Engineering Software
Dlubal

Version
February 2013

Program

RF-TENDON Design

Prestressed Concrete Design

Program Description

© Ing.-Software Dlubal
Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Overview

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.

Before starting the module RF-TENDON Design, it is necessary to enter data in the module RF-TENDON for the prestressed tendons, calculated equivalent loads, performed load balancing, calculated short-term losses and as well as all known internal forces for the members to design. After completing the definition of the sections for the design members, you are ready to start the module RF-TENDON Design.

In RF-TENDON Design all 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 results of all checks can be presented in 2D and 3D graphical figures as well as in text form. Short, standard, or detailed reports can be generated and sent directly to a printer or saved as a file.

1.2 RF-TENDON Design Team

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

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

Section

The program works with individual sections. A section is defined with relation to specific member data and reinforcement data (configurations).

One or more sets of load effects (extremes) are assigned to each section.

One project can contain multiple sections with multiple members, reinforcement configurations and load effects.

Extreme

An extreme is a set of combinations of internal forces, specifically one combination for Ultimate Limit States and three for Serviceability Limit States (Characteristic, Frequent, Quasi-permanent).

More extremes can be assigned to one section. When the check of a single section is performed, the reinforced section is checked only for the current extreme. When the summary check of all sections is performed, each section is checked for all extremes assigned to the section.

Design Member

The design member data defines the information about the whole element (type, exposure class, creep coefficient, etc.) for which a specified section is being checked. User-defined design member data can be assigned to multiple sections. Any change in the design member data is reflected in all related sections which are assigned to the design member.

Reinforced section

The reinforced section defines information about the reinforcement configuration: section geometry, longitudinal reinforcement, shear reinforcement, applied cover(s) and reinforcement materials. The user-defined reinforced section configuration can be assigned to multiple sections. Any change in the reinforced section data is reflected in all sections to which the corresponding data is assigned.

2. Running the Program

2.1 Starting RF-TENDON Design

The module RF-TENDON Design can be started from the module RF-TENDON after selecting **Check Positions** from the navigator and then clicking on the **RF-TENDON Design** icon in the ribbon group.

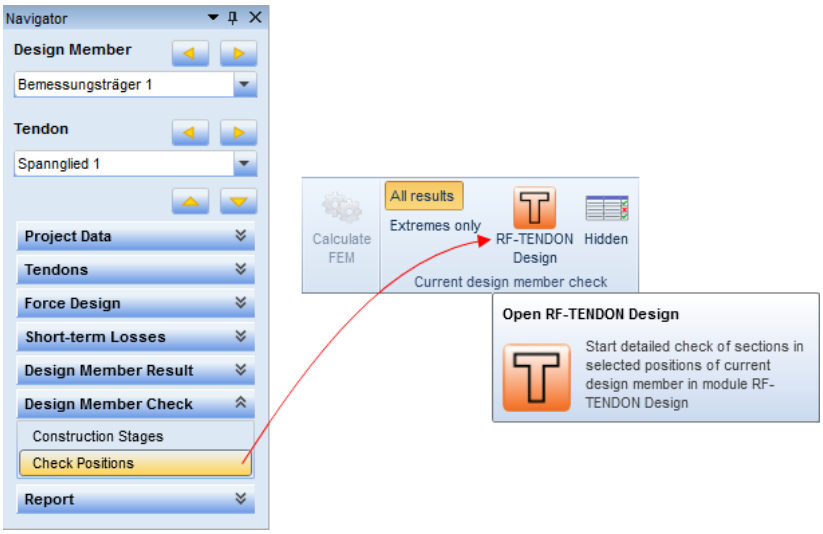


Figure 2.1: Starting RF-Tendon Design

2.2 User Interface

The user interface consists of the following parts:

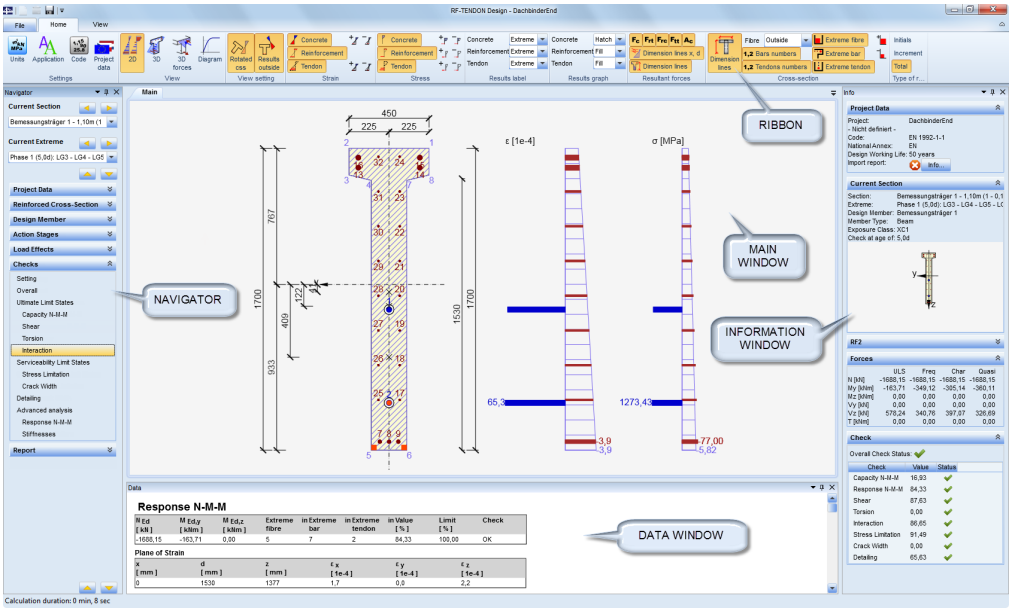


Figure 2.2: Parts of user interface

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 File

In this menu, basic functions are available.

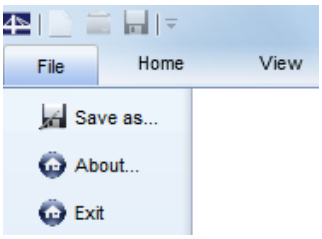


Figure 2.3: File menu

Save as

Save file with another name.

About

Show information about the program version.

Exit

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

2.4 Home

In this menu, settings for the module are available.

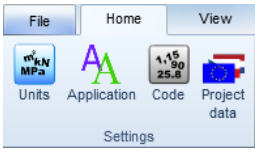


Figure 2.4: Home menu

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.

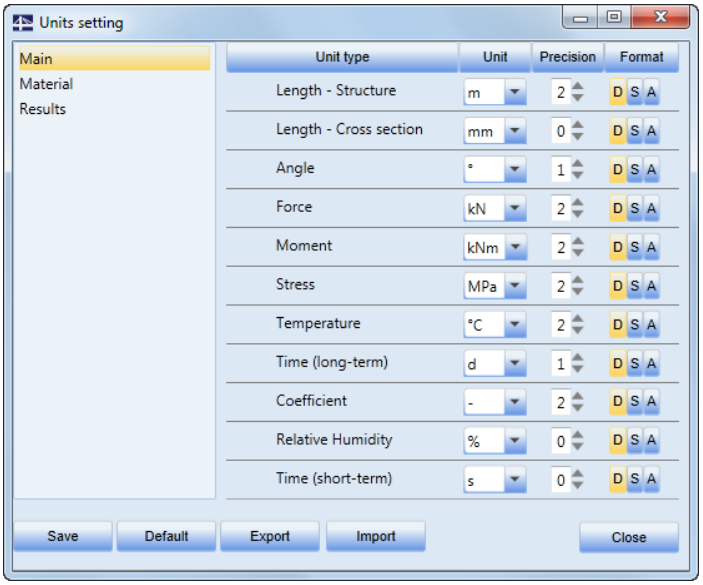


Figure 2.5: Main units

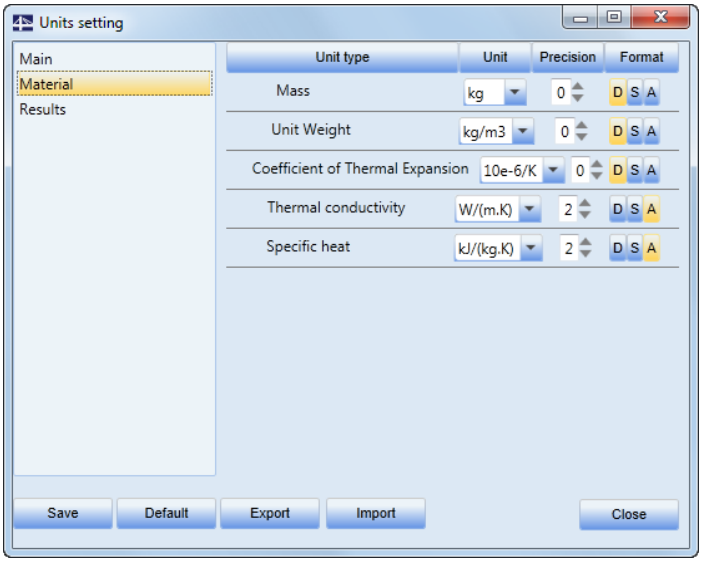


Figure 2.6: Material units

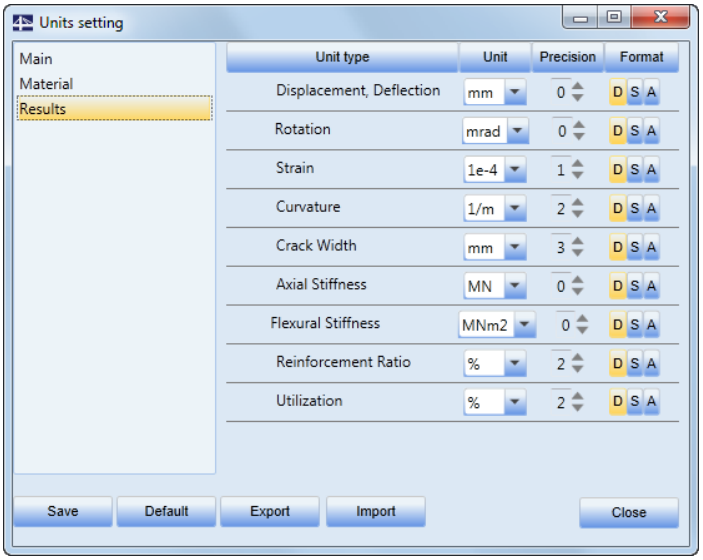


Figure 2.7: Units for Results

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.

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.

Export

Saves the current units settings to a file.

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.

2.4.2 Application

Application settings are available on five tabs by clicking the [Application] button in the *Settings* ribbon group.

Select the **Cross-Section Drawing** tab to open a dialog box to set the outline thickness and cross-section color.

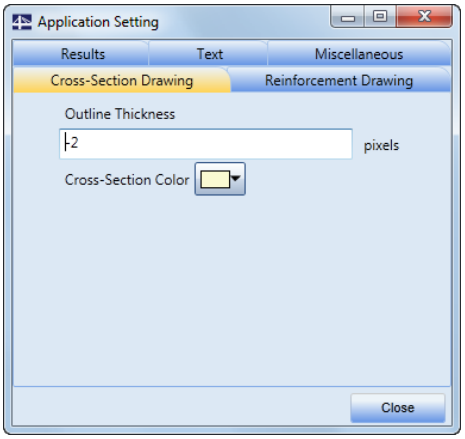


Figure 2.8: Cross-Section Drawing

Select the **Reinforcement Drawing** tab to set the colors of the reinforcements.

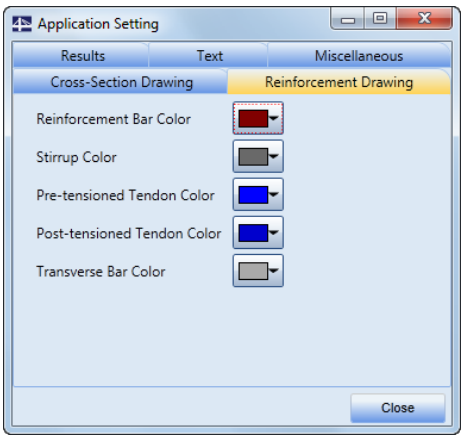


Figure 2.9: Reinforcement Drawing

Select the **Results** tab to set color of drawn results.

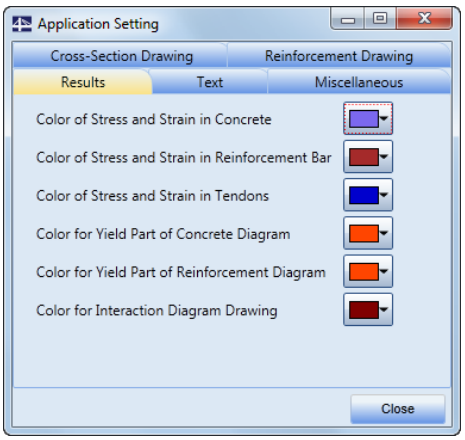


Figure 2.10: Results

Select the **Text** tab to open a dialog box to set the text height used for the reinforced cross-section pictures and the result drawings.

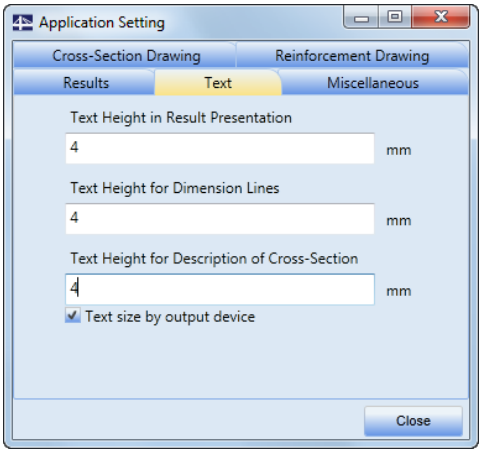


Figure 2.11: Text

Select the **Miscellaneous** tab to define some general parameters.

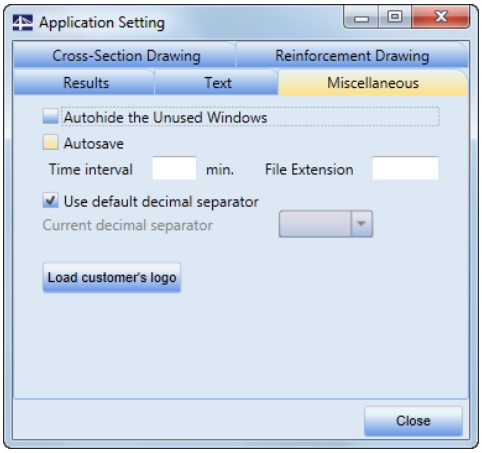


Figure 2.12: Miscellaneous

Autohide the Unused Windows

Turns on/off the setting to automatically hide the unused windows with empty content (*Info window, Data window*). The change is implemented after restarting the program.

Autosave

Turns on/off the setting to automatically save the data in a defined time interval. Automatic saving is only possible when a file extension is set in the textbox.

Use default decimal separator

If the check box is cleared, a decimal separator can be set in the list *Current decimal separator*. Otherwise, the decimal separator specified in the Regional settings is used.

Load customer's logo

Click this command button to select an image file (jpg, gif) to be used on the top right corner of the report.

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.

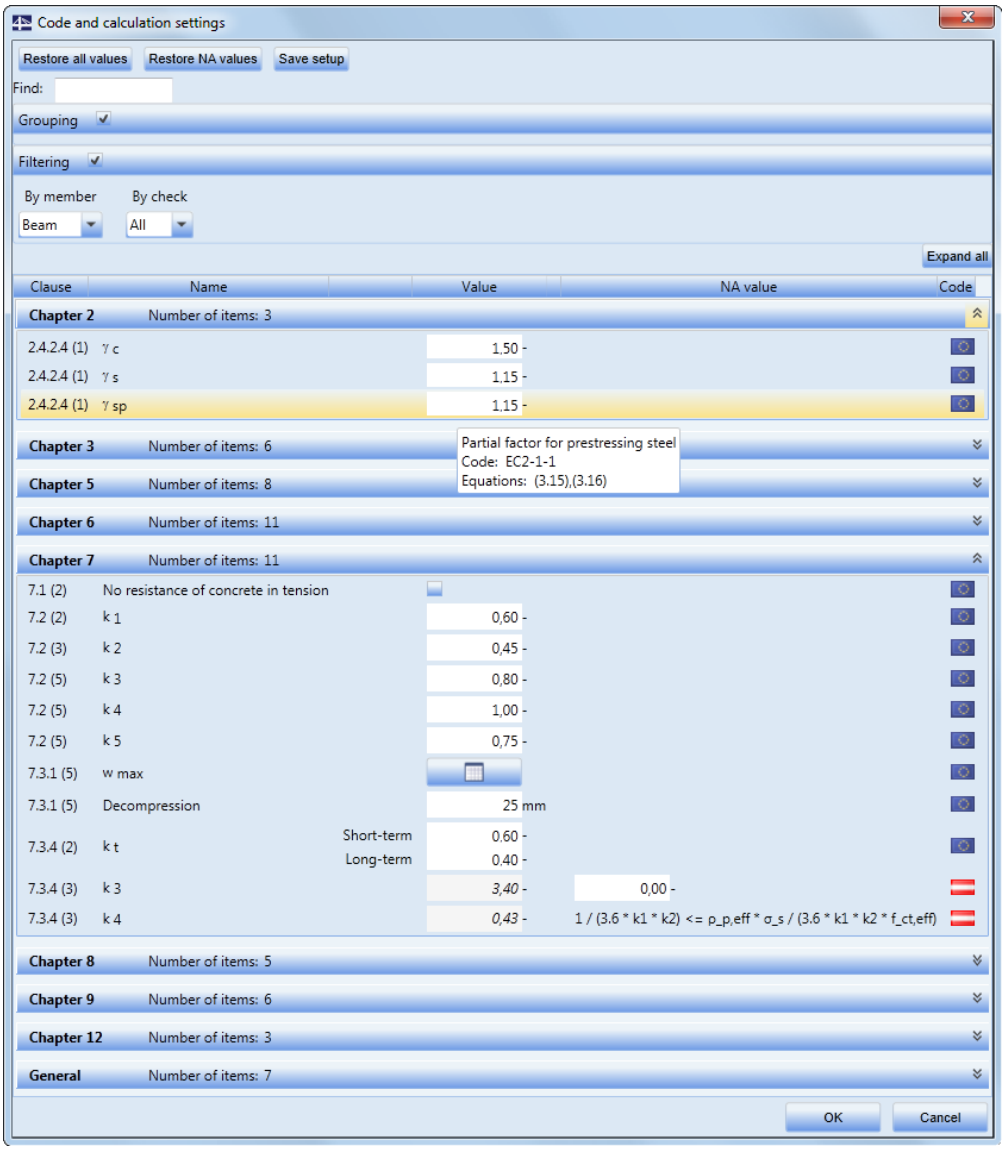


Figure 2.13: Code and calculation settings

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 chapter 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.

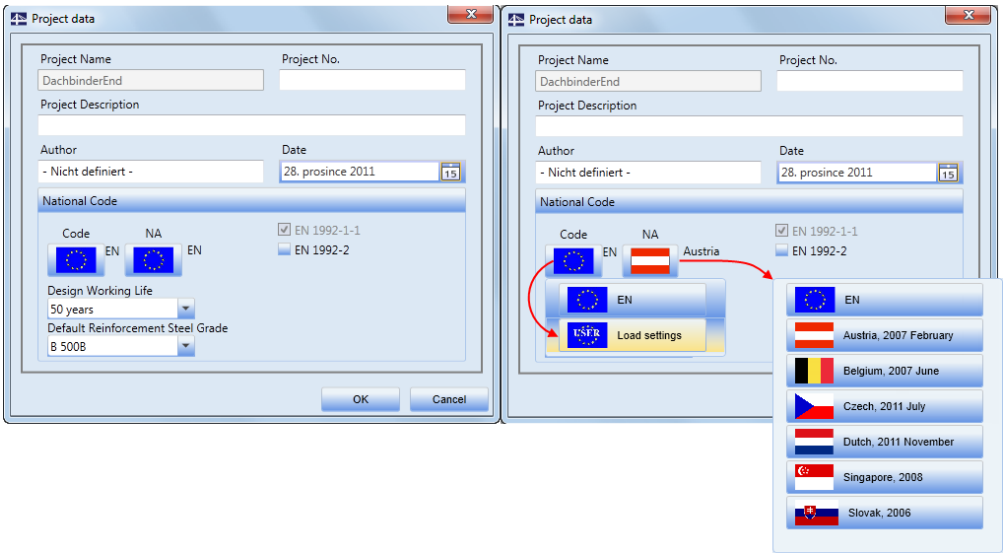


Figure 2.14: Project data, Load settings and National Annex

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.

Design Working Life

Select the value for the design working life.

Default Reinforcement Steel Grade

The default reinforcement grade from the displayed list is assigned to newly entered reinforcement bars and stirrups.

2.5 View

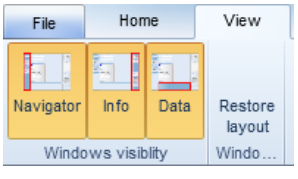


Figure 2.15: View

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*.

Selecting *Restore layout* will return the default settings of *Windows visibility* after restarting the module.

3. Navigator

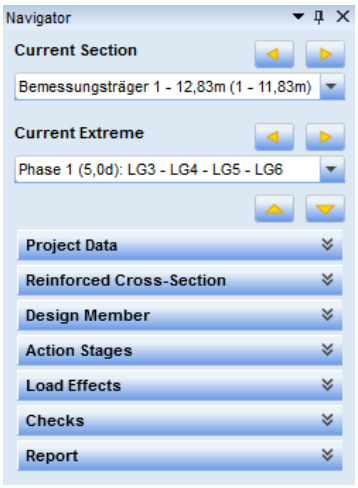


Figure 3.1: Navigator

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 be opened.

3.1 Project Data

3.1.1 Sections

Click the navigator command **Sections** to display a table with all the defined sections in the current project. In the Main window, sections are arranged as they are created (a minimum of one section is always available). The list of extremes assigned to the current section is displayed in the Data window.

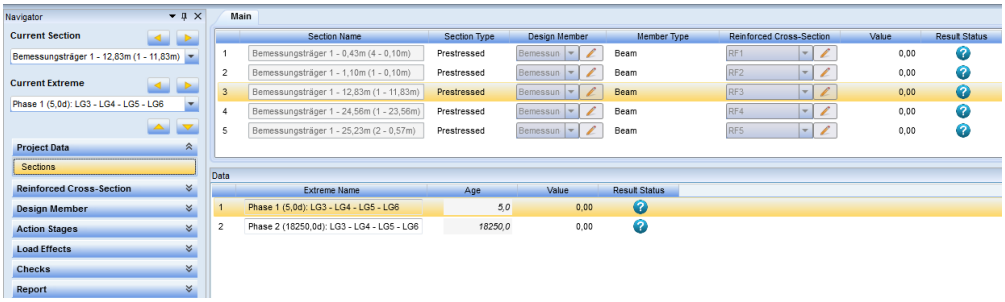


Figure 3.2: Sections

In the **Main** window, the name and distance of the section are in the *Section Name* column. In the *Reinforced Cross-Section* column, the names of reinforced cross-sections are shown. If the section is already calculated, the maximum unity check value from all possible checks for all extremes is displayed in the *Value* column. In the *Result Status* column, the actual check status (OK / NOT OK) is displayed.

The name of the extreme is shown in the *Extreme Name* column in the **Data** window. The age of concrete is displayed in the *Age* column. For each extreme the maximum value from all checks and corresponding status are displayed in the *Value* and *Result Status* columns.

The following ribbon groups are displayed at the top of the screen:

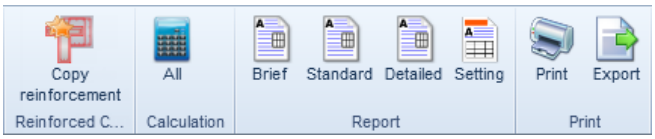


Figure 3.3: Ribbon groups for sections

Copy reinforcement

Makes a copy of an existing reinforced cross-section.

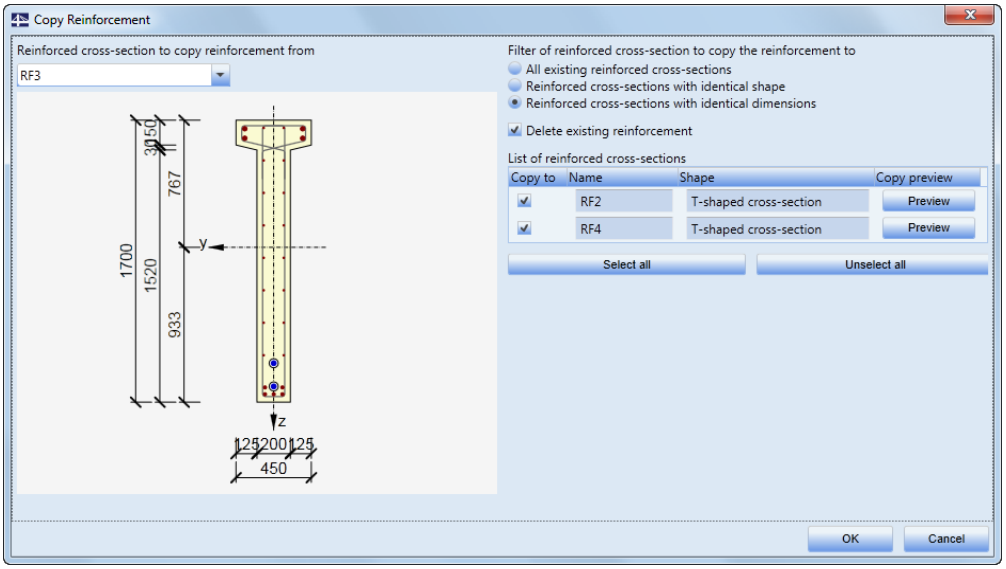


Figure 3.4: Copy reinforcement

All existing reinforced cross-sections

The reinforcement from the original reinforced cross-section will be copied to all existing reinforced cross-sections.

Reinforced cross-sections with identical shape

The reinforcement from the original reinforced cross-section will be copied to reinforced cross-sections with identical cross-section shape. The dimensions of cross-sections can be different.

Reinforced cross-sections with identical dimensions

The reinforcement from the original reinforced cross-section will be copied to reinforced cross-sections with identical cross-section shape and identical dimensions.

Delete existing reinforcement

Deletes existing reinforcement in copied reinforced cross-sections.

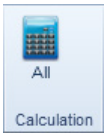


Figure 3.5: All

All

Calculates all sections which have been correctly defined by the user. The overall report is displayed afterwards.

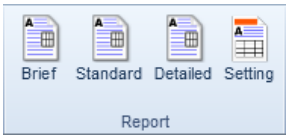


Figure 3.6: Report

Brief

Shows a brief report for sections. The following figure shows the entire generated report.

Brief summary of results of sectional checks

Section Name	Design Member	Member Type	Reinforced Section	Cross-Value	Result Status
Bemessungsträger 1 - 0,43m (4 - 0,10m)	Bemessungsträger 1	Beam	RF1	91,55	✔
Bemessungsträger 1 - 1,10m (1 - 0,10m)	Bemessungsträger 1	Beam	RF2	91,49	✔
Bemessungsträger 1 - 12,83m (1 - 11,83m)	Bemessungsträger 1	Beam	RF3	98,78	✔
Bemessungsträger 1 - 24,56m (1 - 23,56m)	Bemessungsträger 1	Beam	RF4	91,56	✔
Bemessungsträger 1 - 25,23m (2 - 0,57m)	Bemessungsträger 1	Beam	RF5	91,62	✔

Figure 3.7: Report – Brief

Standard

Shows a standard report for sections. The following figure shows only the table of contents from the generated report.

Table of contents

Chapter number	Chapter name
1.	Project data
2.	Brief summary of results of sectional checks
3.	Sectional checks
3.1.	Section Bemessungsträger 1 - 0,43m (4 - 0,10m)
3.2.	Section Bemessungsträger 1 - 1,10m (1 - 0,10m)
3.3.	Section Bemessungsträger 1 - 12,83m (1 - 11,83m)
3.4.	Section Bemessungsträger 1 - 24,56m (1 - 23,56m)
3.5.	Section Bemessungsträger 1 - 25,23m (2 - 0,57m)

Figure 3.8: Report – Standard

Detailed

Shows a detailed report for sections. The following figure shows only the table of contents from the generated report.

Table of contents

Chapter number	Chapter name
1.	Project data
2.	Brief summary of results of sectional checks
3.	Sectional checks
3.1.	Section Bemessungsträger 1 - 0,43m (4 - 0,10m)
3.2.	Section Bemessungsträger 1 - 1,10m (1 - 0,10m)
3.3.	Section Bemessungsträger 1 - 12,83m (1 - 11,83m)
3.4.	Section Bemessungsträger 1 - 24,56m (1 - 23,56m)
3.5.	Section Bemessungsträger 1 - 25,23m (2 - 0,57m)
4.	Symbols explanations
4.1.	Symbols related to check of capacity N-M-M
4.2.	Symbols related to shear checks
4.3.	Symbols related to torsion checks
4.4.	Symbols related to check of interaction of internal forces
4.5.	Symbols related to stress limitation checks
4.6.	Symbols related to crack width checks
4.7.	Symbols related to the calculation of stiffness
4.8.	Symbols related to check of detailing
5.	List of Design Members
5.1.	Design Member Bemessungsträger 1
6.	List of Reinforced Sections
6.1.	Reinforced Section RF1
6.2.	Reinforced Section RF2
6.3.	Reinforced Section RF3
6.4.	Reinforced Section RF4
6.5.	Reinforced Section RF5
7.	List of Used Materials

Figure 3.9: Report – Detailed

Setting

Shows the *Report settings* dialog box where you can select the sections and chapters to be printed for each section. You can use detailed report settings for each single section.

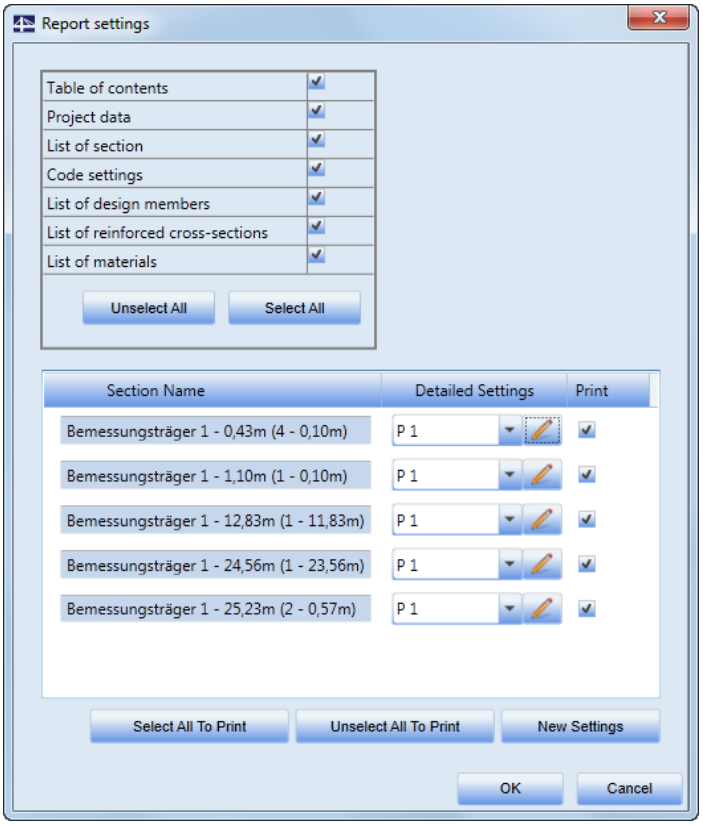


Figure 3.10: Report settings

Table of contents

If selected, a table of contents will be included in the report.

Project data

If selected, a chapter containing general project information from the navigator command 1.1 *Sections* will be included in the report.

List of sections

If selected, a chapter containing the overall results of all checked sections will be included in the report. This setting is taken into account only for *Standard* and *Detailed* report options.

Code setting

If selected, a chapter containing any code dependent variables will be included in the report. This setting is taken into account for *Detailed* reports only.

List of members

If selected, a chapter containing information about the member data will be included in the report. This setting is taken into account for *Detailed* reports only.

List of reinforced cross-sections

If selected, a chapter containing information about the reinforced cross-section will be included in the report. This setting is taken into account for *Detailed* reports only.

List of materials

If selected, a chapter about the material characteristics will be included in the report. This setting is taken into account for *Detailed* reports only.

Select all to print

Output for all sections in the list will be printed to the report.

Unselect all to print

Output for all sections in the list will not be printed to the report.

New setting

Adds new detailed print settings to the existing detailed report settings.

Print

The output for the section will be included in the report.

Detailed setting

Creates detailed print settings independently for each section. The settings are used in the report for the selected section. Selected settings can be changed by clicking the edit button.

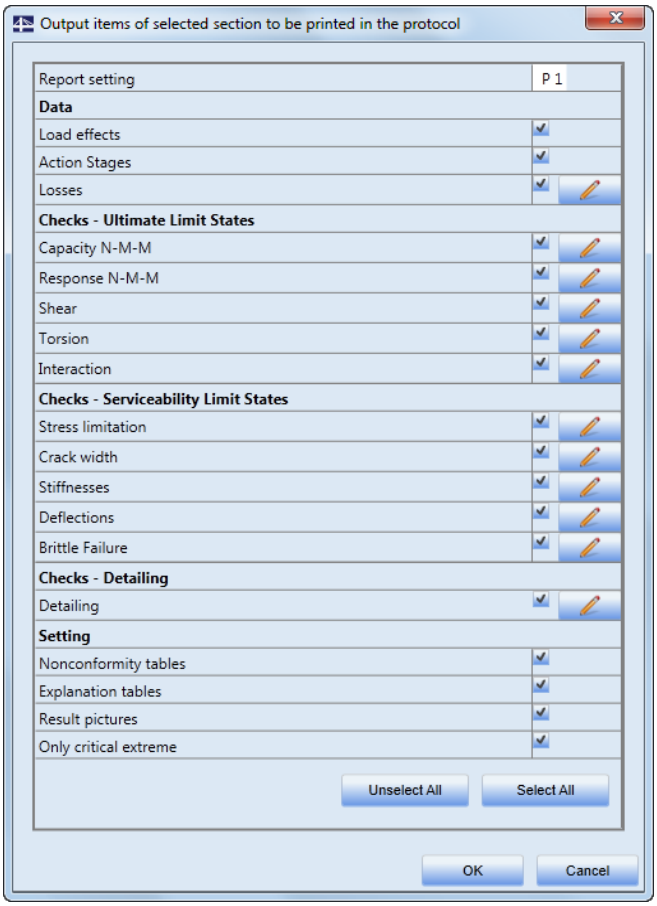


Figure 3.11: Output items

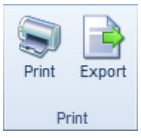


Figure 3.12: Print

Print

Prints the current report.

Export

Exports the document to an RTF file.

3.2 Reinforced Cross-section

3.2.1 Shape

This part of the menu is used for defining the cross-section shape and material properties. The *Cement class* and *Diagram type* can be edited. Basic cross-section characteristics are shown in the table.

Ribbon groups located at the top are in described in the following section, 3.2.2 Reinforcement.

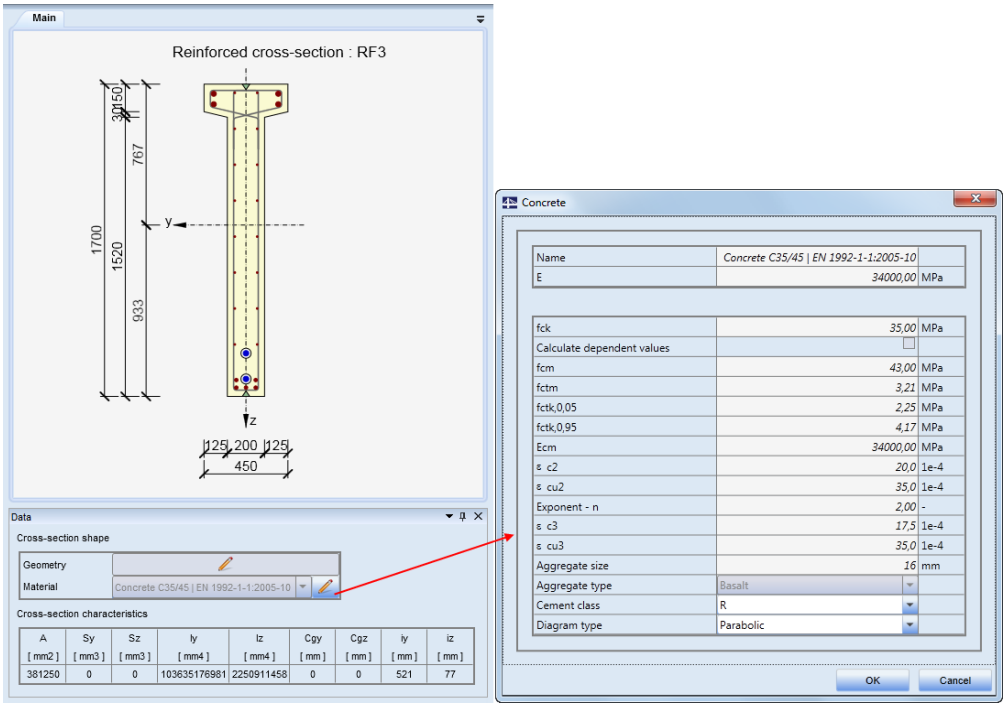


Figure 3.13: Shape

3.2.2 Reinforcement

For the current section (selected in the *Current Section* list at the top part of the Navigator window), the navigator command button **Reinforcement** launches the definition of longitudinal and shear reinforcement.

Click navigator command *Reinforcement* to activate the ribbon groups *Input by reinforcement templates*, *Stirrups*, *Longitudinal reinforcement*, *All reinforcement*, *Import*, *Export*, *Cross-section points*, *Dimension lines*, and *Calculation*.

Figure 3.14: Reinforcement

3.2.2.1 Input by Reinforcement Templates

The *Input by reinforcement templates* ribbon group contains templates for predefined sectional shapes. By clicking the button for the required reinforcement template, a dialog box appears in which you can set the parameters of the template.

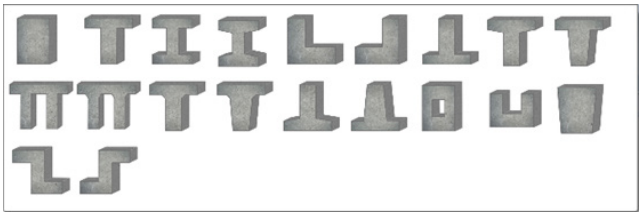


Figure 3.15: Current predefined sectional shapes

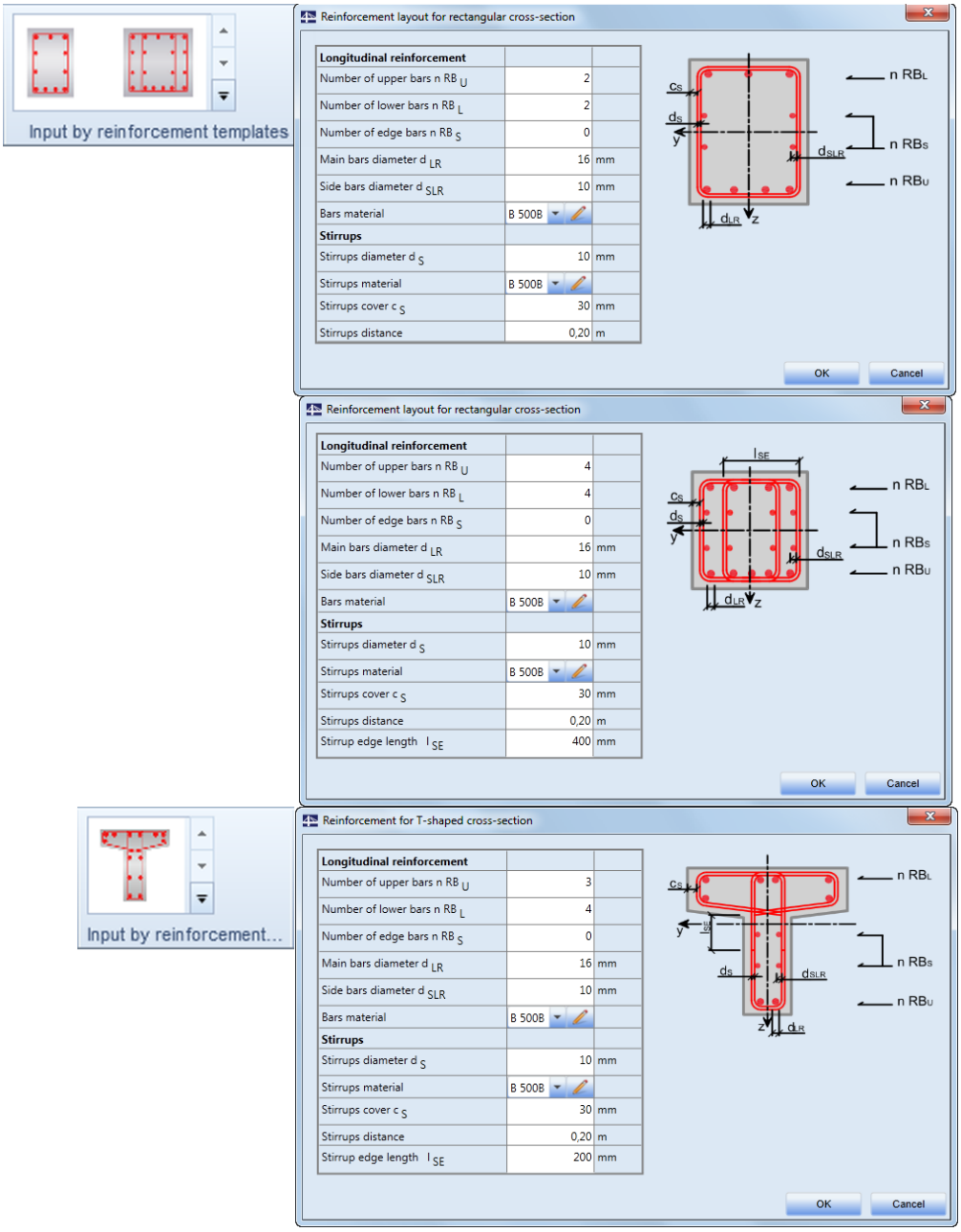


Figure 3.16: Input by reinforcement templates with examples of shapes

Click [OK] to add the reinforcement to the cross-section.

3.2.2.2 Cover

The *Cover* ribbon group contains options to define a cover for all edges of a cross-setion.

Figure 3.17: Cover

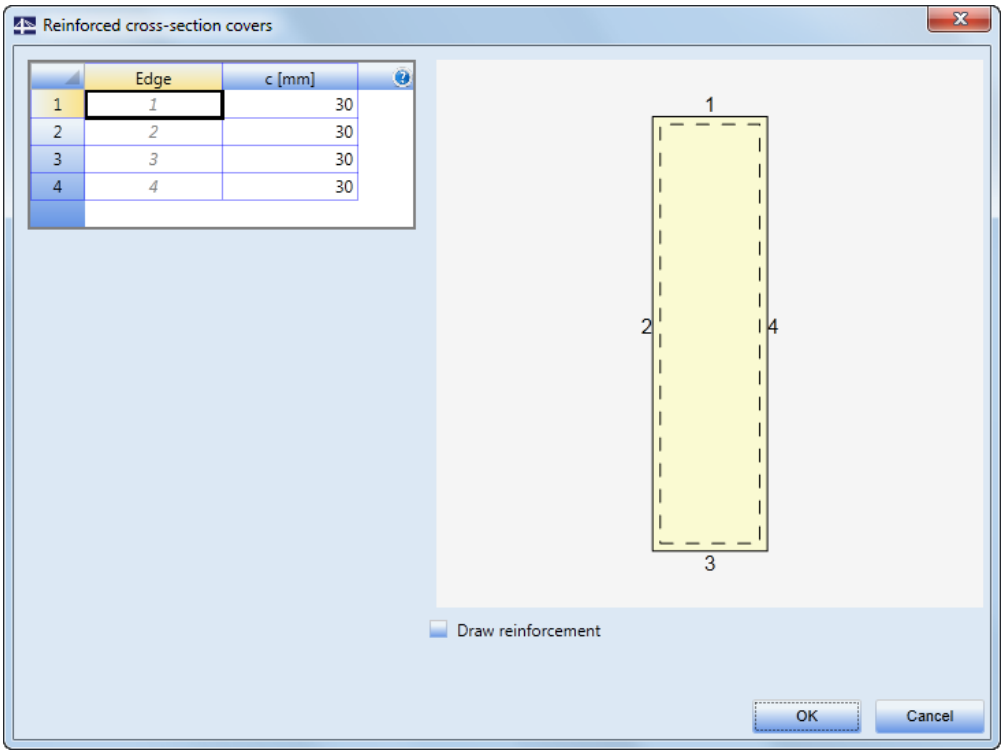


Figure 3.18: Input of cover

3.2.2.3 Stirrups

The *Stirrups* ribbon group gives effective possibilities to reinforce cross-sections with stirrups.

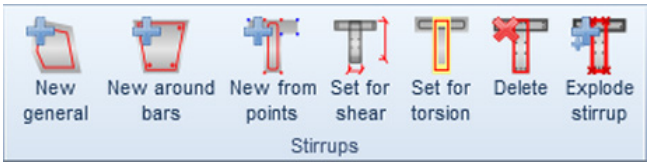


Figure 3.19: Stirrups

New general stirrup input

By selecting *New general* from the Stirrups ribbon group, the stirrup shape is defined by coordinates of the stirrup vertexes. A vertex is the intersection of two stirrup branch axes.

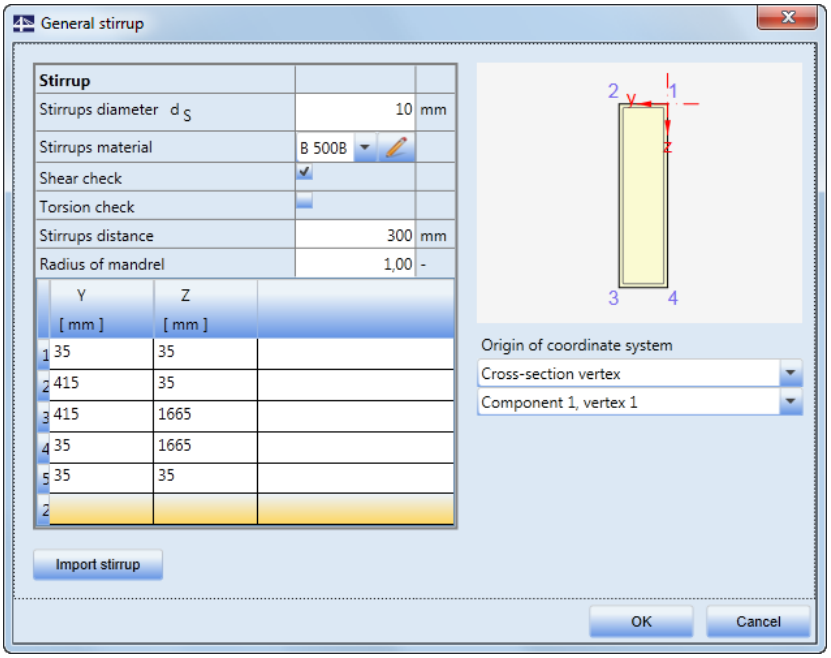


Figure 3.20: New general

The preceding figure shows the input options for a stirrup with a 10mm diameter and a 30mm cover with a rectangular cross-section with dimensions of 450mm x 1700mm.

Stirrup diameter – Input value of stirrup diameter

Stirrups material – Select or edit material of stirrup

Shear check – If the check box is selected, the stirrup is considered in the shear check

Torsion check – If the check box is selected, the stirrup is considered in the torsion check

Stirrups distance – Input value of the longitudinal distance between stirrups

Radius of mandrel – Input value of mandrel radius
(for 10mm diameter and 1.00 radius of mandrel, the actual radius is 10 x 1.00 = 10mm)

Origin of coordinate system – Defined vertex coordinates are related to a point, which can be selected it the list. Choose from the following options:

Point [0,0] – Vertex coordinates are related to origin of cross-section coordinate system.

Cross-section vertex – Vertex coordinates are related to a vertex, which is selected from the drop-down list.

Import stirrup – Click to import the stirrup coordinates from a text file (see chapter 3.2.2.6)

New around bars of main reinforcement

The stirrup shape is defined by a selection of main reinforcement bars.

The following two options are available to create a selection of bars:

- selection of bar numbers in the *Reinforcement bar* list
- selection of bar numbers graphically (click on the desired bars directly in the figure)

The stirrup is generated around the selected bars. Selected bars are listed in the *Reinforcement bar* list. After the selection of bars is finished, the list of bars (stirrup vertexes) can be edited.

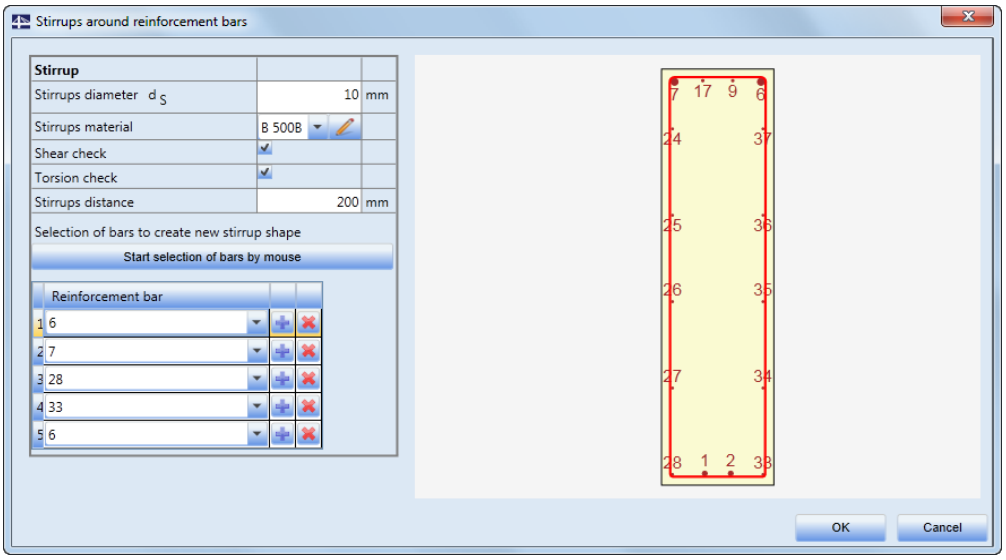


Figure 3.21: New around bars

Stirrup diameter – Input value of stirrup diameter

Stirrups material – Select or edit material of stirrup

Shear check – If the check box is selected, the stirrup is considered in the shear check

Torsion check – If the check box is selected, the stirrup is considered in the torsion check

Stirrups distance – Input value of the longitudinal distance between stirrups

Start selection of bars by mouse – Click to start the selection of bars around which to create the stirrup. If the selection is in progress, the command *Start selection of bars by mouse* is replaced by the commands:

Finish selection of bars – Finishes selection of bars, *Close stirrup* and *Step back* disappear. Stirrup is not closed automatically.

Close stirrup – Closes stirrup, creating a line between the first and last defined point, finishes selection of bars

Step back – Deletes the last defined stirrup line

New from points (cross-section vertexes)

Stirrup shape is defined by a selection of cross-section vertexes. Particular points determine particular vertexes of the stirrup.

The following two options are available to create the selection of points:

- selection of cross-section vertexes in *Points* list
- selection of cross-section vertexes graphically

The selected points are listed in the *Points* list. If vertexes are used for stirrup shape definition, the list of points (stirrup vertexes) can be edited after the points have been selected.

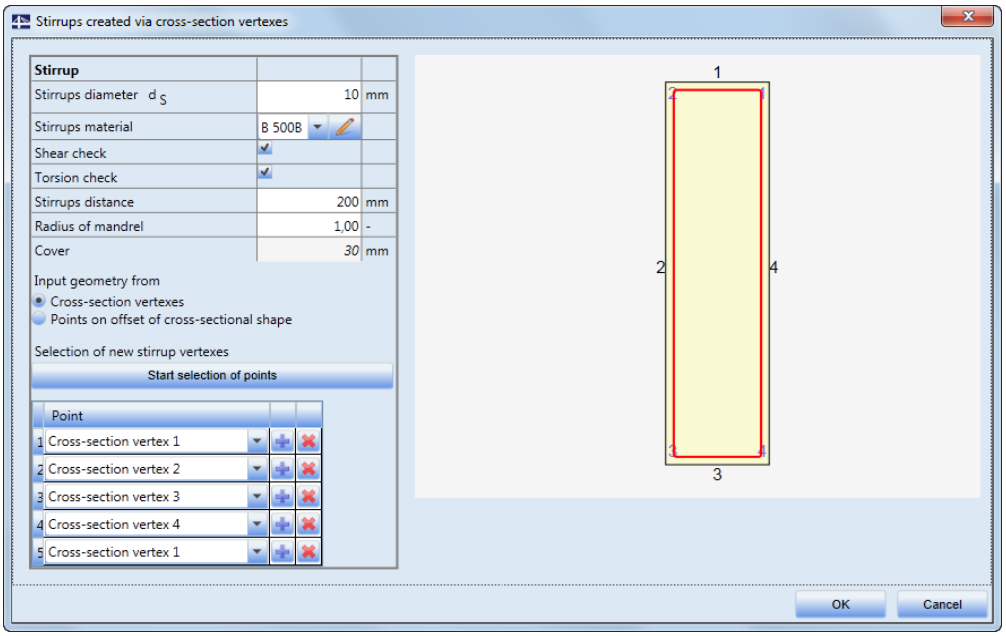


Figure 3.22: New from points

Stirrup diameter – Input value of stirrup diameter

Stirrups material – Select or edit material of stirrup

Shear check – If the check box is selected, the stirrup is considered in the shear check

Torsion check – If the check box is selected, the stirrup is considered in the torsion check

Stirrups distance – Input value of the longitudinal distance between stirrups

Radius of mandrel – Input value of mandrel radius
(for 10mm diameter and 1.00 radius of mandrel, the actual radius is 10 x 1.00 = 10mm.)

Input geometry from

Cross-section vertexes – Points of cross-section vertexes are selected to define the stirrup

Points on offset of cross-sectional shape – Points on outline of cross-section decreased by value of cover are selected to define the stirrup

Start selection of points – Click to start the selection of points to create the stirrup.
If the selection is in progress, the command *Start stirrup shape definition* is replaced by the commands:

Finish selection of points – Finishes selection of points, *Close stirrup* and *Step back* disappear. The stirrup shape is not closed automatically.

Close stirrup – Closes the stirrup creating a line between the first and last defined point, finishes the selection of bars

Step back – Deletes the last defined stirrup line

Set for shear (Input of effective cross-section for shear)

If necessary, the automatically determined values of the effective cross-section for the shear check can be replaced by user-defined values.

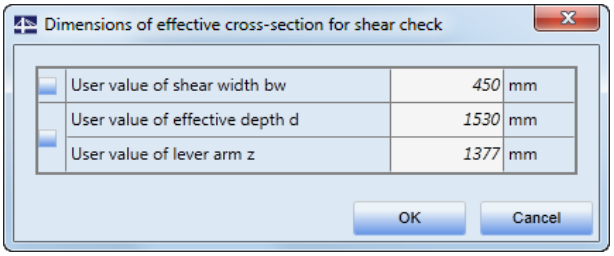


Figure 3.23: Set for shear

The calculated values of the effective cross-section dimensions are displayed in a dialog box. To enable the input of user-defined values, select the check box in the first column.

Set for torsion (Input of equivalent cross-section for torsion)

An equivalent thin-walled section is used for the calculation of torsion. The equivalent cross-section can be calculated using:

- stirrups which are marked as effective for torsion
- area and perimeter of real cross-section
- user-defined values of cross-sectional area and perimeter

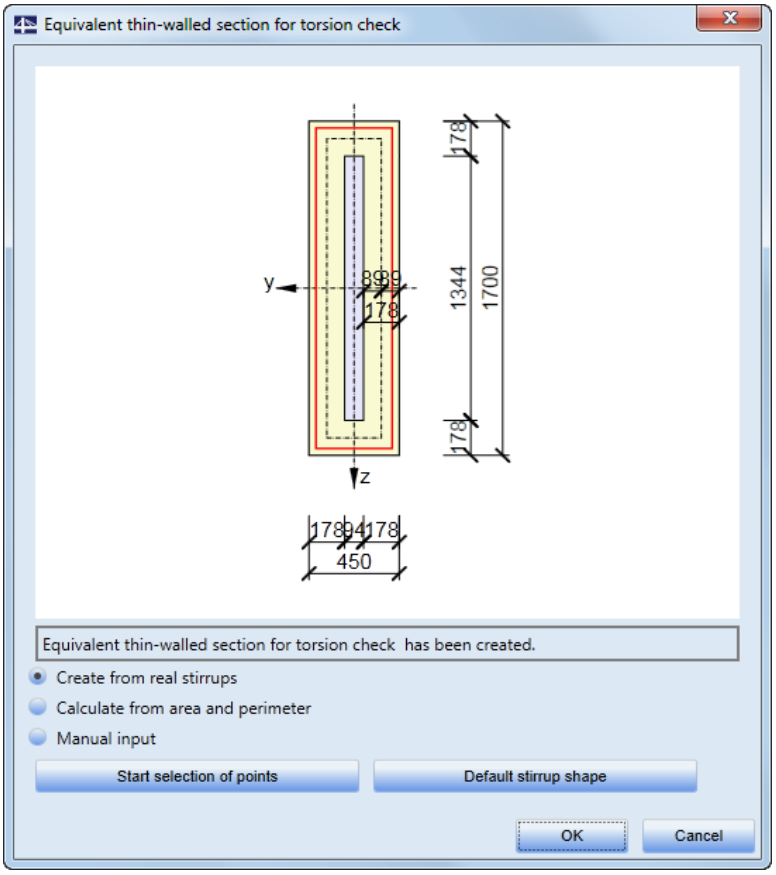


Figure 3.24: Set for torsion

Create from real stirrups – Creates equivalent thin-walled cross-section using outlines of stirrups, which are marked as effective for torsion. If this option is active, it is possible to click *Start stirrup shape definition* and adapt the shape of stirrups for check of torsion.

Start stirrup shape definition – Displays a dialog box, where the shape of a stirrup for determination of equivalent cross-section can be edited. The input of the shape is done similarly to the input of a new stirrup shape using cross-section vertexes.

Default stirrup shape – Restores shape of the stirrup, which was defined as effective for torsion.

Calculate from area and perimeter – Calculates an equivalent thin-walled cross-section using the area and perimeter of the original cross-section. Diameter, material and distance of stirrups are taken from the first stirrup, which is marked as effective for torsion.

Manual input – The values for area, perimeter and thickness of equivalent thin-walled cross-section including diameter, material and distance of stirrups are specified by the user.

Delete stirrup

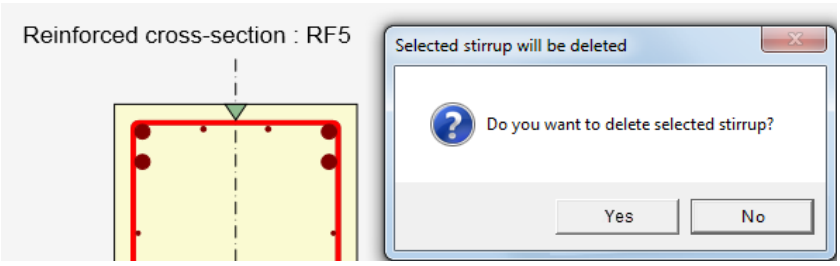


Figure 3.25: Delete (selected stirrup)

Stirrup bars can be selected graphically and removed by clicking *Delete* in the Stirrups ribbon group and then [Yes].

Explode stirrups

Stirrups can be transformed to a generally defined (general) stirrup with editable vertexes. Particular vertexes of stirrup can then be edited in the same way as when using the *New from points* option to insert a new stirrup.

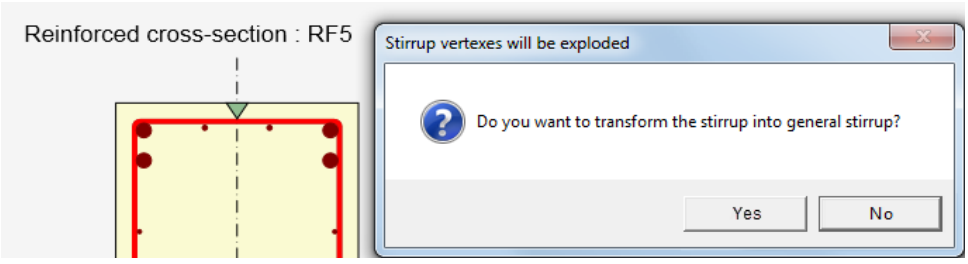
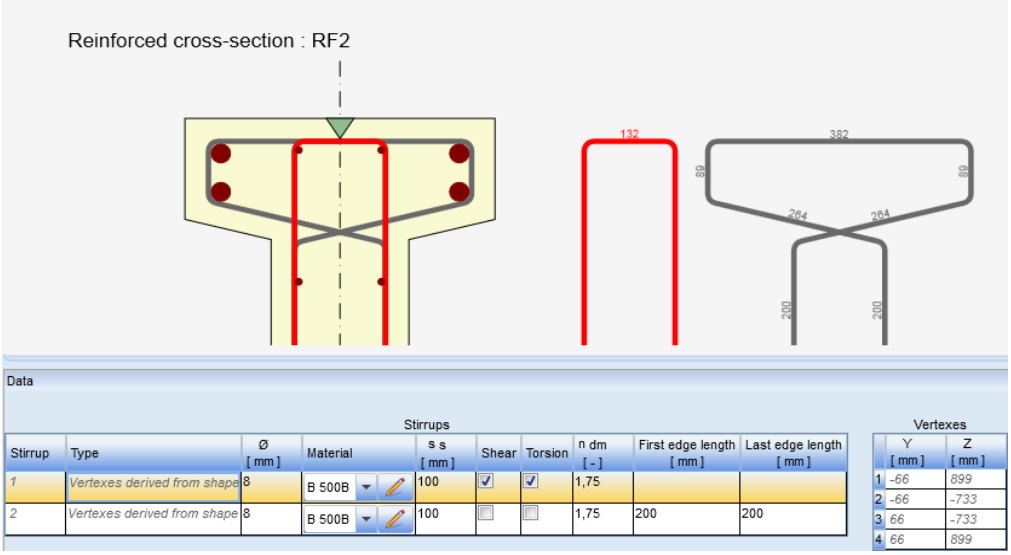


Figure 3.26: Explode (selected stirrup)

Stirrup bars can be selected graphically and transformed by clicking *Explode stirrup* from the Stirrups ribbon group and then [Yes].

Edit stirrup

It is possible to edit the currently selected stirrup. The properties of the selected stirrup are shown in a table in the *Data* window.



Stirrup	Type	Ø [mm]	Material	s s [mm]	Shear	Torsion	n dm [-]	First edge length [mm]	Last edge length [mm]
1	Vertexes derived from shape	8	B 500B	100	☑	☑	1,75		
2	Vertexes derived from shape	8	B 500B	100	☐	☐	1,75	200	200

	Y [mm]	Z [mm]
1	-66	899
2	-66	-733
3	66	-733
4	66	899

Figure 3.27: Edit (selected stirrup)

Depending on the method used for creating stirrups, the following properties can be edited:

For stirrups created from *Template*, *New around bars* and *New from points*, it is possible to adjust: diameter, material, distance, set for shear, set for torsion and radius of mandrel. The coordinates are listed, but cannot be changed by the user.

For stirrups created using *New general,* it is possible to adjust the same parameters and the listed coordinates, in addition.

3.2.2.4 Longitudinal Reinforcement

The **Longitudinal reinforcement** ribbon group contains buttons for the definition of longitudinal reinforcement.

Figure 3.28: Longitudinal reinforcement

New in line

Adds a new layer of longitudinal reinforcement defined by the coordinates of edge bars.

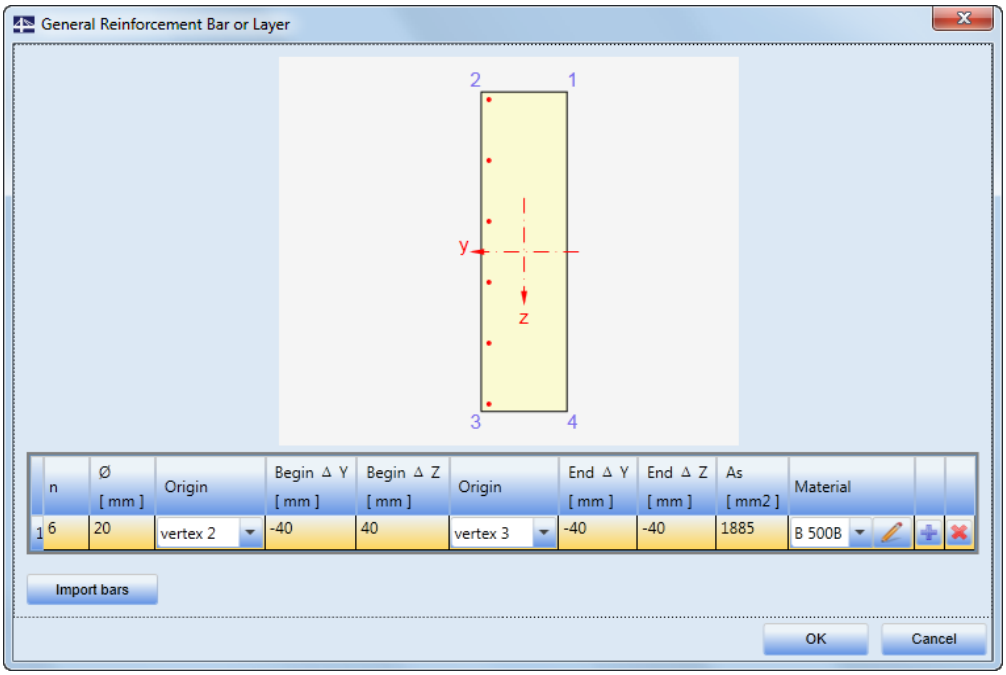


Figure 3.29: New in line

Figure 3.29 shows the input of six reinforcement bars with 20mm diameter and 30mm cover (cover is applied on longitudinal reinforcement).

The reinforcement is defined in layers. A layer is defined by the number of bars in the layer, the coordinates of the first bar in the layer, and the coordinates of the last bar in the layer. The bar diameter and material can be assigned to individual layers.

The columns in the table with longitudinal reinforcement layer are in particular:

n – Input the number of bars in reinforcement layer

Ø – Input the diameter of bars in reinforcement layer

Origin – Select origin to which coordinates of first bar in layer are related. The position of the point can be related to point [0;0] (center of gravity) or a selected cross-section vertex.

Begin Y, Begin Z – Input values of coordinates of the first bar in reinforcement layer related to the selected origin.

Origin – Select origin to which coordinates of last bar in layer are related. The position of the point can be related to point [0;0] (center of gravity) or a selected cross-section vertex.

End Y, End Z – Input values of coordinates of the last bar in reinforcement layer related to selected origin.

As – Calculated value of reinforcement area in layer

Material – In the list of available materials, select the material of bars in the reinforcement layer or click the edit button to edit material properties.

An entire layer can be added or deleted via the blue [+] or red [X] buttons.

Import bars – Click to import bar coordinates from a text file (see chapter 3.2.2.6, page 33).

New on edge

Adds a new layer of longitudinal reinforcement related to the cross-section edge.

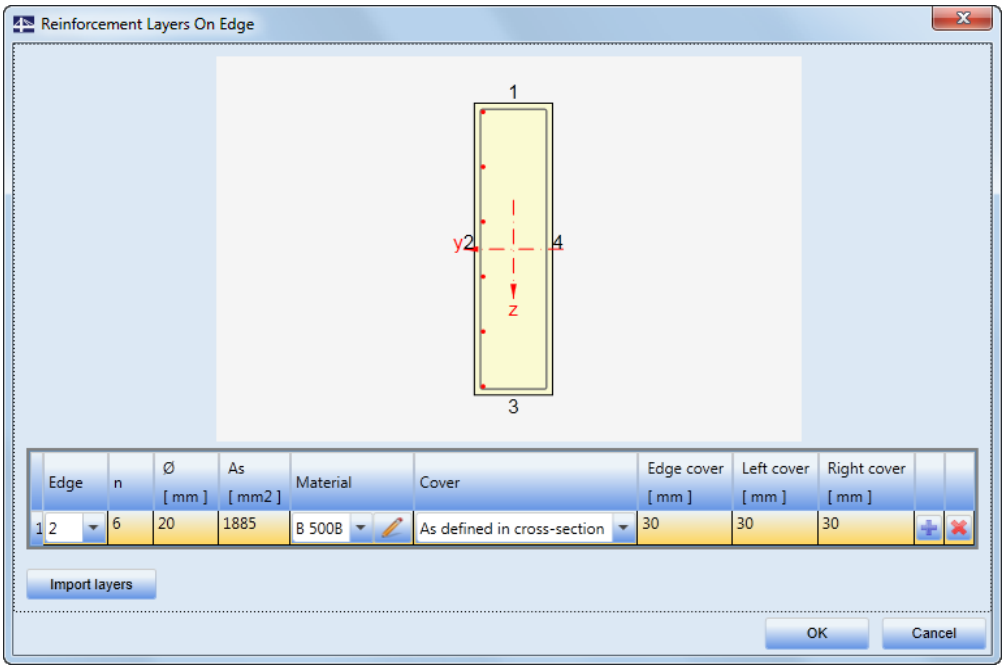


Figure 3.30: New on edge

Shown in the preceding figure is the input of six reinforcement bars with 20mm diameter and 30mm cover (cover is applied on stirrup).

The reinforcement is defined in layers. A layer is defined by the edge, the number of bars in the layer and cover. The bar diameter and material can be assigned to individual layers.

The columns in the table with longitudinal reinforcement layer are in particular:

Edge – Select edge to which layer of reinforcement is related

n – Input the number of bars in layer

Ø – Input the diameter of bars in reinforcement layer

As – Calculated value of reinforcement area in layer

Material – In the list of available materials, select the material of bars in the reinforcement layer or click the edit button to edit the material properties.

Cover – Select mode of cover determination in the list. The following modes are available:

As defined in cross-section – Values of cover are taken from the cross-section shape. Existing stirrups are taken into account.

User defined – Values of cover can be entered in the columns *Edge cover*, *Left cover*, and *Right cover*.

An entire layer can be added or deleted via the blue [+] or red [X] buttons.

Import layers – Click to import bar coordinates from a text file (see chapter 3.2.2.6).

Delete

Removes the selected layer of longitudinal bars from the cross-section.

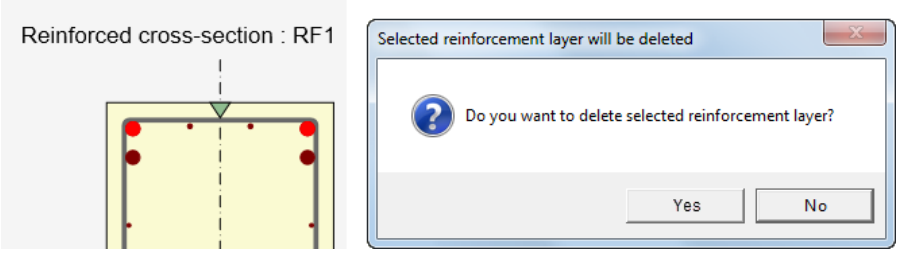


Figure 3.31: Delete (selected layer of bars)

Layer of bars can be selected graphically and removed by clicking *Delete* from the Longitudinal reinforcement ribbon group and then [Yes].

Explode layer

The longitudinal reinforcement can be transformed (i.e. exploded) to separate longitudinal bars with editable coordinates.

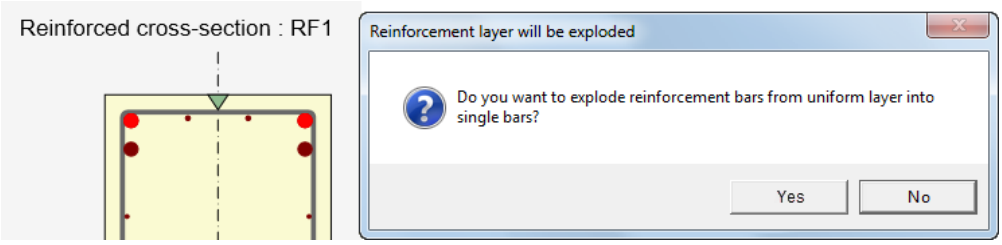


Figure 3.32: Explode layer

Layer of bars can be selected graphically and exploded by clicking *Explode layer* from the Longitudinal reinforcement ribbon group and then [Yes].

Edit longitudinal reinforcement

In order to edit the longitudinal reinforcement, it is possible to make a selection either graphically or in the table as seen in Figure 3.33.

The selected reinforcement is shown in red in the cross-section and is highlighted in yellow in the table, which allows you to easily check which cross-section is selected.

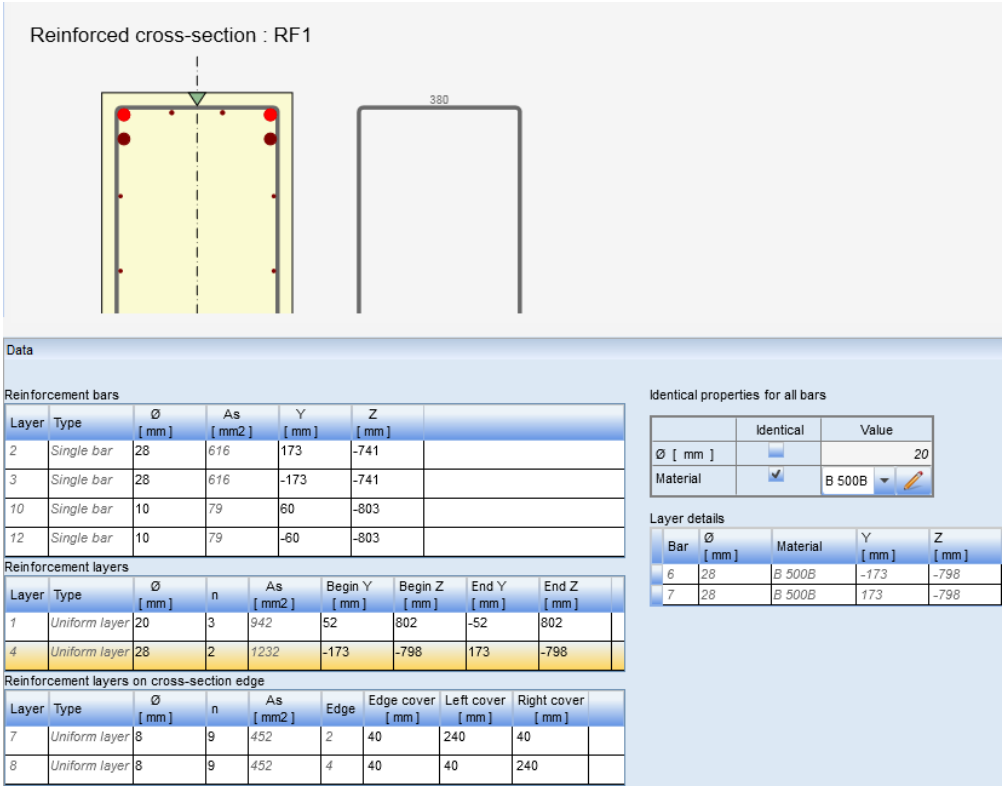


Figure 3.33: Edit longitudinal reinforcement

After the selection, it is possible to edit the following parameters in the tables:

Reinforcement bars (single bars) – diameter (Ø), coordinates (Y,Z)

Reinforcement layers – diameter (Ø), total number (n), begin (Y, Z), end (Y, Z)

Reinforcement layers on cross-section edge – diameter (Ø), total number (n), cover (edge, left, right)

Identical properties for all bars – diameter (Ø), material

3.2.2.5 All Reinforcement

To remove all reinforcement from the current section, click *Delete* in the All reinforcement ribbon group.

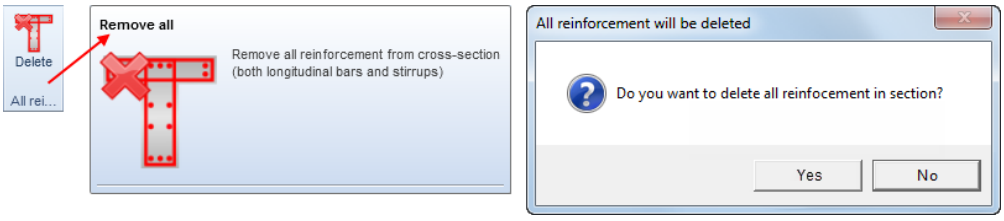


Figure 3.34: All reinforcement

All reinforcement in the section will be removed by clicking *Delete* and then [Yes].

3.2.2.6 Import, Export

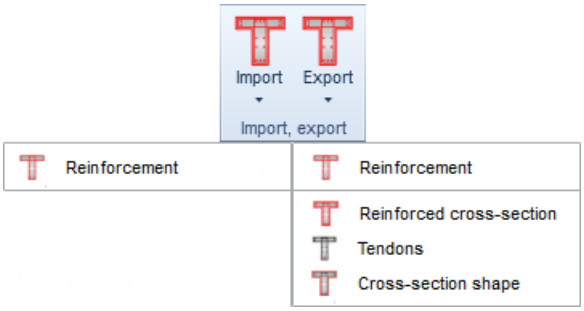


Figure 3.35: Import, Export

The **Import / Export** ribbon group can be used to import and export reinforcement (longitudinal reinforcement and stirrups).

The reinforcement is saved in text format *.NAV which is described in detail in appendix **A *Text Format *.NAV***.

The button *Export* includes additional export options in the text format *.NAV:

Reinforced cross-section – cross-section, longitudinal reinforcement, stirrups, tendons

Tendons – only tendons

Cross-section shape – only cross-section without reinforcement

3.2.2.7 Cross-section Points

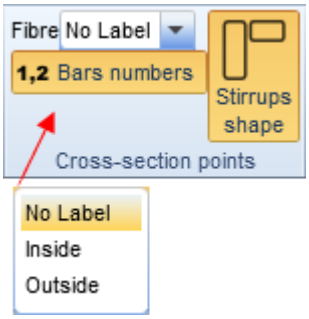


Figure 3.36: Cross-section points

The **Cross-section points** ribbon group can be used to set drawing options of fibre and bar numbers.

Fibre – Select mode for drawing fibres from the list. One of following modes can be chosen:

No labels – Description of fibres is not drawn.

Outside – Fibre numbers are drawn outside the cross-section outline

Inside – Fibre numbers are drawn inside the cross-section outline

Bar numbers – Turns on/off the display of reinforcement bar numbers

Stirrups shape – Turns on/off the display of the stirrup shapes

3.2.2.8 Dimension Lines

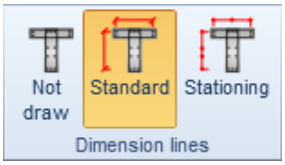


Figure 3.37: Dimension lines

The **Dimension lines** ribbon group controls the mode of drawing dimension lines:

Not draw – Turns off the display of dimension lines

Standard – Turns on the display of standard dimension lines of reinforcement

Stationing – Turns on the display of dimension lines with distances related to a reference point

3.2.2.9 Calculation

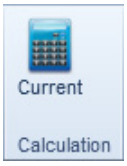


Figure 3.38: Calculation

The **Calculation** ribbon group is used to calculate the current section.

3.2.3 Tendons

In this menu, information about tendons is available.

The ribbon groups located at the top are described in chapter 3.2.2 *Reinforcement*.

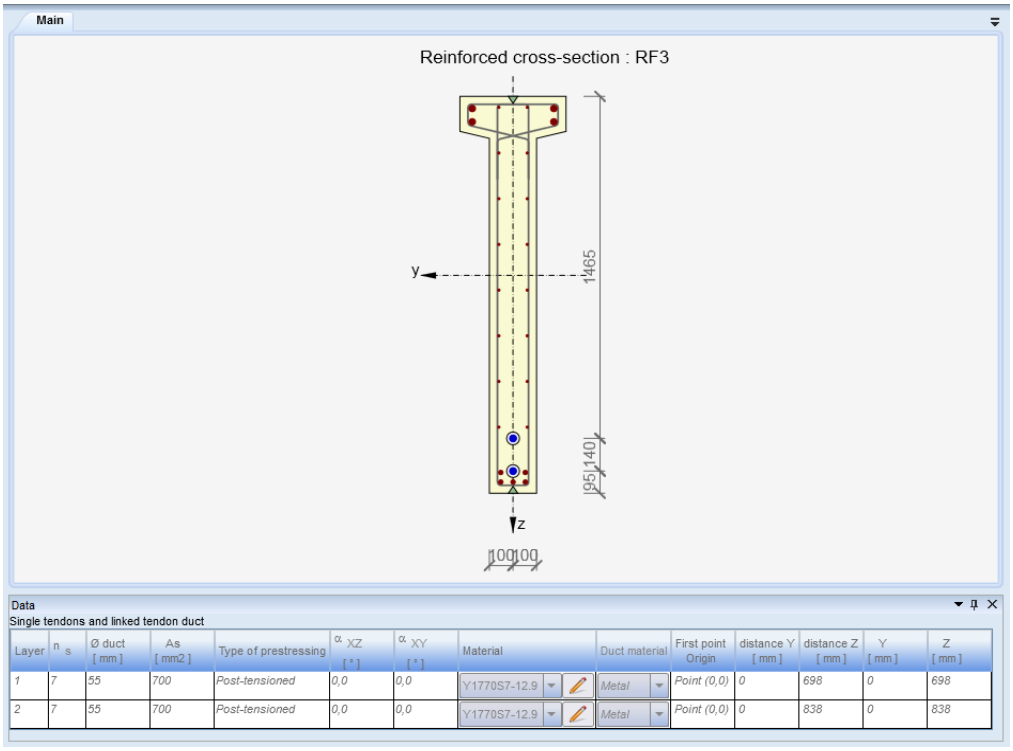


Figure 3.39: Tendons

3.3 Design Member

3.3.1 Member Data

Click navigator command **3.1 Member data** to launch a dialog box containing general member data for the current section.

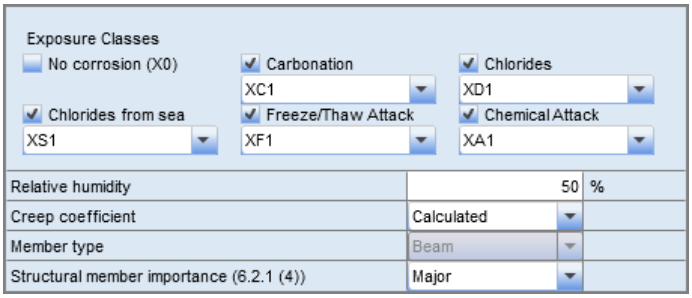


Figure 3.40: Member data

3.3.2 Construction Stages

In this menu, information about construction stages is available.

When initialized in phase 1 (5 days), *Prestressing* is selected, and remains also in phase 2 (18250 days).

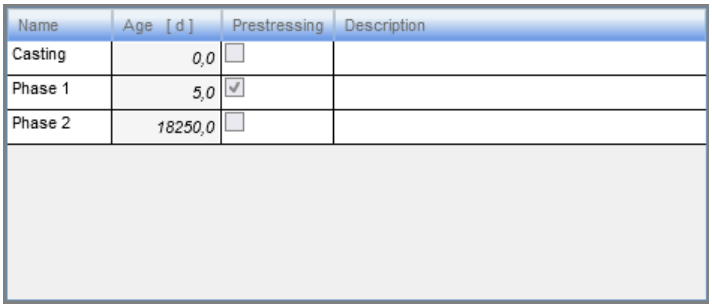


Figure 3.41: Construction stages

3.4 Action Stages in Section

Increments of effects of characteristic permanent load

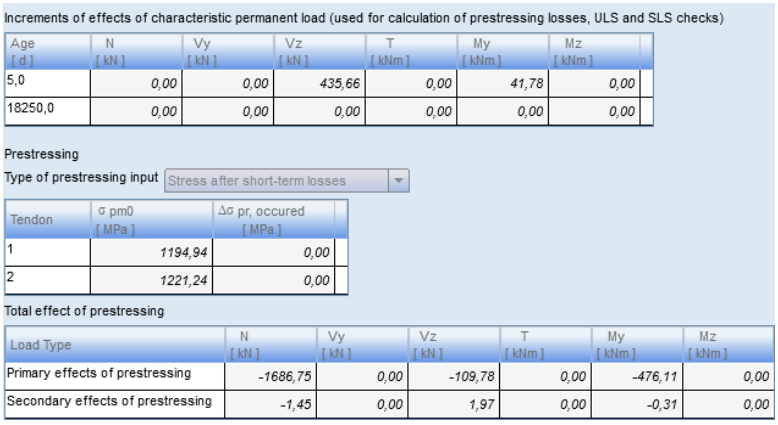
Table which shows the values of load increments for current section and points of time axis.

Stress after short-term losses

In the table below you can find the values of calculated stress in prestressing reinforcement just after the introduction of prestressing and values of relaxation occurred in the past.

Total effect of prestressing

This table displays the calculated values of *Primary effects of prestressing* in cross-section. User-defined values of secondary effects of prestressing can be entered into the *Secondary effects of prestressing* row.



Increments of effects of characteristic permanent load (used for calculation of prestressing losses, ULS and SLS checks)

Age [d]	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

Prestressing

Type of prestressing input: Stress after short-term losses

Tendon	σ pm0 [MPa]	Δσ pr, occured [MPa]
1	1194,94	0,00
2	1221,24	0,00

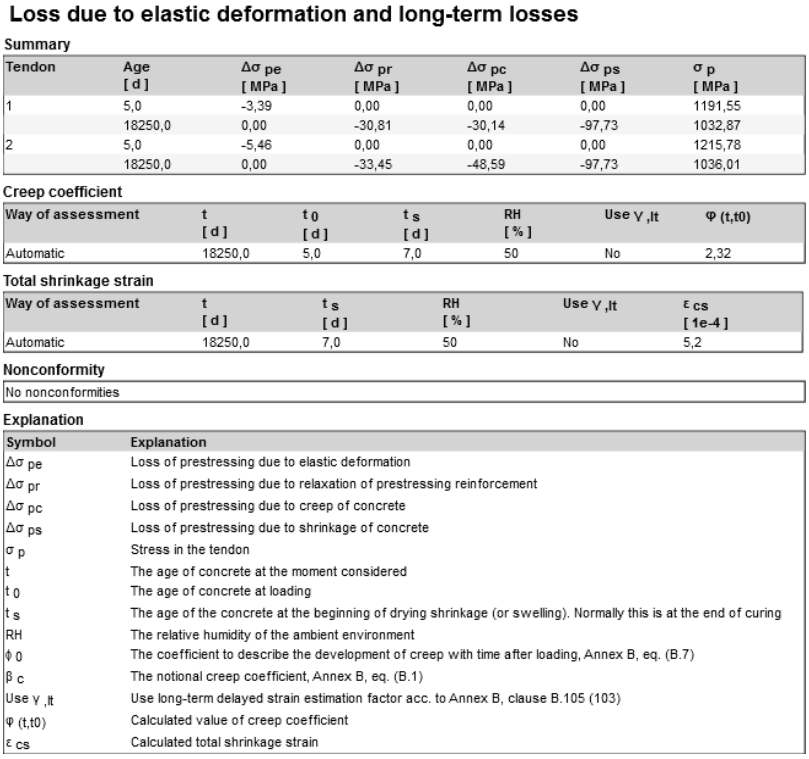
Total effect of prestressing

Load Type	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Primary effects of prestressing	-1686,75	0,00	-109,78	0,00	-476,11	0,00
Secondary effects of prestressing	-1,45	0,00	1,97	0,00	-0,31	0,00

Figure 3.42: Total effect of prestressing

Loss due to elastic deformation and long-term losses

The values are displayed in the *Data* window.



Loss due to elastic deformation and long-term losses

Summary

Tendon	Age [d]	Δσ pe [MPa]	Δσ pr [MPa]	Δσ pc [MPa]	Δσ ps [MPa]	σ p [MPa]
1	5,0	-3,39	0,00	0,00	0,00	1191,55
	18250,0	0,00	-30,81	-30,14	-97,73	1032,87
2	5,0	-5,46	0,00	0,00	0,00	1215,78
	18250,0	0,00	-33,45	-48,59	-97,73	1036,01

Creep coefficient

Way of assessment	t [d]	t_0 [d]	t_s [d]	RH [%]	Use $\gamma_{,lt}$	φ (t,t0)
Automatic	18250,0	5,0	7,0	50	No	2,32

Total shrinkage strain

Way of assessment	t [d]	t_s [d]	RH [%]	Use $\gamma_{,lt}$	ε_{cs} [1e-4]
Automatic	18250,0	7,0	50	No	5,2

Nonconformity

No nonconformities

Explanation

Symbol	Explanation
$\Delta\sigma_{pe}$	Loss of prestressing due to elastic deformation
$\Delta\sigma_{pr}$	Loss of prestressing due to relaxation of prestressing reinforcement
$\Delta\sigma_{pc}$	Loss of prestressing due to creep of concrete
$\Delta\sigma_{ps}$	Loss of prestressing due to shrinkage of concrete
σ_p	Stress in the tendon
t	The age of concrete at the moment considered
t_0	The age of concrete at loading
t_s	The age of the concrete at the beginning of drying shrinkage (or swelling). Normally this is at the end of curing
RH	The relative humidity of the ambient environment
ϕ_0	The coefficient to describe the development of creep with time after loading, Annex B, eq. (B.7)
β_c	The notional creep coefficient, Annex B, eq. (B.1)
Use $\gamma_{,lt}$	Use long-term delayed strain estimation factor acc. to Annex B, clause B.105 (103)
φ (t,t0)	Calculated value of creep coefficient
ε_{cs}	Calculated total shrinkage strain

Figure 3.43: Loss due elastic deformation and long-term losses

3.5 Load Effects

3.5.1 Internal Forces in Section

For the current section and current extreme (set in lists *Current section* and *Current extreme* at the top of the navigator), click the navigator command **Internal forces in Section** to show the internal forces. The forces are shown in the *Combination types* table.

Effects of design load (at the time of check)

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
	Variable Qd1	0,00	0,00	111,60	0,00	10,61	0,00
	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
	Variable Qd1	0,00	0,00	74,40	0,00	7,07	0,00
	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
	Variable Qd1	0,00	0,00	14,88	0,00	1,41	0,00
	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
	Variable Sum Qdi	0,00	0,00	0,00	0,00	0,00	0,00

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
Effect of prestressing	-1682,01	0,00	-107,46	0,00	-474,28	0,00

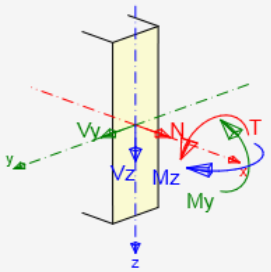


Figure 3.44: Internal forces in section

The combinations may be of various types. Each type is used for different checks:

Fundamental ULS

The values of internal forces defined in this combination type are used to perform ultimate limit state checks (command 6.3 in the navigator) and the detailing check.

Characteristic

The values of internal forces defined in this combination type are used to perform the stress limitation check.

Frequent

The values of internal forces defined in this combination type are used to perform the crack width check and stress limitation check.

Quasi permanent

The values of internal forces defined in this combination type are used to perform the stress limitation check, crack width check, stiffness check and deflection checks.

3.6 Checks

By selecting **Checks** in the navigator, the section for checks becomes available.

Each cross-section check has a graphical and textual representation. Graphically, the presented results are drawn in the *Main* window. The results in textual presentation are displayed in the *Data* window. For the checks whose graphical presentation is not useful, a picture of the reinforced cross-section with overall dimensions and reinforcement data is displayed in the *Main* window.

The graphical presentation of checks is adjustable by a settings dialog box. The settings are saved for the check and later used when the figure is printed to the report.

3.6.1 Settings

For the current section, click the navigator command **Setting** to launch the dialog box to select which checks should be performed for the current section.

Only the selected checks will be executed. Unselected checks are not calculated and the results will be empty, therefore those checks cannot be included in the report.

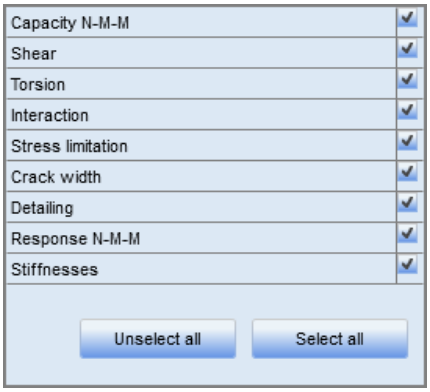


Figure 3.45: Settings

Unselect all – Unselect all selected checks

Select all – Selects all checks

3.6.2 Overall

For the current section and current extreme, click the navigator command **Overall** to display an overview of results for all executed checks in the section. In the *Main* graphical window the reinforced cross-section with information about tendons, longitudinal and shear reinforcement is displayed. The *Components label* ribbon group is available for this navigator command.

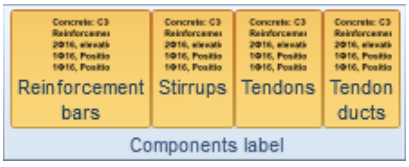


Figure 3.46: Components label

Reinforcement bars – Shows or hides longitudinal reinforcement description in the picture of reinforced cross-section

Stirrups – Shows or hides stirrup descriptions in the picture of reinforced cross-section

Tendons – Shows or hides tendon descriptions in the picture of reinforced cross-section

Tendon ducts – Shows or hides tendon duct descriptions in the picture of reinforced cross-section

An overview of all check results is displayed in tabular form in the *Data* window. The check with the maximum value is labeled as the *Governing type of check*.

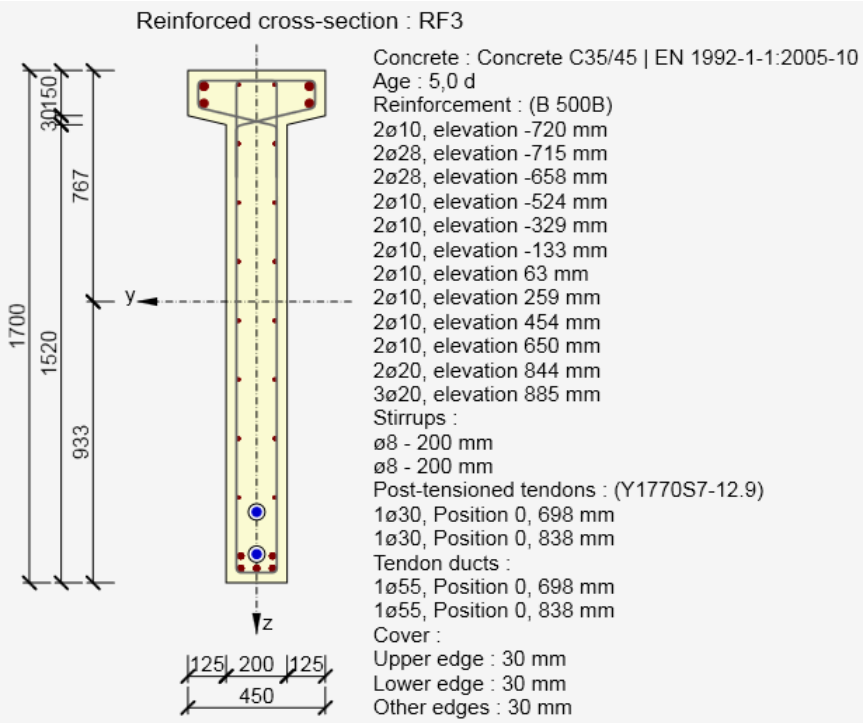


Figure 3.47: Main window

Overall

Governing type of check	N Ed [kN]	M Ed,y [kNm]	M Ed,z [kNm]	V Ed [kN]	T Ed [kNm]	Value [%]	Check
Stress Limitation	-1750,99	1119,41	0,00			98,78	OK

Type of check	N Ed [kN]	M Ed,y [kNm]	M Ed,z [kNm]	V Ed [kN]	T Ed [kNm]	Value [%]	Check
Capacity N-M-M	12,72	3407,41	0,00			83,34	OK
Response N-M-M	-1750,99	2052,38	0,00			89,29	OK
Shear	-1750,99			-19,69		8,92	OK
Torsion					0,00	0,00	OK
Interaction	-1750,99	2052,38	0,00	-19,69	0,00	89,29	OK
Stress Limitation	-1750,99	1119,41	0,00			98,78	OK
Crack Width	-1575,89	879,22	0,00			4,29	OK
Detailing	-1750,99	2052,38	0,00			65,63	OK

Figure 3.48: *Data* window

3.6.3 Ultimate Limit States

For the current section and extreme, click the navigator command **Ultimate Limit States** to display the results overview of all executed ultimate limit state checks in the section.

The content in the *Main* graphical window is the same as described in chapter 3.6.2. An overview of ultimate limit state checks is displayed in tabular form in the *Data* window and the ribbon group *Components label* is also available for this navigator command.

3.6.3.1 Capacity N-M-N

For the current section and extreme, click the navigator command **Capacity N-M-N** to execute the capacity check and display the resulting interaction diagrams.

The command activates the ribbon groups *Diagram type, Interaction surface sections*, and *Draw points*. The interaction diagrams are drawn in the *Main* window and the text representation is available in the *Data* window.

Load effects with prestressing actions are used for interaction diagram calculation of staged sections, but for better graphical visualization, the whole figure is transformed in the origin of the cross-section.

Cross-sectional resistance is determined assuming proportional change of all components of acting internal forces (the eccentricity of normal force remains constant) until interaction surface is reached. The change of acting internal forces can be interpreted as the movement along the line connecting the origin of the coordinate system (0,0,0) and the point of acting internal forces (N_{Ed}, M_{Edy}, M_{Edz}). Two points of intersection of the connecting line and interaction surface, which can be found, represent two sets of forces of resistance. Three resistance forces are determined in each point of intersection by the program: normal force capacity N_{Rd}, and capacities in flexure M_{Rdy} and M_{Rdz}.

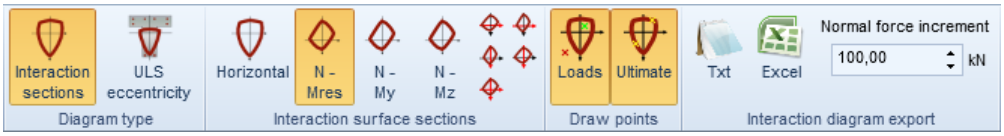


Figure 3.49: Ribbon groups

Diagram type

Interaction sections - Turns on/off drawing of interaction surface sections

ULS eccentricity – Turns on/off drawing of "ULS kern", i.e. interaction diagram recalculated to the eccentricity of normal force. In cases when normal force is zero, a horizontal section in M_y-M_z is drawn.

Interaction surface sections

Horizontal – Turns on/off drawing of horizontal section of intersection surface through the point (N_{Ed},0,0).

N-M result – Turns on/off drawing of the vertical section of intersection surface through the origin of the coordinate system and the result of bending moments $M_{Ed,y}$, $M_{Ed,z}$. If the both sections are zero, the section is drawn in the plane N-M_y.

N-My – Turns on/off drawing of a vertical section of intersection surface through the point (0,0,$M_{Ed,z}$), parallel with the plane N-M_y.

N-Mz – Turns on/off drawing of a vertical section of intersection surface through the point (0,0,$M_{Ed,y}$), parallel with the plane N-M_z.

The buttons on the right side of the ribbon group are used for adjusting the vertical and horizontal scale of the interaction diagram, or respectively for setting default scales in both directions.

Draw points

Loads – Turns on/off the drawing of points representing load effects, i.e. applied design internal forces.

Ultimate – Turns on/off drawing of points representing design resistance forces.

Interaction diagram export

Txt – Exports the points of interaction diagram to a text file.

Excel – Opens Microsoft Excel and exports the points of interaction diagram to the first sheet.

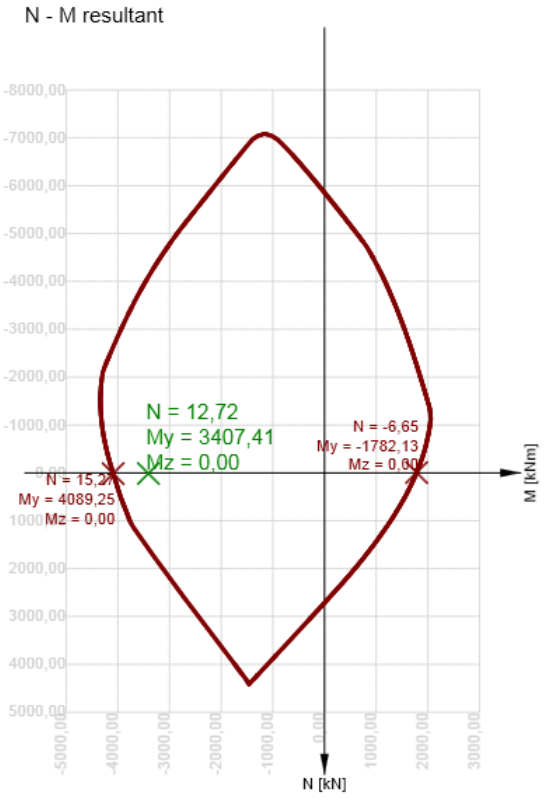


Figure 3.50: Main window – Interaction section - Horizontal

Capacity N-M-M

N Ed [kN]	M Ed,y [kNm]	M Ed,z [kNm]	Type	Value [%]	Limit [%]	Check
12,72	3407,41	0,00	NuMuMu	83,34	100,00	OK

Design resistance of css subjected to bending and axial force

Type	F Ed	F Rd1	F Rd2
N [kN]	12,72	15,27	-6,65
M y [kNm]	3407,41	4088,79	-1781,59
M z [kNm]	0,00	0,00	0,00

Figure 3.51: Data window

3.6.3.2 Shear

In order to execute the shear check for the current section and extreme, select **Shear** from the navigator. In the *Main* graphical window the reinforced cross-section is displayed with the geometrical, material, and overall data about the shear and longitudinal reinforcement. A detailed text representation of the shear check is available in the *Data* window.

The **Components label** ribbon group is available for this navigator command and is described in chapter 3.6.2 on page 38.

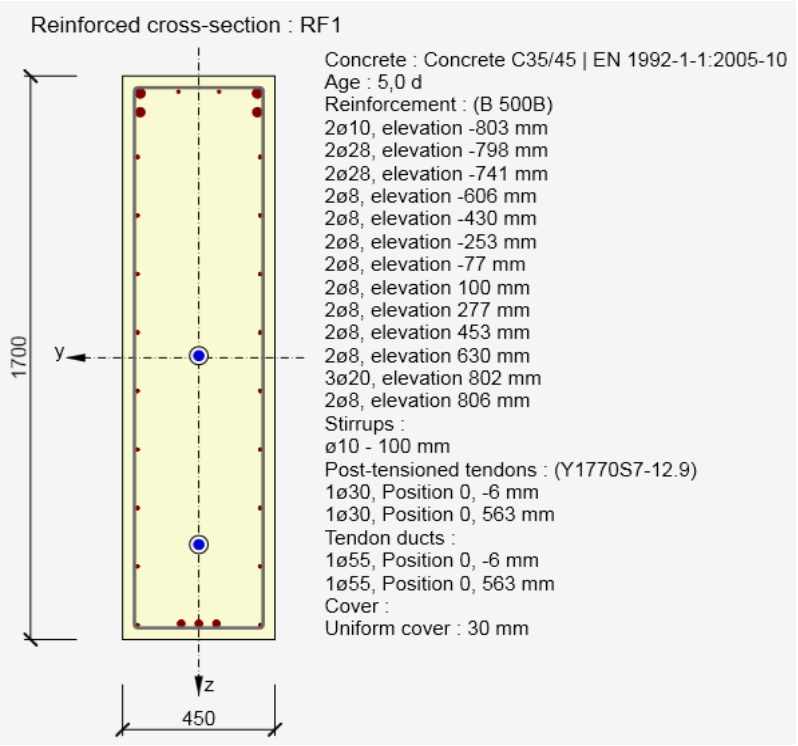


Figure 3.52: *Main* window

Shear

V_{Ed} [kN]	N_{Ed} [kN]	Clause	Value [%]	Limit [%]	Check
592,27	-1682,01	6.2.3(3)	57,44	100,00	OK

Design and resistance shear forces

V_{Ed} [kN]	$V_{Rd,c}$ [kN]	$V_{Rd,max}$ [kN]	$V_{Rd,r}$ [kN]	$V_{Rd,s}$ [kN]	V_{Rd} [kN]
592,27	356,19	2917,75	2617,26	1031,10	1031,10

Input values and intermediate results of shear design

n_c	A_{sw} [mm2]	A_{sl} [mm2]	b_w [mm]	d [mm]	z [mm]	v [°]	α [°]	α_{cw} [-]
2	1571	0	384	1530	1377	40,0	90,0	1,13

$C_{Rd,c}$ [-]	k [-]	k_1 [-]	ρ_l [-]	σ_{cp} [MPa]	σ_{wd} [MPa]	v_{min} [MPa]	v [-]	v_l [-]
0,12	1,36	0,15	0,00	2,20	229,76	0,28	0,54	0,60

Figure 3.53: *Data* window

3.6.3.3 Torsion

For the current section and extreme, click the navigator command **Torsion** to execute the torsion check. In the *Main* graphical window a figure of reinforced cross-section and a figure of equivalent thin-walled cross-section are drawn. This particular check does not have any graphical representation of its own. In the *Data* window a detailed text representation of the torsion check is displayed.

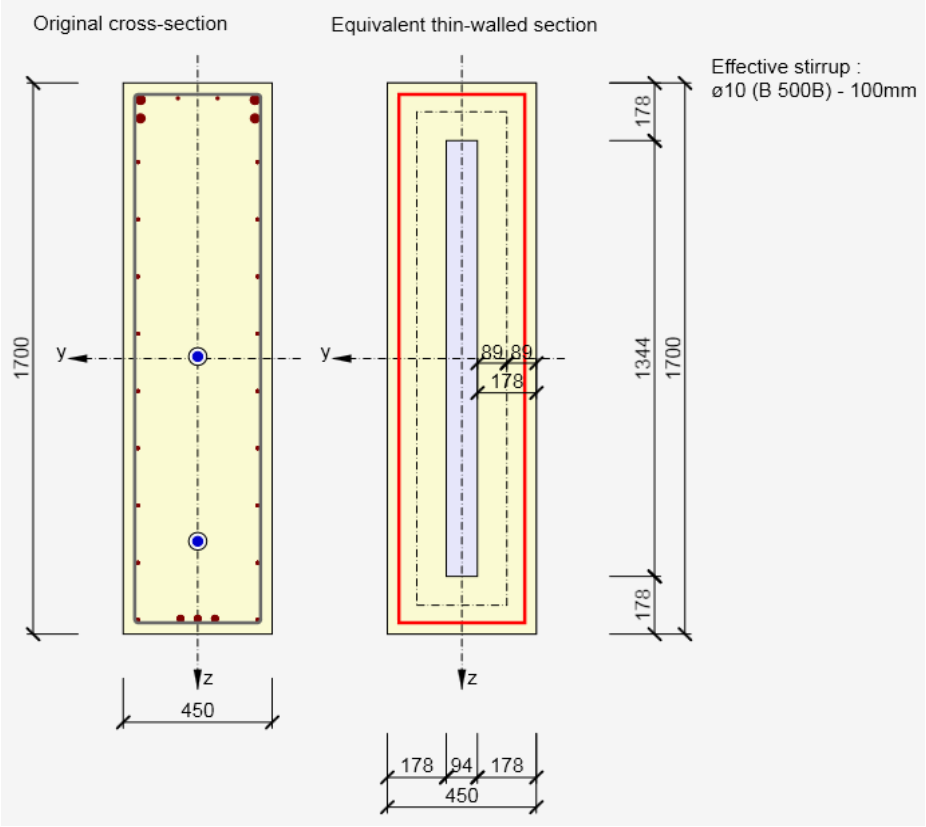


Figure 3.54: *Main* window

Torsion

T_{Ed} [kNm]	Value [%]	Limit [%]	Check
0,00	0,00	100,00	OK

Design and resistance torsional moments

T_{Ed} [kNm]	$T_{Rd,c}$ [kNm]	$T_{Rd,max}$ [kNm]	$T_{Rd,s}$ [kNm]	T_{Rd} [kNm]
0,00	167,96	732,74	0,00	167,96

Input values and intermediate results of shear design

A_k [mm2]	u_k [mm]	t_{eff} [mm]	A_{sw} [mm2]	A_{sl} [mm2]	A_{sp} [mm2]
414151	3588	178	785	4467	1400

Figure 3.55: *Data* window

3.6.3.4 Interaction

For the current section and extreme, click the navigator command **Interaction** (Interaction check of bending, normal force, shear and torsion) to execute the check of response with the interaction of shear, torsion, bending and normal force. In the *Main* graphical window the check is displayed and a detailed text representation of the check is available in the *Data* window. The ribbon groups with settings for drawings are described in chapter 3.6.6.1 on page 52.

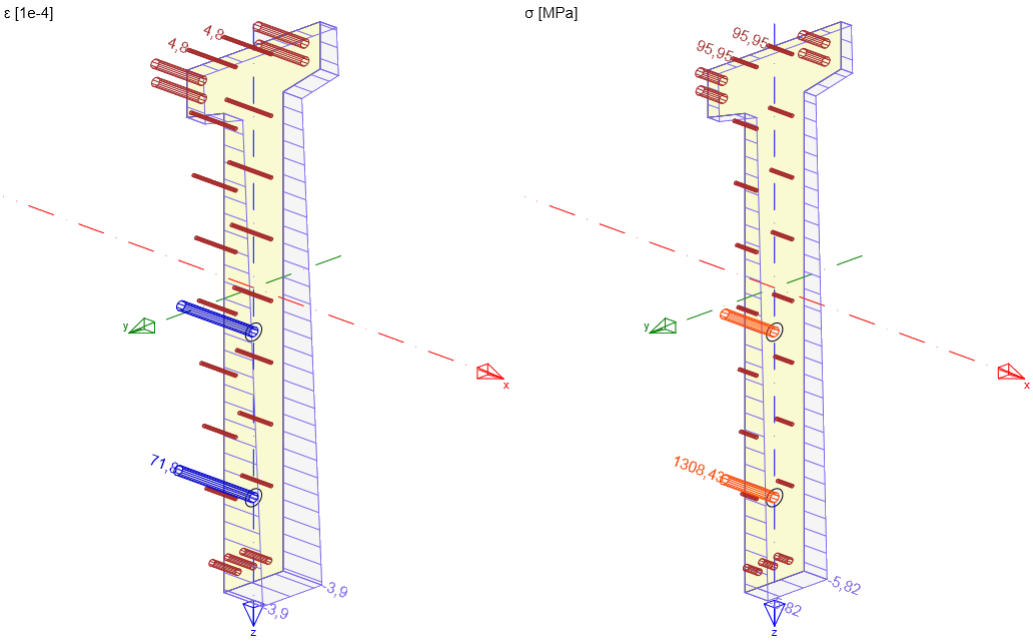


Figure 3.56: *Main* window – View 3D

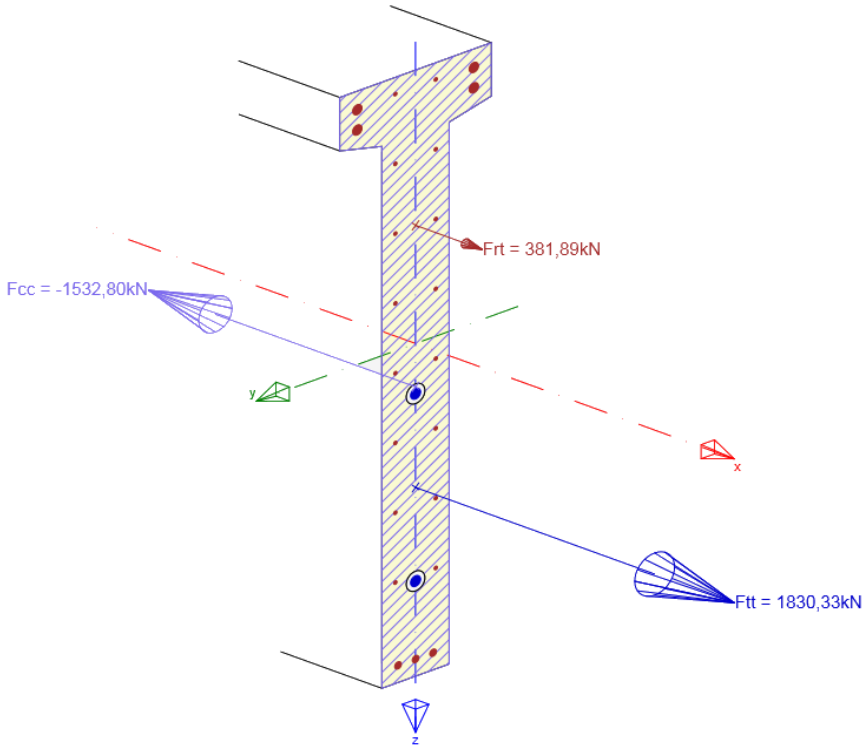


Figure 3.57: *Main* window – View 3D forces

Interaction

N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	V_{Ed} [kN]	T_{Ed} [kNm]	Value $V+T$ [%]	Value $V+T+M$ [%]	Value [%]	Limit [%]	Check
-1688,15	-163,71	0,00	578,24	0,00	80,62	86,65	86,65	100,00	OK

Interaction check of shear and torsion (concrete)

$V_{Rd,c}$ [kN]	$T_{Rd,c}$ [kNm]	$V_{Rd,max}$ [kN]	$T_{Rd,max}$ [kNm]	Eq. 6.31 [%]	Eq. 6.29 [%]	Value [%]	Limit [%]	Check
158,03	36,31	1122,89	174,68	365,91	51,50	365,91	100,00	Not OK

Interaction check of shear and torsion (longitudinal reinforcement)

A_{sl} [mm2]	F_{sl} [kN]	$F_{sl,lim}$ [kN]	Value [%]	Limit [%]	Check
3599	689,12	2805,79	24,56	100,00	OK

Interaction check of shear and torsion (shear reinforcement)

A_{sw} [mm2]	F_{sw} [kN]	$F_{sw,lim}$ [kN]	Value [%]	Limit [%]	Check
503	176,18	218,55	80,62	100,00	OK

Interaction check of shear, torsion, bending and normal force

$\Delta F_{td,s}$ [kN]	$\Delta F_{td,t}$ [kN]	ΔF_{td} [kN]	$\Delta\varepsilon_s$ [1e-4]	$\Delta\varepsilon_t$ [1e-4]	Extreme bar	in Value [%]	Limit [%]	Check
689,12	0,00	689,12	6,5	0,0	32	86,65	100,00	OK

Detailed check of reinforcement

Bar	y_i [mm]	z_i [mm]	$\Delta\varepsilon$ [1e-4]	ε [1e-4]	ε_{lim} [1e-4]	$\Delta\sigma$ [MPa]	σ [MPa]	σ_{lim} [MPa]	Value [%]	Check
7	52	885	6,5	2,7	450,0	130,19	53,19	434,78	12,23	OK
8	0	885	6,5	2,7	450,0	130,19	53,19	434,78	12,23	OK
9	-52	885	6,5	2,7	450,0	130,19	53,19	434,78	12,23	OK
13	173	-658	6,5	4,7	450,0	130,19	94,30	434,78	21,69	OK
14	-173	-658	6,5	4,7	450,0	130,19	94,30	434,78	21,69	OK
15	-173	-715	6,5	4,8	450,0	130,19	95,82	434,78	22,04	OK
16	173	-715	6,5	4,8	450,0	130,19	95,82	434,78	22,04	OK
17	-60	650	6,5	3,0	450,0	130,19	59,45	434,78	13,67	OK
18	-60	454	6,5	3,2	450,0	130,19	64,66	434,78	14,87	OK
19	-60	259	6,5	3,5	450,0	130,19	69,88	434,78	16,07	OK
20	-60	63	6,5	3,8	450,0	130,19	75,09	434,78	17,27	OK
21	-60	-133	6,5	4,0	450,0	130,19	80,31	434,78	18,47	OK
22	-60	-329	6,5	4,3	450,0	130,19	85,52	434,78	19,67	OK
23	-60	-524	6,5	4,5	450,0	130,19	90,73	434,78	20,87	OK
24	-60	-720	6,5	4,8	450,0	130,19	95,95	434,78	22,07	OK
25	60	650	6,5	3,0	450,0	130,19	59,45	434,78	13,67	OK
26	60	454	6,5	3,2	450,0	130,19	64,66	434,78	14,87	OK
27	60	259	6,5	3,5	450,0	130,19	69,88	434,78	16,07	OK
28	60	63	6,5	3,8	450,0	130,19	75,09	434,78	17,27	OK
29	60	-133	6,5	4,0	450,0	130,19	80,31	434,78	18,47	OK
30	60	-329	6,5	4,3	450,0	130,19	85,52	434,78	19,67	OK
31	60	-524	6,5	4,5	450,0	130,19	90,73	434,78	20,87	OK
32	60	-720	6,5	4,8	450,0	130,19	95,95	434,78	22,07	OK

Detailed check of prestressing reinforcement

Tendon	y_i [mm]	z_i [mm]	$\Delta\varepsilon$ [1e-4]	ε [1e-4]	ε_{lim} [1e-4]	$\Delta\varepsilon$ [MPa]	σ [MPa]	σ_{lim} [MPa]	Value [%]	Check
1	0	141	6,5	69,3	315,0	82,37	1306,33	1510,11	86,51	OK
2	0	666	6,5	71,8	315,0	35,00	1308,43	1510,11	86,65	OK

Figure 3.58: *Data* window

3.6.4 Serviceability Limit States

Select **Serviceability Limit States** from the navigator to display an overview of checks for the serviceability limit state executed in the current section. In the *Main* graphical window the check is displayed and a detailed text representation of the check is available in the *Data* window. The **Components label** ribbon group is available for this navigator command and is described in chapter 3.6.2 on page 38.

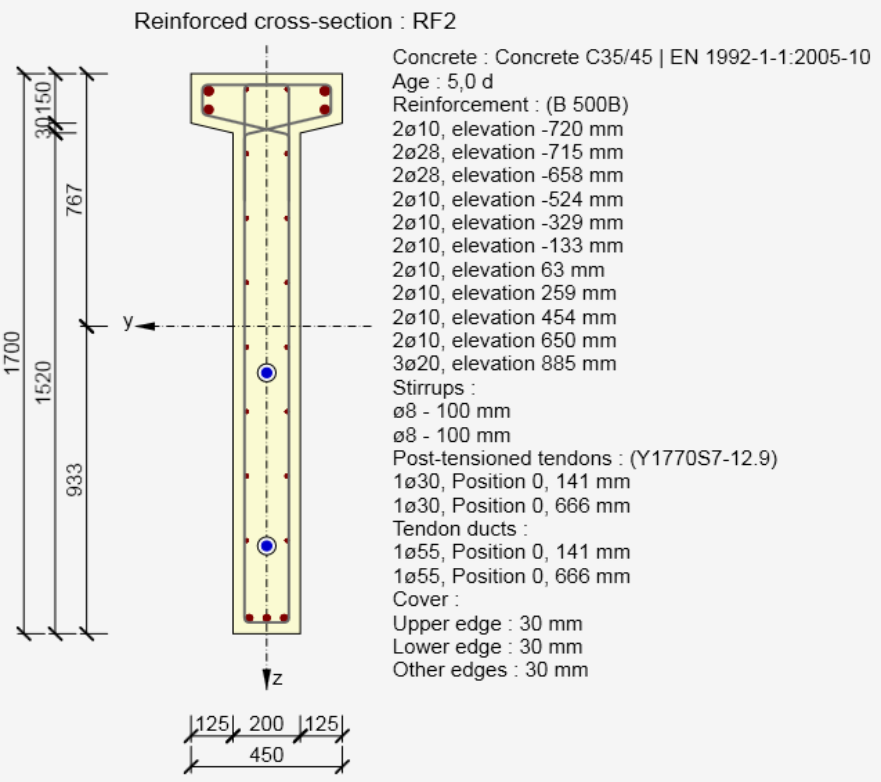


Figure 3.59: *Main* window

Serviceability Limit States

Governing type of check	N_{Ed} [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,z}$ [kNm]	Value [%]	Check
Stress Limitation	-1688,15	-305,14	0,00	91,49	OK

Type of check	N_{Ed} [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,z}$ [kNm]	Value [%]	Check
Stress Limitation	-1688,15	-305,14	0,00	91,49	OK
Crack Width	-1856,56	-417,61	0,00	0,00	OK

Figure 3.60: *Data* window

3.6.4.1 Stress Limitation

For the current section and extreme, click the navigator command **Stress limitation** to execute the stress limitation check. In the *Main* graphical window the check is displayed and a detailed text representation of the check is available in the *Data* window. The ribbon groups with settings for drawings are described in chapter 3.6.6.1 on page 52.

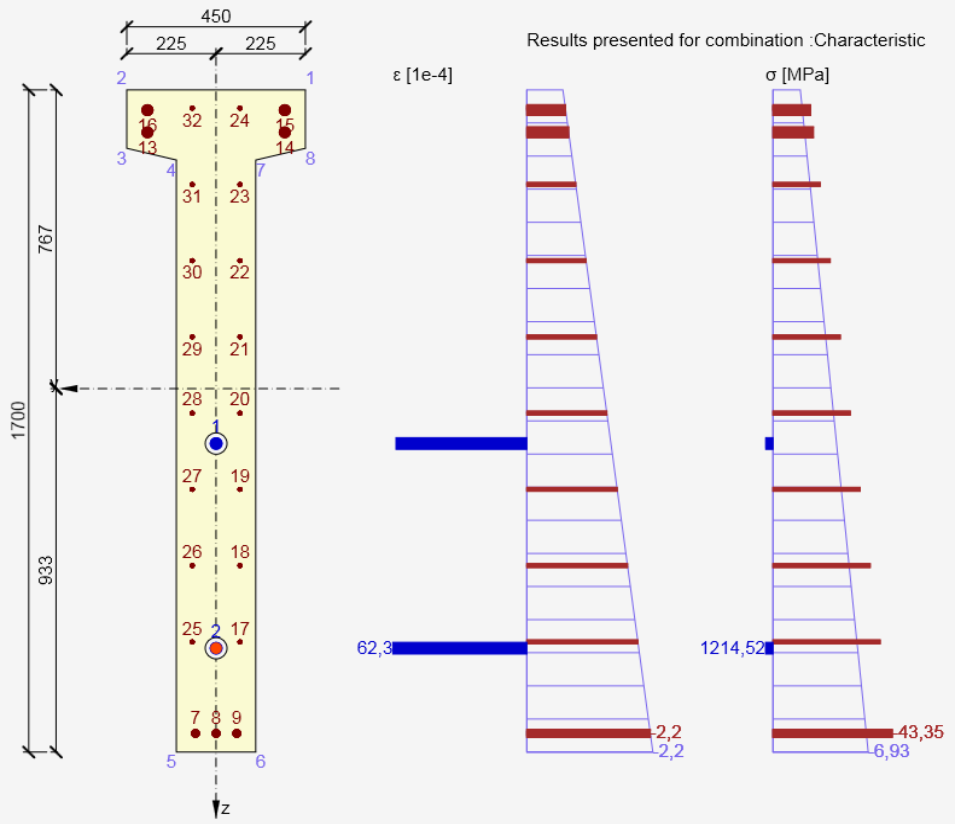


Figure 3.61: *Main* window

Stress limitation

Stress limitation

Type of check	Component type	Index	Value [%]	Limit [%]	Check
$7.2(5)-Char	Tendon	2	91,49	100,00	OK

Detailed check of concrete

Type of check	Fibre	y_i [mm]	z_i [mm]	N [kN]	M_y [kNm]	M_z [kNm]	σ [MPa]	$σ_{lim}$ [MPa]	Value [%]	Check
$7.2(3)-Quasi	5	100	933	-1688,15	-360,11	0,00	-7,37	-11,12	66,24	OK
$7.2(3)-Quasi,sup	6	-100	933	-1856,56	-428,60	0,00	-8,37	-11,12	75,26	OK
$7.2(3)-Quasi,inf	5	100	933	-1519,74	-291,62	0,00	-6,36	-11,12	57,22	OK

Figure 3.62: *Data* window

3.6.4.2 Crack Width

To execute the crack width check for the current section and extreme, select from the navigator **Crack width.** A figure is displayed in the *Main* graphical window and a detailed text representation is available in the *Data* window. The ribbon groups with settings for drawings are described in chapter 3.6.6.1 on page 52.

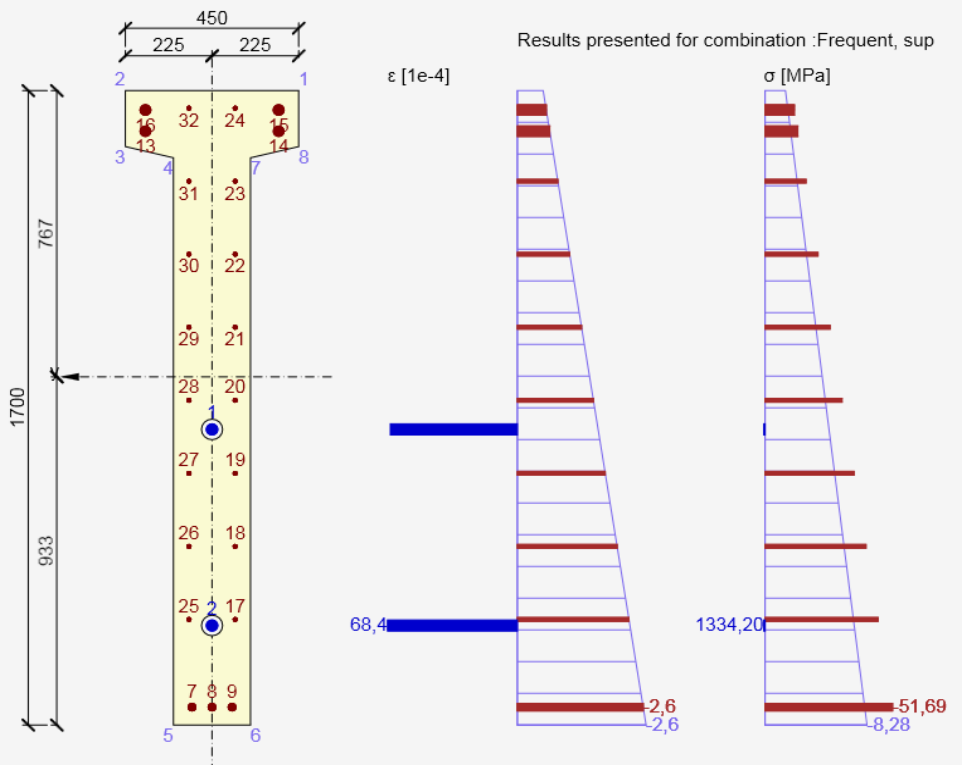


Figure 3.63: *Main* window

Crack width

N [kN]	M y [kNm]	M z [kNm]	w k [mm]	w lim [mm]	Value [%]	Limit [%]	Check
-1856,56	-417,61	0,00	0,000	0,200	0,00	100,00	OK

Result of crack width calculation for combinations including rsup, rinf (5.10.9)

Combination	N [kN]	M y [kNm]	M z [kNm]	Value calc [mm]	Value lim [mm]	Value [%]	Limit [%]	Type of check	Check
Freq,inf	-1519,74	-280,62	0,00	0,000	0,200	0,00	100,00	CW,CNA	OK
Freq,sup	-1856,56	-417,61	0,00	0,000	0,200	0,00	100,00	CW,CNA	OK

Figure 3.64: *Data* window

3.6.4.3 Brittle Failure

This menu is active only for EN 1992-2.

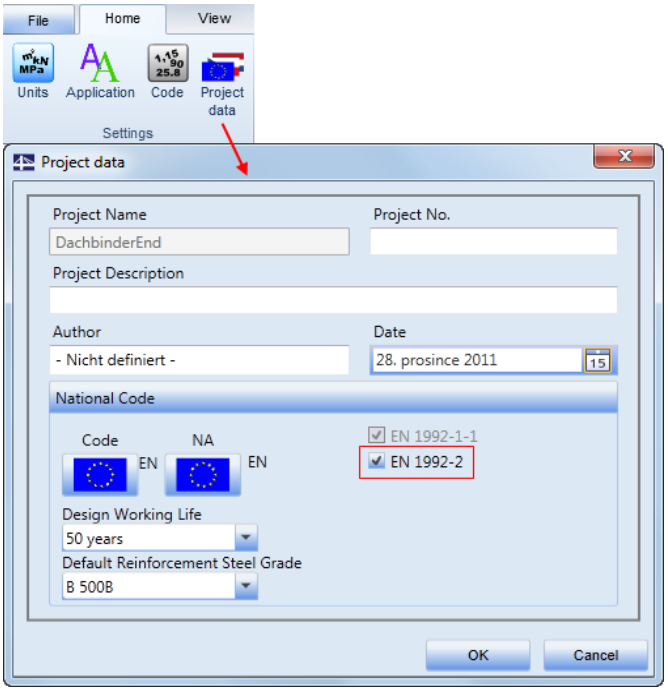


Figure 3.65: Activating of EN 1992-2

For the brittle failure check of the reinforced section, select the navigator command **Brittle failure**. A graphical figure of the reinforced cross-section is drawn in the *Main* window. In the *Data* window, a detailed the text representation of the brittle failure check is available.

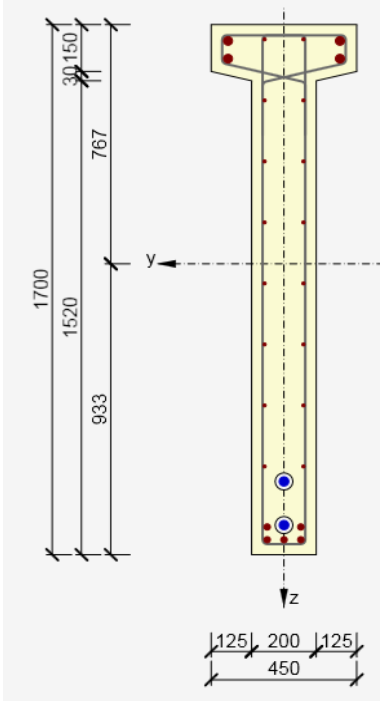


Figure 3.66: *Main* window

Brittle Failure

N Ed [kN]	M Ed,y [kNm]	M Ed,z [kNm]	Value [%]	Limit [%]	Check
-1750,99	744,68	0,00	49,22	100,00	OK

Check according to EN 1992-2, 6.1(109a)

Type	N [kN]	M y [kNm]	M z [kNm]	σ ct [MPa]	f ctm [MPa]
Forces	-1632,16	835,99	0,00		
Resistance	0,00	4246,42	0,00	2,43	2,44

Input parameters for check according to EN 1992-2, 6.1(109a)

Type	N [kN]	M y [kNm]	M z [kNm]	A p [mm2]	Reduction Factor [-]
Original	-1750,99	-1345,45	0,00	1400	
Reduced	-1632,16	-1254,13	0,00	1305	0,93

Figure 3.67: *Data* window

3.6.5 Detailing

For the current section and extreme, select the navigator command **Detailing** to execute the check of the detailing rules for the reinforced section. In the *Main* graphical window the reinforced cross-section is displayed with the geometrical, material, and overall data about the shear and longitudinal reinforcement. A detailed text representation of shear check is located in the *Data* window. The **Components label** ribbon group is available for this navigator command and is described in detail in chapter 3.6.2 on page 38.

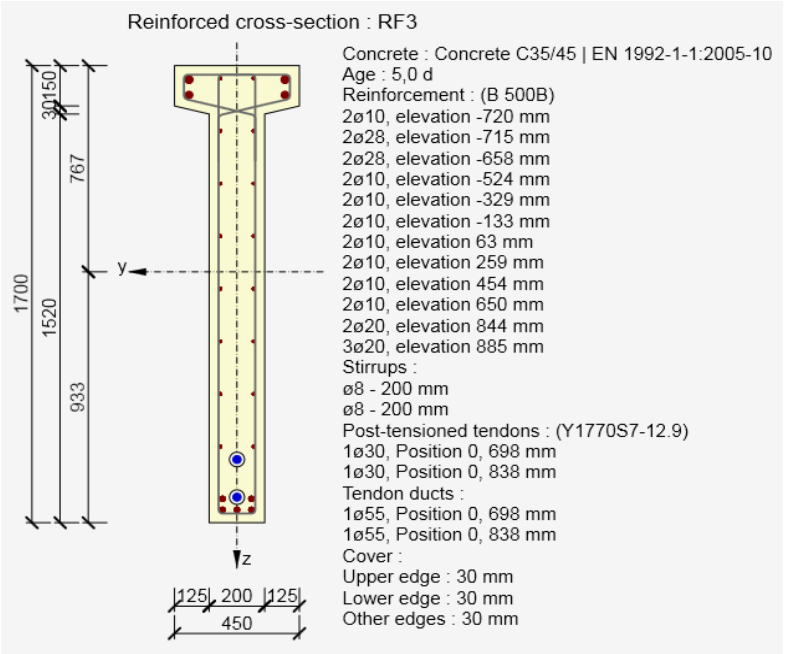


Figure 3.68: *Main* window

Detailing rules

N_{Ed} [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,z}$ [kNm]	Ratio long [%]	Ratio shear [%]	Ratio prestressing [%]	Governing [%]	Limit [%]	Check
-1750,99	2052,38	0,00	65,63	31,65	58,82	65,63	100,00	OK

Check of detailing provisions of longitudinal reinforcement

Type	Value calc	Value lim	Ratio [%]	Check
Minimal reinf. ratio for longitudinal reinforcement (9.2.1.1 (1)) [%]	0,00	0,00	0,00	Off
Maximal reinf. ratio for longitudinal reinforcement (9.2.1.1(3)) [%]	1,39	4,00	34,69	OK
Minimal clear distance of longitudinal reinforcement (8.2 (2)) [mm]	32	21	65,63	OK
Maximal axial distance of longitudinal reinforcement (9.2.3 (4)) [mm]	120	350	34,29	OK

Check detailing provisions of shear reinforcement

Type	Value calc	Value lim	Ratio [%]	Check
Minimal reinf. ratio for shear reinforcement (9.2.2 (5)) [%]	0,25	0,08	31,65	OK
Maximal reinf. ratio for shear reinforcement (6.2.3 (3)) [%]	0,25	1,17	21,40	OK
Maximal distance of stirrups (9.2.2 (6)) [mm]	200	1159	17,25	OK
Maximal transversal distance of branches of stirrups (9.2.2 (8)) [mm]	132	600	22,00	OK

Check detailing provisions of prestressing reinforcement

Type	Value calc	Value lim	Ratio [%]	Check
Minimal clear spacing of tendons (8.10.1.2 (1)) [mm]	0	0	0,00	Off
Minimal clear spacing of ducts (8.10.1.3 (3)) [mm]	85	50	58,82	OK

Input values and intermediate results for detailing

b_w [mm]	d [mm]	A_c [mm2]	f_{yk} [MPa]	f_{yd} [MPa]	f_{ck} [MPa]	f_{ctm} [MPa]	f_{cd} [MPa]
200	1546	381250	500,00	434,78	24,72	2,44	16,48

Figure 3.69: *Data* window

3.6.6 Advanced analysis

3.6.6.1 Response N-M-N

For the current section and extreme, click the navigator command *Response N-M-N* to execute the response check.

The command activates the *View, View setting, Strain, Stress, Results label, Results graph, Resultant forces, Cross-section*, and *Type of results* ribbon groups.

View

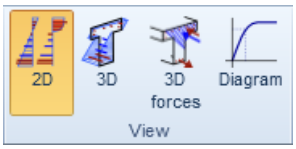


Figure 3.70: View

2D - Turns on drawing of results in 2D picture (cross-sectional areas, stress-strain diagrams)

3D - Turns on drawing of stress and strain for concrete and reinforcement in 3D view

3D forces - Turns on drawing of force resultants in 3D view

Diagram - Turns on/off drawing of check results as stress-strain diagrams, draws stress and strain in concrete fibres and reinforcement bars

> Click *Diagram* in the *View* ribbon group to launch the check using a stress-strain diagram. This option displays the stress-strain of the concrete fibres and reinforcement bars in a stress-strain diagram.
>
> Diagrams can be displayed for tendons, reinforcement bars or concrete fibres by double-clicking on the desired element in the *Main* graphical window. For the selected part, a stress-strain diagram is displayed with a label containing the position of the selected member. Positioning the mouse over the reinforcement bar or the concrete fibre in the figure displays an information tooltip with numerical values for the particular element.

View setting

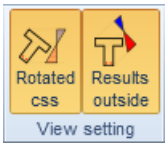


Figure 3.71: View setting

Rotated css – Turns on/off drawing of cross-section and stress and strain distributions on a rotated cross-section in the way that the neutral axis is horizontal to obtain a larger picture on the screen.

Results outside – Turns on/off drawing of a cross-section and stress and strain distributions outside the cross-section. If the stress and strain is drawn inside the cross-section, the stress and strain in concrete and in reinforcement is displayed. The stress in reinforcement bars is not displayed inside the cross-section.

Strain

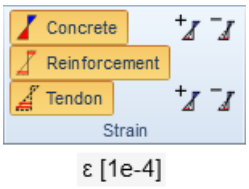


Figure 3.72: Strain

Concrete – Turns on/off drawing of concrete strain distribution

Reinforcement – Turns on/off drawing of strain in the reinforcement bars

Tendon – Turns on/off drawing of strain in the tendons

+/ - – increase / decrease the scale of strain display (the strain of concrete and reinforcement are in the same scale)

+/ - – increase / decrease the scale of tendon strain display

Stress

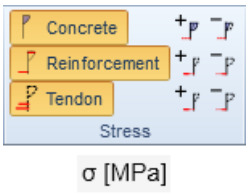


Figure 3.73: Stress

Concrete – Turns on/off display of concrete stress distribution

Reinforcement – Turns on/off display of stress in reinforcement bars

Tendon – Turns on/off display of stress in tendons

+ / - – increase / decrease the scale of the concrete stress drawing (the stress of concrete and reinforcement bars are in different scales)

+ / - – increase / decrease the scale of the stress drawing in the reinforcement bars

+ / - – increase / decrease the scale of the stress drawing in the tendons

Results label

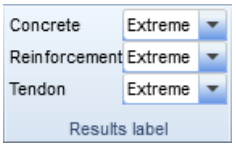


Figure 3.74: Results label

Concrete – Select the labeling style for concrete

No label – Turns off all the labels

Extreme – Labels the minimum and maximum strain/stress values in concrete

All – Labels the strain/stress values in all concrete fibres

Reinforcement – Select the labeling style of reinforcement bar results

No label – Turns off all the labels in the reinforcement bars

Extreme – Labels the minimum and maximum strain/stress values in the reinforcement bars

All – Labels the strain/stress values in all the reinforcement bars

Tendon – Select the labeling style of tendon results

No label – Turns off all the labels in tendons

Extreme – Labels the minimum and maximum strain/stress values in tendons

All – Labels the strain/stress values in all tendons

Results graph

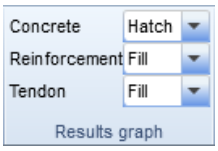


Figure 3.75: Results graph

Concrete – Select the graph style for concrete results

Line – Draws the outline of the graph of stress and strain in concrete

Hatch – Draws a hatched graph of stress and strain in concrete

Fill – Draws a graph of stress and strain in concrete filled with color

Reinforcement – Select the graph style for results in reinforcement

Line – Draws the reinforcement bars in the graph of stress and strain as lines

Outline – Draws the outline of reinforcement bars in the graph of stress and strain

Fill – Draws the filled outline of reinforcement bars in the graph of stress and strain

Tendon – Select the graph style for results in tendons

Line – Draws the tendons in the graph of stress and strain as lines

Outline – Draws the outline of tendons in the graph of stress and strain

Fill – Draws the filled outline of tendons in the graph of stress and strain

Resultant forces

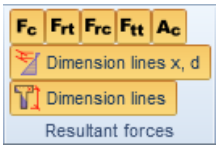


Figure 3.76: Resultant forces

F_c – Shows/hides the position of the resultant of the concrete compression zone

F_{rt} – Shows/hides the position of the resultant of the tensile reinforcement

F_{rc} – Shows/hides the position of the resultant of the reinforcement in compression

A_c – Shows/hides the hatching of the concrete compression zone

Dimension lines x, d – Shows/hides the dimension lines for the depth of the compression zone and the effective depth of cross-section

Dimension lines of results positions – Shows/hides the dimension lines for the positions of resultant forces

Cross-section

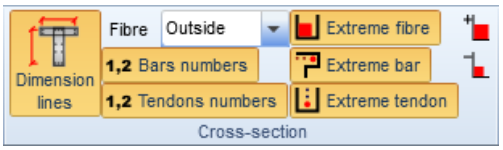


Figure 3.77: Cross-section

Dimension lines – Shows/hides the dimension lines for the cross-section geometry and the center of gravity

Fibre – The drop-down list provides three options to draw the fibre labels:

Outside the outline of the cross-section

Inside the outline of the cross-section

No label

Bars numbers – Shows/hides the bar numbers

Tendons numbers – Shows/hides the tendon numbers

Extreme fibre – Highlights the extremely loaded fibres

Extreme bar – Highlights the extremely loaded reinforcement bars

Extreme tendon – Highlights the extremely loaded tendons

+/ - – makes the concrete fibre and extreme reinforcement/tendon labels larger or smaller

Type of results

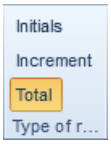


Figure 3.78: Type of results

Initials – Turns on/off drawing of diagrams of stress and strain caused by prestressing and permanent component of load effects

Increment – Turns on/off drawing of diagrams of stress and strain caused by effects of variable loads in the current extreme

Total – Turns on/off drawing of courses of stress and strain caused by all effects (initials and variable loads) in the current extreme

Figure 3.79: Label extremes

Fibre – Shows/hides the stress and strain values at the most loaded fibres

Bar – Shows/hides the stress and strain values at the most loaded reinforcement bar

Tendon – Shows/hides the stress and strain values at the most loaded tendon

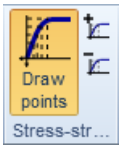


Figure 3.80: Draw points

Draw points – Shows/hides the values of points on parabolic part of stress-strain interaction

+ / - – increase / decrease of points on parabolic part of stress-strain relationship

In the *Main* window, the graphical representation of the checks is drawn. In the *Data* window the text representation is displayed.

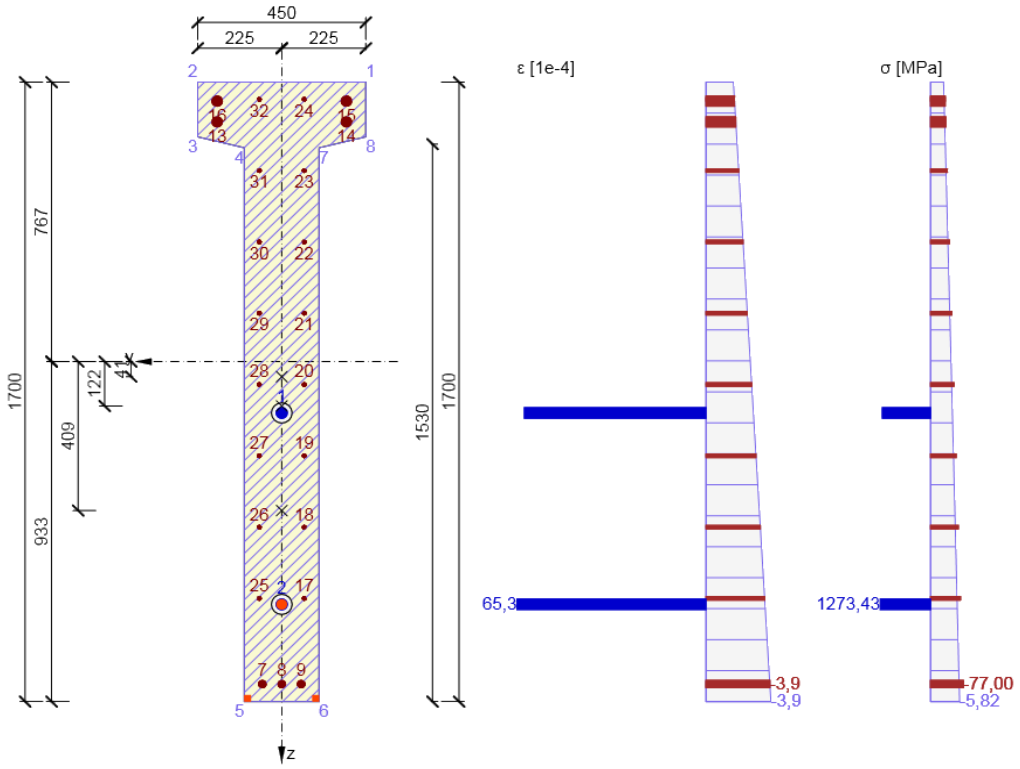


Figure 3.81: *Main* window – View 2D

Response N-M-M

N Ed [kN]	M Ed,y [kNm]	M Ed,z [kNm]	Extreme in fibre	Extreme in bar	Extreme in tendon	Value [%]	Limit [%]	Check
-1688,15	-163,71	0,00	5	7	2	84,33	100,00	OK

Plane of Strain

x [mm]	d [mm]	z [mm]	ε x [1e-4]	ε y [1e-4]	ε z [1e-4]
0	1530	1377	1,7	0,0	2,2

Forces in components of cross-section

Component of css	N [kN]	M y [kNm]	M z [kNm]	A [mm2]	y i [mm]	z i [mm]
Concrete	-1532,80	-186,61	0,00	0	0	122
Reinforcement in compression	-225,07	-9,30	0,00	4662	0	41
Tendons	1748,17	714,58	0,00	1400	0	409
Total	-9,71	518,66	0,00			

Detailed check of concrete

Fibre	y i [mm]	z i [mm]	ε [1e-4]	ε lim [1e-4]	σ [MPa]	σ lim [MPa]	Value [%]	Check
1	-225	-767	-1,6	-35,0	-2,60	-16,48	15,76	OK
abbreviated								
9	-225	-767	-1,6	-35,0	-2,60	-16,48	15,76	OK

Detailed check of reinforcement

Bar	y i [mm]	z i [mm]	ε [1e-4]	ε lim [1e-4]	σ [MPa]	σ lim [MPa]	Value [%]	Check
7	52	885	-3,9	-450,0	-77,00	-434,78	17,71	OK
abbreviated								
32	60	-720	-1,7	-450,0	-34,24	-434,78	7,88	OK

Detailed check of prestressing reinforcement

Tendon	y i [mm]	z i [mm]	ε [1e-4]	ε lim [1e-4]	σ [MPa]	σ lim [MPa]	Value [%]	Check
1	0	141	62,8	315,0	1223,95	1510,11	81,05	OK
2	0	666	65,3	315,0	1273,43	1510,11	84,33	OK

Figure 3.82: *Data* window (abbreviated)

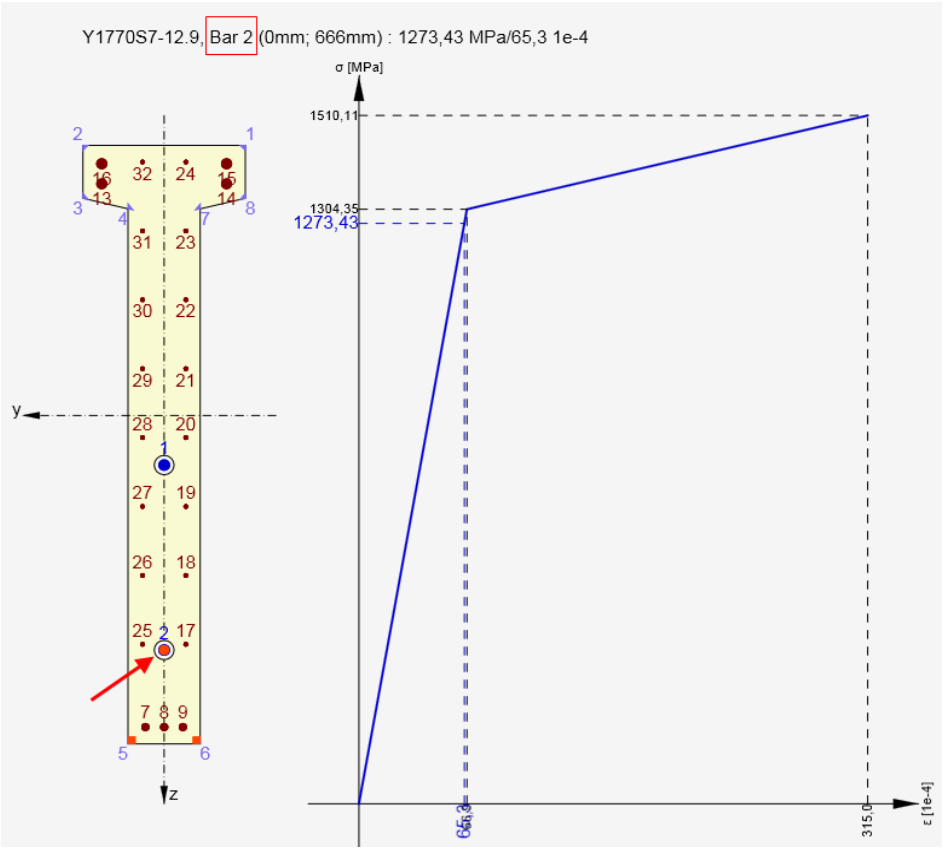


Figure 3.83: Diagram – Tendon 2

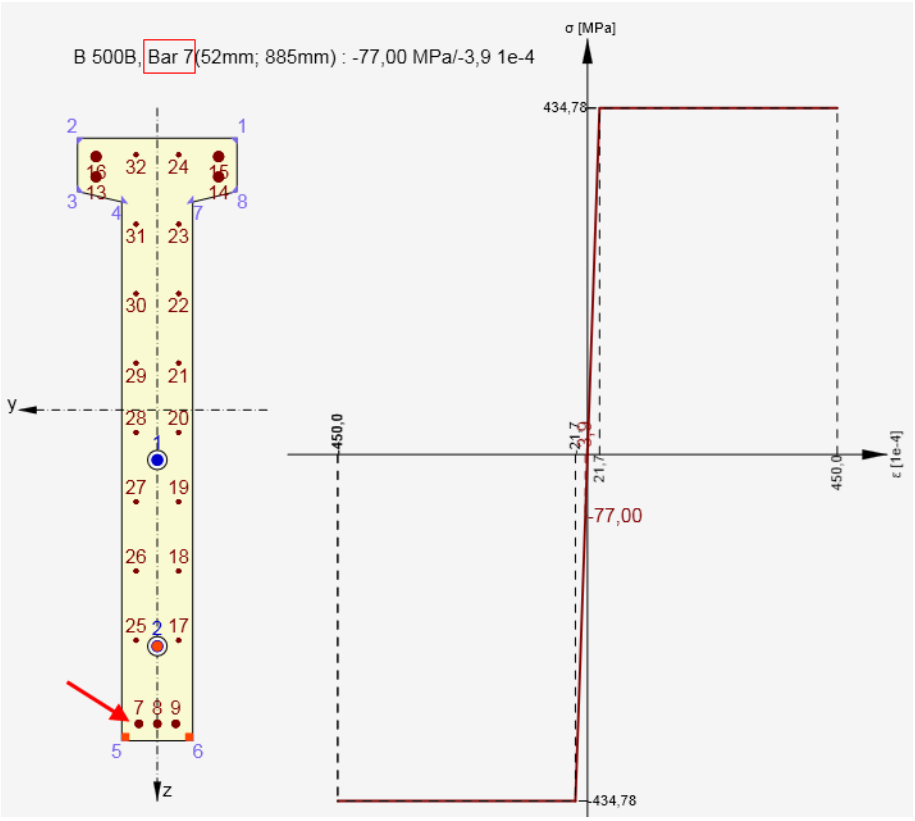


Figure 3.84: Diagram – Reinforcement bar 7

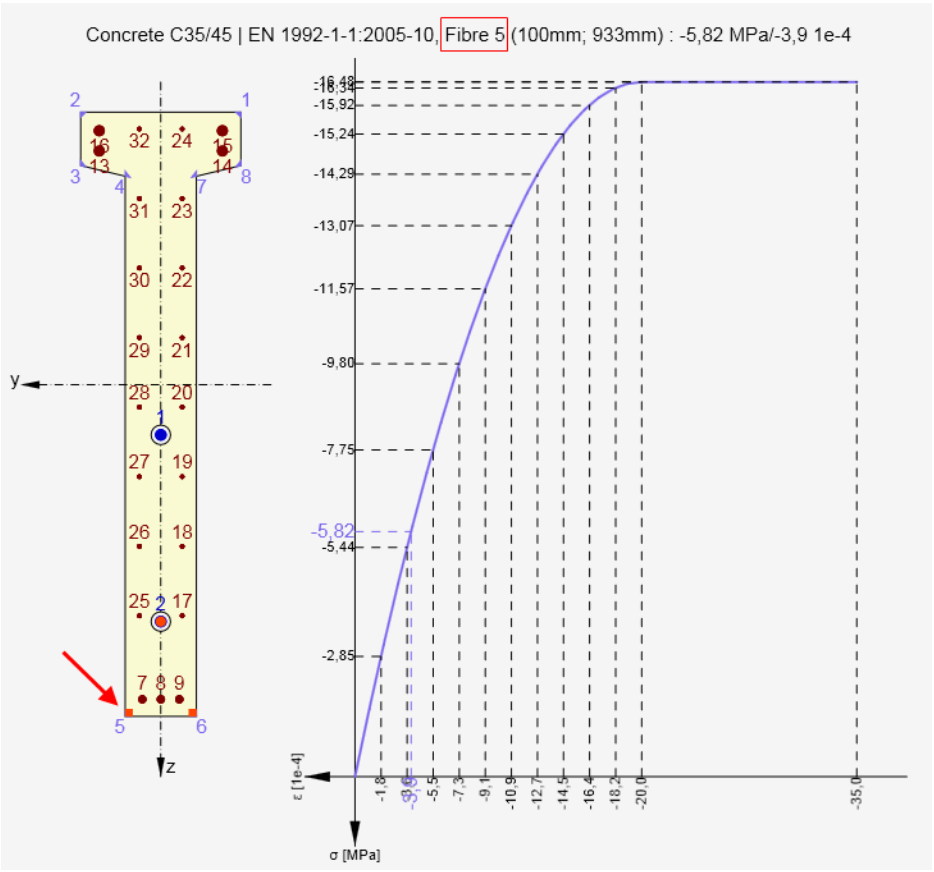


Figure 3.85: Diagram – Concrete fibre 5

3.6.6.2 Stiffnesses

For the current section and extreme, select the navigator command **Stiffnesses** to execute the short and long term stiffnesses calculation of the cross-section. In the *Main* graphical window the check is displayed and a detailed text representation of the check is available in the *Data* window. The ribbon groups with settings for drawings are described in chapter 3.6.6.1 on page 52.

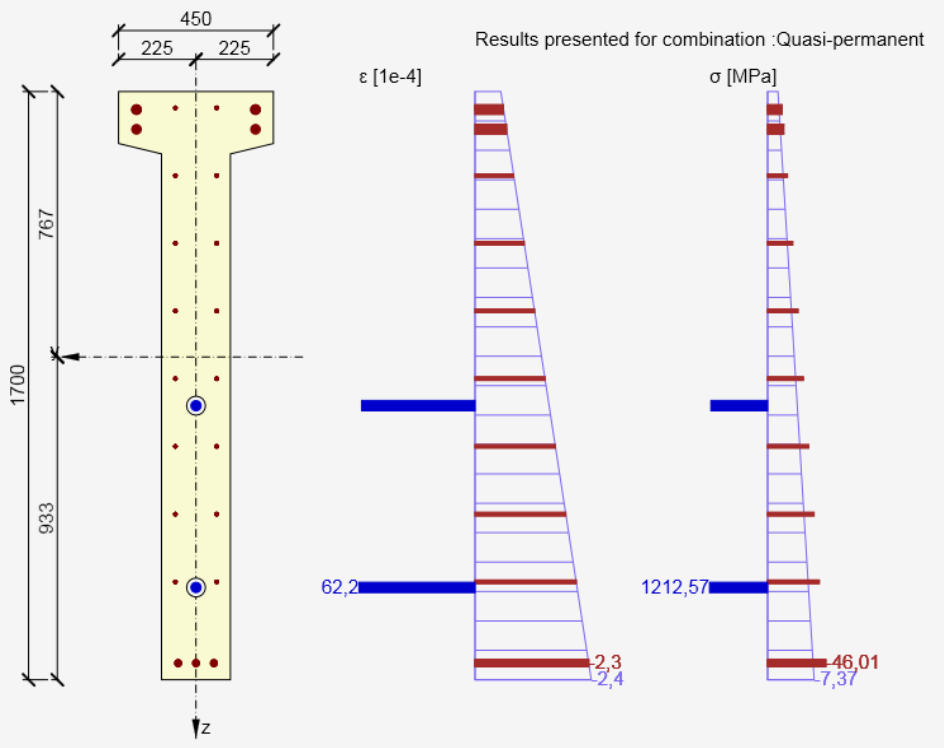


Figure 3.86: *Main* window

Stiffness

Type	N [kN]	M y [kNm]	M z [kNm]	El y [MNm2]	El z [MNm2]	EA x [MN]
Result	-1688,15	-360,11	0,00	3705	86	12999
Type	**Nr [kN]**	**M yr [kNm]**	**M zr [kNm]**	**El y [MNm2]**	**El z [MNm2]**	**EA x [MN]**
Uncracked cross-section	9,25	-307,17	0,00	3705	86	12999

Intermediate results of stiffness calculation

A s [mm2]	A st [mm2]	A sc [mm2]	ζ [-]	β [-]	σ sr [MPa]	σ ss [MPa]
10814	0	4662	0,00	1,00	0,00	0,00

Cross-section characteristics

Type	A [mm2]	S y [mm3]	S z [mm3]	I y [mm4]	I z [mm4]	t y [mm]	t z [mm]	x [mm]
Uncracked cross-section	414981	-4151224	0	118314795291	2761312742	0	-10	0

Figure 3.87: *Data* window

3.7 Report

Select the navigator command **Report** to create a new report.

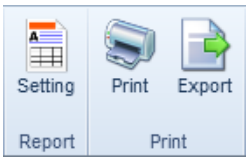


Figure 3.88: Report

Setting

Shows the report settings dialog box for the selection of sections to be printed and the selection of chapters to be printed for each section. You can make a detailed print setting for each section independently. The settings are described in detail in chapter 3.1.1 on page 15.

Print

Print the current report

Export

Export document to an RTF file

3.7.1 Settings

The report settings of the current section are available after clicking on **Setting** in the navigator.

Figure 3.89: Settings

3.7.2 Standard

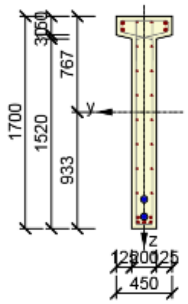
From the navigator, select **Standard** to create a standard report for the current section. Standard reports contain only tables with the overall check results of the selected check types. The following shows a complete standard report.

1. Sectional checks

1.1. Section Bemessungsträger 1 - 12,83m (1 - 11,83m)

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

Design Member	Bemessungsträger 1
Reinforced Cross-Section	RF3
Check at the age of concrete of	5,0 d



Concrete : Concrete C35/45 | EN 1992-1-1:2005-10
Age : 5,0 d
Reinforcement : (B 500B)
2ø10, elevation -720 mm
2ø28, elevation -715 mm
2ø28, elevation -658 mm
2ø10, elevation -524 mm
2ø10, elevation -329 mm
2ø10, elevation -133 mm
2ø10, elevation 63 mm
2ø10, elevation 259 mm
2ø10, elevation 454 mm
2ø10, elevation 650 mm
2ø20, elevation 844 mm
3ø20, elevation 885 mm
Stirrups :
ø8 - 200 mm
ø8 - 200 mm
Post-tensioned tendons : (Y1770S7-12.9)
1ø30, Position 0, 698 mm
1ø30, Position 0, 838 mm

1.1.1.1. Overall

Governing type of check	N_{Ed} [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,z}$ [kNm]	V_{Ed} [kN]	T_{Ed} [kNm]	Value [%]	Check
Stress Limitation	-1750,99	1119,41	0,00			98,78	OK

Type of check	N_{Ed} [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,z}$ [kNm]	V_{Ed} [kN]	T_{Ed} [kNm]	Value [%]	Check
Capacity N-M-M	12,72	3407,41	0,00			83,34	OK
Response N-M-M	-1750,99	2052,38	0,00			89,29	OK
Shear	-1750,99			-19,69		8,92	OK
Torsion					0,00	0,00	OK
Interaction	-1750,99	2052,38	0,00	-19,69	0,00	89,29	OK
Stress Limitation	-1750,99	1119,41	0,00			98,78	OK
Crack Width	-1575,89	879,22	0,00			4,29	OK
Detailing	-1750,99	2052,38	0,00			65,63	OK

Figure 3.90: Standard report

3.7.3 Detailed

Select the navigator command **Detailed** to create a detailed report for the current section. The following shows the table of contents for a detailed report.

Table of contents

Figure 3.91: Detailed report

A Text Format *.NAV

A .NAV file is used to export and import data. It includes XML tags for the defined groups of data. A file in the NAV format enables you to export the whole reinforced cross-section (outline, openings, longitudinal reinforcement, stirrups, tendons and tendon ducts) at once and also import reinforcement which was entered by the module RF-TENDON Design.

The following tags are used:

<ReinforcedCss> </ReinforcedCss> - begin and end tag for reinforced section.

It can include the tags <Css>, <Bars>, <Stirrups>, <Tendons> and <TendonDucts>.

<Css> </Css> - begin and end tag for the definition of a cross-section shape.
It contains the tags <Component> and <Opening>.

<Component> </Component> - begin and end tag for the definition of one cross-section component. The content includes lines with vertex coordinates for the cross-section shape.

<Opening> </Opening> - begin and end tag for one opening in the cross-section. The content includes lines with vertex coordinates for the opening shape.

<Bars> </Bars> - begin and end tag for the definition of the longitudinal reinforcement. The content includes lines with the same reinforcement bars description as defined in the TXT file.

<Stirrups> </Stirrups> - begin and end tag for the definition of one stirrup.
It contains the tags <DataStirrup> and <GeometryStirrup>.

<DataStirrup> </DataStirrup> - it contains the lines with the same general stirrup parameters as defined in the TXT file.

<GeometryStirrup></GeometryStirrup> - it contains the lines with the same vertex coordinates as defined in the TXT file.

<Tendons> </Tendons> - begin and end tag for definition of prestressing tendons.
It contains tags <TendonsInLine> and </TendonsOnCssEdge>.

<TendonsInLine></TendonsInLine> - contains lines with the same tendons defined by coordinates description as defined in the TXT file.

<TendonsOnCssEdge></ TendonsOnCssEdge > - contains lines with the same tendons at cross-section edge description as defined in the TXT file.

Example

An example of a complete reinforced cross-section exported to a .NAV file is as follows:

```
<ReinforcedCss>
	<Css>
		<Component>
			-150 -250
			150 -250
			150 250
			-150 250
			-150 -250
		</Component>
		<Opening>
			-50 -50
			50 -50
			50 50
			-50 50
			-50 -50
		</Opening>
	</Css>
	<Bars>
		2 16 102 202 -102 202
		2 16 -102 -202 102 -202
	</Bars>

	<Stirrups>
		<Stirrup>
			<DataStirrup>
				10 200 1 1.30
			</DataStirrup>
			<GeometryStirrup>
				-115 215
				-115 -215
				115 -215
				115 215
				- 115 215
			</GeometryStirrup>
		</Stirrup>
	</Stirrups>

	<Tendons>
		<TendonsInLine>
			2 6 1 0.0 0.0 -110 210 110 210 1 33
		</TendonsInLine>
		<TendonsOnCssEdge>
			2 6 1 0.0 0.0 1 1 30 30 30 1 33
		</TendonsOnCssEdge>
	</Tendons>

	<TendonDucts>
		<TendonDuctsOnCssEdge>
			2 1.7 1 2 80 30 30
		</TendonDuctsOnCssEdge>
	</TendonDucts>
</ReinforcedCss>
```

Explanation

To export a cross-section shape, one vertex of outline is defined on each line in the text file. The coordinates y and z are separated by a space.

Export of a rectangle cross-section is:

```
-150 -250
150 -250
150 250
-150 250
-150 -250
```

To export an opening, one vertex of opening is defined at each line in the text file. The coordinates y and z are separated by a space.

Export of a rectangle hole:

```
-50 -50
50 -50
50 50
-50 50
-50 -50
```

To export a longitudinal reinforcement layout, one layer is defined at each line in the text file. It is required that the parameters must be defined in the following order: numbers of bars, bar diameter, begin Y coordinate, end Y coordinate, begin Z coordinate, end Z coordinate.

Export of two bar layers is:

```
2 16 352 252 -352 252
2 16 -352 -252 352 -252
```

To export a stirrups layout there is one stirrup layout definition at each line in the text file. It is required that the parameters must be defined in following order: stirrup diameter, the distance between two adjacent stirrups, take into account the torsion check (0=no,1=yes), the radius of mandrel (multiple of stirrup diameter).

Export of one stirrup is as follows:

```
10 300 1 1.30
-365 265
-365 -265
365 -265
365 265
-365 265
```

To export a tendons layer defined by coordinates of first and last tendon in layer, one layer is defined at each line in the text file. It is required that the parameters must be defined in the following order: number of tendons in layer, number of strands in tendon, 1, vertical slope of tendon, horizontal slope of tendon, begin Y, begin Z, end Y, end Z, pre/post-tensioned (1=post-tensioned, 0=pre-tensioned), duct diameter, duct material (0=metal, 1=plastic)

Export of one layer of tendons defined by coordinates:

```
2 6 1 0.0 0.0 -120 -190 120 -190 1 33 0
```

To export a tendons layer defined at cross-section edge, one layer is defined at each line in the text file. It is required that the parameters must be defined in the following order: number of tendons in layer, number of strands in tendon, 1, vertical slope of tendon, horizontal slope of tendon, 1, number of edge, edge cover, left cover, right cover, pre/post-tensioned (1=post-tensioned, 0=pre-tensioned), duct diameter, duct material (0=metal, 1=plastic)

Export of one layer of tendons at cross-section edge

```
2 6 1 0.0 0.0 1 1 30 30 30 1 33 0
```

To export a ducts layer defined by coordinates of first and last duct in layer, one layer is defined at each line in the text file. It is required that the parameters must be defined in the following order: number of ducts in layer, duct diameter, begin Y, begin Z, end Y, end Z, duct material (0=metal,1=plastic)

Export of one layer of ducts defined by coordinates:

```
2 33 -120 -220 120 -220 0
```

To export a ducts layer defined at cross-section edge, one layer is defined at each line in the text file. It is required that the parameters must be defined in the following order: number of ducts in layer, ducts diameter, number of edge, edge cover, left cover, right cover, duct material (0=metal,1=plastic)

Export of one layer of ducts at cross-section edge

```
2 33 1 4 30 30 30 0
```

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, Germany, 2011

C Index