

IDEA StatiCa, s.r.o.

IDEA Beam

Prestressed concrete, reinforced concrete and steel beams design

Version 8

CONTENTS

1 Introduction	5
1.1 Program requirements	5
1.2 Installation guidelines	5
2 Basic terms	6
3 User interface	7
3.1 View controls in the main window	7
3.2 DXF export settings	8
4 Units setting	9
5 Working with project	10
5.1 Creating new project	10
5.1.1 Selecting a beam type	11
5.1.2 Single span beam	13
5.1.3 Beam with cantilevers	14
5.1.4 Continuous beam	15
5.1.5 Continuous beam composed of prefabricated beams made continuous by composite slab	16
5.1.6 Code, cross-section, loads	17
5.1.7 Construction stages	19
5.1.8 Global time axis of beam made continuous by composite slab	21
5.1.9 Production stages of prefabricated beams composing beam made continuous by composite slab	23
5.2 Structure drawing settings	24
5.2.1 Ribbon group View settings	24
5.2.2 Ribbon group CSS drawing	24
5.2.3 Ribbon group View	24
5.2.4 Ribbon group Axo view	24
6 Input of the structure	26
6.1 Project data	27
6.2 Cross-sections	29
6.3 Materials	31
6.4 Geometry	32
6.4.1 Members and haunches	32
6.4.2 Supports	36
6.5 Loads	38
6.5.1 Groups of load cases	38
6.5.2 Load cases of non-staged beams	42
6.5.3 Load cases of staged beams	44
6.5.4 Point forces	45
6.5.5 Point moment loads	47
6.5.6 Uniform loads	48
6.5.7 Line load	50
6.5.8 Settlements of supports	52
6.5.9 User defined internal forces	53
6.5.10 Combinations of load cases	58
6.5.11 Manager of load cases combinations	62
7 Construction stages	65
7.1 General settings of construction stages	65
7.2 History of members	66
7.2.1 Ribbon group Construction stages	67
7.3 Construction stages	68
7.3.1 Ribbon group Construction stages	69
7.3.2 Load cases and combinations in construction stages	69
7.3.3 Manual assignment of combinations to construction stages	70
7.3.4 Temporary supports of prefabricated beams	72
7.3.5 Cast-in-situ post-tensioned beam	74
7.3.6 Cast-in-situ composite reinforced beam	77
7.3.7 Cast-in-situ composite post-tensioned beam	78

7.3.8 Prefabricated reinforced beam	79
7.3.9 Prefabricated pre-tensioned beam	83
7.3.10 Prefabricated post-tensioned beam	85
7.3.11 Prefabricated composite reinforced beam	87
7.3.12 Prefabricated composite pre-tensioned beam	91
7.3.13 Prefabricated composite post-tensioned beam	93
8 Design of tendons	95
9 Results	96
9.1 Results evaluation setup	96
9.1.1 Results evaluation setup of staged beams	97
9.2 Reactions in supports	98
9.2.1 Ribbon group Results	98
9.2.2 Ribbon group System of reactions	98
9.2.3 Ribbon group Extreme	98
9.2.4 Ribbon group Reactions/Bearing stress intensity	99
9.3 Deformations	100
9.3.1 Ribbon group Results	100
9.3.2 Ribbon group Extreme	100
9.3.3 Ribbon group Deformations	100
9.4 Internal forces	102
9.4.1 Ribbon group Results	102
9.4.2 Ribbon group Extreme	102
9.4.3 Ribbon group Transformations	102
9.4.4 Ribbon group Internal forces	103
9.4.5 Ribbon group Fatigue evaluation	103
9.4.6 Ribbon group Resultant of stage	104
10 Design and check of structural items	105
11 Check of concrete members	106
11.1 Settings for section checks and calculation of deflections	107
11.1.1 Code and calculation settings	107
11.1.2 Setting of result class for calculation of deflections	108
11.1.3 Setting of result classes for section checks	108
11.1.4 Editing result class	109
11.2 Design member data	111
11.3 Reinforcement zones	113
11.3.1 Check positions for check of pre-stressed members	115
11.3.2 Zone templates	116
11.3.3 Ribbon group View settings and scale	116
11.3.4 Ribbon group Detailed view	117
11.3.5 Ribbon group Internal forces	117
11.3.6 Ribbon group Bill of material	118
11.4 Editor of reinforcement	119
11.4.1 Editing cover	120
11.4.2 Input of reinforcement by template	121
11.4.3 Input of one-way slabs reinforcement by a template	123
11.4.4 Shear reinforcement	124
11.4.5 Longitudinal reinforcement	133
11.4.6 User settings of reinforced cross-section	141
11.4.7 Deleting reinforcement	143
11.4.8 Import and export of reinforced cross-section	143
11.4.9 View settings of reinforced cross-section	143
11.4.10 User defined reinforcement templates	145
11.5 Input of data for deflections check	149
11.6 Reduction and redistribution of internal forces	151
11.6.1 Supports definition for calculation of redistributions and reductions	151
11.6.2 Ribbon group Internal forces	152
11.7 Detailed check	153
11.8 Results evaluation	154

11.8.1 Ribbon group Concrete design	154
11.8.2 Ribbon group View settings and scale	154
11.8.3 Ribbon group Extreme	154
11.8.4 Ribbon group Calculation	155
11.8.5 Ribbon group Stage	155
11.8.6 Ribbon group Results drawing	155
11.8.7 Drawing of section check results courses	155
11.8.8 Drawing of interaction diagrams	156
11.8.9 Drawing of deflections check results	158
11.8.10 Check report	160
12 Bridge load rating of concrete members	164
12.1 Input data of bridge load rating	164
12.1.1 Check settings	164
12.1.2 Check positions	165
12.1.3 Load cases and combinations for bridge load rating	167
12.1.4 Ribbon group View settings and scale	174
12.2 Results of bridge load rating calculation	175
12.2.1 Ribbon group View settings and scale	175
12.2.2 Ribbon group Extreme	175
12.2.3 Ribbon group LR type	175
12.2.4 Ribbon group Check	175
13 Steel members design	176
13.1 Default check settings	176
13.2 Check settings for the current design group	180
13.3 Design data	182
13.3.1 Point LTB restraint	183
13.3.2 Distributed LTB restraint	183
13.3.3 Not checked field	184
13.3.4 Ribbon group LTB restraints	184
13.3.5 Ribbon group Check data	184
13.3.6 Ribbon group View settings	185
13.4 Buckling lengths	186
13.4.1 Ribbon group 3D View	188
13.4.2 Ribbon group Dimensions drawing	188
13.5 Check results evaluation	189
13.5.1 Ribbon group Steel design	189
13.5.2 Ribbon group Extremes	189
13.5.3 Ribbon group Type of check	190
13.5.4 Ribbon group Type of output	190
14 Report	191
14.1 Standard report	191
14.1.1 Input data	191
14.1.2 Calculation results	192
14.1.3 Concrete 1D design results	192
14.1.4 Bridge load rating results	193
14.1.5 Steel design results	193
14.2 Detailed report	194
14.2.1 Input data	194
14.2.2 Calculation results	195
14.2.3 Concrete 1D design results	196
14.2.4 Bridge load rating results	199
14.2.5 Steel design results	200
14.3 Ribbon group Report view	201

1 Introduction

1.1 Program 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 guidelines

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

2 Basic terms

IDEA Beam is a program for the analysis and design of beams.

IDEA Beam is one from the group of programs developed especially for 2D FEA structural analysis. All these programs work with the same data model. This allows their direct connection with all IDEA design modules.

It is possible to input a straight single-span or multi-span beam with or without cantilevers.

The beam can be straight (2D) or a spatial polygon (3D).

Loads are sorted into load cases. There can be point and line impulses on members, uniform loads and support settlements.

Analysis of internal forces and deformations is done by finite element method. Program calculates internal forces and deformations. Load cases can be combined.

Cross-sections of concrete beams can be one-sectional or composite.

Concrete beams (composite, prestressed) can be analysed with taking into account construction stages and time dependent analysis.

Results of IDEA Beam program can be directly used in design modules:

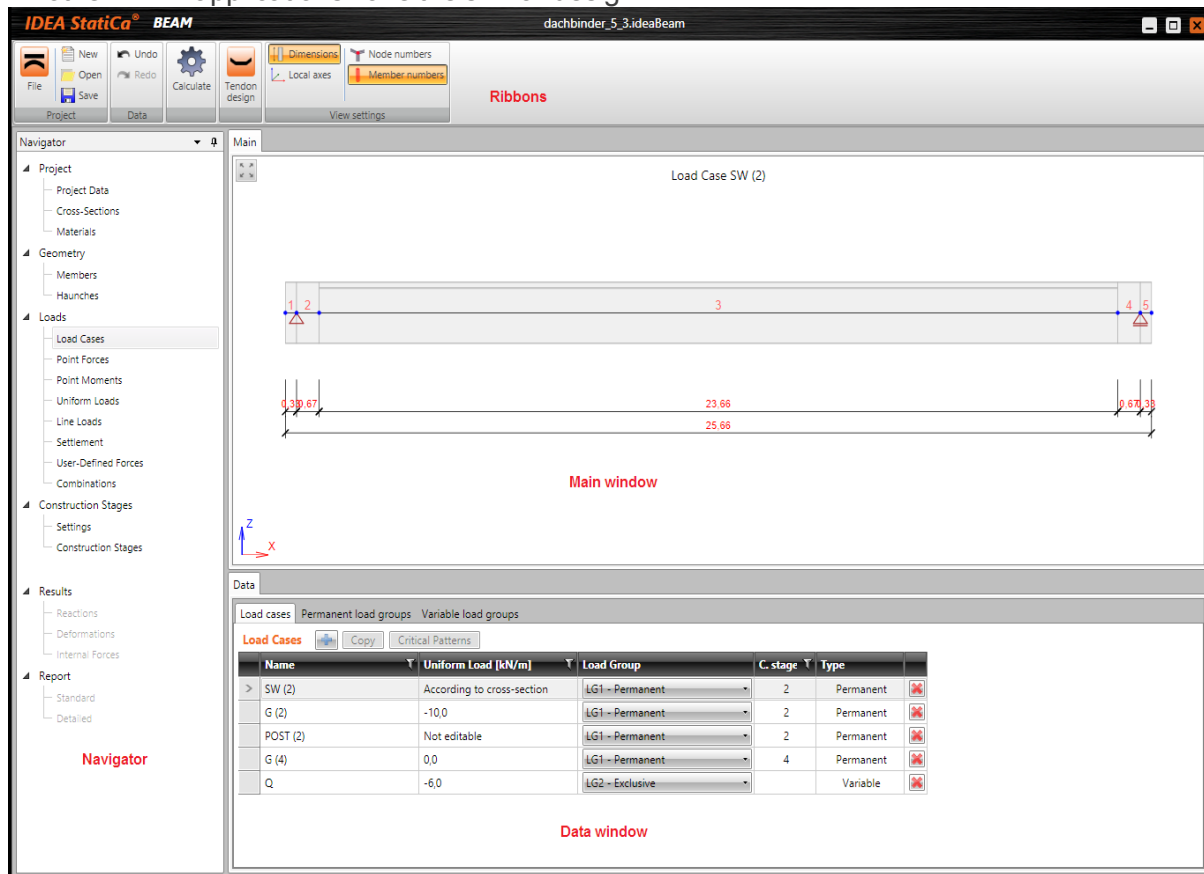
- IDEA RCS where the required reinforcement can be designed
- IDEA Steel where the steel members can be designed
- IDEA Tendon where prestressed members can be designed.

3 User interface

The items of user interface of the application are composed into following groups:


- **Navigator** – it contains main commands for the work with a project.
- **Ribbons** – there are sets of controls. Ribbons content changes according to the current Navigator command.
- **Main window** – it is used mainly for appropriate drawings.
- **Data window** – properties of objects and results of analysis are displayed in this window according to current Navigator command

All other IDEA applications have the similar design.



3.1 View controls in the main 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.

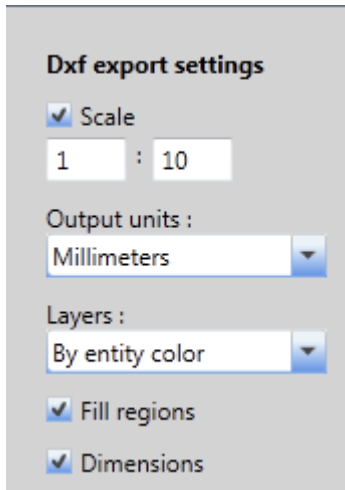
To set the required view using keyboard and mouse following combinations can be used:

- 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 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.

3.2 DXF export settings

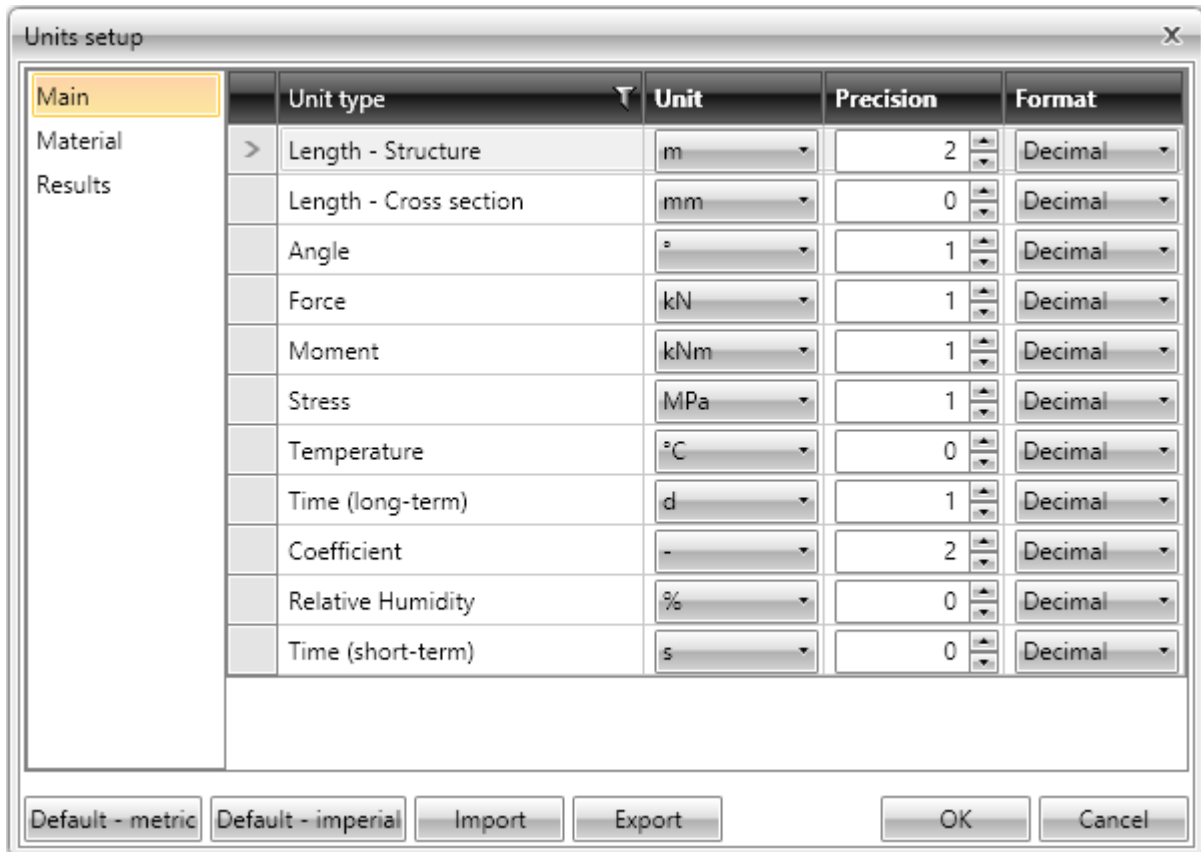


Following export parameters can be set in the Save as dialog when exporting the view to the 2D file:

- **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.

4 Units setting

The units used by the application can be set by menu command **File > Units**.



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:

- **Decimal** – display numbers in standard decimal format (“ddd.ddd...”).
- **Scientific** – display numbers in exponential format (“d.ddd...E+ddd”).
- **Automatic** – 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.
- **Imperial** – display numbers in fractional format (only for imperial unit types).

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

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

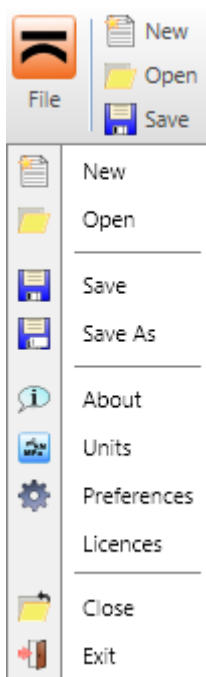
Import - reads the units configuration from a file.

Export - saves the current units settings to a file.

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

5 Working with project

Use commands in ribbon group **Project** to work with project file:



- **New** – create a new project.
- **Open** – open an existing project (files with extension *.ideaBeam or *.wsBeam).
- **Save** – save the current project into the data file.
- **Save as** – save the current project into the data file using a new file name.
- **About** – open the About application dialog.
- **Units** – open dialog for units settings.
- **Preferences** – open a dialog to set the application language or the logo for printed reports.
- **Licences** – launch **Licence manager** application
- **Close** – close the current project.
- **Exit** – close the application.

5.1 Creating new project

To create a new project click **New** in the ribbon group **Project**. New project is created using a wizard in several steps:

- Selecting a beam type;
- Selecting beam supports;
- Setting cross-sections and supports alignment;
- Setting initial cross-sections, material, loads.

5.1.1 Selecting a beam type

The screenshot shows a dialog box titled "Type of beam" from the IDEA StatiCa software. The dialog has a title bar with the IDEA StatiCa logo and the slogan "Calculate yesterday's estimates". The main area contains a list of radio button options for beam types, two dropdown menus for "Beam alignment" and "Supports position", and three buttons at the bottom: "< Previous", "Next >", and "Cancel".

Type of beam

- Cast-in-situ concrete beam
 - Reinforced concrete beam
 - Post-tensioned beam
 - Pre-fabricated concrete beam
 - Reinforced precast beam with the option of subsequent post-tensioning
 - Prestressed precast beam with the option of subsequent post-tensioning
- Steel beam

- Single span beam
- Beam with cantilevers
- Continuous beam

Beam alignment:

Supports position:

< Previous Next > Cancel

The type of a beam is selected in the first step of wizard:

- **Cast-in-situ concrete beam** – the concrete cast-in-situ beam will be designed. Material of all cross-sections must be concrete. Beam can be reinforced by concrete or reinforcement or/and prestressed by prestressing reinforcement. Beam will be designed in the program IDEA RCS. One of following types can be selected for cast-in-situ concrete beam:
 - **Reinforced concrete beam** – reinforced concrete beam is not prestressed. The cross-section of beam can be with or without composite slab. The beam can be reinforced by concrete reinforcement only. Construction stages are taken into account for beam containing composite cross-section.
 - **Post-tensioned beam** - the cross-section of beam can be with or without composite slab. The beam can be prestressed by post-tensioned tendons in construction stages and reinforced by concrete reinforcement.
- **Prefabricated concrete beam** – a beam is precast in the factory and finalized by casting in-situ. Material of all cross-sections must be concrete. Beam can be reinforced by concrete or reinforcement or/and prestressed by prestressing reinforcement. The shape of prefabricated (non-composite) beam is limited to simple beam (one span with cantilevers). The position of supports may vary during the production and assemblage. One of following types can be selected for prefabricated beam:
 - **Reinforced precast beam with option of subsequent post-tensioning** - precast beam is produced as non-prestressed. It is possible to apply post-tensioning in subsequent construction stages. Simply supported precast beam (with cantilevers) can be both with and without composite slab. Continuous beam made of precast beams is

always with composite slab.

- **Prestressed precast beam with option of subsequent post-tensioning** - precast beam is produced as pre- or post-tensioned. It is possible to apply additional post-tensioning in subsequent construction stages. Simply supported precast beam (with cantilevers) can be both with and without composite slab. Continuous beam made of precast beams is always with composite slab.
- **Steel beam** – the steel beam will be designed. Default material is steel, rolled and thin-walled cross-sections can be used. Beam will be designed in the module IDEA Steel.

One of following types of geometry can be selected for cast-in-situ beams:

- **Single span beam** – the beam of one span will be designed. Lengths of spans and type of supports are set in next step of wizard.
- **Beam with cantilevers** – the beam of one span with cantilevers will be designed. Lengths of spans and type of supports are set in next step of wizard.
- **Continuous beam** – the beam of more spans will be designed. Lengths of individual spans can be defined in next step of wizard.

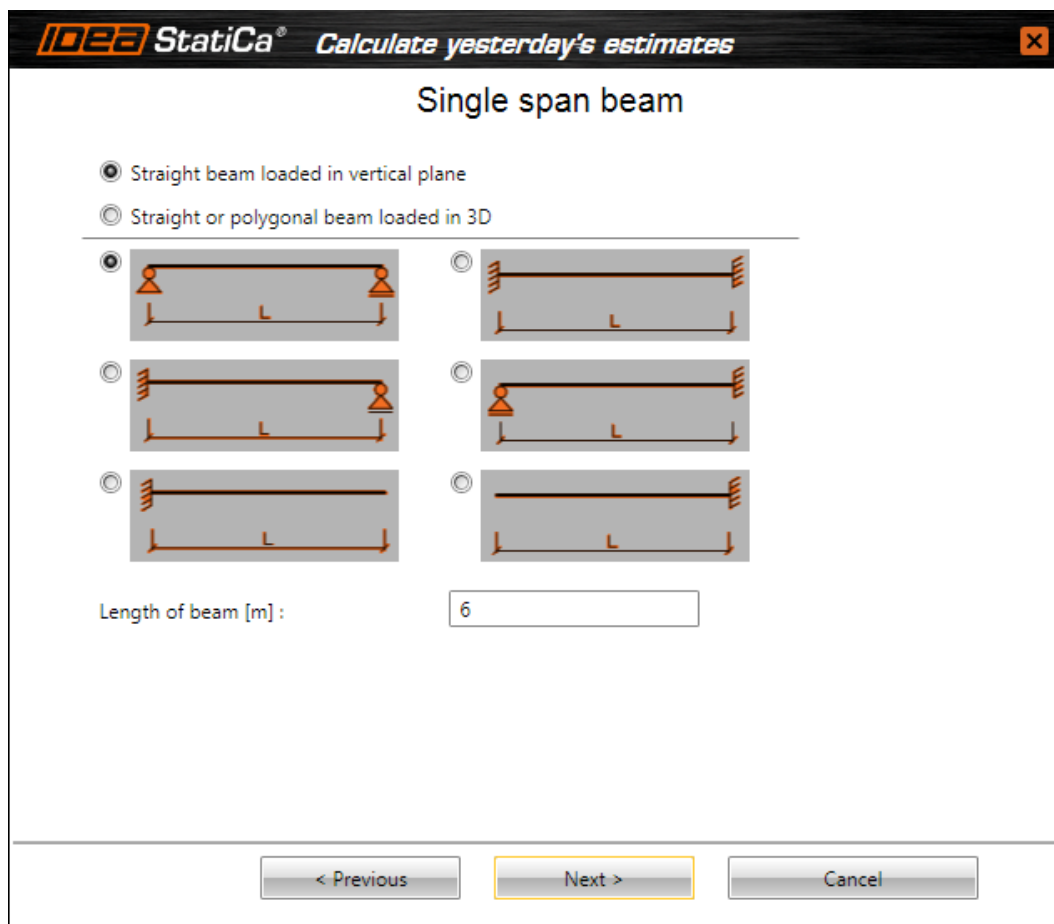
One of following types of geometry can be selected for prefabricated composite beams:

- **Single span beam** – the beam of one span will be designed. Cantilevers can be defined on the beam also. Lengths of spans and type of supports are set in next step of wizard.
- **Continuous beam** – continuous beam composed of prefabricated concrete beams made continuous by cast-in-situ diaphragms above internal supports and composite concrete slab will be designed.

Following alignment modes are available for cross-sections and supports:

- **Beam Alignment** – determines the alignment of cross-sections in particular spans of beam and the alignment of haunches.
 - **Top surface** – the top edges of cross-sections of particular beam spans and the top edges of cross-section of haunches are aligned to the top edge of the cross-section of the first span of the beam.
 - **Center line** – the centroids of cross-sections of particular beam spans and the centroids of cross-sections of haunches are placed on the reference line. The reference curve of the beam meets the centroid of the cross-section of the first beam span.
 - **Bottom surface** - the bottom edges of cross-sections of particular beam spans and the bottom edges of cross-section of haunches are aligned to the bottom edge of the cross-section of the first span of the beam.
 - **Centre line + eccentricities** – adds a possibility to define eccentricities at the ends of member - distance between the centre of gravity of cross-section and the reference curve.
- **Supports position** – determines the position of support to the cross-section in the supported node
 - **Top surface** – supports are aligned to the top edge of the cross-section.
 - **Centre line** – supports are aligned to the reference curve.
 - **Bottom surface** – supports are aligned to the bottom edge of the cross-section.
 - **Centre line + eccentricities** – adds a possibility to define eccentricities of supports - distance between the support and the reference curve.

5.1.2 Single span beam



Single span beam supporting can be selected from several available types. The length of span can be defined in the field **Length of beam**.

One of following geometry types can be selected for single span beam:






- **Straight beam loaded in vertical plane** – the straight horizontal beam can be defined (only X coordinates of nodes can be edited). Loads can act in the YZ plane of global coordinate system.
- **Straight or polygonal beam loaded in 3D** – straight or polygonal beam can be defined (X and Y coordinates of nodes can be edited). Z coordinates of nodes can be edited, if the option Pitch (slope) of beam is selected. Loads can act in all directions of global coordinate system or in directions of local axes of members.

5.1.3 Beam with cantilevers

IDEA StatiCa® Calculate yesterday's estimates

Beam with cantilevers

Straight beam loaded in vertical plane
 Straight or polygonal beam loaded in 3D

Spans of beam [m]:

L:

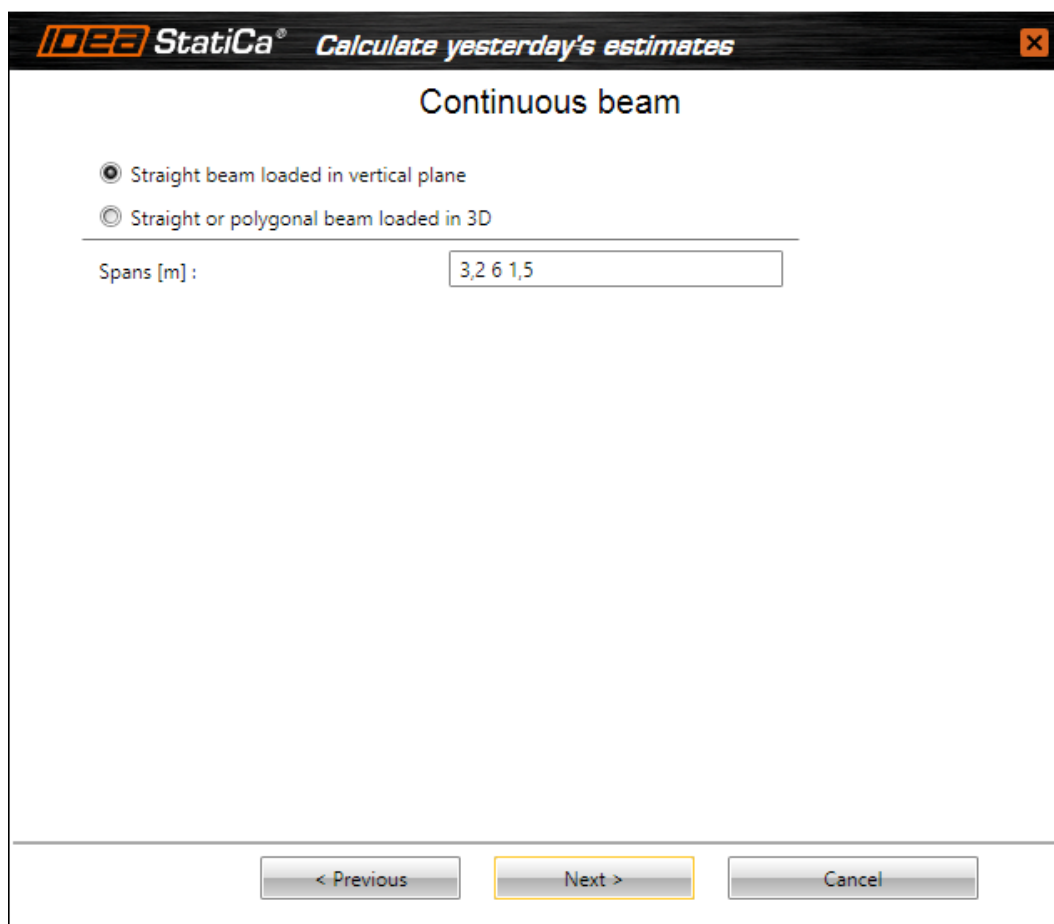
L1: L2:

Single span beam with cantilevers can be selected from several available types of different layouts and supporting. The length of span can be defined in fields **L**, **L1** and **L2**.

One of following geometry types can be selected for beam with cantilevers:

- **Straight beam loaded in vertical plane** – the straight horizontal beam can be defined (only X coordinates of nodes can be edited). Loads can act in the YZ plane of global coordinate system.
- **Straight or polygonal beam loaded in 3D** – straight or polygonal beam can be defined (X and Y coordinates of nodes can be edited). Z coordinates of nodes can be edited, if the option Pitch (slope) of beam is selected. Loads can act in all directions of global coordinate system or in directions of local axes of members.

5.1.4 Continuous beam



IDEA StatiCa® Calculate yesterday's estimates

Continuous beam

Straight beam loaded in vertical plane

Straight or polygonal beam loaded in 3D

Spans [m] :

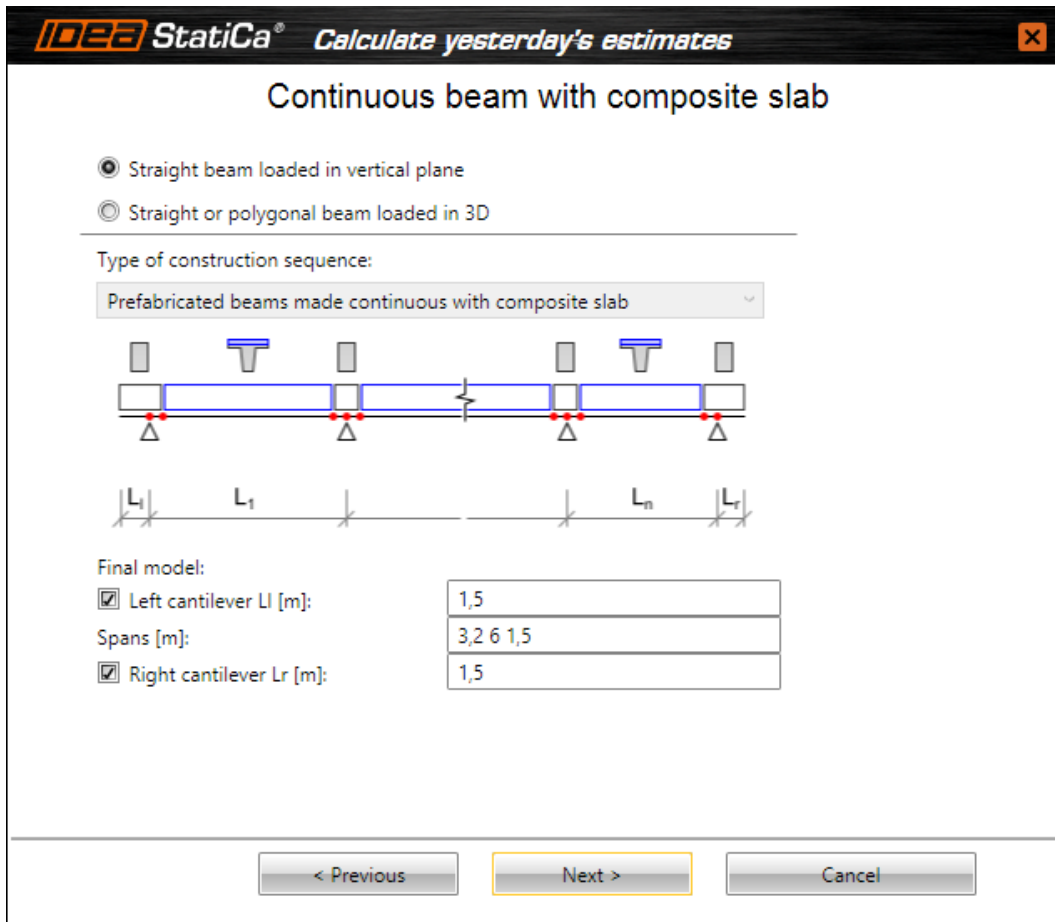
< Previous Next > Cancel

The sequence of lengths of beam spans (space delimited numbers) can be entered in **Spans** edit box.

One of following geometry types can be selected for continuous beam:

- **Straight beam loaded in vertical plane** – the straight horizontal beam can be defined (only X coordinates of nodes can be edited). Loads can act in the YZ plane of global coordinate system.
- **Straight or polygonal beam loaded in 3D** – straight or polygonal beam can be defined (X and Y coordinates of nodes can be edited). Z coordinates of nodes can be edited, if the option Pitch (slope) of beam is selected. Loads can act in all directions of global coordinate system or in directions of local axes of members.

5.1.5 Continuous beam composed of prefabricated beams made continuous by composite slab



For continuous beam with composite slab lengths of spans and lengths of cantilevers can be defined in second step of wizard.

- **Left cantilever L_I** – if the option is selected, the length of left cantilever can be defined.
- **Span** – input sequence of lengths of individual beam spans (space delimited numbers).
- **Right cantilever L_r** – if the option is selected, the length of left cantilever can be defined.

One of following geometry types can be selected for continuous beam with composite slab:

- **Straight beam loaded in vertical plane** – the straight horizontal beam can be defined (only X coordinates of nodes can be edited). Loads can act in the YZ plane of global coordinate system.
- **Straight or polygonal beam loaded in 3D** – straight or polygonal beam can be defined (X and Y coordinates of nodes can be edited). Z coordinates of nodes can be edited, if the option Pitch (slope) of beam is selected. Loads can act in all directions of global coordinate system or in directions of local axes of members.

5.1.6 Code, cross-section, loads

Code, cross-section, loads

Design code: EN 1992-2

National annex: EN

Design working life: 50 years

Groups for bridges

Cross-section CCSS I-shape 1000, 800

Diaphragm (rectangle) [mm] b = 300 / h = 800

Self weight according cross-section

Permanent load [kN/m]: -10

Variable load [kN/m]: -15

Critical load patterns

Ratio for unloaded spans: 0

Summary combination for permanent and superimposed dead load

Summary combination for prestressing and rheological load cases

< Previous Next > Cancel

National code, default cross-section and loads (according to the beam type) are defined in the next step of the wizard.

Particular options of dialog:

- **Design code** – select the national code of the project.
- **National annex** – select the national annex for concrete beams.
- **1992-2** – switch on/off support of EN-1992-2 for concrete beams.
- **Design working life** – select the value of design working life for concrete beams.
- **Groups for bridges** – switch on/off generation of default groups of variable loads (including corresponding values of ψ coefficients) for traffic load on bridges:
 - **Road bridge** – groups of variable loads for road bridges are generated.
 - **Railway bridge** – groups of variable loads for railway bridges are generated.
 - **Footbridge** – groups of variable loads for footbridges are generated.
- **Cross-Section** – click to input the initial cross-section of beam spans is defined.
- **Diaphragm (rectangle)** – input width of cross-section of diaphragm of continuous beam composed of composite beam made continuous by composite slab. The height of cross-section of diaphragm is taken according to height of cross-sections in spans.
- **Material** – click to select material of the cross-section. It is available for a steel rolled cross-sections only.
- **Self-weight according cross-section** – if the option is selected, the load case for the self-weight is added. The self-weight is generated automatically on particular beam members

respecting the current cross-section of the member. The value cannot be changed.

If the option is not selected, the load case named Self weight is created. For such load case, the value of distributed load acting on all beam members is calculated respecting the defined default cross-section and the value is editable. Note, that the calculated value does not change after any change of beam!

- **Permanent load** – input of the size of permanent uniform load in all spans of a beam. These loads will be assigned to one permanent load case.
- **Variable load** – input of the size of variable uniform load in all spans of a beam. These loads will be assigned to one variable load case.
- **Critical load patterns** – if it is switched on, more load cases will be generated from variable load case. Different spans will be loaded or unloaded to get the most critical moments in spans and above supports and maximal reactions in supports. This option is available for multi-spans beams.
- **Ratio for unloaded spans** – the ratio of size of loads in unloaded spans. When 0 is set, there is no load in unloaded spans. When 0,25 is set, 25% of variable load is put into unloaded spans. This option is available for multi-spans beams.
- **Summary combination for permanent and superimposed dead loads** – if the option is selected, a combination containing all permanent and superimposed dead loads is generated per each construction stage. This option is available for staged concrete beams.
- **Summary combination for prestressing and rheological load cases** – if the option is selected, a combination containing all prestressing load cases and rheological load cases is generated per each construction stage. This option is available for staged concrete beams.

5.1.7 Construction stages

For cast-in-situ staged beams and prefabricated single-span beams, basic parameters of construction stages and positions of temporary supports can be defined in the last step of wizard. The dialog content (number of stages, available parameters) differs according to the type of analysed beam. Values defined in this wizard dialog can be modified later in construction stages properties.

Parameters of two additional construction stages can be defined for post-tensioned cast-in situ beams.

Construction stages	Time [d]	POST
<input checked="" type="checkbox"/> Casting	0	
<input checked="" type="checkbox"/> Post-tensioning	5	
<input type="checkbox"/> Construction stage 1	10	<input type="checkbox"/>
<input type="checkbox"/> Construction stage 2	35	<input type="checkbox"/>

Service stages	Time [d]
<input checked="" type="checkbox"/> Superimposed dead load	60
<input checked="" type="checkbox"/> End of design working life	18250

IDEA StatiCa® Calculate yesterday's estimates

Construction stages

Construction stages	Time [d]	PRE	POST	L1 [m]	L [m]	L2 [m]	n
<input checked="" type="checkbox"/> Casting	0						
<input checked="" type="checkbox"/> Prestressing	5	<input type="radio"/>	<input checked="" type="radio"/>	0	6	0	
<input checked="" type="checkbox"/> Storage yard	5,1		<input type="checkbox"/>	1,2	3,6	1,2	0
<input checked="" type="checkbox"/> Transport	25			1,2	3,6	1,2	0
<input checked="" type="checkbox"/> Temporary supports	26		<input type="checkbox"/>	1,2	3,6	1,2	0
<input checked="" type="checkbox"/> Casting of composite slab	28			1,2	3,6	1,2	1
<input checked="" type="checkbox"/> Final supports	35		<input type="checkbox"/>	1,2	3,6	1,2	

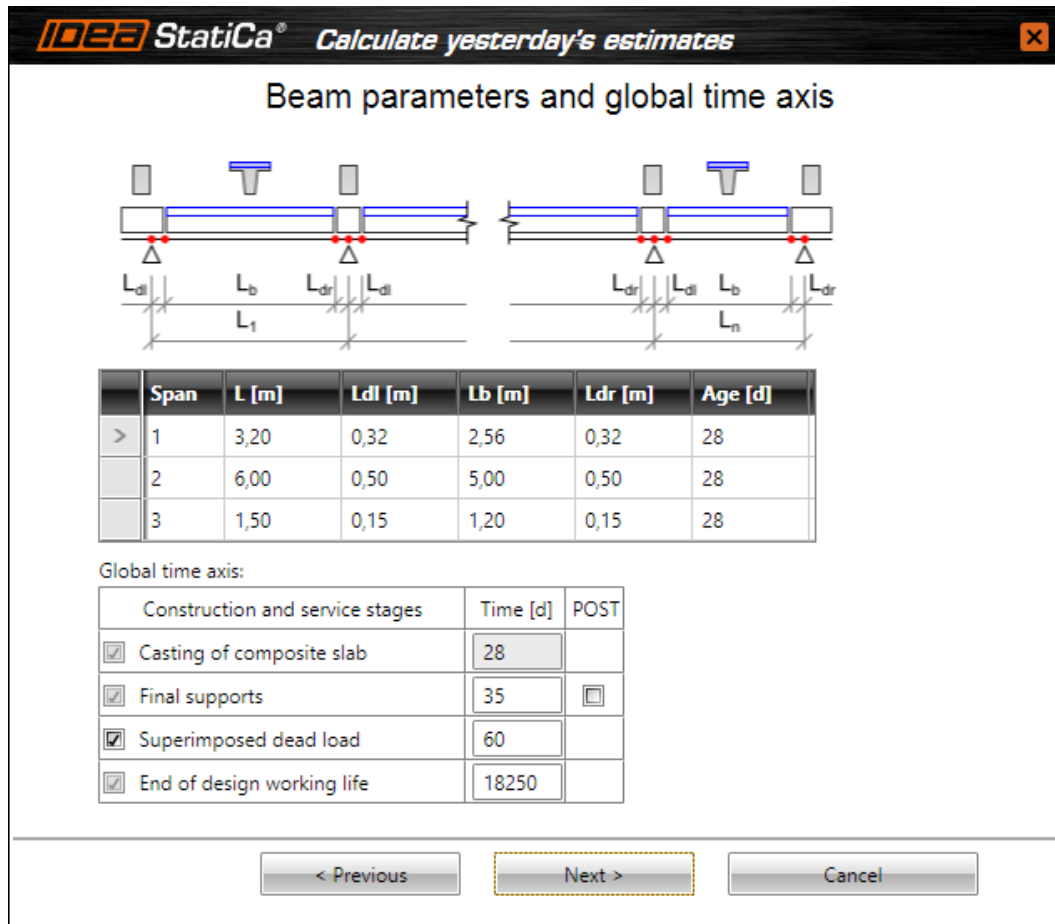
Service stages	Time [d]
<input checked="" type="checkbox"/> Superimposed dead load	60
<input checked="" type="checkbox"/> End of design working life	18250

Columns in the table:

- **Construction stages** – displays names of construction stages, which are generated automatically according to the type of analysed beam. If the checkbox before the name of stage is not selected, the corresponding stage is not taken into account neither in analysis, nor in section check.
- **Time** – input the time of production or service stage.
- **PRE** – turn on application of prestressing by pre-tensioned tendons in the **Prestressing** construction stage of prefabricated beams.
- **POST** – turn on/off application of prestressing by post-tensioned tendons into the corresponding construction stages.
- **L1** – input the length of left cantilever in the appropriate stage. The value determines the position of left temporary support in the appropriate stage. The position of right temporary support is set to the same value. The length of span is modified respectively.
- **L** – input the length of beam span in the appropriate stage. The value determines positions of temporary supports in the appropriate stage.
- **L2** – input the length of right cantilever in the appropriate stage. The value determines the position of right temporary support in the appropriate stage. The length of span is modified respectively.
- **n** – input number of internal supports in mid span for the appropriate stage.

5.1.8 Global time axis of beam made continuous by composite slab

For continuous beam composed of prefabricated concrete beams made continuous by cast-in-situ diaphragms above internal supports and composite concrete slab, lengths of individual prefabricated spans and properties of global time axis are set in the penultimate wizard step.



Beam parameters and global time axis

Span	L [m]	Ldl [m]	Lb [m]	Ldr [m]	Age [d]
1	3,20	0,32	2,56	0,32	28
2	6,00	0,50	5,00	0,50	28
3	1,50	0,15	1,20	0,15	28

Global time axis:

Construction and service stages	Time [d]	POST
<input checked="" type="checkbox"/> Casting of composite slab	28	
<input checked="" type="checkbox"/> Final supports	35	<input type="checkbox"/>
<input checked="" type="checkbox"/> Superimposed dead load	60	
<input checked="" type="checkbox"/> End of design working life	18250	

< Previous Next > Cancel

Columns in table of spans:

- **Span** – displays index of beam span.
- **L** – displays length of span.
- **Ldl** – input of length of adjacent part of left diaphragm. Zero value of length in the first span means that left cantilever is formed by part of prefabricated beam.
- **Lb** – displays length of prefabricated beam calculated automatically from length of span and diaphragms.
- **Ldr** – input of length of adjacent part of right diaphragm. Zero value of length in the last span means that right cantilever is formed by part of prefabricated beam.
- **Age** – input of equivalent age of prefabricated beam at the time of casting of diaphragms and composite slab.

Columns in table **Global time axis**:

- **Construction and service stages** – displays names of construction stages, which are generated automatically according to the type of analysed beam. If the checkbox before the name of stage is not selected, the corresponding stage is not taken into account neither in analysis, nor in section check.
- **Time** – input the time of production or service stage.

- **POST** – turn on/off application of prestressing by post-tensioned tendons into the corresponding construction stages.

5.1.9 Production stages of prefabricated beams composing beam made continuous by composite slab

For continuous beam composed of prefabricated concrete beams made continuous by cast-in-situ diaphragms above internal supports and composite concrete slab, properties of production stages of individual prefabricated beams composing the continuous beam can be set in the last step of wizard.

IDEA StatiCa® Calculate yesterday's estimates

Production stages of prefabricated beams

Identical history of prefabricated beams
 Identical temporary supports

		Beam 1 (Lb = 2,56m)	Beam 2 (Lb = 5m)	Beam 3 (Lb = 1,2m)			
Construction stages	Age [d]	PRE	POST	L1 [m]	L [m]	L2 [m]	n
<input checked="" type="checkbox"/> Casting	0						
<input checked="" type="checkbox"/> Prestressing	5	<input checked="" type="radio"/>	<input checked="" type="radio"/>	0	2,56	0	
<input checked="" type="checkbox"/> Storage yard	5,1		<input type="checkbox"/>	0,512	1,536	0,512	0
<input checked="" type="checkbox"/> Transport	25			0,512	1,536	0,512	0
<input checked="" type="checkbox"/> Temporary supports	26		<input type="checkbox"/>	0,512	1,536	0,512	0
<input checked="" type="checkbox"/> Casting of composite slab	28			0,512	1,536	0,512	1
<input checked="" type="checkbox"/> Final supports	35		<input type="checkbox"/>				
Service stages		Age [d]					
<input checked="" type="checkbox"/> Superimposed dead load	60						
<input checked="" type="checkbox"/> End of design working life	18250						

< Previous
Finish
Cancel

Dialog options:

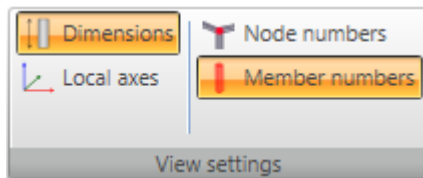
- **Identical history of prefabricated beams** – if the option is selected, history of production stages is adopted from local time axis of first beam – absolute values of the age at Transfer of prestressing and Storage yard, and relatively to the age of Casting of composite slab for stages of Transport and Temporary supports.
- **Identical temporary supports** – if the option is selected, number temporary supports and relative positions of temporary supports to the beam length are adopted from first beam.
- **Tabs Beam x** – properties of local time axis of individual prefabricated beams can be set on tabs. The setting is similar to the definition of construction stages for the one-span composite beam – see **Construction stages**. Application of prestressing by pre-tensioned or post-tensioned tendons into the corresponding construction stages can be turned on in **PRE** and **POST** columns.

5.2 Structure drawing settings

Use commands in ribbon groups **View settings** and **CSS drawing** to set the drawing of the structure. Those ribbon groups are available for all navigator commands.

Ribbon groups **View** and **Axo view** are available for beams loaded in 3D.

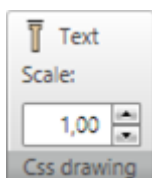
5.2.1 Ribbon group View settings



Commands in ribbon group **View settings**:

- **Dimensions** – switch on/off drawing of beam dimension lines in the picture of structure.
- **Local axes** – switch on/off drawing of local coordinate system of individual members in the picture of structure.
- **Node numbers** – switch on/off the drawing of node numbers in the picture of structure.
- **Member numbers** - switch on/off the drawing of member numbers in the picture of structure.

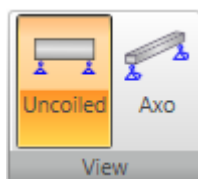
5.2.2 Ribbon group CSS drawing



Commands in ribbon group **Css drawing**:

- **Text** – switch on/ off the displaying of cross-section names.
- **Scale** – input of value of exceeded scale in Z axis direction.

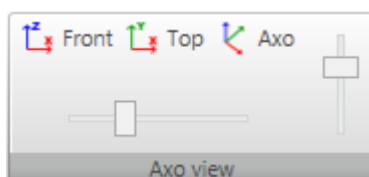
5.2.3 Ribbon group View



Commands in ribbon group **View**:

- **Uncoiled** – switch to draw the beam in uncoiled view.
- **Axo** – switch to draw the beam in axonometric view.

5.2.4 Ribbon group Axo view



Commands in ribbon group **Axo view**:

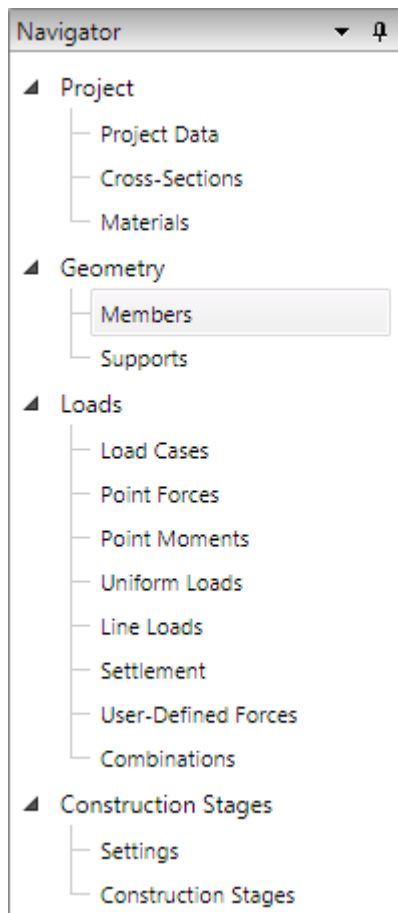
- **Front** – switch to the front view (opposite the direction of Y-axis of global coordinate system)
- **Top** – switch to the top view (opposite the direction of Z-axis of global coordinate system).
- **Axo** – switch to the default axonometric view.

Use the horizontal and vertical scrolls to rotate the view about corresponding axis.

6 Input of the structure

Appropriate structural data can be input using specific navigator commands. Input of data is sorted into following groups of commands: **Project**, **Geometry** and **Loads**. User simply goes through navigator from top to bottom.

All data are edited from keyboard into tables. There is no graphical interactive input like drawing by mouse.



6.1 Project data

Click navigator command **Project Data** to display the table with basic and identification data of the project.

The table content differs according to the type of the current beam.

Project data

Code and type settings	
Design code	EN
National annex	EN
EN 1992-2	<input type="checkbox"/>
Type of bridge	No bridge
Type of beam	Cast in situ concrete beam
Cast in situ concrete beam	Post-tensioned
Design working life	50 years
Bridge load rating	<input type="checkbox"/>
Geometry and loads settings	
Geometry and loads	Straight beam loaded in vertical plane
Flexible supports	<input type="checkbox"/>
Ratio for unloaded spans	0
Beam alignment	Top surface
Supports position	Bottom surface
Identification	
Name	
Number	
Author	
Description	
Date	8. 3. 2017

Properties group **Code and type settings**:

- **Design code** –the national code of the project is displayed.
- **National annex** – select the national annex for concrete members analysed according to EC code.
- **EN 1992-2** – switch on/off the check of concrete members according to EN1992-2.
- **Type of bridge** – displays the selected type of bridge for generation of variable loads groups.
- **Type of beam** – display (steel beam) or select (concrete beams) the type of analysed beam – see [Selecting a beam type](#).
- **Prefabricated concrete beam** – select a type of the prefabricated concrete beam – see [Selecting a beam type](#).
- **Cast-in-situ concrete beam** – select a type of the cast-in-situ concrete beam – see [Selecting a beam type](#).

- **Design working life** – select the considered design working life.
- **Bridge load rating** – switch on/off the calculation of bridge load rating.

Properties group **Geometry and load settings**:

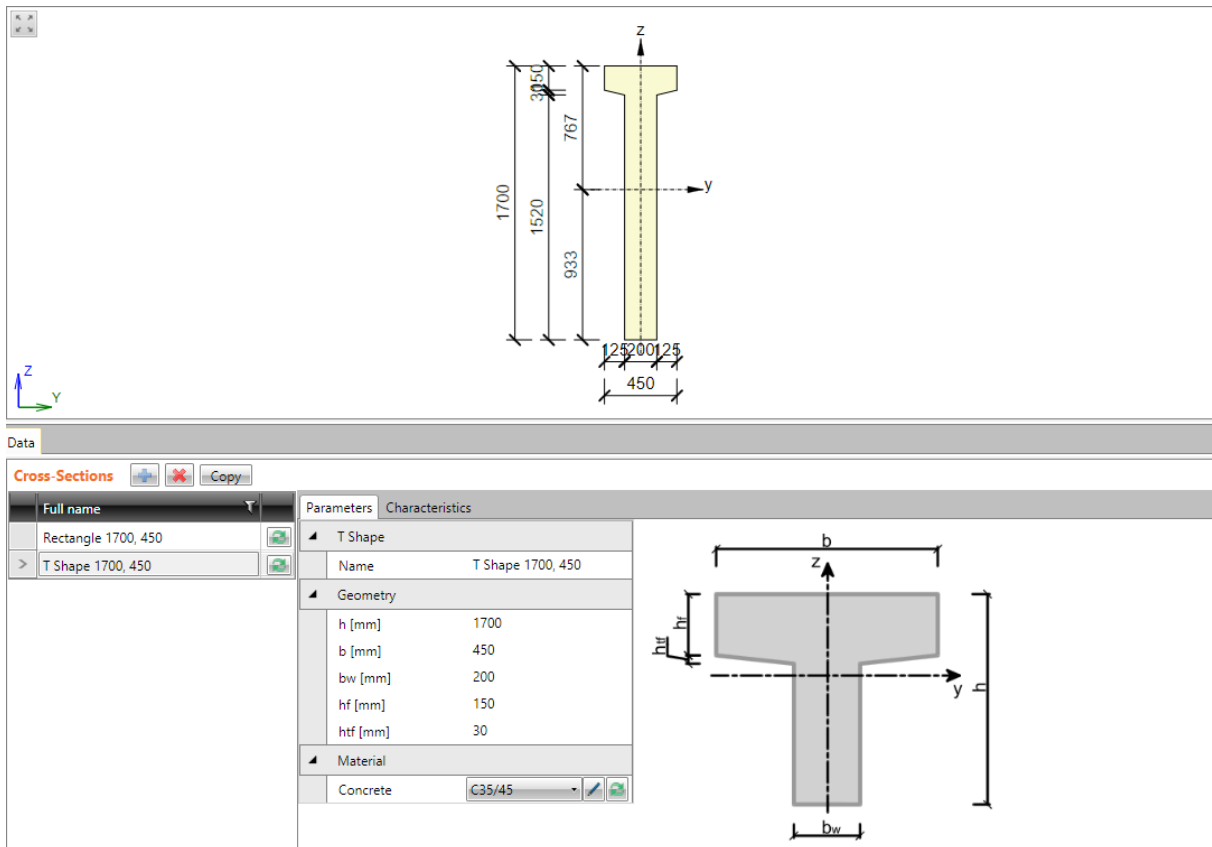
- **Geometry and loads** – select one of types of geometry and load acting modes:
 - **Straight beam loaded in vertical plane** – the straight horizontal beam can be defined (only X coordinates of nodes can be edited). Loads can act in the YZ plane of global coordinate system.
 - **Straight or polygonal beam loaded in 3D** – straight or polygonal beam can be defined (X and Y coordinates of nodes can be edited). Z coordinates of nodes can be edited, if the option Pitch (slope) of beam is selected. Loads can act in all directions of global coordinate system or in directions of local axes of members.
- **Pitch (slope) of beam** – if the option is selected, the slope of beam can be changing the Z-coordinate of selected nodes. This option is available only for Straight or polygonal beam loaded in 3D beam type.
- **Flexible supports** – if the option is selected, translational and rotational stiffness of supports can be defined.
- **Ratio for unloaded spans** – the ratio of size of loads in unloaded spans. When 0 is set, there is no load in unloaded spans. When 0,25 is set, 25% of variable load is put into unloaded spans.
- **Beam alignment** - see [Selecting a beam type](#).
- **Supports position** - see [Selecting a beam type](#).

Properties group **Identification**:

- **Name** – input of the project name.
- **Number** – input of the project identification number.
- **Author** – input name of the project author name.
- **Description** – input of additional information about the structure.
- **Date** – date of calculation.

6.2 Cross-sections

Each structural member has assigned one cross-section. Click navigator command **Project > Cross-sections** to input cross-sections.




The screenshot displays the software interface for defining a cross-section. At the top, a 3D model of a T-shaped cross-section is shown with dimensions: total height 1700 mm, flange thickness 30 mm, flange width 450 mm, web width 200 mm, and web height 933 mm. Below this, a 'Data' window is open, showing a table of cross-sections and a detailed parameter list for the selected 'T Shape 1700, 450'.

Full name	Parameters	Characteristics
Rectangle 1700, 450	<ul style="list-style-type: none"> ▲ T Shape 	
> T Shape 1700, 450	<ul style="list-style-type: none"> Name: T Shape 1700, 450 ▲ Geometry <ul style="list-style-type: none"> h [mm]: 1700 b [mm]: 450 bw [mm]: 200 hf [mm]: 150 htf [mm]: 30 ▲ Material <ul style="list-style-type: none"> Concrete: C35/45 	

To the right of the table, a 2D diagram of the T-section is shown with labels: b for flange width, h for total height, h_f for flange thickness, b_w for web width, and h_{tf} for web height.

Click **Copy** above the table of cross-sections to copy an existing cross-section.

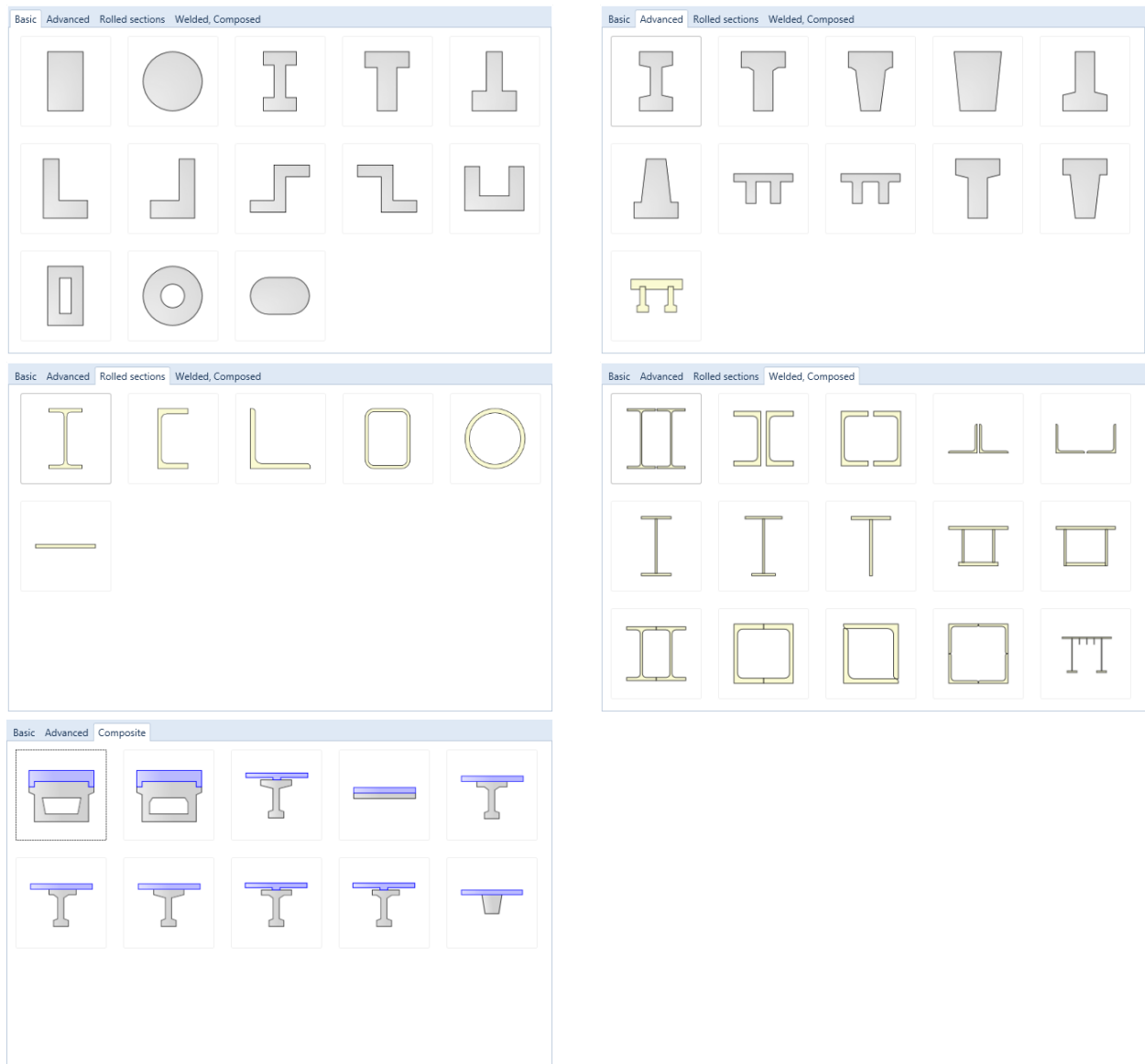
Click  above the table to delete the selected cross-section. Cross-section, which is used anywhere in the structure, cannot be deleted.

Click  above the table to add a new cross-section.


Dialog **Cross-Section Navigator** appears with groups of available cross-sections:

- **Basic** – standard shapes of concrete cross-sections;
- **Advanced** – shapes of concrete cross-sections used mostly in bridge designs, general concrete section;
- **Rolled** – steel rolled cross-sections I, L, U, T, plate, circular, square and rectangular hollow section;
- **Welded, composed** – more complex shapes composed from steel rolled sections, general steel section.
- **Composite** – cross-section shapes for analysis of composite cast-in-situ or prefabricated concrete beams, taking into account time dependent analysis.

Click the picture of required cross-section shape to add a new cross-section. The new cross-section is set as current cross-section and its properties are displayed in the table in the data window.



There is a list of already defined cross-sections in the left part of data window. Following buttons are available for each cross-section:






-  - input a new cross-section. Current cross-section is replaced by a new one.

Properties of current cross-section can be edited on the tab **Parameters** in the right part of data table. Table of cross-sectional characteristics is displayed on the tab **Characteristics**.

6.3 Materials

Click navigator command **Project > Materials** to review or edit material characteristics of the materials in the project.

Materials

Name	Type		
> C25/30	Concrete		
Edit of S 235	Steel	 	




Physical properties

m [kg/m ³]	2500
v	0,2
α [10e-6/K]	10
λ [W/(m.K)]	0,8
c [kJ/(kg.K)]	0,00075

EN 1992-1-1

f _{ck} [MPa]	25,0
Calculate dependence	<input checked="" type="checkbox"/>
E _{cm} [MPa]	31475,8
G [MPa]	13114,9
f _{cm} [MPa]	33,0
f _{ctm} [MPa]	2,6
f _{ctk,0,05} [MPa]	1,8
f _{ctk,0,95} [MPa]	3,3
ε _{c2} [1e-4]	20,0
ε _{cu2} [1e-4]	35,0
Exponent - n	2
ε _{c3} [1e-4]	17,5
ε _{cu3} [1e-4]	35,0
Aggregate size [mm]	16
Aggregate type	Quartzite
Cement class	R
Diagram type	Parabolic

All materials, which are assigned to cross-sections in the project, are listed in the **Materials** table.

-  - click the edit button to convert the material to the editable material. The name of material changes and the particular material characteristics can be edited. The change affects all cross-sections, which have assigned the edited material.
-  - click the button to display the dialog with the list of all materials available in the system material library. When a material from the library is selected, it replaces the edited material. The change affects all cross-sections, which have assigned the edited material.
-  - save current (modified) material to the selected or new materials table into the users database.
- Clean** – the button is available, if there is a material in the project, which is not assigned to any cross-section. Click the button to delete not used materials from the project.

6.4 Geometry

Use commands in the navigator group **Geometry** to input the beam members, haunches and supports.

6.4.1 Members and haunches

Click navigator command **Geometry > Members** to input and edit beam members, eccentricities of members and haunches on members.

Beam members can be edited in the table **Members** in the data window.

The screenshot displays the 'Structural scheme' window showing a beam with a haunch. The beam is divided into five segments labeled 1 to 5. Segment 1 is a small rectangle at the left end, segment 2 is a rectangle, segment 3 is a T-shaped haunch, segment 4 is a rectangle, and segment 5 is a small rectangle at the right end. Dimensions are shown: 0.33, 0.67, 11.66, 23.66, 12.00, 25.66, and 0.67. A coordinate system (X, Y, Z) is shown at the bottom left.

The 'Data' window below shows the 'Members' table and the haunch configuration panel.

Member	Length [m]	Delta X [m]	Delta Y [m]	Cross-Section
1	0,33	0,33	0,00	1 - Rectangle 1
2	0,67	0,67	0,00	1 - Rectangle 1
> 3	23,66	23,66	0,00	2 - T Shape 250
4	0,67	0,67	0,00	1 - Rectangle 1
5	0,33	0,33	0,00	1 - Rectangle 1

The haunch configuration panel shows the following settings:

- Haunch Type: Both ends
- Left end:
 - Cross-section: 3 - T Shape 1700, 450 (T Sha)
 - Length [m]: 11,66
- Right end:
 - Cross-section: 3 - T Shape 1700, 450 (T Sha)
 - Length [m]: 12,00
- Eccentricities:
 - Beg Y [m]: 0,00
 - Beg Z [m]: 0,08
 - End Y [m]: 0,00
 - End Z [m]: 0,08
 - Haunch alignmer: Bottom surface






Click  above the table to add a new member at the end of beam.

Table **Members** contains following columns:

- **Member** – the number of member is displayed.
- **X** – coordinate of end of member from the origin of global coordinate system.
- **Length** – input the length of the member. If the length of one member changes, all following nodes move. Lengths of following members remain unchanged.
- **Delta X** – input the distance between end node and beginning node of the member in the direction of X-axis of global coordinate system (coordinate increment in the X-axis direction). This option is available only for **Straight or polygonal beam loaded in 3D beam type**.
- **Delta Y** – input the distance between end node and beginning node of the member in the direction of Y-axis of global coordinate system (coordinate increment in the Y-axis direction). This option is available only for **Straight or polygonal beam loaded in 3D beam type**.
- **Cross-section** - assign the appropriate cross-section to the member. Cross-section can be selected from the list of available cross-sections. Click

 to change the parameters of the current cross-section. Click  to add a new cross-section into the project. The newly added cross-section replaces the current one.



-  - add new member behind the appropriate member of beam.
-  - delete the appropriate member.

6.4.1.1 Haunches and eccentricities

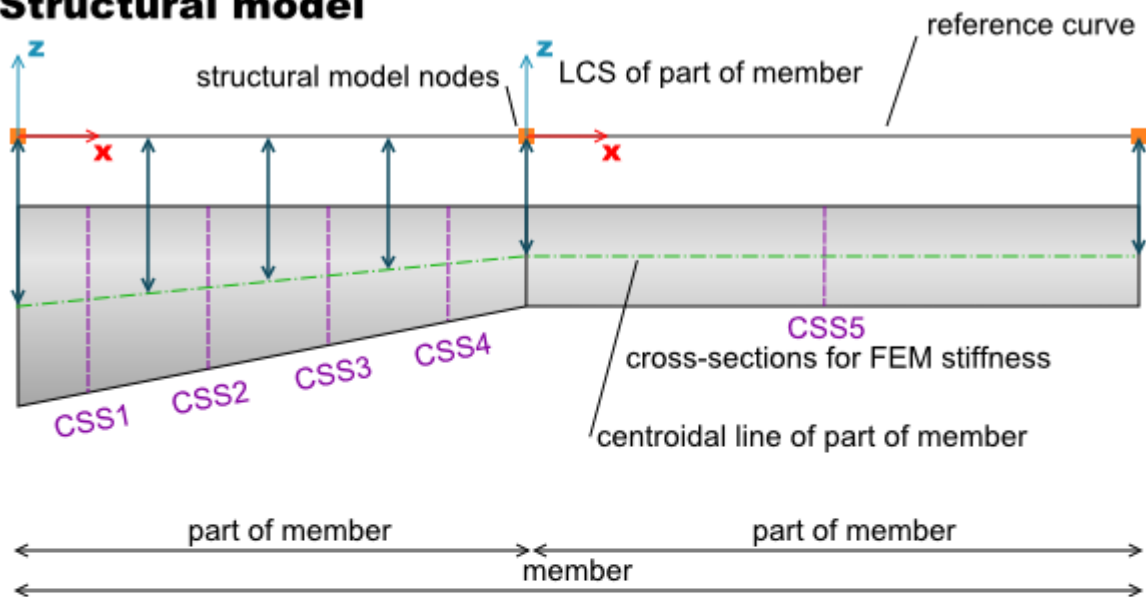
A haunch can be defined on any particular member of the beam. The haunch can be defined at the beginning or at the end or on both ends of the member. The haunch is defined by length and by two cross-sections – by the original cross-section of the member and by the cross-section assigned to the haunch on the appropriate member end.

A table with haunch properties and eccentricities properties is displayed for the current member (selected in **Members** table).

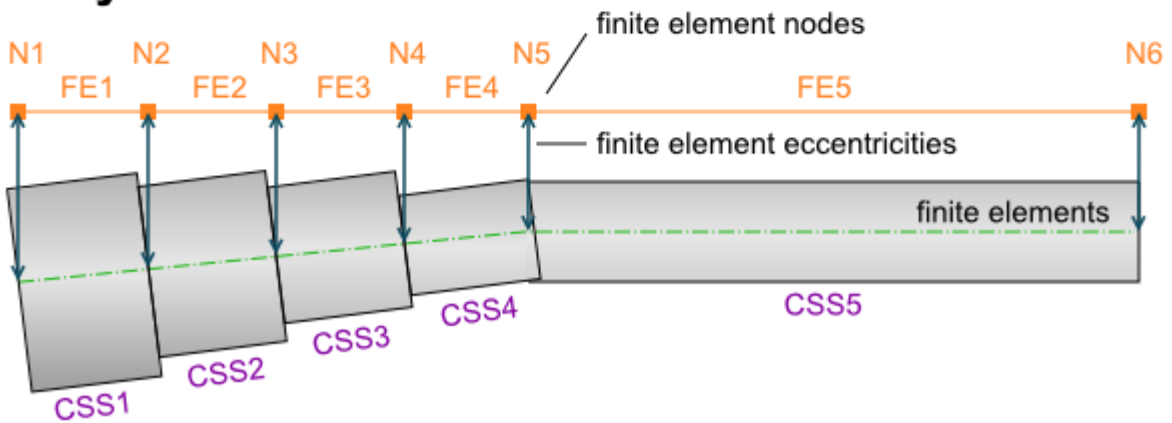
Particular haunch properties:

- **Type** – select the position of the haunch on member.
 - **None** – there is no haunch defined on the member
 - **Symmetrical** – haunches are defined on both ends of member and both haunches have the same properties
 - **Left end** – haunch is defined on the begin of the member
 - **Right end** – haunch is defined on the end of the member
 - **Both ends** – haunches are defined on both ends of the member and both haunches can have different properties.
- **Left (Right) end** – haunch properties on the appropriate end of member
 - **Cross-Section** – select the cross-section of the haunch in the list. The list contains only cross-sections which can create haunch on the current member. Click edit button  to change properties of the haunch cross-section.
Click  button to input new cross-section of haunch. This cross-section is of the same type as the cross-section of the current member.
- **Eccentricities** – properties of eccentricities at the ends of member:
 - **Beg Y** – input distance between the centre of gravity of cross-section and the reference axis at the span beginning in direction of Y-axis.
 - **Beg Z** – input distance between the centre of gravity of cross-section and the reference axis at the span beginning in direction of Z-axis.
 - **End Y** – input distance between the centre of gravity of cross-section and the reference axis at the end of span in direction of Y-axis.
 - **End Z** – input distance between the centre of gravity of cross-section and the reference axis at the end of span in direction of Z-axis.
- **Haunch alignment** – select mode of alignment of the haunch in the span:
 - **Top surface** – the definition cross-sections of haunch in span are aligned to the top edge of first cross-section in span.
 - **Center line** – the centres of gravity of haunch cross-sections are aligned to centre line of the span.
 - **Bottom surface** – the definition cross-sections of haunch in span are aligned to the bottom edge of first cross-section in span.

Structural model



Analysis model



Transformation of the structural model to the analysis model (finite elements for the analysis)

6.4.2 Supports

Click navigator command **Geometry > Supports** to input and edit nodal supports.

Supports								
	Node	Nodal Support						Z [m]
		X	Y	Z	Rx	Ry	Rz	
	1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0,00
>	2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0,03
	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0,08
	4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,92
	5	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,97
	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	2,00

Supports and slope of beam can be modified in the table **Supports** in the data window.

Table **Nodes** contains following columns:

- **X** – switch on/off the support to resist translations in the direction of global X axis.
- **Y** - switch on/off the support to resist translations in the direction of global X axis. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Z** – switch on/off the support to resist translations in the direction of global Z axis.
- **Rx** – switch on/off the support to resist the rotation about the global X axis. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ry** – switch on/off the support to resist the rotation about the global Y axis.
- **Rz** - switch on/off the support to resist the rotation about the global Z axis. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Z** – input the Z-coordinate of node in global coordinate system. The coordinate value can be defined only for nodes unlocked by clicking the button . Coordinates of intermediate nodes (between the unlocked nodes) are interpolated – the unlocked nodes define the slope of the appropriate part of the beam. This option is available only for **Straight or polygonal beam loaded in 3D** beam type and **Pitch (slope) of beam** option selected.

For **Straight or polygonal beam loaded in 3D** beam type the rotation of support can be defined in supported node. Also support stiffness can be defined, if it is enabled in **Project data**.

Supports								
	Node	Nodal Support						Z [m]
		X	Y	Z	Rx	Ry	Rz	
	1	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0,00
>	2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0,03
	3	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0,08
	4	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,92
	5	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	1,97
	6	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	2,00

Support system	
Local system	Global
Rotation	0

Support stiffnesses	
kx [MN/m]	10000000,0
ky [MN/m]	10000000,0
kz [MN/m]	10000000,0
krx [MNm/rad]	10000000,0

Eccentricities	
Y [m]	0,00
Z [m]	0,00

Group **Support system** – definition of support rotation:

- **Local system** – select the mode to determine the rotation of support:
 - **Global** – the final rotation of support is defined by a sequence of rotations about individual axes of global coordinate system.
 - **Local – from previous beam** – the support is rotated in such a way that the directions of individual axes of rotated supports are parallel to corresponding axes of local coordinate system of the previous member prior the support.
 - **Local – from next beam** – the support is rotated in such a way that the directions of individual axes of rotated supports are parallel to corresponding axes of local coordinate system of the member following the support.
 - **Local – average from both beams** – the support is rotated in such a way that the directions of individual axes of rotated supports are parallel (or perpendicular) to the direction of the half of angle between corresponding axes of local coordinate systems of members prior and following the support.
- **Rotation** – input the sequence of rotations about individual axes of global coordinate system using the string in format e. g. X30 Y45 Z20.

Group **Support stiffnesses** – definition of stiffnesses of supports. For flexible supports, the value of stiffness 1e7 MN/m, or MN/rad means the rigid support in the appropriate direction.

- **kx** – input of translational stiffness of support in the x-axis direction of support coordinate system.
- **ky** - input of translational stiffness of support in the y-axis direction of support coordinate system.
- **kz** - input of translational stiffness of support in the z-axis direction of support coordinate system.
- **krx** - input of rotational stiffness of support about the x-axis direction of support coordinate system.
- **kry** - input of rotational stiffness of support about the y-axis direction of support coordinate system.
- **krz** - input of rotational stiffness of support about the z-axis direction of support coordinate system.

Group **Eccentricities** – definition of support eccentricity:

- **Y** – distance between support and the reference curve in global Y axis direction.
- **Z** - distance between support and the reference curve in global Z axis direction.

6.5 Loads

Individual commands of group **Loads** are used for input of load cases, point loads, uniform loads, line loads, settlements and load combinations.

Internal forces defined by user can be entered as a special load type. The courses of internal forces along members can be defined manually or can be imported from XML file.

6.5.1 Groups of load cases

Each load case is assigned to one group of load cases.

Load cases, which are in one group, are considered as one load case when generating load case coefficients for combinations.

Click **Loads > Load cases** to input groups of load cases.

6.5.1.1 Groups of permanent loads

Groups of permanent loads are defined in the table on tab **Permanent load groups**.

Name	$\gamma G_{,sup}$ [-]	$\gamma G_{,inf}$	ξ [-]	
> LG1	1,35	1,00	0,85	

Table **Permanent load groups** contains following columns:

- **Name** – input name of load group.
- **$\gamma G_{,sup}$** – input of partial factor for permanent unfavourable load cases in ULS combinations.
- **$\gamma G_{,inf}$** – input of partial factor for permanent favourable load cases in ULS combinations.
- **ξ** – input of reduction factor of unfavourable permanent loads.

To add new group of permanent loads click above the table.

6.5.1.2 Groups of variable loads


Groups of variable loads are defined in the table on tab **Variable load groups**.

Name	Type	γq [-]	$\Psi 0$ [-]	$\Psi 1$ [-]	$\Psi 2$ [-]	
> LG2	Exclusive	1,50	0,70	0,50	0,30	
LG3	Standard	1,50	0,70	0,50	0,30	

Table **Variable load groups** contains following columns:

- **Name** – input name of load group.
- **Type** – select the type of variable loads group. The type determines the behaviour of load cases from the group in appropriate combinations of load cases.
 - **Standard** – load cases from the group are considered as additional load in ULS and SLS combinations.

- **Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. Only one load case from the group can act in single critical combination.
- **Accidental** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are considered as design value of an accidental action A_d .
- **Accidental, Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are considered as design value of an accidental action A_d . Only one load case from the group can act in single critical combination.
- **Fatigue, Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Fatigue combination the load cases are considered as fatigue load Q_{fat} . Only one load case from the group can act in single critical combination.
- γ_q – input the value of partial load factor of variable load cases in ULS combinations.
- ψ_0 – input the value of partial load factor of variable load cases in ULS and SLS Characteristic combinations.
- ψ_1 – input the value of partial load factor of variable load cases in SLS Frequent combinations.
- ψ_2 – input the value of partial load factor of variable load cases in SLS Quasi-permanent combinations.

To add new group of variable loads click  above the table.

6.5.1.3 Groups of variable loads for bridges

Load cases Permanent load groups Variable load groups							
Variable Load Groups		Generate bridge groups	Road bridge				
Name	Type	Road bridge load group		γ_q	ψ_0	ψ_1	ψ_2
> gr1a - TS	Exclusive	gr1a - TS	gr1a - TS	1,5	0,75	0,75	0
gr1a - UDL	Exclusive	gr1a - UDL	gr1a - UDL	1,5	0,4	0,4	0
gr1a - Pedestr. + cycle trac	Exclusive	gr1a - Pedestr. + cycle track	gr1a - Pedestr. + cycle track	1,5	0,4	0,4	0
gr1b - Single axle	Exclusive	gr1b - Single axle	gr1b - Single axle	1,5	0	0,75	0
gr2 - Horizontal forces	Exclusive	gr2 - Horizontal forces	gr2 - Horizontal forces	1,5	0	0	0
gr3 - Pedestrian loads	Exclusive	gr3 - Pedestrian loads	gr3 - Pedestrian loads	1,5	0	0,4	0
gr4 - Crowd loading	Exclusive	gr4 - Crowd loading	gr4 - Crowd loading	1,5	0	0	0
gr5 - Special vehicles	Exclusive	gr5 - Special vehicles	gr5 - Special vehicles	1,5	0	0	0
Fwk - Persistent	Exclusive	Fwk - Persistent	Fwk - Persistent	1,5	0,6	0,2	0
Fwk - Execution	Exclusive	Fwk - Execution	Fwk - Execution	1,5	0,8	0	0
F**W - Design	Exclusive	F**W - Design	F**W - Design	1,5	1	0	0
Thermal - Tk	Exclusive	Thermal - Tk	Thermal - Tk	1,5	0,6	0,6	0,5
QSn,k - Execution	Exclusive	QSn,k - Execution	QSn,k - Execution	1,5	0,8	0	0
Construction - Qc	Exclusive	Construction - Qc	Construction - Qc	1,5	1	0	1

- **Generate bridge groups** – generates variable loads groups including default values of ψ coefficients for the current type of bridge according to tables in Appendix A2 of EN 1990. The button is available only if no group of variable loads exists.
- **List of bridge types** – select the type of bridge to generate the variable loads groups for. The change of bridge type is available only if no group of variable loads exists – it means that all variable loads has to be changed to permanent and all groups of variable loads must be deleted.
 - **Road bridge** – variable loads groups for road bridges according to table A2.1 of appendix A2 EN 1900 will be defined/generated.

- **Footbridge** – variable loads groups for footbridges according to table A2.2 of appendix A2 EN 1900 will be defined/generated.
- **Railway bridge** - variable loads groups for road bridges according to table A2.3 of appendix A2 EN 1900 will be defined/generated.
- **No bridge** – ordinary (not for bridges) variable loads groups will be defined.

For bridge structures, the table **Variable loads groups** contains additional column:

- **Xxx bridge loads group** ... - select the type of bridge load. The selected type of bridge load determines the possible interaction of loads in the resulting critical combinations.

Loads groups can act together in the resulting critical combination respecting following rules:

Footbridges:

- extreme load from groups of traffic loads (one of groups gr1, gr2, Qfwk);
- wind load Fwk;
- temperature load Tk;
- snow load Qsn,k;
- construction load Qc.

Road bridges:

- extreme load from groups of traffic loads gr1a, gr1b..gr5, where gr1a is evaluated as envelope from gr1a-TS, gr1a-UDL and gr1a-pedestrians and cycle-track (it means that loads from all subgroups of gr1a can act together in the resulting combination);
- extreme load from groups of wind loads (one of groups Fwk, Fw*);
- temperature load Tk;
- snow load Qsn,k;
- construction load Qc.

Railway bridges:

- extreme load of main traffic loads groups (one of groups gr11..gr31);
- extreme load of other operating actions groups (one of groups Aerodynamics actions, Maintenance);
- extreme load of wind forces groups (one of groups Fwk, Fw*);
- temperature load Tk;
- snow load Qsn,k;
- construction load Qc.

Names of user defined bridge load groups can be defined in table **User bridge load group**. The defined names are added to the list of bridge load groups in column **xxx bridge load group** in the table **Variable load groups**. There are no default values of load coefficients defined for the user bridge load group, the required coefficient values must be set in the table **Variable load groups**.

Commands above the table **User bridge load group**:

-  - add new user defined bridge load group.

Columns in **User bridge load group** table:

-
- **Name** – input name of load group.
 - **Traffic** – if selected, loads in the group are considered as the traffic loads when evaluating combinations.

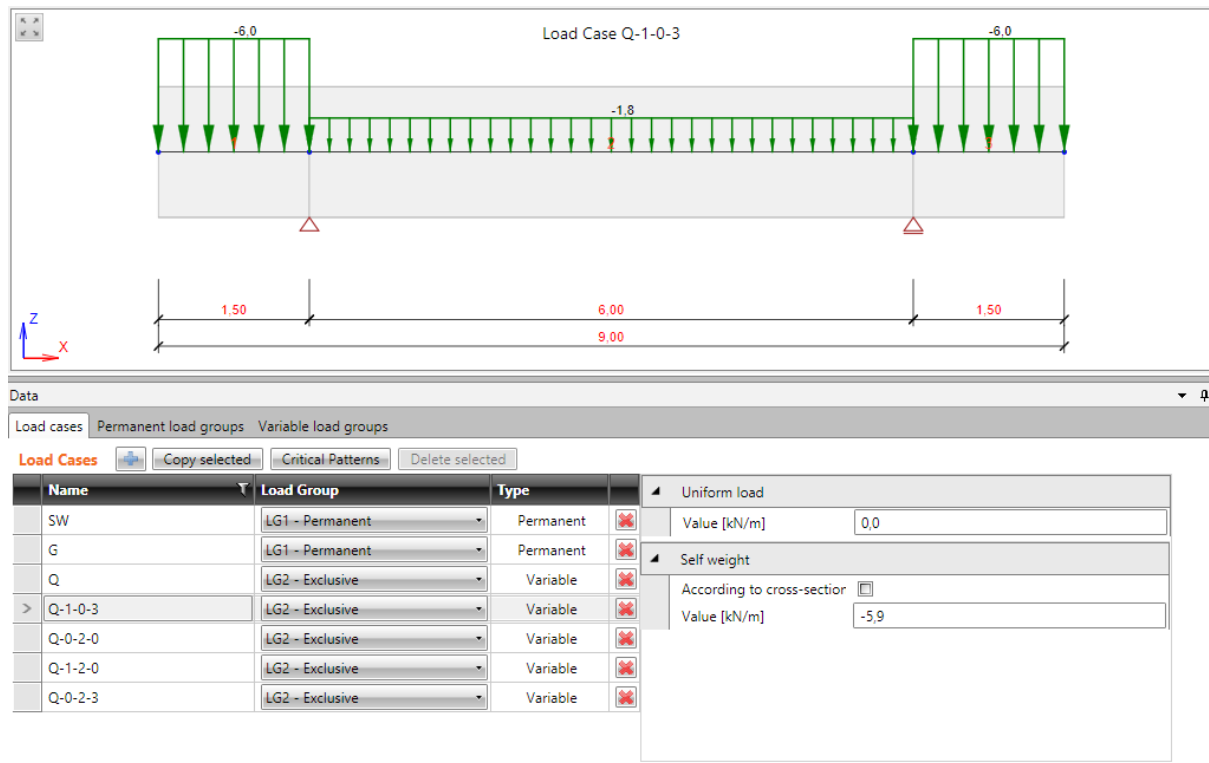
6.5.2 Load cases of non-staged beams

Click navigator command **Loads > Load cases** to input load cases.

Loads are associated to load cases. Load cases can be of type Permanent or Variable. Each case can contain point loads, uniform loads, line loads and settlement loads.

If Prestressing is active in the project, permanent load cases named “**POST**” or “**PRE**” are generated automatically. Those load case are used for transfer of equivalent load actions caused by prestressing tendons into the structural model of the beam. Load case for prestressing cannot be deleted.

Load case of self-weight may be generated automatically.



The screenshot displays a beam model with a coordinate system (Z vertical, X horizontal) and a load case configuration table. The beam has a total length of 9.00 units, with a central span of 6.00 units and two side spans of 1.50 units each. The load case 'Q-1-0-3' is defined with a central uniform load of -1.8 kN/m and two side uniform loads of -6.0 kN/m.

Name	Load Group	Type	Uniform load	Self weight
SW	LG1 - Permanent	Permanent	Value [kN/m] 0,0	
G	LG1 - Permanent	Permanent		
Q	LG2 - Exclusive	Variable		According to cross-section <input type="checkbox"/>
> Q-1-0-3	LG2 - Exclusive	Variable		Value [kN/m] -5,9
Q-0-2-0	LG2 - Exclusive	Variable		
Q-1-2-0	LG2 - Exclusive	Variable		
Q-0-2-3	LG2 - Exclusive	Variable		

Click  above the table of load cases to add new load case.


Click **Copy** above the table to copy selected load case including all loads associated to the load case.

If the current load case is of type Variable, click **Critical Patterns** above the table to generate more load cases to get the most critical moments in spans and above supports and also the most critical reactions in supports.

Click **Self weight** above the table to add load case of self-weight (if the SW load case does not exist).

Table of additional properties is displayed for the current load case.

Table **Load Cases** contains following columns:

- **Name** – enter name of load case.
- **Load group** – select the load group to assign the load case to.
- **Type** - click the cell to change the type of load.
-  - delete the appropriate load case.

Properties group **Uniform load** (not available for load case of self-weight type):

- **Value** - input value of the uniform load, which (in the appropriate load case) acts on each member of the beam.

Properties group **Self weight** (available only for load case of self-weight type) :

- **According to cross-section** – if the option is selected, the self-weight is generated automatically on particular beam members respecting the current cross-section of the member. The value cannot be changed.
- **Value** – if **According to cross-section** is not selected, the value of distributed load acting as self-weight on all beam members is calculated respecting the defined default cross-section and the value is editable. Note, that the calculated value does not change after any change of beam!

6.5.3 Load cases of staged beams

Following load cases are generated for each stage of staged beam:

- Load case to store increments of permanent loads. The load case name is G(n), where n is the number of the appropriate stage. Loads, which are applied in the appropriate stage, can be defined in this load case.
- Load case to store the rheological effects. The load case name is R(n), where n is the number of the appropriate stage. No loads can be defined to these load cases. Actions caused by rheological effects are stored to these load cases during the calculation. These load cases are generated for beams analysed by time dependent analysis.

These automatically generated load cases cannot be deleted.

The number of assigned stage is displayed in the column **C. stage** of **Load cases table**. The load case can be assigned to the required stage during the definition of stages – see **Construction stages**.

Load cases						
Permanent load groups Variable load groups						
Load Cases + Copy Critical Patterns <input type="checkbox"/> Show rheological load cases						
Name	Uniform Load [kN/m]	Load Group	C. stage	Type		
> SW (1)	According to cross-section	LG1 - Permanent	1	Permanent	<input type="checkbox"/>	
G (2)	0,0	LG1 - Permanent	2	Permanent	<input type="checkbox"/>	
PRE (2)	Not editable	LG1 - Permanent	2	Permanent	<input type="checkbox"/>	
G (3)	0,0	LG1 - Permanent	3	Permanent	<input type="checkbox"/>	
G (4)	0,0	LG1 - Permanent	4	Permanent	<input type="checkbox"/>	
G (5)	0,0	LG1 - Permanent	5	Permanent	<input type="checkbox"/>	
SWS (6)	According to cross-section	LG1 - Permanent	6	Permanent	<input type="checkbox"/>	
G (7)	0,0	LG1 - Permanent	7	Permanent	<input type="checkbox"/>	
G (8)	-10,0	LG1 - Permanent	8	Permanent	<input type="checkbox"/>	
G (9)	0,0	LG1 - Permanent	9	Permanent	<input type="checkbox"/>	
Q	-15,0	LG2 - Exclusive		Variable	<input type="checkbox"/>	

Note:
Rheological load cases are not presented in data table.

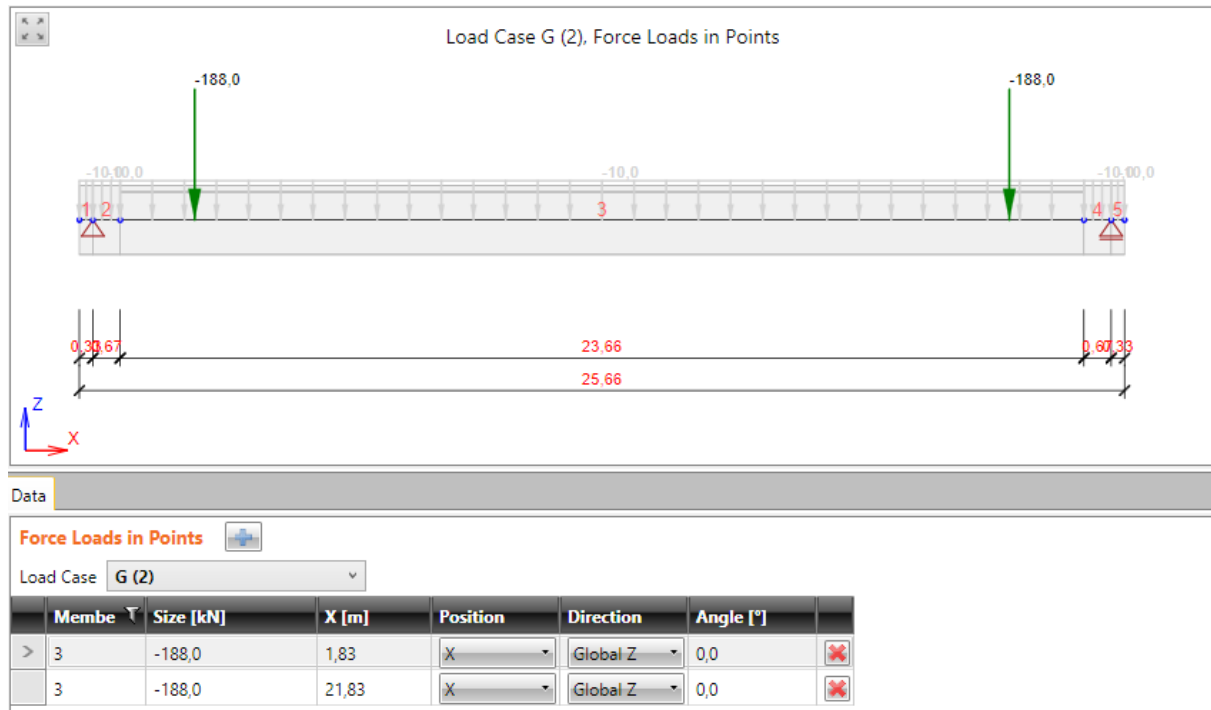
Checkbox **Show rheological load cases** is available for structures analysed using time dependent analysis – switches on/off displaying of load cases for rheological effect in the table of defined load cases.

6.5.4 Point forces

Click navigator command **Loads > Point forces** to input point forces.

Point forces can be defined on any member of the beam. The force can act in the direction of selected axis of the local or global coordinate system. Also inclination can be defined.

Positive value defines the force, which acts in the positive direction of axis.



The screenshot displays a beam model titled "Load Case G (2), Force Loads in Points". The beam is supported at two points, labeled 1 and 5. Two point forces of -188.0 kN are applied at positions 1.83 m and 21.83 m from the left end. The beam is divided into segments of 0.33, 6.7, 23.66, and 0.67, 3.3 m. A coordinate system (X, Y, Z) is shown at the bottom left.

The "Data" section shows the following table:

Membe	Size [kN]	X [m]	Position	Direction	Angle [°]
> 3	-188,0	1,83	X	Global Z	0,0
3	-188,0	21,83	X	Global Z	0,0

Current load case can be set in the list **Load cases**. Point forces already defined in this load case are displayed in the table.



Click  above the table to add new point force to the current load case.

Table **Point forces** contains following columns:

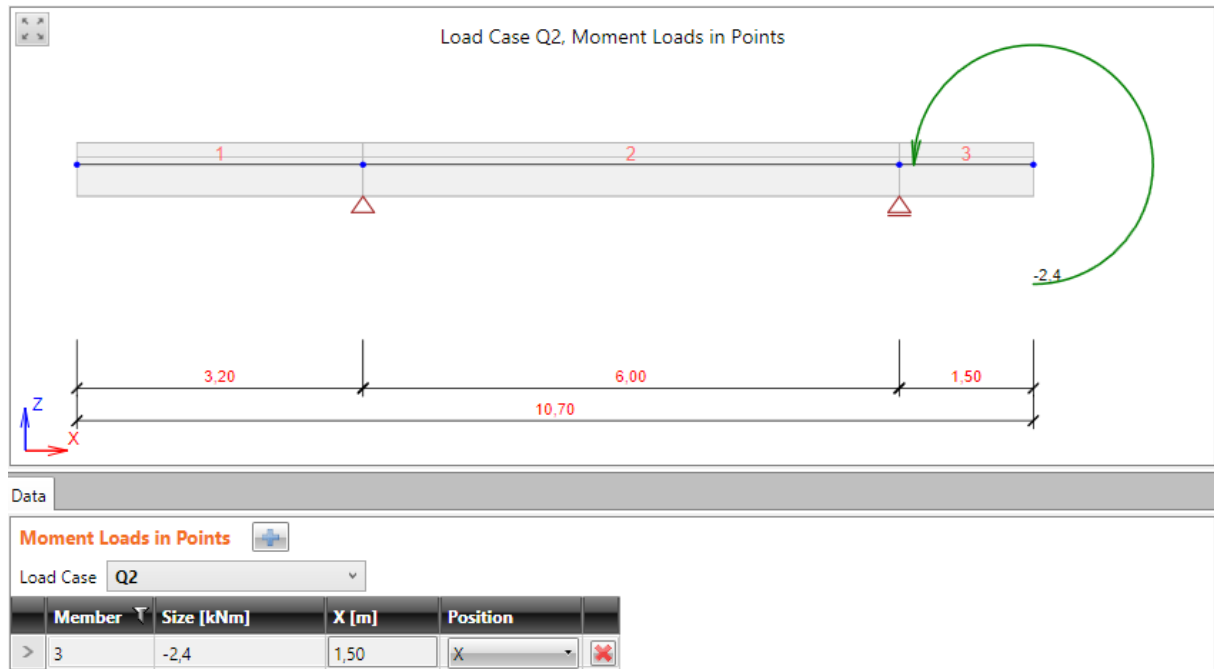
- **Member** – input the number of member loaded by point force.
- **Size** – input the value of point force.
- **X** – input the position of force from the beginning of the member. The value is taken into account only if the value in column **Position** is set to **X**.
- **Position** – select the mode to define the force position. Following modes are available:
 - **X** – one force on the member in the distance defined in the X-column.
 - **P1/2** – one force in the mid of the member.
 - **P1/3** – two forces in the thirds of the member.
 - **P1/4** – three forces in the quarters of the member.
 - **P1/5** – four forces in the fifths of the member.
- **Direction** - select the axis of coordinate system in which point force is defined. Following directions are available:
 - **Global Z** – the load acts in the direction of Z-axis of global coordinate system.
 - **Global Y** – the load acts in the direction of Y-axis of global coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.

- **Global X** – the load acts in the direction of X-axis of global coordinate system.
- **Local z** – the load acts in the direction of z-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Local y** – the load acts in the direction of y-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Local x** – the load acts in the direction of x-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Angle** – input the value of point force inclination to specified direction. This option is available only for **Straight beam loaded in vertical plane** beam type.
- **Ey** – input of load eccentricity in the y-axis direction of member local coordinate system. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ez** – input of load eccentricity in the z-axis direction of member local coordinate system. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
-  - delete the appropriate point force.

6.5.5 Point moment loads

Click navigator command **Loads > Point Moments** to input point moment loads.

Point moment load can be defined on any member of the beam. Moment acts about the selected axis of member local coordinate system.




Load Case Q2, Moment Loads in Points

Member	Size [kNm]	X [m]	Position
3	-2,4	1,50	X

Current load case can be set in the list **Load cases**. Point moment loads already defined in this load case are displayed in the table.

Click  above the table to add new point moment load to the current load case.

Table **Moment loads in points** contains following columns:

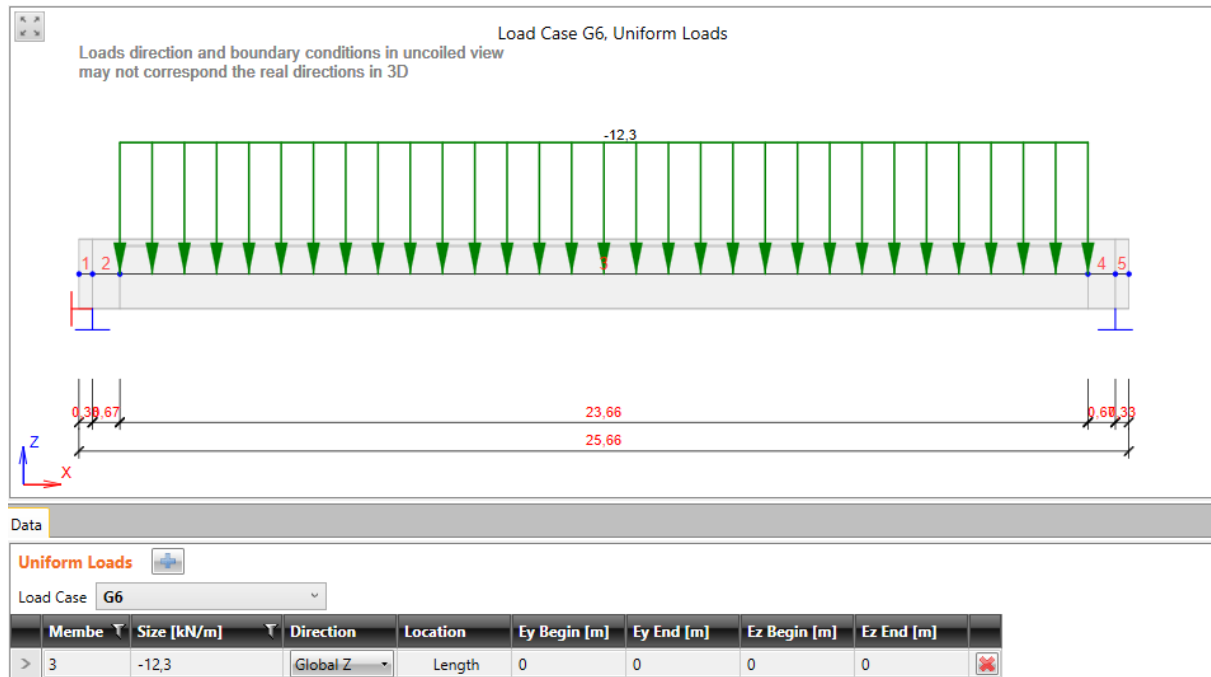
- **Member** – input the number of member loaded by point moment load.
- **Size** – input the value of point moment load.
- **X** – input the position of moment load from the beginning of the member. The value is taken into account only if the value in column **Position** is set to **X**.
- **Position** – select the mode to define the moment position. Following modes are available:
 - **X** – one moment on the member in the distance defined in the X-column.
 - **P1/2** – one moment in the mid of the member.
 - **P1/3** – two moments in the thirds of the member.
 - **P1/4** – three moments in the quarters of the member.
 - **P1/5** – four moments in the fifths of the member.
- **Direction** - select the axis of member local coordinate system in which point moment is defined. This option is available only for **Straight or polygonal beam loaded in 3D** beam type. Following directions are available:
 - **Local x** – the moment acts about the x-axis of the member local coordinate system.
 - **Local y** – the moment acts about the y-axis of the member local coordinate system.
 - **Local z** – the moment acts about the x-axis of the member local coordinate system.
-  - delete the appropriate moment load.

6.5.6 Uniform loads

Click navigator command **Loads > Uniform Loads** to input uniform loads.

Uniform load can be defined on any member of the beam. The load can act in direction of selected axis of local or global coordinate system. Also inclination can be defined. The load acts on the whole member.

Positive value defines the force, which acts in the positive direction of axis.



Load Case G6, Uniform Loads
Loads direction and boundary conditions in uncoiled view may not correspond the real directions in 3D

-12,3

0,39,67 23,66 25,66 0,60,33

Data

Uniform Loads

Load Case G6


Membe	Size [kN/m]	Direction	Location	Ey Begin [m]	Ey End [m]	Ez Begin [m]	Ez End [m]
> 3	-12,3	Global Z	Length	0	0	0	0

Current load case can be set in the list **Load cases**. Uniform loads already defined in this load case are displayed in the table.

Click  above the table to add new uniform load to the current load case.

Table **Uniform Loads** contains following columns:

- **Member** – input the number of member loaded by uniform load.
- **Size** – input the value of uniform load.
- **Direction** - select the axis of coordinate system in which distributed load is defined. Following directions are available:
 - **Global Z** – the load acts in the direction of Z-axis of global coordinate system.
 - **Global Y** – the load acts in the direction of Y-axis of global coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
 - **Global X** – the load acts in the direction of X-axis of global coordinate system.
 - **Local z** – the load acts in the direction of z-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
 - **Local y** – the load acts in the direction of y-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.

- **Local x** – the load acts in the direction of x-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Location** – select the location mode of the load. This option is available only for **Straight or polygonal beam loaded in 3D** beam type. Following modes are available:
 - **Length** – the load is placed on the whole length of the member.
 - **Projection** – the load is placed on the projection of member on the corresponding axis. This mode can be used only for loads acting in global coordinate system.
- **Angle** – input the value of distributed load inclination to specified direction. This option is available only for **Straight beam loaded in vertical plane** beam type.
- **Ey - begin** – input of load eccentricity in the y-axis direction of member local coordinate system at the beginning of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ey – end** – input of load eccentricity in the y-axis direction of member local coordinate system at the end of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ez – begin** – input of load eccentricity in the z-axis direction of member local coordinate system at the beginning of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ez – end** – input of load eccentricity in the z-axis direction of member local coordinate system at the end of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
-  - delete the appropriate uniform load.

6.5.7 Line load

Click navigator command **Loads > Line Loads** to input line loads.

Line load can be defined on any member of the beam. The load can act in direction of selected axis of local or global coordinate system.. Also inclination can be defined. Load acts from position X1 to position X2 and can have different values in points X1 and X2.

Positive value defines the force, which acts in the positive direction of axis.

Loads direction and boundary conditions in uncoiled view may not correspond the real directions in 3D

Load Case G6, Line Loads

Data

Line Loads

Load Case **G6**


Memt	Size p1 [kN/m]	Size p2 [kN/m]	X1 [m]	X2 [m]	Direction	Location	Ey Begin [m]	Ey End [m]	Ez Begin [m]	Ez End [m]	
3	0	-2,36	0	11,88	Global Z	Length	0	0	0	0	
3	-2,36	0	11,88	23,66	Global Z	Length	0	0	0	0	

Current load case can be set in the list **Load cases**. Line loads already defined in this load case are displayed in the table.

Click above the table to add new line load to current load case.

Table **Line loads** contains following columns:

- **Member** – input the number of member loaded by uniform load.
- **Size p1** – input the value of line load at the beginning (in position X1)
- **Size p2** – input the value of line load at the end (in position X2)
- **X1** – the position of the beginning of the line load from the beginning of the member.
- **X2** – the position of the end of the line load from the beginning of the member.
- **Direction** - select the axis of coordinate system in which distributed load is defined. Following directions are available:
 - **Global Z** – the load acts in the direction of Z-axis of global coordinate system.
 - **Global Y** – the load acts in the direction of Y-axis of global coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
 - **Global X** – the load acts in the direction of X-axis of global coordinate system.
 - **Local z** – the load acts in the direction of z-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.

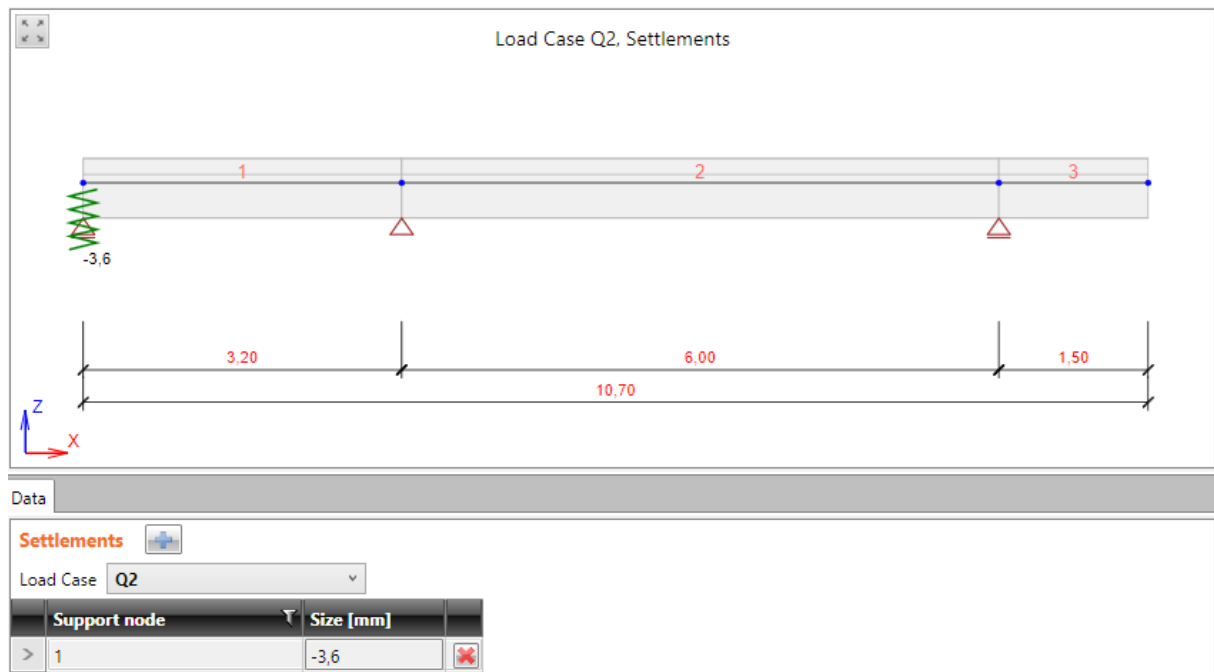
- **Local y** – the load acts in the direction of y-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Local x** – the load acts in the direction of x-axis of the member local coordinate system. This direction is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Location** – select the location mode of the load. This option is available only for **Straight or polygonal beam loaded in 3D** beam type. Following modes are available:
 - **Length** – the load is placed on the whole length of the member.
 - **Projection** – the load is placed on the projection of member on the corresponding axis. This mode can be used only for loads acting in global coordinate system.
- **Angle** – input the value of line load inclination to specified direction. This option is available only for Straight beam loaded in vertical plane beam type.
- **Ey - begin** – input of load eccentricity in the y-axis direction of member local coordinate system at the beginning of member. This column is available only for Straight or polygonal beam loaded in 3D beam type.
- **Ey – end** – input of load eccentricity in the y-axis direction of member local coordinate system at the end of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ez – begin** – input of load eccentricity in the z-axis direction of member local coordinate system at the beginning of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Ez – end** – input of load eccentricity in the z-axis direction of member local coordinate system at the end of member. This column is available only for **Straight or polygonal beam loaded in 3D** beam type.
-  - delete the appropriate line load.

6.5.8 Settlements of supports

To input settlements of supports click navigator command **Loads > Settlements**.

Settlement can be defined in any support. Settlement can act only in global Z-axis direction.

Positive value defines the settlement acting in the positive direction of Z axis.




The screenshot displays the 'Load Case Q2, Settlements' window. It shows a beam model with three segments labeled 1, 2, and 3. Segment 1 has a length of 3,20, segment 2 has a length of 6,00, and segment 3 has a length of 1,50. The total length of the beam is 10,70. A settlement of -3,6 is applied at support node 1. The data table below the beam model is as follows:

Support node	Size [mm]
1	-3,6

Current load case can be set in the list **Load cases**. Settlements already defined in the current load case are displayed in the table.

Click  above the table to add new settlement to the current load case.

Table **Settlements** contains following columns:

- **Supported node** – input the number of node with support, in which the settlement acts.
- **Size** – input the value of settlement.
-  - delete the appropriate settlement.

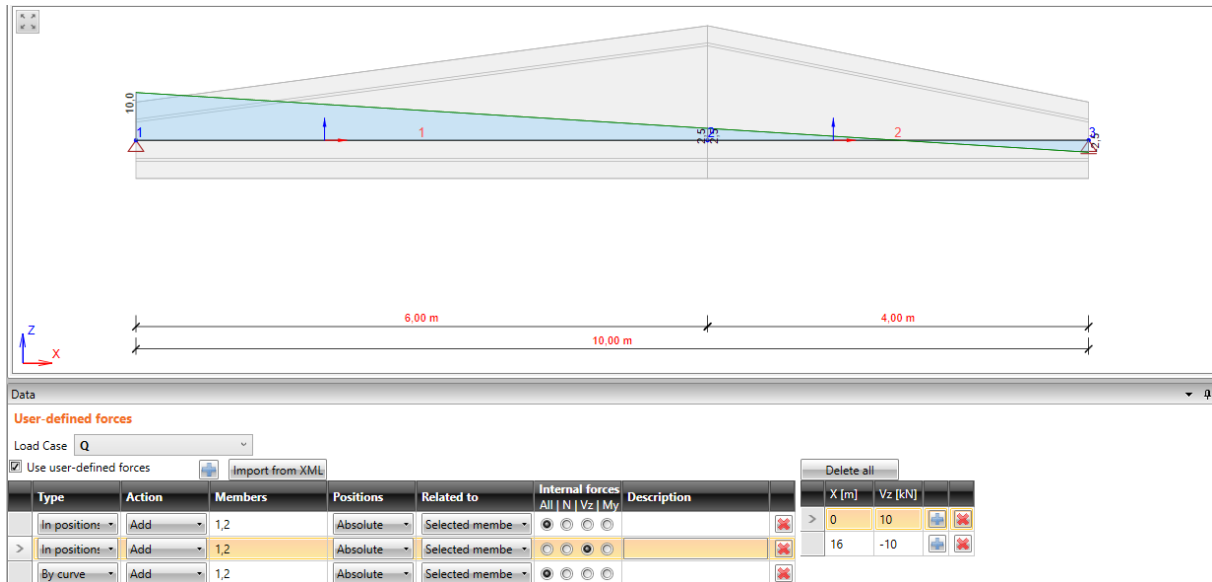
6.5.9 User defined internal forces

To input user defined internal forces click navigator command **Loads > User defined forces**.

Ribbon groups **User defined internal forces** and **Preference** are available.

Values of user defined internal forces are not analysed by FEM solver, but are transformed directly to internal forces in appropriate load cases on appropriate members and then are processed in the same way as internal forces calculated by FEM solver.

User defined internal forces can be defined manually or can be imported from XML file exported from SCIA Engineer.



The screenshot displays a beam model with a peak load of 10.0. The beam is divided into three segments: 6.00 m, 10.00 m, and 4.00 m. The software interface shows a table of user-defined internal forces for the current load case 'Q'.

User-defined forces							Delete all	
Load Case Q							X [m]	Vz [kN]
Type	Action	Members	Positions	Related to	Internal forces	Description		
In positions	Add	1,2	Absolute	Selected membe	All N Vz My		> 0	10
In positions	Add	1,2	Absolute	Selected membe	All N Vz My		16	-10
By curve	Add	1,2	Absolute	Selected membe	All N Vz My			


Current load case can be set in the list **Load cases**. User defined internal forces already defined in the current load case are displayed in the table.

Click  above the table to add new user defined internal force to the current load case.

If the check box **Use user defined forces** is not selected, the user defined forces of appropriate load case are not taken into account during the calculation event though they are defined.





Columns in table **User defined internal forces**:

- **Type** – select mode of user defined force input:
 - **In position** – user defined forces are defined by values of internal force components in specified positions. The values of user defined forces are interpolated linearly between specified positions. Positions and corresponding force values are defined in the next table.
 - **By curve** - user defined forces are defined by curve between specified positions. Course curves are defined in the next table.
- **Action** – select the interaction between user defined internal forces and calculated internal forces caused by other loads in the same load case:
 - **Add** – user defined internal forces are added to calculated internal forces from other loads in the appropriate load case.
 - **Replace** – user defined internal forces replace the calculated internal forces.
- **Members** – input list of numbers of consecutive members to apply the user defined internal forces on.

- **Position** – select evaluation mode of specified positions of user defined forces:
 - **Absolute** – position coordinates are specified in absolute values from the beginning of first selected member or the beginning of whole beam.
 - **Relative** – coordinates are specified relative to the total length of selected members or the whole beam.
- **Related to** – select origin to which the positions of user defined forces are related:
 - **Selected members** – values of positions of user defined forces are related to member (s) defined in column **Members**.
 - **Beam** – values of positions of user defined forces are related to the whole beam.
- **Internal forces** – select components of internal force to be defined:
 - **All** – values in positions or curves of all internal forces components can be defined in the next table.
 - **N** – values in positions or curves of normal force N can be defined in the next table.
 - **Vy** – values in positions or curves of shear force Vy can be defined in the next table. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
 - **Vz** – values in positions or curves of shear force Vz can be defined in the next table.
 - **My** – values in positions or curves of bending moment My can be defined in the next table.
 - **Mz** – values in positions or curves of bending moment Mz can be defined in the next table. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
 - **Mx** – values in positions or curves of torsional moment Mx can be defined in the next table. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
-  - delete the appropriate user defined force.

6.5.9.1 User defined forces in positions



To input the user defined forces in positions, table of values in positions is displayed for the current row of **User-defined forces** table.

Smazat vše					
	X [m]	N [kN]	Vz [kN]	My [kNm]	
>	0	0	0	20	 
	3	0	0	0	 

Click **Delete all** to remove all rows of values in positions.

Columns in table:

- **X** – input position to define values of internal forces. According to the setting in **Position** column the value is evaluated as absolute or relative and according to setting in **Related to** column is related to the whole beam or to the beginning of first selected member.
- **N** – input value of normal force N in appropriate position.
- **Vy** – input value of shear force Vy in appropriate position. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Vz** – input value of shear force Vz in appropriate position.

- **My** – input value of bending moment M_y in appropriate position.
- **Mz** – input value of bending moment M_z in appropriate position. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Mx** – input value of bending moment M_x in appropriate position. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
-  - add new position for input of user defined internal forces.
-  - delete existing position with values of user defined internal forces.

6.5.9.2 User defined internal forces by curves

To input the user defined forces by curves, table of curves definition is displayed for the current row of **User-defined forces** table.

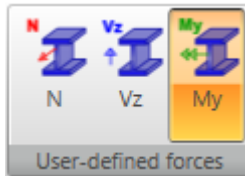
	N [kN]	Vz [kN]	My [kNm]
Type of curve	Not set ▾	Not set ▾	Parabolic ▾
Beg [m]			0
Vertex [m]			6
End [m]			12
Value - begin			50
Value - vertex			70
Value - end			50

The type of curve to define the course of internal force component is selected in row **Type of curve**:

- **Not set** – course of internal force component is not defined.
- **Constant** – the course of internal force component is constant and is defined by following values:
 - **Begin** – input position of beginning point of curve.
 - **End** – input position of end point of curve.
 - **Value – begin** – input magnitude of internal force component, which is constant between beginning and end point of curve.
- **Linear** – the course of internal force component is defined by line segment between two points and is defined by following values:
 - **Begin** – input position of beginning point of curve.
 - **End** – input position of end point of curve.
 - **Value – begin** – input magnitude of internal force component in the beginning point of curve.
 - **Value – end** – input magnitude of internal force component in the end point of curve.
- **Parabolic** – the course of internal force component is defined by parabola and is defined by following values:
 - **Begin** – input position of beginning point of parabola.
 - **Vertex** – calculated position of vertex of parabola.
 - **End** - input position of end point of curve.
 - **Value – begin** – input magnitude of internal force component in the beginning point of parabola.

- **Value – vertex** – input magnitude of internal force component in the vertex point of parabola.
- **Value – end** – input magnitude of internal force component in the end point of parabola.

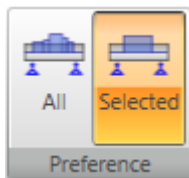
6.5.9.3 Ribbon group User defined forces



Drawing of existing user defined internal force component is set by commands in this group:

- **N** – switch to draw user defined normal force N.
- **Vz** – switch to draw user defined shear force Vz.
- **My** – switch to draw user defined bending moment My.

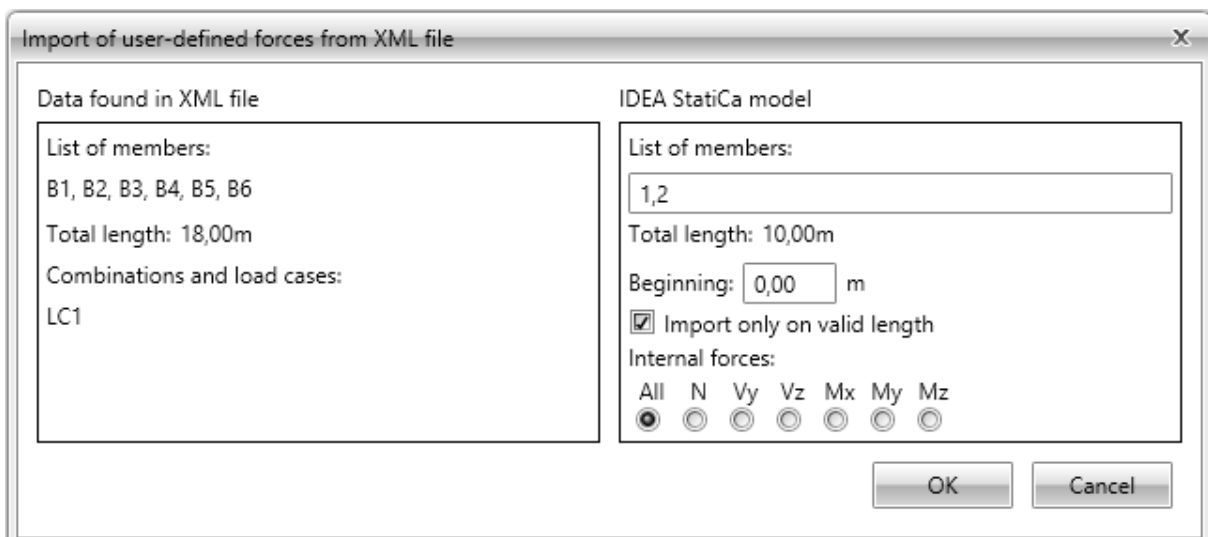
6.5.9.4 Ribbon group Preference



Mode of drawing of user defined forces is set by commands in this group:

- **All** – switch to draw sum of the current user defined force component, taking into account all rows in the **User defined internal forces table**.
- **Selected** – switch to draw the current user defined force component according to selected row in the **User defined internal forces table**.

6.5.9.5 Import of internal forces from XML file



User defined internal forces can be imported from XML file created by SCIA Engineer program.

Members, sections on members and internal forces in sections are read from XML file. Single members are lined up and the resulting sequence of positions with internal forces is applied on

selected members in IDEA model. The shift of imported sequence related to the beginning of first selected member in IDEA model can be specified.

To start the import, click **Import from XML** above the table **User-defined internal forces**. After choosing the imported file an import options dialog appears:

Group **Data found in XML file**:

- **List of members** – list of names of members from XML file is displayed.
- **Total length** – displays the overall length of lined up members from XML file..
- **Combinations and load cases:** - list of load cases and combinations from XML file is displayed. The appropriate load case is created in IDEA Beam model for each imported load case and combination.

Group **IDEA StatiCa model**:

- **List of members** – input list of consecutive members in IDEA model to apply the imported internal forces on.
- **Total length** – the overall length of selected members is displayed.
- **Beginning** – input the offset between beginning point of imported internal forces and beginning of the first selected member in IDEA StatiCa.
- **Import only on valid length** – if the option is selected, only positions in range from defined beginning to the end point of last selected member in IDEA model are imported. Positions out of this range are excluded from import.
- **Internal forces** – select components of internal forces to be imported.

6.5.10 Combinations of load cases

To input combinations click navigator command **Loads > Combinations**.

The structure and loads corresponding to the current combination are drawn in the main window.

Tables to create the combinations or modify the content of combination are drawn in the data window.

Combinations of load cases are important for the determination of extreme effects of loads. It is possible to input different types of combinations, which are used for different designs and checks.

One of following types can be assigned to each combination:

- ULS Fundamental;
- ULS Accidental;
- ULS Fatigue
- SLS Characteristic;
- SLS Quasi-permanent;
- SLS Frequent.



One of following evaluation types can be set for the combination:









- **Linear** – all load cases in the combination are simply added up, taking into account defined load case coefficients.
- **Envelope** – such combinations are searched, which cause maximal and minimal values of evaluated magnitude. Defined load case coefficients are taken into account.
- **Code** – combinations behave similarly to the envelope combinations, but the load coefficients are generated using the values from the national code. According to Eurocode following formulas are used:
 - For **ULS Fundamental combinations** formulas 6.10 or 6.10a,b;
 - For **SLS Characteristic combination** formula 6.14b;
 - For **SLS Frequent combination** formula 6.15b;
 - For **SLS Quasi-permanent combination** formula 6.16b;
 - For **ULS Accidental combination** formula 6.11b;
 - For **ULS Fatigue combination** formula 6.69 EN 1992-1-1.

The automatically determined load coefficients are multiplied by user defined load coefficients.

The content (load cases and the appropriate coefficients) of critical combinations, which were generated from code or envelope combination prescription and caused an extreme of evaluated magnitude, are printed in results evaluation.

6.5.10.1 Load combinations for non-staged beams

User defined combinations  

	Name	Type	Evaluation			Description
>	ULS Fundame	ULS Fundam:	Code (6.10)			1,0*SW + 1,0*G + 1,0*Q
	SLSC	SLS Char	Code			1,0*SW + 1,0*G + 1,0*Q
	SLSF	SLS Frequent	Code			1,0*SW + 1,0*G + 1,0*Q
	SLSQ	SLS Quasi	Code			1,0*SW + 1,0*G + 1,0*Q




Click  above the table of combinations to input new combination.

Table **Combinations** contains following columns:

- **Name** – input the name of combination.
- **Type** – select the type of current combination.
- **Evaluation** – select the evaluation mode of combination.
-  - launch the **Combinations manager** to modify the combination rules – see **Manager of load cases combinations**.
-  - delete the appropriate combination.

Load cases are displayed in following columns. Specific value of coefficient for each load case in combination can be defined.

If the combination evaluation mode is set to code, the automatically determined load coefficients are multiplied by user defined load coefficients.

Click **Delete all** above the combinations table to delete all combinations.

6.5.10.2 Load combinations for staged beams

A set of combinations (ULS fundamental, SLS Characteristic, SLS Frequent, SLS Quasi-permanent) is generated automatically per each stage after creating new project using wizard. Appropriate load cases, which act in individual stages, are automatically assigned to combinations.

In addition to generated combinations, user defined combinations can be also entered.

The combination for construction stages can be regenerated, if necessary.

The combination can be assigned to the required stage during the definition of stages – see

Construction stages.

Combinations for construction stages Edit Update combinations

▲ Evaluation of generated combinations

ULS Fundamental	Code (6.10) ▾
SLS Char	Linear ▾
SLS Frequent	Linear ▾
SLS Quasi	Linear ▾

User defined combinations + Delete all

Name ▾	Type	Evaluation	C. stage ▾	Description ▾
> All dead load:	ULS Fundam: ▾	Linear ▾		1,0*SW (1) + 1,0*G

Table for definition of generated combinations and table for user defined combinations are displayed in the data window.

Evaluation mode of combinations generated for construction stages can be set in the table

Combinations for construction stages:



- **ULS fundamental** – select the evaluation mode for generated combinations of type ULS fundamental.
- **SLS char** – select the evaluation mode for generated combinations of type SLS Characteristic.
- **SLS Frequent** – select the evaluation mode for generated combinations of type SLS Frequent.
- **SLS Quasi** – select the evaluation mode for generated combinations of type SLS Quasi-permanent.

Commands above the table:

- **Edit** – launch the **Combinations manager** to modify the combinations – see **Manager of load cases combinations**. Combinations in the manager are categorized by stages.
- **Update combinations** – updates the content of load cases combinations to correspond to the assignment of permanent load cases to individual construction stages. Combination can be assigned to the construction stage, only if contains all permanent load cases, which have been applied in selected and all previous construction stages..

Columns in **User defined combinations** table:

- **Name** – input name of combination.
- **Type** – select type of combination.

- **Construction stage** – displays number of construction stage, to which is the combination assigned.
- **Evaluation** – select evaluation mode of combination rule.
-  - launch the **Combinations manager** to modify the combinations – see Manager of load cases combinations. Combinations in the manager are categorized by types.
-  - delete appropriate combination.
- **Description** – displays the content of combination rule.

If the combination evaluation mode is set to code, the automatically determined load coefficients are multiplied by user defined load coefficients.

User defined combination of type Linear or Envelope can contain also other combination of type Linear or Envelope.

In particular combination, load case can be assigned to different load group than is the default load group of the load case. This assignment is valid only for the appropriate combination.

Click **Delete all** above the table to delete all user defined combinations.

6.5.11 Manager of load cases combinations

The **Load combinations manager** is launched by commands to modify the combinations.



Combinations of load cases are important to determine the extremes of load actions. It is possible to input several types of combinations, which can be used in appropriate checks.

One of following types can be assigned to each combination:

- Ultimate limit state;
- Serviceability limit state – characteristic;
- Serviceability limit state – quasi-permanent;
- Serviceability limit state – frequent;
- Ultimate limit state – fatigue;
- Ultimate limit state – accidental.

Following evaluation mode can be set for the combination:

- **Linear** – all load cases in the combination are simply added up, taking into account defined load case coefficients.
- **Envelope** – such combinations are searched, which cause maximal and minimal values of evaluated magnitude. Defined load case coefficients are taken into account
- **Code** – combinations behave similarly to the envelope combinations, but the load coefficients are generated using the values from the national code. According to Eurocode following formulas are used:

- For **ULS combinations** formulas 6.10 or 6.10a,b;
- For **SLS Characteristic** combination formula 6.14b;
- For **SLS Frequent** combination formula 6.15b;
- For **SLS Quasi-permanent** combination formula 6.16b;
- For **ULS Accidental** combination formula 6.11b;
- For **ULS Fatigue combination** formula 6.69 EN 1992-1-1.




The automatically determined load coefficients are multiplied by user defined load coefficients.


The content (load cases and the appropriate coefficients) of critical combinations, which were generated from code or envelope combination prescription and caused an extreme of evaluated magnitude, are printed in results evaluation.

Options of **Combinations manager dialog**:

- **Combination** – defined combinations, grouped according to their types, are displayed in the tree view. Properties of selected combination and list of load cases in the combination are displayed in the middle part of dialog.
- **New** – add new combination of load cases.
- **Delete** – delete selected combination of load cases.
- **Expand all items** – expand/collapse all items in the combinations tree view.

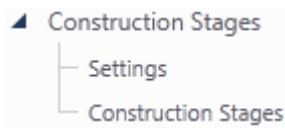
Combination properties:

- **Name** – input name of the current combination.
- **Evaluation** – select the evaluation mode of the current combination.
- **Type** – select the type of the current combination.
- **Stage** – construction stage, to which is the current combination assigned.
- **Load cases in combination** – load cases or combinations assigned to the current combination are displayed in the tree view. Load cases are grouped according to the load groups. The value of load case coefficient can be defined in the **Coeff.** column. If the coefficient value changes in the row with load group name, the new value of coefficient is assigned to all load cases in the affected group.
Remark – the defined load case coefficients are multiplied by automatically determined coefficients for combinations evaluated by national code.
- **Expand all items** – expand/collapse all items in the load cases tree view.
- **List of load groups** – assign other than default load group to selected load cases. The change of load group is valid only for the particular combination. Selected load cases can be moved to the different load group using mouse.
-  - remove selected load case, load cases group or combination from the current combination.
-  - remove all load cases, load groups or combinations from the current combination.
-  - add selected load case or load group, selected in the tree view **Load cases in project** on the tab **Load cases**, or combinations selected in the tree view **Combinations in project** on the tab **Combinations**, to the current combination.

-  - add all load cases from the tree view **Load cases in project** to the current combination.

7 Construction stages

To set general properties of construction stages and to define the individual construction stages use navigator commands **Construction stages**.



7.1 General settings of construction stages

Click navigator command **Construction Stages > Settings** to set common properties of construction stages. Table of common properties is displayed in the data window.

Construction stages settings	
End of curing [d]	7
Use γ_{lt}	<input type="checkbox"/>
Relative humidity [-]	0,65
Maximum length of subzone [m]	1,00
Number of intervals	10
Calculation of non-linear creep	<input type="checkbox"/>
Don't exclude tendons	<input type="checkbox"/>

- **End of curing** – the age of concrete at the beginning of drying shrinkage (or swelling).
- **Use γ_{lt}** – switch on/off taking into account the safety factor for long-term extrapolation of delayed strains in creep calculation.
- **Relative humidity** – input the value of relative humidity.
- **Maximal length of subzone** – input the length of finite element for TDA calculation.
- **Number of intervals** – input the number of generated time nodes per decade.
- **Calculation of non-linear creep** – switch on/off calculation of nonlinear creep if the compressive concrete stress at age t_0 exceeds the limit value.
- **Don't exclude tendons** – switch on/off excluding tendons from the calculation model of reinforced cross-section in cases, when the tendon is positioned out of the cross-section.

7.2 History of members

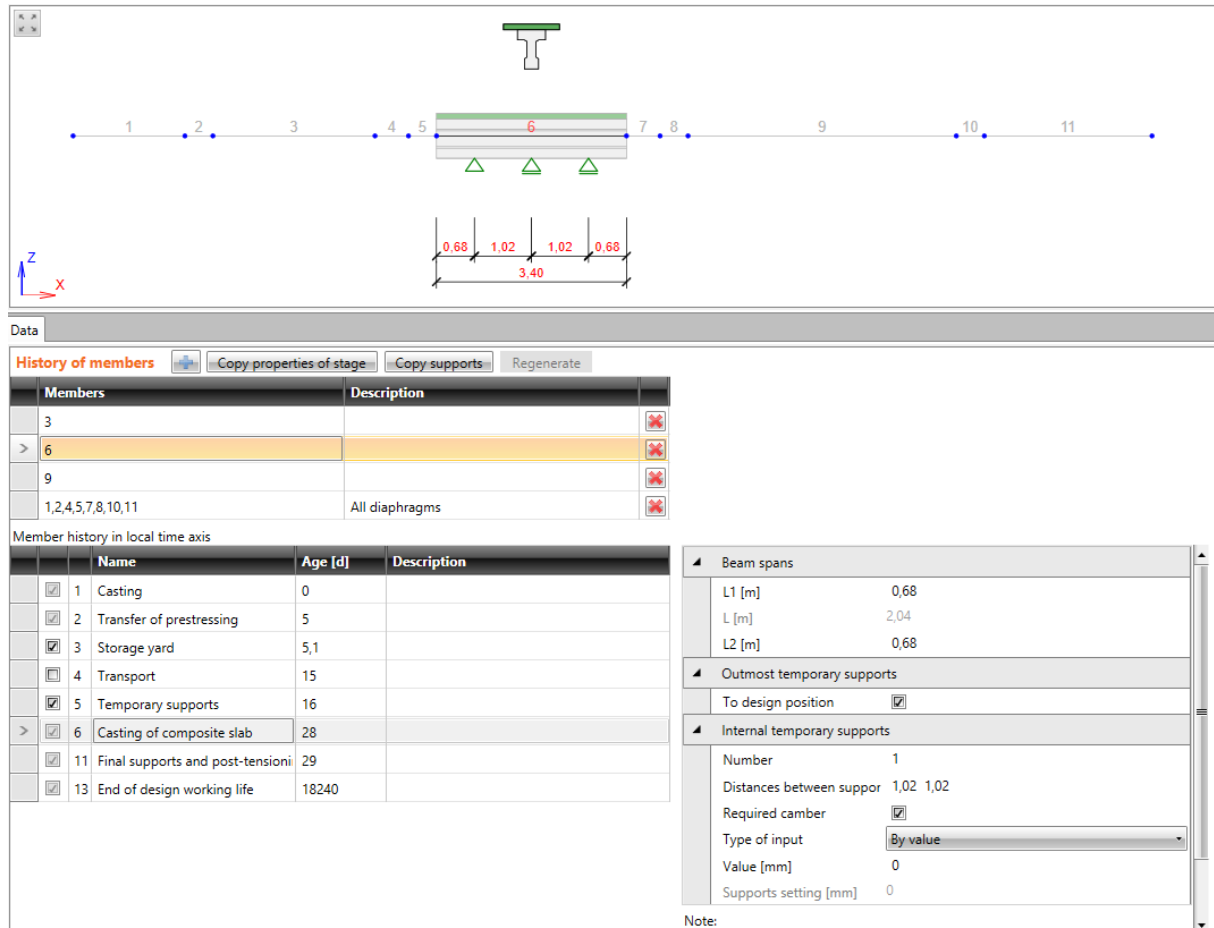
For beam composed of prefabricated beams made continuous by composite slab, parameters of construction stages of individual structural members can be defined.

To modify the history of individual structural members click navigator command **Construction stages > Members history**.

Status of the current item in the current stage or the time history graph is drawn in the main window.

Tables to define the members history are displayed in the data window.

Ribbon group **Construction stages** is displayed.



History of members

Members	Description
3	
> 6	
9	
1,2,4,5,7,8,10,11	All diaphragms

Member history in local time axis

	Name	Age [d]	Description
<input checked="" type="checkbox"/>	1 Casting	0	
<input checked="" type="checkbox"/>	2 Transfer of prestressing	5	
<input checked="" type="checkbox"/>	3 Storage yard	5,1	
<input type="checkbox"/>	4 Transport	15	
<input checked="" type="checkbox"/>	5 Temporary supports	16	
>	6 Casting of composite slab	28	
<input checked="" type="checkbox"/>	11 Final supports and post-tensioning	29	
<input checked="" type="checkbox"/>	13 End of design working life	18240	

▲ Beam spans

L1 [m]	0,68
L [m]	2,04
L2 [m]	0,68

▲ Outmost temporary supports

To design position


▲ Internal temporary supports

Number	1
Distances between support	1,02 1,02
Required camber	<input checked="" type="checkbox"/>
Type of input	By value
Value [mm]	0
Supports setting [mm]	0


Note:

Local time axes for members are defined in the table **Members history**.

Commands above the table:

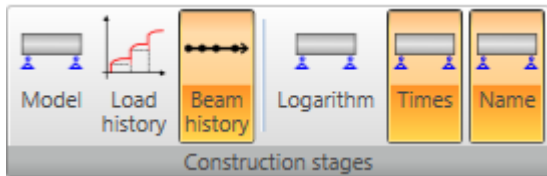
-  - add new local time axis to the table.
- Copy properties of stage** – copies stage status and equivalent concrete age from the current stage of current member to corresponding stages of other members (except diaphragms).
- Copy supports** – copies settings of temporary supports from the current stage of the current member to corresponding stages of other members. The ratio of positions is used for members of different length.
- Regenerate** – regenerates all time axes.

Columns in the table:

- **Members** – input number or sequence of numbers of members, which are assigned to the local time axis. Members on the time axis must be of the same cross-section, prefabricated members must be of the same length also.
- **Description** – input description of the local time axis.
-  - delete the appropriate local time axis.

The construction stages properties for the current member can be modified in the table **Member history in local time axis**. The table of stage properties is displayed for the current stage. Stages settings are described in following chapters.

7.2.1 Ribbon group Construction stages



Commands in ribbon group are used to set the drawing mode in the main window:

- **Model** – switch to draw the structural model. Current stage of current member is drawn.
- **Load history** – switches to draw the history of load application in construction stages. History of load application shows, which load cases contribute to structural response in individual construction stages, and how the loads combinations are composed with respect to construction stages.
- **Beam history** – switch to draw time axes of construction stages.
- **Logarithm** – switch on/off drawing of time axes in logarithmic scale.
- **Times** – switch on/off drawing of times at the time axes nodes.
- **Name** – switch on/off drawing of names of construction stages at the time axes nodes.

7.3 Construction stages

Construction stages are generated automatically according to the type of analysed beam.

For composite continuous beam composed of prefabricated beams made continuous by composite slab, the global time axis of construction stages is generated from the local time axes of individual structural members.

Other stages (service stages) can be added manually. To modify construction stages click navigator command **Construction stages > Construction stages**.

The structure is displayed in the Main window. Supports are drawn for the current stage.

Table of construction stages and table of properties of the current stage are displayed in the Data window.

Ribbon group **Construction stages** is available.

The screenshot shows a beam model with 11 construction stages. The stages are represented by colored segments along the beam: 1 (green), 2 (blue), 3 (red), 4 (green), 5 (blue), 6 (red), 7 (green), 8 (blue), 9 (red), 10 (green), and 11 (blue). Dimensions are shown below the beam, and a coordinate system (X, Y, Z) is visible in the bottom left.

Construction and service stages	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting M(3,6,9)	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Transfer of prestressing M(3,6,9)	5	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	3 Storage yard M(3,6,9)	5,1	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	4 Transport M(3,6,9)	25	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	5 Temporary supports M(3,6,9)	26	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	6 Casting of composite slab M(3,6,9)	28	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	7 Final supports	35	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	8 Superimposed dead load	60	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	9 End of design working life	18250	<input checked="" type="checkbox"/>	

Note:
Select particular table row in order to edit construction stage.

Load cases and combinations:
Load cases: R (6), SWS (6)
Combinations: ULS Fundamental ST(6), SLSC ST(6), SLSF ST(6)

Note:
The moment when fresh concrete of composite slab is cast. Only prefabricated part of the structure resists the load from fresh concrete. It may be temporarily supported by a number of internal supports to limit the range of bending moments caused by the load from fresh concrete.



Columns of **Construction stages** table:

- - switch on/off the stage to be calculated and checked. Only some stages can be switched off.
- **Number** – displays the serial number of construction stage. The number is used in names of generated load cases and combinations.
- **Name** – displays the name of construction stage.
- **Time** – input the time of the stage, related to the stage of casting of first part of the design member. The time cannot be less than the time of previous stage.
- **Check** – switch on/off taking the stage into account when performing checks of reinforced

cross-sections in IDEA RCS module.

- **Description** – input the additional description of the stage.

Commands above the **Construction stages** table:

-  - add new (service) stage to the list of construction stages.
-  - delete selected service stage.
- **Update combinations** – updates the content of load cases combinations to correspond to the assignment of permanent load cases to individual construction stages. Combination can be assigned to the construction stage, only if contains all permanent load cases, which have been applied in selected and all previous construction stages.
- **Generate combinations** – generates combinations, if missing in some construction stage, respecting the assignment of permanent load cases to individual construction stages.
- **Regenerate** – regenerates time nodes on the global time axis.

7.3.1 Ribbon group Construction stages

See **Ribbon group Construction stages**.



7.3.2 Load cases and combinations in construction stages

Load cases and combinations, which are assigned to the current stage, are displayed in the properties group Load cases and combinations.



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.

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

Load cases and combinations		
Load cases	R (9), G (9)	
Combinations	ULS Fundamental ST(9), SLSC ST(9), SLSF ST(9)	

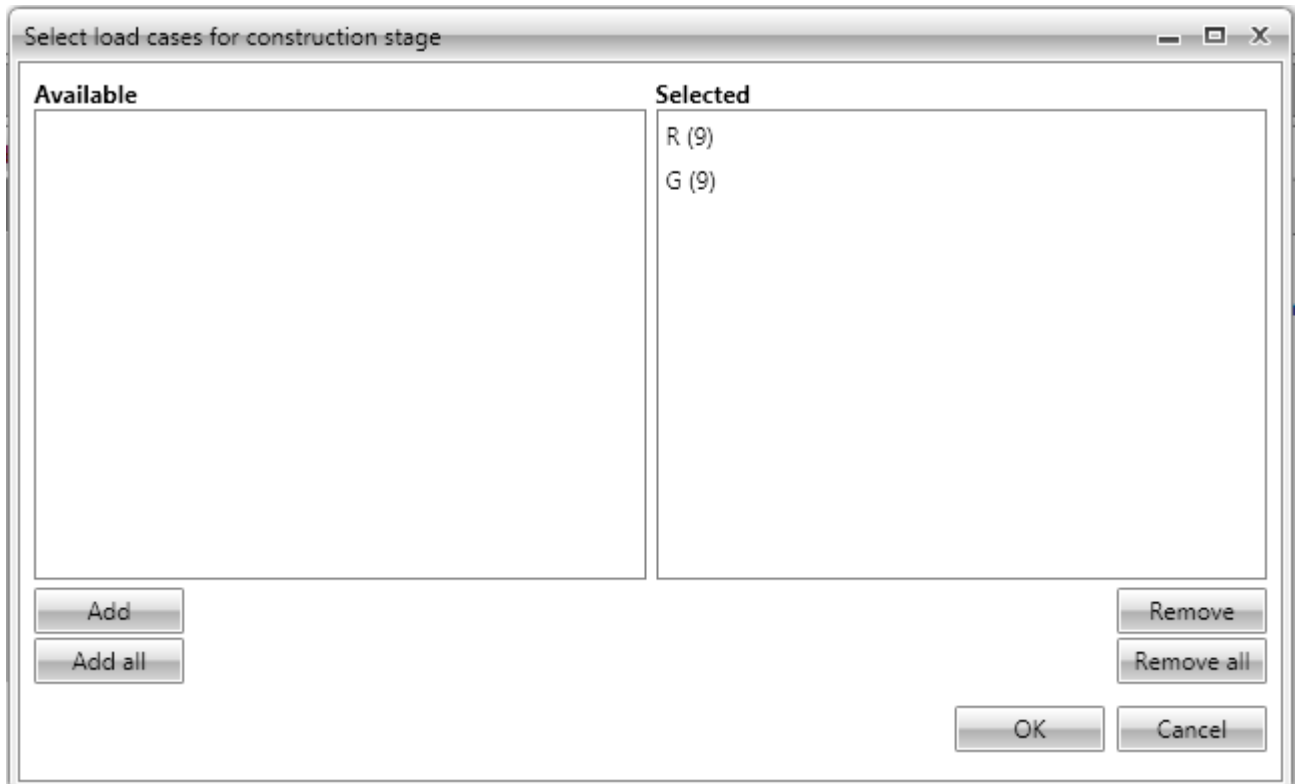
Properties in group **Load cases and combinations**:

- **Load cases** – displays the list of permanent load cases, applied in the current construction stage. To edit assignment of load cases to the stage click edit button .
- **Combinations** – displays the list of combinations assigned to the current construction stage. Those combinations are used for checks of sections in IDEA RCS module. Combinations can be assigned to construction stage either manually or automatically by clicking **Update combinations** above the **Construction stages** table. To assign combinations to construction stage manually click edit button  - see **Manual assignment of combinations to construction stages**.

If combination for some of construction stages is missing, the required combinations can be generated by clicking the **Generate combinations** button above the **Construction stages** table.

The selection of load cases applied in the current stage is performed in the dialog **Select load cases for construction stage**.

Load cases available for the current stage are listed in the column **Available**. Load cases applied in the current stage are listed in the column **Selected**.



7.3.3 Manual assignment of combinations to construction stages

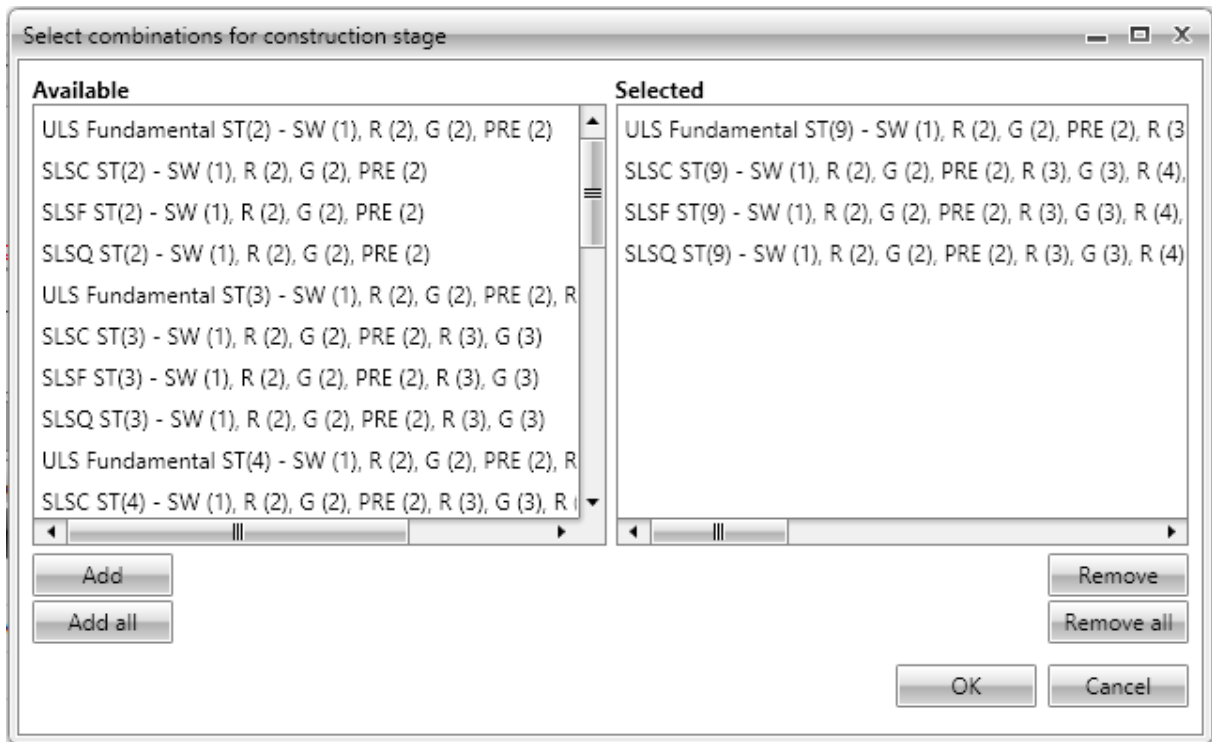
Following rule has to be fulfilled when assigning combination of load cases to construction stage:

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.

The selection of combinations assigned to the current stage is performed in the dialog **Select combinations for construction stage**.

All combinations are listed in the column **Available**. Combinations assigned to the current stage are listed in the column **Selected**.



7.3.4 Temporary supports of prefabricated beams

Properties of outmost temporary supports and inner temporary supports can be defined for some construction stages (storage, transport...) of prefabricated beam. The settings of temporary supports can be different in each stage, which enables to define temporary supports. Properties of temporary supports can be defined in property groups **Outmost temporary supports** and **Internal temporary supports**.

▲ Outmost temporary supports	
To design position	<input type="checkbox"/>
Local system	<input checked="" type="checkbox"/>
Flexible supports	<input checked="" type="checkbox"/>
Ky - left support [MN/m]	10000000,0
Kz - left support [MN/m]	10000000,0
Ky - right support [MN/m]	10000000,0
Kz - right support [MN/m]	10000000,0
▲ Internal temporary supports	
Number	3
Distances between suppor	0,91 0,91 0,91 0,91
Flexible supports	<input type="checkbox"/>
Required camber	<input checked="" type="checkbox"/>
Type of input	By value
Value [mm]	0
Supports setting [mm]	0 0 0

Properties group **Outmost temporary supports** – definition of temporary supports:

- **To design position** - if the option is selected, the supports emerging in this stage are placed to the design position. Thus the deflections, which appeared in positions of new supports during previous stages, will be eliminated.
- **Local system** – switches on/off rotation of temporary supports to the direction, in which the supports will act in construction stage.
- **Flexible supports** – if the option is selected (option **To design position** must be unselected), the stiffness of temporary supports emerging it this stage can be defined. For flexible supports, the value of stiffness $1e7$ MN/m means the rigid support in the appropriate direction.
 - **Ky – left support** - input of translational stiffness of left temporary support in the y-axis direction of support coordinate system.
 - **Kz – left support** - input of translational stiffness of left temporary support in the z-axis direction of support coordinate system.
 - **Ky – right support** - input of translational stiffness of right temporary support in the y-axis direction of support coordinate system.
 - **Kz – right support** - input of translational stiffness of right temporary support in the z-axis direction of support coordinate system.

Properties group **Internal temporary supports** – definition of internal temporary supports:

- **Number** - number of intermediate supports, which are generated in the mid span.
- **Distances between supports** – sequence of distances between supports. Individual distances may differ, but the total sum must match the length of the mid span.
- **Flexible supports** – if the option is selected (option To design position must be unselected), the stiffness of temporary internal supports emerging at this stage can be defined. For flexible supports, the value of stiffness $1e7$ MN/m means the rigid support in the appropriate direction.
- **Kz** – left support - input of sequence of translational stiffness of individual temporary internal supports in the z-axis direction of support coordinate system.
- **Required camber** – if the option is selected, the value of camber of the mid span by means of internal supports can be defined.
- **Type of input** – camber value determination mode:
 - **By fraction of span** – the camber value is defined as the n-th of mid span length entered in the property 1/.
 - **By value** – the required camber value in the mid of the mid span is defined in the property Value.
- **Supports setting** – displays list of calculated camber per each individual internal support in the mid span.

7.3.5 Cast-in-situ post-tensioned beam

Following default construction stages are generated for cast-in-situ post-tensioned beam:

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
	<input type="checkbox"/>	3 Construction stage 1	10	<input type="checkbox"/>	
	<input type="checkbox"/>	4 Construction stage 2	35	<input type="checkbox"/>	
	<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 End of design working life	36500	<input checked="" type="checkbox"/>	

- Casting
- Post-tensioning
- Construction stage 1 – additional construction stage. This stage may be switched off from calculation.
- Construction stage 2 – additional construction stage. This stage may be switched off from calculation.
- Superimposed dead load. This stage may be switched off from calculation.
- End of design working life.

7.3.5.1 Casting

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Casting of composite slab	28	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 End of design working life	18250	<input checked="" type="checkbox"/>	

Load cases

Load cases SW (1)

Note: Casting of fresh concrete of design member. The age of concrete in following

Note: Select particular table row in order to edit construction stage.

This is the default first stage for each beam type. The fresh concrete of first part of the design member is cast in this stage. Times of all following stages are related to this stage.

Time of this stage cannot be changed.

Check cannot be performed for this stage.

For some beam types, load case of self-weight of the first phase of cross-section is assigned to the stage.

Neither other load cases, nor combinations can be assigned to the stage.

7.3.5.2 Post-tensioning

		Name	Time [d]	Check	Description
	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
>	<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
	<input type="checkbox"/>	3 Construction stage 1	10	<input type="checkbox"/>	
	<input type="checkbox"/>	4 Construction stage 2	35	<input type="checkbox"/>	
	<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>	

In-plane restraint

In-plane restraint

Load cases and combinations

Load cases SW (2), G (2), POST (2)

Combinations ULSF ST(2), SLSC ST(2), SLSF ST(2), SLSQ ST(2)

The post-tensioned tendons are tensioned and anchorage permanently in this stage. Post-tensioned reinforcement is a part of the structure since this time and resists all loads.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Load case of increments of permanent loads, load case for transfer of prestressing effects of post-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

For some beam types, load case of self-weight of the first phase of cross-section is assigned to this stage.

Corresponding combinations are assigned to this stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.5.3 Additional construction stage

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
> <input checked="" type="checkbox"/>	3 Construction stage 1	10	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	4 Construction stage 2	35	<input type="checkbox"/>	
<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	6 End of design working life	36500	<input checked="" type="checkbox"/>	

▲ Post-tensioning	
Post-tensioning	<input checked="" type="checkbox"/>
▲ Load cases and combinations	
Load cases	G (3), POST (3) 
Combinations	ULS Fundamental ST(3), SLSC ST(3), SLSF ST(3) 

Additional user defined construction stage. Additional prestressing by post-tensioned tendons can be applied in this stage

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Properties group **Post-tensioning**:

- **Post-tensioning** – if the option is selected, post-tensioned tendons can be defined for the stage to apply additional prestressing.

Load case of increments of permanent loads, load case for transfer of prestressing effects of post-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to this stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.5.4 Superimposed dead load

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	3 Construction stage 1	10	<input type="checkbox"/>	
<input type="checkbox"/>	4 Construction stage 2	35	<input type="checkbox"/>	
> <input checked="" type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>	

In-plane restraint	
In-plane restraint	<input type="checkbox"/>
Load cases and combinations	
Load cases	G (5)
Combinations	ULSF ST(5), SLSC ST(5), SLSF ST(5), SLSQ ST(5)

The final structural system already exists in this stage. All dead load should act since this stage.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Load case of increments of permanent loads in the stage and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.5.5 End of working life

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	3 Construction stage 1	10	<input type="checkbox"/>	
<input type="checkbox"/>	4 Construction stage 2	35	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
> <input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>	

In-plane restraint	
In-plane restraint	<input type="checkbox"/>
Load cases and combinations	
Load cases	G (6)
Combinations	ULSF ST(6), SLSC ST(6), SLSF ST(6), SLSQ ST(6)

Construction stage at the moment of the end of expected working life. All effects of long term losses and internal forces redistribution are taken into account.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Load case of increments of permanent loads in the stage and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.6 Cast-in-situ composite reinforced beam

		Name	Time [d]	Check	Description	
<input checked="" type="checkbox"/>	1	Casting	0	<input type="checkbox"/>		▲ In-plane restraint
						In-plane restraint <input type="checkbox"/>
>	<input checked="" type="checkbox"/>	2	Casting of composite slab	28	<input checked="" type="checkbox"/>	▲ Load cases and combinations
	<input type="checkbox"/>	3	Superimposed dead load	60	<input type="checkbox"/>	Load cases R (2), SWS (2)
	<input checked="" type="checkbox"/>	4	End of design working life	18250	<input checked="" type="checkbox"/>	Combinations ULSF ST(2), SLSC ST(2), SLSF ST(2), SLSQ ST(2)

Following default construction stages are generated for cast-in-situ composite reinforced beam:

- Casting.
- Casting of composite slab.
- Superimposed dead load. This stage may be switched off from calculation.
- End of design working life.

7.3.6.1 Casting

See [Casting](#).

7.3.6.2 Casting of composite slab

		Name	Time [d]	Check	Description	
<input checked="" type="checkbox"/>	1	Casting	0	<input type="checkbox"/>		▲ In-plane restraint
						In-plane restraint <input type="checkbox"/>
>	<input checked="" type="checkbox"/>	2	Casting of composite slab	28	<input checked="" type="checkbox"/>	▲ Load cases and combinations
	<input type="checkbox"/>	3	Superimposed dead load	60	<input type="checkbox"/>	Load cases R (2), SWS (2)
	<input checked="" type="checkbox"/>	4	End of design working life	18250	<input checked="" type="checkbox"/>	Combinations ULSF ST(2), SLSC ST(2), SLSF ST(2), SLSQ ST(2)

Fresh concrete of composite slab is cast in this construction stage. Only the prefabricated part of structure resists the load from fresh concrete.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- In-plane restraint – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Load case of own weight of composite slab and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in [Load cases and combinations in construction stages](#)).

7.3.6.3 Superimposed dead load

See [Superimposed dead load](#).

7.3.6.4 End of working life

See [End of working life](#).

7.3.7 Cast-in-situ composite post-tensioned beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
	<input type="checkbox"/>	3 Construction stage 1	10	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Casting of composite slab	28	<input checked="" type="checkbox"/>	
	<input type="checkbox"/>	5 Construction stage 2	35	<input type="checkbox"/>	
	<input type="checkbox"/>	6 Superimposed dead load	60	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 End of design working life	36500	<input checked="" type="checkbox"/>	

Following default construction stages are generated for cast-in-situ composite post-tensioned beam:

- Casting.
- Post-tensioning.
- Construction stage 1 – additional construction stage. This stage may be switched off from calculation.
- Casting of composite slab.
- Construction stage 1 – additional construction stage. This stage may be switched off from calculation.
- Superimposed dead load. This stage may be switched off from calculation.
- End of design working life.

7.3.7.1 Casting

See [Casting](#).

7.3.7.2 Post-tensioning

See [Post-tensioning](#).

7.3.7.3 Additional construction stage

See [Additional construction stage](#).

7.3.7.4 Casting of composite slab

See [Casting of composite slab](#).

7.3.7.5 Superimposed dead load

See [Superimposed dead load](#).

7.3.7.6 End of working life

See [End of working life](#).

7.3.8 Prefabricated reinforced beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Final supports	35	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>	

Following default construction stages are generated for prefabricated reinforced beam:

- Casting.
- Storage yard.
- Transport. This stage may be switched off from calculation.
- Final supports.
- Superimposed dead load. This stage may be switched off from calculation.
- End of working life.

7.3.8.1 Casting

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Final supports	35	<input checked="" type="checkbox"/>	
	<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 End of design working life	36500	<input checked="" type="checkbox"/>	

▲ Temporary supports	
Local system	<input type="checkbox"/>
▲ Load cases	
Load cases	SW (1) <input checked="" type="checkbox"/>

This is the default first stage for each beam type. The fresh concrete of first part of the design member is cast in this stage. Times of all following stages are related to this stage.

Time of this stage cannot be changed.

Check cannot be performed for this stage.

Properties group **Temporary supports**:

- **Local system** – switches on/off rotation of temporary supports to the direction, in which the supports will act in construction stage.

Load case of self-weight of the first phase of cross-section is assigned to the stage.

Neither other load cases, nor combinations can be assigned to the stage.

7.3.8.2 Storage yard

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Storage yard	5,1	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	3 Transport	25	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	4 Final supports	35	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>	

In-plane restraint	
In-plane restraint	<input type="checkbox"/>
Post-tensioning	
Post-tensioning	<input type="checkbox"/>
Concrete strength in current construction stage	
User-specified concrete strength	<input type="checkbox"/>
fck [MPa]	24,9
Beam spans	
L1 [m]	1,22
L [m]	3,65
L2 [m]	1,22
Outmost temporary supports	
To design position	<input type="checkbox"/>
Local system	<input type="checkbox"/>
Flexible supports	<input checked="" type="checkbox"/>
Ky - left support [MN/m]	10000000,0
Kz - left support [MN/m]	10000000,0
Ky - right support [MN/m]	10000000,0
Kz - right support [MN/m]	10000000,0
Internal temporary supports	
Number	0
Load cases and combinations	
Load cases	R (2), G (2)
Combinations	ULSF ST(2), SLSC ST(2), SLSF ST(2), SLSQ ST(2)

The production of prefabricated part of the design member is finished in this construction stage. The prefabricated element is transferred to a storage yard, where it can be piled one on top of another, separated using wooden or rubber stacking strips between individual layers. Additional prestressing using post-tensioned tendons can be applied in this stage.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Properties group **Post-tensioning**:

- **Post-tensioning** – if the option is selected, post-tensioned tendons can be defined for the stage to apply additional prestressing.

Properties group **Concrete strength in current construction stage** – in the first stage after casting, it enables to calculate the time of current construction stage from the specified concrete strength the and vice versa.

- **User-specified concrete strength** – if the option is selected, the value of concrete strength in the current construction stage can be specified. The time of stage is calculated from the specified value.
- **fck** – input of characteristic compressive cylinder concrete strength determined by tests.

Properties group **Beam spans** – definition of supports position on the storage yard:

- **L1** – input of length of left span
- **L** – displays the length of the mid span, calculated from the overall length of beam.
- **L2** – input of length of right span

Properties group **Outmost temporary supports and Internal temporary supports** – see **Temporary supports of prefabricated beams**.

Load case of increments of permanent loads in the stage, load case for transfer of effects of post-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

For some beam types, load case of self-weight of the first phase of cross-section is assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.8.3 Transport

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Storage yard	5,1	<input checked="" type="checkbox"/>	
> <input checked="" type="checkbox"/>	3 Transport	25	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	4 Final supports	35	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>	

In-plane restraint	
In-plane restraint	<input type="checkbox"/>
Beam spans	
L1 [m]	1,22
L [m]	3,65
L2 [m]	1,22
Outmost temporary supports	
To design position	<input type="checkbox"/>
Local system	<input type="checkbox"/>
Flexible supports	<input checked="" type="checkbox"/>
Ky - left support [MN/m]	10000000,0
Kz - left support [MN/m]	10000000,0
Ky - right support [MN/m]	10000000,0
Kz - right support [MN/m]	10000000,0
Internal temporary supports	
Number	3
Distances between support	0,91 0,91 0,91 0,91
Flexible supports	<input type="checkbox"/>
Required camber	<input checked="" type="checkbox"/>
Type of input	By value
Value [mm]	0
Supports setting [mm]	0 0 0
Load cases and combinations	
Load cases	R (3), G (3)
Combinations	ULSF ST(3), SLSC ST(3), SLSF ST(3), SLSQ ST(3)

The prefabricated element is transported from the storage yard to the building ground in this stage.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Properties group **Beam spans** – definition of supports position during the transport:

- **L1** – input of length of left span
- **L** – displays the length of the mid span, calculated from the overall length of beam.
- **L2** – input of length of right span

Properties group **Outmost temporary supports and Internal temporary supports** – see **Temporary supports of prefabricated beams**.

Load case of increments of permanent loads in the stage and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.8.4 Final supports

	Name	Time [d]	Check	Description	
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>		▲ In-plane restraint
<input checked="" type="checkbox"/>	2 Storage yard	5,1	<input checked="" type="checkbox"/>		In-plane restraint <input type="checkbox"/>
<input checked="" type="checkbox"/>	3 Transport	25	<input checked="" type="checkbox"/>		▲ Post-tensioning
					Post-tensioning <input type="checkbox"/>
> <input checked="" type="checkbox"/>	4 Final supports	35	<input checked="" type="checkbox"/>		▲ Final supports
<input type="checkbox"/>	5 Superimposed dead load	60	<input type="checkbox"/>		To design position <input checked="" type="checkbox"/>
<input checked="" type="checkbox"/>	6 End of design working life	18250	<input checked="" type="checkbox"/>		▲ Load cases and combinations
					Load cases R (4), G (4)
					Combinations ULSF ST(4), SLSC ST(4), SLSF ST(4), SLSQ ST(4)

The design member is placed to the final position in this stage. It is assumed that the structural system will not change since this time. Position of supports corresponds to positions defined during input of beam geometry. Additional prestressing by post-tensioned tendons can be applied in this stage.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Properties group **Post-tensioning**:

- **Post-tensioning** – if the option is selected, post-tensioned tendons can be defined for the stage to apply additional prestressing.

Properties group **Final supports**:

- **To design position** – if selected, the supports emerging in this stage are placed to the design position. Thus the deflections, which appeared in positions of new supports during previous stages, will be eliminated.

Load case of increments of permanent loads in the stage, load case for transfer of effects of post-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.8.5 Superimposed dead load

See **Superimposed dead load**.

7.3.8.6 End of working life

See **End of working life**.

7.3.9 Prefabricated pre-tensioned beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Transfer of prestressing	5	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Final supports	35	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 End of design working life	18250	<input checked="" type="checkbox"/>	

Following default construction stages are generated for prefabricated pre-tensioned beam:

- Casting.
- Transfer of prestressing.
- Storage yard. This stage may be switched off from calculation.
- Transport. This stage may be switched off from calculation.
- Final supports.
- Superimposed dead load. This stage may be switched off from calculation.
- End of working life.

7.3.9.1 Casting

See [Casting](#).

7.3.9.2 Transfer of prestressing

		Name	Time [d]	Check	Description
	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
>	<input checked="" type="checkbox"/>	2 Transfer of prestressing	5	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Final supports	35	<input checked="" type="checkbox"/>	
	<input type="checkbox"/>	6 Superimposed dead load	60	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 End of design working life	18250	<input checked="" type="checkbox"/>	

<ul style="list-style-type: none"> <input type="checkbox"/> In-plane restraint <input type="checkbox"/> In-plane restraint 	
<ul style="list-style-type: none"> <input type="checkbox"/> Concrete strength in current construction stage User-specified concrete strength [MPa] <input type="checkbox"/> 24,7 	
<ul style="list-style-type: none"> <input type="checkbox"/> Beam spans L1 [m] 0,00 L [m] 6,08 L2 [m] 0,00 	
<ul style="list-style-type: none"> <input type="checkbox"/> Temporary supports To design position <input type="checkbox"/> Local system <input type="checkbox"/> Flexible supports <input checked="" type="checkbox"/> Ky - left support [MN/m] <input type="text" value="10000000,0"/> Kz - left support [MN/m] <input type="text" value="10000000,0"/> Ky - right support [MN/m] <input type="text" value="10000000,0"/> Kz - right support [MN/m] <input type="text" value="10000000,0"/> 	
<ul style="list-style-type: none"> <input type="checkbox"/> Load cases and combinations Load cases R (2), G (2), PRE (2) Combinations ULSF ST(2), SLSC ST(2), SLSF ST(2), SLSQ ST(2) 	

The sufficient strength is reached in this stage, thus the prestressing from pre-tensioned tendons can be applied. The effects caused by pre-tensioned reinforcement are considered as permanent since this time.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

As far as the precast element lifts up from the rest during stressing, self-weight is activated and it acts against this cambering. To utilize the self-weight to reduce the effects of prestressing, the supports are placed at the ends of the beam.

Properties group **Concrete strength in current construction stage** – in the first stage after casting, it enables to calculate the time of current construction stage from the specified concrete strength the and vice versa.

- **User-specified concrete strength** – if the option is selected, the value of concrete strength in the current construction stage can be specified. The time of stage is calculated from the specified value.
- **fck** – input of characteristic compressive cylinder concrete strength determined by tests.

Properties group **Temporary supports** – definition of temporary (outmost) supports – see **Temporary supports of prefabricated beams**.

Load case for transfer of effects of pre-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

For some beam types, load case of self-weight of the first phase of cross-section is assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.9.3 Storage yard

See **Storage yard**.

7.3.9.4 Transport

See **Transport**.

7.3.9.5 Final supports

See **Final supports**.

7.3.9.6 Superimposed dead load

See **Superimposed dead load**.

7.3.9.7 End of working life

See **End of working life**.

7.3.10 Prefabricated post-tensioned beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Final supports	35	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 End of design working life	18250	<input checked="" type="checkbox"/>	

Following default construction stages are generated for prefabricated post-tensioned beam:

- Casting.
- Post-tensioning.
- Storage yard. This stage may be switched off from calculation.
- Transport. This stage may be switched off from calculation.
- Final supports.
- Superimposed dead load. This stage may be switched off from calculation.
- End of working life.

7.3.10.1 Casting

See [Casting](#).

7.3.10.2 Post-tensioning

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
>	<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>
	<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>
	<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>
	<input checked="" type="checkbox"/>	5 Final supports	35	<input checked="" type="checkbox"/>
	<input type="checkbox"/>	6 Superimposed dead load	60	<input type="checkbox"/>
	<input checked="" type="checkbox"/>	7 End of design working life	18250	<input checked="" type="checkbox"/>

<ul style="list-style-type: none"> In-plane restraint <ul style="list-style-type: none"> In-plane restraint <input type="checkbox"/> Concrete strength in current construction stage <ul style="list-style-type: none"> User-specified concrete strength <input type="checkbox"/> f_{ck} [MPa] 24,7 Beam spans <ul style="list-style-type: none"> L1 [m] 0,00 L [m] 6,08 L2 [m] 0,00 Temporary supports <ul style="list-style-type: none"> To design position <input type="checkbox"/> Local system <input type="checkbox"/> Flexible supports <input checked="" type="checkbox"/> K_y - left support [MN/m] 10000000,0 K_z - left support [MN/m] 10000000,0 K_y - right support [MN/m] 10000000,0 K_z - right support [MN/m] 10000000,0 Load cases and combinations <ul style="list-style-type: none"> Load cases R (2), G (2), POST (2) Combinations ULSF ST(2), SLSC ST(2), SLSF ST(2), SLSQ ST(2)
--

The post-tensioned tendons are tensioned and anchorage permanently in this stage. Post-tensioned reinforcement is a part of the structure since this time and resists all loads.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

As far as the precast element lifts up from the rest during stressing, self-weight is activated and it acts against this cambering. To utilize the self-weight to reduce the effects of prestressing, the supports are placed at the ends of the beam.

Properties group **Concrete strength in current construction stage** – in the first stage after casting, it enables to calculate the time of current construction stage from the specified concrete strength the and vice versa.

- **User-specified concrete strength** – if the option is selected, the value of concrete strength in the current construction stage can be specified. The time of stage is calculated from the specified value.
- **fck** – input of characteristic compressive cylinder concrete strength determined by tests.

Properties group **Temporary supports** – definition of temporary (outmost) supports – see **Temporary supports of prefabricated beams**.

Load case for transfer of prestressing effects of post-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

For some beam types, load case of self-weight of the first phase of cross-section is assigned to this stage.

Corresponding combinations are assigned to this stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.10.3 Storage yard

See **Storage yard**.

7.3.10.4 Transport

See **Transport**.

7.3.10.5 Final supports

See **Final supports**.

7.3.10.6 Superimposed dead load

See **Superimposed dead load**.

7.3.10.7 End of working life

See **End of working life**.

7.3.11 Prefabricated composite reinforced beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Temporary supports	26	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Casting of composite slab	28	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 Final supports	35	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	8 End of design working life	18250	<input checked="" type="checkbox"/>	

Following default construction stages are generated for prefabricated composite reinforced beam:

- Casting.
- Storage yard. This stage may be switched off from calculation.
- Transport. This stage may be switched off from calculation.
- Temporary supports. This stage may be switched off from calculation.
- Casting of composite slab.
- Final supports.
- Superimposed dead load. This stage may be switched off from calculation.
- End of working life.

7.3.11.1 Casting

See [Casting](#).

7.3.11.2 Storage yard

See [Storage yard](#).

7.3.11.3 Transport

See [Transport](#).

7.3.11.4 Temporary supports

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Transfer of prestressing	5	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
>	<input checked="" type="checkbox"/> 5 Temporary supports	26	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	6 Casting of composite slab	28	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	7 Final supports	35	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	8 Superimposed dead load	60	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	9 End of design working life	18250	<input checked="" type="checkbox"/>	

In-plane restraint	
In-plane restraint	<input type="checkbox"/>
Post-tensioning	
Post-tensioning	<input type="checkbox"/>
Beam spans	
L1 [m]	1,22
L [m]	3,65
L2 [m]	1,22
Outmost temporary supports	
To design position	<input type="checkbox"/>
Local system	<input type="checkbox"/>
Flexible supports	<input checked="" type="checkbox"/>
Ky - left support [MN/m]	10000000,0
Kz - left support [MN/m]	10000000,0
Ky - right support [MN/m]	10000000,0
Kz - right support [MN/m]	10000000,0
Internal temporary supports	
Number	0
Load cases and combinations	
Load cases	R (5), G (5)
Combinations	ULSF ST(5), SLSC ST(5), SLSF ST(5), SLSQ ST(5)

Temporary supporting before casting the composite slab can be defined in this stage. Additional prestressing by post-tensioned tendons can be applied in this stage.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Properties group **Post-tensioning**:

- **Post-tensioning** – if the option is selected, post-tensioned tendons can be defined for the stage to apply additional prestressing.

Properties group **Beam spans** – definition of temporary supports position:

- **L1** – input of length of left span.
- **L** – displays the length of the mid span, calculated from the overall length of beam.
- **L2** – input of length of right span.

Properties group **Outmost temporary supports** and **Internal temporary supports** – see **Temporary supports of prefabricated beams**.

Load case of increments of permanent loads in the stage, load case for transfer of effects of post-tensioning and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.11.5 Casting of composite slab

	Name	Time [d]	Check	Description
<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	2 Transfer of prestressing	5	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	5 Temporary supports	26	<input checked="" type="checkbox"/>	
>	<input checked="" type="checkbox"/> 6 Casting of composite slab	28	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	7 Final supports	35	<input checked="" type="checkbox"/>	
<input type="checkbox"/>	8 Superimposed dead load	60	<input type="checkbox"/>	
<input checked="" type="checkbox"/>	9 End of design working life	18250	<input checked="" type="checkbox"/>	

<ul style="list-style-type: none"> In-plane restraint <ul style="list-style-type: none"> In-plane restraint <input type="checkbox"/> Beam spans <ul style="list-style-type: none"> L1 [m] <input type="text" value="1,22"/> L [m] <input type="text" value="3,65"/> L2 [m] <input type="text" value="1,22"/> Outmost temporary supports <ul style="list-style-type: none"> To design position <input type="checkbox"/> Local system <input type="checkbox"/> Flexible supports <input checked="" type="checkbox"/> Ky - left support [MN/m] <input type="text" value="10000000,0"/> Kz - left support [MN/m] <input type="text" value="10000000,0"/> Ky - right support [MN/m] <input type="text" value="10000000,0"/> Kz - right support [MN/m] <input type="text" value="10000000,0"/> Internal temporary supports <ul style="list-style-type: none"> Number <input type="text" value="1"/> Distances between support <input type="text" value="1,82 1,82"/> Flexible supports <input type="checkbox"/> Required camber <input checked="" type="checkbox"/> Type of input <input type="text" value="By value"/> Value [mm] <input type="text" value="6"/> Supports setting [mm] <input type="text" value="6"/> Load cases and combinations <ul style="list-style-type: none"> Load cases R (6), SWS (6) Combinations ULSF ST(6), SLSC ST(6), SLSF ST(6), SLSQ ST(6)
--

Fresh concrete of composite slab is cast in this construction stage. Only the prefabricated part of structure resists the load from fresh concrete.

The time of stage can be modified.

The checks can be switched on/off for this stage.

Properties group **In-plane restraint**:

- **In-plane restraint** – switch on/off preventing such deformation of member, which would form out-of-plane flexure (displacement in the direction of local Y axis and rotation about X and Z axes) starting from selected construction stage and all subsequent stages.

Properties group **Beam spans** – definition of temporary supports position at composite slab casting:

- **L1** – input of length of left span.
- **L** – displays the length of the mid span, calculated from the overall length of beam.
- **L2** – input of length of right span.

Properties group **Outmost temporary supports** and **Internal temporary supports** – see **Temporary supports of prefabricated beams**.

Load case of own weight of composite slab and load case of rheological effect (for structures calculated using time dependent analysis) in the stage are assigned to the stage.

Corresponding combinations are assigned to the stage.

Assignment of load cases and combinations can be edited (respecting the rules described in **Load cases and combinations in construction stages**).

7.3.11.6 Final supports

See **Final supports**.

7.3.11.7 Superimposed dead load

See **Superimposed dead load**.

7.3.11.8 End of working life

See **End of working life**.

7.3.12 Prefabricated composite pre-tensioned beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Transfer of prestressing	5	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Temporary supports	26	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 Casting of composite slab	28	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 Final supports	35	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	8 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	9 End of design working life	18250	<input checked="" type="checkbox"/>	

Following default construction stages are generated for prefabricated composite pre-tensioned beam:

- Casting.
- Transfer of prestressing.
- Storage yard. This stage may be switched off from calculation.
- Transport. This stage may be switched off from calculation.
- Temporary supports. This stage may be switched off from calculation.
- Casting of composite slab.
- Final supports.
- Superimposed dead load. This stage may be switched off from calculation.
- End of working life.

7.3.12.1 Casting

See **Casting**.

7.3.12.2 Transfer of prestressing

See **Transfer of prestressing**.

7.3.12.3 Storage yard

See **Storage yard**.

7.3.12.4 Transport

See **Transport**.

7.3.12.5 Temporary supports

See **Temporary supports**.

7.3.12.6 Casting of composite slab

See **Casting of composite slab**.

7.3.12.7 Final supports

See **Final supports**.

7.3.12.8 Superimposed dead load

See **Superimposed dead load**.

7.3.12.9 End of working life

See **End of working life**.

7.3.13 Prefabricated composite post-tensioned beam

		Name	Time [d]	Check	Description
>	<input checked="" type="checkbox"/>	1 Casting	0	<input type="checkbox"/>	
	<input checked="" type="checkbox"/>	2 Post-tensioning	5	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	3 Storage yard	5,1	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	4 Transport	25	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	5 Temporary supports	26	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	6 Casting of composite slab	28	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	7 Final supports	35	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	8 Superimposed dead load	60	<input checked="" type="checkbox"/>	
	<input checked="" type="checkbox"/>	9 End of design working life	18250	<input checked="" type="checkbox"/>	

Following default construction stages are generated for prefabricated composite post-tensioned beam:

- Casting.
- Post-tensioning.
- Storage yard. This stage may be switched off from calculation.
- Transport. This stage may be switched off from calculation.
- Temporary supports. This stage may be switched off from calculation.
- Casting of composite slab.
- Final supports.
- Superimposed dead load. This stage may be switched off from calculation.
- End of working life.

7.3.13.1 Casting

See **Casting**.

7.3.13.2 Post-tensioning

See **Post-tensioning**.

7.3.13.3 Storage yard

See **Storage yard**.

7.3.13.4 Transport

See **Transport**.

7.3.13.5 Temporary supports

See **Temporary supports**.

7.3.13.6 Casting of composite slab

See **Casting of composite slab**.

7.3.13.7 Final supports

See **Final supports**.

7.3.13.8 Superimposed dead load

See **Superimposed dead load**.

7.3.13.9 End of working life

See **End of working life**.

8 Design of tendons

Tendons must be defined for prestressed beams (pre-tensioned, post-tensioned, pre/post tensioned).

Input and design of tendons is performed in the module IDEA Tendon. To start design of tendons click **Tendon design**.

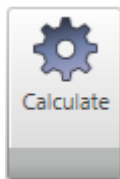


Design of tendons is described in a separate user guide **IDEA Tendon**.

The resulting equivalent effect of prestressing are stored in appropriate load cases, which are generated automatically according to the beam type.

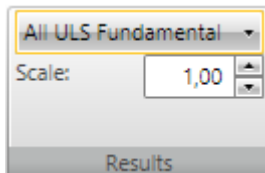
9 Results

When the structure is defined , click **Calculate** to perform structure analysis.

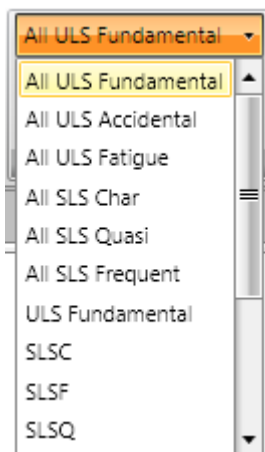


After the analysis finishes, navigator commands in the **Results** group are available.

9.1 Results evaluation setup



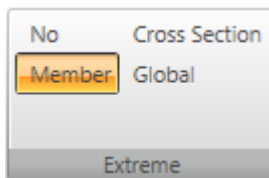
Evaluation of reactions, deformations and internal forces is performed for the current result class, load case or combination. Current result class, load case or combination can be selected in the list in ribbon group **Results**.



Following result classes are generated automatically:

- **All ULS Fundamental** – all existing ULS fundamental combinations are automatically assigned to this result class.
- **All ULS Accidental** - all existing ULS accidental combinations are automatically assigned to this result class.
- **All ULS Fatigue** - all existing ULS fatigue combinations are automatically assigned to this result class.
- **All SLS Char** – all existing SLS Characteristic combinations are automatically assigned to this result class.
- **All SLS Quasi** – all existing SLS Quasi-permanent combinations are automatically assigned to this result class.
- **All SLS Frequent** – all existing SLS Frequent combinations are automatically assigned to this result class.

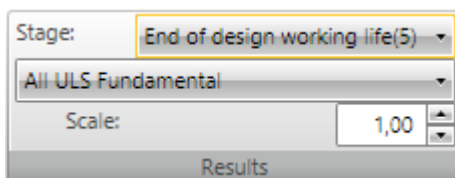
Scale for drawing of evaluated results can be set in edit box **Scale**.



Switches in ribbon group **Extreme** can be used to set the range of evaluated results.

- **No** – all values of evaluated components are printed for each case/combination in each member section/node.
- **Member** – extreme values of evaluated components are found per each member/supported node of the beam.
- **Cross-section** – extreme values of evaluated components are found per each cross-section of the beam.
- **Global** – extreme values of evaluated components are found from all members/ supported nodes of the beam.

9.1.1 Results evaluation setup of staged beams



A current construction stage, for which the results are evaluated, can be selected in the list **Stage**.

The list of available load cases and combinations is filtered according to the current stage. Also the content of automatically generated result classes is modified so that each result class contains only such combinations, which are assigned to the current stage.

The supports are drawn for the current stage too.

9.2 Reactions in supports

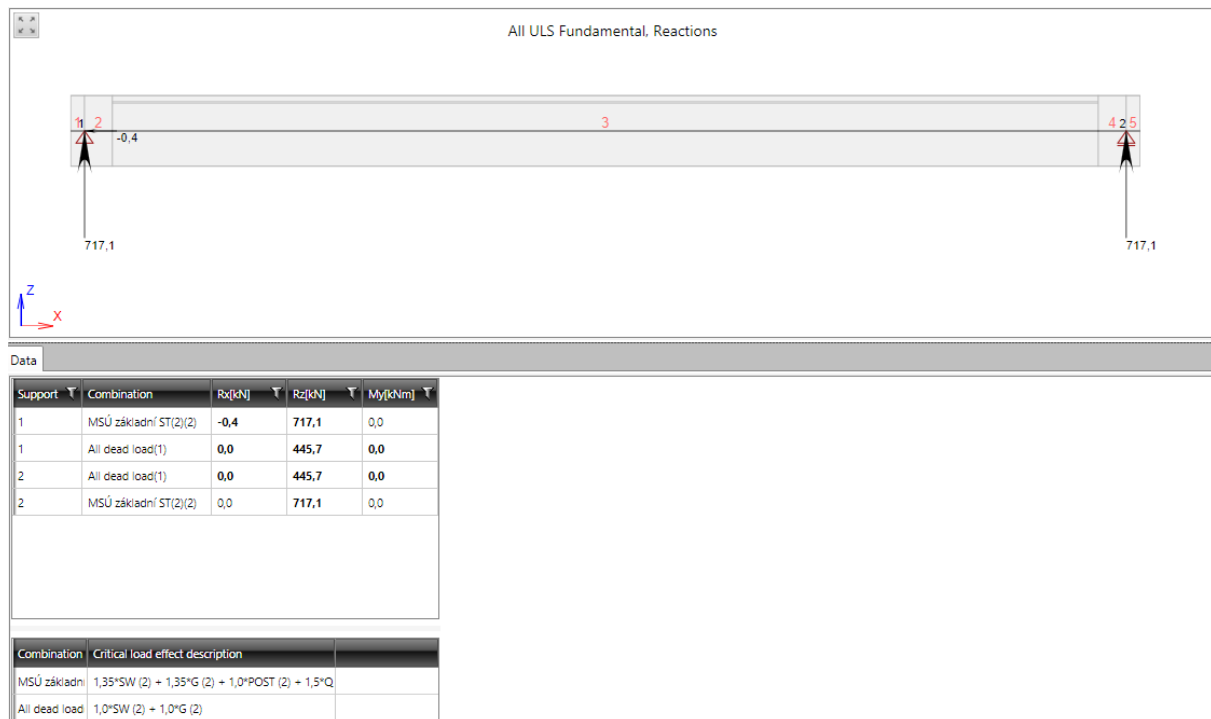
Click navigator command **Results > Reactions** to evaluate reactions in supports.

Calculated reactions are evaluated:

- Graphically – diagrams of reactions are drawn in the Main window.
- Textually – extreme values of reactions are displayed in the table in the Data window.

Reactions are evaluated for current load case or combination.

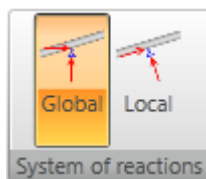
Ribbon groups **Results**, **System of reactions** and **Extreme** are available when evaluating reactions in supports. Ribbon group **Reactions/Bearing stress intensity** is available when evaluating reactions on line supports.



9.2.1 Ribbon group Results

See [Results evaluation setup](#).

9.2.2 Ribbon group System of reactions



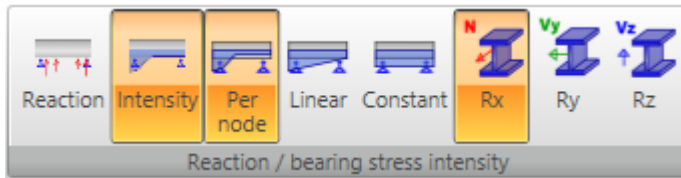
Use commands in ribbon group **System of reactions** to set the coordinate system to evaluate the reactions:

- **Global** – switches to evaluate reactions in global coordinate system.
- **Local** – switches to evaluate reactions in local coordinate system of rotated supports.

9.2.3 Ribbon group Extreme

See [Results evaluation setup](#).

9.2.4 Ribbon group Reactions/Bearing stress intensity



Use commands in ribbon group **Reactions** to set the evaluation mode on line supports:

- **Reaction** – switches to evaluate reactions in individual supported nodes of finite elements mesh.
- **Intensity** – switches to evaluate bearing stress intensity on individual line supports.
- **Per node** – switches to evaluate course of bearing stress falling on individual nodes of finite elements mesh.
- **Linear** – switches to evaluate linear course of bearing stress along the whole line support.
- **Constant** – switches to evaluate constant courses of bearing stress along the whole line support.
- **Rx** – switches to draw course of bearing stress in global X-axis direction.
- **Ry** – switches to draw course of bearing stress in global Y-axis direction. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Rz** – switches to draw course of bearing stress in global Z-axis direction.

9.3 Deformations

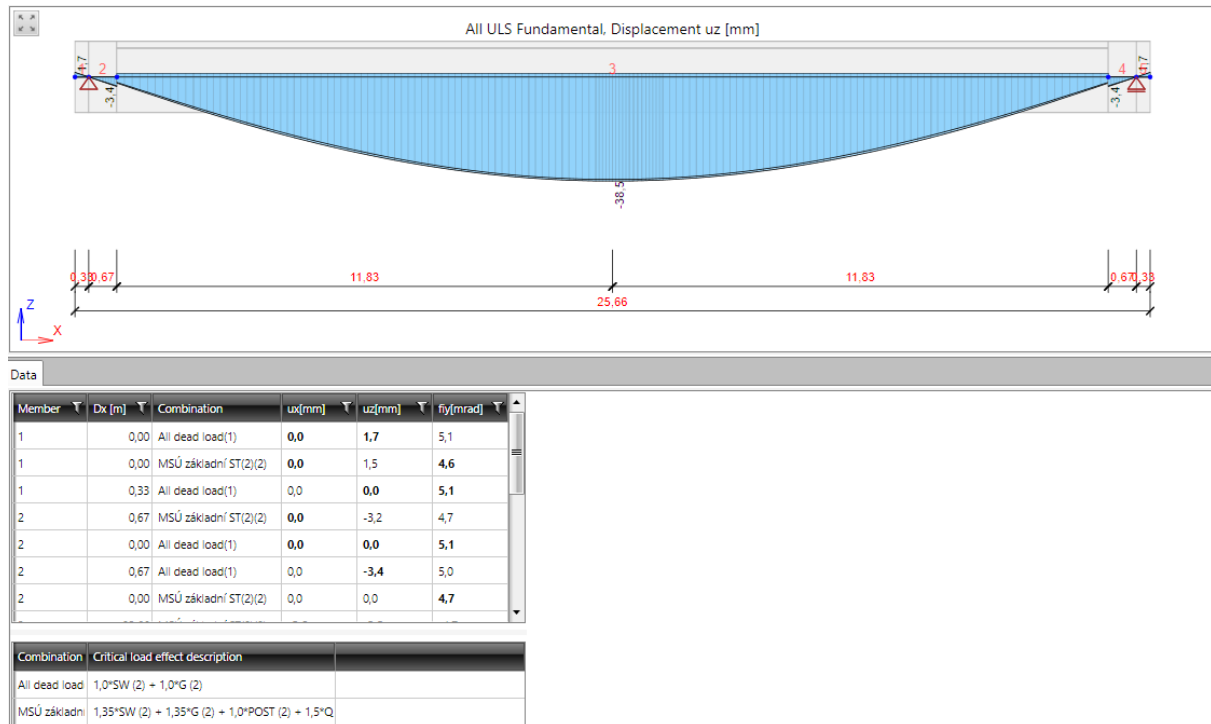
Click navigator command **Results > Deformations** to evaluate beam deformations.

Calculated deformations are evaluated:

- **Graphically** – diagrams of selected component of deformation are displayed in the Main window.
- **Textually** – extreme values of deformations and rotations are displayed in the table in the Data window.

Deformations are evaluated for current load case or combination.

Ribbon groups **Results**, **Extreme** and **Deformations** are available when evaluating deformations.



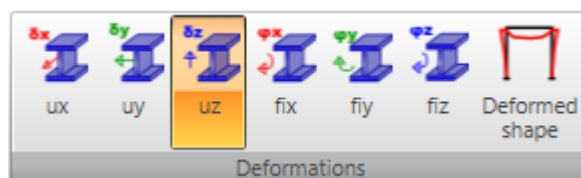
9.3.1 Ribbon group Results

See [Results evaluation setup](#).

9.3.2 Ribbon group Extreme

See [Results evaluation setup](#).

9.3.3 Ribbon group Deformations



Particular options of ribbon group **Deformations**:

- **ux** – switch to graphical evaluation of deformation in direction of local member axis x.
- **uy** – switch to graphical evaluation of deformation in direction of local member axis y. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.

- **uz** – switch to graphical evaluation of deformation in direction of local member axis z.
- **fix** – switch to graphical evaluation of rotation about local member axis x. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **fiy** – switch to graphical evaluation of rotation about local member axis y.
- **fiz** – switch to graphical evaluation of rotation about local member axis z. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Deformed shape** – switch to draw deformed shape (spatial deformation) of the structure.

9.4 Internal forces

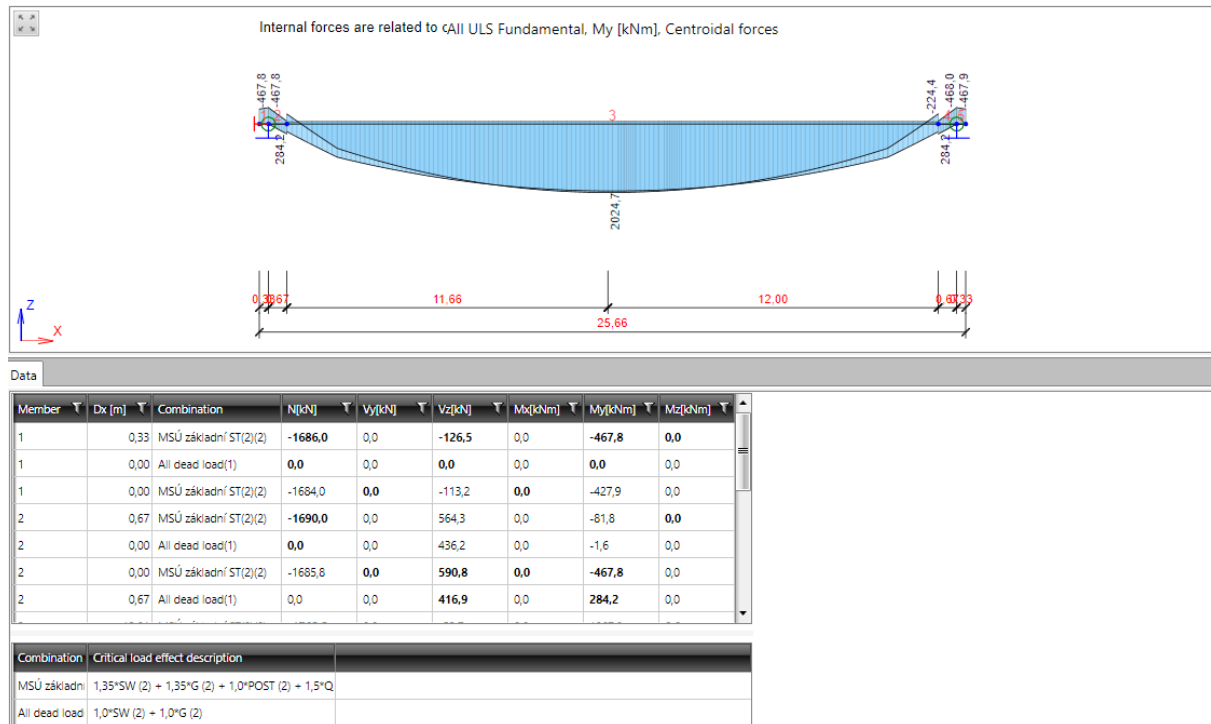
Click navigator command **Results > Internal forces** to evaluate beam internal forces.

Calculated internal forces are evaluated:

- Graphically – diagrams of selected component of internal forces are displayed in the Main window.
- Textually – extreme values of internal forces are displayed in the table in Data window.

Internal forces are evaluated for current load case or combination.

Ribbon groups **Results**, **Extreme**, **Transformations**, **Internal forces** and **Fatigue evaluation** are available when evaluating internal forces.



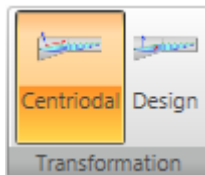
9.4.1 Ribbon group Results

See [Results evaluation setup](#).

9.4.2 Ribbon group Extreme

See [Results evaluation setup](#).

9.4.3 Ribbon group Transformations

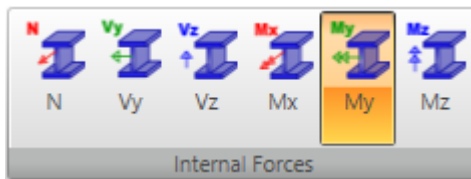


Use switches in ribbon group **Transformations** to set the mode of internal forces evaluation (graphical and textual evaluation).

- **Centroidal** – switch to evaluate internal forces in sections perpendicular to the centroidal axis of members.

- **Design** – switch to evaluate internal forces in sections perpendicular to the reference line.

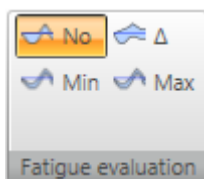
9.4.4 Ribbon group Internal forces



Individual commands in ribbon group **Internal forces**:

- **N** – switch to the graphical evaluation of normal force N .
- **Vy** – switch to the graphical evaluation of shear force V_y . This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Vz** – switch to the graphical evaluation of shear force V_z .
- **Mx** – switch to the graphical evaluation of torsional moment M_x . This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **My** – switch to the graphical evaluation of bending moment M_y .
- **Mz** – switch to the graphical evaluation of bending moment M_z . This option is available only for **Straight or polygonal beam loaded in 3D** beam type.

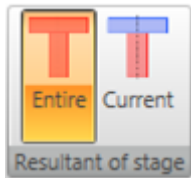
9.4.5 Ribbon group Fatigue evaluation



If result class All ULS fatigue or combination of fatigue type is set to be evaluated, the ribbon group **Fatigue evaluation** is available.

- **No** – switch to evaluate the basic combination of fatigue combination (not considering the cyclic load).
- **Δ** – switch to draw the difference between the minimal and maximal value of evaluated component (amplitude) caused by the cyclic load of the fatigue combination only. The amplitudes and minimal and maximal values of single internal forces components (caused by the complete fatigue combination) are displayed in the table.
- **Min** – switch to draw the courses of minimal amplitudes of evaluated internal forces component caused by the complete fatigue combination (including the cyclic load). The amplitudes and minimal and maximal values of single internal forces components (caused by the complete fatigue combination) are displayed in the table.
- **Max** – switch to draw the courses of maximal amplitude of evaluated internal forces component caused by the complete fatigue combination (including the cyclic load). The amplitudes and minimal and maximal values of single internal forces components (caused by the complete fatigue combination) are displayed in the table.

9.4.6 Ribbon group Resultant of stage



If the evaluation of internal forces is set for individual construction stage and for load case or combination corresponding to selected stage, the mode of internal forces resultant evaluation can be set..

- **Entire** – switches to evaluation of internal forces related to the centroid of concrete cross-section (without the reinforcement) with modulus of elasticity at the age 28 days. All phases of cross-section are taken into account in the case of composite cross-section irrespective of the fact if they currently exist..
- **Current** – switches to evaluation of internal forces related to the centroid of transformed cross-section determined from currently existing phases of cross-section and their prestressing reinforcement. Change of modulus of elasticity due to concrete aging is considered in all phases of cross-section.

10 Design and check of structural items

IDEA Beam is an application for static analysis of the structure. Reactions in support, linear deformations and internal forces caused by acting load are results of this analysis.

In case of concrete beam the reinforcement zones can be designed to reinforce the beam. Reinforcement can be input to the zones using reinforcement templates. Reinforced beam can be checked and results of check can be evaluated along the beam. The detailed check of sections can be performed in IDEA RCS.

The deflections can be calculated and evaluated on the reinforced (not pre-tensioned) beam too.

The design of prestressed tendons is done in IDEA Tendon. After design of tendons is finished, equivalent loads are transferred back to the appropriate load case for prestressing.

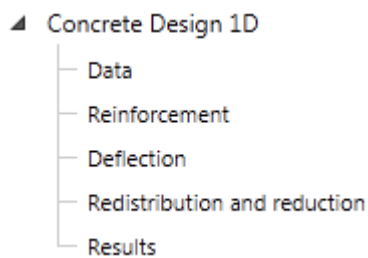
To start IDEA Tendon click **Tendon design**.



In the case of steel beam the buckling parameters and the steel design data can be defined. Afterwards, the section resistance check, the buckling resistance check and deflection check can be performed.

11 Check of concrete members

The input of data for check and evaluation of check results and deflection calculation results is done on the current design member. The whole beam is considered as one design member in IDEA Beam.



Use commands in ribbon group Concrete Design 1D to define reinforcement zones, to input reinforcement to zones, to define supports for deflection calculation, to define conditions for redistribution and reduction calculation and to evaluate results of section check and deflection calculation.

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

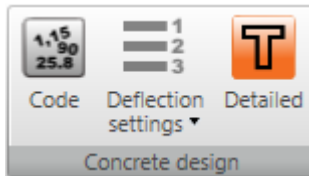
To be able to design the reinforcement, reinforcement zones have to be defined along the design member and reinforcement has to be defined using reinforcement templates. Each zone corresponds to one section and each template corresponds to reinforced section in IDEA RCS.

Extreme internal forces from the whole zone are found for the design.

To be able to design the reinforcement on beam several preconditions have to be fulfilled:

- Beam is of type Concrete Beam.
- There are combinations of ULS, SLS Characteristic, SLS Frequent and SLS quasi-permanent.
- The project is calculated – internal forces are available.
- Calculation of deflections and the check of limit deflections is available only for beams, which are not calculated using time dependent analysis.

11.1 Settings for section checks and calculation of deflections



Check and deflection calculation settings, common for all design members, can be changed using commands in ribbon group **Concrete design**:

- **Code** – code and calculation settings – see **Code and calculation settings**.
- **Deflection settings** – settings of parameters of loads, which are used for calculation of deflections – see **Setting of result class for calculation of deflections**.

11.1.1 Code and calculation settings

To edit code settings and calculation settings click **Code** in ribbon group **Concrete design**.

The screenshot shows the 'Code and calculation settings' dialog box. It is organized into three sections: Chapter 2, Chapter 5, and Chapter 6. Each section contains a list of parameters with corresponding input fields for their values.

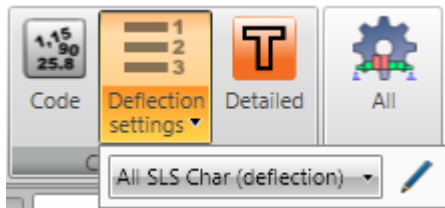
Section	Parameter	Value
Chapter 2	2.4.2.4(1) γ_c - Persistent, transient	1,5
	2.4.2.4(1) γ_c - Accidental	1,2
	2.4.2.4(1) γ_s - Persistent, transient	1,15
	2.4.2.4(1) γ_s - Accidental	1
	2.4.2.4(1) γ_{sp}	1,15
Chapter 5	5.5 k1	0,44
	5.5 k2	$1,25 * (0,6 + 0,0014 / \epsilon_c)$
	5.5 k3	0,54
	5.5 k4	$1,25 * (0,6 + 0,0014 / \epsilon_c)$
	5.5 k5	0,7
	5.5 k6	0,8
Chapter 6	6.2.2 Values for shear check $d = h^*$	0,9
	6.2.2 Values for shear check $z = d^*$	0,9

Only the important values can be set in this setup dialog. The remaining values can be set in the detailed check in IDEA StatiCa RCS.

Buttons: Expand all, Collapse all, OK, Cancel

11.1.2 Setting of result class for calculation of deflections

The deflections are calculated for all combinations assigned to the result class which is set to be used to calculation of deflections. The default result class for calculation of deflections is named **All SLS (Deflections)**. Combinations in this result class are considered as characteristic combinations. In the background, one quasi-permanent combination is generated per each combination in this result class. To determine the portion of long term loads in variable load cases value of coefficient ψ_2 is used, which is taken from the load group, in which the load cases are assigned.



Click **Deflection settings** in ribbon group Settings to set loads for calculation of deflections.

- **Result class** – select the result class for calculation of deflections. Click edit button  to edit the content of the selected result class – see [Editing result class](#).

11.1.3 Setting of result classes for section checks

Result classes are required to be able to perform the sections checks of current design member.

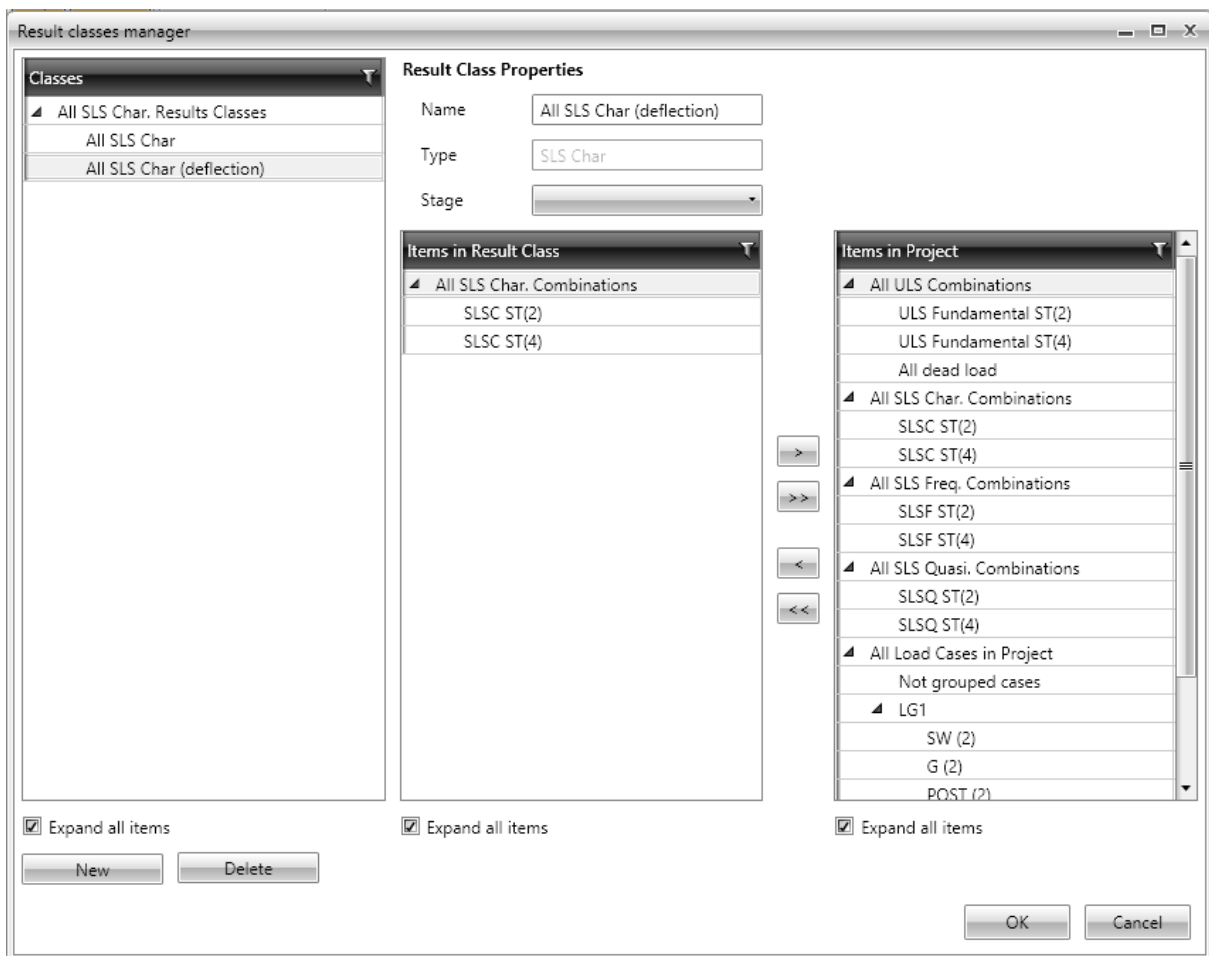
The result classes are generated automatically in the background. All combinations of corresponding type are assigned to appropriate result classes. The content of result classes cannot be edited.

Following combinations of load actions are used for check of concrete sections in IDEA RCS:

- **ULS – fundamental** – contains load effects caused by result class ULS - fundamental, which contains all load combinations of type ULS - fundamental.
- **ULS – accidental** – contains load effects caused by result class ULS - accidental, which contains all load combinations of type ULS - accidental.
- **Min. cyclic load** – contains minimal amplitude of load effects caused by result class ULS - fatigue, which contains all load combinations of type ULS – fatigue.
- **Max. cyclic load** – contains maximal amplitude of load effects caused by result class ULS - fatigue, which contains all load combinations of type ULS – fatigue.
- **SLS – characteristic** – contains load effects caused by result class SLS Char, which contains all load combinations of type SLS Char.
- **SLS – frequent** – contains load effects caused by result class SLS Frequent, which contains all load combinations of type SLS Frequent.
- **SLS – quasi-permanent** – contains load effects caused by result class SLS Quasi, which contains all load combinations of type SLS Quasi.

11.1.4 Editing result class

Click edit button at result classes list box to edit the content of result class. Defined result classes are displayed in the left tree view. The content and the properties of the current result class are displayed in the mid column. All load cases and all combinations available in project are displayed in the right tree view.







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

- **Classes** – defined result classes, grouped according to their types, are displayed in the tree view. Properties of selected result class and list of combinations and load cases in the result class are displayed in the middle part of dialog.
- **New** – add new result class.
- **Delete** – delete selected result class.
- **Expand all items** – expand/collapse all items in the result classes tree view.

Result class properties:

- **Name** – input name of the current result class.
- **Type** – select the type of the current result class.
- **Items in result class** – combinations and load cases, assigned to the current result class, are displayed in the tree view. Combinations are grouped according to their types and load cases are grouped according to the load groups.
- **Expand all items** – expand/collapse all items in the result class items tree view.

-  - remove selected combination or load case or load cases group from the current result class.
-  - remove all items from the current result class.
-  - add combination, load case or load cases group, selected in the tree view **Items in project**, to the current result class.
-  - add all items from the tree view **Items in project** to the current result class.

11.2 Design member data

Click navigator command **Concrete Design 1D > Data** to define design member data relevant for checks.

Ultimate limit state	
Capacity N-M-M	<input checked="" type="checkbox"/>
Shear	<input checked="" type="checkbox"/>
Torsion	<input type="checkbox"/>
Interaction	<input checked="" type="checkbox"/>
Fatigue	<input checked="" type="checkbox"/>
Serviceability limit state	
Stress limitation	<input checked="" type="checkbox"/>
Crack width	<input checked="" type="checkbox"/>
Detailing	
Detailing	<input checked="" type="checkbox"/>
Deflections	
Deflections	Detailed calculation
Reduction and redistribution	
Redistribution of moment:	<input checked="" type="checkbox"/>
Reduction of moments	<input checked="" type="checkbox"/>
Reduction of shear force	<input checked="" type="checkbox"/>
Limited interaction check	<input type="checkbox"/>

Exposure class	
No corrosion (X0)	<input type="checkbox"/>
Carbonation	XC3 - Moderate h
Chlorides	XD1 - Moderate h
Chlorides from sea	No risk of chloride
Freeze/Thaw Attack	No risk of freeze/t
Chemical Attack	No risk of chemica
Properties	
Relative humidity [%]	65
Creep coefficient	Calculated
Member type	Beam
Structural member import.	Major

Particular concrete checks, which should be performed on current design member, can be selected in the left part of the table.

Properties group **Ultimate limit state**:

- **Capacity N-M-M** – switch on/off performing the capacity check.
- **Shear** – switch on/off performing the shear check.
- **Torsion** – switch on/off performing the torsion check.
- **Interaction** — switch on/off performing the check of interaction of normal force, bending, torsion and shear.
- **Fatigue** – switch on/off performing the fatigue check.

Properties group **Serviceability limit state**:

- **Stress limitation** – switch on/off performing the stress limitation check.
- **Crack width** – switch on/off performing the crack width check..

Properties group **Detailing**:

- **Detailing** – switch on/off performing the detailing rules verification.

Properties group **Deflections**:

- **Deflection** – select mode of deflection check:
 - **Do not calculate** – deflections are neither calculated, nor checked.
 - **By limiting the flexural slenderness** – the deflection check is performed as a proof of flexural slenderness limitation according to art. 7.4.2
 - **Detailed calculation** – the detailed calculation of deflections is performed, following 7.4.3 Checking deflections by calculation, without respecting (6) Shrinkage curvatures.

Properties group **Redistributions and reductions**:

- **Redistribution of moments** – switch on/off calculation of moments redistribution according to EN 1992-1-1, art. 5.5.
- **Reduction of moments** – switch on/off calculation of reduced moments in supports according to EN 1992-1-1, art. 5.3.2.2(3) a 5.3.2.2(4).
- **Reduction of shear force** – switch on/off calculation of reduced shear force for members with loads near supports according to EN 1992-1-1, art. 6.2.2(6) and 6.2.3(8).
- **Limited interaction check** – switch on/off limitation of interaction check in distance less than d from position of maximal moment according to EN 1992-1-1 6.2.3(7).

Exposure classes and design member properties can be set in the right part of the table.

Properties group **Exposure class**:

- **No corrosion** – switch on/off exposure class with no risk of corrosion or attack X0.
- **Carbonation** – select exposure class for corrosion caused by carbonation.
- **Chlorides** – select exposure class for corrosion caused by chlorides.
- **Chlorides from sea** – select exposure class for corrosion caused by chlorides from sea.
- **Freeze/Thaw attack** – select exposure class for corrosion caused by freeze/thaw cycles.
- **Chemical attack** – select exposure class for corrosion caused by chemically aggressive environment.
- **Relative humidity** – input value of relative humidity.
- **Creep coefficient** – select mode to determine the creep coefficient value:
 - **Calculated** – creep coefficient value is calculated automatically.
 - **User input** – value of creep coefficient Φ_{inf} can be defined by user.
- **Structural member importance** – select type of structural member importance according to 6.2.1(4).

11.3 Reinforcement zones

Click navigator command **Concrete design 1D > Reinforcement** to input reinforcement zones and reinforcement to zones.

After the zones and reinforcement are defined, either the detailed check in IDEA RCS or the section check along the design member and calculation of deflections along the design member can be performed.

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

Ribbon groups **Concrete design**, **Calculation**, **Zone templates**, **View settings and scale**, **Internal forces**, **Detailed view** and **Bill of material** 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 screenshot displays the main window with a beam model and three reinforcement zones. The zones are labeled A-A, B-B, and A-A. The beam is supported at nodes 1, 2, 3, and 4. The distance between nodes 1 and 2 is 1.50m, between 2 and 3 is 6.00m, and between 3 and 4 is 1.50m. The B-B section is located between nodes 2 and 3. The beam has a diameter of $\phi 10\text{mm} \text{ à } 0.20\text{m}$.

The Data window shows the following table for reinforcement zones:



Reference point	Begin [m]	End [m]	Reinforcement	Check	Division
1	0,00	1,50	A-A	<input checked="" type="checkbox"/>	3
2	0,00	6,00	B-B	<input checked="" type="checkbox"/>	1
3	0,00	1,50	A-A	<input checked="" type="checkbox"/>	3

The Zone properties panel shows:




- Section position: Center
- Cross-section: T tvar 600, 450

The Details window shows a detailed view of the A-A cross-section, which is a T-section with a total height of 600mm, a web height of 440mm, a flange thickness of 105mm, a web width of 240mm, and a flange width of 450mm. The web width is divided into two 125mm segments on either side of the 240mm central section.

Table **Reinforcement zones** 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:
 -  - launches **Reinforcement editor** to input and modify the reinforcement in the current reinforcement template – see **Editor of reinforcement**.
 -  - creates 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. Also the **Reinforcement editor** is launched to define reinforcement in the newly created reinforcement template.

-  - displays dialog to modify the name of reinforcement template.
- **Check** – switch on/off the check of the zone. If the option is turned off, then no sections from this zone are generated in IDEA RCS.
- **Division** – input number of subzones, to which the current zone will be split. Sections for check are generated for each subzone.
-  - insert new zone. The current zone is split to two halves by inserting new zone.
-  - delete the current zone.

Properties group **Zone properties** – additional properties for zone on haunch:

- **Section position** – select the position on the subzone, where the section for check is generated.
- **Cross-section** - 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. Than the reinforcement is interpolated from the governing cross-section to the rest of haunch cross-sections.

11.3.1 Check positions for check of pre-stressed members

Reinforcement zones

Reference point	Begin [m]	End [m]	Reinforcement
1	0,00	1,00	A-A
3	0,00	5,58	B-B
3	5,58	18,10	C-C
3	18,10	23,66	B-B
4	0,00	1,00	A-A

Check position

Name	Reference point	Position [m]	Total position [m]	Check
Section 1	1	12,83	12,83	<input checked="" type="checkbox"/>
Section 2	1	0,43	0,43	<input checked="" type="checkbox"/>
Section 3	1	25,23	25,23	<input checked="" type="checkbox"/>
Section 4	1	1,10	1,10	<input checked="" type="checkbox"/>
Section 5	1	24,56	24,56	<input checked="" type="checkbox"/>

Details

Cross-section Results

C-C

Reinforcement:
 2ø10 (157mm²) (B 500B), z = 720 mm
 2ø28 (1232mm²) (B 500B), z = 715 mm
 2ø28 (1232mm²) (B 500B), z = 658 mm
 2ø10 (157mm²) (B 500B), z = 524 mm
 2ø10 (157mm²) (B 500B), z = 329 mm
 2ø10 (157mm²) (B 500B), z = 133 mm
 2ø10 (157mm²) (B 500B), z = -63 mm
 2ø10 (157mm²) (B 500B), z = -259 mm
 2ø10 (157mm²) (B 500B), z = -454 mm
 2ø10 (157mm²) (B 500B), z = -650 mm
 2ø20 (628mm²) (B 500B), z = -844 mm
 3ø20 (942mm²) (B 500B), z = -885 mm


Stirrups:
 ø8 (B 500B) - 200 mm, closed, for torsion check
 ø8 (B 500B) - 200 mm

As far as there are tendons in pre-stressed design member and the tendon position in the cross-section may vary along the design member, the proper check positions, where the check will be performed, must be defined.

Check positions can be defined in the **Check positions** table.

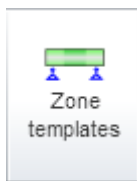
To add a new check position click  above the table.

Columns in the **Check positions** table:

- **Name** – input the name of position. The name is used to generate the name of section in the IDEA RCS module.
- **Reference point** – select the point of the design member, which the distance of section is related to.
- **Position** – input the distance between the check position and the reference point.
- **Total position** – displays the distance between the check position and the beginning of the design member.
- **Check** – switch on/off the check of the check position. If the option is turned off, then no sections for this check position are generated in IDEA RCS.
-  - delete the current check position.

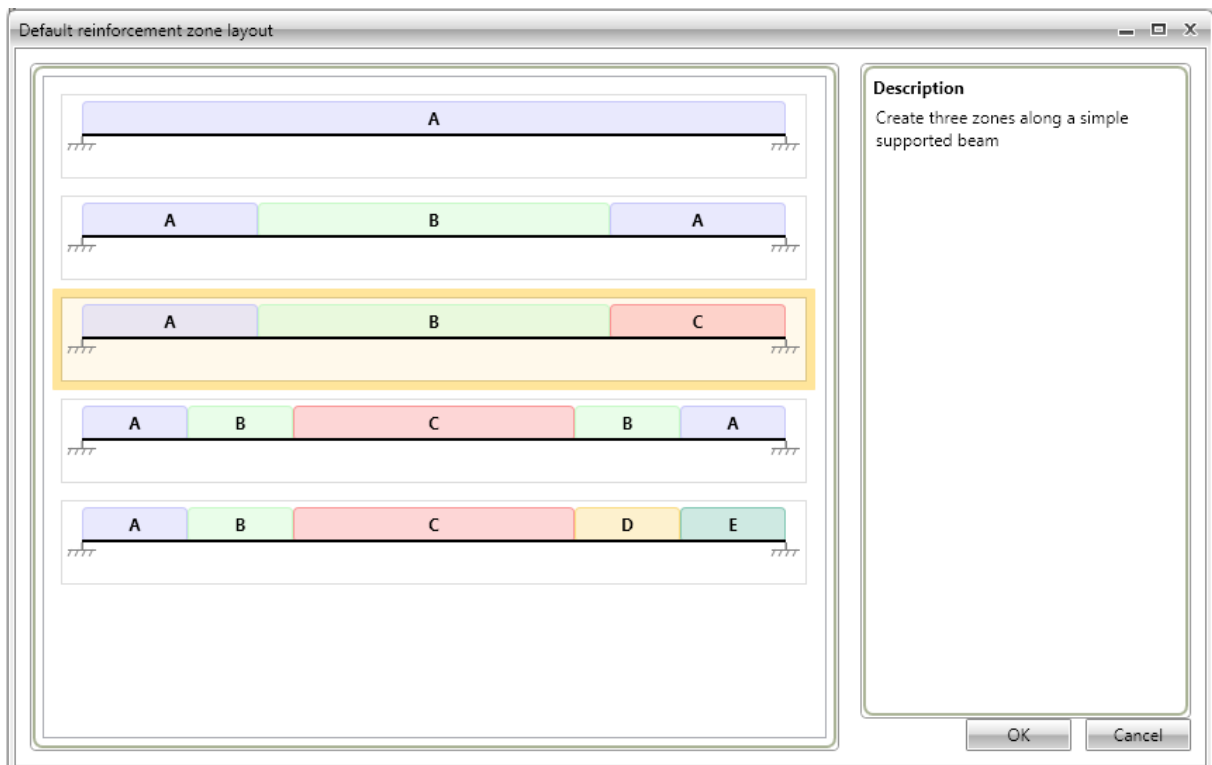
11.3.2 Zone templates

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

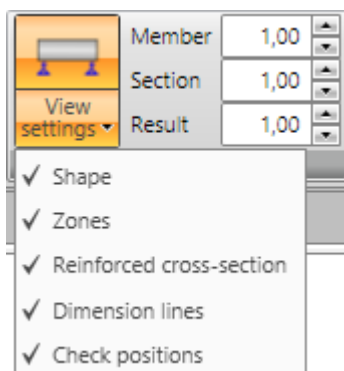


Zone patterns, which are suitable for the current design member, are displayed in the dialog **Default reinforcement zone layout**.

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



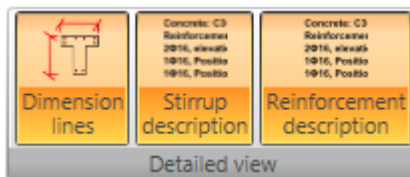
11.3.3 Ribbon group View settings and scale



Use options in ribbon group **View settings and scale** 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.
- **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...)

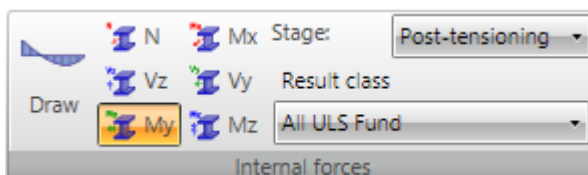
11.3.4 Ribbon group Detailed view



Use options in ribbon group **Detailed view** 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.

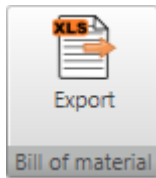
11.3.5 Ribbon group Internal forces



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. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Mx** – switch to drawing of torsional moment Mx. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **My** – switch to drawing of bending moment My.
- **Mz** – switch to drawing of bending moment Mz. This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Stage** – see **Results evaluation setup of staged beams**.
- **Result class** – see **Results evaluation setup**.


11.3.6 Ribbon group Bill of material



- **Export** – start export of bill of material of the current design member to the Microsoft Excel file.

11.4 Editor of reinforcement

Input of current reinforcement template can be started by:

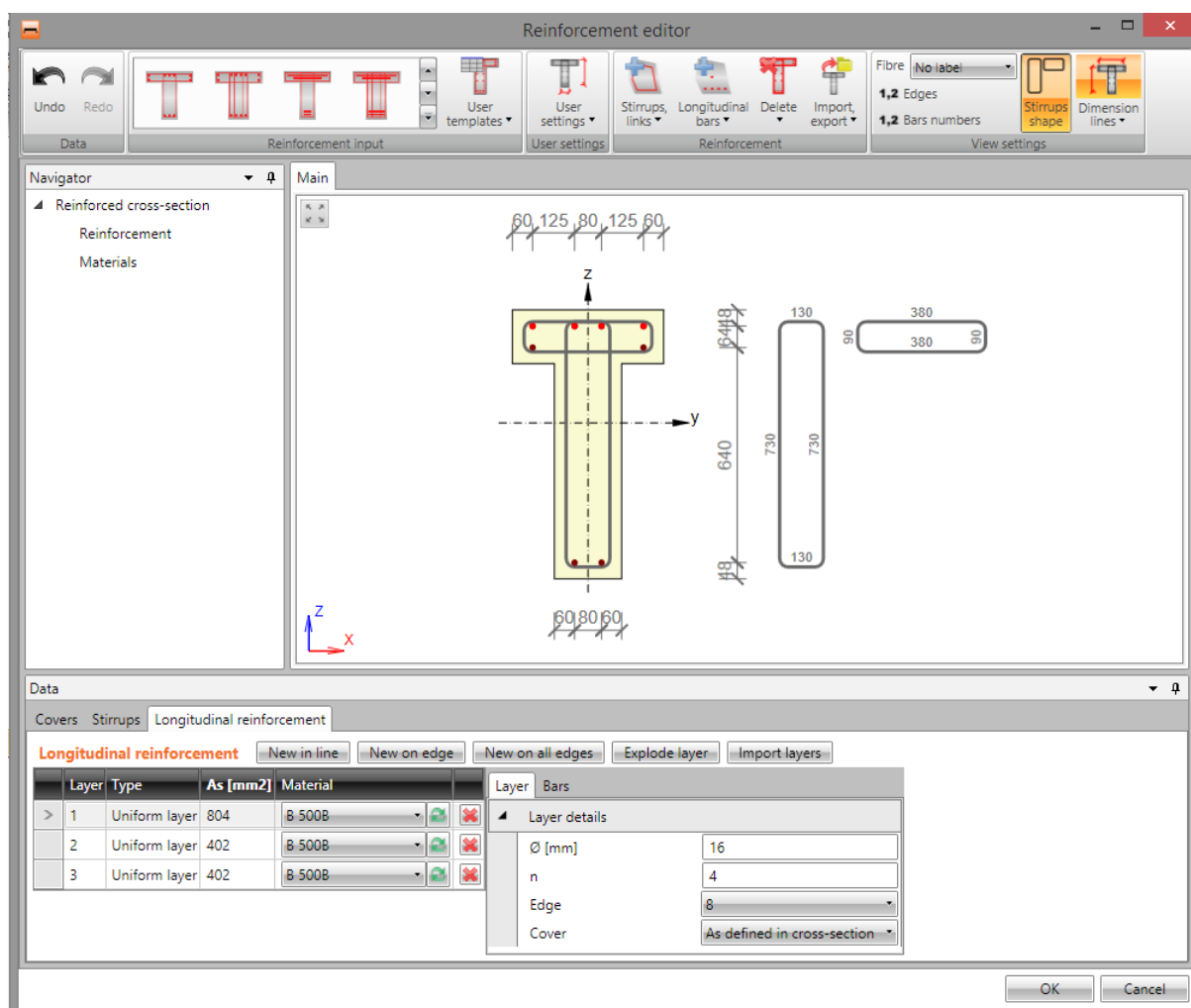
- Clicking the edit button  in the **Reinforcement** column in the zones table in the Data window
- Clicking the picture of section above the zone in the Main window.

Current reinforced section is displayed in the main window of reinforcement editor.

Following tabs are displayed in the data window of reinforcement editor:

- **Cover** – table of concrete cover on individual cross-section edges is displayed.
- **Stirrups** – tables of stirrups properties are displayed.
- **Longitudinal reinforcement** – tables of longitudinal reinforcement properties are displayed.

Ribbon groups **Reinforcement input**, **User settings**, **Reinforcement** and **View settings** are available.



11.4.1 Editing cover

Concrete cover at particular cross-section can be modified in table on tab **Cover**.

Cover can be defined:

- to individual cross-section edges, if option **All edges** is selected.
- to individual cross-section surfaces, if option **All edges** is not selected.

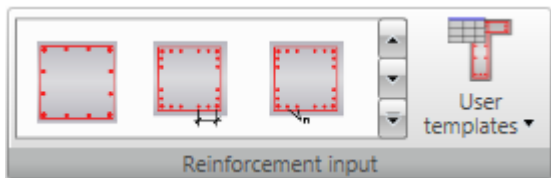
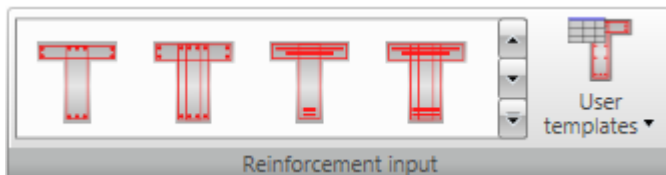
Covers All edges

	Type	Cover [mm]
>	Lower edge	30
	Upper edge	30
	Other edges	30

Covers All edges

	Edge	Cover [mm]
>	1	30
	2	30
	3	30
	4	30
	5	30
	6	30
	7	30
	8	30

11.4.2 Input of reinforcement by template

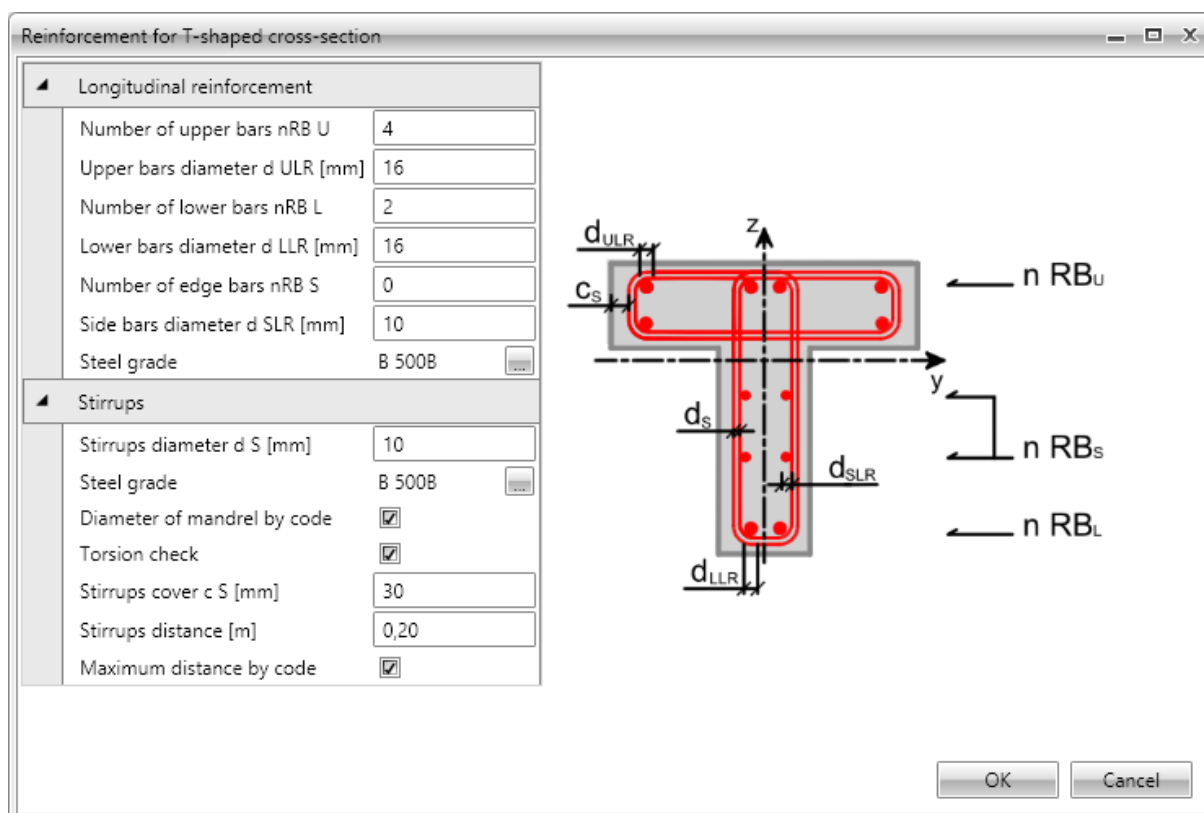


Reinforcement templates are available for some pre-defined sectional shapes. Reinforcement templates available for the current cross-section are displayed in ribbon group **Reinforcement**.

Click button with required reinforcement template to set the parameters of the inserted template in the settings dialog.

- **User templates** – input cross-section reinforcement using user defined templates of reinforcement – see **User defined reinforcement templates**.

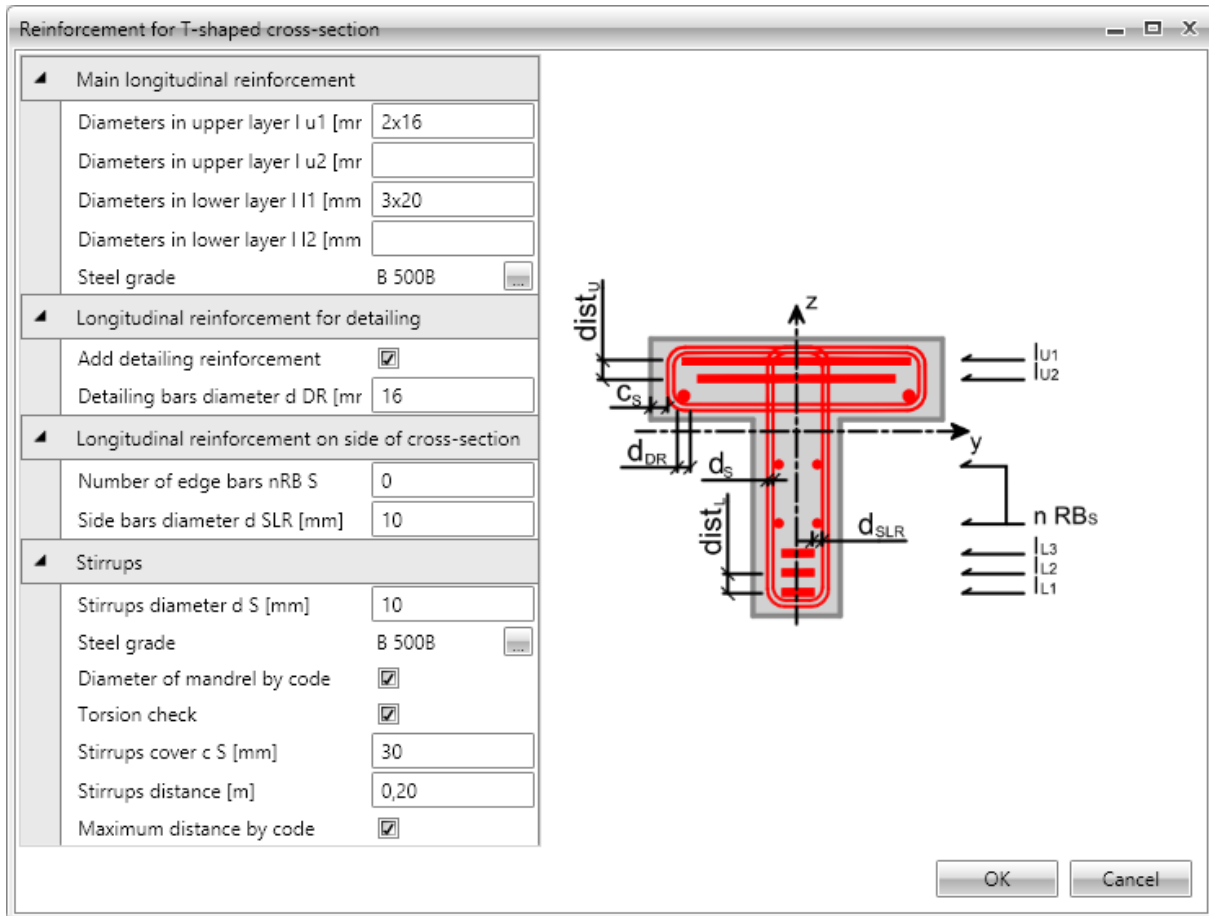
Reinforcement template parameters for T-shaped cross-section :



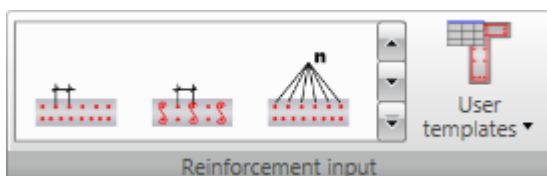
Click **OK** to add the reinforcement into the cross-section.

For some cross-sections templates with special definition of reinforcement layout are available. Those templates enable to input reinforcement bars with different diameters in one reinforcement layer at once.

The reinforcement layer is defined by string, which describes diameters of individual bars in the layer. Individual diameters are separated by space, characters 'x' or '*' can be used to define multiple diameters, e. g. ,20 16 16 20' or ,20 2*16 20'.

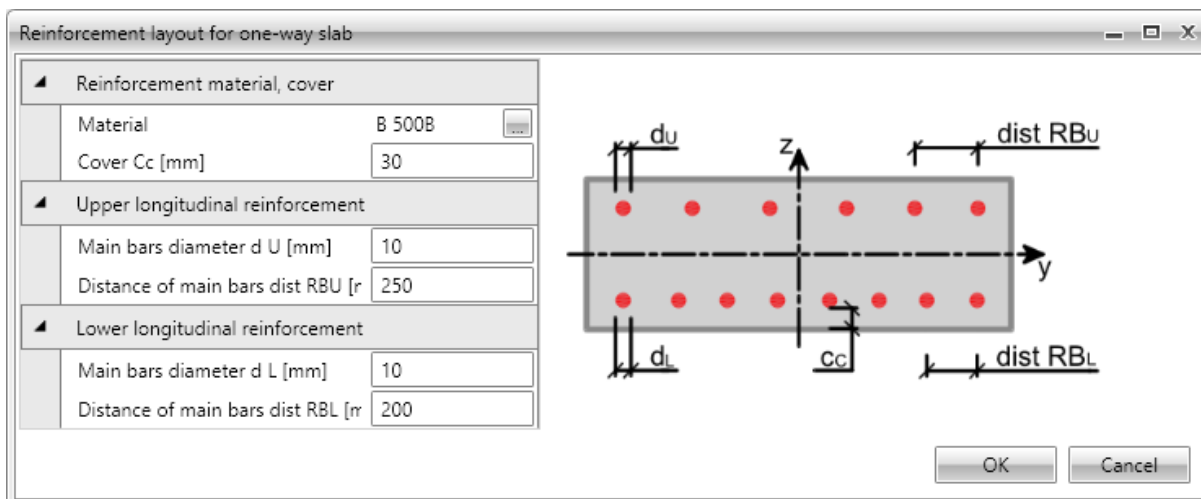


11.4.3 Input of one-way slabs reinforcement by a template



For one way slabs basic templates to define reinforcement at particular faces are available. Available templates are displayed in the ribbon group **Reinforcement**.

- **User templates** – input cross-section reinforcement using user defined templates of reinforcement – see **User defined reinforcement templates**.

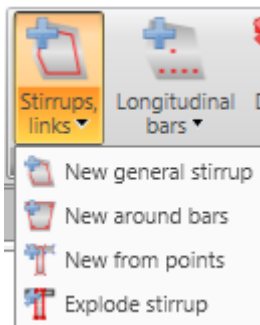


For one way slabs the distance between bars or bars number, bars diameter, bars material and concrete cover are defined in the template dialog.

11.4.4 Shear reinforcement

Shear reinforcement of beams and columns is defined using stirrups. Shear reinforcement of one-way slabs is defined using links.

11.4.4.1 Stirrups



Drop-down button **Stirrups, links** in ribbon group **Reinforcement** collects commands for stirrups operations:

- **New general stirrup** – adds a new stirrup by coordinates of stirrup vertices and stirrup diameter.
- **New around bars** – adds a new stirrup by vertices defined by selection of longitudinal reinforcement bars.
- **New from points** – adds a new stirrup by vertices defined by selection of cross-section vertices.
- **New links** – adds a new layer of links into the cross-section of beam-slab.
- **Explode stirrup** – stirrups defined by templates can be transformed to a generally defined (general) stirrup with editable vertices. Particular vertices of stirrup than can be edited as by stirrup defined by points.

Stirrups defined in cross-section are displayed on tab **Stirrups** in the data window in table **Stirrups**. Properties of selected stirrup are displayed in property table.

Columns in **Stirrups** table:

- **Type** – mode of stirrup definition is displayed.
- **Ø** – input value of stirrup diameter.
- **Material** – select stirrup material.
- **Distance** – input value of longitudinal distance between stirrups.
- **Shear** – if the checkbox is checked, stirrup is taken into account for shear check.
- **Torsion** – if the checkbox is checked, stirrup is taken into account for torsion check.

Data

Covers Stirrups Longitudinal reinforcement

Stirrups New New around bars New from points Explode stirrup

	Type	Ø [mm]	Material	Distance [mm]	Shear	Torsion	Stirrup	Vertexes
>	1	Vertexes derived from shape 8	B 500B	200	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Stirrup detail
	2	Vertexes derived from shape 8	B 500B	200	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	n dm 2,5

11.4.4.1.1 General stirrups

The general stirrup shape is defined by coordinates of the stirrup vertices. A vertex is the intersection of two stirrup branches axes.

To input new general stirrup click **Stirrups, links > New general stirrup** in ribbon group **Reinforcement** or click **New** above the table **Stirrups**.

Properties group **Stirrup detail**:

- **n dm** – input value of mandrel diameter.
- **Closed** – if the option is selected, stirrup branch between first and last vertex is created automatically.
- **Origin** - vertex coordinates are related to point, which is selected in the list:
 - **Point [0,0]** – vertex coordinates are related to origin of cross-section coordinate system.
 - **Cross-section vertex** – vertex coordinates are related to vertex, which is selected in list below.

	Y [mm]	Z [mm]	Ycg [mm]	Zcg [mm]		
>	50	50	-100	-200		
	250	50	100	-200		
	250	450	100	200		
	50	450	-100	200		

Stirrup vertices are defined in the table on **Vertices** tab. Coordinates can be copied from Microsoft Excel table also.



- **Y**,
- **Z** – input vertex coordinate related to the selected origin.
- **Ycg**,
- **Zcg** – vertex coordinate related to the centroid of cross-section is displayed.
- - adds new vertex row to the table..
- - deletes appropriate row from the table.

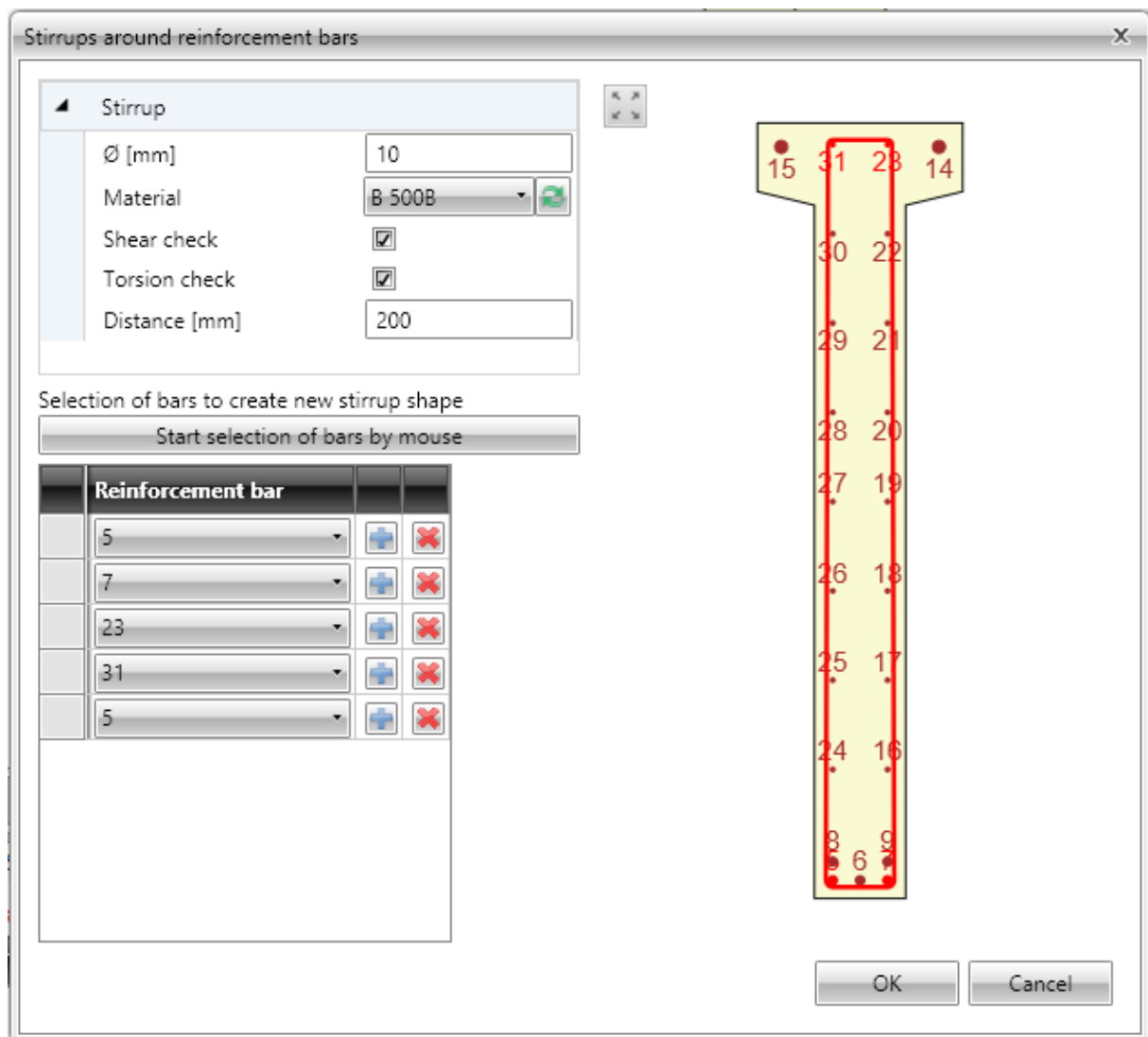
11.4.4.1.2 Stirrups around bars of longitudinal reinforcement

To input stirrup around main reinforcement click **Stirrups, links > New around bars** in ribbon group **Reinforcement** or click **New around bars** above the table **Stirrups**.

Stirrup shape is defined by selection of longitudinal reinforcement bars.

Following two options are available to create selection of bars:

- Sequential selection of bar numbers in **Reinforcement bar** lists .Click  to add new item behind current row. Click  to delete current row.
- Gradually, bars of longitudinal reinforcement are selected by mouse. Stirrup is generated around selected bars. Selected bars are listed in **Reinforcement bars** list. After selection of bars is finished, list of bars (stirrup vertices) can be edited.



Particular dialog options:

- **Ø** – input value of stirrup diameter.
- **Material** – select or edit material of stirrup.
- **Shear check** – if selected, stirrup is taken into account for shear check.
- **Torsion check** – if selected, stirrup is taken into account for torsion check.

- **Distance** – input value of longitudinal distance between stirrups
- **Start selection of bars by mouse** – click to start selection of bars to create stirrup around them.

If the selection is in progress, command **Start selection of bars** is replaced by commands:

- **Finish selection of bars** – finishes selection of bars, Close stirrup and Step back disappear. Stirrup is not closed automatically.
- **Close stirrup** – closes stirrup creating branch between first and last defined point, finishes selection of bars.
- **Step back** – deletes last defined stirrup branch.



11.4.4.1.3 Stirrup by cross-section vertices

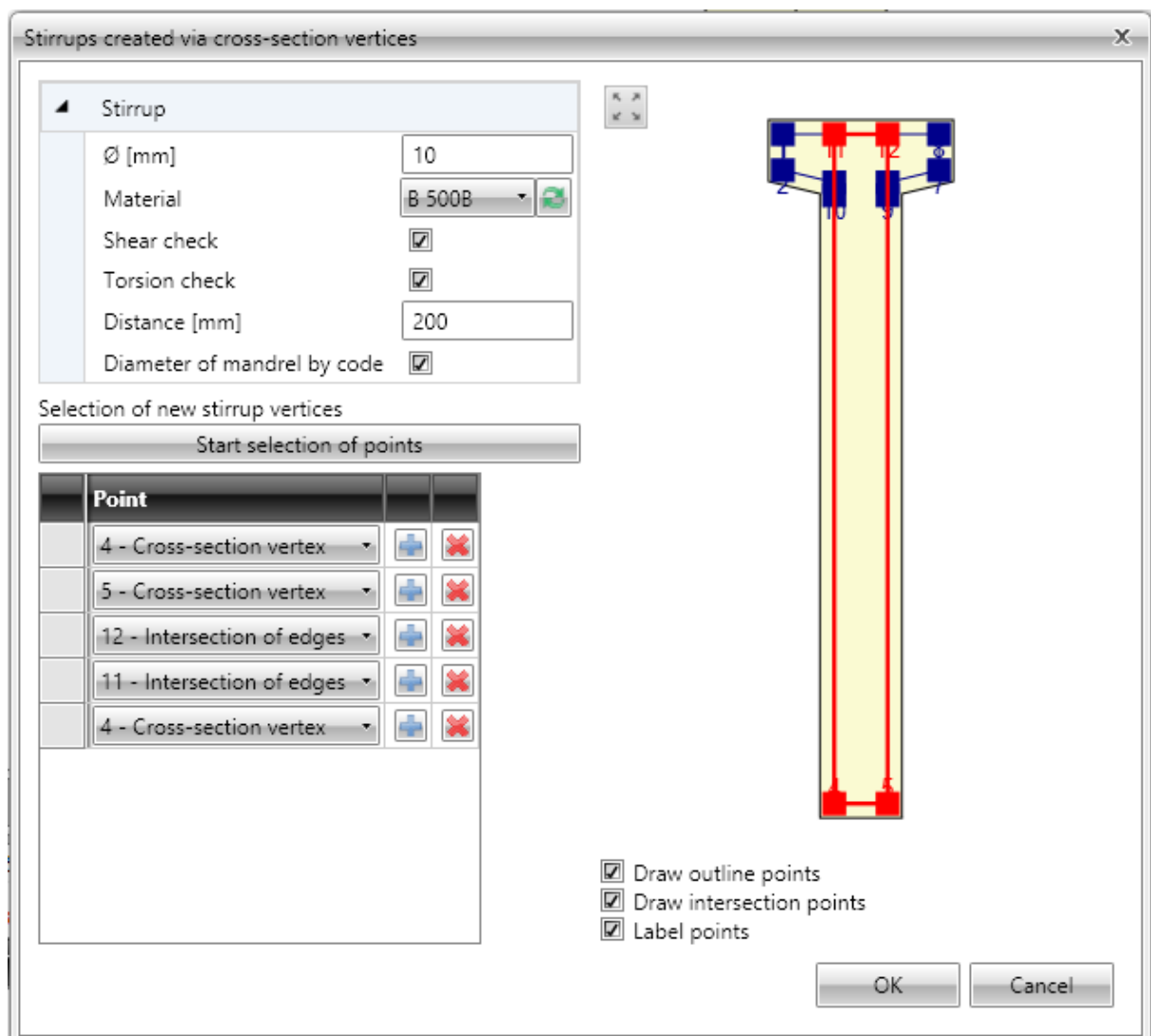
To input stirrup around main reinforcement click **Stirrups, links > New from points** in ribbon group **Reinforcement** or click **New from points** above the table **Stirrups**.

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

Points are selected by mouse in the picture of cross-section. The created stirrup passes through selected points.

Following two options are available to create stirrup vertices:

- Sequential selection of vertex number in **Point** lists .Click  to add new item behind current row. Click  to delete current row.
- Gradually, points are selected by mouse. Stirrup is generated by selected points. Selected points are listed in **Point** list. After selection of points is finished, list of points (stirrup vertices) can be edited.



Particular dialog options:

- **Ø** – input value of stirrup diameter.
- **Material** – select or edit material of stirrup.

- **Shear check** – if selected, stirrup is taken into account for shear check.
- **Torsion check** – if selected, stirrup is taken into account for torsion check.
- **Distance** – input value of longitudinal distance between stirrups.
- **Diameter of mandrel by code** – switch on/off automatic determination of stirrup mandrel diameter by national code.
 - **ndm** – input value of mandrel diameter.
- **Start selection of points** – click to start selection of points to create stirrup.
If the selection is in progress, command **Start stirrup shape definition** is replaced by commands:
 - **Finish selection of points** – finishes selection of points, **Close stirrup** and **Step back** disappear. Stirrup shape is not closed automatically.
 - **Close stirrup** – closes stirrup creating branch between first and last defined point, finishes selection of bars.
 - **Step back** – deletes last defined stirrup branch.
- **Draw outline points** – switch on/off drawing of points in vertices of the cross-section outline offset. The offset corresponds to the cover defined at particular cross-section edges.
- **Draw opening points** – switch on/off drawing of points in vertices of the cross-section opening offset. The offset corresponds to the cover defined at particular opening edges.
- **Draw intersections points** – switch on/off drawing of points in intersections of offset edges of cross-section outline and cross-section opening.
- **Label points** – switch on/off drawing of numbers of points.

11.4.4.1.4 Exploding stirrups


To convert stirrup to general stirrup defined by vertices click **Explode stirrup** in ribbon group **Stirrups**.

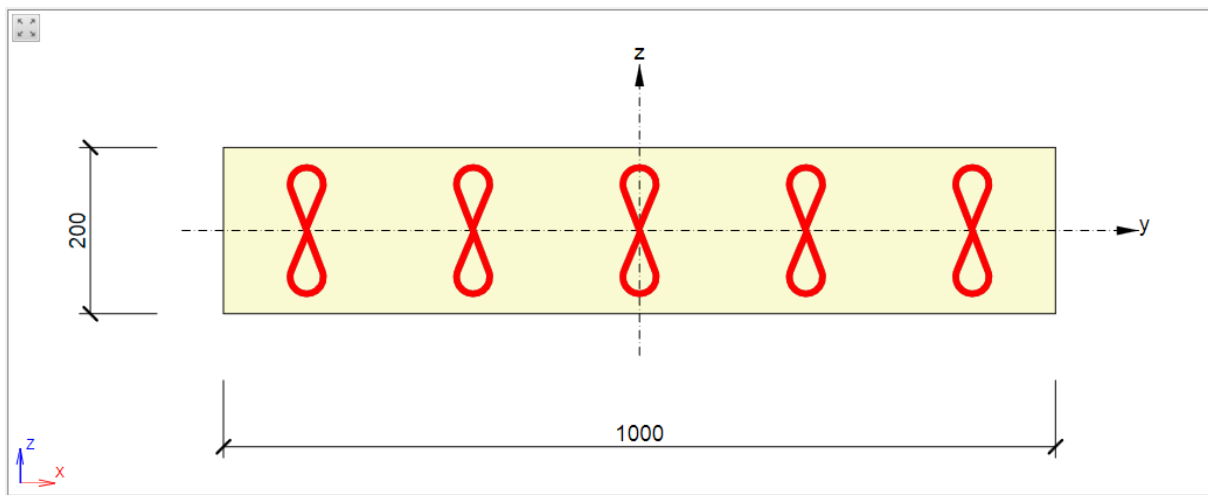
11.4.4.2 Links

Shear reinforcement of one-way slabs is defined using links.

Links defined in cross-section are displayed on tab **Links** in the data window in table **Layers of links**. Properties of selected link are displayed in property table.

Columns in **Link layers** table:

- \emptyset – input diameter of link bar.
- **Distance** – input the distance between axes of links in the plane of cross-section.
- **n** – the calculated number of links per meter is displayed.
- **As** – the reinforcement area of all links in the layer is displayed.
- **Ss** – input the distance between links along the design beam.
- **cu** – input the value of concrete cover at the top edge of the cross-section.
- **cl** – input the value of concrete cover at the lower edge of the cross-section.
- **ndm** – input the requested value of mandrel diameter as multiple of link bar diameter.
- **lbd** – input the requested value of anchorage length.
- **Material** – select material of link bar.
-  - delete links layer.



Data

Covers Longitudinal reinforcement Links

Layers of links

Link	\emptyset [mm]	Distance [mm]	n [-]	As [mm ²]	Ss [mm]	cu [mm]	cl [mm]	ndm [-]	lbd [mm]	Material	Link layer details
> 1	8	200	5,00	251	200	20	20	4,00	50	B 500B	Edge bar specification <input type="button" value="Symmetrically"/>

11.4.4.2.1 Links layer

To input new layer of links click **Stirrups, links > New links** in ribbon group Reinforcement or click **New** above the table **Layers of links**.

Link layer details

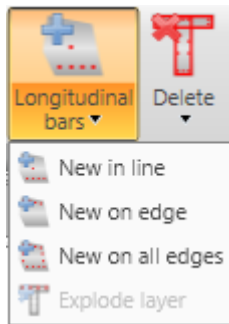
Edge bar specification

Edge bars distance [mm]

Properties group **Link layer details**:

- **Edge bar specification** – select mode of edge bar definition:
 - **Symmetrically** – links are positioned automatically to get the same distance between edge links and cross-section edges.
 - **User input**
 - **Edge bars distance** – input distance of first link from the cross-section edge.

11.4.5 Longitudinal reinforcement



Drop-down button **Longitudinal bars** in ribbon group **Reinforcement** collects commands for longitudinal reinforcement operations:

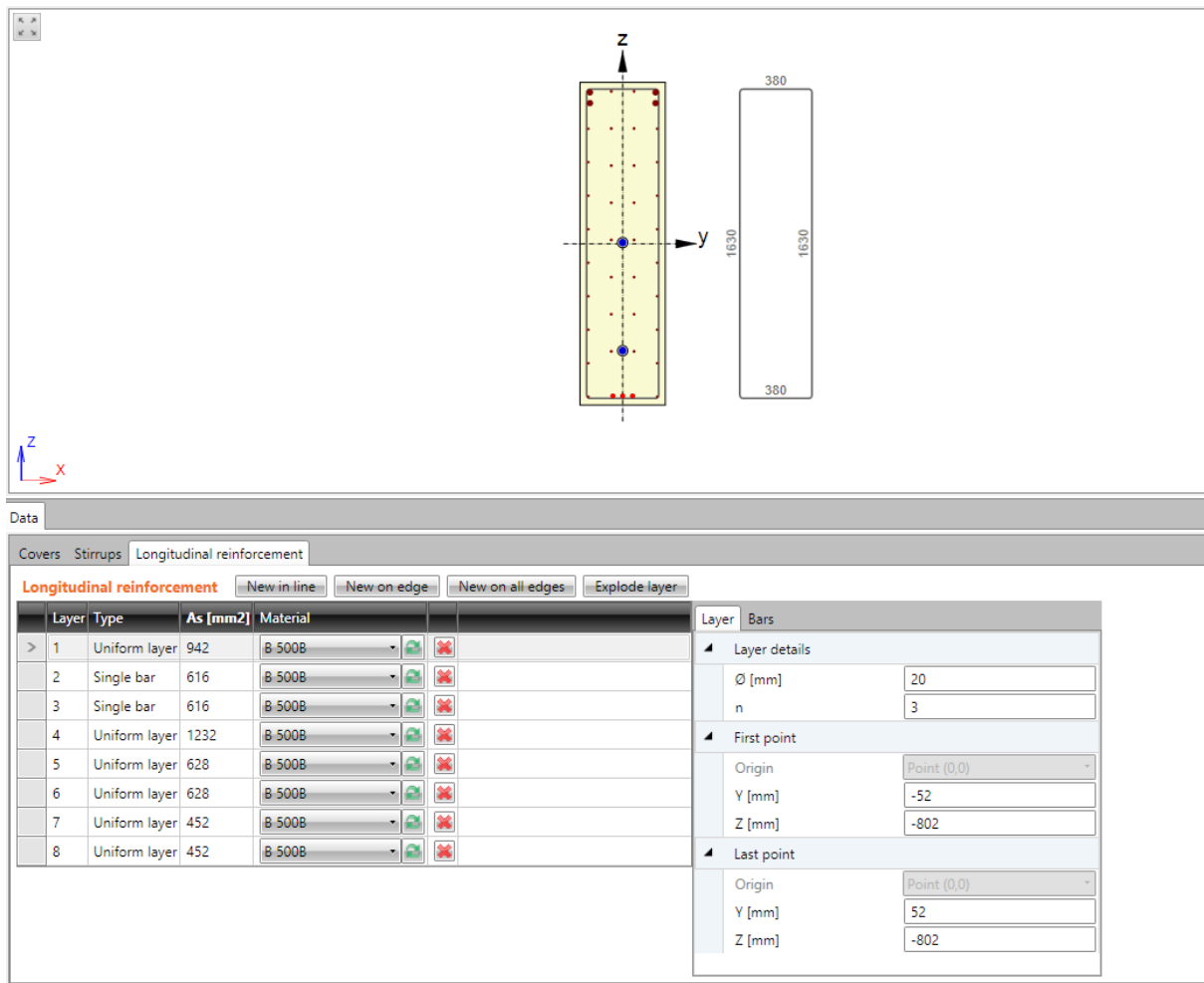
- **New in line** - adds a new layer of longitudinal reinforcement defined by coordinates of edge bars.
- **New on edge** - adds a new layer of longitudinal reinforcement related to cross-section edge.
- **New on all edges** - adds new layers of longitudinal reinforcement on all edges of cross-section.
- **New in waves** - adds a new layer of longitudinal reinforcement into the wave of trapezoidal plate. Command is available only for one-way slabs.
- **New by spacing** – input of new longitudinal reinforcement layer at the edge by the spacing of bars. This input mode is available for one way slabs only.
- **Explode layer** – the longitudinal reinforcement defined from templates can be transformed to separate longitudinal bars with editable coordinates. Exploding of reinforcement layer is not available for reinforcement of 2D members.

Longitudinal reinforcement is defined by layers. A layer is defined by the number of bars in the layer and position. Position can be specified by:

- coordinates of the first bar in the layer, and the coordinates of the last bar in the layer,
- edge, to which the layer is related and offsets of bars layer from the edge.

Bar diameter and material can be assigned to individual layers.

List of defined layers is displayed in the table **Longitudinal reinforcement** on the **Longitudinal reinforcement** tab in the data window. For the selected bars layer a table of properties is displayed.



The screenshot displays the 'Data' window in IDEA Beam, specifically the 'Longitudinal reinforcement' tab. At the top, a 3D model of a beam cross-section is shown with a coordinate system (x, y, z). The beam has a width of 380 mm and a height of 1630 mm. The reinforcement is represented by red dots and blue circles within the cross-section.


Below the model, the 'Data' window contains a table for 'Longitudinal reinforcement' and a 'Layer' details panel.

Layer	Type	As [mm ²]	Material		
> 1	Uniform layer	942	B 500B		
2	Single bar	616	B 500B		
3	Single bar	616	B 500B		
4	Uniform layer	1232	B 500B		
5	Uniform layer	628	B 500B		
6	Uniform layer	628	B 500B		
7	Uniform layer	452	B 500B		
8	Uniform layer	452	B 500B		

The 'Layer' details panel on the right shows the following settings:

- Layer details:**
 - Ø [mm]: 20
 - n: 3
- First point:**
 - Origin: Point (0,0)
 - Y [mm]: -52
 - Z [mm]: -802
- Last point:**
 - Origin: Point (0,0)
 - Y [mm]: 52
 - Z [mm]: -802

Columns in **Longitudinal reinforcement** table:

- **Type** – mode of layer definition is displayed.
- **As** – calculated value of reinforcement area in layer is displayed.
- **Material** – materials select material of bars in reinforcement layer.
-  - delete the appropriate reinforcement layer.

11.4.5.1 Layer of reinforcement by coordinates

To input new layer of reinforcement defined by coordinates click **Longitudinal bars > New in line** in ribbon group **Reinforcement** or click **New in line** above the table **Longitudinal reinforcement**.

Layer	Bars
Layer details	
∅ [mm]	16
n	2
First point	
Origin	vertex 1
Δ Y [mm]	50
Δ Z [mm]	50
Y [mm]	-175
Z [mm]	-800
Last point	
Origin	vertex 2
Δ Y [mm]	-50
Δ Z [mm]	50
Y [mm]	175
Z [mm]	-800

Properties of reinforcement layer specified by coordinates on **Layers** tab:

Properties group **Layer details**:

- **∅** – input diameter of bars in reinforcement layer.
- **n** – input number of bars in reinforcement layer.

Properties group **First point**:

- **Origin** – select origin, to which coordinates of first bar in layer are related. Position of bar can be related to point [0;0] (center of gravity) or to selected cross-section vertex.
- **ΔY**,
- **ΔZ** – input distance of first bar in layer to the selected origin in direction of the corresponding axis.
- **Y**,
- **Z** – coordinates of first bar in layer to the center of gravity in direction of the corresponding axis are displayed.

Properties group **Last point**:

- **Origin** – select origin, to which coordinates of last bar in layer are related. Position of bar can be related to point [0;0] (center of gravity) or to selected cross-section vertex.
- **ΔY**,
- **ΔZ** – input distance of last bar in layer to the selected origin in direction of the corresponding axis.
- **Y**,

- **Z** – coordinates of last bar in layer to the center of gravity in direction of the corresponding axis are displayed

11.4.5.1.1 Layer details

Layer		Bars						
	Bar	\emptyset [mm]	Y [mm]	Z [mm]	Bent-up	sb [m]	α_{xz} [°]	α_{yz} [°]
>	5	16	-175	-800	<input type="checkbox"/>			
	6	16	175	-800	<input type="checkbox"/>			

Properties of individual bars in current layer are displayed on **Bars** tab . Columns in the table:

- **Bar** – index of bar is displayed.
- \emptyset – bar diameter is displayed.
- **Y**,
- **Z** – distance of bar centre from cross-section centroid in direction of appropriate axis is displayed.
- **Bent-up** – switch on/off the bent-up bar.
- **sb** – input distance between individual bent-ups.
- **α_{XZ}** – input angle of bent-up bar in XZ plane of cross-section (to longitudinal axis of member).
- **α_{YZ}** – input angle of bent-up bar in YZ plane of cross-section (to longitudinal axis of member).

11.4.5.2 Layer of reinforcement on edge

To input new layer of reinforcement on cross-section edge click **Longitudinal bars > New on edge** in ribbon group **Reinforcement** or click **New on edge** above the table **Longitudinal reinforcement**.

Layer	Bars
▲ Layer details	
∅ [mm]	16
n	2
Edge	1
Cover	As defined in cross-section

Properties of reinforcement layer on cross-section edge on **Layers** tab:

Properties group **Layer details**:

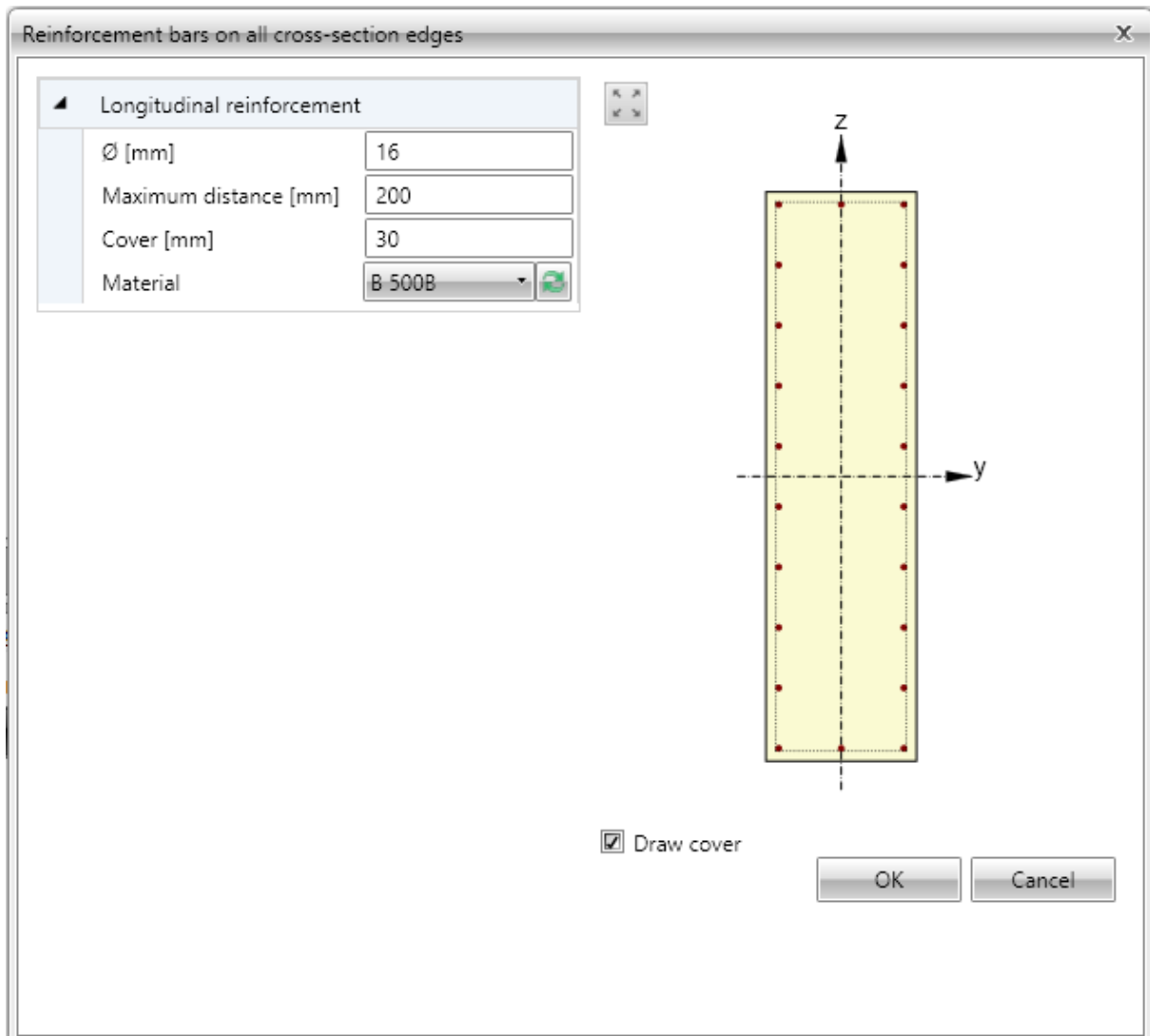
- **∅** – input diameter of bars in reinforcement layer.
- **n** – input number of bars in layer.
- **Edge** – select edge, to which layer of reinforcement is related.
- **Cover** – select mode of cover determination in the list. Following modes can be selected:
 - **As defined in cross-section** – values of cover are taken from cross-section shape. Existing stirrups are taken into account.
 - **User defined** – values of cover can be entered in columns **Edge cover**, **Left cover**, **Right cover**.

Properties of individual bars of the layer are displayed on tab **Bars** – see **Layer details**.

11.4.5.3 Layers of reinforcement on all edges

To input new layers of reinforcement on all cross-section edges click **Longitudinal bars > New on all edges** in ribbon group **Reinforcement** or click **New on all edges** above the table **Longitudinal reinforcement**.

One reinforcement layer is created on each edge of cross-section. Number of bars on the edge is determined automatically respecting the given maximal distance between bars and bars diameter.



Individual dialog options:

- **Ø** – input diameter of bar in layers.
- **Maximal distance** – input the maximal distance between bars to determine the number of bars on edge.
- **Cover** – input the value of concrete cover, common for all edges.
- **Material** – in the list of available materials select material of bars in reinforcement layer or click edit button to edit material properties.
- **Draw cover** – switch on/off drawing of concrete cover.

Properties of individual bars of selected layer are displayed on tab **Bars** - see [Layer details](#).

11.4.5.4 Layer of reinforcement to wave of trapezoidal sheet

To input new layers of reinforcement of one-way slab defined by spacing click **Longitudinal bars > New by spacing** in ribbon group **Reinforcement** or click **New by spacing** above the table **Longitudinal reinforcement**.

Layer	Bars
Layer details	
Ø [mm]	10
n	4
Number of bars in wave	1
Concrete cover [mm]	10

Properties group **Layer details**:

- **Ø** – input diameter of bar in layers.
- **n** – the calculated number of bars per meter in layer is displayed.
- **Number of bars in wave** – input number of bars in each wave of cross-section.
- **Cover** – input the value of concrete cover.

Properties of individual bars of selected layer are displayed on tab **Bars** - see [Layer details](#).

11.4.5.5 Layer of reinforcement by spacing

To input new layers of reinforcement of one-way slab defined by spacing click **Longitudinal bars > New by spacing** in ribbon group **Reinforcement** or click **New by spacing** above the table **Longitudinal reinforcement**.

Layer	Bars
Layer details	
Ø [mm]	10
Distance [mm]	200
n [-]	5,00
Edge bar specification	User input
Edge bars distance [mm]	100
Surface to cover	Lower
Cover [mm]	20

Layer is defined by face, distance between bars, distance of edge bar and concrete cover.

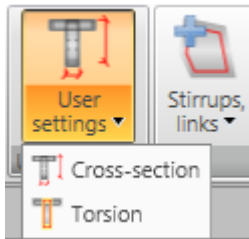
Properties group **Layer details**:

- **Ø** – input the diameter of bar in the layer.
- **Distance** – input the distance between axes of adjacent bars.
- **n** – the calculated number of bars per meter in layer is displayed.
- **Edge bar specification** – select the mode to determine the position of the edge bar. One of following modes can be selected:
 - **Symmetrically** – the distance of the first bar from the edge is calculated in such way, that the distances of both edge bars from the edges is the same.

- **Diameter/2** – the distance of first bar from the edge is set as a half of the bar diameter.
- **User input** – the required value of the edge bar distance can be defined.
 - **Edge bar distance** – input the required value of the edge bar distance (or the calculated value is displayed).
- **Surface to cover** – select the face, to which the layer is defined.
- **Cover** – input the value of concrete cover.

Properties of individual bars of selected layer are displayed on tab **Bars** - see [Layer details](#).

11.4.6 User settings of reinforced cross-section



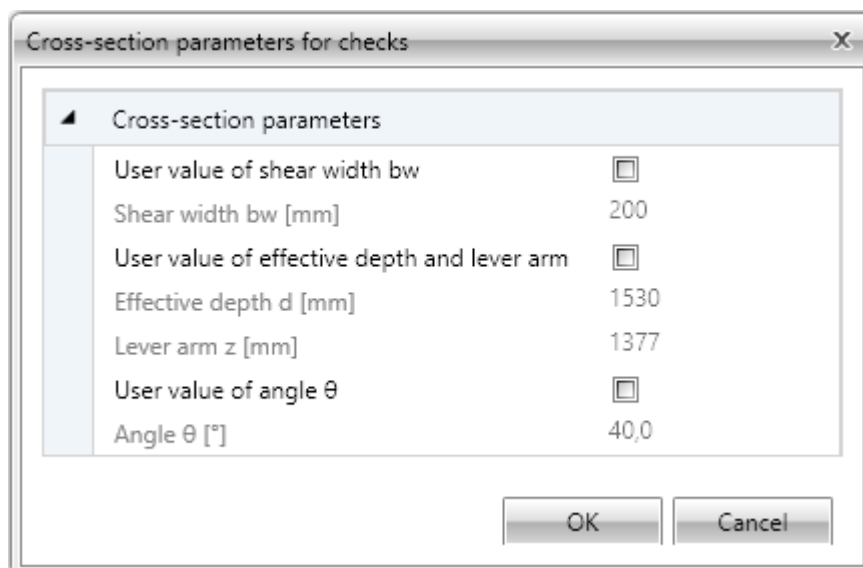
Ribbon group **User setting** contains commands to modify some shear and torsion calculation parameters of reinforced cross-section:

- **Set for shear** – input of user defined dimensions of effective cross-section for shear check.
- **Set for torsion** – input or modification of equivalent thin-shaped cross-section for check of torsion.

11.4.6.1 Input of effective cross-section for shear

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

To input dimensions of effective cross-section for shear click **Set for shear** in ribbon group **User settings**.

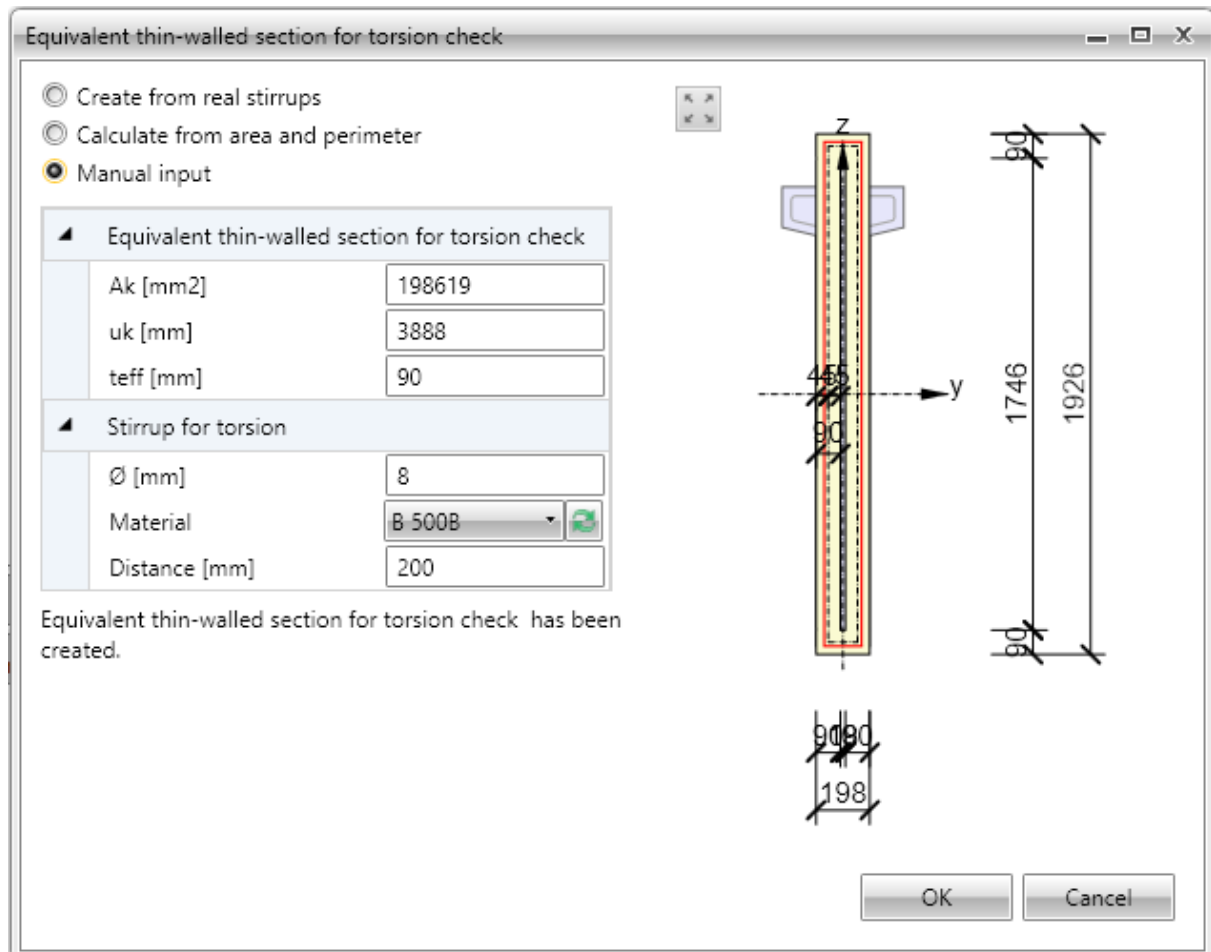


Calculated values of effective cross-section dimensions and overridable values of shear check from code and calculation settings are displayed in dialog. To enable input of user defined values check particular checkbox in first column.

11.4.6.2 Input of equivalent cross-section for torsion

Equivalent thin-walled section is used for calculation of torsion. Equivalent cross-section can be calculated using:

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



Particular options of dialog:

- **Create from real stirrups** – create equivalent thin-walled cross-section using outlines of stirrups, which are marked as effective for torsion. If this option is active, it is possible to click **Start stirrup shape definition** and adapt shape of stirrups for check of torsion.
 - **Start stirrup shape definition** – displays dialog, where shape of stirrup for determination of equivalent cross-section can be edited. Input of shape is done similarly to input of new stirrup shape using cross-section vertices
 - **Default stirrup shape** – restores shape of stirrup, which was defined as effective for torsion.
- **Calculate from area and perimeter** – calculate equivalent thin-walled cross-section using area and perimeter of original cross-section. Diameter, material and stirrups distance are taken from first stirrup, which is marked as effective for torsion.
- **Manual input** – values of area, perimeter and thickness of equivalent thin-walled cross-section including diameter, material and distance of stirrups are specified by user.

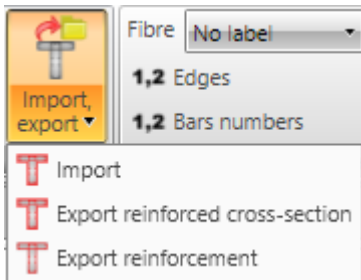
11.4.7 Deleting reinforcement



Use commands in ribbon group **Delete** to delete reinforcement from cross-section:

- **Selected** – delete selected layer or bar of reinforcement.
- **All** – delete all reinforcement.

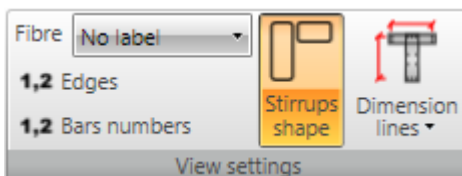
11.4.8 Import and export of reinforced cross-section



Drop-down button **Import, export** in ribbon group **Reinforcement** collects commands for import and export of reinforced cross-section:

- **Import** – starts import of cross-section shape including reinforcement from text file.
- **Export reinforced cross-section** – starts export of reinforced cross-section to file.
- **Export reinforcement** – starts export of reinforcement to file.

11.4.9 View settings of reinforced cross-section





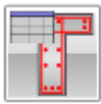
Ribbon group **View settings** can be used to set drawing options of reinforced cross-section:

- **Fibre** – select mode of fibres drawing in the list. One of following modes can be chosen:
 - **No labels** – description of fibres is not drawn.
 - **Outside** – fibre numbers are drawn outside the cross-section outline.
 - **Inside** – fibre numbers are drawn inside the cross-section outline.
- **Edges** – switch on/off drawing of numbers of edges.
- **Bar numbers** – turns on/off drawing of reinforcement bar numbers.
- **Stirrups shape** – turn on/off drawing of dimensioned stirrups outside the cross-section.
- **Dimension lines** – switch on/off drawing of dimension lines:
 - **Standard** – switch to drawing of standard dimension lines of reinforcement.
 - **Stationing** – switch to drawing of dimension lines with distances related to reference point.

11.4.10 User defined reinforcement templates

The existing reinforcement of cross-section can be stored into the database of user defined reinforcement templates. The stored template can be used to reinforce other sections in the current project or sections in other projects.

Following commands in dialog **User templates** are available to work with user templates of reinforcement:

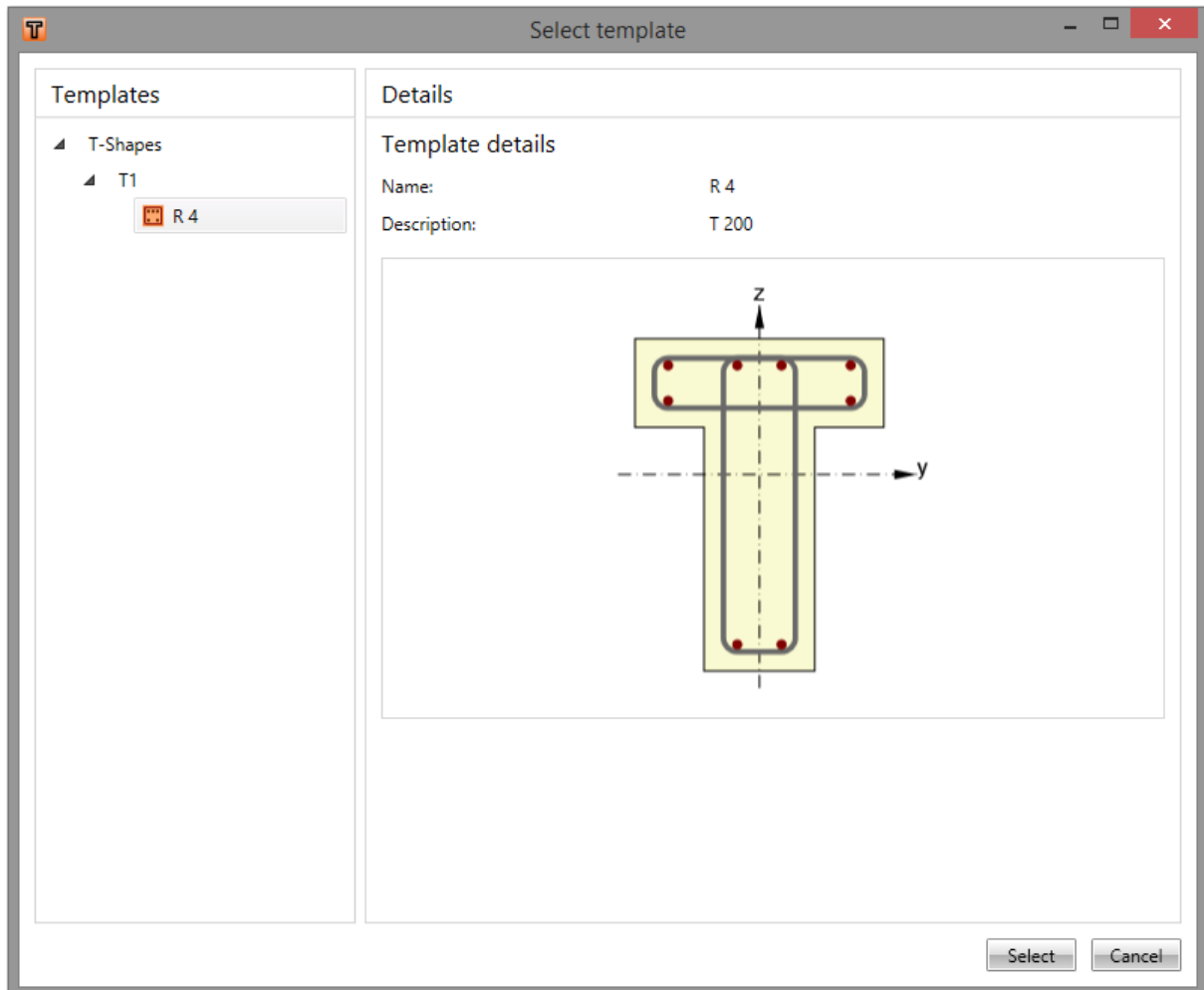
-  – starts input of reinforcement using the user defined reinforcement template – see **Reinforcing by user defined reinforcement template.**
-  – stores the current reinforcement into the database of user reinforcement templates. Dialog Add template appears. The target folder must be selected in the tree control in the left part of dialog. The current reinforcement is stored as a template into the selected folder.
-  – launches templates manager – see **Templates manager.**

11.4.10.1 Reinforcing by user defined reinforcement template

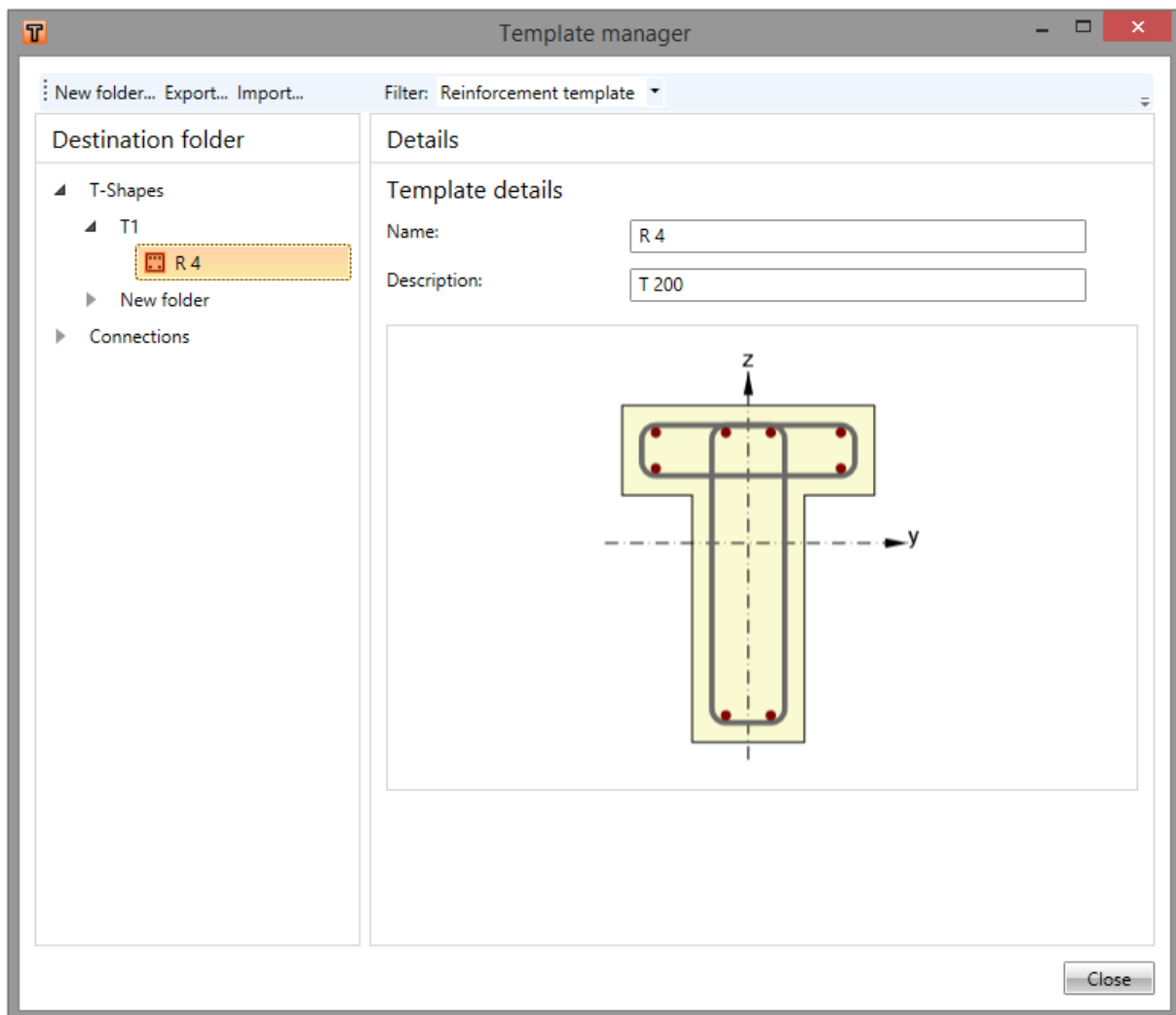
Dialog **Select template** appears after start of reinforcing by user defined reinforcement template.

Only templates, which have the same cross-section type as the reinforced cross-sections, 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 reinforce the cross-section using the selected template.



11.4.10.2 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.

11.5 Input of data for deflections check

Click navigator command **Concrete design 1D > Deflections** to input data for check of limit deflections.

Table with design member spans and a simplified picture of design member with supports is drawn in the Data window.

For each span the parameters of deflection check can be defined.

The screenshot shows the 'Data' window in the IDEA StatiCa software. The top part displays a beam model with spans of 1.00, 5.58, 12.52, 5.56, and 1.00 m, and a total length of 25.00 m. The 'Data' window below shows input fields for 'Effective time [d]' (5.0), 'Long-term losses coefficient' (Calculated), 'Span definition' (Direction: Plane XZ, Current span: 2, nodes 2 - 5), and 'Deflection limits' (Limit value for deflections as length of span / 250, Check acc. 7.4.1 (4) checked, Limit value for deflections as length of span / 500). The 'Required camber' is set to 0 mm. The 'Details' window shows the 'Results' for Plane XY (25.66 m) and Plane XZ (25.00 m).

- **Effective time** – input fictive time of application of all permanent loads selected in such a way that the error of long-term deflection calculation is minimized (for pre-stressed members).
- **Long-term losses coefficient** – select method to determine the coefficient which represents the level of prestressing after long-term losses relatively to the level after short-term losses (for pre-stressed members):
 - **Calculated** – the value of coefficient is determined automatically.
 - **User defined value** – the value of coefficient is specified by user.
- **Direction** – select plane, for which the deflection check settings are defined:
 - **Plane XZ** – parameters of deflection check in global XZ plane are defined (vertical deflection).
 - **Plane XY** – parameters of deflection check in global XY plane are defined (horizontal deflection).
- **Actual span** – set the current span in the list. The current span can be set by clicking the cell in the column **Length of span**.

For the current span and current deflection direction (plane) following parameters can be set in **Deflection limits** properties group:

- **Limit is defined as numerical value** – if selected, the values of limit deflection are defined as absolute value (in length units). Otherwise the limit deflection value is calculated as ratio of current span length.
- **User-defined value of limit deflection** – if selected, the ratio of span length to define the limit deflection can be defined by user.
- **Check acc. to 7.4.1 (4)** – switch on/off the deflection check with respect to possible impairment of the appearance and general utility of the structure acc. 7.4.1 (4).
 - **Limit value for deflection as length of span /** - display or input the ratio of span length to determine the limit value of deflection.
 - **Numerical value of deflection limit** – user-defined absolute value of deflection limit.
- **Check acc. to 7.4.1 (5)** – switch on/off the deflection check with respect to possible impairment of the appearance and general utility of the structure.
 - **Limit value for deflection as length of span /** - display or input the ratio of span length to determine the limit value of deflection.
 - **Numerical value of deflection limit** – user-defined absolute value of deflection limit.
- **Required camber** – input the value of camber at midspan. Deflection limit may be exceeded if camber is provided so that total deflection minus camber does not exceed limit (for pre-stressed members).

11.6 Reduction and redistribution of internal forces

Click navigator command **Concrete design 1D > Redistribution and reduction** to input parameters and calculation of redistributions and reductions of internal forces.

Design member and courses of evaluated values are drawn in the main window. Ribbon group Internal forces is available to modify the evaluation settings.

Tabs for input and evaluation of redistribution and reduction are displayed in the data window:

- **Supports definition** – setting of type of individual supports along the design member.
- **Internal forces** – tabular evaluation of modified internal forces.
- **Intermediate results** – tabular evaluation of intermediate results of modified internal forces calculation.

11.6.1 Supports definition for calculation of redistributions and reductions

Support conditions for calculation of reduction and redistribution of internal forces can be set on the tab Supports definition.

Node	Support width [m]	Beam or slab is
1	0.4	Continuous over a support
2	0.4	Continuous over a support
3	0.4	Continuous over a support
4	0.4	Continuous over a support
5	0.4	Continuous over a support

Settings panel:

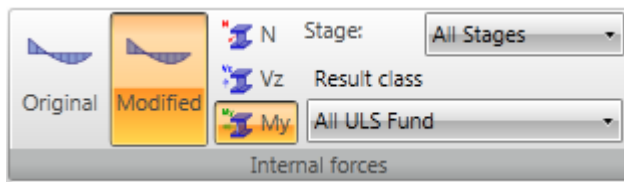
- Redistribution of moments:
- Reduction of moments:
- Reduction of shear force:
- Limited interaction check:
- Type of effective depth d: d = 0.9 * h

Table **Supports definition**:

- **All supports identical** – if the option is selected, all supports are considered to be the same when calculating redistribution and reduction. If the option is not selected, the support conditions can be defined for each support separately.
- **Support width** – input the support width for reduction of internal forces calculation.
- **Beam or slab is** – type of beam for reduction of internal forces calculation:
 - **Monolithic with support** – beam is considered to be monolithic with support.
 - **Continuous over a support** – beam is considered to be continuous over the support.
- **Redistribution of moments** – switch on/off calculation of moments redistribution according to EN 1992-1-1, art. 5.5.
- **Reduction of moments** – switch on/off calculation of reduced moments in supports according to EN 1992-1-1, art. 5.3.2.2(3) a 5.3.2.2(4).
- **Reduction of shear force** – switch on/off calculation of reduced shear force for members with loads near supports according to EN 1992-1-1, art. 6.2.2(6) and 6.2.3(8). The way to determine effective height can be selected in **Type of effective depth d** property:
 - Calculate according to formula $d = 0.9 \cdot h$;
 - **User defined**;
 - Determine according to compression strut angle θ .

- **Limited interaction check** – switch on/off limitation of interaction check in distance less than d from position of maximal moment according to EN 1992-1-1 6.2.3(7).

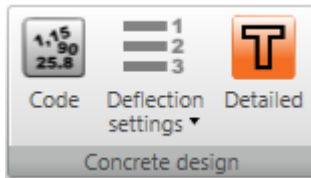
11.6.2 Ribbon group Internal forces



Use commands in ribbon group **Internal forces** to set the drawing mode of modified internal forces along the current design member.

- **Original** – switches on/off drawing of courses of not modified internal forces..
- **Modified** – switches on/off drawing of courses of internal forces taking into account calculated reduction and redistribution.
- **N** – switches to draw the course of axial force N .
- **Vz** – switches to draw the course shear force Vz .
- **Vy** - switches to draw the course shear force Vy . This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Mx** – switches to draw the course of torsional moment Mx . This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **My** – switches to draw the course of bending moment My .
- **Mz** – switches to draw the course of bending moment Mz . This option is available only for **Straight or polygonal beam loaded in 3D** beam type.
- **Stage** – see **Results evaluation setup of staged beams**.
- **Result class** – see **Results evaluation setup**.

11.7 Detailed check



Click **Detailed** in ribbon group **Concrete design** to run the detailed concrete check in the module IDEA RCS.

The data for detailed check in IDEA RCS are generated with respect to the input data (cross-section, zones, reinforcement) and the result classes associated to combinations for check. Besides the detailed section check, the reinforcement can be edited in IDEA RCS. Changes of reinforcement are transferred back to IDEA Beam and are respected in calculation of deflections.

11.8 Results evaluation

Click navigator command **Concrete design 1D > Results** to start the checks and calculation of deflections and its evaluation.

Results can be evaluated:

- Graphically – the courses of evaluated magnitude are drawn in the Main window.
- Textually – the textual presentation of results is printed to tables on tabs in the Data window. Following tabs are available:
 - **Summary** – the summary tables of section check, deflection check and the input data are printed in the table.
 - **Redistribution and reduction** – tables with intermediate results of internal forces redistribution and reduction are printed in the table.
 - **Section check** – the detailed output of reinforced sections check is printed in the table
 - **Deflections check** – the detailed output of deflection calculation and limit deflection check is printed in the table.

Following results may be evaluated graphically:

- course of summary section check along the design member
- courses of individual section check along the design member
- interaction diagrams for individual zones on the design member.

Ribbon groups **Concrete design**, **View settings and scale**, **Extreme**, **Calculation**, **Results drawing** and **Report** are available, when evaluating results.

Ribbon group **Check** and **Stage** are available for evaluation of section check results.

Ribbon groups **Combinations**, **Type of results** and **Stiffness** are available for evaluation of deflection calculations.

Ribbon groups **Stage**, **Interaction surfaces sections**, **Drawing settings** and **Colors settings** are available for evaluation of interaction diagrams.

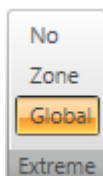
11.8.1 Ribbon group Concrete design

See [Ribbon group Concrete design](#) and [Detailed check](#).

11.8.2 Ribbon group View settings and scale

See [Ribbon group View settings and scale](#).

11.8.3 Ribbon group Extreme

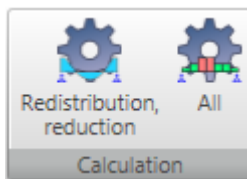


The drawing of check results labels can be modified using commands in this ribbon group.

- **No** – the extreme value of check of each reinforced subzone is found and displayed in the picture of check result course.
- **Zone** – the extreme value of check of each reinforced zone is found and displayed in the picture of check result course.

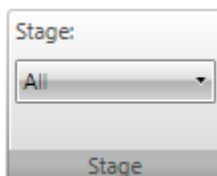
- **Global** – the extreme value of check of the whole design member is found and displayed in the picture of check result course.

11.8.4 Ribbon group Calculation



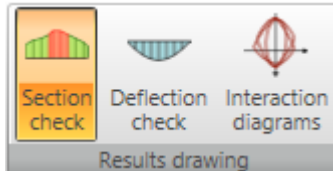
- **Redistribution, reduction** – run calculation of redistributed and reduced internal forces on the current design member. The command is available after change of parameters of redistribution and reduction calculation.
- **All** – run calculation of checks results of the current design member. This command is available, if the check results were deleted e.g. after the calculation settings were changed.

11.8.5 Ribbon group Stage



For staged beams, a current stage can be selected from the **Stage** list. The results are drawn for the selected stage.

11.8.6 Ribbon group Results drawing



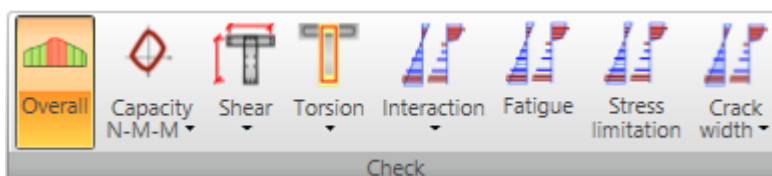
Use options in ribbon group **Results drawing** to set the mode of graphical evaluation of results:

- **Section check** – switch to evaluation of section check results
- **Deflection check** – switch to evaluation of deflection check and calculated stiffnesses.
- **Interaction diagrams** – switch to drawing of interaction diagrams for selected zone or sub-zone of the design member.

11.8.7 Drawing of section check results courses

11.8.7.1 Ribbon group Check

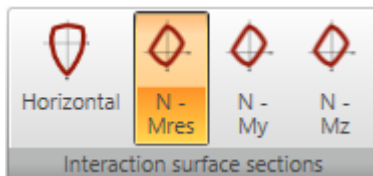
Use options in ribbon group **Check** to set the evaluation of particular check result along the design member.



- **Overall** – switch to drawing of envelope of extremes of all performed checks.
- **Capacity N-M-M** – switch to drawing of course of one of available components of capacity check – check value, bending moments of resistance, axial force of resistance.
- **Shear** – switch to drawing of course of one of available components of shear check – check value, $V_{Rd,c}$, $V_{Rd,max}$, $V_{Rd,s}$.
- **Torsion** – switch to drawing of course of one of available components of torsion check – check value, $T_{Rd,c}$, $T_{Rd,max}$, $T_{Rd,s}$.
- **Interaction** – switch to drawing of course of one of available components of interaction check – check value, check value $V+T$, check value $V+T+M$.
- **Fatigue** – switch to drawing of course of fatigue check results.
- **Stress limitation** – switch to drawing of course of stress limitation check results.
- **Crack width** – switch to drawing of course of one of available components of crack width check – check value, w , w_{lim} , d_e , $d_{e,lim}$.

11.8.8 Drawing of interaction diagrams

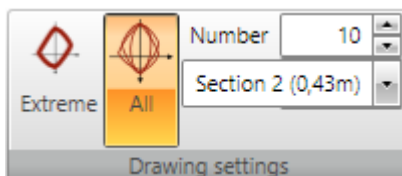
11.8.8.1 Ribbon group Interaction surface sections



Use commands in this ribbon group to switch the interaction diagram to be drawn.

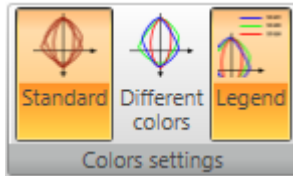
- **Horizontal** – switch to draw the horizontal section of intersection surface through the point $N_{ed}, 0, 0$.
- **N-M res** – switch to draw the vertical section of intersection surface through the origin of coordinate system and the result of bending moments $ME_{d,y}$, $ME_{d,z}$. If the both sections are zero, the section is drawn in the plane N-My.
- **N – My** – switch to draw the vertical section of intersection surface through the point $(0,0,ME_{d,z})$ parallel with the plane N-My.
- **N–Mz** – switch to draw the vertical section of intersection surface through the point $(0,0,ME_{d,y})$ parallel with the plane N-Mz.

11.8.8.2 Ribbon group Drawing settings



- **Extreme** – switch to draw the extremal interaction diagram in the current position.
- **All** – switch to draw all interaction diagrams in the current position.
- **Number** – set the number of interaction diagrams to be drawn. The diagrams with the highest exploitation value are drawn.
- **Position** – set the position on the current design member, for which the interaction diagrams are drawn.

11.8.8.3 Ribbon group Colors settings

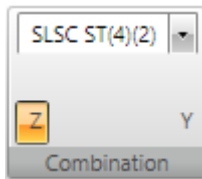


Use commands in this ribbon group to set the drawing colors of interaction diagrams.

- **Standard** – switch to draw all interaction diagrams in one color – default color for drawing of interaction diagrams.
- **Different colors** – switch to draw each interaction diagram in different color.
- **Legend** – switch on/off drawing of legend describing the points which represents the design resistance forces.

11.8.9 Drawing of deflections check results

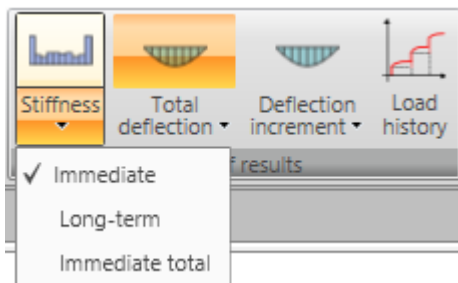
11.8.9.1 Ribbon group Combination



Characteristic combinations, which are assigned to the result class for deflections calculation, are available in the list. The courses of calculated deflections and stiffnesses are drawn for selected combination.

- **Z** – switch to drawing of deflections in direction of Z-axis of local coordinate system of design member.
- **Y** – switch to drawing of deflections in direction of Y-axis of local coordinate system of design member.

11.8.9.2 Ribbon group Type of results

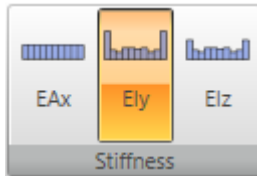


Use the options in ribbon group **Type of results** to set the results mode:

- **Stiffness** – switch to drawing of calculated stiffness for the current combination along the design member:
 - **Immediate** – switch to drawing of stiffness for calculation of immediate effects of long-term load components for the current combination.
 - **Long-term** – switch to drawing of stiffness for calculation of long-term effect of long-term load components for the current combination..
 - **Immediate total** – switch to drawing of stiffness for calculation of immediate effect of total load for the current combination.
- **Total deflection** – switch to drawing of calculated total deflections for the current combination along the design member:
 - **Linear** – switch to drawing of deflections from linear calculation for the current combination.
 - **Immediate** – switch to drawing of immediate deflections (calculated from short-term stiffness) from total load for the current combination.
 - **Long-term** – switch to drawing of long-term deflections (calculated including the effects of creep) from the long-term loads for the current combination.
 - **Total** – switch to drawing of total deflections (calculated including the effects of creep) for the current combination.
 - **Limit** – switch to drawing of limit deflections.

- **Deflection increment** – switch to drawing of calculated deflection increments for the current combination along the design member:
 - **Increment** – switch to drawing of deflection increments for the current combination.
 - **Limit** – switch to drawing of limit of deflection increments.
- **Load history** – switch to drawing of load history of current combination for deflection check. Load history diagram shows load cases and their contribution to deflection calculation and the effective time of their application.

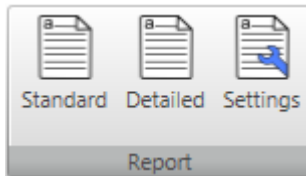
11.8.9.3 Ribbon group Stiffness



Use options in ribbon group **Stiffness** to set the evaluated stiffness.

- **EAx** – switch to drawing of axial stiffness EAx.
- **Ely** – switch to drawing of flexural stiffness Ely.
- **Elz** – switch to drawing of flexural stiffness Elz.

11.8.10 Check report

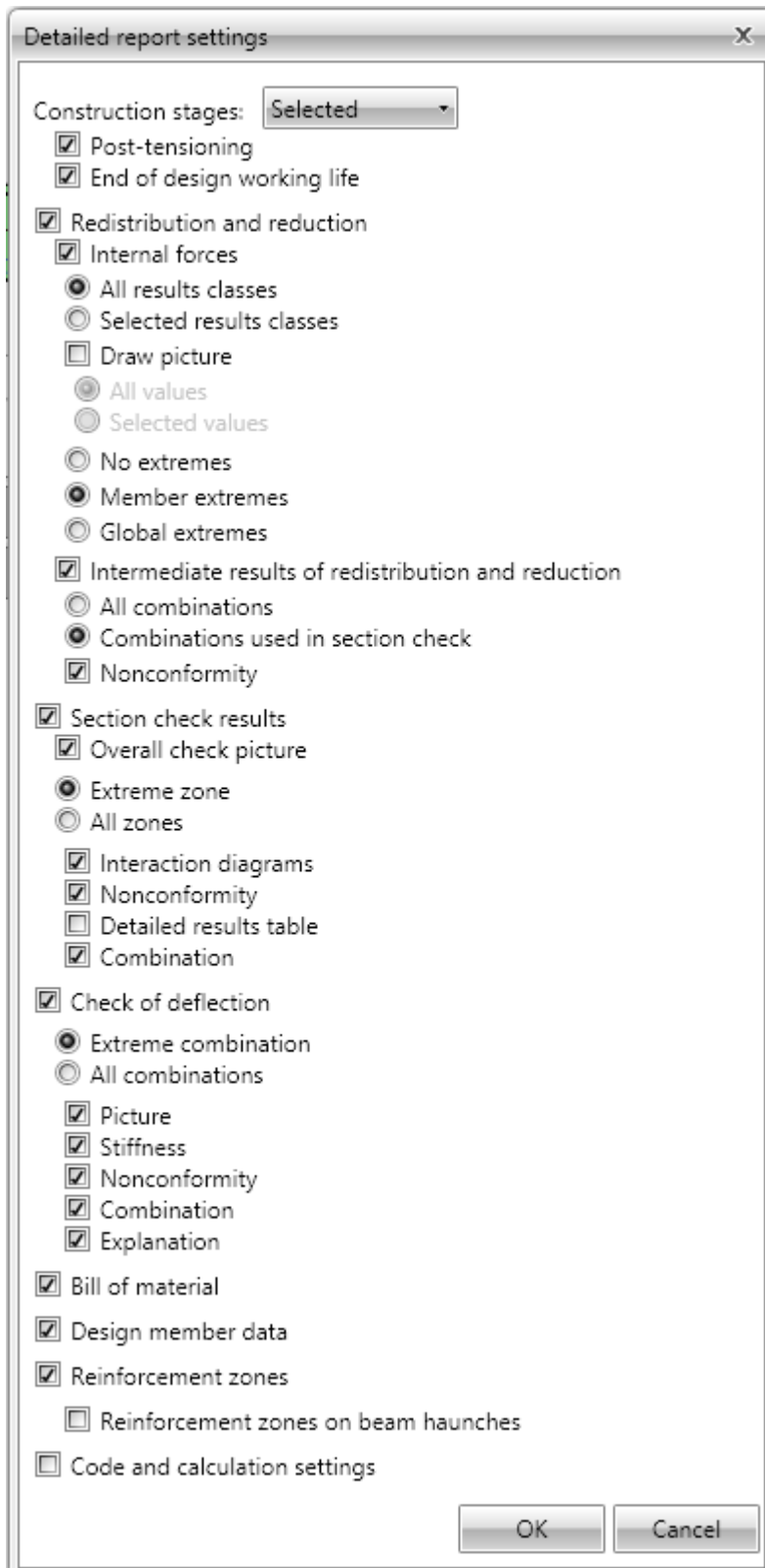


To generate and print the calculation report with section check results and deflection calculation results use options in ribbon group **Report**.

- **Standard** – generate standard check report for the current design member or design group.
- **Detailed** - generate detailed check report for the current design member or design group.
- **Settings** – display dialog to define the content of the detailed report.

11.8.10.1 Report settings

To set the content of report click **Settings** in ribbon group **Report**.



Particular dialog options:

- **Construction stages** – select the mode of stages evaluation.
 - **All** – extreme values of individual check searched from all checked stages will be printed.
 - **Selected** – extreme values of individual check per each stage, selected in the list below, will be printed.
- **Redistributions and reductions** – switch on/off print of all outputs concerning calculation of redistributions and reductions of internal forces.
 - **Internal forces** – switch on/off print of tables of modified internal forces.
 - **All result classes** – switch to print tables of modified internal forces for all result classes.
 - **Selected result class** – switch to print tables of modified internal forces for result classes selected in the list below.
 - **Draw picture** – switch on/off print of pictures of courses of modified internal forces.
 - **All values** – switch to draw pictures of all components of modified internal forces.
 - **Selected values** – switch to draw pictures of components of modified internal forces selected in the list below.
 - **No extremes** – no extreme values of modified internal forces are evaluated.
 - **Member extremes** – extreme values of modified internal forces are evaluated for each single member of the structure.
 - **Global extremes** – extreme values of modified internal forces are evaluated for the whole structure.
 - **Intermediate results of redistributions and reductions** – switch on/off print of tables of intermediate results of calculation of internal forces reductions and redistributions.
 - **All combinations** – switch to print tables of intermediate results for all combinations from result classes available for checks.
 - **Combinations used in section check** – switch to print tables of intermediate results only for combinations, which were used during concrete sections check.
 - **Nonconformity** – switch on/off the print of the table with nonconformities of reduction and redistribution calculation.
- **Section check results** – switch on/off the print of all results of section checks.
 - **Overall check picture** – switch on/off the print of the picture of overall section check results along the design member.
 - **Extreme zone** – if the option is selected, the results are printed only for the reinforcement zone, where the most extreme value of check is found.
 - **All zones** – if the option is selected, the results are printed for each zone on the design member.
 - **Interaction diagrams** – switch on/off the print of interaction diagrams pictures.
 - **Nonconformity** – switch on/off the print of the table with checks nonconformities.
 - **Detailed results table** – switch on/off the print of detailed table of the check results.
 - **Combinations** – switch on/off the print of the table with description of combinations for section checks.
- **Bill of material** – switch on/off the print of the table with bill of material.

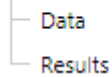
- **Check of deflection** – switch on/off the print of all results of deflection calculations.
 - **Extreme combinations** – if the option is selected, results of deflection check will be printed for the combination, which caused the extreme check value.
 - **All combinations** – if the option is selected, results of deflection check will be printed for all combinations.
 - **Picture** – switch on/off the print of the picture with the courses of deflections.
 - **Stiffness** – switch on/off the print of the tables with stiffnesses.
 - **Nonconformity** - switch on/off the print of the table with nonconformities.
 - **Combination** – switch on/off the print of the table with description of combinations for calculation of deflections.
 - **Explanation** - switch on/off the print of the table with nonconformities.
- **Design member data** – switch on/off the print of the table with design members data.
- **Reinforced zones** – switch on/off the print of tables with reinforcement zones data.
 - **Reinforced zones on beam haunches** - switch on/off the print of tables with reinforcement zones on design members haunches.
- **Code and calculation setup** - switch on/off the print of tables with national code settings and concrete calculation settings.

12 Bridge load rating of concrete members

The bridge load rating calculation determines maximal multiplier of load cases for bridge loads (traffic loads – variable loads), at which all required checks of the structure satisfy.

Bridge load rating can be analysed for project, in which variable loads are assigned to variable loads groups for bridges and the option **Bridge load rating** is selected in the **Project data**.

▲ Bridge load rating



Navigator commands in group **Bridge load rating** are used to define input data, run analysis and evaluate results of bridge load rating calculation.

12.1 Input data of bridge load rating

Click navigator command **Bridge load rating > Data** to start to define input data of bridge load rating calculation. Individual tabs to define the input data are displayed in the Data window.

Ribbon group **View settings and scale** is available.

12.1.1 Check settings

Check settings	Check positions	Permanent load groups	Variable load groups	Load cases	Combinations
▲ Ultimate limit state					
Capacity N-M-M	<input checked="" type="checkbox"/>				
Shear	<input checked="" type="checkbox"/>				
Torsion	<input checked="" type="checkbox"/>				
Interaction	<input checked="" type="checkbox"/>				
▲ Serviceability limit state					
Stress limitation	<input checked="" type="checkbox"/>				
Crack width	<input checked="" type="checkbox"/>				
▲ Reduction and redistribution					
Redistribution of moment:	<input type="checkbox"/>				
Reduction of moments	<input type="checkbox"/>				
Reduction of shear force	<input type="checkbox"/>				
Limited interaction check	<input type="checkbox"/>				
▲ Estimate of rating factor					
nT n [-]	<input type="text" value="1,00"/>				
nT r [-]	<input type="text" value="1,00"/>				
nT e [-]	<input type="text" value="1,00"/>				
▲ Precision					
Value [%]	<input type="text" value="1,0"/>				
▲ Perform the calculation for					
Normal	<input checked="" type="checkbox"/>				
Reserved	<input checked="" type="checkbox"/>				
Exceptional	<input checked="" type="checkbox"/>				

Properties group **Ultimate limit state** – settings of ULS checks to be taken into account when determining the bridge load rating:

- **Capacity N-M-M** – switch on/off taking into account the capacity check during the bridge load rating calculation.
- **Shear** – switch on/off taking into account the shear check during the bridge load rating calculation.
- **Torsion** – switch on/off taking into account the torsion check during the bridge load rating calculation.
- **Interaction** – switch on/off taking into account the interaction check during the bridge load rating calculation.

Properties group **Serviceability limit state** – settings of SLS checks to be taken into account when determining the bridge load rating:

- **Stress limitation** – switch on/off taking into account the stress limitation check during the bridge load rating calculation.
- **Crack width** – switch on/off taking into account the crack width check during the bridge load rating calculation.

Properties group **Reduction and redistribution** – settings of reduction and reduction calculation during the bridge load rating calculation:

- **Redistribution of moments** – switch on/off calculation of bending moments redistribution during the bridge load rating calculation.
- **Reduction of moments** – switch on/off the calculation of reduced moments at supports during the bridge load rating calculation.
- **Reduction of shear force** – switch on/off reduction of shear force for members with loads near supports during the bridge load rating calculation.
- **Limited interaction check** - switch on/off limitation of interaction check in distance less than d from position of maximal moment during the bridge load rating calculation.

Properties group **Estimate of rating factor** – input of expected load ratings at reaching the limit state:

- **nTn** – input of estimated traffic load rating for normal bridge load rating.
- **nTr** – input of estimated traffic load rating for reserved bridge load rating.
- **nTe** – input of estimated traffic load rating for exceptional bridge load rating.

Properties group **Precision**:

- **Value** – limit value of utilisation difference between two consecutive calculation iterations.

Properties group **Perform the calculation for**

- **Normal** – switch on/off performing the calculation for normal bridge load rating.
- **Reserved** – switch on/off performing the calculation for reserved bridge load rating.
- **Exceptional** – switch on/off performing the calculation for exceptional bridge load rating.

12.1.2 Check positions

Checks selected to determine the bridge load rating are executed only in selected sections of design member.


Positions of sections to determine the bridge load rating are defined on tab **Check positions**. For pre-stressed structures the positions to determine the bridge load rating are the same as the positions for the design loads check.

Check settings							Check positions	Permanent load groups	Variable load groups	Load cases	Combinations
Check positions											
Name	Reference point	Position [m]	Total position [m]	Check	LR						
Section 1	1	1,60	1,60	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					
> Section 2	2	3,00	6,20	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					
Section 3	3	0,75	9,95	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>					

The positions to determine the bridge load rating are defined in the table **Check positions**.

Click  above the table to add new check position.

Columns in the **Check positions** table:

- **Name** – input the name of position. The name is used to generate the name of section in the IDEA RCS module.
- **Reference point** – select the point of the design member, which the distance of section is related to.
- **Position** – input the distance between the check position and the reference point.
- **Total position** – displays the distance between the check position and the beginning of the design member.
- **Check** – this column is available only for pre-stressed design members, for which the individual sections for check of design loads are defined. It displays, if the check of design loads is performed in the position.
-  - delete the current check position.

12.1.3 Load cases and combinations for bridge load rating


The bridge load rating analysis uses separate groups of permanent and variable load cases and separate combinations of load cases. As default the groups and combinations for bridge load rating are generated according to load groups and load combinations defined for checks of design loads.

12.1.3.1 Permanent load groups



Permanent load groups for bridge load rating are defined on the tab **Permanent load groups**.

Name	$\gamma G, \text{sup}$ [-]	$\gamma G, \text{inf}$ [-]	ξ [-]
LR LG1	1,35	1,00	0,85

Commands above the table:

-  - add new group of permanent loads for bridge load rating.
- **Synchronize** – updates permanent load groups for bridge load rating according to permanent load groups for check of design loads.

Columns in **Permanent load groups table**:


- **Name** – input name of load group.
- **$\gamma G, \text{sup}$** – input of partial factor for permanent unfavourable load cases in ULS combinations.
- **$\gamma G, \text{inf}$** – input of partial factor for permanent favourable load cases in ULS combinations.
- **ξ** – input of reduction factor of unfavourable permanent loads.
-  - launch **Load groups manager** to assign load cases to load groups and to modify properties of load groups (also multiple) – see **Load groups manager**.
-  - delete the appropriate permanent load group.

12.1.3.2 Variable load groups

Variable load groups for bridge load rating are defined on the tab **Variable load groups**. Also user defined group of bridge loads can be defined.



Check settings								Check positions		Permanent load groups		Variable load groups		Load cases		Combinations	
Variable load groups												Synchronize		User bridge load group			
Name	Type	Bridge load group	γ_q []	Ψ_0 []	Ψ_1 []	Ψ_2 []			Name	Traffic							
> LR gr1a - TS	Exclusive	gr1a - TS	1,35	0,75	0,75	0,00			LR LG100	<input checked="" type="checkbox"/>							
LR gr1a - UDL	Exclusive	gr1a - UDL	1,35	0,40	0,40	0,00											
LR gr1a - Pedestr. + cycle track	Exclusive	gr1a - Pedestr. + cycle track	1,35	0,40	0,40	0,00											
LR gr1b - Single axle	Exclusive	gr1b - Single axle	1,35	0,00	0,75	0,00											
LR gr2 - Horizontal forces	Exclusive	gr2 - Horizontal forces	1,35	0,00	0,00	0,00											
LR gr3 - Pedestrian loads	Exclusive	gr3 - Pedestrian loads	1,35	0,00	0,40	0,00											
LR gr4 - Crowd loading	Exclusive	gr4 - Crowd loading	1,35	0,00	0,00	0,00											
LR gr5 - Special vehicles	Exclusive	gr5 - Special vehicles	1,35	0,00	0,00	0,00											
LR Fwk - Persistent	Exclusive	Fwk - Persistent	1,50	0,60	0,20	0,00											
LR Fwk - Execution	Exclusive	Fwk - Execution	1,50	0,80	0,00	0,00											
LR F**W - Design	Exclusive	F**W - Design	1,50	1,00	0,00	0,00											
LR Thermal - Tk	Exclusive	Thermal - Tk	1,50	0,60	0,60	0,50											
LR QSn.k - Execution	Exclusive	QSn.k - Execution	1,50	0,80	0,00	0,00											
LR Construction - Qc	Exclusive	Construction - Qc	1,50	1,00	0,00	1,00											

Commands above the **Variable loads group** table:

-  - add new group of variable loads for bridge load rating click.
- Synchronize** – updates variable load groups for bridge load rating according to variable load groups for check of design loads.


Columns in **Variable load groups** table:

- Name** – input name of load group.
- Type** – select the type of variable loads group. The type determines the behaviour of load cases from the group in appropriate combinations of load cases.
 - Standard** – load cases from the group are considered as additional load in ULS and SLS combinations.
 - Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. Only one load case from the group can act in single critical combination.
 - Accidental** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are considered as design value of an accidental action Ad.
 - Accidental, Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are considered as design value of an accidental action Ad. Only one load case from the group can act in single critical combination.
 - Fatigue, Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Fatigue combination the load cases are considered as fatigue load Qfat. Only one load case from the group can act in single critical combination.
- Xxx bridge loads group ...** - select the type of bridge load. The selected type of bridge load determines the possible interaction of loads in the resulting critical combinations.
- γ_q** – input the value of partial load factor of variable load cases in ULS combinations.

- ψ_0 – input the value of partial load factor of variable load cases in ULS and SLS Characteristic combinations.
- ψ_1 - input the value of partial load factor of variable load cases in SLS Frequent combinations.
- ψ_2 - input the value of partial load factor of variable load cases in SLS Quasi-permanent combinations.
-  - launch **Load groups manager** to assign load cases to load groups and to modify properties of load groups (also multiple) – see **Load groups manager**.
-  - delete the appropriate variable load group.

Names of user defined bridge load groups can be defined in table **User bridge load group**. The defined names are added to the list of bridge load groups in column **xxx bridge load group** in the table **Variable load groups**. There are no default values of load coefficients defined for the user bridge load group, the required coefficient values must be set in the table **Variable load groups**.

Commands above the table **User bridge load group**:

-  - add new user defined bridge load group.

Columns in **User bridge load group** table:

- **Name** – input name of load group.
- **Traffic** – if selected, loads in the group are considered as the traffic loads when evaluating combinations.

12.1.3.3 Load cases


Variable load groups for bridge load rating can be modified on the tab **Load cases**.

Check settings Check positions Permanent load groups Variable load groups Load cases Combinations					
Load cases Load groups manager					
	Name	Load group	Type	Dynamic factor δ	M1 [t]
>	SW (1)	LR LG1 - Permanent	Permanent		
	R (2)	LR LG1 - Permanent	Permanent		
	SWS (2)	LR LG1 - Permanent	Permanent		
	R (4)	LR LG1 - Permanent	Permanent		
	G (4)	LR LG1 - Permanent	Permanent		
	Q	LR gr1a - UDL - Exclusive	Traffic	1,00	0,0

Commands above the table:

- **Load groups manager** - launch **Load groups manager** to assign load cases to load groups and to modify properties of load groups (also multiple) – see **Load groups manager**.

Columns in **Load cases table**:



























- **Name** – name of load case is displayed.
- **Load group** – select the load group to assign the load case to.
- **Type** – displays type of load – **Permanent** or **Traffic** for variable loads for bridge structures.
- **Dynamic factor** – input value of dynamic factor applied on variable load cases for traffic loads. Factor takes into account dynamic impact of moving vehicles in current load case.
- **M1** – for road bridges only. Total weight of one vehicle, which corresponds with type of rating, and which was specified in current load case in the model for structural analysis. If the value is zero, no weight limit in tons will be calculated, only rating factor will be displayed for the load, which has been input in current load case.
- **m1** – for footbridges only. Input weight of uniformly distributed load in t/m² specified in current load case. If the value is zero, no weight limit in tons will be calculated, only rating factor will be displayed for the load, which has been input in current load case.
-  - delete the appropriate load case.

12.1.3.4 Combinations of load cases


Combinations of load cases for bridge load rating can be modified on the tab **Combinations**.

As default, combinations to determine normal, reserved and exceptional load rating are created for each existing combination of design loads.



To be able to determine the load rating value for certain type of traffic load (load rating), the combination assigned to this type of traffic load (load rating) must be defined.

Check settings Check positions Permanent load groups Variable load groups Load cases Combinations						
Combinations  						
Name	Type	Evaluation	LR			Description
> LR ULSF Normal	ULS Fundam	Code, (6.10 a,b)	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR ULSF Reserved	ULS Fundam	Code, (6.10 a,b)	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR ULSF Exceptional	ULS Fundam	Code, (6.10 a,b)	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSC Normal	SLS Char	Code	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSC Reserved	SLS Char	Code	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSC Exceptional	SLS Char	Code	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSF Normal	SLS Frequent	Code	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSF Reserved	SLS Frequent	Code	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSF Exceptional	SLS Frequent	Code	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSQ Normal	SLS Quasi	Code	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSQ Reserved	SLS Quasi	Code	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
LR SLSQ Exceptional	SLS Quasi	Code	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3

Commands above the table:

-  - add a new combination for bridge load rating.
- Generate** – generates missing combinations required to perform checks selected to determine load rating of design member.

Columns in the **Combinations** table:

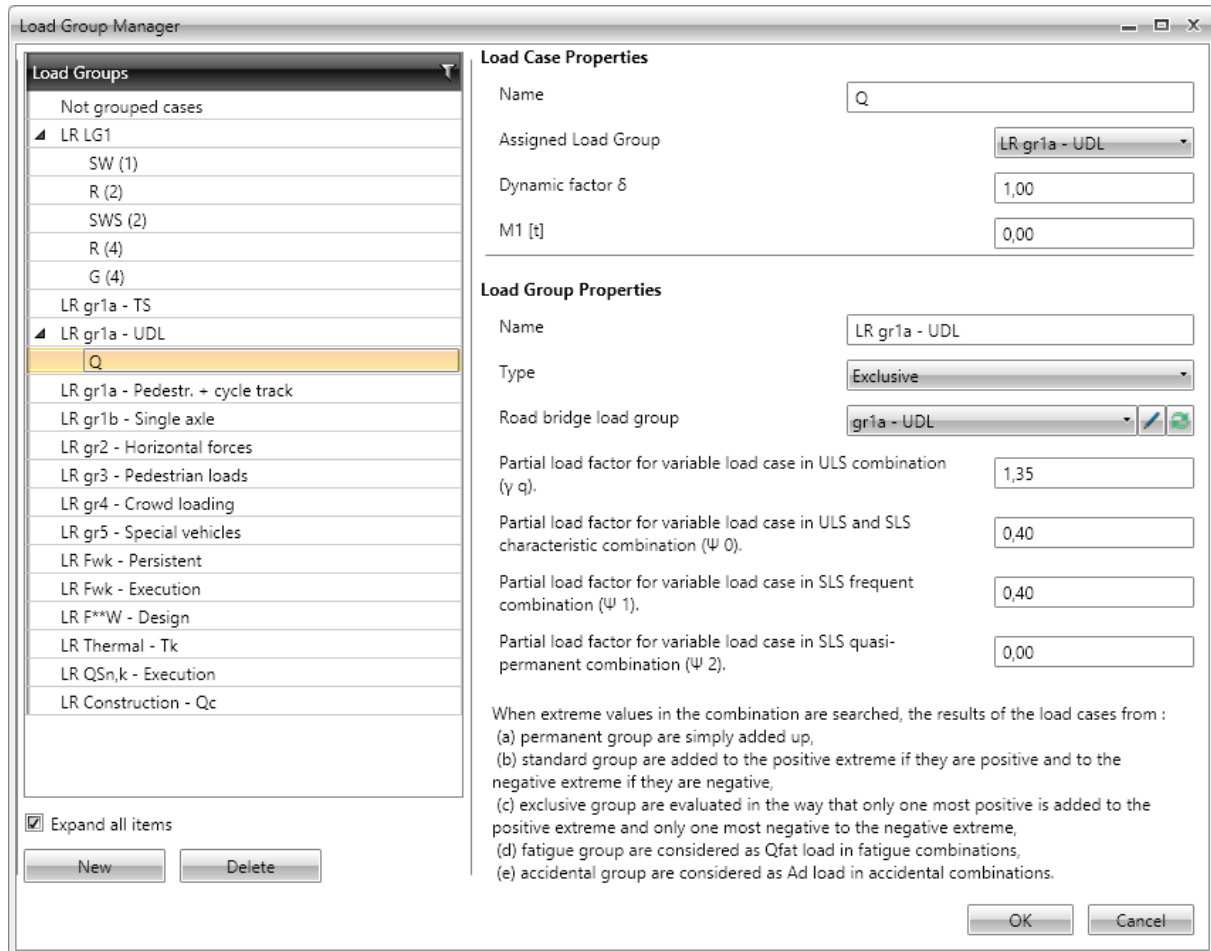
- Name** – input the name of combination.
- Type** – select the type of current combination.
- Evaluation** – select the evaluation mode of combination.
- LR** – select type of traffic load (load rating) assigned to the combination:
 - Normal** - maximum weight of single truck, which can access the bridge with no traffic restrictions, in arbitrary number and with no restrictions for pedestrians and cyclists.
 - Reserved** - maximum weight of single truck, which can access the bridge as the only vehicle with no other traffic restrictions for pedestrians and cyclists.
 - Exceptional** - maximum weight of single truck, which can access the bridge as the only vehicle with no other traffic of pedestrians and cyclists, and subject to further restrictive measures.
-  - launch the **Combinations manager** to modify the combination rules – see **Manager of load cases combinations**.
-  - delete the appropriate combination.
- Description** – displays the content of combination rule.

12.1.3.5 Load groups manager

Each load case is assigned to a load cases group.

Load cases, which are in one load cases group, are considered as one load case when load case coefficients for combinations are generated.

To view or modify load cases groups for bridge load rating determination click **Load groups manager** in ribbon group above the table **Load cases**.



A type of group and values of partial load factors can be set for load group

Particular options of **Load groups manager** dialog:

- **Load groups** – load groups with associated load cases are displayed in the tree view. If a load case is selected in the tree view, properties of load case and properties of the appropriate load group are displayed in the right part of dialog. If a load group is selected in the tree view, properties of the appropriate load group are displayed in the right part of dialog. Load cases can be moved between load groups in the tree view using drag and drop (single or multiple).
- **New** – add a new load cases group.
- **Delete** – delete the current group of load cases.
- **Expand all items** – expand/collapse all items in the tree view.



Load case properties:

- **Name** – input name of the load case.
- **Assigned load group** – select load group, which the current load case is assigned to. Load cases can be moved between load groups in the tree view using drag and drop.
- **Dynamic factor** – input value of dynamic factor applied on variable load cases for traffic loads. Factor takes into account dynamic impact of moving vehicles in current load case.
- **M1** – for road bridges only. Total weight of one vehicle, which corresponds with type of rating, and which was specified in current load case in the model for structural analysis. If the value is zero, no weight limit in tons will be calculated, only rating factor will be displayed for the load, which has been input in current load case.
- **m1** – for footbridges only. Input weight of uniformly distributed load in t/m² specified in current load case. If the value is zero, no weight limit in tons will be calculated, only rating factor will be displayed for the load, which has been input in current load case.

Load group properties:

- **Name** – input name of the current load group.
- **Type** – select type of the load group:
 - **Permanent** – load group for permanent load cases. Effect of load cases in permanent load groups are added up when searching for critical combinations. Following partial load factors can be set for the permanent load cases group:
 - **$\gamma_{qu, sup}$** – input value of partial coefficient for permanent unfavourable loads in ULS combinations.
 - **$\gamma_{qu, inf}$** – input value of partial coefficient for permanent favourable loads in ULS combinations.
 - **ξ** – input value of reduction coefficient for unfavourable permanent actions.
 - **Standard** – load group for standard variable load cases. Effects of load cases in standard group are added up to the positive extreme, if they are positive, and are added up to the negative extreme, if they are negative. Following partial load factors can be set for the variable load cases group:
 - **γ_q** – input value of partial load factor for variable load cases in ULS combinations.
 - **ψ_0** – input value of partial load factor for variable load cases in ULS and SLS Characteristic combinations.
 - **ψ_1** – input value of partial load factor for variable load cases in SLS Frequent.
 - **ψ_2** – input value of partial load factor for variable load cases in SLS Quasi-permanent.
 - **Exclusive** – load group for variable load cases. Only one of load cases of the exclusive group acts in one critical combination. The load case, which causes the maximal positive value of the evaluated component, is added up to the positive extreme and the load case, which causes the minimal value of the evaluated component, is added up to the negative extreme.
 - **Accidental** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are considered as design value of an accidental action A_d .
 - **Accidental, Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are con-

sidered as design value of an accidental action Ad. Only one load case from the group can act in single critical combination.

- **Fatigue, Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Fatigue combination the load cases are considered as fatigue load Qfat. Only one load case from the group can act in single critical combination.
- **Xxx bridge load group** - select the type of bridge load. The selected type of bridge load determines the possible interaction of loads in the resulting critical combinations.
-  - modification of user defined bridge load group.
-  - add new user defined bridge load group. A name and type can be set for the new group. If **Traffic load** is selected, load cases in this group are considered as the traffic loads when evaluating combinations.

12.1.4 Ribbon group View settings and scale

See [Ribbon group View settings and scale](#).

12.2 Results of bridge load rating calculation

To calculate and evaluate results of bridge load rating click navigator command **Bridge load rating > Results**.

Results are evaluated:

- Graphically - the courses of evaluated magnitude are drawn in the Main window.
- Textually - the textual presentation of results is printed to tables on tabs in the Data window. Following tabs are available:
 - **Summary** – tables of overall status of all results for all types of traffic load is displayed in tables on this tab.
 - **Normal** – tables with detailed results of calculation of load rating for normal traffic load are displayed on this tab.
 - **Reserved** – tables with detailed results of calculation of load rating for reserved traffic load are displayed on this tab.
 - **Exceptional** – tables with detailed results of calculation of load rating for exceptional traffic load are displayed on this tab.

In the main window the overall check results or results of individual concrete section checks in defined check positions can be evaluated graphically for the selected type of traffic load (load rating).

Tabs of analysis result tables are displayed in the Data window.

Ribbon groups **View settings and scale**, **Extreme**, **LR type** and **Check** are available.

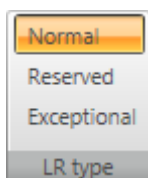
12.2.1 Ribbon group View settings and scale

See [Ribbon group View settings and scale](#).

12.2.2 Ribbon group Extreme

See [Ribbon group Extreme](#).

12.2.3 Ribbon group LR type



- **Normal** – switches to draw results of checks for normal traffic load rating.
- **Reserved** – switches to draw results of checks for reserved traffic load rating.
- **Exceptional** – switches to draw results of checks for exceptional traffic load rating.

12.2.4 Ribbon group Check

See [Ribbon group Check](#).

13 Steel members design



Use commands in navigator group **Steel design** to input design data, buckling lengths, to set design options and to perform and evaluate the check of steel members.

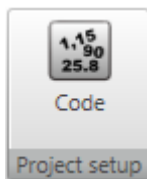
To be able to run the design of steel members, following conditions must be met:

- The beam contains members with steel cross-sections.
- USL and SLS (characteristic) combinations are defined.
- Design members and design groups of steel members are defined.
- The results of linear calculation are available.

The design data are defined for the current design group, which can be set in the navigator **Design elements**. The detailed results evaluation can be performed for the current design group too.

Only one design member is automatically created in IDEA Beam. This design member is automatically assigned to a design group and is set as the current design member.

13.1 Default check settings



Click **Code** in ribbon group **Project setup** to change the default check settings (common for all design groups) and the national code settings.

Code and calculation settings

Check settings	
Stability check	<input checked="" type="checkbox"/>
Deflection check	<input checked="" type="checkbox"/>
Fire resistance check	<input checked="" type="checkbox"/>
Classes 1 and 2 checked as class 3	<input type="checkbox"/>
Class 4 checked as class 3	<input type="checkbox"/>
Check of shear buckling	<input checked="" type="checkbox"/>
EN1993-1-1: Chapter 6	
γ_{M0}	1
γ_{M1}	1
γ_{M2}	1,25
Use always eq. 6.2 in combined strength check	<input type="checkbox"/>
Max slenderness λ acc. to 6.3.1.2 (4)	0,2
Limit value for expression $(\gamma_{M1} N_{Ed})/N_{cr}$	0,04
If possible, determine LTB curve acc. to eq. (6.57)	<input checked="" type="checkbox"/>
$\lambda_{LT,0}$	0,4
Interaction method	Annex B (German method)
Do not take small moments M_z into account in stability check if $M_z E_d / M_z R_d$ is less than limit	0,01
Use art. 6.3.3 also for unsymmetrical cross-section when limit for $M_z E_d / M_z R_d$ is exceeded	<input type="checkbox"/>
Do not take into account bending around minor axis in stability check of unsymmetrical cross-section	<input type="checkbox"/>
EN 1993-1-1: Chapter 7	
Roof structure - Frequently used by residence	250
Floor structure - Bearing the columns	400
Floor structure - Bearing the paving	250
General	250
EN1993-1-2 Structural fire design	
$\gamma_{M,fi}$	1
Calculation model	Strength domain
Required time of fire resistance ($t_{fi,req}$) [s]	900
Time interval of analysis - Unprotected members (Δt) [d]	0,0
Time interval of analysis - Protected members (Δt) [d]	0,0
Fire exposure	All sides
Type of protection	None
Temperature curve	Standard curve
Net heat flux - Configuration factor (Φ)	1
Net heat flux - Surface emissivity of member (ϵ_m)	0,7
Net heat flux - Emissivity of flame (ϵ_f)	1
Fire protection material - Temperature independent specific heat (c_p) [kJ/(kg.K)]	0,0011
Fire protection material - Thickness (d_p) [mm]	10
Fire protection material - Thermal conductivity (λ_p) [W/(m.K)]	0,12
Fire protection material - Unit mass (ρ_p) [kg/m ³]	550

Code and calculation settings dialog options:

Group Check settings:

- **Stability check** – switch on/off the execution of stability check. If the option is not selected, the stability check of designed members is not performed, only the section check is performed.
- **Deflection check** – switch on/off the execution of deflection check. If the option is selected, the deflection check of designed members is performed.

- **Fire resistance check** – switch on/off the execution of check of structural fire design according to EN1993-1-2 is performed. If the option is selected, the fire resistance check is performed.
- **Classes 1 and 2 checked as class 3** – switch on/off the execution of plastic check. If the option is selected, cross-sections classified as class 1 or class2 are checked using articles for class 3.
- **Class 4 cross-checked as class 3** - if the option is selected, cross-sections classified as class 4 are checked using articles for class 3. The check of class 4 cross-sections is not supported. If the option is not selected and the cross-section is classified as class 4, the check value of cross-section is set to 500%.
- **Check of shear buckling** – if the option is selected, the limit slenderness of cross-section web is checked. If the limit slenderness is checked and is exceeded, the check value of cross-section is set to 500% – the check according to EN1993-1-5 should be performed in such case, but it is not supported.

Group **EN1993-1-1:Chapter 6**:

- **$\gamma M0$** – input value of partial factor for resistance of cross-sections whatever the class is.
- **$\gamma M1$** – input value of partial factor for resistance of members to instability assessed by member check.
- **$\gamma M2$** – input value of partial factor for resistance of cross-sections in tension to fracture.
- **Use always eq. 6.2 in combined strength check** – if the option is selected, the equation 6.2 is used when analysing the section resistance. Otherwise the equation 6.41 is used.
- **Max slenderness λ acc. to 6.3.1.2 (4)** – input the limit value of relative slenderness to neglect the buckling resistance check according to 6.3.1.2 (4).
- **Limit value for expression $(\gamma M.NEd)/Ncr$** – input the limit value of formula to neglect the buckling resistance check according to (6.3.1.2(4)).
- **If possible, determine LTB curve acc. to eq. (6.57)** – if the option is selected and it is possible to determine LTB curve according to (6.57), the determined LTB curves are used. If the curves cannot be determined according to (6.57) or if the option is not selected, the buckling curves for LTB are determined according to (6.56).
- **$\lambda L T0$** – input the plateau length of the lateral torsional buckling curves. (6.3.2.3(1)).
- **Interaction method** – select the interaction method to be used in the interaction check according to 6.3.3.
- **Do not take small moments Mz into account in stability check if $MzEd/MzRd$ is less than limit** – input the limit value when the influence of Mz can be neglected for checks of unsymmetrical cross-section under compression and bending checked acc. to art. 6.3.4 or modified method 6.3.3 for mono-symmetrical cross-section where only in plain bending can be used. When limit is exceeded the check fails and error is written in report if the other options do not cover this case.
- **Use art. 6.3.3 also for unsymmetrical cross-section when limit for $MzEd/MzRd$ is exceeded** – if the option is selected, members with unsymmetrical cross-section are checked acc. to art. 6.3.3 when art. 6.3.4 or alternative method for mono-symmetrical cross-section cannot be used. The utilisation reserve to cover possible inaccuracy must be ensured.
- **Do not take into account bending around minor axis in stability check of unsymmetrical cross-section** – if the option is selected, bending around minor axis is neglected. This option allows to use art. 6.3.4 or alternative method 6.3.3 for mono-symmetrical cross-section always. Then utilisation reserve to cover possible inaccuracy must be ensured.

Group **Chapter 7** – input the value of limit deflections for particular types of structural elements related to the length of the member ($1/n$).

Group **EN1993-1-2 – Structural fire design**

- **$\gamma_{M,fi}$** – input the partial factor for the relevant material property, for the fire situation.
- **Calculation model** – select the type of calculation model of structural fire design. The design can be performed using either Strength domain or Temperature domain.
- **Method for critical temperature calculation** – select the method for calculation of critical temperature.
- **Required time of fire resistance** – input the time for which the structure should resist the fire.
- **Time interval of analysis – unprotected members** – input the time interval for calculation of temperature increment on unprotected members.
- **Time interval of analysis – protected members** – input the time interval for calculation of temperature increment on protected members.
- **Fire exposure** – select the mode of cross-section fire exposure. Following modes are available:
 - **All sides** – the cross-section is exposed to the fire on all sides.
 - **Three sides** – the cross-section is exposed to the fire on three sides (one side is protected).
- **Type of protection** – select the fire-protection type of the cross-section:
 - **None** – the member is not protected against the fire.
 - **Board** – the member is protected by fire protection boards.
 - **Spray** – the member is protected by fire protection spray.
- **Temperature curve** – select the temperature curve to calculate the temperature in time. Following curves are available:
 - **Standard curve**
 - **External fire curve**
 - **Hydrocarbon curve**
- **Net heat flux – configuration factor** – input the configuration factor to determine the net heat flux.
- **Net heat flux – surface emissivity of member** – input the member surface emissivity ϵ_m to determine the net heat flux.
- **Net heat flux –emissivity of flame** – input the emissivity of flame ϵ_f to determine the net heat flux.
- **Fire protection material – temperature independent specific heat** – input the specific heat of applied fire protection material.
- **Fire protection material – thickness** – input the thickness of applied fire protection material.
- **Fire protection material – thermal conductivity** – input the thermal conductivity of applied fire protection material.
- **Fire protection material – unit mass** – input the unit mass of applied fire protection material.

Group **General**:

- **Sway YY** – enable/disable sway buckling mode for buckling about y axis considered in stability check.
- **Sway ZZ** – enable/disable sway buckling mode for buckling about z axis considered in stability check.
- **Maximal buckling length coefficient** – input the maximal value of buckling coefficient for coefficients determined from calculation.
- **LT buckling system is equal as ZZ and YZ buckling system** – this option is used when creating a new design group. If the option is selected, the common buckling system is used for LTB, ZZ and YZ buckling.

13.2 Check settings for the current design group

Click navigator command **Steel design > Options** to change the check settings for the current design group.

Check settings	
Use check settings from project setup	<input checked="" type="checkbox"/>
Deflection check	<input checked="" type="checkbox"/>
Stability check	<input checked="" type="checkbox"/>
Fire resistance check	<input checked="" type="checkbox"/>
Classes 1 and 2 checked as class 3	<input type="checkbox"/>
Class 4 checked as class 3	<input type="checkbox"/>
Check of shear buckling	<input checked="" type="checkbox"/>
Buckling settings	
Use buckling settings from project setup	<input checked="" type="checkbox"/>
Sway YY	<input type="checkbox"/>
Sway ZZ	<input type="checkbox"/>
If possible, determine LTB curve acc. to eq. (6.57)	<input checked="" type="checkbox"/>
Effect of load position in the cross-section on LTB behavior of member	destabilizing
Use art. 6.3.3 also for unsymmetrical cross-section when limit for $MzEd/MzRd$ is exceeded	<input type="checkbox"/>
Do not take into account bending around minor axis in stability check of unsymmetrical cross-section	<input type="checkbox"/>
Member type for deflection check	Floor structure - Primary
EN1993-1-2 Structural fire design	
Use fire design settings from project setup	<input checked="" type="checkbox"/>
Calculation model	Strength domain
Required time of fire resistance ($t_{fi,req}$) [s]	900
Fire exposure	All sides
Type of protection	None
Temperature curve	Standard curve
Heat transfer by convection (α) [W/(m ² .K)]	25,0
Net heat flux - Configuration factor (Φ)	1
Net heat flux - Surface emissivity of member (ϵ_m)	0,7

Group **Check settings**:

- **Use check settings from project setup** – if this option is selected, the cross-section resistance check settings are taken from the default project check settings – see **Default check settings**. If the option is not selected, specific cross-section resistance check settings can be set for the current design group.

Group **Buckling settings:**

- **Use buckling settings from project setup** - if this option is selected, the buckling resistance check settings are taken from the default project check settings – see **Default check settings**. If the option is not selected, specific buckling resistance check settings can be set for the current design group.
- **Effect of load position in the cross-section on LTB behaviour of member** – select the position of the load on member. The load position may be either destabilising, neutral or stabilising.
- **Member type for deflection check** – select the type of member to determine the limit value of relative deformation for deflection check.

Group **EN1993-1-2 Structural fire design:**

- **Use fire design from project setup** – if this option is selected, the fire design check settings are taken from the default project check settings – see **Default check settings**. If the option is not selected, specific fire design check settings can be set for the current design group.

13.3 Design data

Click navigator command **Steel design > Design data** to input or edit the design data.

The screenshot shows a beam model with three members labeled 1, 2, and 3, each with a cross-section of Iwn400x(250/160). The beam is supported at points 2 and 3. The lengths of the members are 1.50 m, 6.00 m, and 1.50 m respectively. The data window below the model is divided into two main sections: 'Members of current design member' and 'Restrains, check data'.

Members of current design member				Restrains, check data			
ID	Member	Beginning [m]	Length [m]	Type	Side	Beg. position	End. position
1	1	0,00	1,50	Distributed	Top	0,00	1,50
2	2	1,50	6,00				
3	3	7,50	1,50				

The 'Restrains, check data' section also includes a detailed view of the selected 'Distributed LTB Restraint' with the following properties:

- Type: Distributed
- Location: Top
- Position [m]: 0,00
- End position [m]: 1,50

Following design data can be defined on the members of the design member:

- Point LTB restraint;
- Distributed LTB restraint;
- Field to be checked.

The current design member of the current design group is drawn in the main window.

The tables with design data is displayed in the data window.

Ribbon groups **LTB restraints**, **Check data** and **View settings** are available.

The design data are defined on single members of the current design member.

The current design member can be selected in the combo box **Design member** above the tables.

The list of members of the current design member is displayed in the table **Members of current design member**.

For the selected member, the defined design data are displayed in the table **Restrains, check data**.

Properties of the current design data can be edited in the right part of the table.


This is a close-up view of the 'Restrains, check data' section. It shows the 'Distributed LTB Restraint' tab selected. The table below shows the defined data for the selected member:

Type	Side	Beg. position	End. position
Distributed	Top	0,00	1,50

To the right of the table, the 'LTB Restraint' properties are displayed:

- Type: Distributed
- Location: Top
- Position [m]: 0,00
- End position [m]: 1,50


13.3.1 Point LTB restraint

To add new point LTB restraint on the current member click  above the design data table (the tab **Point LTB restraint** must be active) or click **Point** in the ribbon group **LTB restraints**.


LTB Restraint	
Type	Point
Location	Top
Position [m]	0,00
Repeated	<input checked="" type="checkbox"/>
Repeated	
Count	5
Regularly	<input checked="" type="checkbox"/>
Spacing [m]	0,38

Point restraint properties:

- **Location** – select the location of restraint. Restraint can be defined either on the top flange or on the bottom flange or on both flanges of the cross-section.
- **Position** – input the distance of point restraint related to the beginning of the member.
- **Repeated** – if the option is selected, the restraint contains more points.
- **Count** – input the count of the points in the repeated restraint.
- **Regularly** – if the option is selected, the points of repeated restraint are distributed regularly between the Position and the end of the current member.
- **Spacing** – input the spacing between points of the irregular repeated restraint.

Click  in the appropriate row of point restraints table to delete the point restraint.

13.3.2 Distributed LTB restraint

To add new distributed LTB restraint on the current member click  above the design data table (the tab **Distributed LTB restraint** must be active) or click **Distributed** in the ribbon group **LTB restraints**.

LTB Restraint	
Type	Distributed
Location	Top
Position [m]	0,00
End position [m]	1,50


Properties of distributed LTB restraint:

- **Location** – select the location of restraint. Restraint can be defined either on the top flange or on the bottom flange or on both flanges of the cross-section.
- **Position** – input the position of the distributed LTB restraint beginning related to the beginning of the member.

- **End position** – input the position of the distributed LTB restraint end related to the beginning of the member.

Click  in the appropriate row of distributed restraints table to delete the point restraint.

13.3.3 Not checked field

To add new field, where the check is not performed, on the current member click  above the design data table (the tab **Check data** must be active) or click **New** in the ribbon group **Check data**.

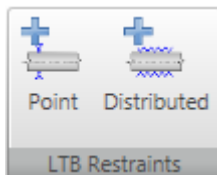
Point LTB Restraint	Distributed LTB Restraint	Check data
Member		
Member length [m]		1,500
Not checked field		
From beginning [m]		0,500
From end [m]		0,500

Properties of not checked field:

- **From beginning** – input the length at the beginning of the member, where the check is not performed.
- **From end** – input the length at the end of the member, where the check is not performed.

Click **Delete** in the ribbon group **Check data** to delete the not checked field.

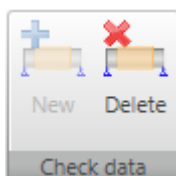
13.3.4 Ribbon group LTB restraints



Commands in ribbon group:

- **Point** – add new point LTB restraint on the current member.
- **Distributed** – add new distributed LTB restraint on the current member.

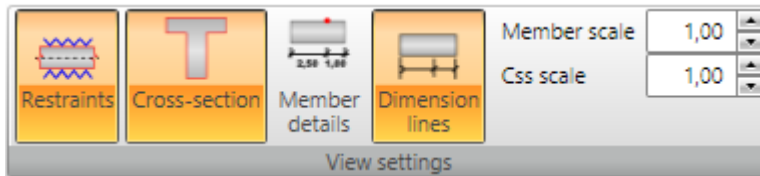
13.3.5 Ribbon group Check data



Commands in ribbon group:

- **New** – add new not checked field on the current design member.
- **Delete** – delete the current not checked field.

13.3.6 Ribbon group View settings



Use commands in ribbon group **View settings** to set the view options of the design member in uncoiled view:

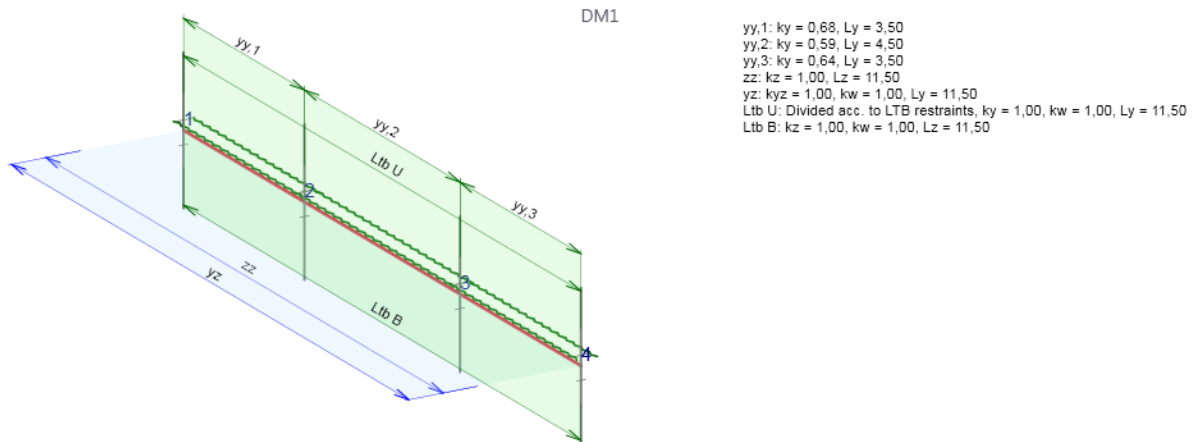
- **Restraints** – switch on/off the drawing of LTB restraints.
- **Cross-section** – switch on/off the drawing of the cross-section picture above the current design member.
- **Member details** – switch on/off the detailed drawing of the current member of the design member.
- **Dimension lines** – switch on/off the drawing of the dimension lines of the current design member.
- **Member scale** – set the value of the exceeded scale to draw the members of the design member.
- **Css scale** – set the value of the exceeded scale to draw the cross-section picture above the design member.

13.4 Buckling lengths

Click navigator command **Steel design > Buckling lengths** to input buckling lengths coefficients required for the check of the buckling resistance.

Ribbon groups **3D View** and **Dimensions drawing** are available for this navigator command.

Buckling lengths parameters are defined for the current design member.



yy.1: ky = 0,68, Ly = 3,50
 yy.2: ky = 0,59, Ly = 4,50
 yy.3: ky = 0,64, Ly = 3,50
 zz: kz = 1,00, Lz = 11,50
 yz: kyz = 1,00, kw = 1,00, Ly = 11,50
 Ltb U: Divided acc. to LTB restraints, ky = 1,00, kw = 1,00, Ly = 11,50
 Ltb B: kz = 1,00, kw = 1,00, Lz = 11,50

Design member: DM1

Buckling coefficients definition

Node	yy	ky	Length	Ltb U	kz	kw	Mcr	Ltb B	kz	kw	Mcr	Def y	Def z	Node
1	<input checked="" type="checkbox"/>	0,68	3,50	<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1
2	<input checked="" type="checkbox"/>	0,59	4,50	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	<input type="checkbox"/>	2
3	<input checked="" type="checkbox"/>	0,64	3,50	<input type="checkbox"/>				<input type="checkbox"/>				<input type="checkbox"/>	<input type="checkbox"/>	3
4	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	4

User Input - Factor User Input - Factor User Input - Factor

Mcr - Calculated Mcr - Calculated

Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB.
 The buckling system is limited by the length of the Design member. If you need to extend it by more members they must be added to the Design Member.
 Unit of length is [m] and unit of Mcr is [kN.m].

The current design member is drawn in the main window. The picture of the design member contains drawing of defined system lengths for individual buckling modes. Values of buckling coefficients are drawn too.

The table to define the system lengths and to input buckling lengths coefficients is displayed in the data window.

It is possible to configure the system length for flexural buckling yy, for flexural buckling zz, for torsional buckling yz, for lateral torsional buckling of upper flange, for lateral torsional buckling of lower flange and system lengths for limit deflections Defy and Defz.

Following coefficients can be assigned - to system lengths:

- For flexural buckling:
 - **yy** for flexural buckling around the axis yy (either use the calculated value of the coefficient or input the value of coefficient or input the value of buckling length).
 - **zz** for flexural buckling around the axis zz (either use the calculated value of the coefficient or input the value of coefficient or enter the value of buckling length)

- for torsional buckling
 - **kw** - either input the value of coefficient or the buckling length
- for LT buckling the coefficients for top and bottom flange of the section:
 - **kz**
 - **kw**
 - **Mcr**

Design member ◀ ▶

Buckling coefficients definition

Node	yy	ky	Length	Ltb U	kz	kw	Mcr	Ltb B	kz	kw	Mcr	Def y
1	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>
2	<input checked="" type="checkbox"/>	0,68	3,50	<input type="checkbox"/>				<input type="checkbox"/>				<input type="checkbox"/>
3	<input checked="" type="checkbox"/>	0,57	4,50	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>
4	<input checked="" type="checkbox"/>	0,72	3,50	<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>
		User Input - Factor		User Input - Factor			User Input - Factor					
				Mcr - Calculated			Mcr - Calculated					

Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB.

The buckling system is limited by the length of the Design member. If you need to extend it by more members they must be added.
Unit of length is [m] and unit of Mcr is [kN.m].

The current design member can be selected in the combo box **Design member** above the table.

In the table in the data window data about the buckling are entered. The first and last column of the table contains numbers of nodes which represent the nodes of individual members of the current design member.

For each type of buckling to the table consists of three or more columns:

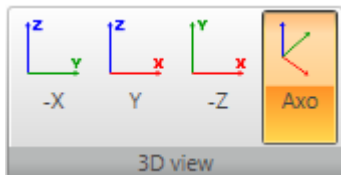
- column of checkboxes – selecting the checkboxes at individual nodes determines nodes, between which the system length is measured. The value of the buckling length of the member for flexural and torsional buckling is then calculated as the product of the coefficient of buckling length and the system length of the member. The course and shape of the moment curves is evaluated along the whole system length.
- column of buckling length coefficients **ky**, **kz** - values of coefficient for individual spans. If the mode of input is set to **Calculated**, the calculated values of buckling coefficient are displayed. In case of **User input** – factor the user defined value of buckling length coefficient can be entered. Option **Calculated** is only available for flexural plane buckling yy and zz and only for projects imported from the Ida Nexis.
- column of specified buckling lengths **Length** - if the mode of input is set to **Input – length**, the total value of the buckling length can be entered in this column.
- columns for input of coefficients **kz** and **kw**- if the mode of input of coefficients is set to **User input – factor**, the values of coefficients **kz** and **kw** can be set.
- column **Mcr**- if the mode of coefficients input for LT buckling is set to **Mcr – input**, the values of critical moment **Mcr** can be set..

Use the checkboxes in columns **Defy**, **Defz** to set the system length for calculation of deflections in the similar way as for buckling assessment.

If the option **Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB** is selected, groups for input of buckling coefficients for flexural buckling zz and for torsional buckling yz are not available. The calculation of flexural buckling zz and torsional buckling yz in this case takes the values of coefficients kz and kw specified for the check of LTB.

If there are LTB restraints on a member, they are taken into account when determining the buckling lengths and the coefficients kz and kw cannot be changed.

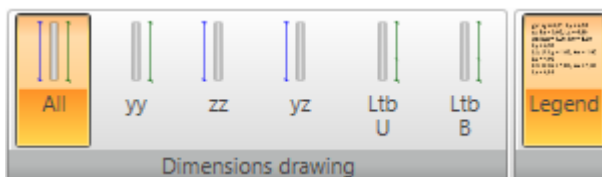
13.4.1 Ribbon group 3D View



Commands in ribbon group **3D View**:

- **-X** – switch the view direction against the global X axis direction.
- **Y** – switch the view direction to the global Y axis direction.
- **-Z** – switch the view direction against the global Z axis direction.
- **Axo** – switch to the axonometric view.

13.4.2 Ribbon group Dimensions drawing



Use commands in this group to set the drawing of dimension lines of system lengths:

- **All** – switch to draw dimension lines of system lengths for all types of buckling.
- **yy** – switch to draw dimension lines of system length of buckling yy.
- **zz** – switch to draw dimension lines of system length of buckling zz.
- **yz** – switch to draw dimension lines of system length of torsional buckling.
- **Ltb, U** – switch to draw dimension lines of system length of lateral torsional buckling at the upper flange.
- **Ltb, B** – switch to draw dimension lines of system length of lateral torsional buckling at the bottom flange.
- **Legend** – switch on/off the drawing of descriptions of system and buckling lengths.

13.5 Check results evaluation

Click navigator command **Steel design > Results** to perform the check and to evaluate the check results.

The graphical course of check results along the current design member is drawn in the Main window.

Tables with text presentation of the check results are displayed in the Data window.




Table with check results overview is displayed in the Details view.

Ribbon groups **Steel design**, **Extremes**, **Type of check** and **Type of output** are available when evaluating check results.

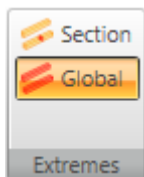
13.5.1 Ribbon group Steel design



Use commands in ribbon group **Steel design** to set or edit the result class for steel check or to set the national code coefficients for all design groups.

- **Code** - see [Default check settings](#).
- **List of ULS result classes** - select the result class for section check and buckling check. Click edit button  to modify the content of result class – see [Editing result class](#).
- **List of SLS result classes** - select the result class for deflection check. Click edit button  to modify the content of result class – see [Editing result class](#).
- **List of ULS result classes** - select the result class for fire resistance check. Click edit button  to modify the content of result class – see [Editing result class](#).

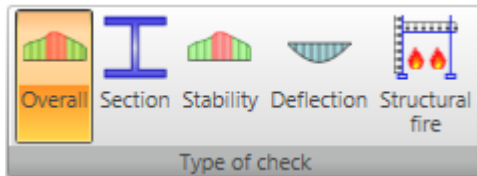
13.5.2 Ribbon group Extremes



The mode for evaluation of extremes can be set using commands in ribbon group **Extremes**. Following evaluation modes can be set:

- **Section** – extreme values of individual checks are searched in each section of the current design member. One result is printed for each type of check and for each section of the current design member.
- **Global** – extreme values of individual checks are searched from all design members. One result is printed for each type of check.

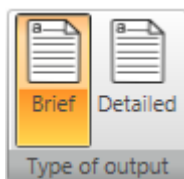
13.5.3 Ribbon group Type of check



Use commands in ribbon group **Type of check** to set the check type to be evaluated:

- **Overall** – switch to evaluation of overall results of all checks. Courses are drawn and tables are printed for the main checks – section resistance check, buckling resistance check and deflection check (if the appropriate checks are set to be performed).
- **Section** – switch to evaluation of results of section resistance check. The course of resistance check is drawn and the tables with results of all partial section resistance checks are printed.
- **Stability** – switch to evaluation of results of buckling resistance check. The course of buckling resistance check is drawn and the tables with results of all partial buckling resistance checks are printed.
- **Deflections** – switch to evaluation of results of the deflection check. The course of deflection check is drawn and the tables with results of deflection check are printed.
- **Structural fire** - switch to evaluation of results of the structural fire design. The course of structural fire design is drawn and the tables with results of structural fire design are printed.

13.5.4 Ribbon group Type of output



Use commands in ribbon group **Type of output** to set the range of printed results:

- **Brief** – switch to print table with brief summary results only.
- **Detailed** – switch to print tables with detailed outputs.

14 Report



Input data and calculation results can be printed in report. Report can contain texts, tables and pictures. Structure of protocol is fixed, it is only possible to set, which tables and which pictures should be generated.

Use commands in ribbon group **Report** to set and print the report.

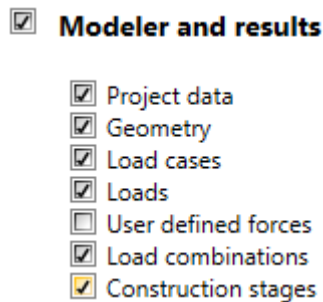
Click **Setup** to set the content of the report. In dialog **Report setup** particular items can be checked to be printed in the report.

14.1 Standard report

To generate the standard report click navigator command **Report > Standard**.

The content of standard report can be modified in the Data window.

14.1.1 Input data



Options to set the content of input data standard report:

- **Modeler and results** – switch on/off print of tables of frame input data and linear calculation results.
- **Project data** – switch on/off print of tables of project identification data, cross-sections and materials.
- **Structure** – switch on/off print of tables of nodes, members and print of the picture of the structure.
- **Load cases** – switch on/off print of tables of load cases and loads groups.
- **Loads** – switch on/off print of tables of load actions in individual load cases.
- **User defined forces** – switch on/off print of tables of basic information about user defined internal forces.
- **Load combinations** – switch on/off print of tables of load cases combinations definitions.
- **Construction stages** – switch on/off print of table with definition of construction stages.

14.1.2 Calculation results

Results

- Global Extreme
 Member Extreme
 Cross-section Extreme
 Construction stages:
- Post-tensioning
 End of design working life
 Internal forces
 Deformations
 Reactions
 Internal forces - fatigue

Options to set the content of calculation results standard report:

- **Global extreme** – if the option is selected, tables of global extremes of evaluated magnitudes will be printed to the report.
- **Member extreme** – if the option is selected, the tables of extreme values of evaluated magnitudes on each member will be printed to the report.
- **Cross-section** – extreme values of evaluated components are found per each cross-section of the beam.
- **Construction stages** – select the mode of stages evaluation.
 - **All** – extreme values of individual check searched from all checked stages will be printed.
 - **Selected** – extreme values of individual check per each stage, selected in the list below, will be printed.
- **Internal forces** – switch on/off print of tables of internal forces on members.
- **Deformations** – switch on/off print of tables of deformations on members.
- **Reactions** – switch on/off print of tables of reactions in supports.
- **Internal forces – fatigue** – switch on/off print of tables of minimal and maximal values and amplitude of internal forces.

14.1.3 Concrete 1D design results

- Concrete Design 1D**
 Construction stages:
- Post-tensioning
 End of design working life
 Bill of material

Options to set the content of concrete 1D check results standard report:

- **Concrete design 1D** – switch on/off print of tables and pictures of check results of concrete design groups.
- **Construction stages** – select the mode of stages evaluation.
 - **All** – extreme values of individual check searched from all checked stages will be printed.

- **Selected** – extreme values of individual check per each stage, selected in the list below, will be printed.
- **Bill of material** – switch on/off print of table of bill of material of reinforced members.

14.1.4 Bridge load rating results

Bridge load rating

- All types
- Calculated

Options to set the content of bridge load rating results standard report:

- **Bridge load rating** – switch on/off print of tables and pictures of bridge load rating calculation results.
- **All types** – switch to print results for all types of bridge load rating.
- **Calculated types** – switch to print results only for calculated types of bridge load rating (according to defined combinations for bridge load rating).

14.1.5 Steel design results

Steel Design

- Bill of material

Options to set the content of steel check results standard report:

- **Steel design** - switch on/off print of tables of steel check results.
 - **Bill of material** – switch on/off print of table of bill of material of steel members.

14.2 Detailed report

To generate the detailed report click navigator command **Report > Detailed**.

The content of detailed report can be modified in the Data window.

14.2.1 Input data

Modeler and results

- Project data
- Geometry
 - Draw picture
- Load cases
- Loads
 - Draw picture
 - Effects of prestressing
- User defined forces
 - Detailed setting
 - Pictures
- Load combinations

- **Modeler and results** – switch on/off print of tables of frame input data and linear calculation results.
- **Project data** – switch on/off print of tables of project identification data, cross-sections and materials.
- **Geometry** – switch on/off print of tables of nodes and members
 - **Draw picture** – switch on/off print of the picture of the structure.
- **Load cases** – switch on/off print of tables of load cases and loads groups.
- **Loads** – switch on/off print of tables of load actions in individual load cases.
 - **Draw picture** – switch on/off print of the pictures of loads in individual load cases.
 - **Effects of prestressing** – switch on/off print of the table with loads caused by prestressing elements.
- **User defined forces** – switch on/off print of tables and pictures of user defined internal forces.
 - **Detailed setting** – switch on/off print of detailed tables of input data of user defined internal forces in individual load cases.
 - **Pictures** – switch on/off print of pictures of user defined internal forces in individual load cases.
- **Load combinations** – switch on/off print of tables of load cases combinations definitions.
- **Construction stages** – switch on/off print of table with definition of construction stages.

14.2.2 Calculation results

Results

Global Extreme
 Member Extreme
 Cross-section Extreme

Construction stages:

Post-tensioning
 End of design working life

All load cases
 All combinations
 Envelope

Internal forces

- Entire centroid
- Current centroid
- Draw picture

Deformations

- Draw picture

Reactions

- Draw picture

Internal forces - fatigue

Options to set the content of calculation results detailed report:

- **Global extreme** – if the option is selected, tables of global extremes of evaluated magnitudes will be printed to the report.
- **Member extreme** – if the option is selected, the tables of extreme values of evaluated magnitudes on each member will be printed to the report.
- **Cross-section** – extreme values of evaluated components are found per each cross-section of the beam.
- **Construction stages** - select the mode of stages evaluation.
 - **All** - extreme values of individual check searched from all checked stages will be printed.
 - **Selected** - extreme values of individual check per each stage, selected in the list below, will be printed.
- **All load cases** – if the option is selected, the tables and pictures of results are printed for all defined load cases.
- **All combinations** – if the option is selected, the tables and pictures of results are printed for all defined combinations.
- **Envelope** – if the option is selected, the tables and pictures of results are printed for an envelope of combinations.
- **Internal forces** – switch on/off print of tables of internal forces on members.
 - **Draw picture** – switch on/off print of pictures of internal forces on members.
- **Deformations** – switch on/off print of tables of deformations on members.
 - **Draw picture** – switch on/off print of pictures of deformations on members.
- **Reactions** – switch on/off print of tables of reactions in supports.
 - **Draw picture** – switch on/off print of picture of reactions in supports.
- **Internal forces – fatigue** – switch on/off print of tables of minimal and maximal values and amplitude of internal forces.

14.2.3 Concrete 1D design results

Concrete Design 1D

- Construction stages:
- Post-tensioning
 - End of design working life
 - Redistribution and reduction
 - Internal forces
 - All results classes
 - Selected results classes
 - Draw picture
 - All values
 - Selected values
 - No extremes
 - Member extremes
 - Global extremes
 - Intermediate results of redistribution and reduction
 - All combinations
 - Combinations used in section check
 - Nonconformity
 - Section check results
 - Overall check picture
 - Extreme zone
 - All zones
 - Interaction diagrams
 - Nonconformity
 - Detailed results table
 - Combination
 - Check of deflection
 - Extreme combination
 - All combinations
 - Picture
 - Stiffness
 - Nonconformity
 - Combination
 - Explanation
 - Bill of material
 - Design member data
 - Reinforcement zones
 - Code and calculation settings

Options to set the content of concrete 1D check results detailed report:

- **Concrete design 1D** – switch on/off print of the table with overall check result and the picture of reinforcement scheme.
- **Construction stages** – select the mode of stages evaluation.
 - **All** – extreme values of individual check searched from all checked stages will be printed.
 - **Selected** – extreme values of individual check per each stage, selected in the list below, will be printed.

- **Redistributions and reductions** – switch on/off print of all outputs concerning calculation of redistributions and reductions of internal forces.
 - **Internal forces** – switch on/off print of tables of modified internal forces.
 - **All result classes** – switch to print tables of modified internal forces for all result classes.
 - **Selected result class** – switch to print tables of modified internal forces for result classes selected in the list below.
 - **Draw picture** – switch on/off print of pictures of courses of modified internal forces.
 - **All values** – switch to draw pictures of all components of modified internal forces.
 - **Selected values** – switch to draw pictures of components of modified internal forces selected in the list below.
 - **No extremes** – no extreme values of modified internal forces are evaluated.
 - **Member extremes** – extreme values of modified internal forces are evaluated for each single member of the structure.
 - **Global extremes** – extreme values of modified internal forces are evaluated for the whole structure.
 - **Intermediate results of redistributions and reductions** – switch on/off print of tables of intermediate results of calculation of internal forces reductions and redistributions.
 - **All combinations** – switch to print tables of intermediate results for all combinations from result classes available for checks.
 - **Combinations used in section check** – switch to print tables of intermediate results only for combinations, which were used during concrete sections check.
 - **Nonconformity** – switch on/off the print of the table with nonconformities of reduction and redistribution calculation.
- **Section check results** – switch on/off print of all chapters of section check results
 - **Overall check picture** – switch on/off print of the picture of overall check course along the design member.
 - **Extreme zone** – if the option is selected, the results are printed only for the reinforcement zone, where the most extreme value of check is found.
 - **All zones** – if the option is selected, the results are printed for each zone on the design member.
 - **Interaction diagrams** – switch on/off the print of interaction diagrams pictures.
 - **Nonconformity** – switch on/off the print of the table with checks nonconformities.
 - **Detailed results table** - switch on/off the print of the table with detailed results of section check.
 - **Combinations** – switch on/off the print of the table with description of combinations.
- **Check of deflection** – switch on/off the print of all results of deflection calculations.
 - **Extreme combinations** – if the option is selected, results of deflection check will be printed for the combination, which caused the extreme check value.
 - **All combinations** – if the option is selected, results of deflection check will be printed for all combinations.
 - **Picture** – switch on/off the print of the picture with the courses of deflections.

- **Stiffness** – switch on/off the print of the tables with stiffnesses.
- **Nonconformity** - switch on/off the print of the table with nonconformities.
- **Combination** – switch on/off the print of the table with description of combinations for calculation of deflections.
- **Explanation** - switch on/off the print of the table with nonconformities.
- **Bill of material** – switch on/off print of the table with bill of reinforcement and concrete.
- **Design member data** – switch on/off print of the tables with calculation settings of individual design members.
- **Reinforced zones** – switch on/off print of the tables and the pictures of reinforced zones and the reinforcement along the design member.
 - **Reinforced zones on beam haunches** - switch on/off the print of tables with reinforcement zones on design members haunches.
- **Code and calculation setup** – switch on/off the print of table with national code and calculation coefficients values.

14.2.4 Bridge load rating results

- Bridge load rating**
 - All types
 - Calculated
 - Section check results
 - Overall check picture
 - Critical position
 - All positions
 - Nonconformity
 - Combinations and summary load
 - Detailed results table
 - Code and calculation settings
 - Load cases
 - Combinations

Options to set the content of bridge load rating results detailed report:

- **Bridge load rating** – switch on/off print of tables and pictures of bridge load rating calculation results.
- **All types** – switch to print results for all types of bridge load rating.
- **Calculated types** – switch to print results only for calculated types of bridge load rating (according to defined combinations for bridge load rating).
- **Section check results** – switch on/off print of all chapters of section check results.
 - **Overall check picture** – switch on/off print of the picture of overall check in check positions for bridge load rating calculation.
 - **Extreme position** – if the option is selected, the results are printed only for the check position, where the most extreme value of check is found.
 - **All positions** – if the option is selected, the results are printed for all check positions for bridge load rating calculation on the design member.
 - **Nonconformity** – switch on/off the print of the table with checks nonconformities.
 - **Combinations and summary load** – switch on/off print of tables of critical combinations for bridge load rating calculation and corresponding internal forces.
 - **Detailed results table** - switch on/off the print of the table with detailed results of section check.
- **Code and calculation settings** – switch on/off the print of table with national code and calculation coefficients values for bridge load rating calculation.
- **Load cases** – switch on/off the print of table with load cases and load groups used for bridge load rating calculation.
- **Combinations** – switch on/off the print of table with combinations used for bridge load rating calculation.

14.2.5 Steel design results

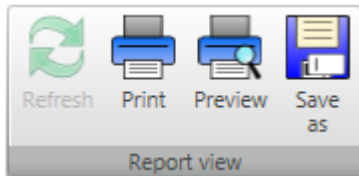
Steel Design

- Cross-section picture
- Internal forces picture
- Strength check picture
- Stability check picture
- Buckling length picture
- Deflection check picture
- Fire resistance check picture
- Print detailed tables
- Bill of material

Options to set the content of steel check results detailed report:

- **Steel design** - switch on/off print of tables of steel check results.
- **Cross-section picture** – switch on/off the print of table with cross-sections picture and cross-section characteristics.
- **Internal forces picture** - switch on/off the print of pictures with course of internal forces.
- **Strength check picture** – switch on/off the print of pictures with course of section resistance check.
- **Stability check picture** – switch on/off the print of pictures with course of buckling resistance check.
- **Buckling lengths picture** – switch on/off the print of pictures of buckling lengths.
- **Deflection check picture** – switch on/off the print of pictures with course of deflection check.
- **Fire resistance check picture** – switch on/off the print of pictures with course of fire resistance check.
- **Print detailed tables** – if the option is selected, the detailed tables of check with intermediate results are printed. If the option is not selected, only the resulting utilisations for particular checks are printed.
- **Bill of material** – switch on/off print of table of bill of material of steel members.

14.3 Ribbon group Report view



To print and export the report use commands in ribbon group Report view.

- **Refresh** – regenerate the report according to the current setting of report content
- **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.

