21 | CO3 | $V_z$ | -28.15 | -26.46 | 26.46 | kN | 1.064 | ✗ | |
| 192 | 1 | Member End - End | 21 | CO3 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 193 | 1 | Member End - End | 21 | CO3 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 194 | 1 | Member End - End | 21 | LC4 | $V_y$ | 0.01 | -15.72 | 15.72 | kN | 0.000 | ✓ | |
| 195 | 1 | Member End - End | 21 | LC4 | $V_z$ | -1.31 | -29.77 | 29.77 | kN | 0.044 | ✓ | |
| 196 | 1 | Member End - End | 21 | LC4 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 197 | 1 | Member End - End | 21 | LC4 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 198 | 1 | Member End - End | 21 | CO4 | $V_y$ | 0.00 | -15.72 | 15.72 | kN | 0.000 | ✓ | |
| 199 | 1 | Member End - End | 21 | CO4 | $V_z$ | -29.13 | -29.77 | 29.77 | kN | 0.978 | ✓ | |
| 200 | 1 | Member End - End | 21 | CO4 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 201 | 1 | Member End - End | 21 | CO4 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 202 | 1 | Member End - End | 21 | CO5 | $V_y$ | 0.00 | -15.72 | 15.72 | kN | 0.000 | ✓ | |
| 203 | 1 | Member End - End | 21 | CO5 | $V_z$ | -26.15 | -29.77 | 29.77 | kN | 0.878 | ✓ | |
| 204 | 1 | Member End - End | 21 | CO5 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 205 | 1 | Member End - End | 21 | CO5 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |
| 206 | 1 | Member End - End | 21 | CO6 | $V_y$ | 0.00 | -13.97 | 13.97 | kN | 0.000 | ✓ | |
| 207 | 1 | Member End - End | 21 | CO6 | $V_z$ | -9.71 | -26.46 | 26.46 | kN | 0.367 | ✓ | |
| 208 | 1 | Member End - End | 21 | CO6 | b | 140.00 | 100.00 | 100.00 | mm | 0.714 | ✓ | |
| 209 | 1 | Member End - End | 21 | CO6 | h | 440.00 | 220.00 | 220.00 | mm | 0.500 | ✓ | |

**Figure 3.6** Module window 2.5 *Limit Checks by Member*

The maximum utilization ratios of all members are shown here. In contrast to module window 2.2 *Limit Checks by Set of Members*, it is also possible to check the cross-section widths or depths.

## 3.6 Limit Checks by Node

2.6 Limit Checks by Node

| No. | A Limit No. | B Object Type | C Node No. | D Loading No. | E Value Symbol | F Value Exist | G Value Minimum | H Value Maximum | I Value Unit | J Ratio | K |
|---|---|---|---|---|---|---|---|---|---|---|---|
| 44 | 2 | Nodal Support | 15 | LC1 | Pz | 6.48 | -70.42 | 70.42 | kN | 0.092 | ✓ |
| 45 | 2 | Nodal Support | 15 | CO1 | Pz | 8.75 | -70.42 | 70.42 | kN | 0.124 | ✓ |
| 46 | 2 | Nodal Support | 15 | LC2 | Pz | 31.09 | -70.42 | 70.42 | kN | 0.442 | ✓ |
| 47 | 2 | Nodal Support | 15 | CO2 | Pz | 55.40 | -70.42 | 70.42 | kN | 0.787 | ✓ |
| 48 | 2 | Nodal Support | 15 | LC3 | Pz | 34.31 | -70.42 | 70.42 | kN | 0.487 | ✓ |
| 49 | 2 | Nodal Support | 15 | CO3 | Pz | 81.12 | -70.42 | 70.42 | kN | 1.152 | ✗ |
| 50 | 2 | Nodal Support | 15 | LC4 | Pz | 2.12 | -70.42 | 70.42 | kN | 0.030 | ✓ |
| 51 | 2 | Nodal Support | 15 | CO4 | Pz | 82.71 | -70.42 | 70.42 | kN | 1.175 | ✗ |
| 52 | 2 | Nodal Support | 15 | CO5 | Pz | 56.99 | -70.42 | 70.42 | kN | 0.809 | ✓ |
| 53 | 2 | Nodal Support | 15 | CO6 | Pz | 60.21 | -70.42 | 70.42 | kN | 0.855 | ✓ |
| 54 | 2 | Nodal Support | 15 | CO7 | Pz | 63.39 | -70.42 | 70.42 | kN | 0.900 | ✓ |
| 55 | 2 | Nodal Support | 15 | CO8 | Pz | 11.93 | -70.42 | 70.42 | kN | 0.169 | ✓ |
| 56 | 2 | Nodal Support | 15 | CO9 | Pz | 92.86 | -70.42 | 70.42 | kN | 1.319 | ✗ |
| 57 | 2 | Nodal Support | 15 | CO10 | Pz | 96.04 | -70.42 | 70.42 | kN | 1.364 | ✗ |
| 58 | 2 | Nodal Support | 16 | LC1 | Pz | 0.25 | -70.42 | 70.42 | kN | 0.004 | ✓ |
| 59 | 2 | Nodal Support | 16 | CO1 | Pz | 0.34 | -70.42 | 70.42 | kN | 0.005 | ✓ |
| 60 | 2 | Nodal Support | 16 | LC2 | Pz | 0.95 | -70.42 | 70.42 | kN | 0.014 | ✓ |
| 61 | 2 | Nodal Support | 16 | CO2 | Pz | 1.77 | -70.42 | 70.42 | kN | 0.025 | ✓ |
| 62 | 2 | Nodal Support | 16 | LC3 | Pz | -5.72 | -70.42 | 70.42 | kN | 0.081 | ✓ |
| 63 | 2 | Nodal Support | 16 | CO3 | Pz | -2.52 | -70.42 | 70.42 | kN | 0.036 | ✓ |
| 64 | 2 | Nodal Support | 16 | LC4 | Pz | 1.17 | -70.42 | 70.42 | kN | 0.017 | ✓ |
| 65 | 2 | Nodal Support | 16 | CO4 | Pz | -1.65 | -70.42 | 70.42 | kN | 0.023 | ✓ |
| 66 | 2 | Nodal Support | 16 | CO5 | Pz | 2.64 | -70.42 | 70.42 | kN | 0.038 | ✓ |
| 67 | 2 | Nodal Support | 16 | CO6 | Pz | -8.24 | -70.42 | 70.42 | kN | 0.117 | ✓ |
| 68 | 2 | Nodal Support | 16 | CO7 | Pz | -6.49 | -70.42 | 70.42 | kN | 0.092 | ✓ |
| 69 | 2 | Nodal Support | 16 | CO8 | Pz | 2.09 | -70.42 | 70.42 | kN | 0.030 | ✓ |
| 70 | 2 | Nodal Support | 16 | CO9 | Pz | -7.24 | -70.42 | 70.42 | kN | 0.103 | ✓ |
| 71 | 2 | Nodal Support | 16 | CO10 | Pz | -5.49 | -70.42 | 70.42 | kN | 0.078 | ✓ |
| 72 | Custom material 3 \| \| | | | | | | | | | |
| 73 | 3 | Node | 6 | LC1 | uz | 5.02 | 0.00 | 73.00 | mm | 0.069 | ✓ |
| 74 | 3 | Node | 6 | CO1 | uz | 6.77 | 0.00 | 73.00 | mm | 0.093 | ✓ |
| 75 | 3 | Node | 6 | LC2 | uz | 26.41 | 0.00 | 73.00 | mm | 0.362 | ✓ |
| 76 | 3 | Node | 6 | CO2 | uz | 46.40 | 0.00 | 73.00 | mm | 0.636 | ✓ |
| 77 | 3 | Node | 6 | LC3 | uz | 15.57 | 0.00 | 73.00 | mm | 0.213 | ✓ |

**Figure 3.7** Module window 2.6 *Limit Checks by Node*

The last module window lists the check results for each node respectively support node. The corresponding limit values are described in Chapter 2.2.

# 4 Printout

## 4.1 Printout Report

Printout reports are created in the same way as in RFEM or RSTAB. First, a printout is generated for the data of the add-on module RF-/LIMITS to which you can add graphics and explanations. The selection in the printout report determines which data from the design module will be included in the final printout.

The printout report is described in the RFEM or RSTAB manual. Chapter 10.1.3.5 *Selecting Data of Add-on Modules* explains how to prepare input and output data of add-on modules for the printout.

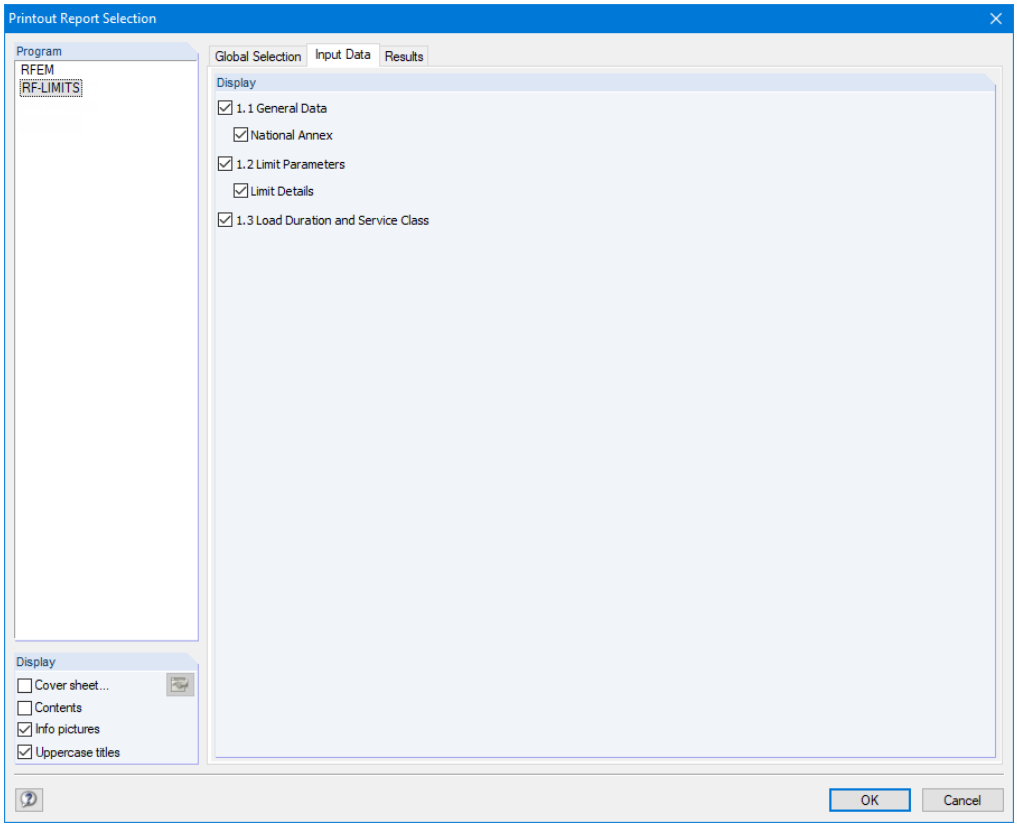



**Figure 4.1** Dialog box *Printout Report Selection* for RF-LIMITS

For large structural systems with many design cases, it is recommended to split the data into several printout reports, thus allowing for a clearly-arranged printout.

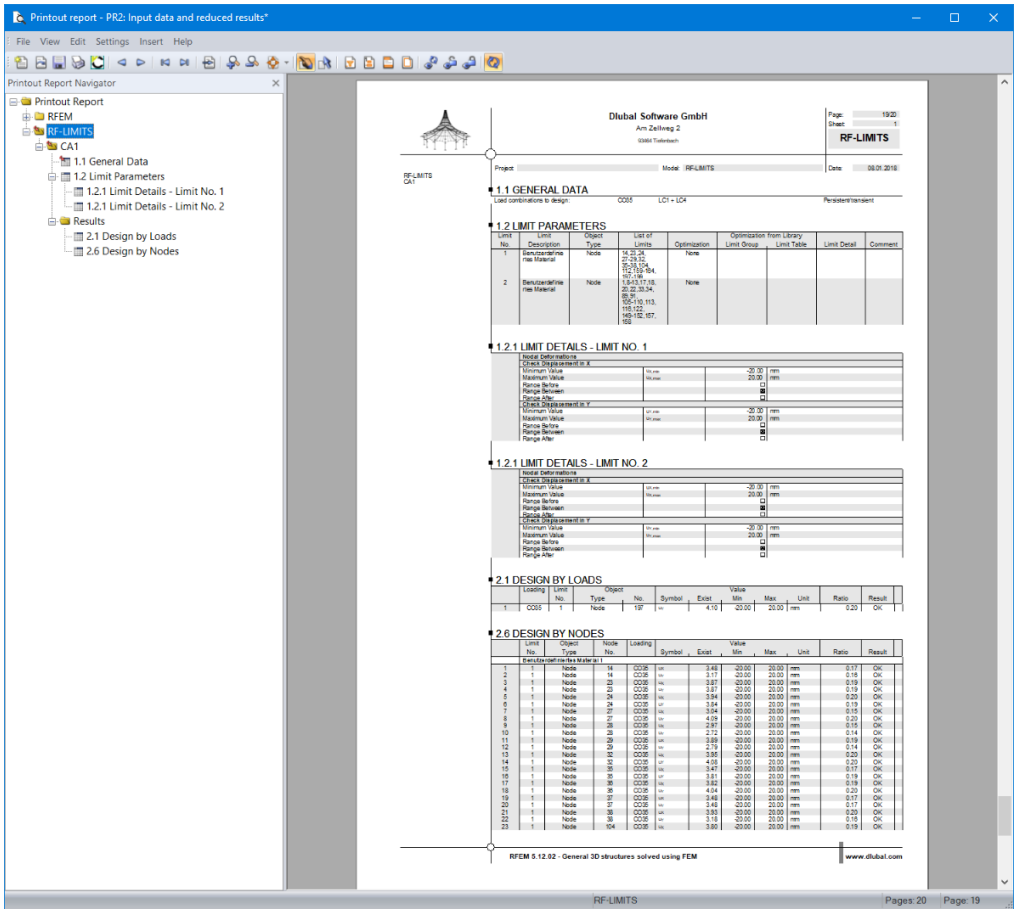



**Figure 4.2** Printout report with data from RF-LIMITS

# 5 Literature

[1] Eurocode 5: Design of timber structures - Part 1-1: General - Common rules and rules for buildings; EN 1995-1-1:2010-12

[2] DIN 1052:2008-12: Design of timber structures - General rules and rules for buildings. Beuth Verlag GmbH, Berlin, 2008.

[3] *Eurocode 3: Design of steel structures - Part 1-8: Design of joints*; EN 1993–1–8:2005 + AC:2009