Version
June 2021

Add-on Module

RF-/STEEL HK

Design of Steel Members according to the Code of Practice for the Structural Use of Steel

Program Description

© Dlubal Software GmbH 2021
Am Zellweg 2
93464 Tiefenbach
Germany

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

Contents

1 Introduction

1.1 Add-on Module RF-/STEEL HK

The Code of Practice for the Structural Use of Steel [1] describes the design, analysis and construction of steel structures relevant to Hong Kong. With the add-on modules RF-STEEL HK (for RFEM) and STEEL HK (for RSTAB), DLUBAL provides a powerful tool for designing steel members according to the regulations published by the Building Department of Hong Kong.

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

RF-/STEEL HK performs all typical ultimate limit state designs as well as stability and deflection analyses. In the ultimate limit state design, the add-on module considers the effect of various loadings. An essential part of the verification is the classification of the cross-sections to be designed into the classes 1 to 4. This way, the limitation of the capacity to withstand stresses as well as the rotational capacity due to local buckling of cross-section parts is checked. RF-/STEEL HK determines the c/t-ratios of the cross-section parts subjected to compression and carries out the classification automatically.

In the stability analyses, you can specify separately for each member or set of members whether flexural buckling in y- and/or z-direction is possible. You can also define additional lateral restraints in order to represent the model close to reality. Based on the boundary conditions, RF-/STEEL HK determines the slenderness ratios and elastic critical buckling loads. The elastic critical moment for flexural-torsional buckling required for the lateral-torsional buckling analysis is determined automatically. Optionally, the program takes into account the load application point of transverse loads, which has a decisive effect on the torsional resistance.

The serviceability limit state represents an important design for structures with slender cross-sections. Load cases, load combinations and result combinations can be assigned to different design situations. The limit deflections are preset by the Code, but can be adjusted, if necessary. In addition, it is possible to specify reference lengths and precambers that are considered accordingly in the deflection design.

The add-on module provides an automatic cross-section optimization with the possibility to export modified cross-sections to RFEM or RSTAB. Separate design cases allow for the flexibility to analyze individual structural components within complex structures.

Like other add-on modules, RF-/STEEL HK is completely integrated in RFEM and RSTAB. Thus, the design-relevant input data is preset when you start the add-on module. After the design, you can use the graphical user interface of the main program to evaluate the results. As they are also included in the global printout report, the entire verification can be presented in a consistent and appealing form.

We wish you ease and success with RF-/STEEL HK.

Your DLUBAL 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-/STEEL HK add-on module.

The descriptions in this manual follow the sequence and structure of the module's input and result windows. In the text, the described **buttons** are given in square brackets, for example, [View Mode]. At the same time, they are pictured on the left. Expressions appearing in dialog boxes, windows, and menus are set in *italics* to clarify the explanation.

At the end of the manual, you find the index. However, if you don't find what you are looking for, you can go to the Knowledge Base to find related articles about the steel add-on modules or consult the FAQs.

1.3 Opening RF-/STEEL HK Add-on Module

RFEM and RSTAB provide the following options to open the RF-/STEEL HK add-on module.

Menu

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

Add-on Modules → **Design - Steel** → **RF-/STEEL HK**.

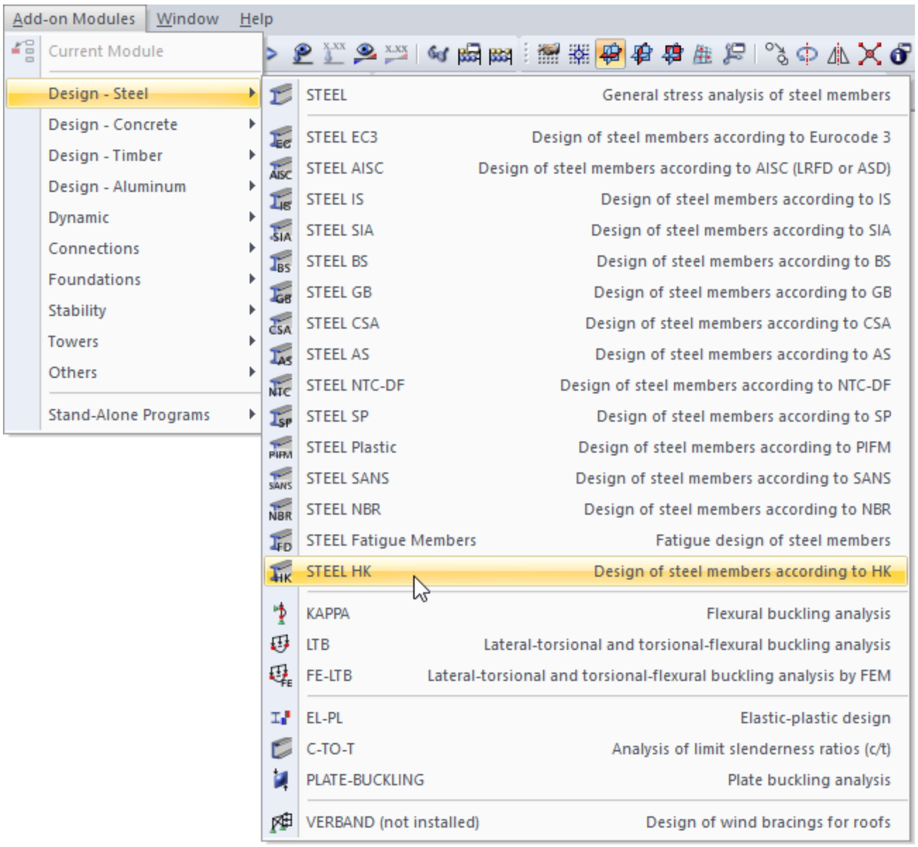


Figure 1.1: Menu *Add-on Modules* → *Design - Steel* → *STEEL HK*

Navigator

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

Add-on Modules → **RF-/STEEL HK**.

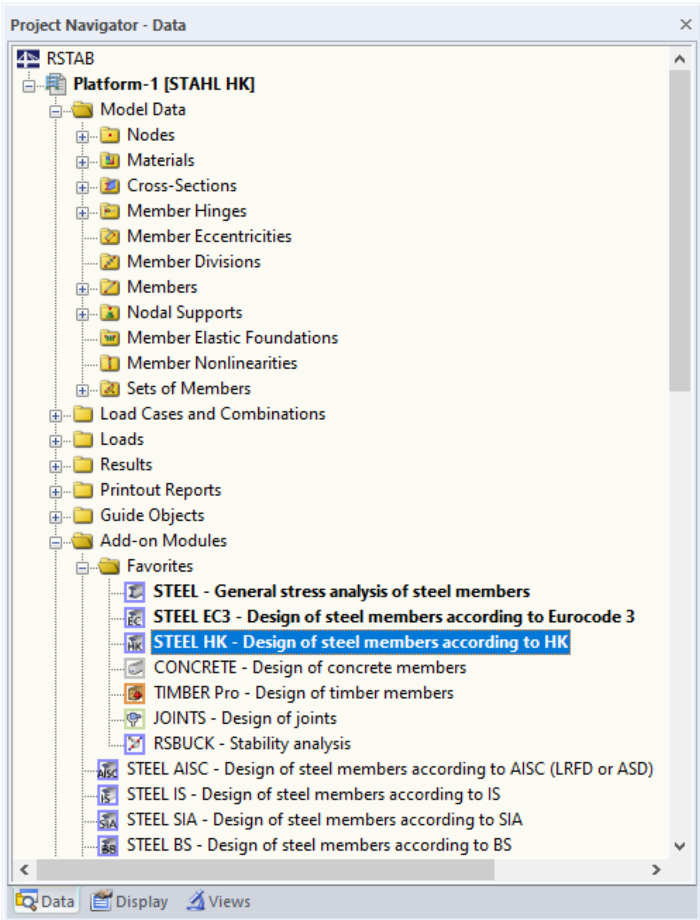
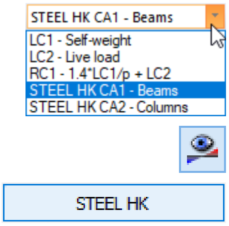


Figure 1.2: Data navigator: *Add-on Modules* → *RF-STEEL HK*

Panel



If any results from RF-/STEEL HK are already available in the model, you can open the design module in the panel:

Set the relevant design case in the load case list of the menu bar. Click the [Show Results] button to display the design criterion graphically on the members.

When the results display is activated, the panel appears showing the [RF-/STEEL HK] button which you can use to open the add-on module.

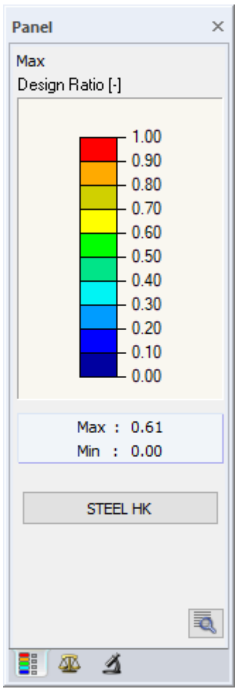


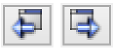
Figure 1.3: Panel with [STEEL HK] button

2 Input Data

When you start the add-on module, a new window appears. 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 (see Chapter 7.1, page 53).

The design-relevant data must be defined in several input windows. The following parameters are imported automatically when you open RF-/STEEL HK for the first time:

- Members and sets of members
- Load cases, load combinations, and result combinations
- Materials
- Cross-sections
- Buckling lengths
- Internal forces (in background, if calculated)



To select a 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] window.

To save the entered data, click [OK]. Thus, you exit RF-/STEEL HK and return to the main program. Click [Cancel] to exit the add-on module without saving the new data.

2.1 General Data

In Window *1.1 General Data*, you select the members, sets of members, and actions that you want to design. The three tabs manage the load cases, load combinations, and result combinations for the ultimate limit state, the serviceability limit state, and the fire protection design.

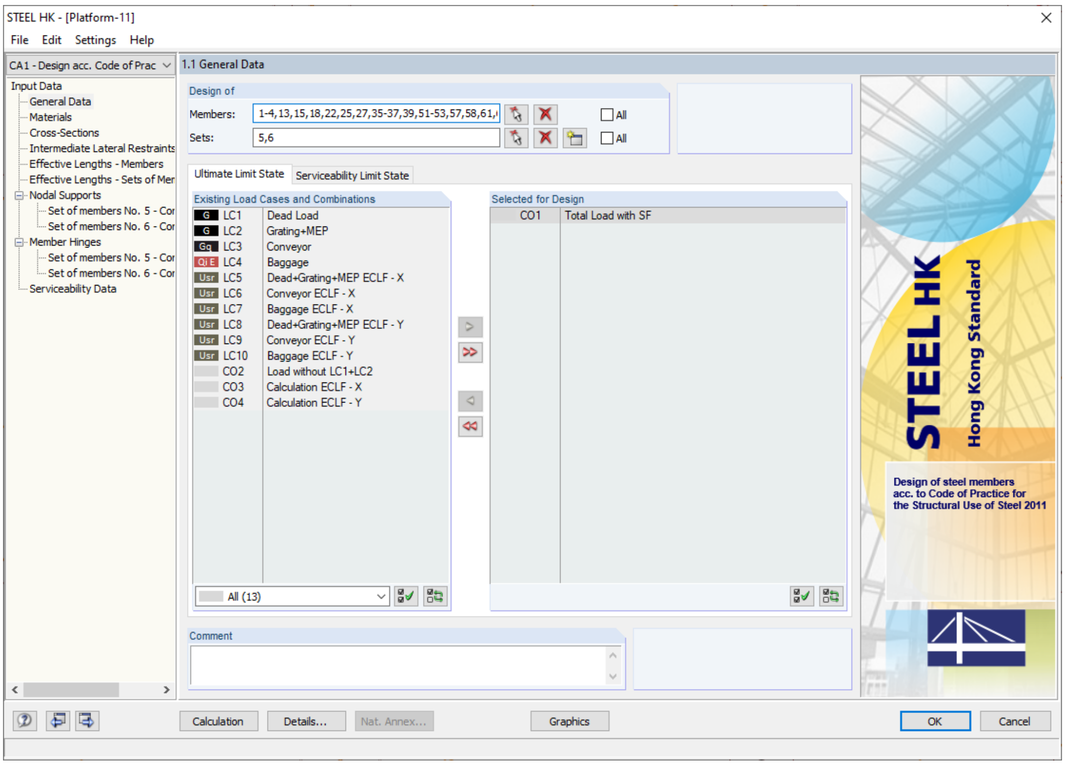


Figure 2.1: Window *1.1 General Data*

Design of

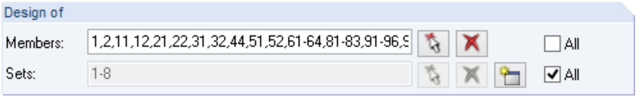


Figure 2.2: Design of members and sets of members

You can design *Members* as well as *Sets* of members. If you want to design only selected objects, clear the *All* check box: Then, you can access the text boxes to enter the numbers of the relevant members or sets of members. Use the [Delete] button to clear the list of preset numbers. Use the [Select] button to define objects graphically in the RFEM or RSTAB work window.

When you design a set of members, the program determines the extreme values of the designs of all members contained in this set of members and takes into account the boundary conditions due to connected members for stability analyses. The results are shown in the results windows *2.3 Design by Set of Members*, *3.2 Governing Internal Forces by Set of Members*, and *4.2 Parts List by Set of Members*.

To define a new set of members, click the [New] button. The dialog box known from RFEM or RSTAB appears where you can enter the parameters for the set of members.

Comment

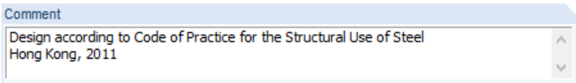


Figure 2.3: User-defined comment

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

2.1.1 Ultimate Limit State

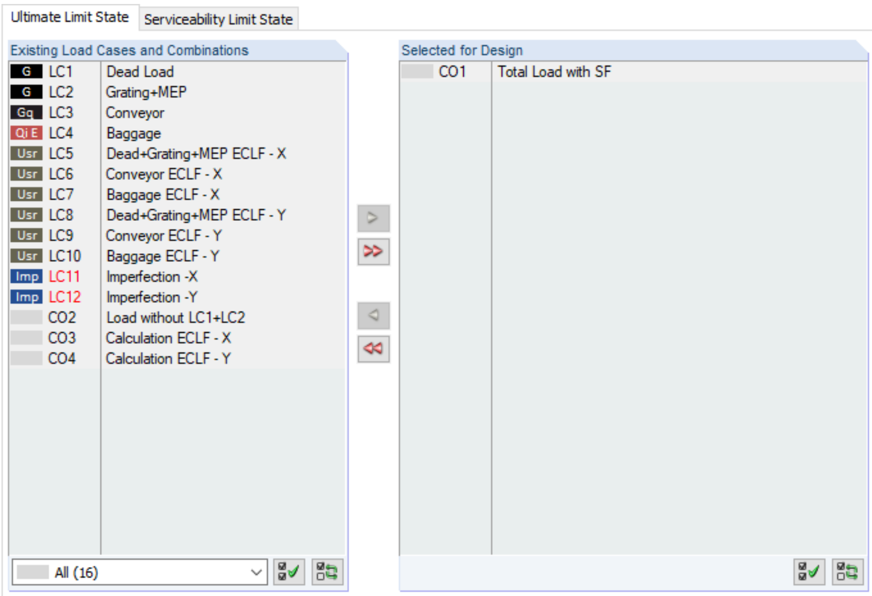


Figure 2.4: Window *1.1 General Data*, tab *Ultimate Limit State*

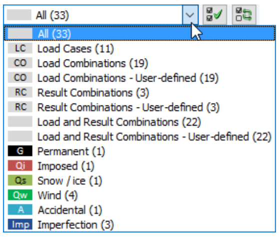
Existing Load Cases and Combinations

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 such as LC11 or LK12 in Figure 2.4, 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 *Ultimate Limit State* tab

Selected for Design

The column on the right lists the load cases as well as 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 [<<].

Result combination

Designing an enveloping max/min result combination (RC) is faster than designing all contained load cases and load combinations, but the analysis of a result combination also has disadvantages: First, the influence of the contained actions is difficult to discern. Second, for the determination of the elastic critical moment M_{cr} for lateral-torsional buckling, the envelope of the moment distributions is analyzed, from which the most unfavorable distribution (max or min) is taken. This distribution, however, only rarely reflects the moment distribution that is available in the individual load combinations. Thus, for an RC design, more unfavorable values for M_{cr} are expected, leading to higher ratios.

Result combinations should be selected for design only in case of dynamic combinations. For "usual" combinations, it is recommended to use load combinations because here the actual moment distributions are applied for the determination of M_{cr}.

Details...

In the *General* tab of the *Details* dialog box, you can define how result combinations of the 'OR' type are handled in the design (see Chapter 3.1.4, page 30).

2.1.2 Serviceability Limit State

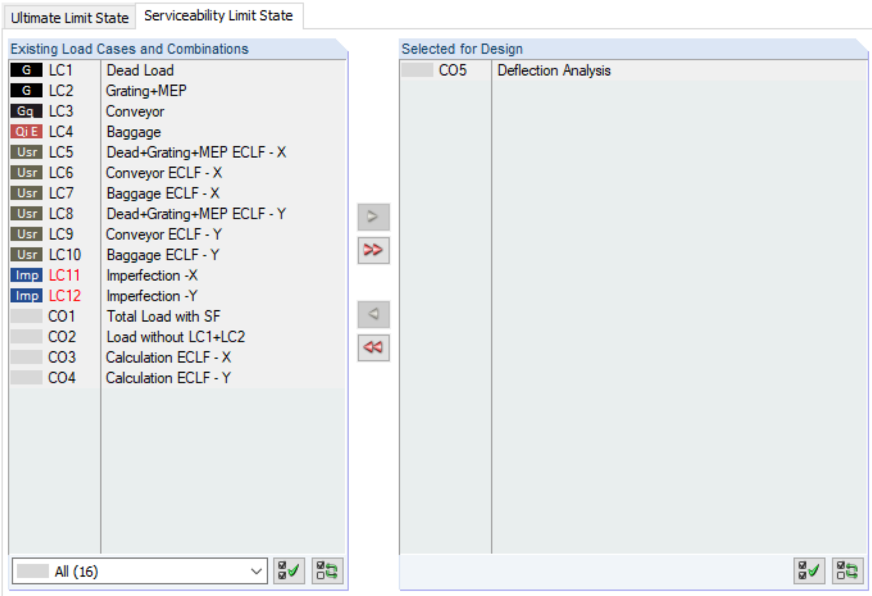


Figure 2.5: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

Nat. Annex...

You can add or remove load cases as well as load and result combinations as described in Chapter 2.1.1.

The limit values of the deformations are defined in the Code of Practice [1] Table 5.1. They can be adjusted for the design situations in the *Serviceability* tab of the *Details* dialog box (see Figure 3.3, page 29) that you open with the [Details] button.

In Window *1.9 Serviceability Data*, the reference lengths applying to the deformation analysis are managed (see Chapter 2.9, page 26).

2.2 Materials

This module window consists of two parts. The upper part lists all materials created in RFEM or RSTAB. The *Material Properties* section shows the properties of the current material, that is, the table row currently selected in the upper section.

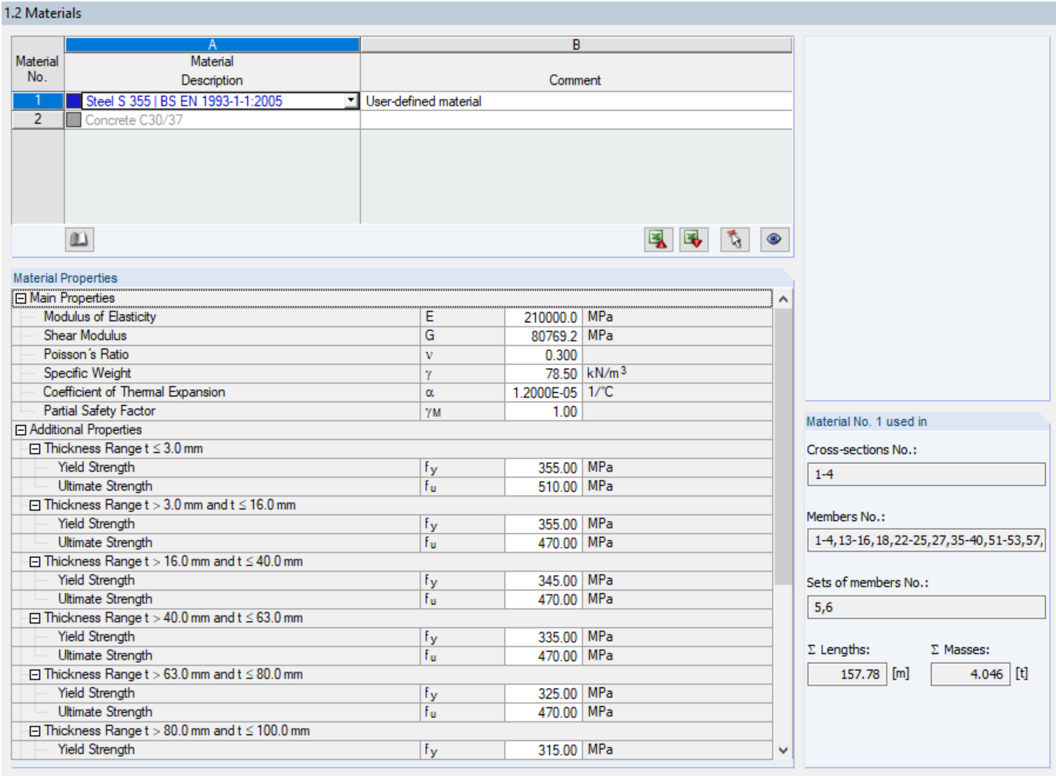


Figure 2.6: Window *1.2 Materials*

Materials that won't be used in the design are grayed out. Materials that are not allowed are highlighted in red. Modified materials are displayed in blue.

Chapter 4.3 of the RFEM manual, or Chapter 4.2 of the RSTAB manual, describes the material properties that are used for the determination of the internal forces (*Main Properties*). The properties of the materials that are required for the design are also stored in the global material library. These values are preset (*Additional Properties*).

To adjust the units and decimal places of the properties and stresses, select on the module menu **Settings** → **Units and Decimal Places** (see Chapter 7.3, page 57).

Material Description

The materials defined in RFEM or RSTAB are preset but you can modify them anytime: Click the material in column A to activate the box. Then, click the button, or press the function key [F7] to open the material list.

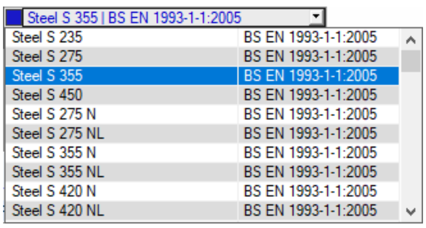


Figure 2.7: List of materials

According to the design concept of the Code [1], only materials of the *Steel* category can be selected.

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

If you change the material description manually and the new entry is already listed in the material library, RF-/STEEL HK will import the material properties as well.

The material properties are generally not editable in the RF-/STEEL HK add-on module.

Material Library

Many materials are stored in the database. To open the material library, click on the module menu

Edit → Material Library

or use the button shown on the left.

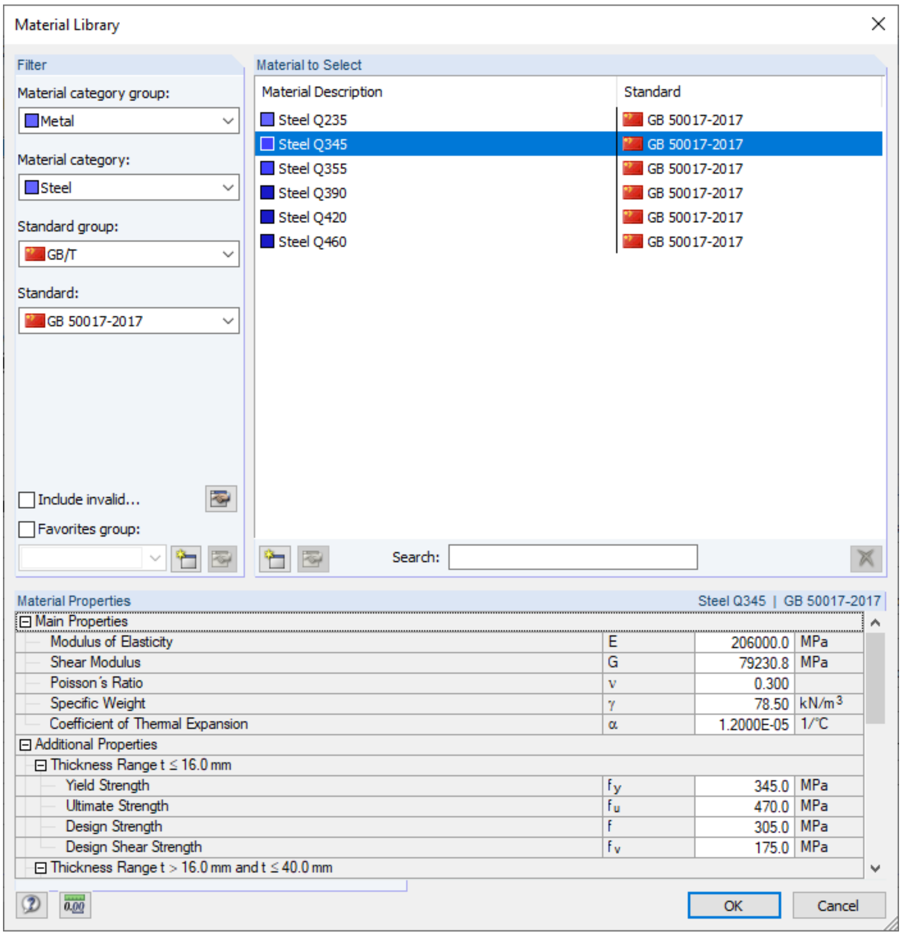


Figure 2.8: Dialog box *Material Library*

The *Steel* material category is preset in the *Filter* section. You can select the desired material grade from the *Material to Select* list; then you can check the properties in the dialog section below.

OK

Click [OK] or use [↵] to transfer the selected material to Window 1.2 of RF-/STEEL HK.

Chapter 4.3 of the RFEM manual, or Chapter 4.2 of the RSTAB manual, describes how to filter, add, or reorganize materials.

In the library, you can also select materials of the categories *Cast Iron* and *Stainless Steel*. However, please check whether these materials are allowed by the design concept of the Code [1].

2.3 Cross-Sections

This window lists the cross-sections used for the design. In addition, you can specify optimization parameters.

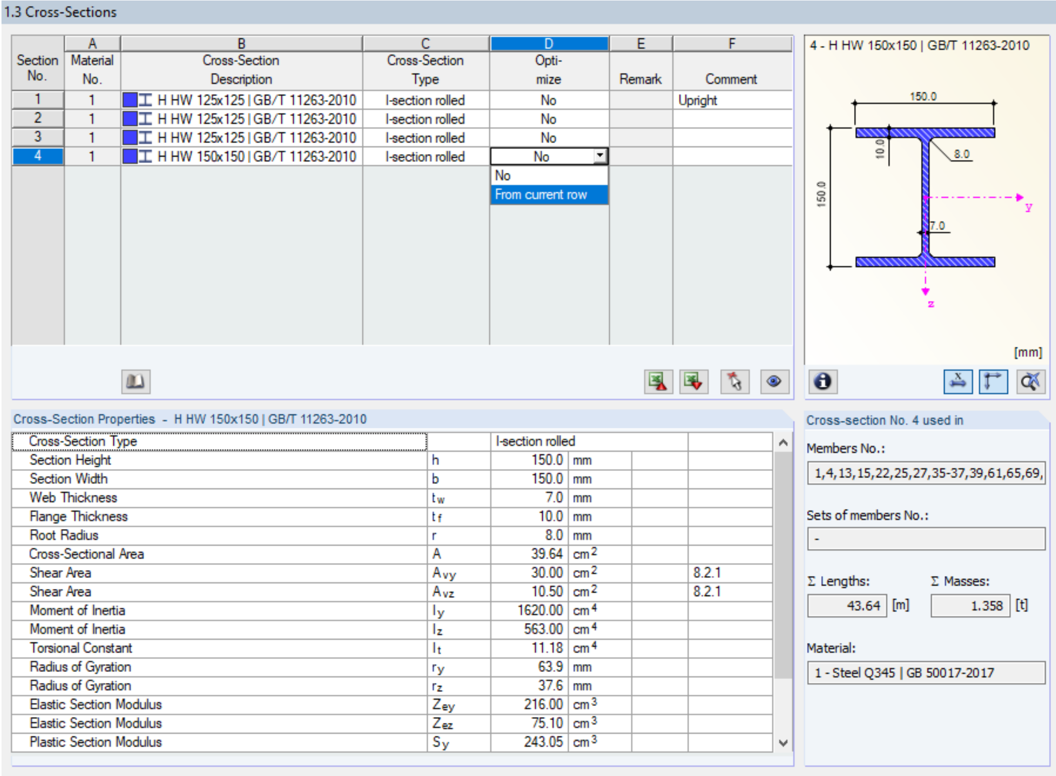


Figure 2.9: Window *1.3 Cross-Sections*

Cross-Section Description

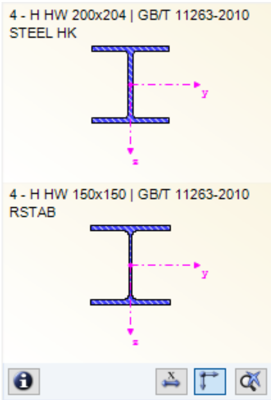
The cross-sections defined in RFEM or RSTAB are preset together with the assigned material numbers.

To modify a cross-section, click the entry in column B. Thus, you set the cell active. Then, open the cross-section table of the current input field by clicking the [Cross-Section Library] button or the [...] button at the end of the box. You can also use the function key [F7] (see Figure 2.10).

In this dialog box, you can choose a different cross-section or even a different cross-section table. If you want to select a completely different cross-section category, click the [Back to Cross-Section Library] button. Then, the general cross-section library opens.

Chapter 4.13 of the RFEM manual, or Chapter 4.3 of the RSTAB manual, describes how to select cross-sections from the library.

You can also enter a new cross-section description directly into the input field in column B. If the entry is already listed in the database, RF-/STEEL HK will import the cross-section properties. A modified cross-section is highlighted in blue.



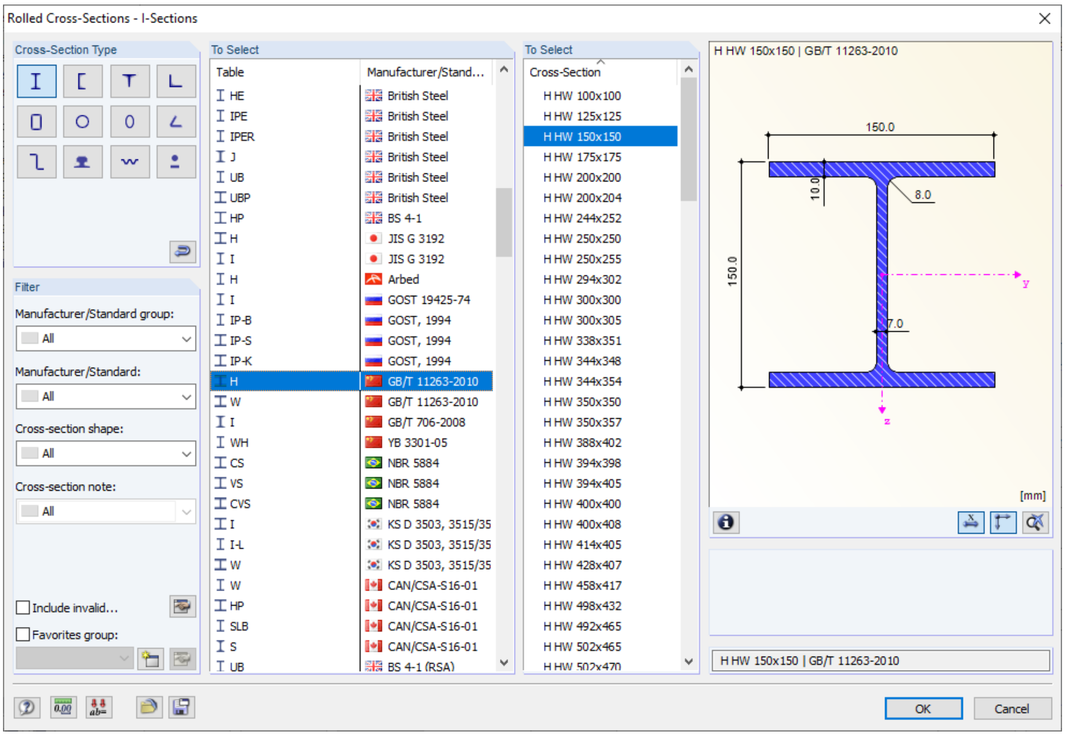


Figure 2.10: Rolled I-section types of cross-section library

If cross-sections set in RF-/STEEL HK are different from the ones used in RFEM or RSTAB, both cross-sections are displayed in the graphic to the right. The designs will be performed with the internal forces from RFEM or RSTAB for the cross-section selected in RF-/STEEL HK.

Cross-Section Type

This column shows the cross-section type that is used for the classification. The cross-sections referred to in [1], Section 7 must be designed plastically or elastically depending on their class. Non-typical cross-sections are classified as *General* and can only be designed elastically, which means Class 3 or 4.

Max. Design Ratio

This column is displayed only after the calculation. It is intended to be a decision support for the optimization: Looking at the design ratios and colored relation scales, you can clearly see which cross-sections are hardly utilized and thus oversized, or extremely stressed and thus undersized.

Optimize

Each cross-section of the library can pass through an optimization process: For the internal forces from RFEM or RSTAB, the program searches the cross-section that comes as close as possible to a user-defined maximum ratio that can be defined in the *General* tab of the *Details* dialog box (see Figure 3.4, page 30).

To optimize a cross section, open the drop-down list in column E or F, and select the relevant entry: *From current row* or, if available, *From favorites* 'Description'. Recommendations for optimizing cross-sections can be found in Chapter 7.2 on page 55.

Remark

This column shows remarks in the form of footnotes. They are explained below the cross-section list.

If the warning *Incorrect type of cross-section!* appears before calculating, a cross-section is set which is not listed in the database. This may be a user-defined cross-section or a SHAPE-THIN cross-section that has not yet been calculated. To select an appropriate cross-section for the design, click the [Library] button (see description below Figure 2.9).

Member with tapered cross-section

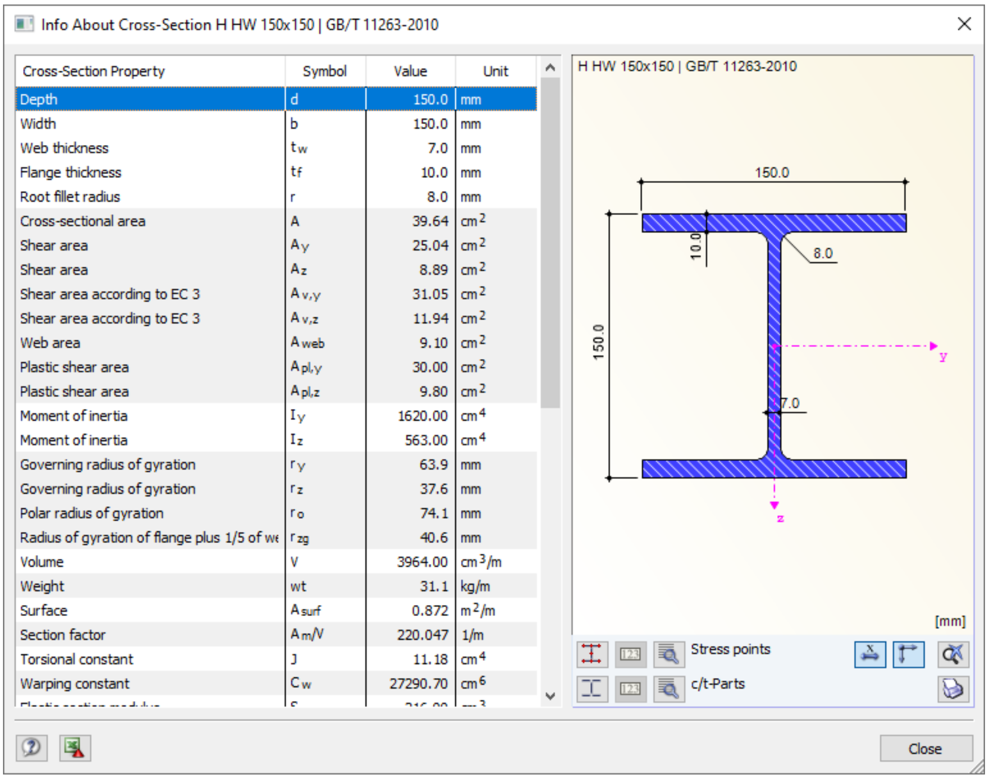
For tapered members with different cross-sections at the member start and end, both cross-section numbers are shown in two rows, in accordance with the definition in RFEM or RSTAB.

RF-/STEEL HK also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the end. Normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the cross-sections at the start and end of a tapered member have different numbers of stress points, the intermediate values cannot be interpolated. The calculation is neither possible in RFEM or RSTAB nor in RF-/STEEL HK.

The cross-section's stress points including numbering can be checked graphically: Select the cross-section in Window 1.3, and then click the button. The dialog box shown in Figure 2.11 appears.

Info About Cross-Section

Below the cross-section graphic, you find the [Info] button. Click it to open the *Info About Cross-Section* dialog box where you can see the cross-section properties, stress points and c/t-parts.



Cross-Section Property	Symbol	Value	Unit
Depth	d	150.0	mm
Width	b	150.0	mm
Web thickness	t_w	7.0	mm
Flange thickness	t_f	10.0	mm
Root fillet radius	r	8.0	mm
Cross-sectional area	A	39.64	cm^2
Shear area	A_y	25.04	cm^2
Shear area	A_z	8.89	cm^2
Shear area according to EC 3	$A_{v,y}$	31.05	cm^2
Shear area according to EC 3	$A_{v,z}$	11.94	cm^2
Web area	A_{web}	9.10	cm^2
Plastic shear area	$A_{pl,y}$	30.00	cm^2
Plastic shear area	$A_{pl,z}$	9.80	cm^2
Moment of inertia	I_y	1620.00	cm^4
Moment of inertia	I_z	563.00	cm^4
Governing radius of gyration	r_y	63.9	mm
Governing radius of gyration	r_z	37.6	mm
Polar radius of gyration	r_o	74.1	mm
Radius of gyration of flange plus 1/5 of we	r_{zg}	40.6	mm
Volume	V	3964.00	cm^3/m
Weight	wt	31.1	kg/m
Surface	A_{surf}	0.872	m^2/m
Section factor	A_m/V	220.047	1/m
Torsional constant	J	11.18	cm^4
Warping constant	C_w	27290.70	cm^6

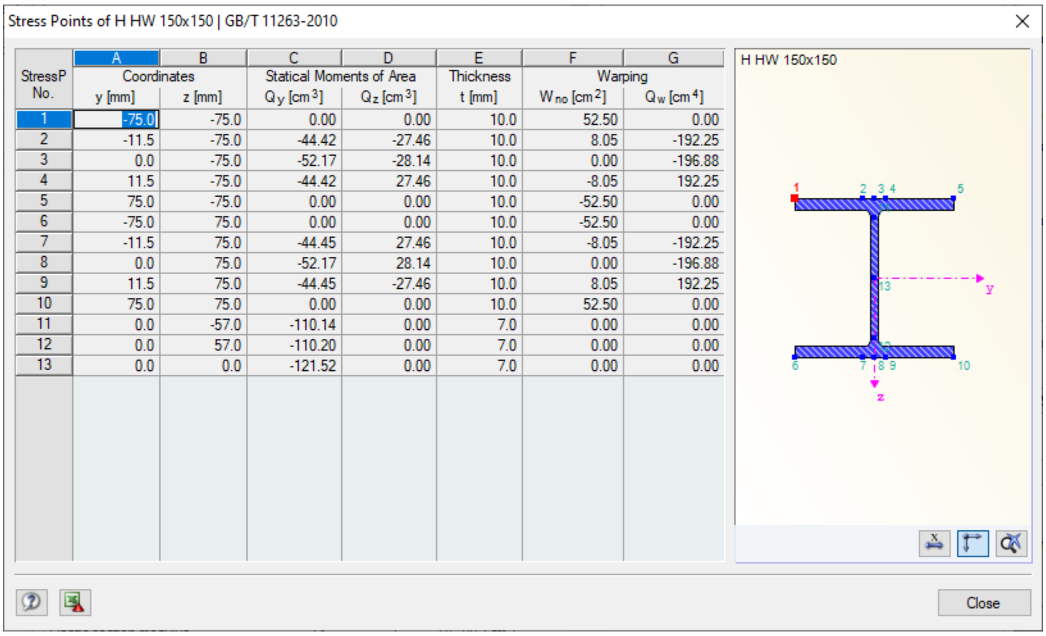
Figure 2.11: Dialog box *Info About Cross-Section*

The buttons below the cross-section graphic have the following functions:

Button	Function
	Displays or hides stress points
	Displays or hides c/t-parts
	Displays or hides numbers of stress points or c/t-parts
	Shows details of stress points or c/t-parts (see Figure 2.12)
	Displays or hides dimensions of cross-section
	Displays or hides principal axes of cross-section
	Resets full view of cross-section

Table 2.2: Buttons of cross-section graphic

Use the [Details] buttons to call up specific information about stress points (centroid distances, statical moments of area, warping ordinates, etc.) and c/t-parts.


Stress Points of H HW 150x150 | GB/T 11263-2010

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_w [cm⁴]
1	-75.0	-75.0	0.00	0.00	10.0	52.50	0.00
2	-11.5	-75.0	-44.42	-27.46	10.0	8.05	-192.25
3	0.0	-75.0	-52.17	-28.14	10.0	0.00	-196.88
4	11.5	-75.0	-44.42	27.46	10.0	-8.05	192.25
5	75.0	-75.0	0.00	0.00	10.0	-52.50	0.00
6	-75.0	75.0	0.00	0.00	10.0	-52.50	0.00
7	-11.5	75.0	-44.45	27.46	10.0	-8.05	-192.25
8	0.0	75.0	-52.17	28.14	10.0	0.00	-196.88
9	11.5	75.0	-44.45	-27.46	10.0	8.05	192.25
10	75.0	75.0	0.00	0.00	10.0	52.50	0.00
11	0.0	-57.0	-110.14	0.00	7.0	0.00	0.00
12	0.0	57.0	-110.20	0.00	7.0	0.00	0.00
13	0.0	0.0	-121.52	0.00	7.0	0.00	0.00

Figure 2.12: Dialog box *Stress Points of H HW 150x150*

2.4 Intermediate Lateral Restraints

In Window 1.4, you can define lateral intermediate restraints for members. RF-/STEEL HK always assumes this kind of support to be perpendicular to the cross-section's minor axis z (see Figure 2.11). Thus, it is possible to influence the members' effective lengths (only for *Lateral and torsional* restraint type) which are important for the stability analyses concerning flexural buckling and lateral-torsional buckling.

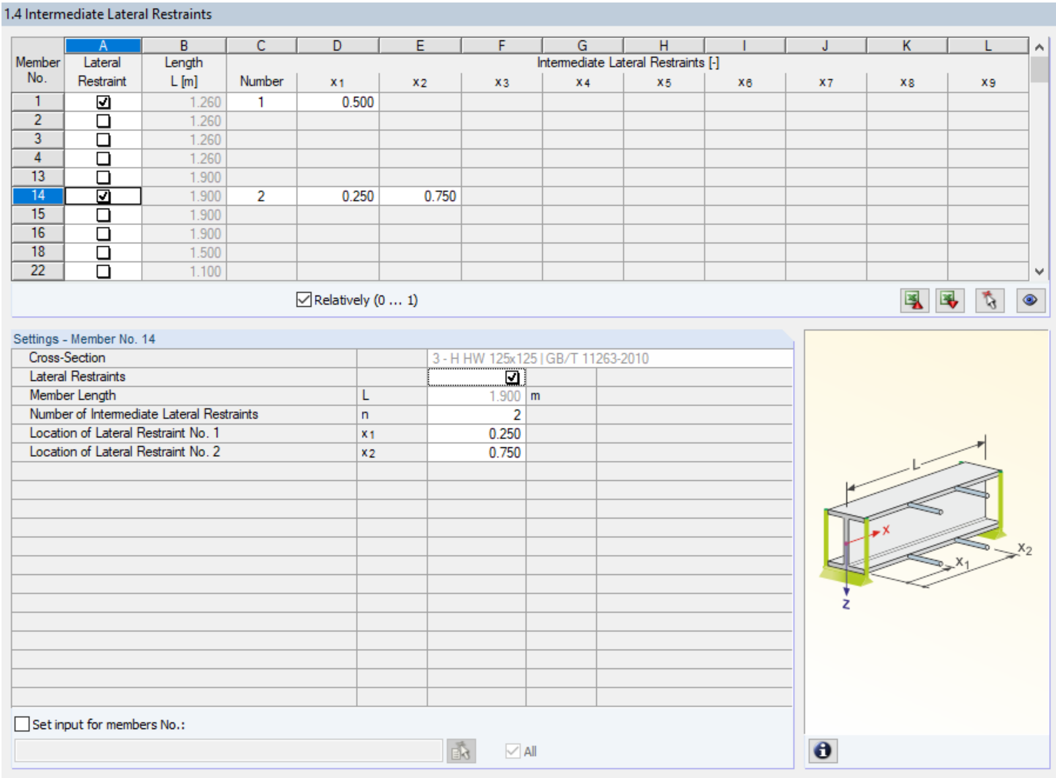


Figure 2.13: Window *1.4 Intermediate Lateral Restraints*

In the upper part of the window, you can assign up to nine lateral supports to each member. The *Settings* section shows the entry displayed in a column view for the member selected above.

To define the intermediate restraints of a member, select the check box for *Lateral Restraints* in column A. With the button you can select the member graphically to activate its row in the table. When the check box is selected, the other columns become available, and you can enter the parameters.

In column C, you can define the *Number* of intermediate restraints. Depending on the setting, you can access one or more of the following *Intermediate Lateral Restraints* columns for defining the x-locations.

☑ Relatively (0 ... 1)

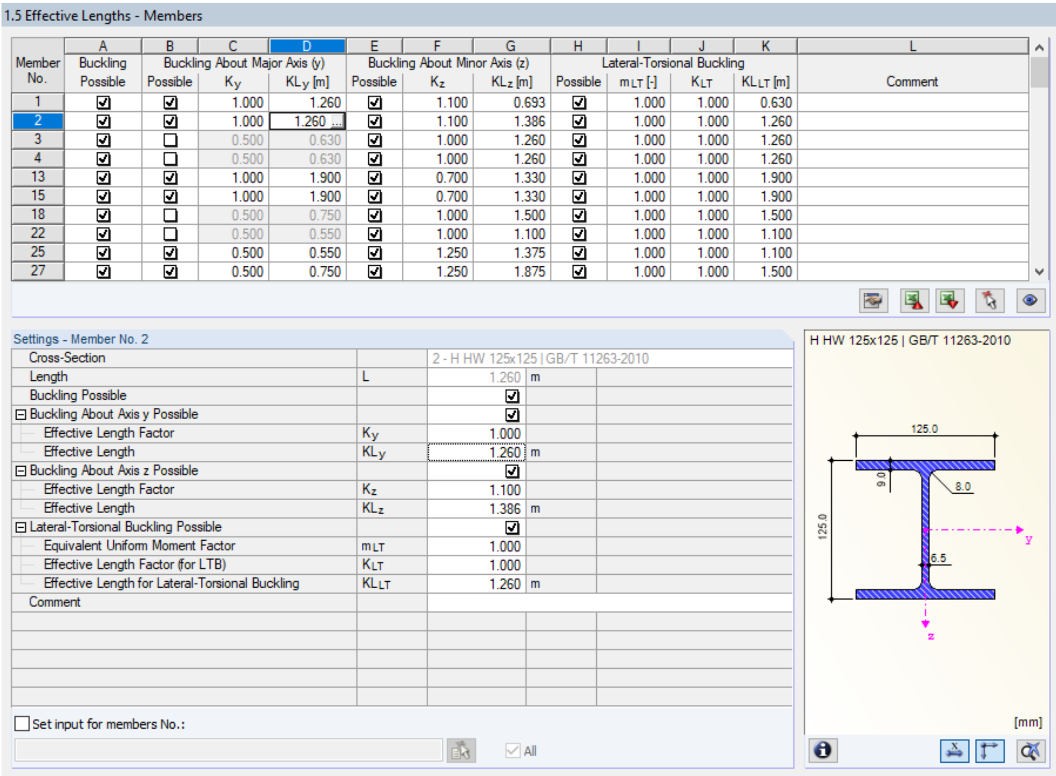
When the check box for *Relatively (0 ... 1)* is selected, you can define the support points by relative input: The locations of the intermediate supports result from the member length and the relative distances to the member start. It is also possible to define the distances manually in the table, if the *Relatively (0 ... 1)* check box is cleared.

2.5 Effective Lengths - Members

This module window is subdivided into two parts. The table in the upper part shows summary information about buckling length factors and equivalent member lengths for buckling and lateral-torsional buckling of all members to be designed. The effective lengths defined in RFEM or RSTAB are preset. In the *Settings* section, you can see additional information about the member whose table row is selected in the upper part.

With the button you can select a member graphically to activate its row in the table.

Changing entries is possible in the table as well as the *Settings* tree.



Member No.	A Buckling Possible	B Buckling About Major Axis (y) Possible	C K_y	D KL_y [m]	E Buckling About Minor Axis (z) Possible	F K_z	G KL_z [m]	H Lateral-Torsional Buckling Possible	I m_{LT} [-]	J K_{LT}	K KL_{LT} [m]	L Comment
1	☑	☑	1.000	1.260	☑	1.100	0.693	☑	1.000	1.000	0.630	
2	☑	☑	1.000	1.260	☑	1.100	1.386	☑	1.000	1.000	1.260	
3	☑	☐	0.500	0.630	☑	1.000	1.260	☑	1.000	1.000	1.260	
4	☑	☐	0.500	0.630	☑	1.000	1.260	☑	1.000	1.000	1.260	
13	☑	☑	1.000	1.900	☑	0.700	1.330	☑	1.000	1.000	1.900	
15	☑	☑	1.000	1.900	☑	0.700	1.330	☑	1.000	1.000	1.900	
18	☑	☐	0.500	0.750	☑	1.000	1.500	☑	1.000	1.000	1.500	
22	☑	☐	0.500	0.550	☑	1.000	1.100	☑	1.000	1.000	1.100	
25	☑	☑	0.500	0.550	☑	1.250	1.375	☑	1.000	1.000	1.100	
27	☑	☑	0.500	0.750	☑	1.250	1.875	☑	1.000	1.000	1.500	

Figure 2.14: Window *1.5 Effective Lengths - Members*

The effective lengths for buckling about the minor axis z are aligned automatically with Window *1.4 Intermediate Lateral Restraints*. If the intermediate supports divide the member into segments of different lengths, no values are displayed in columns G, K, and L of Window 1.5.

You can enter the effective lengths manually in the table and the *Settings* tree. You can also define them graphically in the work window by using the [...] button that becomes active when the cursor is placed in the text box (see Figure 2.14).

The effective lengths of each member have to be defined manually on the basis of its boundary conditions.

The *Settings* tree includes the following parameters:

- *Cross-Section*
- *Length* of member
- *Buckling Possible* for member (corresponds to columns B, E, and H)
- *Buckling About Axis y* (corresponds to columns C and D)
- *Buckling About Axis z* z (corresponds to columns F and G)
- *Lateral-Torsional Buckling* (corresponds to columns I to K)

For the selected member, you can define whether a buckling or a lateral-torsional buckling analysis is generally to be carried out. In addition, you can adjust the *Effective Length Factor* for the respective directions. When changing a factor, the equivalent member length will be adjusted automatically, and vice versa.

It is also possible to define the effective length of a member in a dialog box that you open with the button [Select effective length factor]. You can find the button below the table.

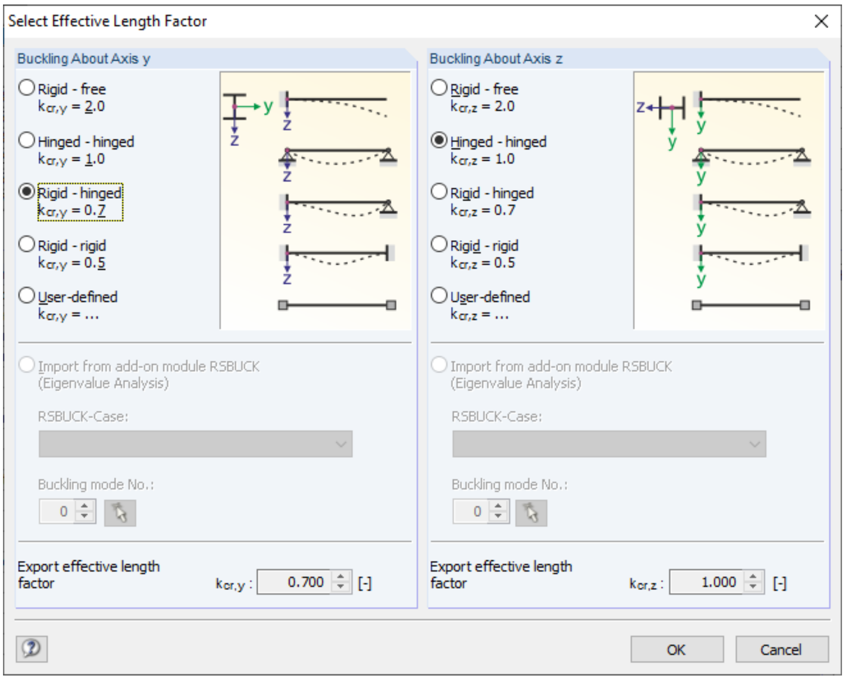


Figure 2.15: Dialog box *Select Effective Length Factor*

For each direction, you can select one of the four Euler buckling modes. You can also set a *User-defined* effective length factor. If an eigenvalue analysis has been carried out by the RF-STABILITY or RSBUCK add-on module, it is also possible to define a *Buckling mode* for the determination of the factor.

Buckling Possible

The stability analyses for flexural and lateral-torsional buckling require the ability to absorb compressive forces. Therefore, members for which such an absorption is not possible due to the member type (for example, tension members, elastic foundations, rigid connections) are excluded from the outset. The rows are grayed out in the table, and a corresponding note is shown in the *Comment* column.

The *Buckling Possible* check boxes in table row A and in the *Settings* tree offer a control option for the stability analyses: They determine if these analyses are performed or omitted for the member.

Buckling About Axis y or Axis z

With the check box in the *Possible* column, you decide if a member has the risk of buckling about the axis y and/or z. These axes represent the local member axes, with axis y being the "major" and axis z the "minor" member axis. The effective length factors K_y and K_z for buckling about the major or minor axis can be selected freely.

In Window *1.3 Cross-Sections*, you can check the position of the member axes in the cross-section graphic (see Figure 2.9, page 13). With the [Jump to graphic] button you can also access the RFEM or RSTAB work window. There, you can display the local member axes by using the member's shortcut menu or the *Display* navigator.

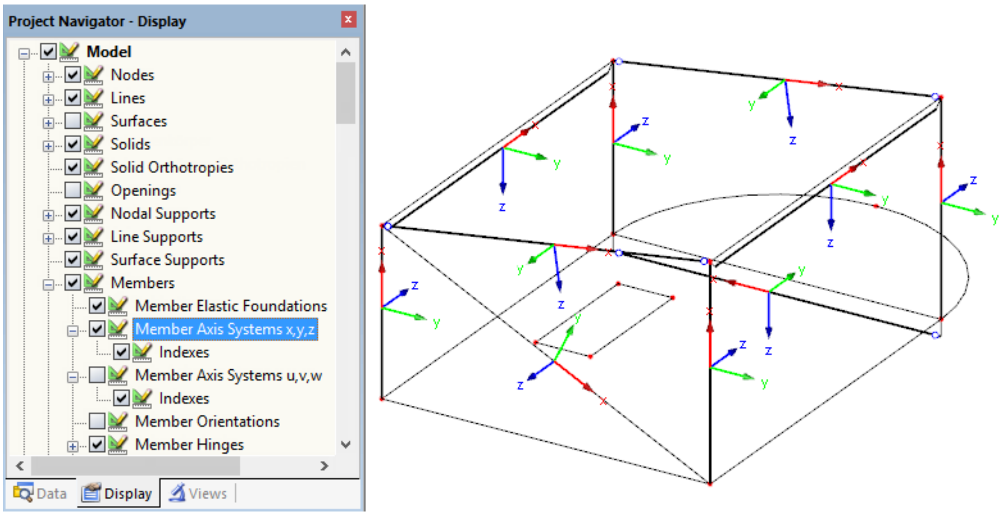


Figure 2.16: Activating the member axis systems in *Display* navigator of RFEM

If buckling is possible about one or both member axes, you can enter the effective length factors in columns C and F, and the effective lengths in columns D and G. The same is possible in the *Settings* tree.

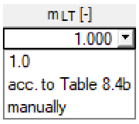
To define the effective lengths graphically in the work window, use the [...] button that becomes available when the cursor is placed in a KL text box (see Figure 2.14).

When you specify the effective length factor K, the program determines the effective length KL by multiplying the member length *L* by this factor. The K and KL text boxes are interactive.

Lateral-Torsional Buckling Possible

Column H shows which members are included in the analysis of lateral-torsional buckling.

Equivalent Uniform Moment Factor m_{LT}



The factor m_{LT} represents the equivalent uniform moment factor for lateral-torsional buckling. For cantilevers, m_{LT} is equal to 1. The equivalent uniform moment factors for typical moment distributions are specified in [1] Table 8.4a and Table 8.4b. The list of this column provides an option for the program to apply this factor automatically *according to Table 8.4b*. Alternatively, you can define this factor *manually*.

Effective Length Factor for LTB K_{LT}

The effective length factor K_{LT} controls the effective length for lateral-torsional buckling according to [1] 8.3.4. For a beam with its compression flange restrained against lateral movement at the end supports, but free to rotate on plan and with ends under nominal torsional restraint about the longitudinal axis of the beam at the end supports, this factor is 1.0. By increasing or reducing this factor, you can adjust the factor to the boundary conditions of each member.

If the effective length for lateral-torsional buckling KL_{LT} differs from the member length, you can also define it manually in column K or graphically with the [...] button that appears when you select a cell. Columns J and K are interactive, i.e. the effective length factor is updated automatically.

Comment

In the final column, you can enter user-defined notes to describe, for example, the equivalent member lengths.

Set input for members No.

Below the *Settings* table, you find the check box *Set input for members No.* If you select it, the subsequent settings will apply to *All* members or to selected members (enter the member numbers manually or select them graphically with). This option is useful if you want to assign the same boundary conditions to several members (see also DLUBAL article https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000726).

Settings which have already been defined cannot be changed subsequently with this function.

2.6 Effective Lengths - Sets of Members

Details...

This window appears when at least one set of members has been set for design in Window *1.1 General Data*.

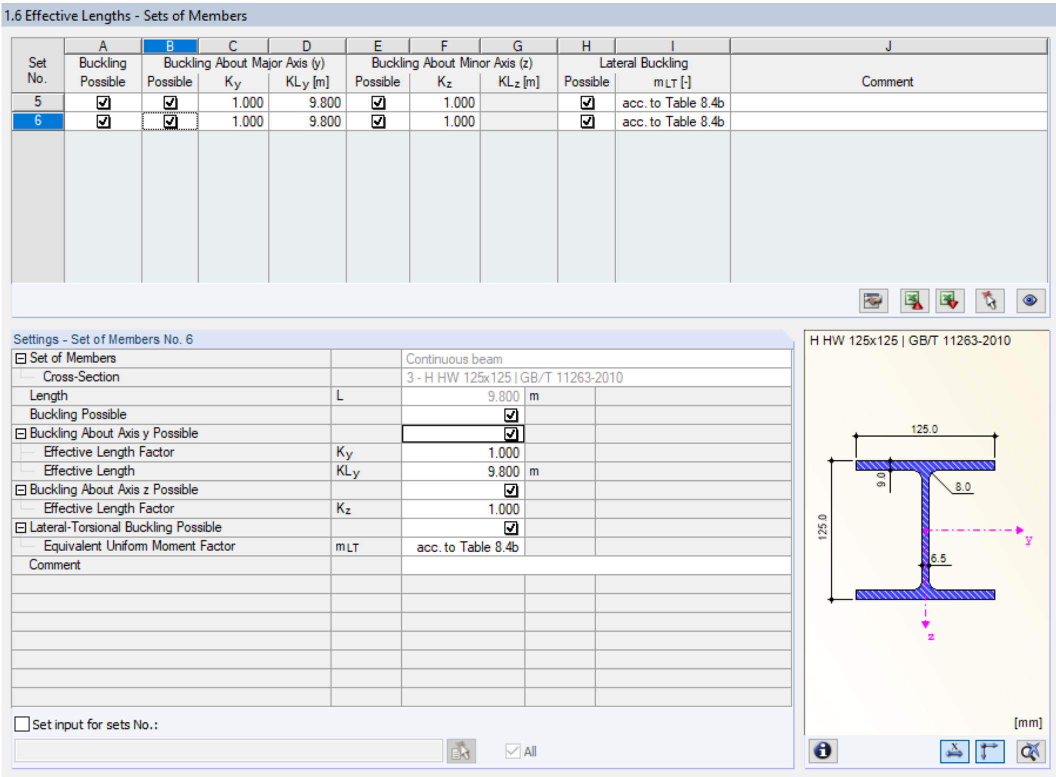


Figure 2.17: Window *1.6 Effective Lengths - Sets of Members*

The concept of this window is similar to the previous Window *1.5 Effective Lengths - Members*. Here, you can enter the effective lengths for buckling about both principal axes of the set of members as well for lateral-torsional buckling, as described in Chapter 2.5. They define the boundary conditions of the set of members that is handled in its entirety.

2.7 Nodal Supports - Sets of Members

This window is displayed if at least one set of members has been selected for design in Window *1.1 General Data*.

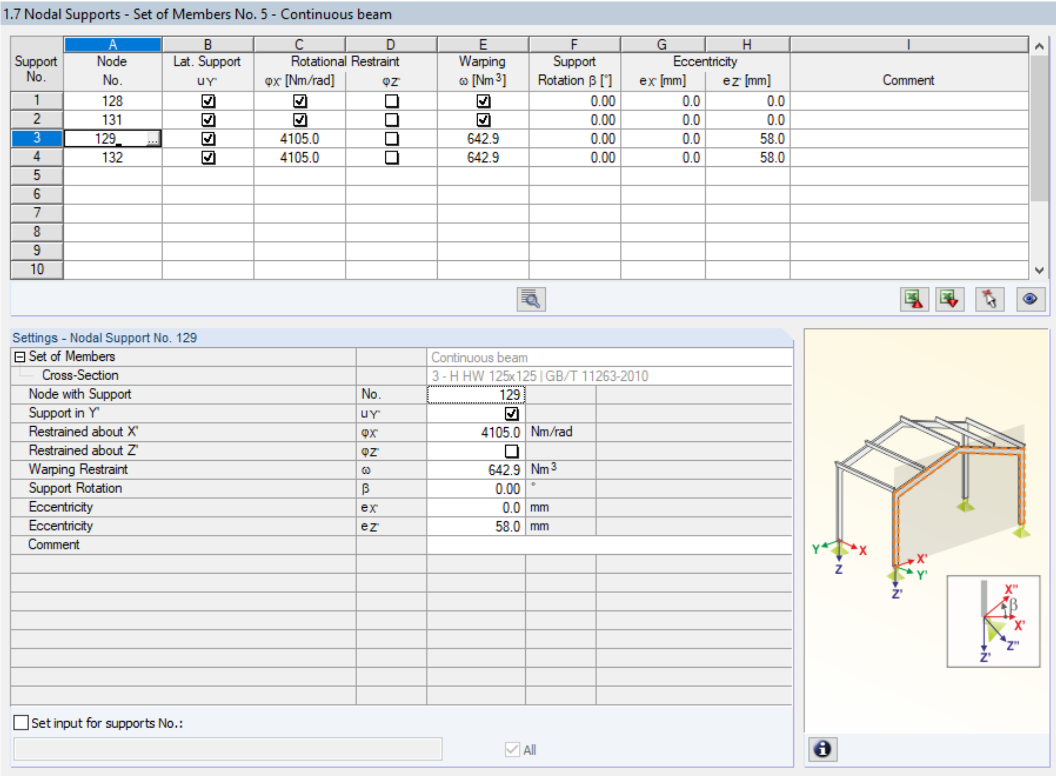


Figure 2.18: Window *1.7 Nodal Supports - Set of Members*

The current table manages the boundary conditions of the set of members that is selected on the left in the navigator.

The supports defined in RFEM or RSTAB (for example, in Z for a continuous beam) are not relevant in this window: The distributions of moments and shear forces for the determination of the amplification factor are automatically imported from RFEM/RSTAB. Here, you define the support conditions affecting the stability failure (buckling, lateral-torsional buckling).

Supports on the start and end nodes of the set of members are preset. Any other supports, for example due to connected members, must be added manually. Use the button to select nodes graphically in the RFEM/RSTAB work window.

To determine the elastic critcal moment M_{cr}, the program creates a planar framework with four degrees of freedom for each node.

The orientation of the axes in the set of members is important for the nodal support definition. The program checks the position of the nodes and internally defines the axes of the nodal supports for Window 1.7 according to Figure 2.19 to 2.22. The [Local Coordinate System] button below the model graphic can help you with the orientation: Use it to display the set of members in a partial view where the axes are clearly visible.

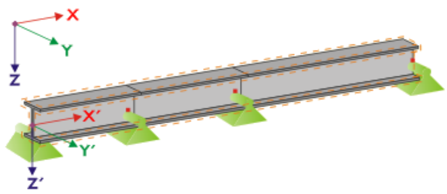


Figure 2.19: Auxiliary coordinate system for nodal supports - straight set of members

If all members of a set of members rest on a straight line, as shown in Figure 2.19, the local coordinate system of the first member in the set of members corresponds to the equivalent coordinate system of the entire set of members.

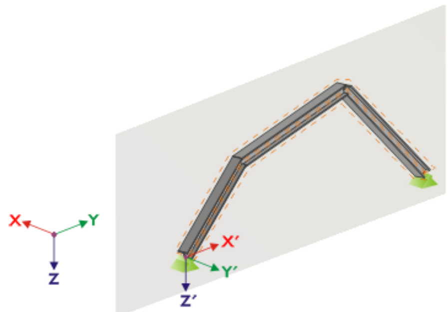


Figure 2.20: Auxiliary coordinate system for nodal supports - set of members in vertical plane

If the members of a set of members do not rest on a straight line, they still have to be located in the same plane. In Figure 2.20, the members rest in a vertical plane. In this case, the X'-axis is horizontal and oriented in the direction of the plane. The Y'-axis is horizontal as well and defined perpendicular to the X'-axis. The Z'-axis is oriented perpendicular downwards.

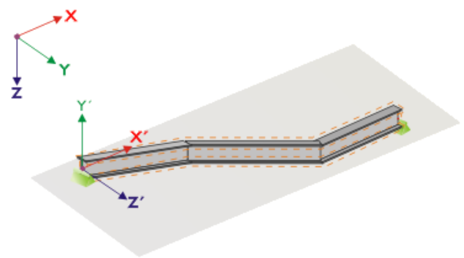


Figure 2.21: Auxiliary coordinate system for nodal supports - set of members in horizontal plane

If the members of a buckled set of members rest in a horizontal plane, the X'-axis is defined parallel to the X-axis of the global coordinate system. Thus, the Y'-axis is oriented in the opposite direction to the global Z-axis, and the Z'-axis is directed parallel to the global Y-axis.

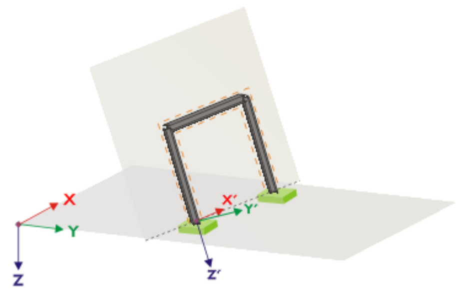


Figure 2.22: Auxiliary coordinate system for nodal supports - set of members in inclined plane

Figure 2.22 shows the general case of a buckled set of members: The members do not rest on a straight line, but in an inclined plane. The definition of the X'-axis results from the intersection line between the inclined and the horizontal plane. Thus, the Y'-axis is perpendicular to the X'-axis and in vertical position to the inclined plane. The Z'-axis is defined perpendicular to the X'-axis and Y'-axis.

The buttons below the graphic have the following functions:

Button	Function
	Shows model or system sketch
	Shows members as 3D rendering or wire-frame model
	Shows current set of members or entire model
	Displays irrelevant members of model as transparent or opaque
	Shows set of members with local coordinate system or entire model
	Shows view in direction of X-axis
	Shows view in opposite direction of Y-axis
	Shows view in direction of Z-axis
	Sets isometric view

Table 2.3: Buttons for cross-section graphic

With the [Edit warp stiffener] button it is possible to determine the constant of a warp spring by the program.

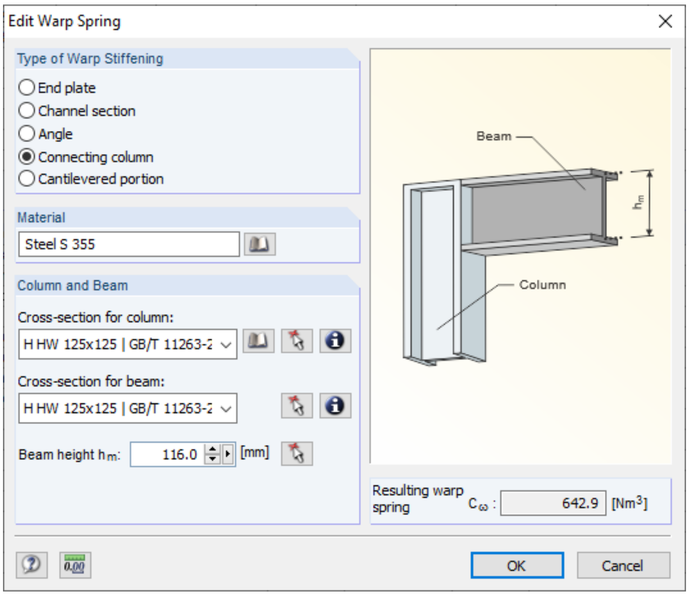


Figure 2.23: Dialog box *Edit Warp Spring*

The following warp stiffening types are available in the *Edit Warp Spring* dialog box:

- End plate
- Channel section
- Angle
- Connecting column
- Cantilevered portion

Materials and cross-sections can be selected by using the lists and [Library] buttons. With the button you can select them also graphically in the RFEM/RSTAB model.

Based on the parameters, RF-/STEEL HK determines the *Resulting warp spring* C_{ω} which can then be imported with [OK] in Window 1.7.

2.8 Member Hinges - Sets of Members

This window is displayed if at least one set of members is selected for design in Window *1.1 General Data*. Here, you can define hinges for members within the set of members that, for structural reasons, don't transfer the degrees of freedom locked in Window 1.7 as internal forces. Make sure that no double hinges are generated in coaction with Window 1.7.

The table manages the hinge parameters of the set of members selected in the navigator on the left.

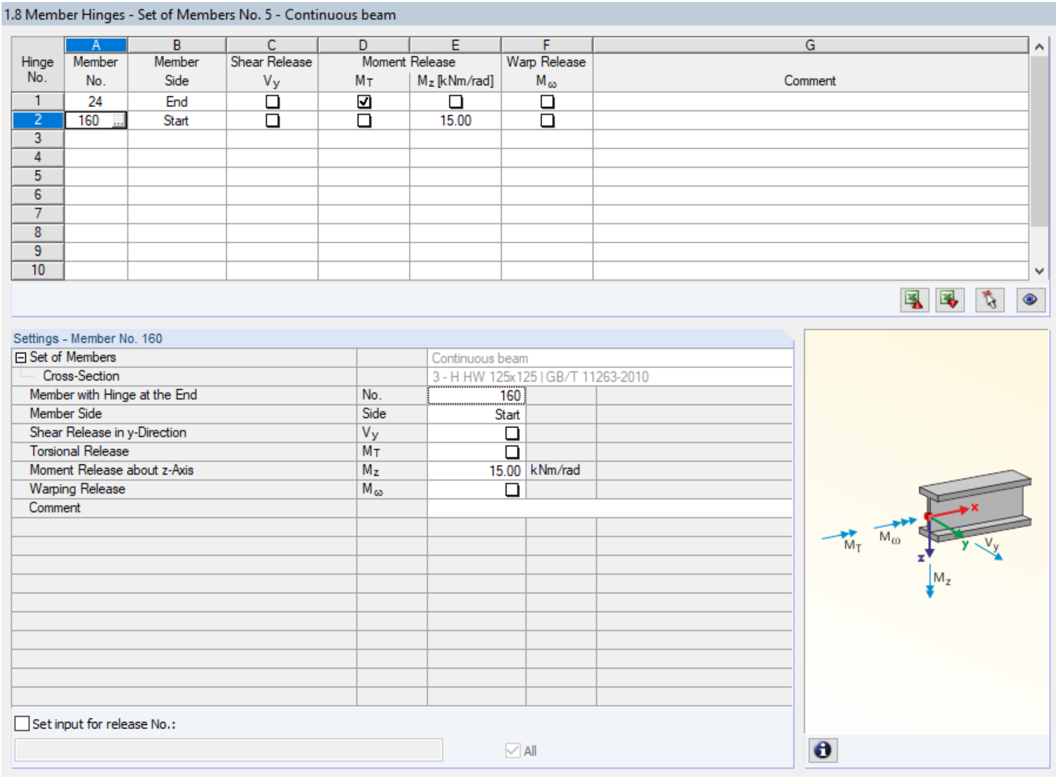
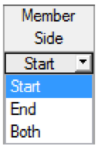


Figure 2.24: Window *1.8 Member Hinges - Set of Members*

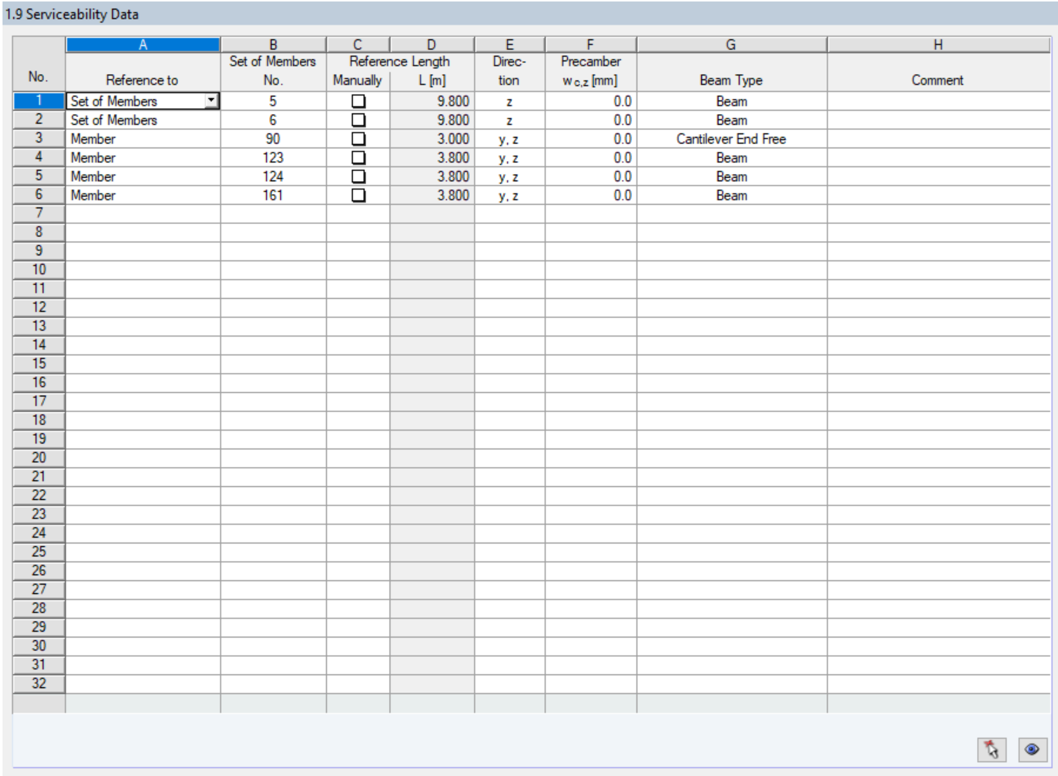


In column B, you specify the *Member Side* where the hinge is located, or if there are hinges on both member sides.

In columns C to F, you can define the releases or spring constants in order to adjust the set of members model to the support conditions of Window 1.7.

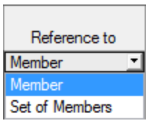
2.9 Serviceability Parameters

This window controls various settings for the serviceability limit state design. It is displayed if corresponding data has been set in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 10).


1.9 Serviceability Data

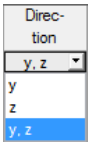
No.	A Reference to	B Set of Members No.	C Reference Length Manually	D Reference Length L [m]	E Direction	F Precamber $w_{c,z}$ [mm]	G Beam Type	H Comment
1	Set of Members	5	☐	9.800	z	0.0	Beam	
2	Set of Members	6	☐	9.800	z	0.0	Beam	
3	Member	90	☐	3.000	y, z	0.0	Cantilever End Free	
4	Member	123	☐	3.800	y, z	0.0	Beam	
5	Member	124	☐	3.800	y, z	0.0	Beam	
6	Member	161	☐	3.800	y, z	0.0	Beam	
7								
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

Figure 2.25: Window *1.9 Serviceability Data*



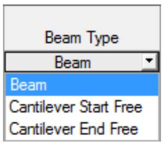
In column A, you decide whether the deformation refers to single members or sets of members. For a set of members, it is necessary that a uniform member orientation and rotation of all included members is given. Only in this way can the deformation components be determined correctly.

In column B, you enter the numbers of the members or sets of members that you want to design. You can also use the ... button to select them graphically in the RFEM/RSTAB work window. Then, the *Reference Length* appears automatically in column D. The column presets the lengths of the members and sets of members. You can adjust the values *Manually* after ticking the check box in column C.



In column E, you define the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z (or u and v for unsymmetrical cross-sections).

In column F, a *Precamber* can be taken into account. The precamber's general direction is defined in the *Serviceability* tab of the *Details* dialog box (see Figure 3.3, page 29). If the precamber is related to the "major" principal axis y or u, the column title $w_{c,z}$ changes to $w_{c,y}$ or $w_{c,u}$.



For a correct application of limit deformations, the *Beam Type* is of vital importance. In column G, you can specify whether a beam or a cantilever is to be designed and which end is free of support.

The setting in the *Serviceability* tab of the *Details* dialog box indicates whether the deformations are related to the undeformed system or to shifted members ends/set of members ends (see Figure 3.3, page 29).

3 Calculation

3.1 Detailed Settings

The designs are based on the internal forces determined in RFEM or RSTAB.

Before you start the calculation, it is recommended to check the design details. You can access the corresponding dialog box in all windows of the add-on module by using the [Details] button.

The *Details* dialog box has the following tabs:

- Ultimate limit state
- Stability
- Serviceability
- General

3.1.1 Ultimate Limit State

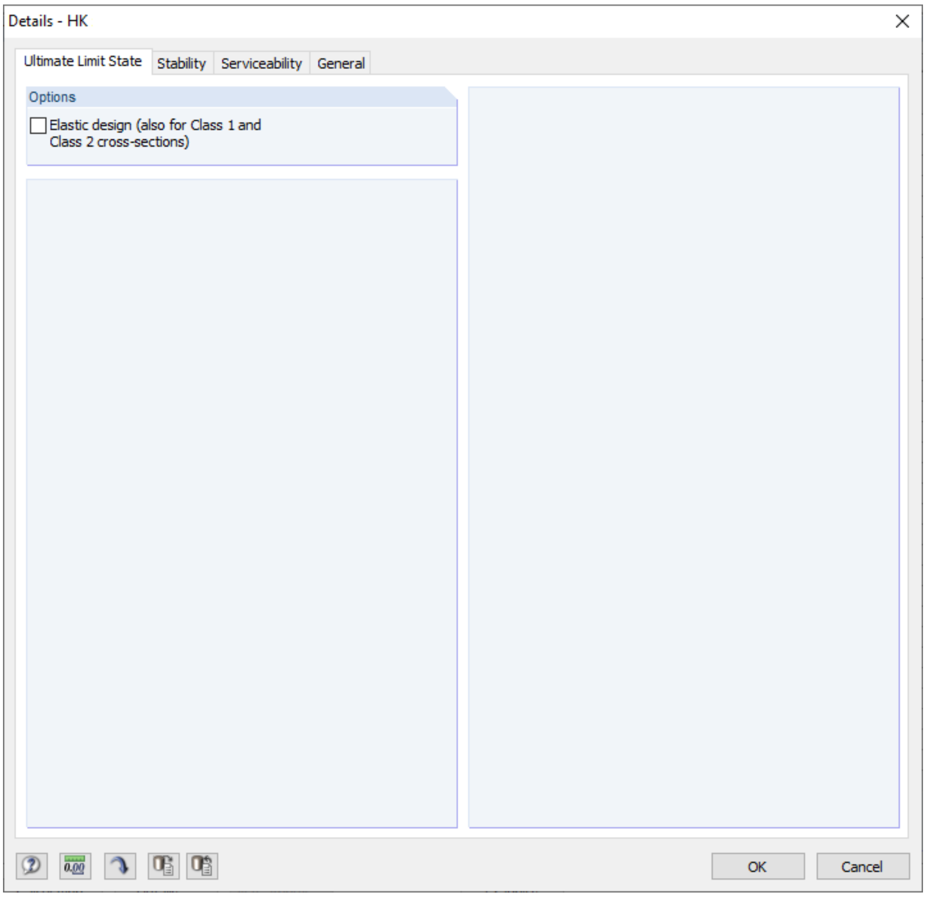


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

Options

Cross-sections assigned to class 1 or 2 are designed plastically by RF-/STEEL HK. If this is not desired, you can activate the *Elastic design* also for those cross-section classes.

3.1.2 Stability

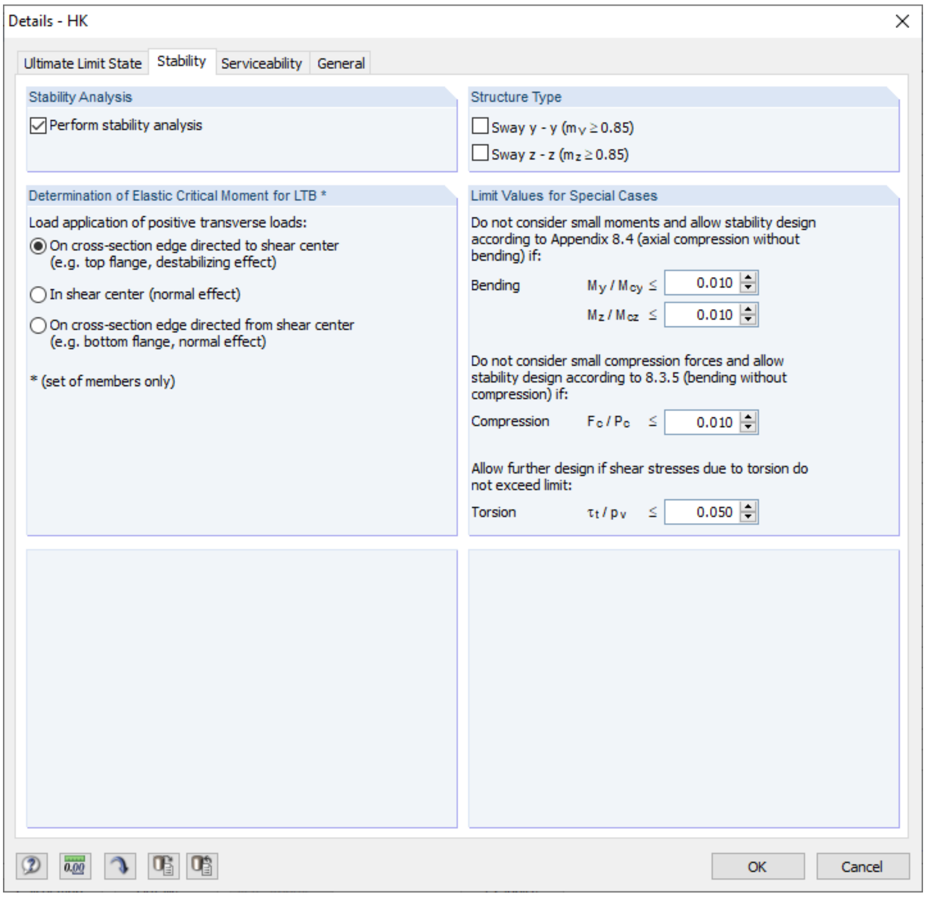


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

The *Perform stability analysis* check box controls whether to run a stability analysis in addition to the cross-section designs. If you clear the check box, Windows 1.4 through 1.8 are not displayed.

Determination of Elastic Critical Moment for LTB

For sets of members, RF-/STEEL HK determines the elastic critical moment by an eigenvalue analysis. For the calculation, the program uses a finite member model to determine M_{cr} taking into account the following items:

- Dimensions of gross cross-section
- Load type and position of load application point
- Effective distribution of moments
- Lateral restraints (by support conditions)
- Effective boundary conditions

If *transverse loads* are available, it is important to define the location where these forces are acting on the cross-section: Depending on the load application, transverse loads can be stabilizing or destabilizing, and thus have a major impact on the elastic critical moment.

The signs of the eccentricities are related to the cross-section's shear center M. The following DLUBAL article provides more information about the sign convention for transverse loads:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000880

Structure Type

If *sway* of structural components is relevant, the equivalent uniform moment factor for flexural buckling about x- and y-axis can be applied in a customary manner.

Limit Values for Special Cases

To design unsymmetrical cross-sections with the intended axial compression according to [1] 8.7, you can neglect *small moments* about the major and the minor axis by the settings defined in this dialog section.

Analogously, it is possible for the pure check of bending according to [1] 8.3 to neglect *small compression forces* by defining a limit ratio of F_c / P_c.

The intended *Torsion* is not clearly specified in [1]. If there is a torsional stress not exceeding the shear stress ratio of 5% preset by default, it is neglected for the stability design; only results for flexural and lateral-torsional buckling are displayed.

If one of the limits in this dialog section is exceeded, a note appears in the results window and the program won't perform any stability analysis. However, the cross-section designs are performed independently. These limit settings are not part of the Code [1]. Modifying the limits is the user's responsibility.

3.1.3 Serviceability

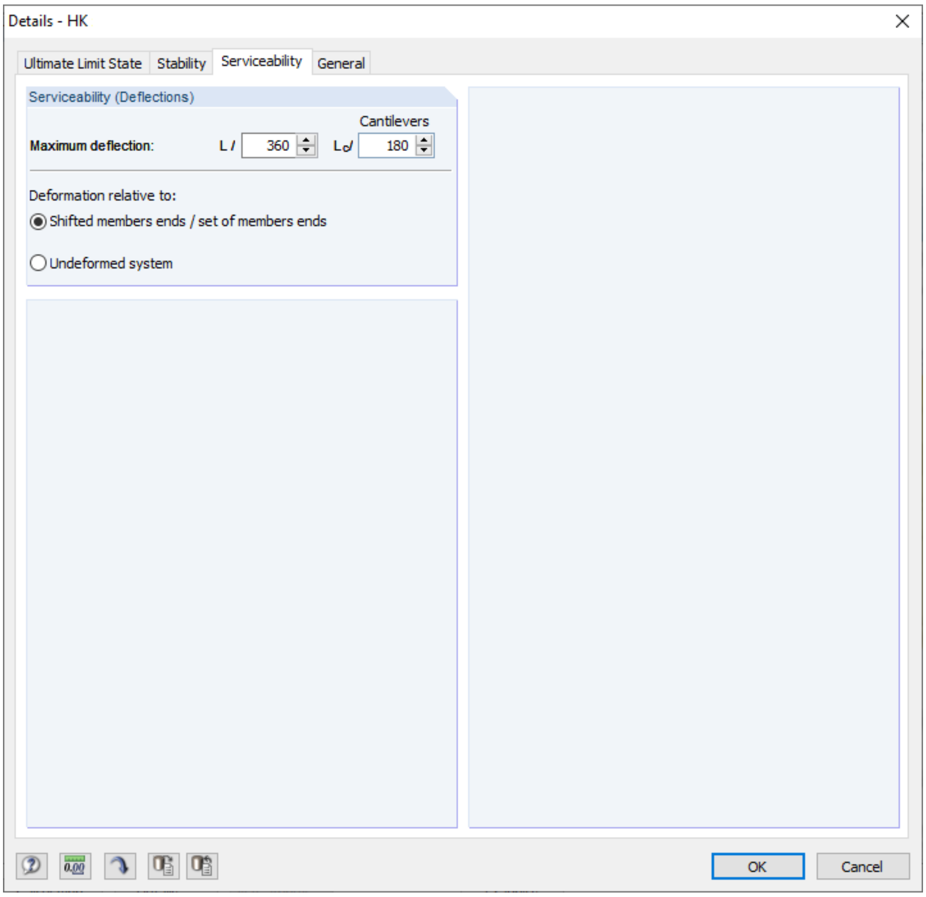


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Serviceability (Deflections)

You can check and adjust, if necessary, the *maximum deflection* relevant to beams and cantilevers. The deflection limits recommended by [1] Table 5.1 are preset.

The *Deformation relative to* options control whether the maximum deflections are related to the shifted ends of members or sets of members (connection line between start and end nodes of the deformed system) or to the undeformed initial system. Generally, the deflections are designed relative to the displacements in the entire structural system.

The following DLUBAL article presents an example describing the relation of deformations.
https://www.dlubal.com/en/support-and-learning/support/knowledge-base/001081

3.1.4 General

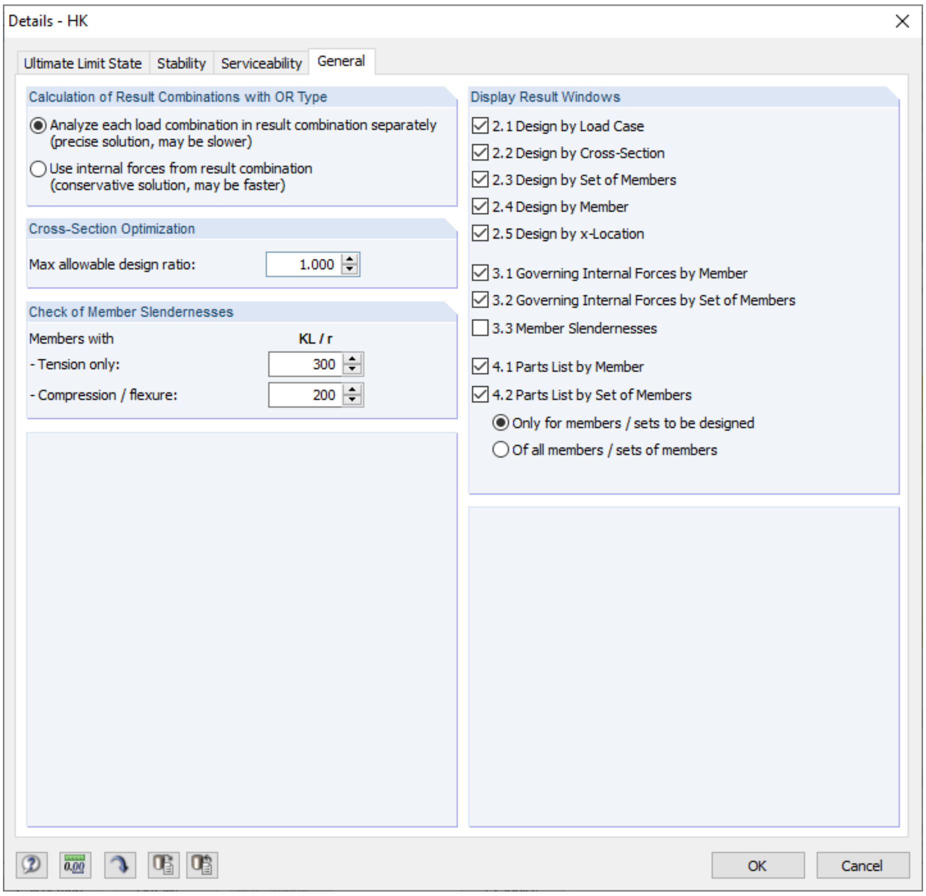


Figure 3.4: Dialog box *Details*, tab *General*

Calculation of Result Combinations with OR Type

If combinations are created automatically, usually many load combinations (CO) are produced. Generally, these combinations are summarized in a result combination (RC) as alternatively acting in an 'OR' connection which provides the envelope: CO1/p or CO2/p or CO3/p or CO4/p etc. For the design of these result combinations, you have two possibilities in RF-/STEEL HK.

The *load components* of the contained combinations can be analyzed *separately*. Thus, the elastic critical moments for lateral-torsional buckling are determined separately for each constellation, and the designs are performed accordingly. This approach provides exact results. However, it is very time-consuming and requires a high computational effort.

Alternatively, it is possible to *Analyze result combinations generally*. This calculation runs considerably faster because RF-/STEEL HK uses only the extreme values with the corresponding internal forces for the design. However, the result may be incorrect if the RC includes a combination where several internal forces (such as N and M_y) are together just below the extreme values.

Cross-Section Optimization

By default, the optimization is targeted on the maximum allowable design ratio of 100%. If necessary, you can set a different design ratio in this text box.

Check of Member Slendernesses

In the two text boxes, you can specify the limit values KL / r in order to define the member slendernesses. Separate specifications are possible for members with tension forces only and for members with compression and flexure. The slenderness ratios recommended by [1] 6.6.4 are preset.

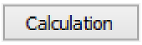
In Window 3.3, the limit values are compared to the real member slendernesses. This window is available after the calculation (see Chapter 4.8, page 40) if the corresponding check box in the *Display Result Windows* section to the right is selected.

Display Result Windows

In this dialog section, you can select which result windows including parts list are displayed. The windows are described in Chapter 4.

Window *3.3 Member Slendernesses* is deactivated by default.

3.2 Starting the Calculation



In all input windows of the RF-/STEEL HK add-on module, you can start the calculation by clicking the [Calculation] button.

RF-/STEEL HK searches for the results of the load cases, load combinations and result combinations to be designed. If they cannot be found, the program starts the RFEM or RSTAB calculation to determine the design-relevant internal forces.

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

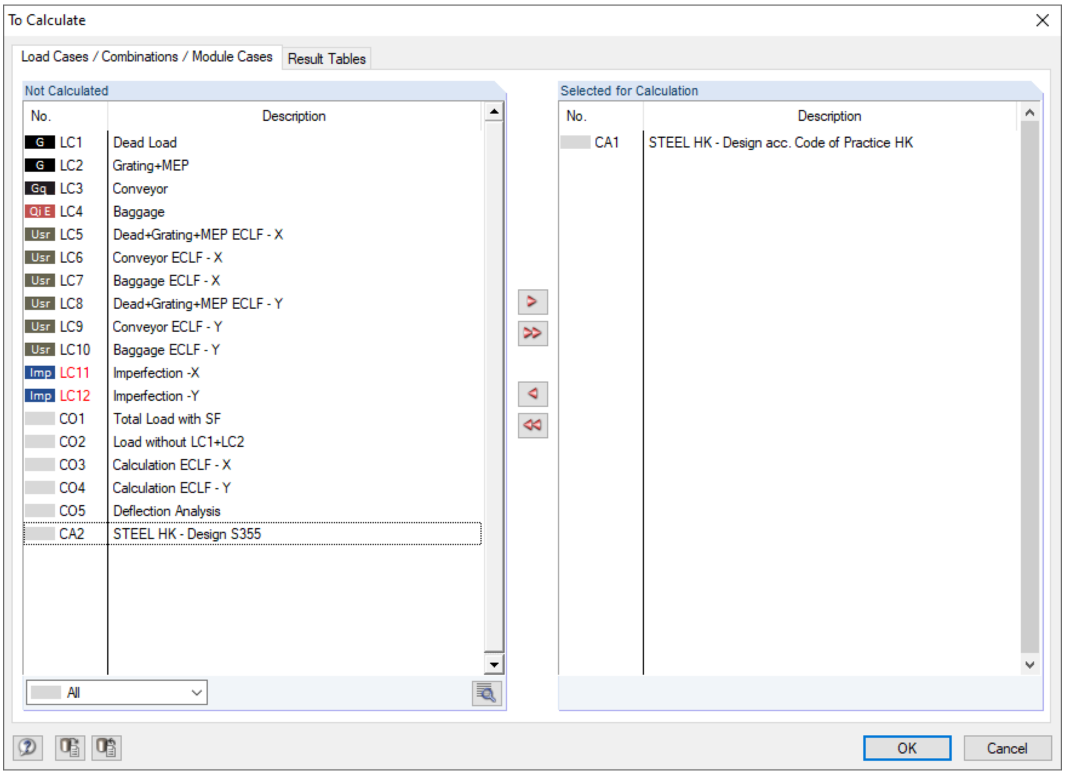
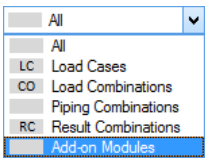
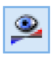


Figure 3.5: Dialog box *To Calculate*



If the RF-/STEEL HK design cases are missing in the *Not Calculated* section, select *All* or *Add-on Modules* in the drop-down list below the list.

To transfer the selected RF-/STEEL HK cases to the list on the right, use the [>] button. Then, click [OK] to start the calculation.



You can also calculate a design case directly by using the list in the toolbar: Set the RF-/STEEL HK case and click the [Show Results] button.

Figure 3.6: Direct calculation of a STEEL HK design case in RSTAB

Subsequently, you can observe the calculation process in the solver dialog box.

4 Results

Window *2.1 Design by Load Case* appears immediately after the calculation.

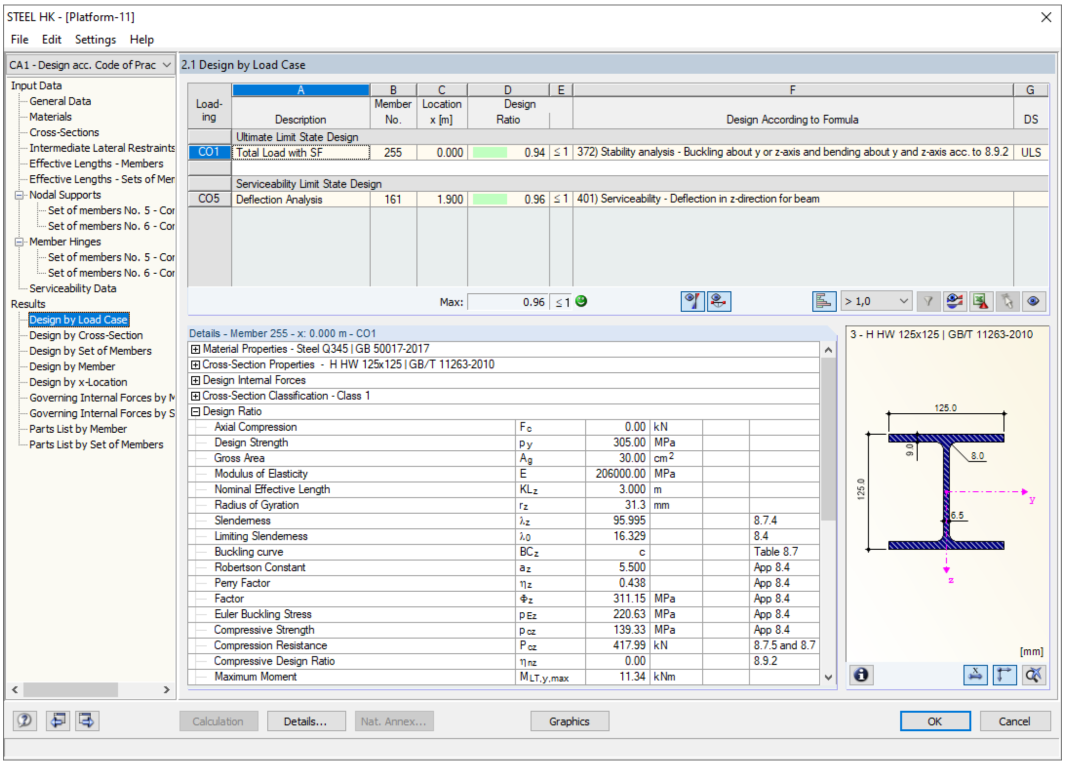
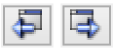


Figure 4.1: Result window with designs and intermediate values

The designs are shown in the result windows 2.1 through 2.5, sorted by different criteria.

Windows 3.1 and 3.2 list the governing internal forces; Window 3.3 gives information on member slendernesses.

Windows 4.1 and 4.2 show the parts lists by members and sets of members.



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

Click [OK] to save the results. Then, you exit RF-/STEEL HK and return to the main program.

Chapter 4 describes the result windows one by one. Evaluating and checking results is described in Chapter 5, page 43.

4.1 Design by Load Case

The upper part of the window shows a summary of the governing designs, sorted by load case, load combination, and result combination. In addition, the table is subdivided into ultimate and serviceability limit state design results.

The lower part includes detailed information on the cross-section properties, analyzed internal forces, and design parameters for the load case selected above.

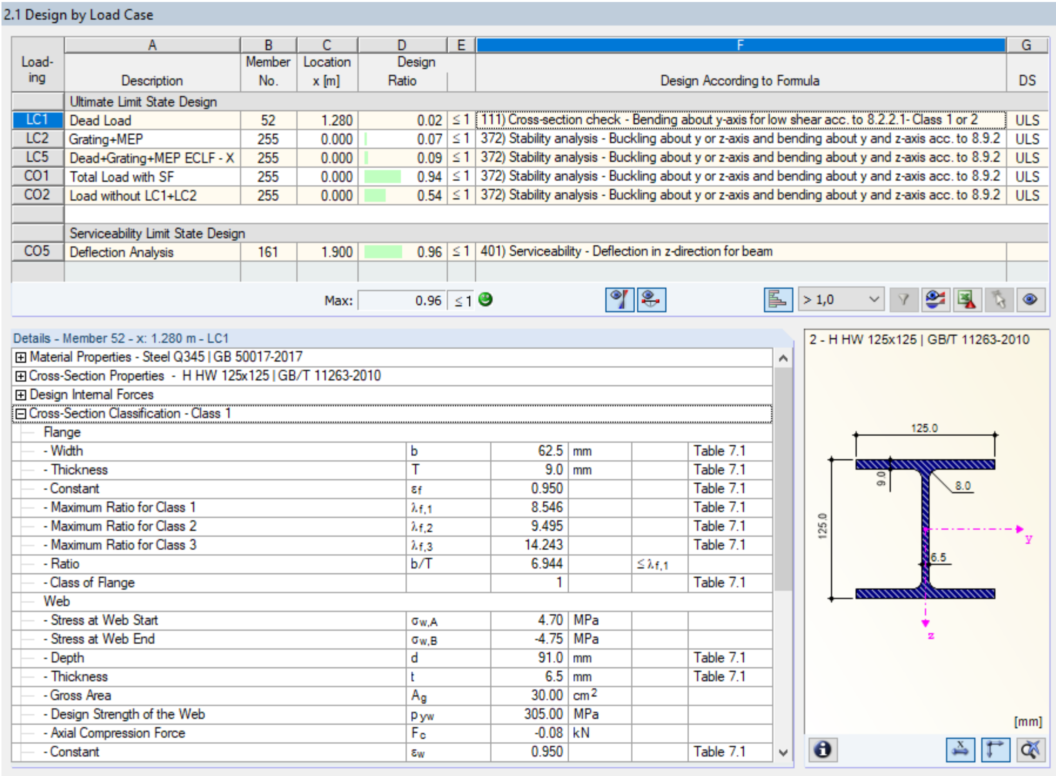


Figure 4.2: Window *2.1 Design by Load Case*

Description

This column shows the descriptions of the load cases, load and result combinations for which the designs have been performed.

Member No.

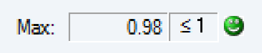
This column shows the number of the member with the maximum design ratio for the designed action.

Location x

This column shows the respective x-location of the member where the maximum design ratio occurs. The following member locations x are used for the table output:

- Start and end node
- Division points according to possibly defined member division (see RFEM table 1.16 or RSTAB table 1.6)
- Member division according to specification for member results (*Calculation Parameters* dialog box of RFEM/RSTAB, *Global Calculation Parameters* tab)
- Extreme values of internal forces

Design Ratio



Columns D and E show the design conditions according to [1].

The length of the colored bar represents graphically the respective design ratio.

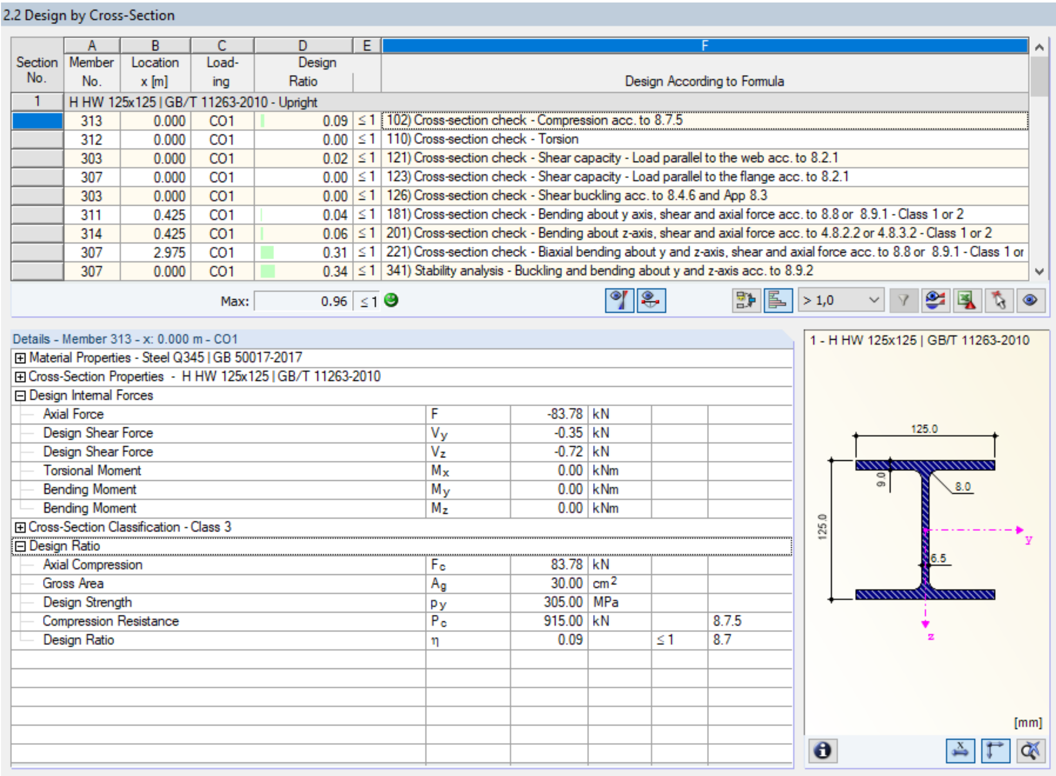
Design According to Formula

This column displays the Code's equations from which the designs have been performed.

DS

Column G provides information on the design relevant situations (DS): *ULS* for the ultimate limit state or *SLS* for the serviceability limit state design.

4.2 Design by Cross-Section



Section No.	A Member No.	B Location x [m]	C Load-ing	D Design Ratio	E	F Design According to Formula
1	H HW 125x125 | GB/T 11263-2010 - Upright					
	313	0.000	CO1	0.09	≤ 1	102) Cross-section check - Compression acc. to 8.7.5
	312	0.000	CO1	0.00	≤ 1	110) Cross-section check - Torsion
	303	0.000	CO1	0.02	≤ 1	121) Cross-section check - Shear capacity - Load parallel to the web acc. to 8.2.1
	307	0.000	CO1	0.00	≤ 1	123) Cross-section check - Shear capacity - Load parallel to the flange acc. to 8.2.1
	303	0.000	CO1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 8.4.6 and App 8.3
	311	0.425	CO1	0.04	≤ 1	181) Cross-section check - Bending about y axis, shear and axial force acc. to 8.8 or 8.9.1 - Class 1 or 2
	314	0.425	CO1	0.06	≤ 1	201) Cross-section check - Bending about z-axis, shear and axial force acc. to 4.8.2.2 or 4.8.3.2 - Class 1 or 2
	307	2.975	CO1	0.31	≤ 1	221) Cross-section check - Biaxial bending about y and z-axis, shear and axial force acc. to 8.8 or 8.9.1 - Class 1 or
	307	0.000	CO1	0.34	≤ 1	341) Stability analysis - Buckling and bending about y and z-axis acc. to 8.9.2
			Max:	0.96	≤ 1	

Details					
Material Properties - Steel Q345 | GB 50017-2017					
Cross-Section Properties - H HW 125x125 | GB/T 11263-2010					
Design Internal Forces					
Axial Force	F	-83.78	kN		
Design Shear Force	V_y	-0.35	kN		
Design Shear Force	V_z	-0.72	kN		
Torsional Moment	M_x	0.00	kNm		
Bending Moment	M_y	0.00	kNm		
Bending Moment	M_z	0.00	kNm		
Cross-Section Classification - Class 3					
Design Ratio					
Axial Compression	F_c	83.78	kN		
Gross Area	A_g	30.00	cm²		
Design Strength	p_y	305.00	MPa		
Compression Resistance	P_c	915.00	kN		8.7.5
Design Ratio	η	0.09		≤ 1	8.7

Figure 4.3: Window *2.2 Design by Cross-Section*

In this results window, the maximum design ratios of all members and actions selected for design are listed by cross-section. The results are sorted by cross-section design and stability analysis as well as serviceability limit state design.

If there is a tapered member, the cross-sections of the member start and end are listed separately.

4.3 Design by Set of Members

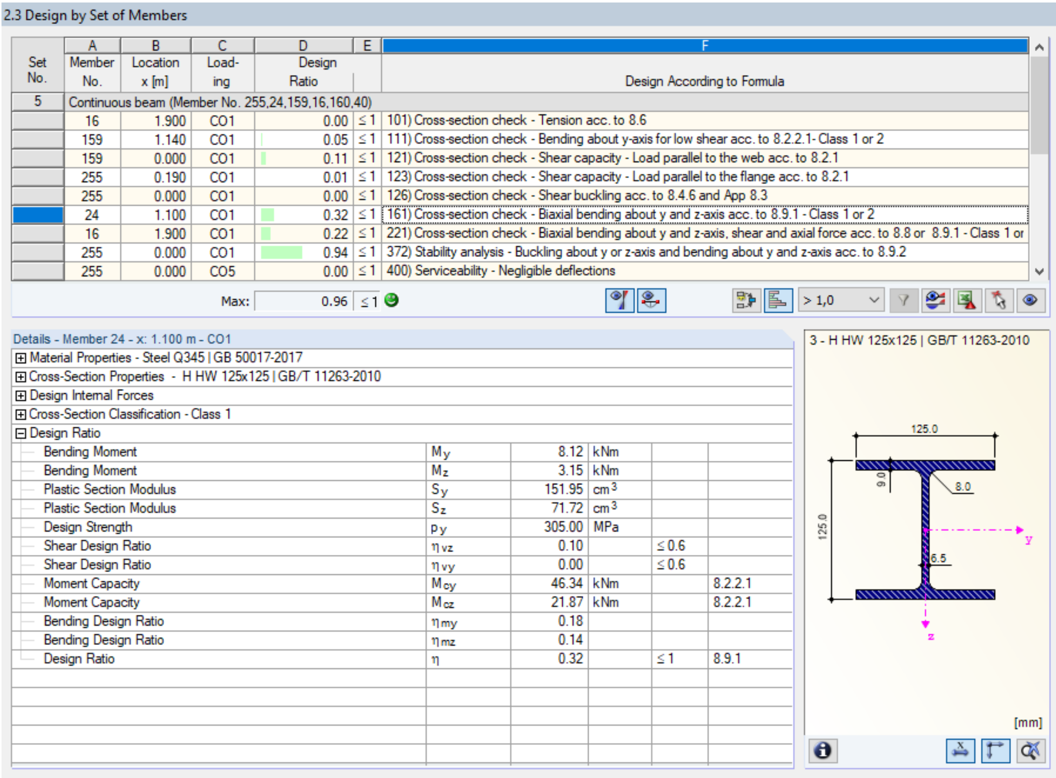


Figure 4.4: Window *2.3 Design by Set of Members*

This results window is displayed if at least one set of members has been selected for design. The window lists the maximum design ratios sorted by set of members.

The *Member No.* column shows the number of the member within the set of members that bears the maximum ratio for the individual design criteria.

The output by set of members allows you to clearly present the design of an entire structural group (a frame, for example).

4.4 Design by Member

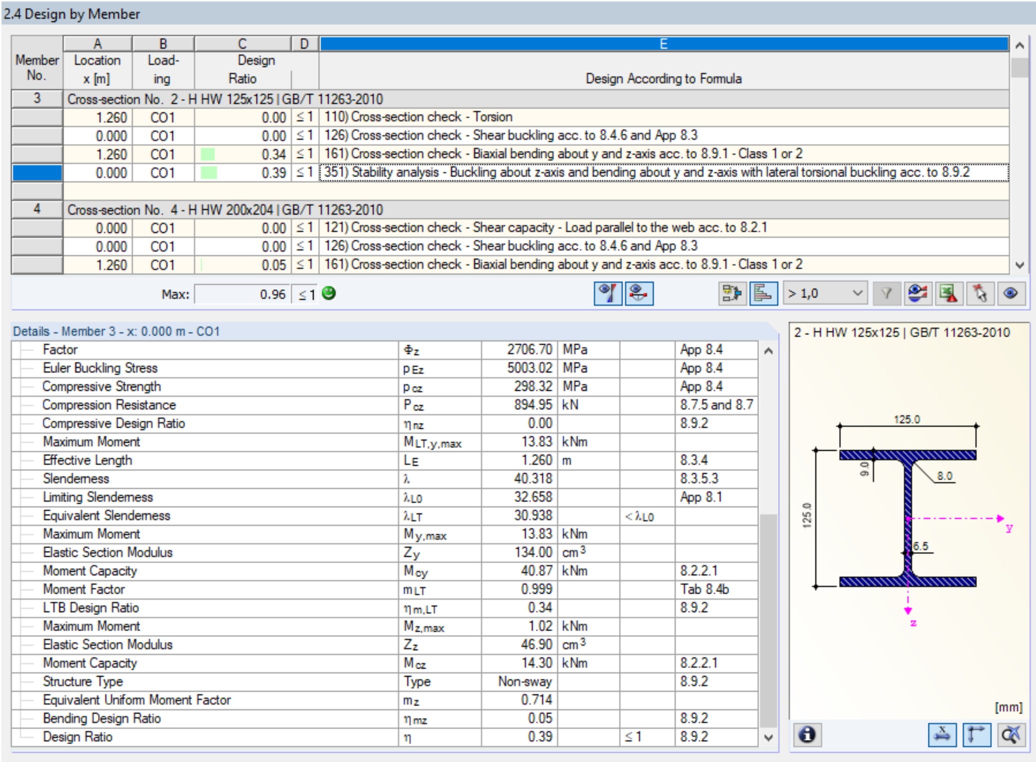


Figure 4.5: Window *2.4 Design by Member*

This results window shows the maximum design ratios for the individual designs sorted by member number. The columns are described in detail in Chapter 4.1 on page 34.

4.5 Design by x-Location

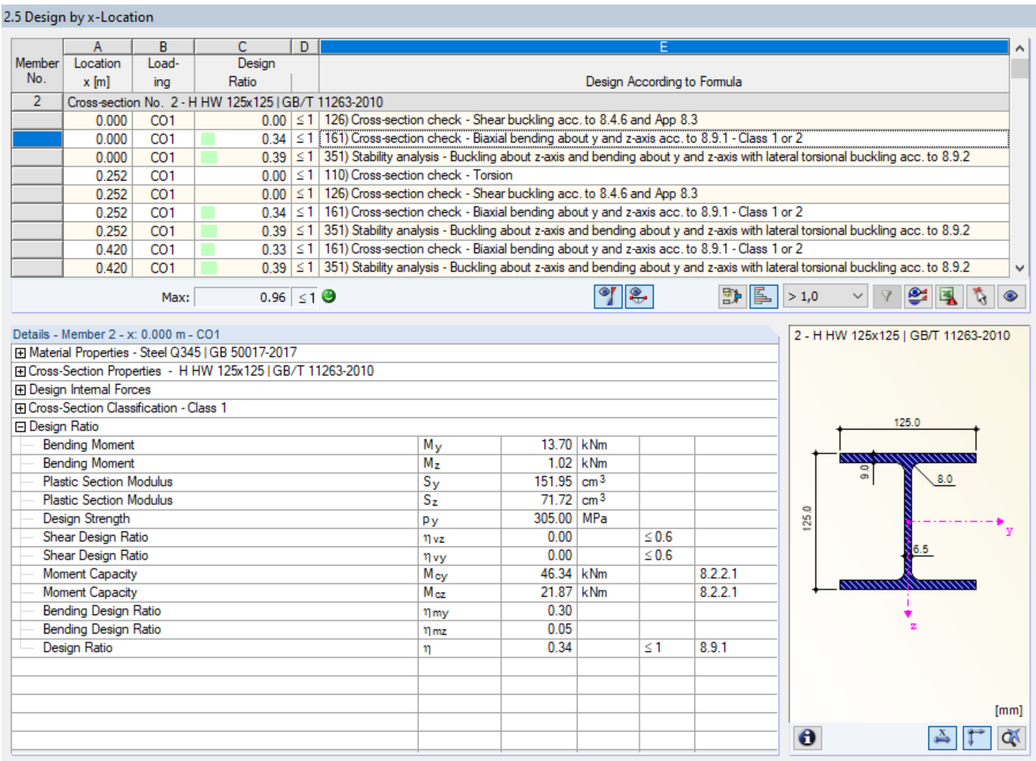
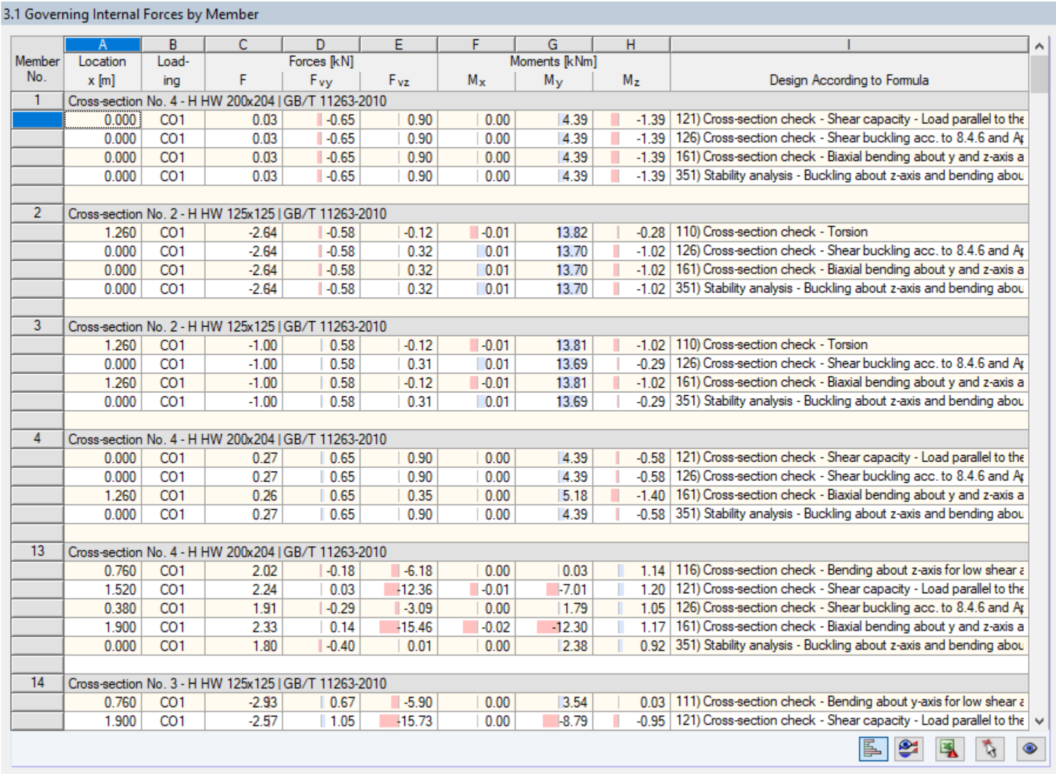


Figure 4.6: Window *2.5 Design by x-Location*

This results window lists the maxima for each member at all locations ***x***, resulting from the division points defined in RFEM or RSTAB:

- Start and end node
- Division points according to possibly defined member division (see RFEM table 1.16 or RSTAB table 1.6)
- Member division according to specification for member results (*Calculation Parameters* dialog box of RFEM/RSTAB, *Global Calculation Parameters* tab)
- Extreme values of internal forces

4.6 Governing Internal Forces by Member


3.1 Governing Internal Forces by Member

Member No.	A: Location x [m]	B: Loading	C: Forces [kN] F	D: F_{vy}	E: F_{vz}	F: Moments [kNm] M_x	G: M_y	H: M_z	I: Design According to Formula
1	Cross-section No. 4 - H HW 200x204 | GB/T 11263-2010								
	0.000	CO1	0.03	-0.65	0.90	0.00	4.39	-1.39	121) Cross-section check - Shear capacity - Load parallel to the
	0.000	CO1	0.03	-0.65	0.90	0.00	4.39	-1.39	126) Cross-section check - Shear buckling acc. to 8.4.6 and A
	0.000	CO1	0.03	-0.65	0.90	0.00	4.39	-1.39	161) Cross-section check - Biaxial bending about y and z-axis a
	0.000	CO1	0.03	-0.65	0.90	0.00	4.39	-1.39	351) Stability analysis - Buckling about z-axis and bending abou
2	Cross-section No. 2 - H HW 125x125 | GB/T 11263-2010								
	1.260	CO1	-2.64	-0.58	-0.12	-0.01	13.82	-0.28	110) Cross-section check - Torsion
	0.000	CO1	-2.64	-0.58	0.32	0.01	13.70	-1.02	126) Cross-section check - Shear buckling acc. to 8.4.6 and A
	0.000	CO1	-2.64	-0.58	0.32	0.01	13.70	-1.02	161) Cross-section check - Biaxial bending about y and z-axis a
	0.000	CO1	-2.64	-0.58	0.32	0.01	13.70	-1.02	351) Stability analysis - Buckling about z-axis and bending abou
3	Cross-section No. 2 - H HW 125x125 | GB/T 11263-2010								
	1.260	CO1	-1.00	0.58	-0.12	-0.01	13.81	-1.02	110) Cross-section check - Torsion
	0.000	CO1	-1.00	0.58	0.31	0.01	13.69	-0.29	126) Cross-section check - Shear buckling acc. to 8.4.6 and A
	1.260	CO1	-1.00	0.58	-0.12	-0.01	13.81	-1.02	161) Cross-section check - Biaxial bending about y and z-axis a
	0.000	CO1	-1.00	0.58	0.31	0.01	13.69	-0.29	351) Stability analysis - Buckling about z-axis and bending abou
4	Cross-section No. 4 - H HW 200x204 | GB/T 11263-2010								
	0.000	CO1	0.27	0.65	0.90	0.00	4.39	-0.58	121) Cross-section check - Shear capacity - Load parallel to the
	0.000	CO1	0.27	0.65	0.90	0.00	4.39	-0.58	126) Cross-section check - Shear buckling acc. to 8.4.6 and A
	1.260	CO1	0.26	0.65	0.35	0.00	5.18	-1.40	161) Cross-section check - Biaxial bending about y and z-axis a
	0.000	CO1	0.27	0.65	0.90	0.00	4.39	-0.58	351) Stability analysis - Buckling about z-axis and bending abou
13	Cross-section No. 4 - H HW 200x204 | GB/T 11263-2010								
	0.760	CO1	2.02	-0.18	-6.18	0.00	0.03	1.14	116) Cross-section check - Bending about z-axis for low shear a
	1.520	CO1	2.24	0.03	-12.36	-0.01	-7.01	1.20	121) Cross-section check - Shear capacity - Load parallel to the
	0.380	CO1	1.91	-0.29	-3.09	0.00	1.79	1.05	126) Cross-section check - Shear buckling acc. to 8.4.6 and A
	1.900	CO1	2.33	0.14	-15.46	-0.02	-12.30	1.17	161) Cross-section check - Biaxial bending about y and z-axis a
	0.000	CO1	1.80	-0.40	0.01	0.00	2.38	0.92	351) Stability analysis - Buckling about z-axis and bending abou
14	Cross-section No. 3 - H HW 125x125 | GB/T 11263-2010								
	0.760	CO1	-2.93	0.67	-5.90	0.00	3.54	0.03	111) Cross-section check - Bending about y-axis for low shear a
	1.900	CO1	-2.57	1.05	-15.73	0.00	-8.79	-0.95	121) Cross-section check - Shear capacity - Load parallel to the

Figure 4.7: Window *3.1 Governing Internal Forces by Member*

For each member, this window displays the governing internal forces, that is, the forces and moments that result in the maximum utilization in the individual designs.

Location x

This column shows the respective x-location of the member where the maximum design ratio occurs.

Loading

This column shows the numbers of the load case as well as the load or result combination whose internal forces result in the maximum design ratio.

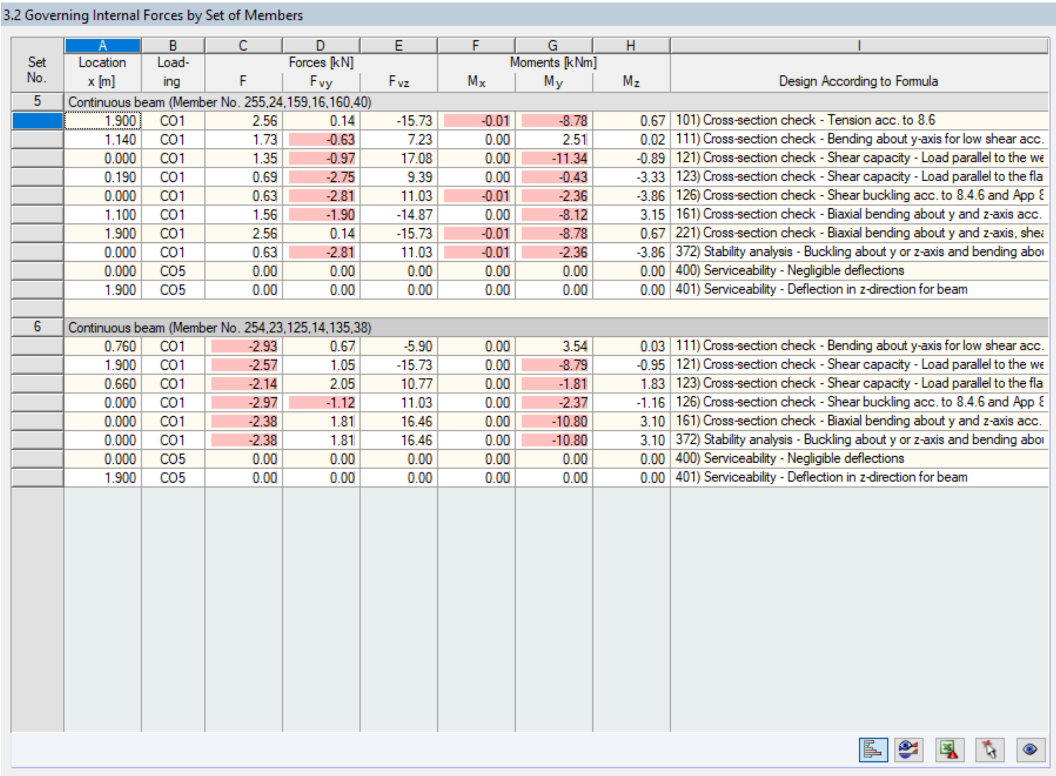
Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing the maximum ratios in the respective cross-section designs, stability analyses, and serviceability limit state designs.

Design According to Formula

The final column gives information on the design types and equations used for performing the designs according to the Code [1].

4.7 Governing Internal Forces by Set of Members


3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [m]	B Loading	C Forces [kN] F	D F_{vy}	E F_{vz}	F Moments [kNm] M_x	G M_y	H M_z	I Design According to Formula
5	Continuous beam (Member No. 255,24,159,16,160,40)								
	1.900	CO1	2.56	0.14	-15.73	-0.01	-8.78	0.67	101) Cross-section check - Tension acc. to 8.6
	1.140	CO1	1.73	-0.63	7.23	0.00	2.51	0.02	111) Cross-section check - Bending about y-axis for low shear acc.
	0.000	CO1	1.35	-0.97	17.08	0.00	-11.34	-0.89	121) Cross-section check - Shear capacity - Load parallel to the we
	0.190	CO1	0.69	-2.75	9.39	0.00	-0.43	-3.33	123) Cross-section check - Shear capacity - Load parallel to the fla
	0.000	CO1	0.63	-2.81	11.03	-0.01	-2.36	-3.86	126) Cross-section check - Shear buckling acc. to 8.4.6 and App 8
	1.100	CO1	1.56	-1.90	-14.87	0.00	-8.12	3.15	161) Cross-section check - Biaxial bending about y and z-axis acc.
	1.900	CO1	2.56	0.14	-15.73	-0.01	-8.78	0.67	221) Cross-section check - Biaxial bending about y and z-axis, shea
	0.000	CO1	0.63	-2.81	11.03	-0.01	-2.36	-3.86	372) Stability analysis - Buckling about y or z-axis and bending abo
	0.000	CO5	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deflections
	1.900	CO5	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Deflection in z-direction for beam
6	Continuous beam (Member No. 254,23,125,14,135,38)								
	0.760	CO1	-2.93	0.67	-5.90	0.00	3.54	0.03	111) Cross-section check - Bending about y-axis for low shear acc.
	1.900	CO1	-2.57	1.05	-15.73	0.00	-8.79	-0.95	121) Cross-section check - Shear capacity - Load parallel to the we
	0.660	CO1	-2.14	2.05	10.77	0.00	-1.81	1.83	123) Cross-section check - Shear capacity - Load parallel to the fla
	0.000	CO1	-2.97	-1.12	11.03	0.00	-2.37	-1.16	126) Cross-section check - Shear buckling acc. to 8.4.6 and App 8
	0.000	CO1	-2.38	1.81	16.46	0.00	-10.80	3.10	161) Cross-section check - Biaxial bending about y and z-axis acc.
	0.000	CO1	-2.38	1.81	16.46	0.00	-10.80	3.10	372) Stability analysis - Buckling about y or z-axis and bending abo
	0.000	CO5	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deflections
	1.900	CO5	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Deflection in z-direction for beam

Figure 4.8: Window *3.2 Governing Internal Forces by Set of Members*

For each set of members, this window shows the internal forces that result in the maximum ratios for the individual designs.

4.8 Members Slendernesses

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C Major Axis y k_y [-]	D i_y [mm]	E λ_y [-]	F Minor Axis z k_z [-]	G i_z [mm]	H λ_z [-]	I
52	Compression / Flexure	1.280	0.500	52.9	12.102	0.500	31.3	20.479	
53	Compression / Flexure	1.500	0.500	52.9	14.182	0.500	31.3	23.999	
57	Compression / Flexure	1.280	0.500	52.9	12.102	0.500	31.3	20.479	
58	Compression / Flexure	1.500	0.500	52.9	14.182	0.500	31.3	23.999	
61	Compression / Flexure	1.500	0.500	83.4	8.989	0.500	48.8	15.384	
65	Compression / Flexure	1.280	0.500	83.4	7.670	0.500	48.8	13.128	
69	Compression / Flexure	1.500	0.500	83.4	8.989	0.500	48.8	15.384	
72	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
76	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
84	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
90	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
122	Compression / Flexure	1.900	0.500	83.4	11.386	0.500	48.8	19.487	
123	Compression / Flexure	3.800	1.000	52.9	71.856	1.000	31.3	121.594	
124	Compression / Flexure	3.800	1.000	52.9	71.856	1.000	31.3	121.594	
126	Compression / Flexure	3.800	1.000	52.9	71.856	1.000	31.3	121.594	
128	Compression / Flexure	1.900	0.500	83.4	11.386	0.500	48.8	19.487	
132	Compression / Flexure	1.100	0.500	83.4	6.592	0.500	48.8	11.282	
133	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
134	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
136	Compression / Flexure	1.100	0.500	83.4	6.592	0.500	48.8	11.282	
140	Compression / Flexure	1.500	0.500	52.9	14.182	0.500	31.3	23.999	
141	Compression / Flexure	1.500	0.500	52.9	14.182	0.500	31.3	23.999	
142	Compression / Flexure	1.500	0.500	83.4	8.989	0.500	48.8	15.384	
151	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	
155	Compression / Flexure	1.280	0.500	83.4	7.670	0.500	48.8	13.128	
156	Compression / Flexure	1.280	0.500	52.9	12.102	0.500	31.3	20.479	
157	Compression / Flexure	1.280	0.500	52.9	12.102	0.500	31.3	20.479	
158	Compression / Flexure	1.280	0.500	83.4	7.670	0.500	48.8	13.128	
161	Compression / Flexure	3.800	1.000	52.9	71.856	1.000	31.3	121.594	
162	Compression / Flexure	3.000	1.000	52.9	56.728	1.000	31.3	95.995	

Members with compression / flexure:

Max KL_y / r_y: 71.856 ≤ 200

Max KL_z / r_z: 121.594 ≤ 200

Figure 4.9: Window *3.3 Member Slendernesses*

Details...

This result window is displayed if the corresponding check box is ticked in the *General* tab of the *Details* dialog box (see Figure 3.4, page 30).

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They have been determined as a function of the load type. Below the list, you see a comparison with the limit values defined in the *General* tab of the *Details* dialog box (see Figure 3.4, page 30).

Members of the "Tension" or "Cable" type are not displayed in this window.

This window is only of an informative nature. It provides no stability analysis of slendernesses.

4.9 Parts List by Member

Finally, there is a summary of all cross-sections included in the design case.

4.1 Parts List by Member

	A	B	C	D	E	F	G	H	I
Part No.	Cross-Section Description	Number of Members	Length [m]	Total Length [m]	Surface Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1	4 - H HW 200x204 | GB/T 11263-2010	2	1.26	2.52	2.95	0.02	56.15	70.75	0.142
2	2 - H HW 125x125 | GB/T 11263-2010	2	1.26	2.52	1.82	0.01	23.55	29.67	0.059
3	4 - H HW 200x204 | GB/T 11263-2010	8	1.90	15.20	17.78	0.11	56.15	106.69	0.853
4	2 - H HW 125x125 | GB/T 11263-2010	8	1.50	12.00	8.68	0.04	23.55	35.33	0.283
5	4 - H HW 200x204 | GB/T 11263-2010	8	1.10	8.80	10.29	0.06	56.15	61.77	0.494
6	4 - H HW 200x204 | GB/T 11263-2010	8	1.50	12.00	14.04	0.09	56.15	84.23	0.674
7	4 - H HW 200x204 | GB/T 11263-2010	4	1.28	5.12	5.99	0.04	56.15	71.87	0.287
8	2 - H HW 125x125 | GB/T 11263-2010	4	1.28	5.12	3.70	0.02	23.55	30.14	0.121
9	2 - H HW 125x125 | GB/T 11263-2010	8	3.00	24.00	17.36	0.07	23.55	70.65	0.565
10	2 - H HW 125x125 | GB/T 11263-2010	4	3.80	15.20	10.99	0.05	23.55	89.49	0.358
11	1 - H HW 125x125 | GB/T 11263-2010	12	2.97	35.70	25.82	0.11	23.55	70.06	0.841
Sum		68		138.18	119.42	0.60			4.677

Figure 4.10: Window *4.1 Parts List by Member*

Details...

By default, this list contains only the designed members. If you need a parts list for all members of the model, you can set it in the *General* tab of the *Details* dialog box (see Figure 3.4, page 30).

Part No.

The program assigns part numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

This column shows how many similar members are used for each part.

Length

This column shows the respective length of an individual member.

Total Length

The values in this column are the product from the previous two columns.

Surface Area

For each part, the program displays the surface areas relative to the total length. The surface area is determined from the *Surface* of the cross-section that can be found in windows 1.3 and 2.1 through 2.5 in the info about cross-section (see Figure 2.11, page 15).

Volume

The volume of a part is determined from the cross-sectional area and the total length.

Unit Weight

The *Unit Weight* represents the cross-section weight relative to the length of one meter. For tapered cross-sections, the program averages both cross-section weights.

Weight

The values of this column are determined from the product of the entries in columns C and G.

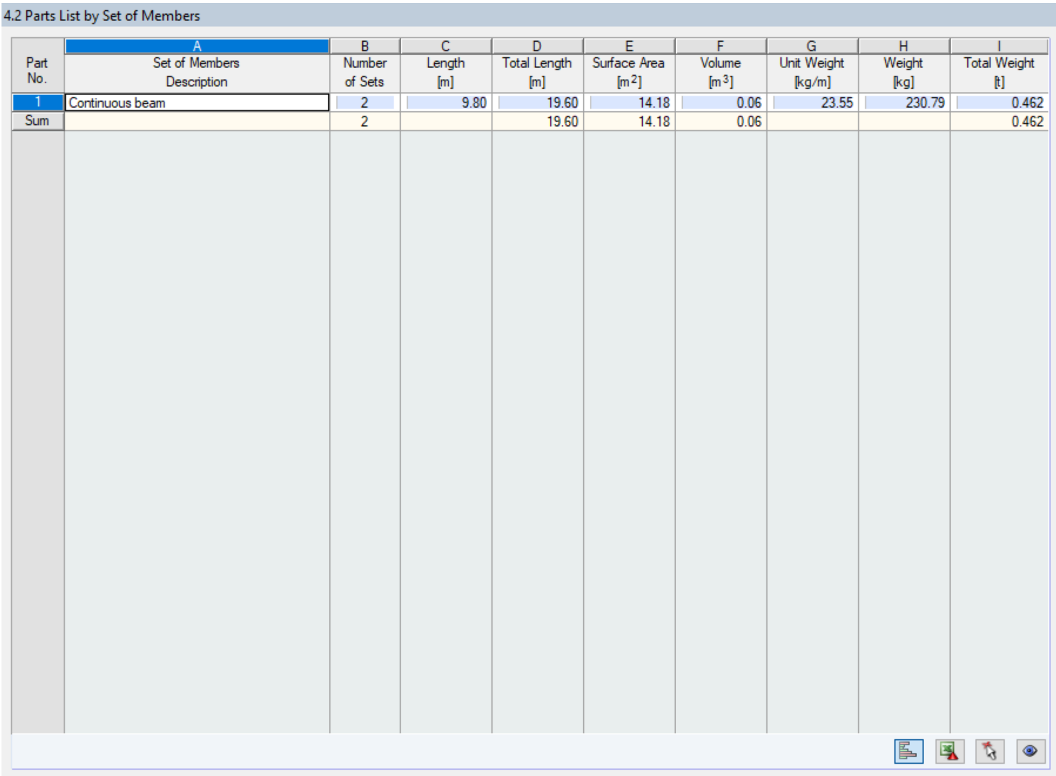
Total Weight

The final column indicates the total weight of each part.

Sum

At the bottom of the list, you find a summary of the values shown in columns B, D, E, F, and I. The last row of the *Total Weight* column shows the required total amount of steel.

4.10 Parts List by Set of Members


4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Continuous beam	2	9.80	19.60	14.18	0.06	23.55	230.79	0.462
Sum		2		19.60	14.18	0.06			0.462

Figure 4.11: Window *4.2 Parts List by Set of Members*

The last result window is displayed if at least one set of members has been selected for design. It gives an overview of the steel parts of entire structural groups such as horizontal beams.

The columns are described in the previous chapter. If there are different cross-sections within a set of members, the program averages the surface area, the volume and the cross-section weight.

5 Results Evaluation

You can evaluate the design results in different ways. The buttons below the upper table may help you.

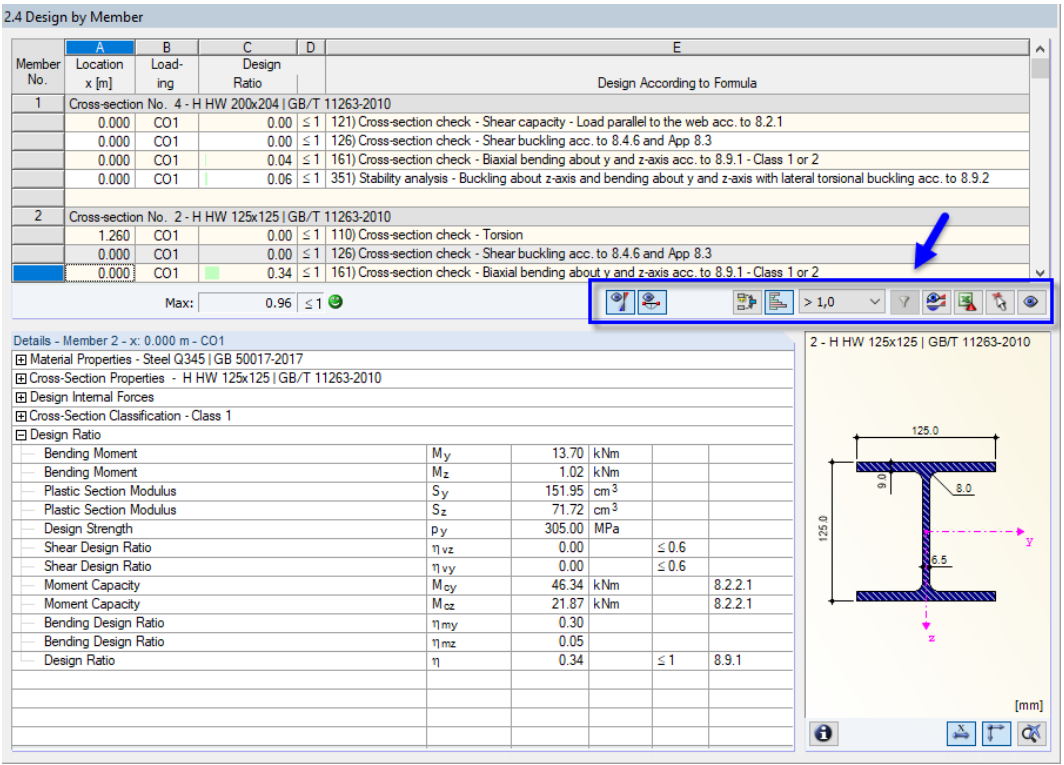


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

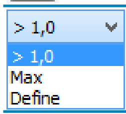
Button	Description	Function
	Ultimate limit state	Displays or hides results of ultimate limit state design
	Serviceability	Displays or hides results of serviceability limit state design
	Result combination	Creates a new result combination from governing load cases and load combinations
	Color bars	Displays or hides colored relation scales in result windows
	Filter parameters	Describes criterion by which results are filtered in tables: ratios greater than 1, maximum value, or user-defined limit
	Apply filter	Shows only rows to which filter parameters apply (ratio > 1, maximum, defined value)
	Result diagrams	Opens *Result Diagram on Member* window → Chapter 5.2, page 47
	Excel export	Exports table to MS Excel → Chapter 7.4.2, page 58
	Select member	Selects a member graphically to display its results in the table
	View mode	Jumps to RFEM or RSTAB work window to change the view

Table 5.1: Buttons in the result windows 2.1 through 2.5

5.1 Results on RFEM/RSTAB Model

You can evaluate the design results also in the work window of RFEM or RSTAB.

Background graphic and view mode

The RFEM/RSTAB work window in the background is useful when you want to find the position of a particular member in the model: The member selected in the result window of RF-/STEEL HK is highlighted in color in the background graphic. Moreover, an arrow indicates the member's x-location selected in the active table row.

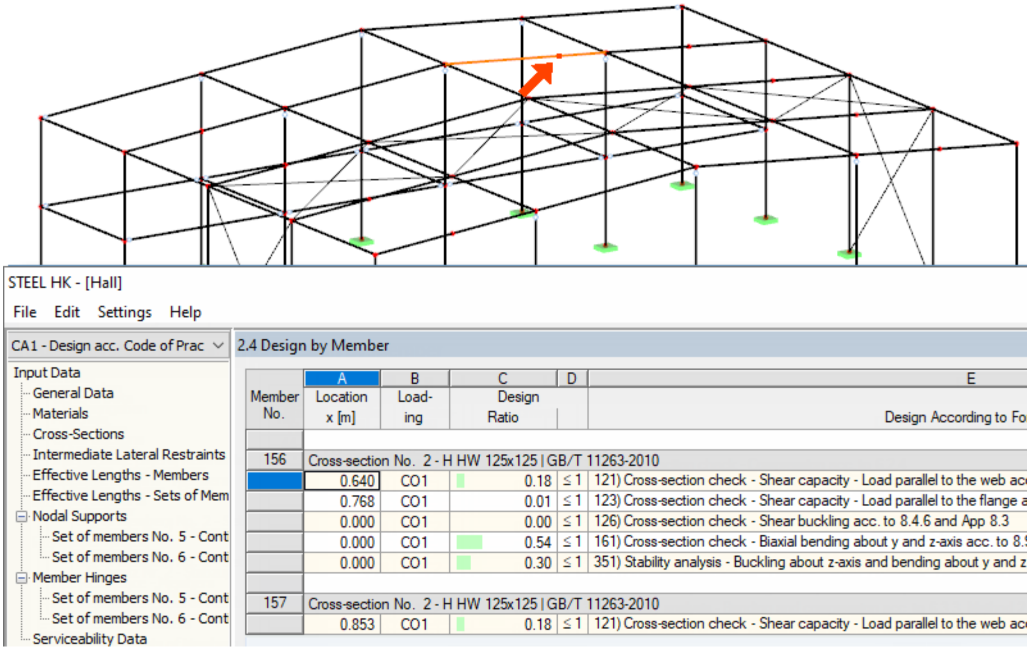
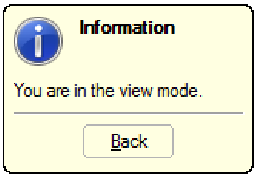


Figure 5.2: Indication of member and current *Location x* in RSTAB model



In case you cannot improve the model display by moving the RF-/STEEL HK module window, click the [Jump to graphic] button to activate the *view mode*: The program hides the module window so that you can adjust the view in the RFEM/RSTAB work window. The view mode provides the functions of the*View* menu, for example, zooming, moving, or rotating the model view. The indicating arrow remains visible.

Click [Back] to return to the RF-/STEEL HK add-on module.

RFEM/RSTAB work window

You can check the design ratios also graphically in the model: Click the [Graphics] button to exit the design module. In the work window of RFEM or RSTAB, the design ratios are now displayed like the internal forces of a load case.

In the *Results* navigator, you can select the design ratios separately for the ultimate and the serviceability limit state design as well as the fire protection design. It is also possible to check the classifications of cross-sections.

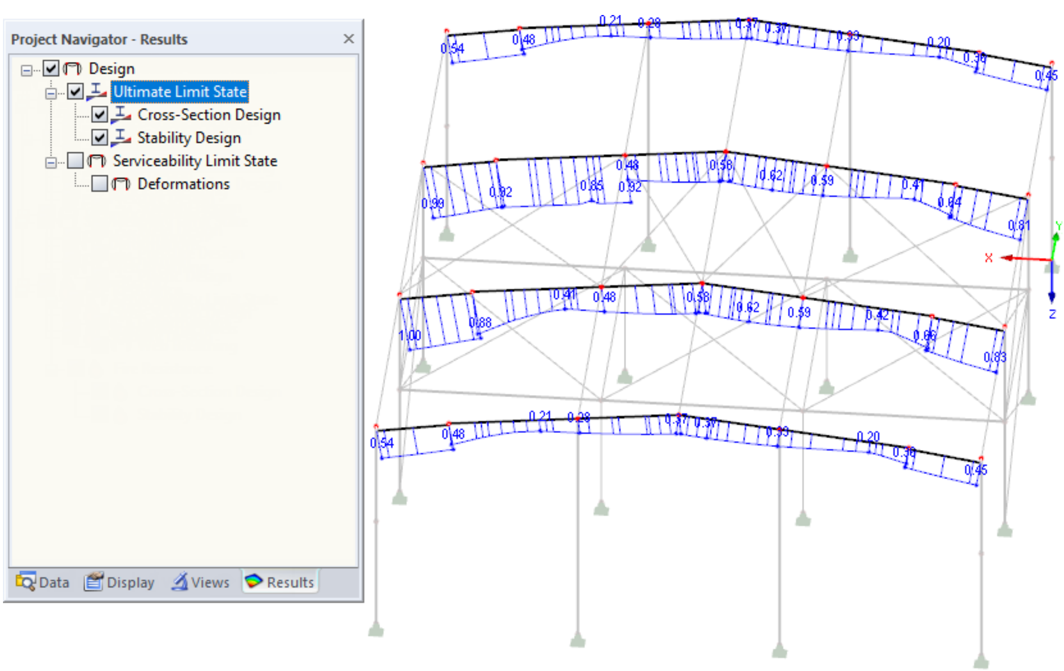
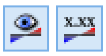
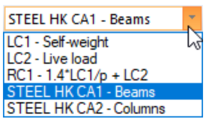


Figure 5.3: *Results* navigator for RF-/STEEL HK



To turn on and off the display of design results, click the [Show Results] button that you know from the display of internal forces in RFEM/RSTAB. Click the [Show Result Values] button to the right to display the result values.

The RFEM/RSTAB tables are not relevant for the evaluation of the design results.



You can set the design cases in the drop-down list of the RFEM/RSTAB menu bar.

To adjust the graphical representation of results, you can use the **Results** → **Members** entry in the *Display* navigator. The display of design ratios is *Two-Colored* by default.

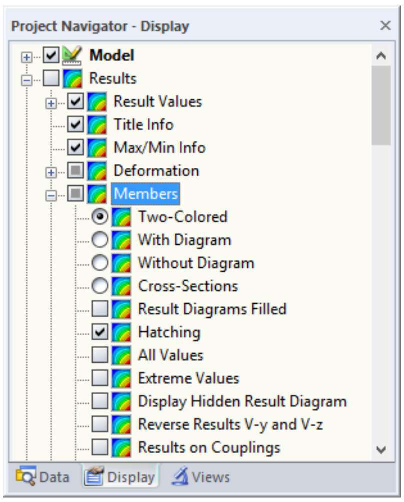


Figure 5.4: *Display* navigator options for *Results* → *Members*

If you select a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color panel becomes available, providing common control functions. The functions are described in Chapter 3.4.6 of the RFEM or RSTAB manual.

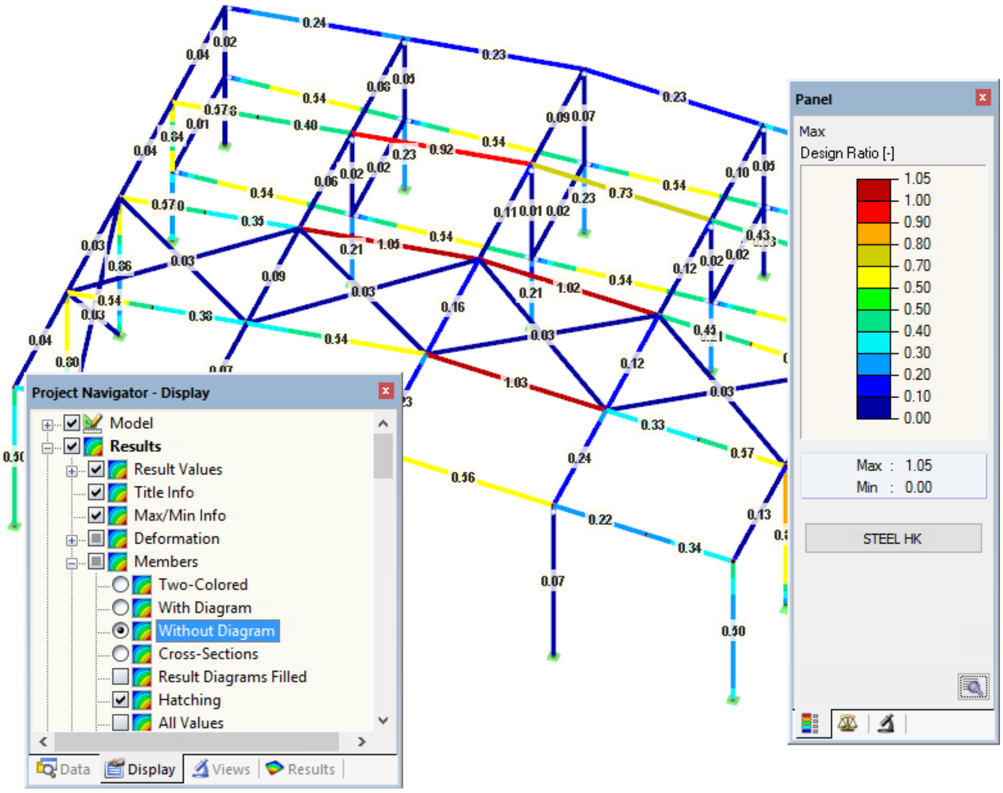


Figure 5.5: Design ratios with display option *Without Diagram*

It is possible to transfer the graphics of design results to the printout report (see Chapter 6.2, page 51).

STEEL HK

To return to the add-on module, click the [RF-/STEEL HK] button in the panel.

5.2 Result Diagrams

You can also evaluate the member results graphically in the form of result diagrams.

Select the member (or set of members) in the RF-/STEEL HK result window by clicking in the member's table row. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. You can find it below the upper result table (see Figure 5.1, page 43).

To access the result diagrams in the RFEM/RSTAB graphic, select on the menu

Results → Result Diagrams for Selected Members

or use the corresponding button in the toolbar of RFEM or RSTAB.

A window opens which shows graphically the distribution of the design values on the member or set of members.

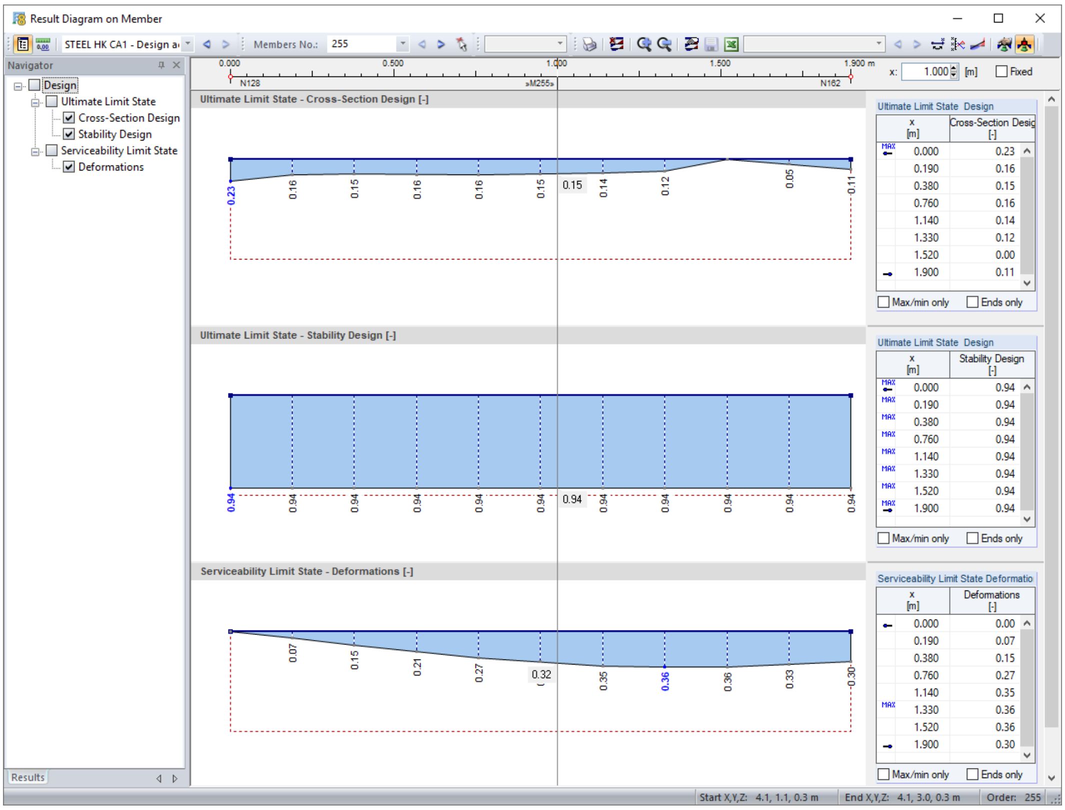
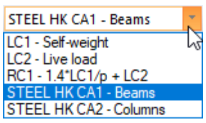


Figure 5.6: Dialog box *Result Diagram on Member*

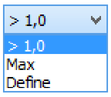
Again, the *Results* navigator allows for a targeted selection among classifications and the designs of the ultimate and the serviceability limit state as well as of fire resistance.



Use the list in the toolbar to switch between the RF-/STEEL HK design cases.

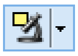
The *Result Diagram on Member* dialog box is described in Chapter 9.5 of the RFEM or RSTAB manual.

5.3 Filter for Results



The arrangement of the RF-/STEEL HK result windows already provides a selection by various criteria. In addition, there are filter options for the tables (see Figure 5.1, page 43) to limit the numerical output by design ratios. This function is also described in a Dlubal article: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000733

Furthermore, you can use the filter options described in Chapter 9.9 of the RFEM manual, or Chapter 9.7 of the RSTAB manual, to evaluate the results graphically.



The possibilities offered by the *Visibility* function (see Chapter 9.9.1 in RFEM manual, or Chapter 9.7.1 in RSTAB manual) are also available for RF-/STEEL HK to filter the members for the evaluation.

Filtering designs

The design ratios can easily be used as filter criteria in the work window of RFEM or RSTAB that you can access with the [Graphics] button. To apply this function, the panel must be displayed. If it is not active, select on the RFEM/RSTAB menu

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

or use the toolbar button shown on the left.

The panel is described in Chapter 3.4.6 of the RFEM or RSTAB manual. The filter settings for the results must be defined in the first panel tab (Color scale). As this tab is not available for the two-colored results display, you have to set the display options *With/Without Diagram* or *Cross-Sections* in the *Display* navigator.

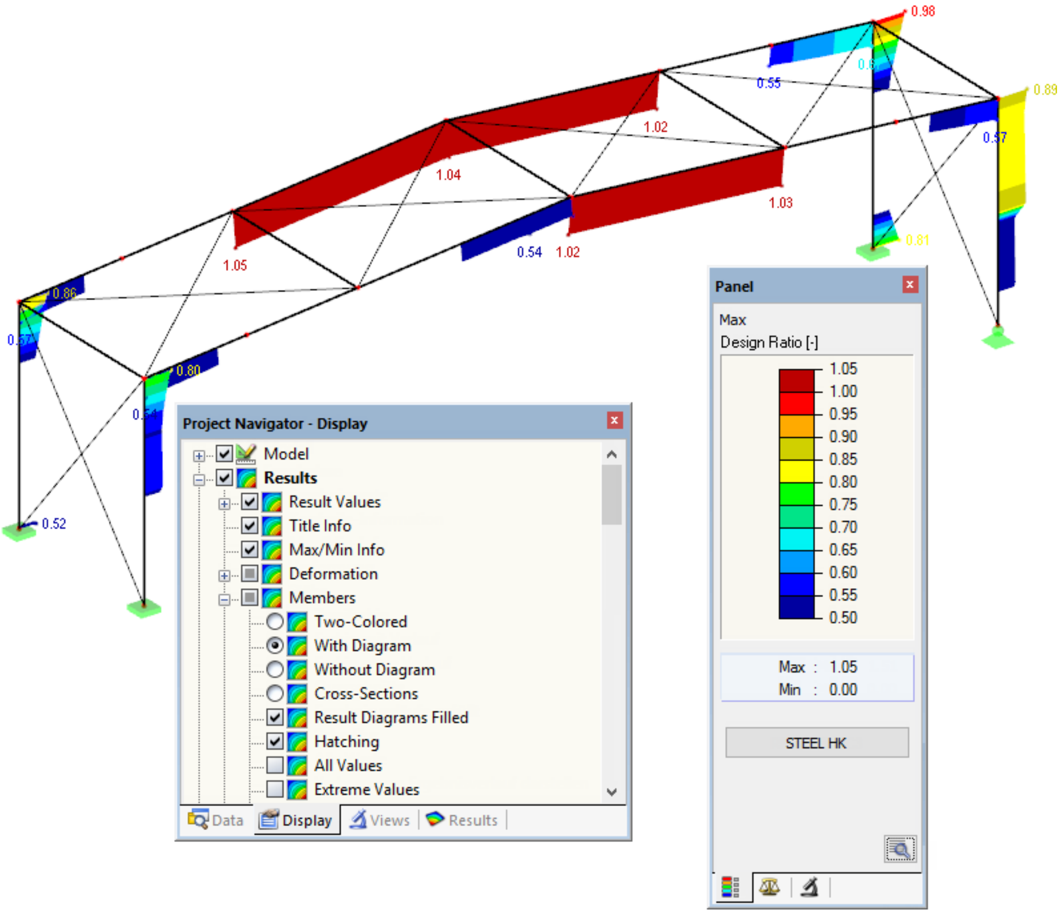


Figure 5.7: Filtering design ratios with adjusted color scale

As shown in Figure 5.7, the panel's scale of values can be set in such a way that only design ratios greater than 0.50 are displayed in a color range between blue and red.

The function *Display Hidden Result Diagram* in the *Display* navigator (**Results** → **Members**) shows all design ratios which are beyond the value spectrum. Those diagrams are represented by dotted lines.

Filtering members

In the *Filter* tab of the control panel, you can specify the numbers of particular members to display their results filtered. The function is described in Chapter 9.9.3 of the RFEM manual or in Chapter 9.7.3 of the RSTAB manual.

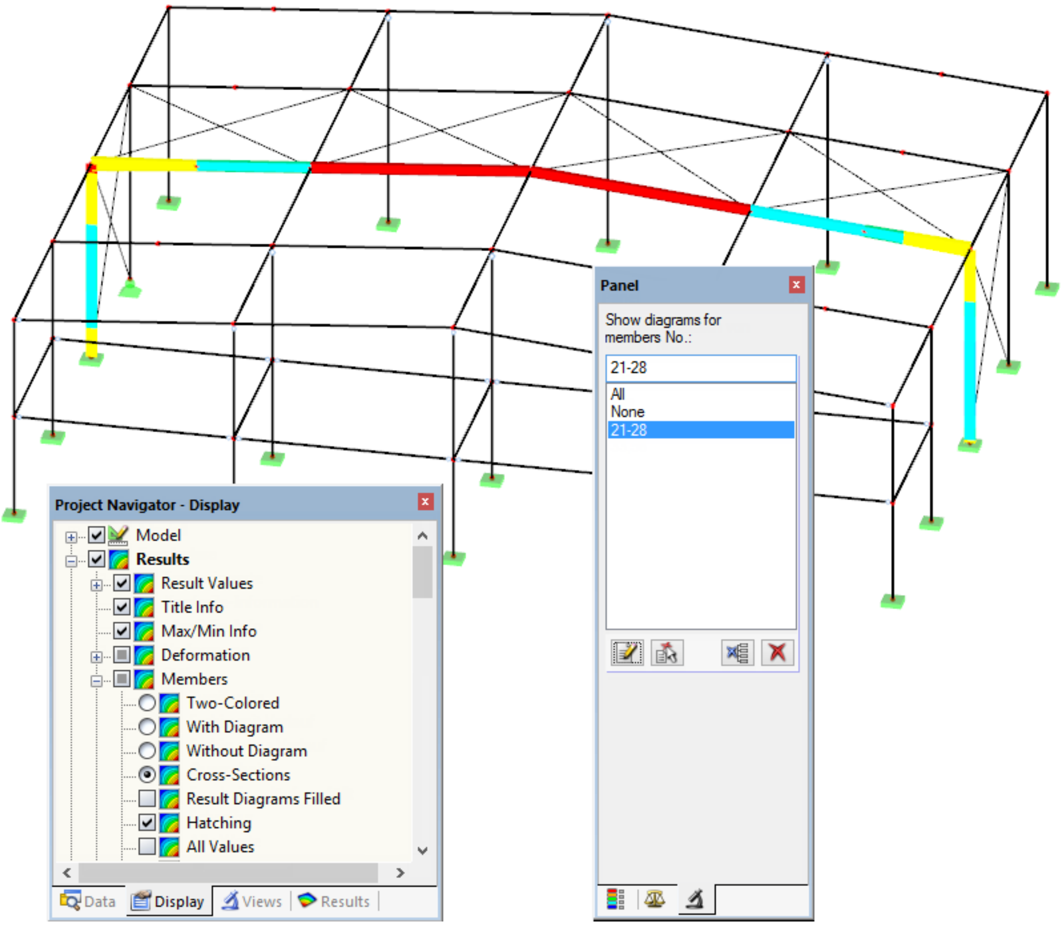


Figure 5.8: Member filter for design ratios of a hall frame

In contrast to the visibility function, the model will be displayed completely in the graphic. The figure above shows the design ratios of a hall frame. The remaining members are displayed in the model but are shown without design ratios.

6 Printout

6.1 Printout Report

A printout report is generated for the data of the RF-/STEEL HK add-on module, like in RFEM or RSTAB, to which you can add graphics and descriptions. The selection in the printout report determines which data from the design module will finally be included in the printout.

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

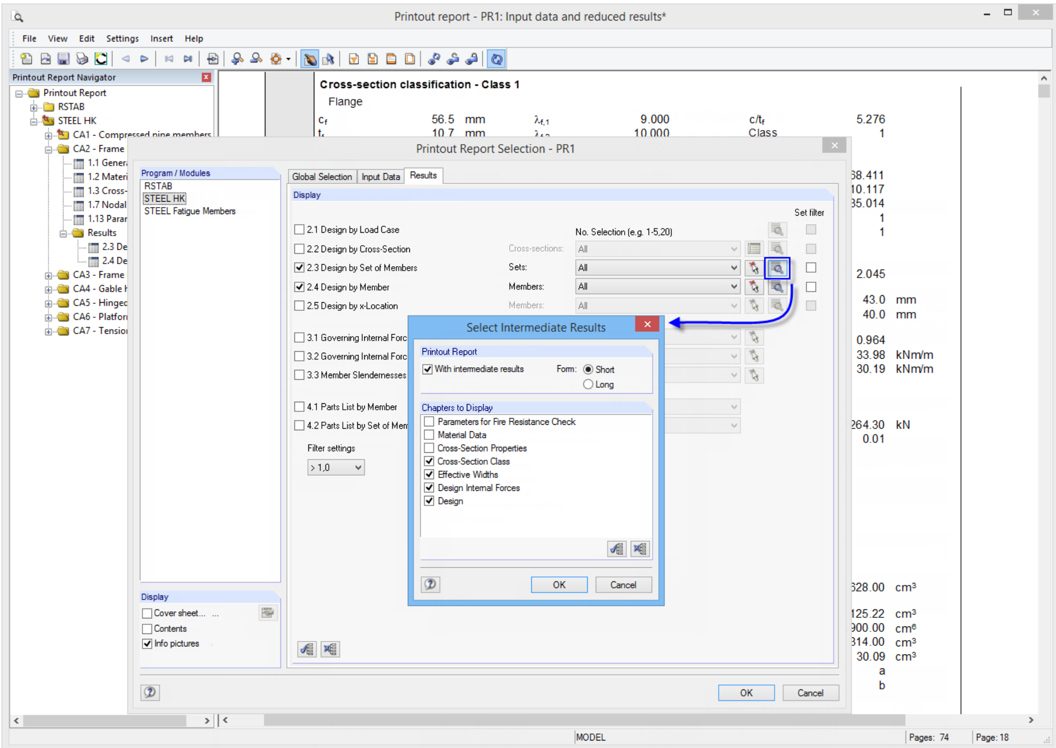


Figure 6.1: Selecting designs and intermediate results in the printout report

Click the [Details] button to specify if the printout also includes intermediate results. They can be defined in a list and documented in a *Short* (compact representation) or *Long* form (list representation).

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

6.2 Graphic Printout

In RFEM and RSTAB, you can transfer every image displayed in the work window to the printout report. It is also possible to send it directly to the printer. Thus, the design ratios displayed in the model can be prepared for the printout, too.

The printing of graphics is described in Chapter 10.2 of the RFEM or RSTAB manual.

Designs in RFEM/RSTAB model

To print the current graphic of design ratios, select on the menu

File → **Print Graphic**

or use the toolbar button shown on the left.

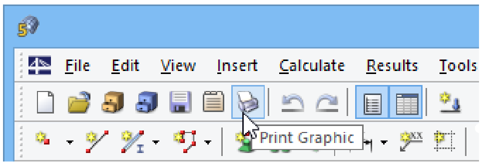


Figure 6.2: *Print Graphic* button in RFEM toolbar

Result diagrams

Also in the *Result Diagram on Member* dialog box, you can send the graphic with design values to the report by clicking the [Print] button. Alternatively, you can print it directly.

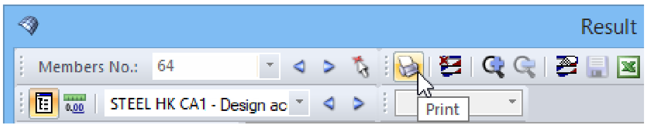


Figure 6.3: *Print* button in *Result Diagram on Member* dialog box

The following dialog box opens:

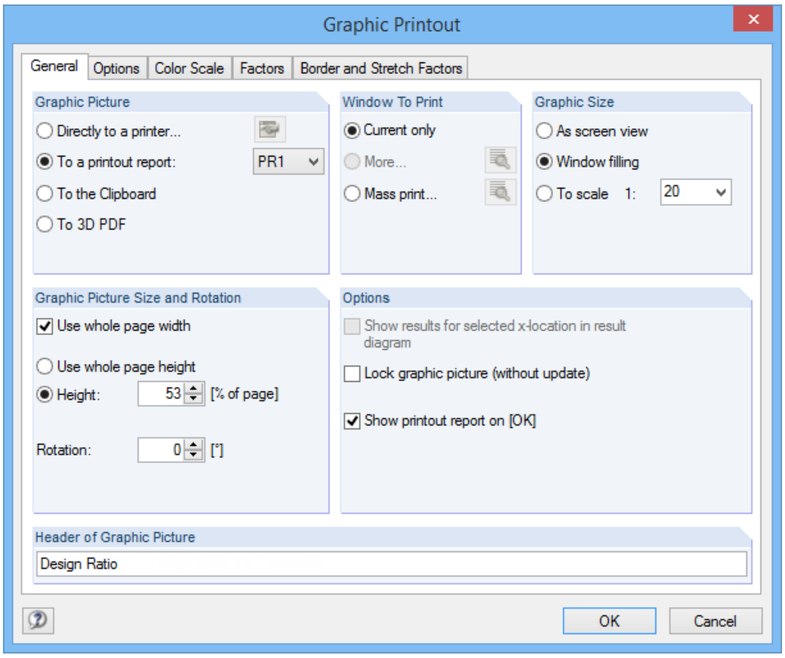
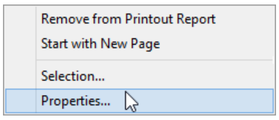


Figure 6.4: Dialog box *Graphic Printout*, tab *General*

The *Graphic Printout* dialog box is described in Chapter 10.2 of the RFEM or RSTAB manual. There, you find also descriptions of the other dialog tabs.



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

To adjust a graphic subsequently in the printout report, right-click the relevant entry in the report navigator. The *Properties* option in the shortcut menu again opens the *Graphic Printout* dialog box, offering various options for adjustment.

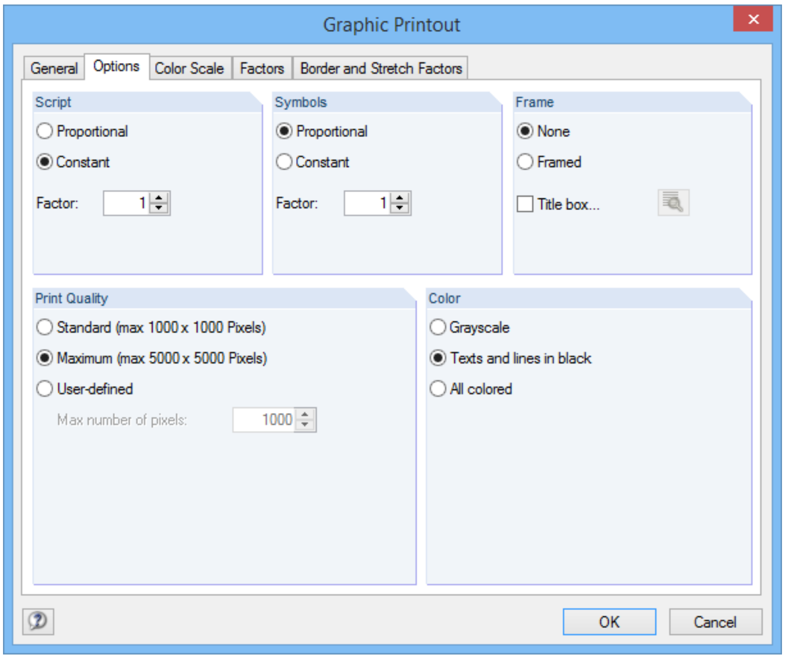


Figure 6.5: Dialog box *Graphic Printout*, tab *Options*

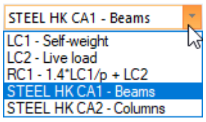
7 General Functions

This chapter describes useful menu functions as well as export options for the designs.

7.1 Design Cases

Design cases allow you to group members for the designs. This way, you can consider groups of structural components or analyze members with particular design specifications (for example, changed materials, partial safety factors, optimization).

It is no problem to analyze the same member or set of members in different design cases.



You can access the design cases of RF-/STEEL HK also in RFEM or RSTAB by using the load case list of the toolbar.

Create a new design case

To create a new design case, select on the RF-/STEEL HK menu

File → **New Case**.

The following dialog box appears.

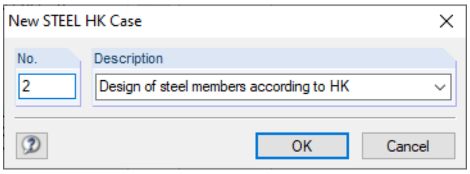


Figure 7.1: Dialog box *New STEEL HK Case*

In this dialog box, enter a *No.* (one that is not yet assigned) for the new design case. A *Description* will make the selection in the load case list easier.

After clicking [OK], the RF-/STEEL HK Window *1.1 General Data* opens for you to enter the design data.

Rename a design case

To change the description of a design case, select on the RF-/STEEL HK menu

File → **Rename Case**.

The following dialog box appears.

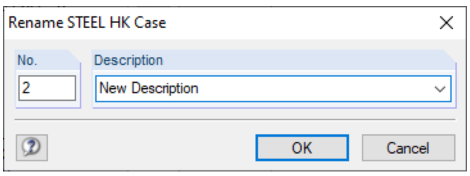


Figure 7.2: Dialog box *Rename STEEL HK Case*

In this dialog box, you can specify a different *Description* as well as a different *No.* for the design case.

Copy a design case

To copy the input data of the current design case, select on the RF-/STEEL HK menu

File → **Copy Case**.

The following dialog box appears.

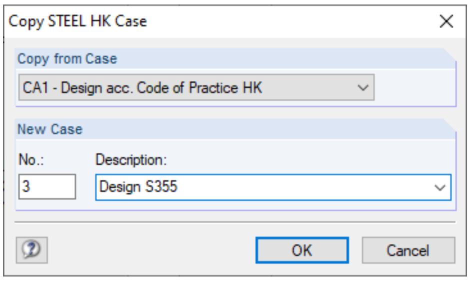


Figure 7.3: Dialog box *Copy STEEL HK Case*

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

Delete a design case

To delete a design case, select on the RF-/STEEL HK menu

File → **Delete Case**.

The following dialog box appears.

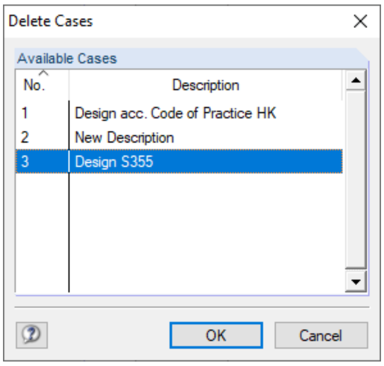
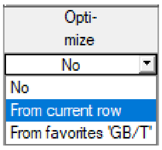


Figure 7.4: Dialog box *Delete Cases*

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

7.2 Cross-Section Optimization



The design module offers you the possibility to optimize overloaded or little utilized cross-sections: Define the relevant sections in Window *1.3 Cross-Sections* by opening the drop-down list in column E or F where you decide if the cross-sections are to be determined *From current row* or from user-defined *Favorites* (see Figure 2.9, page 13). You can also start the optimization in the result windows by using the shortcut menu.

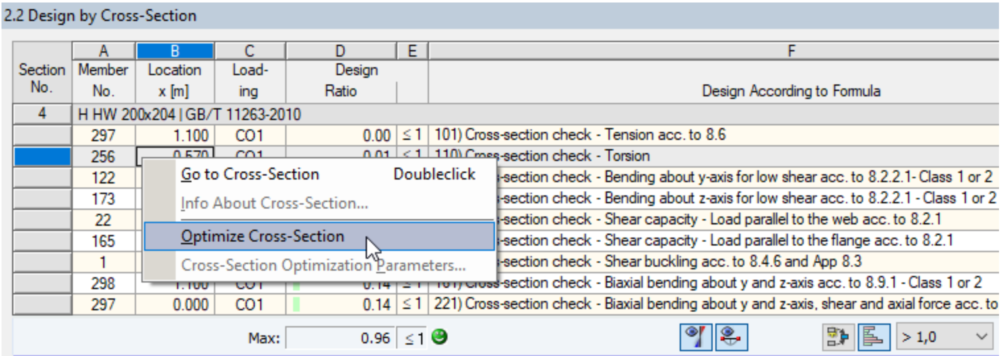


Figure 7.5: Shortcut menu for cross-section optimization

During the optimization process, RF-/STEEL HK determines the cross-section that fulfills the ultimate limit state design in the most "optimal" way, that is, it comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.4, page 30). The required cross-section properties are determined with the internal forces as available from RFEM or RSTAB. If another cross-section proves to be more favorable, it is used for the design. Then, the graphic in Window 1.3 shows two cross-sections – the original cross-section from RFEM or RSTAB and the optimized cross-section (see Figure 7.7).

If you select the *Optimize* option for a parametric cross-section, the following dialog box appears.

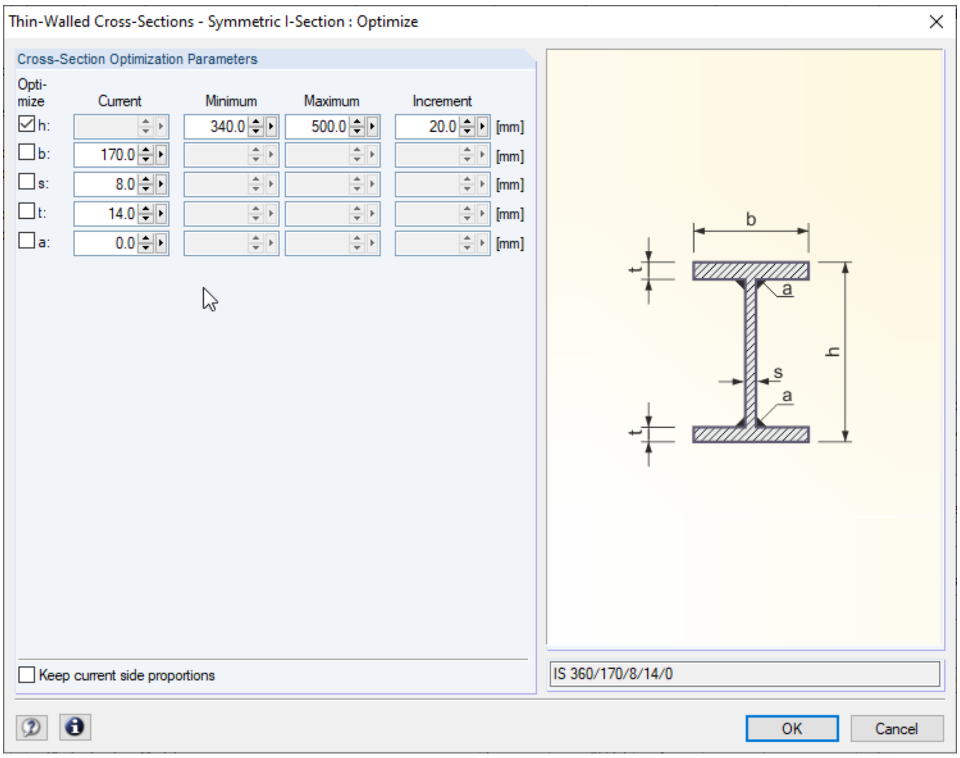


Figure 7.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section : Optimize*

You determine the parameter(s) that you want to modify by selecting the *Optimize* check box(es). This enables the *Minimum* and *Maximum* columns where you can define the upper and lower limits of the parameter. The *Increment* column controls the interval in which the size of the parameter varies during the optimization process.

If you want to *Keep current side proportions*, activate the corresponding check box. In addition, you have to select at least two parameters for optimization.

Cross-sections composed of rolled cross-sections cannot be optimized.

Please note that during the optimization the internal forces won't be automatically recalculated with the modified cross-sections: It is up to you to decide which cross-sections should be transferred to RFEM or RSTAB for recalculation. As a result of optimized cross-sections, the internal forces may differ significantly because of the modified stiffnesses in the structural system. Therefore, it is recommended to recalculate the internal forces with the modified cross-sections after the first optimization, and then to optimize the cross-sections once again.

You can export the modified cross-sections to RFEM or RSTAB: Go to Window *1.3 Cross-Sections* and select on the menu

Edit → Export All Cross-Sections to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.3 to export optimized cross-sections to RFEM or RSTAB.

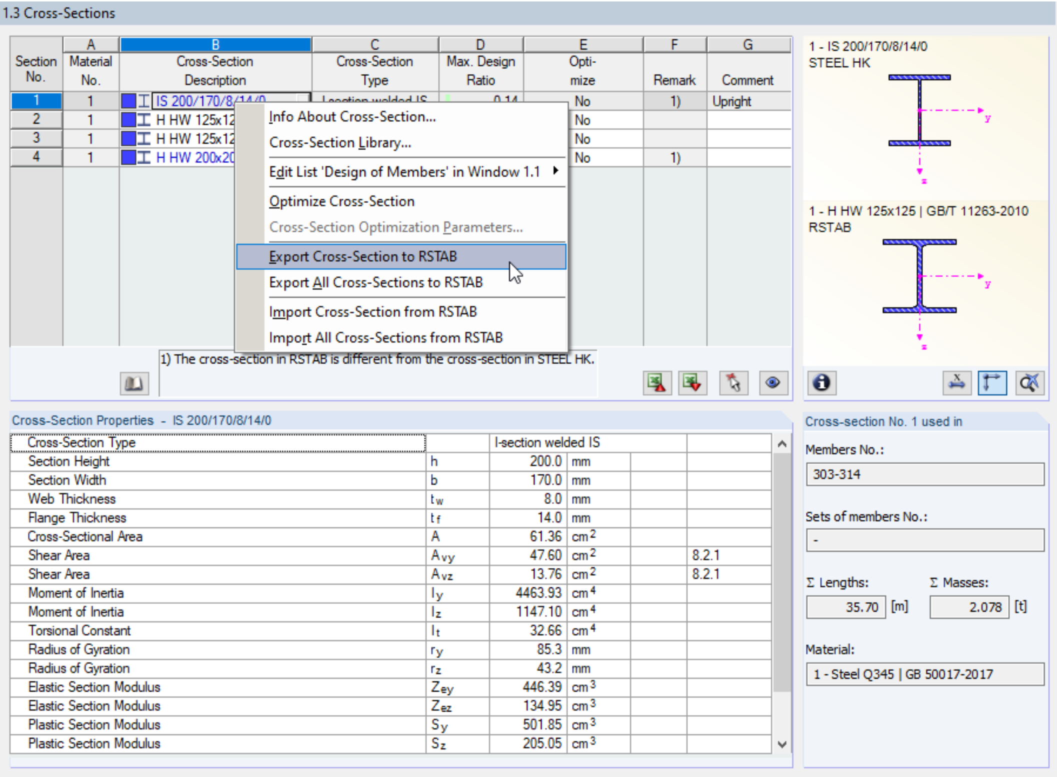


Figure 7.7: Shortcut menu in Window *1.3 Cross-Sections*

Before the modified cross-sections are transferred, a query appears asking if the results of RFEM or RSTAB should be deleted.

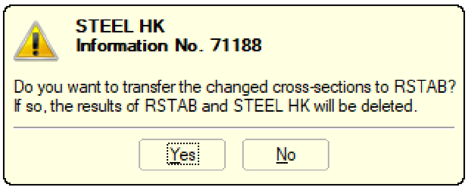


Figure 7.8: Query before transfer of modified cross-sections to RSTAB

Calculation

After starting the [Calculation] in RF-/STEEL HK, the internal forces and design ratios are determined in one calculation run.

If the modified cross-sections have not yet been exported to RFEM or RSTAB, you can reimport the original cross-sections to the design module by using the options shown in Figure 7.7. Please note that this possibility is only available in Window *1.3 Cross-sections*.

If you optimize a tapered member, the program modifies the member start and end and linearly interpolates the second moments of area for the intermediate locations. Since these moments are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-section differ considerably. In such a case, it is recommended to divide the taper into several members, thus modeling the taper layout manually.

7.3 Units and Decimal Places

The units and decimal places are managed for RFEM/RSTAB and the add-on modules in one dialog box. In RF-/STEEL HK, you can access this dialog box for adjusting the units by selecting on the menu

Settings → **Units and Decimal Places**

The dialog box known from RFEM or RSTAB appears. The RF-/STEEL HK add-on module is preset in the *Program / Module* list.

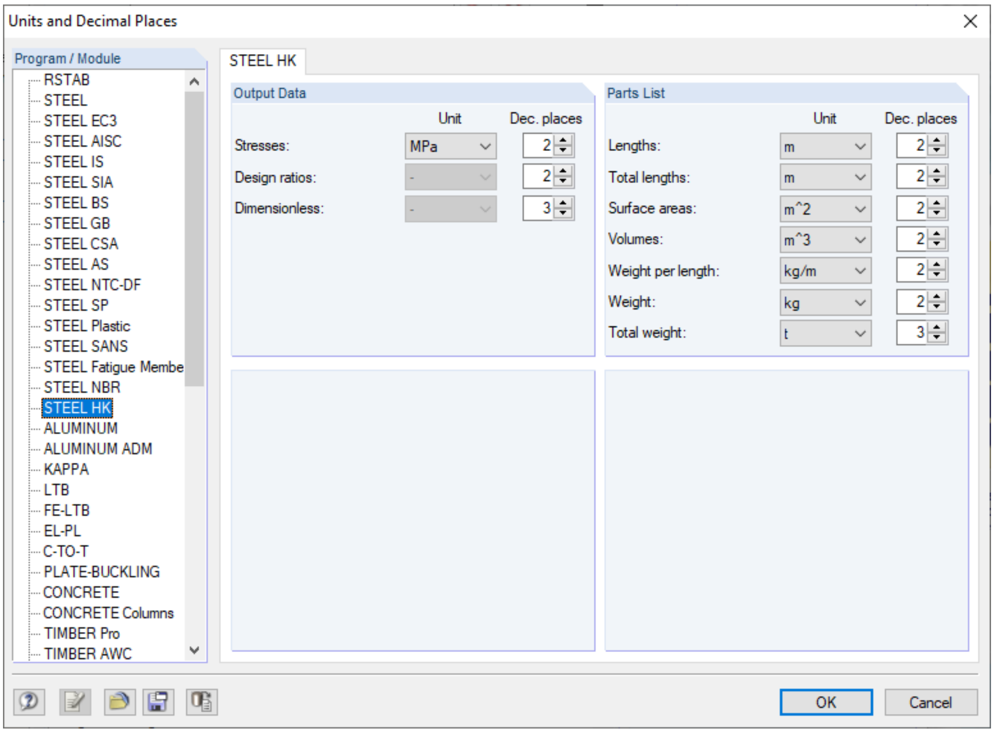


Figure 7.9: Dialog box *Units and Decimal Places*

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.

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

If the materials have been adjusted in RF-/STEEL HK for the design, you can export the modified materials to RFEM or RSTAB in a similar way as you export cross-sections: Open Window *1.2 Materials*, and then select on the menu

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

You can also use the shortcut menu in Window 1.2 to export materials to RFEM/RSTAB.

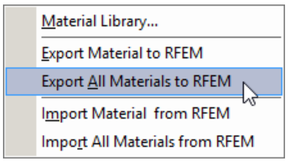
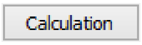


Figure 7.10: Shortcut menu of Window *1.2 Materials*



Before the modified cross-sections are transferred, a query appears asking if the results of RFEM or RSTAB should be deleted. After starting the [Calculation] in RF-/STEEL HK, the internal forces and design ratios are determined in one calculation run.

If the modified materials have not yet been exported to RFEM or RSTAB, you can reimport the original materials to the design module by using the options shown in Figure 7.10. Please note that this possibility is only available in Window *1.2 Materials*.

7.4.2 Export of Results

The RF-/STEEL HK results can also be used by other programs.

Clipboard

To copy cells selected in the results windows to the clipboard, use the keys [Ctrl]+[C]. To insert them, for example, in a word processing program, press [Ctrl]+[V]. The headers of the table columns won't be transferred.

Printout report

The data of RF-/STEEL HK can be printed into the printout report (see Chapter 6.1, page 50) where they can be exported. Then, in the printout report, select on the menu

File → **Export to RTF**.

This function is described in Chapter 10.1.11 of the RFEM or RSTAB manual.

Excel/CSV export

RF-/STEEL HK provides a function for directly exporting data to MS Excel or to the CSV file format. To access this function, select on the menu

File → **Export Tables**.

The following export dialog box opens.

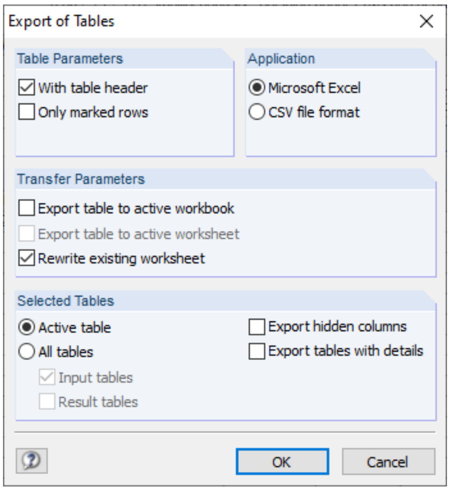


Figure 7.11: Dialog box *Export of Tables*

When you have selected the relevant data, you can start the export with [OK]. Excel will be started automatically, that is, you do not need to open the programs first.

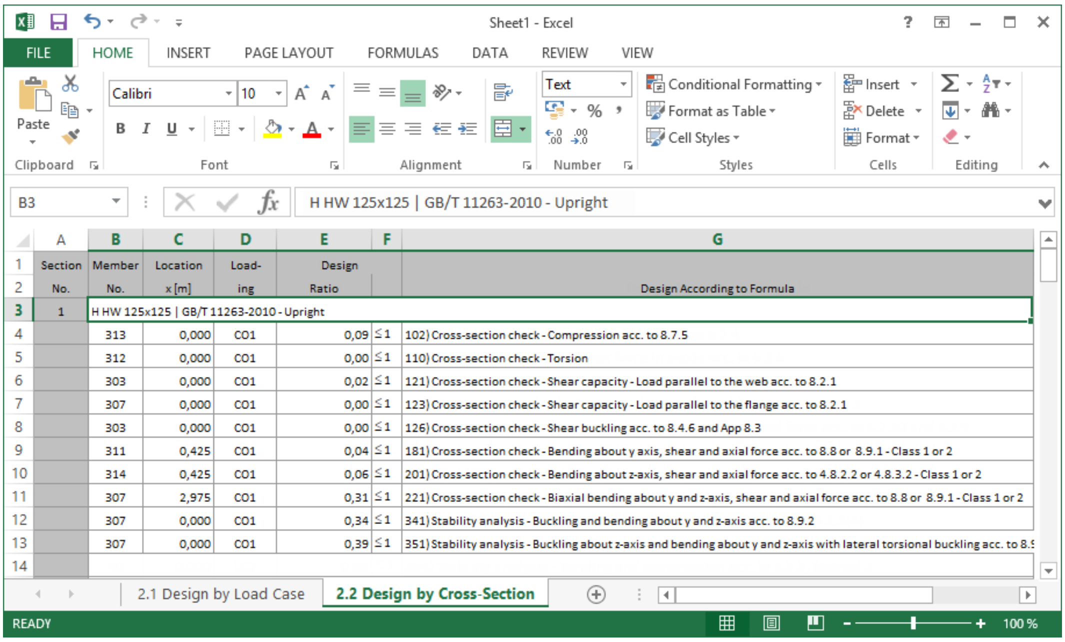


Figure 7.12: Results in *Excel*

Literature

[1] *Code of Practice for the Structural Use of Steel*. Building Department, Hong Kong, 2011.

Index