Ing. Software
Dlubal

Version
September 2011

Add-on Module

RF-TIMBER Pro

Design of Timber Members According to DIN 1052, EN 1995 and SIA 265

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Add-on Module RF-TIMBER Pro

NAs according to EC 5

RF-TIMBER Pro is one of the add-on modules integrated in the RFEM user interface. Use the program to perform the design of timber members and sets of members, based on the design concepts of DIN 1052:2008, Eurocode 5 (EN 1995-1-1:2010-12 + A1:2008) or SIA 265:2003. Country-specific regulations concerning Eurocode 5 are taken into account by national annexes (NAs). You can take advantage of the national documents that are already implemented, but you can also define your own limit values or create new NAs. The list shown on the left includes the NAs available for EN 1995-1-1 and is constantly being expanded.

RF-TIMBER Pro performs all typical ultimate limit state designs as well as stability and deformation analyses. The stability analysis is carried out according to the equivalent member method or the second-order analysis. When the equivalent member method is applied, the program considers intended axial compression along the grain, bending without compression force, bending and compression, shear from transverse force as well as bending and tension. Fire resistance is designed according to DIN 4102-4, EN 1995-1-2 or SIA 265.

In timber construction the serviceability limit state represents an important factor for structural calculations. Therefore, you can assign load cases, groups and combinations individually to various design situations. The limit deformations are preset, for example by the national annex, and can be adjusted, if necessary. In addition, it is possible to specify reference lengths and cambers that will be considered accordingly in the design.

As RF-TIMBER Pro like other add-on modules is completely integrated in RFEM, the design-relevant input data is already preset when you have started the program. Subsequent to the design, you can use the graphical RFEM user interface to evaluate the results. The entire design process is presented properly and in a consistent form in the global printout report.

Finally, RF-TIMBER Pro offers you an automatic cross-section optimization including export option for transferring modified cross-sections to RFEM. Separate design cases allow for a flexible analysis of several structural components in complex structures.

We hope you will enjoy working with RF-TIMBER Pro.

Your team from ING. SOFTWARE DLUBAL

1.2 RF-TIMBER Pro - Team

The following people were involved in the development of RF-TIMBER Pro:

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) René Flori
Ing. Jiří Hanzálek
Dipl.-Ing. (FH) Bastian Kuhn

Programming

Ing. Zdeněk Kosáček
Dipl.-Ing. Georg Dlubal
Ing. Roman Svoboda
DiS. Jiří Šmerák

Cross-section and material database

Ing. Ph.D. Jan Rybín
Jan Brnušák

Program design, dialog figures and icons

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

Program supervision

Ing. Jiří Hanzálek
Zdenek Kodera
Dipl.-Ing. (FH) Bastian Kuhn

Manual, help system and translation

Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Robert Vogl
Dipl.- Ü. Gundel Pietzcker
Mgr. Petra Pokorná
Jan Jeřábek

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Bastian Kuhn
M.Sc. Dipl.-Ing. (FH) Frank Lobisch
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

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

The descriptions in this manual follow the sequence of the module's input and results tables as well as their structure. The text of the manual shows the described **buttons** in square brackets, for example [Apply]. At the same time, they are pictured on the left. **Expressions** appearing in dialog boxes, tables and menus are set in *italics* to clarify the explanations.

At the end of the manual, you find the index. However, if you don't find what you are looking for, please check our website **www.dlubal.com** where you can go through our *FAQ* pages by selecting particular criteria.

1.4 Open the Add-on Module RF-TIMBER Pro

RFEM provides the following options to start the add-on module RF-TIMBER Pro.

Menu

To start the program in the menu bar,

point to **Design - Timber** on the **Additional Modules** menu, and then select **RF-TIMBER Pro**.

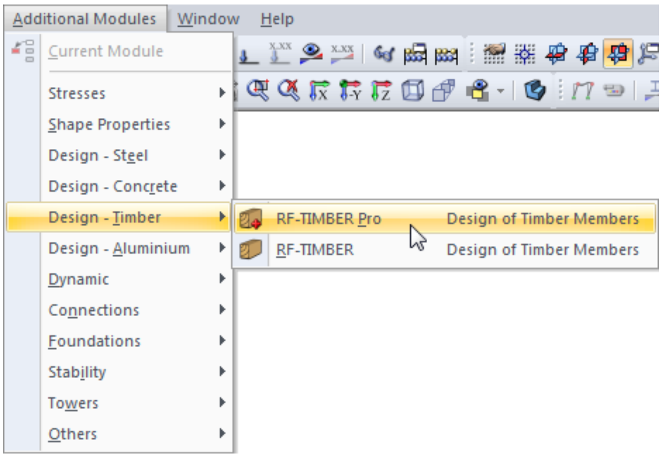


Figure 1.1: Menu *Additional Modules* → *Design - Timber* → *RF-TIMBER Pro*

Navigator

To start RF-TIMBER Pro in the *Data* navigator,

select **RF-TIMBER Pro** in the **Additional Modules** folder.

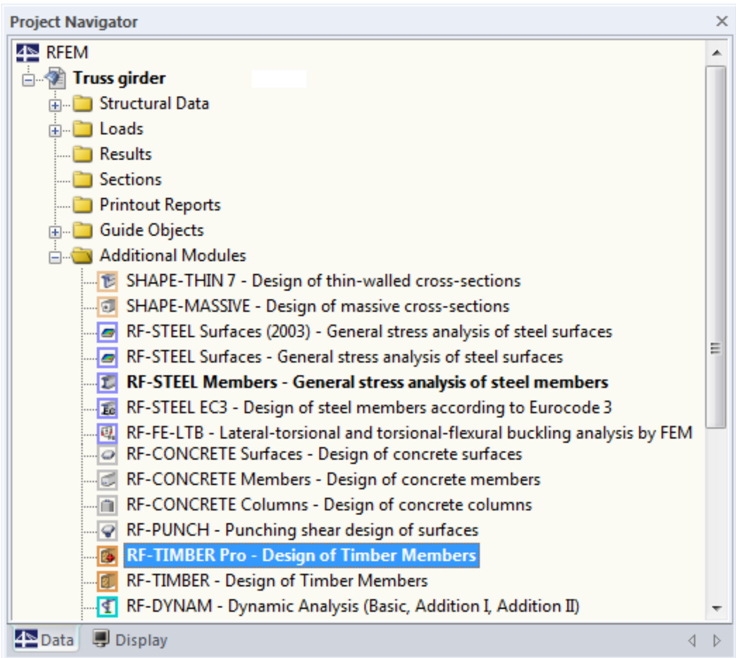
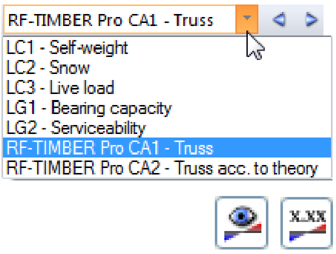


Figure 1.2: Data navigator: *Additional Modules* → *RF-TIMBER Pro*

Panel



In case RF-TIMBER Pro results are already available in the RFEM structure, set the relevant RF-TIMBER Pro design case in the load case list of the RFEM menu bar. Use the button [Results on/off] to display the design criteria on the members graphically.

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

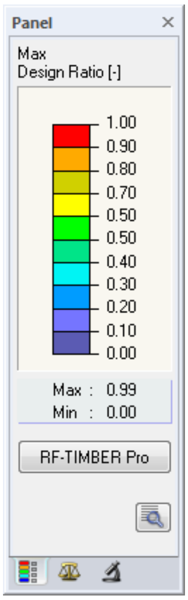


Figure 1.3: Panel button [RF-TIMBER Pro]

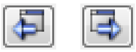
2. Input Data

The design cases are defined in several input tables. For members and sets of members you can use the [Pick] function to select them graphically.

When you have started RF-TIMBER Pro, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently. The pull-down list above the navigator contains the design cases that are already available (see chapter 7.1, page 54).

If you open RF-TIMBER Pro in an RFEM structure for the first time, the module imports the following design relevant data automatically:

- Members and sets of members
- Load cases (LC), load groups (LG) and load combinations (CO)
- Materials
- Cross-sections
- Internal forces (in background, if calculated)



To select a table, click the corresponding entry in the RF-TIMBER Pro navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.

Click [OK] to save the entered data and quit the add-on module RF-TIMBER Pro. When you click [Cancel], you quit the module but without saving the data.

2.1 General Data

In table 1.1 *General Data*, you select the members, sets of members and actions that you want to design. The tabs are managing the load cases, groups and combinations for the different designs.

2.1.1 Ultimate Limit State

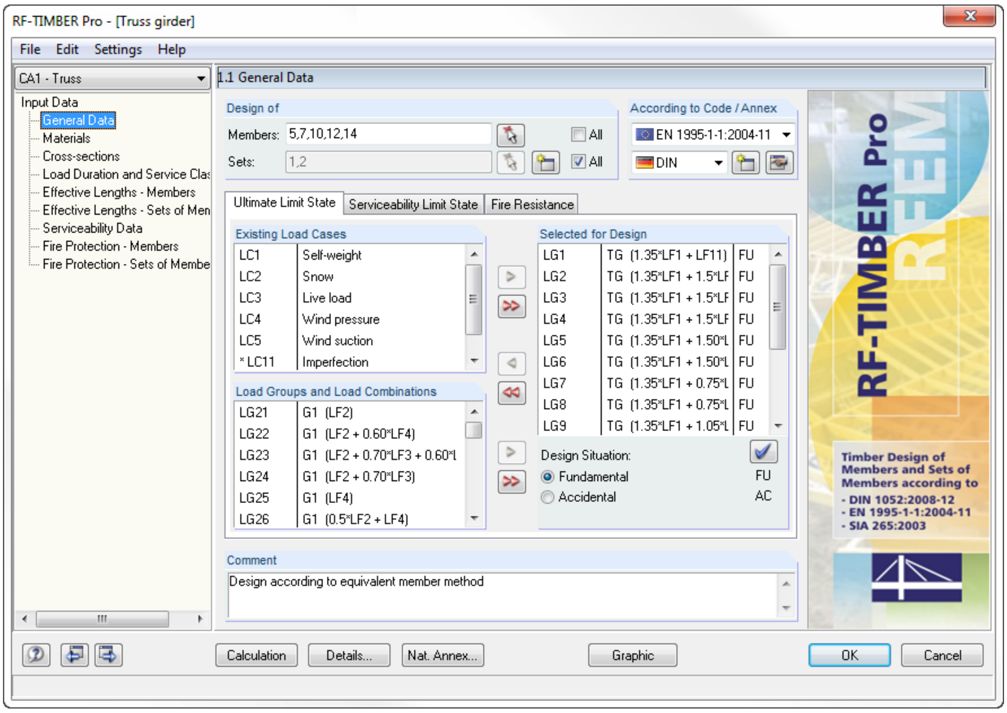


Figure 2.1: Table 1.1 *General Data*, tab *Ultimate Limit State*

Design of

The design can be carried out for *Members* as well as for *Sets* of members. If you want to design only selected objects, clear the *All* check boxes. Then you can access the two input fields to enter the numbers of the relevant members or sets of members. To select the objects graphically in the RFEM work window, use the [Pick] button. The list of the preset member numbers can be selected quickly by double click and overwritten by entering the data manually.

In case no set of members has yet been defined in RFEM, you can create a new set of members in RF-TIMBER Pro by using the [New] button. The dialog box that you already know from RFEM appears where you can specify the data for a new set of members.

When you design a set of members, the program determines the extreme values of the designs of all members contained in the set of members and takes into account the boundary conditions of connected members for the stability analysis. Subsequent to the calculation, the additional results tables 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* will be displayed.

According to Code / Annex

In the selection field on the top right, you specify the standard by which you want to perform the design. The following codes can be selected:

- DIN 1052:2008-12
- EN 1995-1:2004-11
- SIA 265:2003

When you select the European standard Eurocode 5, you also have to specify the national annex whose parameters will be used for the design and the deformation's limit values.

Use the [Edit] button to open a dialog box where you can check and adjust, if necessary, the parameters of the standard or the currently selected NA. The dialog box is described in chapter 2.1.4 on page 12.

Existing Load Cases / Load Groups and Load Combinations

These two dialog sections list all load cases, groups and combinations defined in RFEM that are relevant for the design. Use the button [▶] to transfer selected load cases, groups or combinations to the list *Selected for Design* on the right. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].

When load cases or combinations are marked by an asterisk (*) like load case 11 in Figure 2.1, they cannot be calculated. Those load cases are cases without load data or pure imperfection cases.

Please note that only load combinations for which the program can determine clear minima and maxima (which means alternative combinations with the superposition criterion *Permanent*) are allowed for the design. When you select an unacceptable load combination, the following error message will be displayed:

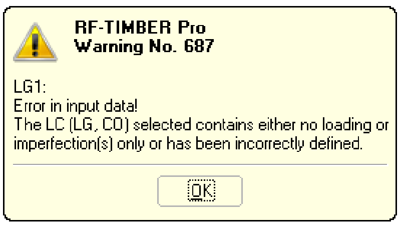


Figure 2.2: Warning appearing when an unallowable load combination is selected

A multiple selection of load cases is possible by means of the common Windows function holding down the keyboard key [Ctrl]. In this way, you can transfer several load cases simultaneously.

Selected for Design

The column on the right lists the loads selected for the design. Use the button [◀] to remove selected load cases, groups or combinations from the list. You can also double-click the entries. With the button [◀◀], you can transfer the entire list to the left.

The analysis of an enveloping *Or* load combination is often carried out more quickly than the design of all load cases and groups that have been globally set. However, please be aware of the restriction mentioned above: The load combination must contain only load cases, groups or combinations that are superimposed with the *Permanent* criterion. Furthermore, when you design a load combination, please keep in mind that it is difficult to see the effect of the actions contained in a CO.

2.1.2 Serviceability Limit State

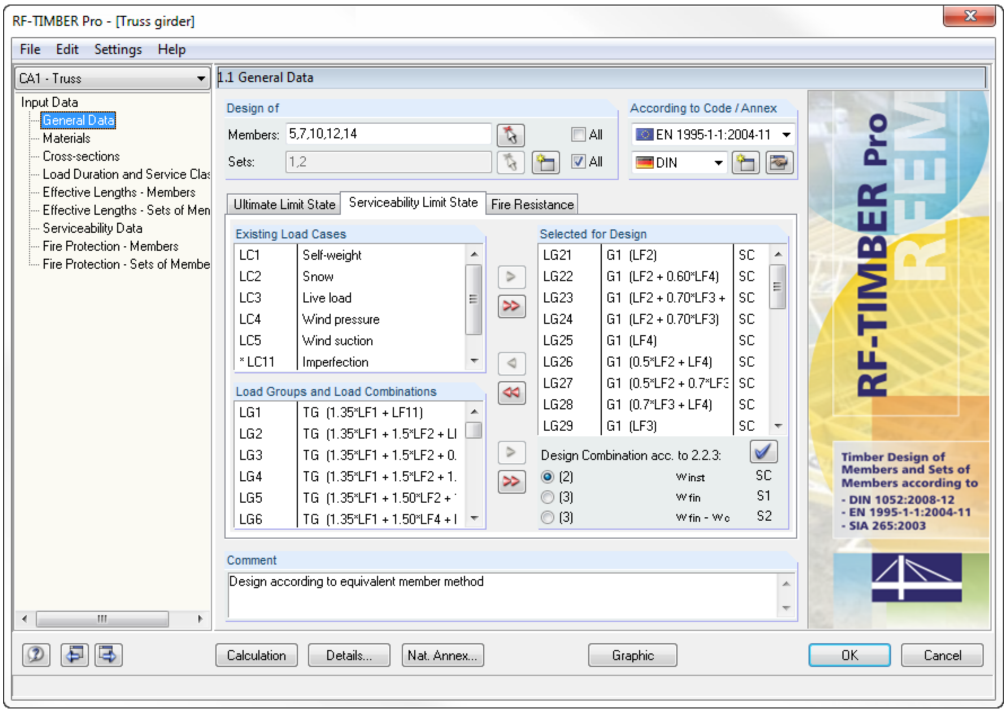


Figure 2.3: Table 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases / Load Groups and Load Combinations

In these two sections, all load cases, load groups and load combinations are listed that have been created in RFEM.

Selected for Design

Adding and removing load cases, load groups and load combinations is described in the previous chapter 2.1.1.

Design Combination

It is possible to assign different limit values for the deflection to the individual load cases, groups and combinations. First, select the relevant action in table *Selected for Design*. To assign the selected design combination, click the blue checkmark button [☑]. The following criteria are available for EN 1995-1-1:

- w_{inst} for a permanent action (SC)
- w_{fin} for a leading variable action (S1)
- w_{fin} - w_c for a leading variable action with camber (S2)

The limit values of the deformations are specified in the standards or NAs and can be adjusted for the different design combinations in the corresponding parameter settings (see Figure 2.5, page 13).

The reference lengths that are decisive for the serviceability limit state design are managed in table 1.8 (see chapter 2.8, page 27).

Comment

In this input field, you can enter user-defined notes describing for example the action combinations of the current design case.

2.1.3 Fire Resistance

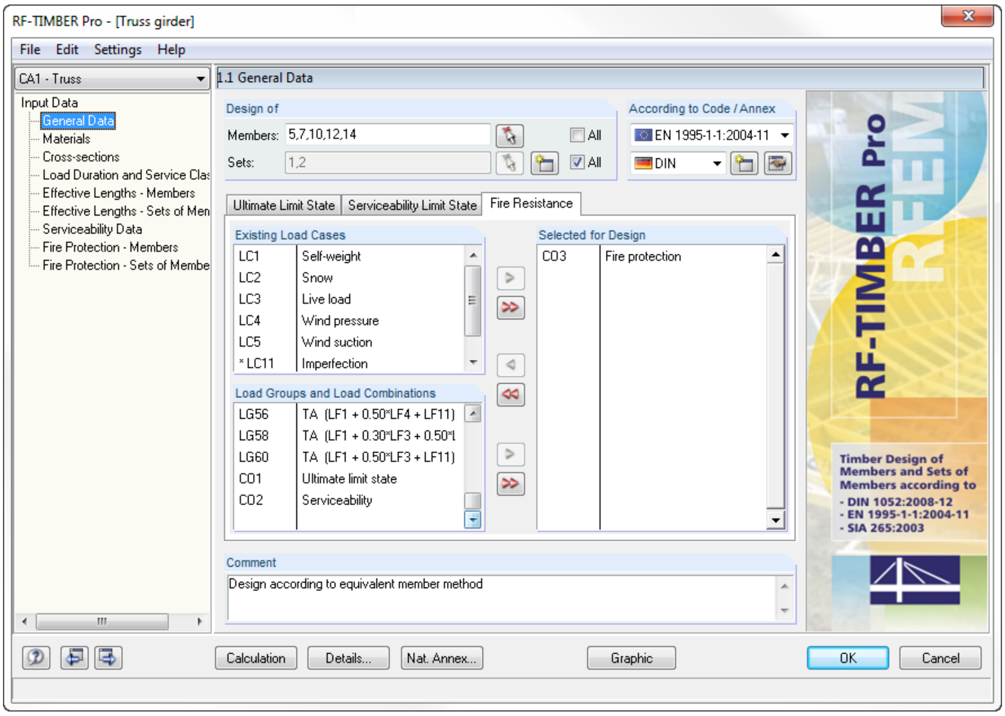


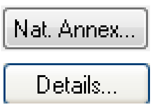
Figure 2.4: Table 1.1 *General Data*, tab *Fire Resistance*

Existing Load Cases / Load Groups and Load Combinations

In these two sections, all load cases, load groups and load combinations are listed that have been created in RFEM. Thus, it is required for the fire protection design that the superposition in an appropriate combination must have been done already in RFEM.

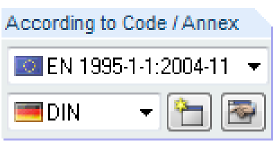
Selected for Design

Adding and removing load cases, load groups and load combinations is described in chapter 2.1.1. The load selection must include the actions and combinations that have been created in RFEM specifically for the fire protection design.



The fire protection design is performed by means of a reduced cross-section. In the dialog box *National Annex Settings* (see Figure 2.5) as well as in the *Details* dialog box, tab *Fire Resistance* (see Figure 3.4, page 32), you can define detailed specifications for the fire resistance design.

2.1.4 Standard Parameters



With the lists of the selection fields in the upper right corner of table 1.1 *General Data*, you can define the standard or NA whose parameters you want to apply to the design and the limit values of the deformation.

Use the [Edit] button to check the preset parameters of the active code or NA. If necessary, you can adjust the settings (see Figure 2.5).

To create a user-defined national annex, use the [New] button.

The buttons in the bottom left corner of the NA *Settings* dialog box allow you to save modified values as default setting. Furthermore, you can use the buttons to import saved parameters or to restore the program presettings.

A user-defined national annex can be deleted by using the [Delete] button.

Moreover, all input tables provide the button [Code] or [Nat. Annex] which you can use any time to access the *Settings* dialog box.

A three-tab dialog box opens.

General

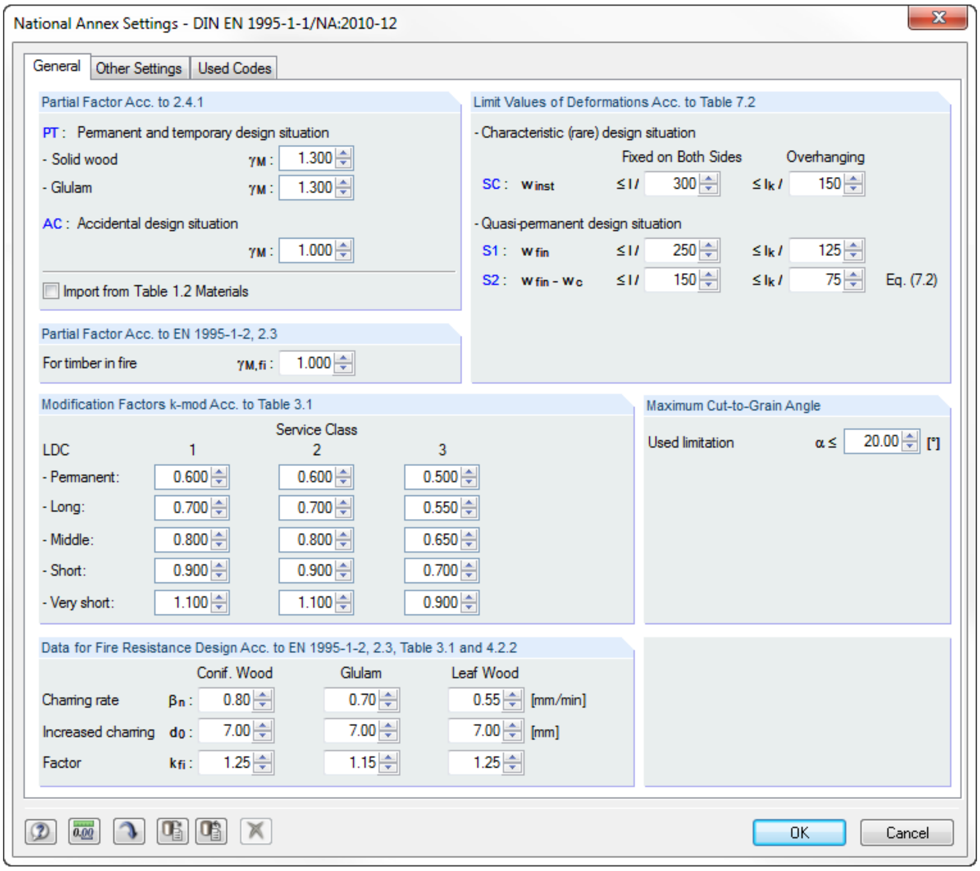


Figure 2.5: Dialog box *National Annex Settings*, tab *General*

Partial and modification factors, limit values of deformations

The dialog sections above allow you to check and, if necessary, to adjust the *Partial Factors* and *Limit Values of Deformations* as well as the *Modification Factors* k_{mod} for the different design situations.

In addition, it is possible to change the *Maximum Cut-to-Grain Angle*, if required.

Data for fire protection

The design for the event of fire is performed with an ideal remaining cross-section. The parameters *Charring Rate* β_n and *Increased Charring* d_0 used to determine the remaining cross-section are predefined for the material types coniferous, glued-laminated and leafy timber but can be adjusted, if necessary.

The *Factor* k_{fi} is used for the determination of the 20%-quantile of the strength and stiffness from the 5%-quantile.

Ing. Software Dlubal

Other Settings

In the second tab of the dialog box *National Annex Settings* you find different factors significant for the design.

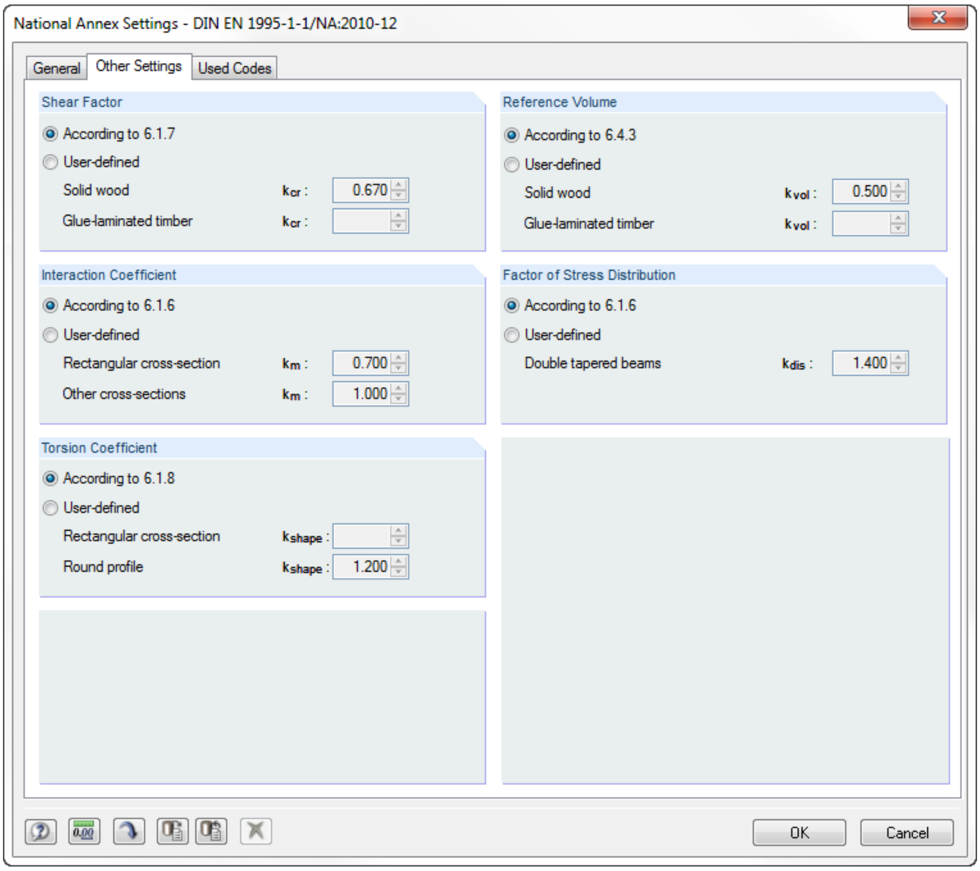
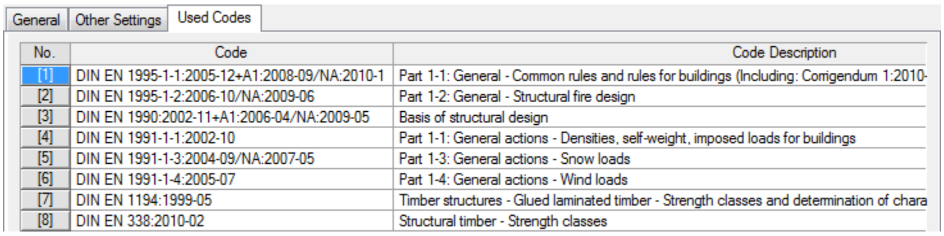


Figure 2.6: Dialog box *National Annex Settings*, tab *Other Settings*

Used Codes

The third tab of the dialog box *National Annex Settings* informs you about the codes according to which the design will be performed.



General | Other Settings | Used Codes

No.	Code	Code Description
[1]	DIN EN 1995-1-1:2005-12+A1:2008-09/NA:2010-1	Part 1-1: General - Common rules and rules for buildings (Including: Corrigendum 1:2010-
[2]	DIN EN 1995-1-2:2006-10/NA:2009-06	Part 1-2: General - Structural fire design
[3]	DIN EN 1990:2002-11+A1:2006-04/NA:2009-05	Basis of structural design
[4]	DIN EN 1991-1-1:2002-10	Part 1-1: General actions - Densities, self-weight, imposed loads for buildings
[5]	DIN EN 1991-1-3:2004-09/NA:2007-05	Part 1-3: General actions - Snow loads
[6]	DIN EN 1991-1-4:2005-07	Part 1-4: General actions - Wind loads
[7]	DIN EN 1194:1999-05	Timber structures - Glued laminated timber - Strength classes and determination of chara
[8]	DIN EN 338:2010-02	Structural timber - Strength classes

Figure 2.7: Dialog box *National Annex Settings*, tab *Used Codes*

2.2 Materials

The table is subdivided into two parts. In the upper part, the materials used for the design are listed. In the *Material Properties* section below, the properties of the current material, i.e. the table row currently selected in the upper section, are displayed.

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

The material properties required for the determination of internal forces in RFEM are described in detail in chapter 5.3 of the RFEM manual. The design relevant material properties are stored in the global material library and preset automatically.

To change the units and decimal places of material properties and stresses, select **Units and Decimal Places** on the module's **Settings** menu (see chapter 7.4, page 59).

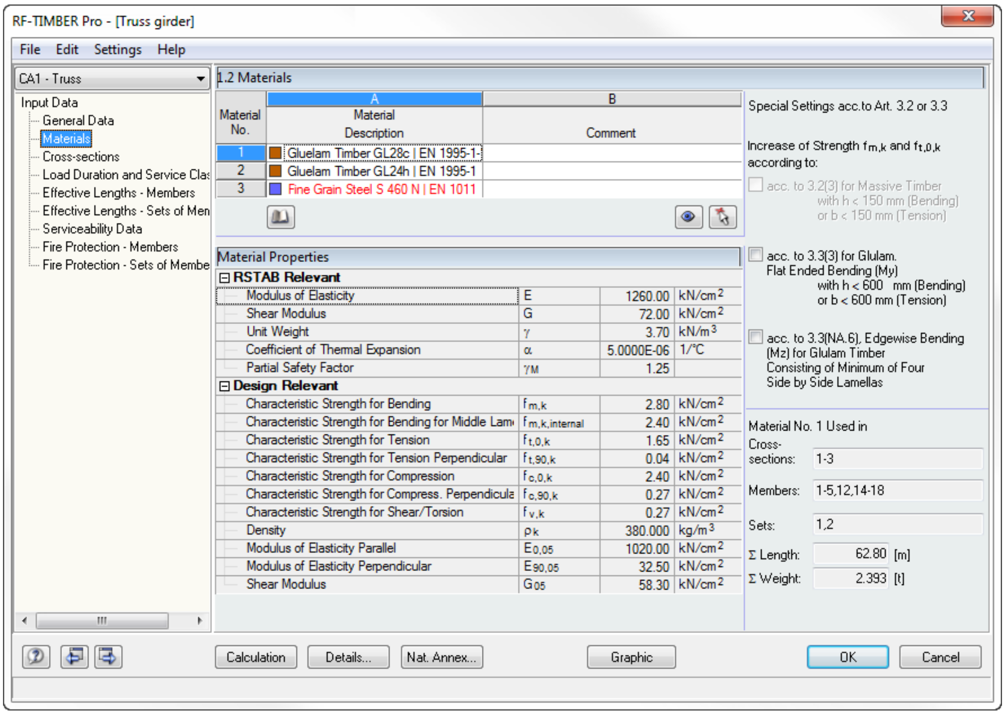
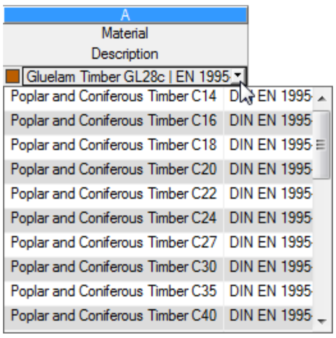


Figure 2.8: Table 1.2 *Materials*

Material Description



The materials defined in RFEM are already preset. If you modify the *Material Description* and the manually specified entry is listed in the material library, RF-TIMBER Pro will import the corresponding material properties.

You can modify the material by using the list: Place the cursor in a table row of column A, and then click the button [▼] or use the function key [F7] to open the list shown on the left. Subsequent to the selection, the program will update the design-relevant properties.

In accordance with the design concept of the timber codes, the list includes only materials of the **Timber** category. The import of materials from the library is described below.

Basically, it is not possible to edit the material properties in RF-TIMBER Pro. Therefore, if you want to apply modified stiffnesses, you have to define a new material with the relevant characteristic values by entering new data in the material library (see chapter 5.3 in the RFEM manual).

Material Properties

The partial safety factor γ_M represents the safety factor used to calculate the design values of the material stiffnesses. By means of this factor the program reduces the characteristic strength value for bending $f_{m,k}$, tension parallel $f_{t,0,k}$, tension perpendicular $f_{t,90,k}$, compression parallel $f_{c,0,k}$, compression perpendicular $f_{c,90,k}$ as well as for shear and torsion $f_{v,k}$ according to Equation 2.1.

The factor γ_M, as described for example in EN 1995-1-1, eq. (2.14), must be taken into account for the ultimate limit state design.

$$X_d = k_{mod} \cdot \frac{X_k}{\gamma_M}$$

Equation 2.1: Reduction of characteristic strength by modification and partial safety factors

Nat. Annex...

The partial safety factor can be adjusted in the *Settings* dialog box, tab *General* (see Figure 2.5, page 13).

Material Library

Numerous materials are available in the library. To open the library,

select **Material Library** on the **Edit** menu

or use the button shown on the left.

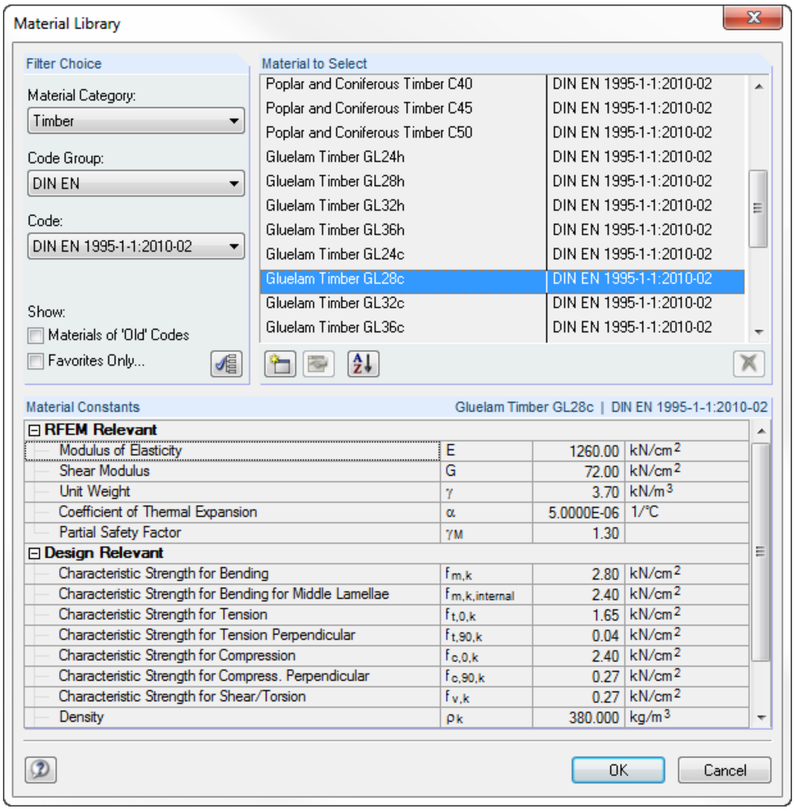


Figure 2.9: Dialog box *Material Library*

In the *Filter Choice* section, *Timber* is preset as material category. Select the timber grade that you want to use for the design in the list *Material to Select*. The corresponding properties can be checked in the dialog section below.

OK

Click [OK] or use the [↵] button to import the selected material to table 1.2 of RF-TIMBER Pro.

Chapter 5.3 of the RFEM manual describes in detail how materials can be filtered, added or rearranged.

A member whose material neither belongs to the "timber" category nor corresponds to the selected standard cannot be designed. A corresponding message will be displayed when you start the calculation.

2.3 Cross-sections

This table lists the cross-sections that are relevant for the design. In addition, the table allows for the definition of optimization parameters.

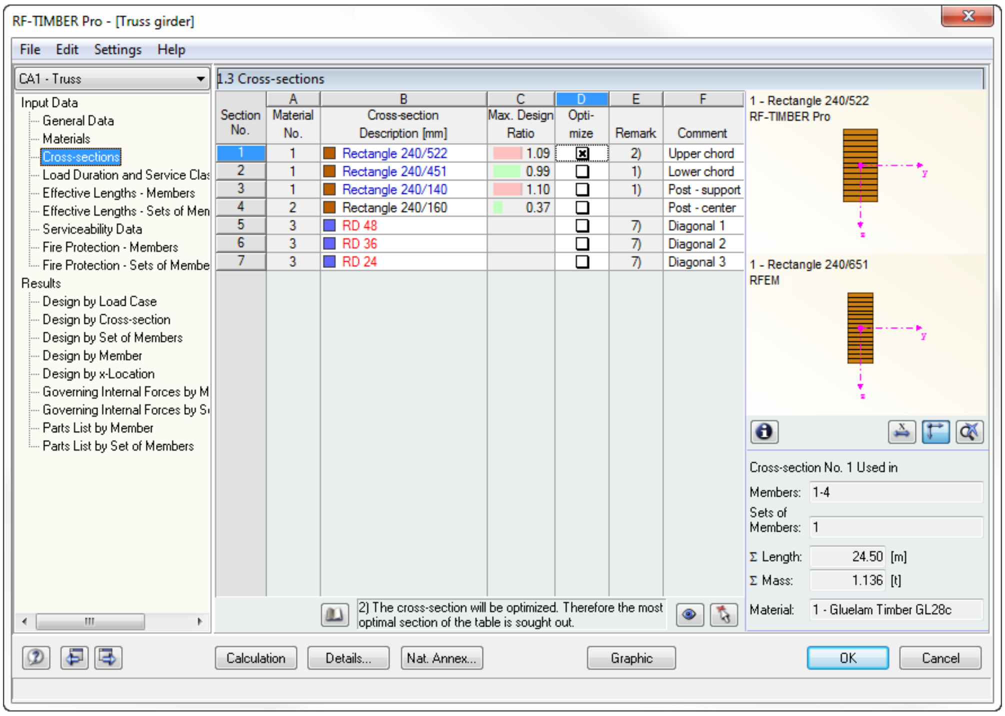


Figure 2.10: Table 1.3 *Cross-sections*

Cross-section Description

The cross-sections used in RFEM are preset together with the assigned material numbers. Currently, the RF-TIMBER Pro design is restricted to rectangular and circular cross-sections of the category *Solid Sections*.

It is always possible to modify the cross-sections for the design. The description of a modified cross-section is highlighted in blue.

To modify a cross-section, enter the new cross-section description directly into the corresponding table row. You can also select a new cross-section from the library. To open the library, use the button [Import Cross-section from Library] below the table. Alternatively, place the pointer in the respective table row and click the [...] button, or use the function key [F7]. The RFEM cross-section library appears, or better to say the cross-section table corresponding to the entry in the input field.

The selection of cross-sections from the library is described in detail in chapter 5.13 of the RFEM manual.

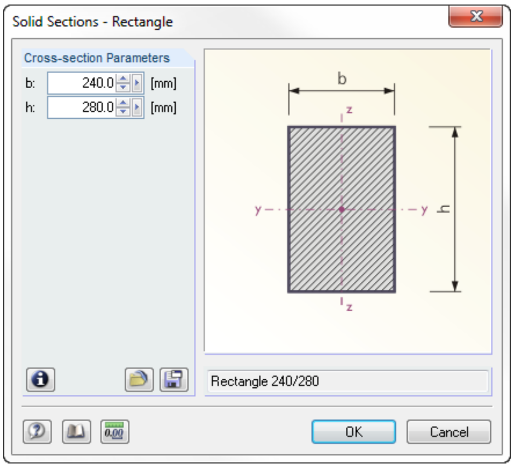
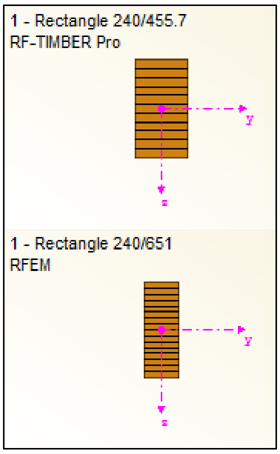


Figure 2.11: Dialog box *Solid Sections - Rectangle*



If the cross-sections set in RF-TIMBER Pro are different from the ones used in RFEM, both cross-sections are displayed in the graphic in the right part of the table. The designs for the cross-section selected in RF-TIMBER Pro will be performed with the internal forces of RFEM.

Member with tapered cross-section

For tapered members with different cross-sections at the member start and member end, the module displays both cross-section numbers, in accordance with the definition in RFEM, in two table rows.

RF-TIMBER Pro is also able to design tapered members provided that the same cross-section type is defined for the start and the end cross-section. A taper design requires additional specifications in table 1.7 (see chapter 2.7, page 26).

Info about cross-section

The cross-section properties and stress points including numbering can be accessed in a separate dialog box: Select the relevant cross-section in table 1.3, and then click the [Info] button to open the dialog box *Info about cross-section*.

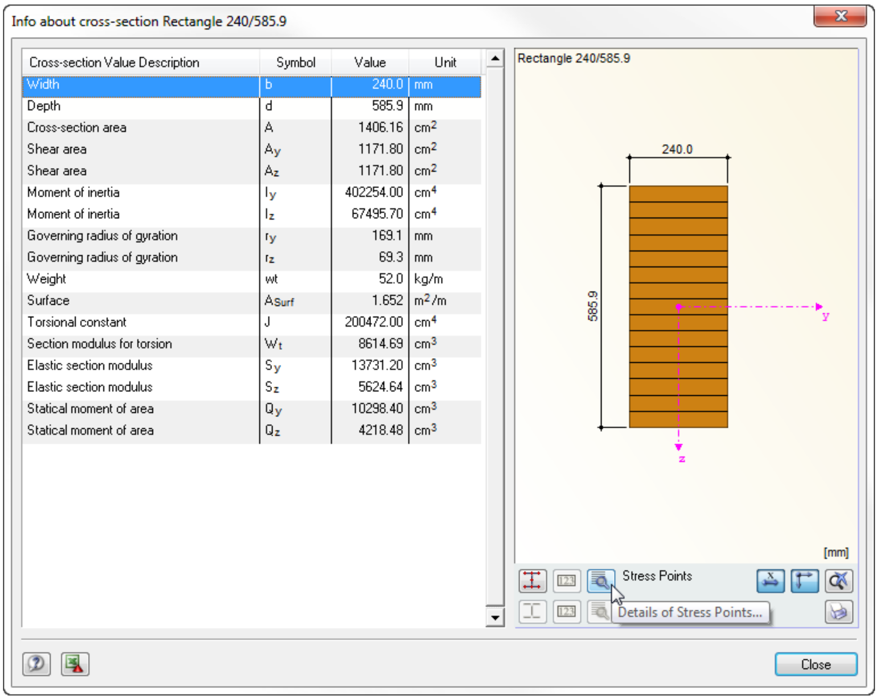


Figure 2.12: Dialog box *Info about cross-section*

In the right part of the dialog box, the currently selected cross-section is displayed. The buttons below the graphic are reserved for the following functions:

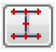
Button	Function
	Displays or hides the stress points.
	Displays or hides the numbering of stress points.
	Displays or hides the details of stress points.
	Displays or hides the dimensions of the cross-section.
	Displays or hides the principal axes of the cross-section.
	Resets the full view of the cross-section graphic.

Table 2.1: Buttons of cross-section graphic

Max. Design Ratio

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

Optimize

Each cross-section of the library can be improved by an optimization process. Using the internal forces from RFEM, the program determines the cross-section of the cross-section table that comes as close as possible to the maximum stress ratio specified in the *Other* tab of the *Details* dialog box (see Figure 3.5, page 33).

If you want to optimize a cross-section, tick the corresponding check box in column C or D. Recommendations for optimizing cross-sections can be found in chapter 7.2 on page 56.

Remark

This column shows remarks in the form of footers that are described in detail below the cross-section list.

2.4 Load Duration and Service Class

The load duration is indicated in table 1.4. In addition, it is possible to assign structural objects to different service classes to consider different climatic conditions for the design.

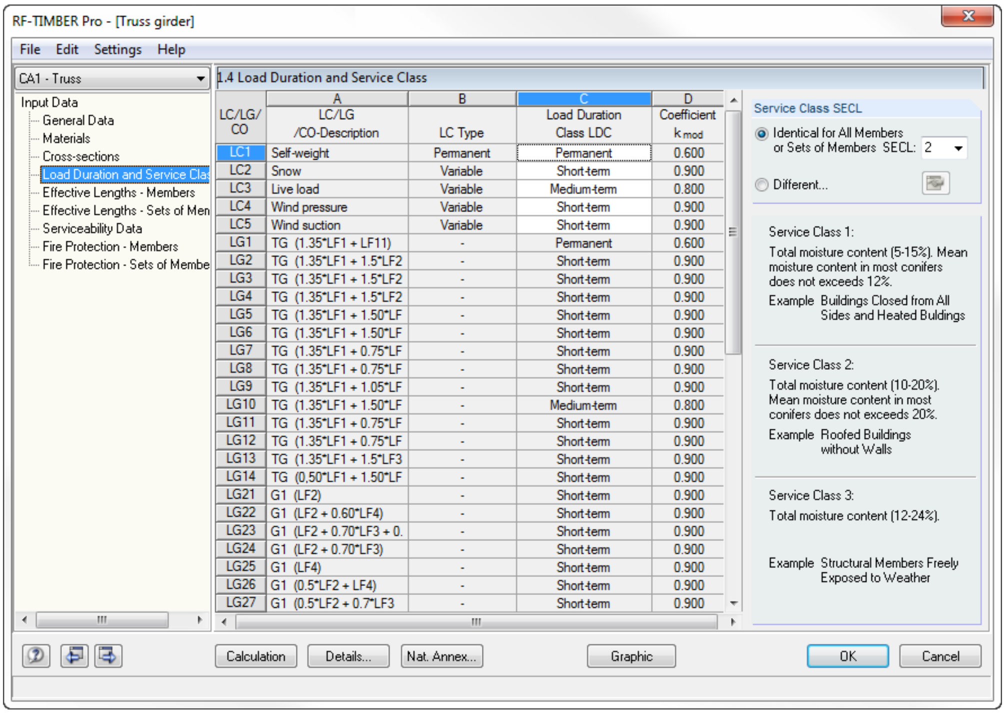


Figure 2.13: Table 1.4 *Load Duration and Service Class*

LC/LG/CO-Description

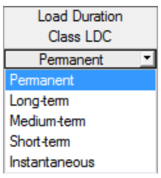
The load case descriptions defined in RFEM make the classification easier.

LC Type

This table column shows the load cases' types of action according to the RFEM definition. They are the basis for the program's presettings in the subsequent table column.

Load Duration Class LDC

Both the ultimate and the serviceability limit state design require the assignment of loads and their superpositions to particular load-duration classes. The classification of actions is specified for example in [1] table 4 or [4] table 2.1.



The load duration for load cases can be adjusted by using the list whereas load groups and load combinations are classified automatically taking into account the relevant leading action.

The modification factor k_{mod} is determined by means of the LDC. The factor is also considered for the material stiffness.

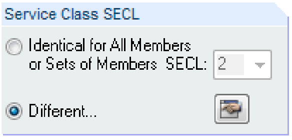
Coefficient k_{mod}

The impact of load duration and service class on the strength properties is taken into account by means of the modification factor (see [4] table 3.1).

The factors can be checked and, if necessary, adjusted in the dialog box *National Annex Settings* (see Figure 2.5, page 13).

Service Class SECL

The classification into service classes makes it possible to assign strength parameters and to calculate deformations by taking into account environmental conditions. The service classes are specified for example in [4] section 2.3.1.3.



By default the program assigns a standard service class to the entire structure. If you want to classify members or sets of members into different service classes, activate the selection field *Different*. To open the following dialog box, use the [Assign] button displayed to the right.

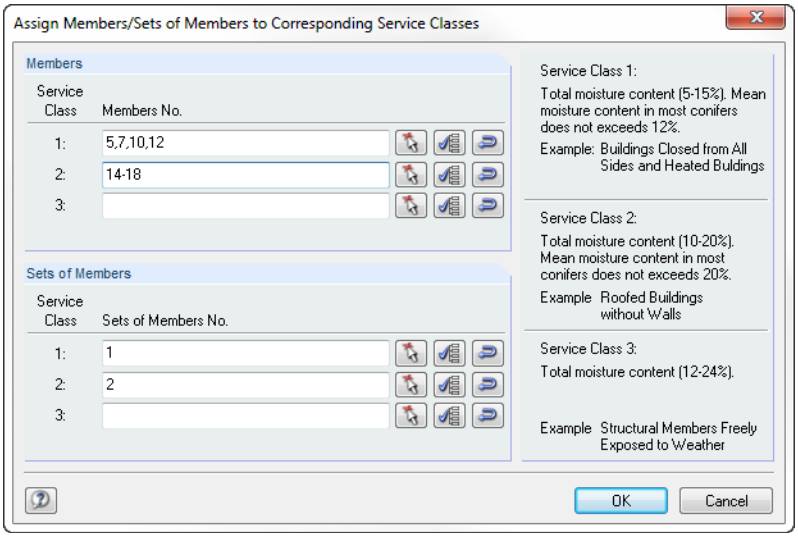


Figure 2.14: Dialog box *Assign Members/Sets of Members to Corresponding Service Classes*

The dialog box allows you to assign members and sets of members individually to service classes. The buttons are reserved for the following functions:

Button	Function
	Select members/sets of members graphically for service class
	Assign all members/sets of members to respective service class
	Assign all members/sets of members not yet assigned to respective service class

Table 2.2: Buttons in the dialog box *Assign Members/Sets of Members to Corresponding Service Classes*

2.5 Effective Lengths - Members

Table 1.5 is divided into two parts in order to have a better data overview. The table in the upper part contains summarized information about the buckling length coefficients and the equivalent member lengths for local and lateral buckling of the members to be designed. The table part below shows detailed information for the member that is selected in the table part above.

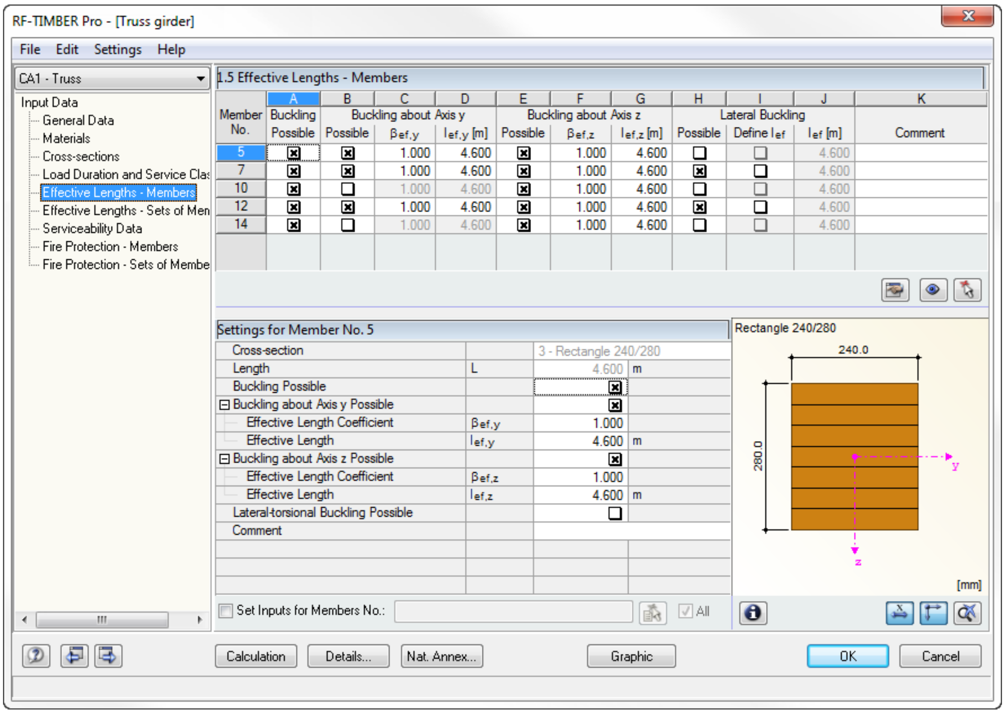


Figure 2.15 Table 1.5 *Effective Lengths - Members*

Changes to the buckling length coefficients can be made in the summarizing table in the upper part as well as in the detailed settings in the lower part. The specifications will be updated automatically in the respective other part of the table. The buckling length of a member can be defined also graphically by using the [Pick] function.

The tree structure in the lower table part *Settings* contains the following parameters:

- *Cross-section*
- *Length* of member
- *Buckling Possible* for member (cf. columns B, E and H)
- *Buckling about Axis y* (effective lengths, cf. columns C and D)
- *Buckling about Axis z* (effective lengths, cf. columns F and G)
- *Lateral-torsional Buckling* (lateral buckling length coefficients, cf. columns H to J)

It is possible to adjust the *Effective Length Coefficient* for the respective directions. Furthermore, you can specify if you generally want to perform a buckling analysis or a lateral torsional buckling design. When the buckling length coefficient is modified, the equivalent member length will be adjusted automatically.

The buckling length of a member can also be defined in a dialog box that you open by clicking the button shown on the left. In the table, you find the button below the upper table part on the right.

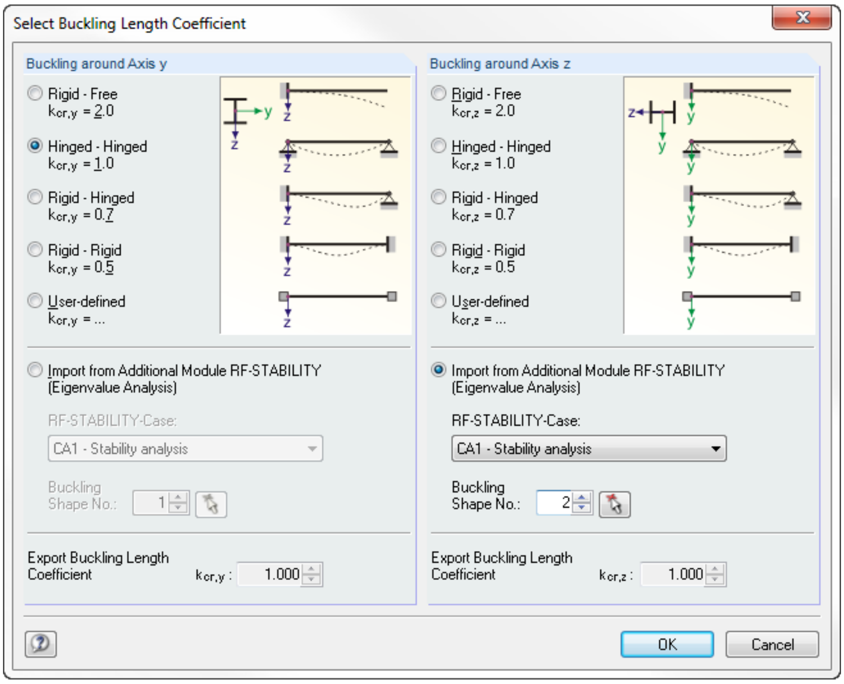


Figure 2.16: Dialog box *Select Buckling Length Coefficient*

In the dialog box *Select Buckling Length Coefficient*, you can choose one of the four famous Euler buckling modes, define the buckling length coefficient manually or, if a calculated RF-STABILITY case is already available, select the governing eigenmode to determine the coefficient.

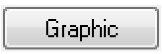
Buckling Possible

A stability analysis for flexural buckling and lateral-torsional buckling requires the ability of members to absorb compressive forces. Therefore, members for which such an absorption is not possible because of the defined member type (for example tension members, elastic foundations, rigid connections) are excluded from the RF-TIMBER Pro design in the first place. The corresponding rows are displayed in gray and a note is indicated in the *Comment* column.

The table column *Buckling Possible* offers you a control option to classify members as compression members or to exclude them from the design. Thus, with the check boxes in table column A and in the table *Settings for Member*, you decide for each member if the input fields for defining the buckling length parameters can be accessed.

Buckling around Axis y or Axis z

With the check boxes in the *Possible* table columns, you decide if a member has a risk of buckling about the axis y and/or z. These axes represent the local member axes, with axis y being the "strong" and axis z the "weak" member axis. The buckling length coefficients $\beta_{ef,y}$ and $\beta_{ef,z}$ for buckling around the strong or the weak axis can be selected freely.



The position of the member axes can be checked in the cross-section graphic in table 1.3 *Cross-sections* (see Figure 2.12). In the RFEM work window that you can always access by using the [Graphic] button, it is possible to display the local member axes by means of the *Display* navigator (see figure below).

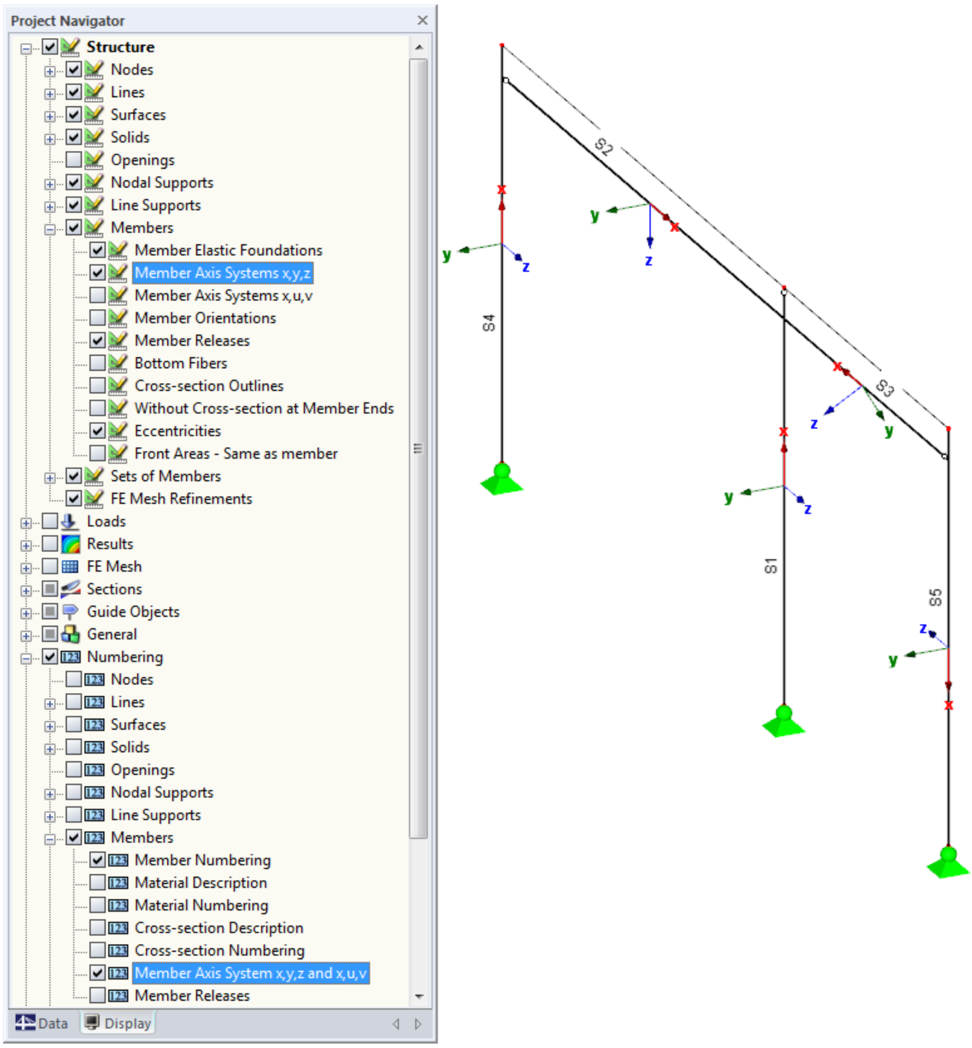


Figure 2.17: Selecting the local member axis systems in the *Display* navigator of RFEM

When buckling is possible about one or even both member axes, you can enter the buckling length coefficients as well as the buckling lengths in the columns C and D or F and G. The same is possible for the table *Settings for Member*.

When you specify the buckling length coefficient β_{ef}, the program determines the effective length l_{ef} by multiplying the member length L by the buckling length coefficient. The input fields are interactive.

To define the buckling length, you can use the [...] button appearing at the end of the l_{ef} input fields to determine the distance between two nodes graphically in the RFEM work window.

Lateral Buckling Possible

Table column H shows you for which members the program performs an analysis of lateral-torsional buckling.

The equivalent member length l_{ef} for this type of design can be defined *manually* or graphically by the spacing of lateral supports.

Below the *Settings* table you find the check box *Set Inputs for Members No*. If it is ticked, the settings entered afterwards will be applied to the selected or even to *All* members. Members can be selected by entering the member number or by selecting them graphically with the [Pick] button. This option is useful when you want to assign the same boundary conditions to several members. Please note that settings that have been already defined cannot be changed subsequently with this function.

Comment

In the final table column, you can enter user-defined specifications for each member to describe for example the selected equivalent member lengths.

2.6 Effective Lengths - Sets of Members

Table 1.6 manages the effective lengths of sets of members. It is displayed only when you have selected at least one set of members for the design in table 1.1 *General Data*.

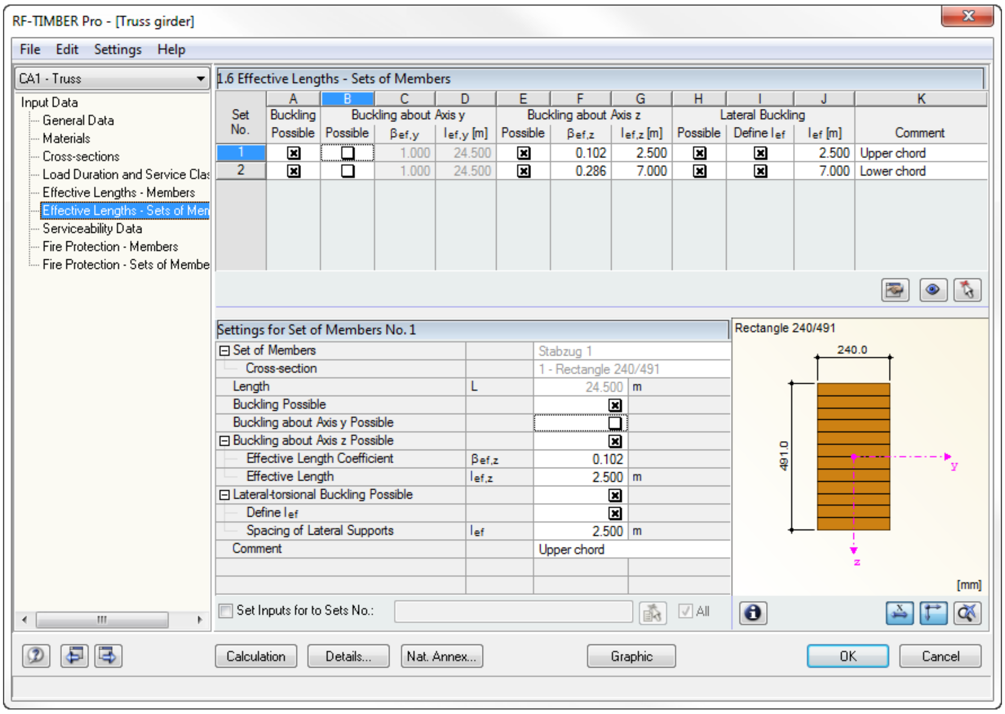


Figure 2.18 Table 1.6 *Effective Lengths - Sets of Members*

The structure of this table is similar to the previous table 1.5. Enter the effective lengths for local as well as for lateral-torsional buckling about the two principal axes of the set of members as described above. They determine the boundary conditions of the set of members that is treated at large as equivalent member.

The detail settings also specify the cross-sections that are contained in the set of members.

2.7 Tapered Members

This table is only available when you have selected at least one member with different cross-sections on both member ends for the design in table 1.1 *General Data*. Here, you can check the definition criteria of the tapered members.

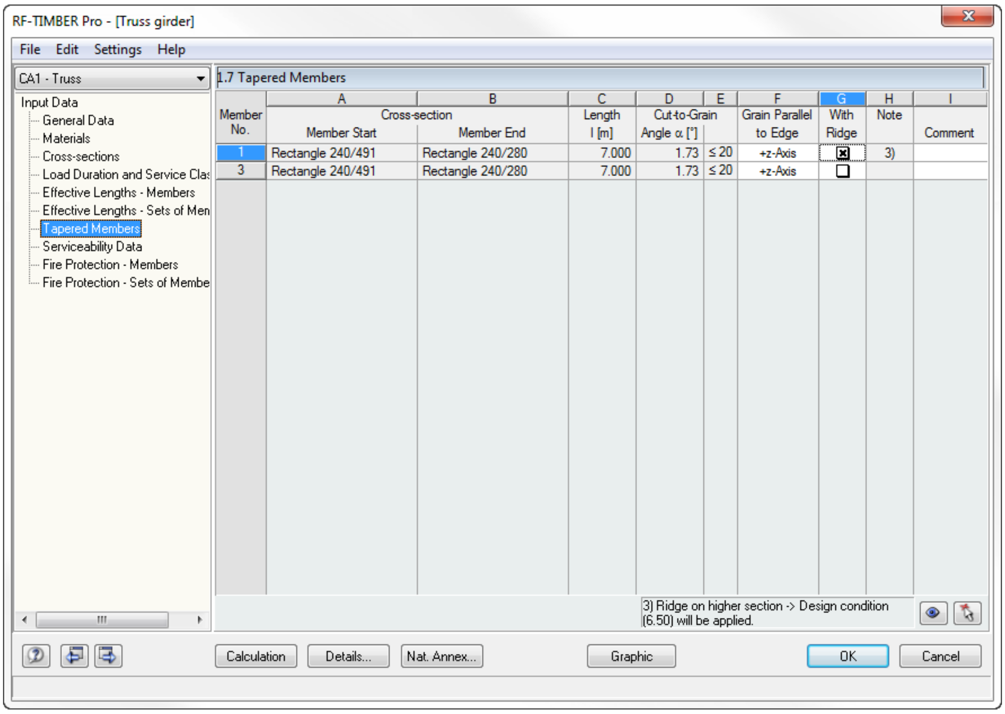


Figure 2.19: Table 1.7 *Tapered Members*

Cross-section

Table columns A and B list the cross-sections that are defined on the *Member Start* and *Member End*.

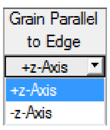
Length

The length of the tapered member is also displayed for checking reasons.

Cut-to-Grain Angle α

RF-TIMBER Pro determines the cut-to-grain angle on the basis of the geometric conditions. The design equations used in the program apply only to the cutting angle $\alpha \leq 20°$ for EN 1995-1-1 and SIA 265 or $\alpha \leq 10°$ for DIN 1052. The limit values can be checked and, if necessary, adjusted in the dialog box *National Annex Settings* (see Figure 2.5, page 13).

Grain Parallel to Edge



In table column F you define the member edge to which the timber's grain direction is running parallel. The "top" or "bottom" edge can be clearly determined by means of the orientation of the local member axis **z** (see Figure 2.17).

With Ridge

When the check box is ticked in table column G *With Ridge*, the program performs the design for the maximum tension stresses perpendicular to the grain and shear from transverse force in the ridge cross-section, for example according to [1] condition (85) or [4] condition (6.50).

2.8 Serviceability Data

This input table controls several settings for the serviceability limit state design. It is only available when you have set relevant entries in the *Serviceability Limit State* tab of table 1.1 (see chapter 2.1.2, page 10).

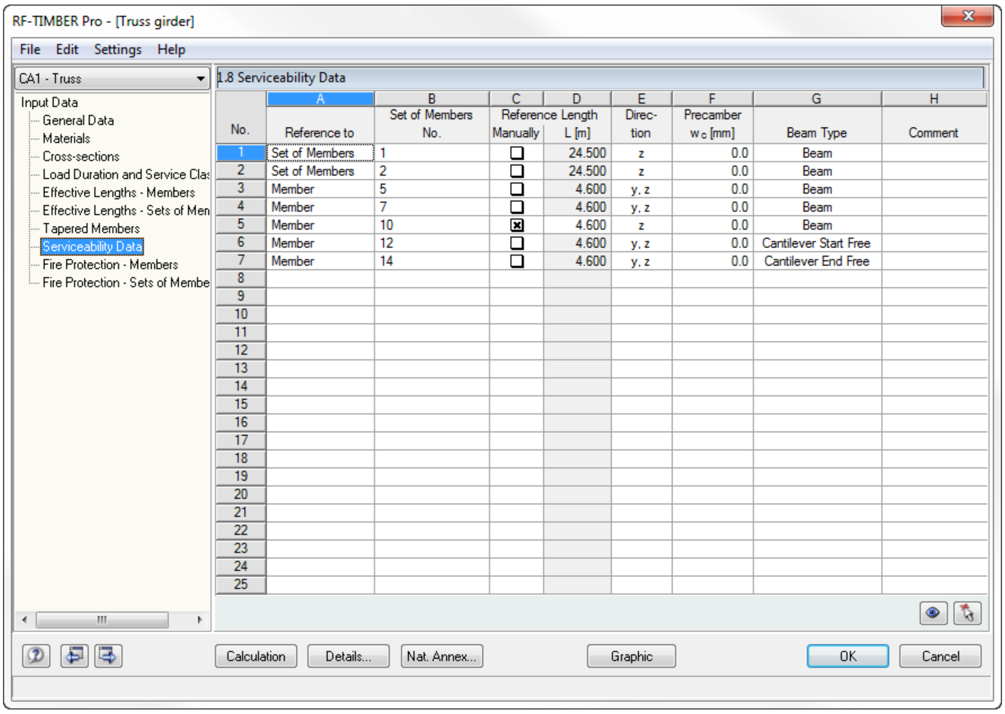


Figure 2.20: Table 1.8 *Serviceability Data*

Reference to

In column A, you decide whether you want to apply the deformation to single members, lists of members or sets of members.

Member / Set No.

In table column B, you enter the numbers of the members or sets of members that you want to design. You can also use the [Pick] function to select them graphically in the RFEM work window. The respective *Reference Length* will be entered automatically in column D. This column presets the lengths of the members, sets of members or member lists. If required, you can adjust these values after ticking the *Manually* check box in column C.

Reference Length

The reference lengths L are represented by the lengths of members or sets of members that are preset in table column D. If required, you can adjust these values after ticking the *Manually* check box in column C.

Direction

Table column E defines the decisive direction for the deformation analysis. The member orientations y and z are available for selection.

Precamber

If you want to consider a precamber w_c, enter a value for the rise in table column F.

Beam Type

The beam type is of vital importance for the correct application of limit deformations. In table column G, you can select the girder to be a beam or a cantilever.

The settings specified in the *Serviceability* tab of the *Details* dialog box determine whether the deformations are related to the undeformed initial structure or to the shifted ends of members or sets of members (see Figure 3.3, page 32).

2.9 Fire Protection - Members

Table 1.9 manages the fire resistance parameters of the members selected for the design. It is only available if you have set relevant entries in the *Fire Resistance* tab of table 1.1 (see chapter 2.1.2, page 10).

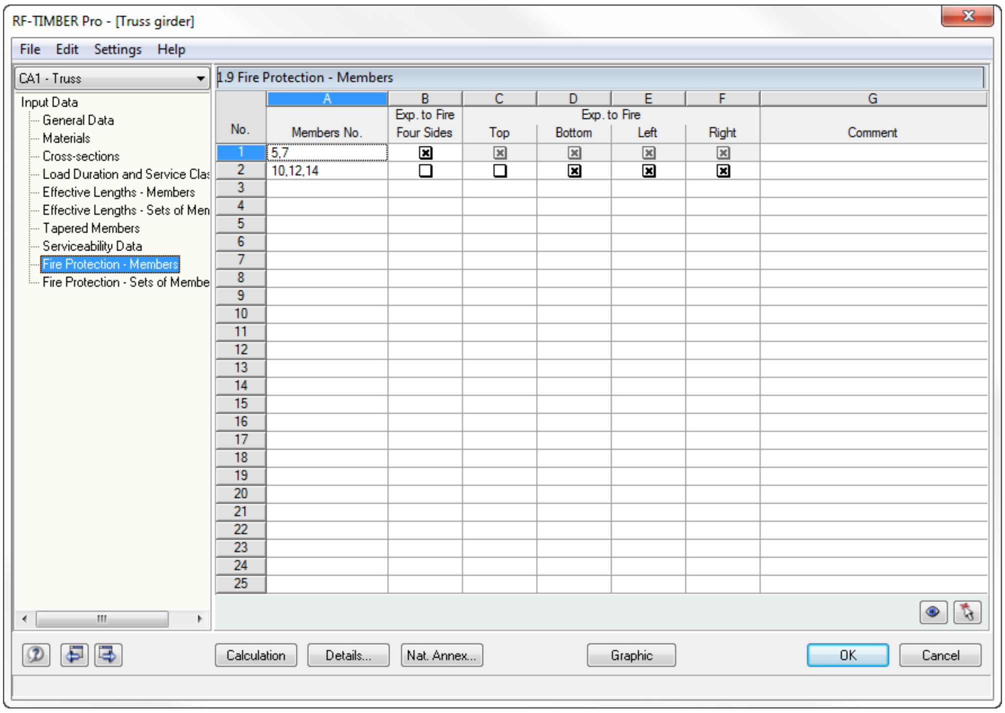


Figure 2.21: Table 1.9 *Fire Protection - Members*

Members No.

Table column A contains the members that are taken into account by the design. Click the [...] button to use the pick function for a graphical member selection in the RFEM work window.

Exposed to Fire - Four Sides

RF-TIMBER Pro assumes that the cross-section is exposed to fire on all sides. If the assumption is wrong and the given circumstances are different, clear the check box. Then it is possible to access the subsequent table columns.

Exposed to Fire

In case the charring does not occur on all four sides, you can define the cross-section sides exposed to fire in the columns C to F. The entries are required in order to determine the ideal remaining cross-section with the fire resistance settings specified in the *Details* dialog box (see chapter 3.1.4, page 32) correctly.

2.10 Fire Protection - Sets of Members

The final input table 1.10 manages the fire resistance parameters of sets of members. It is displayed only when you have selected at least one set of members for the design in table 1.1 *General Data*.

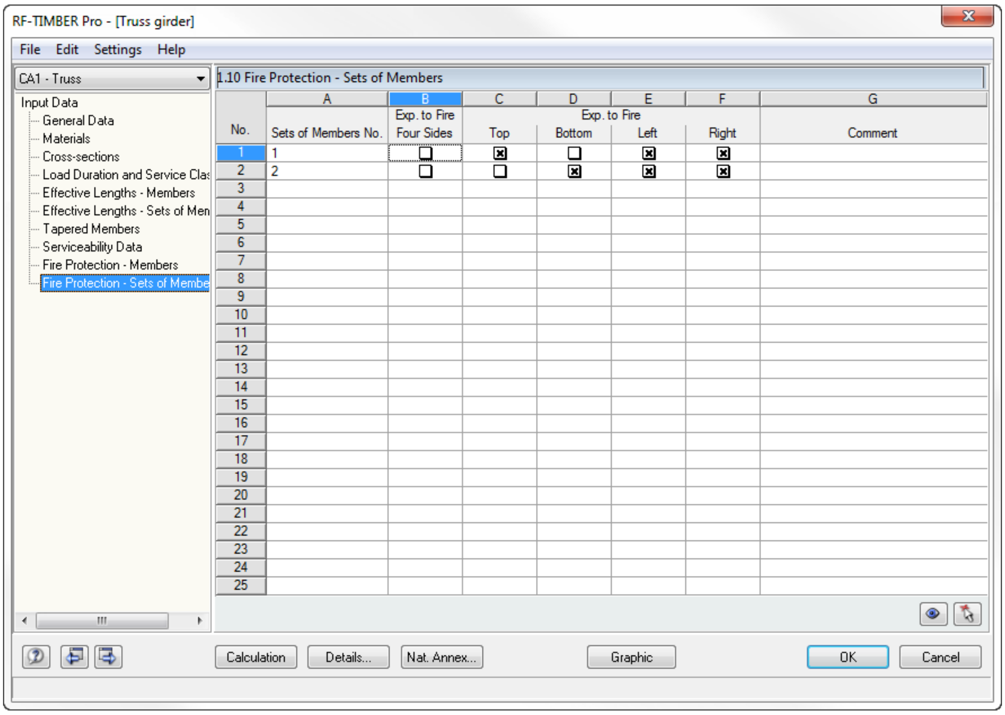
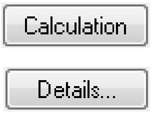


Figure 2.22: Table 1.10 *Fire Protection - Sets of Members*

The structure of the table is similar to the previous table 1.9. Enter the charring sides of the cross-section as described above.

3. Calculation

3.1 Detail Settings



The individual designs are carried out by using the internal forces determined in RFEM. Before you start the calculation by clicking the [Calculation] button, it is recommended to check the design details. The corresponding dialog box can be accessed in each input und output table by using the [Details] button.

The *Details* dialog box provides the following tabs:

- Resistance
- Stability
- Serviceability
- Fire Resistance
- Other

3.1.1 Resistance

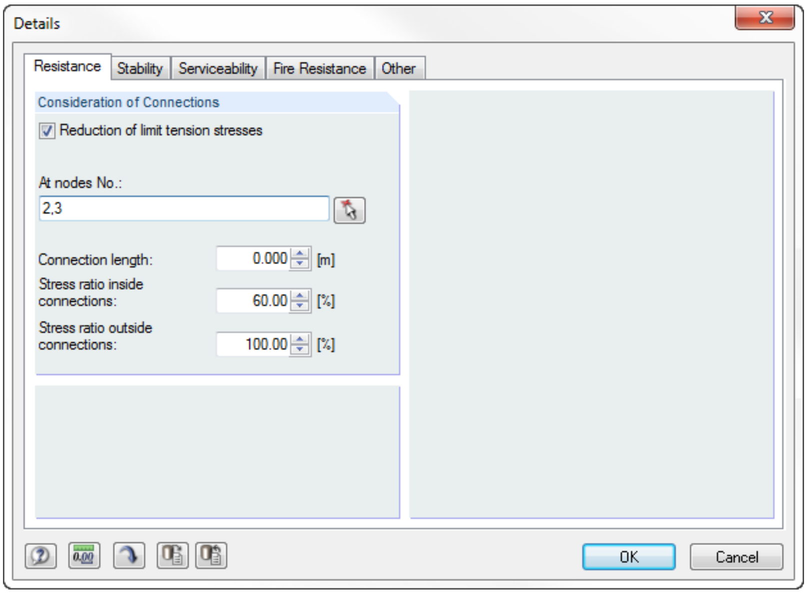


Figure 3.1: Dialog box *Details*, tab *Resistance*

Consideration of Connections

Often, zones near member connections show weakenings of the cross-section. It is possible to take into account this effect by a *Reduction of limit tension stresses*.

The numbers of the relevant *nodes* can be entered manually or selected graphically by using the [Pick] button.

The *Connection length* defines the zone on the member where reduced stresses are considered. In the input field below, enter the allowable *Stress ratio* in percentage.

The maximum allowable stresses are applied outside the connection zone. If you want to reduce the allowable stresses also there, it is possible to enter a limit in the input field *Stress ratio outside connections*.

3.1.2 Stability

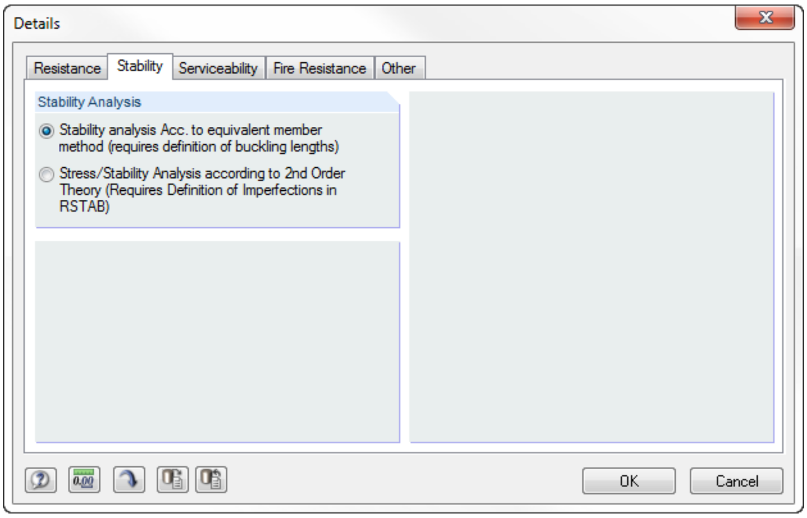


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

You can select between equivalent member method and a design according to the second-order analysis.

When the *equivalent member method* is selected, the internal forces determined according to the linear static analysis will be used for the design. Thus, the effective lengths of the members and sets of members must be specified in table 1.5 and 1.6 to perform the stability analysis.

When the bearing capacity of a structural system is significantly affected by its deformations, it is recommended to select a calculation according to the *2nd Order Theory*. This approach requires additionally the definition of imperfections in RFEM taken into account for the load groups.

Please note that you also have to perform the design for lateral torsional buckling for a calculation according to the second-order analysis. Thus, the lengths for lateral-torsional buckling of members or sets of members must be specified manually in table 1.5 or 1.6 *Effective Lengths*. In this way, we can make sure that the lateral buckling analysis will be performed with the appropriate factors (for example 1.0).

3.1.3 Serviceability

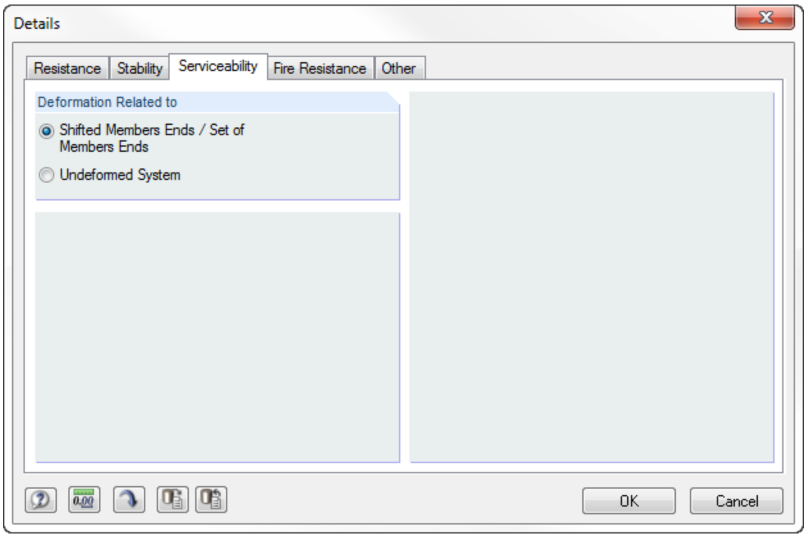


Figure 3.3: Dialog box *Details*, tab *Serviceability*

With the options in the dialog section *Deformation Related to*, you can decide if the maximum deformations are related to the undeformed initial structure or to an imaginary connection line between member start and member end in the deformed system, which means the shifted ends of the member or member set.

The limit deformations can be checked and, if necessary, adjusted in the dialog box *National Annex Settings* (see chapter 2.1.4, page 12).

3.1.4 Fire Resistance

This tab manages the settings for the fire protection design.

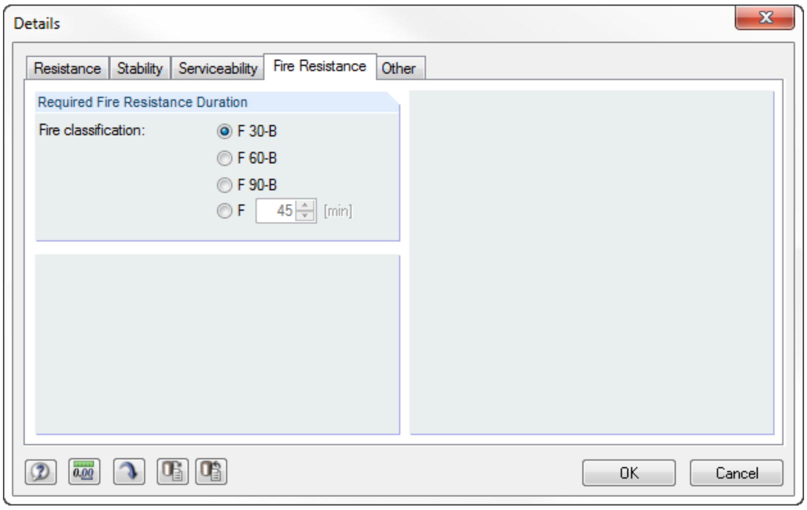


Figure 3.4: Dialog box *Details*, tab *Fire Resistance*

The *Fire classification* can be selected directly or defined individually by specifying a time for the fire duration.

The dialog box *National Annex Settings* manages the standard-specific parameters that are significant for the fire protection design (see Figure 2.5, page 13).

3.1.5 Other

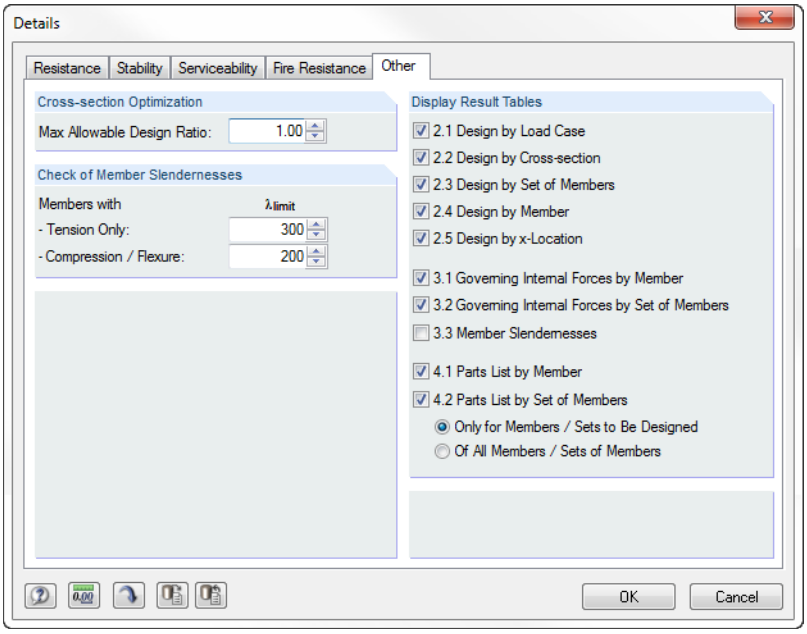


Figure 3.5: Dialog box *Details*, tab *Other*

Cross-section Optimization

In case the optimization is not targeted on the maximum design ratio of 100 %, you can specify another limit value in this input field.

Check of Member Slendernesses

The two fields allow for the input of limit values λ_{limit} in order to define member slendernesses. It is possible to enter specifications separately for members with pure tension forces and members with bending and compression.

In the results table 3.3, the limit values are compared to the real member slendernesses. This table is available only after the calculation (see chapter 4.8, page 42) when the corresponding check mark has been set in the dialog section *Display Result Tables* to the right.

Display Result Tables

In this dialog section, you can select the results tables including parts lists that you want to display in the output. The individual results tables are described in chapter 4.

Results table 3.3 *Member Slendernesses* is set inactive by default but can be activated for a specified evaluation by ticking the check box.

3.2 Start Calculation

To start the calculation, click the [Calculation] button that is available in all input tables of the add-on module RF-TIMBER Pro.

RF-TIMBER Pro searches for the results of the load cases, load groups and load combinations that you want to design. If they cannot be found, the program starts the RFEM calculation to determine the design relevant internal forces. In this determination process, the calculation parameters preset in RFEM are applied.

If cross-sections should be optimized (see chapter 7.2, page 56), the program determines the required cross-sections and performs the corresponding designs.

It is also possible to start the calculation for RF-TIMBER Pro results in the RFEM user interface. The add-on modules are listed like load cases and load groups in the dialog box *To Calculate*. To open the dialog box in RFEM,

select **To Calculate** on the **Calculate** menu.

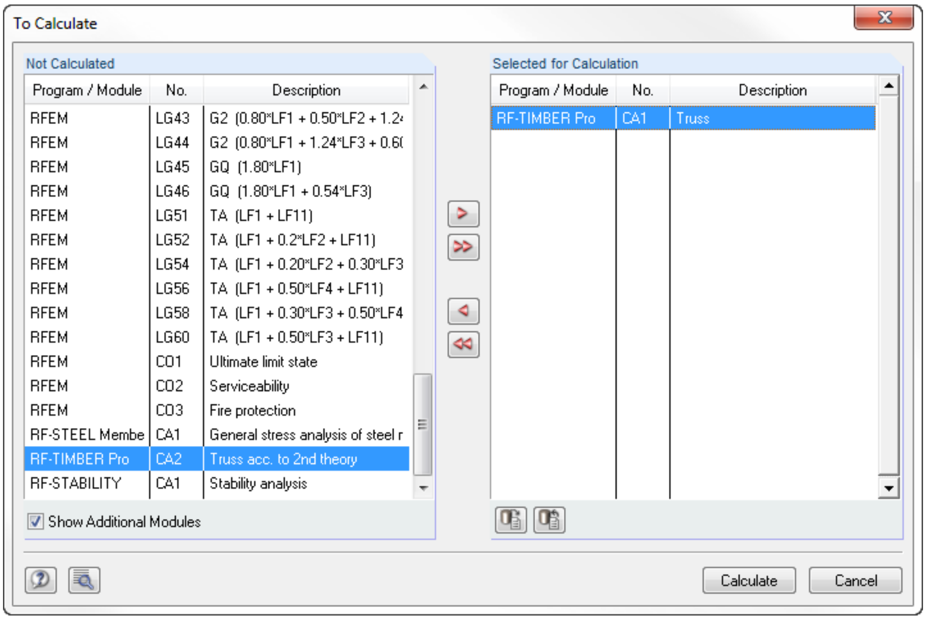


Figure 3.6: Dialog box *To Calculate*

If the RF-TIMBER Pro design cases are missing in the *Not Calculated* list, tick the check box *Show Additional Modules* below the list.

To transfer the selected RF-TIMBER Pro cases to the list on the right, use the button [▶]. Start the calculation by using the [Calculate] button.

You can also use the list in the RFEM toolbar to calculate a design case directly: Select the RF-TIMBER Pro case and click the button [Results on/off].

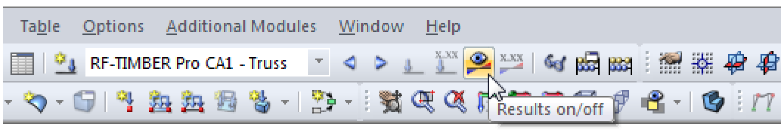


Figure 3.7: Direct calculation of a RF-TIMBER Pro design case in RFEM

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

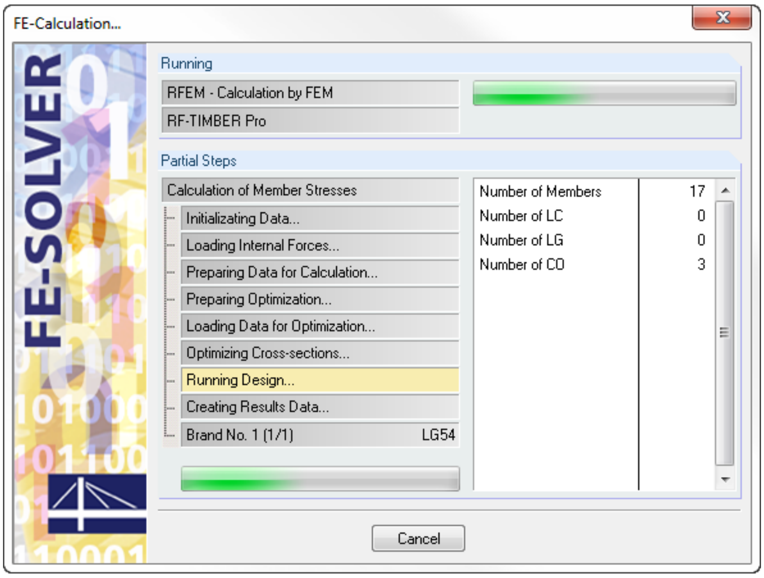
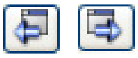


Figure 3.8: RF-TIMBER Pro calculation

4. Results

Table 2.1 *Design by Load Case* is displayed immediately after the calculation. The upper part of the results table shows you a summary of designs for the individual load cases, groups and combinations. The lower part contains detailed information about the cross-section characteristics, analyzed internal forces and performed designs for the load case that is selected above.

The designs are shown in the results tables 2.1 to 2.5, sorted by different criteria. The subsequent tables 3.1 and 3.2 display the governing internal forces. Table 3.3 informs you about the member slendernesses. The two final results tables 4.1 and 4.2 contain the parts lists referring to members and sets of members.



Each results table can be selected directly in the RF-TIMBER Pro navigator. You can also use the two buttons shown on the left or the function keys [F2] and [F3] to select the previous or subsequent table.

Click the [OK] button to save the results. The add-on module RF-TIMBER Pro will be closed and you will return to the RFEM work window.

Chapter 4 *Results* describes the different results tables one after the other. Evaluating and checking results is described in detail in chapter 5 *Results Evaluation* on page 45.

4.1 Design by Load Case

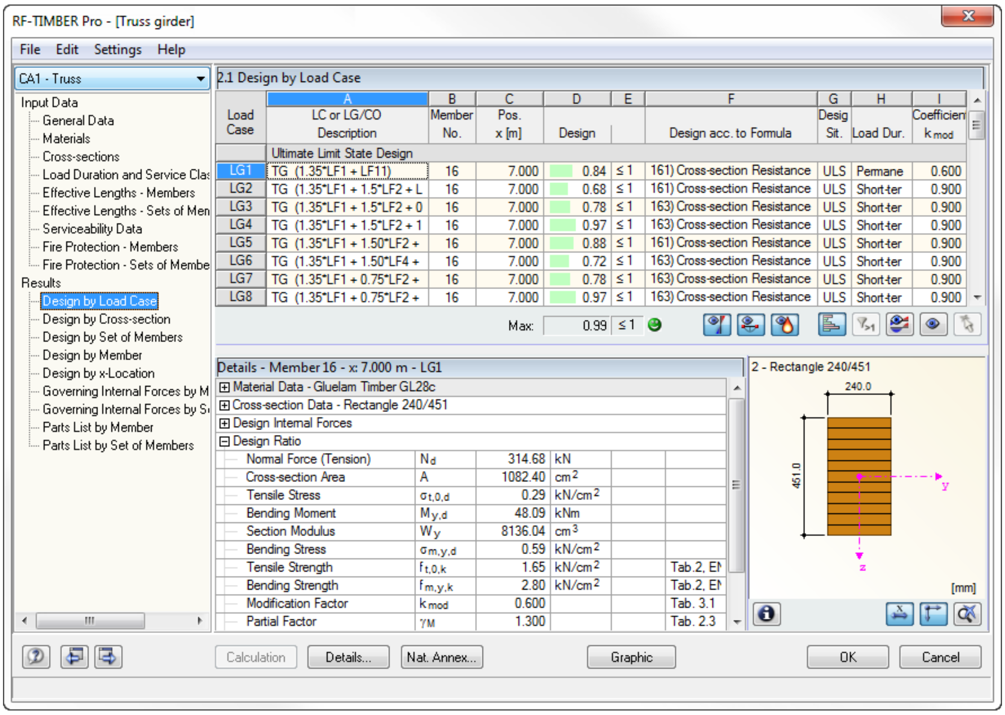


Figure 4.1: Table 2.1 *Design by Load Case*

Description

This column displays for information the descriptions of the load cases, groups and combinations that are decisive for the respective designs.

Member No.

Table column B shows for each designed load case, group or combination the number of the member that has the maximum design ratio.

Position x

The column shows the respective x-location for which the program has determined the member's maximum design ratio. For the table output, the program uses the following RFEM member locations *x*:

- Start and end node
- Partition points according to possibly defined member division
- Member division according to specification for member results (*Options* tab of RFEM dialog box *Calculation Parameters*)
- Extreme values of internal forces

Design

The output shows for each type of design and each load case, group or combination the design criteria according to the selected standard.

The colored scales represent the design ratios due to the individual load cases.

Design according to Formula

This column lists the code's equations by which the designs have been performed.

Design Situation

Table column G contains information about the respective design-relevant *Design Situation*: *ULS* (ultimate limit state) or one of the three design situations for serviceability (*SC*, *S1*, *S2*) according to the specification set in table 1.1 *General Data* (see Figure 2.3, page 10) or *ULS* (fire protection).

Load Duration Class

This table column indicates the load duration classes defined in table 1.4 (see chapter 2.4, page 20).

Coefficient k_{mod}

The final table column informs you about the modification factors by which the impact of load duration and service class on the strength properties is taken into account (see chapter 2.4, page 20).

4.2 Design by Cross-section

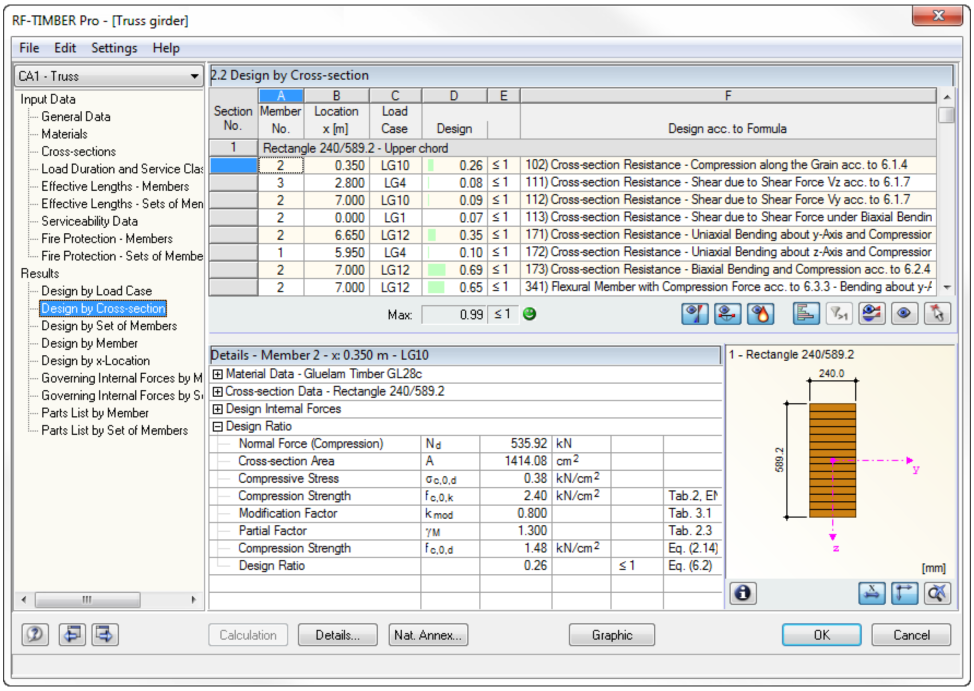


Figure 4.2: Table 2.2 *Design by Cross-section*

This table lists the maximum ratios of all members and actions selected for design, sorted by cross-sections.

If a member is a tapered member, both cross-section descriptions are displayed in the table row next to the section number.

4.3 Design by Set of Members

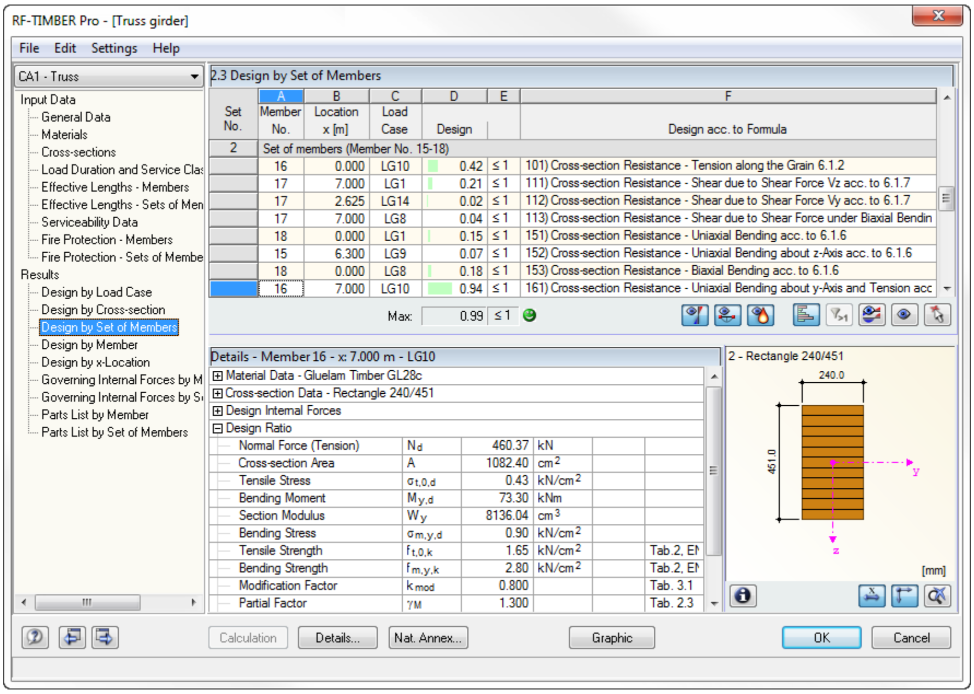


Figure 4.3: Table 2.3 *Design by Set of Members*

This results table is displayed when you have selected at least one set of members for the design. The maximum design ratios are listed by sets of members. Table column *Member No.* shows the number of the member that has the maximum ratio within the set of members.

The results output by sets of members presents the design for an entire structural group (for example a chord) clearly in one results table.

4.4 Design by Member

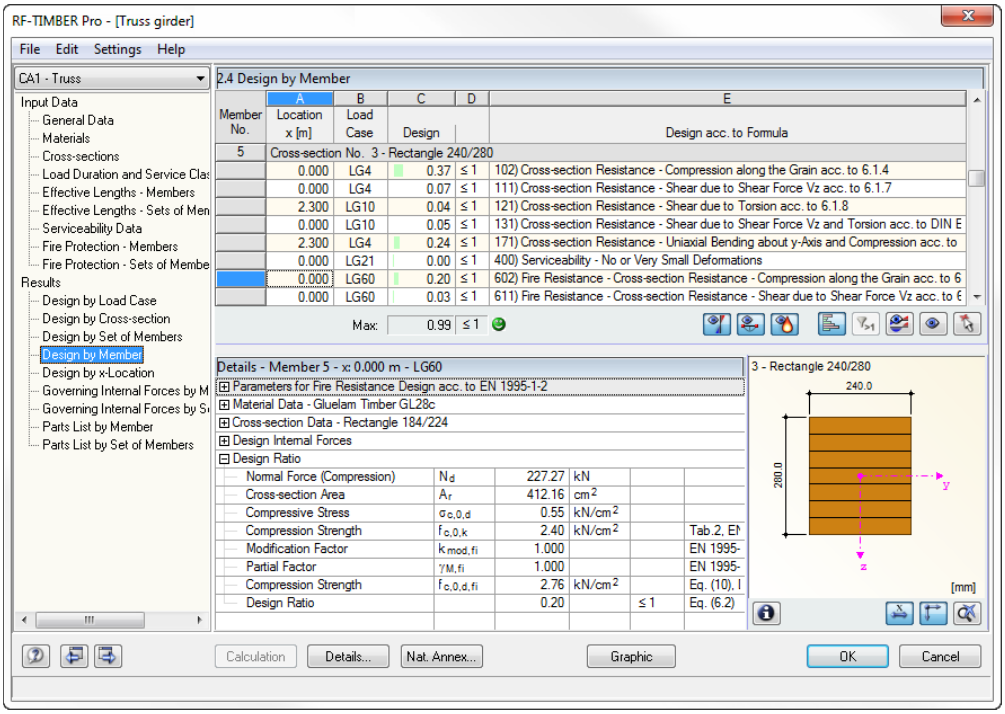


Figure 4.4: Table 2.4 *Design by Member*

This results table shows the maximum design ratios sorted by member numbers. The different columns are described in detail in chapter 4.1 on page 36.

4.5 Design by x-Location

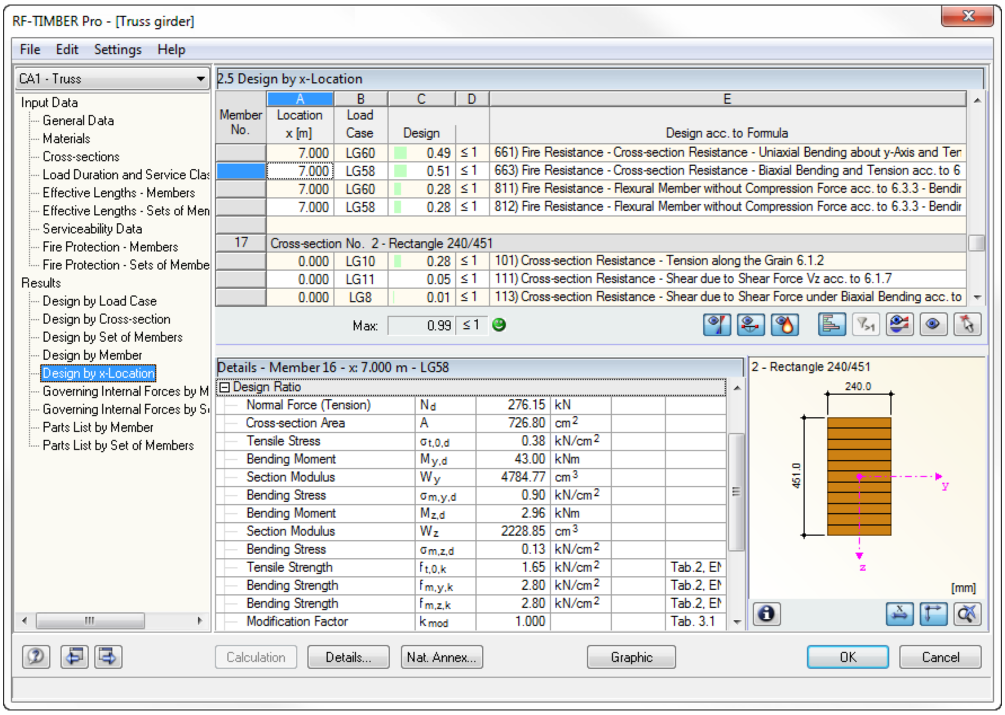


Figure 4.5: Table 2.5 *Design by x-Location*

This results table lists the maxima for each member at the locations ***x*** resulting from the division points defined in RFEM:

- Start and end node
- Partition points according to possibly defined member division
- Member division according to specification for member results (*Options* tab of RFEM dialog box *Calculation Parameters*)
- Extreme values of internal forces

4.6 Governing Internal Forces by Member

This table displays for each member the governing internal forces that result in the maximum design ratio.

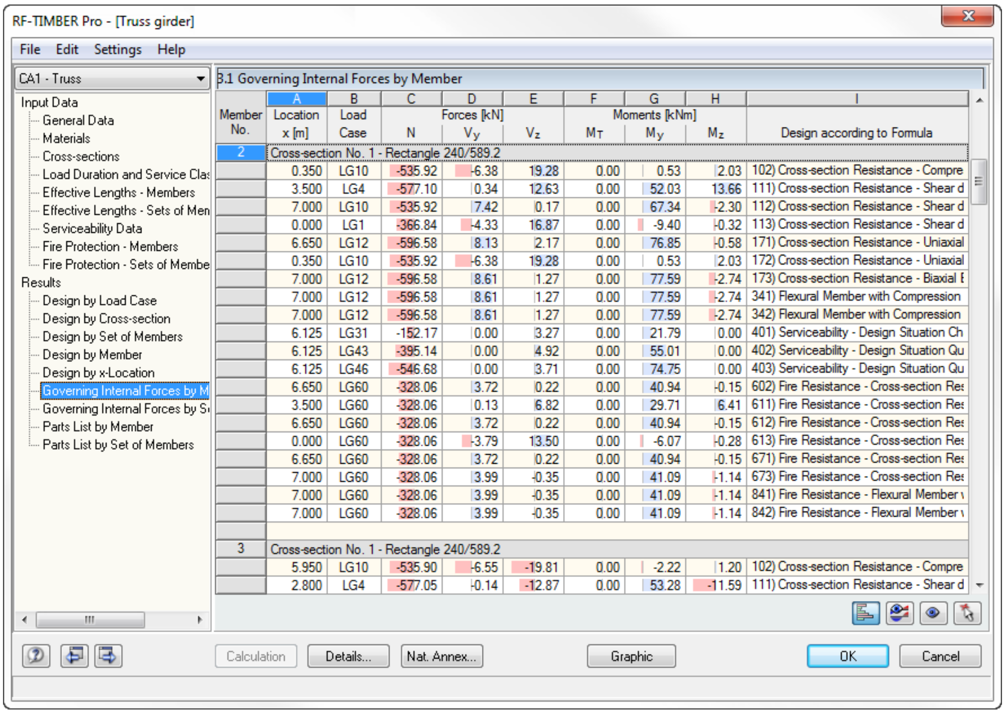


Figure 4.6: Table 3.1 *Governing Internal Forces by Member*

Location x

The column shows the respective x-location where the member's maximum ratio occurs.

Load Case

This column indicates the numbers of the load case, load group or load combination whose internal forces result in the maximum design ratio on the member.

Forces / Moments

For each member, the governing normal and shear forces as well as the torsional and bending moments are displayed.

Design according to Formula

The final column informs you about the design types and equations by which the designs have been performed according to the selected standard.

4.7 Governing Internal Forces by Set of Members

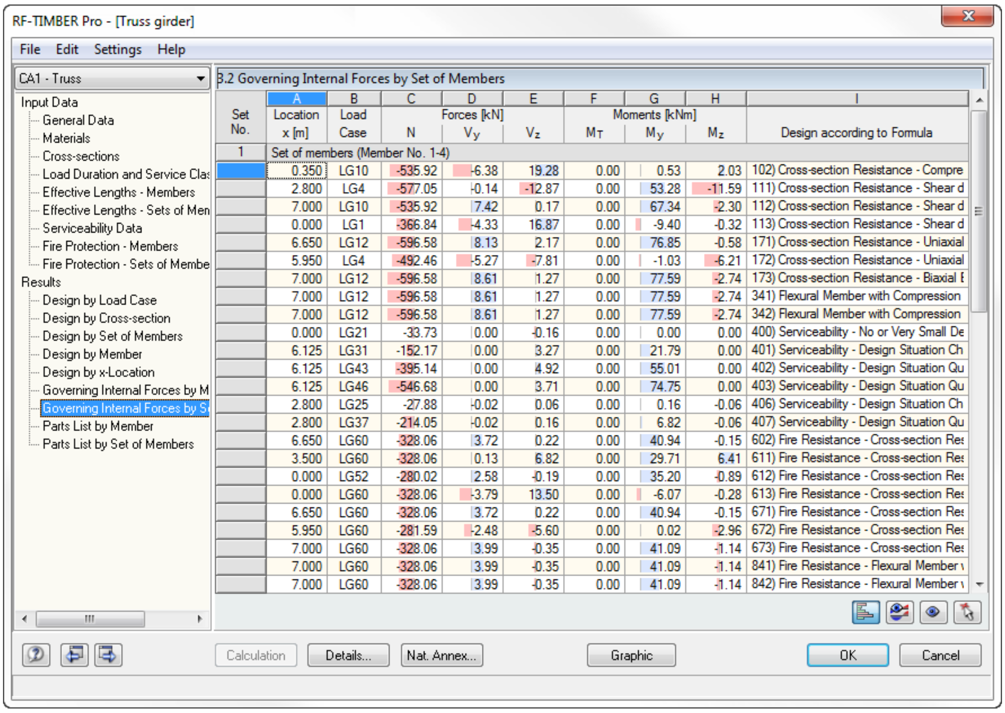


Figure 4.7: Table 3.2 *Governing Internal Forces by Set of Members*

This table contains the governing internal forces that result in the maximum ratios in each set of members.

4.8 Member Slendernesses

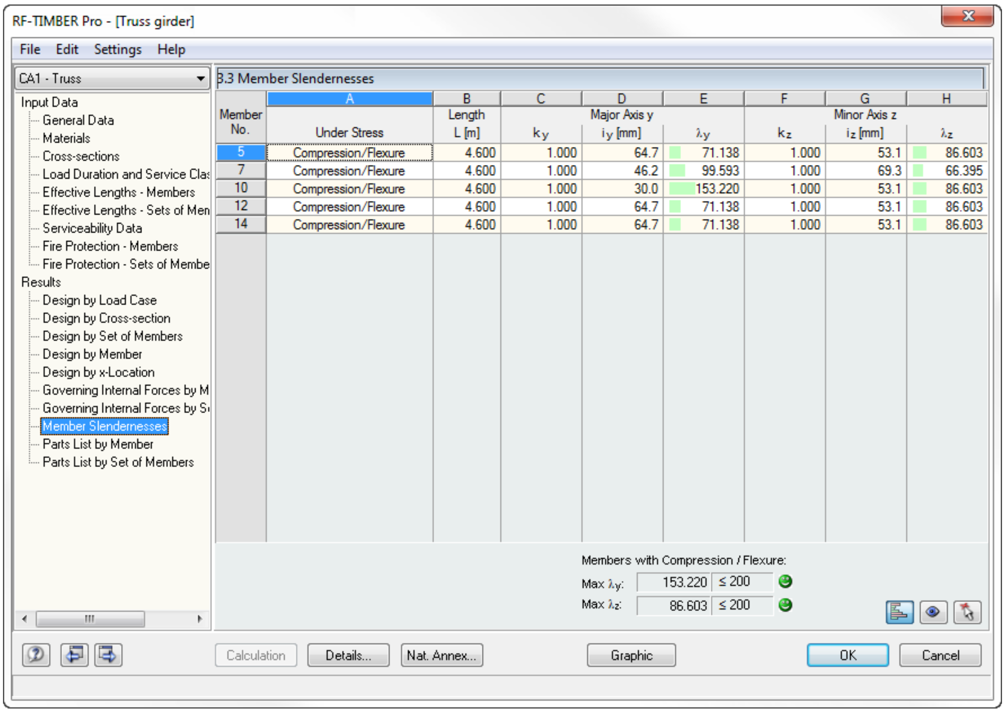


Figure 4.8: Table 3.3 *Member Slendernesses*

The table lists the effective slendernesses of the designed members for both principal directions depending on the type of load and compares them to the values that have been defined in the *Details* dialog box (see chapter 3.1.5, page 33). This table is displayed only for information because no stability design of slendernesses is intended.

Members of the member type "Tension" are not included in this table.

4.9 Parts List by Member

Finally, RF-TIMBER Pro provides a summary of cross-sections contained in the design case.

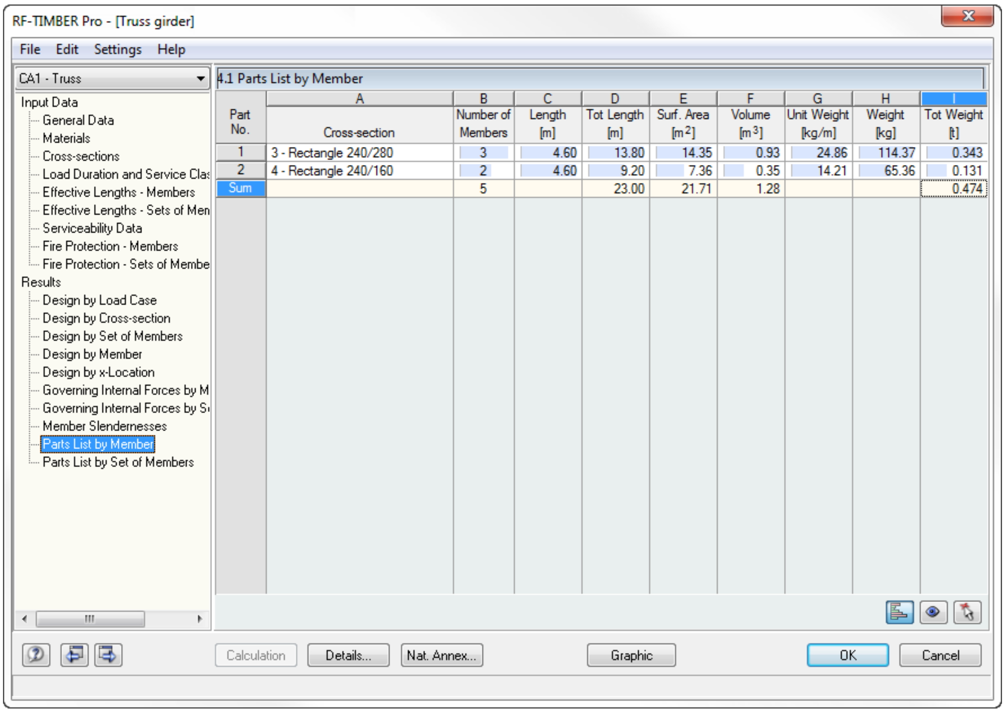


Figure 4.9: Table 4.1 *Parts List by Member*

By default, the list contains only the designed members. If you want to display a parts list for all members of the structure, select the corresponding option in the *Other* tab of the *Details* dialog box (see Figure 3.5, page 33). To open the dialog box, use the [Details] button shown on the left.

Part No.

The program assigns automatically part numbers for similar members.

Cross-section

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column displays the length of an individual member.

Total Length

This column shows the product that is determined from the two previous columns.

Surface Area

The program indicates the surface area of the respective parts in relation to the total length. The surface area is determined from the *Surface* of the cross-sections. You can find and check the relevant entry in the cross-section information, available in table 1.3 and tables 2.1 to 2.5 (see Figure 2.12, page 18).

Volume

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

Unit Weight

The *Unit Weight* of the cross-section represents the mass in relation to the length of one meter. For tapered cross-sections, the program averages both cross-section properties.

Weight

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

Total Weight

The final column indicates the total weight of the respective part.

Sum

At the bottom of the list, you find a summary showing the sums of column B, D, E, F and I. The results output in the *Tot Weight* column gives information about the overall timber mass that is required.

4.10 Parts List by Set of Members

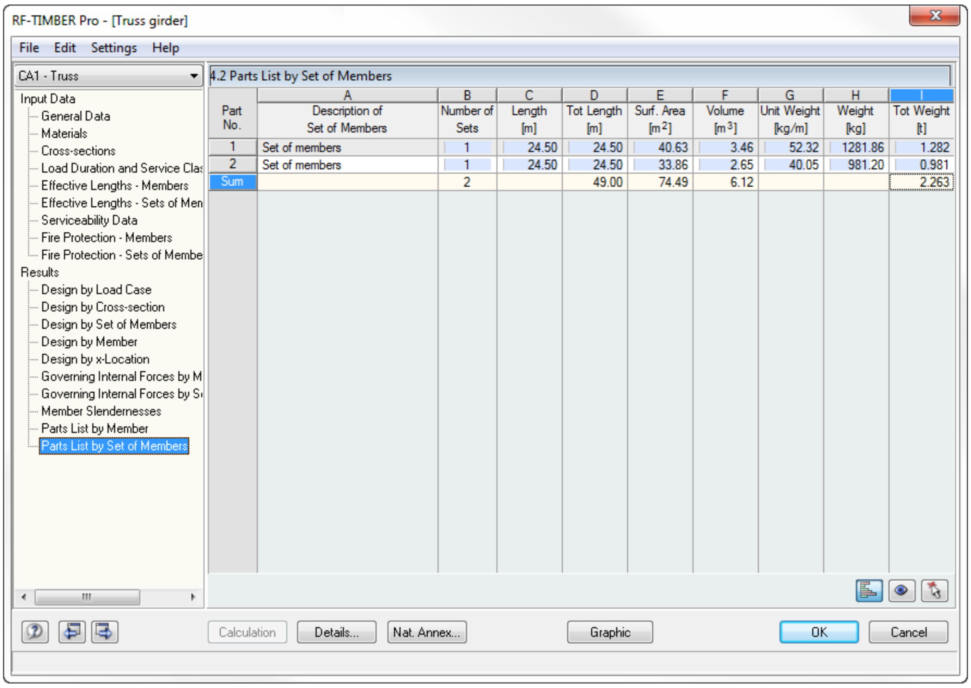


Figure 4.10: Table 4.2 *Parts List by Set of Members*

The final table is displayed when you have selected at least one set of members for the design. It offers you a summarizing parts list for an entire structural group, e.g. a chord).

Details on the table columns can be found in the previous chapter 4.9. When different cross-sections are used in the set of members, the program averages the surface area, the volume and the unit weight.

5. Results Evaluation

When the design is complete, several options are available for the results evaluation. The buttons below the first table part can be helpful for the evaluation process.

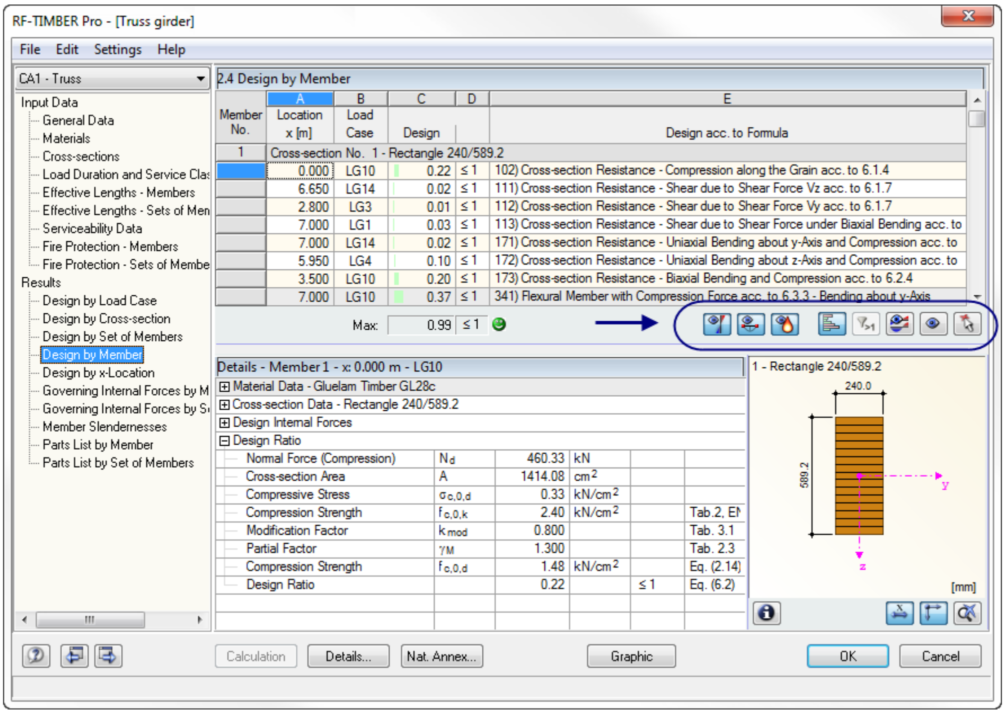


Figure 5.1: Buttons for results evaluation

The buttons are reserved for the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Turns on and off the results of the ultimate limit state design
	Serviceability Limit State Designs	Turns on and off the results of the serviceability limit state design
	Fire Protection Designs	Turns on and off the results of the fire protection design
	Show Color Bars	Turns on and off the colored reference scales in the results tables
	Exceeding	Displays only rows where the ratio is more than 1, thus the design is failed
	Show Result Diagrams	Opens the window *Result Diagram on Member* → chapter 5.2, page 49
	View mode	Jumps to the RFEM work window to change the view
	Member selection	Enables the selection of a member in the RFEM work window to show its results in the table

Table 5.1: Buttons of results tables 2.1 to 2.5

5.1 Results in the RFEM Model

To evaluate the design results, you can also use the RFEM work window.

RFEM background graphic and view mode

The RFEM graphic in the background may be useful when you want to check the position of a particular member in the model. The member that is selected in the RF-TIMBER Pro results table is highlighted in the selection color in the RFEM background graphic. In addition, an arrow indicates the member's x-location that is displayed in the active table row.

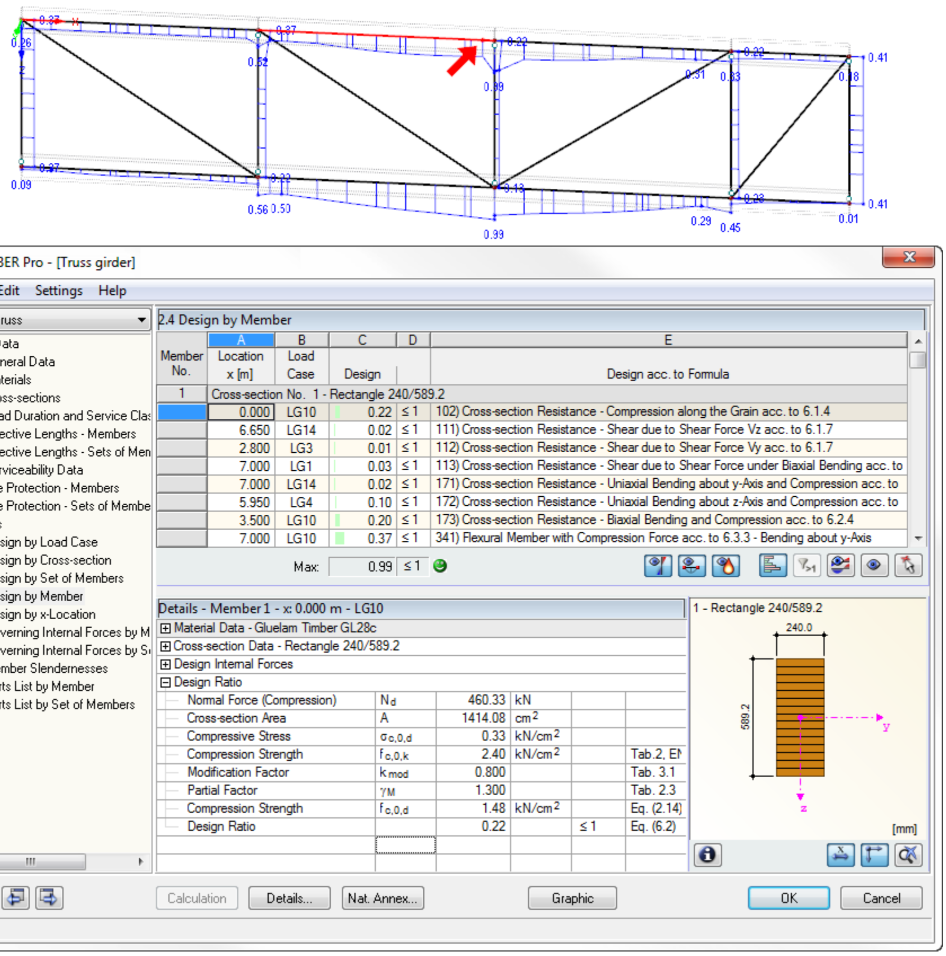


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model

If you move the RF-TIMBER Pro window to another place in the display and you still cannot see the graphic clearly, use the button [Jump to Graphics] to activate the *view mode*: The RF-TIMBER Pro window will be hidden so that you can adjust the view appropriately in the RFEM user interface. The view mode provides the functions of the *View* menu, for example zooming, moving or rotating the display.

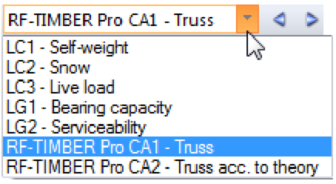
Click [Back] to return to the add-on module RF-TIMBER Pro.

RFEM work window

It is also possible to visualize the design ratios directly in the structural model: Click the [Graphic] button to quit the add-on module RF-TIMBER Pro. The ratios are displayed in the RFEM work window like internal forces of a load case.

To turn the display of design results on or off, use the button [Results on/off] shown on the left. To display the result values in the graphic, use the toolbar button [Show Result Values] to the right.

As the RFEM tables are of no relevance for the evaluation of the RF-TIMBER Pro results, you may deactivate them.



The design cases can be set as usual by means of the list in the RFEM menu bar.

The graphical display of results can be set in the *Display* navigator, by opening the *Results* folder and selecting the *Members* entry. The ratios are drawn *Two-Colored* by default.

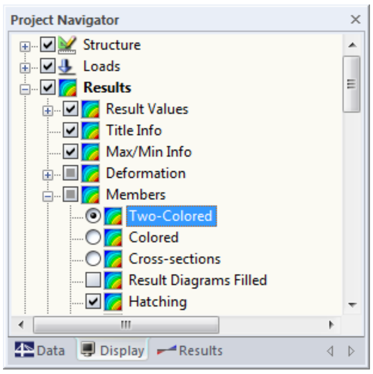


Figure 5.3: *Display* navigator: Results → Members

In case of a multicolor representation (options *Cross-sections* or *Colored*), the color panel is available, providing common control functions. The panel functions are described in detail in the RFEM manual, chapter 4.4.6, page 77.

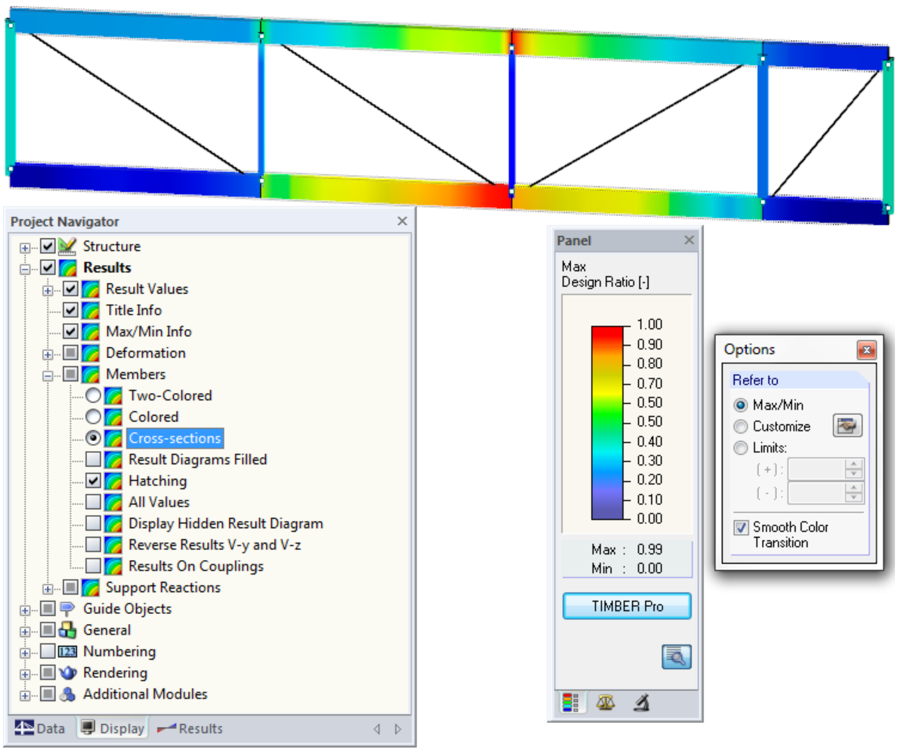


Figure 5.4: Design ratios with display option *Cross-sections*

In the *Factors* tab of the panel, you can scale the design results, as you know it from the member internal forces. If you enter the factor *0* in the input field *Member Diagrams*, the results will be represented without scaling but with an increased line thickness.

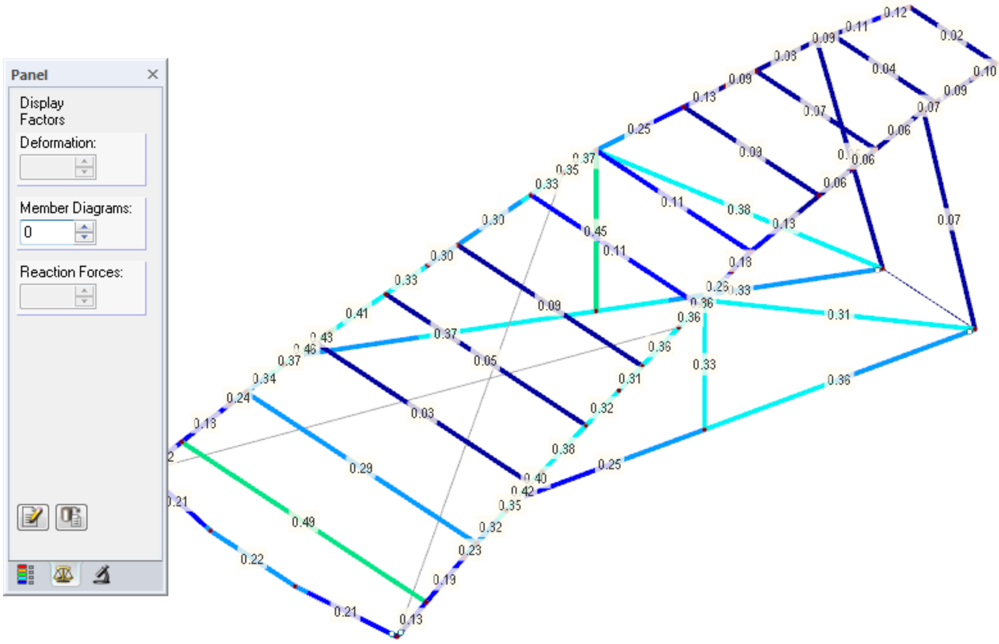


Figure 5.5: Panel tab *Factors*

The graphics can be transferred to the printout report like RFEM graphics (see chapter 6.2, page 52).

To return to the add-on module RF-TIMBER Pro, use the [RF-TIMBER Pro] button in the panel.

5.2 Result Diagrams

The result diagram can be useful if you want to see a member's result distribution displayed graphically. Select the member (or set of members) in the RF-TIMBER Pro results table by placing the cursor in the corresponding table row and open the result diagram by clicking the button shown on the left. You find the button below the first table part of the results table (see Figure 5.1, page 45).

The result diagrams are available in the RFEM graphic. To display the diagrams,

select **Result Diagrams on Selected Members** on the **Results** menu,

or use the button in the RFEM toolbar shown on the left.

A window opens showing the distribution of design results on the selected member or set of members.

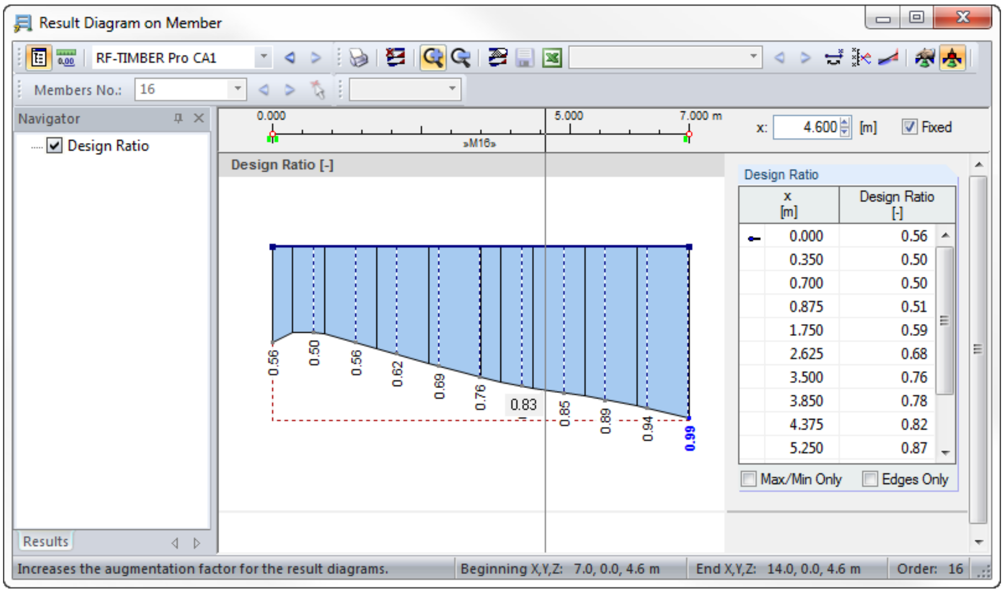
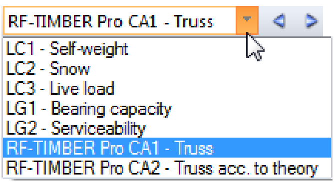


Figure 5.6: Dialog box *Result Diagram on Member*



Use the list in the toolbar above to choose the relevant RF-TIMBER Pro design case.

For more detailed information on the dialog box *Result Diagram on Member*, see the RFEM manual, chapter 10.5, page 311.

5.3 Filter for Results

In addition to the RF-TIMBER Pro results tables which already allow for a particular selection according to certain criteria because of their structure, you can use the filter options described in the RFEM manual to evaluate the RF-TIMBER Pro design results graphically.

Generally, you can take advantage of already defined partial views (see RFEM manual, chapter 10.9, page 321) used to group objects appropriately.

Filtering designs

The ratios can be used easily as filter criteria in the RFEM workspace. To apply this filter function, the panel must be displayed. If the panel is not active,

select **Control Panel (Colour scale, Factors, Filter)** on the **View** menu of RFEM

or use the toolbar button shown on the left.

The panel is described in the RFEM manual, chapter 4.4.6, page 77. The filter settings for the results must be defined in the panel tab *Color spectrum*. As this tab is not available for the two-colored results display, you have to set the display option *Colored* or *Cross-sections* in the *Display* navigator.

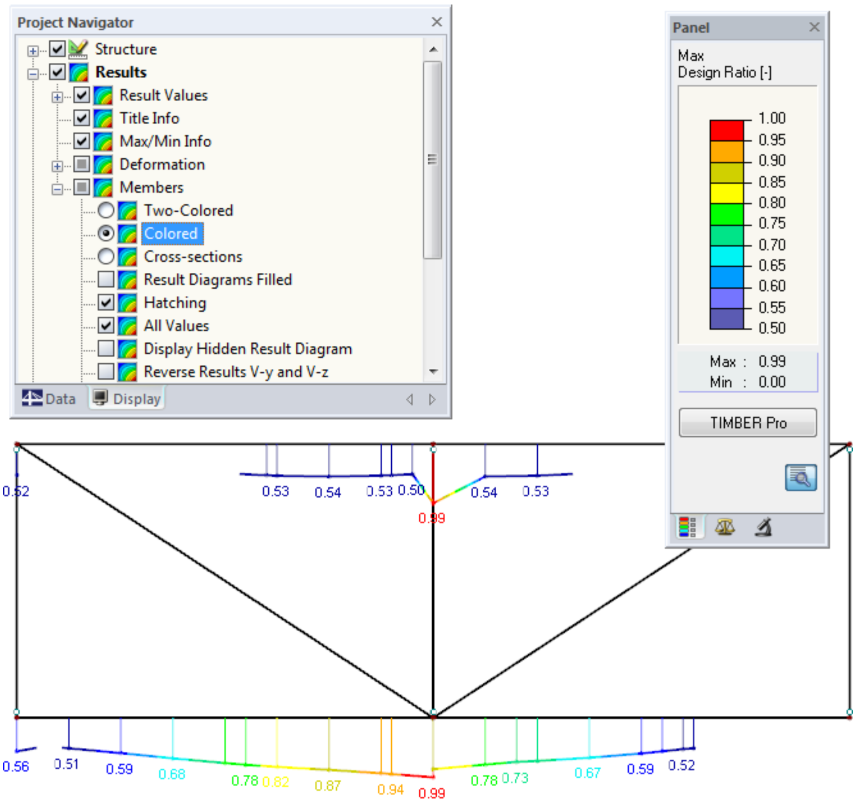


Figure 5.7: Filtering design ratios with adjusted color spectrum

For example, if you use a colored results display, you can use the panel to determine that only design ratios higher than 0.50 are displayed. Furthermore, the color spectrum can be adjusted in such a way that a color range covers a design ratio of 0.05 as shown in the figure above.

When you select the option *Display Hidden Result Diagram* (under *Results* → *Members* in the *Display* navigator), you can even display all stress ratio diagrams that do not fulfill the conditions. Those diagrams will be represented by dotted lines.

Filtering members

In the *Filter* tab of the control panel, you can define the numbers of the members whose results you want to be shown exclusively, which means filtered. The function is described in detail in the RFEM manual, chapter 10.9, page 326.

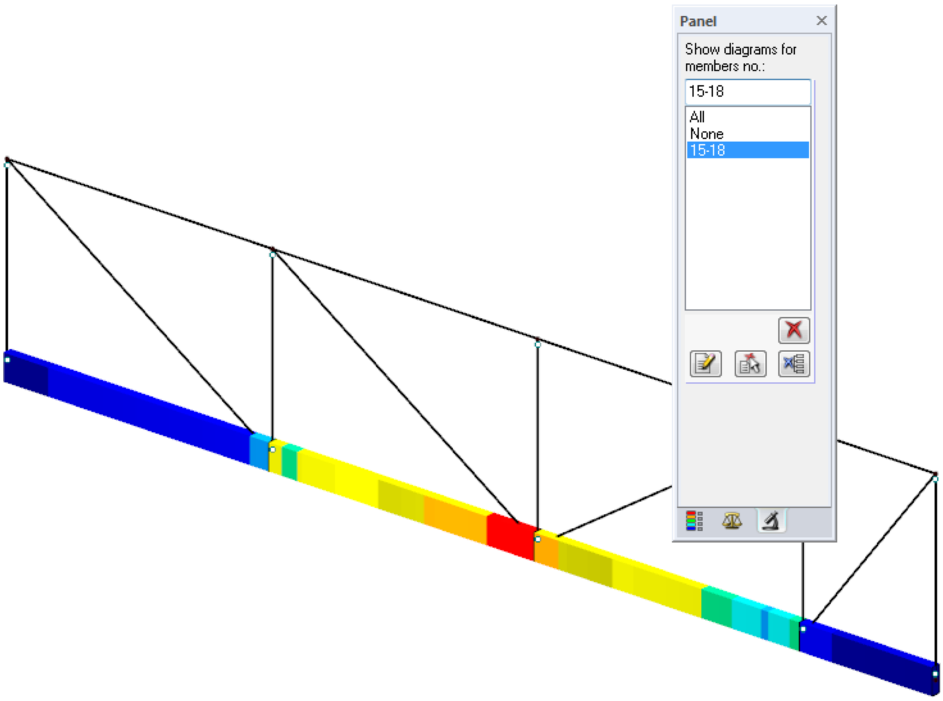


Figure 5.8: Member filter for ratios of bottom chord

Unlike the partial view function, the structure is now displayed completely in the graphic. The figure above shows the ratios in the bottom chord of a truss girder. The remaining members are displayed in the model but are shown without design ratios.

6. Printout

6.1 Printout Report

Similar to RFEM, the program generates a printout report for the RF-TIMBER Pro results that can be completed by graphics and descriptions. In the report you can also decide which input and output tables appear in the printout.

The printout report is described in detail in the RFEM manual. In particular, chapter 11.1.3.4 *Selecting Data of Add-on Modules* on page 338 provides information concerning the selection of input and output data in add-on modules.

For complex structures with a high number of design cases, it is recommended to split the data into several small printout reports which allows for a clearly-arranged printout and a faster work.

6.2 RF-TIMBER Pro Graphic Printout

Every picture that is displayed in the graphic window of the main program RFEM can be included into the printout report. This means that the design ratios displayed in the RFEM model can be prepared for the printout, too. The graphics can be integrated in the global printout report or sent directly to the printer. Printing graphics is described in detail in the RFEM manual, chapter 11.2.

Designs in the RFEM model

To print the RF-TIMBER Pro graphic currently displayed in the RFEM work window,

select **Print** on the **File** menu

or use the toolbar button shown on the left.

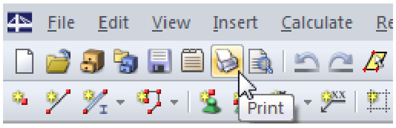


Figure 6.1: Button *Print* in the toolbar of the main window

Result diagrams

In the same way, you can integrate the result diagrams of members into the report by using the [Print] button. It is also possible, to print them directly.

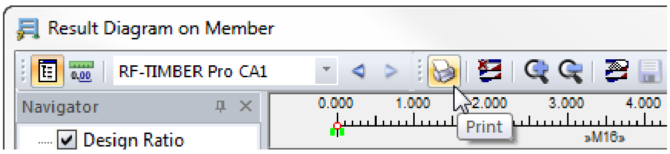


Figure 6.2: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog box opens:

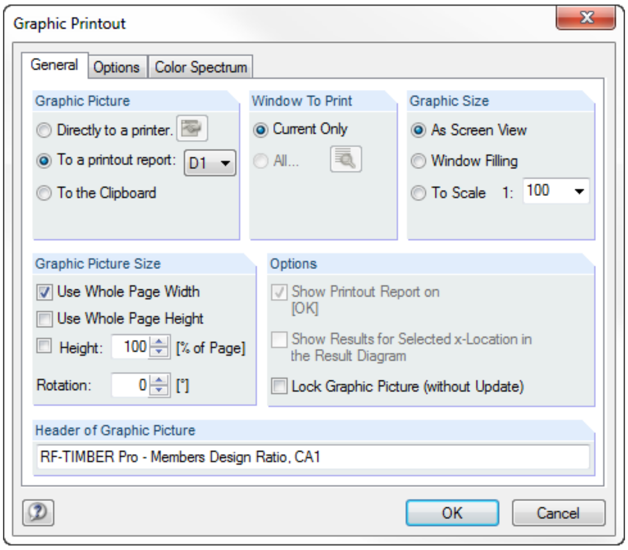
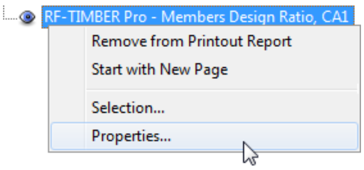


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in the RFEM manual, chapter 11.2, page 354. The RFEM manual also describes the *Options* and *Color Spectrum* tab.



A graphic from RF-TIMBER Pro that has been integrated in the printout report can be moved anywhere within the report by using the drag-and-drop function. In addition, it is possible to adjust imported graphics subsequently: Right-click the relevant entry in the navigator of the printout report and select *Properties* in the context menu. The dialog box *Graphic Printout* appears again, offering various options for adjustment.

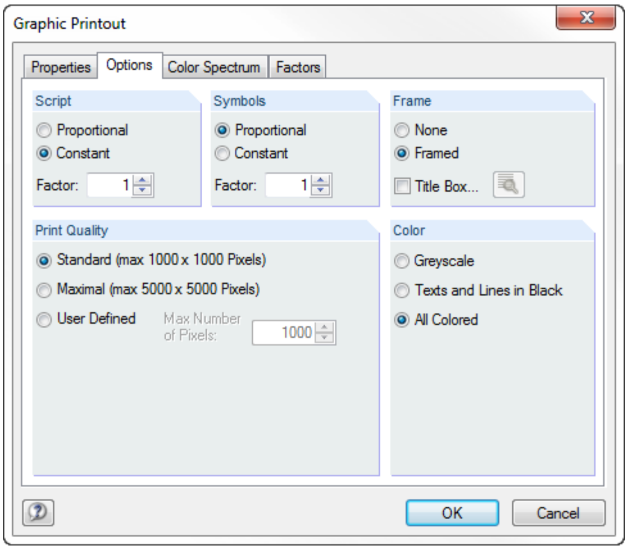


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

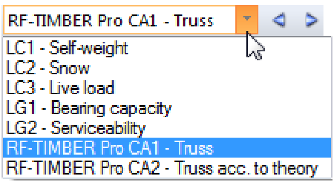
7. General Functions

This chapter describes some menu functions as well as export options for the design results.

7.1 RF-TIMBER Pro Design Cases

Members can be arranged in groups for different design cases. In this way, you can combine 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.



The design cases of RF-TIMBER Pro are available in the RFEM work window and can be displayed like a load case or load group by means of the toolbar list.

Create a new RF-TIMBER Pro case

To create a new design case,

select **New Case** on the **File** menu in the RF-TIMBER Pro add-on module.

The following dialog box appears.

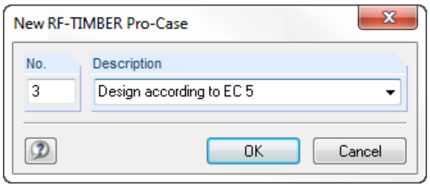


Figure 7.1: Dialog box *New RF-TIMBER Pro-Case*

In this dialog box, enter a *No.* (which is not yet assigned) and a *Description* for the new design case. When you click [OK], table 1.1 *General Data* opens where you can enter the new design data.

Rename an RF-TIMBER Pro case

To change the description of a design case subsequently,

select **Rename Case** on the **File** menu in the RF-TIMBER Pro add-on module.

The dialog box *Rename RF-TIMBER Pro-Case* appears.

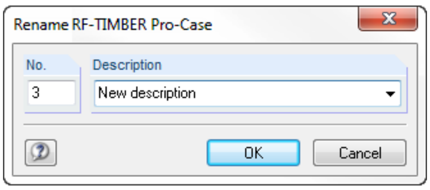


Figure 7.2: Dialog box *Rename RF-TIMBER Pro-Case*

Copy an RF-TIMBER Pro case

To copy the input data of the current design case,

select **Copy Case** on the **File** menu in the RF-TIMBER Pro add-on module.

The dialog box *Copy RF-TIMBER Pro-Case* appears where you can specify the number and description of the new case.

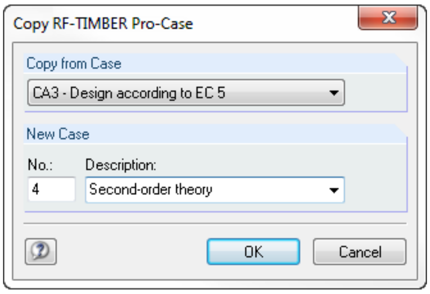


Figure 7.3: Dialog box *Copy RF-TIMBER Pro-Case*

Delete an RF-TIMBER Pro case

To delete design cases,

select **Delete Case** on the **File** menu in the RF-TIMBER Pro add-on module.

In the dialog box *Delete Cases*, you can select the relevant design case in the *Available Cases* list to delete it by clicking [OK].

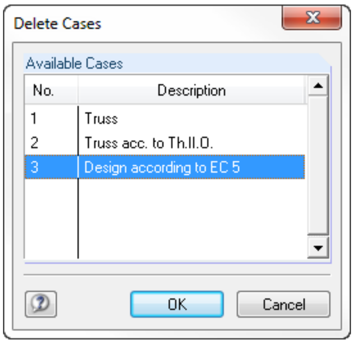
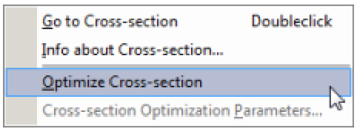


Figure 7.4: Dialog box *Delete Cases*

7.2 Cross-section Optimization

As mentioned in chapter 2.3, RF-TIMBER Pro offers you the possibility to optimize cross-sections. Select the relevant cross-section by ticking its check box in column C or D in table 1.3 *Cross-sections* (see Figure 2.10, page 17).



You can also start the cross-section optimization out of the results tables by using the context menu.

During the optimization process, RF-TIMBER Pro determines the cross-section within the same cross-section table that fulfills the analysis requirements in the most optimal way, that means comes as close as possible to the maximum possible ratio specified in the *Details* dialog box (see Figure 3.5, page 33). The required cross-section properties will be determined with the RFEM internal forces. If another cross-section proves to be more favorable, it will be used for the design. In this case, two cross-sections will be displayed on the right of table 1.3 as shown in Figure 7.6, the original cross-section from RFEM and the optimized one.

A dialog box with detailed specifications appears when you tick the check box for optimization.

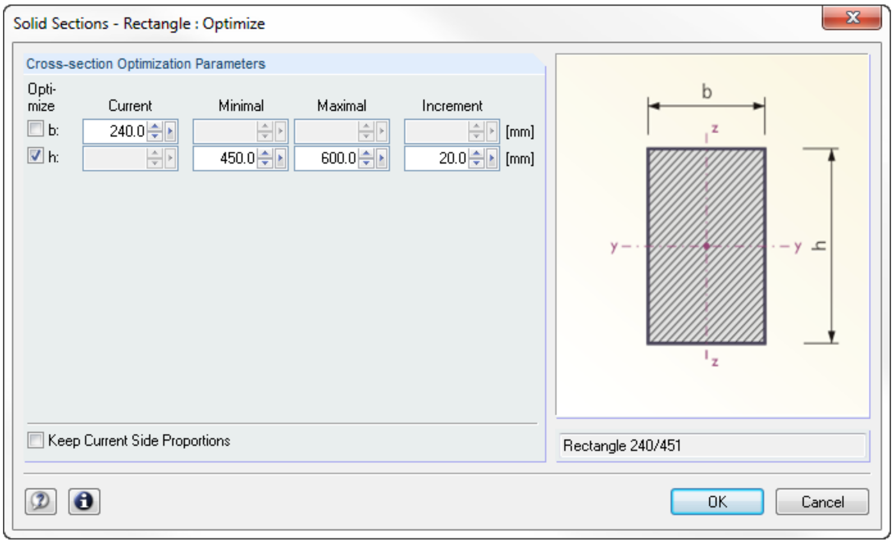


Figure 7.5: Dialog box *Solid Sections - Rectangle : Optimize*

By ticking the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. The ticked check box enables the *Minimal* and *Maximal* columns where you specify the upper and lower limits of the parameter. The *Increment* column determines the interval in which the size of this parameter varies during the optimization process.

If you want to *Keep Current Side Proportions*, tick the corresponding check box. In addition, you have to select both parameters for the optimization.

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

You do not need to transfer the modified cross-sections to RFREM manually: Set table 1.3 *Cross-sections*, and then

select **Export All Cross-sections to RFEM** on the **Edit** menu.

Also the context menu available in table 1.3 provides options to export optimized cross-sections to RFEM.

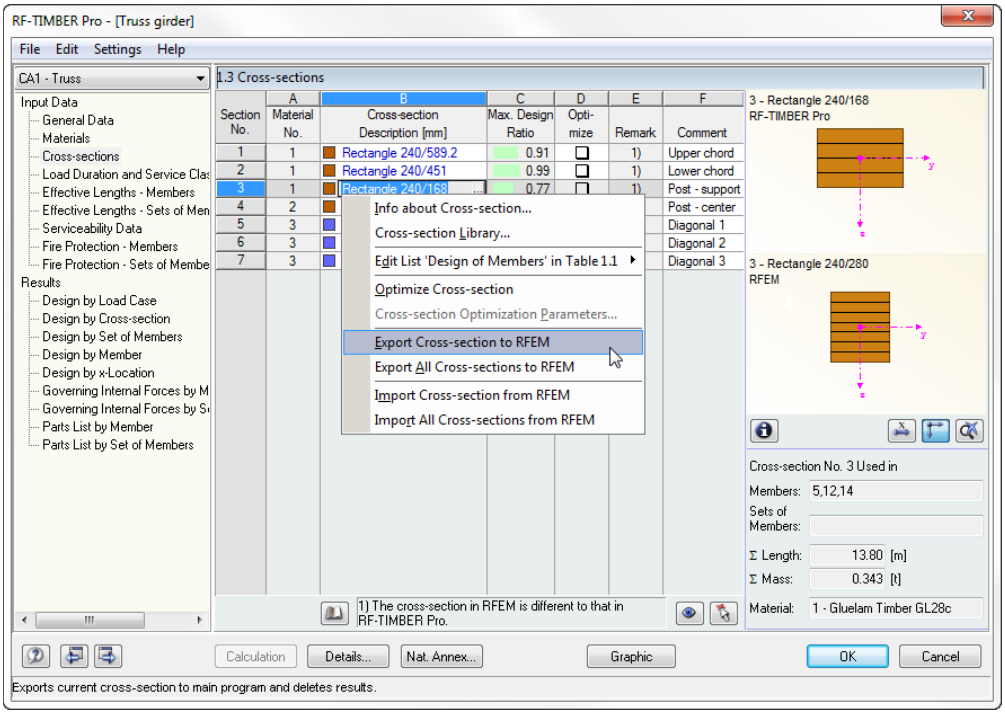


Figure 7.6: Context menu in table 1.3 *Cross-sections*

Before the changed cross-sections are transferred to RFEM, a security query appears, because the transfer requires the deletion of results. When you confirm the query and start the [Calculation] in RF-TIMBER Pro, the RFEM internal forces as well as the RF-TIMBER Pro designs will be determined and performed in one single calculation run.

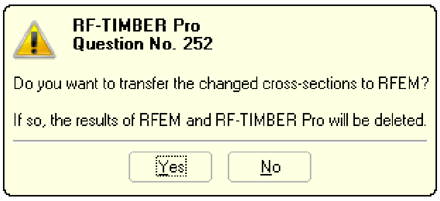


Figure 7.7: Query before transfer of modified cross-sections to RFEM

By using the menu functions described above, you can also reimport the original RFEM cross-sections to RF-TIMBER Pro. Please note that this option is only available in table 1.3 *Cross-sections*.

If you optimize a tapered member, the program modifies the member's start and end and interpolates the second moments of area for the intermediate locations linearly. As 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 this case, it is recommended to divide the taper into several single members whose start and end cross-sections have minor cross-section differences.

7.3 Material Export to RFEM

When you have changed materials in table 1.2 of RF-TIMBER Pro, you can export the modified materials to RFEM, similar to the export of modified cross-sections. It is also possible to import the originally used materials from RFEM. Materials modified in the add-on module are highlighted in blue.

You do not need to transfer the modified materials manually to RFEM.
Set table 1.2 *Materials*, and then

select **Export All Materials to RFEM** on the **Edit** menu.

Also the context menu of table 1.2 provides options to transfer modified materials to RFEM.

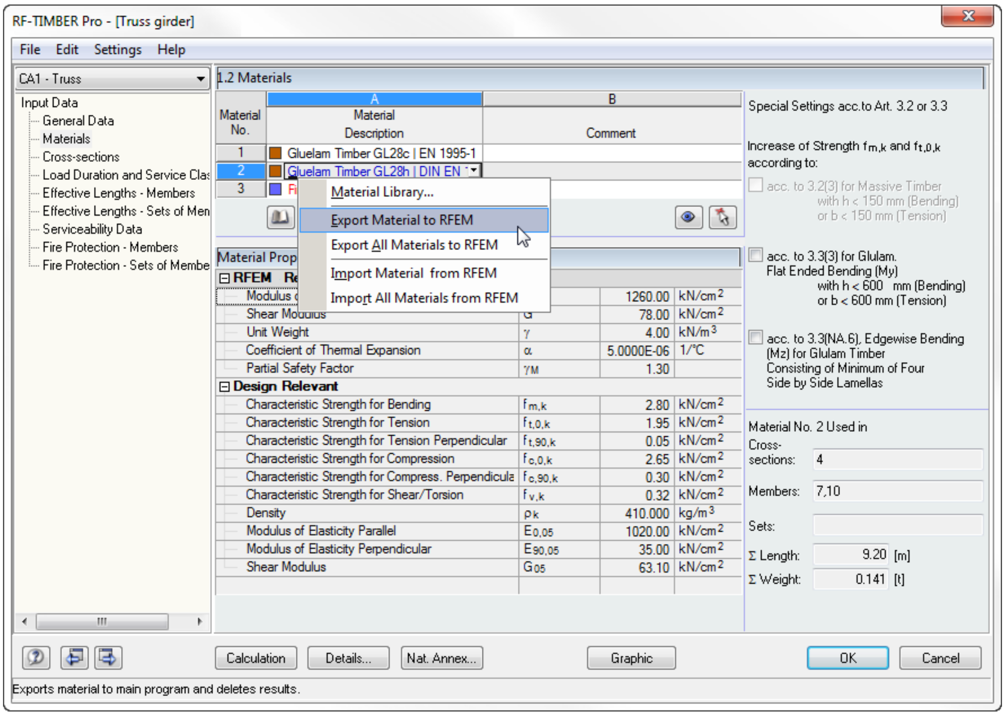


Figure 7.8: Context menu of table 1.2 *Materials*

Before the changed materials are transferred to RFEM, a security query appears because the transfer requires the deletion of results. When you confirm the query and start the [Calculation] in RF-TIMBER Pro, the RFEM internal forces as well as the TIMBER Pro designs will be determined and performed in one single calculation run.

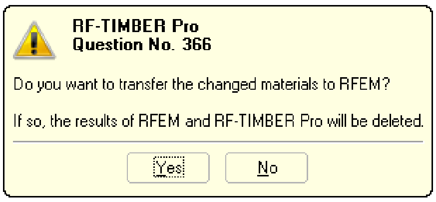


Figure 7.9: Query before transfer of modified materials to RFEM

7.4 Units and Decimal Places

The units and decimal places for RFEM and all add-on modules are managed in one global dialog box. In the add-on module RF-TIMBER Pro, you can use the menu to define the units. To open the corresponding dialog box,

select **Units and Decimal Places** on the **Settings** menu.

The program opens the following dialog box that you already know from RFEM. The add-on module RF-TIMBER Pro is preset.

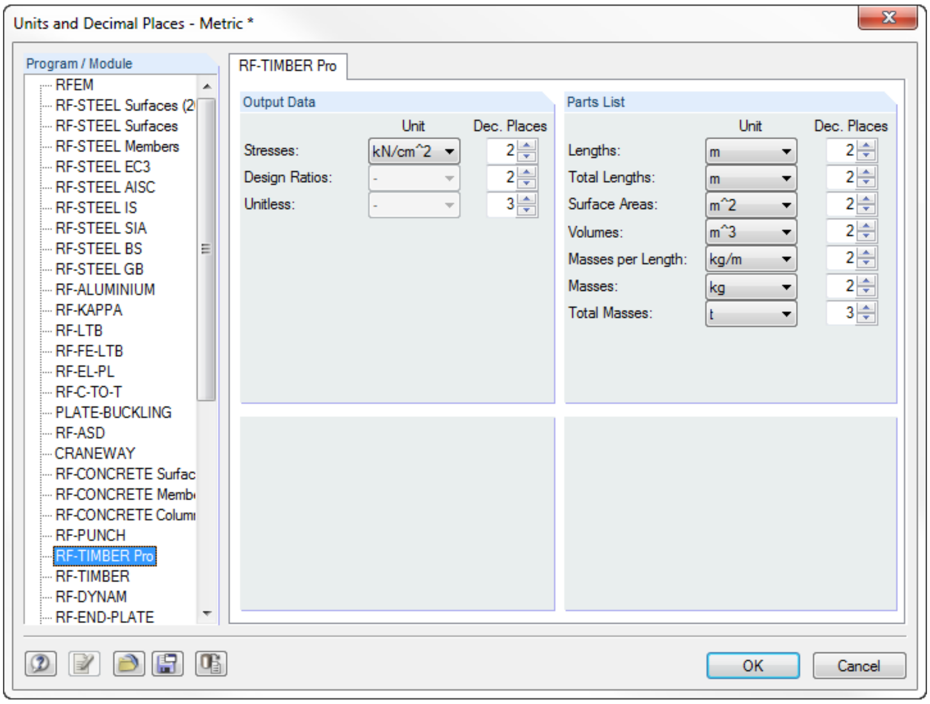


Figure 7.10: Dialog box *Units and Decimal Places*

The settings can be saved as user profile to reuse them in other structures. The functions are described in the RFEM manual, chapter 12.6.2, page 453.

7.5 Export of Results

The designs can be provided for other programs in various ways.

Clipboard

To copy cells selected in the RF-TIMBER Pro results tables to the clipboard, use the keyboard keys [Ctrl]+[C]. To insert the cells, 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 the RF-TIMBER Pro add-on module can be printed into the global printout report (see chapter 6.1, page 52) to export them subsequently. Then, in the printout report,

select **Export to RTF File or BauText** on the **File** menu.

The function is described in detail in the RFEM manual, chapter 11.1.11, page 350.

Excel / OpenOffice

RF-TIMBER Pro provides a function for the direct data export to MS Excel, OpenOffice.org Calc or the file format CSV. To open the corresponding dialog box,

select **Export Tables** on the **File** menu in the RF-TIMBER Pro add-on module.

The following export dialog box appears.

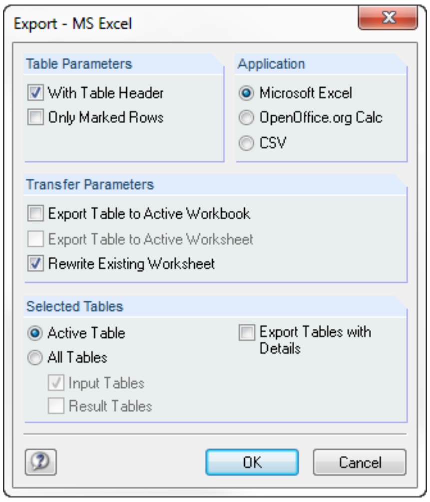


Figure 7.11: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking [OK]. Excel or OpenOffice will be started automatically. It is not necessary to run the programs in the background.

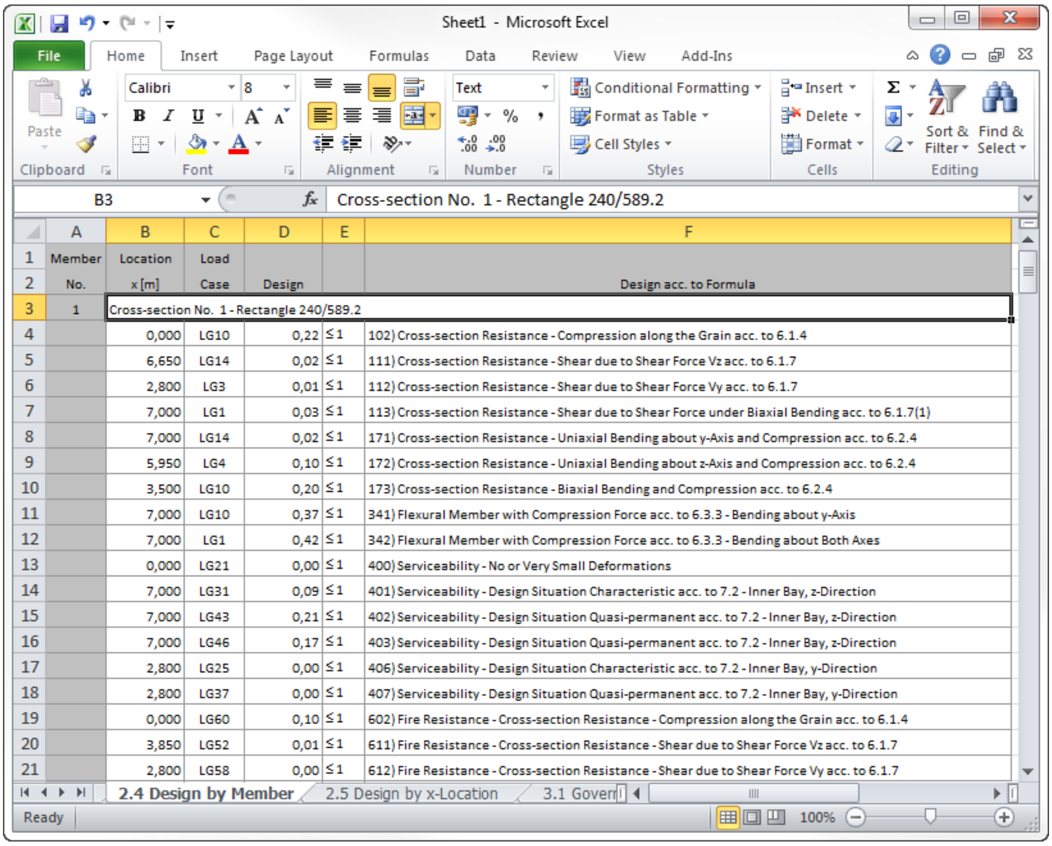


Figure 7.12: Results in *Excel*

8. Example

We perform the design according to EN 1995-1-1 for a wood column that is restrained and stressed by compression and bending. Moreover, it is supported on the free end in direction Y. The example is described in the German timber construction book [8], page 236.

8.1 Structure and Loads

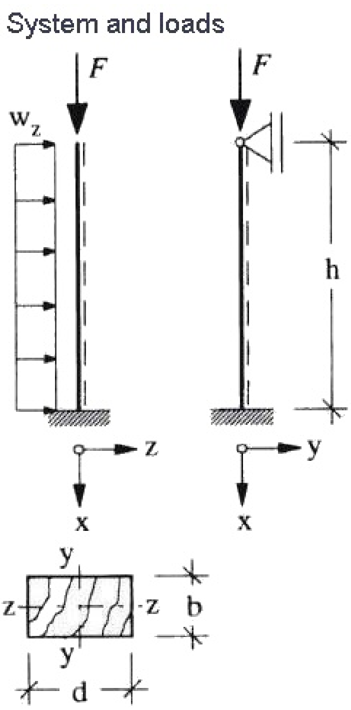


Figure 8.1: System and loads according to [8]

Structure

Cross-section:	b/d = 14/22 cm
Material:	CT C24
Depth:	h = 3.20 m
Service class:	1
LDC:	permanent

Load

LC 1 Self-weight:	F = 45 kN
LC 2 Wind:	w = 1.5 kN/m

Design values

$N_d = 1.35 \cdot F = 1.35 \cdot 45 \text{ kN} = 60.75 \text{ kN}$ ($k_{mod} = 0.6$)

$q_d = 1.5 \cdot w = 1.5 \cdot 1.5 \text{ KN/m} = 2.25 \text{ kN/m}$ ($k_{mod} = 0.9$)

8.2 Calculation with RFEM

The system as well as the loads in both load cases are modeled in RFEM as 3D structure. We deactivate the automatic consideration of the self-weight when we create LC 1 because it is also neglected in the example of the German timber construction book.

We superimpose the load cases for the basic combination with the corresponding partial safety factors in a load combination. They will be calculated according to the linear static analysis. For the design in RF-TIMBER Pro it is important to define both load cases with the "permanent" criterion.

RFEM determines the following diagrams of internal forces.

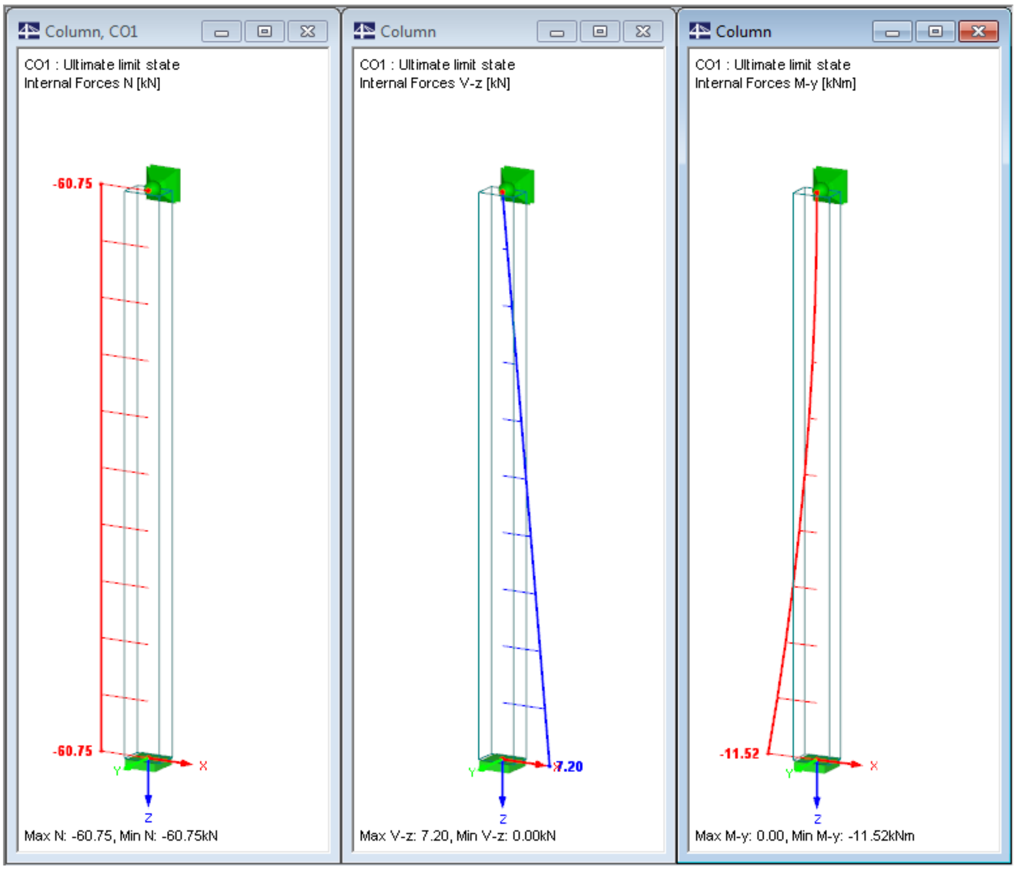


Figure 8.2: Internal forces N, V_z and M_y

The analyzed internal forces are equivalent to the ones mentioned in [8], page 237.

8.3 Design with RF-TIMBER Pro

8.3.1 Ultimate Limit State Design

In table 1.1 *General Data*, we select the load combination **CO1** for the *Ultimate Limit State* design.

We perform the design according to **EN 1995-1-1** with the German national annex **DIN**.

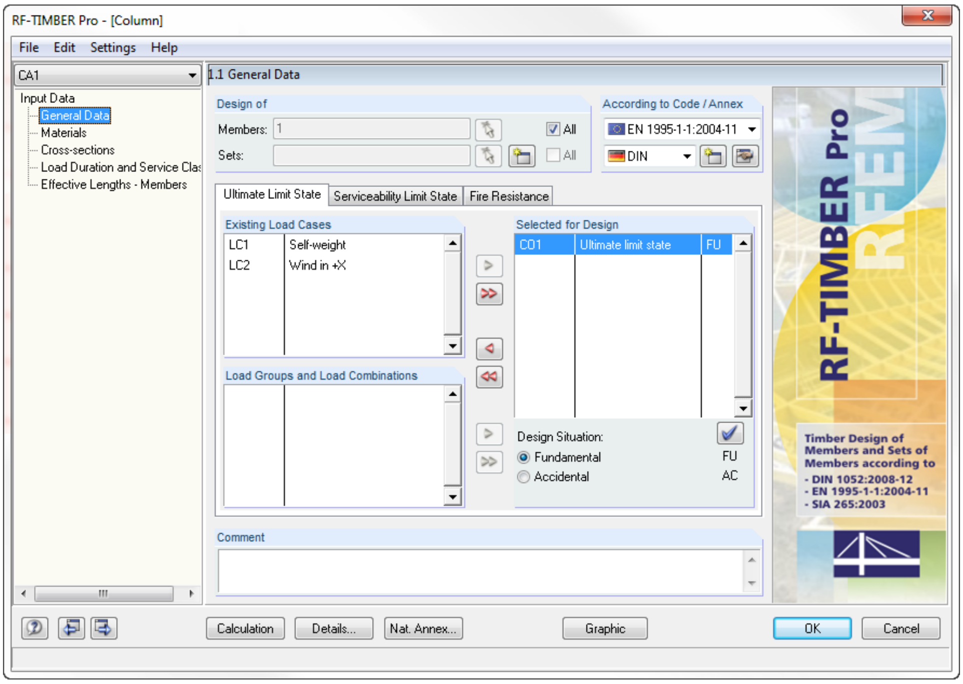


Figure 8.3: Table 1.1 *General Data*

Table 1.2 *Materials* and table 1.3 *Cross-sections* present the characteristic stiffnesses of the selected material as well as the available cross-section.

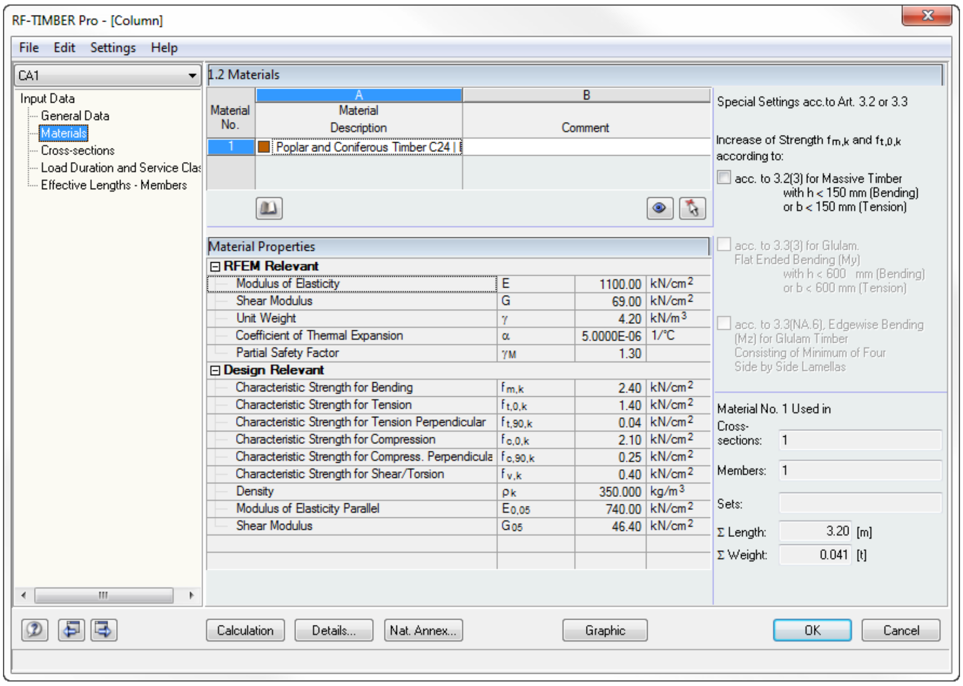


Figure 8.4: Table 1.2 *Materials*

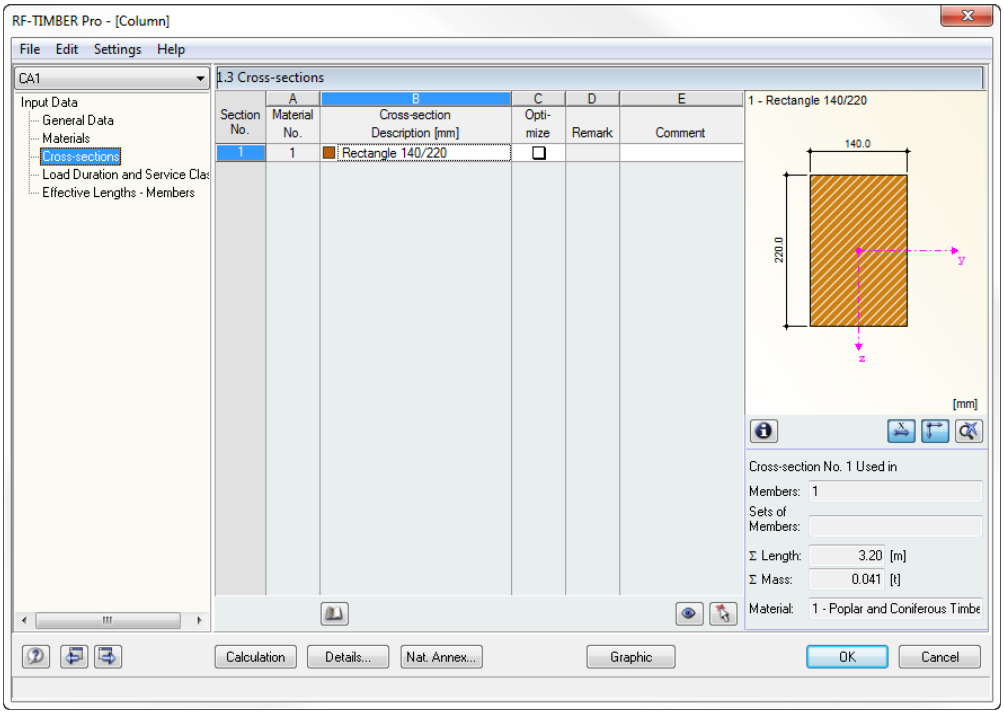


Figure 8.5: Table 1.3 *Cross-sections*

In table 1.4 we define the load duration and the service class. The factor k_{mod} of CO1 will be calculated from the LDC of the contained load cases by taking into account the service class.

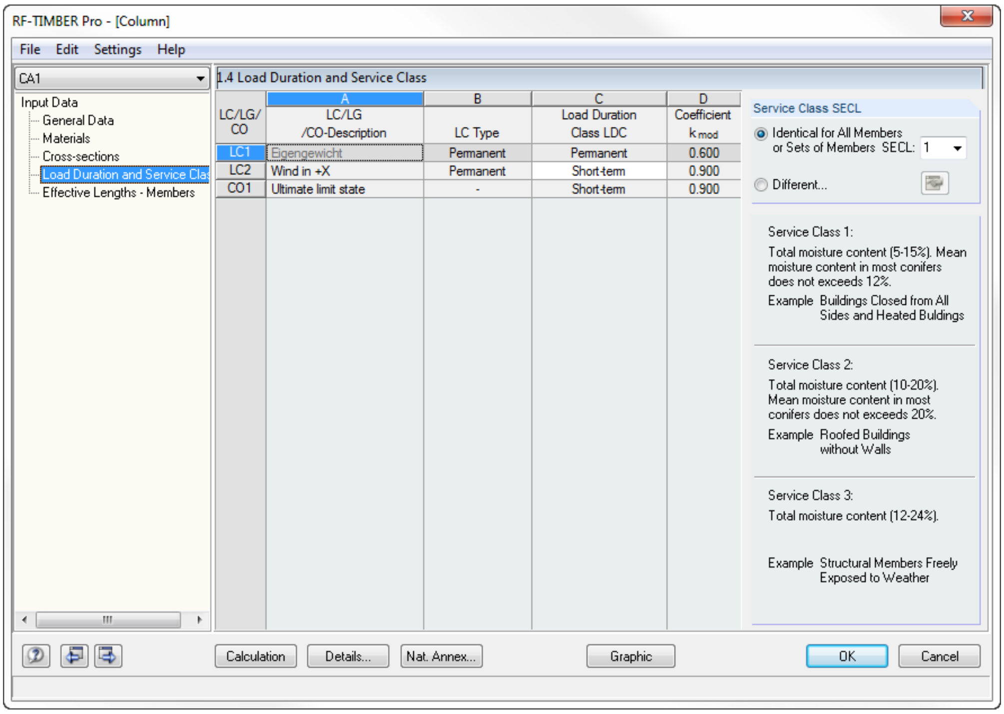


Figure 8.6: Table 1.4 *Load Duration and Service Class*

In table 1.5 *Effective Lengths* we specify the buckling lengths of the column. The example provides the Euler buckling modes 1 and 3 with the coefficients for effective length $\beta_{ef,y}$ = **2.0** and $\beta_{ef,z}$ = **0.7**.

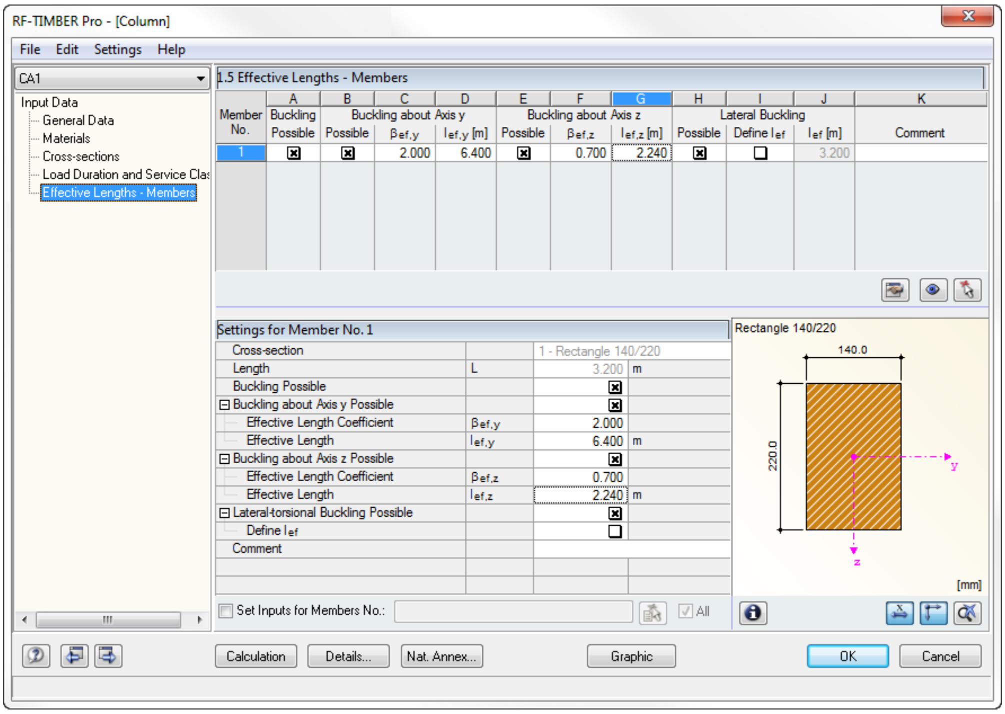


Figure 8.7 Table 1.5 *Effective Lengths - Members*

We start the calculation by clicking the [Calculation] button.

After the calculation table 2.1 *Design by Load Case* appears showing the governing design results.

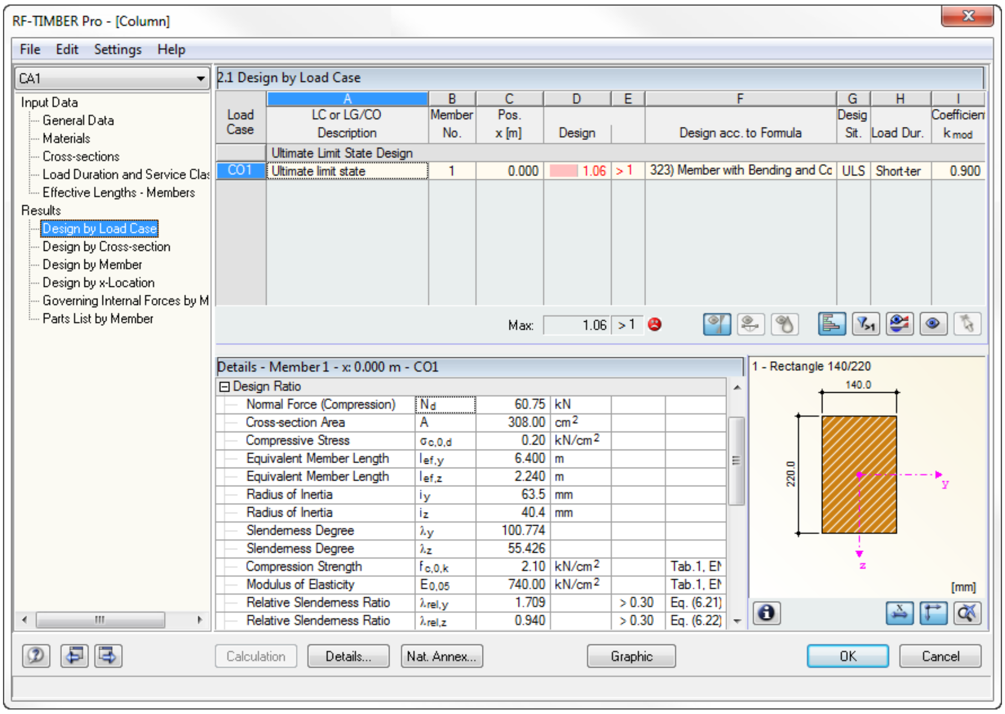


Figure 8.8: Table 2.1 *Design by Load Case*

The *Details* displayed in the lower part of the table correspond to the designs described in [8].

$$\sigma_{m,d} = \frac{M_y}{S_y} = \frac{1152\,kN/cm}{1129.33\,cm^3} = 1.02\,kN/cm^2$$

$$\sigma_{c,0,d} = \frac{N}{A} = \frac{60.75\,kN}{308\,cm^2} = 0.197\,kN/cm^2$$

According to [4] we have to reduce the allowable compressive stress for the stability analysis (buckling design) by the buckling coefficient k_c that depends on the slenderness λ.

$$i_y = \frac{d}{\sqrt{12}} = \frac{22\,cm}{\sqrt{12}} = 6.35\,cm$$

$$i_z = \frac{b}{\sqrt{12}} = \frac{14\,cm}{\sqrt{12}} = 4.04\,cm$$

The slenderness according to [4] is calculated as follows:

$$\lambda_z = \frac{S_k}{i_z} = \frac{224\,cm}{4.04\,cm} = 55.4$$

$$\lambda_y = \frac{S_k}{i_y} = \frac{640\,cm}{6.35\,cm} = 100.8$$

Buckling coefficient k_c according to [4] section 6.3.2 (intermediate values can be interpolated linearly):

$$k_{c,z} = 0.733$$
$$k_{c,y} = 0.301$$

Stability analysis

Design according to [4] eq. (6.23):

$$f_{m,d} = \frac{f_{m,k} \cdot k_{mod}}{\gamma_m} = \frac{2.4 \cdot 0,9}{1.3} = 1.66\,kN/cm^2$$

Design: $$\eta_1 = \frac{\frac{N}{A}}{k_{c,y} \cdot f_{c,o,d}} + \frac{\frac{M}{S}}{k_m \cdot f_{m,y,d}} = \frac{0.197}{0.301 \cdot 1.45} + \frac{1.02}{1.66} = 1.066 > 1$$

Shear design

Design of shear from transverse force according to [4] section 6.1.7:

$$f_{v,d} = \frac{f_{v,k} \cdot k_{mod}}{\gamma_m} = \frac{0.4 \cdot 0.9}{1.3} = 0.277\,kN/cm^2$$

The shear stresses can be determined by the stress point details with the corresponding statical moments of area (see Figure 2.12, page 18).

$$\tau_d = \frac{V_y \cdot Q_{z,i}}{I_z \cdot t_i} + \frac{V_z \cdot Q_{y,i}}{I_y \cdot t_i} = \frac{7.2\,kN \cdot 847\,cm^3}{5030.67\,cm^4 \cdot 22\,cm} = 0.055\,kN/cm^2$$

Design: $$\frac{\tau_d}{f_{v,d}} = \frac{0.055}{0.277} = 0.199 \le 1$$

8.3.2 Serviceability Limit State Design

We create another load combination in RFEM with different partial safety factors for the serviceability limit state design.

CO2 = 1.0 · LC1/s + 1.0 · LC2/s

In the add-on module RF-TIMBER Pro, we select the load combination **CO2** for the design in the *Serviceability Limit State* tab of table 1.1 *General Data*.

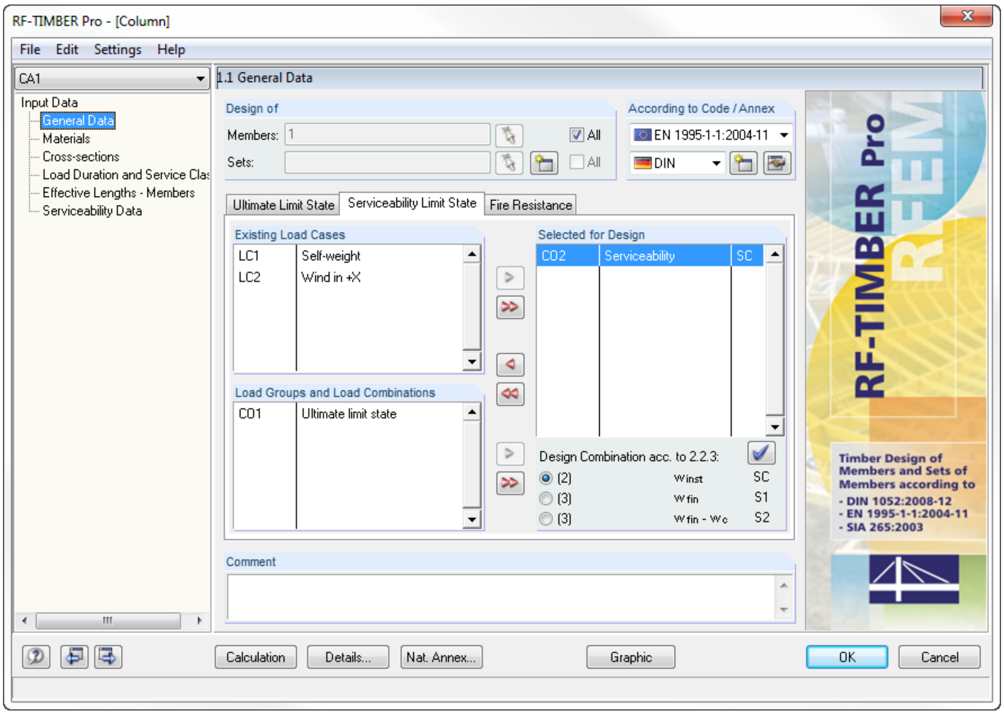


Figure 8.9: Table 1.1 *General Data*, tab *Serviceability Limit State*

We click the blue check mark button to assign the *Design Combination acc. to 2.2.3* **(2)** w_{inst} *SC* to the selected load combination.

Then, we enter member **1** in table 1.8 *Serviceability Data* (see the following Figure 8.10).

We do not modify the reference length but we restrict the direction to **z**. As the beam has no support in this direction, we select the beam type **Cantilever End Free** in the list.

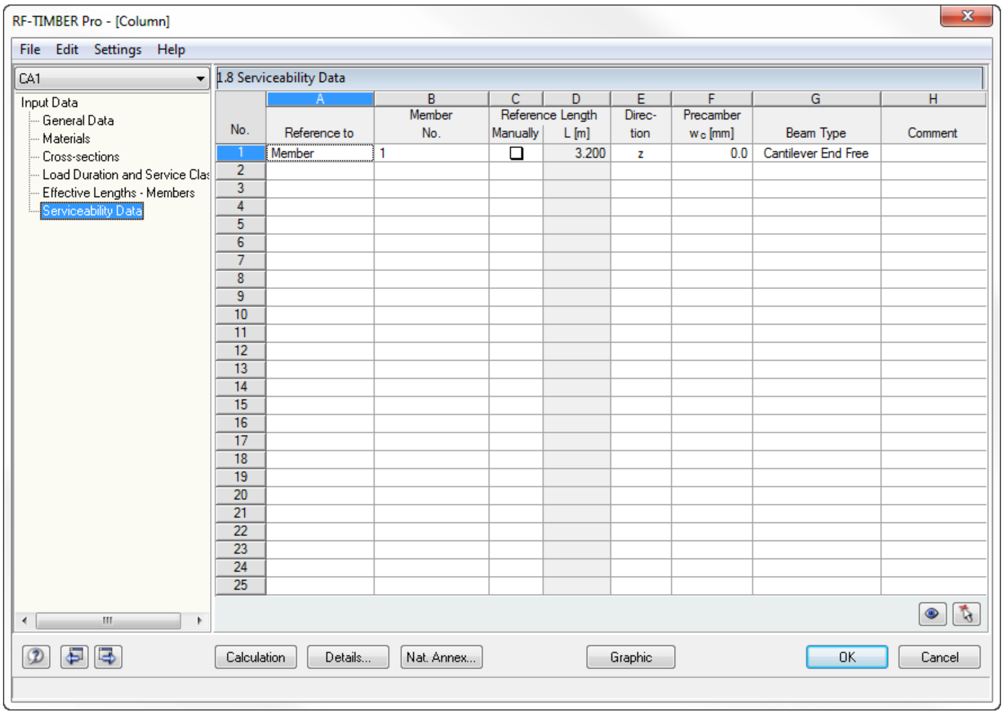


Figure 8.10: Table 1.8 *Serviceability Data*

We have to change a setting for the control calculation. We use the [Details] button to open the dialog box below where we specify our selection in the *Serviceability* tab: We want the deformation to be related to the **Undeformed System**.

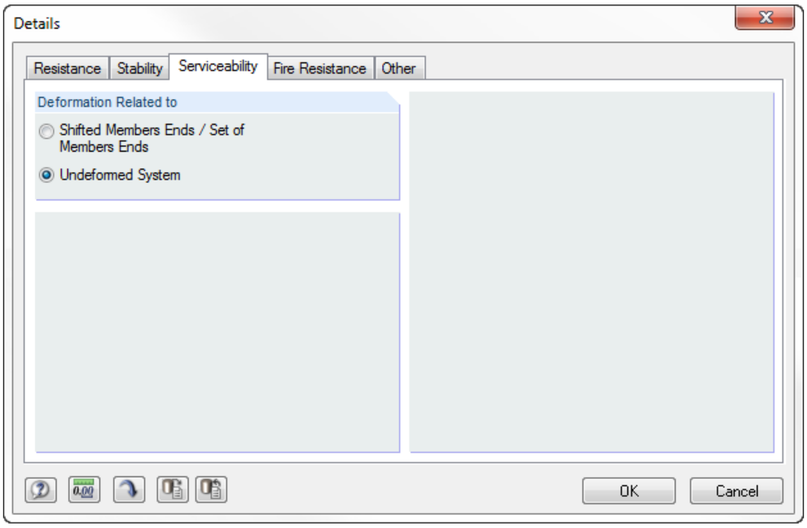


Figure 8.11: Dialog box *Details*, tab *Serviceability*

In the example of [8] an elastic modulus of 10,000 MN/m² is applied. Thus, a new material with corresponding characteristics would have to be defined in RFEM. However, to simplify matters, we use the default value of 11,000 MN/m² for the following control calculation.

$$w_{inst} = \frac{w \cdot h^4}{8 \cdot E \cdot I_y} \leq \frac{l}{150}$$

$$w_{inst} = \frac{1.5 \cdot 3.2^4}{8 \cdot 11000 \cdot 12422.70} \cdot \frac{10^{-1}}{10^{-8}} = 1.44\,\text{cm} < 2.13\,\text{cm} = \frac{320}{150}$$

Design: $\frac{w_{inst}}{w_{limit}} = \frac{1.44\,\text{cm}}{2.13\,\text{cm}} = 0.676 < 1$

This result is also displayed in the RF-TIMBER Pro results table 2.1 *Design by Load Case* under the table entry *Serviceability Limit State Design*.

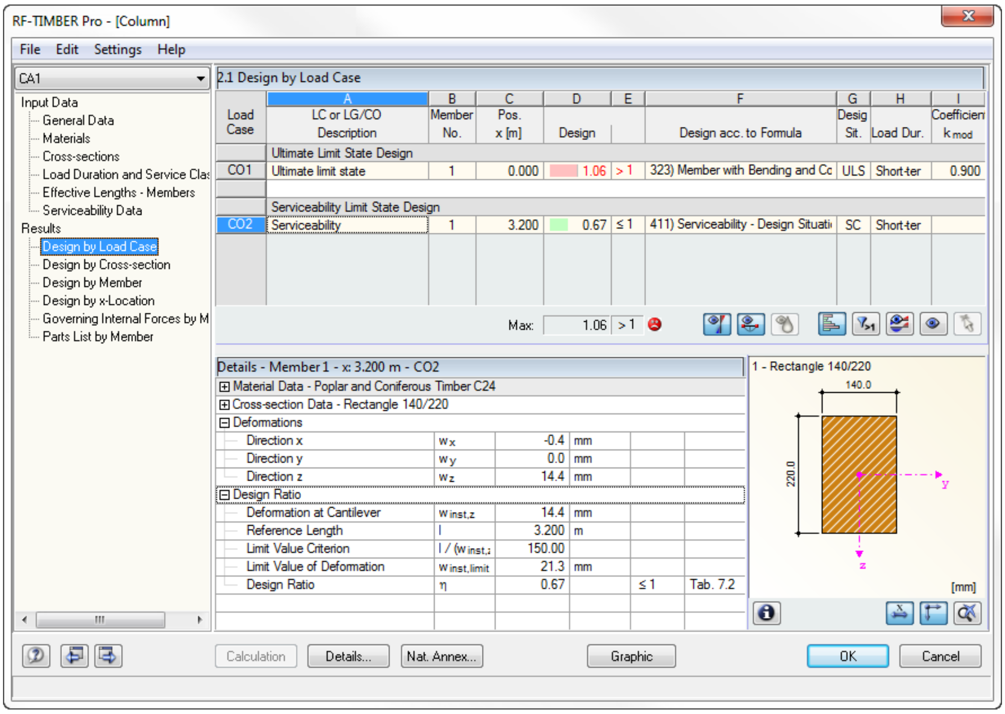


Figure 8.12: Table 2.1 *Design by Load Case*

A Literature

[1] DIN 1052:2008-12: Entwurf, Berechnung und Bemessung von Holzbautragwerken – Allgemeine Bemessungsregeln und Bemessungsregeln für den Hochbau, Dezember 2008

[2] DIN 4102-22:2004-11: Brandverhalten von Baustoffen und Bauteilen, Teil 22: Anwendungsnorm zu DIN 4102-4 auf der Bemessungsbasis von Teilsicherheitsbeiwerten

[3] Erläuterungen zur DIN 1052:2004-08, DGfH Innovations- und Service GmbH, München, 2. Auflage März 2005

[4] DIN EN 1995-1-1:2010-12: Bemessung und Konstruktion von Holzbauten
Teil 1-1 Allgemeines – Allgemeine Regeln und Regeln für den Hochbau, Dezember 2010

[5] DIN EN 1995-1-2:2006-10: Bemessung und Konstruktion von Holzbauten
Teil 1-2 Allgemeines – Tragwerksbemessung für den Brandfall, Oktober 2006

[6] SIA 265:2003: Holzbau, 2003

[7] SCHNEIDER Bautabellen, Werner Verlag, 19. Auflage 2010

[8] Holzbau-Taschenbuch, Band 1, 9. Auflage, Ernst & Sohn

B Index