Version
February 2013

Program

RF-TENDON

Definition of Tendons in Pre-stressed Concrete Members

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.de

Contents

1. Introduction

1.1 Overview

RF-TENDON is an add-on module for the program RFEM for calculation of pre-tensioned and post-tensioned prestressed concrete beams according to EN 1992-1-1 and 1992-2, with or without a national application document.

It is necessary to first define a model in RFEM. The model can contain concrete members, cross-sections and materials, external loads, load cases including load cases for prestressing and load combinations. Once RF-TENDON is started, the user selects the concrete members to be pre-stressed. Then the user is navigated by going through individual design steps:

- input of tendon layout, material and other characteristics of prestressing
- calculation of loads equivalent to the effects of prestressing
- design of prestressing forces using load-balancing method
- calculation of short-term losses of prestressing due to friction, anchorage set and steel relaxation
- export of equivalent loads to RFEM and structural analysis

A detailed check of input data for sections of design members can continue after starting add-on module **RF-TENDON Design**. The preparation of inputs to perform a detailed check is described in chapter 3.6.2 Check Positions (preparation for RF-TENDON Design) and a short description of the module possibilities is also enclosed in that chapter.

1.2 RF-TENDON Team

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

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Alexander Meierhofer
IDEA RS s.r.o

Programming

IDEA RS s.r.o.
Dis. Jiří Šmerák

Program supervision

IDEA RS s.r.o.
Dipl.-Ing. (FH) Alexander Meierhofer
M.Sc. Dipl.-Ing. (FH) Frank Lobisch
Ing. Jan Fráňa
Ing. Bohdan Šmid

Manual, help system and translation

IDEA RS s.r.o
Dipl.-Ing. (FH) Alexander Meierhofer
Dipl.-Ing. (FH) Robert Vogl
Ing. Bohdan Šmid
Ing. Chelsea Jennings
Dipl.-Ü. Gundel Pietzcker

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
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) Andreas Hörold
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
Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl

1.3 Getting Started

Before beginning the installation of both RF-TENDON and RF-TENDON Design, it is necessary to check if .NET Framework 4 is installed on your computer. The installation cannot be launched without .NET Framework 4 being installed.

Notice:

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.

1.4 Terminology

1.4.1 General

Part of Member – is a basic entity which is imported from the analytical model. It is not a finite element. Each *part of the member* has its own geometry, and is therefore one geometrical entity (line, circle arc, parabolic arc). This geometrical entity contains definition of its local coordinate system (LCS). The cross-section including its rotation and eccentricity is defined at the beginning and the ending of the member part.

Example:

Member P1 is defined by a polygon in RFEM. The polygon is defined by points 1 to 5 and it consists of four segments. The member in RF-TENDON will consist of four *member parts*.

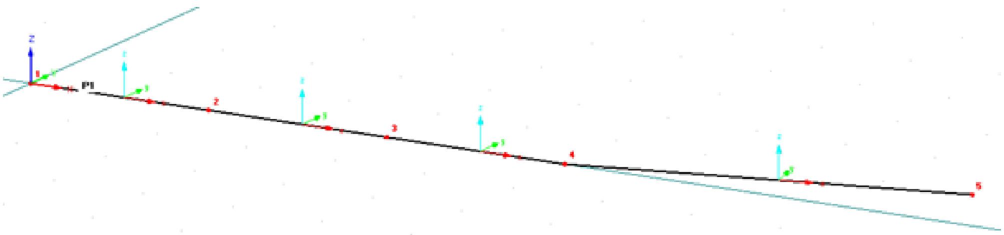


Figure 1.1: Member P1 - Polygon

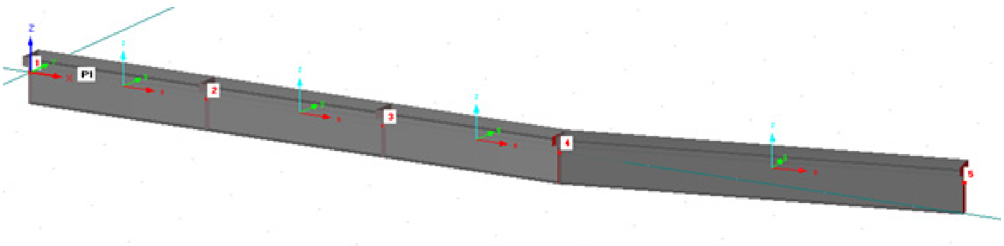


Figure 1.2: Member P1

LCS is defined as follows:

X-axis is vector identical with tangent in any point of the part of member and with orientation identical with the geometrical entity.

The direction of the y-axis or z-axis is defined according to the type of project or according to the settings. E.g. the z-axis of the LCS is parallel to the Z-axis of the global coordinate system or the z-axis is defined by a vector. The third axis is calculated to be perpendicular to those two axes.

The coordinate system is right-handed.

Identical LCS – Two local coordinate systems are identical if both start at the same point and the angle between the corresponding axes is zero.

Member – A 1D element of the analytical (static) model, which consists of at least one member part. If a member consists of more member parts, all parts of the member are connected in a row so that the ending point of one part is also the beginning point of the following member part. The local coordinate systems of individual parts of the member in this point may (but not necessarily) be identical.

Design Member – A one 1D element or group of consecutive 1D elements of analytical model (members). Consecutive members must have a common node of the analytical model and must have the same orientation – the ending point of one member is the beginning point of the following member. A design member is analyzed as a whole and prestressing reinforcement is designed for the design member.

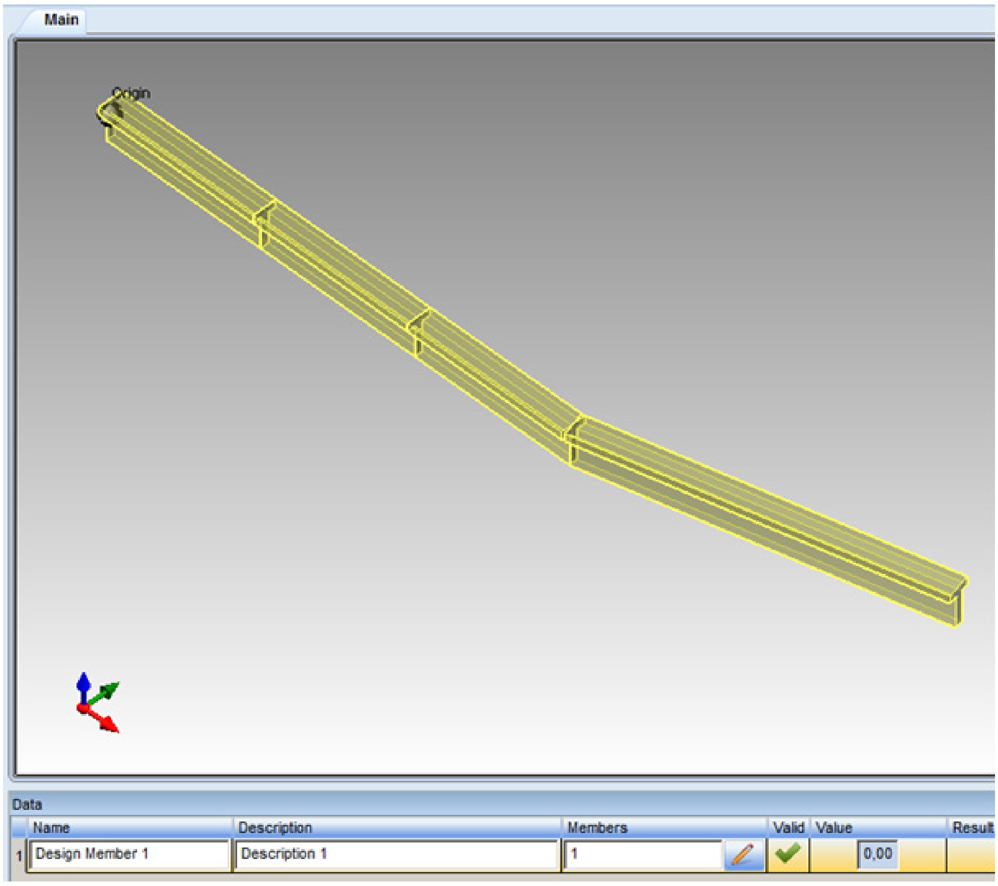


Figure 1.3: Design Member P1 – 3D view

Member P1 has been exported from RFEM to RF-TENDON. Design Member 1 has been created, which consist of one member. The member consists of four member parts.

Coordinate system of member – It is a right-handed cartesian coordinate system, which is taken from RFEM. The coordinate system of a member consists of the coordinate systems of individual member parts.

Reference axis – The connector of nodes of members or parts of members.

Coordinate system of design member – A design member does not have its own coordinate system. The geometry of a design member is defined by the sequence of coordinate systems of consecutive members.

Reference curve/polygon – A general spatial curve or polygon. Its shape or edges consist of a sequence of the x-axes of the members (tendon can be allocated to them). A spatial curve cannot be smooth and coordinate systems at the ends on particular members cannot be identical.

Uncoiled design member – Tendon design is performed with the uncoiled view of design member. Individual members of a design member are put to one straight line to create an uncoiled view as follows:

If member or member part are not straight, it is straightened to the x-axis of the uncoiled view in such a way that the distance x between the beginning and ending of the member is equal to the real length of the member axis or member part axis.

Together the LCS over the whole length of the member or its parts are straightened. If the LCS at the beginning and ending of the design member are different, the LCS of the member is moved to have the x-axis and z-axis of the LCS at the beginning of the member in the same plane as the x-axis and z-axis of the LCS at the ending of the member.

The following member is connected to the previous in such a way that the LCS at the beginning of the following member is identical to the LCS at the ending of previous member. The rules from previous paragraphs are applied.

This defines the coordinate system in uncoiled view in which tendons are designed.

Uncoiled view of design member – is drawn in two planes (XY or XZ) of the corresponding design member LCS. Cross-sections including their rotations are positioned to those planes in such a way that real positions of tendons can be calculated in relation to the cross-sectional edges. Eccentricity and rotation of each member part is also applied.

Figure 1.4: Uncoiled Design Member in planes XY and XZ

Uncoiled view coordinate system – The uncoiled view contains the coordinate system described in the definition of an *uncoiled design member*. Local x-axis vectors of coordinate systems of all members of a design member have the same direction and all x-axes of all local coordinate systems of all members are in one straight line.

Primary geometry – primary uncoiled view – Primary geometry defines the primary uncoiled view to define tendon geometry. It is used in cases when the position of points in the second definition geometry depends on the position of points in the primary definition geometry.

1.4.2 Tendon Geometry

Tendon geometry component – basic geometrical entity (line, parabola, circle)

Tendon segment – group of consecutive tendon geometry components in one plane. Neighboring segments are interdependent.

Segment parameters – input values related to segment geometry (tendon distance from top/bottom edge or cross-section center of gravity, length of straight part, arc diameter)

Stand-alone segment – type of segment which cannot be joined to another segment

Closing segment - type of segment which can be used at the beginning or end of a tendon. It is followed by an internal segment or another end segment.

Inner segment - type of segment which can be placed only between two other segments

Editing point - point used to change segment parameters

Closing point – type of editing point which is placed at the beginning (or end) of an end segment

Intermediate point – editing point inside a segment

Connecting point – point at the connection of two segments

Characteristic points of tendon segment – editing points which determine the tendon segment geometry. The tendon segment contains two or three points depending on the segment shape.

1.5 Limitations and Assumptions

The structure does not change its structural system during construction stages. Structural analysis is performed with one structural model only - all tendons are assumed to be prestressed in one moment.

It is possible to apply pre-tensioned tendons only to straight and statically determinate design members.

No external load is applied to the prestressed part of the structure before it is prestressed, external loading or self-weight can be applied at the same time as prestressing.

A prestressed beam makes one (integral) structural system or part of such a system (not a set of independent members) at the stage of the structure for which the design of tendon is performed. Examples: one structural system = simply supported beam or continuous beam, part of structural system = primary beam of portal frame.

Only Eurocode concrete and steel materials are supported. Other materials which are export from RFEM to RF-TENDON are marked as general.

1D members are exported, but 2D members (walls, slabs) are not exported from RFEM to RF-TENDON.

Only 1D members with the following cross-sections are used for design members:

- Solid concrete cross-sections (not composite)
- General concrete sections created in the module SHAPE-MASSIVE
- Cross-sections cast in one construction stage

Figure 1.5: Cross-sections Type

Variable cross-sections are not exported from RFEM to RF-TENDON. Beams with variable cross-sections are exported as prismatic beams.

Rigid links are not used because a design member cannot be created if eccentricity is defined using rigid links.

All members in the design member must have the same orientation. It means that the local x-axes of two consecutive members must not be oriented against each other – in other words two members in one design member cannot have a common ending point.

The beginning node of the following member must be the finishing node of the current member by creating a design member.

A load case of type *Prestress* must be defined in RFEM. This load case is used to transfer effects of equivalent loads from prestressing to the analytical model.

The method of analysis for load combinations has to be set to *Geometrically Linear Static Analysis.*

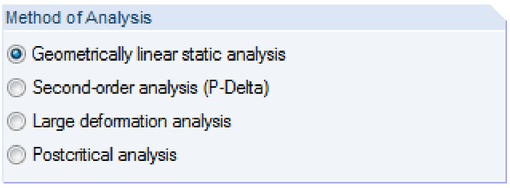


Figure 1.6: Method of Analysis

Only results from a linear calculation can be used for the design of tendons.

1.6 Demo Version Limitations

The RF-TENDON demo version enables the input and check of the comparative example "RFEM-Example08", which is described in the document "RF-TENDON Tutorial".

There are limited possibilities of the input of design members and tendons in the demo version. The check of design member including check of tendons is not limited.

Demo version limitations:

Only two cross-sections can be used for design member cross-sections:

Rectangle with dimensions 450 / 1700mm

T shape with haunch on flange of width 450mm, total cross-section height 1700 mm and web width 200mm

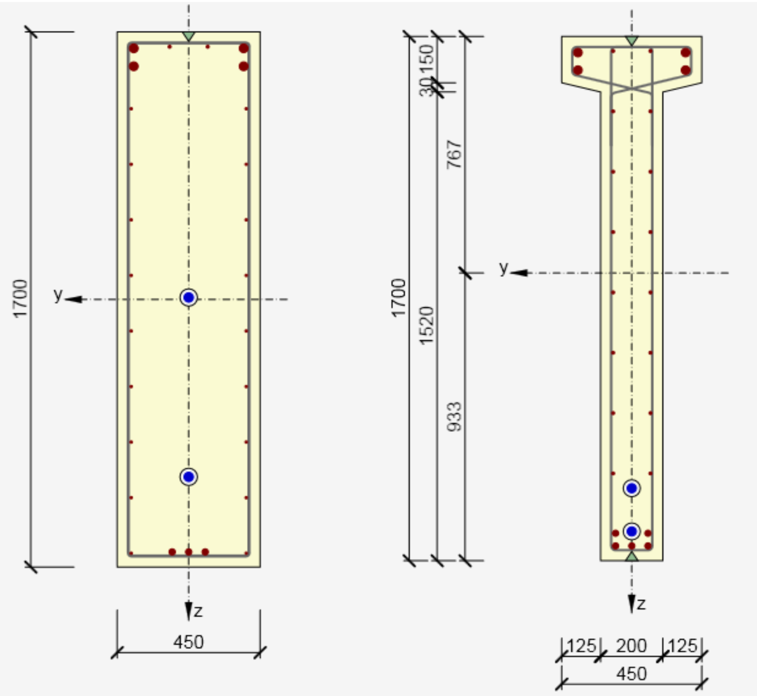


Figure 1.7: Demo – Cross sections

Only one design member can be defined. The number of members in a design member is not limited.

A maximum of two (2) tendons can be defined for a design member. Tendons cannot be copied, exported and imported.

A maximum of five (5) check positions (sections) can be defined for a design member.

2. Running the Application

2.1 Starting RF-TENDON

The module RF-TENDON can be started from RFEM after selecting Modules, External Modules and then clicking on the **RF-TENDON** icon.

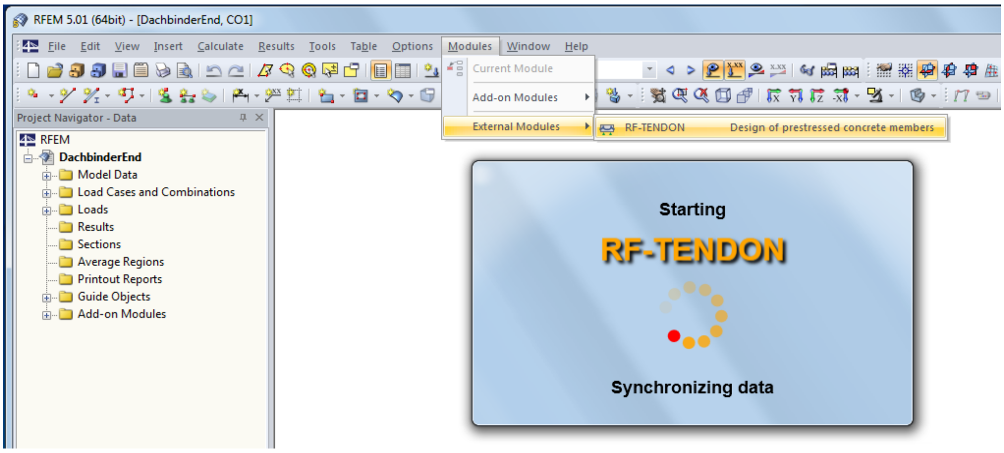


Figure 2.1: Start of RF-TENDON

2.2 User Interface

The user interface consists of the following parts:

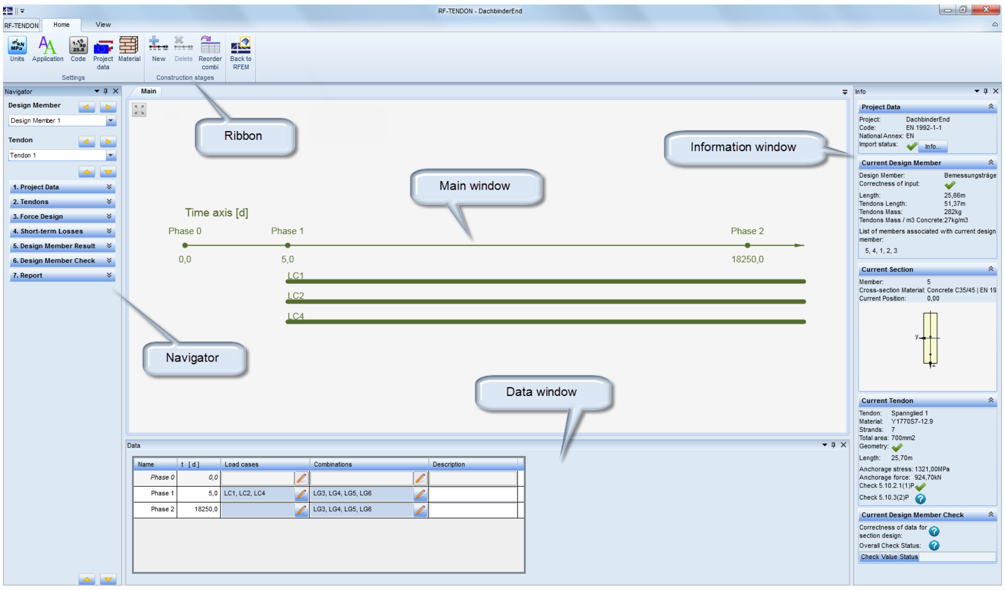


Figure 2.2: Parts of user interface

Navigator (left)

Set of commands logically ordered, starting first from the input, through the check options, and ending with output and reporting.

Ribbon groups (top)

Shows editing functions relevant to the section of the *Navigator* that is currently selected.

Main window (center)

Shows the section of the *Navigator* that is selected and the result of the editing functions selected and defined from the *Ribbon group options*, displayed as a graphical image, diagram, or text dialog.

Data window (bottom)

Shows information from the *Navigator*, or the selected object in the *Main window*, with different tables or properties.

Information window (right)

Information related to the project is shown for quick user reference.

2.3 RF-TENDON

Basic functions are available in this menu.

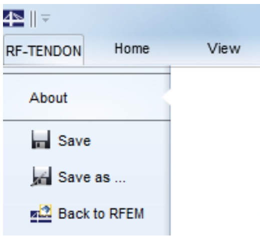


Figure 2.3: File menu

About

Show information about the program version.

Save

Save a file.

Save as

Save a file with another name.

Back to RFEM

Save data, exit RF-TENDON and go back to RFEM.

2.4 Home

Settings for the module are available in this menu.

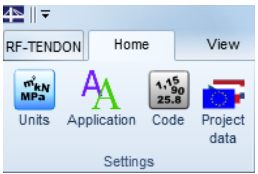


Figure 2.4: Home menu

2.4.1 Units

The units used by the program can be set by clicking the [Units] button in the *Settings* ribbon group. The settings for units must be saved in order to apply the configuration the next time the program is opened. However, the settings configuration will not be automatically applied to a project when opened in another instance.

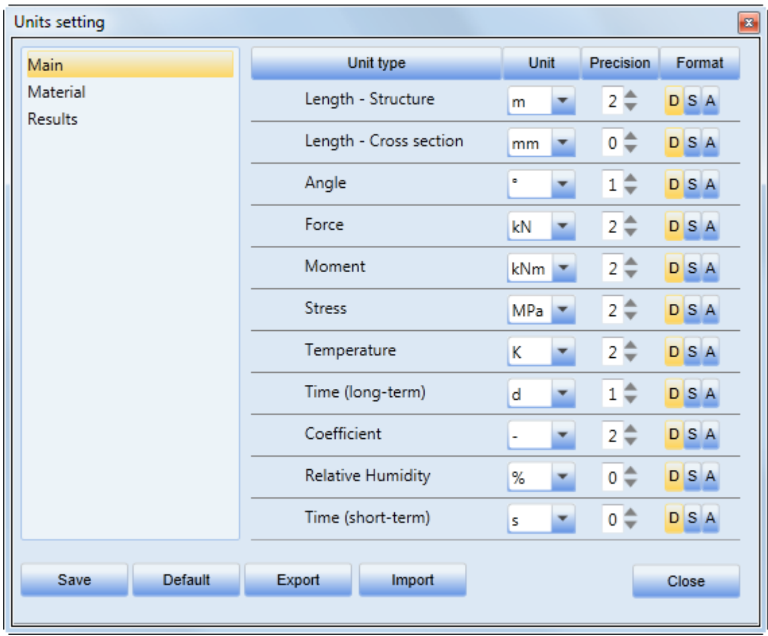


Figure 2.5: Main units

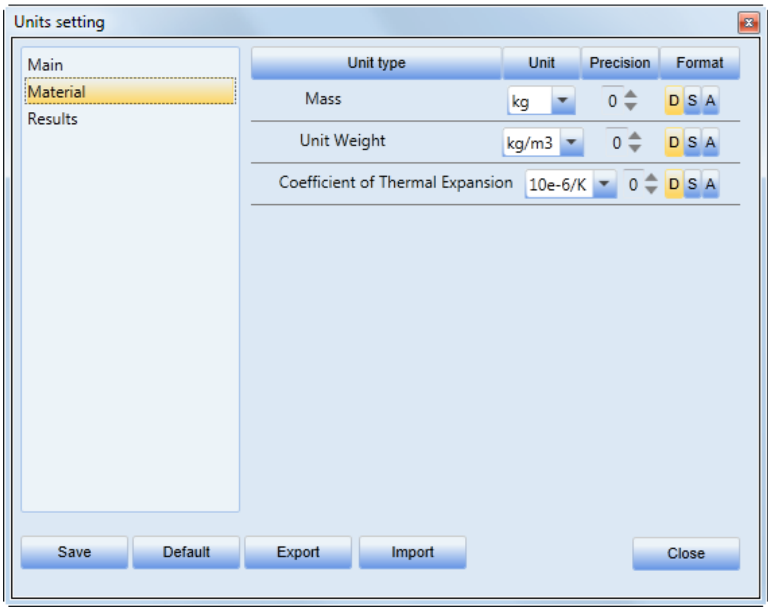


Figure 2.6: Material units

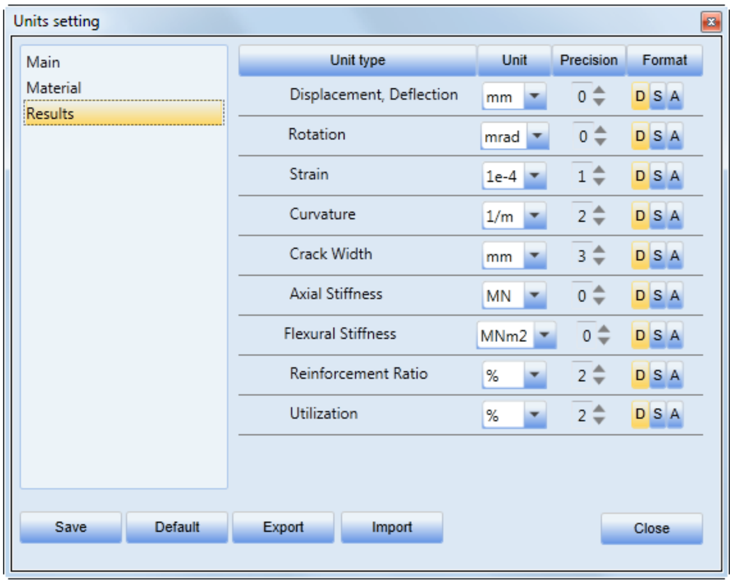


Figure 2.7: Units for Results

Variables for which you can set the units are grouped into various categories: *main, material,* and *results,* which are displayed in the column on the left side of the dialog box. The selected group is shown in a table of variable values for which user-defined units are displayed. For each variable in the *Unit* column, one of the available units can be set.

For each value the number of applied decimal places can be set in the *Precision* column.

For each value the format of number can be set via the buttons in the *Format* column:

D

Displays numbers in standard *decimal format* ("-ddd.ddd…"). The precision specifier indicates the desired number of decimal places.

S

Displays numbers in *scientific (exponential) format* ("-d.ddd…E+ddd"). The precision specifier indicates the desired number of decimal places.

A

Automatic format automatically determines to display number either in decimal or in exponential format according to the length of the resulting string. The precision specifier defines the maximum number of significant digits that can appear in the result string.

In order to apply the changes to the unit settings for the next program run, it is necessary to save them by clicking the [Save] button.

Save

Click this button to save the current configuration of units to a file with user settings. The saved settings for units are applied the next time you run the program.

Default

Sets the current units setting as the default units. These units are stored and distributed within the program. To use default units in the next program run, you must save the configuration by clicking the [Save] button.

Export

Saves the current units settings to a file.

Import

Reads the units configuration from a file. To use the imported configuration in the next program run, you must save them by clicking the [Save] button.

2.4.2 Application

To change the application environment (colors, fonts, lines) click **Application** in the **Settings** ribbon group. The settings are grouped into several tabs. The settings can be stored as a file or loaded from a file using the following commands:

Save – store current application settings to a specified file

Load – load application settings from a specified file

Default – restore default application settings

To set the colors for the drawing of a model in the 3D view, click the **Palette setting** tab.

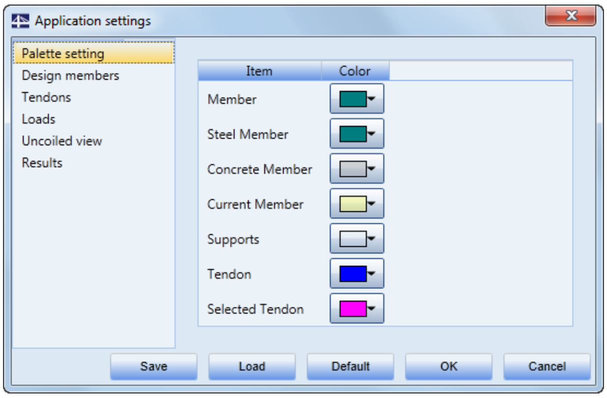


Figure 2.8: Palette setting

Colors for the following elements can be set in the **Palette setting** tab:

Member – select a color for drawing of design members

Steel member – select a color for drawing of steel members

Concrete member – select a color for drawing of concrete members

Current member – select a color for drawing of the current design member

Supports – select a color for drawing of supports

Tendon – select a color for drawing of tendons

Selected Tendon – select a color for drawing of the selected tendon

To set the drawing of a design member in uncoiled views click the **Design members** tab.

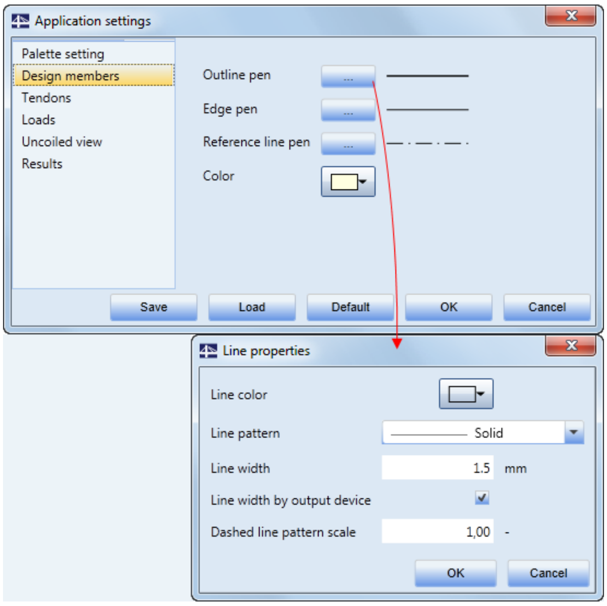


Figure 2.9: Design members and line properties

Outline pen – line style setting for drawing of design member outline

Edge pen – line style setting for drawing of design member edges. All edges (visible and hidden) of a design member in uncoiled views XY and XZ are considered as edges.

Reference line pen – line style setting for drawing of design member reference axis

Color – select color for drawing of design member fill

Particular options of the *Line properties* dialog box:

Line color – set color of line

Line pattern – set line pattern

Line width – value of line width in length units or in number of pixels

Line width by output device – if selected, a corresponding line is drawn in the specified width in corresponding length units. If not selected, the line is drawn in the specified width in pixels.

Dashed line pattern scale – value of scale for drawing of dashed lines

To set drawing of tendons in uncoiled views click the **Tendons** tab.

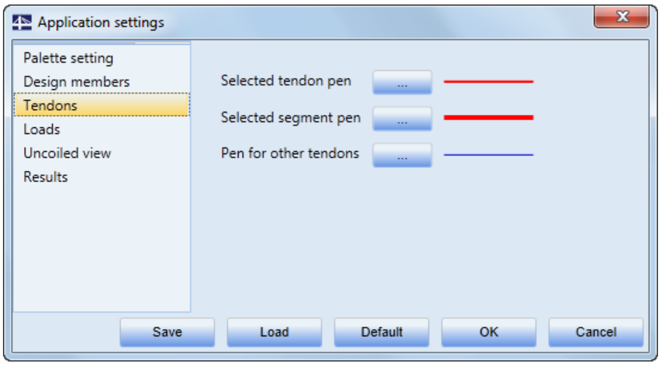


Figure 2.10: Tendons

Selected tendon pen – line style setting for drawing of the selected tendon

Selected segment pen – line style setting for drawing of the selected tendon segment

Pen for other tendons – line style setting for drawing of not selected tendons

To set drawing of equivalent loads and unbalanced loads click the **Loads** tab.

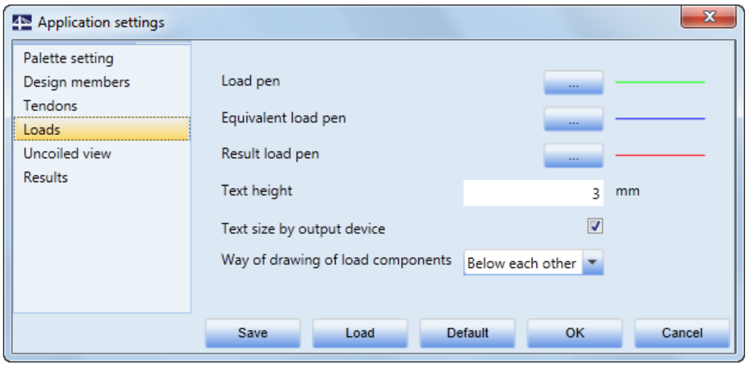


Figure 2.11: Loads

Load pen – line style setting for drawing of external loads

Equivalent load pen – line style setting for drawing of tendon equivalent load

Result load pen – line style setting for drawing of resulting unbalanced load

Text height – value of text size of load labels

Text size by output device – set evaluation mode of text height. If the option is selected, the real height of text on output device (2D window, report, printer) is the specified value in millimeters (length units).

Way of drawing of load component – select mode of drawing load graphs

Side by side – graphs of loads in individual uncoiled views are drawn side by side

Below each other – graphs of loads for all uncoiled views are drawn below each other

To set drawing in the uncoiled view click the **Uncoiled view** tab.

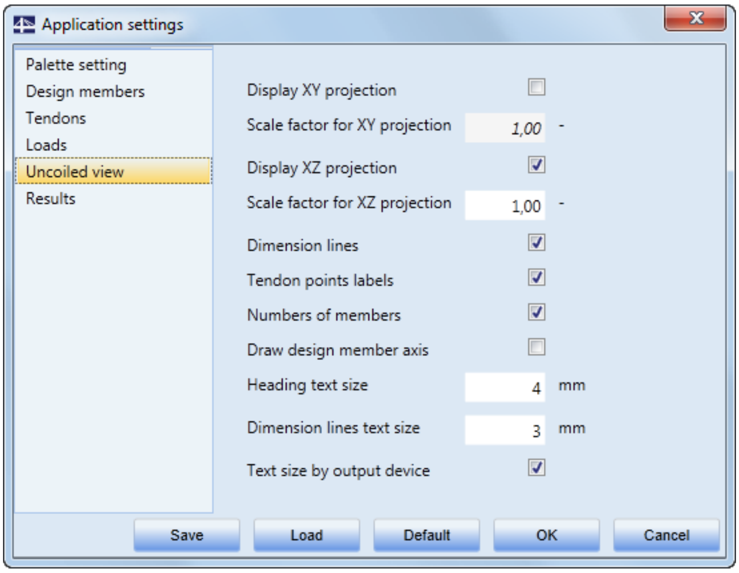


Figure 2.12: Uncoiled view

Display XY projection – turn on/off drawing of design member uncoiled view in plane XY

Scale factor for XY projection – value of the scale for drawing of design member in uncoiled view in plane XY. The scale enables more a clear drawing of tendons in design members which the x-axis length exceeds the size in the y-axis in the uncoiled view XY.

Display XY projection – turn on/off drawing of design member uncoiled view in plane XZ

Scale factor for XZ projection – value of the scale for drawing of design member in uncoiled view in plane XZ

Dimension lines – turn on/off drawing of dimension lines

Tendon points labels – turn on/off description of tendon editing points

Numbers of members – turns on/off drawing of numbers of members in a design member

Draw design member axis – turns on/off drawing of axis of a design member

Heading text size – value of text size of headings of uncoiled views

Dimension lines text size – value of text size of dimension lines texts

Text size by output device – set evaluation mode of text height. If the option is selected, the real height of text on an output device (2D window, protocol, printer) is the specified value in millimeters (length units).

To set drawing and descriptions of internal forces and tendon losses click the **Results** tab.

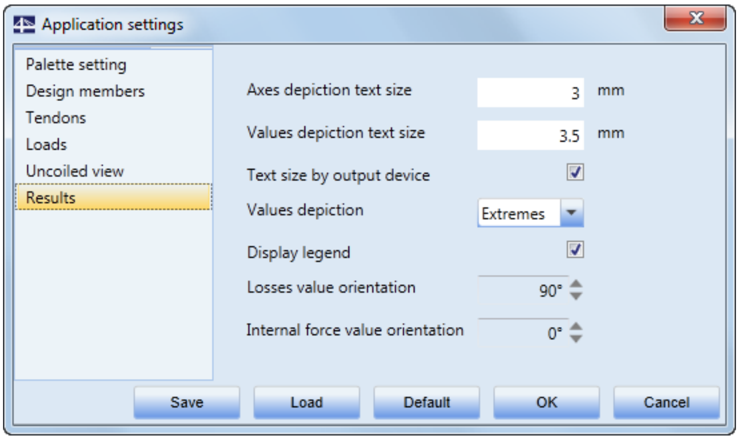


Figure 2.13: Results

Axes depiction text size – value of text size for axes depiction

Values depiction text size – value of text size for result values depiction

Text size by output device – set evaluation mode of text height. If the option is selected, the real height of text on output device (2D window, report, printer) is the specified value in milli-metres (length units).

Values depiction – mode of tendon losses graph depiction drawing

- **No depiction** – no values in graphs are depicted
- **Extremes** – extreme values in graphs are depicted
- **All** – all values in graphs are depicted

Display legend – turn on/off drawing of legend in tendon losses graph

Losses value orientation – value of slope for depiction of tendon losses

Internal force value orientation – value of slope for depiction of internal forces

2.4.3 Code

Click the [Code] button in the *Settings* ribbon group to set the National Code values and calculation variables.

Code dependent variables are grouped according to chapters and articles (clauses) of the code. The last group, *General,* contains settings of general (not code dependent) calculation values.

If a National Annex (NA) is enabled (the [Project data] button in the *Settings* ribbon group), the values of a national annex can be changed or default values of the Eurocode can be used.

To display a tooltip containing detailed information about a code variable, point to the row containing the code variable.

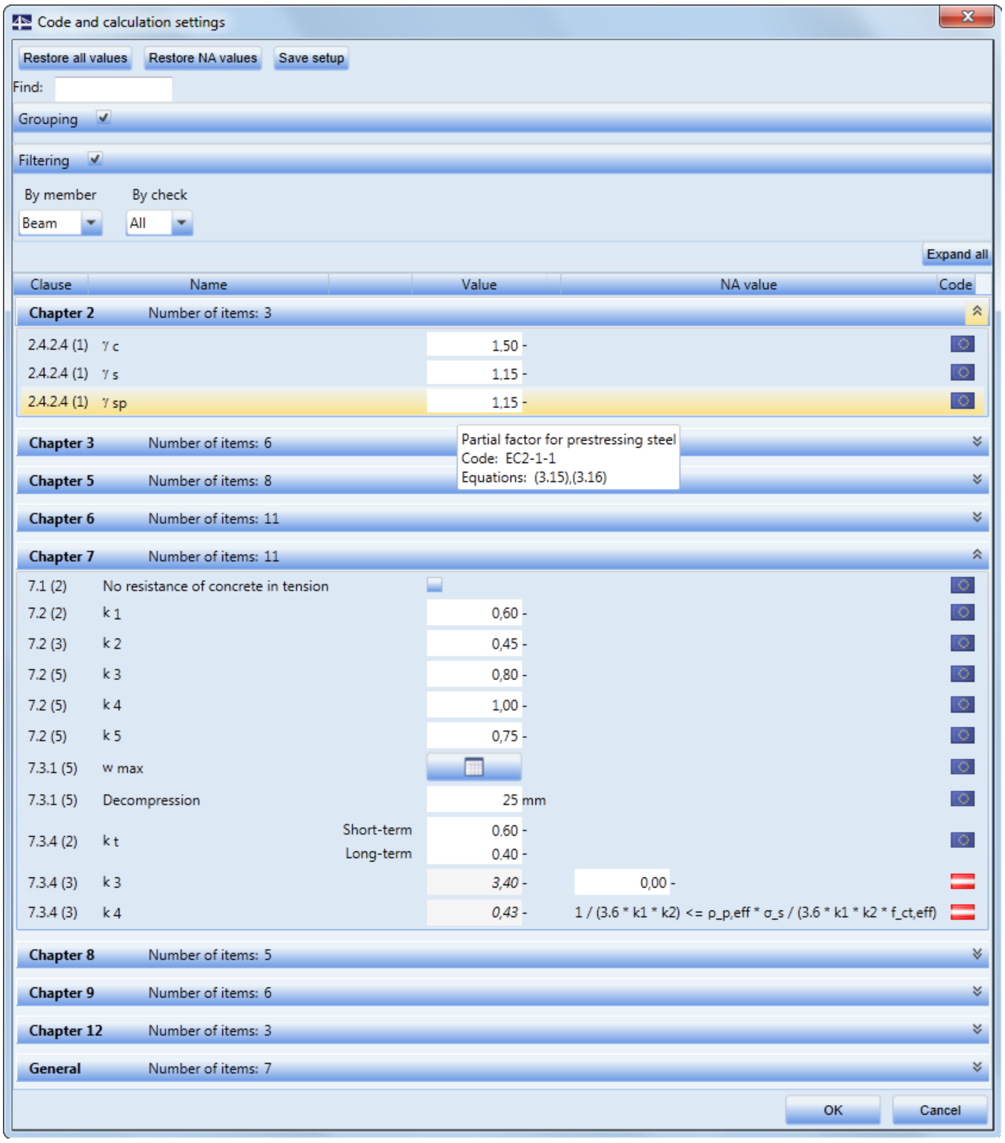


Figure 2.14: Code

Restore all values

Resets all values of code settings for Eurocode to the default code and resets all settings of the current national annex to the default annex values.

Restore NA values

Resets all settings of the current national annex to the default annex values.

Save setup

Saves the current code settings to a file. Saved settings can be loaded by opening *Project Data* in the *Settings* ribbon group and clicking the [Code] button (with flag), see 2.4.4.

Find

After entering a value in the text box, this function filters out those available code variables that contain the entered value of the article number.

Grouping

Turns on/off the grouping of code variables by chapter. When *Grouping* is on, you can collapse or expand individual chapters of code variables.

Filtering

Turns on/off the filtering of code variables by chapter. When *Filtering* is on, you can choose filtering criteria *By member* or *By check*.

Expand all / Collapse all

When *Grouping* is on, you can expand or collapse all the code variable chapters.

Clause Column

The numbers of particular code clauses are displayed in this column.

Name Column

The names of code variables are displayed in this column.

Value Column

The code variable values can be edited in this column. If there is check box at code value, it is possible to determine whether the value should be considered or neglected in the check. The values of code variables can be edited only if the *Code* column is set to EN.

Value NA Column

The values of a national annex can be edited in this column if a national annex value is available for the particular code setting item. Values of annex variables can be edited only if the *Code* column is set to a national annex.

Code Column

The flag in this column indicates which code is active for the particular code setting item. Click the flag icon to switch between a National Annex and Eurocode.

2.4.4 Project Data

To change the project data and select default materials, click **Project data** in the *Settings* ribbon group. The dialog box for *Project data* appears with project details and a section containing options for the *National Code* to be used. Project identification data is available in the header.

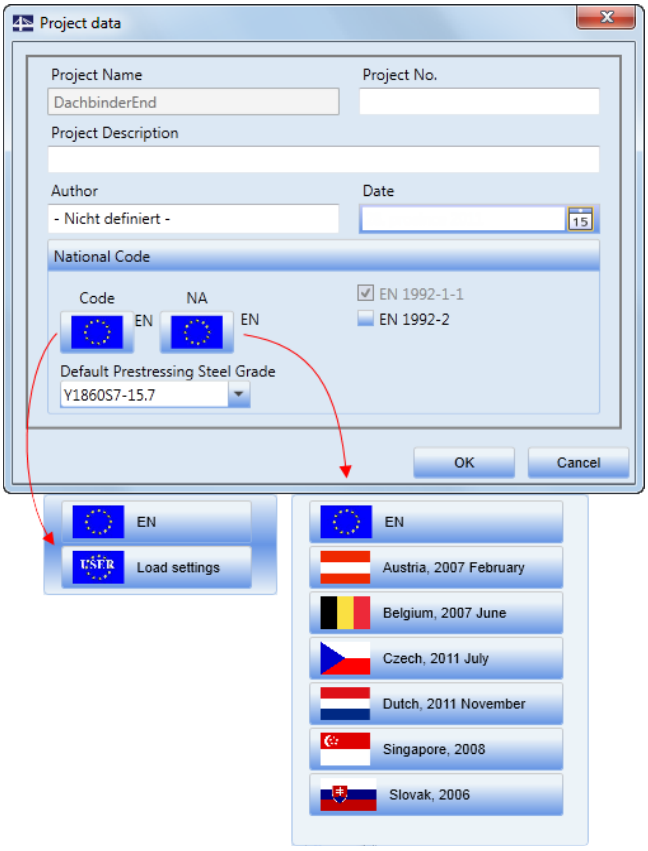


Figure 2.15: Project data

Code

Click to set the current code to EN or to load user-defined settings of code parameters. (To save the current code settings to a file, see 2.4.3)

NA

Click to load one of the available sets of National Annex parameters.

EN 1992-2

Turn on/off the option to check a cross-section according to EN 1992-2.

Default Reinforcement Steel Grade

The default prestressing reinforcement grade from the displayed list is assigned to newly entered prestressing tendons.

2.4.5 Material

To add, delete or edit materials, click **Material** in the *Settings* ribbon group. The *Material Library* dialog box will open.

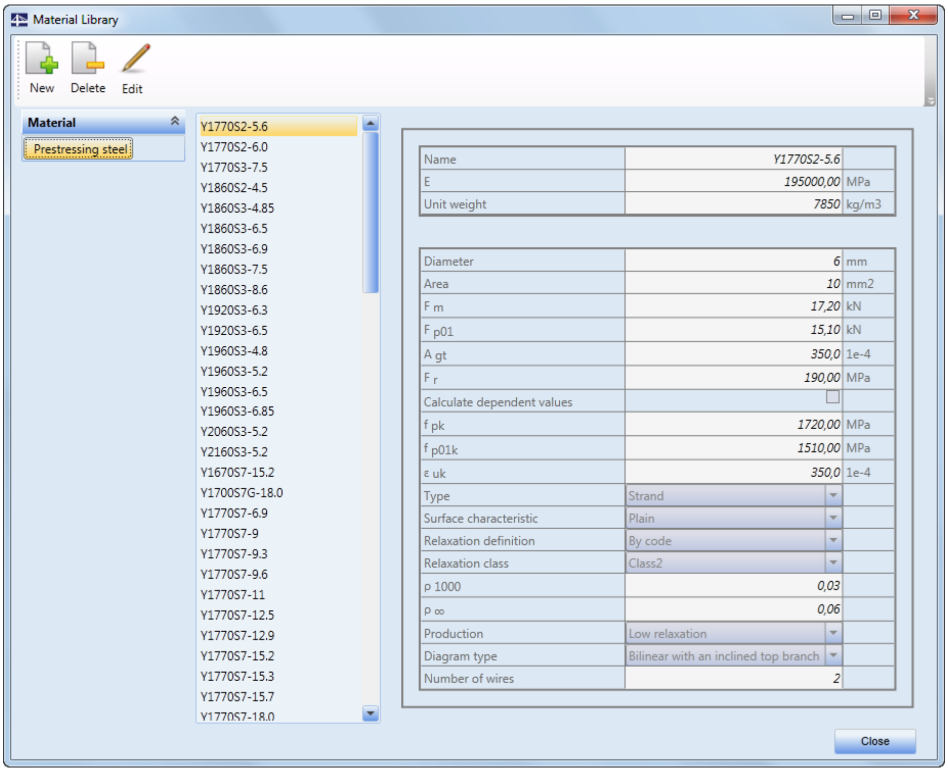


Figure 2.16: Material Library

Click the **New** button to select a new material from the material library. A new material can be added to the library by changing the material properties of an existing material.

Material Y1860S3-4.85 is selected in the picture below. After clicking **Select**, a copy of the material will be added to the list. Then click the **Edit** button to change the name and material properties of the selected material. Click **Delete** to delete the selected material.

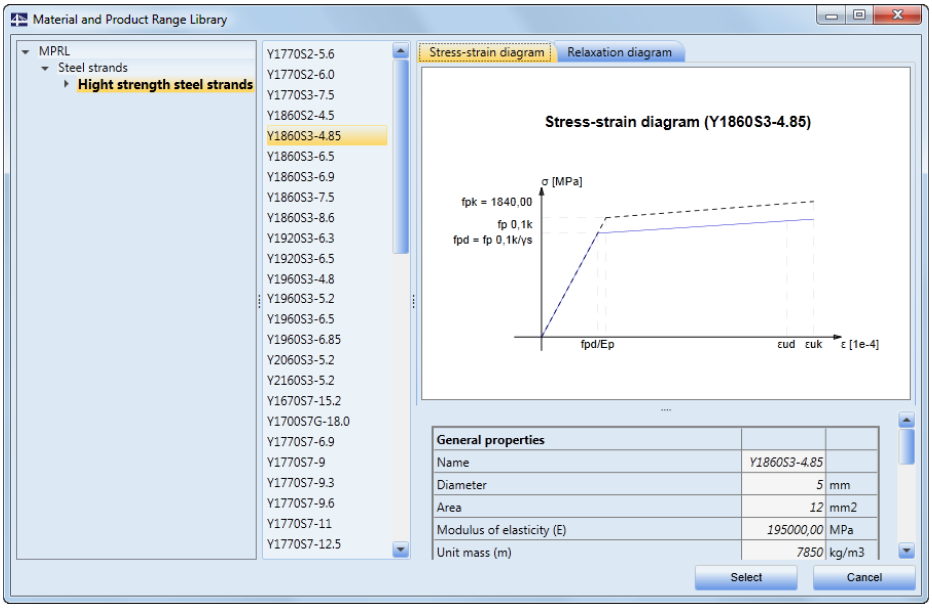


Figure 2.17: New Material – Stress-strain diagram

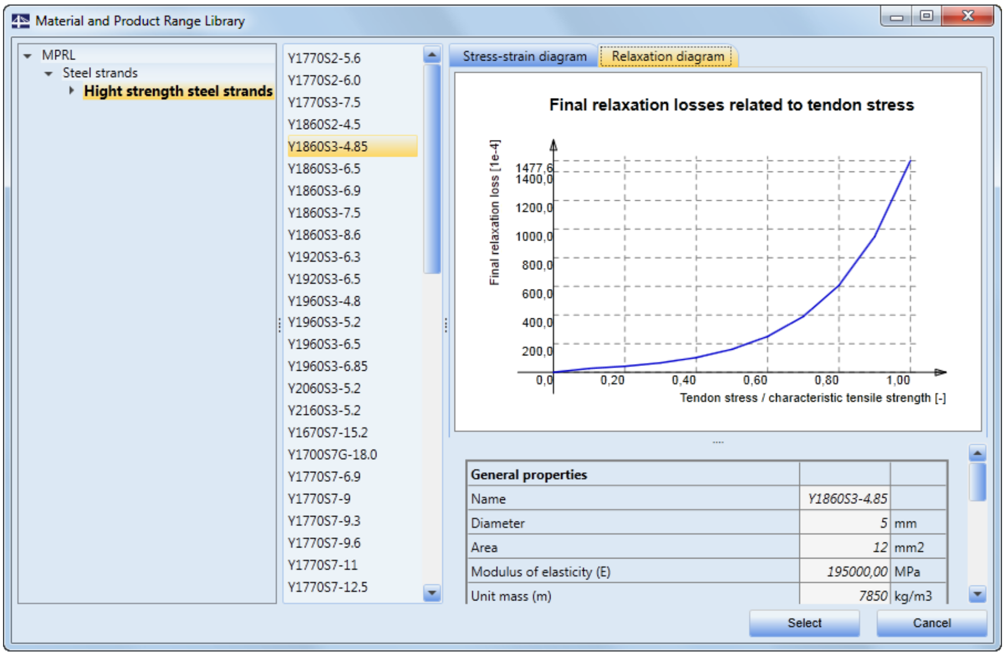


Figure 2.18: New Material – Ralaxation diagram

2.5 View

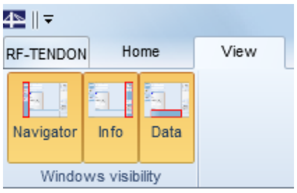


Figure 2.19: View

By using the function *Windows visibility,* it is possible to show or hide the **Navigator** (on the left), the **Info** window (on the right) and the **Data** window (on the bottom), see also chapter 2.2 *User Interface*.

2.6 Info Window

The info window (to the right) contains the following groups.

2.6.1 Project Data

Figure 2.20: Project data

The following information is displayed in **Project data** group:

Name of the project

Current National Code

Current National Annex

Information about status of import from RFEM. If some problems during import were found, click **Info** to display a detailed report of the import status.

2.6.2 Current Design Member

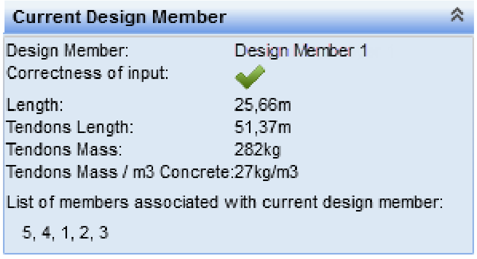


Figure 2.21: Current Design Member

The following information is displayed in the **Current Design Member** group:

Name of the current design member

Validity status of the current design member

Length of the current design member

Total length of all tendons in the design member

Total weight of all tendons in the design member

Total weight of all tendons in the design member per volume of current design member

List of members in the current design member

2.6.3 Current Section

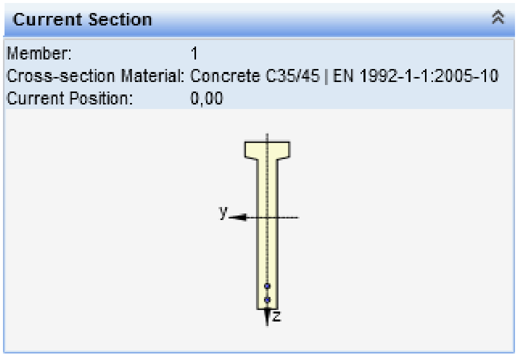


Figure 2.22: Current section

The following information is displayed in the **Current Section** group:

Number of the member, in which current the position is located

Material of cross-section

Current position on the design member. The distance is measured from the beginning of the design member.

Picture of cross-section in the current section including tendons defined in this section

2.6.4 Current Tendon

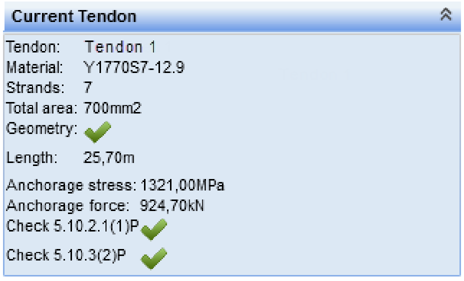


Figure 2.23: Current tendon

The following information is displayed in the **Current tendon** group:

Name of the current tendon

Material of the current tendon

Number of strands in the current tendon

Total area of the current tendon

Geometry validity status of the current tendon

Length of the current tendon

Anchorage stress of the current tendon

Anchorage force of the current tendon

Result of the maximum prestressing force check

Result of the check of prestressing force after anchoring

2.6.5 Check of Current Design Member

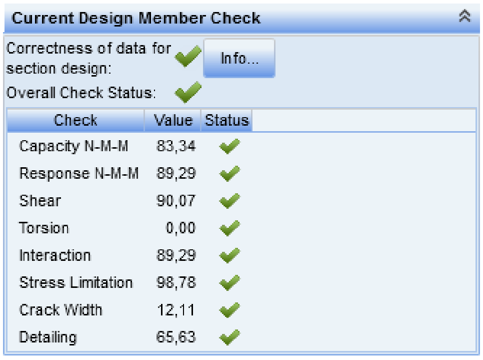


Figure 2.24: Check of current Design member

The **Current Design Member Check** group contains information about the status and check results of the current design member:

Correctness of data for section design displays the status of internal forces for check preparation. If the internal forces or other data for the check was not prepared correctly, click the [Info] button to display a detailed report of problems that occurred during the check.

Overall Check Status of all positions defined for the current design member.

Table with results of particular checks performed in defined positions. Each value represents an extreme value of a particular check from all positions defined for the design member.

Overall Check Status and individual checks are not visible until calculation in the add-on module **RF-TENDON Design** (see also chapter 3.6.2, page 76).

3. Navigator

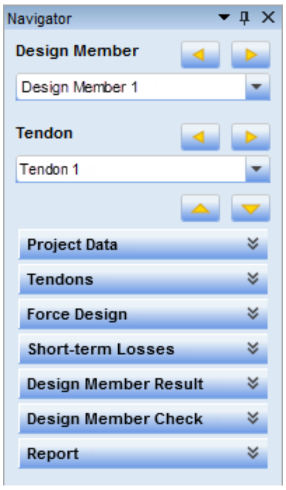


Figure 3.1: Navigator

The **Current Section** and **Current Extreme** are located in the Navigator.

By selecting one of the sections from **Project Data** to **Report**, additional parts of the program will become available.

3.1 Project Data

3.1.1 Construction Stages

To input or edit construction stages for the entire structure (all design members) click **Construction Stages** in the navigator.

Construction stages are defined by a **Global time axis**. The global time axis is drawn in the main window.

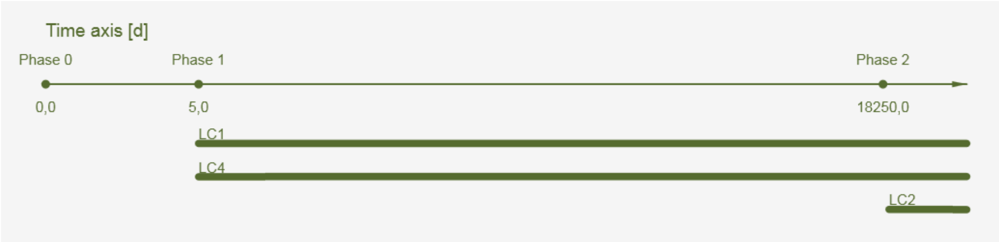


Figure 3.2: Global time axis

Each stage is defined by its age on the global time axis, by a list of load cases and load combinations. A project must contain at least 3 stages (phases). The name and description of a stage can also be edited.

The definition of construction stages is available in the **Construction stages** ribbon group.

Figure 3.3: Construction stages

The following commands are available in the *Construction stages* ribbon group:

New – create a new construction stage

Delete – delete the selected construction stage

Reorder combi – assign load combinations to construction stages automatically according to permanent load cases in construction stages

A table with the defined construction stages is displayed in the Data window.

Name	t [d]	Load cases	Combinations	Description
Stage 0	0,0			
Stage 1	5,0	LC1, LC4	LG1	
Stage 2	18250,0	LC2	LG2	

Figure 3.4: Table with construction stages

Permanent load cases applied to particular construction stages are listed in the **Load cases** column. In the *Combinations* column, the **load combinations** applied to a particular construction stages are listed.

To assign load cases to a particular construction stage click the edit button in the *Load cases* column.

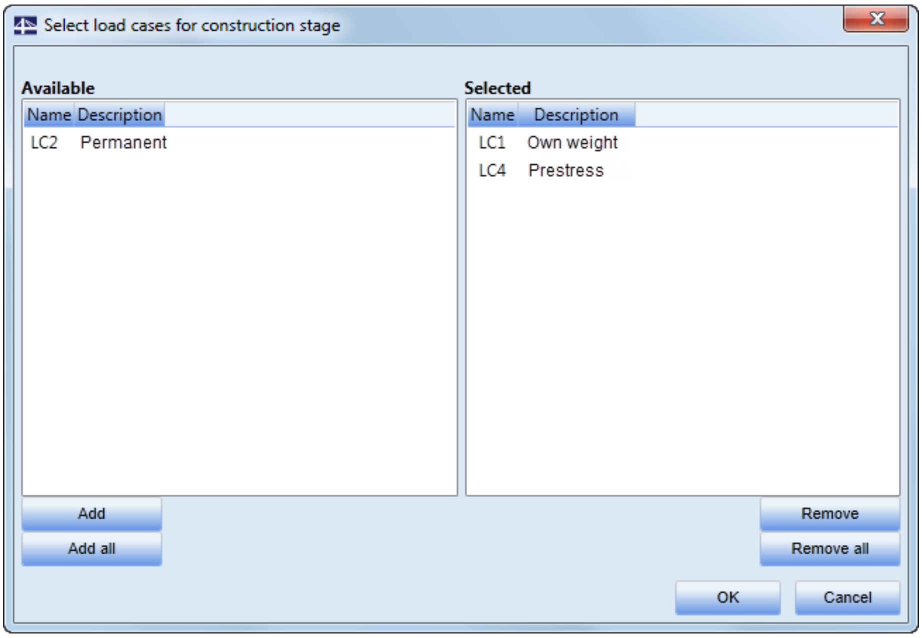


Figure 3.5: Select load cases

In the *Available* column, only **permanent** load cases are displayed. Permanent load cases can be selected and assigned directly to construction stages. The load case is applied to the selected construction stage and is automatically applied to all following construction stages. Therefore, a load case cannot be added again in another stage. Load cases defined in RFEM as **Prestress** type loads must be applied to the **first** construction stage.

Variable load cases cannot be directly assigned to construction stages. Variable load cases can act in construction stages only in load combinations.

To assign load combinations to a particular construction stage, click the edit button in the *Combinations* column.

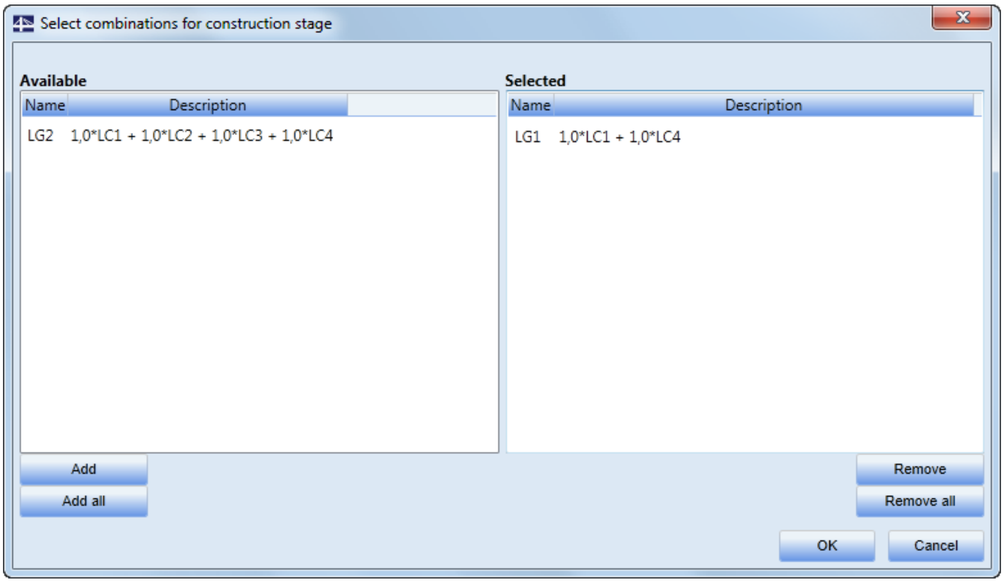


Figure 3.6: Select load combinations

For a selected construction stage it is possible to add only load combinations which contain permanent load cases that have been applied to the *Load cases* column. **Example** (see previous pictures in this chapter):

Load cases: LC1 – Self-weight, LC2 – Permanent, LC3 – Variable, LC4 – Prestress

LC1 and LC4 is added to Stage 1. LC2 is added to Stage 2. LC3 cannot be added to a stage directly, because it is a variable load (not displayed). LC4 must be added to Stage 1 because it is a Prestress load type.

LC1 and LC4 are applied to Stage 1 and therefore automatically to Stage 2. LC2 is applied only to Stage 2. LC3 was not added and at the time is not applied, it will be applied by using a load combination.

Stage 1 = LC1, LC4

Stage 2 = LC1, LC2, LC4

Load combinations:

CO1 = LC1 + LC4 , CO2 = LC1 + LC2 + LC3 + LC4

CO1 can be added only to Stage 1, because Stage 1 contains LC1 and LC4, but not to Stage 2, because Stage 2 contains also LC2.

CO2 can be added only to Stage 2, because Stage 2 contains LC1 and LC4 from Stage 1 and also LC2, which was added to Stage 2. CO 2 cannot be added to Stage 1 because Stage 1 does not contain LC2. After adding CO2 to Stage 2, LC3 will be applied to Stage 2.

3.1.2 Design Members

A design member is a basic entity to design tendons. A design member consists of one member or a group of consecutive members in the analytical model.

To define or edit design members click the **Design members** command in the navigator.

The ribbon groups *Design Member*, *Design Member View* (Uncoiled view, 3D view), *Calculate FEM*, *Check*, *Report* and *Print* are available when working with design members.

3.1.2.1 Design member

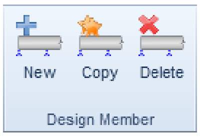


Figure 3.7: Design member

The following commands are available in the *Design member* ribbon group:

New – create a new empty design member. The new design member is added to the table of design members. A newly created design member is set as the current design member.

Copy – copy the whole design member

Delete – delete the current design member including all defined tendons

3.1.2.2 Design member view

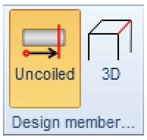


Figure 3.8: Design member view

The following commands are available in the *Design member view* ribbon group:

Uncoiled – draw uncoiled views of the current design member according to the current settings. Uncoiled views display the current design member in the XY and XZ planes.

3D – draw 3D view of the whole imported structure

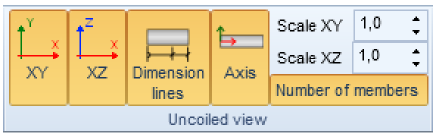


Figure 3.9: Design member view - Uncoiled view

This ribbon group is available if the view is set to **Uncoiled** in the *Design member view* ribbon group.

XY– turn on/off drawing of the uncoiled view in the XY plane

XZ – turn on/off drawing of uncoiled view in the XZ plane

Dimension lines – turn on/off drawing of design member dimension lines in uncoiled views

Axis – turn on/off drawing of design member axis in uncoiled views

Scale XY, XZ – value of the scale of y-axis (or z-axis) in uncoiled view XY (or XZ). The scale enables a more clear drawing of tendons and shapes of long design members. The scale of the x-axis always equals 1.

Number of members – turn on/off display of member numbers in uncoiled views

After changing from uncoiled view to 3D view:

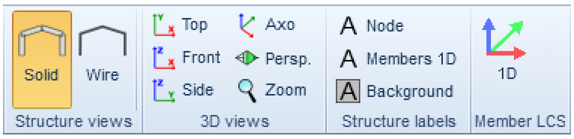


Figure 3.10: Design member view - 3D view

These ribbon groups are available if the view is set to **3D** in the *Design member view* ribbon group.

Solid – draws all structural members as solids

Wire – draws all structural members as wires

Top – sets the view from the top of the structure (opposite the positive Z-semi-axis of global coordinate system)

Front – sets the view from the front of the structure (opposite the positive Y-semi-axis of global coordinate system)

Side – sets the view from the side of the structure (opposite the positive X-semi-axis of global coordinate system)

Axo – sets the default 3D view point and performs zoom to fit the whole structure into 3D window

Persp. – turns on/off the perspective view on the structure

Zoom – performs zoom to fit the current member or design group into the 3D window

Node – switches on/off drawing of node numbers

Members 1D – switches on/off drawing of numbers of 1D members

Background – switches on/off drawing of background under numbers

1D – turns on/off drawing of local coordinate systems on 1D members

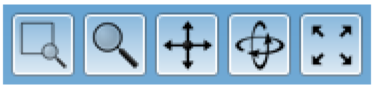


Figure 3.11: Manipulating 3D view

For manipulating in 3D view the following functions are available:

Zoom window – Click this button and drag the mouse while holding the left mouse button to draw window to zoom.

Increasing/decreasing view – Click this button and drag the mouse while holding the left mouse button to increase/decrease the view.

Pan the view – Click this button and drag the mouse while holding the left mouse button to pan the view.

Rotate the view – Click this button and drag the mouse while holding the left mouse button to rotate the view.

Zoom all – Click this button to fit the whole model in the 3D window (used also by Uncoiled view)

3.1.2.3 Calculate FEM

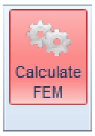


Figure 3.12: Calculate FEM

To recalculate internal forces in RFEM click the **Calculate FEM** button. The background of the button is set to red if recalculation is required – for example after changes to the geometry of tendons.

3.1.2.4 Check

Figure 3.13: Check

The following commands are available in the *Check* ribbon group:

All results – If this option is selected, a check of design members is performed for all combinations of internal forces.

Extremes only – If this option is selected, a check of design members is performed only for extreme values of internal forces. Extremes are determined from all combinations (cases) in associated result class. Maximum and minimum values are determined for each component of internal forces and all corresponding values are stored with the extreme value.

This option is recommended for debugging of the model to save time with calculation. For the final calculation it is necessarily to use the option *All results*.

All design members – run check of all positions in all design members.

3.1.2.5 Report (all design members)

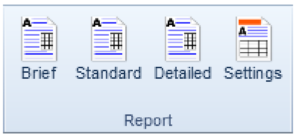


Figure 3.14: Report

Three types of reports can be generated for the selected design members:

Brief – Contains only a table with a description and overall check results of design members in the project. Content of a brief report can be affected only by the selection of design members to be printed, not by report settings.

Standard – Contains basic project data information, design members information, prestressing information and check results. Content of a standard report can be affected by selection of design members and by report settings.

Detailed – Contains detailed project data information, detailed design members information, detailed equivalent load information, detailed prestressing information and check results. Content of a detailed report can be affected by selection of design members and by report settings.

Settings – To set the content of a report click *Settings*. Report settings consist of global settings and detailed settings.

Click the edit button to select which tables and pictures should be printed in particular chapters. For chapters with graphical representation, particular pictures can be selected to be printed and the size of pictures can be set.

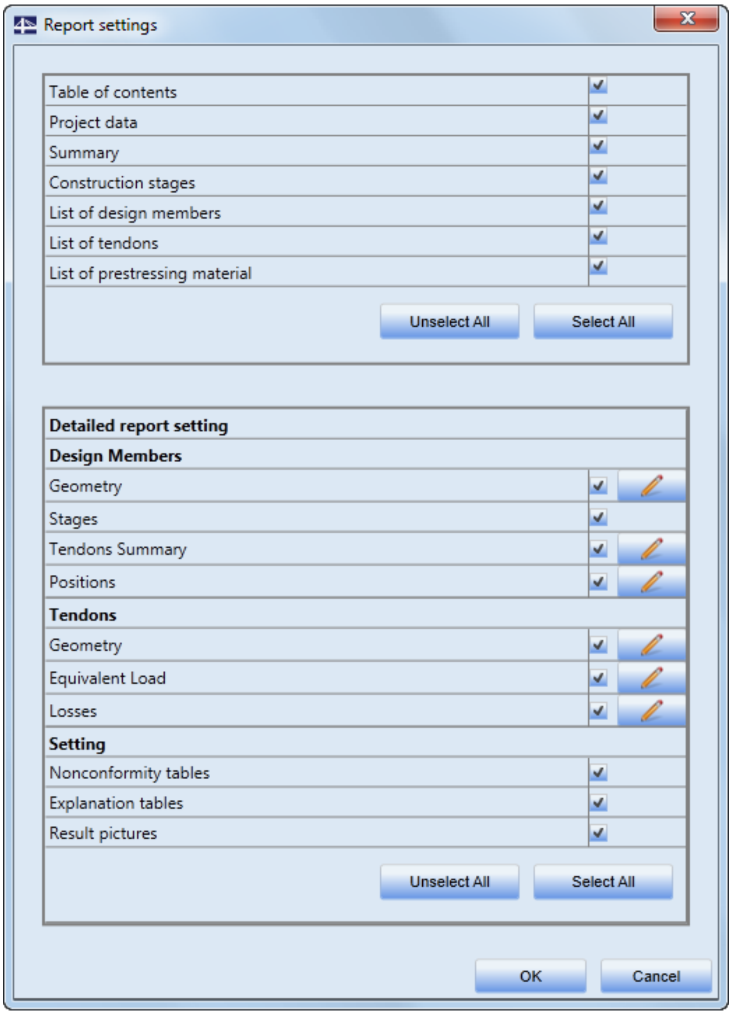


Figure 3.15: Report - Settings

The options in the **Design members** group enables adding report chapters with design member information and pictures. It is possible to turn on/off printing of the design member geometry table, construction stages table, tendons summary table and positions for check table.

The options in the **Tendons** group enables adding report chapters with tendon information and pictures. It is possible to turn on/off printing of the tendon geometry table, equivalent loads table and losses table.

Options in the **Settings** group:

Nonconformities tables – If the option is off, no nonconformity table is printed in the report. Otherwise the nonconformity tables are printed if they haven't been switched off in the detailed settings.

Explanation tables – If the option is off, no explanation table is printed in the report. Otherwise the explanation tables are printed if they haven't been switched off in the detailed settings.

Result pictures – If the option is off, no picture with graphical presentation of results is printed in the report. Otherwise pictures are printed if they haven't been switched off in the detailed settings.

By clicking the edit buttons, additional settings are available. For example:

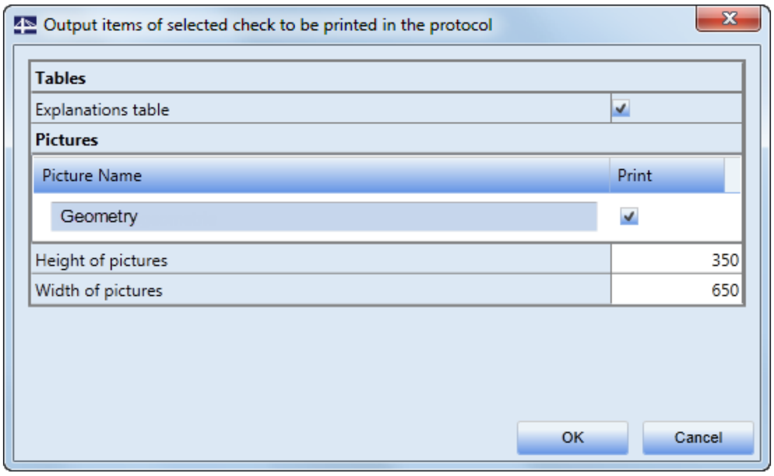


Figure 3.16: Group Tendons – Geometry – edit

Explanation table – turns on/off the printing of a table with an explanation of symbols to the report for edited check

Pictures – list of available graphical representations of results for the selected chapter. The picture name and option to print or not is available.

Height of pictures – value of picture height for pictures in current document chapter

Width of pictures – value of picture width for pictures in current document chapter

Design members are selected to print results by clicking the *Print* icon in the Data window.

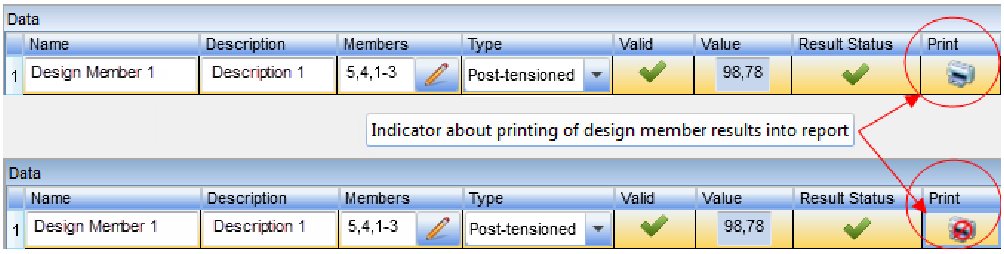


Figure 3.17:Print ON / OFF

3.1.2.6 Print

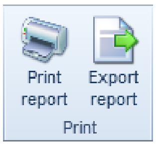


Figure 3.18: Print

The following commands are available in the *Print* ribbon group:

Print – click to print the report

Export – click to export the current report to a *.rtf file.

3.1.2.7 Creating a new design member

Click on **New** to create a new design member.

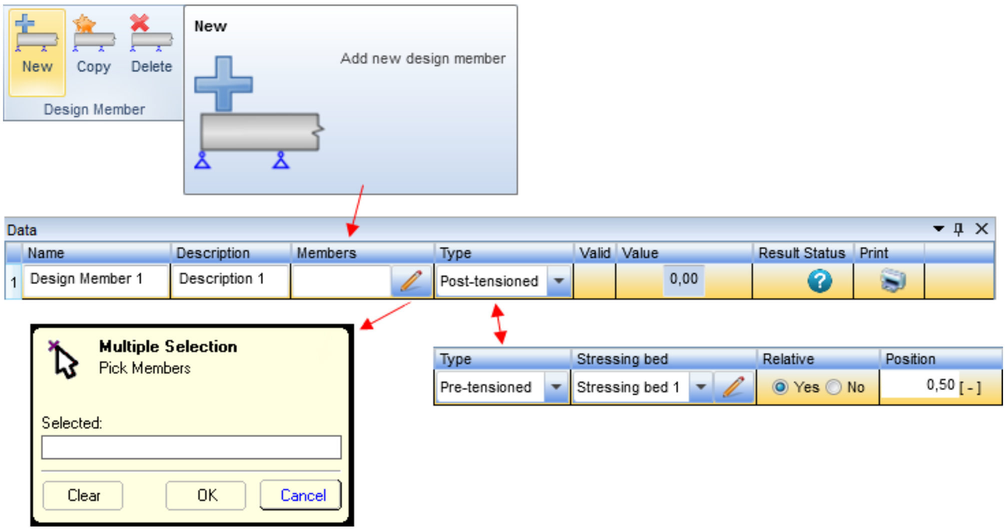


Figure 3.19: New Design Member

Click the edit button in the *Members* column to display the model in RFEM and use the **Multiple Selection** window to select members graphically or input the numbers of members manually (for example: Numbers 1,4-6 defines a design member created by Members 1, 4, 5, 6).

The table of new design members will contain the following columns:

Name – name of the design member

Description – description of design member

Members – list of members which create the design member

Type – define the type of member – Pre-tensioned or Post-tensioned

- **Pre-tensioned** – only pre-tensioned tendons are applied to the design member
- **Post-tensioned** – only post-tensioned tendons are applied to the design member

Stressing bed – define parameters of the stressing bed for pre-tensioned design member (see Figure 3.20)

- **Code coefficient** – value according to Equation (10.3), EN 1992
- **T_{max}** – maximum temperature of concrete near the tendons
- **T_0** – initial temperature of concrete near the tendons

Relative – relative or absolute definition of the position

Position – position of the section for the definition of pre-tensioned tendons

Valid – display design member validity status – it means if design member fulfills conditions to be created from the defined list of members

Value – display extreme value of check from all positions checked on design member

Result status – display overall status of design member check

Print – turn on/off design member to be printed in the report

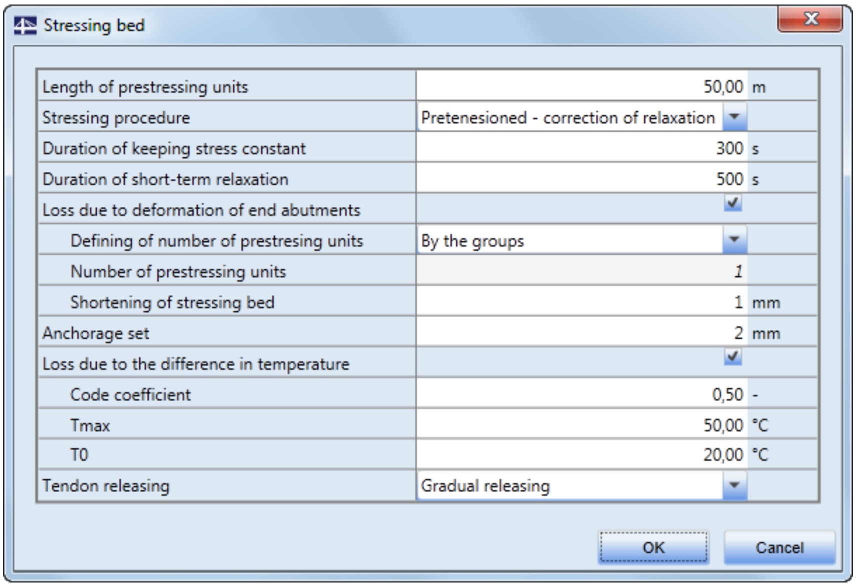


Figure 3.20: Stressing bed

3.1.2.8 Check of created design member

When RF-TENDON is launched for first time for a particular project, the application attempts to create a new design member from the imported members. All imported members are checked and if those members follow each other, one design member is created. It is not necessary that members lie in one line.

The following rules are checked during the creation of a design member:

- entire design member is of a concrete cross-section
- all members in the design member have the same orientation of local x-axes
- beginning node of the following member is the ending node of the current member

When RF-TENDON is re-launched for the same project, a new design member is not created. Individual members of an existing design members are checked, whether the geometry or material has been changed or deleted in RFEM. Validity of the design member is displayed in the *Valid* column in the design member properties table or in the Info window for the design member.

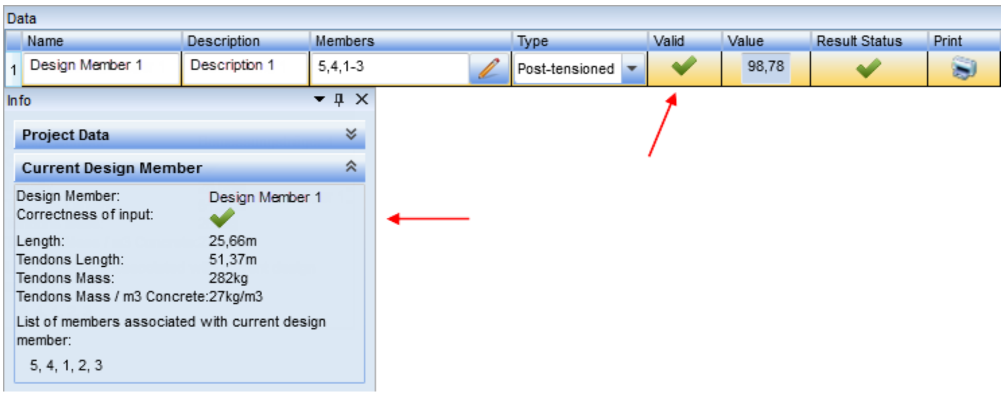


Figure 3.21: Status of design member in Data window and in Info window

3.2 Tendons

To define and edit tendons, click **Tendons layout** in the navigator.

Uncoiled views of the current design member are drawn in the **Main window**.

Tabs with tendon properties and tendon geometry properties are displayed in the **Data window**. Particular tabs:

Tendons – properties of tendons in the current design member. After tendon properties change the equivalent forces are updated automatically.

Tendon geometry XY – editing of tendon geometry in uncoiled view XY.

Tendon geometry XZ – editing of tendon geometry in uncoiled view XZ.

3.2.1 Tendon Layout

The ribbon groups New tendon, Tendon tools, Import and Export, Design member views (Uncoiled view, 3D view, Cross-section) are available when working with tendons.

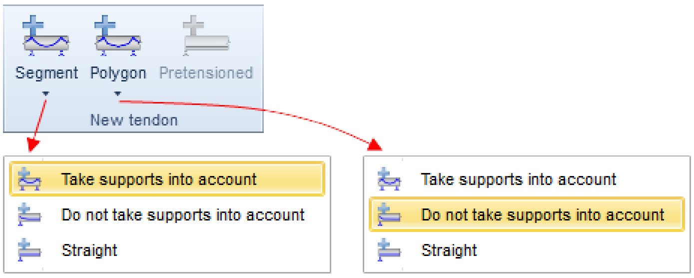


Figure 3.22: New tendon – post-tensioned

There are two options to create post-tensioned tendons:

Segment – new tendon is created by parabolic segments

Polygon – new tendon is created by straight line polygons

With both options it is possible to create:

- a new tendon respecting positions of supports
- a new tendon without respecting positions of supports
- a new straight tendon

Create a new tendon respecting positions of supports

Respecting supports means that the tendon is at the bottom of the cross-section in the span between supports and at the top of the cross-section above the supports. The tendon consists of at least one segment in each plane.

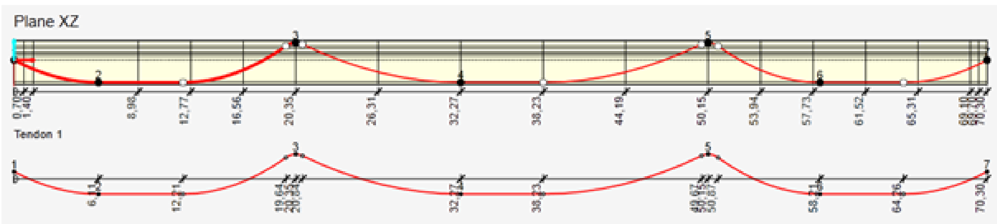


Figure 3.23: Tendon - with positions of supports

Create new tendon without respecting positions of supports

A new tendon is created which consists of exactly one segment in each plane. The tendon is straight in plane XY (ground plan) over the entire design member length and parabolic in the vertical plane XY.

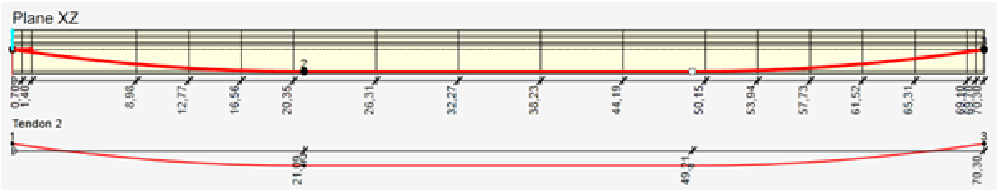


Figure 3.24: Tendon – without positions of supports

Create new straight tendon

The new tendon consists of exactly one straight segment in both planes.

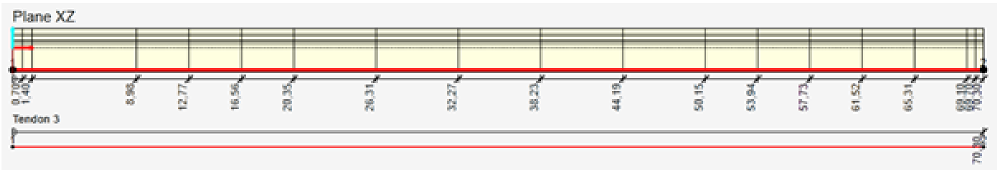


Figure 3.25: Tendon – straight

Figure 3.26: New tendon – pre-tensioned

Click **Pretensioned** to create a new group of straight pretensioned tendons.

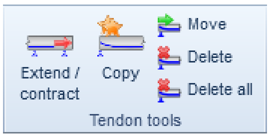


Figure 3.27: Tendon tools

Extend / contract – adapt tendon length

If the closing point of the last segment lies inside the design member, the closing point is moved to have the same X-coordinate as the last point of the design member. Other coordinates remain unchanged.

Closing points are described in 3.2.1.5 Description of tendon definition geometry points.

If the closing point of the last segment lies outside of the design member (tendon is extruding from design member), the tendon will be shortened. At first whole segments lying outside the design member are deleted. Afterward the length of last segment is shortened to have the X-coordinate of the closing point identical with the X-coordinate of the last design member point.

Copy – copy current tendon

Move – move current tendon

Delete – delete current tendon

Delete all – delete all tendons in the current design member

Copy current tendon

The tendon created by copying can move in the Y- and Z-axis of the member coordinate system.

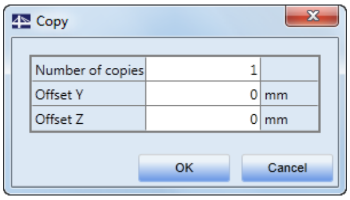


Figure 3.28: Tendon – Copy

Number of copies – required number of copies of the current tendon

Offset Y – offset value between copies in the Y-axis

Offset Z – offset value between copies in the Z-axis

The copies have identical properties as the source tendon, including characteristic points in uncoiled views. But the tendon geometry in the uncoiled views may be different because the tendon segment characteristic points may be related to points on the cross-section edges.

Move current tendon

The tendon can move in the Y- and Z-axis of the design member coordinate system.

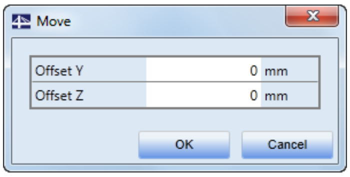


Figure 3.29: Tendon – Move

Offset Y – offset value in the Y-axis

Offset Z – offset value in the Z-axis

If characteristic points are related to points at the cross-section edges, the tendon geometry of the moved tendon may not fully correspond with the original tendon.

Delete current tendon

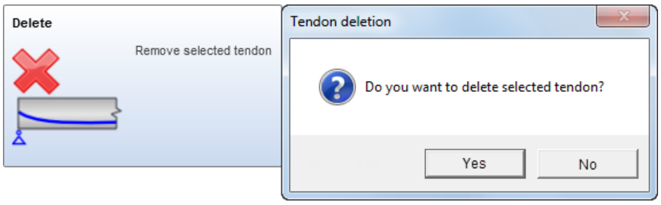


Figure 3.30: Tendon – Delete

The tendon can be selected and removed by clicking Delete and then [Yes].

Delete all – delete all tendons in current design member

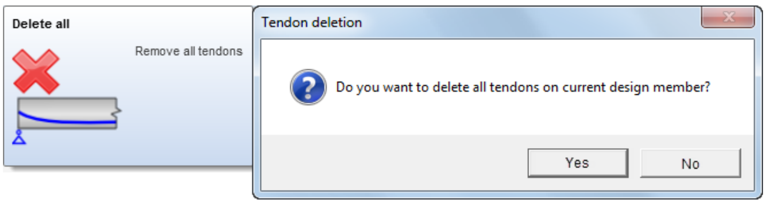


Figure 3.31: Tendon – Delete all

All tendons can be selected and removed by clicking Delete and then [Yes].

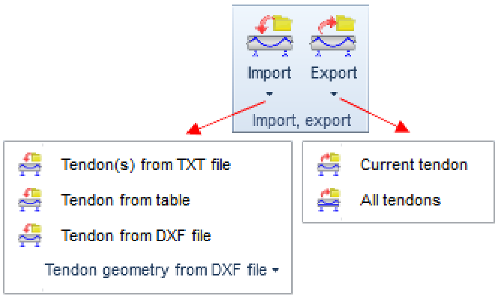


Figure 3.32: Tendon tools

The geometry of the created tendon can be stored as a file. To run tendon export and import use the following commands in the *Tools* ribbon group.

Import – read tendon geometry from a *.nav text file, a table or a *.dxf file.
If the imported tendon is longer than the target design member, the imported tendon is shortened automatically.

Export -

Current tendon – save the geometry definition of the current tendon to a *.nav text file

All tendons – save the geometry definition of all tendons in the current design member to a *.nav text file

For a description of text format *.nav with an example, see chapter A *Text Format *.nav*.

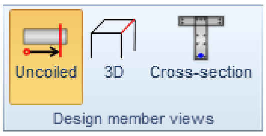


Figure 3.33: Design member views

The following commands are available in the *Design member view* ribbon group:

Uncoiled – draw uncoiled views of the current design member according to the current settings. Uncoiled views display the current design member in the XY and XZ planes.

3D – draw 3D view of the whole imported model

Cross-section – draw cross-section of the current position

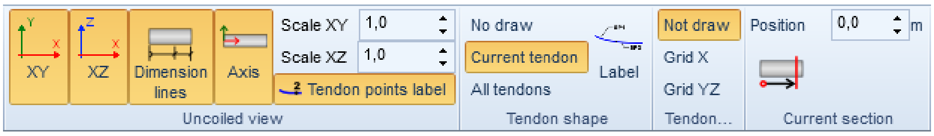


Figure 3.34: Design member views – Uncoiled view

Uncoiled view

Tendon points label – set on/off labeling of tendon points

Tendon shape

No draw – shape of tendon is not drawn

Current tendon – shape of tendon is drawn for current tendon

All tendons – shape of tendon is drawn for all tendons

Label – detail parameters of tendon geometry will be drawn

Tendon spacers

Not draw –shape of tendon is not drawn

Grid X – Y or Z tendon coordinates are calculated and drawn in user defined equidistant X-locations

Current section

Set current section to be displayed in the Info window. Define the position by value or click on the icon and input it by using the mouse in the Main window.

For description of remaining functions of this ribbon group, see chapter 3.1.2.2 *Design member view*.

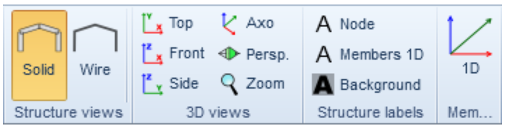


Figure 3.35: Design member views – 3D view

A description of this ribbon group can be found in chapter 3.1.2.2 *Design member view*.

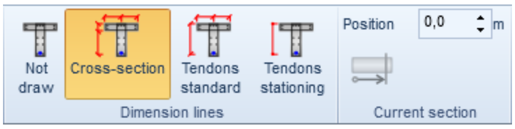


Figure 3.36: Design member views – Cross-section

Not draw – dimesion lines will not be drawn

Cross section – dimension lines will be drawn for the cross-section

Tendons standard – dimension lines will be drawn for tendons

Tendons stationing – dimension lines with distances related to certain (reference) points will be drawn for tendons

Current section – set current section to be displayed in the Main window and in the Info window, define position by value

3.2.1.1 3D Tendon geometry

Tendon geometry is created from so called definition geometry. **Tendon definition geometry** DGY or DGZ is tendon geometry defined in the **uncoiled view** YX (or XZ) of the design member. Definition geometry in the XZ-plane (or XY-plane) is defined as the horizontal (or vertical) projection of the tendon transformed by uncoiling the design member to the XZ-plane (or XY-plane) of the uncoiled view coordinate system. Both definition geometries are created independently except the total length of segments, which consist of straight and parabolic components. Segments are defined by characteristic points. Definition geometries carry information about e.g. arc diameters or tangent length.

3D tendon geometry is created by the composition of **tendon definition geometries** to spatial polygon and its backward winding on **reference curve/polygon** (spatial transformation of definition geometry into the coordinate system of each point of the reference curve in such way, that the x-coordinate of definition geometry corresponds to the curve ordinate of the reference curve). The final 3D tendon geometry is only a set of points without information about arc, radii, etc.

Figure 3.37: Example 1 : Design member uncoiled views in planes XY and XZ including straight tendon

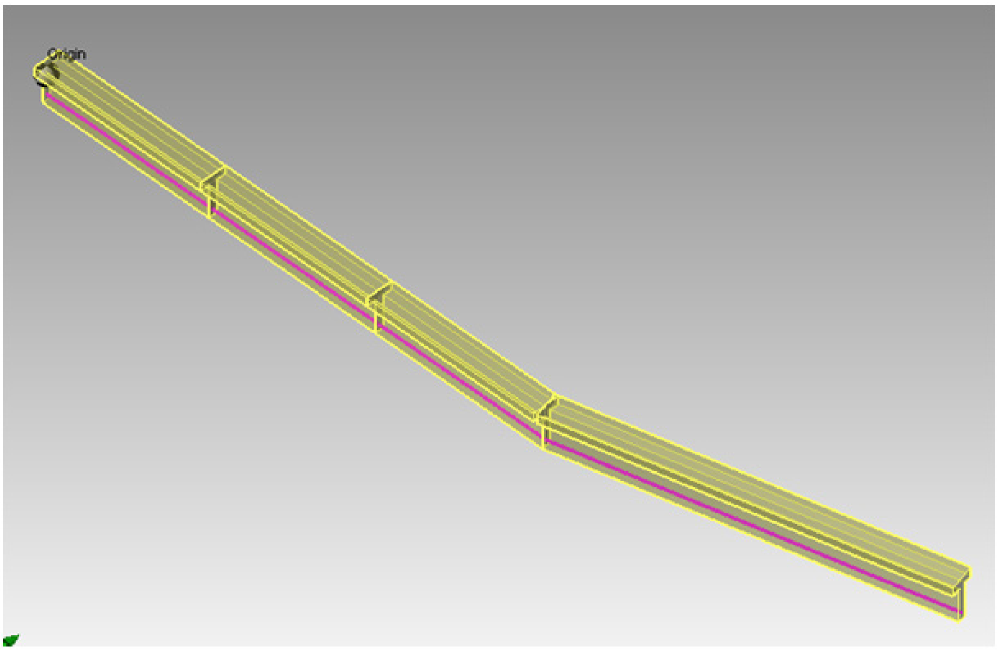


Figure 3.38: Example 1 : Final 3D tendon geometry

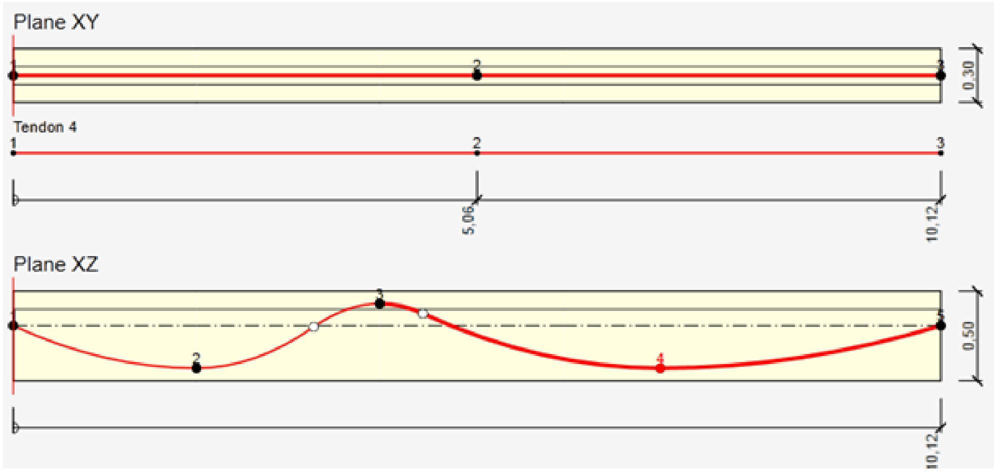


Figure 3.39: Example 2 : Design member uncoiled view XZ including parabolic tendon

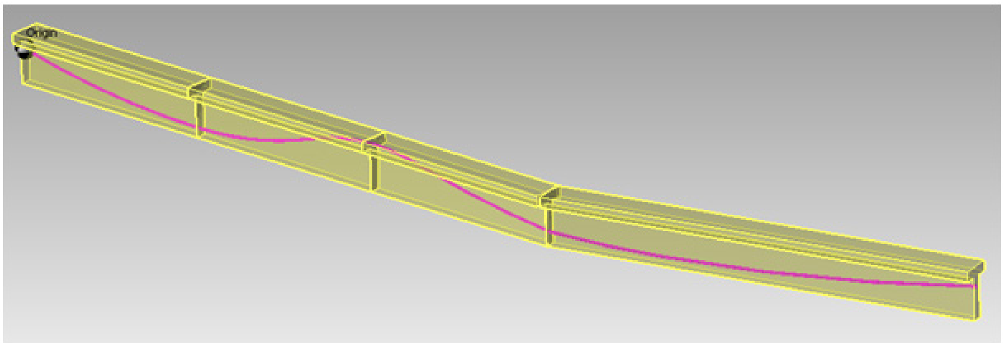


Figure 3.40: Example 2 : Final 3D tendon geometry

3.2.1.2 Description of tendon definition geometry segments

A total of **7 segment types** can be used to define geometry. Their usage depends on the segment position in the tendon geometry to keep continuity of particular segments as well as termination of tendons.

Segment type 1 – Straight stand-alone

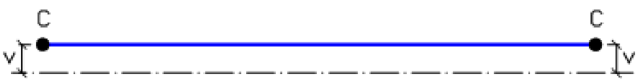


Figure 3.41: Segment type 1

This segment consists of one geometrical entity only – a straight line. It cannot be connected to another segment and can be used only as a stand-alone. The shape is defined using two **C points (Closing points)**. Point C is always on the beginning or on the ending of the segment and its position is defined by distance *v* from the member reference line in plane XY or XZ.

Segment type 2 – Parabolic with straight stand-alone

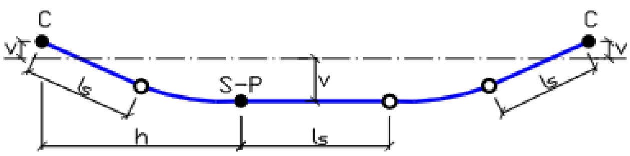


Figure 3.42: Segment type 2

This is the default segment for new tendon. This type cannot be connected to another segment. However, if the segment is split, it is automatically replaced by a corresponding segment, which enables it to connect to another segment. Geometrically it consists of three curves (parabola, straight line and parabola). A straight line can be omitted. If parameters of the parabolic part define s straight line, the straight line is used instead of the parabola. Straight lines can replace the appropriate part of the parabola at the beginning or ending of the segment.

The segment is defined using two **C points** and an intermediate **point S-P (Straight-Parabolic – intermediate point between straight and parabolic component)**. The position of the **S-P point** is defined by a distance *h* from the left or right ending point or from the center of the segment and by a distance *v* the member reference line in the XY or XZ plane. The distance *ls* is the length of the straight part between the parabolas. The coordinates of the white-filled points in the picture are not entered, but calculated from the entered parameters.

Segment type 3 – Parabolic with straight, closing left

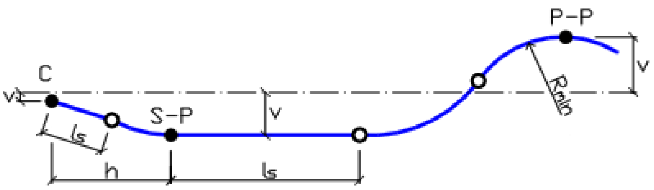


Figure 3.43: Segment type 3

This type can be used as a beginning tendon segment and the next follow-up segment can be connected to it. This segment consists of up to five curves – straight line, parabola, straight line, parabola and parabola. A straight line can be omitted. Entering parabola can be partially replaced with a straight part. The last two parabolas have opposite orientation, rotated by 180 degrees.

The segment is defined using an entering **point C**, intermediate **point S-P** and **point P-P (Parabolic–Parabolic – connecting point between two parabolas). Point P-P** describes the transition from a segment of type 3 to a follow-up segment. The position of **point P-P** is defined by a distance *v* from the beam reference line and the minimum radius of parabolas.

Segment type 4 – Parabolic with straight, closing right

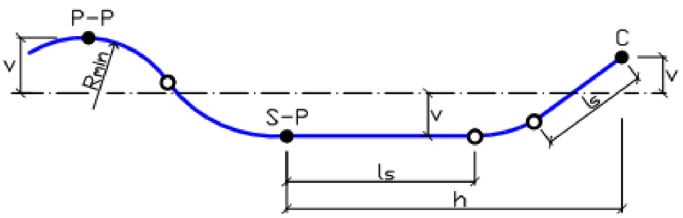


Figure 3.44: Segment type 4

This segment type is almost a mirror type to segment type 3. This segment type can be used as the last segment in a tendon and it follows-up the previous segments.

Segment type 5 – Parabolic with straight inner

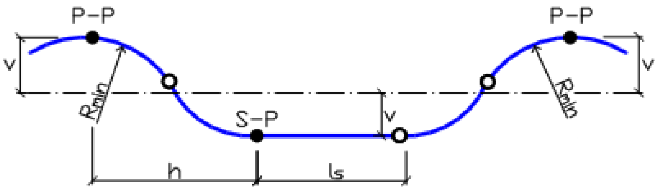


Figure 3.45: Segment type 5

This segment type can be placed only between two other segments, so it is an inner tendon segment. The segment consists of five curves - parabola, parabola, straight line, parabola and parabola. The straight part can be omitted, also the parabolas can change to lines according to the entered parameters.

The segment is defined by **point P-P** at the beginning and **point P-P** at the ending, and by an intermediate **point S-P**.

Segment type 6 – Straight closing left

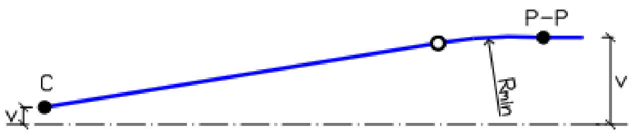


Figure 3.46: Segment type 6

This segment can be used as a beginning segment of tendon geometry. It starts with a straight part, which changes to a parabolic part to connect the following segment.

The segment is defined by starting **point C** and ending **point P-P**.

Segment type 7 – Straight closing right

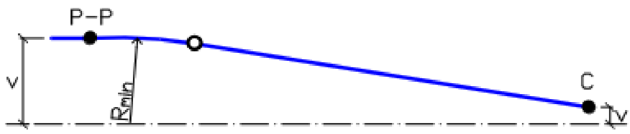


Figure 3.47: Segment type 7

This segment type is mirror type to segment type 6 and can be used as a last tendon segment, which follows the previous segments.

3.2.1.3 Composition of segments to create tendon geometry

Possibilities of how to compose tendon geometry using several numbers of segments are described in this section.

Tendons consisting of one segment

If the tendon geometry in the uncoiled view consists of one segment only, two types of stand-alone segments can be selected:

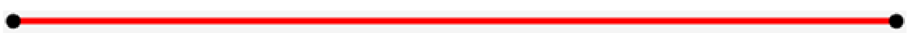


Figure 3.48: Tendon consisting of one segment - Type 1 – straight stand-alone

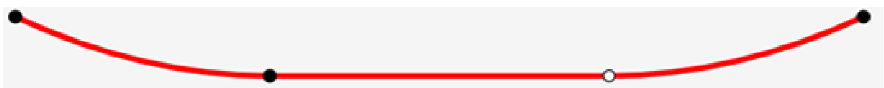


Figure 3.49: Tendon consisting of one segment - Type 2 – parabolic with straight stand-alone

Neither of these two segments can be combined with another segment type.

Tendons consisting of two segments

Four segment types can be used to define tendons composed from two segments – two types for the first segment and two types for the second segment.

The following segment types can be used for the first segment:

Type 3 – parabolic with straight left

Type 6 – straight closing left

The following segment types can be used for the second segment:

Type 4 – parabolic with straight right

Type 6 – straight closing right

Possible combinations of segment types are displayed in the following table:

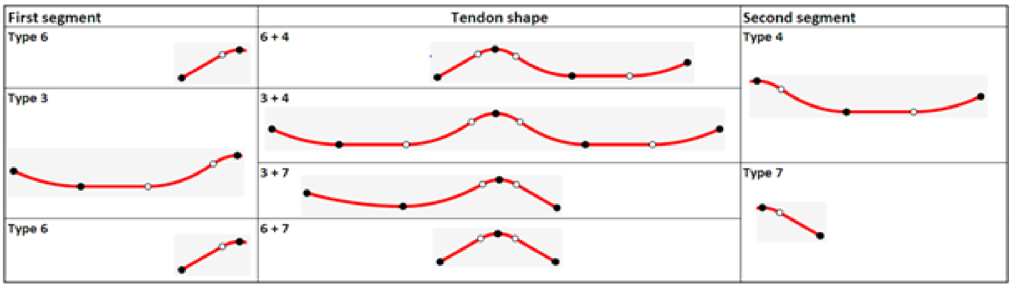


Figure 3.50: Tendons consisting of two elements

Tendon containing three and more segments

The geometry of a tendon consisting of three and more segments is composed similarly as the geometry of a tendon consisting of two segments. Identical segment types can be used for the outer segments, and for internal segment(s) only segments of type 5 can be used.

Possible combinations of segment types are displayed in the following table:

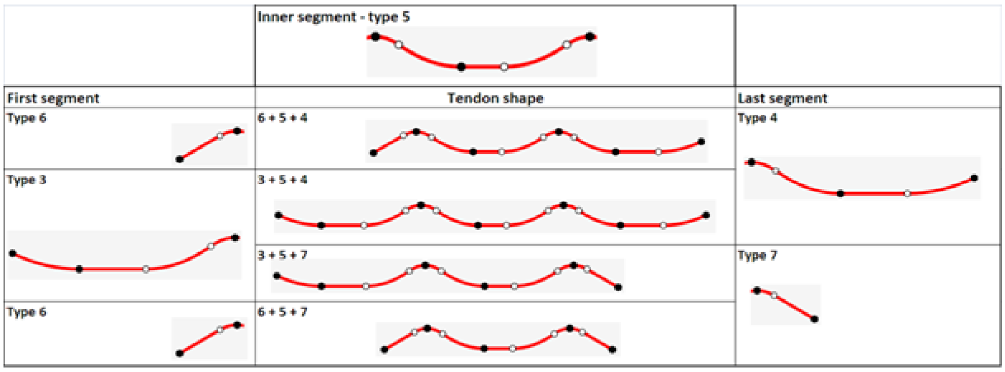


Figure 3.51: Tendons containing three and more segments

3.2.1.4 Rules and limitations for compositions of segments

All combinations of the segments listed above have the following limitations:

Neighboring segments have a common tangent at the segments border (point P-P). The tangent at this point is parallel to the x-axis.

The minimum radius of the parabola in point P-P has the same values for both parabolas from the left and from the right.

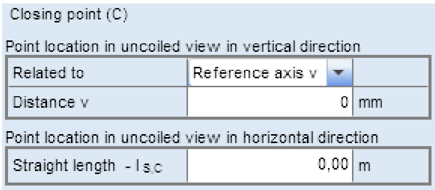
The inner straight parts of the segments are always parallel to the reference x-axis of the member. This is not true for straight closing segments and stand-alone straight segments.

3.2.1.5 Description of tendon definition geometry points

The geometry of each tendon segment is defined by two or three characteristic points, depending on the type of segment. Those points are drawn as filled black circles. The current point (selected to be edited) is drawn as a filled red circle. Points can be selected in the picture by using the mouse. Other points, which are necessary to define the geometry, are calculated automatically. Those points are for example points at the endings of straight tendon segments or points in the transition between inverted parabolas. Those points are drawn with a black circle filled with white color and they cannot be selected and edited. Their position depends on the defined length of the straight parts.

Point C – tendon closing points

Point *C* is always located at the beginning of the first segment or at the ending of the last segment. Therefore only the distance *v* from the member reference curve in plane XY or plane XZ is defined. The properties of point C can be edited in the following table:



Closing point (C)

Point location in uncoiled view in vertical direction

Related to	Reference axis v	
Distance v	0	mm

Point location in uncoiled view in horizontal direction

Straight length - $l_{s,c}$	0,00	m

Figure 3.52: Point C

Related to – specify the origin for determination of final vertical tendon point coordinate v. The following options can be selected (e.g. in plane XZ – see Figure 3.53):

Maximum Z+ – maximum positive coordinate Z

Edge intersection Ze+ – maximum positive Z+ coordinate of intersection of the line parallel to the Z-axis drawn in Y coordinate of tendon with the edge of the cross-section

Reference axis v – distance from the reference axis

Centre of gravity Zcg – vertical distance from the center of gravity

Edge intersection Ze- – minimum negative Z- coordinate of the intersection of the line parallel to the Z-axis drawn in the Y coordinate of the tendon with the edge of the cross-section

Minimum Z- – minimum negative coordinate Z

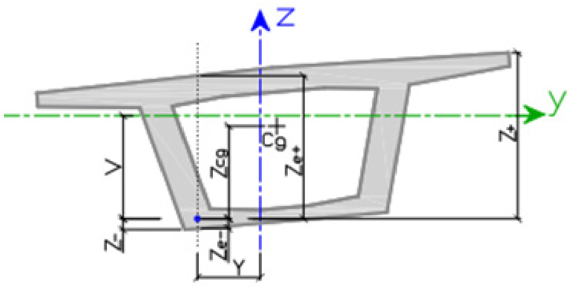


Figure 3.53: Point relations

Distance v – point distance measured from the defined origin, positive value is in the positive direction of beam Z-axis (Y-axis)

Straight length - $l_{s,c}$ - length of the straight part of the tendon measured from the beginning (ending) point of the segment

Point S-P – inner point between straight and parabolic segment

Point *S-P* is always an inner point of a tendon. The point properties can be edited in the following table:

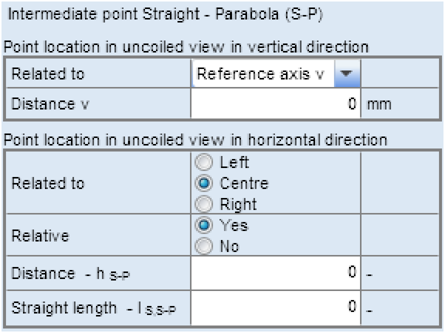


Figure 3.54: Point S-P

Related to – specify the origin for determination of the final vertical tendon point coordinate v

Distance v – point distance measured from the defined origin, positive value is in the positive direction of the beam Z-axis (Y-axis)

Related to – specify the origin for the input of the horizontal point position. The following points can be used as a reference point for the horizontal coordinate $h_{S\text{-}P}$:

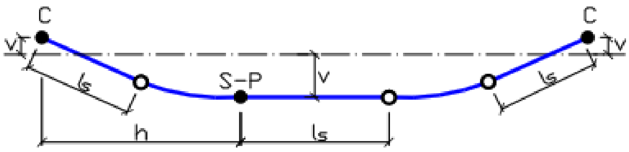


Figure 3.55: Beginning point of segment

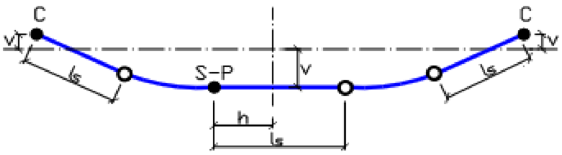


Figure 3.56: Middle point of segment

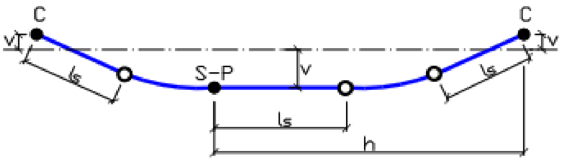


Figure 3.57: Ending point of segment

Relative – switch of input mode for input of distance h and straight length ls

Distance $h_{S\text{-}P}$ – horizontal distance h of selected point

Straight length $l_{s,S\text{-}P}$ – length of inner straight part of tendon

Point P-P – connection point between parabolas

Point *P-P* is always located in connection of two segments and it defines the transition between parabolas. Point properties can be edited in the following table:

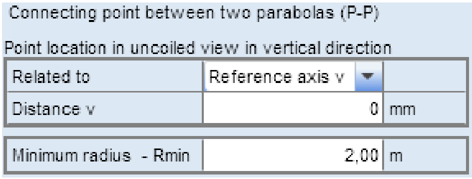


Figure 3.58: Point P-P

Related to – specify the origin for determination of final vertical tendon point coordinate v

Distance v – point distance measured from defined origin, positive value is in positive direction of beam Z-axis (Y-axis)

Minimum radius - Rmin – minimum radius of parabola

3.2.1.6 Tendon properties – editing

To edit general tendon properties click the **Tendons** table in the Data window.

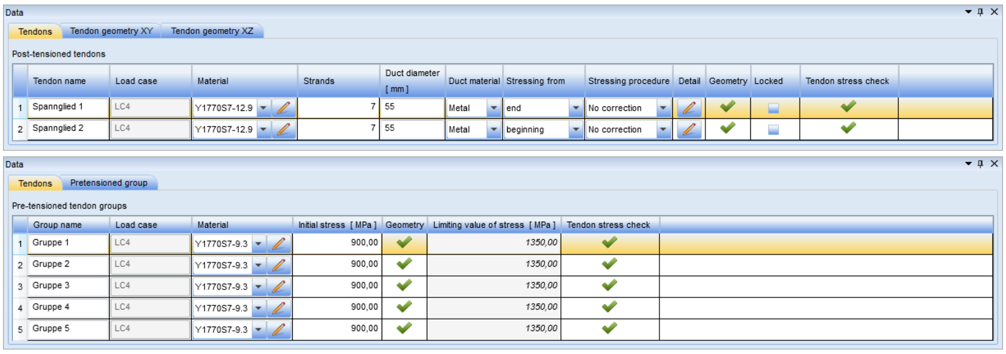


Figure 3.59: Tendons

The following properties are displayed and some can be edited:

Material – select current material of tendon. Click the edit button to change the properties of the current material.

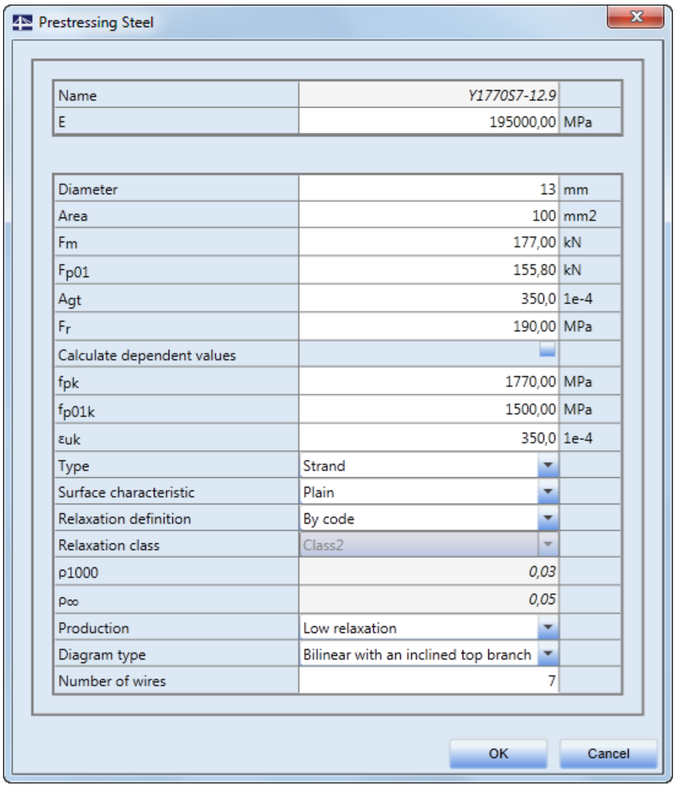


Figure 3.60: Material – edit properties

Strands – number of strands in tendon

Duct diameter – value of minimum duct diameter. Default value of minimum duct diameter is calculated according to the area of the tendon.

Duct material – select material of tendon duct. Two materials are available – Plastic or Metal.

Stressing from – select stressing mode. Tendons can be stressed from the beginning of the design member, from the ending of the design member or from both ends of the design member.

Stressing procedure – select stressing procedure. A stressing procedure with or without correction of relaxation can be selected.

No Correction – tendon is prestressed and is anchored immediately after prestressing

Correction of relaxation – tendon is prestressed for some time (usually 2- 10 minutes) before anchoring and is kept at the same tension and after this time is anchored. This process reduces relaxation of steel of tendon.

Detail – Click the edit button to display a dialog box with detailed tendon properties.

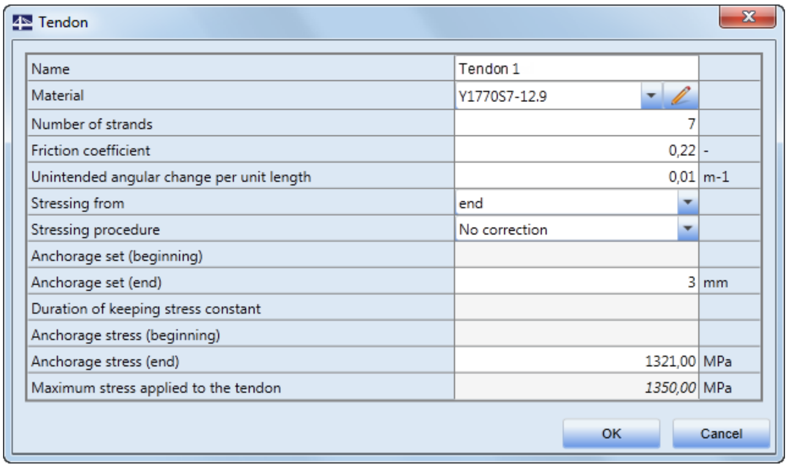


Figure 3.61: Detail – edit properties

Tendon detail dialog box:

Name – name of selected tendon

Material – select current material of tendon. Click the edit button to change the properties of the current material.

Number of strands – number of strands in tendon

Friction coefficient – value of tendon friction coefficient

Unintended angular change per unit length – value expressing the increase of tendon friction losses due to unintended ripple of tendon

Stressing from – select stressing mode. Tendons can be stressed from the beginning of the design member, from the ending of the design member or from both ends of the design member.

Stressing procedure – select stressing procedure. Stressing procedure with or without correction of relaxation can be selected.

No Correction – tendon is prestressed and is immediately anchored after prestressing.

Correction of relaxation – tendon is prestressed for some time (usually 2- 10 minutes) before anchoring and is kept at the same tension and after this time is anchored. This process reduces relaxation of steel of tendon.

Anchorage set (beginning) – value of anchorage set at the beginning of the tendon. Value is available if stressing from beginning is set.

Anchorage set (end) – value of anchorage set at the ending of the tendon. Value is available if stressing from end is set.

Duration of keeping stress constant – value of time to keep the stress constant during stressing. Value is available only for stressing with correction of relaxation.

Anchorage stress (beginning) – value of anchorage stress at the beginning of the tendon. Value is available if stressing from beginning is set.

Anchorage stress (end) – value of anchorage stress at the ending of the tendon. Value is available if stressing from end is set.

Maximum stress applied in tendon – value of maximum stress in tendon

Geometry – Display the status of the tendon geometry. Result value depends on partial results of checks of all tendon segments in both uncoiled views.

The geometry of the tendon is valid if the following assumptions are fulfilled:

Geometry of all segments is valid

Continuity of segments must be smooth, which means that the tangent of the angle in segment transitions has to be equal to zero.

Geometry of the design member must be valid, which means that all members of the design member must continue correctly.

If the tendon geometry is not valid the tendon cannot be analyzed, tendon losses and equivalent loads cannot be calculated, and the corresponding design member cannot be checked. Tendons with invalid geometry cannot be exported from the application.

Locked – If selected, the tendon is locked and the tendon properties cannot be edited.

Tendon stress check – Display result of maximum stress in the tendon check according to EN 1992-1-1 5.10.2.1(1)P.

Initial stress – editable initial stress for pretensioned group of tendons

Limiting stress – limiting stress for pretensioned group of tendons

3.2.1.7 Post-tensioned tendon geometry – editing

Tendon geometry is edited separately for both uncoiled views XY and XZ. A table with tendon geometry is displayed in the *Data* window.

Tendon geometry XY – edit tendon geometry in uncoiled view XY

Tendon geometry XZ – edit tendon geometry in uncoiled view XZ

Editing tables correspond to drawing of uncoiled views where the following entities are drawn in different colors:

Selected tendon segment (thick red line in default settings)

Selected characteristic point (filled red circle in default settings)

The following items are available for both uncoiled views on particular tabs:

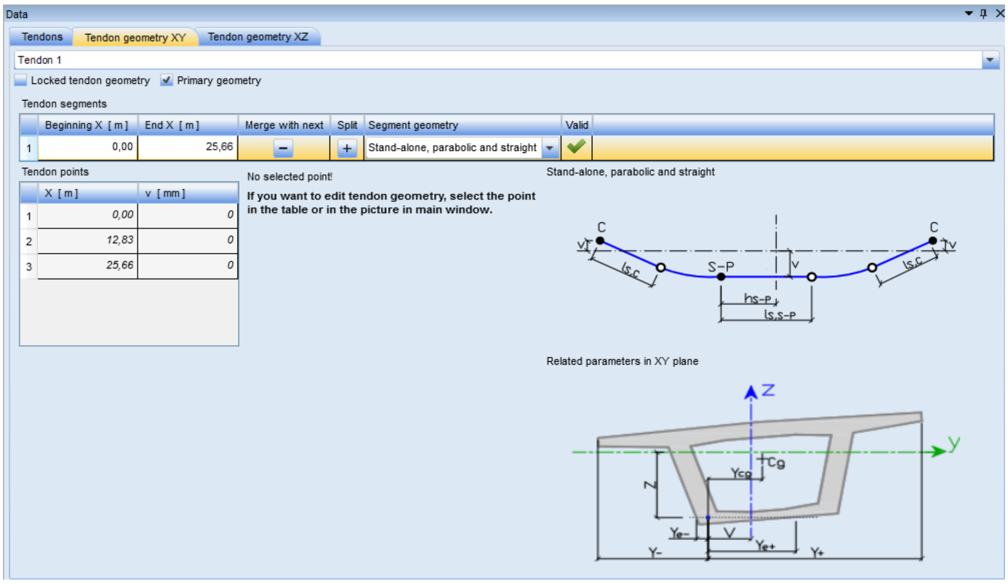


Figure 3.62: Tendon geometry

List of existing tendons – current tendon can be set in the list

Locked tendon geometry – if selected, the tendon geometry cannot be edited

Primary geometry – if selected, the tendon geometry in the appropriate plane is assumed to be the primary tendon geometry (it is switching between geometry XY and XZ)

Tendon segments – individual tendon segments can be edited in this table

Tendon points – points of the current tendon segment are listed in this table

Tendon point properties – edit the current tendon point properties in this table

Primary geometry

The primary tendon geometry determines the primary uncoiled view for the input of tendon position in the cross-section. According to the principle of tendon geometry input using two independent uncoiled views it is necessary to determine the primary uncoiled view, if characteristic tendon points refer to intersections with the cross-section edges in the second uncoiled view. Those intersections are determined using:

The vertical line drawn in the **Y**-coordinate of the tendon with the edge of the cross-section in the primary uncoiled view XY. The **Z**-position of the tendon in the cross-section can refer to the intersection of this vertical line with the cross-section edges. All available reference points for the input of tendon **Z**-position are displayed in Figure 3.63. Intersections of the vertical line in the **Y**-coordinate of the tendon with the cross-section edges are depicted with dimension lines **Ze**$^+$ and **Ze**$^-$.

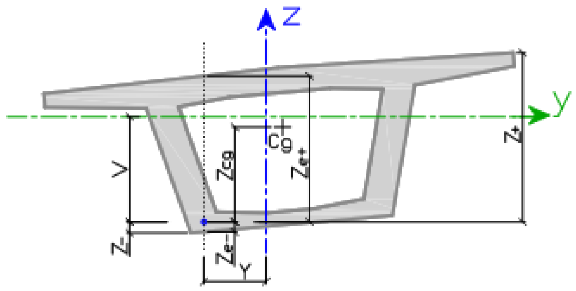


Figure 3.63: Vertical line

The horizontal line drawn in the **Z**-position of the tendon in the cross-section in the primary uncoiled view XZ. The **Y**-position of the tendon in the cross-section can refer to the intersection of this horizontal line with the cross-section edges. All available reference points for the input of tendon **Y**-position are displayed in the following picture. Intersections of the horizontal line in the **Z**-coordinate of the tendon with the cross-section edges are depicted with dimension lines **Ye**$^+$ and **Ye**$^-$.

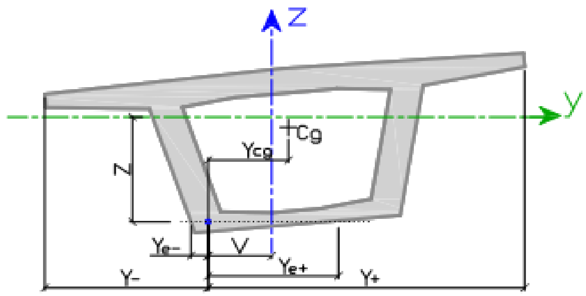


Figure 3.64: Horizontal line

The primary uncoiled view does not allow defining reference points on the cross-section edges; it can use only minimum or maximum coordinates of the cross-section.

Tendon segments

Tendon segments

	Beginning X [m]	End X [m]	Merge with next	Split	Segment geometry	Valid
1	0,00	25,66	–	+	Stand-alone, parabolic and straight	✔

Figure 3.65: Tendon segments

All segments of the current tendon are listed in the *Tendon segments* table. The table contains following columns:

Beginning – position of the beginning of the tendon segment measured in the axis of the uncoiled view from the beginning of the design member

End – position of the end of the tendon segment measured in the axis of the uncoiled view from the beginning of the design member

Merge with next – click the [-] button to remove segment by merging the current segment with the following one. The merge of segments causes a change to the segment geometry and the length of the segment is the sum of segment lengths before merging.

Example: By merging the following segments one straight parabolic segment is created.

	Beginning X [m]	End X [m]	Merge with next	Split	Segment geometry	Valid
1	0,00	12,83	−	+	Closing, parabolic and straight left	✔
2	12,83	25,66	−	+	Closing, parabolic and straight right	✔

Figure 3.66: Before merging

	Beginning X [m]	End X [m]	Merge with next	Split	Segment geometry	Valid
1	0,00	25,66	−	+	Stand-alone, parabolic and straight	✔

Figure 3.67: After merging

Split – click the [+] button to split the current tendon segment into two segments of the same length. Depending on the position of the current segment the geometry of the segment can change.

Description of tendon definition geometry segments

Segment geometry – Select a segment type from the available tendon segment types. All types of tendon segment geometry are described in chapter 3.2.1.2 *Description of tendon definition geometry segments*. The content of the list is filtered automatically to display only the allowed segment types. E.g. if a tendon consists of one segment only, its geometry can be defined using segment types 1 or 2. The geometry of the current segment including the described characteristic points is drawn below this table.

Valid – The geometry of each tendon segment is checked automatically. The validity of the segment geometry can be verified in the tendon segments table. Probable causes of invalid geometry are:

- Parabola with the minimum radius cannot be inserted
- Entered lengths of straight segments are longer than the length of the segment
- The whole segment or part of it is outside of the design member

Tendon points

Coordinates of characteristic points of the current tendon segment are listed in the*Tendon points* table. Coordinates cannot be edited because they are calculated from the uncoiled tendon geometry.

Tendon points

	X [m]	v [mm]
1	0,00	0
2	12,83	0
3	25,66	0

Closing point (C)

Point location in uncoiled view in vertical direction

Related to	Reference axis v	
Distance v	0	mm

Point location in uncoiled view in horizontal direction

Straight length - $l_{s,C}$	0,00	m

Figure 3.68: Tendon points

The *Tendon points* table contains the following columns:

Point number – number of the characteristic tendon point in the uncoiled view

X – The position of the point measured in the uncoiled view from the beginning of the design member.

v – The position of point Y or Z for the uncoiled view XY or XZ relative to the cross-section coordinate system origin.

Tendon point properties

Next to the *Tendon points* table is a table for editing the current characteristic point parameters. The current characteristic point can be set by selecting the appropriate row in the *Tendon points* table or by clicking on the drawing of uncoiled view. Tables for all available types of characteristic points are described in chapter 3.2.1.5 *Description of tendon definition geometry points*.

3.2.1.8 Non continuous tendons

On members of polygonal shape

In the point of the design member fracture "tendon tearing" appears, because the corresponding local coordinate systems of members (or member parts) are not identical in the point of fracture. In the picture the fracture of the design member is visible, and the local coordinate systems of the following members are not identical.

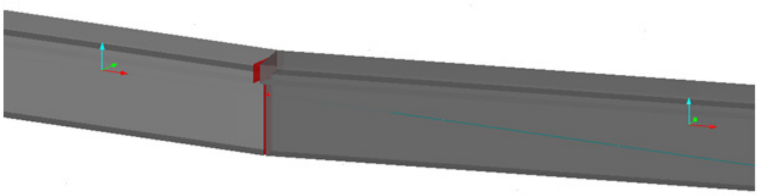


Figure 3.69: Fracture of design memeber

In this case the particular tendon segments begin or end at a point which lies in the plane perpendicular to the reference axis in the point of fracture. If this point lies on the outside of the break, the tendon can appear to be ruptured.

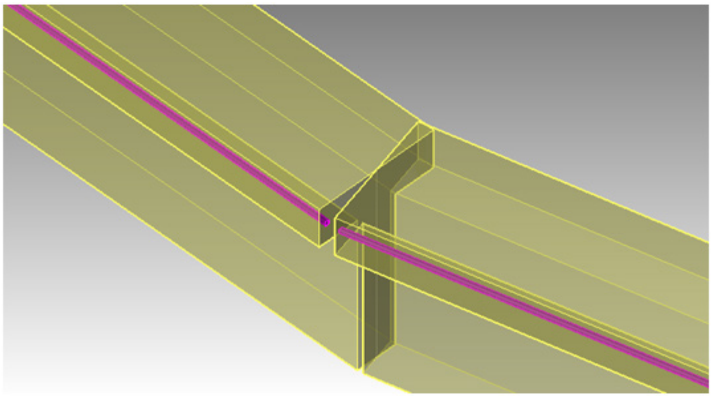


Figure 3.70: Polygonal shape - point lies outside of the break

If the point lies on the inside of the break, tendon segments can cross.

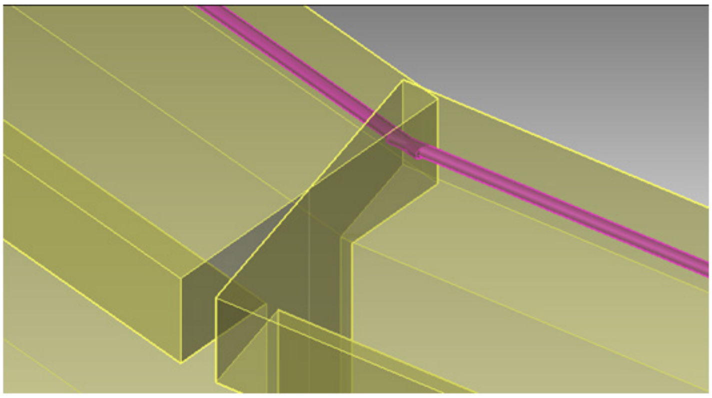


Figure 3.71: Polygonal shape - point lies inside of the break

The angular change between the tangents of the tendon ends at the point of rupture is assumed in tendon analysis.

On rotated members

An identical case appears if two neighboring member parts do not have an identical local coordinate system, but the LCS differs only in rotation about the x-axis, and therefore the angle between the Y-axes does not equal zero. The tendon is broken in this point too, but both end points lie in one plane, which is also perpendicular to the reference line. This difference is not taken into account in the calculation. It is assumed that the rotation between members is very small (the order of degrees). If it is not the case, the analytical model should be adapted.

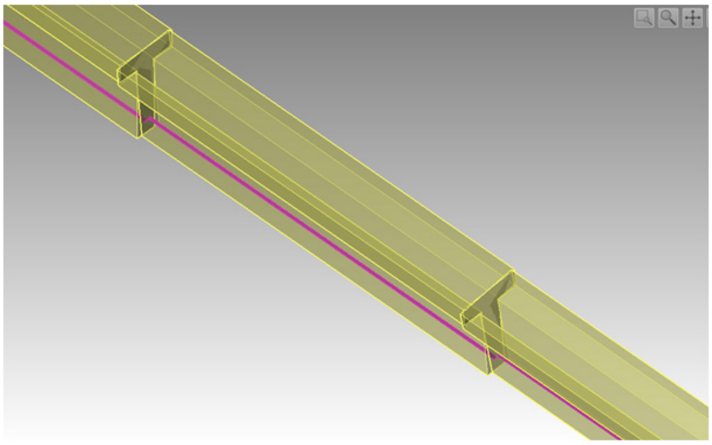


Figure 3.72: Rotated members

3.2.1.9 Pretensioned tendon groups geometry – editing

To edit the geometry of a pretensioned tendon group click the **Pretensioned group** tab in the Data window. The selected group is displayed in the Data window and the parameters can be edited. The group can contain more tendons. The location of tendons is same in the whole design beam and is defined by the cover relative to the selected edge and controlled by dimension lines. By using blanketed (debonding) length it is possible to define the length without coherence between tendon and concrete.

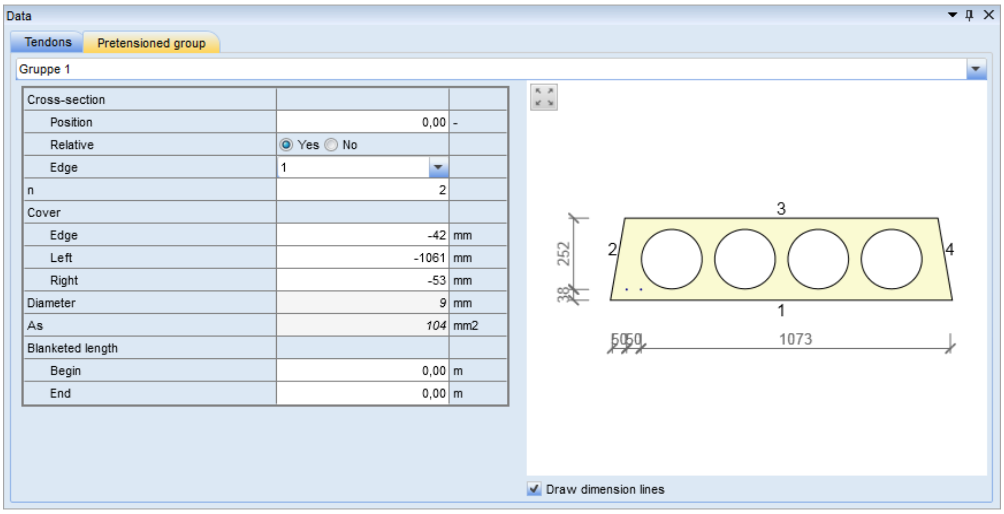


Figure 3.73: Pretensioned group

3.3 Force Design

The program enables the evaluation actions caused by tendons on the concrete member and balancing the effects of external loads by tendon layout design.

3.3.1 Equivalent Load

If the distribution of the prestressing force along the length of tendon is known, then the effect of the tendon on the structure can be examined and the equivalent load of prestressing can be established. If the equivalent load is applied on a given structure, the distribution of internal forces due to prestressing is obtained.

To evaluate actions caused by a tendon on the concrete member (equivalent load) for the current design member click **Equivalent Load** in the navigator.

The ribbon groups *Load setting*, *Component of equivalent load*, *System*, *Extreme*, *Load view*, *Uncoiled view*, *Load display* and *Current section* are available for evaluation of equivalent loads.

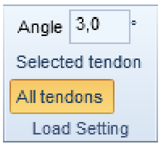


Figure 3.74: Load setting

Calculation precision and evaluated tendons can be set in the *Load setting* ribbon group.

Angle – value of the maximum angular change of the tendon (geometrical discretization) for tendon losses and equivalent load calculation. Default value is 3,0°. The set angle also influences the equivalent load in the Prestress load case in RFEM.

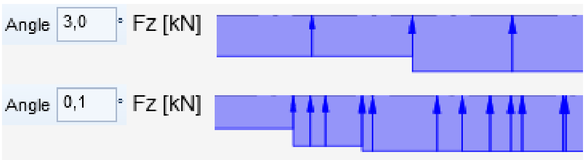


Figure 3.75: Comparison different values of Angle

Selected tendon – turns on evaluation of equivalent load graphs only for the current tendon in the current design member

All tendons – turns on evaluation of equivalent load graphs for all tendons in the current design member

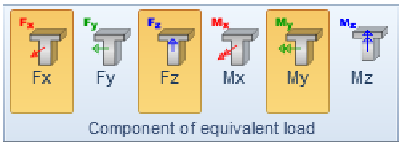


Figure 3.76: Component of equivalent load

The components of the equivalent load to be drawn can be set in the *Component of equivalent load* ribbon group.

Fx – turn on/off drawing of force Fx in coordinate system specified in the *System* ribbon group

Fy – turn on/off drawing of force Fy in coordinate system specified in the *System* ribbon group

Fz – turn on/off drawing of force Fz in coordinate system specified in the *System* ribbon group

Mx – turn on/off drawing of moment Mx in coordinate system specified in the *System* ribbon group

My – turn on/off drawing of moment My in coordinate system specified in the *System* ribbon group

Mz – turn on/off drawing of moment Mz in coordinate system specified in the *System* ribbon group

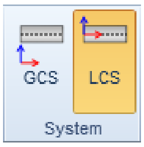


Figure 3.77: System

The coordinate system for evaluation of equivalent loads can be set in the *System* ribbon group.

GCS – turn on evaluation of equivalent loads in the global coordinate system

LCS – turn on evaluation of equivalent loads in the local coordinate system of the design member

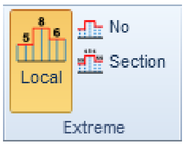


Figure 3.78: Extreme

A description of equivalent load values can be set in the *Extreme* ribbon group:

Local – Values of local extremes of equivalent load are depicted along the design member.

No – No values of equivalent load are depicted.

Section – Values of equivalent loads are depicted in each section.

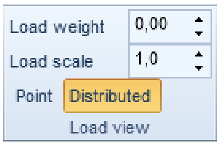


Figure 3.79: Load view

The *Load view* ribbon group can be used to set drawing options of equivalent loads.

Load weight (for drawing of loads) – Load weight suppresses real scale of load drawing. It can have values between 1,0 to -1,0. If the load weight is equal 0, the real scale of the load proportional to its value will be maintained. Positive weight accentuates small values of the load and suppresses large values. If the load weight is equal to 1, all loads are drawn in equal size. Negative value suppresses the drawing of that load which is smaller than the product of the maximum load and the absolute value of the load weight. This means for a load weight equal to -1 only the maximum load is drawn, for a load weight equal to -0,5 all loads greater than half of the maximum load are drawn.

Load scale – Value of multiplier for drawing of load effects. Default value is 1,0.

Point – Draw calculated equivalent loads as point loads in points generated by discretization of the tendon.

Distributed – Draw calculated equivalent loads as distributed load along the whole length of the design member.

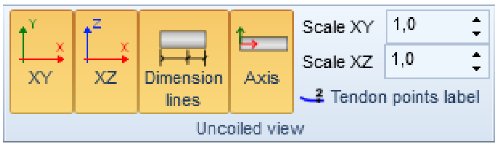


Figure 3.80: Uncoiled view

See chapter 3.1.2.2 for a description of the function of this ribbon group.

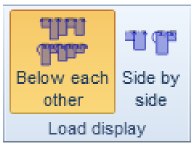


Figure 3.81: Load display

The pattern of load pictures can be set in the *Load display* ribbon group.

Below each other – Draw individual components of equivalent loads below each other.

Side by side – Draw components Fx, Fy and Mz in one column and components Mx, Fz and My in the second column.

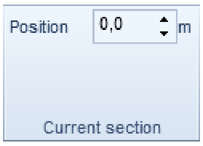


Figure 3.82: Current section

Position – The value of the distance of the current section from the beginning of the design member. Current section details are displayed in the *Info* window.

The graph of **equivalent load** according to the current settings is drawn in the *Main* window.

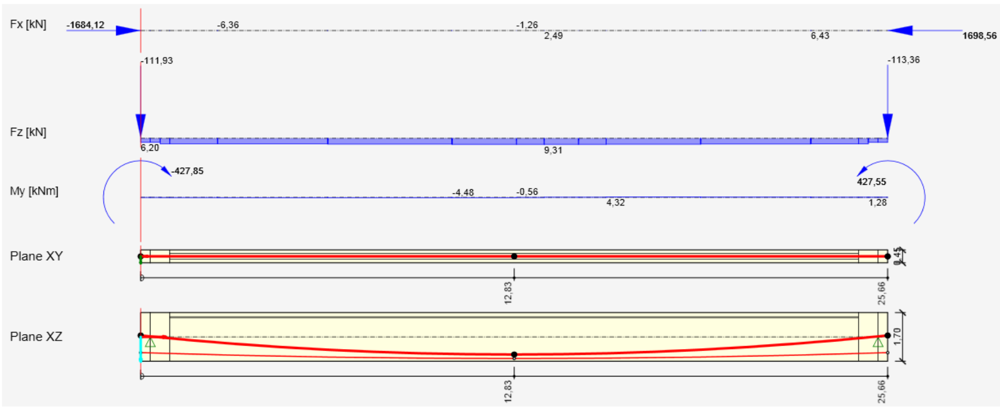


Figure 3.83: Main window - Equivalent Load

Tabs for tendons editing and textual presentation of equivalent loads are displayed in the *Data* window.

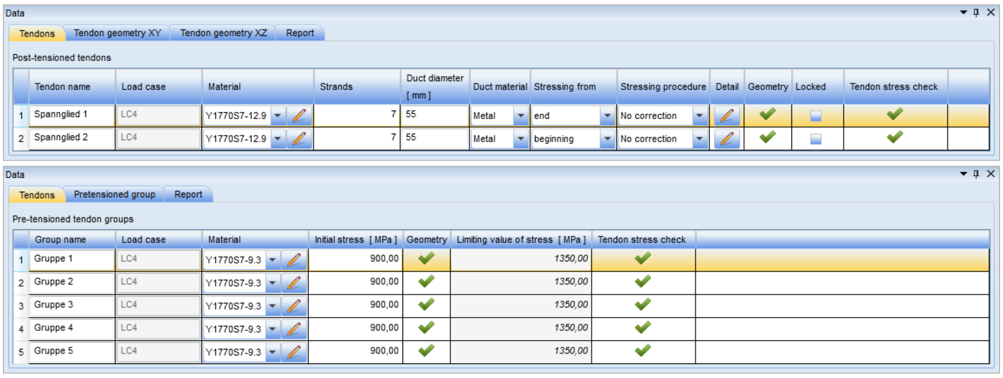


Figure 3.84: Data window – Tabs for tendons

Individual tabs in the *Data* window are:

Tendons – table with post- or pre- tensioned tendon properties. Equivalent load graphs update automatically after the tendon properties change.

Tendon geometry XY – table with post-tensioned tendon geometry properties in the XY-plane

Tendon geometry XZ – table with post-tensioned tendon geometry properties in the XZ-plane

Pretensioned group – table with pre-tensioned tendon geometry properties in the section

Report – textual presentation of equivalent loads

3.3.2 Load Balancing

The load balancing method is used for design of prestressing. It will find the balance between the equivalent loads from prestressed tendons and the external permanent loads. For balancing it is recommended to use 80-100% of all permanent loads. It depends the required level of prestressing and quality of the calculation method used for analysis of deflections. The level of prestressing should not be lower than 80% of all permanent loads even when partial prestressing is introduced and nonlinear analysis taking into account the effect of cracks on the stiffness of the structure is applied.

To display equivalent loads together with external loads actions, click **Load Balancing** in the navigator.

The ribbon groups *Load, Load setting, Direction, Extreme, Load view, Uncoiled view* and *Current section* are available for load balancing.

The ribbon groups *Load setting, Extreme, Load view, Current section* are described in chapter 3.3.1 *Equivalent Load*, the ribbon group *Uncoiled view* in chapter 3.1.2.2 *Design member view*.

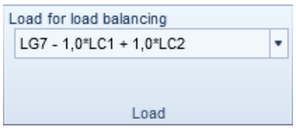


Figure 3.85: Load

Use the ribbon group *Load* to select a load for load balancing.

The load combination for load balancing was prepared in RFEM.

Example (see Figure 3.86)

Load cases : LC1 – Self-weight, LC2 – Permanent load

Load combinations usually used for load balancing : LG7 = 1,0*LC1 + 1,0*LC2

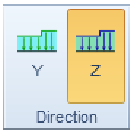


Figure 3.86: Direction

Use the *Direction* ribbon group to set the direction for the graphical evaluation of load balancing results.

Y – draw load balancing results in the direction of the Y-axis of the current design member local coordinate system

Z – draw load balancing results in the direction of the Z-axis of the current design member local coordinate system

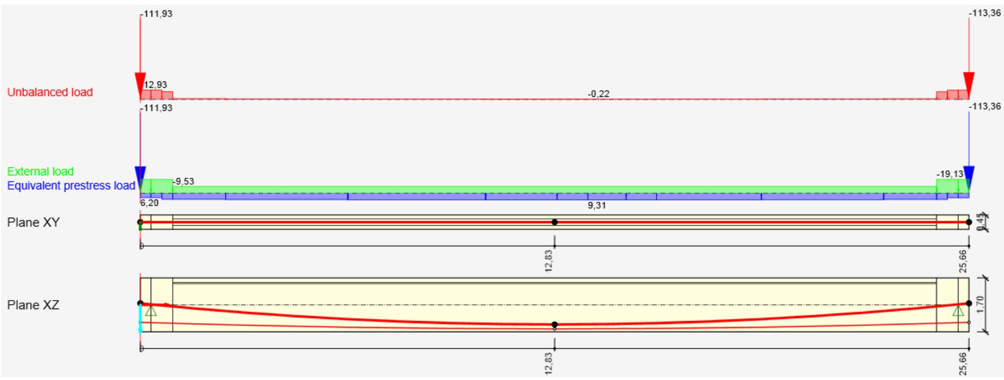


Figure 3.87: Main window - Load balancing

Loads along the design member are drawn in the Main view:

Unbalanced load – graph of the difference between actions of external loads caused by the current result class and prestressing actions caused by the current tendon or all tendons in the current design member

External load – graph of actions of external loads caused by the current result class

Equivalent prestress load – graph of equivalent load caused by the current tendon or all tendons

Tabs for tendons editing and textual presentation of load balancing are shown in the *Data* window.

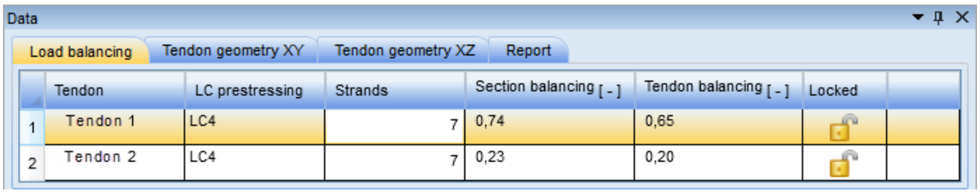


Figure 3.88: Data window – Tabs for tendons

Particular tabs in the *Data* window:

Load balancing – display table with basic information about load balancing. The table contains the following columns:

Tendon – name of the evaluated tendon

LC prestressing – name of the load case which is determined to transfer the effect of prestressing into the analytical model

Strands – number of strands per tendon

Section balancing – ratio between external load / equivalent prestress load in the selected section

In Figure 3.88:

Tendon 1 ext. load / eq. prestress load = 0,74

Tendon 2 ext. load / eq. prestress load = 0,23

0,74 + 0,23 = 0,97 … 97 % of external load is balanced in the selected section

Tendon balancing – ratio between external load / equivalent prestress load over the whole design member (the integral of the loading is calculated)

In Figure 3.88:

Tendon 1 ext. load / eq. prestress load = 0,65

… 65 % of the tendon is on average used in the whole design member

Tendon 2 ext. load / eq. prestress load = 0,20

… 20 % of the tendon is on average used in the whole design member

Locked – indication of locked tendon

Tendon geometry XY – table with tendon geometry properties in the XY-plane

Tendon geometry XZ – table with tendon geometry properties in the XZ-plane

Report – textual presentation of unbalanced loads

3.4 Short-term Losses

In this part of calculation the following short-term losses are calculated:

Frictional loss – Friction between the tendon and walls of the tendon duct occurs during stressing. This loss consists of two components: curvature and wobble effect frictional losses. The curvature frictional loss (in a curved part of the tendon) results from (intended, designed) change of direction of the tendon. From an unintentional change of the direction (deformation of the ducts fixed by spacers due to self-weight) results also wobble effect frictional losses.

It is present especially in post-tensioned prestressed concrete. In pre-tensioned prestressed concrete the loss due to friction occurring at the draping points of the prestressing reinforcement is considered.

Anchorage set loss – The slip of the wedge and the strand (anchorage set) in the anchor head results in the reduction of stress in the prestressing reinforcement. This is termed *anchorage set loss*.

It takes place in both pre-tensioned and post-tensioned prestressed concrete.

Relaxation loss (of prestressing reinforcement) – Relaxation of prestressing steel reduces stress over time in the steel that is subjected to a constant deformation load (elongation).

That relaxation increases with increasing stress in the steel and there exists such a stress level, under which the relaxation of steel no longer occurs. This limit reaches approximately 50 % of the characteristic 0,2 % proof stress.

Relaxation of prestressing reinforcement is negative (it reduces prestress) and it is minimized by the application of special materials (low-relaxation strands) and a special process by prestressing termed *correction of relaxation*.

Tendon losses are calculated on tendon analysis geometry. Tendon analysis geometry is slightly different from drawn geometry, which consists only of points.

The analysis geometry is composed in a similar way as described in chapter 3.2.1.1 *3D Tendon geometry*. The information about tendon curvature between particular calculation points and real lengths between those points is stored. Moreover information required for calculation of tendon on refracted member is added in chapter 3.2.1.8 *Non continuous tendons*.

Analysis geometry depends on tendon segment division. Parts similar to finite elements are created during division. Calculation of losses is performed on those divided parts. The value of maximum angular change between spatial tangents to tendon segment ends determines the size of tendon parts. It is the maximum angle, so some parts may have a smaller angle. The tendon parts must correspond with member parts, so they have to begin and end at the beginning or the end of the member.

Example: Straight tendon on a straight member. This member has three parts of the member. Even if no angular change along the member exists, the tendon is split into three segments.

Analysis values of individual tendons are specified during the input of tendons in **Tendons Layout** in the navigator.

To edit analysis values click the edit button in the *Detail* column. According to the origin of stressing, values are entered for the beginning or the end or both ends of the tendon. When stressing procedure with relaxation correction is set, the time to keep stress has to be entered.

The maximum angular change is valid for all tendons. The recommended basic value is approximately 3°. Lesser values do not have significant influence on calculation precision.

3.4.1 Summary

To display the overall results of tendon losses click **Summary** in the navigator. The *Uncoiled view* ribbon group is described in 3.1.2.2 Design member view and the ribbon group *Tendon shape* is described in chapter 3.2.1 Tendon Layout.

The uncoiled view of the design member or detailed drawing of the current or of all tendons is drawn in the Main window.

An overall report of tendon losses on the current design member is shown in the *Data* window:

Table with particular tendon values (area, length, cumulative angular change, minimum radius, theoretical tendon elongation before anchoring etc.).

Summary table of minimum and maximum stress in tendons with value of the maximum allowed stress acc. to EN 1992-1-1 5.10.3(2).

Prestressing

Name	Material	A p [mm2]	Length [m]	L s [m]	L arc [m]	R min [m]	θ [°]
	Strands	σ a [MPa]	σ min [MPa]	σ max [MPa]	e ba [mm]	e aa [mm]	L set [m]
Tendon 1	Y1770S7-12.9	700	25,71	0,00	25,71	123,83	11,8
	7	1321,00	1192,90	1265,65	166	163	10,79
Tendon 2	Y1770S7-12.9	700	25,66	0,00	25,66	401,47	3,7
	7	1310,00	1219,79	1264,09	166	163	12,98

Name	σ ini,max [MPa]	σ lim [MPa]	Check 5.10.2.1(1)P	σ min [MPa]	σ max [MPa]	σ pm0 [MPa]	Check 5.10.3(2)P
Tendon 1	1321,00	1350,00	✓	1192,90	1265,65	1275,00	✓
Tendon 2	1310,00	1350,00	✓	1219,79	1264,09	1275,00	✓

Symbol	Explanation
A p	Area of tendon
Length	Length of tendon
L s	Sum of lengths of straight parts of tendon
L arc	Sum of lengths of curved parts of tendon
θ	Cumulative angular change
R min	Minimum radius
σ a	Anchorage stress
σ min	Minimum stress
σ max	Maximum stress
e ba	Theoretical tendon elongation before anchoring
e aa	Theoretical tendon elongation after anchoring
L set	Length affected by anchorage set
σ ini,max	Maximum initial stress in tendon
σ lim	Limit tendon stress
Check 5.10.3(2)P	Maximum stress check applied to the tendon acc. 5.10.2.1 (1)P
σ pm0	Stress after anchoring
Check 5.10.3(2)P	Maximum stress check applied to the tendon acc. 5.10.3 (2)P

Figure 3.89: Data window – Prestressing summary

Notice:

Length Total length of tendon consists of straight and curved parts $L = L_s + L_{arc}$

Θ Summation of all angular changes of the tendon

R_{min} Minimum radius of curved parts of the tendon which is useful to compare with the minimum allowable radius of selected type of tendon

σ_a Stress by anchoring of tendon $\sigma_a = \sigma_{ini,max}$

σ_{min} Minimum stress in tendon after all short-term losses

σ_{max} Maximum stress in tendon after all short-term losses

e_{ba} Tendon elongation by prestressing of tendon before anchoring

e_{aa} Tendon elongation by prestressing of tendon after anchoring

$e_{ba} - e_{aa} = w$ ... elongation after anchorage set loss
(By prestressing of tendon from both ends it is the summation of elongations)

L_{set} Length of the tendon which is affected by short-term loss of anchorage set

$\sigma_{ini,max}$ Initial prestressing stress of tendon - $\sigma_{ini,max} \leq \sigma_{lim}$

3.4.2 Tendon Stress / Losses

To evaluate short-term losses in detail click **Tendon Stress/Losses** in the navigator.

A graph of the stress before and after anchoring along the tendon is drawn in the *Main* window.

Tables with detailed information about the current tendon are displayed in the *Data* window:

Table of the minimum and maximum stress in the current tendon with the maximum allowable tendon stress acc. to EN 1992-1-1 5.10.3(2).

Table with a detailed description of the current tendon (tendon area, tendon length, cumulative angular change, minimum radius, theoretical elongation before anchoring, etc.)

Table with detailed output of losses in sections according to the specified distance for evaluation. In addition to those sections results are presented in characteristic sections points of anchorage set impact or points of intersection of frictional losses for stressing from both ends.

The ribbon groups *Losses, Labels* and *Labels orientation* are available when evaluating tendon losses.

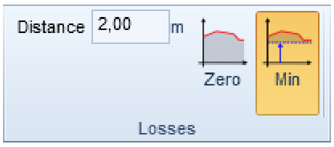


Figure 3.90: Losses

Use the *Losses* ribbon group to set the distance of the sections and then the mode of the graph drawing.

Distance – Value of the distance between sections for graphical and textual evaluation of tendon losses. This value does not affect the precision of calculation.

Zero – Sets the minimum value on the stress axis in the graph to 0.

Min – Sets the minimum value on the stress axis in the graph to a suitable value according to the minimum value of stress in the tendon (e.g. if the minimum tendon stress is 739,3 MPa, the minimum stress value in graph is set to 700 MPa).

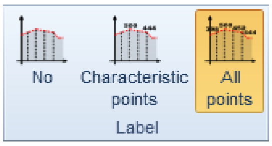


Figure 3.91: Label

Use the *Label* ribbon group to set depiction of sections in the graph.

No – turn off labels for all sections

Characteristic points – turn on labeling of the graph in characteristic points – points of anchorage set impact or points of intersection of frictional losses for stressing from both ends

All points – turns on labelling in characteristic points and in all points according to the specified distance for evaluation of losses

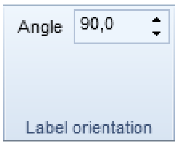


Figure 3.92: Label orientation

Use the *Label orientation* ribbon group to set the angle of labels.

Angle – value of the angle to draw labels of the stress values in the graph

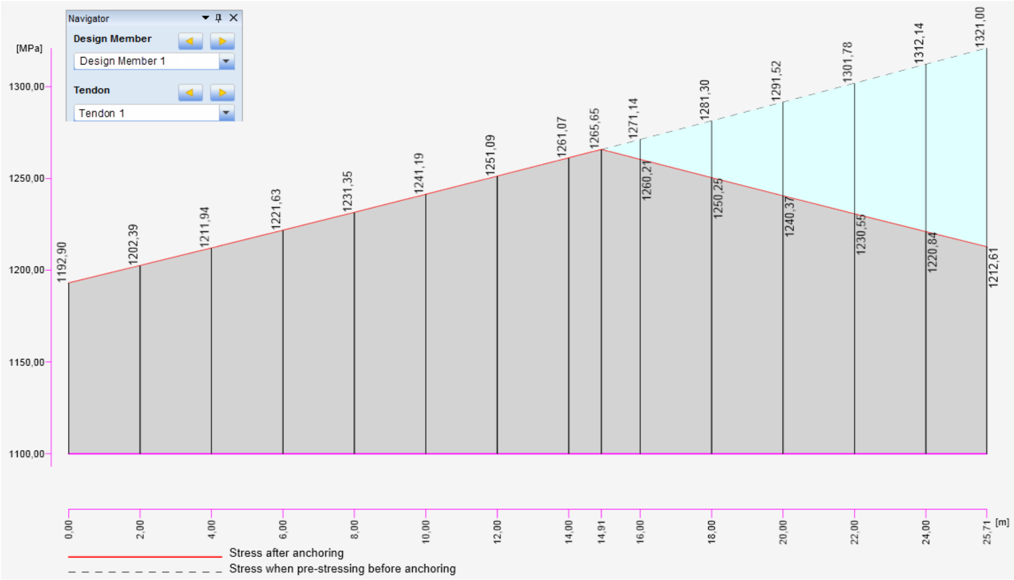


Figure 3.93: Main window - Graph of stress of post-tensioned tendon

Calculation of short-term losses

Tendon : Tendon 1

Maximum stress allowed in tendon immediately after tensioning or transfer acc. 5.10.3(2)

Minimum stress [MPa]	Maximum stress [MPa]	Stress after anchoring σ pm0 [MPa]	Check of stress
1192,90	1265,65	1275,00	✔

Input values and intermediate results

Area of tendon	700 mm2
Length of tendon	25,71 m
Sum of lengths of straight parts of tendon	0,00 m
Sum of lengths of curved parts of tendon	25,71 m
Cumulative angular change	11,8 °
Minimum radius	123,83 m
Anchorage stress	1321,00 MPa
Minimum stress	1192,90 MPa
Maximum stress	1265,65 MPa
Theoretical tendon elongation before anchoring	166 mm
Theoretical tendon elongation after anchoring	163 mm
Length affected by anchorage set - end	10792 mm

Short-term losses

d x [m]	Δσ pμ [MPa]	Δσ pw [MPa]	Δσ pr [MPa]	σ pr,cor [MPa]	σ p0 [MPa]	Δσ pr [MPa]	Δσ pr,cap [MPa]
0,00	-128,10	0,00	0,00	0,00	1192,90	0,00	-41,46
2,00	-118,61	0,00	0,00	0,00	1202,39	0,00	-42,81
4,00	-109,06	0,00	0,00	0,00	1211,94	0,00	-44,18
6,00	-99,37	0,00	0,00	0,00	1221,63	0,00	-45,65
8,00	-89,65	0,00	0,00	0,00	1231,35	0,00	-47,14
10,00	-79,81	0,00	0,00	0,00	1241,19	0,00	-48,71
12,00	-69,91	0,00	0,00	0,00	1251,09	0,00	-50,33
14,00	-59,93	0,00	0,00	0,00	1261,07	0,00	-52,00
14,91	-55,35	0,00	0,00	0,00	1265,65	0,00	-52,79
16,00	-49,86	-10,93	0,00	0,00	1260,21	0,00	-51,86
18,00	-39,70	-31,05	0,00	0,00	1250,25	0,00	-50,19
20,00	-29,48	-51,15	0,00	0,00	1240,37	0,00	-48,58
22,00	-19,22	-71,24	0,00	0,00	1230,55	0,00	-47,01
24,00	-8,86	-91,30	0,00	0,00	1220,84	0,00	-45,52
25,71	0,00	-108,39	0,00	0,00	1212,61	0,00	-44,28

Symbols related to short-term losses

Symbol	Explanation
Δσ pμ	Frictional loss
Δσ pw	Anchorage set loss
Δσ pr	Relaxation loss
σ pr,cor	Stress after short-term relaxation
σ p0	Stress after short-term losses except the loss due to immediate elastic strain of concrete
Δσ pr	Sigma relaxation passed Relaxation that already took place (occurred)
Δσ pr,cap	Remaining relaxation capacity, i.e. potential stress decrease in prestressing reinforcement due to relaxation at infinite time

Figure 3.94: Data window - Table with detailed information about current post-tensioned tendon

$\Delta\sigma_{p\mu}$ – frictional loss of prestressing $\Delta\sigma_{p\mu,l} = - \sigma_{p0,0}\,(1 - e^{-\mu(\alpha+kl)})$

$\sigma_{p0,0}$ – stress in tendon introduced by a jack during stressing (before anchoring) in section „0“ under the anchor at the stressed end (anchorage stress)

μ – coefficient of friction, depends on the roughness of the duct surface and prestressing reinforcement, type and diameter of reinforcement, degree of filling of the ducts with prestressing reinforcement, for multi-strand tendons with ducts made of steel tubes is usually specified within the range 0,05 – 0,5.

α – total intended angular change along tendon length l

k – unintentional angular change, usually is in the interval 0,005 - 0,01m^{-1}

l – total tendon length from the stressed end to the investigated point

$\Delta\sigma_{pw}$ – anchorage set loss – the calculation in the post-tensioned prestressed concrete including the friction between the tendon and the walls of its duct, the calculation is different by stressing from one end only or by stressing from both ends (in pre-tensioned prestressed concrete the calculation of the anchorage set loss is very simple, because the loss due to friction practically does not occur, set *w* decreases elongation *Δl* of the stressed tendon with the length of l, E_p is modulus of elasticity of prestressing steel, therefore, the anchorage set loss can be written as:

$\Delta\sigma_{pw} = - w\, E_p / l$)

$\Delta\sigma_{pr}$ – relaxation loss of prestressing steel, it depends on the level of introduced prestressing and the time, the calculation takes into account the procedure of stressing, anchoring and losses of prestressing due to other effects.

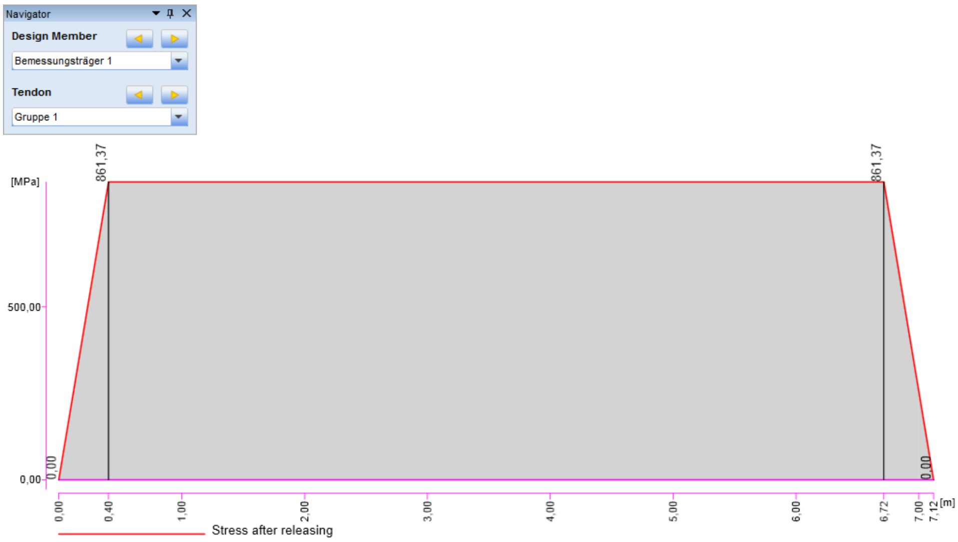


Figure 3.95: Main window - Graph of stress of pre-tensioned group

Maximum stress allowed in tendon immediately after transfer acc. 5.10.3(2)

Maximum stress after transfer [MPa]	Limit value of tendon stress σ pm0 [MPa]	Check of stress
861,37	1275,00	✔

Input values and intermediate results

Area of tendon	52 mm2
Length of tendon	7,12 m
Maximum stress during tensioning	900,00 MPa
Maximum stress after transfer	861,37 MPa
Theoretical tendon elongation before anchoring	231 mm
Theoretical tendon elongation after anchoring	229 mm
Transmission length - begin	0,40 m
Transmission length - end	6,72 m
Blanketed length - begin	0,00 m
Blanketed length - end	0,00 m

Transmission length - begin

f ctd (t) [MPa]	η p1 [-]	η 1 [-]	α 1 [-]	α 2 [-]	ϕ [mm]	σ pm0 [MPa]	f bpt [MPa]	l pt [m]	l pt1 [m]	l pt2 [m]
1,18	3,20	1,00	1,00	0,19	9	861,37	3,76	0,40	0,32	0,49

Transmission length - end

f ctd (t) [MPa]	η p1 [-]	η 1 [-]	α 1 [-]	α 2 [-]	ϕ [mm]	σ pm0 [MPa]	f bpt [MPa]	l pt [m]	l pt1 [m]	l pt2 [m]
1,18	3,20	1,00	1,00	0,19	9	861,37	3,76	6,72	0,32	0,49

Symbol	Explanation
f_{ctd} (t)	The design tensile value of strength at time of release; fctd(t) = αct 0,7 fctm(t) / γc (see also 3.1.2 (8) and 3.1.6 (2)P)
η_{p1}	A coefficient that takes into account the type of tendon and the bond situation at release
η_1	A coefficient related to the quality of the bond condition and the position of the bar during concreting (see 8.4.2)
α_1	1.0 - for gradual release, 1.25 - for sudden release
α_2	0.25 - for tendons with circular cross section, 0.19 - for 3 and 7-wire strands
ϕ	The nominal diameter of tendon
σ_{pm0}	The tendon stress just after release
f_{bpt}	Constant bond stress acc. to 8.10.2.2 (1)
l_{pt}	The basic value of the transmission length acc. to 8.10.2.2 (2)
l_{pt1}	0.8 l_{pt}
l_{pt2}	1.2 l_{pt}

Figure 3.96: Data window - Table with detailed information about current pre-tensioned group (part 1)

Short-term losses

d_x [m]	$\Delta\sigma_{pw}$ [MPa]	$\Delta\sigma_{pA}$ [MPa]	$\Delta\sigma_{pr}$ [MPa]	σ_{pr}^{cor} [MPa]	σ_{pT} [MPa]	σ_{pa} [MPa]	$\Delta\sigma_{pr}^{occur}$ [MPa]	$\Delta\sigma_{pr}^{cap}$ [MPa]
0,00	-7,80	-1,56	-0,02	890,62	-29,25	0,00	-0,06	-12,97
0,40	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
1,00	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
2,00	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
3,00	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
4,00	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
5,00	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
6,00	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
6,72	-7,80	-1,56	-0,02	890,62	-29,25	861,37	-0,06	-12,97
7,00	-7,80	-1,56	-0,02	890,62	-29,25	255,45	-0,06	-12,97
7,12	-7,80	-1,56	-0,02	890,62	-29,25	0,00	-0,06	-12,97

Symbols related to short-term losses

Symbol	Explanation
$\Delta\sigma_{pw}$	Anchorage set loss
$\Delta\sigma_{pA}$	Loss due the deformation of ends abutments of the stressing bed
$\Delta\sigma_{pr}$	Relaxation loss
σ_{pr}^{cor}	Stress after short-term relaxation
$\Delta\sigma_{pT}$	Loss due to the difference in temperature of prestressing steel and stressing bed
σ_{pa}	Stress after short-term losses except the loss due to immediate elastic strain of concrete
$\Delta\sigma_{pr}^{occur}$	Relaxation that already took place (occurred)
$\Delta\sigma_{pr}^{cap}$	Remaining relaxation capacity, i.e. potential stress decrease in prestressing reinforcement due to relaxation at infinite time

Figure 3.97: Data window - Table with detailed information about current pre-tensioned group (part 2)

3.5 Design Member Result

After recalculation of the structure, the internal forces of design members can be evaluated and considered in construction stages.

3.5.1 Internal Forces

To evaluate internal forces of design members click **5.1 Internal Forces** in the navigator.

The ribbon groups *Result class, Internal forces, Prestressing, Label orientation, Calculate FEM* and *Current section* are available.

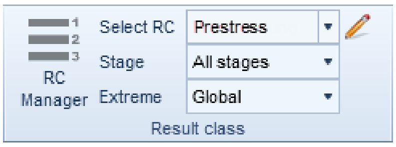


Figure 3.98: Result class

Use the *Result class* ribbon group to set the current result class and construction stage to evaluate results of the current design member.

RC manager – add, delete or edit result class

Select RC – select result class from the list, for which evaluation of the internal forces is performed. Click the edit button to edit the content of the current result class.

Stage – filter out from the current result class only combinations which are not defined in the selected stage.

- **All stages** – evaluate results from all combinations (load cases, load combinations) in the current result class without respecting construction stages
- **„Stage“** – evaluate results only for those combinations (load cases, load combinations) in the current result class, which are defined in the selected stage

Extreme – select the mode of evaluation of extremes:

No – no extreme is evaluated

Section – extreme values of the evaluated components are determined for each section

Member – extreme values of the evaluated components are deteremined for each particular member of the design member

Global – extreme values of the evaluated components are determined for the whole design member

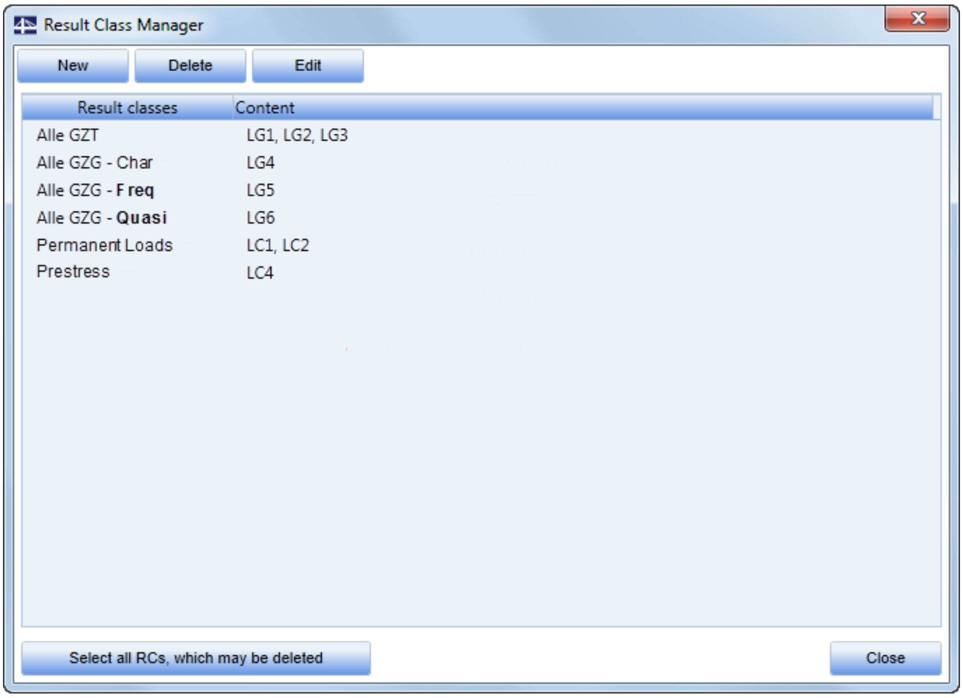


Figure 3.99: RC (Result class) manager

Detailed options of the *Result class manager* dialog box:

New – add a new result class to project

Delete – delete the selected result class from the project if this class is not assigned to a list of classes for the evaluation of results or check of the model

Edit – start edit of the selected result class

Result classes – names of the existing result classes are displayed in this column

Content – names of load cases or combinations in a particular result class are displayed in this column

Select all RCs, which may be deleted – select all result classes which can be deleted, meaning classes which are not assigned to be evaluated or checked

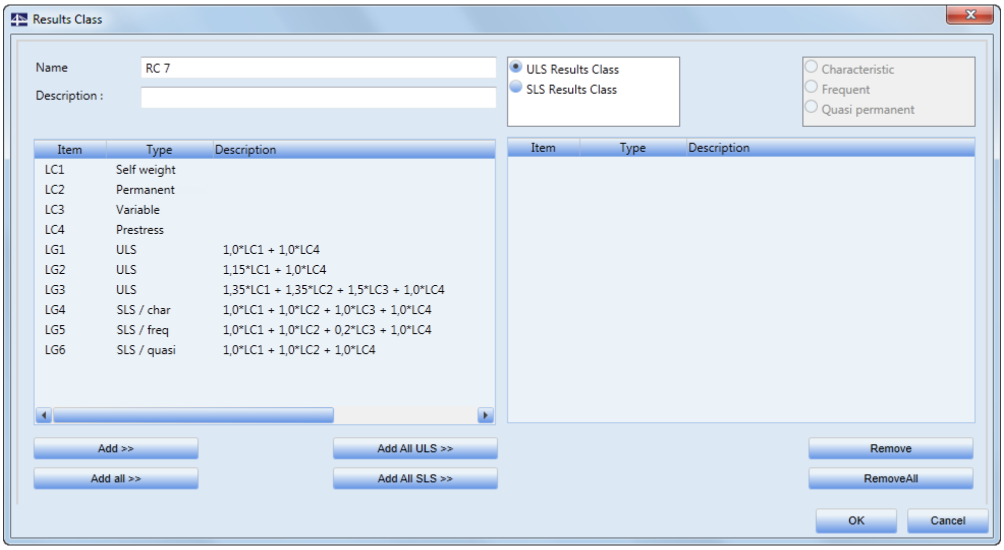


Figure 3.100: New result class

The **Result Class** dialog box contains the following options:

Name – name of the new class

Description – input of additional description of class

Add >> – add a load case or load combination from the list on the left into the class content. A case or combination can be added also by double-clicking on item in the list on the left.

Add All >> – add all load combinations from the list on the left into the class content

Add All ULS >> – add all ULS load combinations from the list into the class content

Add All SLS >> – add all SLS load combinations from the list into the class content

Result Class ULS – set the class type as ULS

Result class SLS – set the class type as SLS. For this type, the sub-type must be defined:

- **Characteristic**
- **Frequent**
- **Quasi-permanent**

Remove – delete selected items (load cases or combinations) from the list on the right. Load cases or combinations can be deleted also by double-clicking on an item in the list.

Remove All – delete all load cases or load combinations from the list on the right

The **Edit Result Class** dialog box is the same as the dialog box for creating a new one. The content of the class can be changed in the list on the right, but the class type (ULS, SLS) cannot be changed.

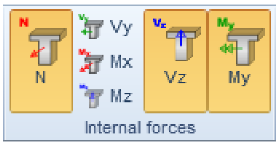


Figure 3.101: Internal forces

Components of internal forces to be drawn are set in the *Internal forces* ribbon group:

N – turn on/off drawing of axial force N

Vy – turn on/off drawing of shear force Vy

Vz – turn on/off drawing of shear force Vz

Mx – turn on/off drawing of torsion moment Mx

My – turn on/off drawing of bending moment My

Mz – turn on/off drawing of bending moment Mz

Figure 3.102: Prestressing

For result classes which contain **only load cases with prestressing**, the following can be set and displayed:

Effects of prestressing – evaluate total effects of prestressing

Primary forces – evaluate statically determinable internal forces (primary effects) of prestressing

Secondary forces – evaluate statically indeterminable internal forces (secondary effects) of prestressing. Negligible secondary internal forces can appear in statically determined structures due to numerical inaccuracy.

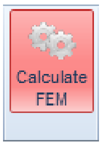


Figure 3.103: Calculate FEM

Click **Calculate FEM** to recalculate the analytical model in RFEM. The values of the Prestressed load case are updated in RFEM and the analysis is performed.

The remaining ribbon groups *Label orientation* and *Current section* are described in chapters 3.4.2 *Tendon Stress / Losses* and 3.2.1 *Tendon Layout*.

A recalculation is required after changes are made to the tendon layout because values of the equivalent loads change due to tendon changes.

The graphs of internal forces for the current result class and the current design members are drawn in the *Main* window (see Figure 3.104).

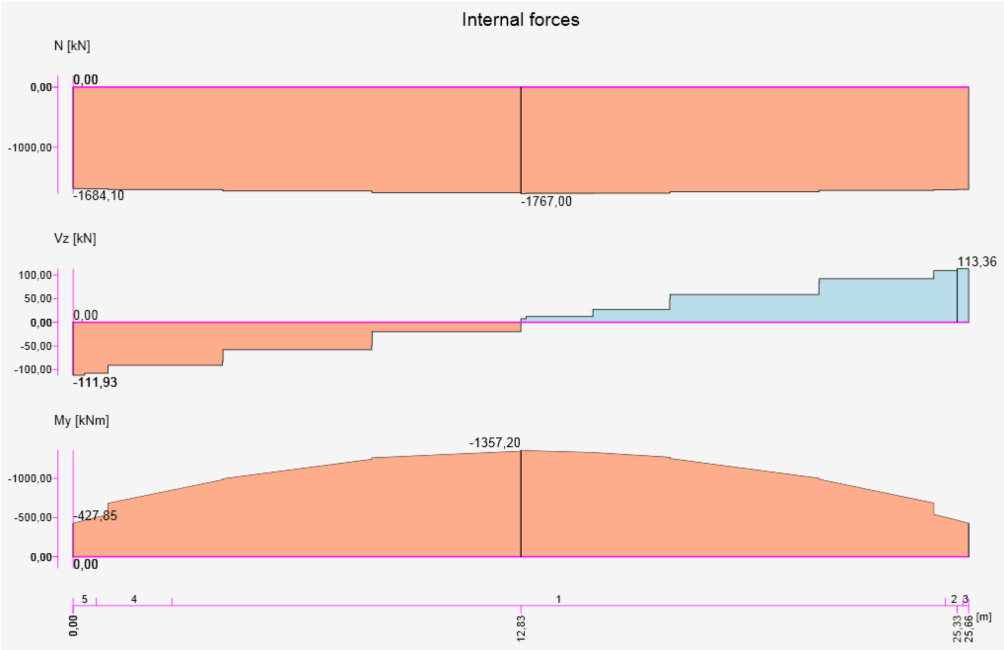


Figure 3.104: Main window – Internal forces

A textual presentation of the internal forces is printed in the *Data* window.

Internal Forces

Vorspannung, Axes: Local, Global Extreme

Member	Position [m]	Combination	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
1	11,83	LC4	-1767,00	0,00	7,51	0,00	-1357,20	0,00
5	0,00	LC4	0,00	0,00	0,00	0,00	0,00	0,00
2	0,67	LC4	-1698,60	0,00	113,36	0,00	-464,96	0,00
5	0,00	LC4	-1684,10	0,00	-111,93	0,00	-427,85	0,00

Figure 3.105: Data window – Internal forces

3.6 Design Member Check

3.6.1 Construction Stages

To enter detailed check properties of the design member click **6.1 Construction Stages** in the navigator. Default values are taken from the construction stages of the whole project.

The time axis of the current design member is drawn in the Main window.

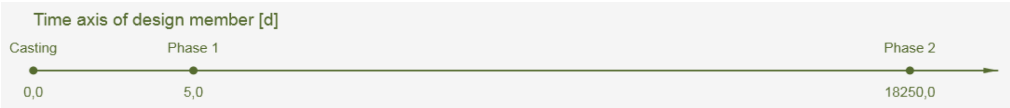


Figure 3.106: Main window – Time axis

The properties table of the current design member is displayed in the Data window.

The value of **Time shift Δt of birth of design member** can decrease or increase the time difference between Stage 0 and Stage 1. The time shift must be less than or equal to the time of the first stage minus 3 days, because required material characteristics are not specified for concrete younger than 3 days.

This option can be useful by changing the start of prestressing for only one design member. It is possible to change it for the whole model in 6.1 Construction Stages.

Example :

Stage 0 – 0 days, Stage 1 – 28 days, Stage 2 – 100 days, Stage 3 – 1000 days

By setting the time shift to Δt = 5 days, the time between Stage 0 and Stage 1 will decreased from 28 days to 23 days.

Stage 0 – 0 days, Stage 1 – 23 days, Stage 2 – 95 days, Stage 3 – 995 days

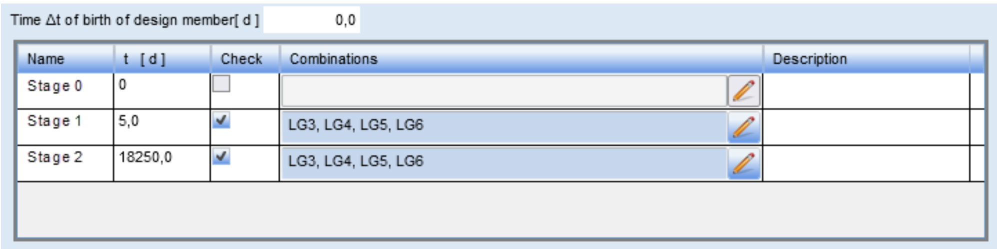
Columns of the design member properties table:

Name – name of stage

t (d) – local time of stage, which is calculated from the specified value of time shift Δt

Check – turn on/off whetger that particular stage is taken into account in the current design member check

Combinations – list of combinations assigned to particular stages in 1.1 Construction Stages for the entire structure (all design members) can be edited for the current (one) design member.


Time Δt of birth of design member[d] 0,0

Name	t [d]	Check	Combinations	Description
Stage 0	0	☐		
Stage 1	5,0	☑	LG3, LG4, LG5, LG6	
Stage 2	18250,0	☑	LG3, LG4, LG5, LG6	

Figure 3.107: Data window – Properties table

3.6.2 Check Positions (preparation for RF-TENDON Design)

The check of the design member is performed in specified positions. For each specified position a section, reinforced cross-section, construction stages and load extremes are generated. Such generated data are then checked in the add-on module **RF-TENDON Design**.

RF-TENDON Design is an add-on module for the calculation of prestressed concrete section checks according to EN 1992-1-1 and EN 1992-2, with or without a national application document.

The module **RF-TENDON Design** is opened from within the module **RF-TENDON** as a continuation for detailed checks of selected sections of the current design member.

In **RF-TENDON Design** all defined sections will be reinforced by longitudinal reinforcement and stirrups. Stirrups will be set for shear and torsion. Then all data for all members such as exposure class, relative humidity, creep coefficient, etc. will be manually entered. The module will calculate the losses of prestressing due to elastic deformation, relaxation of prestressing reinforcement, creep and shrinkage of concrete. The calculation will continue by ultimate limit state design (ULS) checks for capacity N-M-M, response N-M-M, shear, torsion and interaction and service limit state design (SLS) checks for stress limitation and crack widths. Finally, there are also controlled detailed provisions of longitudinal and shear reinforcement.

The final results from the calculation of all positions in **RF-TENDON Design** are displayed also in **RF-TENDON** after closing RF-TENDON Design in the Data window in the *Positions* table in the *Value* and *Result Status* columns, and overall check status of all calculated checks for the whole design member is displayed in the Info window as described in this chapter.

To define check positions (= sections on the design member) click **6.2 Check positions** in the navigator.

The ribbon groups *Check, Positions, Design member views, Uncoiled view, Calculation FEM* and *Current design member check* are available when defining positions for the check.

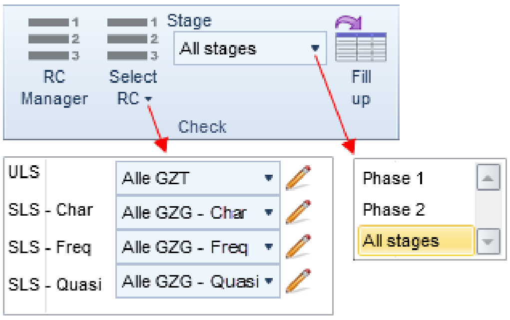


Figure 3.108: Check

Result classes and stages for the check of the current design member can be set in the *Check* ribbon group.

RC manager – add, delete and edit result class, see 3.5.1 Internal Forces

Select RC – display lists where particular result classes can be assigned as combination types for the check:

ULS – a result class selected in this list is used to generate the content of ULS – fundamental combination for the check of the reinforced concrete section. To edit the content of the current result class click the edit button.

SLS – Char – a result class selected in this list is used to generate the content of SLS – Characteristic combination for the check of the reinforced concrete section. To edit the content of the current result class click the edit button.

SLS – Freq – a result class selected in this list is used to generate the content of SLS – Frequent combination for the check of the reinforced concrete section. To edit the content of the current result class click the edit button.

SLS – Quasi – a result class selected in this list is used to generate the con tent of ULS – Quasi-permanent combination for the check of the reinforced concrete section. To edit the content of the current result class click the edit button.

Stage – filter out from the current result class only combinations which are not defined in the selected stage:

All stages – evaluate results from all combinations (load cases, load combinations) in the current result class for all construction stages

„Stage“ – evaluate results only for those combinations (load cases, load groups) in the current result class which are defined in the selected stage

Fill up – fill the content of basic result classes (All ULS, All SLS-char, All SLS-freq, All SLS-QP) automatically. Combinations of the appropriate type are assigned to the corresponding result classes.

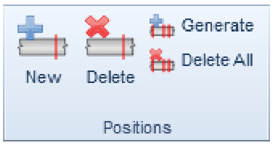


Figure 3.109: Positions

Use the commands in the *Positions* ribbon group to create or remove positions.

New – add a new single position

Delete – delete the selected positions

Generate – run a mass generation of positions

Delete all – delete all existing positions on the current design member

To input a new single position click **New** in the *Positions* ribbon group.

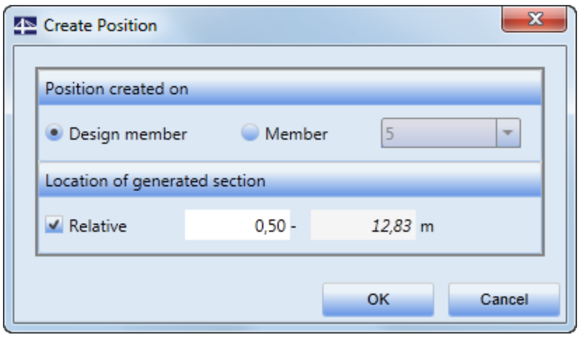


Figure 3.110: New position

The location of the new position is specified in the *Create position* dialog box.

Design member – If this option is selected, the value of the distance is relative to the beginning of the current design member.

Member – If this option is selected, the value of the distance is relative to the beginning of the member selected in the list.

Relative – If selected, the relative value to the length of the current design member or selected member can be entered. Otherwise the absolute value of the distance from the beginning of the current design member or selected member is entered.

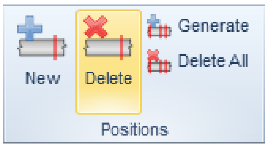


Figure 3.111: Delete selected position

To delete the selected (marked red) position click **Delete** in the *Positions* ribbon group. If this position contains a reinforced cross-section from the module **RF-TENDON Design**, then this reinforcement will also be deleted.

To define more positions at once click **Generate** in the *Positions* ribbon group.

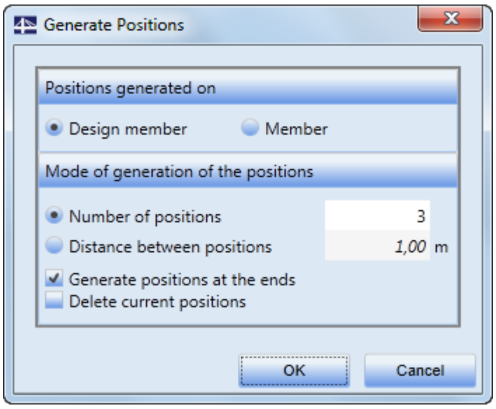


Figure 3.112: Generate positions

Design member – If this option is selected, the settings in the *Location of generated positions* group are applied to the current design member as a whole.

Member – If this option is selected, the settings in the *Location of generated positions* group are applied to each individual member in the current design member.

Number of positions – If selected, the defined number of sections is generated.

Distance between positions – If selected, the defined distance between generated sections is used.

Generate positions at the ends – If selected, sections are generated at the ends.

Delete current positions – If selected, all existing positions are deleted during the generation of new positions.

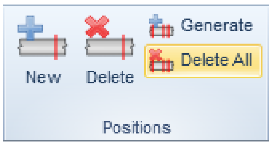


Figure 3.113: Delete all positions

To delete all positions click **Delete All** in the *Positions* ribbon group. If these positions contain reinforced cross-sections from the module **RF-TENDON Design**, then this reinforcement will also be deleted.

Find more information on the ribbon groups *Design member views* in chapter 3.2.1 *Tendon Layout*, *Uncoiled view* in chapter 3.1.2.2 *Design member view* and *Calculate FEM* in chapter 3.5.1 *Internal Forces*.

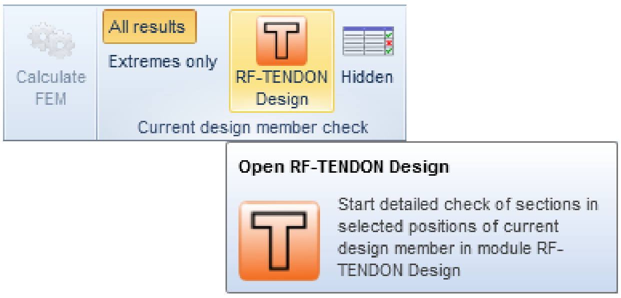


Figure 3.114: Current design member check

A detailed check of the current design member can be started after FEM analysis (the *Calculate FEM* button is grey, not red).

Open RF-TENDON Design – Run the module to perform a detailed check of positions (sections) of the current design member.

Hidden – Run the check of the current design member in the background.

All results – If this option is selected, the extreme is generated for each combination in the appropriate result class.

Extreme only – If this option is selected, extreme values of internal forces are determined from all combinations in the result class. A maximum of 12 extremes are generated for each section.

Defined positions are displayed in the *Main* window, the selected (current) position is marked red.

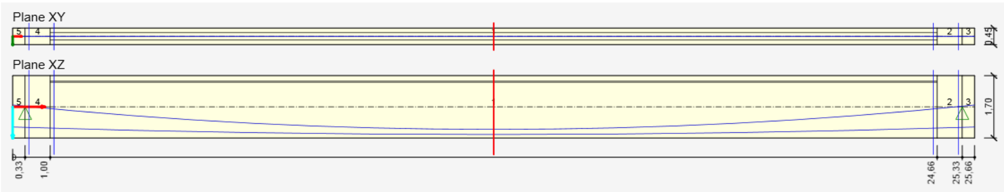


Figure 3.115: Main window - Defined positions (blue) and selected position (red)

The defined positions can be edited it the table under the *Positions* tab in the *Data* window.

Positions | Internal forces

	Description	Position on	Relative	Position	Position on design member	Value	Result Status
1	Design Member 1 - 0,43m (4 - 0,10m)	Design member / Member	Yes / No	0,43 [m]	0,43 [m]	0,00	
2	Design Member 1 - 1,10m (1 - 0,10m)	Design member / Member	Yes / No	1,10 [m]	1,10 [m]	0,00	
3	Design Member 1 - 12,83m (1 - 11,83m)	Design member / Member	Yes / No	0,50 [-]	12,83 [m]	0,00	
4	Design Member 1 - 24,56m (1 - 23,56m)	Design member / Member	Yes / No	24,56 [m]	24,56 [m]	0,00	
5	Design Member 1 - 25,23m (2 - 0,57m)	Design member / Member	Yes / No	25,23 [m]	25,23 [m]	0,00	

Figure 3.116: Data window – Positions – before starting the module RF TENDON Design

Positions | Internal forces

	Description	Position on	Relative	Position	Position on design member	Value	Result Status
1	Design Member 1 - 0,43m (4 - 0,10m)	Design member / Member	Yes / No	0,43 [m]	0,43 [m]	91,55	
2	Design Member 1 - 1,10m (1 - 0,10m)	Design member / Member	Yes / No	1,10 [m]	1,10 [m]	91,49	
3	Design Member 1 - 12,83m (1 - 11,83m)	Design member / Member	Yes / No	0,50 [-]	12,83 [m]	98,78	
4	Design Member 1 - 24,56m (1 - 23,56m)	Design member / Member	Yes / No	24,56 [m]	24,56 [m]	91,56	
5	Design Member 1 - 25,23m (2 - 0,57m)	Design member / Member	Yes / No	25,23 [m]	25,23 [m]	91,62	

Figure 3.117: Data window – Positions – after calculation in the module RF TENDON Design

The *Positions* table (Figure 3.117) contains the following columns:

Description – Display the generated name of the position. The description contains the distance from the design member origin, the name of the member and the distance of the position from the beginning of the member.

Position on – Set the origin to which the location of the position is relative:

Design member – If the option is selected, the value in the *Position* column is relative to the origin of the design member.

Member – If the option is selected, the value in the *Position* column is relative to the origin of the corresponding member.

Relative – Set the mode of evaluation of the distance in the *Position* column.

Position – The value of the distance relative to the origin specified in the *Position* column.

Position on design member – Display the absolute distance of the position from the beginning of the design member.

Value – Display the maximum check value of all the calculated checks in the position from the module RF-TENDON Design.

Result status – Displays and indicator of the check status of all the calculated checks in the position from the module RF-TENDON Design.

Combinations of internal forces for the check of the design member positions can be reviewed on the *Internal forces* tab in the *Data* window after clicking [Calculate FEM]. These forces are prepared and used for calculation in the add-on module **RF-TENDON Design**.

Positions | Internal forces

Design Member 1 - 0,43m (4 - 0,10m)

Section: Design Member 1 - 0,43m (4 - 0,10m)

Extreme: Phase 1 (5,0d): LG3 - LG4 - LG5 - LG6

Total internal forces with influence of prestressing

Combination type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Fundamental ULS	-1682,01	0,00	592,27	0,00	-407,27	0,00
Characteristic	-1682,01	0,00	402,59	0,00	-425,43	0,00
Frequent	-1682,01	0,00	343,07	0,00	-431,09	0,00
Quasi-permanent	-1682,01	0,00	328,19	0,00	-432,50	0,00

Internal forces without prestressing

Combination type	Load type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Fundamental ULS	Permanent Sum Gdj	0,00	0,00	588,14	0,00	56,41	0,00
Fundamental ULS	Variable Qd1	0,00	0,00	111,60	0,00	10,61	0,00
Fundamental ULS	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Characteristic	Permanent Sum Gdj	0,00	0,00	435,66	0,00	41,78	0,00
Characteristic	Variable Qd1	0,00	0,00	74,40	0,00	7,07	0,00
Characteristic	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Frequent	Permanent Sum Gdj	0,00	0,00	435,66	0,00	41,78	0,00
Frequent	Variable Qd1	0,00	0,00	14,88	0,00	1,41	0,00
Frequent	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Quasi-permanent	Permanent Sum Gdj	0,00	0,00	435,66	0,00	41,78	0,00
Quasi-permanent	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Quasi-permanent	Variable Qd1	0,00	0,00	0,00	0,00	0,00	0,00

Internal forces caused by prestressing

Load type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Primary effects of prestressing	-1680,57	0,00	-109,43	0,00	-473,97	0,00
Secondary effects of prestressing	-1,45	0,00	1,96	0,00	-0,31	0,00

Extreme: Phase 2 (18250,0d): LG3 - LG4 - LG5 - LG6

Total internal forces with influence of prestressing

Combination type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Fundamental ULS	-1445,51	0,00	606,96	0,00	-337,06	0,00
Characteristic	-1445,51	0,00	417,28	0,00	-355,22	0,00
Frequent	-1445,51	0,00	357,76	0,00	-360,88	0,00
Quasi-permanent	-1445,51	0,00	342,88	0,00	-362,30	0,00

Internal forces without prestressing

Combination type	Load type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Fundamental ULS	Permanent Sum Gdj	0,00	0,00	588,14	0,00	56,41	0,00
Fundamental ULS	Variable Qd1	0,00	0,00	111,60	0,00	10,61	0,00
Fundamental ULS	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Characteristic	Permanent Sum Gdj	0,00	0,00	435,66	0,00	41,78	0,00
Characteristic	Variable Qd1	0,00	0,00	74,40	0,00	7,07	0,00
Characteristic	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Frequent	Permanent Sum Gdj	0,00	0,00	435,66	0,00	41,78	0,00
Frequent	Variable Qd1	0,00	0,00	14,88	0,00	1,41	0,00
Frequent	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Quasi-permanent	Permanent Sum Gdj	0,00	0,00	435,66	0,00	41,78	0,00
Quasi-permanent	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00
Quasi-permanent	Variable Qd1	0,00	0,00	0,00	0,00	0,00	0,00

Internal forces caused by prestressing

Load type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Primary effects of prestressing	-1444,27	0,00	-94,47	0,00	-403,81	0,00
Secondary effects of prestressing	-1,24	0,00	1,69	0,00	-0,27	0,00

Increments of effects of characteristic permanent load

Age	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
5,0	0,00	0,00	435,66	0,00	41,78	0,00
18250,0	0,00	0,00	0,00	0,00	0,00	0,00

Figure 3.118: *Data* window – Internal forces

The list of internal forces (Figure 3.118) contains:

List of defined positions. The current position, for which the list of internal forces is displayed, can be selected from this list.

Section – generated name of section

Extreme – generated name of extreme, which consists of:

(Extremes are generated separately for every phase by using extremes max N, min N, max My, min My … from all combinations, max 12 extremes, if the same extremes are found then only one of them is used)

Phase 1 (5,0d) – The name of the construction stage and its time, as defined in the local time axis of the design member.

LG3 – The name of load case combination from the **ULS** result class. The name of the ULS combination is always in the first position of the extreme name. If the ULS combination is not specified, the text "not filled" is printed instead of the combination name.

LG4 – The name of the load case combination from the **SLS-Char** result class. The name of the SLS-Char combination is always in the second position of the extreme name.

LG5 – The name of the load case combination from the **SLS-Freq** result class. The name of the SLS-Freq combination is always in the third position of the extreme name.

LG6 – The name of load case combination from the **SLS-Quasi** result class. The name of the SLS-Quasi combination is always in the third position of the extreme name.

The **Total internal forces with influence for prestressing** table contains total internal forces for **ULS** and **SLS** combinations (names of combinations are included in the extreme name) for the current extreme including the effects of prestressing.

The **Total internal forces without prestressing** table contains internal forces for the same combinations without the effects of prestressing. Internal forces are split to internal forces for permanent load components Sum Gdj and variable load components Qd1 and Sum Qdi.

The **Internal forces caused by prestressing** table contains primary and secondary effects of prestressing.

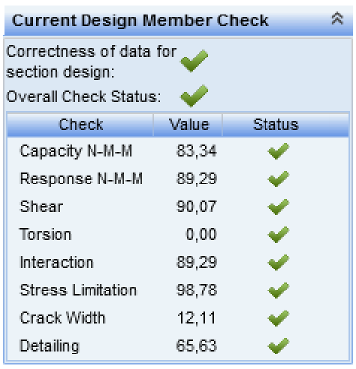


Figure 3.119: *Info* window – Results of all check after calculation in the module RF TENDON Design

Overall check status of all calculated checks from the module **RF-TENDON Design** for the whole design member is displayed in **RF-TENDON** in the Info window.

3.7 Report (current design member)

A report can be generated for design members which have the Print option enabled in the design member properties, see also chapter 3.1.2.5 *Report (all design members)*.

A Report for all design members can be generated in the **Design members** navigator command.

A Report for current design member can be generated in the **Report** navigator command.

3.7.1 Setting

Open **Setting** in the navigator to display detailed report settings for the report which is generated by clicking 7.3 Detailed.

For a full description of the detailed report settings, see chapter 3.1.2.5 *Report (all design members)*.

3.7.2 Standard

Click **Standard** in the navigator to generate a standard report for the current design member.

The standard printout report contains basic project data information, design member information, prestressing information and check results. The content of a standard printout report cannot be adjusted in the report settings.

Table of contents

Chapter number	Chapter name
1.	Project data
2.	Brief summary of results of design member
3.	Construction Stages
4.	Design Members
4.1.	Design Member 1

1. Project data

Name	DachbinderEnd
Author	
Created on	
Description	

National code

National code	EN 1992-1-1
National annex	EN

2. Brief summary of results of design member

Design Name	Member Description	Members	Tendons	Valid	Value [%]	Result Status
Design Member 1	Description 1	5, 4, 1, 2, 3	Tendon 1 Tendon 2	✓	98,78	✓

3. Construction Stages

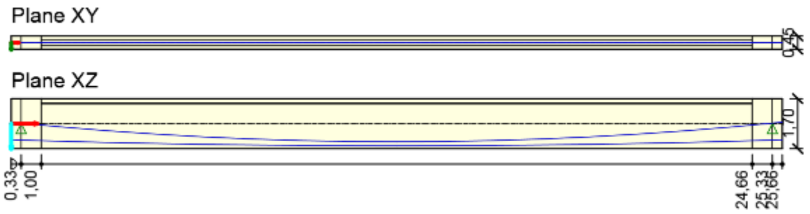
Name	Time [d]	Load Cases	Combinations	Description
Phase 0	0,0			
Phase 1	5,0	LC1, LC2, LC4	LG3, LG4, LG5, LG6	
Phase 2	18250,0		LG3, LG4, LG5, LG6	

Figure 3.120: Standard – Part 1

4. Design Members

4.1. Design member 1

Description	Members	Tendons	Valid	Value [%]	Result Status
Description 1	5, 4, 1, 2, 3	Tendon 1, Tendon 2	✓	98,78	✓



4.1.1. Prestressing

Name	Material	A p [mm2]	Length [m]	L s [m]	L arc [m]	R min [m]	θ [°]
	Strands	σ a [MPa]	σ min [MPa]	σ max [MPa]	e ba [mm]	e aa [mm]	L set [m]
Tendon 1	Y1770S7-12.9	700	25,71	0,00	25,71	123,83	11,8
	7	1321,00	1192,90	1265,65	166	163	10,79
Tendon 2	Y1770S7-12.9	700	25,66	0,00	25,66	401,47	3,7
	7	1310,00	1219,79	1264,09	166	163	12,98

Name	σ ini,max [MPa]	σ lim [MPa]	Check 5.10.2.1(1)P	σ min [MPa]	σ max [MPa]	σ pm0 [MPa]	Check 5.10.3(2)P
Tendon 1	1321,00	1350,00	✓	1192,90	1265,65	1275,00	✓
Tendon 2	1310,00	1350,00	✓	1219,79	1264,09	1275,00	✓

4.1.2. Positions

Description	Position on design member [m]	Value [%]	Result Status
Design Member 1 - 0,43m (4 - 0,10m)	0,43	91,55	✓
Design Member 1 - 1,10m (1 - 0,10m)	1,10	91,49	✓
Design Member 1 - 12,83m (1 - 11,83m)	12,83	98,78	✓
Design Member 1 - 24,56m (1 - 23,56m)	24,56	91,56	✓
Design Member 1 - 25,23m (2 - 0,57m)	25,23	91,62	✓

Figure 3.121: Standard – Part 2

3.7.3 Detailed

Click **Detailed** in the navigator to generate a detailed report for the current design member.

The detailed printout report contains detailed project data information, detailed design member information, detailed equivalent load information, detailed prestressing information and check results. The content of detailed printout report can be adjusted in the report settings.

Table of contents

Figure 3.122: Detailed (only table of content displayed)

A Text Format *.nav

Format of text file for import and export of tendons

Tendon geometry is defined in section <**BondedTendons**> </**BondedTendons**>. This section contains information about all imported/exported tendons. One tendon data is defined in section <**BondedTendon**> </**BondedTendon**>
Section <**BondedTendon**> must contain 3 basic tags:

<**BondedTendonData**> - contains tendon data

<**BondedTendonSpansXY**> - contains tendon geometry in XY plane

<**BondedTendonSpansXZ**> - contains tendon geometry in XZ plane

<**BondedTendonData**> </**BondedTendonData**> contains two lines. The name of the tendon is in the first line. The second line contains gradually: number of strands, primary geometry (XY or XZ) to determine the tendon position in the cross-section, tendon duct diameter, tendon duct material (1=metal, 2=plastic). The next parameters describe the type of tendon stressing (1=stressing from the beginning, 2=stressing from the end, 3=stressing from both sides with anchoring at the beginning, 4=stressing from both sides with anchoring at the end) and stressing proce dure (3=with correction of relaxation, 4= without correction).

<**BondedTendonSpansXY**></**BondedTendonSpansXY**> contains the next 2 tags -<**SpansData**> and <**SpansPoints**>. It describes tendon geometry in plane XY.

<**SpansData**></**SpansData**> describes tendon geometrical segments in the XY-plane. The number of rows corresponds with the number of tendon geometrical segments in the XY-plane. Each row consists of identification of the segment type, the beginning point and the ending point referred to the member reference axis.

<**SpansPoints**></**SpansPoints**> describes the points determining the geometry of tendon segments. Each row specifies one point. For each straight segment two points have to be specified, for other segments 3 points have to be specified. One point definition contains:
The number of the segment, on which point lies,

Type of point (1= point at the beginning or the ending of the whole tendon – point C, 2=point between straight part and parabola– point S-P, 3= point between two parabolas –point P-P),

Type of reference point for input of vertical segment point position (1= maximum Y-coordinate, 2=origin of reference axis, 3= minimum Y-coordinate, 4= maximum coordinate of intersection of horizontal line through tendon centre with cross-section edges, 5=minimum coordinate of intersection of horizontal line through tendon centre with cross-section edges, 6=centre of gravity of cross-section – must not always be identical with reference axis)

vertical distance from reference point

for **point-C**: length of end straight part

for point S-P:

type of reference point for definition of the segment point horizontal position (1=input related to left segment edge, 2=input related to middle of segment, 3=input related to right segment edge)

horizontal distance from reference point

length of straight tendon part

mode of input values (1=relative, distances refer to tendon segment length, 0=absolute distances)

for **point P-P**: minimum radius of parabola

<**BondedTendonSpansXZ**></**BondedTendonSpansXZ**> contain identical tags and data as tags <**BondedTendonSpansXY**></**BondedTendonSpansXY**>, but with description of the geometry in XZ-plane.

Example of text file for tendon import.

<BondedTendons> … beginning of the section for all tendons definition
<BondedTendon> … beginning of the section for one tendon definition
<BondedTendonData> … beginning of the tendon data section
Tendon 6 … the tendon name
1 XY 14 1 1 4 … gradually: **1** strand in tendon, primary geometry **XY**, tendon duct diameter **14** mm, tendon duct material **1**(metal), stressing from the beginning **1**, stressing procedure without relaxation correction **4**
</BondedTendonData> … end of the tendon data section
<BondedTendonSpansXY> …beginning of the section for input of geometry in uncoiled view XY
<SpansData> … beginning of the section for tendon segments input
1 0.00000 30.00000 … gradually segment type **1** (straight standalone), x-coordinate of segment beginning, x-coordinate of segment ending
</SpansData> … end of the section for tendon segments input
<SpansPoints> … beginning of the section for tendon segment characteristic points input
1 1 2 0.00 0.00000… gradually: point lies on first segment – **1**, point type **C** - **1**, vertical position is related to the origin of the member reference axis– **2**, vertical distance from the reference point is **0** mm, length of straight part is **0** m

1 1 2 0.00 0.00000... parameters are identical as for the previous point.

</SpansPoints> … end of the section for tendon segment characteristic points input
</BondedTendonSpansXY> … end of the section for input of geometry in uncoiled view XY
<BondedTendonSpansXZ> … beginning of the section for input of geometry in uncoiled view XZ
<SpansData> … beginning of the section for tendon segments input
3 0.00000 10.00000 … gradually: segment type **3** (parabolic with straight, left), x-coordinate of the segment the beginning, x-coordinate of segment ending
5 10.00000 20.00000 … gradually: segment type **5** (parabolic with straight, inner), x-coordinate of the segment beginning, x-coordinate of the segment ending
4 20.00000 30.00000 … gradually: segment type **4** (parabolic with straight, right), x-coordinate of the segment beginning, x-coordinate of the segment ending
</SpansData> … end of the section for tendon segments input
<SpansPoints> … beginning of the section for tendon segment characteristic points input
1 1 2 0.00 0.00000… gradually point lies on first segment – **1**, point type **C** - **1**, vertical position is related to origin of member reference axis – **2**, vertical distance from reference point is **0** mm, length of straight part is **0** m
1 2 1 -70.00 1 0.30000 0.30000 1 … gradually: point lies on the first segment – **1**, point type **S-P** - **2**, vertical position is related to the maximum cross-section coordinate in the Z-axis – **1**, vertical distance from the reference point is -**70** mm, horizontal position is related to the left segment edge – **1**, horizontal distance from the reference point is **0.3**, length of the straight tendon part is **0.3**, values of the horizontal distance and length of the straight part are relative to the tendon segment length - **1**
1 3 3 70.00 2.00000 … gradually: point lies on the first segment – **1**, point type **P-P** - **3**, vertical position is related to the minimum cross-section coordinate in the Z-axis-**3**, vertical distance from the reference point is **70** mm, parabolas diameter is **2** m
2 3 3 70.00 2.00000 … gradually: point lies on the second segment – **2**, point type **P-P** - **3**, vertical position is related to the minimum cross-section coordinate in Z-axis– **3**, vertical distance from the reference point is **70** mm, parabola diameter is **2** m
2 2 1 -70.00 1 0.40000 0.20000 1 … gradually: point lies on the second segment – **2**, point type **S-P** - **2**, vertical position is related to the maximum cross-section coordinate in the Z-axis – **1**, vertical distance from the reference point is -**70** mm, horizontal position is related to the left segment edge – **1**, horizontal distance from the reference point is **0.4**, length of the straight tendon part is **0.2**, values of the horizontal distance and length of the straight part are relative to the tendon segment length - **1**
2 3 3 70.00 2.00000 … gradually: point lies on the second segment – **2**, point type **P-P** - **3**, vertical position is related to the minimum cross-section coordinate in the Z-axis-**3**, vertical distance from the reference point is **70** mm, parabolas diameter is **2** m
3 3 3 70.00 2.00000 …gradually: point lies on the third segment – **3**, point type **P-P** - **3**, vertical position is related to the minimum cross-section coordinate in the Z-axis-**3**, vertical distance from the reference point is **70** mm, parabolas diameter is **2** m

3 2 1 -70.00 1 0.40000 0.30000 1 … gradually: point lies on the third segment – **3**, point type **S-P** - **2**, vertical position is related to the maximum cross-section coordinate in the Z-axis – **1**, vertical distance from the reference point is -**70** mm, horizontal position is related to the left segment edge – **1**, horizontal distance from the reference point is **0.4**, length of the straight tendon part is **0.3**, values of the horizontal distance and length of the straight part are relative to the tendon segment length - **1**

3 1 2 0.00 0.00000 ...gradually: point lies on the third segment – **3**, point type **C** - **1**, vertical position is related to the origin of the member reference axis – **2**, vertical distance from the reference point is **0** mm, length of the straight part is **0** m

</SpansPoints> … end of the section for tendon segment characteristic points input
</BondedTendonSpansXZ> … end of the section for input of geometry in uncoiled view XZ
</BondedTendon> … end of the section for one tendon definition
</BondedTendons> … end of the section for all tendons definition

B Literature

[1] Deutscher Beton- und Bautechnik-Verein e.V.: Beispiele zur Bemessung nach Eurocode 2. (Beispiel 8: Vorgespannter Dachbinder)
Verlag für Architektur und technische Wissenschaft, Berlin, Deutschland, 2011

[2] EN 1992 Design of concrete structures
Part 1-1 General rules and rules for buildings

[3] EN 1992 Design of concrete structures
Part 2 Concrete bridges

[4] Jaroslav Navrátil : Prestressed concrete structures

C Index