IDEA Tendon 8

User guide

Content

1 Getting started

1.1 System requirements

Application requires .NET Framework 4.5 to be installed on the computer. It can be downloaded from web pages of Microsoft Company (https://www.microsoft.com/en-US/download/details.aspx?id=30653).

In case of a missing .NET Framework the installation is not launched.

1.2 Installation

IDEA Tendon program is installed as a part of IDEA StatiCa package.

2 Introduction

IDEA Tendon and IDEA RCS are external add-on modules of AxisVM, IDEA Beam or IDEA Frame programs, which enable the user to design pre-tensioned and post-tensioned prestressed concrete beams according to EN 1992-1-1 and EN 1992-2 codes.

The precondition is that a project in AxisVM, IDEA Beam or IDEA Frame (superior linked application) has been input. The structure can contain 1D and 2D concrete members, cross-sections and materials, external loads, load cases including load cases for pre-tensioning and post-tensioning, and load groups. Once IDEA Tendon is started, the user selects 1D concrete members to be prestressed. Then he is navigated by going through individual design steps:

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

IDEA RCS is an effective tool to perform the design pre-stressed concrete sections according to EN codes, based on the results calculated in IDEA Tendon and AxisVM, IDEA Beam or IDEA Frame. The following features are provided:

- the evaluation of extreme internal forces based on selected strategy,
- comfortable automatic or manual input of additional non-prestressed reinforcement,
- calculation of short and long-term losses of prestressing (due to elastic deformation of concrete, steel relaxation, creep and shrinkage of concrete,
- design for axial force, biaxial bending, shear, torsion and combined internal forces,
- ultimate and serviceability limit state design for relevant design situations
- detailed results documentation with reference to design equations used and described in the standard,
- descriptive graphics in printout report.

2.1 Limitations

- The structure does not change its structural system during construction stages. Structural analysis is performed with one structural model only - all tendons are assumed to be prestressed in one moment.
- No external load is applied to prestressed part of the structure before it is prestressed, external load or self-weight can be applied at the same time as prestressing.
- Prestressed beam makes one (integral) structural system or part of such system (not a set of independent members) at the stage of the structure, for which the design of tendon is performed. Examples: one structural system = simply supported beam or continuous beam, part of structural system = primary beam of portal frame.
- Cross-section of concrete 1D members is solid (not composite) and it is cast in one construction stage.
- Pre-tensioned tendons can be defined only for straight and statically determined design members.

3 Terminology

3.1 General

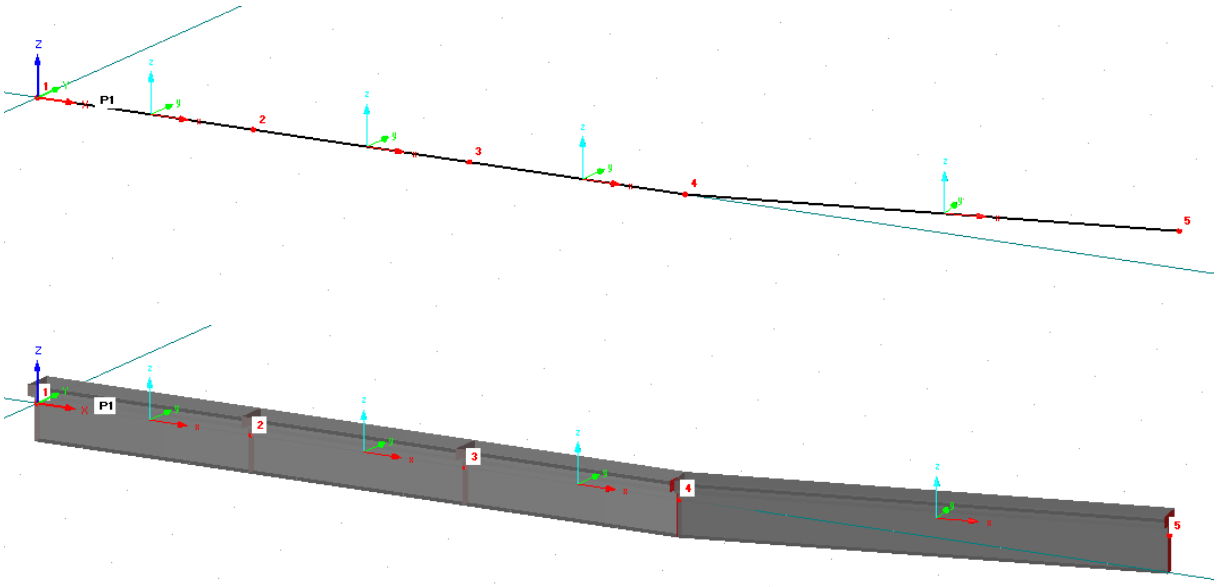
Part of Member – is a basic entity which is imported from structural model. It is not a finite element. Each part of member is linked to one basic geometrical entity (line, circle arc, parabolic arc). This geometrical entity contains definition of its local coordinate system (LCS). In relation to the geometrical entity the part of member can be defined eccentric with different eccentricities at the beginning and at the end and rotated with constant rotation along the part of member.

Reference curve – it is defined as a union of basic geometrical entities of parts of members. Reference curve goes through nodes of structural model. If the basic geometrical entity is a straight line, the reference curve is defined as a connecting line of the nodes, see below.

Nodes of structural model – points, to which the position of part of member is defined. Part of member can be eccentric to the beginning and end node.

Example:

Member P1 is defined by polygon in FEM program (superior linked application). Polygon is defined by five points 1 to 5 and it consists of four segments. Member in IDEA Tendon will consist of four Parts of member.



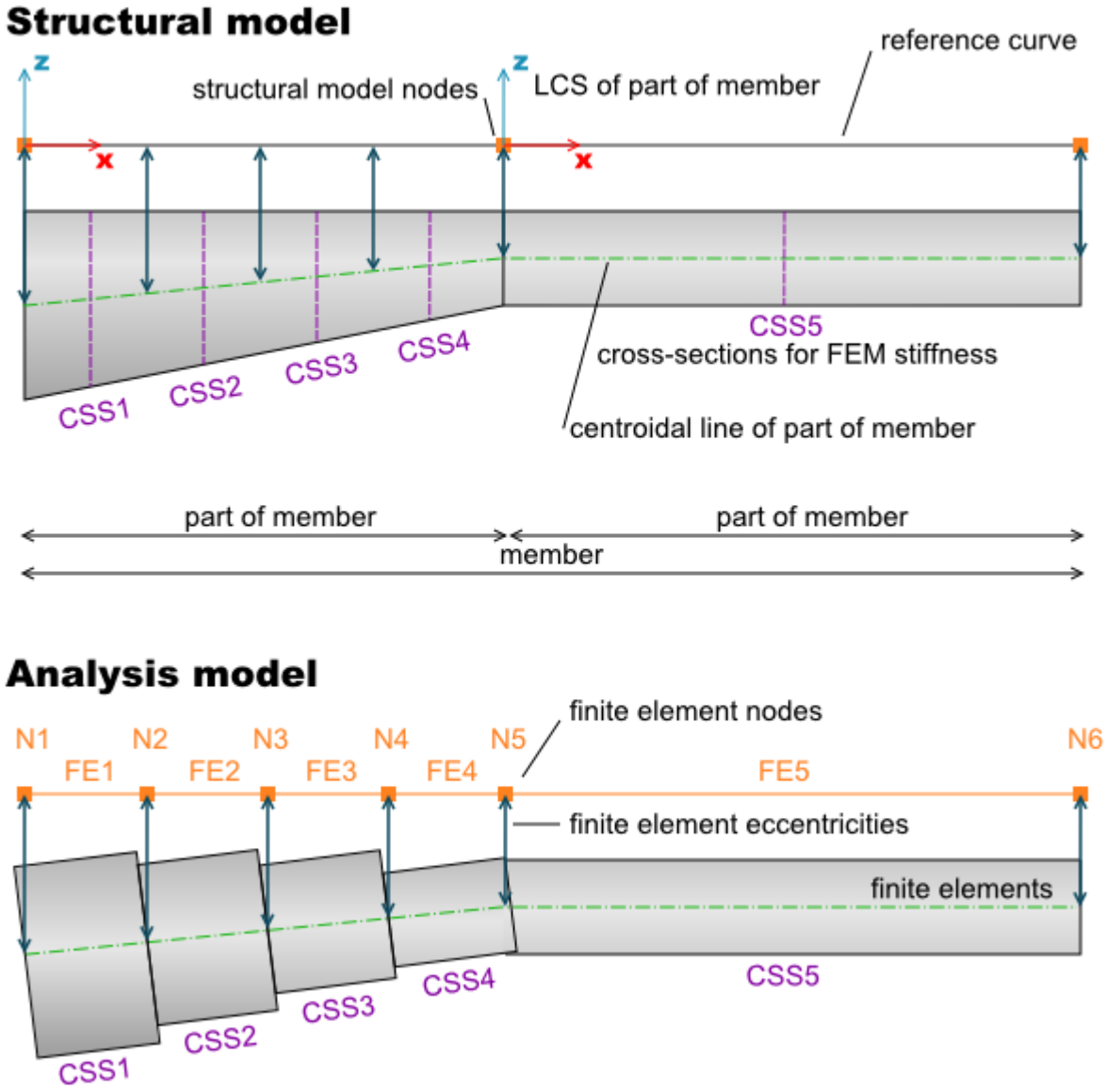
Local coordinate system of part of member is defined as follows:

- x-axis is defined depending on basic geometrical entity of part of member (straight line, circle, parabolic arc) as a vector identical with tangent in any point of part of member and with orientation identical with geometrical entity.

- According to settings direction of y-axis or z-axis is defined. E.g. z-axis of LCS is parallel to Z-axis of global coordinate system or z-axis is defined by vector. The third axis is calculated to be perpendicular to those two axes.
- Coordinate system is right-handed.

Identical LCS – two local coordinate systems are identical, if both of them have the beginning in the same point and angle between corresponding axes is zero.

Member – is 1D element of structural model, which consist of at least one **Part of Member**. If member consist of more parts of member, all parts of member are connected in a row, it means that ending point of one part of member is also beginning point of following member. Local coordinate systems of particular parts of member in this point may (but not must) be identical.

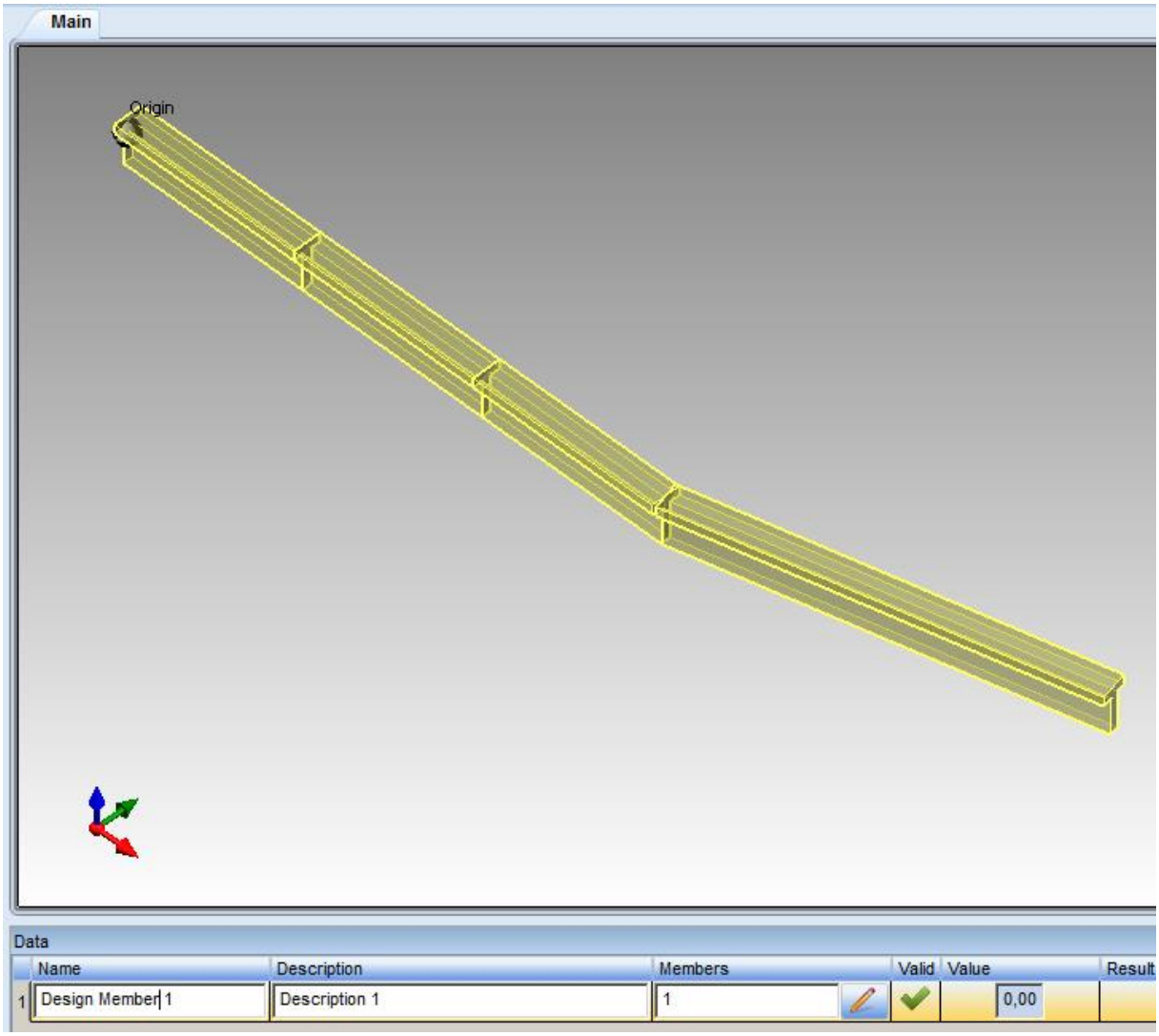


Design Member – one or group of consecutive structural model members. Consecutive members must have common node of structural model and must have the same orientation –

ending point of ne member is beginning point of following member. Design member is analysed as the whole and prestressing reinforcement is designed on Design Member.

Remark.: in IDEA Designer: ***Design Member*** *– representative of design group.*

Example:



Member P1 has been exported from FEM program (superior linked application) to IDEA Tendon. **Design Member 1** has been created, which consist of one **Member** (1). Member consists of four **Parts of member**.

Coordinate system of member – it is right-handed Cartesian coordinate system, which is taken from superior application. Coordinate system of member consists of coordinate systems of particular parts of member.

Coordinate system of Design member – design member does not have its own coordinate system. Geometry of design member is defined by sequence of coordinate systems of consecutive members of design members.

Uncoiling of Reference Curve – is performed sequentially for particular members of design member. Uncoiling begins with the second member in the order, separately for XZ-plane and for-XY plane. For example uncoiling to XZ plane consists of following steps:

- A line is created, which is parallel to z-axis of coordinate system of the first member in design member and which leads through node, in which the member being uncoiled neighbours with the uncoiled part of design member.
- A surface (in case of curve) or plane (in case of polygon) interleaving this straight line and the local x-axis of member being uncoiled, is generated.
- The surface is uncoiled /plane rotated (including the member of design member and tendons allocated to it) to be parallel with XZ plane of coordinate system of the first member in design member
- In similar way local x-axis of the member being uncoiled and all corresponding entities are uncoiled in XY plane (the reference curve is straightened).
- The coordinate system of the member being uncoiled (and all corresponding entities) is rotated about x-axis to be identical with the local coordinate system of the first member in design member.
- Eventual translation due to eccentricity of member in Y-direction is not performed.

Uncoiled view (of member, tendon, design member) – is obtained by uncoiling of the reference curve/polygon

Example:

Uncoiled Design member in planes XY and XZ

Uncoiled view coordinate system – it is the coordinate system of the first member in design member.

3.2 Tendon geometry

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

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

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

Stand-alone segment – type of segment, which cannot be join to other segment.

Closing segment - type of segment, which can be used at the beginning or ending of tendon. It is followed by internal segment or other end segment.

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

Editing point - point used to change segment parameters.

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

Intermediate point – editing point inside the segment.

Connecting point – point at the connection of two segments.

Characteristic points of tendon segment – editing points determining tendon segment geometry. Tendon segment contains two or three points depending on tendon segment shape.

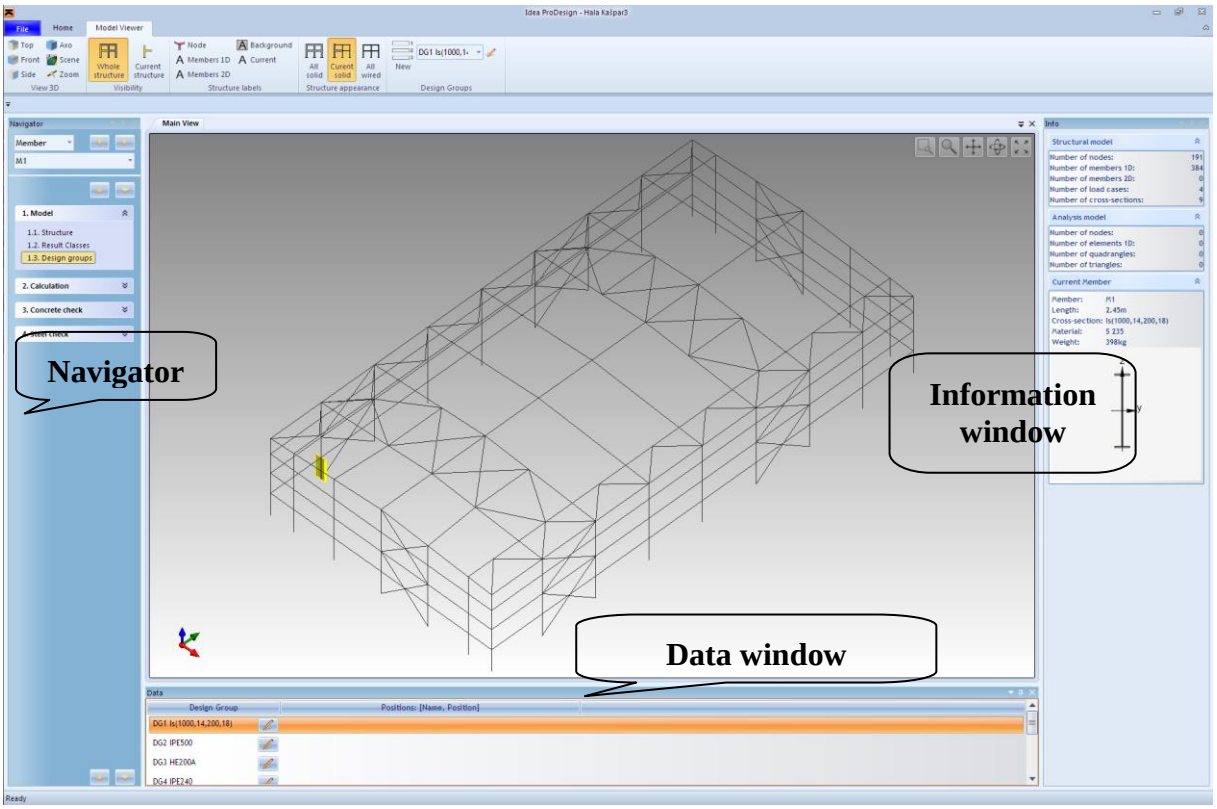
Tendon definition geometry – it is the tendon geometry defined in uncoiled view of design member XY or XZ.

Primary geometry – one of user specified definition geometries. It is used in cases, when position of points in second definition geometry depends on position of points in primary definition geometry.

4 User interface

The user interface consists of several co-operating parts.

- Navigator – set of commands logically ordered, starting first from the input, through the check options to output and reporting.
- Ribbon groups – shows commands related to the current navigator command.
- Main window – shows the image, diagram or text dialog related to the current navigator command.
- Data window – shows the info related to the current navigator command, or the selected object in the Main window, with different tables or properties.
- Information window– actual information related to project are shown for quick user reference



4.1 Info window

Info window of IDEA Tendon contains following groups:

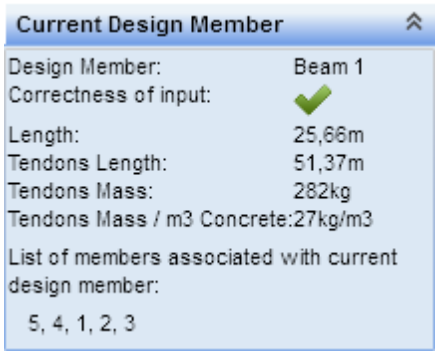
4.1.1 Project data

Following information is displayed in **Project data** group:

- Name of **Project**
- Current National **Code**
- Current **National Annex**
- Information about status of import from superior application to IDEA Tendon. If some problems during import were found, click **Info** to display detailed report of import status.

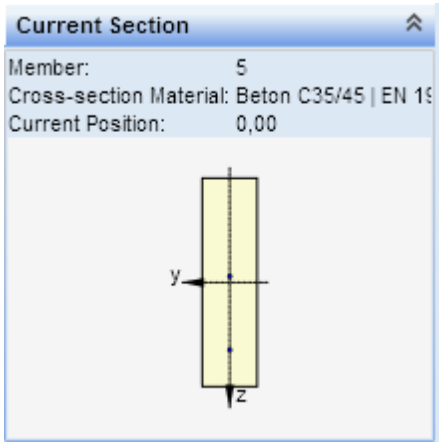
4.1.2 Current Design member

Following information is displayed in **Current Design member** group:



- Name of current **Design member**
- Validity status of current Design member
- **Length** of current Design member
- Total length of all tendons in Design Member
- Total weight of all tendons in Design member
- Total weight of all tendons in Design Member per volume of current Design Member
- List of Members in current Design Member

4.1.3 Current section



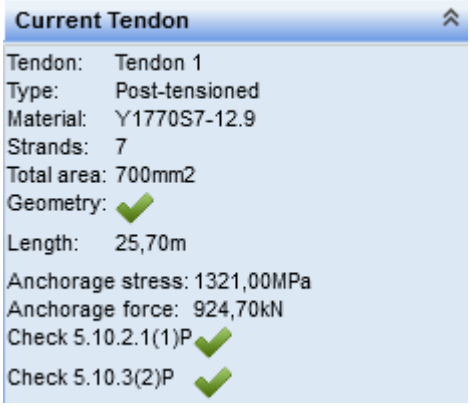
Following information is displayed in group **Current Section**:

- Number of Member, in which current position takes place
- Material of cross-section
- Current position on Design member. Distance is measured from the beginning of Design Member.
- Picture of cross-section in current section including tendons defined in this section.

4.1.3.1 Setting the current section

Use ribbon group **Current section** to set the section position on the design member, for which information about the cross-section is displayed in the Info window. The value of position can be entered into the edit box **Position** or can be set graphically clicking  .

4.1.4 Current tendon

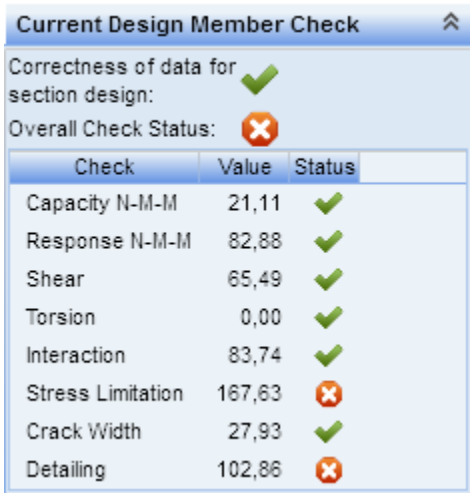


Following information is displayed in group **Current tendon**:

- Name of current tendon
- Type of current tendon
- Material of current tendon
- Number of strands in current tendon
- Total area of current tendon
- Geometry validity status of current tendon
- Length of current tendon
- Anchorage stress of current tendon
- Anchorage force of current tendon
- Result of maximal prestressing force check

- Result of check of prestressing force after anchoring.

4.1.5 Check of current Design member



Group **Current Design Member Check** contains information about status and check results of current Design Member:

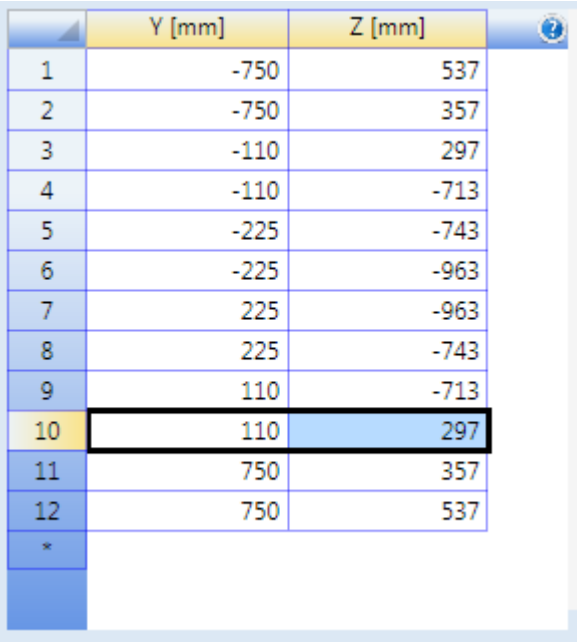
- **Correctness of data for section design** displays status of internal forces for check preparation. If internal forces or other data for check was not prepared correctly, click **Info** to display detailed report of problems during check
- **Overall Check Status** of all positions defined on current Design Member.
- **Table with results** of particular checks performed in defined positions. Each value represents extreme value of particular check from all positions defined on design member.

4.2 Table editor

Some input data (vertexes, values of internal forces etc.) can be entered using table editor.

Copy to clipboard and paste from clipboard can be used to enter the value to single cell or to fill the range of cells (using shortcuts CTRL–C (CTRL-INS) and CTRL–V (SHIFT-INS).

- Cells (ranges) can be pasted to the table from the Microsoft Excel table
- When pasting the data to the table the data are inserted to the current position in the table.
- If the number of columns in the clipboard is greater than the number of columns in the target table, the redundant columns are ignored.
- If the number of rows in the clipboard is greater than one, the rows following the current row in the target table are overwritten. If the number of inserted rows is greater than the number of rows in the target table, the required number of new rows is inserted to the target table



	Y [mm]	Z [mm]
1	-750	537
2	-750	357
3	-110	297
4	-110	-713
5	-225	-743
6	-225	-963
7	225	-963
8	225	-743
9	110	-713
10	110	297
11	750	357
12	750	537
*		

- If a range is selected in the target table and the clipboard contains only value of one cell, all cells in the selected range are filled with the same value when pasting from the clipboard.
- To add a new row to the table click cell * in the indexes column or use the keyboard shortcut **CTRL + ENTER** (the last row of the table must be set as current row)

Following keyboard shortcuts can be used when working with the table editor:

CTRL + + - insert a row before the current row.

CTRL + ENTER - append a row to the current row.

CTRL + - delete the current row.

CTRL + A- select the whole table

CTRL + C (**CTRL + INS**) – copy the selected cells to the clipboard.

CTRL + V (**SHIFT + INS**) – paste the clipboard content to the table

TAB – change the current cell by moving forwards through the cells

SHIFT + TAB – change the current cell by moving backwards through the cells

<, >, ∧, ∨ - change the current cell by moving left, right, up, down

F2 – switch to edit mode of the cell and place the cursor to the end of the current cell. Move to the other cell to finish the edit mode with preserving the changes or push **ESC** to discard the changes.

ESC – close the edit mode discarding the changes.

4.3 Control of view in the 2D window

The view in 2D window can be set by mouse or by tool in the left upper corner of the window.

 - zoom all. Click this button to fit the whole structure to the 2D window.

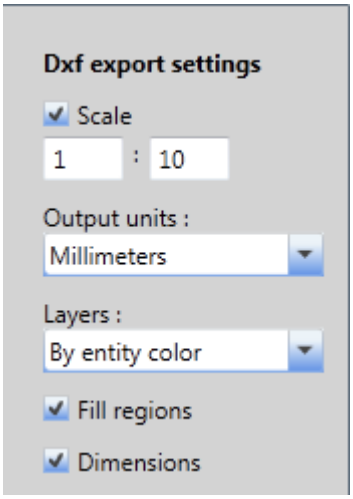
To set the required view using keyboard and mouse following combinations can be used:

- Click and hold mid mouse button – moving the mouse pans the view.
- Roll with mid mouse button – moving the mouse increases/decreases the view.
- Push CTRL+SHIFT and hold mid mouse button – moving the mouse defines the window for zoom.

Click on right mouse button over 2D window shows context menu with following commands:

- **Zoom all** – zoom to show the whole current structure in the 2D window.
- **Print** – start printing of the current content of 2D window on selected printer.
- **To bitmap** – start export of the current content of 2D window to the raster graphics file (PNG, GIF, BMP, JPEG, TIFF).
- **To clipboard** – copy of the current content of 2D window to the Windows clipboard.
- **To DXF** – start export of the current content of 2D window to the 2D DXF file.

4.3.1 DXF export settings

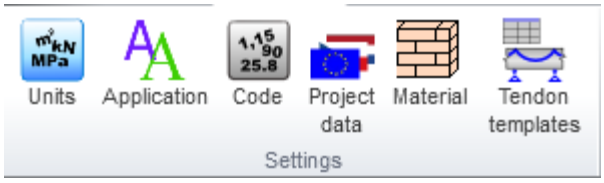


Following export parameters can be set in the Save as dialog when exporting the view to the 2D file:

- **Scale** – if the option is selected, the scale ratio used to create the drawing in exported DXF can be set.
- **Output units** – select units of the drawing in the exported DXF file.
- **Layers** – select the mode of layers generation. Layers can be generated according to the line type, the line thickness, the entity type or the entity color.
- **Fill regions** – switch on/off export of filled regions (otherwise only outlines are exported).
- **Dimensions** – switch on/off export of dimension lines.

5 Application settings

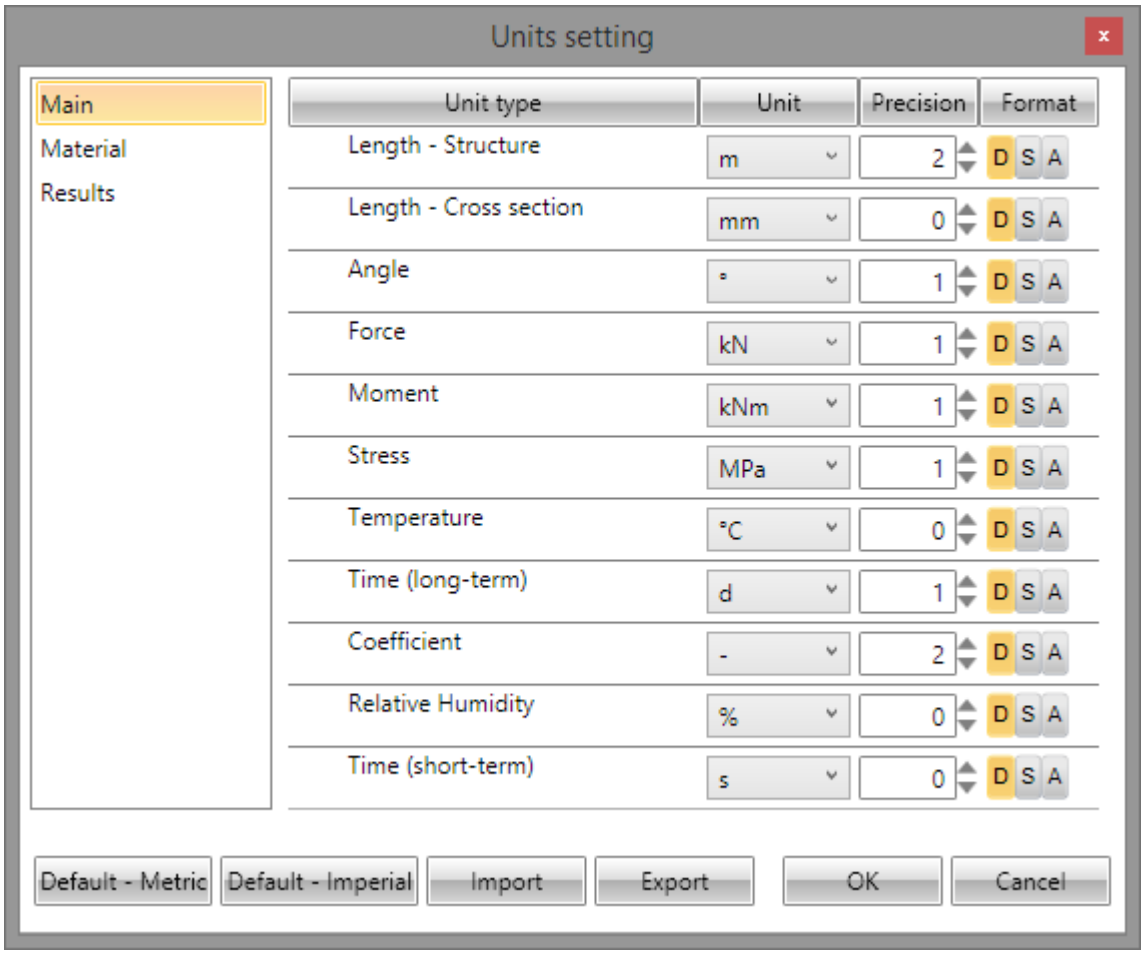
Several application settings can be defined using buttons in ribbon group **Settings**.



- **Units** – change of units settings – see **5.1 Units setting**.
- **Application** –settings of colours, line styles and item description styles – see **5.2 Application environment settings**.
- **Code** – setting of values of national code and calculation coefficients, which are used during check of reinforced cross-sections – see **5.3 Code and calculation settings**.
- **Project data** – input of project identification data and selection of national annex for check of reinforced cross-section – see **5.4 Project data**.
- **Material** – view and edit of project library of prestressing reinforcement materials – see **5.5 Materials library**.
- **Tendon templates** – launch manager of user defined templates of tendon shapes – see **9.10.2 Templates manager**.

5.1 Units setting

The units used by the application can be set by clicking button **Units** in ribbon group **Settings** of the page **Home**.



Magnitudes, for which the units can be set, are grouped into categories Main, Material and Results. The categories are displayed in the column on the left of the dialog. For the selected

category the table of corresponding magnitudes is displayed. For each magnitude, which is listed in column **Unit type,** one of the available units can be set in the column **Unit**.

For each magnitude the number of digits to be displayed after decimal point can be set in the column **Precision**.

Style of numbers presentation can be set in **Format** column:

- **D** – display numbers in standard decimal format ("-ddd.ddd…").
- **S** – display numbers in exponential format ("-d.ddd…E+ddd").
- **A** – according to length of resulting string it is automatically chosen whether to use decimal or exponential format. In this mode value specified in **Precision** column means number of significant digits in the resulting string.

Default – metric – loads default units settings for metric units system.

Default – imperial – loads default units settings for imperial units system.

Import - reads the units configuration from a file.

Export - saves the current units settings to a file.

Click **OK** to apply the changes and to be used at next application start.

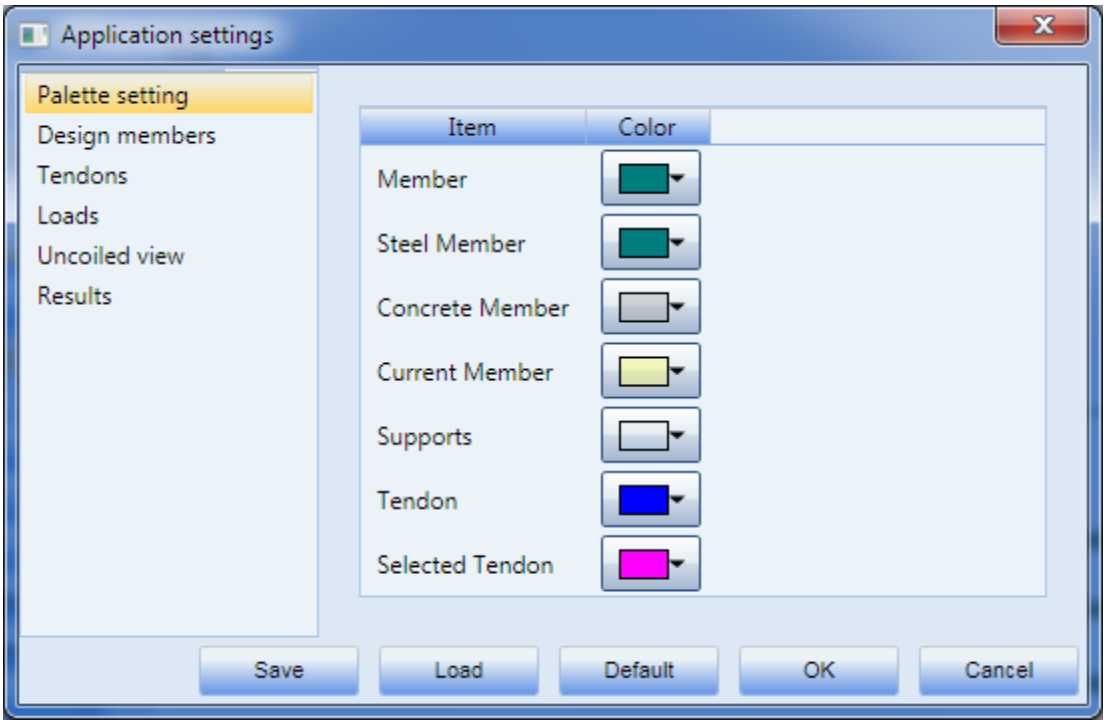
5.2 Application environment settings

To change application environment (colours, fonts, lines) click **Application** in ribbon group **Settings**. The settings are grouped to several tabs. The whole settings can be stored to file or to be read from file using commands:

- **Save** – store current application settings to a specified file
- **Load** – load application settings from specified file
- **Default** – restore default application settings

5.2.1 3Ddrawing colours setting

To set colours for drawing of model in 3D view click tab **Palette setting**.

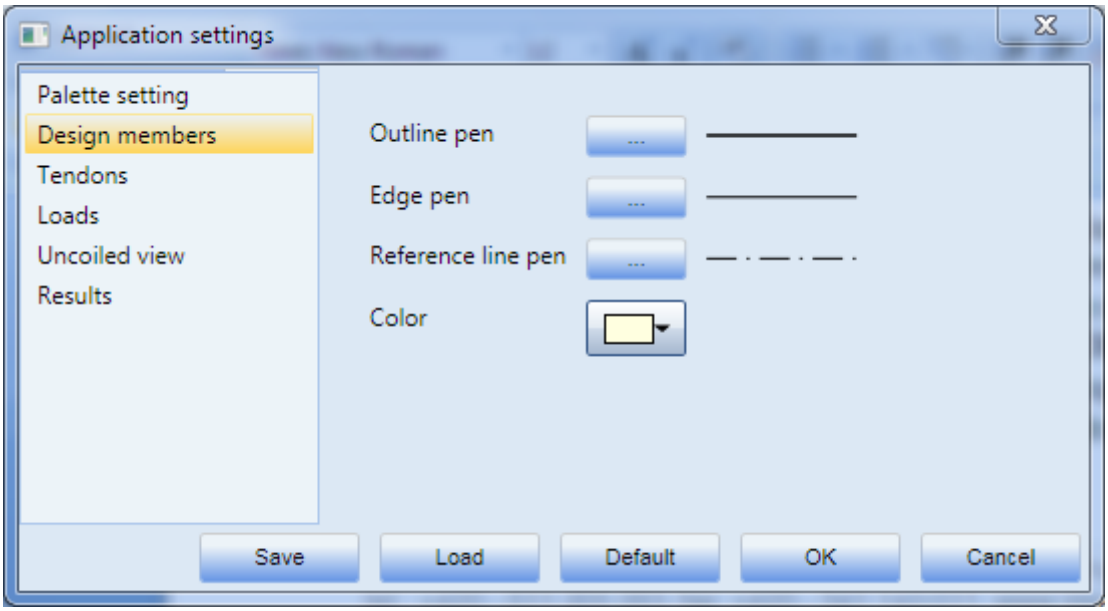


Colours of following entities can be set on tab **Palette setting**:

- **Member** – select a colour for drawing of design members
- **Steel member** – select a colour for drawing of steel members
- **Concrete member** – select a colour for drawing of concrete members
- **Current member** – select a colour for drawing of current design member
- **Supports** – select a colour for drawing of supports
- **Tendons** – select a colour for drawing of tendons
- **Current tendon** –select a colour for drawing of selected tendon

5.2.2 Design member drawing

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

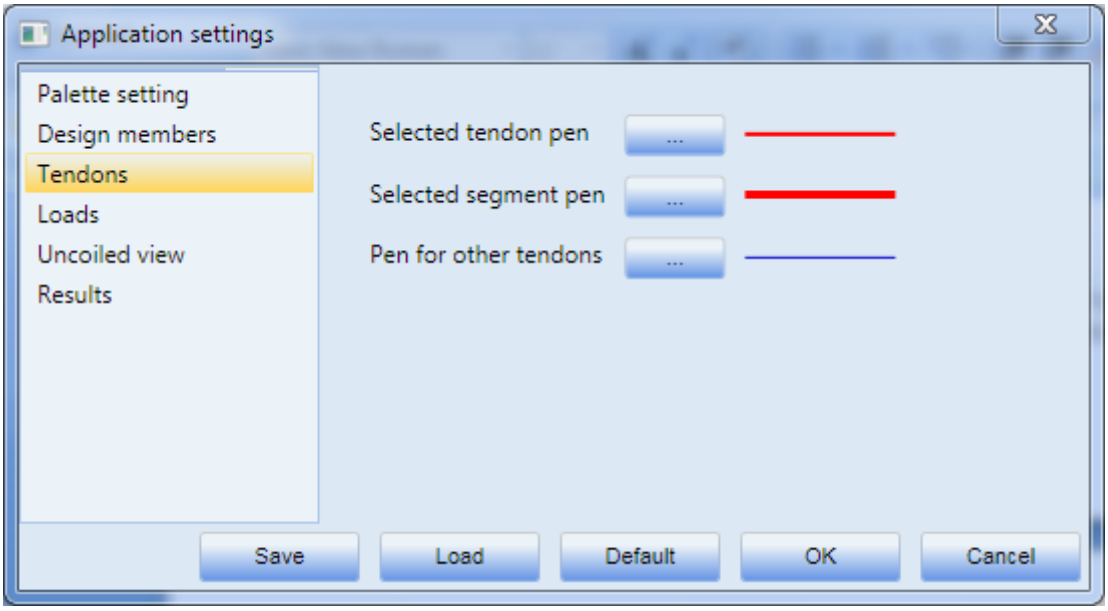


Particular dialog options:

- **Outline pen** – line style setting for drawing of design member outline. See **5.2.7 Line drawing style setting**.
- **Edge pen** – line style setting for drawing of design member edges. All edges (visible and hidden) of design member in uncoiled views XY and XZ are considered as edges. See **5.2.7 Line drawing style setting**.
- **Reference curve pen** – line style setting for drawing of reference curve. See **5.2.7 Line drawing style setting**
- **Colour** – select colour for drawing of design member fill.

5.2.3 Tendons drawing

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



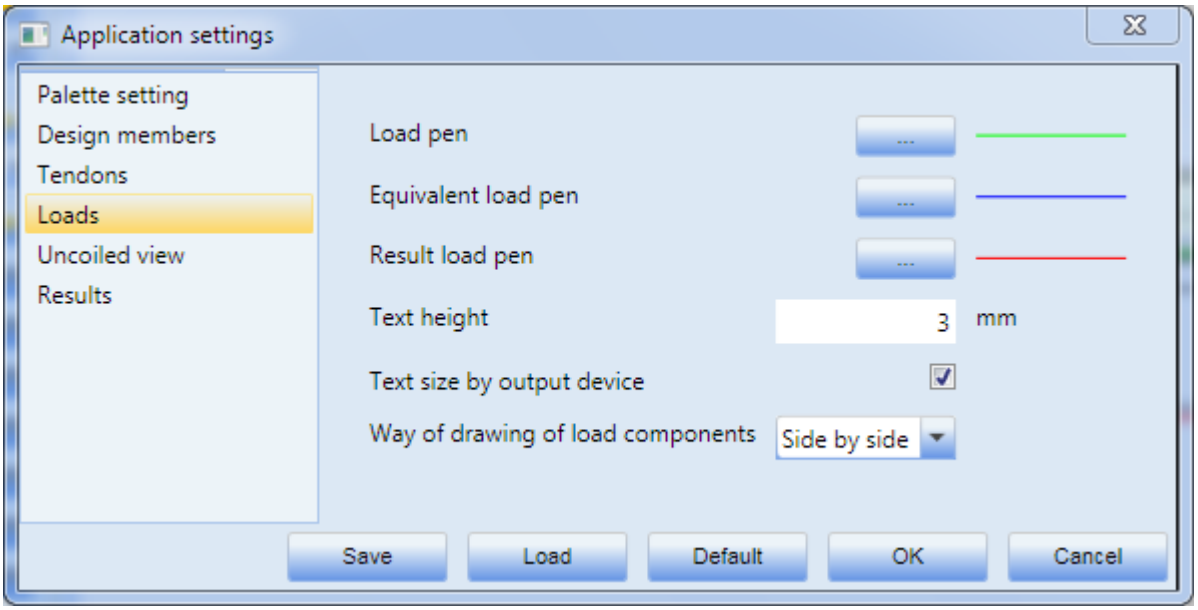
Particular dialog options:

- **Selected tendon pen** – line style setting for drawing of selected tendon. See **5.2.7 Line drawing style setting**.

- **Selected segment pen** – line style setting for drawing of selected tendon segment. See **5.2.7 Line drawing style setting**.
- **Pen for other tendons** – line style setting for drawing of not selected tendons. See **5.2.7 Line drawing style setting**.

5.2.4 Loads drawing

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

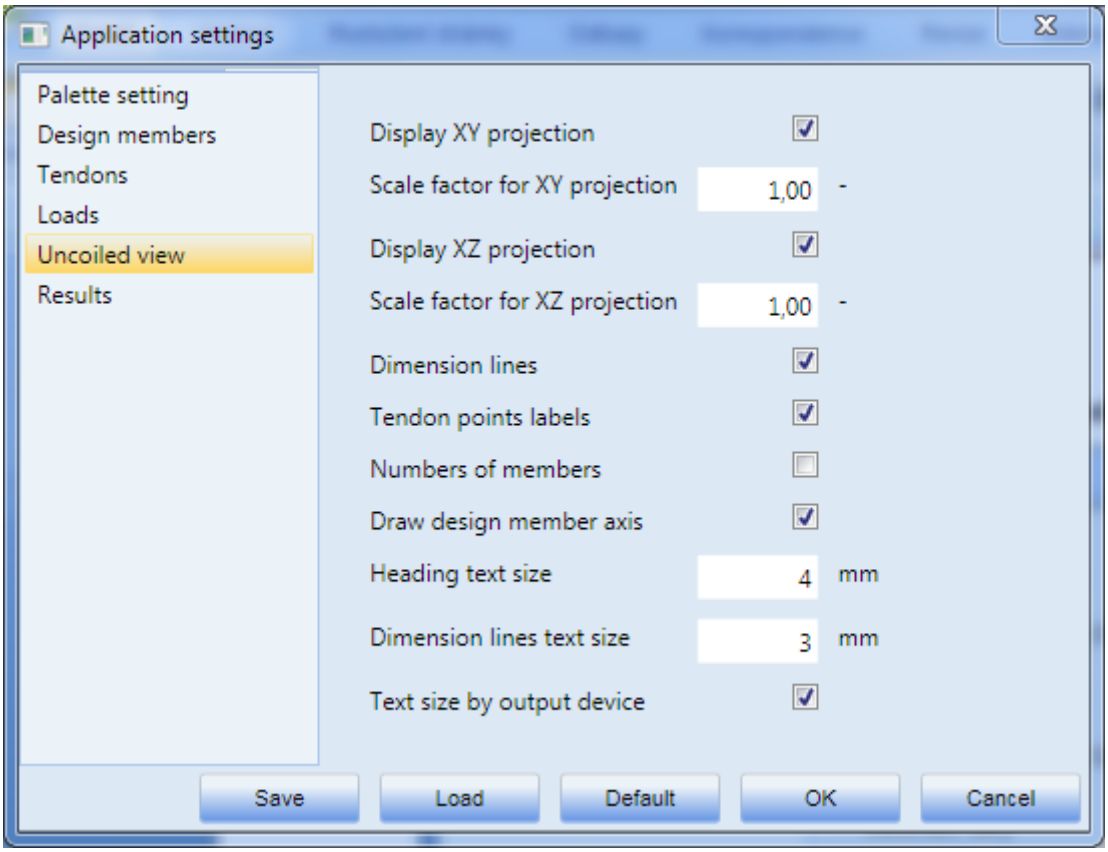


Particular dialog options:

- **Load pen** – line style setting for drawing of external loads. See **5.2.7 Line drawing style setting**.
- **Equivalent load pen** – line style setting for drawing of tendon equivalent load. See **5.2.7 Line drawing style setting**.
- **Result load pen** – line style setting for drawing of resulting not balanced load. See **5.2.7 Line drawing style setting**.
- **Text height** – value of text size of loads labels
- **Text size by output device** – set evaluation mode of text height. If the option is on, the real height of text on output device (2D window, report, printer) is the specified value in millimetres (length units).
- **Way of drawing of load component** – choose mode of load graphs drawing
  - **Side by side** – graphs of loads in particular uncoiled views are drawn side by side
  - **Below each other** – graphs of loads for all uncoiled views are drawn below each other

5.2.5 Uncoiled view

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

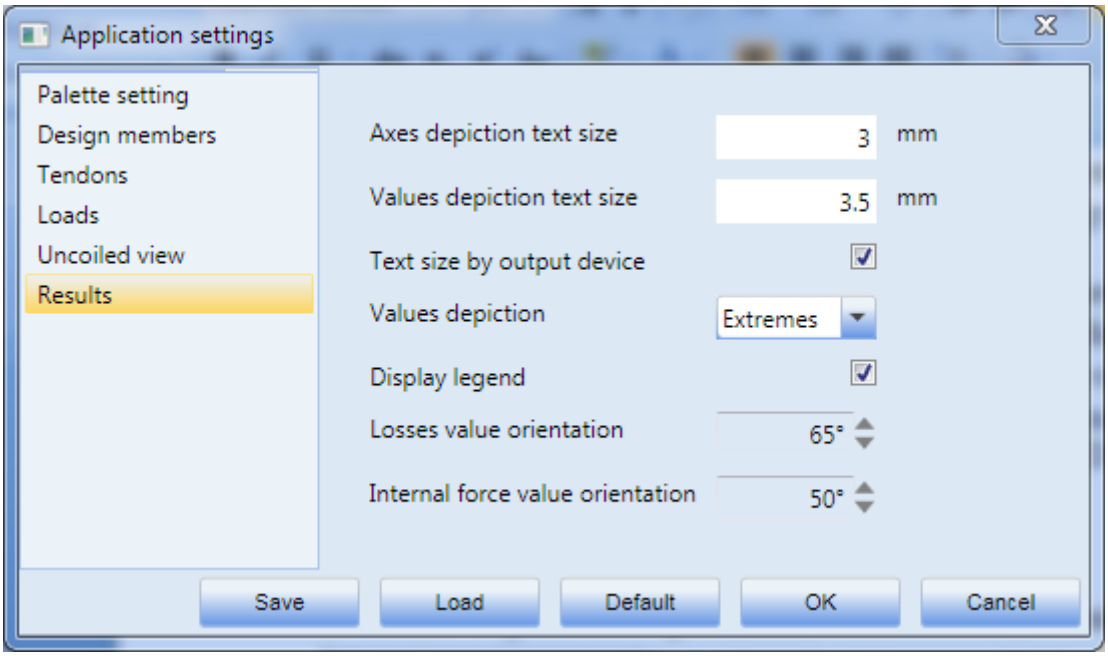


Particular dialog options:

- **Display XY projection** – turn on/off drawing of design member uncoiled view in plane XY.
- **Scale factor for XY projection** – value of exceeded scale for drawing of design member in uncoiled view in plane XY. Exceeded scale enables more clear drawing of tendons in design members, which x-axis length exceeds the size in y-axis in uncoiled view XY.
- **Display XY projection** – turn on/off drawing of design member uncoiled view in plane XZ.
- **Scale factor for XZ projection** – value of exceeded scale for drawing of design member in uncoiled view in plane XZ.
- **Dimension lines** – turn on/off drawing of dimension lines.
- **Tendon points labels** – turn on/off description of tendon editing points.
- **Numbers of members** – turns on/off drawing of numbers of members in design member.
- **Draw design member axis** – turns on/off drawing of axis of design member
- **Heading text size** – value of text size of headings of uncoiled views
- **Dimension lines text size** – value of text size of dimension lines texts
- **Text size by output device** – set evaluation mode of text height. If the option is on, the real height of text on output device (2D window, report, printer) is the specified value in millimetres (length units).

5.2.6 Results drawing and descriptions

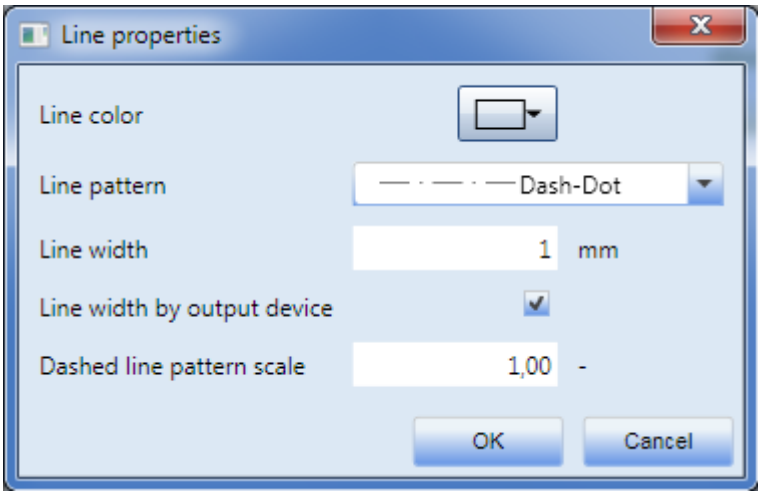
To set drawing and descriptions of internal forces and tendon losses click tab **Results**.



Particular dialog options:

- **Axes depiction text size** – value of text size of axes depiction
- **Values depiction text size** – value of text size of result values depiction
- **Text size by output device** – set evaluation mode of text height. If the option is on, the real height of text on output device (2D window, report, printer) is the specified value in millimetres (length units).
- **Values depiction** – mode of tendon losses graph depiction drawing
  - **No depiction** – no values in graphs are depicted
  - **Extremes** – extreme values in graphs are depicted
  - **All** – all values in graphs are depicted
- **Display legend** – turn on/off drawing of legend in tendon losses graph
- **Losses value orientation** –value of slope for depiction of tendon losses
- **Internal force value orientation** – value of slope for depiction of internal forces

5.2.7 Line drawing style setting



Particular options of **Line properties** dialog:

- **Line colour** – set colour of line

- **Line pattern** – set line pattern
- **Line width** – value of line width in length units or in number of pixels
- **Line width by output device** – if the option is on, corresponding line is drawn in specified width in corresponding length units. If the option is off, the line is drawn in specified with in pixels
- **Dashed line pattern scale** – value of scale for drawing of dashed lines.

5.3 Code and calculation settings

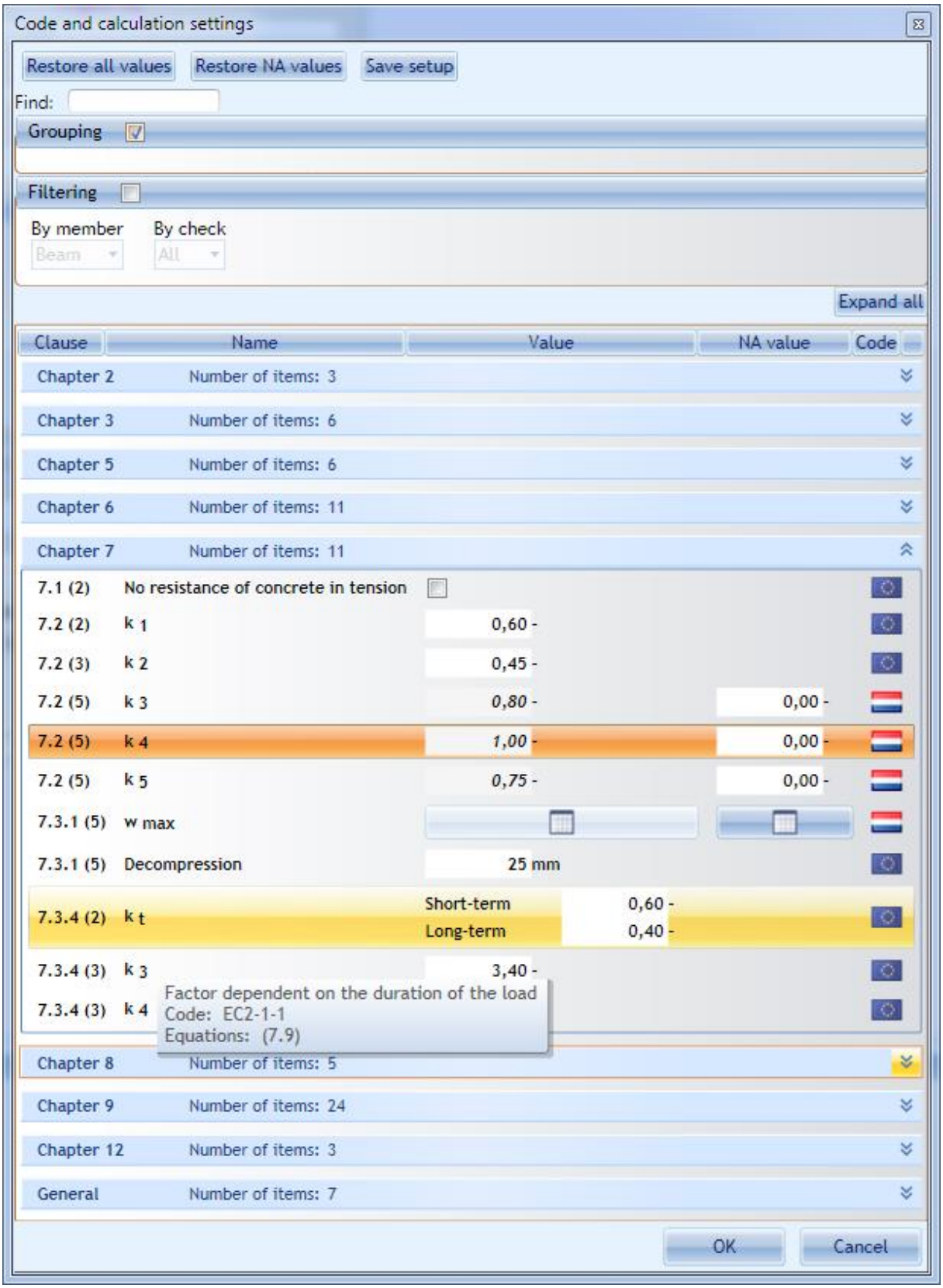
Click button **Code** in ribbon group **Settings** to set the national code values and calculation variables.

Code and calculation settings are taken into account when performing checks of prestressed reinforced section in module IDEA RCS.

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

If national annex is enabled (to change national annex, click **NA** in **Project data** dialog), values of national annex can be changed or default value of EC code can be used.

To display tooltip with detailed information about code variable point mouse cursor on row with code variable.



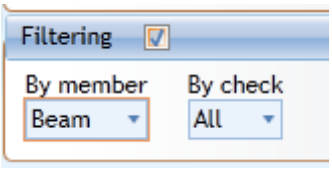
Restore all values – resets all values of code settings for EC and current national annex to default code and annex values.

Restore NA values – resets values of current national annex to default annex values.

Save setup – saves current code settings to file. Saved settings can be loaded into other project clicking **Code** button in **Project data** dialog. To display **Project data** dialog click **Project data** in ribbon group **Settings**.

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.

Column **Clause** – numbers of particular code clauses are displayed it this column.

Column **Name** – names of code variables are displayed in this column.

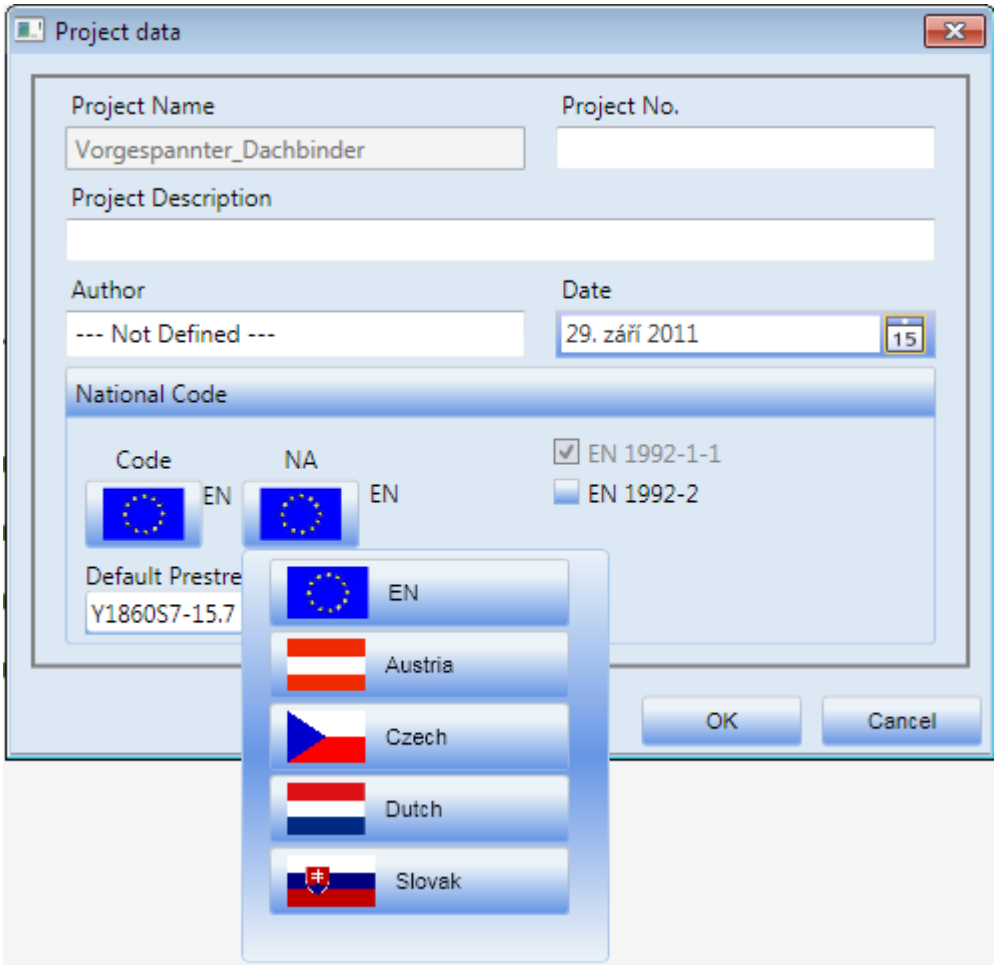
Column **Value** – code variable values can be edited in this column. If there is checkbox at code value, it can turn the value on/off to be taken into account or neglected in the check. Values of code variables can be edited only if column **Code** is set to EC-EN.

Column **Value NA** – value of national annex can be edited in this column if national annex value is available for particular code setting item. Values of annex variables can be edited only if column **Code** is set to national annex

Column **Code** – flag in column indicates, which code is active for particular code setting item. Click flag icon to switch between national annex and EC code.

5.4 Project data

To change project data, choose national annex and choose default materials click **Project data** in ribbon group **Settings**. Dialog **Project data** with project details and table National code appears. Project identification data are printed in the header of report.



Particular options of dialog **Project data**:

- **Code** – click to set current code to EC-EN or to load user defined settings of code parameters**.**
- **NA** – click to load one of available National Annex parameter sets.
- **EN 1992-2** –turn on/off option to check a cross-section according to code EN 1992-2 in IDEA RCS.
- **Default prestressing steel grade** – (list-box) the default prestressing reinforcement grade from the displayed list is assigned to newly input prestressing tendons.

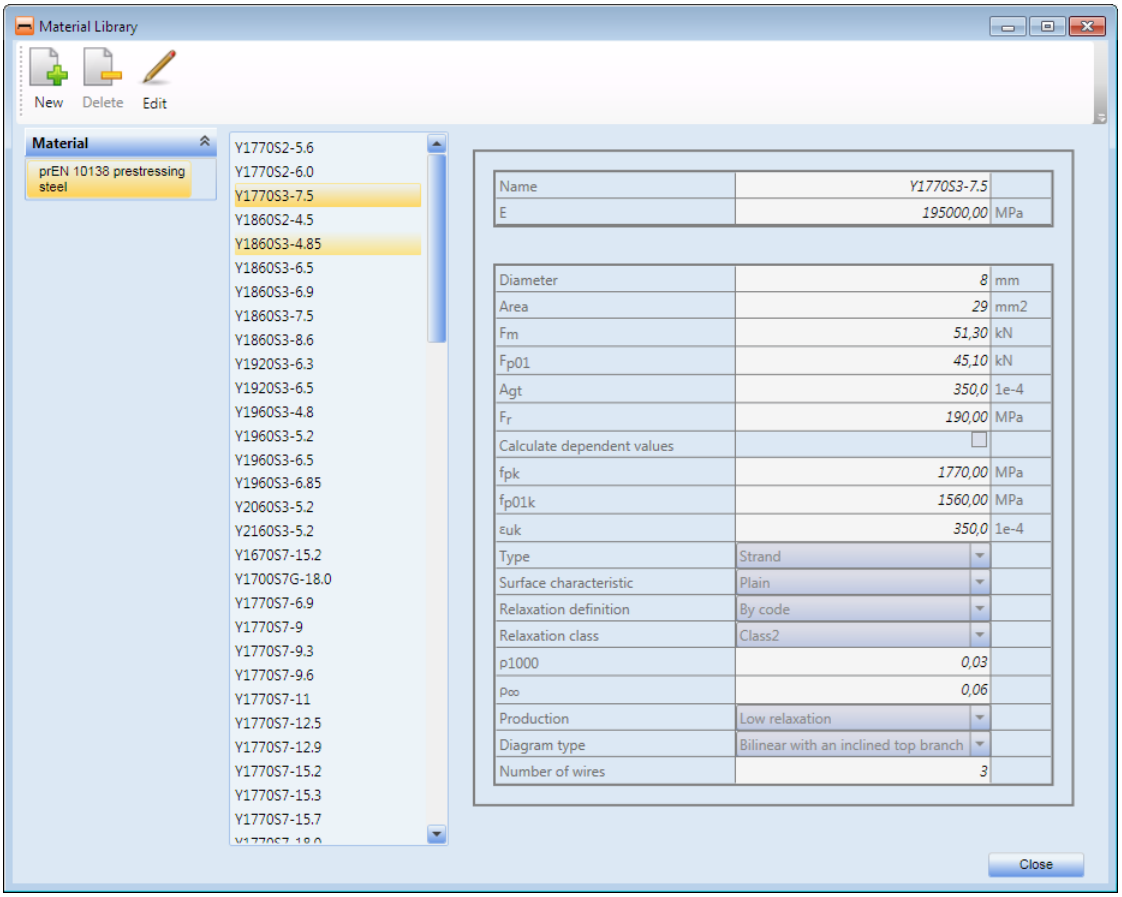
5.5 Materials library

To open Material library dialog click **Material** in ribbon group **Settings**.

The first column contains list of available material types in the project.

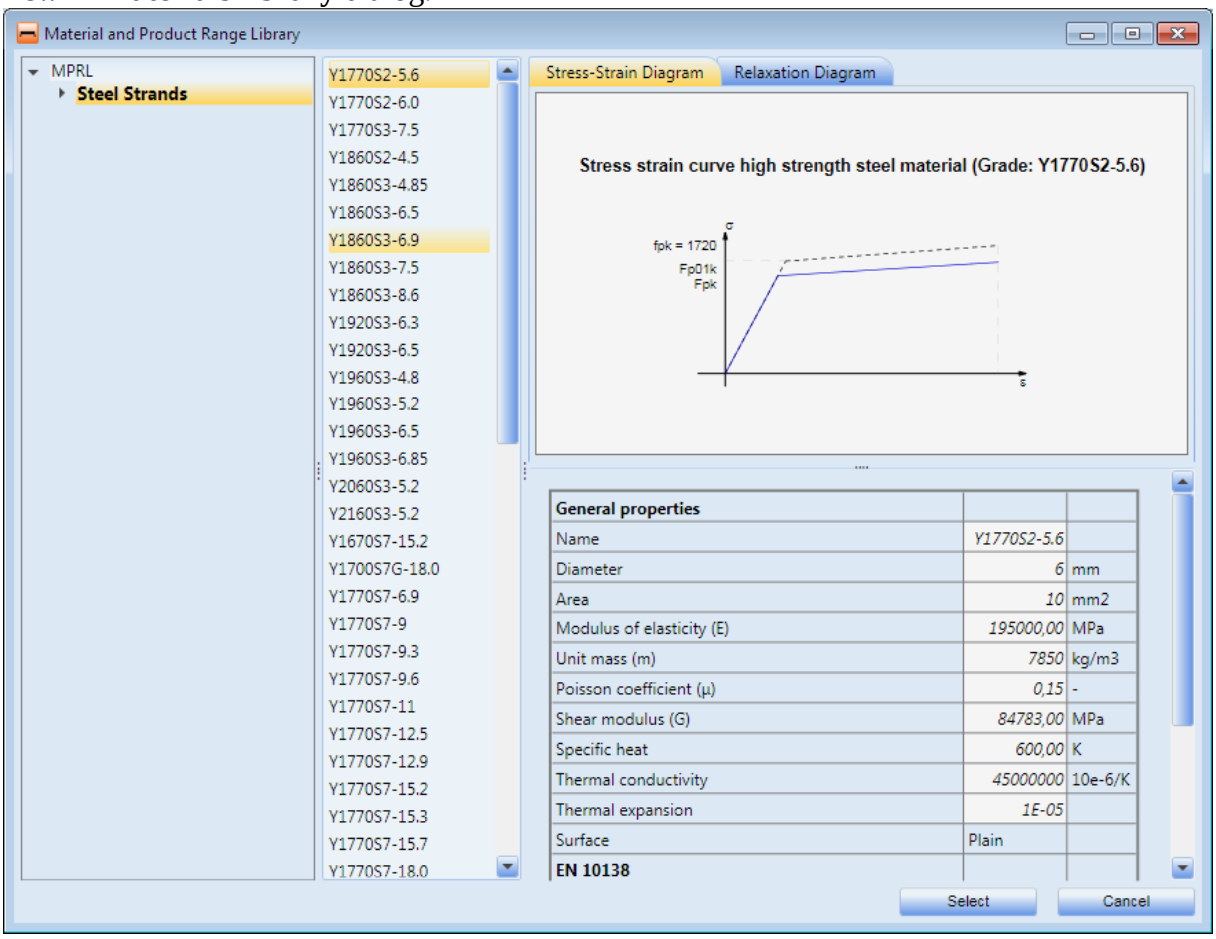
The second column contains list of particular materials in the project.

The third column contains table with material properties of selected material.



5.5.1 New material

To add a new material from the system material library into the project material library click **New** in **Materials library** dialog.



The first column of dialog contains tree list of available items group in system database of cross-sections and materials.

The second column contains list of particular items, which are available in the selected items group.

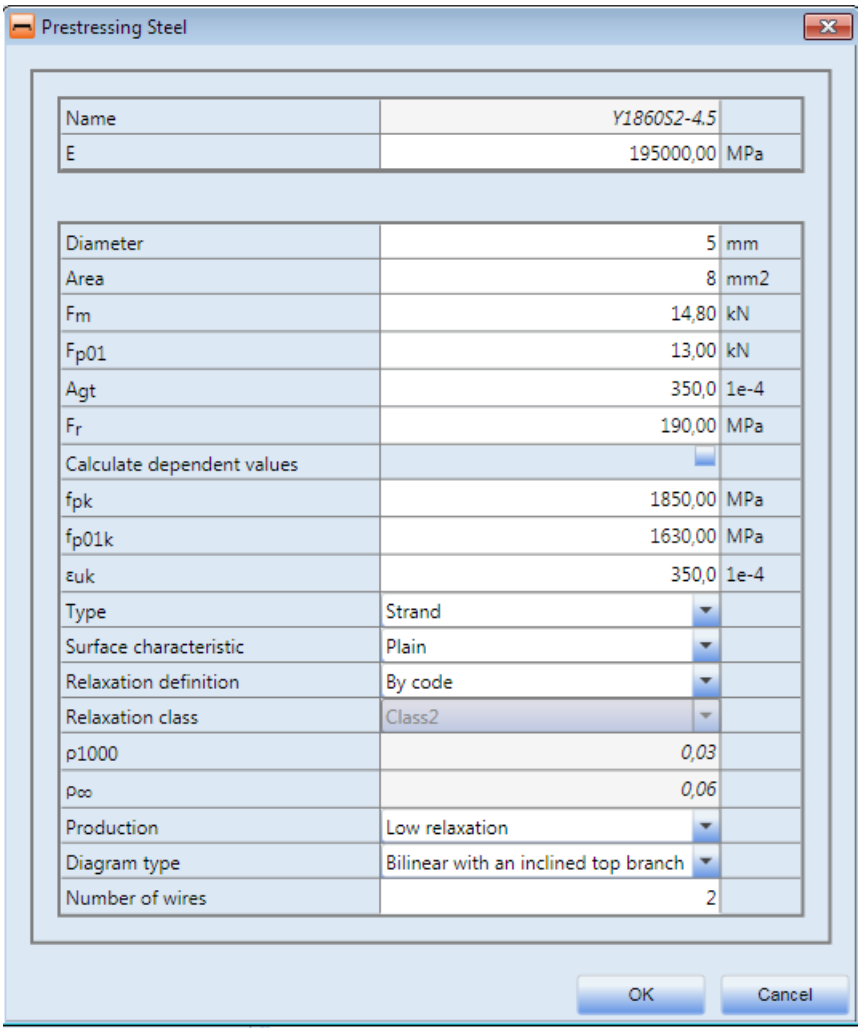
The third column contains table with properties of selected library item.

To add selected material into the project library click **Select**.

5.5.2 Material edit

To edit properties of material, which is selected in **Material library** dialog, click **Edit**.

Dialog **Prestressing steel** appears, in which characteristics of edited material can be changed.

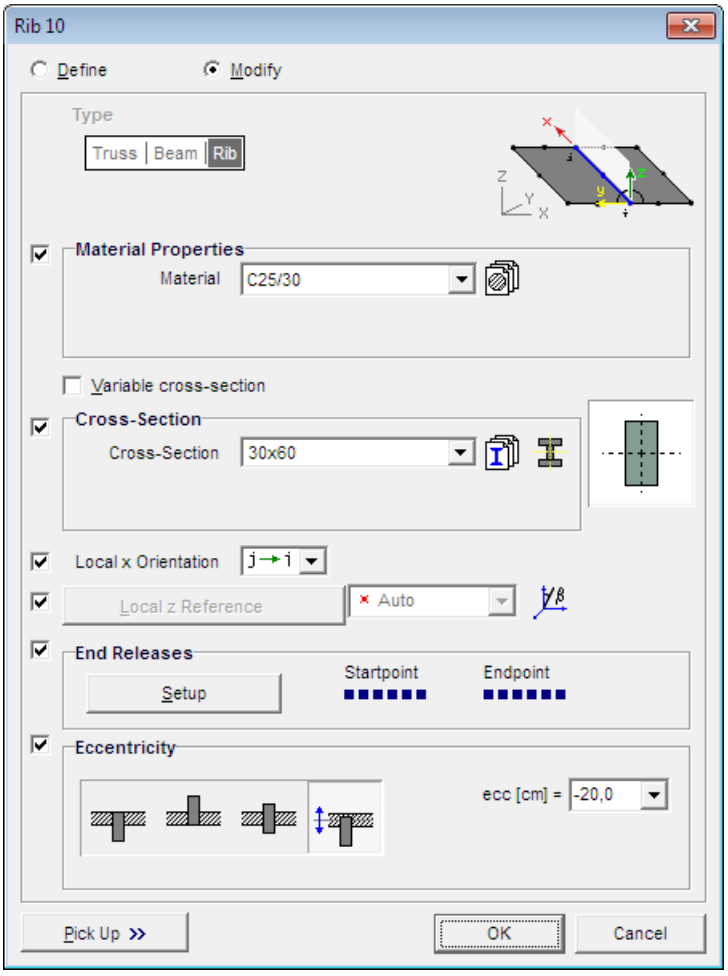

Prestressing Steel
Name
Y1860S2-4.5
E
195000,00
MPa
Diameter
5
mm
Area
8
mm2
Fm
14,80
kN
Fp01
13,00
kN
Agt
350,0
1e-4
Fr
190,00
MPa
Calculate dependent values
fpk
1850,00
MPa
fp01k
1630,00
MPa
εuk
350,0
1e-4
Type
Strand
Surface characteristic
Plain
Relaxation definition
By code
Relaxation class
Class2
ρ1000
0,03
ρ∞
0,06
Production
Low relaxation
Diagram type
Bilinear with an inclined top branch
Number of wires
2
OK
Cancel

6 Export of structure from AxisVM to IDEA Tendon

6.1 Requirements and limitations of export from AxisVM 11

6.1.1 Not supported structural elements

- Only Euro code materials are supported. Materials of other codes cannot be used.
- **Shell surfaces** are not exported
- **Rigid links** – design member cannot be created from members, which eccentricity is defined using rigid links. Eccentricity has to be defined as rib eccentricity. To be able to create a Design member the neighbouring members and ribs must be connected in common node. It is not possible to connect them using rigid link.

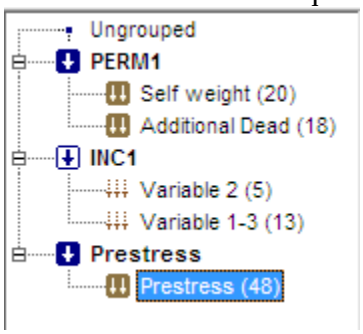


6.1.2 Load cases / combinations

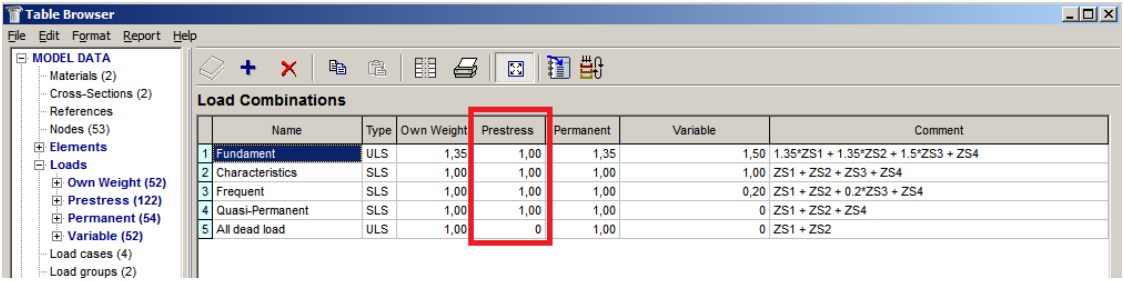
Following limitations has to be considered when exporting data from AxisVM to IDEA Tendon:

- First load case of type **Prestress**, which has been defined in AxisVM project, is used for transfer of equivalent loads caused by prestressing tendons.

- If more load cases of type **Prestress** are defined in the AxisVM project, only the first one is used to transfer equivalent loads.



- **Coefficient of load case** has to be set to 1 or 0. Thus values of partial load factors in appropriate load group have to be corrected. It is recommended to create separate load group only for load case "Prestress".



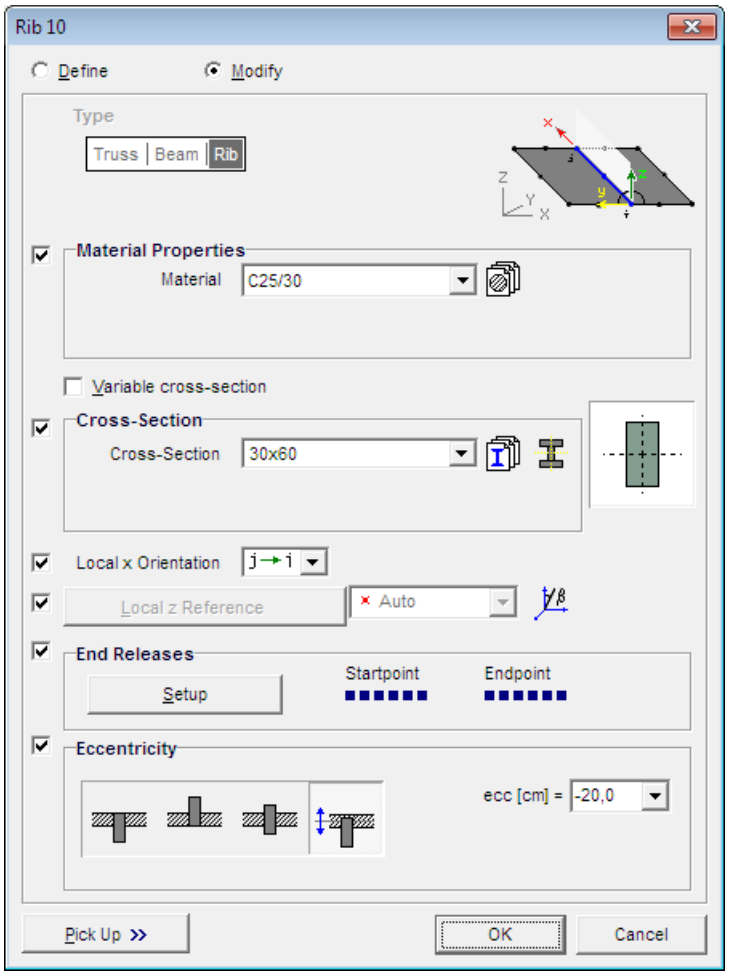
6.1.3 Analysis

- Only results of linear calculation can be used for design of tendons in IDEA Tendon.
- To obtain adequate results the members of structural model has to be refined using mesh parameters for line members.

6.2 Requirements and limitations of export from AxisVM 10

6.2.1 Not supported structural elements

- Only Euro code materials are supported. Materials of other codes cannot be used.
- **Shell surfaces** are not exported
- **Rigid links** – design member cannot be created, if eccentricity is defined using rigid links. Eccentricity has to be defined as rib eccentricity.

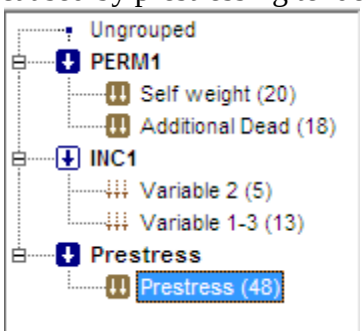


6.2.2 Load cases / combinations

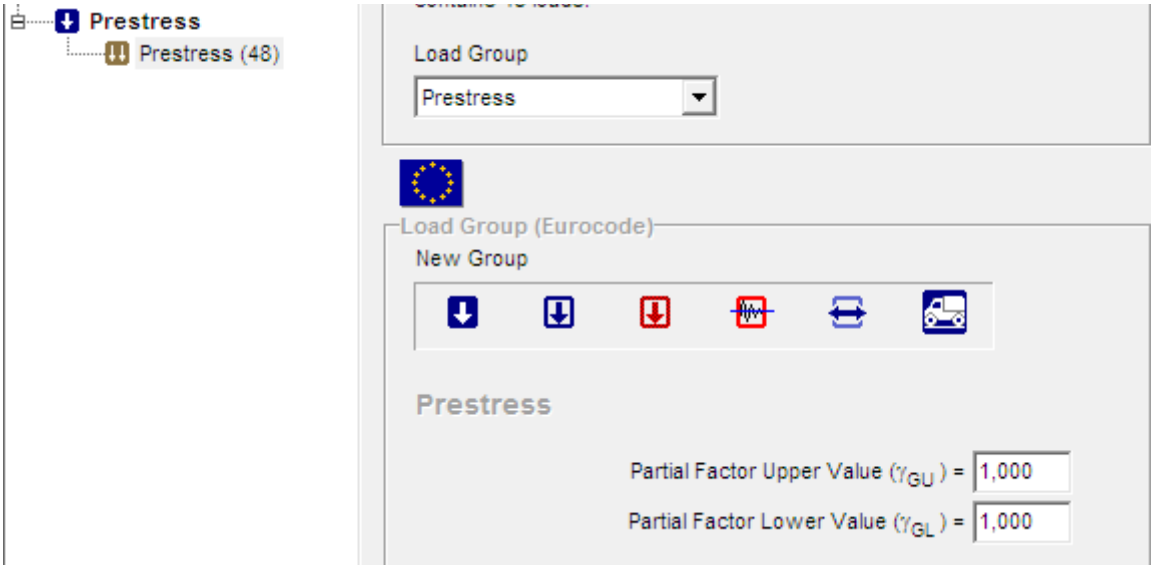
Following limitations has to be considered when exporting data from AxisVM to IDEA Tendon:

- Load case of type Prestress, which can be defined in AxisVM, cannot be used for transfer of equivalent loads caused by prestressing tendons. Permanent load case with name containing string “Prestress” has to be defined to transfer equivalent loads

caused by prestressing tendons into structural model in AxisVM



- Only one load case with name containing string “Prestress” can be defined in project. If more load cases with name containing string “Prestress” are defined, the first is used for transfer of equivalent loads.
- **Coefficient of load case** has to be set to 1 or 0. Thus values of partial load factors in appropriate load group have to be corrected. It is recommended to create separate load group only for load case “Prestress”.



- None of load cases can be renamed in AxisVM during working with structural model. Names of load cases are used to match load case in AxisVM with load case in IDEA Tendon.
- None of combinations of load cases can be renamed in AxisVM during working with structural model. Names of combinations are used to match combination in AxisVM with combination in IDEA Tendon.

6.2.3 Calculation

- Only results of linear calculation can be used for design of tendons in IDEA Tendon.

7 Global time axis

Global time axis defines construction and service stages of structure.

Project must contain at least 3 stages. Each stage is defined by its time on global time axis, by list of load cases and by list of combinations. Name and description of stage can be edited too.

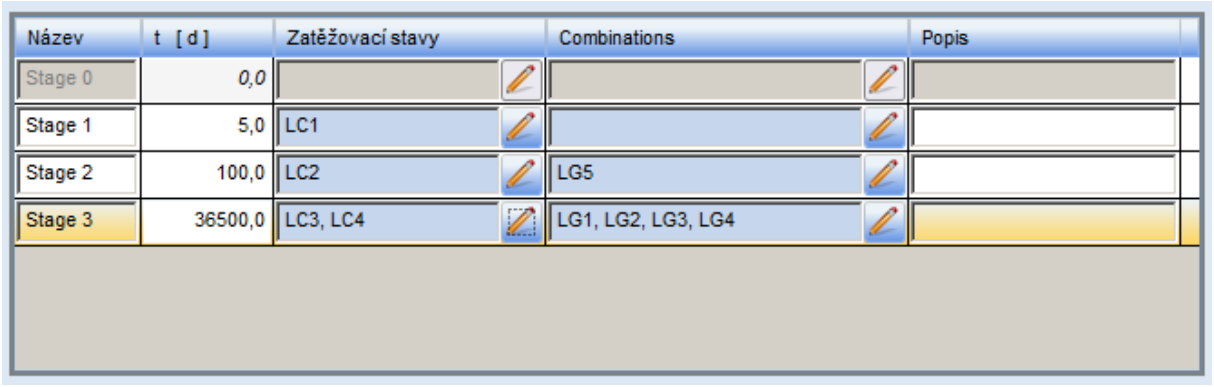
To input or edit construction stages click navigator command **Project data > Construction stages**.

- **If IDEA Tendon is launched from IDEA Beam, the modifications of global time axis (stages, assigning load cases and combinations to stages etc.) are not available.**

Global time axis with assigned load cases is drawn in **Main window**.

Table with defined construction stages is displayed in **Data window**.

Ribbon group **Construction stages** in available when working with construction stages.



7.1 List of load cases applied in particular construction stage

Only **permanent** load cases can be assigned to construction stages. **Permanent** load case can be applied only in **one** construction stage.

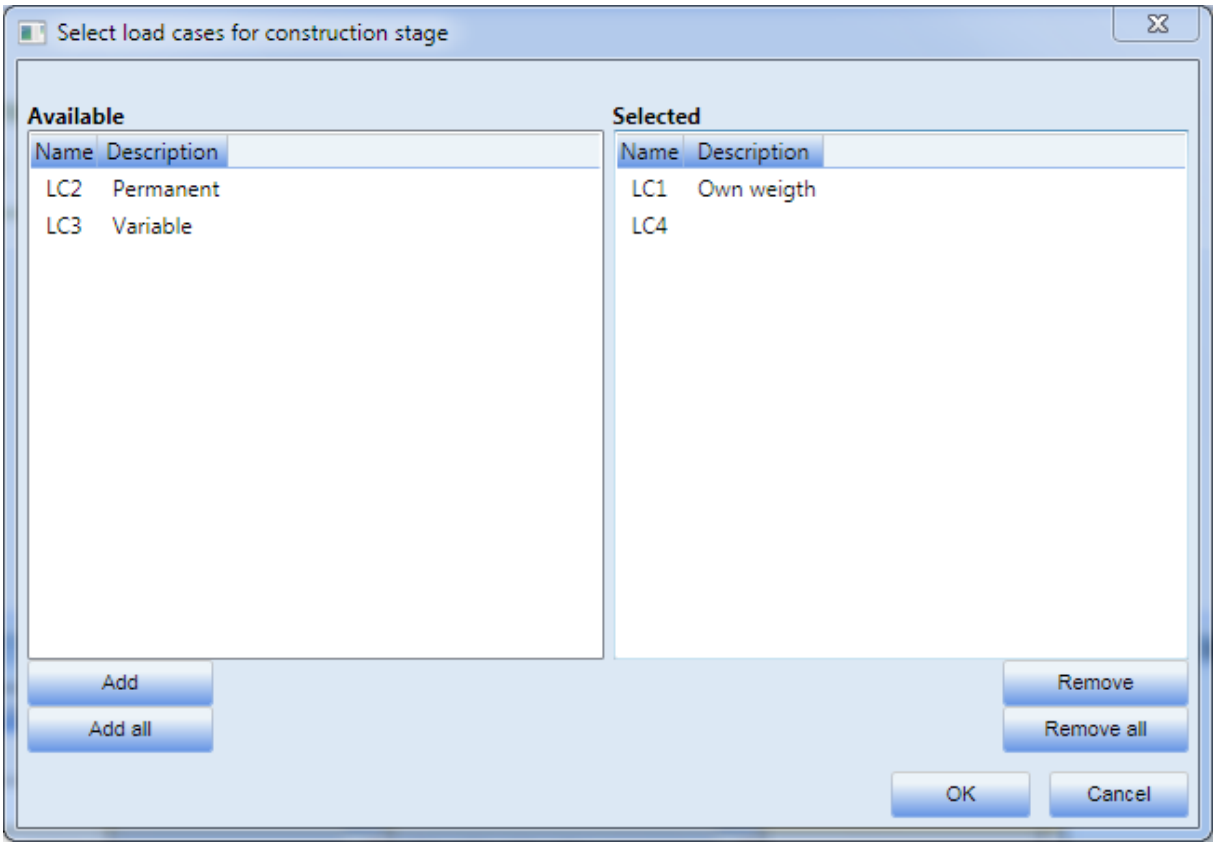
Permanent load cases are assigned to that one construction stage, in which the first occurrence of the load case is assumed. If permanent load case has been assigned to some construction stage, it cannot be assigned again to any later construction stage. Load case for prestressing has to be applied to first construction stage.

Variable load cases cannot be assigned to construction stages. Variable load cases can act in construction stages only in combinations, as will be explained later.

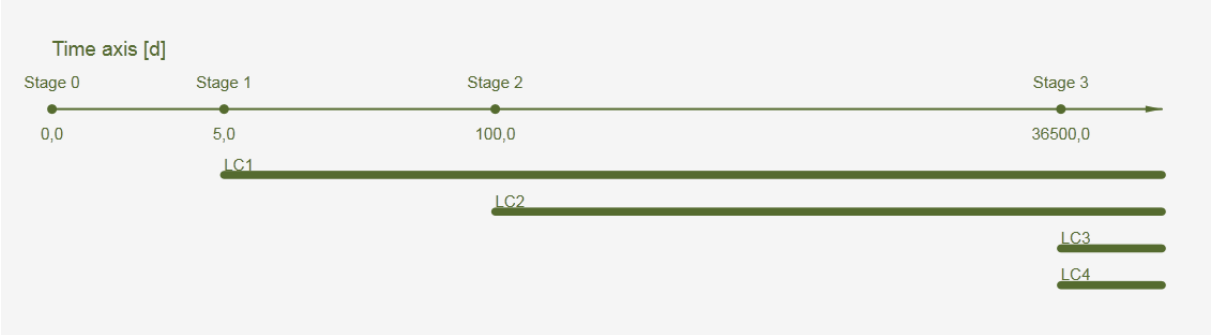
Load cases applied in particular construction stages are listed in column **Load cases** in table **Construction stages**. To assign load cases to particular construction stage click edit button  in column **Load cases**.

Required load cases to be applied to construction stage can be selected in dialog **Select load cases for construction stage**.

Load cases, which can be assigned to construction stage, are listed in column **Available**. Load cases, which have been assigned to edited stage, are listed in column **Selected**.



Application of load cases can be verified in the picture of global time axis.



7.2 List of combinations applied in construction stage

Combinations applied in particular construction stages are listed in column **Combinations** in table **Construction stages**. Those combinations are used for checks of sections in module IDEA RCS. Combinations can be assigned to construction stage either manually or automatically by clicking **Reorder combi** in ribbon group **Construction stages**.

7.2.1 Manual assignment of combinations to construction stages

Following rule has to be fulfilled when assigning combination of load cases to construction stage:

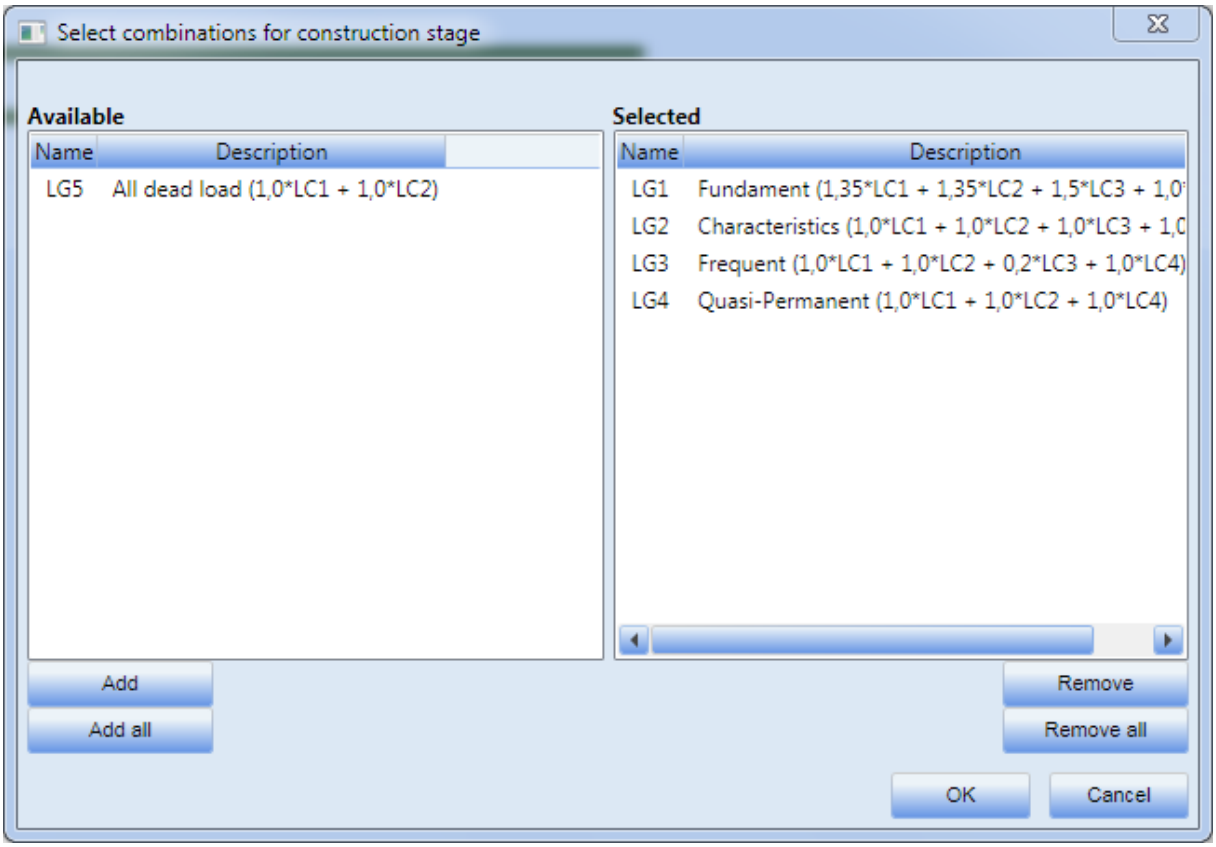
Combination must contain all permanent load cases, which have been applied in selected and all previous construction stages.

Combinations, which do not fulfil this rule, are not taken into account during checks in IDEA RCS.

Combinations applied in particular construction stages are listed in column **Combinations** in table **Construction stages**. To assign combinations to construction stage click edit button  in **Combinations** column.

Required load cases to be applied to construction stage can be selected in dialog **Select load cases for construction stage**.

Combinations, which can be assigned to construction stage, are listed in column **Available**. Combinations, which have been assigned to edited stage, are listed in column **Selected**



7.2.2 Assigning combinations to construction stages automatically

To assign combinations to construction stages automatically click **Reorder combi** in ribbon group **Construction stages**. Rule described in previous chapter is applied during automatic assignment.

7.3 Ribbon group Construction stages

Following commands are available in ribbon group **Construction stages**:

- **New** – create new construction stage
- **Delete** – delete selected construction stage.
- **Reorder combi –** assign combinations to construction stages automatically according to permanent load cases in construction stages.

8 Design members

Design member is basic entity to design tendons. Design member consists of one member or group of consecutive members of structural model.

- **If IDEA Tendon is launched from IDEA Beam or IDEA Frame, design members cannot be neither created, nor deleted. The list of members in design member cannot be modified too – the definition of design members is taken from the superior linked application.**

To input or edit design members click navigator command **Project data > Design members**.

Ribbon groups **Design member**, **Design member views**, **Uncoiled view**, **Calculate FEM**, **Check** and **Report** are available when working with design members.

Current design member uncoiled views are drawn in the Main window.

Table with list of defined design members is displayed in the Data window.

Table of design members contains following columns:

- **Name** –input name of design member.
- **Description** – input description of design member.
- **Members** – input list of members, which create design member.
- **Type** – set process of prestressing application on the design member:
  - **Pre-tensioned** – only pre-tensioned tendons can be defined on the design member
  - **Post-tensioned** – only post-tensioned tendon can be defined on the design member
  - **Pre/Post-tensioned** – both pre-tensioned and post-tensioned tendons can be defined on the design member.
- **Valid** – display design member validity status – it means if design member fulfils conditions to be created from defined list of members.
- **Value** – display extreme value of check from all positions checked on design member
- **Result status** – display overall status of design member check.
- **Print** – turn on/off design member to be printed in report.

Following columns are available for design member, which can be post-tensioned:

- **Stressing bed** – select stressing bed or edit properties of the current stressing bed.
- **Relative** – set mode, how to determine the position of section, where properties of prestressing units are defined.
- **Position** – input the distance of the section, where properties of prestressing units are defined, from the beginning of the design member

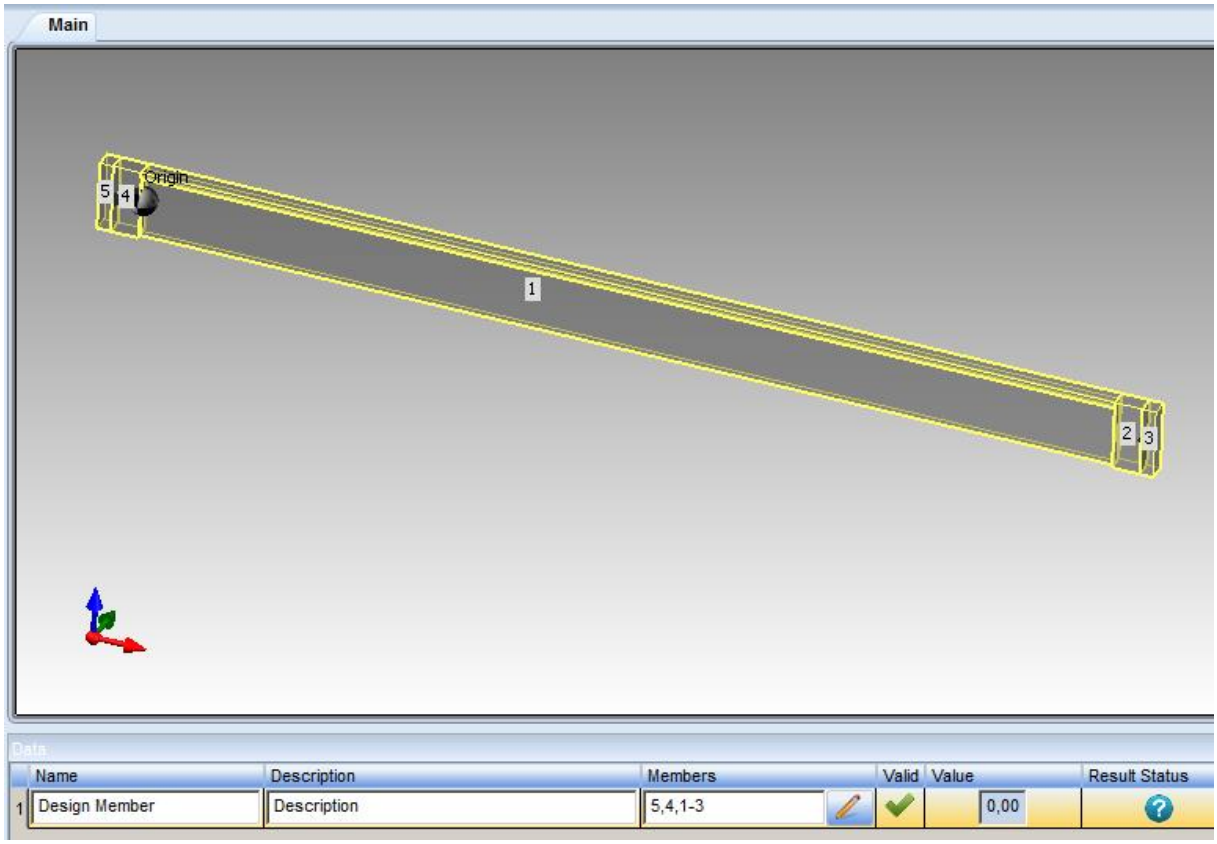
8.1 Creating Design member

To create new design member click **New** in ribbon group **Design member – see 8.1.4 Ribbon group Design member**. Newly created design member does not contain any members.

8.1.1 Creating Design member by input of member sequence

Design member can be defined by input of member numbers into edit box in column Members of table with design member properties. Numbers separated by comma or sequence defined by two numbers separated by dash can be entered – e.g. 1, 4-6 defines design member created by members 1, 4, 5, 6.

Number of members can be displayed in 3D structural view. To switch between uncoiled view and 3D structural view buttons in ribbon group **Design member views** can be used.**– see 8.1.5 Ribbon group Design member view**. Drawing of member numbers can be turned on in 3D structural view.



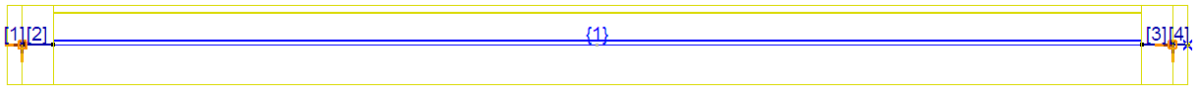
8.1.1.1 Syntax of member numbers for design member created from AxisVM

If IDEA Tendon is launched as plug-in from AxisVM, prefixes have to be used when creating design members from structural members of AxisVM analysis model to distinguish between standard members and eccentric members – ribs:

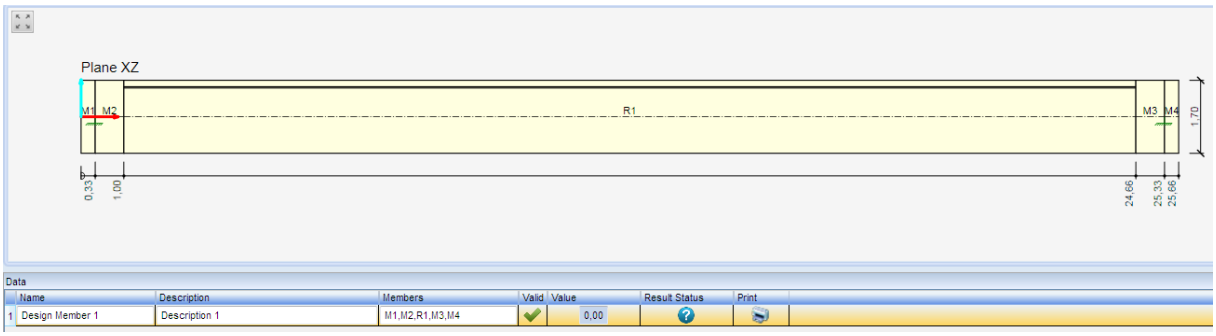
- To enter indexes of standard members use prefix M (e.g. ‚M1-5‘ or ‚M1, M3, M6‘ or ‚M1, 2,3‘)
- To enter indexes of ribs use prefix R (e.g. ‚R1-3‘)

Both prefixes can be combined in at once, e. g. ‚M1-2, R1, M3-4‘

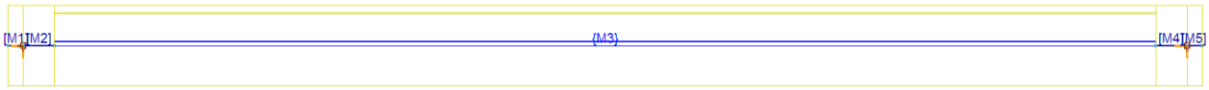
For older projects, created in AxisVM release 10, member names could have changed during synchronisation between AxisVM and IDEA Tendon – the prefix ‘M’ was added to the index of AxisVM structural member. In such case to add the standard member to the design member use format MM1’ and to add a rib member to the design member use format ‘RM1’.



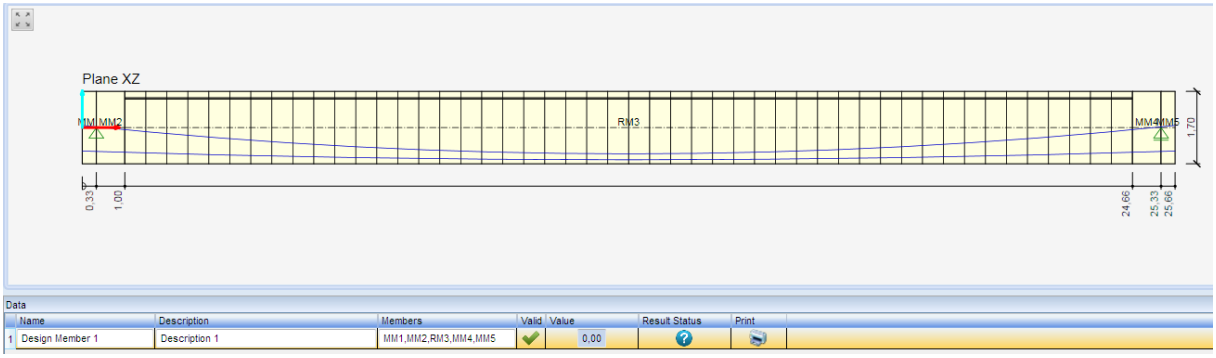
Model of beam (containing standard members and member with eccentricity - rib) created in AxisVM 11



Design member generated from AxisVM 11 data



Model of beam (containing standard members and member with eccentricity - rib) created in AxisVM 10



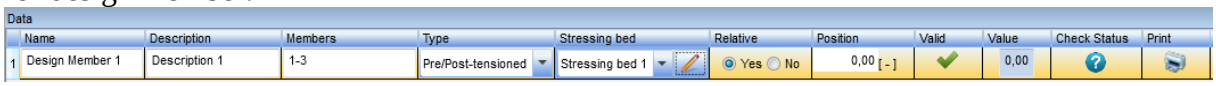
Design member generated from AxisVM 10 data

8.1.2 Assumptions to create Design member

When IDEA Tendon is launched for first time for particular project, application attempts to create new design member from imported members. All imported members are checked and if those members follow each other, one design member is created. Particular members do not need to lie in one line. Following rules are checked during design member creation:

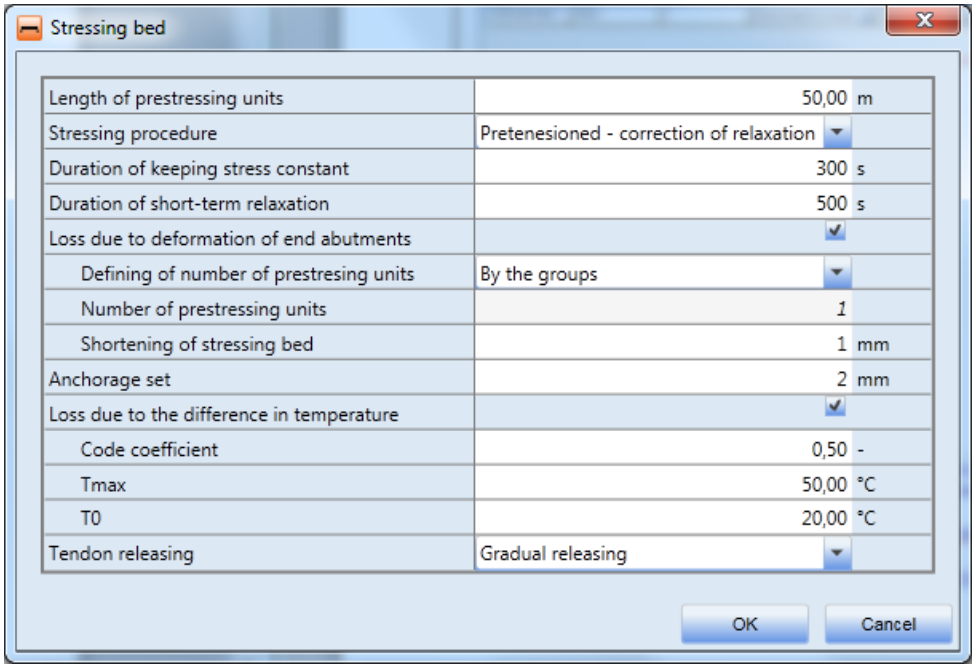
- The whole design member must be made of concrete. It means that all members in Design member must have assigned concrete cross-section.
- All members in design member must have the same orientation. It means that local x-axes of two consecutive members must not be oriented against each other – in other words two members in one design member cannot have common ending point.
- Beginning node of following member must be finishing node of current member.
- Design member for pre-tensioning must be straight and statically determined.

When IDEA Tendon is re-launched for the same project, new design member is not created. Particular members of existing design members are checked, whether geometry or material has been changed or whether has been deleted in superior application. Validity of design member is displayed in column Valid in design member properties table or in **Info window** for design member.



8.1.3 Stressing bed properties

To edit the properties of stressing bed click edit button  in column Stressing bed.



- **Length of prestressing units** – input the length of prestressing tendons between the anchors of anchorage abutments.
- **Stressing procedure** – choose the procedure of stressing, anchoring and transfer of prestressing.
- **Duration of keeping stress constant** – input the time of keeping stress constant during correction of relaxation

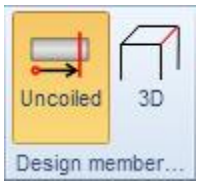
- **Duration of short-term relaxation** – input the time of short-term relaxation before transfer of prestressing.
- **Loss due to deformation of end abutments** – turn on/off the calculation of losses due to deformation of end abutments of the stressing bed.
    - **Defining the number of prestressing units** – choose the mode, how to determine the number of prestressing units for calculation of losses.
    - **Number of prestressing units** – for user defined number of prestressing units input the number of prestressing units
    - **Shortening of stressing bed** – input the value of shortening of stressing bed due to stressing of all prestressing units.
- **Anchorage set** – input the value of anchorage set.
- **Loss due to the difference in temperature** – turn on/off the calculation of losses due to the temperature difference of the prestressing units and the stressing bed.
    - **Code coefficient** – the value of coefficient in equation (10.3) of EN 1992
    - **Tmax** – input the value of maximal temperature of concrete near the tendons.
    - **T0** – input the initial temperature of concrete near the tendons
- **Tendon releasing** – set mode of tendon releasing.

8.1.4 Ribbon group Design member

Following commands are available in ribbon group **Construction stages**:

- **New** – create new empty design member. New design member is added to table of design members. Newly created design member is set as current design member.
- **Copy** – copy the whole design member
- **Delete** – delete current design member including all defined tendons

8.1.5 Ribbon group Design member view

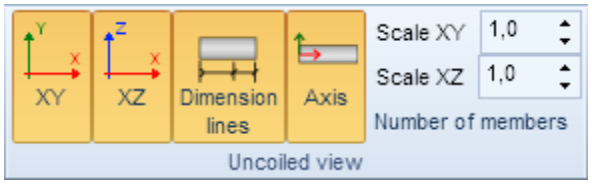


Following commands are available in ribbon group Design member view:

- **Uncoiled** – draw uncoiled views of current design member according to current settings. Uncoiled views display current design member in planes XY and XZ.
- **3D** – draw 3D view of the whole imported structure.

8.1.6 Ribbon group Uncoiled view

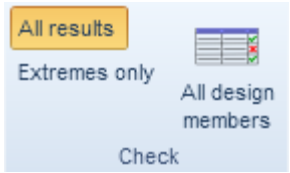
This ribbon group is available only if the view is set to **Uncoiled** in ribbon group **Design member view**.



- **XY**- turn on/off drawing of uncoiled view in plane XY
- **XZ** – turn on/off drawing of uncoiled view in plane XZ
- **Dimension lines** – turn on/off drawing of design member dimension lines in uncoiled views.
- **Axis** – turn on/off drawing of design member axis in uncoiled views
- **Scale XY, XZ** – value of exceeded scale of y-axis (or z-axis) in uncoiled view XY (or XZ). Exceeded scale enables more clear drawing of tendons and shapes of long design members. The scale of x-axis always equals 1.
- **Number of members** – turn on/off display of member numbers in uncoiled views.

8.1.7 Ribbon group Check

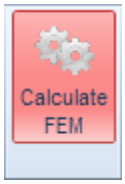
➢ **If IDEA Tendon is launched from IDEA Beam or IDEA Frame, the checks cannot be performed in IDEA Tendon. Checks are performed in superior linked application.**



Following commands are available in ribbon group **Check**:

- **All results** – if this option is on, check of design members is performed for all combinations of internal forces.
- **Extremes only** – if this option is on, check of design members is performed only for extreme values of internal forces. Extreme sextuplets are searched from all combinations (cases) in associated result class. Maximal and minimal values are searched for each component of internal forces and all corresponding values are stored with the extreme value.
- **All design members** - run check of all positions in all design members.

8.1.8 Ribbon group Calculate FEM



To recalculate internal forces in superior application click button **Recalculate FEM**. Background of the button is set to red colour, if the recalculation is required – for example after geometry of tendon changes.

8.1.9 3D view of the structure

If 3D view of the structure is active, following ribbon groups are available: **Structure views**, **3D views**, **Structure labels** and **Member LCS**.

8.1.9.1 Manipulating 3D view

To set the required view point in 3D window use commands in right top corner of 3D window or keyboard shortcuts with mouse keys.

Commands in 3D window:

 - zoom window. Click this button and drag mouse with holding left mouse button to draw window to zoom.

 - increasing/decreasing view. Click this button and drag mouse with holding left mouse button to increase/decrease the view.

 - pan the view. Click this button and drag mouse with holding left mouse button to pan the view.

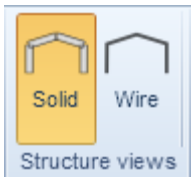
 - rotate the view. Click this button and drag mouse with holding left mouse button to rotate the view.

 - zoom all. Click this button to fit the whole structure to the 3D window.

To set the required view using keyboard and mouse following combinations can be used:

- Click and hold mid mouse button – moving the mouse pans the view.
- push CTRL and hold mid mouse button – moving the mouse rotates the view
- push SHIFT and hold mid mouse button – moving the mouse increases/decreases the view

8.1.10 Ribbon group Structure views



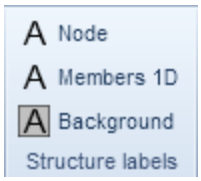
- **Solid** – draw all structural members as solids
- **Wire** – draw all structural members as wires

8.1.11 Ribbon group 3D views

- **-Z** – set the view from the top of the structure (opposite the positive Z-semi-axis of global coordinate system)
- **Y** - set the view from the front of the structure (opposite the positive Y-semi-axis of global coordinate system)
- **-X** - set the view from the side of the structure (opposite the positive X-semi-axis of global coordinate system)
- **Axo** – set the default 3D view point and performs zoom to fit the whole structure into 3D window

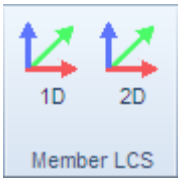
- **Zoom** – perform zoom to fit the current member or design group into the 3D window.
- **Persp.** – turn on/off the perspective view on the structure.

8.1.12 Ribbon group Structure labels



- **Node** – switches on/off drawing of node numbers.
- **Members 1D** – switches on/off drawing of numbers of 1D members
- **Background** – switches on/of drawing of background under numbers.

8.1.13 Ribbon group Member LCS



- **1D** – turn on/off drawing of local coordinate systems on 1D members
- **2D** – turn on/off drawing of local coordinate systems on 2D members

9 Tendon geometry

9.1 3D tendon geometry

Tendon geometry is created from so called definition geometry. **Tendon Definition geometry** DGY or DGZ is tendon geometry defined in **uncoiled view** YX (or XZ) of Design member. Definition geometry in XZ-plane (or XY-plane) is defined as horizontal (or vertical) projection of tendon transformed by uncoiling of reference curve to XZ-plane (or XY-plane) of coordinate system of uncoiled view.

Two types of tendon definitions can be used:

- **Segments** – the tendon is created using single segments, which are defined by coordinates of characteristics points
- **Polygons** – the tendon is created using polygons, which are defined by coordinates of polygon vertexes.

Both definition geometries are created independently, respecting following rules:

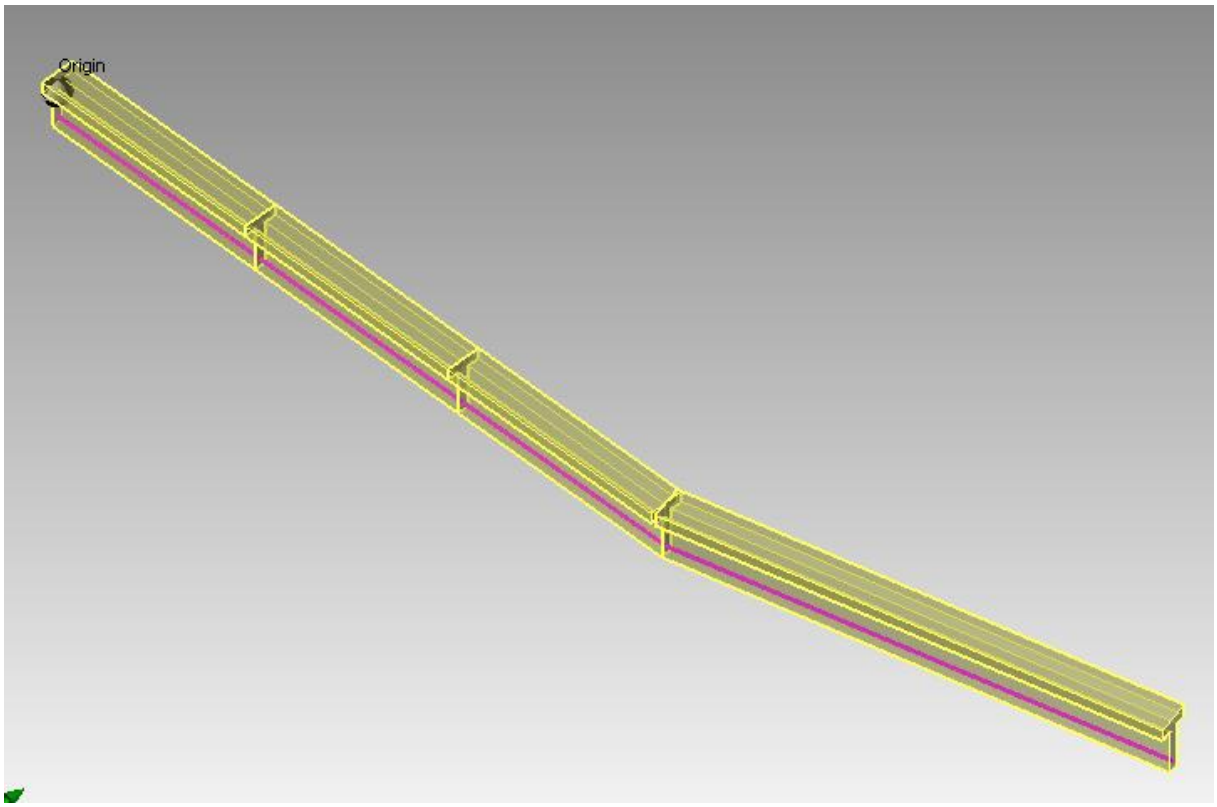
- x-coordinates of beginning points of both definition geometries are identical
- x-coordinates of end points of both definition geometries are identical

Both segments and polygons are defined by characteristic points. Definition geometries carry information about e.g. arc diameters, tangent lengths or angular changes in polygon vertexes.

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

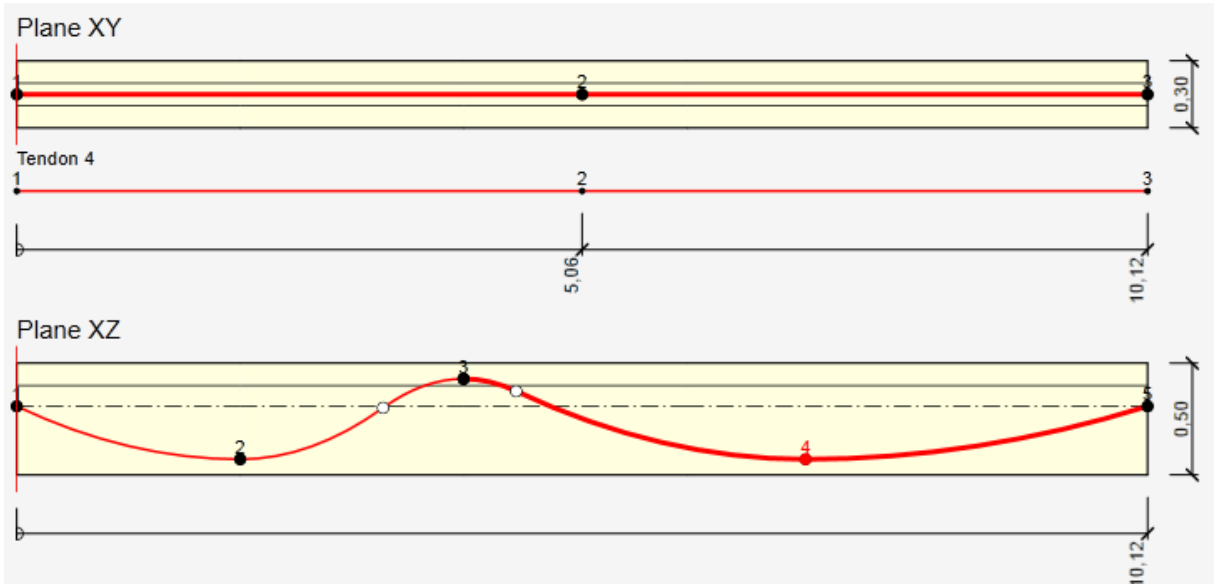
Example 1:

Design member uncoiled views in planes XY and XZ including straight tendon.

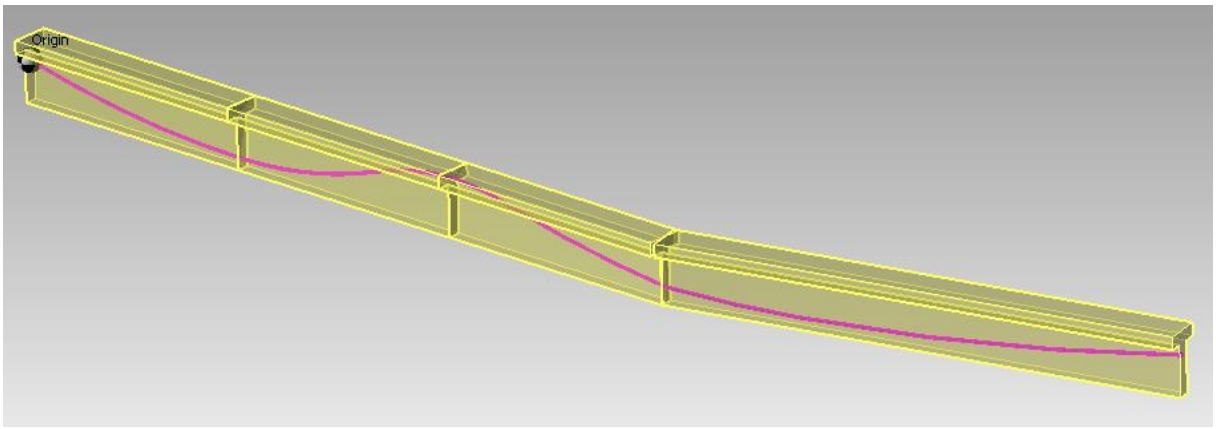


Final 3D tendon geometry

Example 2:



Design member uncoiled view XZ including parabolic tendon.



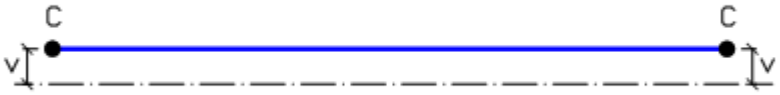
Final 3D tendon geometry

9.2 Description of tendon definition geometry segments

9.2.1 Type of segments to define tendon geometry

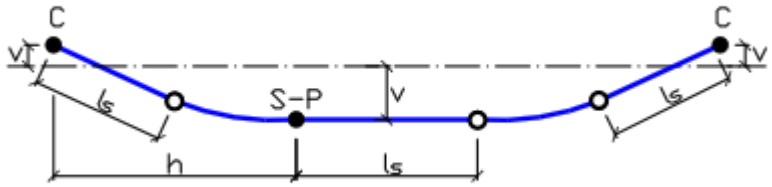
7 segment types can be used to define geometry. Their usage depends on segment position in tendon geometry to keep continuity of particular segments and termination of tendon too.

9.2.1.1 Segment type 1 – Straight stand-alone



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

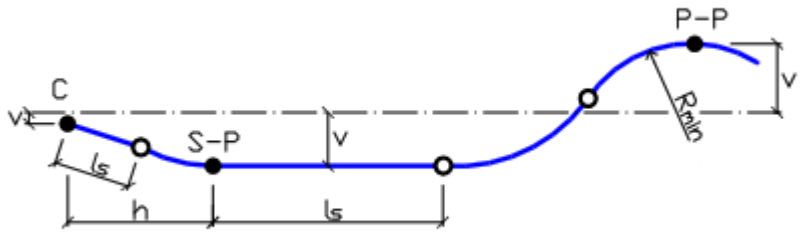
9.2.1.2 Segment type 2 – Parabolic with straight stand-alone



Default segment for new tendon. Neither this type can be connected to other segment. But if the segment is split, it is automatically replaced by corresponding segment, which enables to connect other segment. Geometrically it consists of three curves (parabola, straight line and parabola). Straight line can be omitted. If parameters of parabolic part define straight line, the straight line is used instead of the parabola. Straight lines can replace the appropriate part of parabola at the segment beginning or at the segment ending.

Segment is defined using two **C** points and intermediate **S-P** point (**S**traight-**P**arabolic – intermediate point between straight and parabolic component). Position of **S-P** point is defined by distance **h** from the left or right ending point or from the centre of segment and by distance **v** the member reference curve in plane XY or XZ. The distance **ls** is the length of straight part between parabolas. Coordinates of white-filled points in the picture are not entered, but calculated from entered parameters.

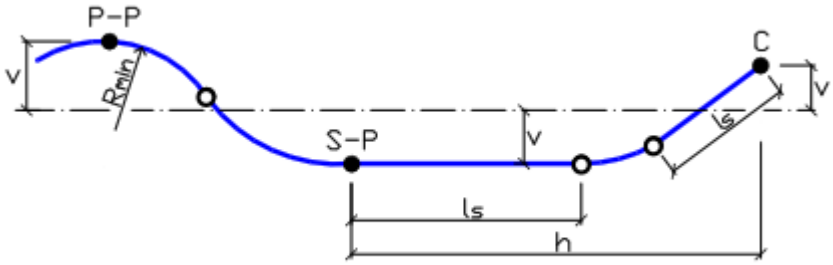
9.2.1.3 Segment type 3 - Parabolic with straight, closing left



This type can be used as a beginning tendon segment and next follow-up segment can be connected to it. This segment consists of up to five curves – straight line, parabola, straight line, parabola and parabola. Beginning straight line can be omitted. Beginning parabola can be partially replaced with straight part. Last two parabolas are rotated against each other by 180 degrees.

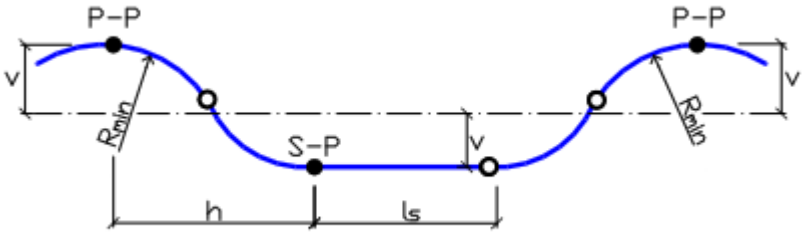
Segment is defined using beginning **C** point, intermediate **S-P** point and **P-P** point (**P**arabolic–**P**arabolic – connecting point between two parabolas). P-P point describes the transition from segment of type 3 to follow-up segment. Position of **P-P** is defined by distance v from the beam reference curve and minimal radius of parabolas.

9.2.1.4 Segment type 4 – Parabolic with straight, closing right



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

9.2.1.5 Segment type 5 – Parabolic with straight inner

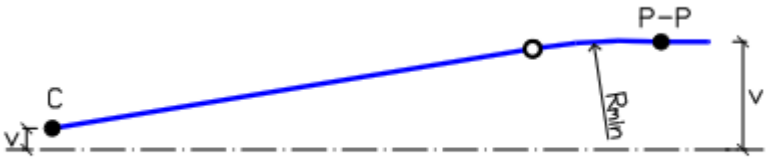


Segment type 5 can be placed only between two other segments, so it is inner tendon segment. The segment consists of five curves - parabola, parabola, straight line, parabola and parabola.

Straight part can be omitted, also the parabolas can change to lines according to entered parameters.

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

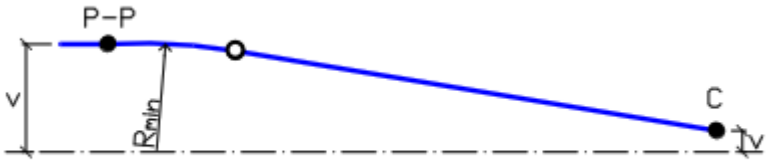
9.2.1.6 Segment type 6 – Straight closing left



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

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

9.2.1.7 Segment type 7 – Straight closing right



Segment type 7 is mirror type to segment type 6 and can be used as last tendon segment, which follows the previous segments.

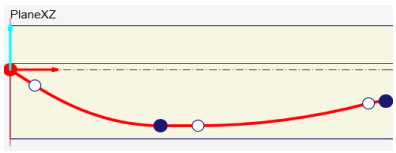
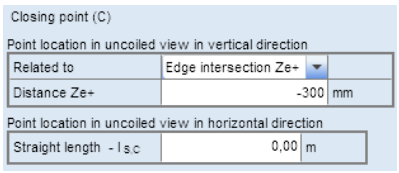
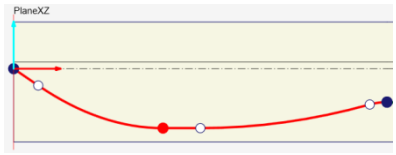
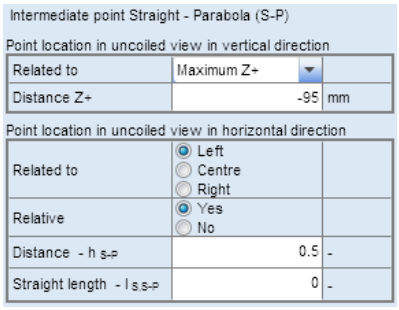
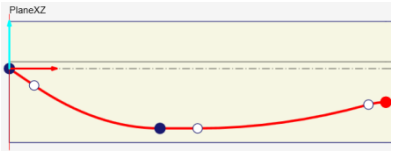
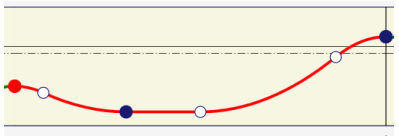
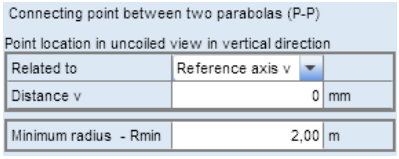
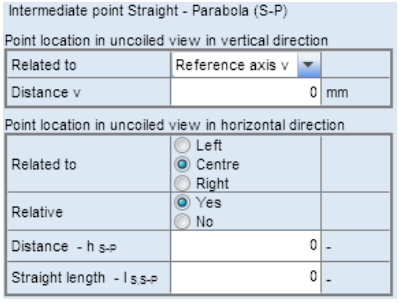
9.2.2 Rules and limitations for segments definition

All segment types listed above have following limitations:

- Neighbouring segments have common tangent at the segments border (**P-P** point). Tangent of angle of this tangent equals zero in current version (tangent is parallel to x-axis).
- Minimal radius of parabola in **P-P** point has the same values for both parabolas from the left and from the right.
- Inner straight parts of segments are always parallel to the reference curve of the member. This is not true for straight closing segments and standalone straight segment.

9.2.3 Detailed description of geometrical properties of definition geometries

Segment	Editing	Description

Segment	Editing	Description
 Type 3 in the picture. Mirrored type 4. First half is identical with type 2 including straight part, ending should be mirrored to the beginning.		Closing point. Vertical offset to bottom or top edge or centre can be defined. Length of straight part specifies initial length of the straight line. From the straight line parabola continues in tangent.
 Type 3		Intermediate point of segment. In fact it is pair of points, which define straight part of segment. Value of vertical offset, relative beginning and length of straight part can be defined. If length is zero, straight part is missing. For this pair of points, the parabola (from the left or from the right) has its vertex in this point.
 Type 3  Type 5		Connecting point. This point type is used in connection of two segments. Vertical offset and minimal radius of connected parabolas can be defined. Both parabolas have vertex in this point and tangent in this vertex is parallel to x-axis of member. Both parabolas are created with the minimal diameter specified and they are continued with “inverted” parabolas, which end in vertex in inner point.
 Type 5		Intermediate point, identical as at types 2, 3, 4.

Segment	Editing	Description
 Type 7, mirrored type 6	Connecting point between two parabolas (P-P) Point location in uncoiled view in vertical direction Related to: Reference axis v Distance v: 0 mm Minimum radius - Rmin: 2,00 m	Connecting point with identical properties as at types 3 and 5. The difference is that the parabola for segments of type 6 and 7 does not continue with "inverted" parabola but with straight line which ends in closing point. Thus the length of following straight line cannot be specified directly, but it depends on defined minimal radius of parabola.
 Type 7	Closing point (C) Point location in uncoiled view in vertical direction Related to: Reference axis v Distance v: 0 mm Point location in uncoiled view in horizontal direction Straight length - l s,c: 1,25 m	Closing point for segments 6 and 7. Vertical offset to top or bottom edge or centre can be specified. Length of straight part is meaningless.

9.2.4 Description of tendon definition geometry points

Geometry of each tendon segment is defined by two or three characteristic points, depending on type of segment. Those points are drawn as black filled circle. Current point (selected to be edited) is drawn as red filled circle. Points can be selected in the picture by mouse.

Other points, which are necessary to define the geometry, are calculated automatically. Those points are for example points at the endings of straight tendon segments or points in the transition between inverted parabolas. Those points are drawn with black circle filled with white colour and they cannot be selected and edited. Their position depends e.g. on defined length of straight parts.

9.2.4.1 Point „C" – tendon closing point

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

Closing point (C)

Point location in uncoiled view in vertical direction

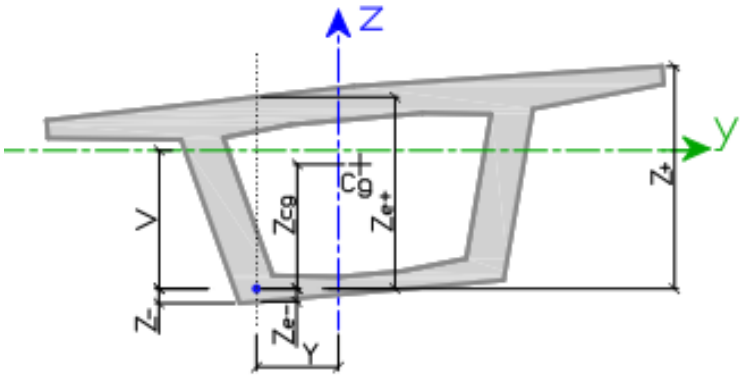
Related to	Reference axis v	
Distance v	0	mm

Point location in uncoiled view in horizontal direction

Straight length - l s,c	0,00	m

- **Related to** – specify the origin for determination of final vertical tendon point coordinate **v**. Following options can be chosen (for example in plane XZ – see following picture) :

- Maximal positive coordinate Z – **Maximum Z+,**
- Maximal positive Z+ coordinate (in local coordinate system of design member) of the intersection of the line parallel to Z axis drawn in Y coordinate of tendon with the edge of cross-section – **Edge intersection Ze+,**
- Distance from reference axis – **Reference axis v**,
- Vertical distance from the centre of gravity – **Centre of gravity Zcg**
- Minimal negative Z- coordinate (in local coordinate system of design member) of the intersection of the line parallel to Z axis drawn in Y coordinate of tendon with the edge of cross-section – **Edge intersection Ze-,**
- Minimal negative coordinate Z – **Minimum Z-,**



- **Distance** – point distance measured from defined origin, positive value is in positive direction of beam Z-axis (Y-axis)
- **Straight length $l_{s,c}$** - length of straight part of tendon measured from beginning (ending) point of segment

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

S-P point is always inner point of tendon segment. Point properties can be edited in following table:

Intermediate point Straight - Parabola (S-P)

Point location in uncoiled view in vertical direction

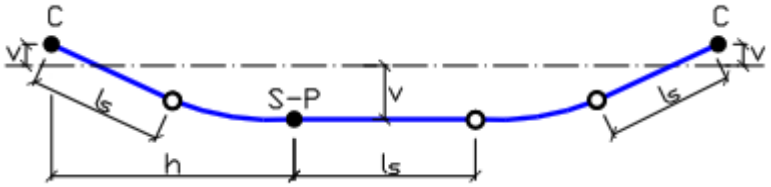
Related to	Reference axis v	
Distance v	0	mm

Point location in uncoiled view in horizontal direction

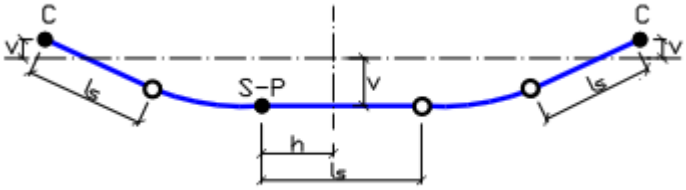
Related to	○ Left ◉ Centre ○ Right	
Relative	◉ Yes ○ No	
Distance - h_{s-p}	0	-
Straight length - $l_{s,s-p}$	0	-

- **Related to** – specify the origin for determination of final vertical tendon point coordinate **v** – see previous chapter,
- **Distance** – point distance measured from defined origin, positive value is in positive direction of beam Z-axis (Y-axis),
- **Related to** – specify the origin for input of horizontal point position. Following points can be used as reference point for horizontal coordinate **h_{S-P}**:

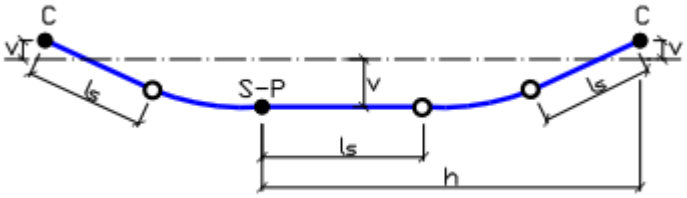
- Beginning point of segment



- Middle of segment



- Ending point of segment



- **Relative** – switch of input mode for input of distance h and straight length ls
- **Distance h $_{S-P}$ –** horizontal distance h of selected point.
- **Straight length l $_{S,S-P}$** – length of inner straight part of tendon

9.2.4.3 Point „P-P“ – connecting point between parabolas

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

Connecting point between two parabolas (P-P)

Point location in uncoiled view in vertical direction

Related to	Reference axis v	
Distance v	0	mm
Minimum radius - Rmin	2,00	m

- **Related to** – specify the origin for determination of final vertical tendon point coordinate **v** – see previous chapter,
- **Distance** – point distance measured from defined origin, positive value is in positive direction of beam Z-axis (Y-axis)
- **Minimum radius Rmin** – minimal radius of parabola

9.2.5 Composition of segments to create tendon geometry in uncoiled view

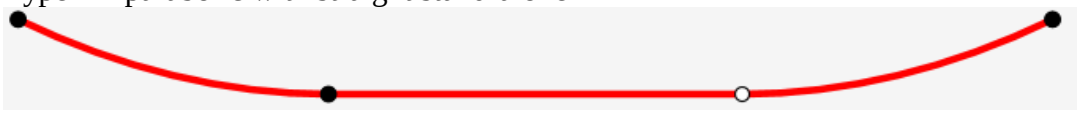
Possibilities how to compose tendon geometry using several numbers of segments, are described in following chapters.

9.2.5.1 Tendon consisting of one segment

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

- Type 1 – straight stand-alone

- Type 2 – parabolic with straight stand-alone



None of those two segments can be combined with another segment type.

9.2.5.2 Tendon consisting of two segments

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

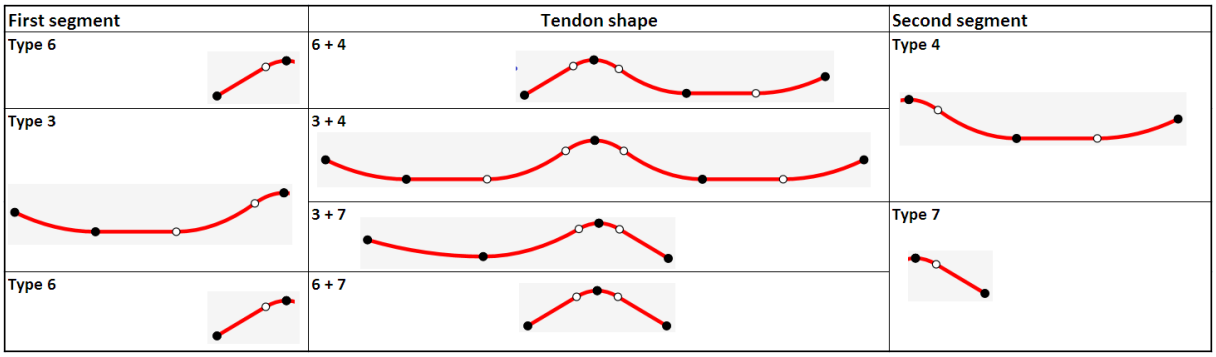
Following segment types can be used for first segment:

- Type 3 – parabolic with straight left
- Type 6 – straight closing left

Following segment types can be used for second segment:

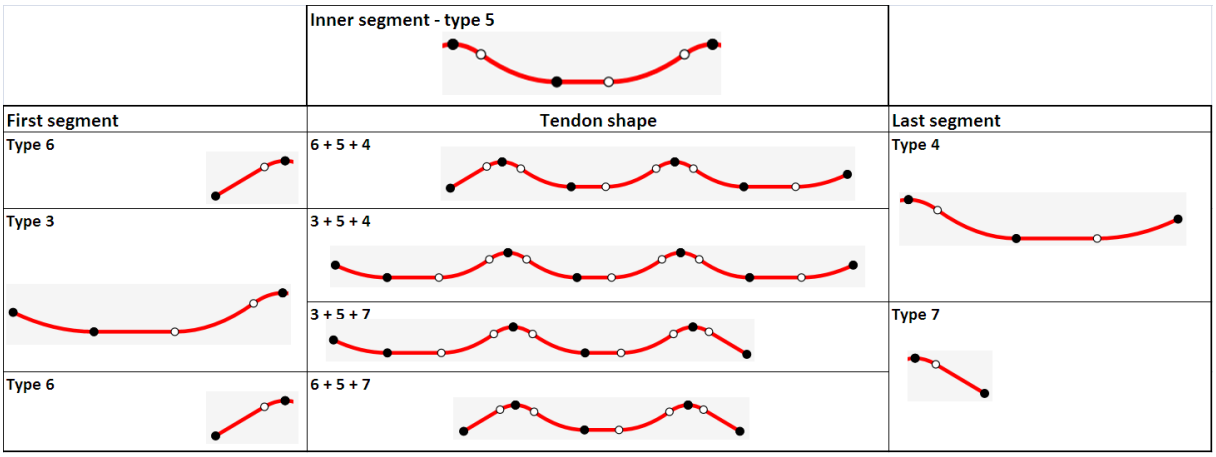
- Type 4 – parabolic with straight right
- Type 6 – straight closing right

Possible combinations of segment types are displayed in following table:



9.2.5.3 Tendon containing three and more segments

Geometry of tendon, consisting of three and more segments, is composed similarly as geometry of tendon consisting of two segments. Identical segment types can be used for outer segments, for internal segment(s) only segment of type 5 can be used. Possible combinations of segment types are displayed in following table:


Inner segment - type 5
First segment
Tendon shape
Last segment
Type 6
6 + 5 + 4
Type 4
Type 3
3 + 5 + 4
3 + 5 + 7
Type 7
Type 6
6 + 5 + 7

9.3 Tendons input and edit

To input and edit tendons click navigator command **Tendons > Tendons layout.**

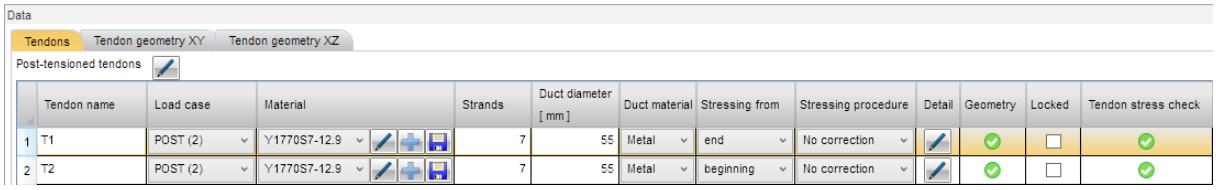
Uncoiled views of current design member are drawn in the **Main window**.

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

- **Tendons** – properties of tendons in current design member. After tendon properties change the equivalent forces are updated automatically.
- **Tendon geometry XY** – editing of post-tensioned tendon geometry in uncoiled view XY.
- **Tendon geometry XZ** – editing of post-tensioned tendon geometry in uncoiled view XZ.
- **Pretensioned group** – editing properties of the group of pre-tensioned tendons in the cross-section, which corresponds with the section in the position defined in design member properties.

9.3.1 Tendon properties

Properties of post-tensioned tendons can be edited in the table **Post-tensioned tendons** on tab **Tendons** in the Data window.



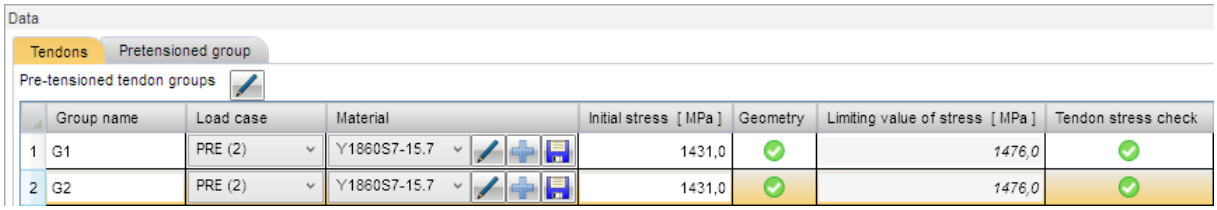
The table **Post-tensioned tendons** contains following columns:

- **Tendon name** – edit name of tendon.
- **Load case** – display name of appropriate load case, to which the prestressing effects from tendon are transferred.
- **Material** – select current tendon material from materials in the project. Required materials must be added to the project from the materials library.
  -  - start modification of properties of current material.
  -  - add new material to the project selecting from materials library.
  -  - store current material to the user materials library.
- **Strands** – number of strands in tendon.
- **Duct diameter** – value of minimal duct diameter. Default value of minimal duct diameter is calculated according to area of tendon.
- **Duct material –** select material of tendon duct. Two materials are available – Plastic or Metal.
- **Stressing from** – select stressing mode. Tendon can be stressed from the beginning of design member, from the ending of design member or from both ends of design member is specified order.
- **Stressing procedure –** select stressing procedure. Stressing procedure with or without correction of relaxation can be selected.
- **Detail** – click edit button to display dialog with detailed tendon properties.
- **Geometry** – display status of tendon geometry. Result value depends on partial results of checks of all tendon segments in both uncoiled views. If tendon

geometry is not valid, tendon losses cannot be calculated and corresponding design member cannot be checked.

- **Locked** – if the switch is on, tendon is locked and tendon properties cannot be edited.
- **Tendon stress check** – display result of maximal stress in tendon check according to EN 1992-1-1 5.10.2.1(1)P.

Properties of pre-tensioned tendons can be edited in the table **Pre-tensioned tendons** on tab **Tendons** in the Data window.

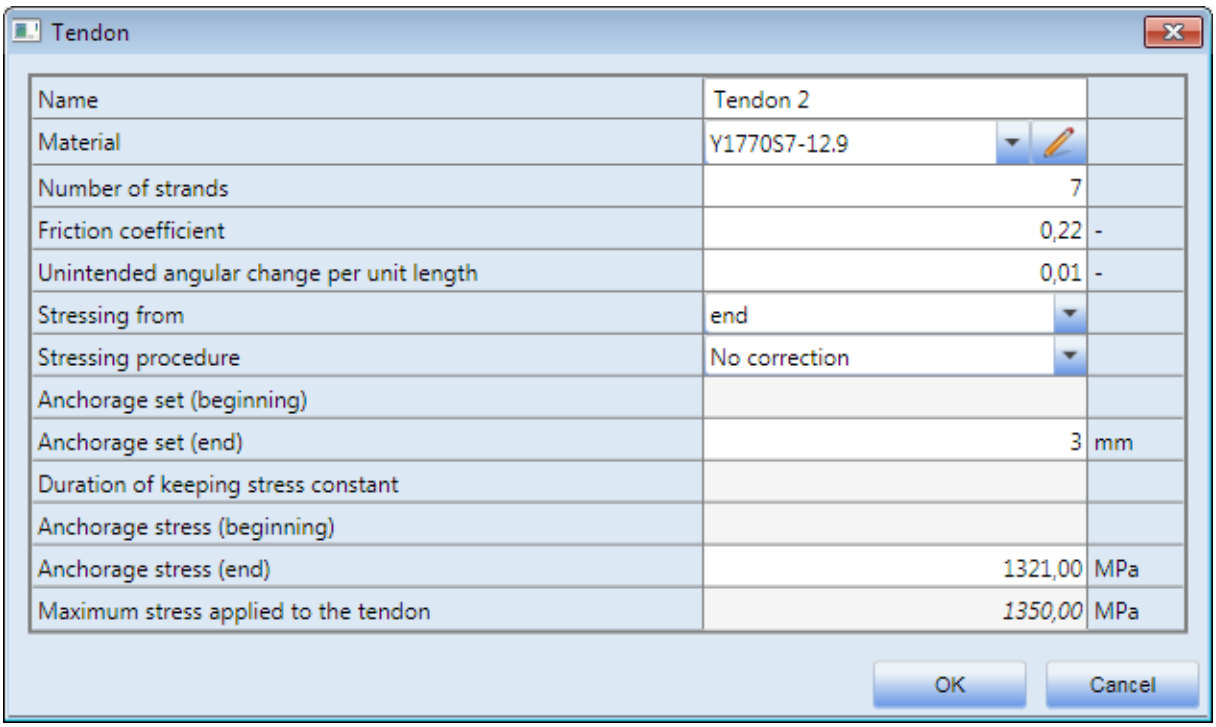


The table **Pre-tensioned tendons** contains following columns:

- **Tendon name** – edit name of tendon.
- **Load case** – display name of appropriate load case, to which the prestressing effects from tendon are transferred.
- **Material** – select material of current prestressing group from materials in the project. Required materials must be added to the project from the materials library.
  -  - start modification of properties of current material.
  -  - add new material to the project selecting from materials library.
  -  - store current material to the user materials library.
- **Initial stress** – edit value of initial tendon stress.
- **Geometry** – display status of tendon geometry. Result value depends on partial results of checks of all tendon segments in both uncoiled views. If tendon geometry is not valid, tendon losses cannot be calculated and corresponding design member cannot be checked.
- **Limiting value of stress** – display the value of maximal stress applied in the tendon.
- **Tendon stress check** – display result of maximal stress in tendon check according to EN 1992-1-1 5.10.2.1(1)P.

9.3.1.1 Detailed tendon properties

To edit detailed tendon properties click edit button  in column **Detail** in table of tendon properties.

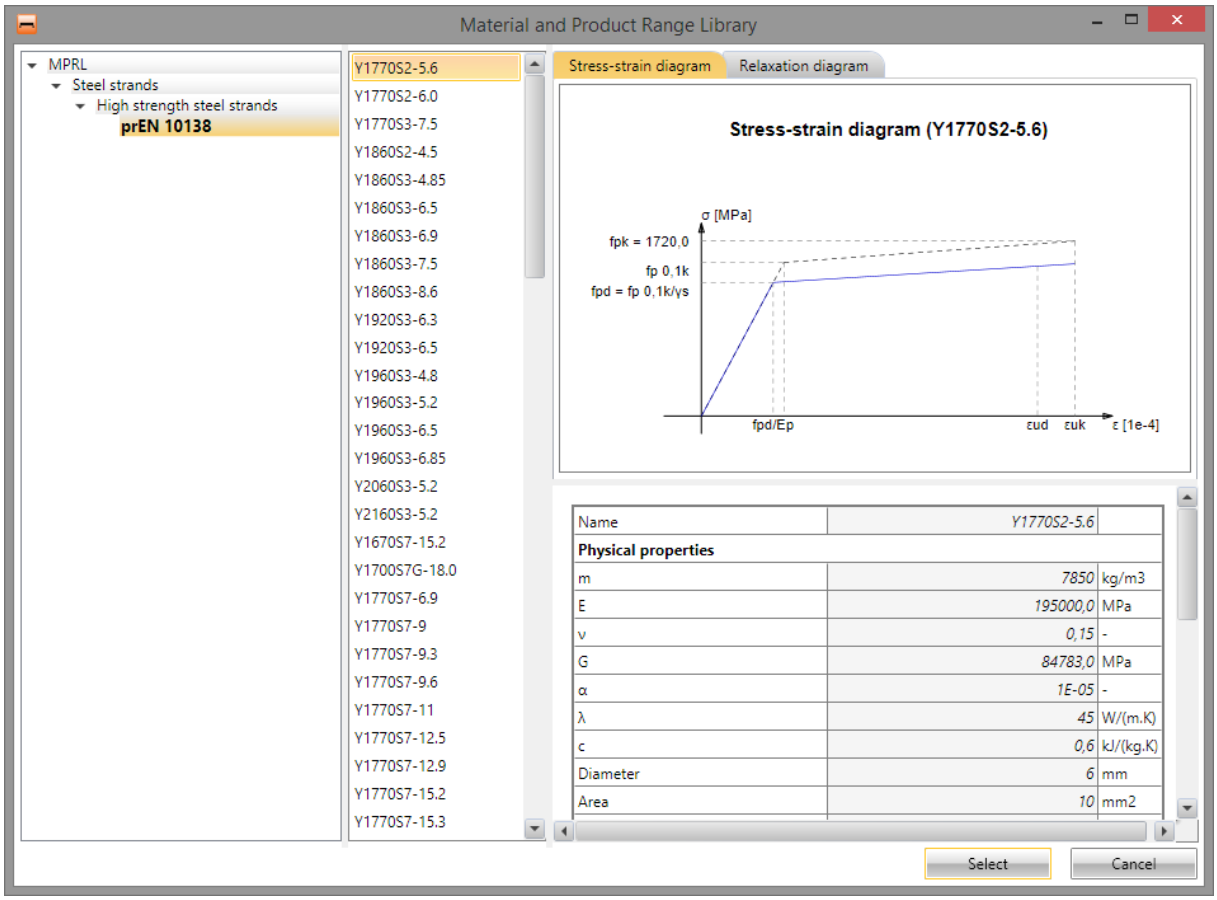


Particular dialog options of **Tendon** dialog:

- **Name** – name of edited tendon.
- **Material** – select current material of tendon. Click edit button  to change properties of current material.
- **Number of strands** – number of strands in tendon.
- **Friction coefficient** – value of tendon friction coefficient.
- **Unintended angular change per unit length** – value expressing the increase of tendon friction losses due to not intended ripple of tendon.
- **Stressing from** – select stressing mode. Tendon can be stressed from the beginning of design member, from the ending of design member or from both ends of design member is specified order.
- **Stressing procedure –** select stressing procedure. Stressing procedure with or without correction of relaxation can be selected.
- **Anchorage set (begin)** – value of anchorage set at the beginning of the tendon. Value is available if stressing from begin is set.
- **Anchorage set (end)** – value of anchorage set at the ending of the tendon Value is available if stressing from end is set.
- **Duration of keeping stress constant** – value of time to keep the stress constant during stressing. Value is available only for stressing with correction of relaxation
- **Anchorage stress (begin)** – value of anchorage stress at the beginning of the tendon. Value is available if stressing from begin is set.
- **Anchorage stress (end)** – value of anchorage stress at the ending of the tendon. Value is available if stressing from end is set.
- **Maximal stress applied in tendon** – value of maximal stress in tendon.

9.3.1.2 Adding prestressing material into the project

To add new prestressing material into the project click  next to the materials drop-down in the tendon properties table.

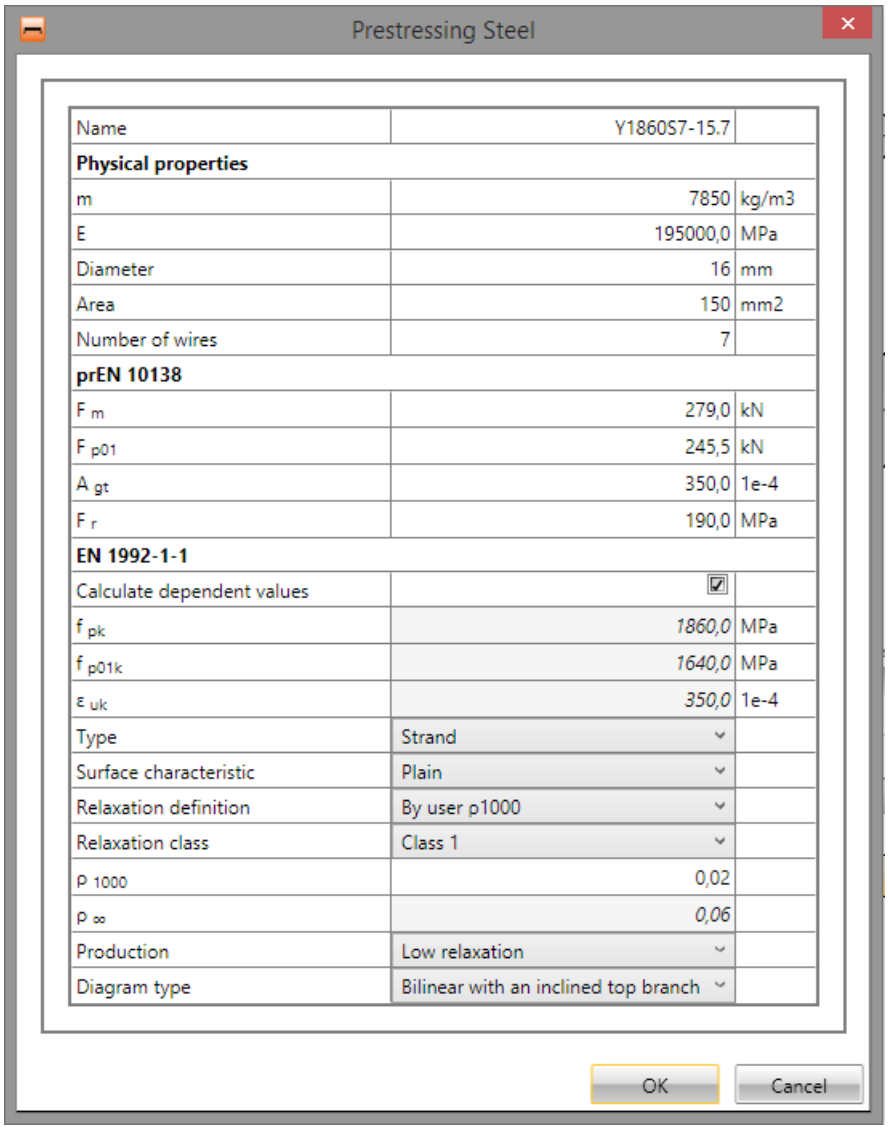


Select the required pre-stressing material in the **Material and product range library** dialog. Material properties of the selected material and the stress-strain diagram are printed on **Stress-strain diagram** tab. The course of final relaxation loss related to the relative tendon stress is printed on the tab Relaxation diagram.

Click **Select** to add the selected material into the project.

9.3.1.3 Modifying the prestressing reinforcement

To modify properties of prestressing material  next to the materials drop-down in the tendon properties table.



Properties of prestressing reinforcement:

- **Name** – input name of prestressing reinforcement material.

Properties group **Physical properties**

- **m** – input unit weigh of prestressing material.
- **E** – input modulus of elasticity of prestressing material.
- **Diameter** – input nominal diameter of prestressing material.
- **Area** – input area of prestressing material.
- **Number of wires** – input number of wires of prestressing reinforcement.

Group **prEN1038**

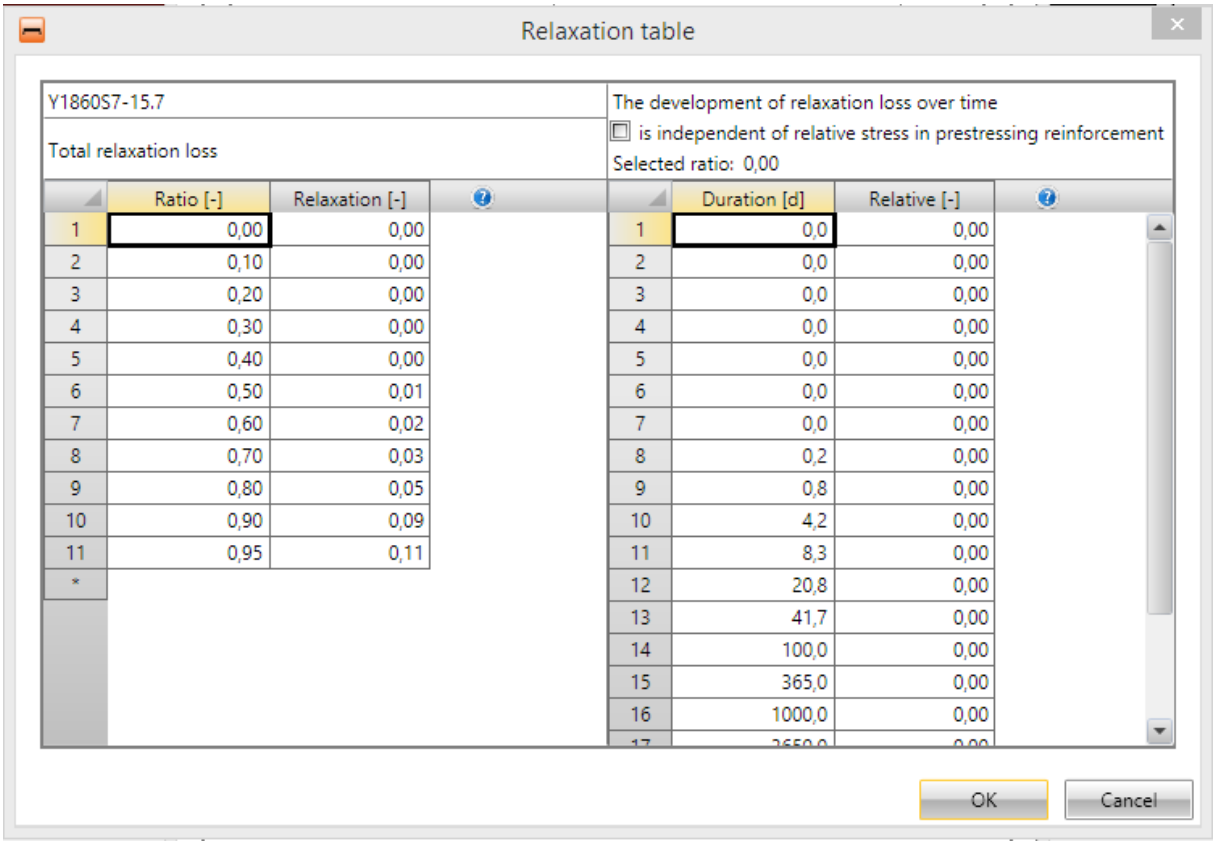
- **Fm** – input characteristic value of maximum force.
- **Fp0.1** – input characteristic value of 0.1% proof force.
- **Agt** – input total elongation at maximum force.

- **Fr** – input fatigue stress range.

Group **EN-1992-1-1**

- **Calculate dependent values** – if selected, values of fpk, fp01k and εuk are calculated automatically Otherwise the values can be defined by user.
- **fpk** – input characteristic tensile strength.
- **fp01k** – input characteristic 0.1% proof stress.
- **εuk** – input characteristic strain of reinforcement at maximum load.
- **Type** – select type of prestressing reinforcement:
  - **Plain round wire**
  - **Indented wire**
  - **Strand**
  - **Compacted strand**
  - **Plain round bar**
  - **Ribbed bar**
- **Surface characteristic** – select type of surface of prestressing reinforcement:
  - **Plain**
  - **Indented**
  - **Ribbed**
- **Relaxation definition** – select mode to determine the relaxation of prestressing reinforcement:
  - **By code** – relaxation is determined according to the national code.
  - **By user ρ1000** – relaxation is determined according to the selected relaxation class and the defined ratio of relaxation loss at 1000 hours after tensioning.
  - **By user table** – relaxation is defined by the user defined table.
    - **Edit** – starts editor of user defined relaxation table.
- **Relaxation class** – displays/select current relaxation class.
- **ρ1000 –** input / displays calculated relaxation loss ratio at 1000 hours after tensioning.
- **ρ∞** - displays calculated relaxation loss ratio at 50000 hours after tensioning.
- **Production** – select production method of prestressing reinforcement:
  - **Hot rolled and processed**
  - **Patented**
  - **Cold drawn**
  - **Stress relieved**
  - **Low relaxation**
- **Diagram type** – select type of stress-strain diagram of prestressing reinforcement:
  - **Bilinear with an inclined top branch**
  - **Bilinear with horizontal top branch**

9.3.1.4 User defined relaxation table of prestressing reinforcement



	Ratio [-]	Relaxation [-]
1	0,00	0,00
2	0,10	0,00
3	0,20	0,00
4	0,30	0,00
5	0,40	0,00
6	0,50	0,01
7	0,60	0,02
8	0,70	0,03
9	0,80	0,05
10	0,90	0,09
11	0,95	0,11
*		

	Duration [d]	Relative [-]
1	0,0	0,00
2	0,0	0,00
3	0,0	0,00
4	0,0	0,00
5	0,0	0,00
6	0,0	0,00
7	0,0	0,00
8	0,2	0,00
9	0,8	0,00
10	4,2	0,00
11	8,3	0,00
12	20,8	0,00
13	41,7	0,00
14	100,0	0,00
15	365,0	0,00
16	1000,0	0,00

Ratios of the total loss of stress for relative stress in tendon are defined in the left table.

- **Ratio** – the ratio of stress in tendon and characteristic tensile strength of prestressing steel fpk.
- **Relaxation** – the ratio of total loss of stress at infinite time and the stress in tendon.

The development of relaxation loss over time is defined in the right table. If the option **Is independent of relative stress in prestressing reinforcement** is selected, the defined development of relaxation loss over time is valid for each relative stress in tendon. If the option is not selected, the development of relaxation loss over time can be defined for each relative stress separately.

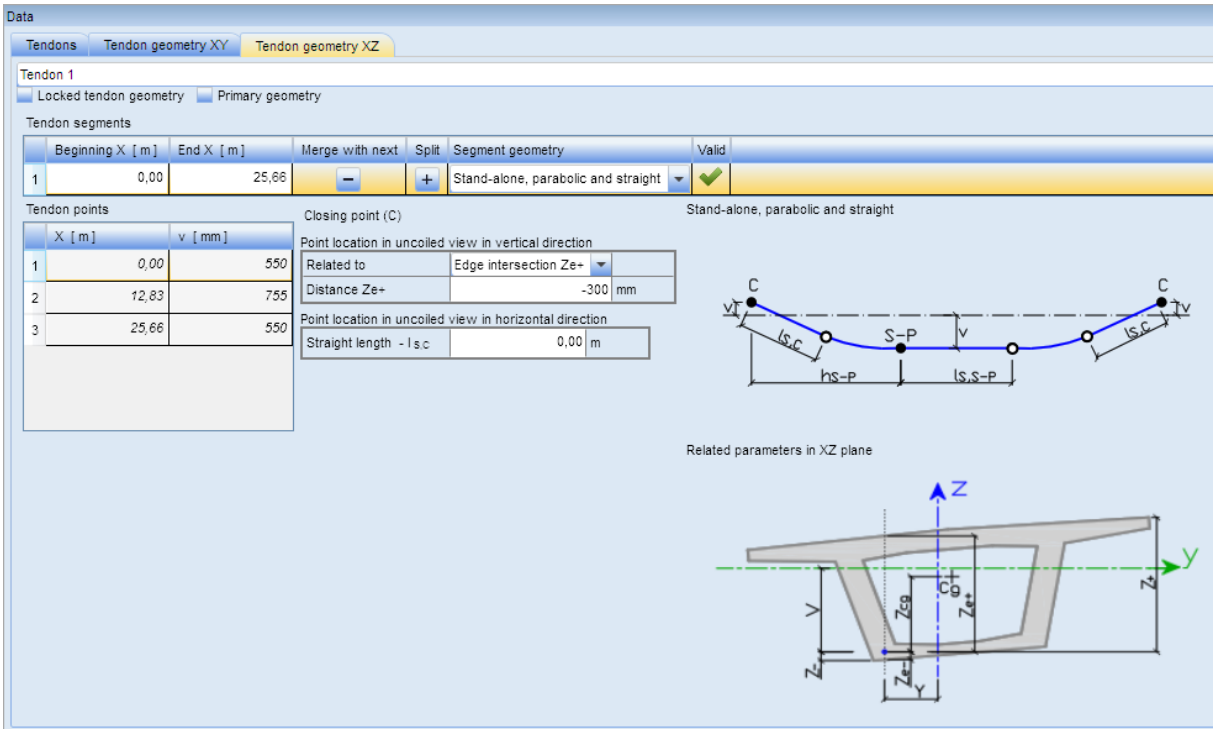
- **Duration** – tendon stress duration.
- **Relative** – ratio of current to total relaxation loss.

9.3.2 Tendon segments geometry editing

Tendon segments geometry is edited separately for both uncoiled views XY and XZ in tables on tables **Tendon geometry XY** and **Tendon geometry XZ**. Editing tables correspond to drawing of uncoiled views, where following entities are drawn in different colours:

- Selected tendon segment (thick red line in default settings);
- Selected characteristic point (red filled circle).

Following items are available for both uncoiled views on particular tabs:



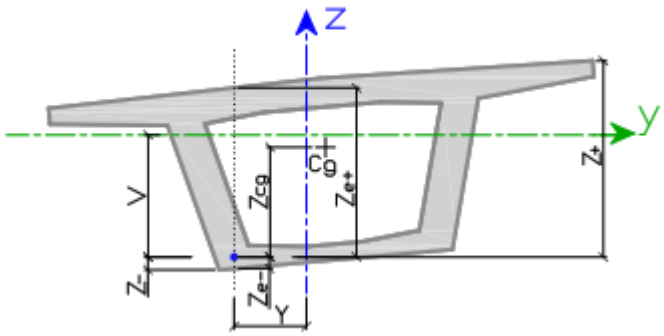
- **List of existing tendons –** current tendon can be set in the list.
- **Tendon geometry is locked** – if the switch is on, tendon geometry cannot be edited.
- **Primary geometry** – if the switch is on, tendon geometry in the appropriate plane is assumed to be the primary tendon geometry.
- **Tendon segments** – particular tendon segments can be edited in this table.
- **Tendon points** – points of current tendon segment are listed in this table.
- **Tendon point properties** – properties of current tendon point can be edited in this table.

9.3.2.1 Primary tendon geometry

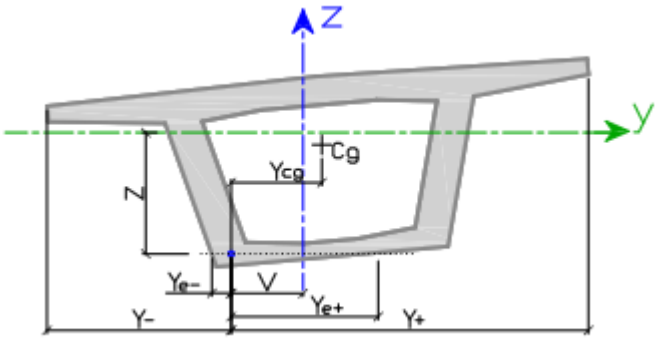
Primary tendon geometry determines primary uncoiled view for input of tendon position in cross-section. According to principle of tendon geometry input using two independent uncoiled views it is necessary to determine primary uncoiled view, if characteristic tendon points refer to intersections with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

- Vertical line drawn in **Y**-coordinate of tendon with the edge of cross-section in primary uncoiled view XY. **Z** -position of tendon in cross-section can refer to intersection of this vertical line with cross-section edges. All available reference points for input of tendon **Z**-position are displayed in the following

picture. Intersections of vertical line in **Y**-coordinate of tendon with cross-section edges are depicted with dimension lines **Ze**$^{+}$ and **Ze**$^{-}$.



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



Primary uncoiled view does not allow inputting reference points on cross-section edges; it can use only minimal or maximal coordinates of cross-section.

9.3.2.2 Tendon segments table

Tendon segments

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

All segments of current tendon are listed in table. Table contains following columns:

- **Beginning** – position of the beginning of the tendon segment measured in the axis of the uncoiled view from the beginning of the design member.
- **End** – position of the end of the tendon segment measured in the axis of the uncoiled view from the beginning of the design member.
- **Merge with next** – click − to remove segment by merging the current segment with the following one. The merge of segments causes the change of segment geometry and the length of the segment is sum of segment lengths before merging. For example by merging of following segments:

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

one straight parabolic segment is created:

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

- **Split –** click + to split current tendon segment into two segments of the same length. Depending on the position of the current segment the geometry of segment can change.
- **Segment geometry** – select segment type from available tendon segment types. All types of tendon segment geometry are described in **9.2.1Type of segments to define tendon geometry**. The content of the list is filtered automatically to display only allowed segment types. E.g. if tendon consist of one segment only, its geometry can be defined using segment types 1 or 2. Geometry of current segment including described characteristic points is drawn below this table.

9.3.2.3 Tendon points table

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

	X [m]	v [mm]
1	0,00	-50
2	12,83	615
3	25,66	-50

- **Point number** – number of characteristic tendon point in the uncoiled view.
- **X** – The position of the point measured in the uncoiled view from the beginning of the design member.
- **v** – Position of point **Y** or **Z** for uncoiled view **XY** or **XZ** related to cross-section coordinate system origin.

9.3.2.4 Table of current tendon characteristic point parameters

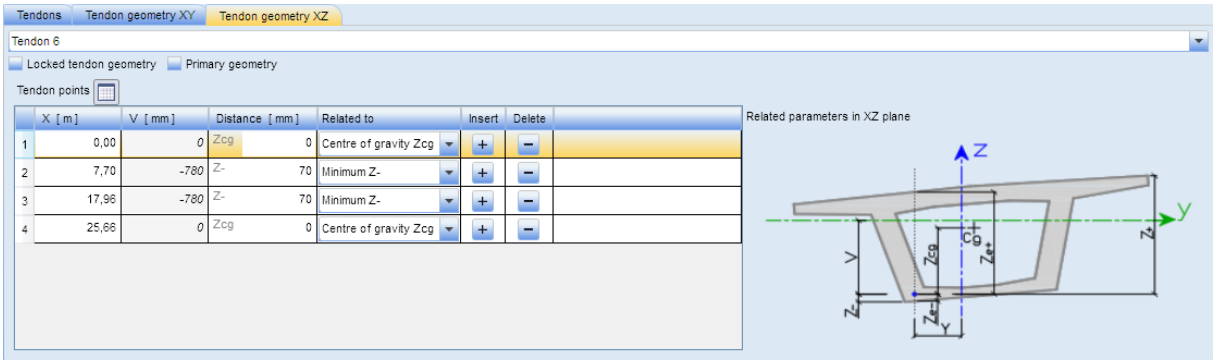
Next to the table **Tendon points** table for editing of current characteristic point parameters is displayed. Current characteristic point can be set by selection of appropriate row in table **Tendon points** or by mouse in the drawing of uncoiled view. Tables for all available types of characteristic points are described in chapter **9.2.4 Description of tendon definition geometry points**.

9.3.3 Tendon polygons geometry editing

Geometry of polygonal tendon is edited similarly as geometry of tendon segments.

Selected vertex of polygon is drawn in red in uncoiled views

Following items are available for both uncoiled views on particular tabs:



- **List of existing tendons –** current tendon can be set in the list
- **Tendon geometry is locked** – if the switch is on, tendon geometry cannot be edited
- **Primary geometry** – if the switch is on, tendon geometry in the appropriate plane is assumed to be the primary tendon geometry
-  – if all tendon polygon vertexes have the identical value of property 'Related to' in particular uncoiled views, click the button to edit the tendon polygon geometry in the table editor
- **Tendon points** – points of current tendon polygon are edited in this table

9.3.3.1 Table Tendon points

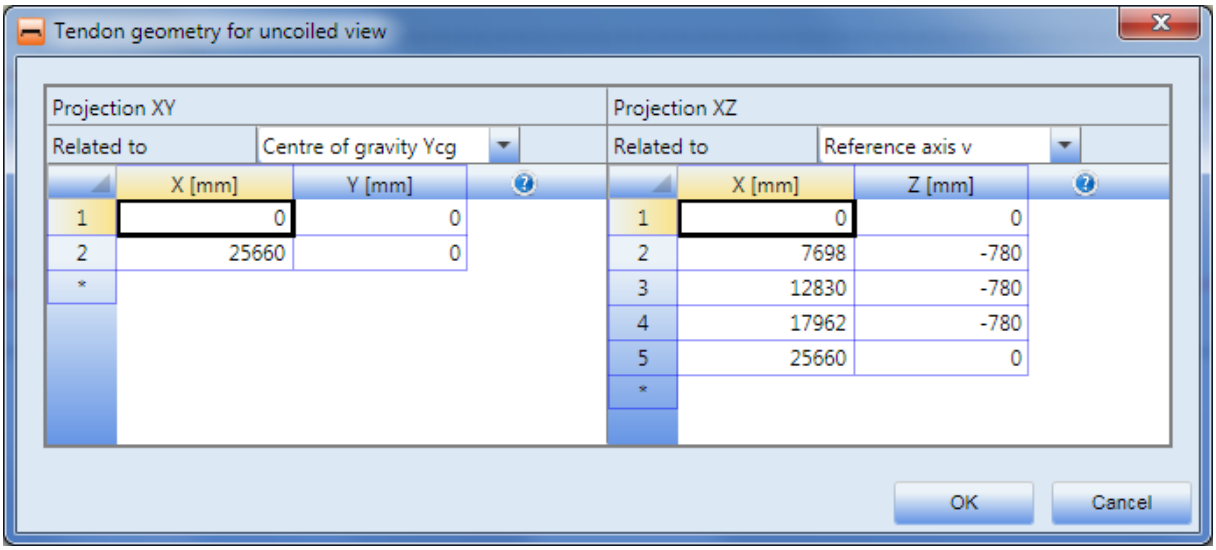
	X [m]	V [mm]	Distance [mm]	Related to	Insert	Delete
1	0,00	0	Zcg 0	Centre of gravity Zcg	+	−
2	7,70	-780	Z- 70	Minimum Z-	+	−
3	17,96	-780	Z- 70	Minimum Z-	+	−
4	25,66	0	Zcg 0	Centre of gravity Zcg	+	−

Vertexes of tendon polygon are listed in the table. The table contains following columns:

- **X** – input of distance from the beginning of the design member.
- **v** – the calculated distance between the polygon vertex and the reference curve.
- **Distance** – input of distance between the vertex and the point specified in the column '**Related to'.**
- **Related to** – choose the point of cross-section, to which the entered distance is related.
- **Insert –** click  to insert new polygon vertex next to the current polygon vertex.
- **Delete –** click  to delete the current polygon vertex.

9.3.3.2 Editing tendon polygon vertexes using the table editor

If all polygon vertexes in the particular uncoiled view are related to the same point, click  to display the table editor to edit the vertexes of both uncoiled views simultaneously.



For the both uncoiled views following properties can be set:

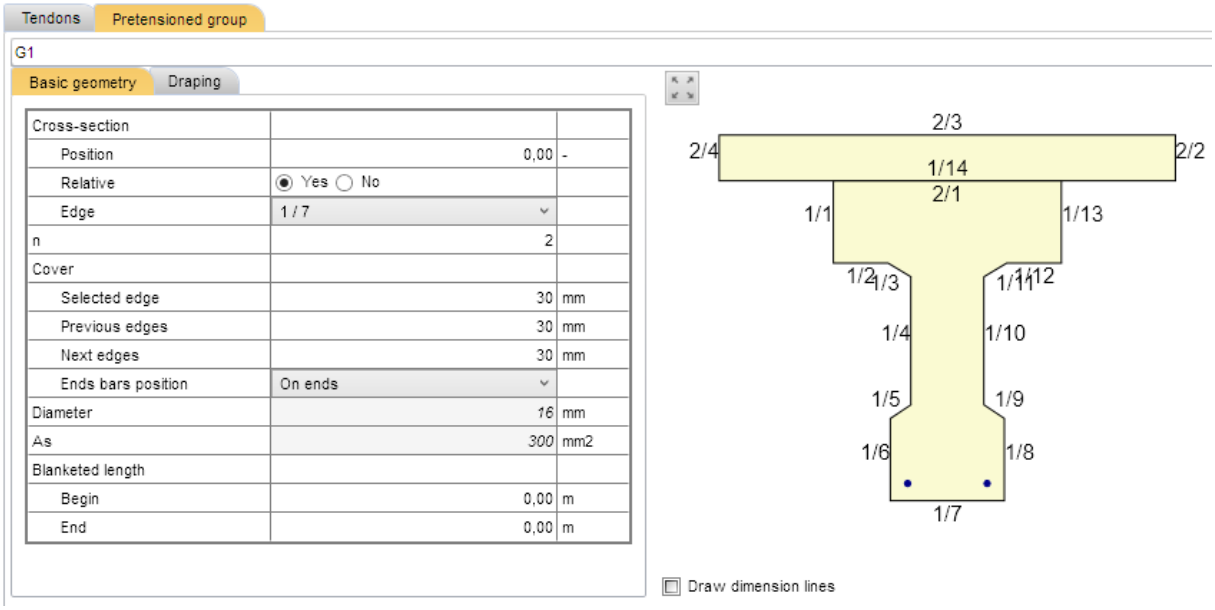
- **Related to** – choose the point of cross-section, to which the entered distance is related.
- To input coordinates of vertexes in planes XY or XZ the table editor is used – see **4.2 Table editor**.

9.3.4 Editing geometry of groups of pre-stressed tendons

Pre-stressed tendons are joined to groups. Properties of group of pre-stressed tendons can be edited on the table **Pretensioned group**, which is common for both uncoiled views of pre-tensioned tendon. Offsets from the straight direction can be defined for the group of pretensioned tendons.

9.3.4.1 Pre-stressed tendons on cross-section edge

Following properties of pre-tensioned group defined on cross-section edge can be edited on the tab **Pretensioned group**:



- **List of existing pre-tensioned groups** – choose the current pre-tensioned group in the list.

Tab **Basic geometry**:

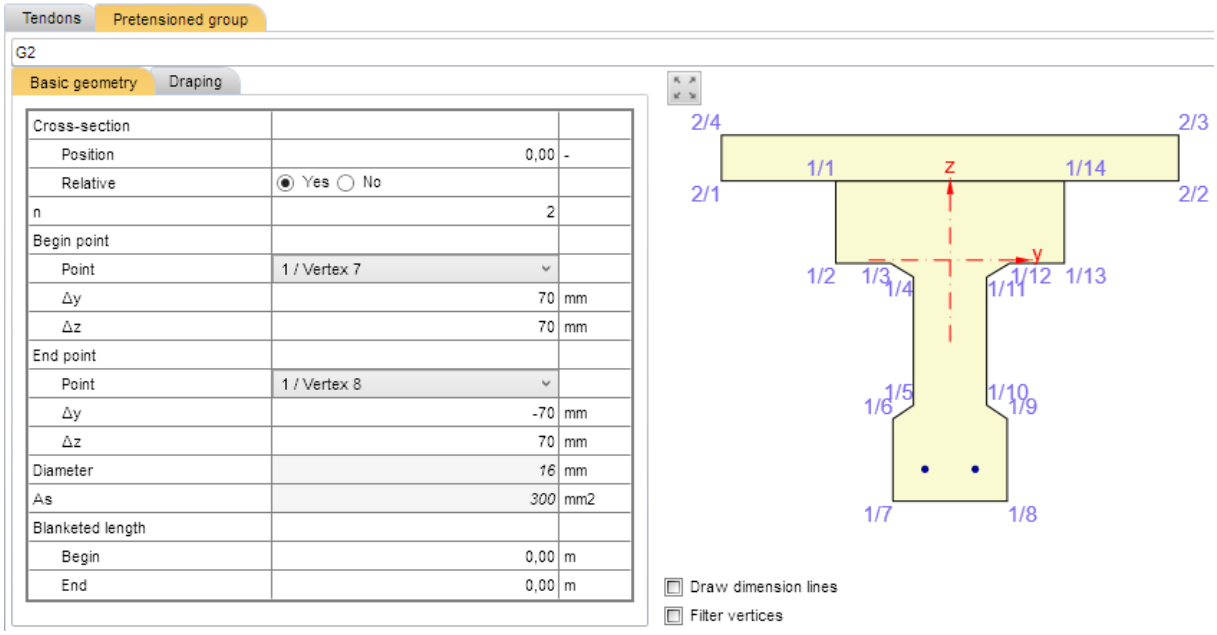
- **Position** – input the position of cross-section, for which the group of pre-stressed tendons is defined. The position is related to the beginning of the design member.
- **Relative** – if the option is set to '**Yes**', the value of position is considered as relative, if it is set to '**No**', the value of position is absolute to the beginning of the design member.
- **Edge** – select the edge of cross-section, to which the group of tendons is related to.
- **n** – edit the number of tendons in group.
- **Cover** – define the cover related to the edges of cross-section.
  - **Selected edge** – edit the value of cover to the current edge of cross.-section.
  - **Previous edges** – edit the value of cover to the edges which are previous to the current edge.
  - **Next edges** – edit the value of cover to the edges, which follow the current edge.
- **Diameter** – display value of tendon diameter.
- **As** – display the area of tendons.
- **Blanketed length**
  - **Begin** – edit the value of blanketed length at the beginning of the tendon.
  - **End** – edit the value of blanketed length at the end of the tendon.

Tab **Draping** – see **9.3.4.3 Draping of pre-tensioned tendons**.

Draw dimension lines – turn on/off the drawing of dimension lines of the cross-section and pre-tensioned tendons.

9.3.4.2 Pre-stressed tendons in line

Following properties of pre-tensioned group defined in the line can be edited on the tab **Pretensioned group**:



- **List of existing pre-tensioned groups** – choose the current pre-tensioned group in the list.

Tab **Basic geometry**:

- **Position** – input the position of cross-section, for which the group of pre-stressed tendons is defined. The position is related to the beginning of the design member.
- **Relative** – if the option is set to '**Yes**', the value of position is considered as relative, if it is set to '**No**', the value of position is absolute to the beginning of the design member.
- **n** – edit the number of tendons in group.
- **Begin point** – definition of the beginning point of the tendons line.
  - **Point** – select the reference point for the input of the position of the first tendon in the line.
  - **Δy** – input the distance between the first tendon in line and the reference point in the direction of y-axis of the cross-section..
  - **Δz** – input the distance between the first tendon in line and the reference point in the direction of z-axis of the cross-section.
- **End point** – definition of the end point of the tendons line.
  - **Point** – select the reference point for the input of the position of the last tendon in the line.
  - **Δy** – input the distance between the last tendon in line and the reference point in the direction of y-axis of the cross-section..
  - **Δz** – input the distance between the lst tendon in line and the reference point in the direction of z-axis of the cross-section.
- **Diameter** – display value of tendon diameter.
- **As** – display the area of tendons.
- **Blanketed length**
  - **Begin** – edit the value of blanketed length at the beginning of the tendon.

- **End** – edit the value of blanketed length at the end of the tendon.

Tab **Draping** – see **9.3.4.3 Draping of pre-tensioned tendons**.

Draw dimension lines – turn on/off the drawing of dimension lines of the cross-section and pre-tensioned tendons.

9.3.4.3 Draping of pre-tensioned tendons

The deviation of pre-stressed tendon from the straight direction is defined on tab **Draping**. The deviation is defined by list of points and offsets in individual points.

-  - add new row into the table to define the point of tendon draping.
-  - remove the current row defining the point of tendon draping from the table.
- **Position** – input the distance of point from the selected origin.
- **Relative** – if selected, the defined position value is taken as relative to the design member length, otherwise the position value is taken as absolute to the defined origin.
- **Measured from** – select the origin, to which is the defined point related:
  - **Beginning** – defines one draping point in specified distance from the beginning of the design member.
  - **End** – defines one draping point in specified distance from the beginning of the design member.
  - **Both symmetrical** – defines two draping points in the specified (identical) distance from the beginning and from the end of the design member.
- **Δy** – input the tendon offset value in y-axis direction of the cross-section.
- **Δz** – input the tendon offset value in z-axis direction of the cross-section.

9.4 Tendon segments geometry validity check

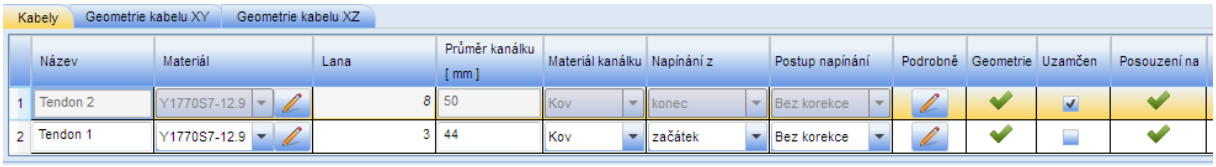
Geometry of each tendon segment is checked automatically. The validity of segment geometry can be verified in tendon segments table. Probable causes of invalid geometry are:

- Parabola with the minimal radius cannot be inserted
- Entered lengths of straight segments are longer than the length of segment
- The whole segment or its part is out the Design member

Similarly to check of tendon segments the geometry of the whole tendon is checked. The validity of tendon geometry can be verified in tendons table. Geometry of tendon is valid if following assumptions are fulfilled:

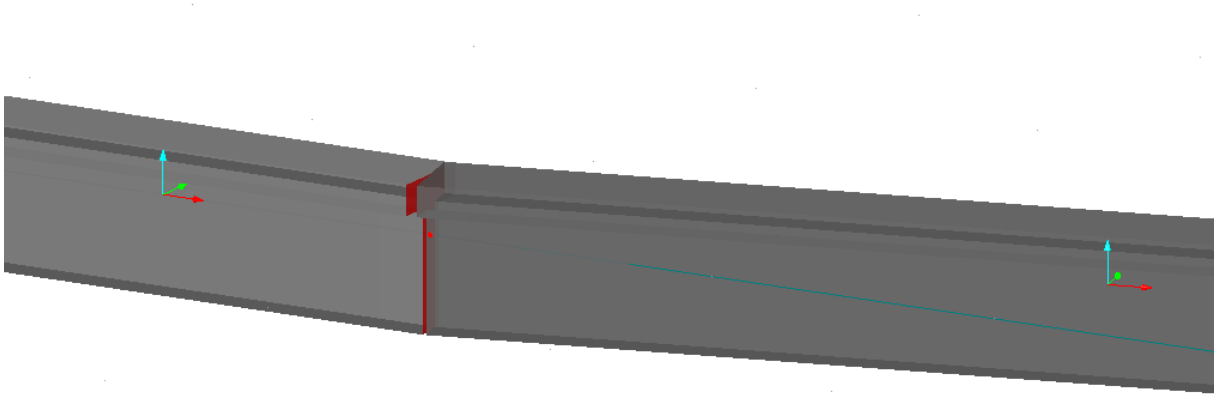
- Geometry of all segments is valid
- Continuity of segments must be smooth; it means that tangent of angle in segment transitions has to be equal zero.
- Geometry of Design member must be valid, it means that all members of design member must continue correctly

If the tendon geometry is invalid, the tendon cannot be analysed. Neither tendon losses, nor equivalent loads can be calculated. Thus tendon with invalid geometry cannot be exported to superior application.

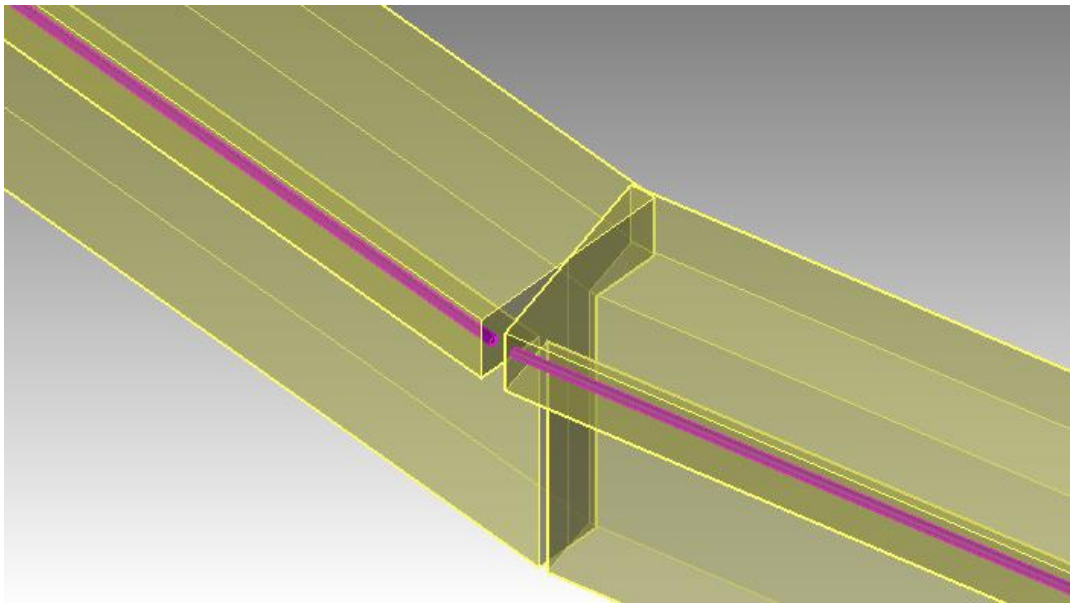


9.5 Discontinuous tendons on member of polygonal shape

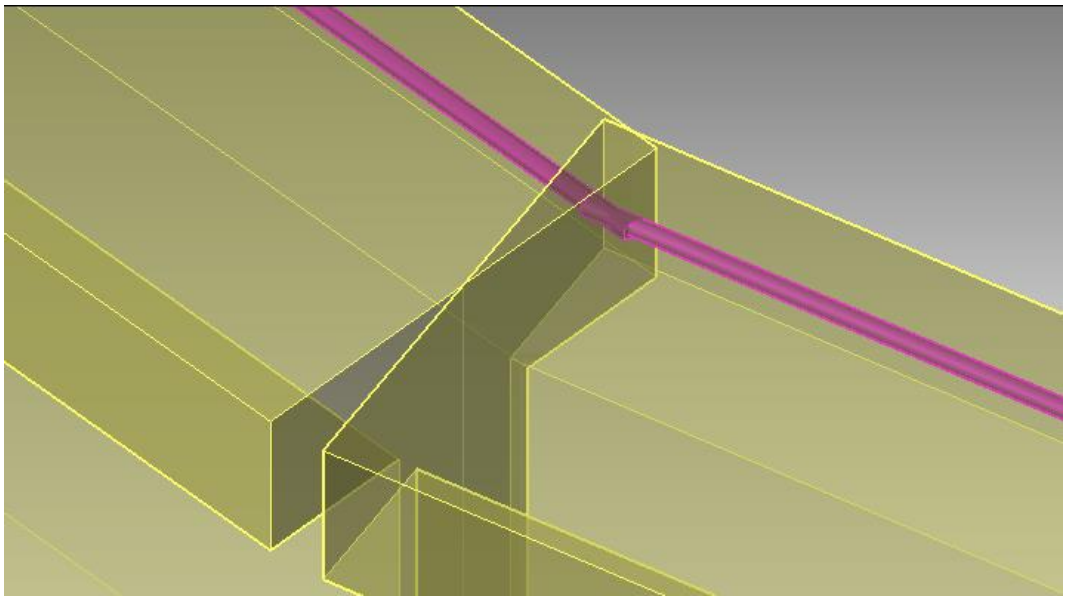
In the point of design member fracture seeming tendon tearing appears, because corresponding local coordinate systems of members (or part of members) are not identical in the point of fracture. In the picture the fracture of design member is visible, with not identical local coordinate systems of following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



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



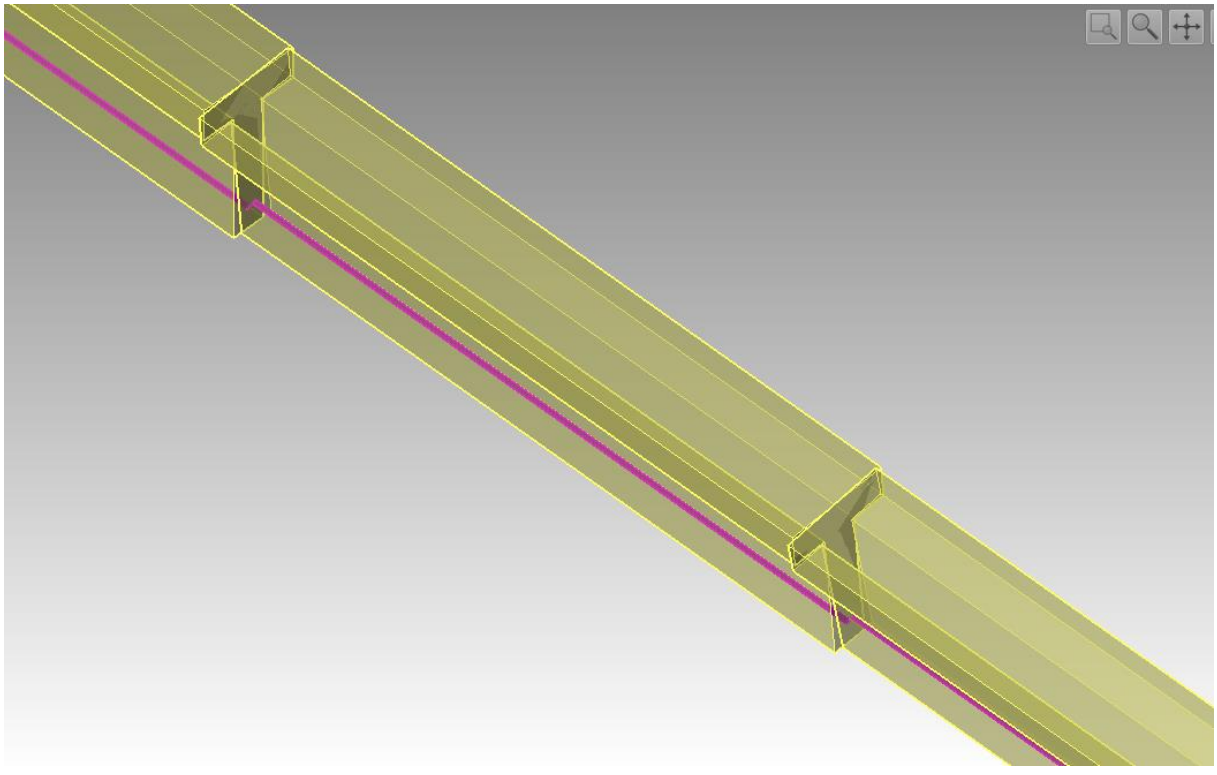
If the point lies on the inside of the break, tendon segments can seem to cross.



Angular change between tangents of tendon ends in the point of rupture is assumed in tendons analysis.

9.6 Not continuous tendons on rotated members

Identical case appears, if two neighbouring parts of member do not have the identical local coordinate system, but the LCS differ only in rotation about x-axis, it means that angle between Y-axes does not equal zero. The tendon is ruptured in this point too, but both end points lie in one plane, which is perpendicular to the x-axis of local coordinate system of part of member. This rotation is not taken into account in the calculation. It is assumed that the rotation between members is very small (the order of degrees). If it is not the case, the analysis model should be adapted.



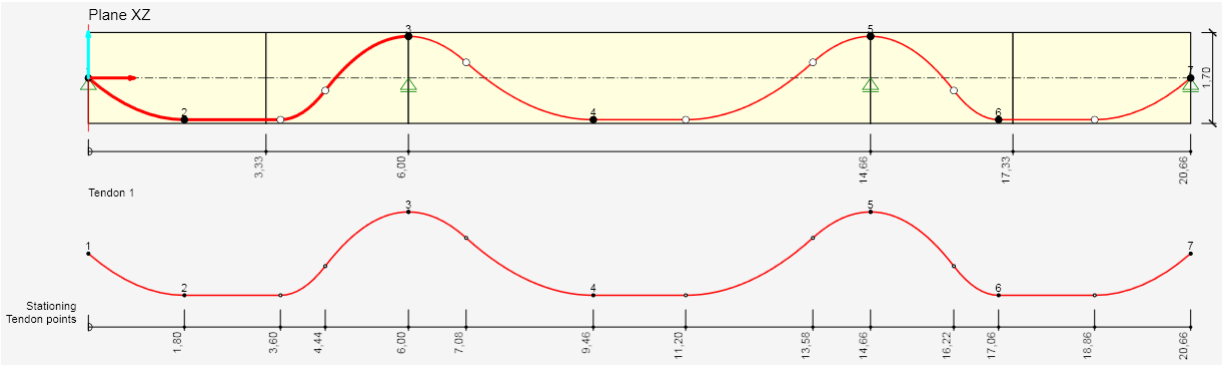
9.7 Input of new tendon

To input new tendon use commands in ribbon group **New tendon**:

- **Segment > Take supports into account** – create new tendon defined by segments respecting positions of supports of the design member.
- **Segment > Do not take supports into account** – create new tendon defined by segments without respecting positions of supports of the design member.
- **Segment > Straight** – create new straight tendon defined by segments.
- **Polygon > Take supports into account** – create new tendon defined by coordinates of polygon vertexes respecting positions of supports of the design member.
- **Polygon > Do not take supports into account** – create new tendon defined by coordinates of polygon vertexes without respecting positions of supports of the design member.
- **Polygon > Straight** – create new straight tendon defined by coordinates of polygon vertexes.

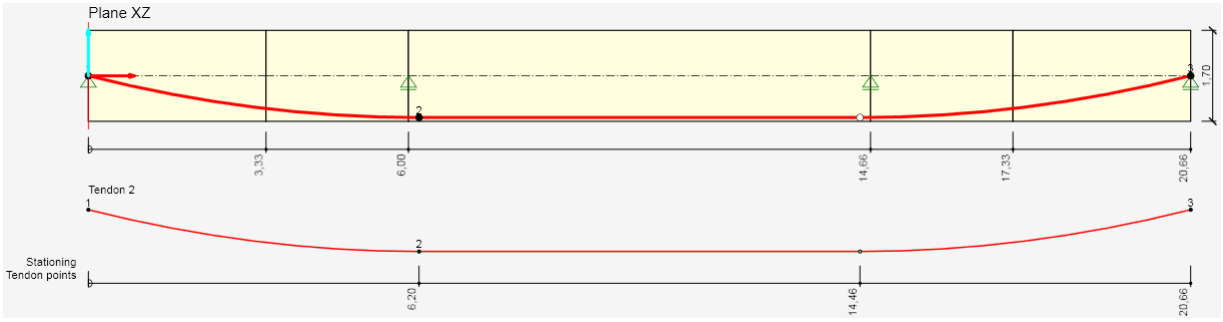
9.7.1 Pretensioned on edge – add new group of pre-stressed tendons related to the cross-section edge.Tendon defined by segments with respecting supports

To create new tendon defined by segments with respecting positions of design member supports click **Segment > Take supports into account** in ribbon group **New tendon**. Respecting supports means that the tendon goes at the bottom of cross-section in the span between supports and at the top of cross-section above the supports. The tendon consists of at least one segment in each plane.



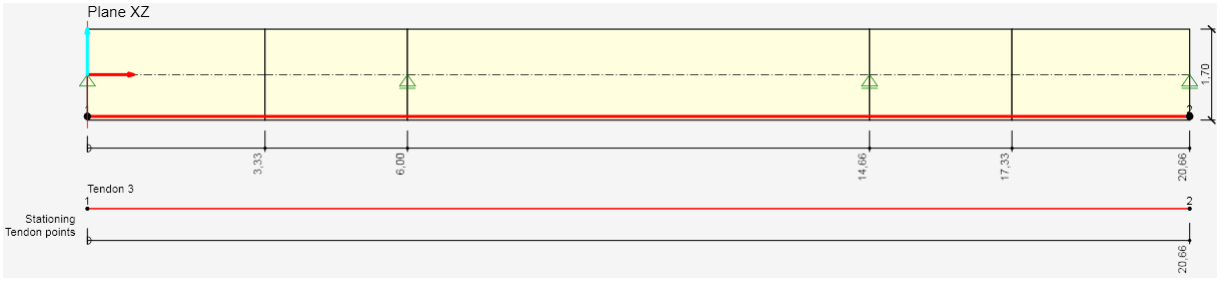
9.7.2 Tendon defined by segments without respecting supports

To create new tendon defined by segments without respecting positions of design member supports click **Segment > Do not take supports into account** in ribbon group **New tendon**. New tendon consists of exactly one segment in each plane. The tendon is straight in plane XY (ground plan) over the whole design member length and parabolic in vertical plane XZ.



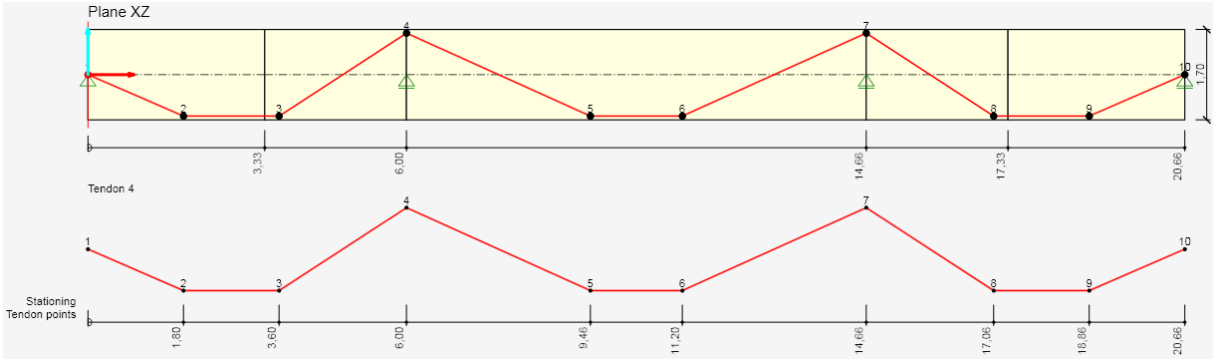
9.7.3 Straight tendon defined by segments

To create new straight tendon defined by segments click **Segment > Straight** in ribbon group **New tendon**. New tendon consists of exactly one straight segment in both planes



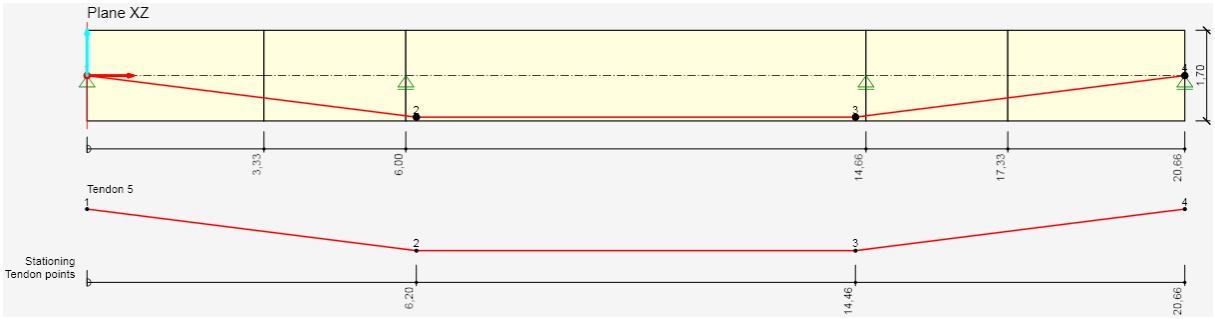
9.7.4 Polygonal tendon with respecting supports

To create new polygonal tendon with respecting positions of design member supports click **Polygon > Take supports into account** in ribbon group **New tendon**. Respecting supports means that the tendon goes at the bottom of cross-section in the span between supports and at the top of cross-section above the supports.



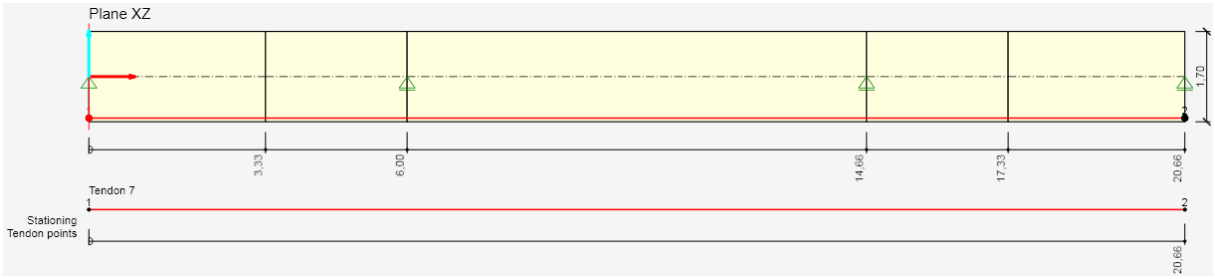
9.7.5 Polygonal tendon without respecting supports

To create new polygonal tendon without respecting positions of design member supports click **Polygon > Do not take supports into account** in ribbon group **New tendon**. The tendon is straight over the whole design member length and is defined by two points n plane XY (ground plan) and is defined by four points in plane XZ and goes from the centroidal axis to the bottom edge of cross-section.

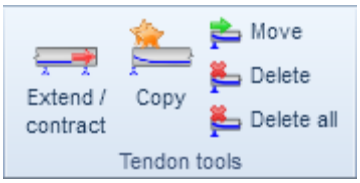


9.7.6 Straight polygonal tendon

To create new straight polygonal tendon click **Polygon > Straight** in ribbon group **New tendon**. New tendon consists of exactly one straight segment in both planes



9.8 Tendon tools

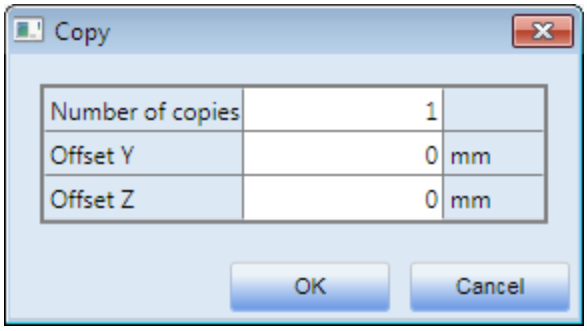


Ribbon group **Tendon tools** contains following commands:

- **Extend/contract** – adapt the length of current tendon according to the length of current design member.
- **Copy** –copy current tendon.
- **Move** – move current tendon.
- **Delete** – delete current tendon.
- **Delete all** – delete all tendons in current design member.

9.8.1 Copying tendons in design member

To copy current tendon click **Copy** in ribbon group **Tendon tools**. Tendon created by copying may move in Y and Z axes of design member coordinate system.



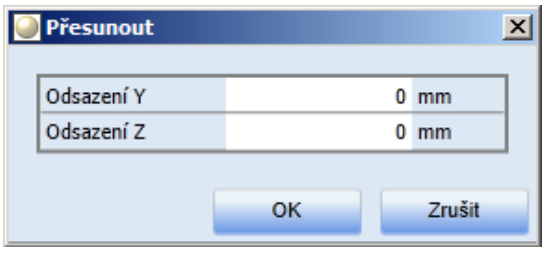
Particular dialog options:

- **Number of copies** – required number of copies of current tendon.
- **Offset Y** – offset value between copies in Y-axis
- **Offset Z** – offset value between copies in Z-axis.

Tendons created by copying have identical properties as source tendon including characteristic points in uncoiled views. But the tendon geometry in uncoiled views must not be identical completely, because the tendon segment characteristic points may be related to points on cross-section edges.

9.8.2 Moving tendon in cross-section

To move tendon click **Move** in ribbon group **Tendon tools**. Tendon can move in Y and Z axis of design member coordinate system. Particular dialog options:



- **Offset Y** – offset value in Y-axis.
- **Offset Z** – offset value in Z-axis.

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

9.9 Import and export of tendons

Geometry of created tendons can be stored to file.

New tendon can be created by import from file.

To run tendon export and import use commands in ribbon group **Import, Export**.

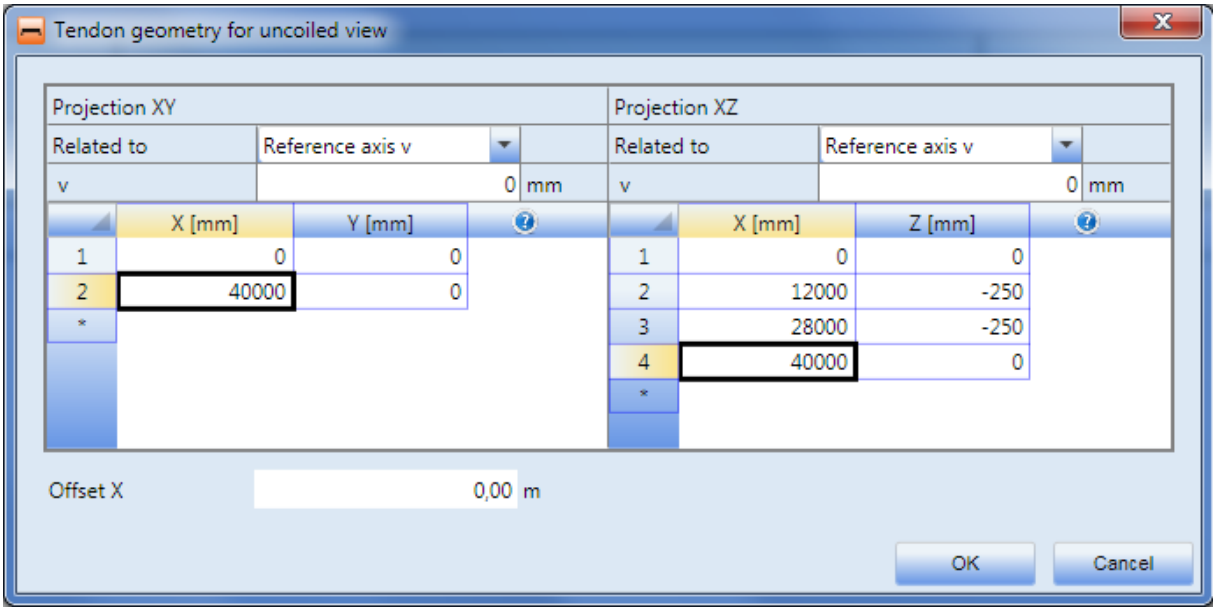
- **Import > New tendon(s) from TXT file** – create new tendons importing geometry from text file. If the imported tendon is longer than target design member, the imported tendon is shortened automatically.
- **Import > New tendon from table** – create new tendon using the table editor to define tendon polygon vertexes.

- **Import > New tendon from template** – create new tendon selecting required tendon shape from the database of user defined templates of tendon shapes – see **9.10 User defined templates of tendon** .
- **Import > New tendon(s) from DXF file** – create new polygonal tendons importing the tendon geometry from DXF file
- **Import > Change geometry from DXF file > Geometry XY** –import tendon geometry into the uncoiled view XY of the current tendon from DXF file.
- **Import > Change geometry from DXF file > Geometry XZ** – import tendon geometry into the uncoiled view XZ of the current tendon from DXF file.

- **Export > Current tendon** – save geometry definition of current tendon to text file.
- **Export > All tendons** –save geometry definition of all tendons in current design member to text file.
- **Export > Save as template** – save geometry definition of the current tendon into the database of user templates. Dialog **Add template** appears. The target folder must be selected in the tree control in the left part of dialog. The current tendon is stored as a template into the selected folder (see **9.10 User defined templates of tendon**).

9.9.1 Input of tendon by the table editor

To create new polygonal tendon using the table editor click **Import > New tendon from table** in ribbon group **Import, export**.



Vertexes coordinates of polygonal tendons are defined in tables for uncoiled views XY and XZ. An offset of the origin point of the tendon can be defined.

- **Related to** – choose the reference point, to which the coordinates of vertexes are defined.
- **v (Y-, Y+,Ycg,** or **Z-, Z+, Zcg)** – the distance between the beginning point of the tendon and the 'Related to' point
- **Offset X** – the distance between the beginning point of tendon and the beginning point of the design member

Particular coordinates of vertexes are defined for both planes XY and XZ in the tables using the built-in table editor– see **4.2 Table editor**.

9.9.2 Import of tendons from DXF file

The import from DXF file can be used:

- To create a new tendon. Only one definition geometry (in selected plane) is created during the import from DXF file. The second definition geometry can be adapted by import of DXF file to the existing tendon.
- Adapt the selected definition geometry of the existing current tendon.

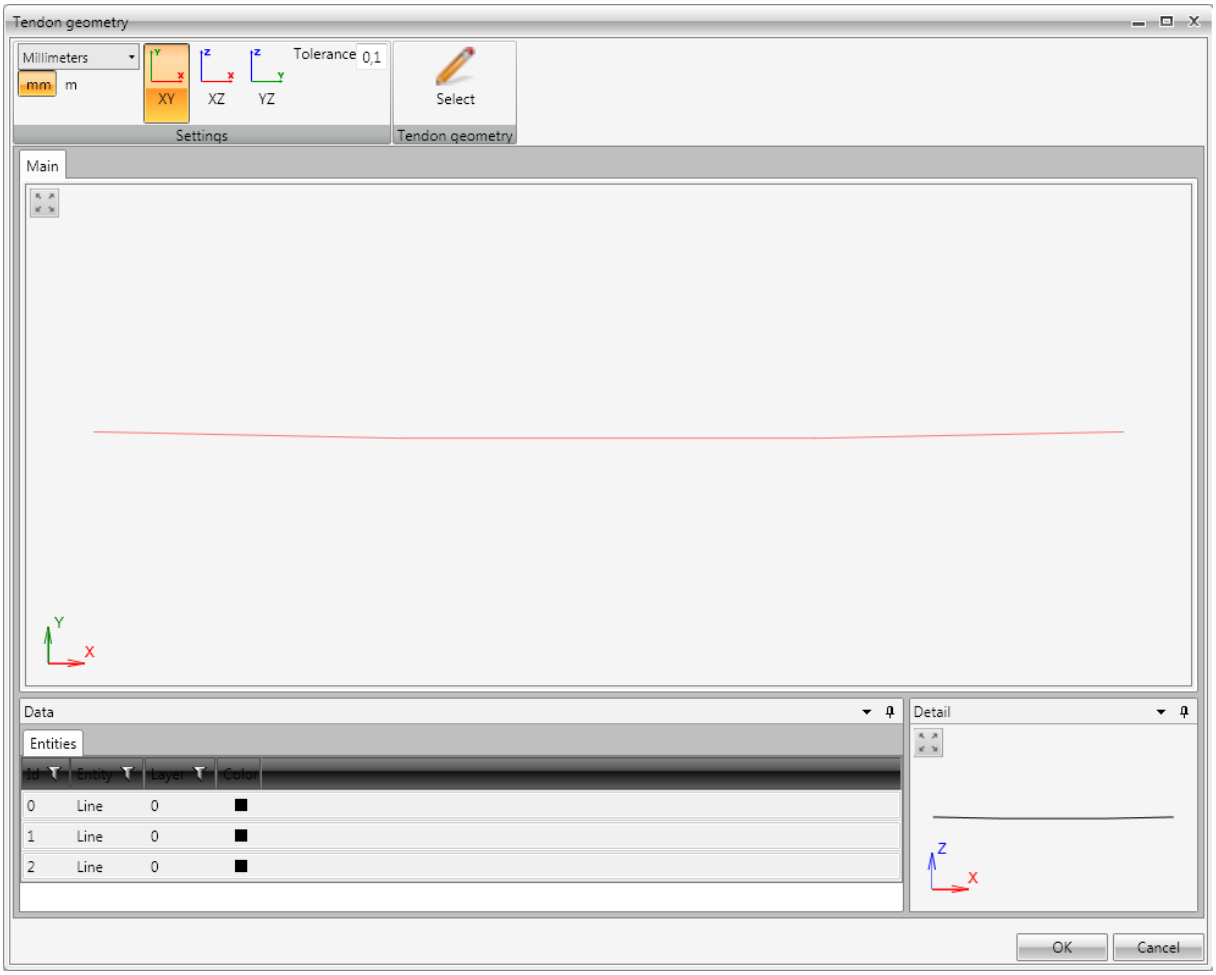
9.9.3 Creating a new cable by import from DXF file

To create new tendon importing DXF file click **Import** > **Tendon from DXF file**.

The tendon import consist of following steps

- Selection of lines, which define tendon in DXF file
- Insertion of polygon into the required uncoiled view.

Following entities types are read from DXF file:LINE, POLYLINE, SPLINE, ARC, CIRCLE, TEXT. Blocks are not imported. Blocks must be exploded before import.



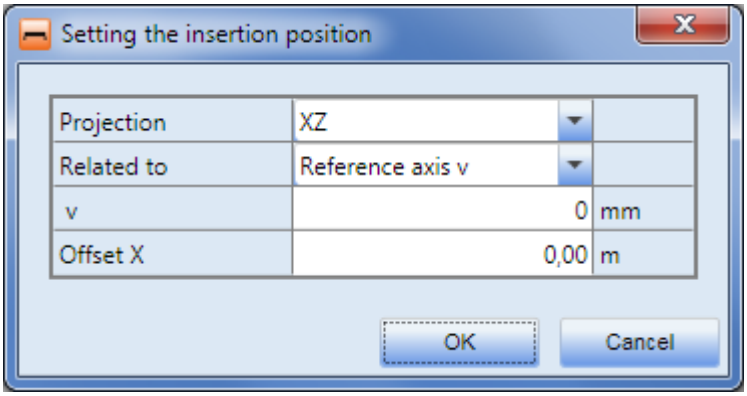
The content of imported DXF file is displayed in dialog **Tendon geometry**.

Lines, which create tendon polygon, must be selected in the main window. Lines can be selected like standard irregular selections in Windows applications – hold CTRL and select single lines.

After finishing the selection, click **Select** in ribbon group **Tendon geometry**. A shape created from selected lines, is displayed in the Detail window.

After clicking **OK** dialog **Setting the insertion point** appears to set the insertion details.

The coordinates of tendon polygon vertexes are transformed during the import from DXF in such way, that the utmost left vertex of tendon polygon has coordinate [0;0].



Options of dialog **Setting the insertion point:**

- **Projection** – choose the destination uncoiled view plane, to which the polygonal tendon will be inserted
- **Related to** – – choose the reference point, to which the offset of the tendon beginning is defined
- **v** (**Y-, Y+,Ycg,** resp. **Z-, Z+, Zcg**) – the distance between the beginning point of the tendon and the 'Related to' point
- **Offset X** – the distance between the beginning point of the tendon and the beginning of the design member

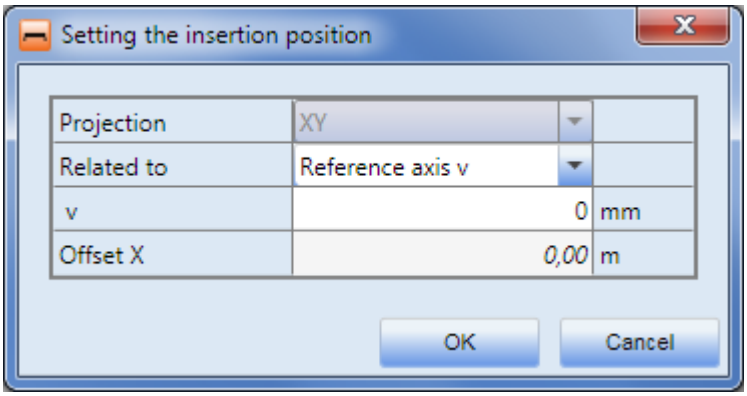
9.9.4 Modification of tendon geometry using the import from DXF file

To modify the geometry of existing polygonal tendon by import of tendon geometry from DXF file following commands can be used:

- **Import > Tendon geometry from DXF file > Geometry XY** to modify the tendon geometry in uncoiled view XY,
- **Import > Tendon geometry from DXF file > Geometry XZ** to modify the tendon geometry in uncoiled view XZ

The tendon defined by segments cannot be modified using import of tendon geometry from DXF file.

The steps to modify the tendon geometry using import from DXF are similar to steps to import new tendon. But the plane of uncoiled view and the offset from the beginning of the design member cannot be changed in the dialog **Setting the insertion point**



After clicking OK in **Setting the insertion point** dialog the tendon geometry in appropriate uncoiled view is overwritten by the tendon geometry imported from DXF file.

9.10 User defined templates of tendon geometry

The existing tendon geometry can be stored into the database of user defined templates. The stored tendon geometry template can be used to pre-stress other design members in the current project or design members in other projects.

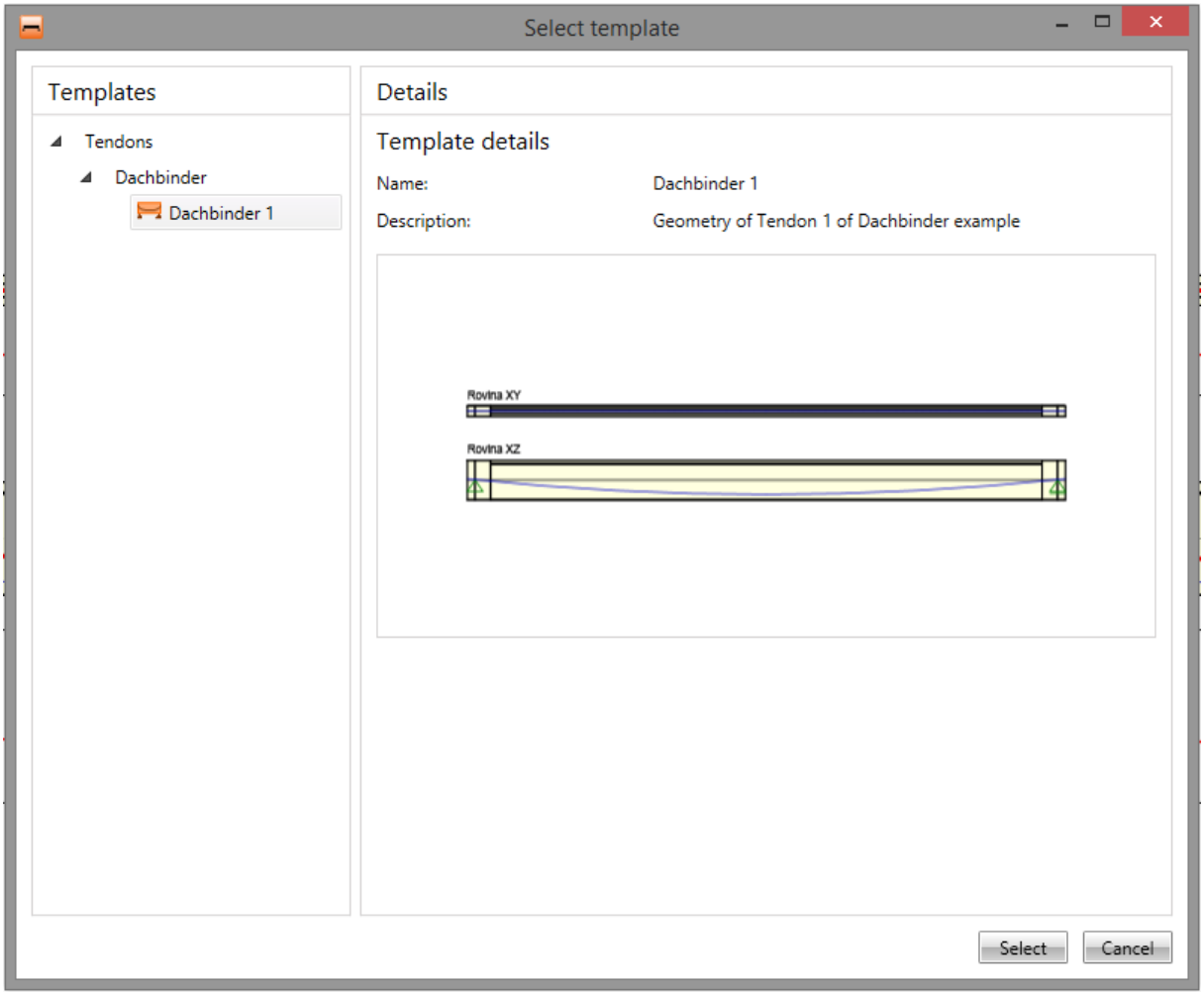
Commands **Import > New tendon from template** and **Export > Save as template** in ribbon group **Import, export** and **Tendon templates** in ribbon group **Settings** can be used to work with tendon geometry templates.

9.10.1 New tendon by user defined tendon geometry template

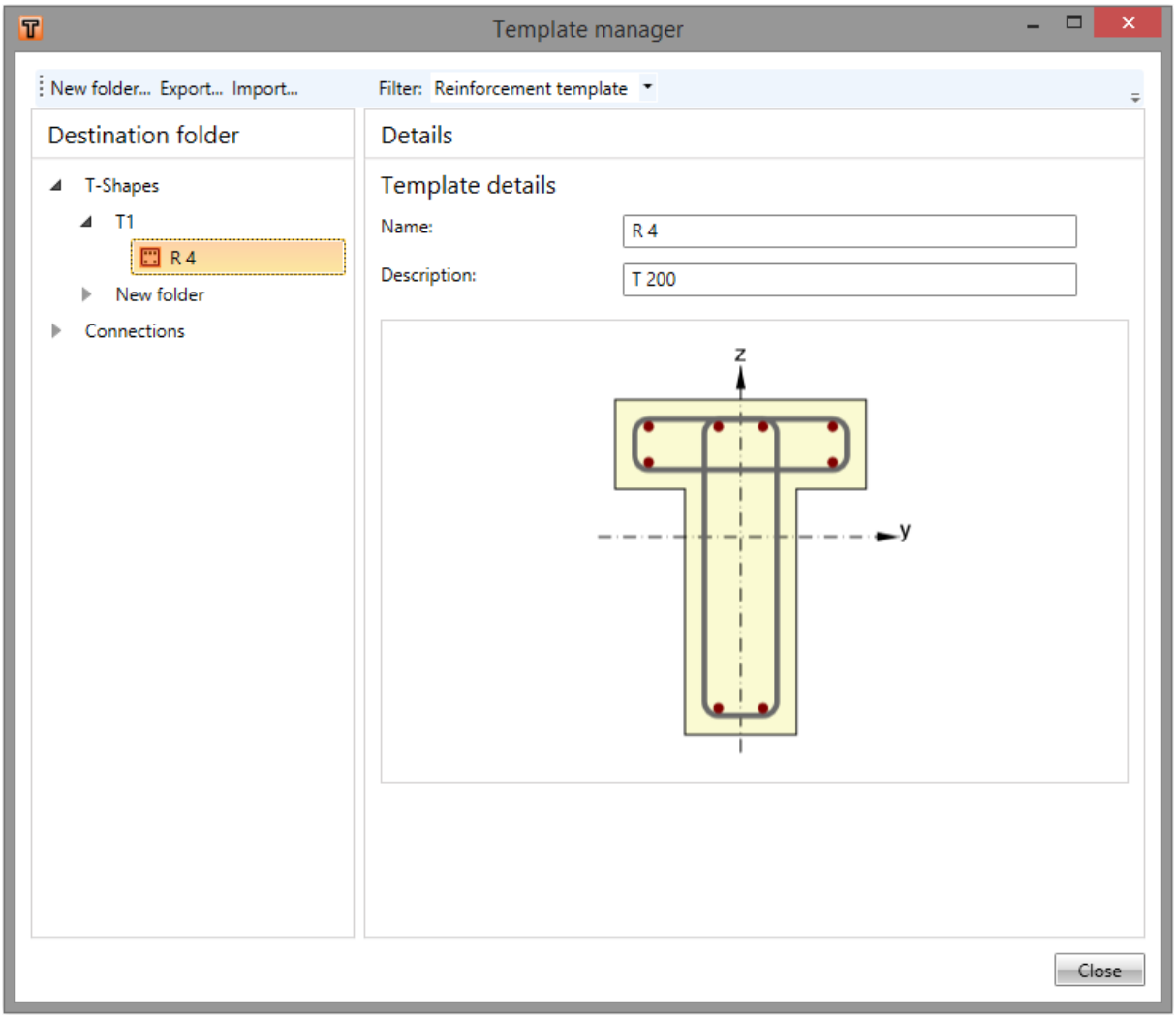
Dialog **Select template** appears after start of creating new tendon by user defined tendon geometry template.

Only templates, which have the same number of design member spans as number of spans on the current design member, are available in the tree control in the left part of the dialog.

Select the required template in the tree of available templates. Click **Select** to create new tendon using the selected template.



9.10.2 Templates manager



Template manager is used to manage templates in the database. The templates database collects templates for:

- Reinforcement templates;
- Templates of tendon shapes;
- Templates of connection manufacturing operations.

Template types to be displayed can be selected in the combo box **Filter**.

The templates are stored using the structure of folders and items in folders (similar to the structure of folders and files on drive).

The database structure (with respect to the filter settings) is displayed in the left part of the dialog. Details of selected template or selected folder are displayed in the right part of dialog.

Following actions can be performed in the templates manager:

- **Create new folder** – by command **New folder…** in the main menu to create new folder in the root folder or in the current subfolder.
- **Rename folder** – by command **Edit** in the context menu by right mouse click above the required folder.
- **Move folder** – drag and drop selected folder(s) to the required target folder.

- **Remove folder (s)** – by command **Delete** in the context menu by right mouse click above the selected folder (s). The folder is removed including all subfolders and all templates in removed folders and subfolders.
- **Edit template name and description** – template name and description of selected template is displayed in the right part of the dialog. The template name and description can be modified.
- **Move template** – drag and drop selected template(s) by mouse to the required target folder.
- **Delete template(s)** – by command **Delete** in the context menu by right mouse click above the selected template.
- **Export templates** – by command **Export…** in the main menu. Selected templates are stored into the file with extension *.EXP. Exported templates can be e.g. used on other computer.
- **Import templates** – by command **Import…** in the main menu. Templates from the selected file with extension *.EXP are imported into the database of templates.

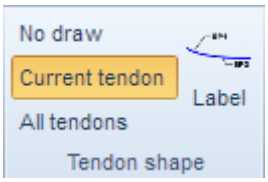
9.11 Extracting and shortening tendon

To extend or shorten current tendon to design member end click **Extract/contract** in ribbon group **Tendon tools**.

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

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

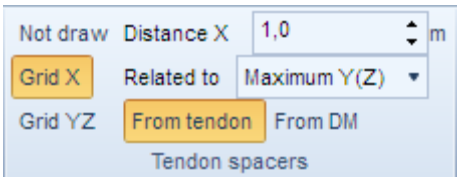
9.11.1 Tendons drawing settings



Use commands in ribbon group **Tendon shape** to set the detailed drawing of tendons shape:

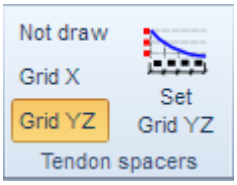
- **Not draw** – switch off the detailed drawing of tendon shape.
- **Current tendon** – switch on the detailed tendon shape drawing of current tendon.
- **All tendons** – switch on the detailed tendon shape drawing of all tendons.
- **Label** – switch on/off labels of detailed description of tendon parts.

9.11.2 Tendon spacers



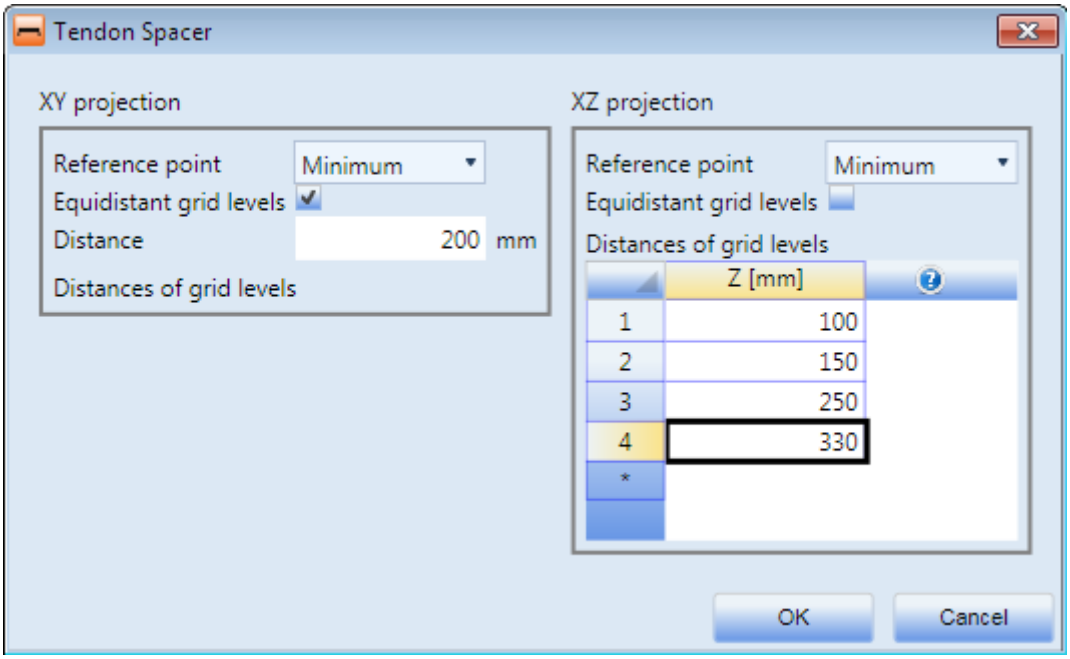
Tendon spacers grids can be displayed along the design member.

Use commands in ribbon group **Tendon spacers** to set the drawing of tendon spacers grids.



- **Not draw** – switch off the drawing of tendon spacer grid
- **Grid X** – switch on drawing of tendon spacers grid, for which coordinates Y and Z are calculated in equidistant points on x-axis. For this grid mode following options can be set:
  - **Distance X** – value of equidistant distance of points on x-axis, where tendon coordinates Y and Z are calculated.
  - **Related to** – choose the reference point, to which the tendon coordinates Y and Z are calculated.
  - **From tendon** – set the beginning of distances measurement on x-axis to the beginning of tendon.
  - **From DM** – set the beginning of distances measurement on x-axis to the beginning of design member.
- **Grid YZ** – switch on drawing of tendon spacers grid to calculate such distances on X-axis, for which user defined Y and Z tendon point coordinates are reached.
- **Set grid YZ** – start definition of tendon coordinates Y and Z for drawing of **Grid YZ**.

9.11.2.1 Tendon spacer grid YZ definition



For both uncoiled views following options can be set:

- Reference point – choose origin, to which the tendon point coordinates are related.
- **Equidistant grid levels** – if this option is checked, the equidistant levels of grid with the defined value of **Distance** are generated in the appropriate uncoiled view.
  If the option is not checked, user defined values of levels can be defined in the table **Distances of grid levels** – see –**4.2 Table editor**.

9.12 Format of text file for import and export of tendons

Tendons geometry is defined in section <**BondedTendons**> <**/BondedTendons**>. This section contains information about all imported/exported tendons. One tendon data is defined in section <**BondedTendon**> <**/BondedTendon**>

Section <**BondedTendon**> must contain 3 basic tags:

- <**BondedTendonData**> - contains tendon data
- <**BondedTendonSpansXY**> - contains tendon geometry in XY plane
- <**BondedTendonSpansXZ**> - contains tendon geometry in XZ plane

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

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

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

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

- Number of segment, on which point lies,
- type of point (1= point at the beginning or the ending of the whole tendon – C point, 2=point between straight part and parabola– S-P point, 3= point between two parabolas –P-P point),
- type of reference point for input of vertical segment point position (1= maximal Y-coordinate, 2=origin of reference curve, 3= minimal Y-coordinate, 4= maximal coordinate of intersection of horizontal line through tendon centre with cross-section edges, 5=minimal coordinate of intersection of horizontal line through tendon centre with cross-section edges, 6=centre of gravity of cross-section – must not always be identical with reference curve)
- vertical distance from reference point
- for **C**-point: length of end straight part
- **for S-P** point:
    - type of reference point for input of segment point horizontal position (1=input related to left segment edge, 2=input related to middle of segment, 3=input related to right segment edge)
    - horizontal distance from reference point
    - length of straight tendon part

  - mode of input values (1=relative, distances refer to tendon segment length, 0=absolute distances)
- for **P-P** point: minimal radius of parabola

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

9.12.1 Example of text file for tendon import

<BondedTendons> … beginning of the section for all tendons definition

<BondedTendon> … beginning of the section for one tendon definition

<BondedTendonData> … beginning of the tendon data section

Tendon 6 … the tendon name

1 XY 14 1 1 4 … gradually: **1** strand in tendon, primary geometry **XY**, tendon duct diameter **14** mm, tendon duct material **1**(metal), stressing from the beginning **1**, stressing procedure without relaxation correction **4**

</BondedTendonData> … end of the tendon data section

<BondedTendonSpansXY> …beginning of the section for input of geometry in uncoiled view XY

<SpansData> … beginning of the section for tendon segments input

1 0.00000 30.00000 … gradually segment type **1** (straight standalone), x-coordinate of segment beginning, x-coordinate of segment ending

</SpansData> … end of the section for tendon segments input

<SpansPoints> … beginning of the section for tendon segment characteristic points input

1 1 2 0.00 0.00000 … gradually: point lies on first segment – **1**, point type **C - 1**, vertical position is related to origin of reference curve– **2**, vertical distance from reference point is **0** mm, length of straight part is **0** m

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

</SpansPoints> … end of the section for tendon segment characteristic points input

</BondedTendonSpansXY> … end of the section for input of geometry in uncoiled view XY

<BondedTendonSpansXZ> … beginning of the section for input of geometry in uncoiled view XZ

<SpansData> … beginning of the section for tendon segments input

3 0.00000 10.00000 … gradually: segment type **3** (parabolic with straight, left), x-coordinate of the segment the beginning, x-coordinate of segment ending

5 10.00000 20.00000 … gradually: segment type **5** (parabolic with straight, inner), x-coordinate of the segment beginning, x-coordinate of the segment ending

4 20.00000 30.00000 … gradually: segment type **4** (parabolic with straight, right), x-coordinate of the segment beginning, x-coordinate of the segment ending

</SpansData> … end of the section for tendon segments input

<SpansPoints> … beginning of the section for tendon segment characteristic points input

1 1 2 0.00 0.00000 … gradually point lies on first segment – **1**, point type **C - 1**, vertical position is related to origin of reference curve – **2**, vertical distance from reference point is **0** mm, length of straight part is **0** m

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

1 3 3 70.00 2.00000 … gradually: point lies on first segment – **1**, point type **P-P - 3**, vertical position is related to minimal cross-section coordinate in Z-axis-**3**, vertical distance from reference point is **70** mm, parabolas diameter is **2** m

2 3 3 70.00 2.00000 … gradually: point lies on second segment – **2**, point type **P-P - 3**, vertical position is related to minimum cross-section coordinate in Z-axis– **3**, vertical distance from reference point is **70** mm, parabola diameter is **2** m

2 2 1 -70.00 1 0.40000 0.20000 1 … gradually: point lies on second segment – **2**, point type **S-P - 2**, vertical position is related to maximal cross-section coordinate in Z-axis – **1**, vertical distance from reference point is **-70** mm, horizontal position is related to left segment edge – **1**, horizontal distance from reference point is **0.4**, length of straight tendon part is **0.2**, values of horizontal distance and length of straight part are relative to tendon segment length - **1**

2 3 3 70.00 2.00000 … gradually: point lies on second segment – **2**, point type **P-P - 3**, vertical position is related to minimal cross-section coordinate in Z-axis-**3**, vertical distance from reference point is **70** mm, parabolas diameter is **2** m

3 3 3 70.00 2.00000 …gradually: point lies on third segment – **3**, point type **P-P - 3**, vertical position is related to minimal cross-section coordinate in Z-axis-**3**, vertical distance from reference point is **70** mm, parabolas diameter is **2** m

3 2 1 -70.00 1 0.40000 0.30000 1 … gradually: point lies on third segment – **3**, point type **S-P - 2**, vertical position is related to maximal cross-section coordinate in Z-axis – **1**, vertical distance from reference point is **-70** mm, horizontal position is related to left segment edge – **1**, horizontal distance from

	reference point is **0.4**, length of straight tendon part is **0.3**, values of horizontal distance and length of straight part are relative to tendon segment length - **1**
3 1 2 0.00 0.00000	gradually: point lies on third segment – **3**, point type **C** - **1**, vertical position is related to origin of reference curve – **2**, vertical distance from reference point is **0** mm, length of straight part is **0** m
</SpansPoints>	… end of the section for tendon segment characteristic points input
</BondedTendonSpansXZ>	… end of the section for input of geometry in uncoiled view XZ
</BondedTendon>	… end of the section for one tendon definition
</BondedTendons>	… end of the section for all tendons definition

10 Prestressing forces design

The program enables to evaluate actions caused by tendon on concrete member and to balance effects of external loads by tendon layout design and prestressing forces design.

10.1 Equivalent load

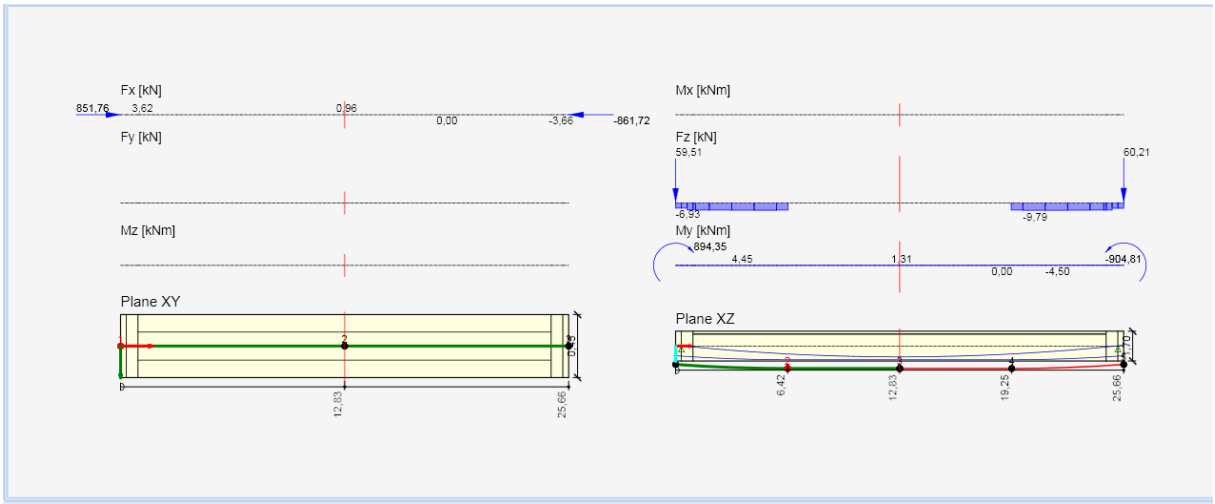
To evaluate actions caused by tendon on concrete member (equivalent load) for current design member click navigator command **Force design > Equivalent loads**.

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

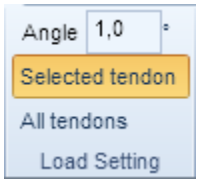
Tabs for tendons editing and textual presentation of equivalent loads are displayed in the **Data window**. Particular tabs in the **Data window**:

- **Tendons** – table with tendons properties. Equivalent load courses update automatically after tendon properties change – see **9.3 Tendons input and edit**.
- **Tendon geometry XY** – table with tendon geometry properties in XY-plane.
- **Tendon geometry XZ** – table with tendon geometry properties in XZ-plane.
- **Pretensioned group** – table with properties of pre-tensioned tendons.
- **Report** – textual presentation of equivalent loads

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



10.1.1 Ribbon group Load setting

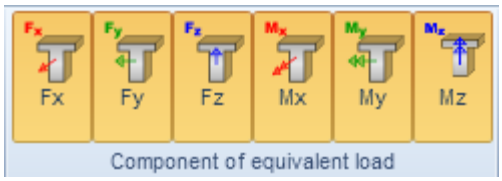


Calculation precision and evaluated tendons can be set in ribbon group Load setting.

- **Angle** – value of tendon maximal angular change (geometrical discretisation) for tendon losses and equivalent load calculation.
- **Selected tendon** – turn on evaluation of equivalent load courses only for current tendon in current design member.

- **All tendons** - turn on evaluation of equivalent load courses for all tendons in current design member.

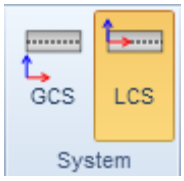
10.1.2 Ribbon group Component of equivalent load



Components of equivalent load to be drawn can be set in ribbon group **Component of equivalent load**.

- **Fx** – turn on/off drawing of force Fx in coordinate system specified in ribbon group **System**.
- **Fy** – turn on/off drawing of force Fy in coordinate system specified in ribbon group **System**.
- **Fz** turn on/off drawing of force Fz in coordinate system specified in ribbon group **System**.
- **Mx** – turn on/off drawing of moment Mx in coordinate system specified in ribbon group **System**.
- **My** – turn on/off drawing of moment My in coordinate system specified in ribbon group **System**.
- **Mz** – turn on/off drawing of moment Mz in coordinate system specified in ribbon group **System**.

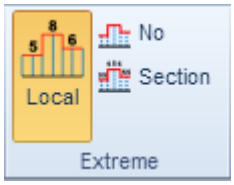
10.1.3 Ribbon group System



Coordinate system for evaluation of equivalent loads can be set in ribbon group **System**.

- **GCS** – turn on evaluation of equivalent loads in global coordinate system.
- **LCS** – turn on evaluation of equivalent loads in local coordinate system of design member.

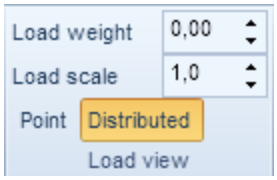
10.1.4 Ribbon group Extreme



Description of equivalent load values can be set in ribbon group **Extreme**:

- **Local** – values of local extremes of equivalent load are depicted along the design member.
- **No** – no values of equivalent load are depicted.
- **Section** – values of equivalent loads are depicted in each section.

10.1.5 Ribbon group Load view



Ribbon group **Load view** can be used to set drawing options of equivalent load.

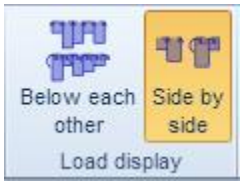
- **Load weight** – number, whose positive value defines size ratio of drawn loads to maximal load value. E.g. when the value is 0, all loads are drawn in their real size, when the value is 1, all loads are drawn in the same size. When negative value is entered, loads, which are less than maximal load value multiplied by absolute value of load weight, are not drawn. It means that for load weight value -0,5 only loads, which are greater than half of maximal load, are drawn, for value -1 only maximal load is drawn.
- **Load scale** – value of multiplier for drawing of load effects.
- **Point** – draw calculated equivalent loads as point loads in points generated by discretisation of tendon.

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

10.1.6 Ribbon group Uncoiled view

See **8.1.6 Ribbon group Uncoiled view**

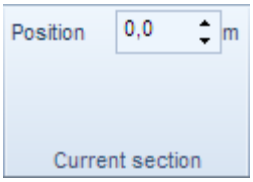
10.1.7 Ribbon group Load display



Pattern of load pictures can be set in ribbon group Load display.

- **Below each other** – draw particular components of equivalent loads below each other.
- **Side by side** – draw components Fx, Fy and Mz in one column and components Mx, Fz and My in the second column.

10.1.8 Ribbon group Current section



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

10.2 Load balancing

To display equivalent loads together with external loads actions click navigator command **Force design > Load balancing**.

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

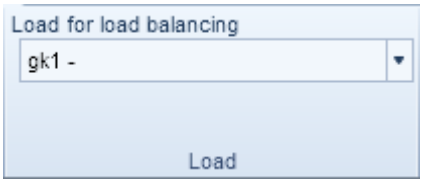
- Unbalanced load – course of difference between actions of external loads caused by current load case/combination and prestressing actions caused by current tendon or all tendons in current design member
- External load – course of actions of external loads caused by current load case/combination
- Equivalent prestressing load – course of equivalent load caused by current tendon or all tendons.

Tabs for tendons editing and textual presentation of load balancing are displayed in the **Data window**. Particular tabs in the **Data window**:

- **Load balancing** – display table with basic information about load balancing. Table contains following columns:
  - **Tendon** – number of evaluated tendon or pre-tensioned group.
  - **LC prestressing** – name of load case, which is determined to transfer effect of corresponding prestressing into analysis model in superior application
  - **Strands** – number of strands in post-tensioned tendon.
  - **Section balancing** –ratio between effects of external loads and effect of prestressing in current section
  - **Tendon balancing** – ratio between effects of external loads and effect of prestressing over the whole design member
  - **Locked** – indication of locked tendon
- **Tendon geometry XY** – table with tendon geometry properties in XY-plane.
- **Tendon geometry XZ** – table with tendon geometry properties in XZ-plane.
- **Pretensioned group** – table with parameters of pre-tensioned tendons.
- **Report** – textual presentation of unbalanced loads

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

10.2.1 Ribbon group Load



Use ribbon group **Load** to set current load case/combination to be balanced.

- **List of load cases/combinations**– select current load case or combination

10.2.2 Ribbon group Load setting

See **10.1.1 Ribbon group Load setting**.

10.2.3 Ribbon group Direction

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

- **Y** – draw load balancing results in direction of Y-axis of current design member local coordinate system.
- **Z** – Draw load balancing results in direction of Z-axis of current design member local coordinate system.

10.2.4 Ribbon group Extreme

See **10.1.4 Ribbon group Extreme**

10.2.5 Ribbon group Load view

See **10.1.5 Ribbon group Load view**

10.2.6 Ribbon group Uncoiled view

See **8.1.6 Ribbon group Uncoiled view**

10.2.7 Ribbon group Current section

See **10.1.8 Ribbon group Current section**

10.3 Calculation of linear elastic stress

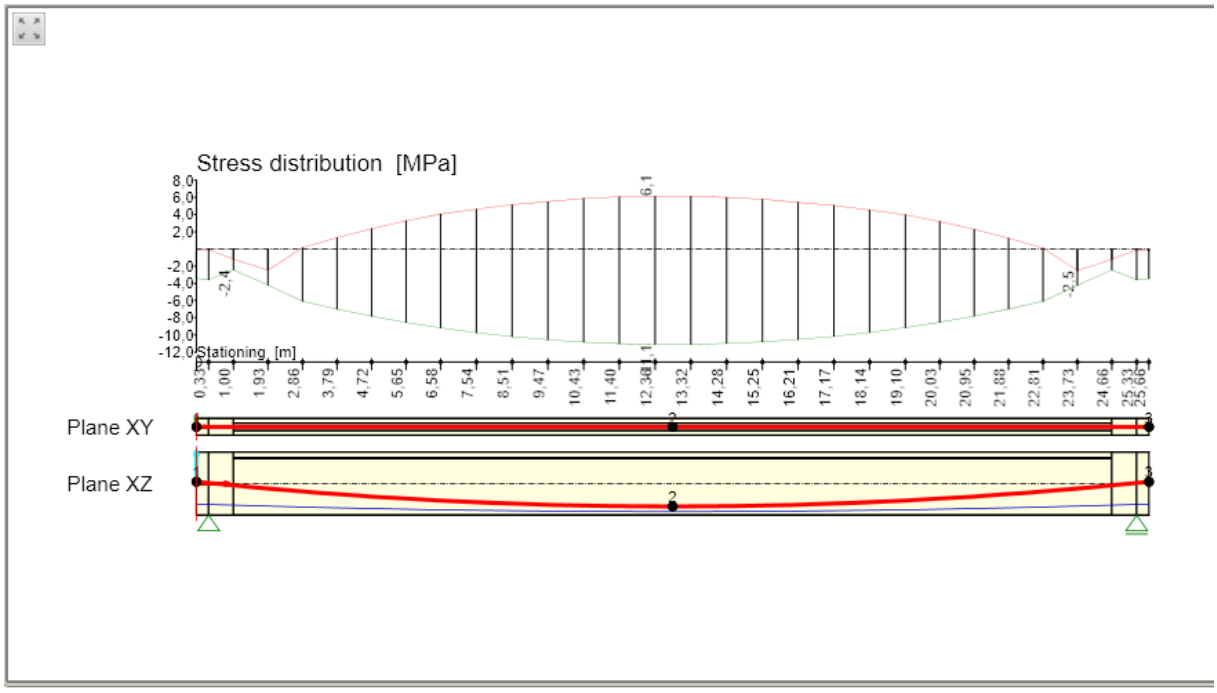
To run calculation of linear elastic stress along the design member click navigator command **Force design > Linear elastic stress**.

Course of calculated linear elastic stress along the current design member is displayed in the main window.

Tabs for tendons editing and textual presentation of calculated stress are displayed in the **Data window**. Particular tabs in the **Data window**:

- **Tendons** – table with tendons properties. Equivalent load courses update automatically after tendon properties change – see **9.3 Tendons input and edit**.
- **Tendon geometry XY** – table with tendon geometry properties in XY-plane.
- **Tendon geometry XZ** – table with tendon geometry properties in XZ-plane.
- **Pretensioned group** – table with properties of pre-tensioned tendons.
- **Report** – textual presentation of calculated linear stress.

Ribbon groups **Result class**, **Uncoiled view** and **Linear elastic stress** are available for calculation of linear stress.



Data

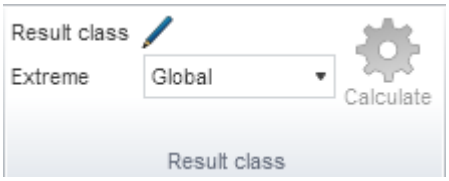
Tendons | Tendon geometry XY | Tendon geometry XZ | Report

Linear elastic stress

Member	Dx [m]	Combination	N [kN]	M_y [kNm]	M_z [kNm]	σ [MPa]	Extreme point
2	1,00	SLS Char ST(6)(5)	-1435,0	-122,6	0,0	-2,4	Min
3	12,36	SLS Char ST(6)(5)	-1549,7	1257,7	0,0	6,1	Max
3	12,36	SLS Char ST(2)(4)	-1761,1	1095,9	0,0	-11,1	Min
3	23,73	SLS Char ST(6)(5)	-1391,8	142,3	0,0	-2,5	Max

Combination	Critical load effect description
SLS Char ST(6)(5)	SW (2) + G (2) + POST (2) + G (6) + Q
SLS Char ST(2)(4)	SW (2) + G (2) + POST (2) + Q

10.3.1 Ribbon group Result class

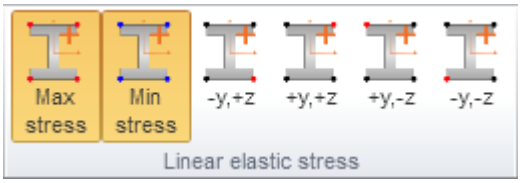


- - load modification of content of result class to calculate linear elastic stress.
- **Extreme** – select mode to evaluate linear elastic stress:
  - **No** – all calculated stresses in all sections are printed.
  - **Section** – extreme stress from all fibres is found for each individual section.
  - **Member** – extreme stress from all sections is found for each individual member of the design member.
  - **Global** – extreme stress from the whole design member is found.
- **Vypočítat** – spustí výpočet lineárních elastických napětí.

10.3.2 Ribbon group Uncoiled view

See **8.1.6 Ribbon group Uncoiled view**.

10.3.3 Ribbon group Linear elastic stress



- **Max. stress** – switch on evaluation of maximum positive stress (max. tension) calculated in all fibers of cross-section.
- **Min. stress** – switch on evaluation of minimum negative stress (max. compression) calculated in all fibers of cross-section.
- **-y, +z** – switch on/off evaluation of stress calculated in the fiber with the maximum positive z- and maximum negative y-coordinates. To find this fiber all phases of cross-section are considered, which already exist in respective construction stage.
- **+y,+z** - switch on/off evaluation of stress calculated in the fiber with the maximum positive z- and y-coordinates. To find this fiber all phases of cross-section are considered, which already exist in respective construction stage..
- **+y,-z** – switch on/off evaluation of stress calculated in the fiber with the maximum negative z- and maximum positive y-coordinates. To find this fiber all phases of cross-section are considered, which already exist in respective construction stage..
- **-y,-z** - switch on/off evaluation of stress calculated in the fiber with the maximum negative z- and y-coordinates. To find this fiber all phases of cross-section are considered, which already exist in respective construction stage.

11 Tendon losses calculation

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

Analysis geometry is composed similar as described in **9.1 3D tendon geometry**. The information about tendon curvature between particular calculation points and real lengths between those points is stored. Moreover information required for calculation of tendon on refracted member is added– see **9.5 Discontinuous tendons on member of polygonal shape**.

Tendon analysis geometry depends on tendons segment division. Elements are created by this division, on which calculation of losses is performed. Value of maximal angular change between tangents to tendon segment ends determines size of tendon parts. It is maximal angle, so some parts may have the angle smaller. The tendon parts must correspond with member parts, so they have to begin and to end at the beginning or the ending of the member.

Example: Straight tendon on straight member. This member has three parts of member. Even no angular change along the member exist, tendon is split to three segments.

Analysis values of particular tendons are specified during input of tendons in navigator **Tendons > Tendons layout**. To edit analysis values click edit button  in column **Detail**. According to origin of stressing values are entered for the beginning or the ending or both ends of tendon. When stressing procedure with relaxation correction is set, time to keep stress has to be entered.

Maximal angular change is valid for all tendons. The recommended basic value is approx. 3°. Lesser values do not have significant influence to calculation precision.

11.1 Overall evaluation of tendon losses on the design member

To display overall results of tendon losses click navigator command **Short term losses > Overall**.

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

Overall report of tendon losses on current design member is displayed it the **Data window**:

- Table with particular tendon values (area, length, cumulative angular change, minimal radius, theoretical tendon elongation before anchoring etc.).
- Summary table of minimal and maximal stress in tendons with value of maximal allowed stress acc. to EN 1992-1-1 5.10.3(2).

Prestressing

Name	Material	A_p [mm2]	Length [m]	L_s [m]	L_{arc} [m]	R_{min} [m]	θ [°]
	Strands	σ_a [MPa]	σ_{min} [MPa]	σ_{max} [MPa]	e_{ba} [mm]	e_{aa} [mm]	L_{set} [m]
Tendon 1	Y1770S7-12.9	700	25,67	12,83	12,84	89,46	8,2
	7	1310,00	1219,76	1264,08	167	164	18,12
Tendon 2	Y1770S7-12.9	700	25,71	0,00	25,71	123,78	11,8
	7	1321,00	1227,11	1273,94	168	165	12,65
Tendon 6	Y1770S7-12.9	700	25,67	12,83	12,84	89,46	8,2
	7	1310,00	1219,76	1264,08	167	164	18,12

Name	$\sigma_{ini,max}$ [MPa]	σ_{lim} [MPa]	Check 5.10.2.1(1)P	σ_{min} [MPa]	σ_{max} [MPa]	σ_{pm0} [MPa]	Check 5.10.3(2)P
Tendon 1	1310,00	1350,00	✔	1219,76	1264,08	1275,00	✔
Tendon 2	1321,00	1350,00	✔	1227,11	1273,94	1275,00	✔
Tendon 6	1310,00	1350,00	✔	1219,76	1264,08	1275,00	✔

Symbol	Explanation
A_p	Area of tendon
Length	Length of tendon
L_s	Sum of lengths of straight parts of tendon
L_{arc}	Sum of lengths of curved parts of tendon
θ	Cumulative angular change
R_{min}	Minimum radius
σ_a	Anchorage stress
σ_{min}	Minimum stress
σ_{max}	Maximum stress
e_{ba}	Theoretical tendon elongation before anchoring
e_{aa}	Theoretical tendon elongation after anchoring
L_{set}	Length affected by anchorage set
$\sigma_{ini,max}$	Maximum initial stress in tendon
σ_{lim}	Limit tendon stress
Check 5.10.3(2)P	Maximum stress check applied to the tendon acc. 5.10.2.1 (1)P
σ_{pm0}	Stress after anchoring
Check 5.10.3(2)P	Maximum stress check applied to the tendon acc. 5.10.3 (2)P

Ribbon groups **Uncoiled view** and **Tendon shape** are available for tendon losses evaluation.

11.1.1 Ribbon group Uncoiled view

See **8.1.6 Ribbon group Uncoiled view**

11.1.2 Ribbon group Tendon shape

Set the drawing of tendon next to picture of design member.

- **Not draw** – do not draw tendon shape outside the design member.
- **Current tendon**- draw only the current tendon shape outside the design member.
- **All tendons** – draw all tendons shapes outside the design member.

11.2 Detailed evaluation of short-term losses

To evaluate short-term losses in details click navigator command **Short-term losses** > **Tendon stress/Losses**.

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

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

- Table of minimal and maximal stress in tendon with maximal allowable tendon stress acc. to EN 1992-1-1 5.10.3(2).
- Table with detailed description of current tendon (tendon area, tendon length, cumulative angular change, minimal radius, theoretical elongation before anchoring etc.)
- Table with detailed output of losses in sections according to specified distance for evaluation. In addition to those sections results are presented in characteristic sections – points of anchorage set impact or points of intersection of frictional losses for stressing from both ends.

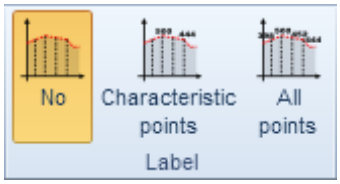
Ribbon groups **Losses**, **Labels** and **Labels orientation** is available when evaluating tendon losses.

11.2.1 Ribbon group Losses

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

- **Distance** – value of distance between sections for graphical and textual evaluation of tendon losses. This value does not affect the precision of calculation.
- **Nula** – set the minimal value on stress axis in graph to 0.
- **Min** – minimal value on stress axis in graph to suitable value according to minimal value of stress in tendon (e.g. if minimal tendon stress is 739,3 MPa, minimal stress value in graph is set to 700 MPa).

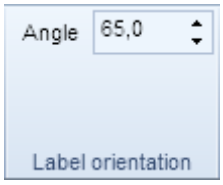
11.2.2 Ribbon group Labels



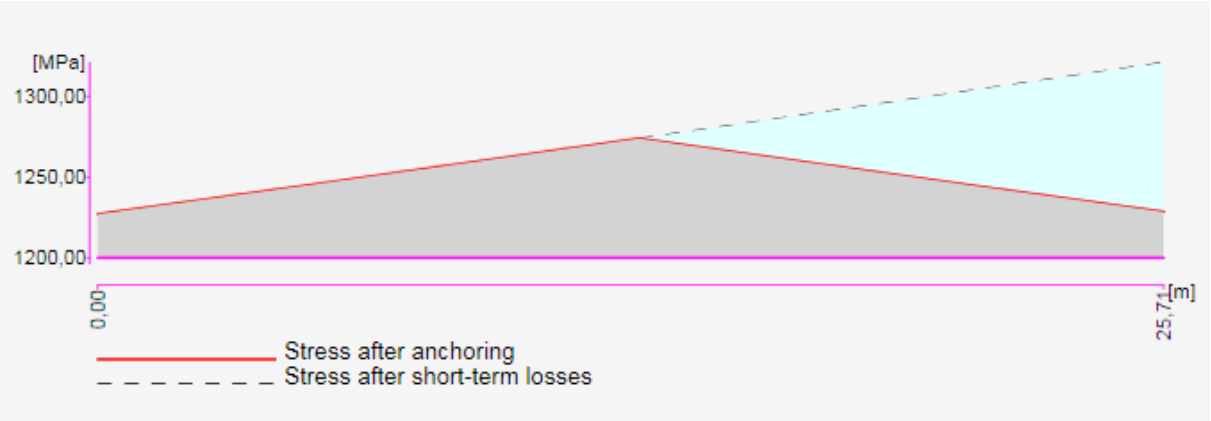
Use ribbon group Labels to set depiction of sections in graph.

- **No** – turn off labels for all sections
- **Characteristic points** – turn on labelling of graph in characteristic points – points of anchorage set impact or points of intersection of frictional losses for stressing from both ends.
- **All sections** – turn on labelling in characteristic points and in all points according to specified distance for evaluation of losses.

11.2.3 Ribbon group Labels orientation



- **Angle** – value of angle to draw labels of stress values.



Calculation of short-term losses

Tendon : Tendon 2

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

Minimum stress [MPa]	Maximum stress [MPa]	Stress after anchoring σ pm0 [MPa]	Check of stress
1227,11	1273,94	1275,00	✔

Input values and intermediate results

Area of tendon	700 mm2
Length of tendon	25,71 m
Sum of lengths of straight parts of tendon	0,00 m
Sum of lengths of curved parts of tendon	25,71 m
Cumulative angular change	11,8 °
Minimum radius	123,78 m
Anchorage stress	1321,00 MPa
Minimum stress	1227,11 MPa
Maximum stress	1273,94 MPa
Theoretical tendon elongation before anchoring	168 mm
Theoretical tendon elongation after anchoring	165 mm
Length affected by anchorage set - end	12649 mm

Short-term losses

d x [m]	Δσ pμ [MPa]	Δσ pw [MPa]	Δσ pr [MPa]	σ pr,cor [MPa]	σ p0 [MPa]	Δσ pr [MPa]	Δσ pr,cap [MPa]
0,00	-93,89	0,00	0,00	0,00	1227,11	-1,87	-66,21
1,00	-90,39	0,00	0,00	0,00	1230,61	-1,93	-66,96
2,00	-86,87	0,00	0,00	0,00	1234,13	-2,00	-67,71
3,00	-83,33	0,00	0,00	0,00	1237,67	-2,07	-68,48
4,00	-79,78	0,00	0,00	0,00	1241,22	-2,15	-69,26
5,00	-76,22	0,00	0,00	0,00	1244,78	-2,23	-70,05
6,00	-72,64	0,00	0,00	0,00	1248,36	-2,31	-70,85
7,00	-69,05	0,00	0,00	0,00	1251,95	-2,39	-71,66
8,00	-65,45	0,00	0,00	0,00	1255,55	-2,48	-72,48
9,00	-61,84	0,00	0,00	0,00	1259,16	-2,57	-73,31
10,00	-58,21	0,00	0,00	0,00	1262,79	-2,66	-74,15
11,00	-54,58	0,00	0,00	0,00	1266,42	-2,76	-75,01
12,00	-50,93	0,00	0,00	0,00	1270,07	-2,86	-75,87
13,00	-47,27	0,00	0,00	0,00	1273,73	-2,96	-76,75

12 Internal forces evaluation

After recalculation of the structure in superior FEM program internal forces on design members can be evaluated, taken into account construction stages.

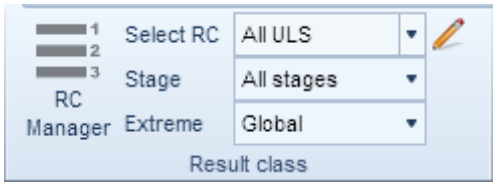
To evaluate internal forces on design member click navigator command **Design member result > Internal forces**.

Courses of internal forces for current result class and current design members are drawn in the **Main window**.

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

To set evaluation of internal forces ribbon groups **Result class**, **Internal forces**, **Prestressing** and **Labels orientation** are available.

12.1 Ribbon group Result class



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

- **RC manager** – add or edit result class – see **13.3.8 Result class manager**.

- **Select TV** – select from the list the current class, for which evaluation of internal forces is performed. Click edit button  to edit content of current result class.
- **Stage** –filter out from current result class only combinations, which are not defined in selected stage.
  - **All stages** – evaluate results from all combinations (load cases, load groups) in current result class without respecting construction stages
  - **„Stage“** – evaluate results only for those combinations (load cases, load groups) in current result class, which are defined in selected stage.
- **Extreme** – select mode of evaluation of extremes:
  - **No** – no extreme is evaluated
  - **Section** – extreme values of evaluated components are searched for each section
  - **Member** – extreme values of evaluated components are searched for each particular member of design member
  - **Global** – extreme values of evaluated components are searched for the whole design member

12.2 Ribbon group Internal forces

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

- **N** –turn on/off drawing of axial force N
- **Vy**- turn on/off drawing of shear force Vy
- **Vz** – turn on/off drawing of shear force Vz
- **Mx**- turn on/off drawing of torsion moment Mx
- **My** – turn on/off drawing of bending moment My
- **Mz** – turn on/off drawing of bending moment Mz

12.3 Ribbon group Prestressing

For result class, which contains only load case for effects of prestressing, effects to be evaluated can be set.

- **Effects of prestressing** – evaluate total effects of prestressing
- **Primary forces** – evaluate statically determinable internal forces (primary effects) of prestressing
- **Secondary forces** – evaluate secondary (statically indeterminable) effects of prestressing

12.4 Ribbon group Labels orientation

See **11.2.3 Ribbon group Labels orientation**

12.5 Ribbon group Calculate

Click Calculate FEM to recalculate analysis model in superior FEM application. Values in load case for transfer of prestressing effects are updated and analysis is performed after clicking this button.

Recalculation is required after tendons layout changes, because values of equivalent load changes due to tendon changes.

13 Design member check

- **If IDEA Tendon is launched from IDEA Beam or IDEA Frame, it is not possible to input positions and to perform checks in IDEA Tendon – the checks are performed in the superior linked application.**

13.1 Design member construction stages

To enter detailed check properties of design member click navigator command **Design member check > Construction stages**. Default values are taken from construction stages of the whole project.

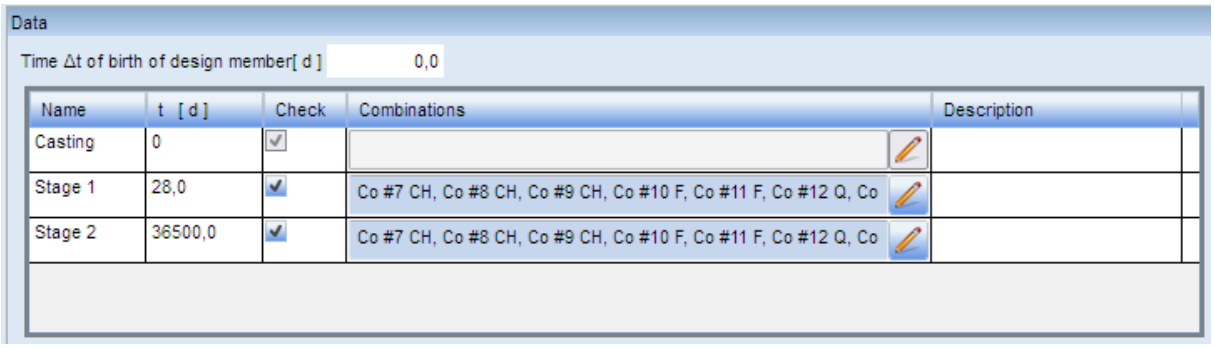
Time axis of current design member is drawn in the **Main window**.

Current design member properties table is displayed in the **Data window**.

Value of **Time shift Δt of birth of design member** can be specified for current design member. Time shift must be less or equal the time of first stage minus 3 days, because required material characteristics are not specified for concrete younger than 3 days

Particular columns in design member properties table:

- **Name** – name of stage
- **t** – local time of stage, which is calculated from specified value of time shift Δt
- **Check** – turn on/off, if particular stage is taken into account in current design member check
- **Combinations** – list of combinations assigned to particular stages can be edited for current design member check.



13.2 Reinforcement zones

Program IDEA RCS is used to check the design member. IDEA RCS designs and checks the reinforced sections. Each section has associated one reinforced cross-section.

To be able to design the reinforcement, reinforcement zones have to be defined along the design member and reinforcement has to be defined using reinforcement templates. Each zone corresponds to one section and each template corresponds to reinforced section in IDEA RCS.

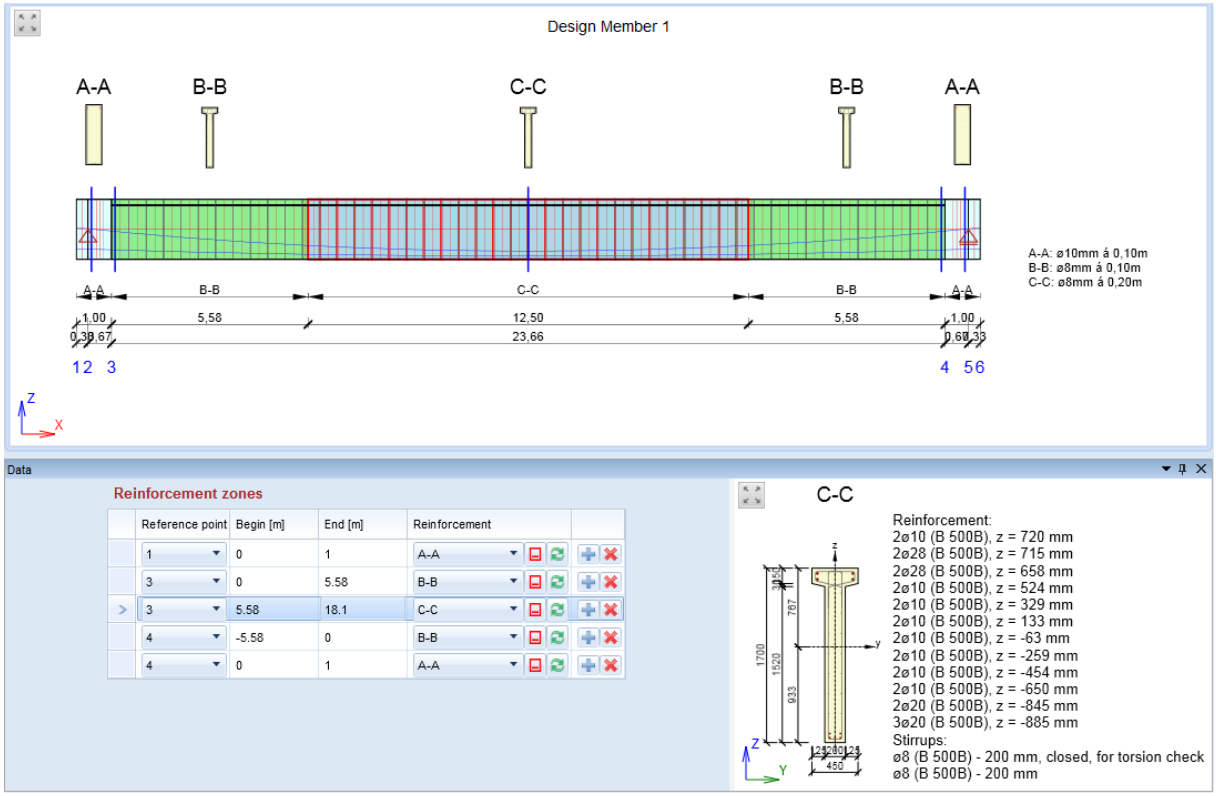
Click navigator command **Design member check** > **Reinforcement** to input reinforcement zones and reinforcement to zones.

After the zones are defined, they can be reinforced by concrete reinforcement using reinforcement templates. Or after the check positions are defined, the detailed check in IDEA RCS can be started and the concrete reinforcement can be defined in IDEA RCS.

To generate reinforcement zones according to patterns the zone templates can be used – see **13.2.1 Zone templates**.

Ribbon groups **Zone templates**, **View settings**, **Scale**, **Internal forces** and **Detailed view** are available during the input of reinforcement zones.

The design member with defined reinforcement zones is drawn in the main window. A table for zones and reinforcement editing is displayed in the Data window. The detailed picture of reinforced section of the current reinforcement zone is drawn in the right part of the Data window.

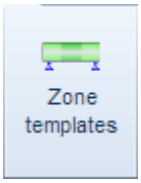


The **Reinforcement zones** table contains following columns:

- **Reference point** – set the number of node. The coordinates defined in columns **Begin** and **End** are related to this point.
- **Begin** – position of zone beginning measured from reference point.
- **End** - position of zone beginning measured from reference point.

- **Reinforcement** – select the reinforcement template associated to the zone. Click edit button to edit the reinforcement in the zone. Click to input new reinforcement template, which is assigned to the current zone. Such created reinforcement template is than available for all reinforcement zones, which have the same cross-section.
- - insert new zone. The current zone is split to two halves by inserting new zone.
- - delete the current zone.

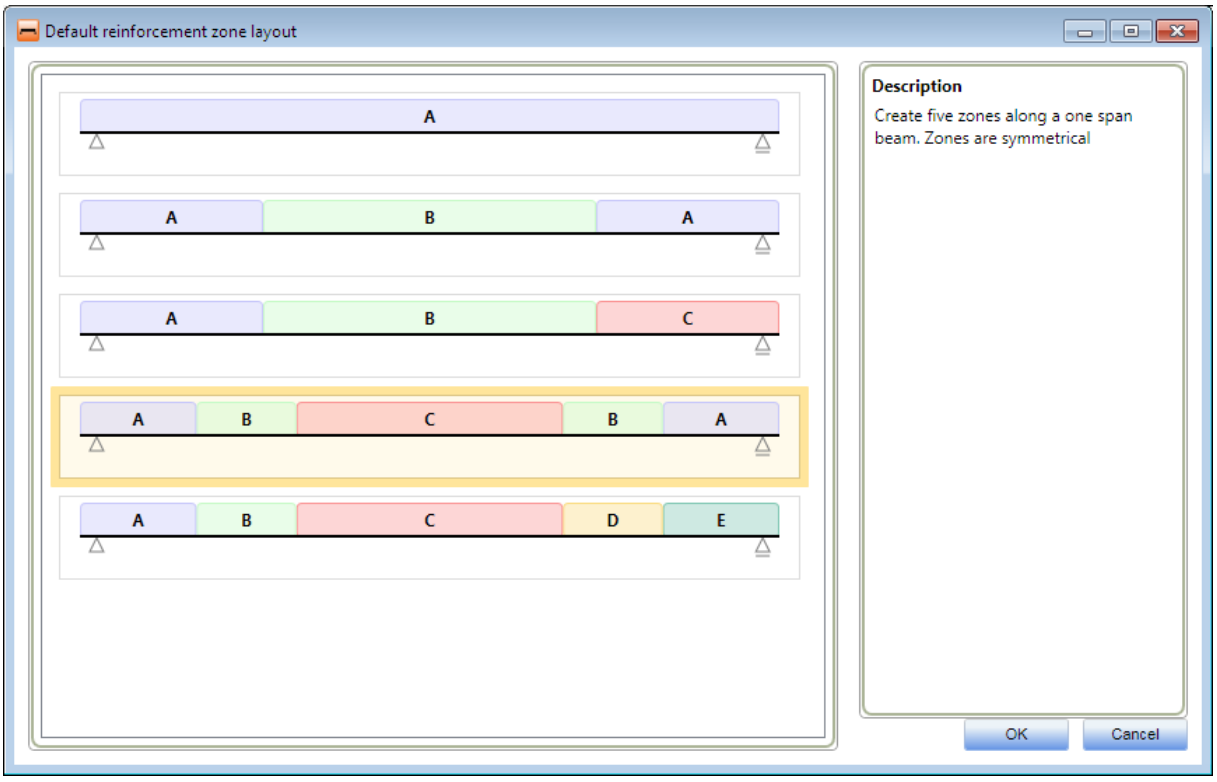
13.2.1 Zone templates



Zone templates can be used to generate zones along the design member. To generate zones according to templates click **Zone templates**.

Zone patterns, which are suitable for the current design member, are displayed in the dialog **Default reinforcement zone layout**.

Click **OK** to generate zones on design member according to the chosen template.

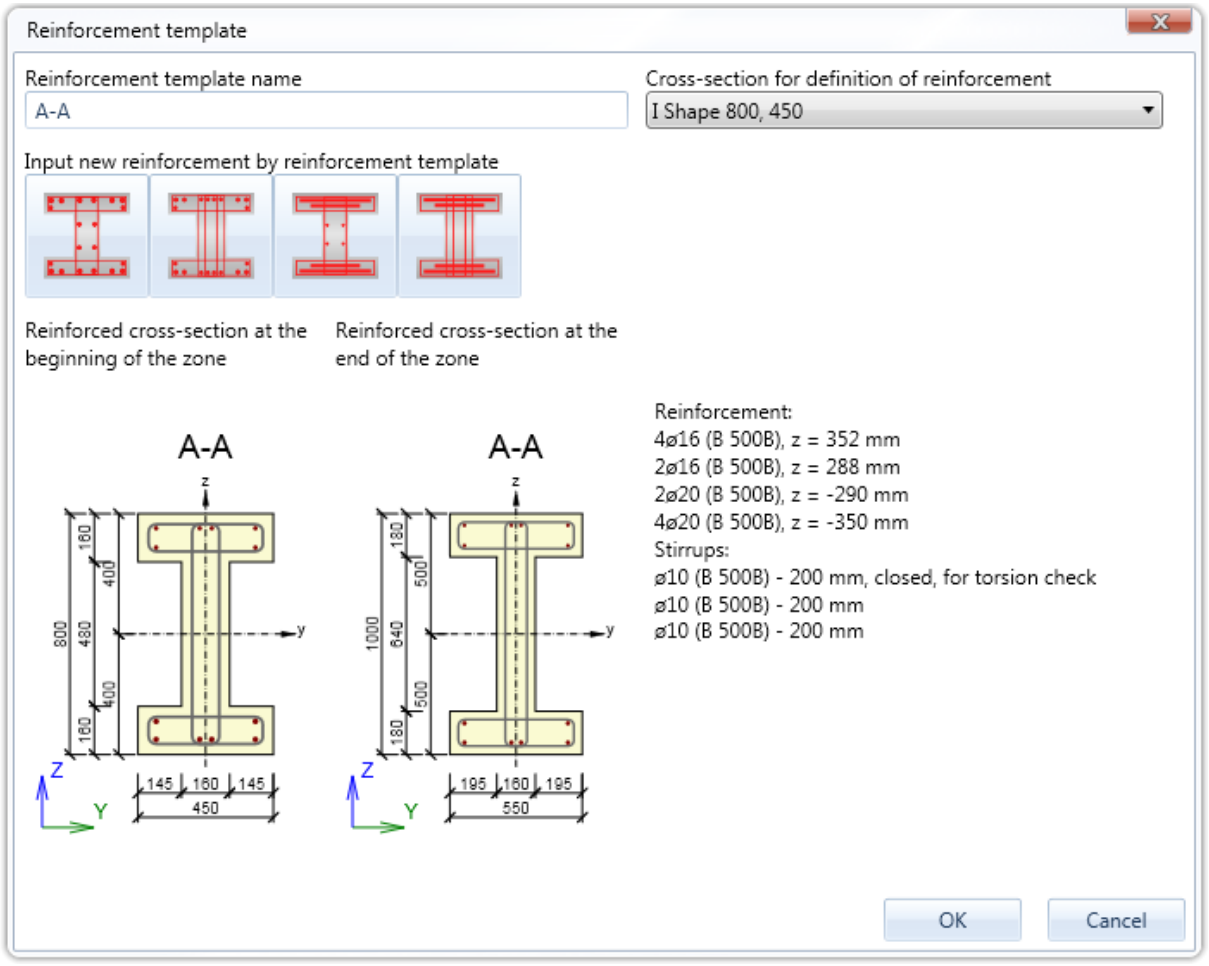


13.2.2 Editing the reinforcement in the zone

The reinforcement in the zone is defined using reinforcement templates. The already defined reinforcement can be edited either by new input using reinforcement template (all reinforcement in the zone is replaced by new one) or IDEA RCS can be run to perform the detailed check of the design member and the reinforcement can be edited.

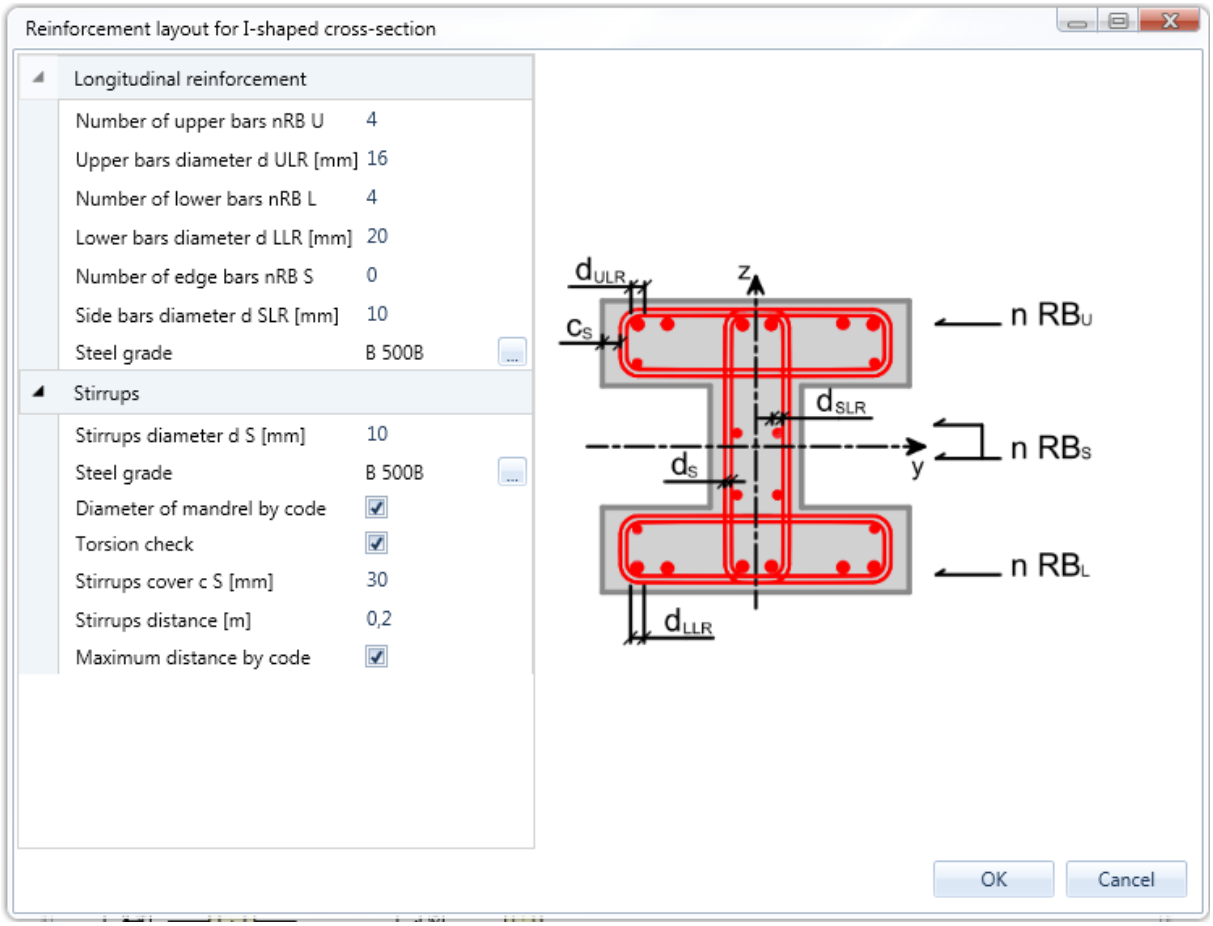
Input of reinforcement can be started

- Clicking the edit button in the **Reinforcement** column it the zones table in the Data window
- Clicking the picture of section above the zone in the Main window.

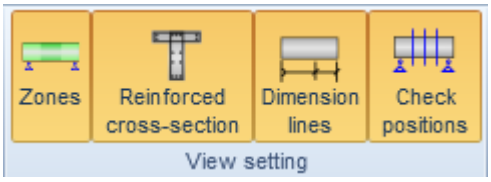


Particular dialog options:

- **Reinforcement template name** – the name of reinforcement template assigned to the current reinforcement zone, is displayed and can be changed in the text box. The name of reinforcement template is included into the name of reinforced cross-section in IDEA RCS.
- **Cross-section for definition of reinforcement** – the list is available, when reinforcing haunches. If the haunch is defined using cross-sections of the not identical shape, the governing cross-section should be selected in the list. The reinforcement template is input into the selected section. The reinforcement is interpolated from the governing section into the haunch than.
- **Input new reinforcement by reinforcement template** – depending on the shape of cross-section in the current zone buttons with pictures of suitable reinforcement templates are displayed. Click the button with required reinforcement pattern to display **Reinforcement for cross-section** dialog. After defining the reinforcement parameters and closing both dialogs by clicking OK the reinforcement is input to the zone.



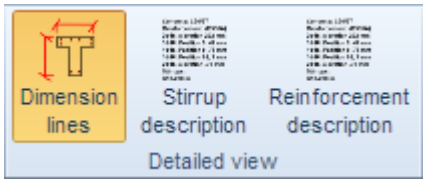
13.2.3 Ribbon group View settings



Use options in ribbon group **View settings** to change the drawing mode of current design member:

- **Zones** – turn on/off the drawing of reinforcement zones in the picture of the design member.
- **Reinforced cross-section** – turn on/off the drawing of reinforced cross-section above the particular zones.
- **Dimension lines** – turn on/off the drawing of dimension lines of the current design member.
- **Check positions** – turn on/off the drawing of defined check positions on the current design member. This option is available only for prestressed design members.

13.2.4 Ribbon group Detailed view

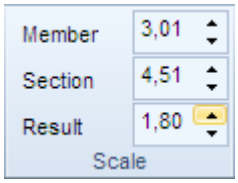


Use options in ribbon group **Detailed drawing** to change the drawing of the detailed reinforced cross-section in the right part of the Data window.

- **Dimension lines** – turn on/off drawing of dimension lines in the detailed picture of reinforced cross-section
- **Stirrup description** – turn on/off drawing of stirrups description in the detailed picture of reinforced cross-section

- **Reinforcement description** – turn on/off drawing of main reinforcement description in the detailed picture of reinforced cross-section

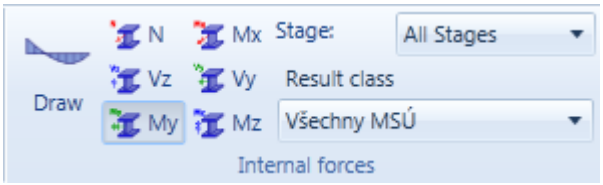
13.2.5 Ribbon group Scale



Use options in ribbon group Scale to set scale ratios for drawing of parts of design member.

- **Member** – set value of exceed scale for drawing of members of design member.
- **Section** – set value of exceed scale for drawing of cross-section pictures above the zones.
- **Results** – set value of scale for drawing of result courses (internal forces, check results...)

13.2.6 Ribbon group Internal forces



Use options in ribbon group **Internal forces** to set drawing mode of internal forces.

- **Draw** – turn on/off the drawing of internal forces along the current design member.
- **N** – switch to drawing of axial force N.
- **Vz** – switch to drawing of shear force Vz.
- **Vy** - switch to drawing of shear force Vy.
- **Mx** – switch to drawing of torsional moment Mx. **My** – switch to drawing of bending moment My.
- **Mz** – switch to drawing of bending moment Mz.
- **Result class** – select the result class to draw the courses of internal forces.

13.3 Positions and check of positions

➢ **If IDEA Tendon is launched from IDEA Beam or IDEA Frame, it is not possible to input positions and to perform checks in IDEA Tendon – the checks are performed in the superior linked application.**

Check of design member is performed in specified positions. For each specified position a section, reinforced cross-section and, construction stages and load extremes are generated. Such generated data are than checked in module IDEA RCS.

To define check positions click navigator command **Design member check > Check positions**

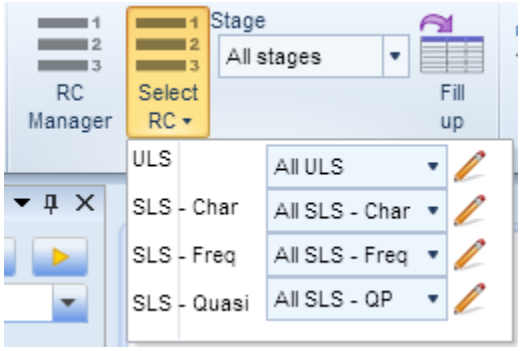
Uncoiled views of current design member are drawn in the **Main window**.

Following tabs are displayed in the **Data window**:

- **Positions** –table of defined positions is displayed on this tab
- **Internal forces** – table with generated load actions for sections generated according to defined sections is displayed on this tab. Presented data are used for check of concrete sections

Ribbon groups **Check**, **Position**, **Design member views, Uncoiled view, Calculation** and **Current design member check** are available when defining positions for check.

13.3.1 Check settings



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

- **RC manager** – add or edit result class – see **13.3.8 Result class manager**.
- **Select RC** – display lists where particular result classes can be assigned to combination types for check:
  - **ULS** –result class selected in this list is used to generate the content of **ULS – fundamental** combination for check of reinforced concrete section. To edit content of current result class click edit button  .
  - **SLS - Char** – result class selected in this list is used to generate the content of **SLS – Characteristic** combination for check of reinforced concrete section. To edit content of current result class click edit button  .
  - **SLS – Freq** – result class selected in this list is used to generate the content of **SLS – Frequent** combination for check of reinforced concrete section. To edit content of current result class click edit button  .
  - **SLS – Quasi** – result class selected in this list is used to generate the content of **ULS – Quasi-permanent** combination for check of reinforced concrete section. To edit content of current result class click edit button  .
- **Stage** –filter out from current result class only combinations, which are not defined in selected stage.
  - **All stages** – evaluate results from all combinations (load cases, load groups) in current result class without respecting construction stages
  - **„Stage“** – evaluate results only for those combinations (load cases, load groups) in current result class, which are defined in selected stage.

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

13.3.2 Ribbon group Positions

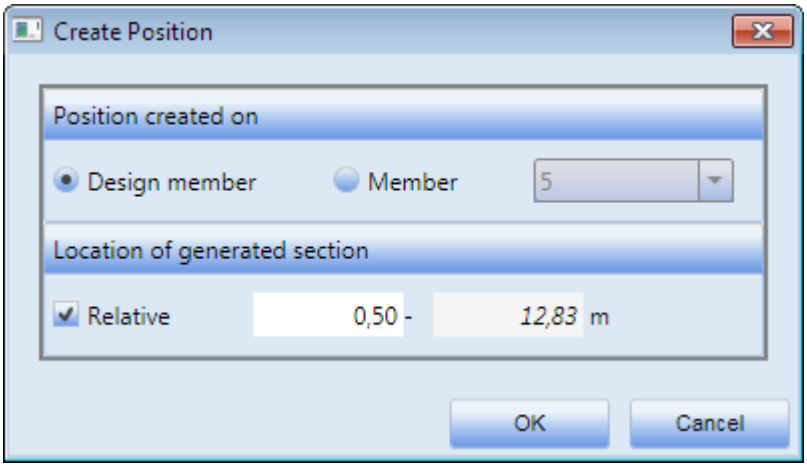
Use commands in ribbon group **Positions** to create or remove positions.

- **New** – add new single position.
- **Delete** – delete selected positions
- **Generate** – run mass generation of positions

- **Delete all** – delete all existing positions on current design member

13.3.2.1 New single position

To input new single position click **New** in ribbon group **Position**.

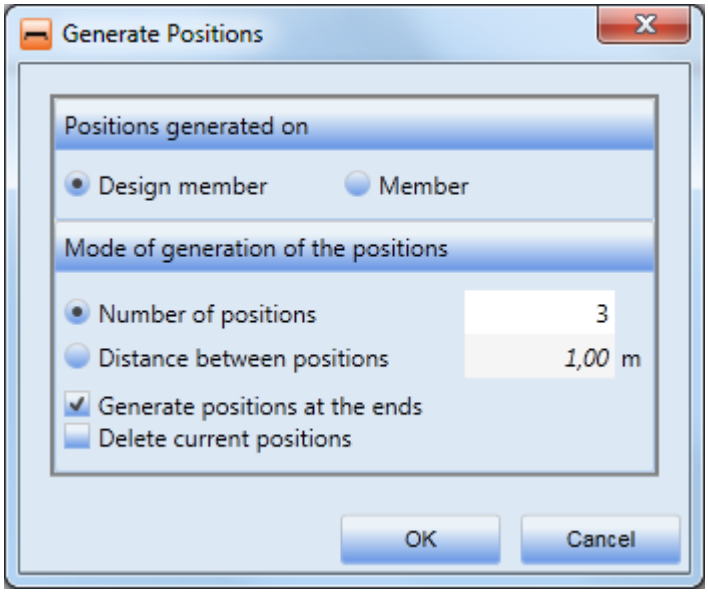


Position of new position is specified in dialog **Create check position**.

- **Design member** – if this option is on, the value of distance is related to the beginning of the current design member.
- **Member** – if this option is on, the value of distance is related to the beginning of member selected in the list
- **Relative** – if this option is on, the relative value to the length of current design member or selected member can be entered. Otherwise the absolute value of distance from beginning of current design member or selected member is entered.

13.3.2.2 Mass generation of positions

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



- **Design member** – if this option is on, settings in group **Mode of generation of positions** are applied on the current design member as whole
- **Member** – if this option is on, settings in group **Mode of generation of positions** are applied each particular member in the current design member.
- **Number of positions** – if this option is on, the entered number of positions is generated regularly along the member or design member.

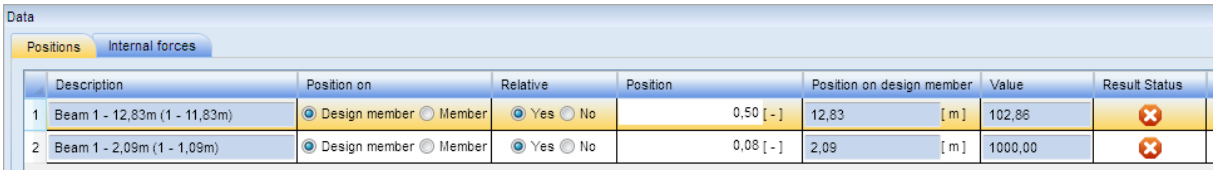
- **Distance between positions** - if this option is on, the positions are generated in entered distance along the whole member or design member.
- **Generate positions at the ends** – if this option is on, positions for check are generated on both ends of current design member or each member in design member.
- **Delete current positions** – if this option is on, all existing positions are deleted during generation of new positions.

13.3.3 Editing positions

Defined positions can be edited it the table on tab **Positions** in the **Data window**.

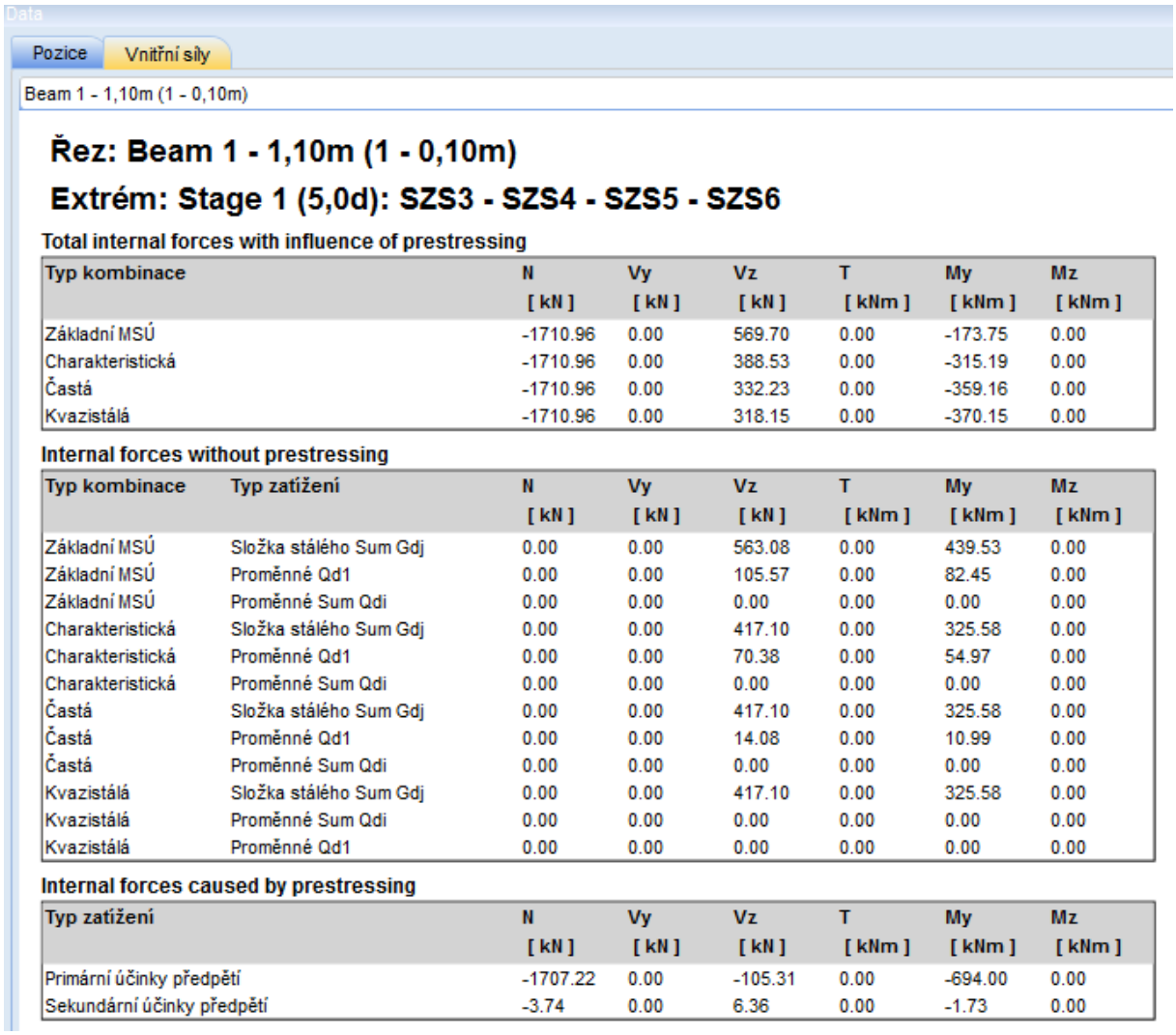
Table **Positions** contains following columns:

- **Description** – display generated name of position. Description contains distance from design member origin, name of member and distance of position from the beginning of the member.
- **Position on** – set origin, which the position of position is related to:
  - **Design member** – if the option is on, the value in column **Position** is related to the origin of design member
  - **Member** – if the option is on, the value in column **Position** is related to the origin of corresponding member
- **Relative** – set mode of evaluation of distance in column **Position**.
- **Position** – distance value related to origin specified in column **Position on**
- **Position on design member** – display the absolute distance of position from the beginning of design member.
- **Value** – display maximal check value in the position.
- **Result status** – display indicator of check status in the position



13.3.4 Internal forces in check positions

Combinations of internal forces for check of design member positions can be reviewed on tab **Internal forces** in the **Data window**.



Data

Pozice | Vnitřní síly

Beam 1 - 1,10m (1 - 0,10m)

Řez: Beam 1 - 1,10m (1 - 0,10m)

Extrém: Stage 1 (5,0d): SZS3 - SZS4 - SZS5 - SZS6

Total internal forces with influence of prestressing

Typ kombinace	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Základní MSÚ	-1710.96	0.00	569.70	0.00	-173.75	0.00
Charakteristická	-1710.96	0.00	388.53	0.00	-315.19	0.00
Častá	-1710.96	0.00	332.23	0.00	-359.16	0.00
Kvazistálá	-1710.96	0.00	318.15	0.00	-370.15	0.00

Internal forces without prestressing

Typ kombinace	Typ zatížení	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Základní MSÚ	Složka stálého Sum Gdj	0.00	0.00	563.08	0.00	439.53	0.00
Základní MSÚ	Proměnné Qd1	0.00	0.00	105.57	0.00	82.45	0.00
Základní MSÚ	Proměnné Sum Qdi	0.00	0.00	0.00	0.00	0.00	0.00
Charakteristická	Složka stálého Sum Gdj	0.00	0.00	417.10	0.00	325.58	0.00
Charakteristická	Proměnné Qd1	0.00	0.00	70.38	0.00	54.97	0.00
Charakteristická	Proměnné Sum Qdi	0.00	0.00	0.00	0.00	0.00	0.00
Častá	Složka stálého Sum Gdj	0.00	0.00	417.10	0.00	325.58	0.00
Častá	Proměnné Qd1	0.00	0.00	14.08	0.00	10.99	0.00
Častá	Proměnné Sum Qdi	0.00	0.00	0.00	0.00	0.00	0.00
Kvazistálá	Složka stálého Sum Gdj	0.00	0.00	417.10	0.00	325.58	0.00
Kvazistálá	Proměnné Sum Qdi	0.00	0.00	0.00	0.00	0.00	0.00
Kvazistálá	Proměnné Qd1	0.00	0.00	0.00	0.00	0.00	0.00

Internal forces caused by prestressing

Typ zatížení	N [kN]	Vy [kN]	Vz [kN]	T [kNm]	My [kNm]	Mz [kNm]
Primární účinky předpětí	-1707.22	0.00	-105.31	0.00	-694.00	0.00
Sekundární účinky předpětí	-3.74	0.00	6.36	0.00	-1.73	0.00

List of internal forces contains:

- **List** of defined positions. Current position, for which list of internal forces is displayed, can be selected in this list.
- **Section** – generated name of section
- **Extreme** – generated name of extreme, which consists of:
  - **Stage 1 (5,0d)** – name of construction stage and its time, as is defined in local time axis of design member
  - **SZZ3** – name of load cases combination from **ULS** result class. Name of ULS combination is always on first position of extreme name. If ULS combination is not specified, the text "**not filled**" is printed instead of combination name
  - **SZZ4** – name of load cases combination from **SLS-Char** result class. Name of SLS-Char combination is always on second position of extreme name.
  - **SZZ5** – name of load cases combination from **SLS-Freq** result class. Name of SLS-Freq combination is always on third position of extreme name.
  - **SZZ6** – name of load cases combination from **SLS-Quasi** result class. Name of SLS-Quasi combination is always on third position of extreme name.
- **Table of internal forces including effect of prestressing** contains total internal forces for **ULS** and **SLS** combinations (names of combinations are included in extreme name) for current extreme including effects of prestressing.

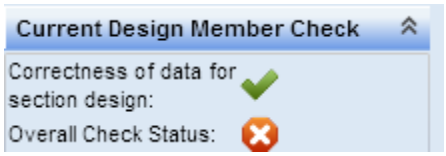
- **Table of internal forces without effects of prestressing** contains internal forces for the same combinations without effects of prestressing. Internal forces are split to internal forces for permanent load components **Sum Gdj** and variable load components **Qd1** and **Sum Qdi**.
- **Table of internal forces caused by prestressing** contains primary and secondary effects of prestressing. Long-term tendon losses are taken into account in effect caused by prestressing.

13.3.5 Sections and extreme for check

After positions are defined and check properties are set, concrete check can be performed. For the concrete check **sections**, **extremes** and appropriate **internal forces** are generated.

Number of generated section equals the number of positions defined on the design member.

Number of generated extremes for one section equals to maximal number of combinations in current result class, whose are valid for selected stage. If the option **Only extremes** is set in ribbon group **Check of current design member**, the maximum of 12 extremes is generated. Extremes and internal forces for selected position can be reviewed in the **Data window** on the tab **Internal forces**.

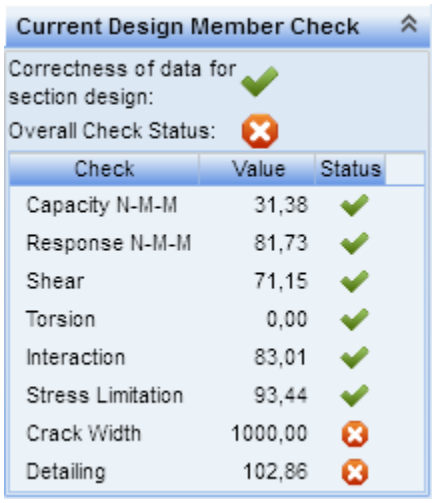


Status of sections generation is displayed in the **Info window**. Click **Info** to display detailed information.

13.3.6 Check of current design member

Use ribbon group **Check of current design member** to perform check of positions in design member.

- **Detailed** – run IDEA RCS module to perform detailed check of current design member.
- **Hidden –** run check of current design member in IDEA RCS on the background.
- **All results** – if this option is on, load extreme is generated for each combination in appropriate result class
- **Extreme only** – if this option is on, extreme values of internal forces are searched from all combinations in result class. The maximum of 12 load extremes is generated for each section.



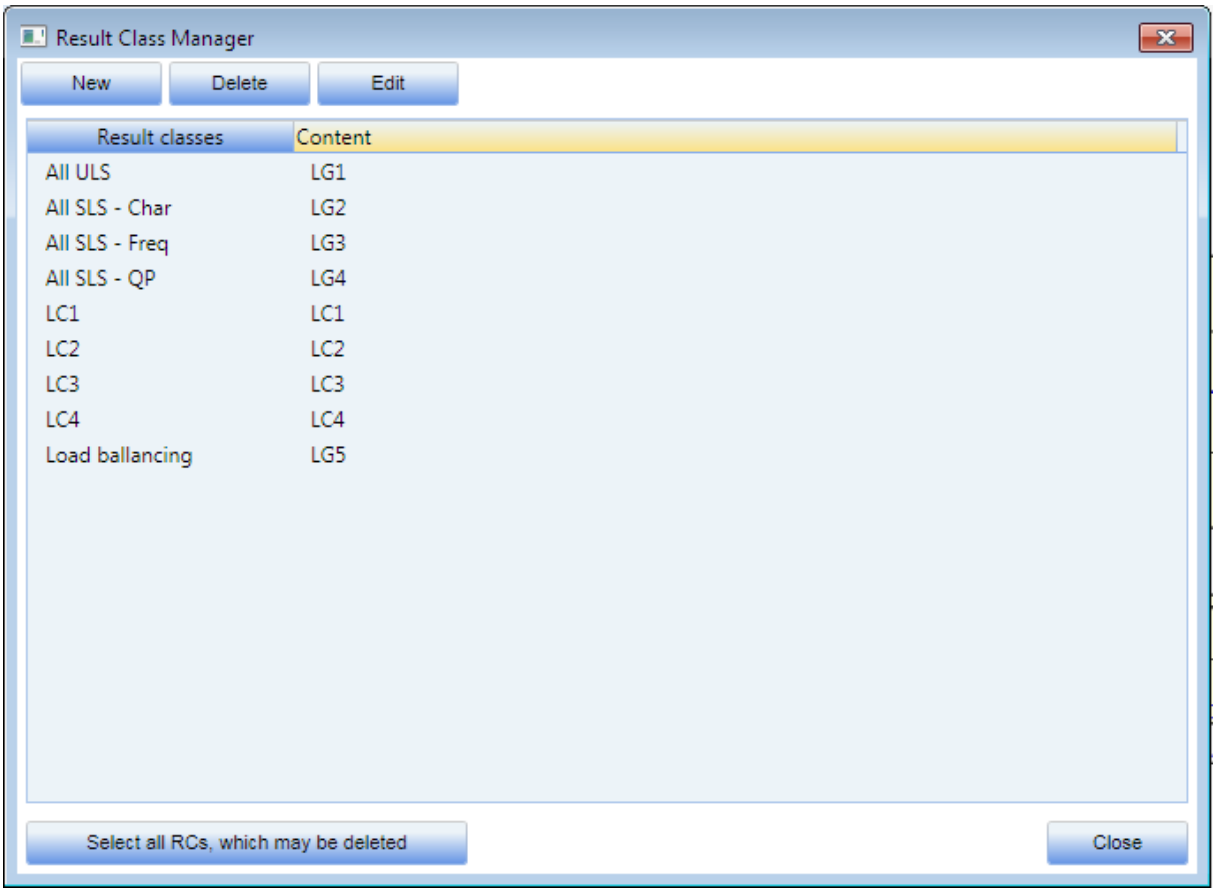
Results of checks of particular positions are displayed in **Data window** in the table with position properties on tab **Positions**. Maximal exploitation value from all particular checks and total status of check is displayed there. Overall check status for the whole design member is displayed in **Info window**.

13.3.7 Result classes

Result class is a group of several load cases or combinations within which extreme values of load effects are found.

13.3.8 Result class manager

Result class manager enables to add new result class, to edit or to delete an existing result class.

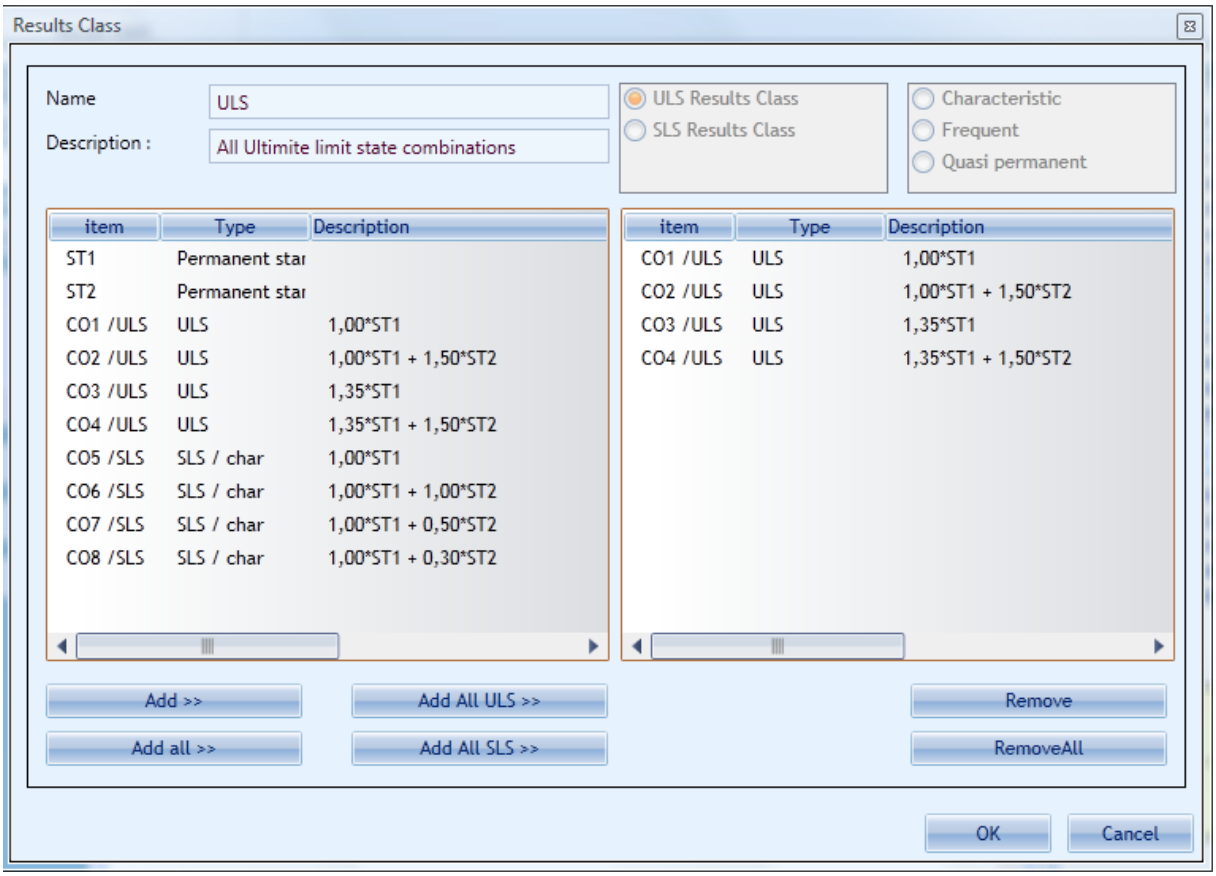


Particular options of **Result class manager** dialog:

- **New** – add new result class to project
- **Delete** – delete selected result class from project, if this class is not assigned to list of classes for evaluation of results or check of the structure
- **Edit** – start edit of selected result class
- **Result classes** – names of existing result classes are displayed in this column
- **Content** – names of load cases or combinations in particular result class are displayed in this column.
- **Select all RCs, which may be deleted** – select all result classes which can be deleted – it means classes, which are not assigned to be evaluated or checked.

13.3.9 New Result Class

To create new result class click **New** in the result class manager. All load cases and all combinations available in project are listed in left list. The content of current result class is displayed in the right list.



Dialog **New Result Class** contains following options:

- **Name** – name of a new class
- **Description** – input of additional description of class
- **Add** >> - add load case or load combination from the left list into the class content. Case or combination can be added also by double-click on item in the left list.
- **Add All** >> - add all load combination from the left list into the class content.
- **Add All ULS** >> - add all ULS load combination from the left list into the class content.
- **Add All SLS** >> - - add all SLS load combination from the left list into the class content.
- **Result Class ULS** – set the class type as ULS.
- **Result class SLS** – set the class type as SLS. For this type also a sub-type has to be defined:
  - **Characteristic**
  - **Frequent**
  - **Quasi-permanent**
- **Delete** – delete selected items (load cases or load combinations) from the right list. Load case or combination can be deleted also by double-click on item in the right list.
- **Delete All** – delete all load cases or load combinations from the right list.

13.3.10 Edit Result Class

To edit result class click Edit in result class manager. Editing of result class is done in the same dialog as for creating a new one. The content of the class can be changed in the right list, but the class type cannot be changed (ULS, SLS).

13.3.11 Limitations of IDEA RCS

If **IDEA RCS** is run from **IDEA Tendon** to check reinforced sections, some functionality is limited or not available.
Following operations are limited:

- Operations with sections, extremes and reinforced sections;
- Operations with cross-section shape;
- Manipulations with tendons and tendon ducts;
- Operations with construction stages;
- Operations with load stages;
- Operations with load actions.

Limited is:

- import of the whole reinforced cross-section, shape of cross-section, tendons and tendon ducts;
- XML import.

Functionality, which is required for working with IDEA RCS and cannot be overtaken from superior application, is not limited – especially:

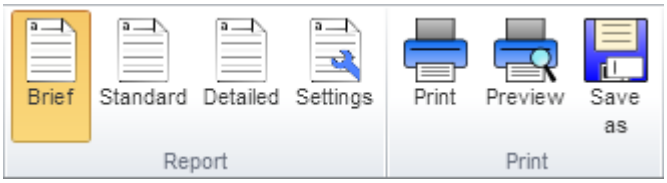
- Application settings, units settings, code settings, project data settings;
- Report, print and report export;
- Input of openings;
- import of reinforcement;
- export of reinforced section, shape of cross-section, reinforcement, tendons and tendon ducts;
- operations with longitudinal reinforcement and shear reinforcement;
- editing of design member data, imperfection data, buckling data and deflection data;
- calculation;
- storing project data for standalone IDEA RCS application.

14 Report

Report can be generated for design members, which have **Print** option enabled in design member properties.

Report for all design members can be generated in navigator **Project data > Design members**. Report for current design member can be generated in navigator **Report**.

14.1 Report for all design members in project



Report for all design members can be generated in navigator **Project data > Design members**. To work with report use ribbon groups **Report** and **Print**.

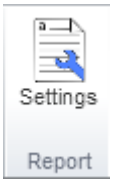
Commands in ribbon group **Report**:

- **Brief** – switch to display brief report for all design members.
- **Standard** – switch to display standard report for all design members.
- **Detailed**– switch to display detailed report for all design members.

Commands in ribbon group **Print**:

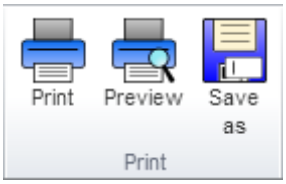
- **Print** –print of the report to the selected print device
- **Preview** – display print preview of the report
- **Save as** – save the report to the file of HTML, MHT (web archive including pictures) or TXT format.

14.2 Current design member report



To generate report for current design member use navigator **Report**. Ribbon groups **Report** and **Print** are available for this navigator.

When navigator command **Report > Settings** is active, click **Settings** in ribbon group **Report** to edit report settings for all design members in project.



When navigator command **Report > Standard** or **Report > Detailed** is active, ribbon group **Print** is available.

Commands in ribbon group **Print**:

- **Print** –print of the report to the selected print device
- **Preview** – display print preview of the report
- **Save as** – save the report to the file of HTML, MHT (web archive including pictures) or TXT format.

14.3 Report types

14.3.1 Brief report

To generate brief report for all design members click **Brief** in ribbon group **Report** in navigator **Project data > Design members**. Brief report contains only table with description and overall check result of design members in project. The content of brief report can be

affected only by selection of design members to be printed, not by report settings.

1. Brief summary of results of design member

Design Name	Member Description	Members	Tendons	Valid	Value [%]	Result Status
Beam 1	Description 1	5, 4, 1, 2, 3	Tendon 2, Tendon 1, Tendon 6	✓	1000,00	✗

14.3.2 Standard report

To generate standard report for all design members click **Standard** in ribbon group **Report** in navigator **Project data >. Design members** or click navigator command **Report > Standard** to generate standard report for current design member only.

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

Table of contents

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

1. Project data

Name	Vorgespannter_Dachbinder
Author	--- Not Defined ---
Created on	9/29/2011 10:55:31 AM
Description	

National code

National code	EN 1992-1-1
National annex	EN

2. Brief summary of results of design member

Design Name	Member Description	Members	Tendons	Valid	Value [%]	Result Status
Beam 1	Description 1	5, 4, 1, 2, 3	Tendon 2, Tendon 1, Tendon 6	✓	1000,00	✗

3. Construction Stages

Name	Time [d]	Load Cases	Combinations	Description
Stage 0	0,0			
Stage 1	5,0	LC1, LC2, LC4, LC3	LG1, LG2, LG3, LG4	
Stage 2	36500,0		LG1, LG2, LG3, LG4	

4. Design Members

4.1. Beam 1

Description	Members	Tendons	Valid	Value [%]	Result Status
Description 1	5, 4, 1, 2, 3	Tendon 2, Tendon 1, Tendon 6	✓	1000,00	✗

14.3.3 Detailed report

To generate detailed report for all design members click **Detailed** in ribbon group **Report** in navigator **Project data > Design members** or click navigator command **Report > Detailed** to generate detailed report for current design member only.

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

5. Tendons

5.1. Tendon: Tendon 2

Material	Number of strands	Load case	Area [mm2]	Ø [mm]	Max. initial stress [MPa]	Limit stress [MPa]	Stress check
Y1770S7-12.9	7	LC4	700	30	1321,00	1350,00	✔

5.1.1. Geometry

Plane XY

Index	Begin [m]	End [m]	Length [m]	Type	Valid
1	0,00	25,66	25,66	Stand-alone, parabolic and straight	✔

Index	x [m]	y [mm]	Type	L s [m]	L s,rel [-]	r [m]
1	0,00	0	Closing point (C)	0,00		
2	12,83	0	Intermediate point Straight - Parabola (S-P)		0,00	
3	25,66	0	Closing point (C)	0,00		

Plane XZ

Index	Begin [m]	End [m]	Length [m]	Type	Valid
1	0,00	25,66	25,66	Stand-alone, parabolic and straight	✔

Index	x [m]	z [mm]	Type	L s [m]	L s,rel [-]	r [m]
1	0,00	-50	Closing point (C)	0,00		
2	12,83	615	Intermediate point Straight - Parabola (S-P)		0,00	
3	25,66	-50	Closing point (C)	0,00		

5.1.2. Equivalent prestress load

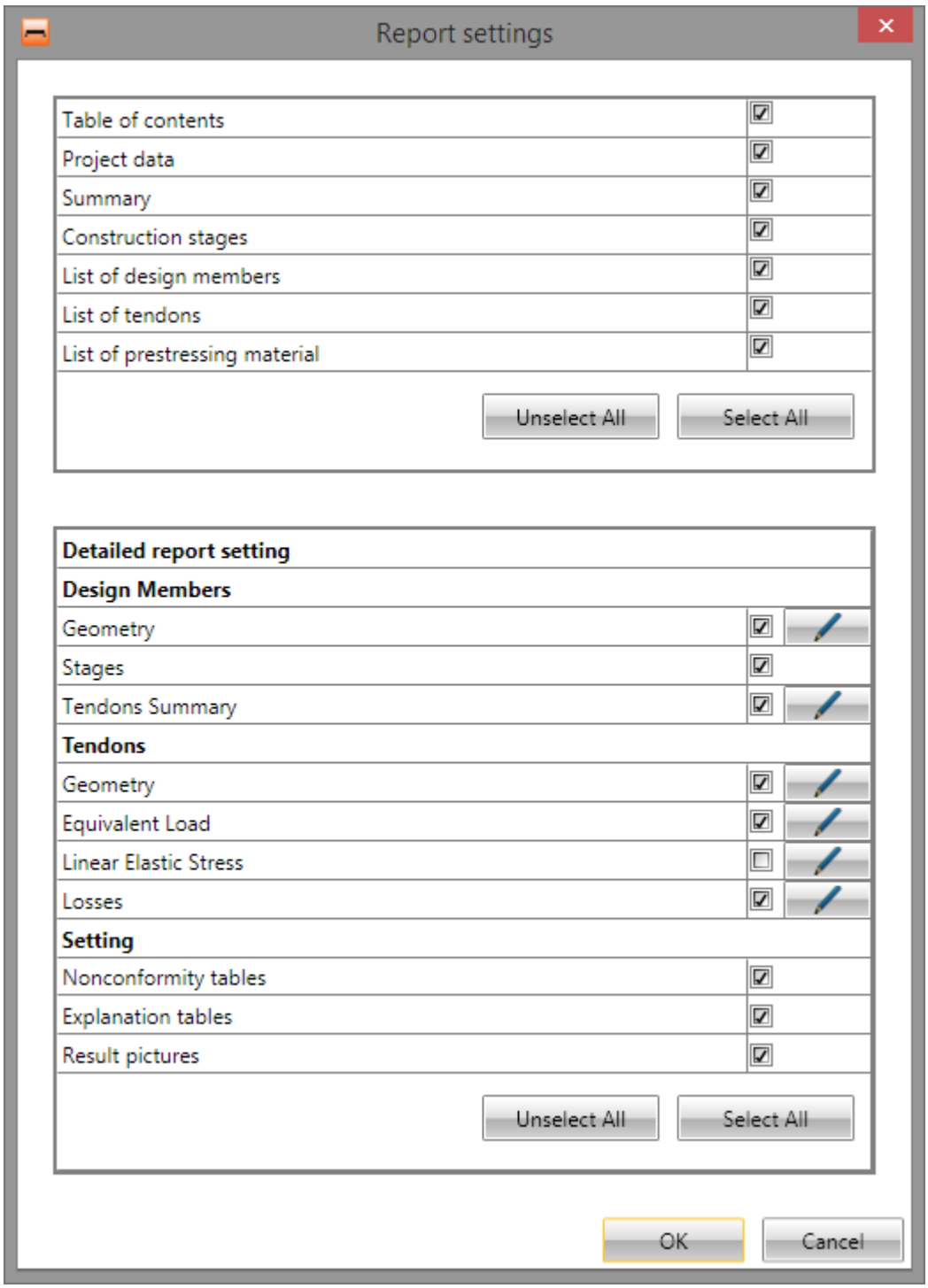
Tendon name	Index	x [m]	F x [kN]	F y [kN]	F z [kN]	M x [kNm]	M y [kNm]	M z [kNm]
Tendon 2	1	0,00	854,51	0,00	-87,44	0,00	42,73	0,00
	2	25,66	-855,52	0,00	-87,55	0,00	-42,78	0,00

Tendon name	Index	x begin [m]	x end [m]	p x [kN/m]	p y [kN/m]	p z [kN/m]	m x [kNm/m]	m y [kNm/m]	m z [kNm/m]
Tendon 2	1	0,00	0,33	2,97	0,00	4,90	0,00	0,05	0,00
	2	0,33	0,67	2,97	0,00	4,90	0,00	0,05	0,00
	3	0,67	1,00	3,11	0,00	6,64	0,00	-0,15	0,00
	4	1,00	1,57	3,11	0,00	6,64	0,00	-0,15	0,00
	5	1,57	3,22	3,06	0,00	6,71	0,00	-0,73	0,00
	6	3,22	5,36	2,97	0,00	6,80	0,00	-1,20	0,00
	7	5,36	7,50	2,88	0,00	6,91	0,00	-1,53	0,00
	8	7,50	9,63	2,77	0,00	7,01	0,00	-1,73	0,00
	9	9,63	11,76	2,67	0,00	7,11	0,00	-1,82	0,00
	10	11,76	12,93	2,59	0,00	7,18	0,00	-1,80	0,00
	11	12,93	14,00	-2,11	0,00	7,18	0,00	1,47	0,00
	12	14,00	16,03	-2,68	0,00	7,12	0,00	1,82	0,00
	13	16,03	18,16	-2,78	0,00	7,02	0,00	1,74	0,00
	14	18,16	20,30	-2,88	0,00	6,92	0,00	1,53	0,00
	15	20,30	22,44	-2,98	0,00	6,81	0,00	1,20	0,00

14.4 Report settings

To set content of report click **Settings** in ribbon group **Report** or navigator command **Report > Settings**. Report settings consist of global settings and detailed settings.

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



14.4.1 Group Design members

Options in group **Design members** enable to add report chapters with design member information and pictures. It is possible to turn on/off print of design member geometry table, construction stages table, tendons summary table and positions for check table.

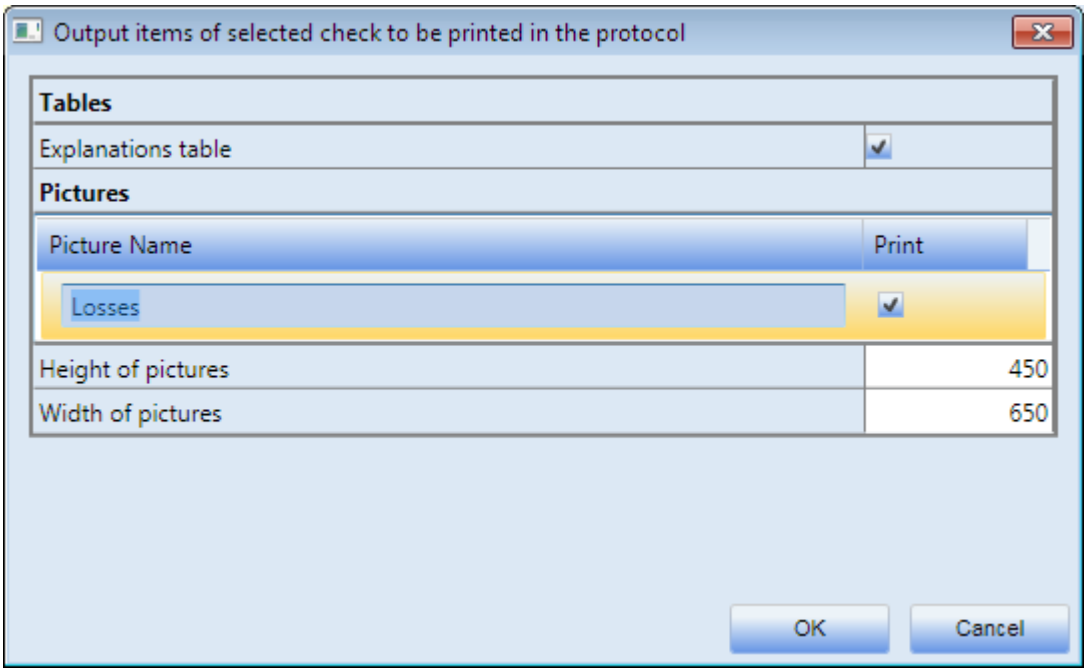
14.4.2 Group Tendons

Options in group **Tendons** enable to add report chapters with tendons information and pictures. It is possible to turn on/off print of tendon geometry table, equivalent loads table and losses table.

14.4.3 Group Settings

- **Nonconformities tables** – if the option is off, no nonconformity table is printed in the report. Otherwise the nonconformity tables are printed if they haven't been switched off in detailed setting.
- **Explanation tables –** if the option is off, no explanation table is printed in the report. Otherwise the explanation tables are printed if they haven't been switched off in detailed setting.
- **Result pictures –** if the option is off, no picture with graphical presentation of results is printed in the report. Otherwise pictures are printed if they haven't been switched off in detailed setting.

14.4.4 Detailed report settings for particular chapters



- **Tables** – all available tables for current document chapter are listed in group Tables. Particular tables can be set on/off to be printed
  - **Nonconformity table** – turns on/off the print of nonconformity table into the report for edited check.
  - **Explanation table** – turns on/off the print of table with explanation of symbols into report for edited check.
- **Pictures –**list of available graphical presentations of results for edited chapter. The picture name and option to print or not is available.

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

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

15 Coordinate systems and convention of internal forces

All coordinate systems used are right-handed.

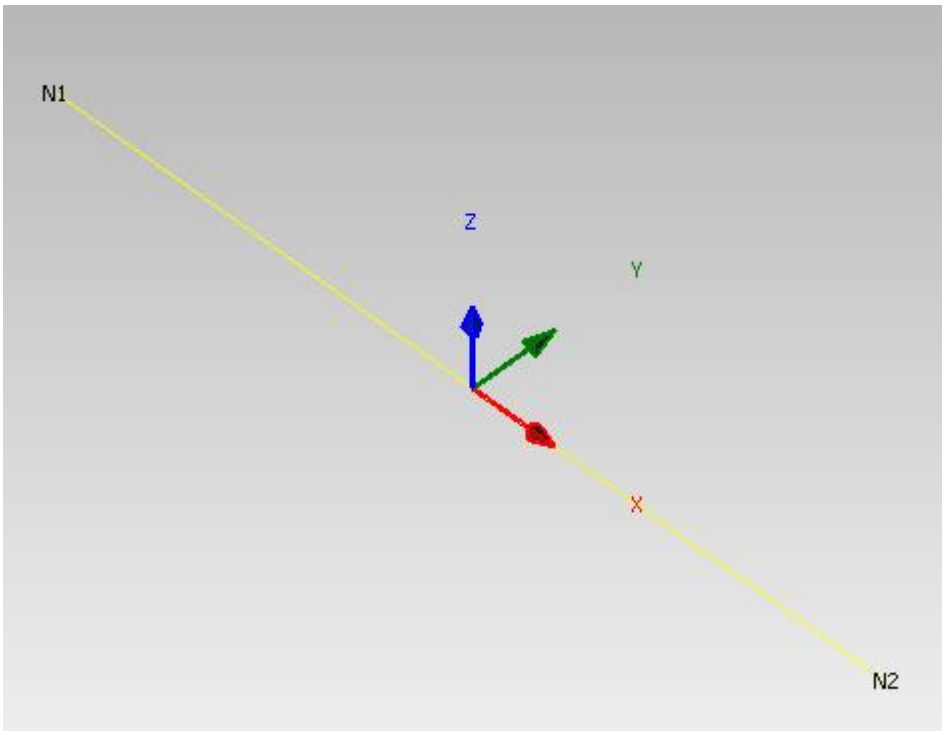
15.1.1 Global coordinate system

X-axis of global coordinate system is horizontal and leads from left to right.

Y-axis of global coordinate system is horizontal and leads backward.

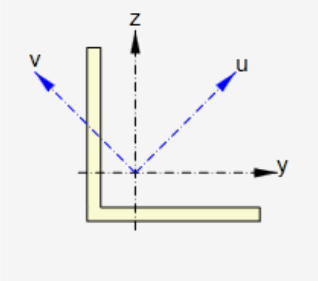
Z-axis of global coordinate system is vertical and leads upwards.

15.1.2 Local coordinate system of the part of member



Each part of member is defined by the beginning and end node. Each part of member has local coordinate system, which origin is in begin node of the part of member. Local x-axis of the part of member is identical with its axis and is oriented from being to end of the part of member. Local y-axis of the part of member is horizontal in general and local z-axis leads upwards.

15.1.3 Coordinate system of cross-sections



Cross-section axes are denoted y (horizontal) and z (vertical).

Principal axes of cross-section are marked with u and v.

If reference axes are identical with main central axes of cross-section, only reference axes are drawn.

.

15.1.4 Convention of internal forces on members 1D (Axis)

Internal forces on 1D members cause following actions:

- positive bending moment My causes tension in cross-section fibres with negative z-coordinate.
- positive bending moment Mz causes tension in fibres' with negative y-coordinate
- positive torsional moment Mx acts about x-axis of 1D member
- positive axial force N acts in direction of x-axis of member and causes tension in cross-section fibres'.
- positive shear force Vz acts in direction of z-axis of cross-section.
- positive shear force Vy acts in direction of y-axis of cross-section.