

# **IDEA Designer 8**

## **User Guide**

## Content

1 Getting started .....	8
1.1 System requirements .....	8
1.2 Installation .....	8
2 Theoretical background.....	9
2.1 Result Class .....	10
2.2 Design member .....	11
2.3 Design group .....	12
2.3.1 Design groups for the design of reinforcement in 1D members .....	13
2.3.2 Positions for the design of reinforcement in 2D members.....	15
2.4 Coordinate systems and convention of internal forces.....	16
2.4.1 Global coordinate system .....	16
2.4.2 Local coordinate system of 1D member .....	16
2.4.3 Coordinate system of cross-sections .....	16
2.4.4 Convention of internal forces on 1D members .....	17
2.4.5 Convention of internal forces on 2D members .....	17
3 User interface .....	18
3.1 3D view in main window .....	18
3.1.1 Manipulating 3D view.....	19
3.1.2 Structure drawing settings .....	19
3.1 Control of view in the 2D window .....	21
3.1.1 DXF export settings .....	21
4 Working with IDEA Designer.....	22
4.1 Opening a project .....	22
4.1.1 Imported parts selection .....	24
4.1.2 Settings for generation of design members during the import .....	26
4.1.3 Creating a new project importing EsaPrimaWin data.....	28
4.1.4 Creating a new project importing AxisVM data .....	28
4.1.5 Creating a new project importing SCIA Engineer data.....	29
4.1.6 Creating a new project importing Midas civil data .....	29
4.1.7 Update of IDEA projects.....	30
4.1.8 Update of projects imported from Esa Prima Win.....	30
4.1.9 Update of projects imported from SCIA Engineer, Midas, RFEM.....	30
4.1.10 Update of projects imported from AxisVM .....	30
4.1.11 Change the path to the source file .....	31
4.2 Model viewer.....	32
4.3 Load cases, combinations and result classes .....	33

4.3.1 Groups of load cases .....	34
4.3.2 Combinations of load cases .....	37
4.3.3 Result classes.....	39
4.4 Working with members and design groups.....	41
4.5 Design members.....	42
4.5.1 New design member.....	43
4.5.2 Creating new design member by pattern.....	44
4.5.3 Generating new design members .....	45
4.6 Design groups.....	48
4.6.1 New Design Group.....	49
4.6.2 Generation of design groups .....	50
4.7 Calculation .....	51
4.7.1 Results drawing settings.....	51
4.7.2 Internal forces on members .....	51
4.7.3 Internal forces on 2D members .....	52
4.7.4 Reactions .....	52
4.7.5 Deformations on 1D members .....	53
4.7.6 Ribbon group Member 1D deformations .....	53
4.7.7 Deformations on 2D members .....	53
4.7.8 Mesh.....	53
4.8 User defined internal forces .....	54
5 2D concrete members design .....	57
5.1 Definition of points for reinforcement design.....	57
5.2 Result Classes.....	60
5.3 Ribbon group Concrete check result classes .....	60
5.4 Ribbon group Concrete design.....	60
5.5 Design.....	61
6 1D concrete members design .....	62
6.1 Check of the design group.....	62
6.2 Settings for section checks and calculation of deflections .....	63
6.2.1 Code and calculation settings.....	63
6.2.2 Settings of result class for calculation of deflections.....	64
6.2.3 Setting of result classes for section checks .....	64
6.3 Design member data.....	65
6.4 Reinforcement zones .....	67
6.4.1 Check positions for check of pre-stressed members .....	68
6.4.2 Zone templates .....	70

6.4.3 Ribbon group View settings and scale .....	70
6.4.4 Ribbon group Detailed view .....	71
6.4.5 Ribbon group Internal forces.....	71
6.5 Editor of reinforcement .....	72
6.5.1 Editing cover .....	73
6.5.2 Input of reinforcement by template .....	74
6.5.3 Input of one-way slabs reinforcement by a template .....	76
6.5.4 Shear reinforcement .....	77
6.5.5 Longitudinal reinforcement.....	86
6.5.6 User settings of reinforced cross-section .....	94
6.5.7 Deleting reinforcement.....	96
6.5.8 Import and export of reinforced cross-section .....	96
6.5.9 View settings of reinforced cross-section .....	96
6.5.10 User defined reinforcement templates.....	97
6.6 Data for buckling effects calculation and deflection check .....	101
6.6.1 Data for deflections check of beam.....	101
6.6.2 Data for buckling effects calculation and deflection check for one span columns	103
6.6.3 Data for buckling effects calculation and deflection check for multi-span columns .....	106
6.7 Reduction and redistribution of internal forces.....	109
6.7.1 Supports definition for calculation of redistributions and reductions .....	109
6.7.2 Ribbon group Internal forces.....	110
6.8 Detailed check .....	111
6.9 Results evaluation .....	112
6.9.1 Ribbon group Concrete design.....	112
6.9.2 Ribbon group View settings and scale .....	112
6.9.3 Ribbon group Extreme .....	112
6.9.4 Ribbon group Recalculate .....	112
6.9.5 Ribbon group Results drawing.....	113
6.9.6 Ribbon group Check.....	113
6.9.7 Drawing of interaction diagrams .....	114
6.9.8 Drawing of deflections check results .....	115
6.9.9 Check report .....	116
7 Bridge load rating of concrete members .....	120
7.1 Input data of bridge load rating .....	120
7.1.1 Check settings .....	120
7.1.2 Check positions .....	121

7.1.3 Load cases and combinations for bridge load rating .....	123
7.1.4 Ribbon group View settings and scale .....	130
7.2 Results of bridge load rating calculation .....	131
7.2.1 Ribbon group View settings and scale .....	131
7.2.2 Ribbon group Extreme .....	131
7.2.3 Ribbon group LR type .....	131
7.2.4 Ribbon group Check .....	131
8 Steel members design .....	132
8.1 Default check settings .....	132
8.2 Check settings for the current design group .....	136
8.3 Design data .....	139
8.3.1 Ribbon group Setup .....	140
8.3.2 Point LTB restraint .....	140
8.3.3 Distributed LTB restraint .....	140
8.3.4 Not checked field .....	141
8.3.5 Ribbon group LTB restraints .....	141
8.3.6 Ribbon group Check data .....	141
8.3.7 Ribbon group View settings .....	141
8.4 Buckling lengths .....	143
8.4.1 Mass input of buckling lengths coefficients .....	145
8.4.2 Ribbon group 3D view .....	145
8.4.3 Ribbon group Structure .....	145
8.4.4 Ribbon group Dimensions drawing .....	146
8.5 Check results evaluation .....	147
8.5.1 Ribbon group Evaluation mode .....	147
8.5.2 Ribbon group Design member .....	147
8.5.3 Ribbon group Steel design .....	147
8.5.4 Ribbon group Evaluation .....	147
8.5.5 Ribbon group Type of check .....	148
8.5.6 Ribbon group Type of output .....	148
9 Checks summary, optimisation of steel cross-sections .....	149
9.1 Evaluation on design members .....	149
9.1.1 Optimisation of steel cross-sections .....	150
9.1.2 Ribbon group Concrete design .....	151
9.1.3 Ribbon group Steel design .....	151
9.1.4 Ribbon group Check .....	151
9.1.5 Ribbon group Check results evaluation .....	151

9.1.6 Ribbon group Material .....	152
9.2 Bill of material .....	153
10 Design and check of joints .....	154
10.1 Ribbon group Joints .....	154
10.2 New joint for design.....	155
10.3 Modification of joint for design .....	155
10.4 Ribbon group Steel design .....	156
11 Report.....	157
11.1 Standard report .....	157
11.2 Detailed report.....	158
11.2.1 Input data.....	158
11.2.2 Concrete 1D design results.....	159
11.2.3 Bridge load rating results .....	161
11.2.4 Steel design results .....	162
11.3 Ribbon group Report view .....	163
12 Cooperating programs .....	164
12.1 Program AxisVM .....	164
12.2 Program Esa Prima Win.....	168
12.3 Program SCIA Engineer.....	169
12.3.1 Nodes.....	170
12.3.2 Materials.....	170
12.3.3 Cross-sections.....	171
12.3.4 Members 1D .....	178
12.3.5 Haunches .....	178
12.3.6 Arbitrary beams.....	179
12.3.7 Openings in beams .....	179
12.3.8 Members 2D.....	179
12.3.9 Openings in members 2D.....	179
12.3.10 Load cases .....	179
12.3.11 Groups of load cases .....	179
12.3.12 Combinations .....	180
12.3.13 Internal forces on 1D members .....	180
12.4 Program Midas Civil .....	181
12.4.1 Materials.....	181
12.4.2 Steel cross-sections .....	181
12.4.3 Concrete cross-sections .....	182
12.4.4 Structural members .....	182

12.4.5 Internal forces.....	183
12.4.6 Combinations .....	183
13 Application setting .....	185
13.1 Units setting.....	185
13.2 3D view settings .....	187
13.3 General settings .....	188
13.4 Bill of material .....	189
13.5 Colours setting.....	190
13.6 Import settings.....	192

## 1 Getting started

### 1.1 System requirements

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

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

### 1.2 Installation

Administrator rights are required for an adequate installation.



## 2 Theoretical background

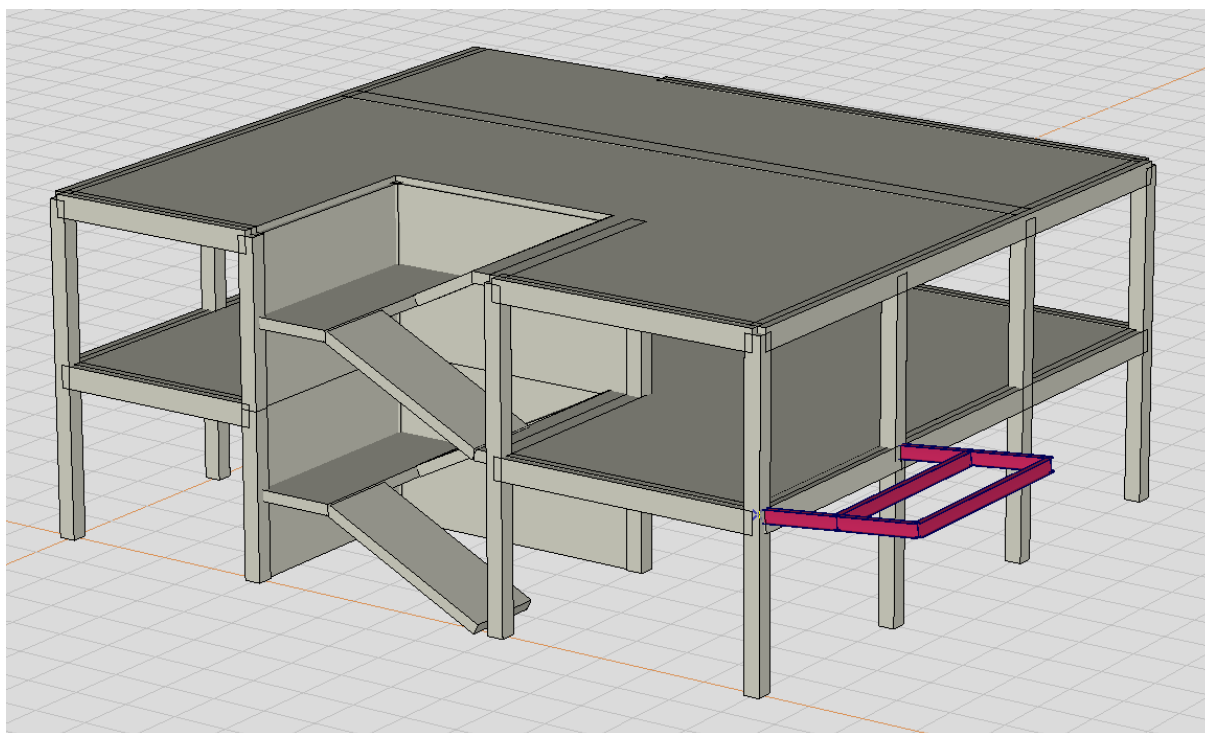
IDEA Designer is a core IDEA program for designing and checking of structural members of 3D models. IDEA Designer links IDEA checking modules with various 3D FEA programs used for analysis of internal forces and deformations.

IDEA Designer works with following FEA programs:

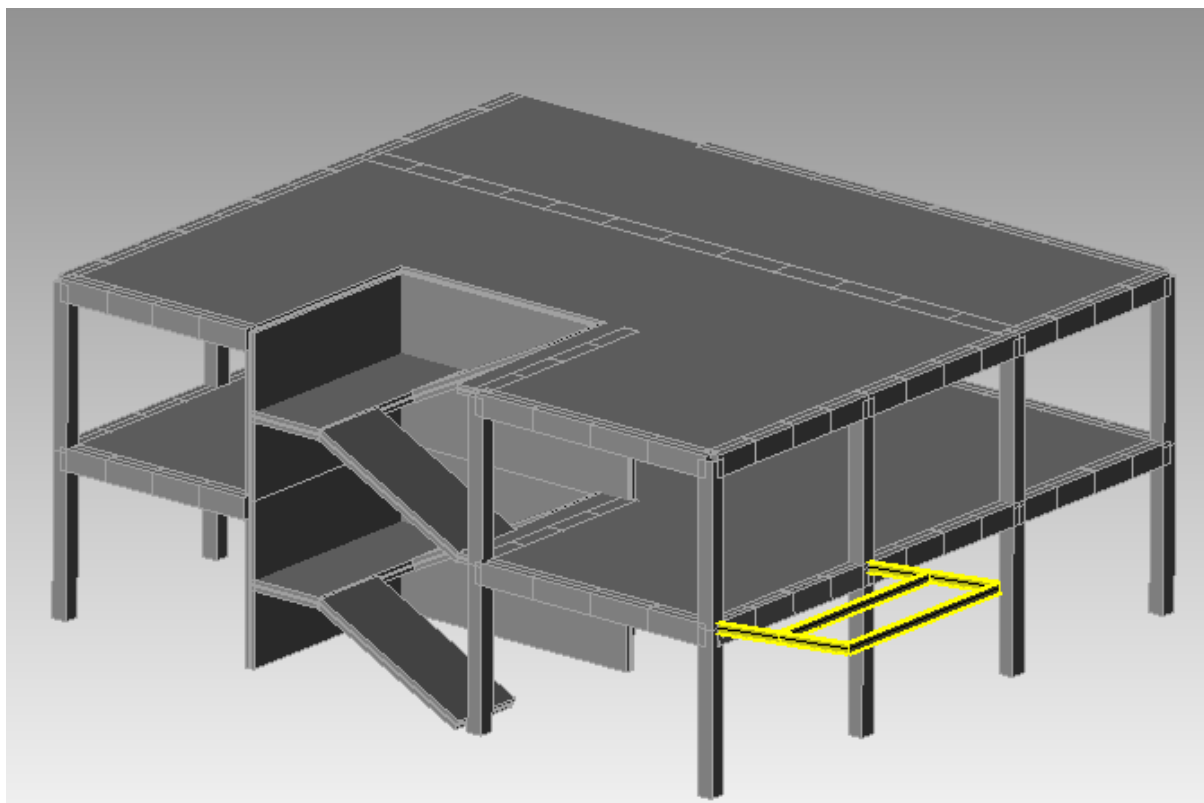
- AxisVM – Members 1D (beams, columns, ribs), Members 2D (slabs, walls, shells). Internal forces and deformations on 1D and 2D members.
- Nexis32 - members 1D (beams, columns, ribs). Internal forces and deformations on 1D members.
- SCIA Engineer - Members 1D (beams, columns, ribs), Members 2D (slabs, walls, shells).

Program IDEA Designer itself does not perform static analysis of the structure. It uses analysis model and calculated results from other FEA programs.

An example of a structure in AxisVM program and the same structure imported in the IDEA Designer:



AxisVM



IDEA Designer

First, IDEA Designer can be used to review the structural model. There are several types of presentation modes of the structure, its parts and internal forces. Next step is to prepare all data needed for the following design and check of structural members. **Result Classes** which are required for the specific checks can be defined. All structural members can be merged into **Design Groups**. All members included in one Design Group are designed according to same preconditions.

## 2.1 Result Class

IDEA Designer works with all the load cases and load combinations, which have been input or generated in the original FEA program.

A group of several load combinations or cases can be created and a check can be performed for all the members at once. Check modules search for extreme values within the appropriate Result Class.

There are Result Classes of different types:

- Ultimate Limit State (ULS)
- Serviceability Limit State (SLS)
  - Characteristic
  - Frequent
  - Quasi-permanent

Specific design and check methods works with appropriate types of Result Classes.

During import from FEA program, IDEA Designer automatically creates default result classes. These include one class of all ULS and one class of all SLS combinations. can edit them and add any number of his own Result Classes.

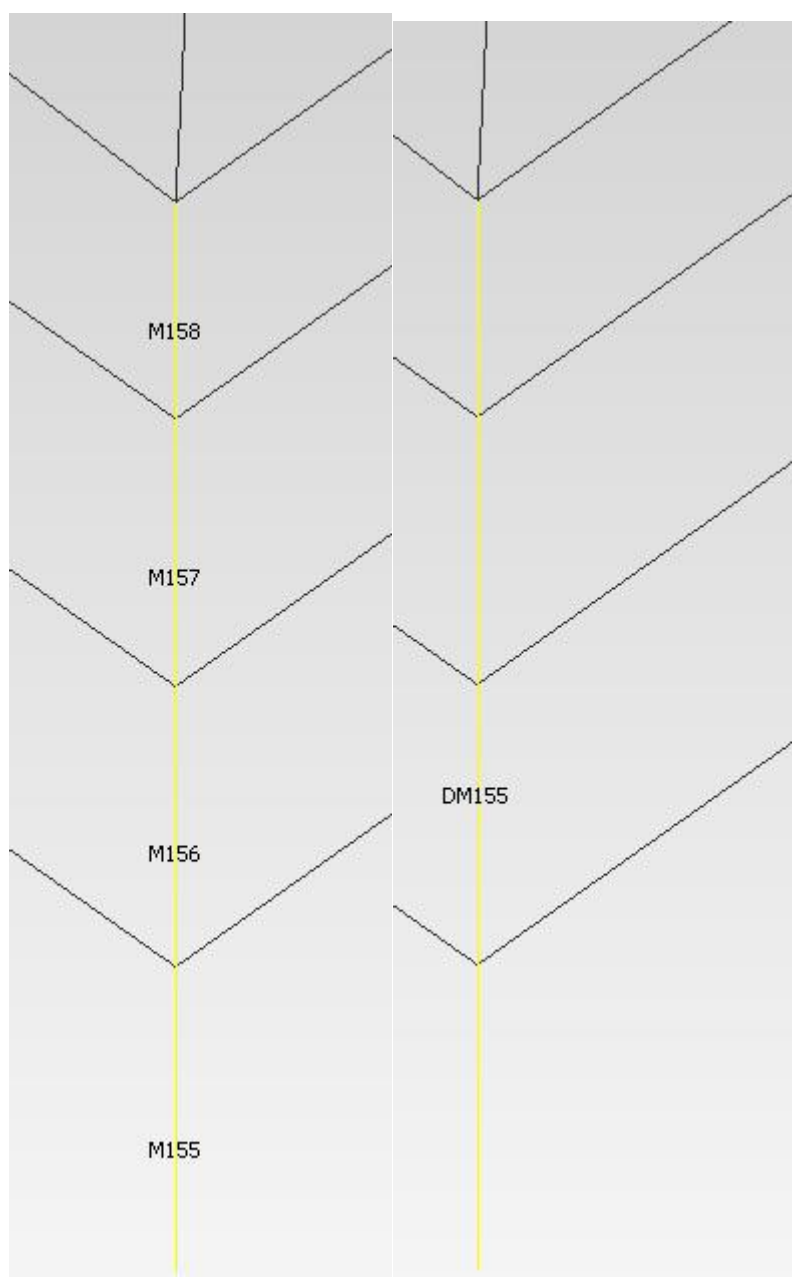
## 2.2 Design member

A design member is one or group of consecutive structural model members. Consecutive members must have common node of structural model and must have the same orientation – ending point of one member is beginning point of following member. All members of the design member must have assigned the same material.

The design member is analysed as the whole and e.g. reinforcement is designed on the design member.

The design member should collect structural members, which should be designed as the whole, but it was necessary to split it to more members in the structural model.

There is a concrete column, which consists of three structural members, in the following pictures. To design and check the column as the whole, structural members can be joined to one design member.



## 2.3 Design group

Even though there are unique members in structures, most of them consist of common members. Those are beams and columns of the same shape with the same reinforcement in concrete structures and members with the same cross-section in steel structures. Having sets of the same foundation blocks or the same beam-column connections is also typical for many structures. It is not effective to work with every member separately. However, this is the way how majority of current programs work. It leads to huge and complex models with too many detailed members. Such models cause slow program response and it is difficult to find desired object. The user can “get lost” in the program.

IDEA Designer usually operates with groups of members. That makes work with the structure much faster and more explanatory. All members of the group have the same check preconditions and all additional data have to be defined only once. Such approach saves both the amount of work and data requirements even if there is only one member in the group.

Following rules are applied in IDEA Designer:

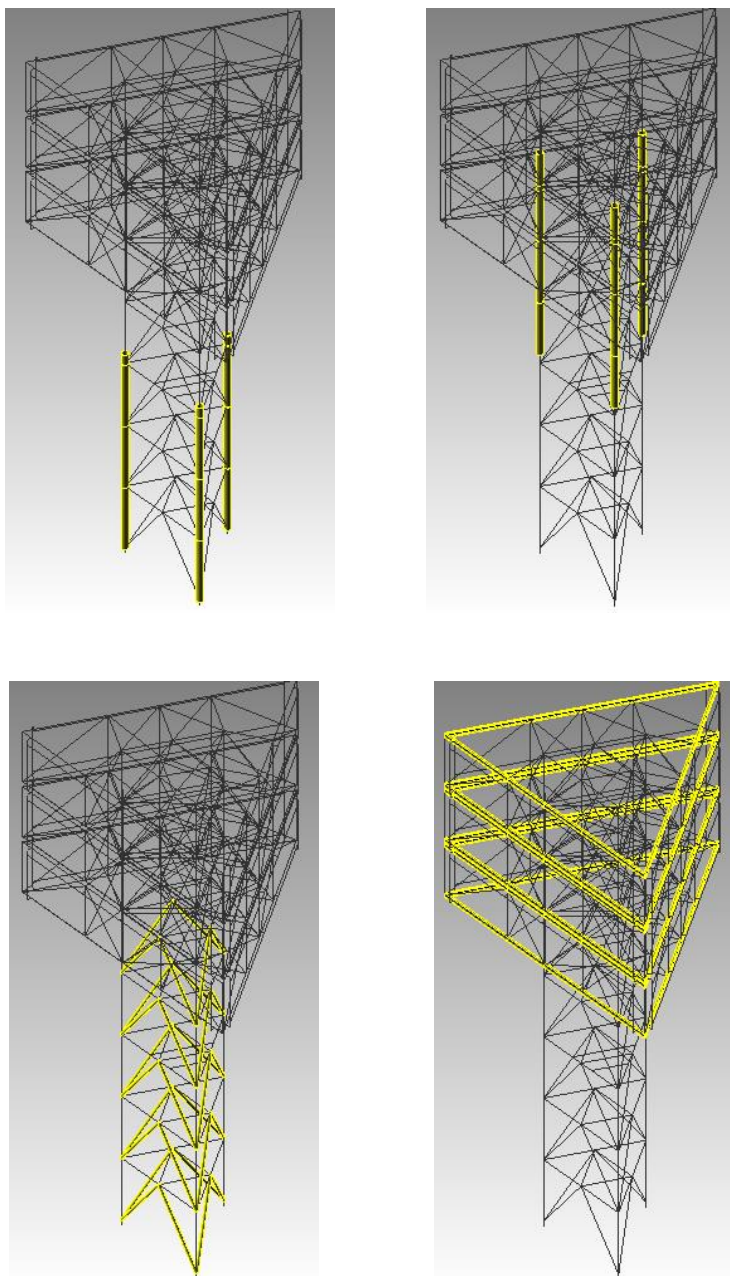
- each structural member must be assigned to a design member.
- structural members in the design member must to have the same material and must be consecutive.
- each structural member belongs to just one design member (cannot belong to more design members).
- the design group for steel members can contain design members, which:
  - Have the same cross-section (each structural member must have the same cross-section, each part of structural member must have the same cross-section at the beginning and at the end);
  - If the cross-section is the same along the whole design member, the design members in the design group may have different number of members.
- the design group for 1D concrete members can contain design members, which:
  - Have the same cross-section (each structural member must have the same cross-section, each part of structural member must have the same cross-section at the beginning and at the end);
  - Have the same length;
  - Have the same element type (rib, column, beam).
  - The number of structural members in the design member is the same

During the import from FEA program, IDEA Designer automatically creates default design members and design groups. By default, each structural member creates one design member. Design members are grouped to design groups.

1D members are sorted out according to their cross-section. Each 2D structural member creates separate design member and is assigned to one design group.

Respecting above-mentioned rules existing members and design groups can be edited and structural members moved from one group to another. Moreover, required number of new design members and design groups can be created.

An example of design groups of a structure (advertising pylon):



### 2.3.1 Design groups for the design of reinforcement in 1D members

In the case of “concrete member” design group collects all design members with the same cross-section and the same length (optional length tolerance can be defined ), which are assumed to be reinforced using the same reinforcement pattern.

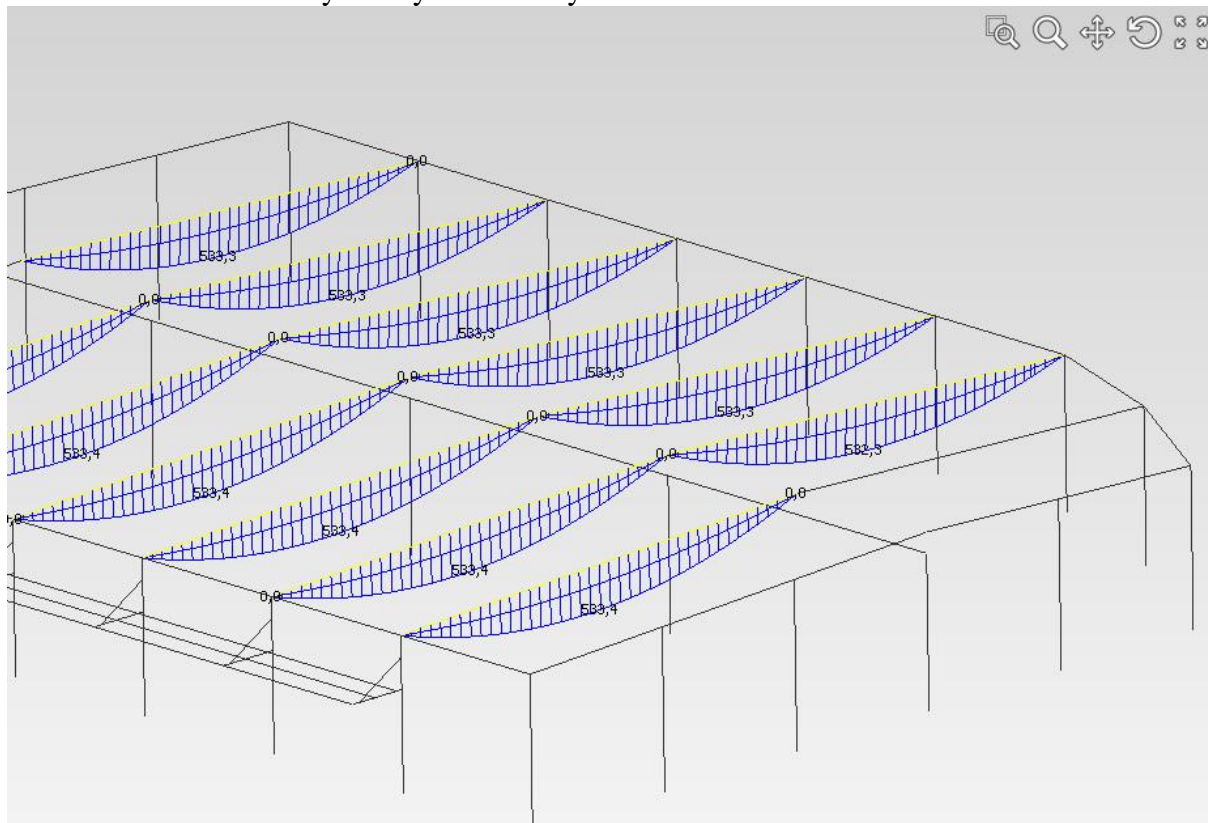
The reinforcement zones have to be defined on the representative design member of the design group. The reinforcement zone is an area along the design member, on which the reinforcement can be defined (and is “constant”).

Typical examples of Design Group are “a line of middle columns”, “edge columns”, “primary beams of side frames”, etc.

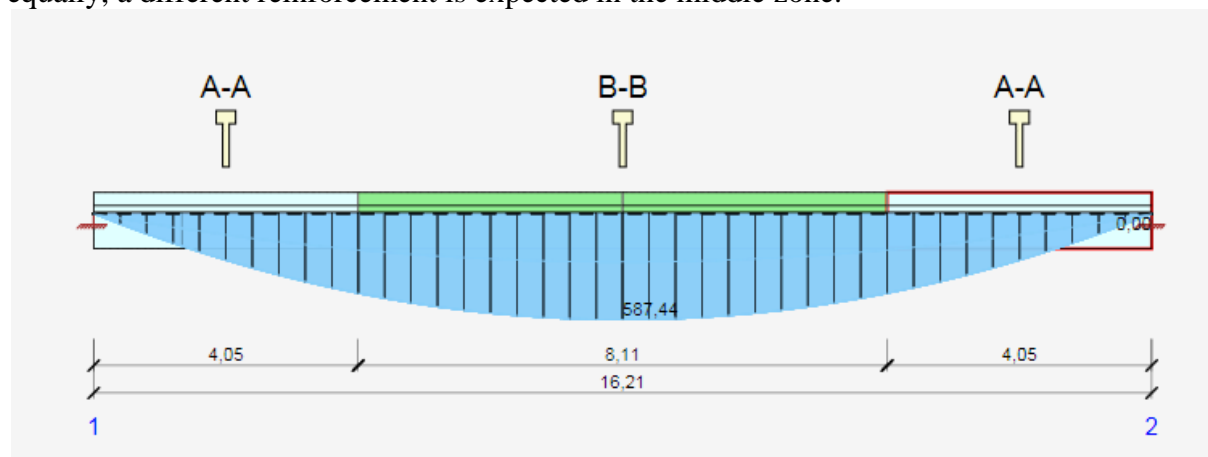
Big FEA programs usually design desired reinforcement in all the members and then locate the biggest one. IDEA Designer works in the opposite way. Firstly, engineer decides which design members are reinforced identically and afterwards he defines the reinforcement zones and the reinforcement in zones. IDEA Designer can find out the critical extreme internal forces in corresponding positions from all members in the group.

Let's show it on an example.

Design members in the picture are assigned to one design group. Engineer decides to reinforce them identically and symmetrically.



On the representative design member of the design group the engineer creates 3 reinforcement zones – one for the middle and two for the edges. The first and the last zone will be reinforced equally, a different reinforcement is expected in the middle zone.



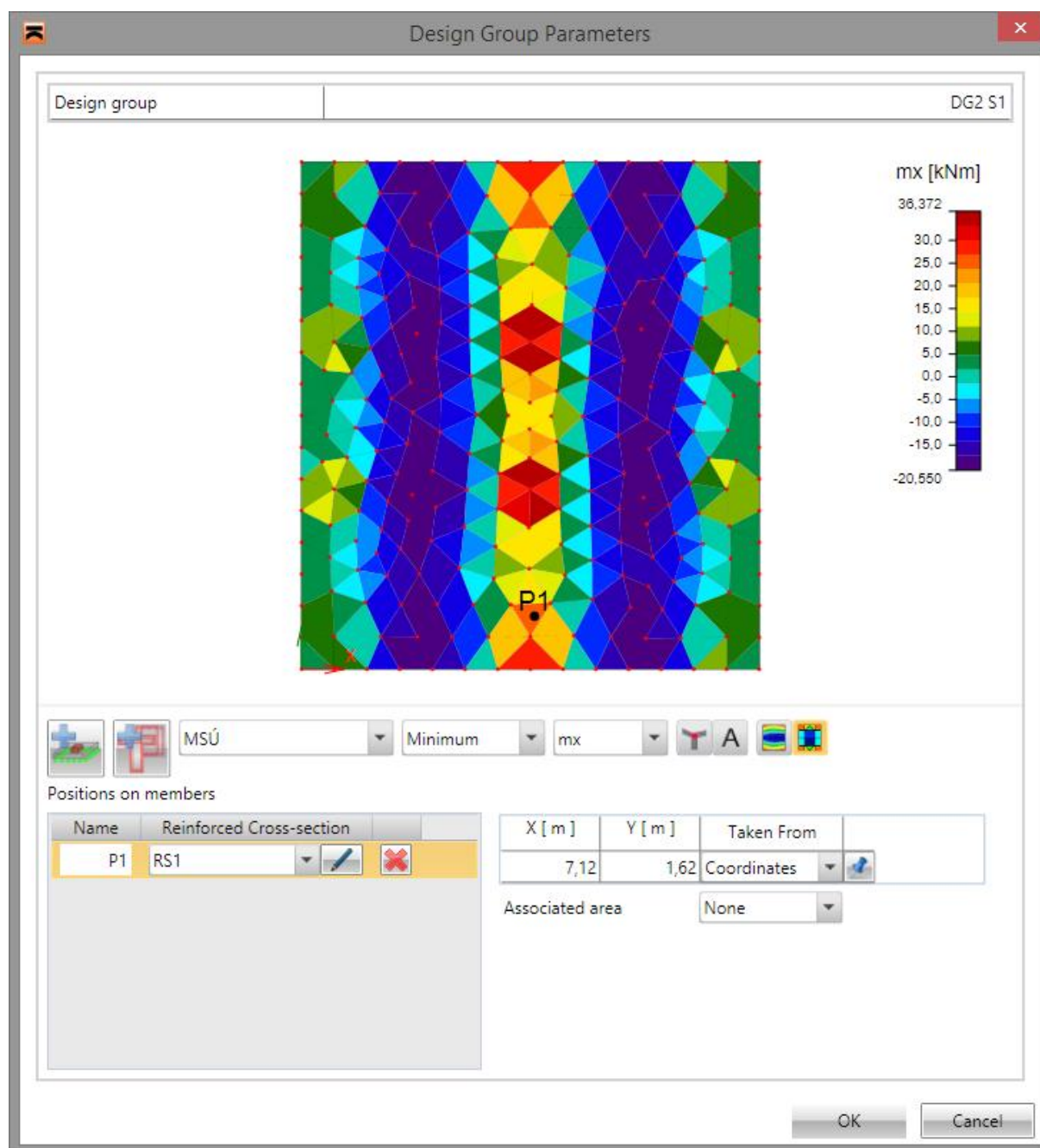
The concrete check in IDEA RCS is performed in 3 sections with 2 reinforced sections, corresponding with zones A-A at the beginning, B-B in the middle and AA at the end of the design member. Extremes of internal forces are found automatically from all design members in the design group.



	Section Name	Section Type	Design Member	Member Type	Reinforced Cross-Section
1	A-A (0,00 - 4,05m)	Reinforced	DG5 Vázni	Beam	A-A
2	B-B (4,05 - 12,16m)	Reinforced	DG5 Vázni	Beam	B-B
3	A-A (12,16 - 16,21m)	Reinforced	DG5 Vázni	Beam	A-A

### 2.3.2 Positions for the design of reinforcement in 2D members

In the case of flat slab/wall element, points are input instead of sections. Logic is the same – all points with the same type of reinforcement are unified, extreme values of internal forces in them are determined. The reinforcement is done only once and is used for all members accordingly.



## 2.4 Coordinate systems and convention of internal forces

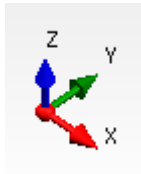
All coordinate systems used are right-handed.

### 2.4.1 Global coordinate system

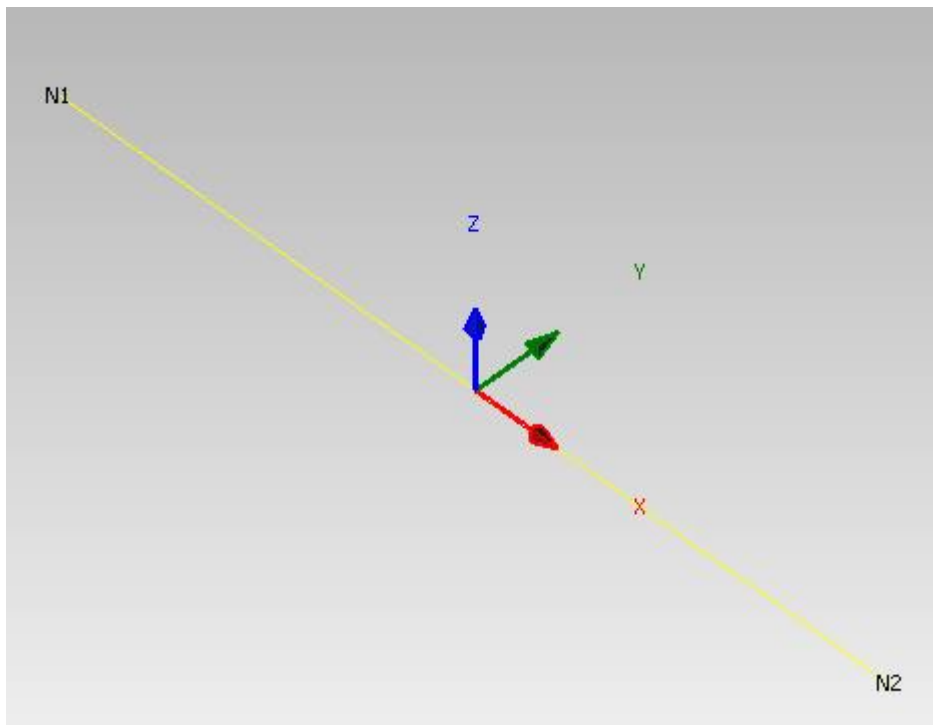
X-axis of global coordinate system is horizontal and leads from left to right.

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

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

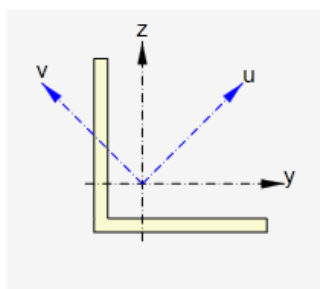


### 2.4.2 Local coordinate system of 1D member



Each 1D member is defined with begin and end node. Each 1D member has local coordinate system, which origin is in begin node of member. Local x axis of 1D member is identical with member axis and is oriented from being to end node of 1D member. Local Y axis of member is horizontal in general and local z axis leads upwards.

### 2.4.3 Coordinate system of cross-sections



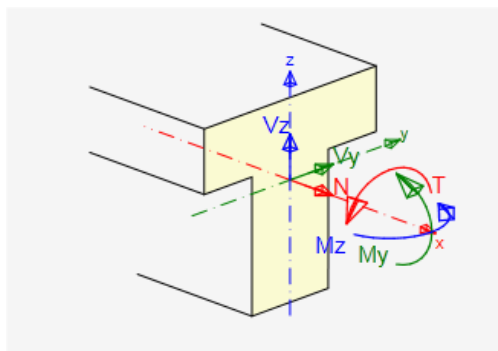
Cross-section has reference axes y (horizontal) and z (vertical).

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

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



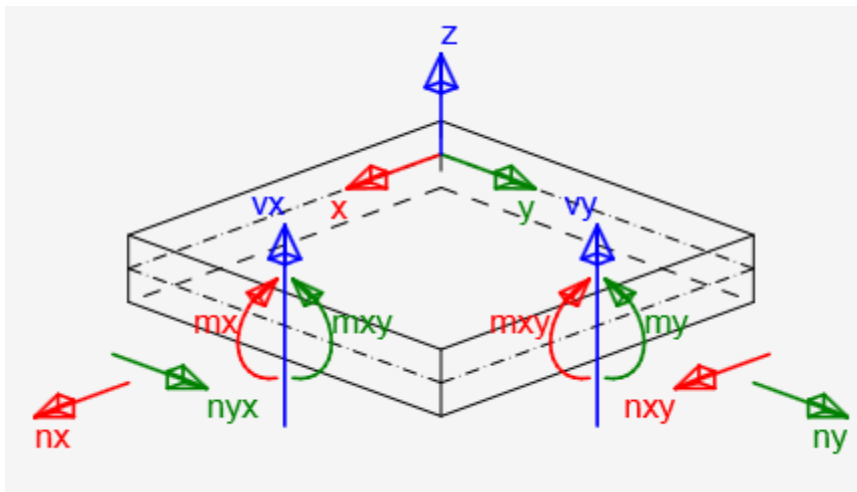
### 2.4.4 Convention of internal forces on 1D members



Internal forces on 1D members perform following actions:

- positive bending moment  $M_y$  causes tension in cross-section fibers with negative  $z$ -coordinate.
- positive bending moment  $M_z$  causes tension in fibers with negative  $y$ -coordinate
- positive torsional moment  $M_x$  acts about  $x$ -axis of 1D member
- positive axial force  $N$  acts in direction of  $x$ -axis of member and causes tension in cross-section fibers.
- positive shear force  $V_z$  acts in direction of  $z$ -axis of cross-section.
- positive shear force  $V_y$  acts in direction of  $y$ -axis of cross-section.

### 2.4.5 Convention of internal forces on 2D members



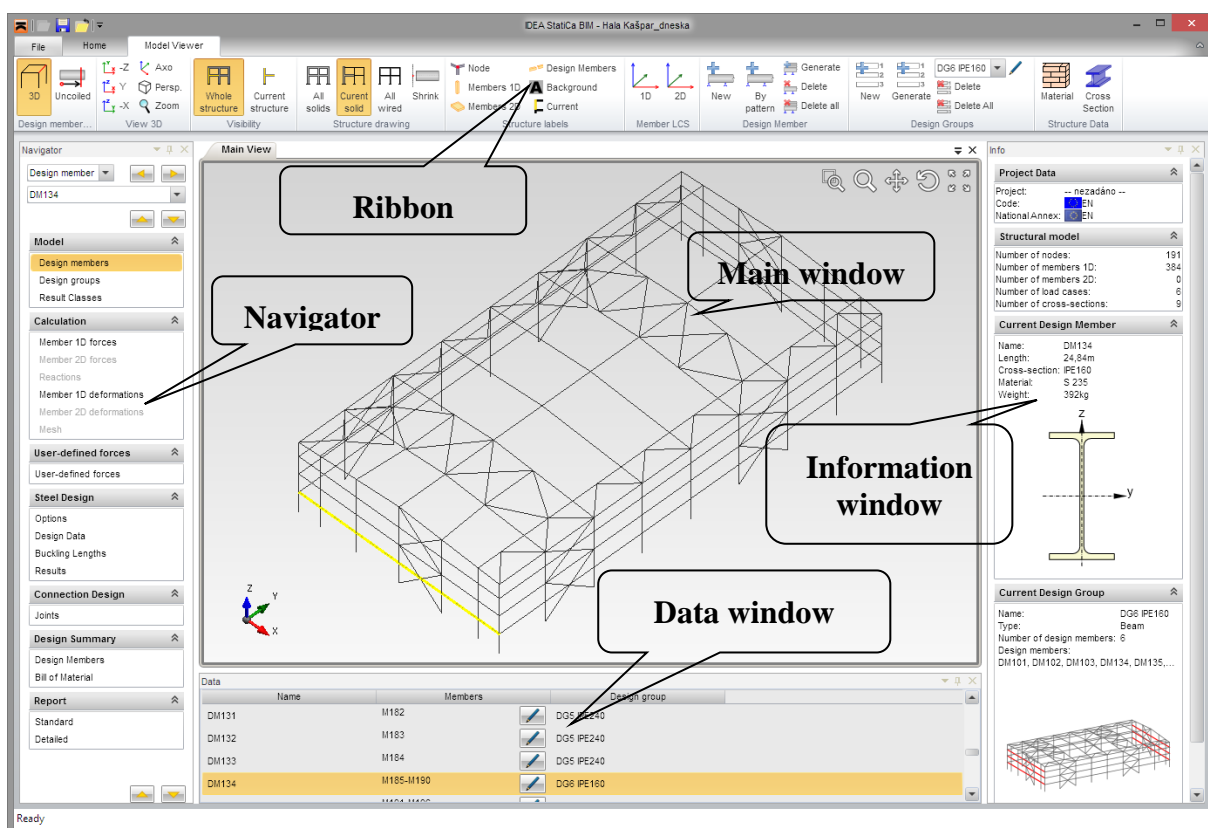
Internal forces on 2D members perform following actions:

- $m_{x(y)}$  – bending moment in direction of  $x(y)$ -axis. Positive value causes tension at bottom surface of 2D element
- $m_{xy(yx)}$  – torsional moment about  $y(x)$ -axis action on edge parallel to axis  $x(y)$ . Positive value causes tensional shear stress at bottom surface of 2D element.
- $n_{x(y)}$  – normal force in direction of  $x(y)$ -axis. Positive value acts in direction of  $x(y)$  axis and causes tension in section.
- $n_{xy(yx)}$  - normal force acting in centre plane in direction of  $y(x)$ -axis on edge parallel to  $x(y)$  axis. Positive value acts in direction of  $x(y)$ -axis.
- $v_{x(y)}$  – shear force acting perpendicular to centre plane on edge parallel to  $x(y)$ -axis. Positive value acts in direction of  $z$ -axis.

### 3 User interface

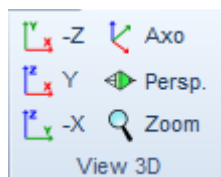
The user interface consists of several co-operating parts.

- Navigator – set of logically ordered commands, from input and check options to output and reporting.
- Tab bar (Ribbon) – set of tabs of control elements grouped by different functionalities. Tabs are changing according to selected command in the Navigator.
- Main window – graphical window displays content based on the selected command. It can be a graphical image, diagram or text dialog.
- Data window – it shows different tables or properties, based on selected command in the Navigator or selected object in the Main window,
- Information window – shows current project information for quick user reference



#### 3.1 3D view in main window

The 3D view of the structure is displayed in the main window. To manipulate the structure in 3D view use commands in group **View 3D** or combination of hotkeys and mouse key



Particular commands in group **View 3D**:

- **-Z** – set the view from the top of the structure (opposite the positive Z-semi-axis of global coordinate system)
- **Y** - set the view from the front of the structure (in the direction of positive Y-semi-axis of global coordinate system)
- **-X** - set the view from the side of the structure (opposite the positive X-semi-axis of global coordinate system)

- **Axo** – set the default 3D view point and performs zoom to fit the whole structure into 3D window
- **Zoom** – perform zoom to fit the current member or design group into the 3D window.
- **Persp.** – switch on/off the perspective view on the structure.

### 3.1.1 Manipulating 3D view

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

Commands in 3D window:



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



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



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



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



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

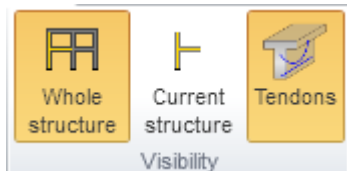
To set the required view using keyboard and mouse following combinations can be used:

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

Clicking right mouse button on 3D window displays context menu with command **Zoom all**.

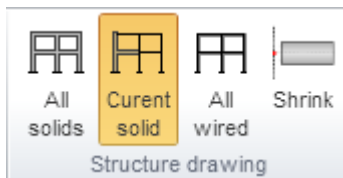
### 3.1.2 Structure drawing settings

There are some general drawing settings for displaying structure in 3D window. To set the drawing commands in groups **Visibility**, **Structure labels** and **Structure appearance** can be used.



Use switches in group **Visibility** to reduce the number of members to be drawn:

- **Whole structure** – switch on drawing of all members in the structure
- **Current structure** – switch on drawing of current member or design group only.
- **Tendons** – switch on/off drawing of prestressing reinforcement



Use switches in group **Structure appearance** to set the display mode of members:

- **All solids** – switch the mode of members drawing to draw all members as solids.
- **Current solid** – switch the mode of members drawing to draw the current member or design group as solid, the rest members are drawn in wired mode.
- **All wired** – switches the mode of members drawing to draw all members in wired mode.
- **Shrink** – switch on/off the drawing of shrunk ends of 1D members to be able to see the separated 1D members.



Use switches in group **Structure labels** to display descriptions of members:

- **Node** – switch on/off drawing of node numbers.
- **Members 1D** – switches on/off drawing of numbers of 1D members
- **Members 2D** – switch on/off drawing of numbers of 2D members
- **Design members** – switch on/off drawing of numbers of design members
- **Background** – switch on/off drawing of background under numbers.
- **Current** – switch between drawing numbers on all members in the structure or only on current member or design group.

### 3.1 Control of view in the 2D window

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



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

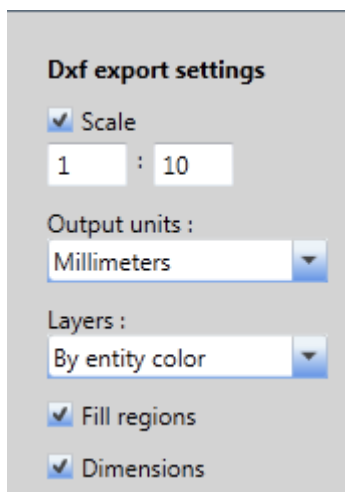
To set the required view using keyboard and mouse following combinations can be used:

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

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

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

#### 3.1.1 DXF export settings



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

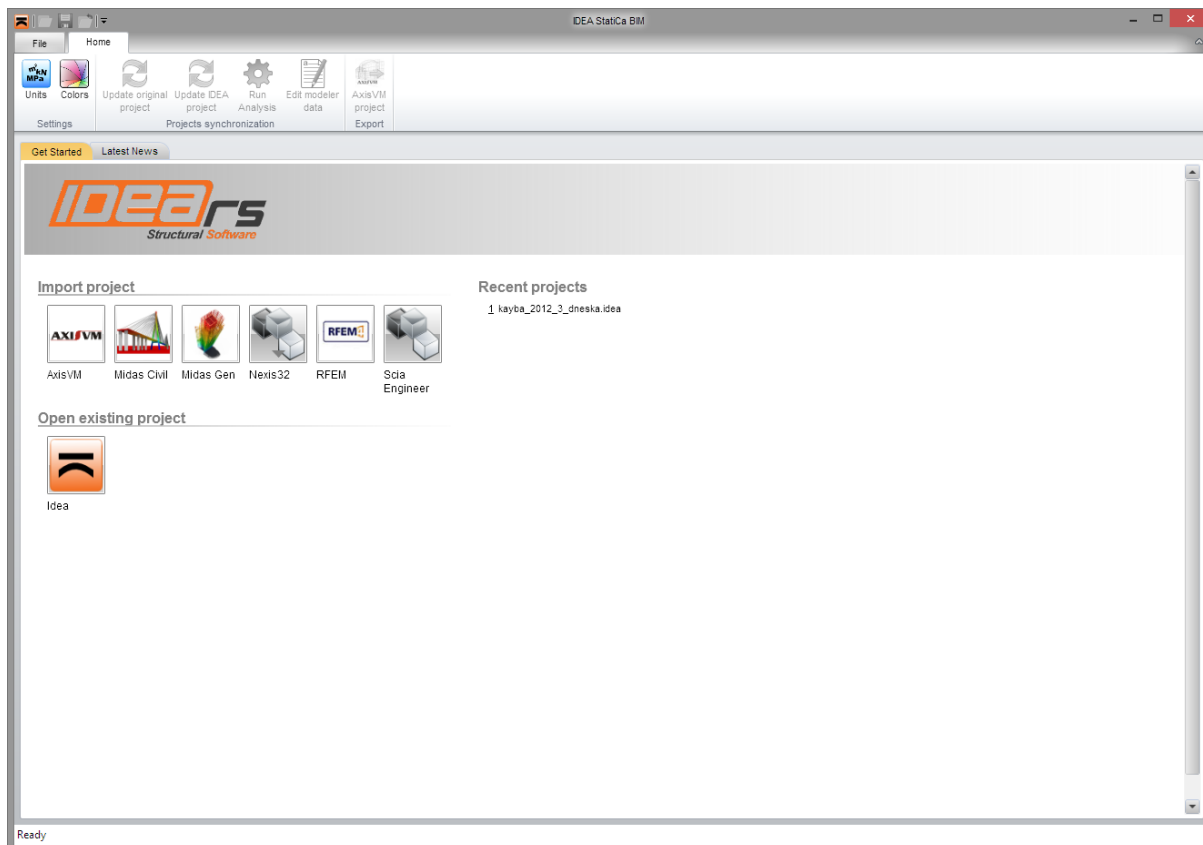
- **Scale** – if the option is selected, the scale ratio used to create the drawing in exported DXF can be set.
- **Output units** – select units of the drawing in the exported DXF file.
- **Layers** – select the mode of layers generation. Layers can be generated according to the line type, the line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

**Dimensions** – switch on/off export of dimension lines.

## 4 Working with IDEA Designer

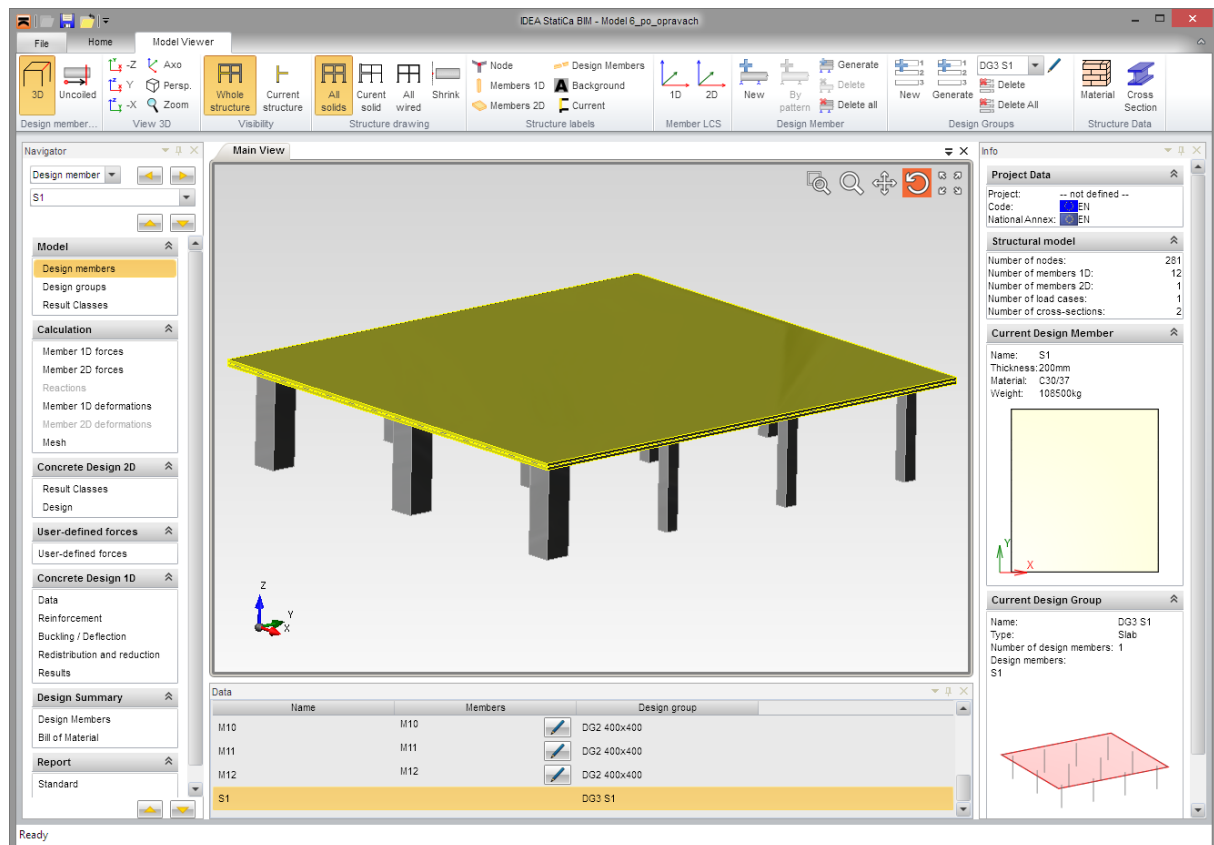
### 4.1 Opening a project

When the application is started an starting screen appears. An existing IDEA project can be opened or a project from any collaborating FEA program can be imported.



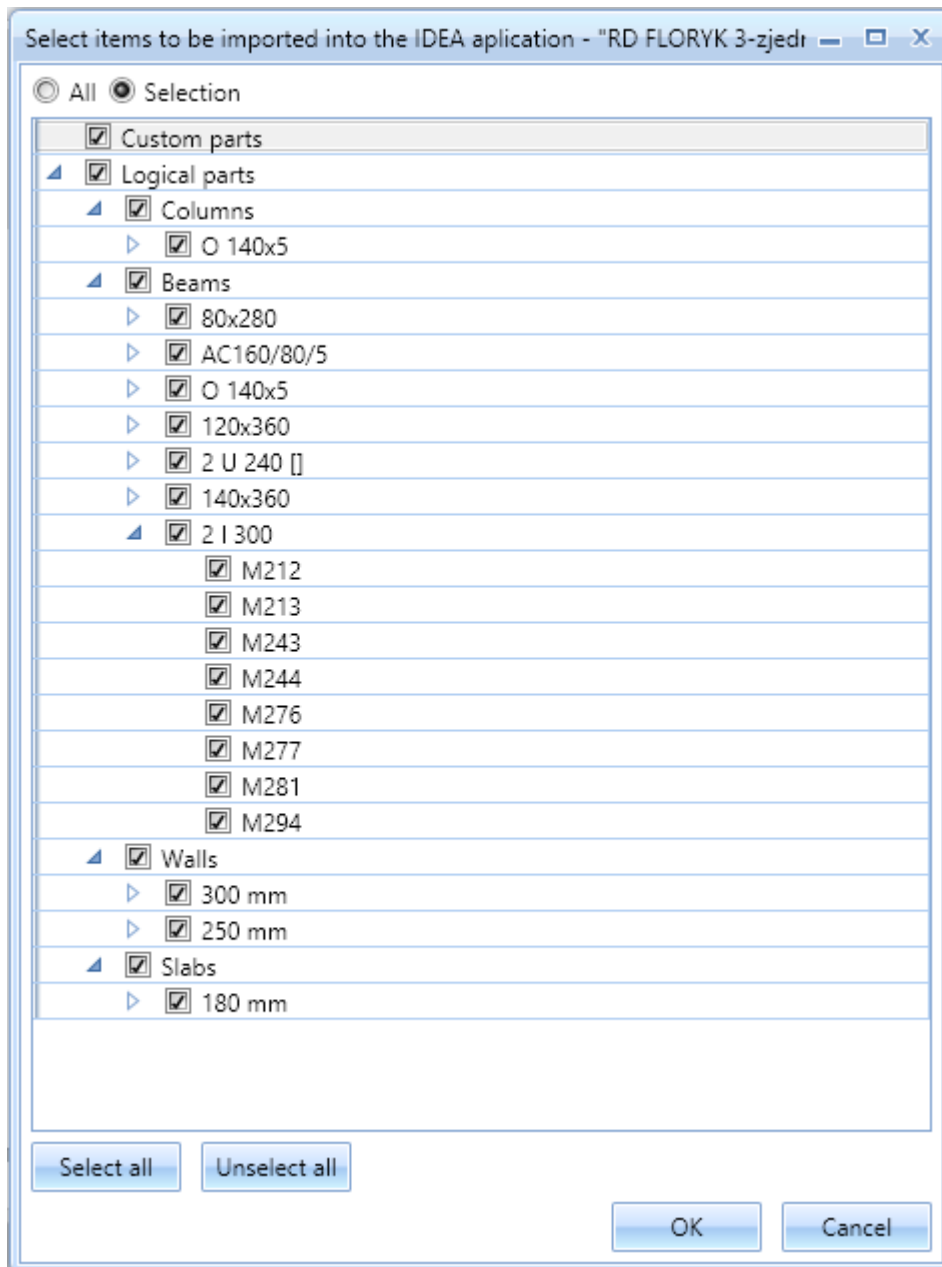
- **AxisVM** – import a structure modelled and calculated in AxisVM. The import requires the installation of AxisVM version 3e or higher.
- **Midas** - import a structure modelled and calculated in Midas Civil. The \*.MCT and \*.MID files generated by Midas Civil are used to import the structure.
- **Nexis 32** – import a structure modeled and calculated in Esa Prima Win. The import requires the installation of Esa Prima Win version 3.30 or higher.
- **SCIA Engineer** –import a structure modeled and calculated in SCIA Engineer. A XML file generated by a program SCIA Engineer is used to import the structure.
- **Idea** – open an IDEA file stored in IDEA Designer.

The ribbon tab **Model viewer** is activated when the project is opened. The drawing of the structure appears in main 3D window.



### 4.1.1 Imported parts selection

Structural parts and members to be imported can be selected when importing data from AxisVM. If member is not selected to be imported, only its axis is drawn, internal forces cannot be evaluated on the member and the member cannot be designed.



After choosing the imported file a dialog **Select items to be imported into IDEA application** appears.

Dialog options:

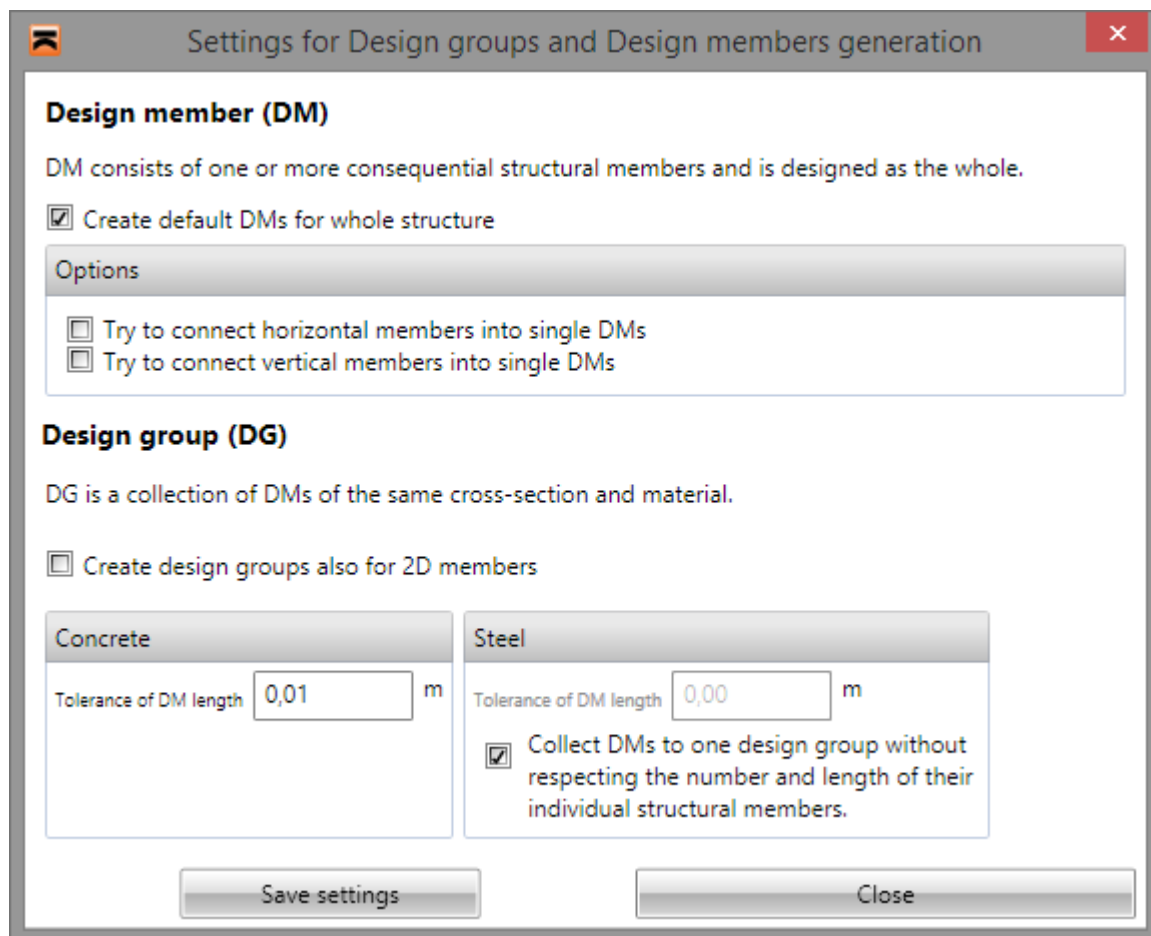
- **All** – if the option is selected, all structural parts and members are imported from AxisVM data.
- **Selection** - if the option is selected, structural parts and members can be selected to be imported from AxisVM data.



- **Data tree** – a tree view of structural parts, which corresponds to parts defined in AxisVM, is displayed in the dialog. Each individual part of member can be selected or unselected to be available or unavailable after import.
- **Select all** – selects all parts and members to be imported.
- **Unselect all** – de-selects all groups and members.

### 4.1.2 Settings for generation of design members during the import

In the moment of initial import of the structure from FEA programs the import options for creation of design members and design groups can be set.



Particular dialog options:

- **Generate default DMs member for whole structure** – if the option is checked, the design members are created from all imported structural members. If the option is off, the design members are not created during the import and they have to be created manually to be able to design them.
- **Try to connect horizontal members into single DMs** – if the option is checked, it is evaluated during the import, if more consecutive horizontal members can be joined to one design member. If it is possible, the design members containing more structural members are created.
- **Try to connect vertical members into single DMs** – if the option is checked, it is evaluated during the import, if more consecutive vertical members can be joined to one design member. If it is possible, the design members containing more structural members are created.
- **Tolerance of DM length (concrete)** – input the permissible length difference to assign the design members into the design group. The lengths of individual structural members in the design members must be identical within the defined tolerance to assign the design members to one design group.
- **Collect DMs to one design group without respecting the number and length of their individual structural members** – if the option is selected, the design groups are created per each cross-section. Design members of different lengths are assigned to

the design group. If the option is not selected, the length tolerance can be defined. The design members of the same length within the specified tolerance and the same cross-section are assigned to one design group.

- **Tolerance of DM length (steel)** – input the permissible length difference of design members to assign the design members to the design group of steel members.
- 
- **Save settings** – save the current settings as the default values to be used for next import of the structure.

### 4.1.3 Creating a new project importing EsaPrimaWin data

IDEA Designer can import data including results of linear calculation from program Esa Prima Win

Import can be performed when following conditions are met:

- Application Esa Prima Win has to be installed on the same computer where IDEA Designer is installed.
- Project, which should be imported, must be calculated and stored including calculation results.
- To import cross-sections correctly, input data should be exported from Esa Prima Win to StepSteel file (using command resp. File >Export>PSS). Options Analysis model-level 2 and To use DSTV conversion for cross-section names should be checked in export dialog.  
StepSteel file has to be stored into the same directory where EWP file of original structure is stored and has to have the same name as name of original file. To import eq. project with name MyProject, files MyProject.epw and MyProject.stp has to be in one directory.

To start import from Esa Prima Win click **Nexis32** on the starting screen of IDEA Designer.

It is possible to choose one of two modes how 1D elements are converted from Esa Prima Win to IDEA Designer:

- **IDEA member = EPW macro** – converts each 1D macro from EPW data to one member in IDEA Designer.
- **IDEA member = EPW member** – converts each 1D member from EPW data to one member in IDEA Designer.


### 4.1.4 Creating a new project importing AxisVM data

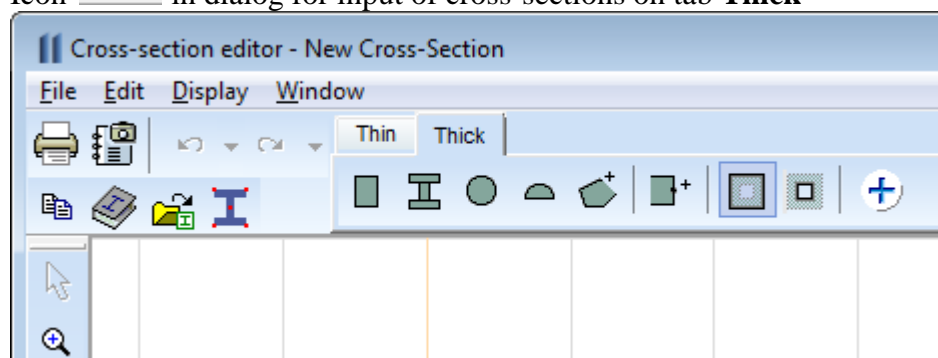
To enjoy all possibilities of connection between AxisVM and IDEA Designer, AxisVM version 3e and higher has to be installed on the same computer where IDEA Designer is installed.

To start import of AxisVM project click **AxisVM** on IDEA Designer starting screen.

Some add-ons are installed to AxisVM during setup of IDEA Designer. Those add-ons enable to use reinforcement templates for concrete sections during reinforcement design in IDEA RCS or to export data to IDEA Designer directly from AxisVM.

If the add-ons are installed correctly, you can find:

- Icon  in dialog for input of cross-sections on tab **Thick**



Clicking the icon displays dialog with cross-sections of the same shapes and parameters as have the cross-sections used in IDEA RCS. If the cross-section is defined using this add-on, it can be reinforced using appropriate reinforcement template in IDEA RCS. Cross-sections defined using other ways are exported to IDEA Designer as general cross-sections and have to be reinforced with general reinforcement

- Command Idea Prestressing in menu Plugins. This command runs module IDEA Tendon, which enables to define tendons and introduce effects of prestressing into calculated structure.

#### 4.1.5 Creating a new project importing SCIA Engineer data

To import data from SCIA Engineer XML file with appropriate data has to be exported from SCIA Engineer. Content of XML file is defined in XML document composer.

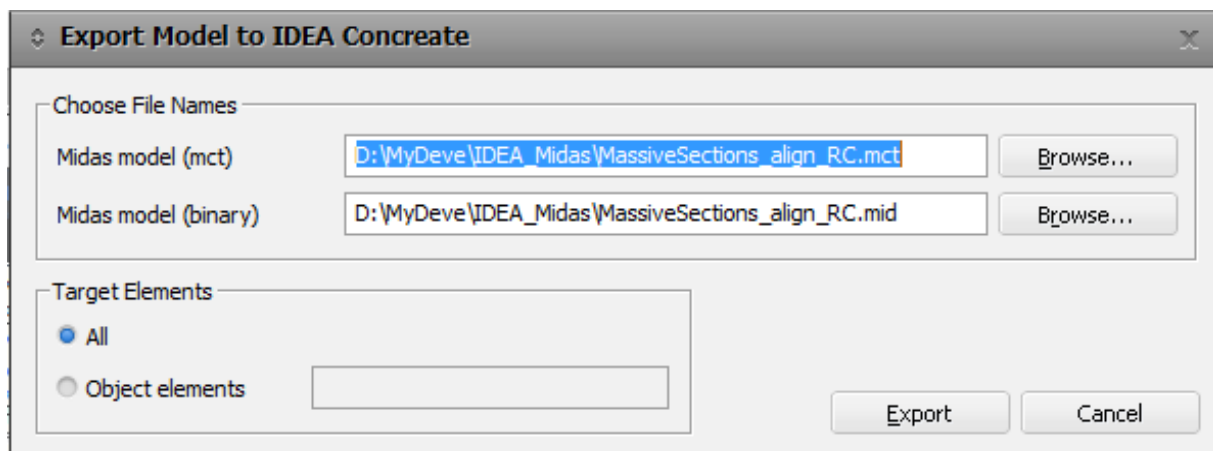
To start import from **SCIA Engineer** click **SCIA** on the starting screen of IDEA Designer.

To read more about import from SCIA Engineer see **12.3 Program SCIA Engineer**.

#### 4.1.6 Creating a new project importing Midas civil data

Files \*.MCT and \*.MID must be exported from Midas Civil to be able to import the structure to IDEA Designer.

The files contain data describing either the whole structure or only the selected part of the structure.



To read more about import from Midas Civil see 12.4 Program Midas Civil.

#### 4.1.7 Update of IDEA projects

IDEA Designer projects, which were created importing data from other programs, can be updated to reflect changes in original (source) projects.

IDEA Designer project created from AxisVM supports bi-directional update. It means that changes due to design in IDEA Designer can be transferred to original AxisVM project.



To update projects use commands in ribbon group **Projects synchronisation** on ribbon tab **Home**.

#### 4.1.8 Update of projects imported from Esa Prima Win

Update of IDEA Designer projects, which were created by import of data from Esa Prima Win, is limited to update of results of linear analysis. No other changes can be transferred to IDEA Designer project.

It means that if cross-section was changed during design in IDEA Designer, this change can be done manually in original Esa Prima Win project. Than original project can be recalculated and IDEA Designer project can be updated to refresh internal forces from original project.

To update IDEA Designer project click **Update IDEA project** in ribbon group **Projects synchronisation** in ribbon tab **Home**.

#### 4.1.9 Update of projects imported from SCIA Engineer, Midas, RFEM

Update of projects imported from SCIA Engineer, Midas or RFEM consist of following steps (which are performed automatically):

- The current state of IDEA project is backed up – file *„ProjectName.X.idea“* is created, where X is the order number of import.
- New project is created by fresh import of changed source data (exported from SCIA Engineer, Midas or RFEM).
- Changes of data from original IDEA project are applied to the newly imported project.

The log file of update can be displayed after the update finishes.

To update IDEA Designer project click **Update IDEA project** in ribbon group **Projects synchronisation** in ribbon tab **Home**.

#### 4.1.10 Update of projects imported from AxisVM

Following updates can be performed for IDEA Designer projects created by import from AxisVM:

- **Update of original (AxisVM project)** – transfer changes due to design in IDEA Designer (dimensions of cross-sections) to original AxisVM project. To update original AxisVM project click **Update original project** in ribbon group **Projects synchronisation**.  
**Note: This update performs changes in original AxisVM project file and saves the changes!**
- **Update of IDEA Designer project** – transfer changes (dimensions of cross-sections, updated calculation results...) undertaken in AxisVM project file to current IDEA

Designer project. To update IDEA Designer project click **Update IDEA project** in ribbon group **Projects synchronisation**.

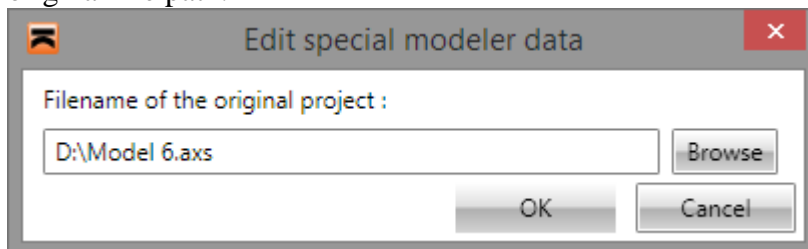
- **Recalculation of project in AxisVM** – transfer changes due to design in IDEA Designer (e.g. cross-section dimensions) from IDEA Designer project to original AxisVM project. Calculation of AxisVM original project is performed on background. After calculation changes from AxisVM project are transferred to IDEA Designer project.

To recalculate project click **Run analysis** in ribbon group **Projects synchronisation**.

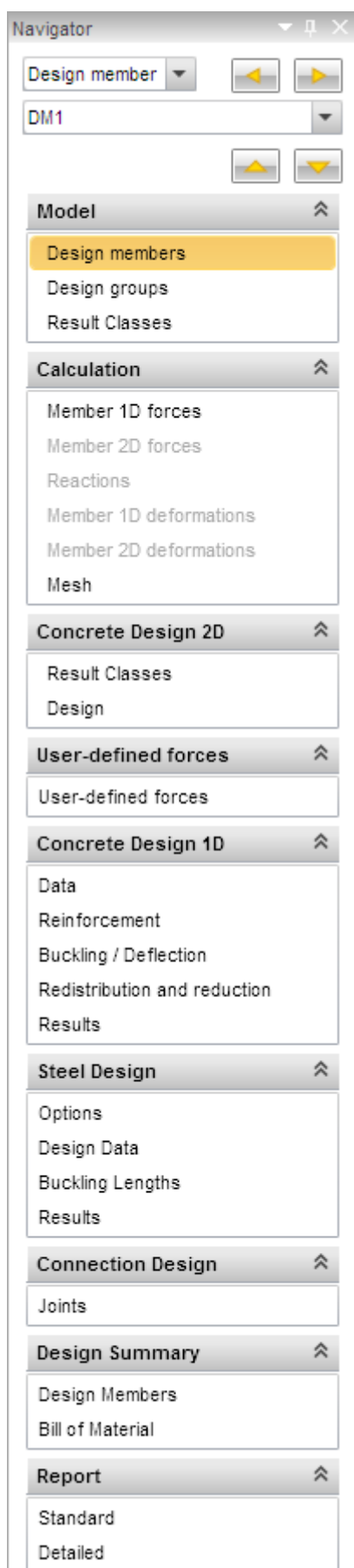
**Note: This update performs changes in original AxisVM project file and saves the changes!**

#### 4.1.11 Change the path to the source file

The path to the original (source) \*.axs file can be changed for project imported from AxisVM. The path could be changed e.g. after moving the original source file to the other directory. Click **Edit modeller data** in the ribbon group **Projects synchronisation** to change the original file path.



## 4.2 Model viewer



All user activities are conducted through Navigator. User goes through specific items, view presented data and controls his work by actions called from the ribbon. Commands for looking over the model and for the preparation of data for checks are collected into 4 sets of the Navigator:

- **Model** – check of imported model, edit of result classes and design groups.
- **Calculation** – check of results of FEM analysis of a structure. Internal forces or deformations on specific structural members or on the whole structure can be reviewed.
- **Concrete design 2D** - preparation of the check of 2D concrete members in IDEA RCS.
- **User defined forces** – input of user defined internal forces on structural members.
- **Concrete design 1D** – definition of reinforcement zones, check of reinforcement along the design member, calculation of deflections of the design member.
- **Steel design** – input of buckling parameters, setting the design data, checking the steel members.
- **Connection design** – export of data for design and check of joint in IDEA Connection application (members, cross-sections, internal forces).
- **Design summary** – displaying the check results of the whole structure, optimization of the steel members and generating the bill of material.
- **Report** – generating and printing the output report.



### 4.3 Load cases, combinations and result classes


Definitions of load cases and combinations are imported from the source application data. According to the content of imported combinations default result classes are generated.

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

Click navigator command **Model > Result Classes** to display the list of all existing result classes in the project in the Main window.



Ribbon group **Result Manager** is available for this navigator command:

- **Load groups** – input and modification of load cases groups– see **4.3.1 Groups of load cases**.
- **Combinations** – input and modification of combinations of load cases – see **4.3.2 Combinations of load cases**.
- **Result classes** – input and modification of result classes – see **4.3.3 Result classes**.
- **Current class** – name of the current result class, for which internal forces and deformations can be evaluated, is displayed. Click edit button  to display the dialog to select the current result class, combination or load case.

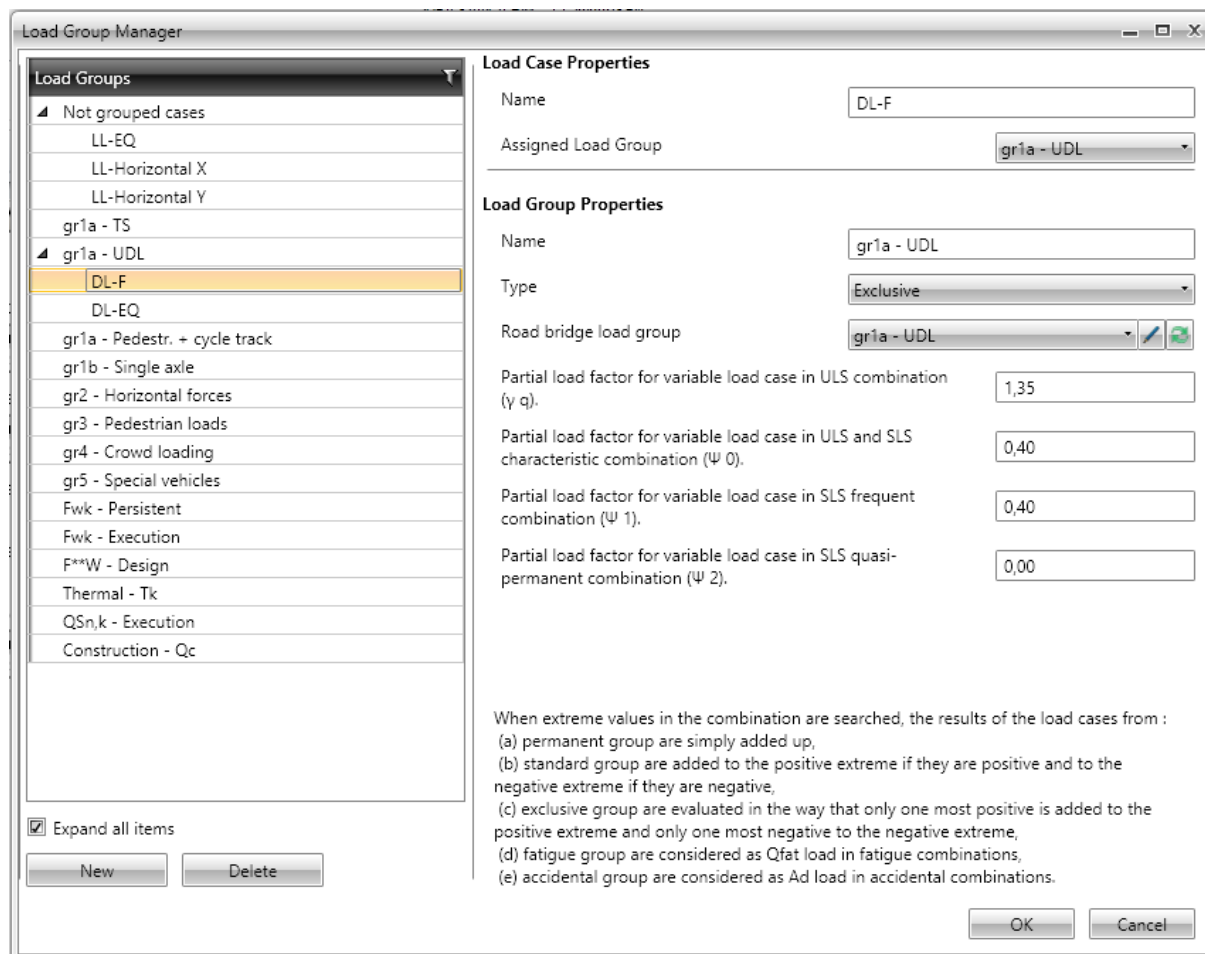
### 4.3.1 Groups of load cases

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.

Load cases groups, imported from AxisVM, cannot be edited in IDEA Designer. Load cases groups have to be modified in AxisVM, afterwards the IDEA project should be updated.

To view or modify load cases groups click **Load groups** in ribbon group **Results manager**.



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.
- **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.

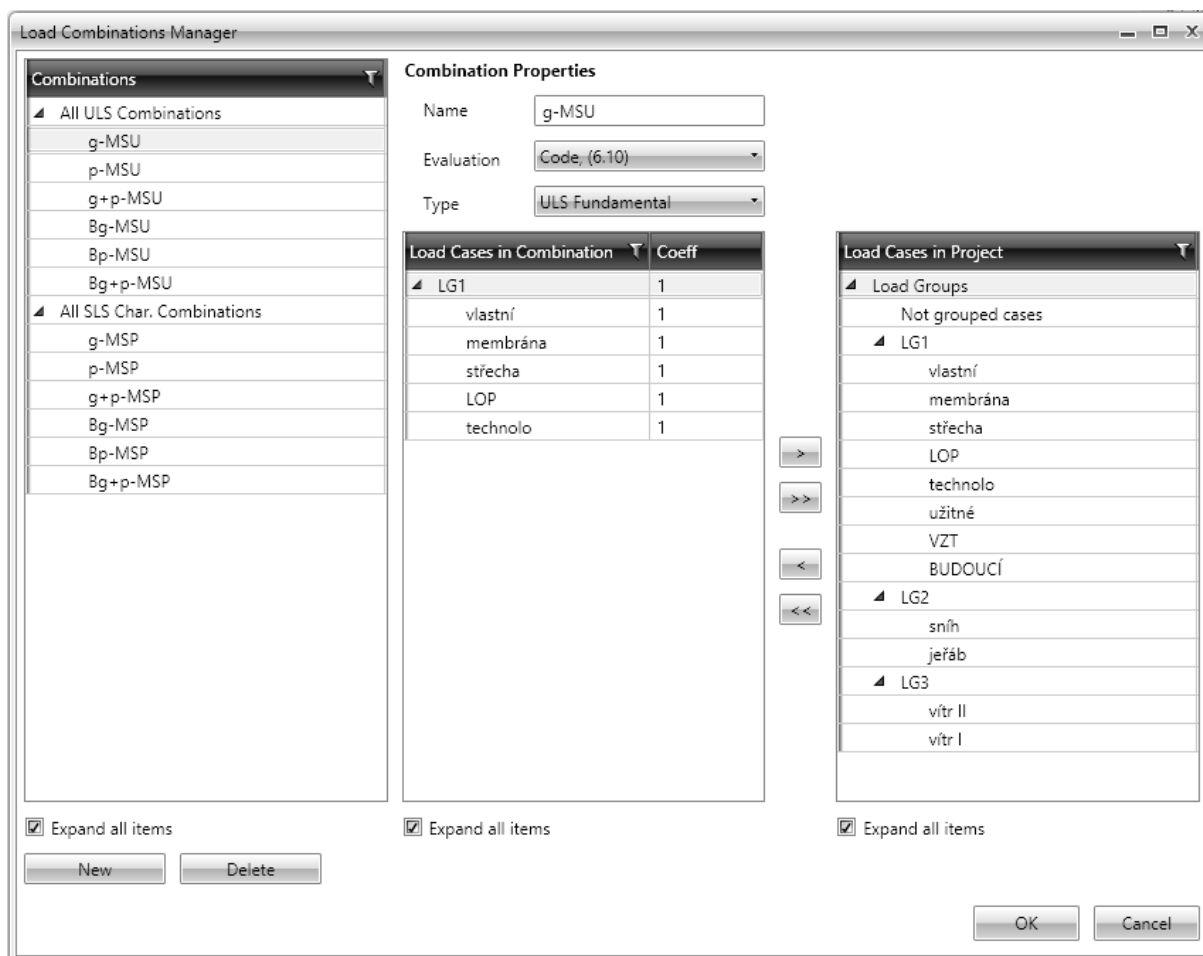
#### Load group properties:

- **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.
    - $\xi$  – input value of reduction coefficient for unfavourable permanent actions.
  - **Standard** – load group for 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:
    - $\gamma_q$  – input value of partial load factor for variable load cases in ULS combinations.
    - $\psi_0$  – input value of partial load factor for variable load cases in ULS and SLS Characteristic combinations.
    - $\psi_1$  – input value of partial load factor for variable load cases in SLS Frequent.
    - $\psi_2$  – 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.
  - **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 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  $Q_{fat}$ . 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.

### 4.3.2 Combinations of load cases

To view or modify combinations of load cases click **Combinations** in ribbon group **Results manager**.



Combinations of load cases are important to determine the extremes of load actions. It is possible to input four 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.

Following type of evaluation 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.





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.

Particular 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.
- **Load cases in combination** – load cases 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.
-  - remove selected load case or load cases group from the current combination.
-  - remove all load cases from the current combination.
-  - add load case or load group, selected in the tree view **Load cases in project**, to the current combination.
-  - add all load cases from the tree view **Load cases in project** to the current combination.

### 4.3.3 Result classes

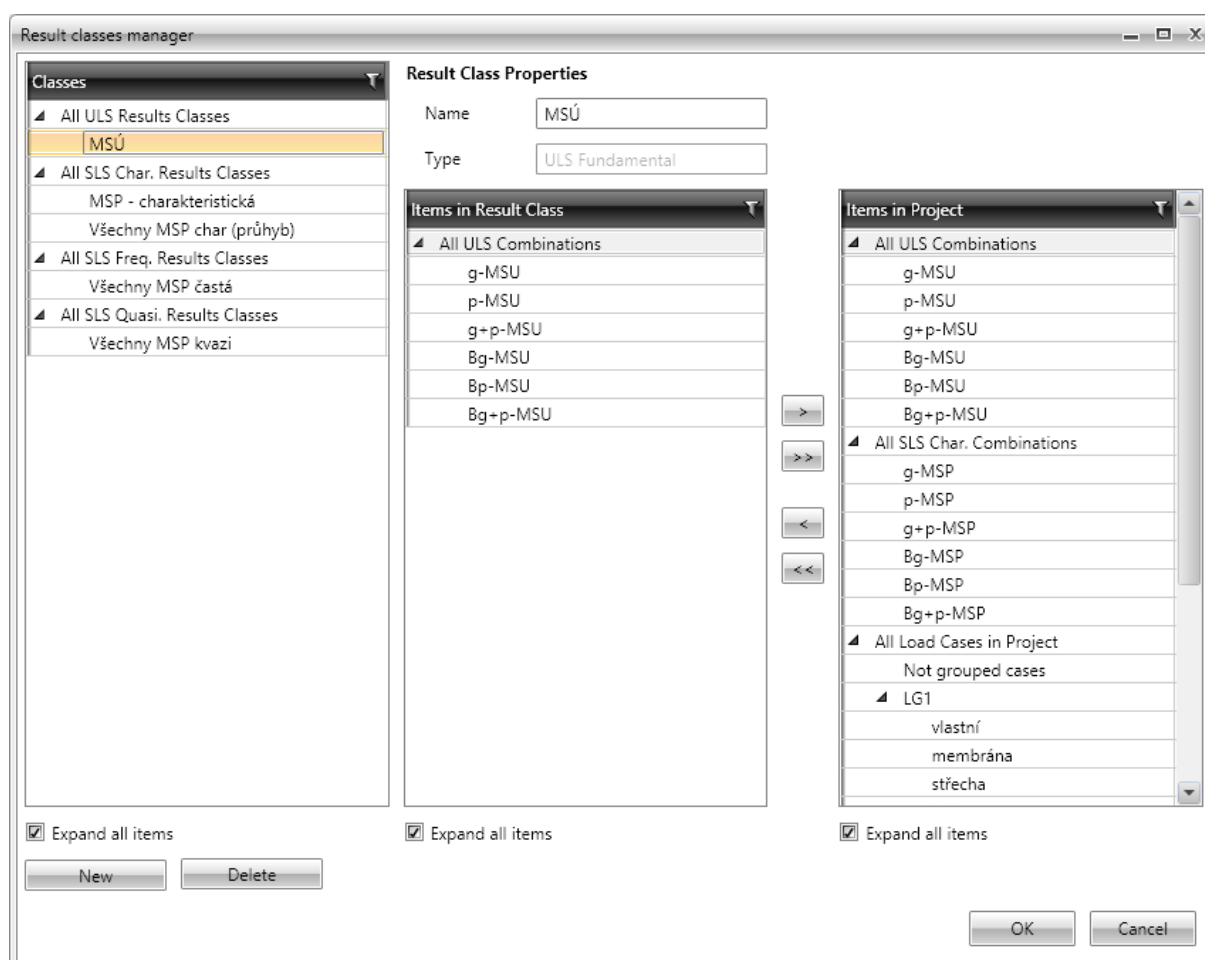
To view or modify result classes click **Result classes** in ribbon group **Results manager**.

Load cases or combinations are grouped in result classes. The extreme values of evaluated magnitude from all items in result class are searched when evaluating results for the current result class.

One of four types can be assigned to each result class. According to its type the result class can be used for appropriate checks.

One of following types can be assigned to each result class:





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



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.

**Combination properties:**

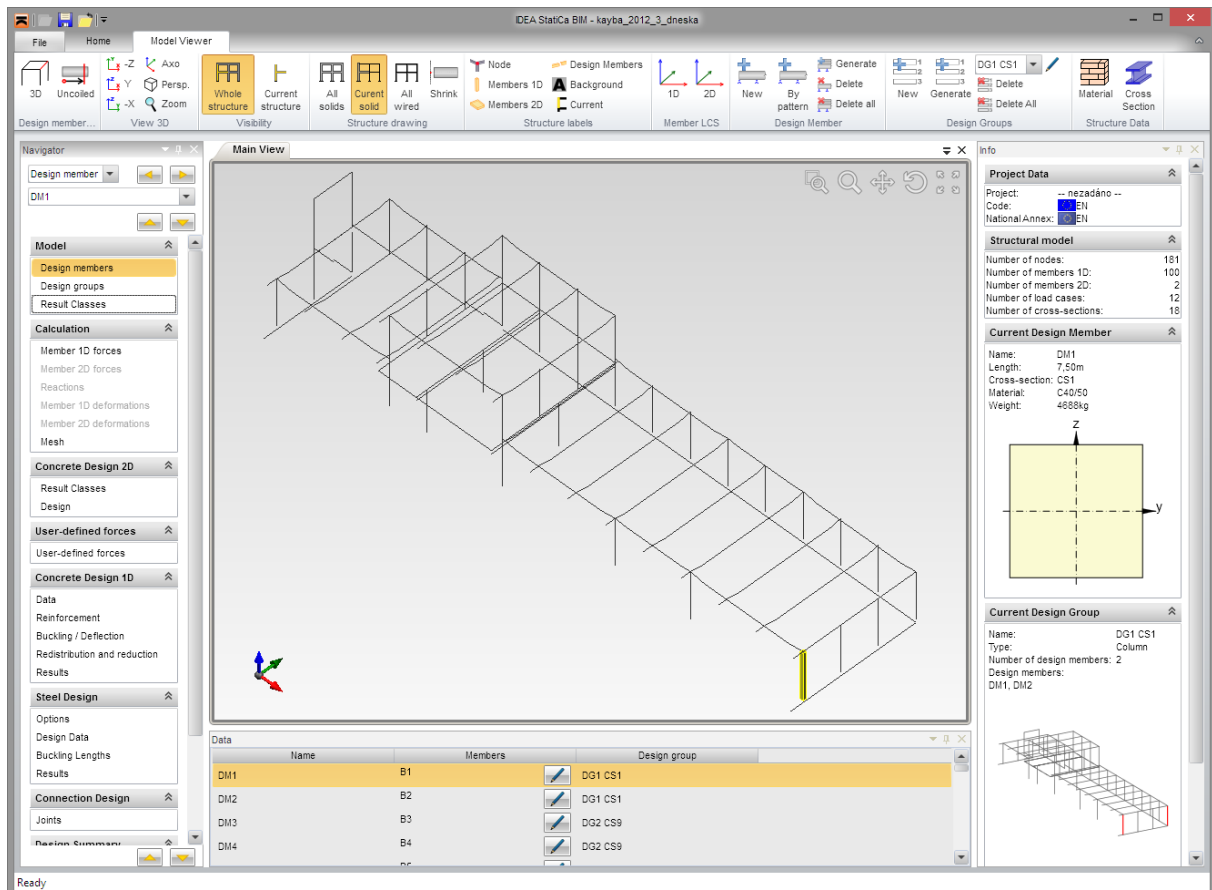
- **Name** – input name of the current result class.
- **Type** – select the type of the current result class.
- **Items in result class** – combinations and load cases, assigned to the current result class, are displayed in the tree view. Combinations are grouped according to their types and load cases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



## 4.4 Working with members and design groups


Program can work in either **Design member** or **Design group** mode. To switch between these two modes, use combo-box on the top of **Navigator**.

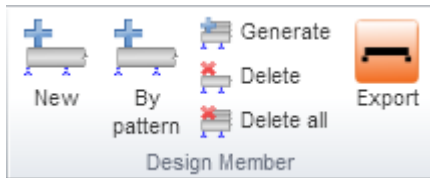
The second combo-box contains the list of all design members or design groups. One design member or design group is chosen as current. Current design member or group is drawn as highlighted in the main window. Brief information about current design member and design group is displayed in the Info window.



## 4.5 Design members

Click navigator command **Model > Design members** to work with design members.

The table with all design members is displayed in the Data window. Click edit button  in the column **Members** to display dialog with design member properties.



Use commands in ribbon group **Design member** to create or edit design members:

- **New** – create new design member – see **4.5.1 New design member**.
- **By pattern** – create new design member according to the pattern design member – see **4.5.2 Creating new design member by pattern**.
- **Generate** – automatically generate design members from all structural members, which are not assigned to any design member.
- **Delete** – delete current design member. Structural members, which were assigned to the deleted design member, stay not assigned – it means that they are not a part of any design member.
- **Delete all** – delete all design members, thus no structural member is assigned to design member.
- **Export** – start wizard to export the current design member to data file of IDEA Beam application. Export is enabled only for concrete members assigned to design group of beam member type.

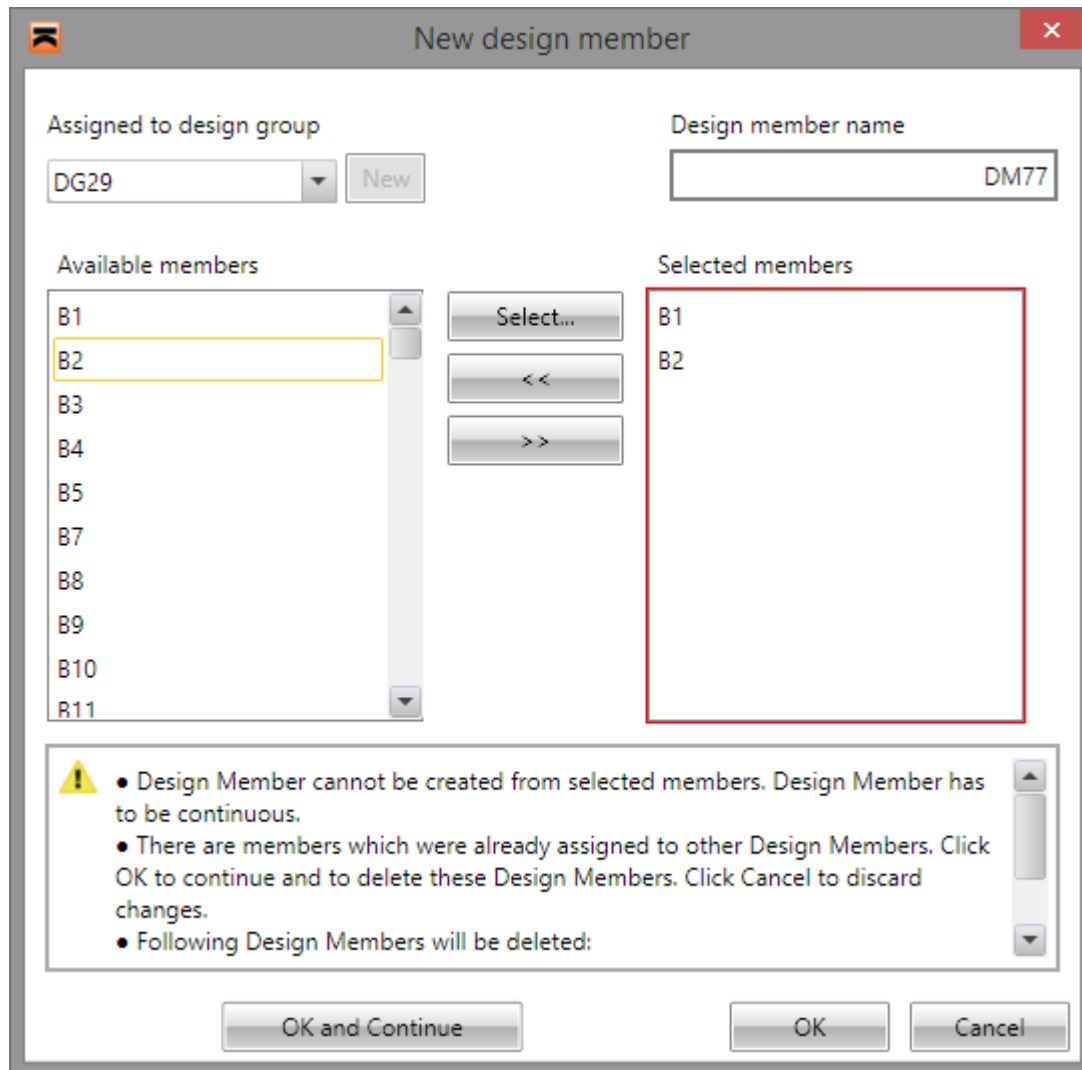
### 4.5.1 New design member

Click **New** in ribbon group **Design member** to create new design members.

To be able to create a design member, the structural members selected to create the design member have to be consequent – must have common nodes.

Newly created design member may be assigned either to newly created design group or an existing design group can be selected from the list of suitable existing design groups (after first structural member is assigned to design member).

Nonconformities, which were found during creation of a design member, are displayed in the bottom part of the dialog.



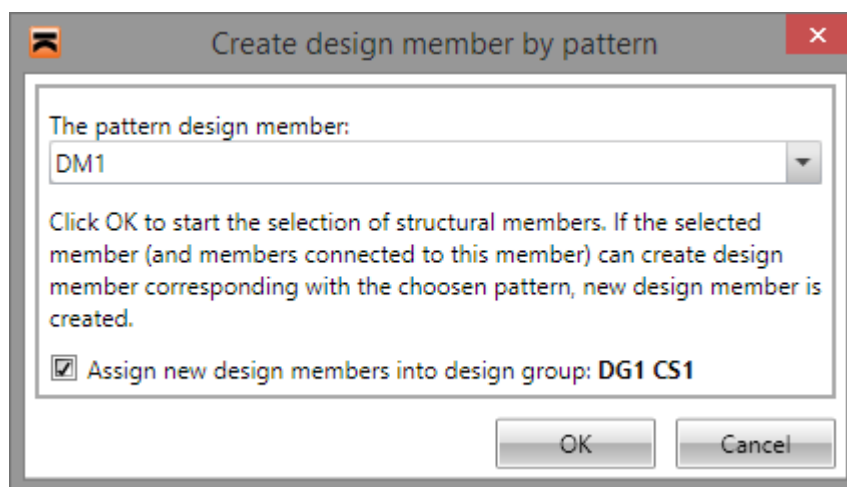
Particular dialog options:

- **Assigned to design group** – select design group from the list, to which newly created design member will be assigned.
- **New** – create new design group. The newly created design member can be assigned to the newly created design group.
- **Design member name** – input name of the new design member.
- The list **Available members** contains numbers of structural members, which can be assigned to newly created design member

- The list **Selected members** contains numbers of structural members, which will create a new design member.
- >> - add structural members, which are selected in the list **Available members**, to the newly created design member.
- << - remove structural members, which are selected in the list **Selected members**, from the newly created design member.
- **Select** – start the interactive selection of structural members in 3D window – see **4.5.3.1 Selecting members in 3D window**.
- **OK and continue** – create new design member according to the content of the list **Selected members** and display dialog New design member again.

#### 4.5.2 Creating new design member by pattern

To create new design member according to the pattern design member click **By pattern** in ribbon group **Design member**.



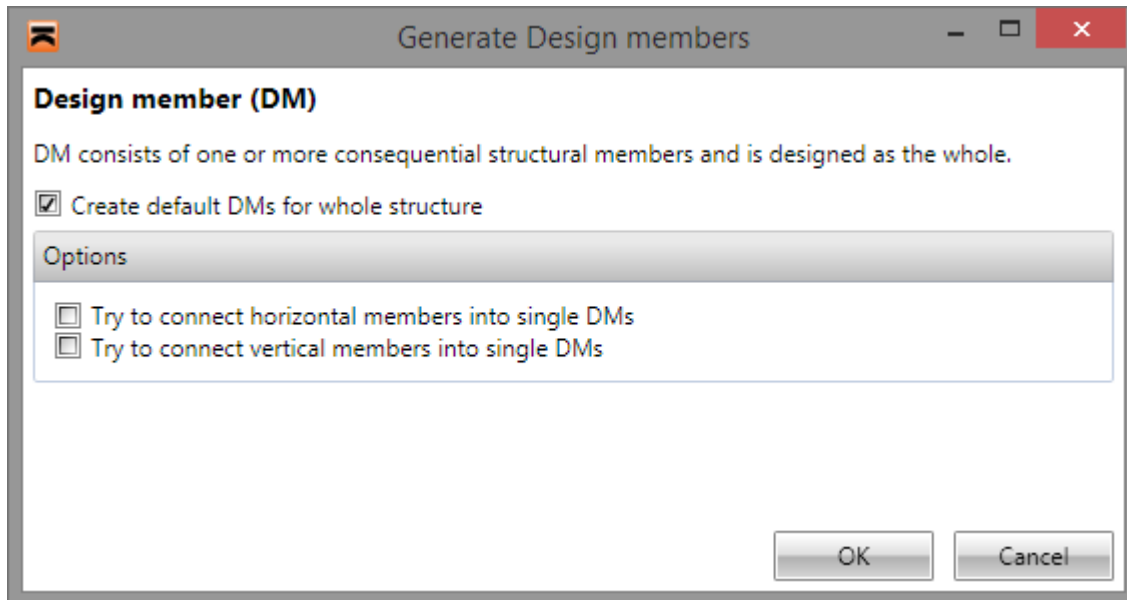
- **The pattern design member** – select a design member in the list, which properties are compared with selected structural members. If the selected structural member fulfils the conditions to create the new design member according to the pattern design member, the new design member is created.
- **Assign new design members to design group** – if the option is checked, the newly created design members are assigned to the design group of the pattern design member.

Click OK to start the selection of structural members in 3D window (see **4.5.3.1 Selecting members in 3D window**). The starting structural members of new design members should be created.

### 4.5.3 Generating new design members

The design members can be generated en masse. The generation of design members processes only the structural members, which haven't been assigned to any design member yet.

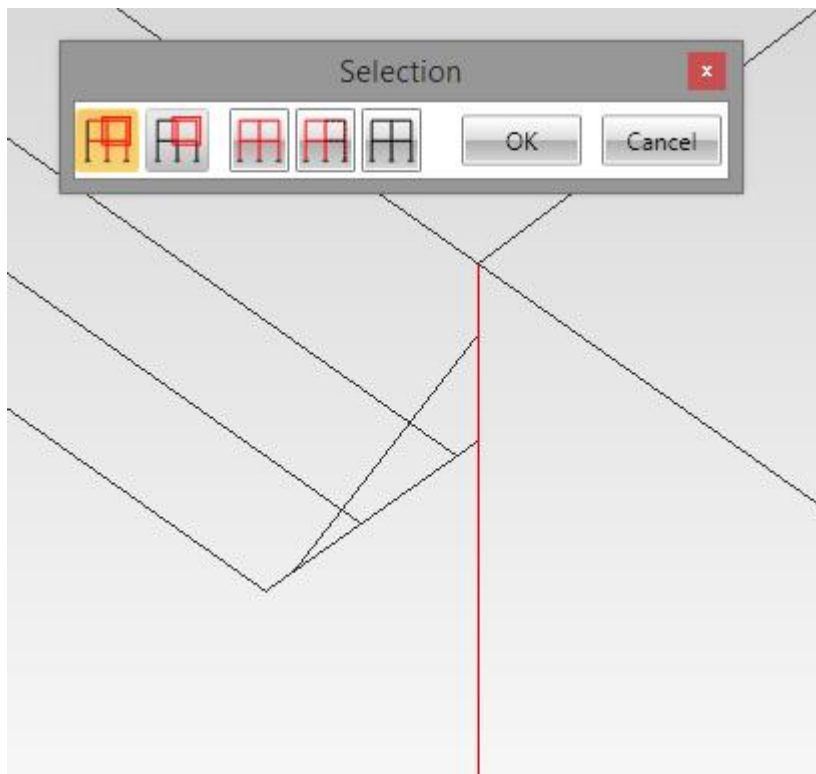
To generate design members click **Generate** in ribbon group **Design member**.



- **Generate default DMs for whole structure** – if the option is selected, the design members are created from all structural members, which haven't been assigned to any design member yet.
- **Try to connect horizontal members into single DMs** – if the option is selected, it is evaluated during the generation of design members, if more consecutive horizontal members can be joined to one design member. If it is possible, the design members containing more structural members are created.
- **Try to connect vertical members into single DMs** – if the option is selected, it is evaluated during the generation of design members, if more consecutive vertical members can be joined to one design member. If it is possible, the design members containing more structural members are created.
- **Save settings** – save the current settings as the default values to be used for next import of the structure.

#### 4.5.3.1 Selecting members in 3D window

A toolbar to set the selection mode appears in the left top part of the 3D window after starting the selection in 3D window.



To switch to one of following selection modes click appropriate icon:



- single selection. Click the member to add it to the selection. Click the selected member to remove it from the selection.



- selection by rectangle. Draw a rectangle by mouse to add members inside the rectangle to the selection. More than a half of the length of rectangle must be inside the rectangle.



- Select all. Select all structural members.



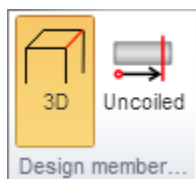
- invert the selection.



- remove selection. Remove all members from selection.

After the selection is finished, click OK to add the numbers of selected members to the list of structural members, which are assigned to the design member.

#### 4.5.3.2 Display modes of design members



When navigator **Model > Design members** is active, display mode of design member can be changed using commands in ribbon group **Design member view**.


- **3D** – switch to display the whole structure in 3D window. The current design member is drawn according to settings in ribbon group **Structure**

**appearance.**

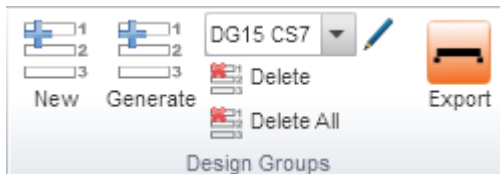
- **Uncoiled** – switch to drawing of the current design member only in uncoiled views.

## 4.6 Design groups

Click navigator command **Model > Design groups** to work with design groups.


There is a list of all design groups in the data window. Clicks edit button  beside the name of a group to open the edit dialog of appropriate group.

When the navigator command **Model > Design group** is active, besides the ribbon groups for view settings a ribbon group **Design Groups** is available. Following commands are available:



- **New** – add a new group into the list of existing ones – see **4.6.1 New Design Group**
- **Generate** - automatically generate design groups. The generation assigns to design groups only those design members, which are not assigned to any

design group.

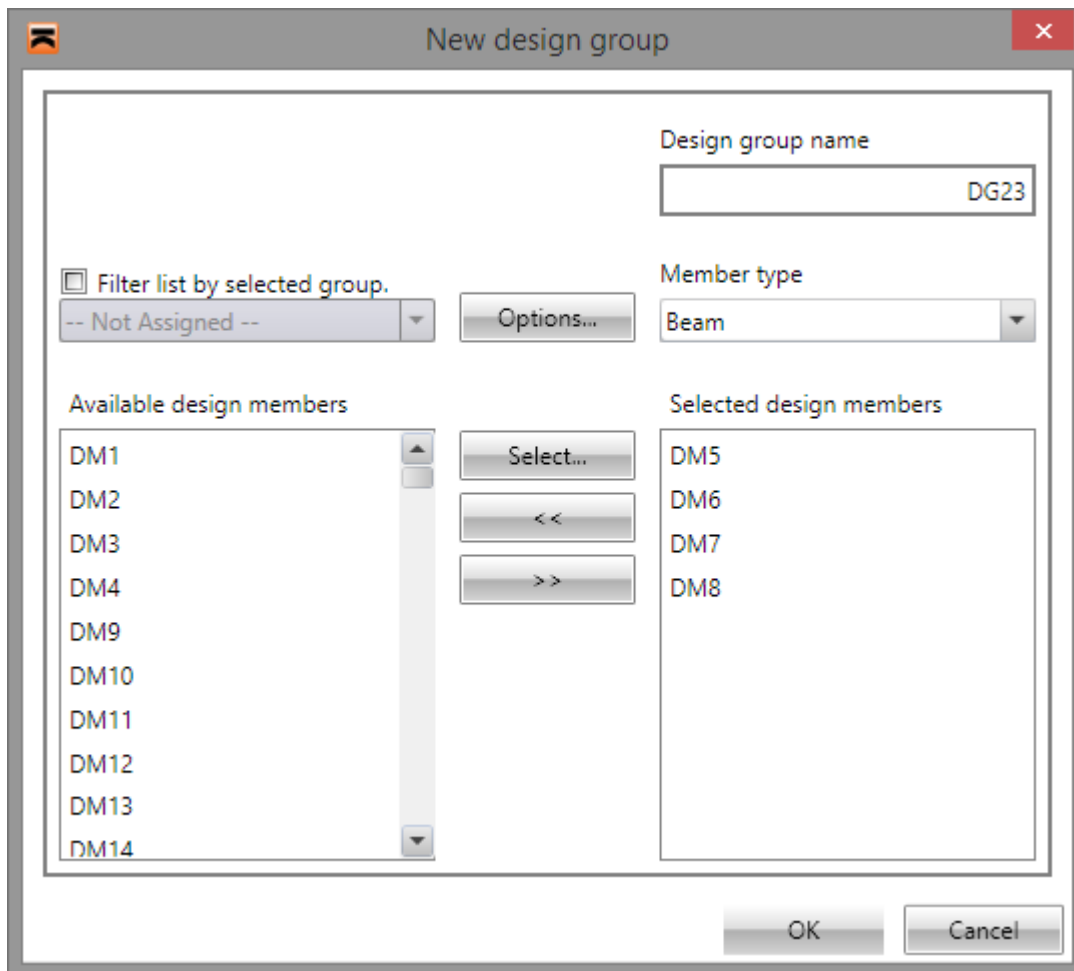
- **List of groups** –select the current group. To edit group click on edit button  beside the list, the edit dialog of this group is opened – see **4.6.1 New Design Group**.
- **Delete** – delete the current design group. Design members from the deleted group are not assigned to any designed design group.
- **Delete All** – delete all design groups. No design member is assigned to any design group.
- **Export** – start wizard to export the representative design member of the current design group to data file of IDEA Beam application. Export is enabled only for design groups of beam member type, containing concrete members.



### 4.6.1 New Design Group

Click **New** in ribbon group **Design Group** to start input of a new design group.

The design group is defined by its name and the list of design members, which are assigned to the design group.



Particular dialog options:

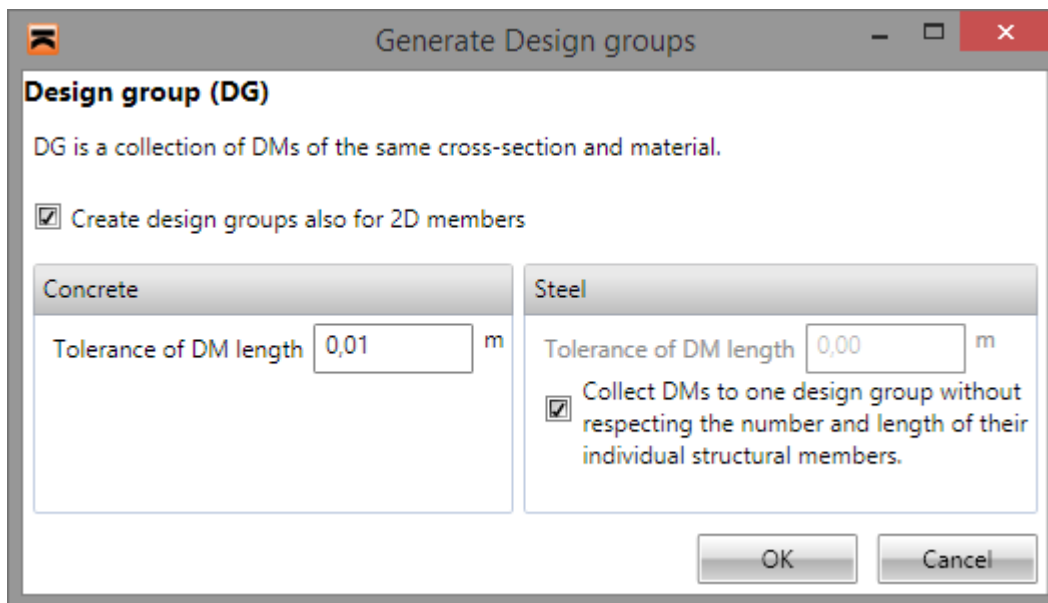
- **Available design members** - list contains all design members which can be used in a new group.
- **Selected design members** -list contains all design members which are currently added into the group.
- **Filter list by selected group** – if the option is switched on, only members of design group selected in combo box below are visible in the **Available members** list
- **Design group name** – input the name of a new design group
- **>>** - move items selected in the list **Available members** into a new design group. Items can be moved also by double-click on specific item of the list **Available design members**.
- **<<** - remove selected items from the list **Selected design members**. Items can be removed also by double-click on specific item of the list **Selected design members**.
- **Member type** – set the type of member for the design group. According to the member type the appropriate checks are available for the design group.

- **Options** – setting of length tolerance, which is taken into account when assigning design members to design group. The tolerance value is valid for the current design group.

After assigning all required design members into a new design group, click **OK**. The check is performed, if all selected design members can be assigned to the new design group.

#### 4.6.2 Generation of design groups

To generate design groups from design members, which are not assigned to any design group, click **Generate** in ribbon group **Design groups**.



**Design groups** dialog options:

- **Tolerance of DM length (concrete)** – input the permissible length difference to assign the design members into the design group. The lengths of individual structural members in the design members must be identical within the defined tolerance to assign the design members to one design group.
- **Collect DMs to one design group without respecting the number and length of their individual structural members** – if the option is selected, the design groups are created per each cross-section. Design members of different lengths are assigned to the design group. If the option is not selected, the length tolerance can be defined. The design members of the same length within the specified tolerance and the same cross-section are assigned to one design group.
- **Tolerance of DM length (steel)** – input the permissible length difference of design members to assign the design members to the design group of steel members.

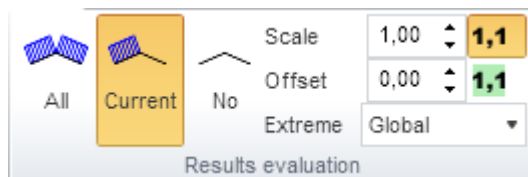
## 4.7 Calculation

Use particular navigator commands in **Calculation** group can to evaluate internal forces and deformations on 1D members and internal forces on 2D members.

Graphical courses of evaluated components for current member or design group are drawn in the main window. Numerical presentation of results is displayed into Data window.

Evaluation of calculation results is performed for the current Result class, which has been set in list in group **Result class** - see **4.3 Load cases, combinations and result classes**.

### 4.7.1 Results drawing settings



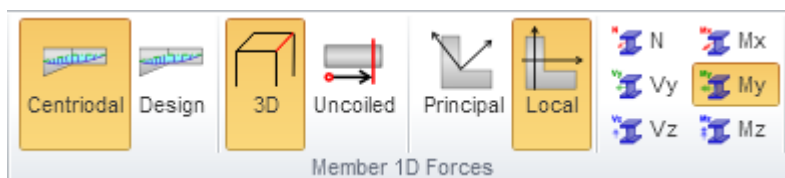
Results evaluation mode can be set using commands in group **Results evaluation**.

- **All** - switch to evaluate results on all members in the structure.
- **Current** - switch to evaluate results on current design group or design member only.
- **No** - do not draw evaluate results.
- **1,1** - switch on/off drawing of value numbers for evaluated components.
- **1,1** - switch on/off drawing of background under value numbers.
- **Scale** – to increase or decrease scale of results drawing, enter a number into edit box or click arrows on the right side of edit box.
- **Offset** – to increase or decrease the distance between graphs of results and drawing of structure, enter a number into edit box or click arrows on the right side of edit box.
- **Extreme** – select the type of evaluation of extremes of internal forces or deformations.
  - **Member** – extreme values of evaluated quantities will be searched on current member or design group.
  - **Global** – extreme values of evaluated quantities will be searched on the whole structure or its active part.

### 4.7.2 Internal forces on members

Click navigator command **Calculation > Member 1D forces** to evaluate internal forces on 1D members. To setup graphical representation of courses of evaluated components use commands in group **Member 1D forces**.

#### 4.7.2.1 Ribbon group Member 1D forces



- **Centroidal** –switch to evaluate internal the forces in sections perpendicular to the centroidal axis of members.
- **Design** –switch evaluate the internal forces in sections perpendicular to the reference line.
- **3D** – switch to evaluate the internal forces in 3D window.
- **Uncoiled** – switch to evaluate the internal forces on the uncoiled shape of the current design member(s).

- **Principal** – switch to evaluation of internal forces in principal axes of cross-section.
- **Local** – switch to evaluation of internal forces in local axes of particular members 1D
- **N, Vy, Vz, Mx, My, Mz** – switch to drawing of particular internal force component.

#### 4.7.3 Internal forces on 2D members

Click navigator command **Calculation > Member 2D forces** to evaluate internal forces on 2D members. To setup graphical representation of courses of evaluated components use commands in group **Member 2D forces, Result values** and **Analysis model labels**.

##### 4.7.3.1 Ribbon group Member 2D forces



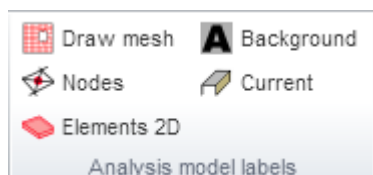
- **mx, my, mxy, vxz, vyz, nx, ny, nxy** – switch to drawing of particular internal force component.

##### 4.7.3.2 Ribbon group Result values



- **Isobands** – switch to drawing of courses of evaluated component in form of isobands.
- **Isolines** - switch to drawing of courses of evaluated component in form of isolines.
- **Maximum/Minimum** – select appropriate value in list to evaluate draw envelope either of **minimal** or of maximal values of evaluated component for current result class.
- **In centre** – switch to evaluation of internal forces in centres of finite elements.
- **In nodes** – switch to evaluation of internal forces in nodes of finite elements with averaging on current 2D member.

##### 4.7.3.3 Ribbon group Analysis model labels



- **Draw mesh** – switch on/off drawing of finite elements mesh
- **Nodes** – switch on/off drawing of node numbers. If drawing of mesh is switched on, numbers of mesh nodes are displayed too.
- **Elements 1D** – switch on/off drawing of 1D mesh elements numbers.
- **Element 2D** – switch on/off drawing of 2D mesh elements member.
- **Background** - switch on/off drawing of background under numbers.
- **Current** – switch between drawing mesh and numbers on the whole structure or on the current 2D member only.

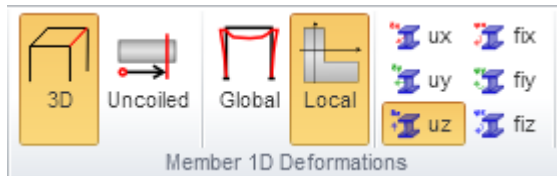
#### 4.7.4 Reactions

Reactions in supports are not available now. It is prepared to be used for check of foundation blocks, settlements etc.

#### 4.7.5 Deformations on 1D members

Click navigator command **Calculation > Deformations on 1D members** to start evaluation of deformations on 1D members To setup graphical representation of courses of evaluated components use commands in group **Member 1D deformations**..

#### 4.7.6 Ribbon group Member 1D deformations



- **3D** – switch to evaluate the deformations in 3D window.
- **Uncoiled** – switch to evaluate the deformations on the uncoiled shape of the current design member(s).
- **Global** – switch to evaluation of deformations in global coordinate system.
- **Local** – switch to evaluation of deformations in local coordinate system of particular 1D members.
- **ux, uy, uz, fix, fiy, fiz** – switch to drawing of particular deformation component.

#### 4.7.7 Deformations on 2D members

Member 2D deformations are not available now. It is prepared to be used for check of slab deflections.

#### 4.7.8 Mesh

It can be useful to review mesh and its parameters, especially when calculating structures with 2D members.

Click navigator command **Calculation > Mesh** to start drawing of mesh To setup graphical representation of mesh use commands in group **Labels of analysis model**.

##### 4.7.8.1 Ribbon group Labels of analysis model

See 4.7.3.3 group Analysis model labels.

## 4.8 User defined internal forces

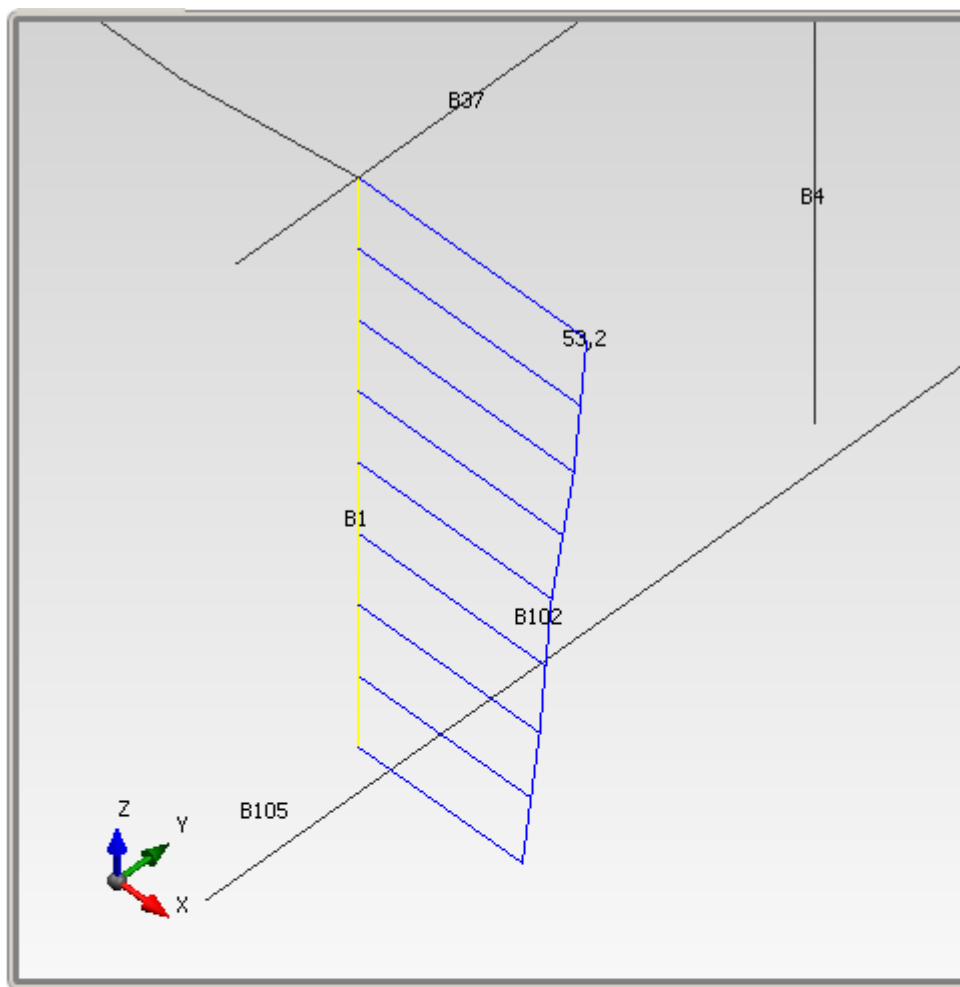
Additional internal forces, defined by user, can be added to original internal forces, imported with the structural model.

Values of user defined internal forces are transformed directly to internal forces in appropriate load cases on appropriate members and then are processed in the same way as internal forces imported with the structural model.

User defined internal forces can be defined manually (individual input or table copy eg. From Microsoft Excel).

Click navigator command **User defined forces** > **User defined forces** to define values of internal forces.

Ribbon group **Member 1D forces** is available.









**Data**

**User-defined forces**


Load case: vlastní

☒ Use user-defined forces   Positions: Relative   Action: Add

 Delete all Sort positions Generate positions

Members	X [-]	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]	
B1	0,00	-120,0	0,0	0,0	0,0	38,2	0,0	
B1	0,25	-120,0	0,0	0,0	0,0	42,2	0,0	
B1	0,50	-120,0	0,0	0,0	0,0	45,0	0,0	
B1	0,75	-120,0	0,0	0,0	0,0	50,2	0,0	
B1	1,00	-120,0	0,0	0,0	0,0	53,2	0,0	

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**:

- **Delete all** – remove all lines in the table of user defined internal forces (in the current load case).
- **Sort positions** – sort table lines in ascendant order according to the names of members and positions X.
- **Generate positions** – generate values of X-positions for each member name in the table. The position coordinates are generated depending of the number of rows per the member to split the positions regularly along the member.
- **Positions** – select the mode of internal force position definition:
  - **Absolute** – value of X-position is defined absolutely to the beginning of the member.
  - **Relative** – value of X-position is defined relative to the member length (0-1).
- **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.

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

- **Members** – name of member, on which the user force is defined.
- **X** – input of position of internal force.
- **N** – input of value of normal force N.
- **Vy** – input of value of shear force Vy.
- **Vz** – input of value of shear force Vz.
- **Mx** – input of value of torsional moment Mx.
- **My** – input of value of bending moment My.
- **Mz** – input of value of bending moment Mz.
-  - remove the corresponding user defined force.



## 5 2D concrete members design

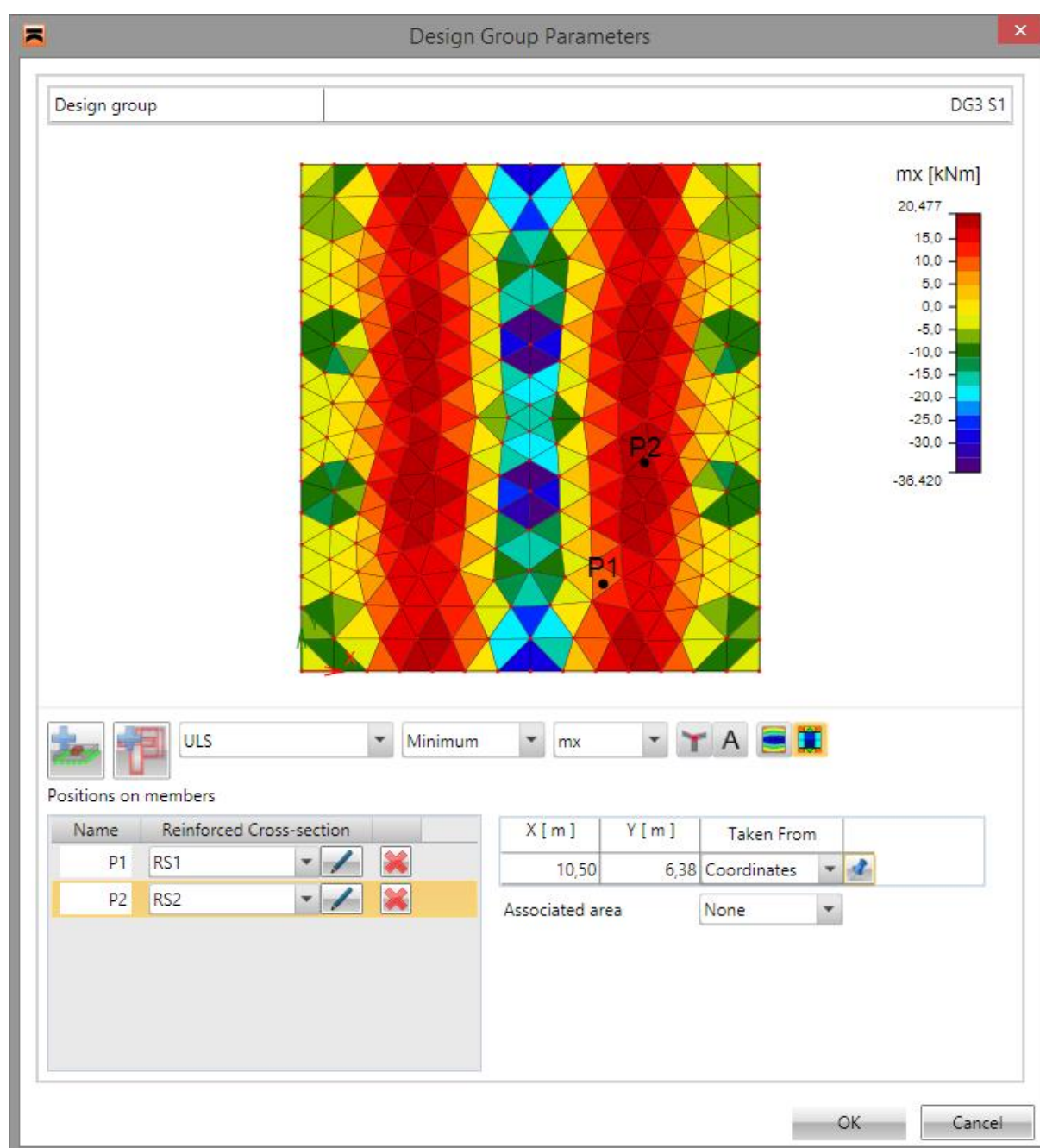
The check of concrete members is done using the module IDEA RCS.

Positions for design of reinforcement (points of 2D members) are defined on the 2D members. Type of reinforcement is assigned to each position.







### 5.1 Definition of points for reinforcement design

To input points for reinforcement design of 2D members design group click the edit button of the design group in the data window either in the navigator Model > Design groups or in the navigator **Concrete design 2D > Design**.



Position for design of reinforcement on 2D member is defined as a point on 2D member. Point can be defined using coordinates, number of mesh node or number of mesh finite element.




Dialog **Design Group Parameters** contains following options:

- **Design group** – input name of design group
- **ULS** – select current result class. For selected result class isobands of internal forces are drawn.
- **Minimum/Maximum** – select mode of internal forces drawing. It is possible to draw envelope either of maximal values or of minimal values of internal forces.
- **my** – select component of internal forces to be drawn.
-  - turn on/off drawing of mesh node numbers.
-  - turns on/off drawing of mesh element numbers.
-  - turns on drawing of courses of internal forces in nodes, averaged on element 2D
-  - turns on drawing of courses of internal forces in centres of finite elements
-  - add new Position to design group
-  – add new reinforced section into the list of available reinforced sections.
- **Group Positions on members** – contains definition of points, where design of reinforcement will be performed. Each Position has assigned name of reinforced section. Positions with identical reinforced section have assigned the same reinforcement pattern in concrete design. If the positions should have different reinforcement, it is necessary to assign them different names of reinforced cross-section.

For each position following properties can be changed in columns:

- **Name** – name of Position. Name of Position is used as a part of section name in module IDEA RCS.
- **Reinforced cross-section** – select name of reinforced section assigned to Position. Click edit button  to change name of reinforced section.
-  – delete Position from design group.

For selected Position (in list of existing positions) coordinates of Position are displayed and can be edited:

- **X [m] , Y [m]** – coordinates of current Position
- **Taken from** – choice of mode to specify coordinates of point for design of reinforcement. Following modes are available:
  - **Coordinates** – direct input of point coordinates defined by values in edit boxes X and Y. After clicking  can the point be determined by click on required position in the picture of internal forces.
  - **Mesh node** – position of point for reinforcement design is determined by input of mesh node number into edit box **Mesh node Id.** After input of node number coordinates of point in columns X and Y are recalculated (and can be edited)
  - **Mesh element** – position of point for reinforcement design is determined by input of mesh element number into edit box **Mesh element Id.** After input of element number coordinates of point in columns X and Y are recalculated (and can be edited)

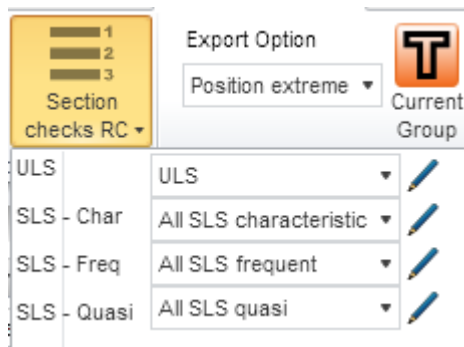
- **Associated area** – definition of area, on which extremes of internal forces are searched. Following options are available in list:
  - **None** – no associated area is taken into account.
  - **Rectangular** – dimensions of rectangular associated area can be defined in edit boxes **Width** and **Height**.
  - **Circular** – radius of circular associated area can be defined in input box **Radius**.

## 5.2 Result Classes

Click navigator command **Concrete design 2D > Result Classes** to set result classes needed for 2D members concrete check. Ribbon groups **Concrete check result classes** and **Concrete check** are available.





The contents of result classes needed for specific concrete checks in IDEA RCS are displayed in the main view.

## 5.3 Ribbon group Concrete check result classes



The check of concrete sections requires with 4 types of result classes – ULS, SLS characteristic, SLS Frequent and SLS quasi-permanent. 4 result classes (existing in current project) have to be set in appropriate combo-boxes before the concrete check is started. IDEA Designer prepares needed load effects for IDEA RCS according to current classes.

- **RC manager** – run **Result classes manager** to input or edit result classes - see **4.3.3 Result classes**.

- **ULS** –result class selected in this list is used to generate the content of **ULS – fundamental** combination for check of reinforced concrete section. To edit content of current result class click edit button .
- **SLS - Char** – result class selected in this list is used to generate the content of **SLS – Characteristic** combination for check of reinforced concrete section. Click edit button  to edit the content of current result class.
- **SLS – Freq** – result class selected in this list is used to generate the content of **SLS – Frequent** combination for check of reinforced concrete section. Click edit button  to edit the content of current result class.
- **SLS – Quasi** – result class selected in this list is used to generate the content of **ULS – Quasi-permanent** combination for check of reinforced concrete section. Click edit button  to edit the content of current result class.

## 5.4 Ribbon group Concrete design



Those commands can be used for the proceeding of cross-section checks in the program IDEA RCS.

- **Export options** – choose strategy for generation of data for module IDEA RCS.
- **Current group** – data needed for all check for the current group are prepared and the program IDEA RCS is launched.
- **All groups** – data needed for all check for all groups in the project are prepared and the program IDEA RCS is launched.

Program can prepare data for IDEA RCS using one of 3 different strategies. It depends on where and how the extreme sets of 6 internal forces are investigated.

IDEA Designer collects more members into one group. All members of one design group are than designed equally. It means that program works with one member, which is representative for the whole design group.

Positions for design of reinforcement are defined on the design group. Each position has assigned its type of reinforcement. Next advantage is that one type of reinforcement can be assigned to more positions (when the same reinforcement is required on the beginning and on the end of member).


Extreme values of internal forces are searched from all load cases (combinations) in appropriate design group. Minimal and maximal values are searched for each component of internal forces and values of corresponding components are stored with those extremes.

There are following strategies:

- **No extremes** – data for IDEA RCS are generated sequentially for all members in design group in positions specified. No extremes of internal forces are being searched for, sextuplets for each case/combination in appropriate result class is generated.
- **Position extreme** – Extremes are investigated sequentially in all design group positions on each member of group. Eq. for position 0.0 m are evaluated internal forces for each particular member in group in position 0.0 m and extreme values are stored. It means that extreme value in particular section is generated across collection of members.
- **Reinforced CSS extreme** – the same investigation as above, but extremes are collected according to reinforced sections.

## 5.5 Design

If navigator command **Concrete design 2D > Design** is active, all defined design groups are listed in data window. Design groups contain all data needed for checks in IDEA RCS.

Click edit button  to modify properties of design group – see **5.1 Definition of points for reinforcement design**.

## 6 1D concrete members design

The check of concrete members is done using the module IDEA RCS.

Reinforcement zones have to be defined on the representative design member of the design group to be able to design concrete 1D members. The reinforcement zones are reinforced with the reinforcement. Then the check of reinforced sections along the representative design member of the design group can be performed. Also the calculation of deflections of all design members in the design group can be performed.

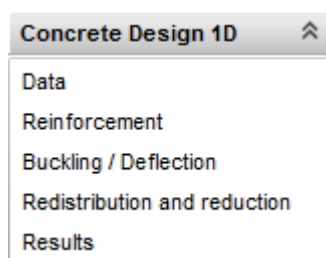
The 1D concrete members design is performed for the current design group. The current design group can be set in the list at top of the navigator.

The reinforcement zones with reinforcement are defined for the representative design member of the current design group. The bearing conditions can be adapted for calculation of deflections.

For the reinforced design members following calculations can be performed:

- Sections check along the representative design member (extremes of internal forces are searched on all design members in the current design group)
- Deflections calculation, which is performed for each design member in the current design group.

### 6.1 Check of the design group



Use commands in navigator group **Concrete design 1D** to define reinforcement zones, input reinforcement, input data for calculation of deflections, to define conditions for redistribution and reduction calculation, check of reinforced sections and calculation of deflections.

The input of reinforcement and data for check is done on the representative design member of the current design group. The current design group can be set in the list at the top of the navigator.

Program IDEA RCS is used to design the design members. 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 of design group several preconditions have to be fulfilled:

- The structure contains concrete elements
- Design members and design groups were defined
- ULS and SLS (characteristics, frequent, quasi-permanent) combinations were defined
- The calculation was performed – results of calculation can be evaluated.

If commands in navigator **Concrete design 1D** are active, the ribbon tab **Concrete design 1D** is available. Content of this ribbon tab depends on the current command in the navigator.

## 6.2 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 **Settings**:



**Code** – code and calculation settings – see 6.2.1 Code and calculation settings.

- **Deflection settings** – settings of parameters of loads, which are used for calculation of deflections – see 6.2.2

**Settings of result class for calculation of deflections.**

- **Section checks RC** – settings of result classes, which are used for section checks – see 6.2.3 Setting of result classes for section check.

### 6.2.1 Code and calculation settings

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

Chapter	Setting	Value
Chapter 2	2.4.2.4(1) $\gamma_c$ - Persistent, transient	1,5
	2.4.2.4(1) $\gamma_c$ - Accidental	1,2
	2.4.2.4(1) $\gamma_s$ - Persistent, transient	1,15
	2.4.2.4(1) $\gamma_s$ - Accidental	1
	2.4.2.4(1) $\gamma_{sp}$	1,15
Chapter 5	5.5 k1	0,44
	5.5 k2	$1,25 * (0,6 + 0,0014 / \epsilon_{ct})$
	5.5 k3	0,54
	5.5 k4	$1,25 * (0,6 + 0,0014 / \epsilon_{ct})$
	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

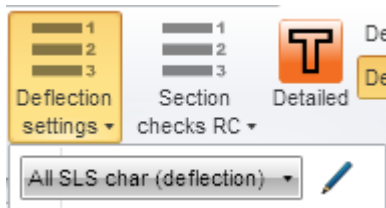
Only the important values can be set in this setup dialog. The remaining values can be set in the detailed check in IDEA StatiCa RCS.

Expand all Collapse all OK Cancel




### 6.2.2 Settings of result class for calculation of deflections

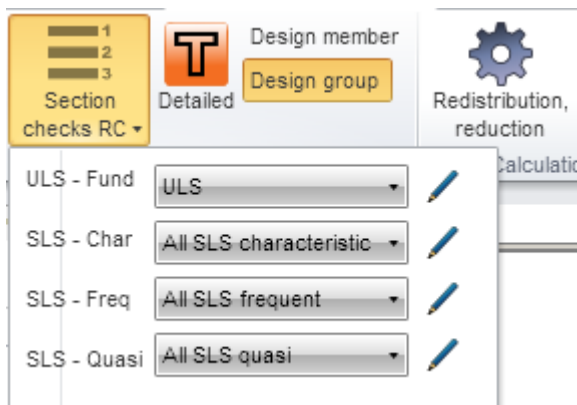
The deflections are calculated for all combinations assigned to the result class which is set to be used to calculation of deflections. The default result class for calculation of deflections is named **All SLS (Deflections)**. Combinations in this result class are considered as characteristic combinations. 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  $\psi_2$  is used, which is taken from the load group, in which the load cases are assigned.









Click **Deflection settings** in ribbon group **Concrete design** 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 result class for calculation of deflection – see 4.3.3 Result classes.

### 6.2.3 Setting of result classes for section checks



Result classes for sections check of current design member have to be set before performing sections check. According to selected result classes appropriate combinations in IDEA RCS are generated. Click **Section checks RC** in ribbon group **Settings** to set result classes for section checks. Lists of result classes appear which can be assigned to particular combination types.

- **ULS – Fund** - result class selected in this list is used to generate the content of **ULS – fundamental** combination for check of reinforced concrete section. To edit content of current result class click edit button .
- **ULS – Accid** –result class selected in this list is used to generate the content of **Accidental** combination for check of reinforced concrete section. To edit content of current result class click edit button .
- **ULS – Fat** - result class selected in this list is used to generate the content of **Min. cyclic loads** and **Max. cyclic loads** combinations for check of reinforced concrete section. To edit content of current result class click edit button .
- **SLS - Char** – result class selected in this list is used to generate the content of **SLS – Characteristic** combination for check of reinforced concrete section. Click edit button  to edit the content of current result class.
- **SLS – Freq** – result class selected in this list is used to generate the content of **SLS – Frequent** combination for check of reinforced concrete section. Click edit button  to edit the content of current result class.
- **SLS – Quasi** – result class selected in this list is used to generate the content of **ULS – Quasi-permanent** combination for check of reinforced concrete section. Click edit button  to edit the content of current result class.



## 6.3 Design member data

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

<b>Ultimate limit state</b>		<b>Exposure class</b>	
Capacity N-M-M	<input checked="" type="checkbox"/>	No corrosion (X0)	<input type="checkbox"/>
Shear	<input checked="" type="checkbox"/>	Carbonation	XC3 - Moderate humidity
Torsion	<input checked="" type="checkbox"/>	Chlorides	XD1 - Moderate humidity
Interaction	<input checked="" type="checkbox"/>	Chlorides from sea	No risk of chlorides from sea
Fatigue	<input checked="" type="checkbox"/>	Freeze/Thaw Attack	No risk of freeze/thaw attack
<b>Serviceability limit state</b>		<b>Chemical Attack</b>	
Stress limitation	<input checked="" type="checkbox"/>	No risk of chemical attack	
Crack width	<input checked="" type="checkbox"/>		
<b>Detailing</b>		<b>Relative humidity [%]</b>	
Detailing	<input checked="" type="checkbox"/>	65	
<b>Deflections</b>		<b>Creep coefficient</b>	
Deflections	Detailed calculation	Calculated	
<b>Reduction and redistribution</b>		<b>Member type</b>	
Redistribution of moments	<input type="checkbox"/>	Beam	
Reduction of moments	<input type="checkbox"/>	<b>Structural member importance</b>	
Reduction of shear force	<input type="checkbox"/>	Major	
Limited interaction check	<input type="checkbox"/>		

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  $\Phi_{inf}$  can be defined by user.
- **Structural member importance** – select type of structural member importance according to 6.2.1(4).

## 6.4 Reinforcement zones

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

After the zones and reinforcement is 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 **6.4.2 Zone templates**.

Ribbon groups **Concrete design**, **Zone templates**, **View setting and scale**, **Internal forces** and **Detailed view** are available during the input of reinforcement zones.

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

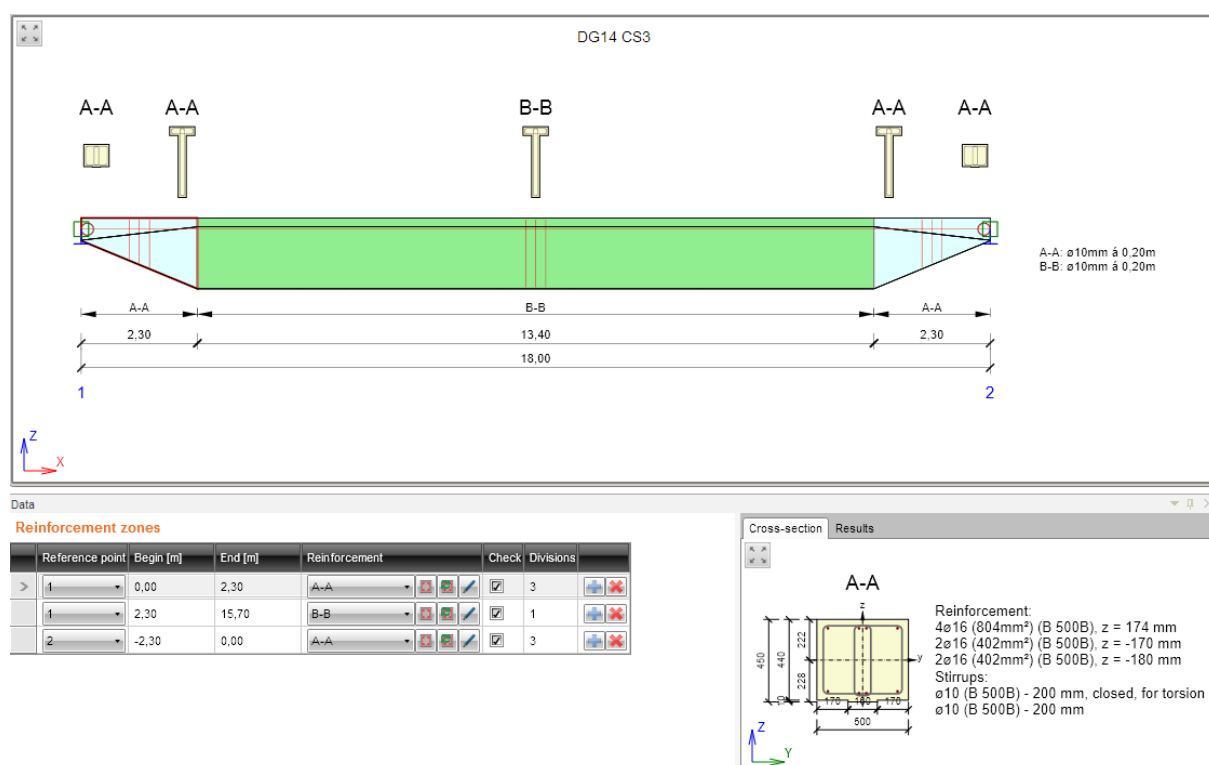





Table **Reinforcement zones** contains following columns:

- **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 **6.5 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

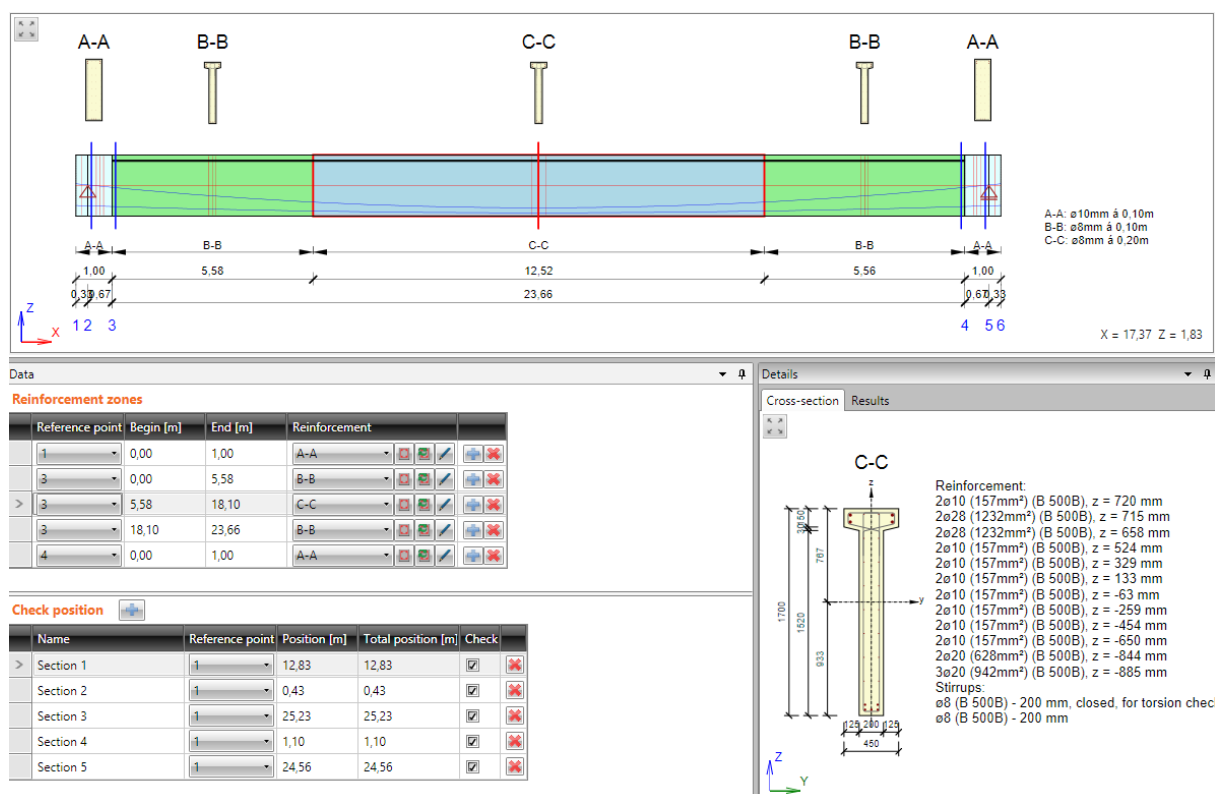
**Reinforcement editor** is launched to define reinforcement in the newly created reinforcement template.

-  - displays dialog to modify the name of reinforcement template.
- **Check** – option, if the zone is checked or not. 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. The 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. Then the reinforcement is interpolated from the governing cross-section to the rest of haunch cross-sections.

#### 6.4.1 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"/>

**Cross-section C-C**

Reinforcement:


- 2ø10 (157mm<sup>2</sup>) (B 500B), z = 720 mm
- 2ø20 (1232mm<sup>2</sup>) (B 500B), z = 715 mm
- 2ø20 (1232mm<sup>2</sup>) (B 500B), z = 658 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = 524 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = 329 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = 133 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = -63 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = -259 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = -454 mm
- 2ø10 (157mm<sup>2</sup>) (B 500B), z = -650 mm
- 2ø20 (628mm<sup>2</sup>) (B 500B), z = -844 mm
- 3ø20 (942mm<sup>2</sup>) (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.

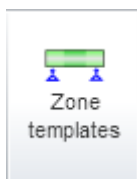
Check positions can be defined in the **Check positions** table.

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.

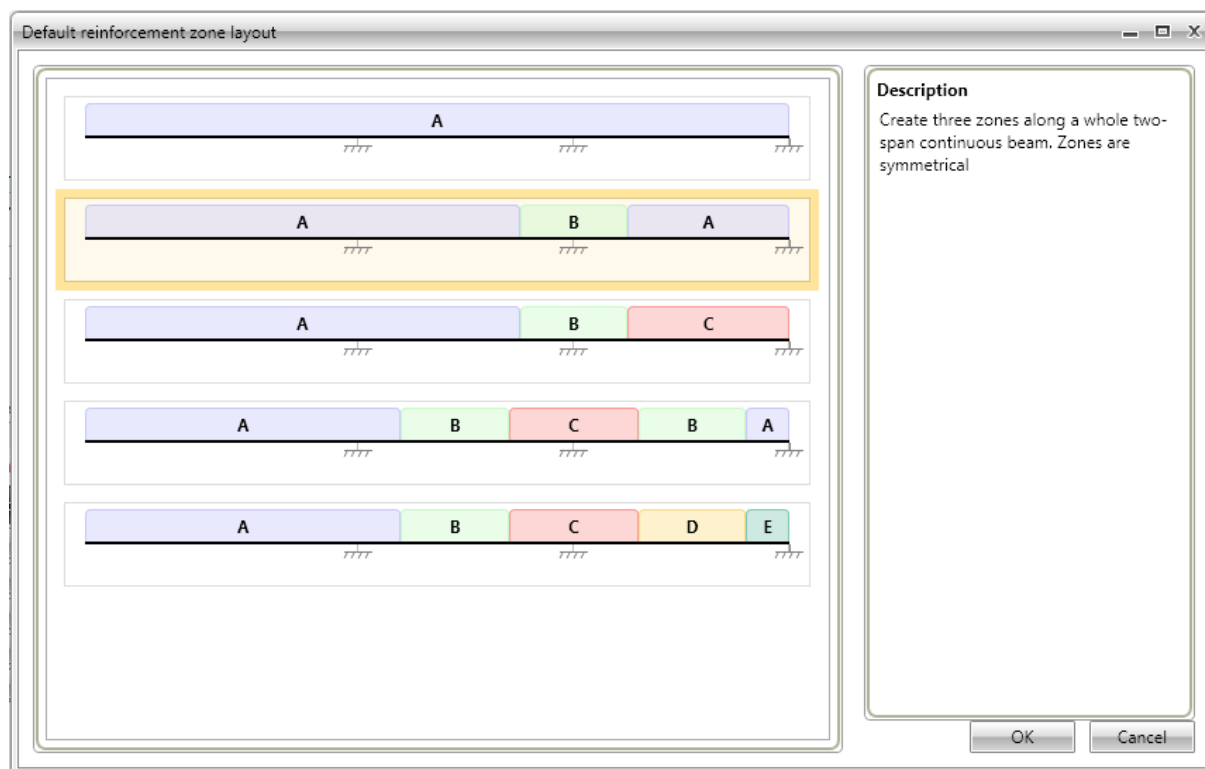
### 6.4.2 Zone templates



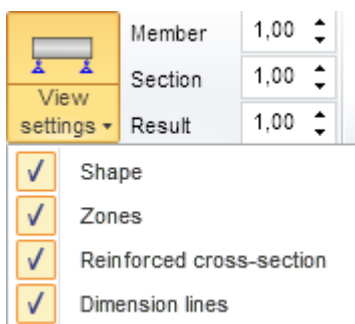
Zone templates can be used to generate zones along the design member. To generate zones according to templates click **Zone templates**.

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.



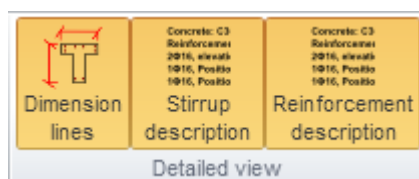
### 6.4.3 Ribbon group View settings and scale



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...)

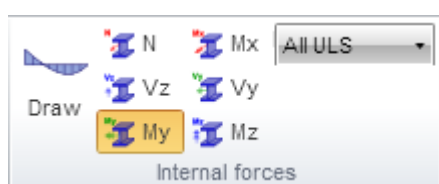
### 6.4.4 Ribbon group Detailed view



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

### 6.4.5 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
- **Vy**,
- **Vz** – switch to drawing of shear force Vz. or Vy
- **Mz**,
- **My** – switch to drawing of bending moment My or Mz.
- **Mx** – switch to drawing of torsional moment Mx.
- **Result class** – select the current result class, for which the internal forces courses are drawn.

## 6.5 Editor of reinforcement

Input of current reinforcement template can be started

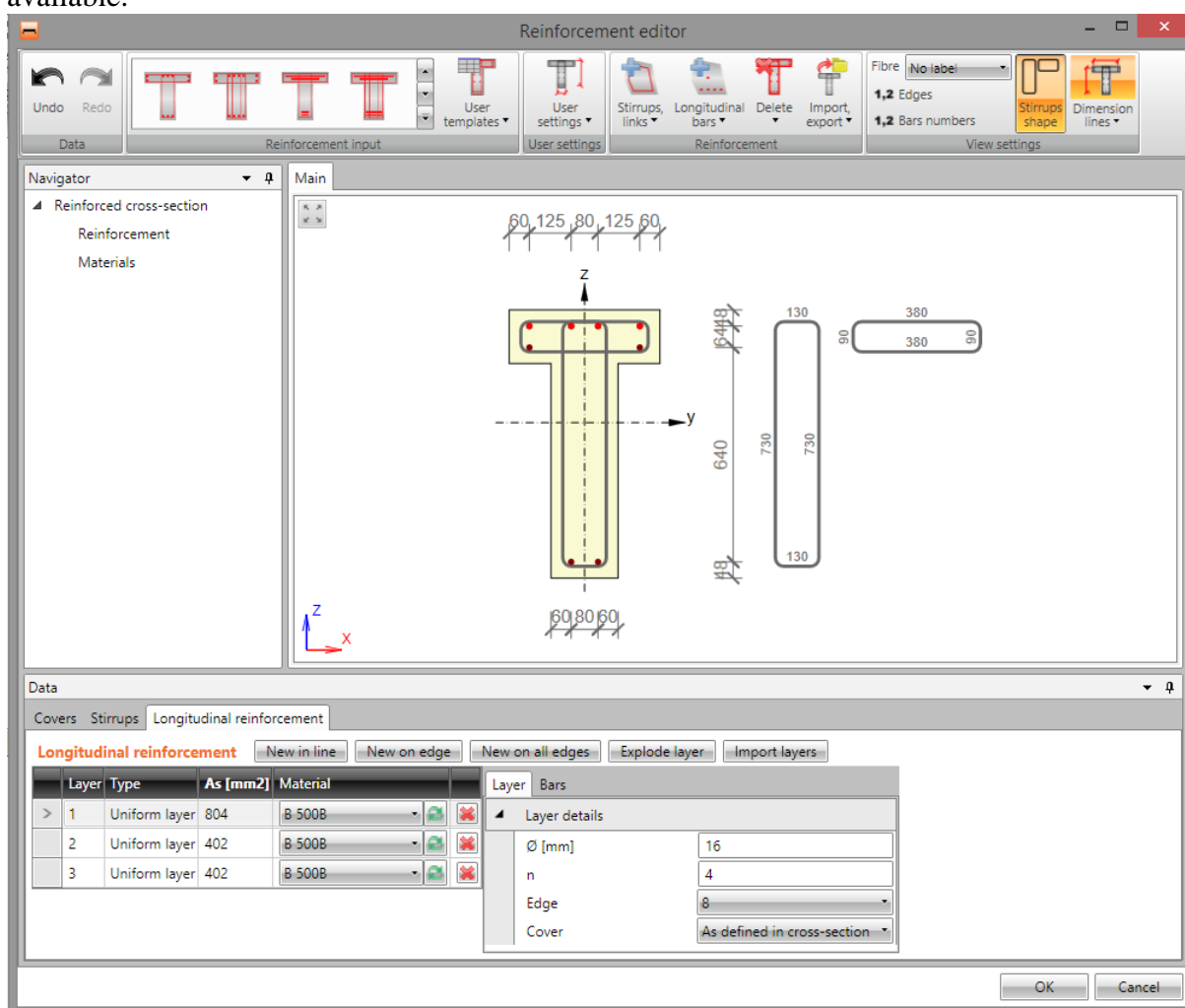
- Clicking the edit button  in the **Reinforcement** column in the zones table in the Data window
- Clicking the picture of section above the zone in the Main window.

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.





### 6.5.1 Editing cover

To edit concrete cover at particular cross-section edges click **Cover** in ribbon group **Reinforcement**.

Values of cover related to particular cross-section edges can be changed in table.

To switch drawing of existing reinforcement on/off select the option **Draw reinforcement**.

**Covers** ☐ All edges

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

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

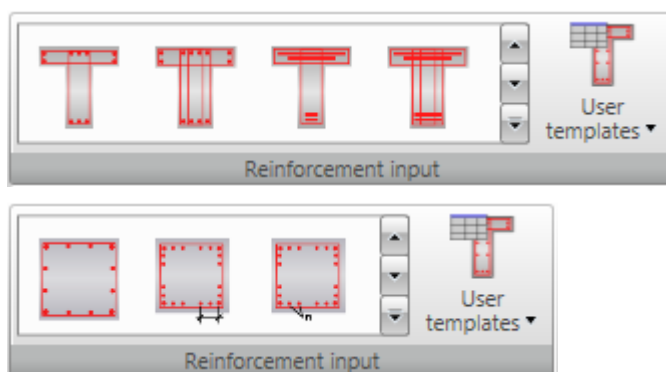
Cover can be defined:

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

**Covers** ☒ All edges

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

### 6.5.2 Input of reinforcement by template

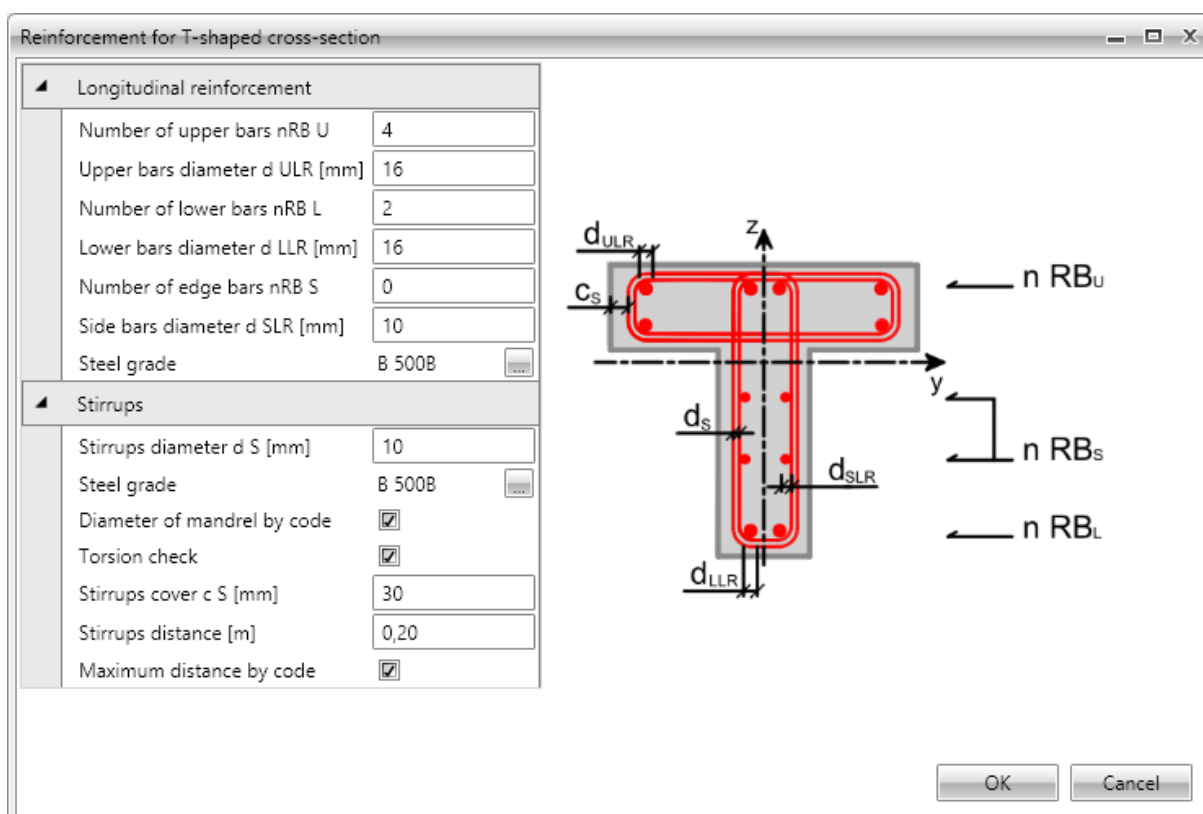


Reinforcement templates are available for some pre-defined sectional shapes. Reinforcement templates available for the current cross-section are displayed in ribbon group **Reinforcement**.

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 **6.5.10 User defined reinforcement templates**.

Reinforcement template parameters for T-shaped cross-section (including the reinforcement design):



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.

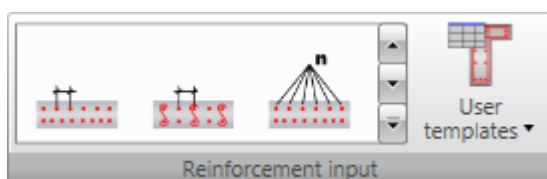
The reinforcement layer is defined by string, which describes diameters of individual bars in the layer. Individual diameters are separated by space, characters 'x' or '\*' can be used to define multiple diameters, e. g. ,20 16 16 20' or ,20 2\*16 20'.

Reinforcement for T-shaped cross-section

Main longitudinal reinforcement	
Diameters in upper layer l u1 [mr]	2x16
Diameters in upper layer l u2 [mr]	
Diameters in lower layer l l1 [mm]	3x20
Diameters in lower layer l l2 [mm]	
Steel grade	B 500B
Longitudinal reinforcement for detailing	
Add detailing reinforcement	<input checked="" type="checkbox"/>
Detailing bars diameter d DR [mr]	16
Longitudinal reinforcement on side of cross-section	
Number of edge bars nRB S	0
Side bars diameter d SLR [mm]	10
Stirrups	
Stirrups diameter d S [mm]	10
Steel grade	B 500B
Diameter of mandrel by code	<input checked="" type="checkbox"/>
Torsion check	<input checked="" type="checkbox"/>
Stirrups cover c S [mm]	30
Stirrups distance [m]	0,20
Maximum distance by code	<input checked="" type="checkbox"/>

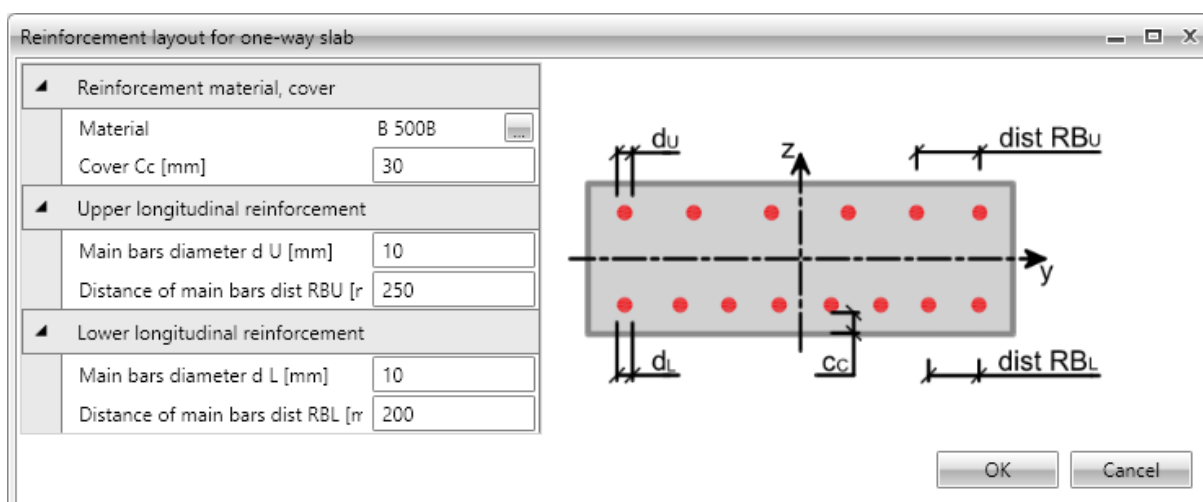
OK Cancel

### 6.5.3 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 **6.5.10 User defined reinforcement templates**.

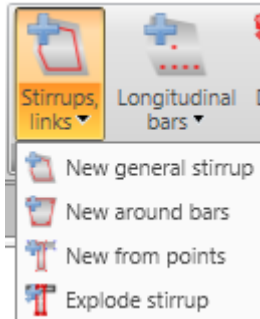


For one way slabs the distance between bars or bars number, bars diameter, bars material and concrete cover are defined in the template dialog.

### 6.5.4 Shear reinforcement

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

#### 6.5.4.1 Stirrups



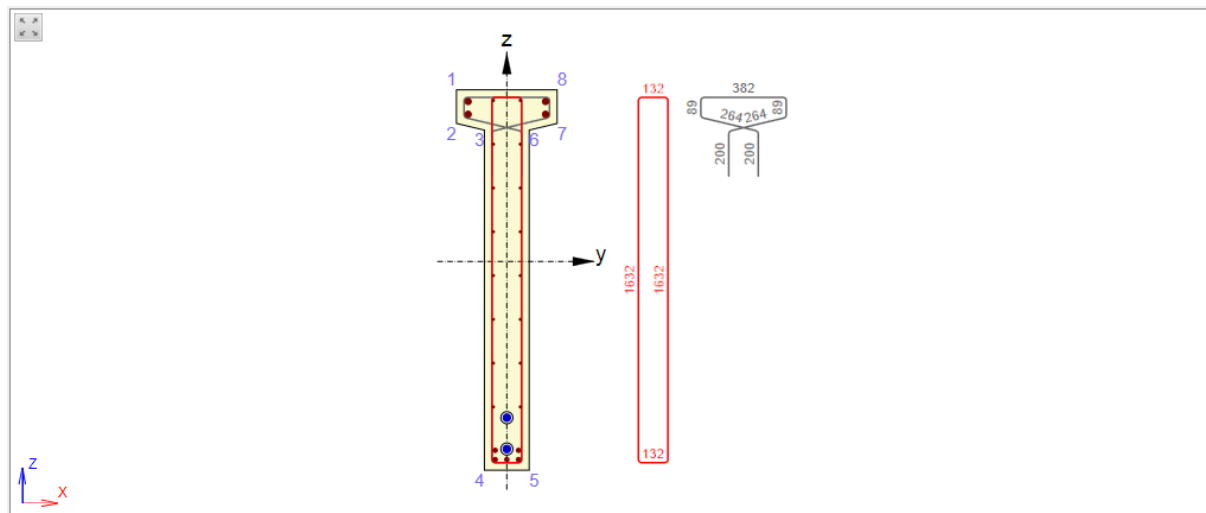
Drop-down button **Stirrups, links** in ribbon group **Reinforcement** collects commands for stirrups operations:

- **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

### 6.5.4.1.1. 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.

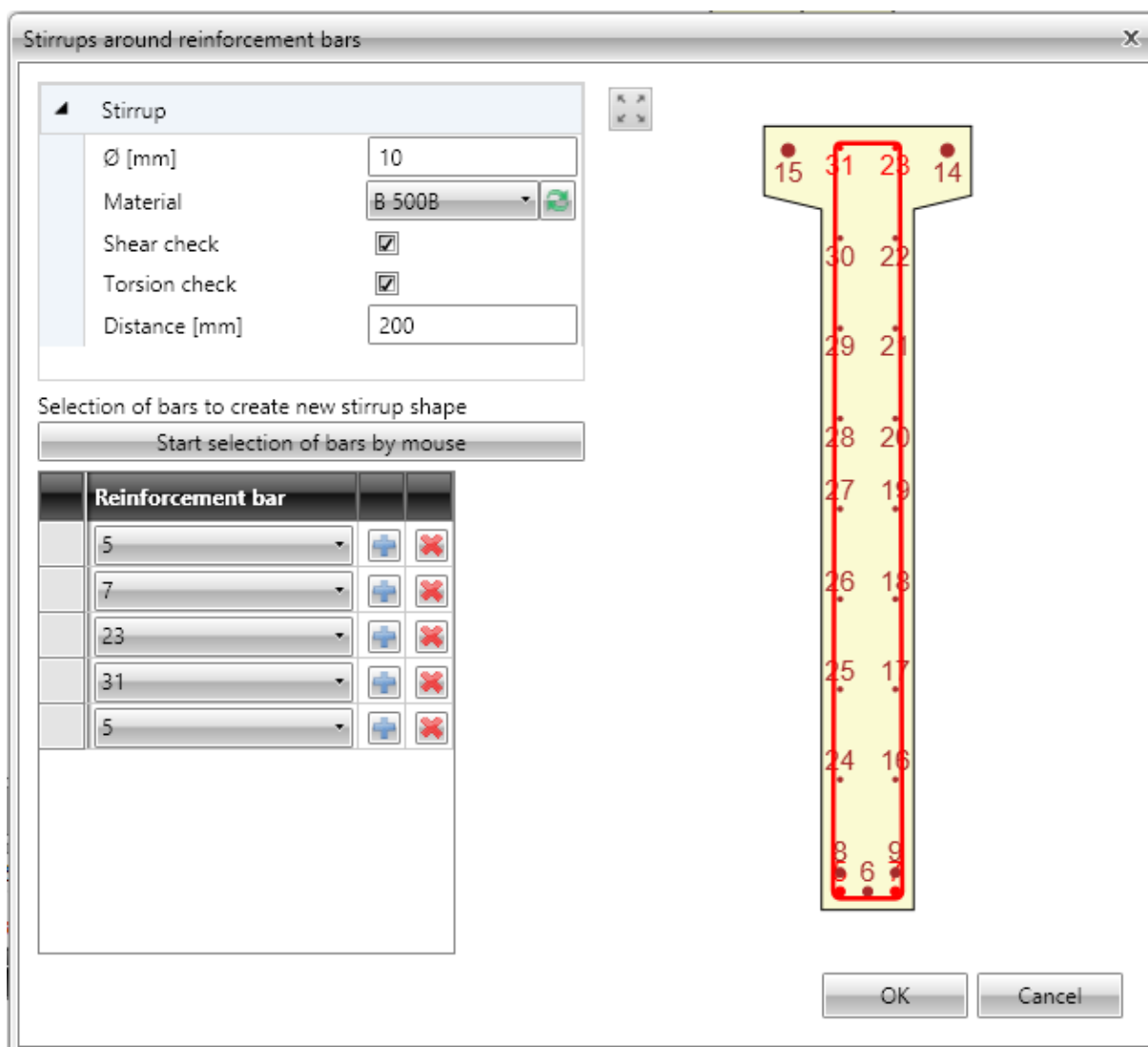
### 6.5.4.1.2. Stirrups around bars of longitudinal reinforcement

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.



### 6.5.4.1.3. Stirrup by cross-section vertices

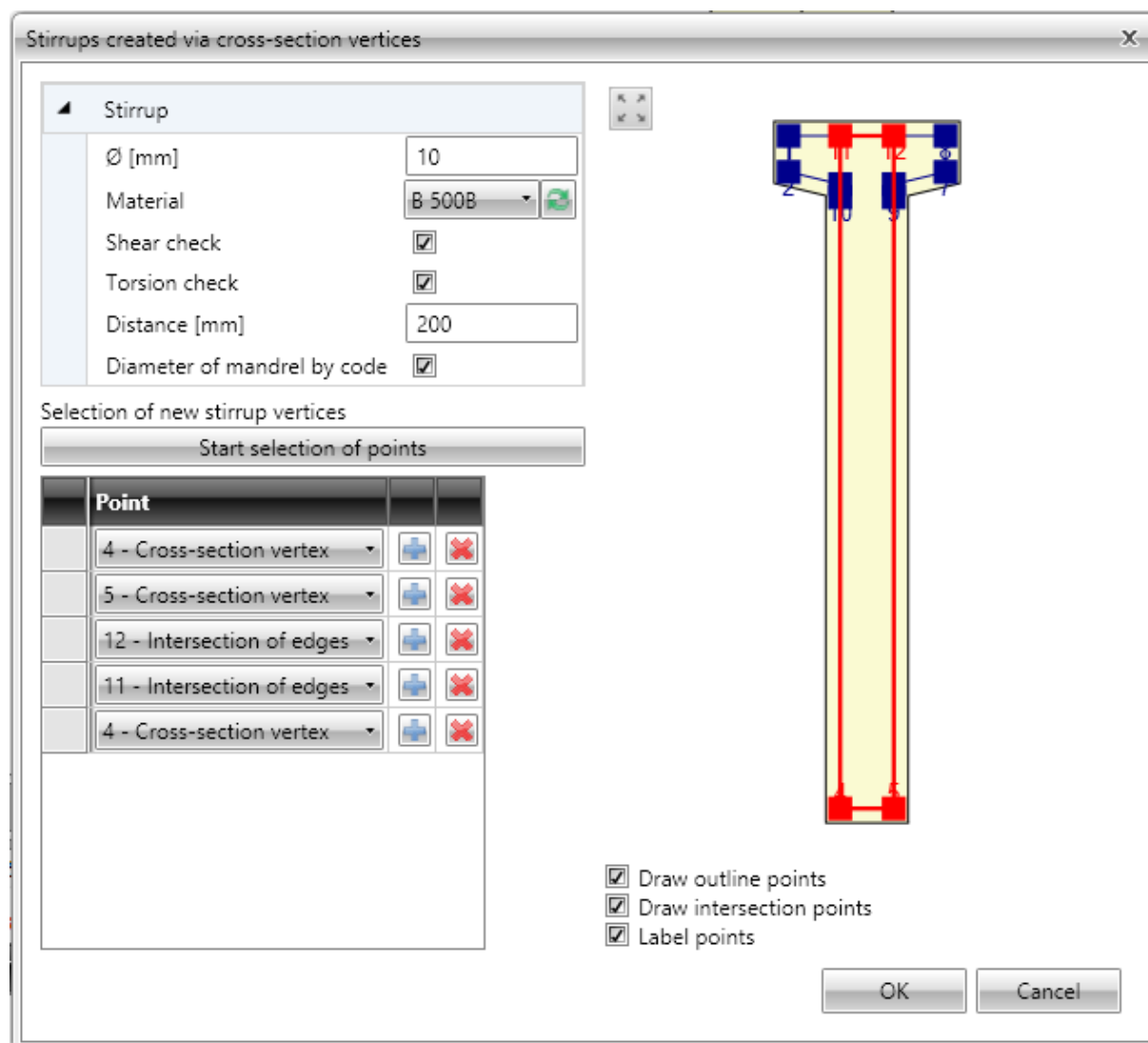
To input stirrup around main reinforcement click **Stirrups, links > New from points** in ribbon group **Reinforcement** or click **New from points** above the table **Stirrups**.

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.

#### 6.5.4.1.4. Exploding stirrups


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

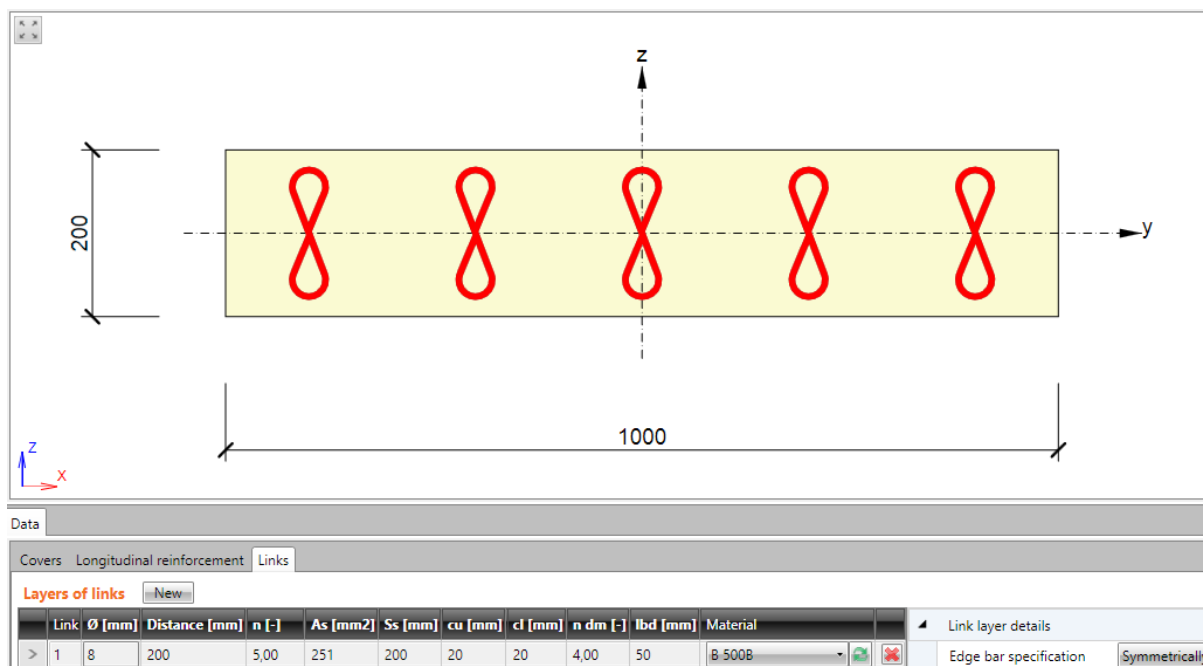
### 6.5.4.2 Links

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.

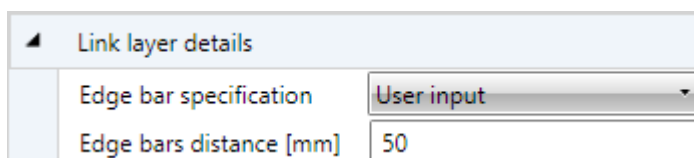
Columns in **Link layers** table:

- **Ø** – 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.



#### 6.5.4.2.1. 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**.



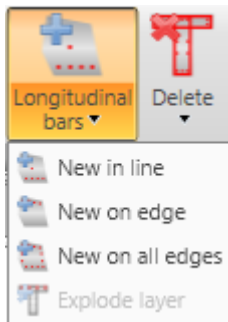
Properties group **Link layer details**:

- **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.

### 6.5.5 Longitudinal reinforcement

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



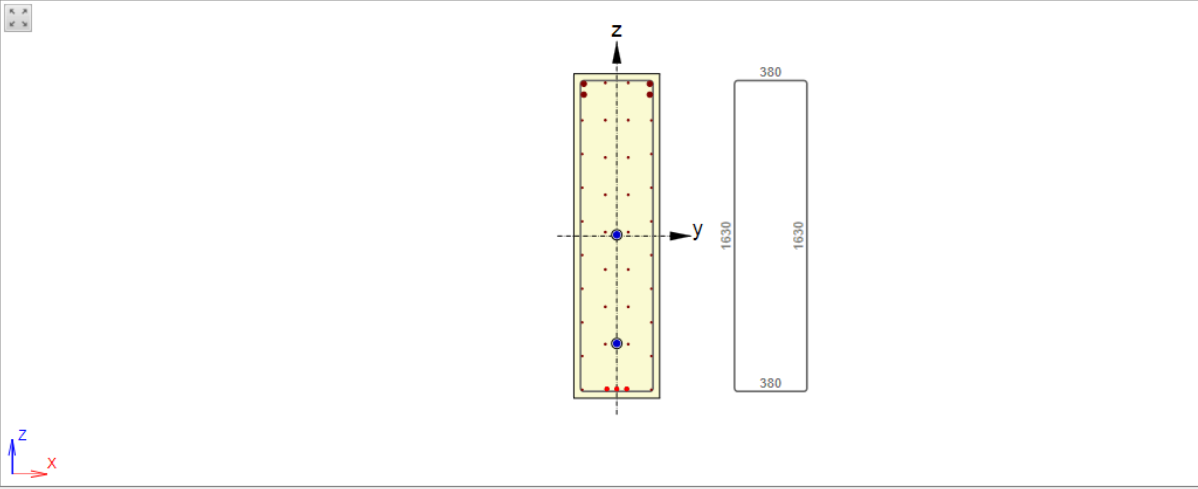
- **New in line** - adds a new layer of longitudinal reinforcement defined by coordinates of edge bars.
- **New on edge** - adds a new layer of longitudinal reinforcement related to cross-section edge.
- **New on all edges** - adds new layers of longitudinal reinforcement on all edges of cross-section.
- **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.



Data

Covers   Stirrups   Longitudinal reinforcement

**Longitudinal reinforcement**   New in line   New on edge   New on all edges   Explode layer

Layer	Type	As [mm <sup>2</sup> ]	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		

Layer   Bars

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 **Tendons** 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.

### 6.5.5.1 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
<b>Layer details</b>	
Ø [mm]	16
n	2
<b>First point</b>	
Origin	vertex 1
Δ Y [mm]	50
Δ Z [mm]	50
Y [mm]	-175
Z [mm]	-800
<b>Last point</b>	
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**:

- **Ø** – 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

#### 6.5.5.1.1. Layer details

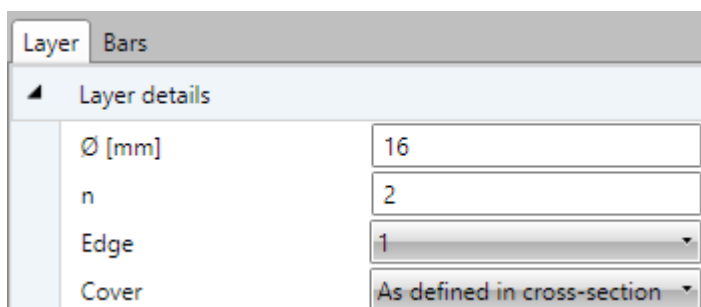
Layer		Bars						
	Bar	$\varnothing$ [mm]	Y [mm]	Z [mm]	Bent-up	$s_b$ [m]	$\alpha_{xz}$ [°]	$\alpha_{yz}$ [°]
>	5	16	-175	-800	<input type="checkbox"/>			
	6	16	175	-800	<input type="checkbox"/>			

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

- **Bar** – index of bar is displayed.
- **$\varnothing$**  – 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.
- **$s_b$**  – input distance between individual bent-ups.
- **$\alpha_{xz}$**  – input angle of bent-up bar in XZ plane of cross-section (to longitudinal axis of member).
- **$\alpha_{yz}$**  – input angle of bent-up bar in YZ plane of cross-section (to longitudinal axis of member).

### 6.5.5.2 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	
Layer details	
Ø [mm]	16
n	2
Edge	1
Cover	As defined in cross-section

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

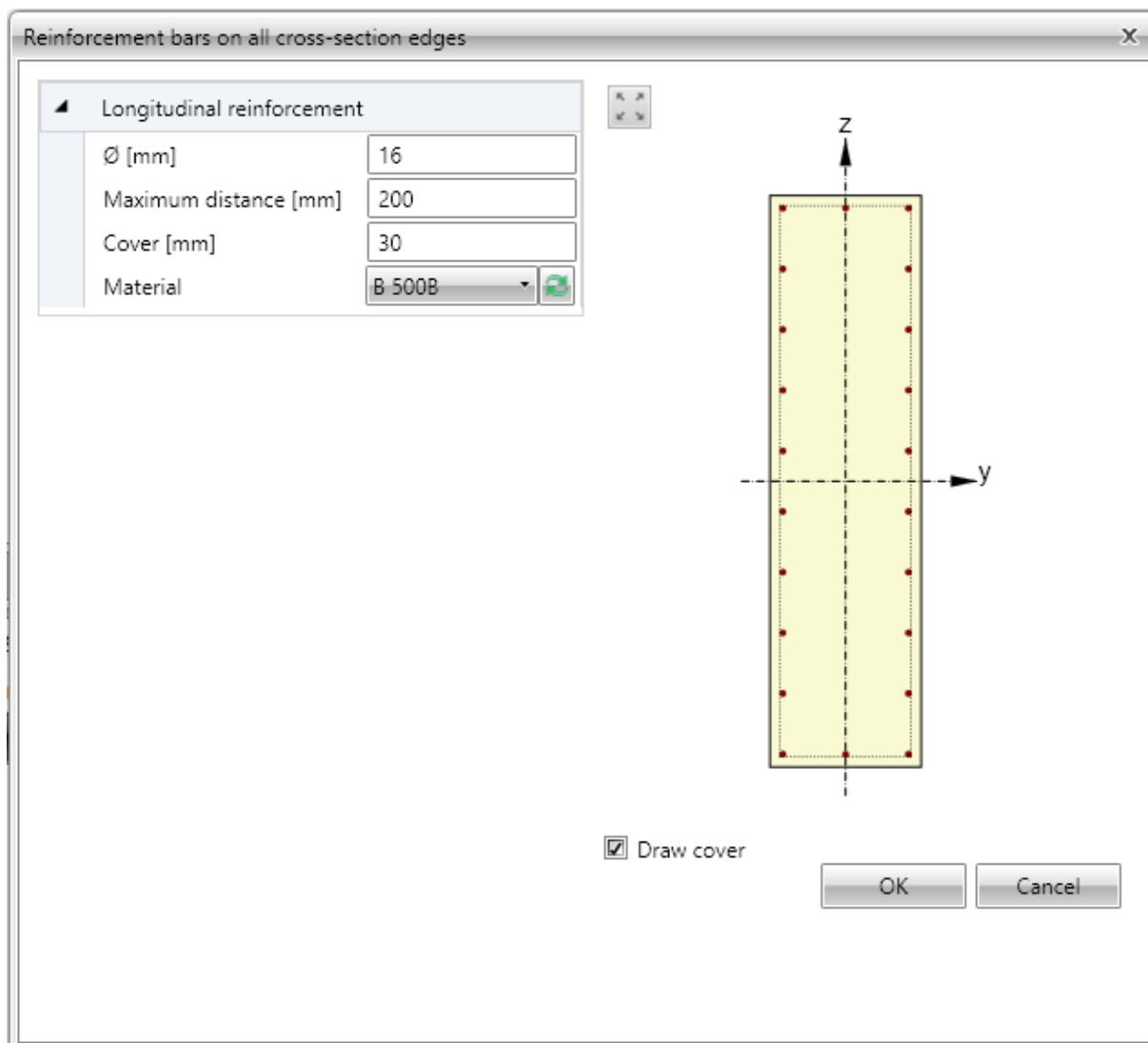
Properties group **Layer details**:

- **Ø** – 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 **6.5.5.1.1 Layer details**.

### 6.5.5.3 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 **6.5.5.1.1 Layer details**.

#### 6.5.5.4 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
<div> <div>Layer details</div> <div> <div>Ø [mm]</div> <div>10</div> </div> <div> <div>n</div> <div>4</div> </div> <div> <div>Number of bars in wave</div> <div>1</div> </div> <div> <div>Concrete cover [mm]</div> <div>10</div> </div> </div>	

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 6.5.5.1.1 **Layer details**.

#### 6.5.5.5 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
<div> <div>Layer details</div> <div> <div>Ø [mm]</div> <div>10</div> </div> <div> <div>Distance [mm]</div> <div>200</div> </div> <div> <div>n [-]</div> <div>5,00</div> </div> <div> <div>Edge bar specification</div> <div>User input</div> </div> <div> <div>Edge bars distance [mm]</div> <div>100</div> </div> <div> <div>Surface to cover</div> <div>Lower</div> </div> <div> <div>Cover [mm]</div> <div>20</div> </div> </div>	

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

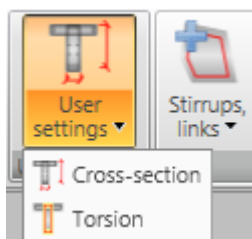
Properties group **Layer details**:

- **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 **6.5.5.1.1 Layer details**.

### 6.5.6 User settings of reinforced cross-section



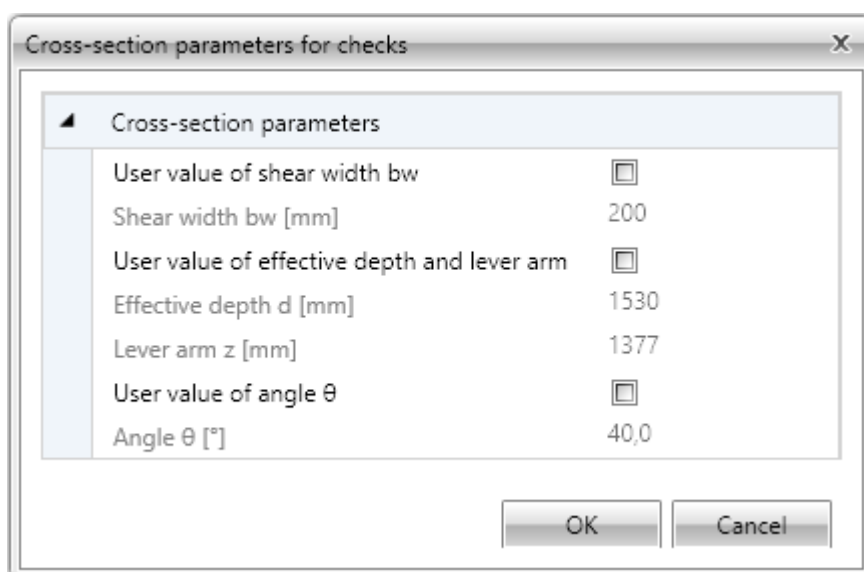
Ribbon group **User setting** contains commands to modify some shear and torsion calculation parameters of reinforced cross-section:

- **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.

#### 6.5.6.1 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.

### 6.5.6.2 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.

Equivalent thin-walled section for torsion check

☐ Create from real stirrups  
☐ Calculate from area and perimeter  
☒ Manual input

▲ Equivalent thin-walled section for torsion check  
 Ak [mm<sup>2</sup>] 198619  
 uk [mm] 3888  
 teff [mm] 90

▲ Stirrup for torsion  
 Ø [mm] 8  
 Material B 500B  
 Distance [mm] 200

Equivalent thin-walled section for torsion check has been created.

OK Cancel

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.

### 6.5.7 Deleting reinforcement

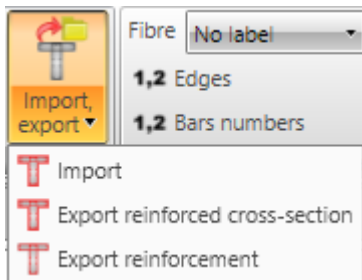


Use commands in ribbon group **Delete** to delete reinforcement from cross-section:

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

### 6.5.8 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.

### 6.5.9 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.



### 6.5.10 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:

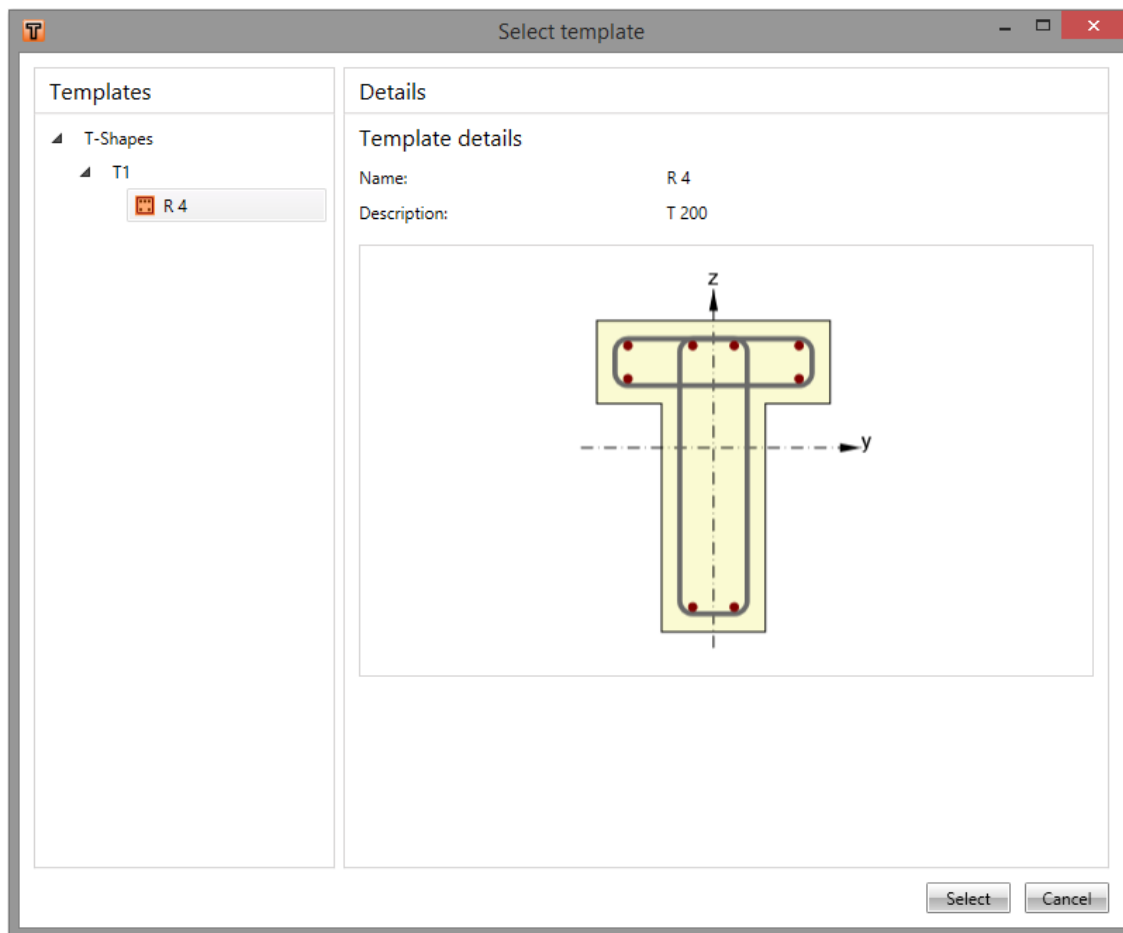
-  – starts input of reinforcement using the user defined reinforcement template – see **6.5.10.1 Reinforcing by user defined reinforcement template**.
-  – stores the current reinforcement into the database of user reinforcement templates. Dialog **Add template** appears. The target folder must be selected 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 **6.5.10.2 Templates manager**.

#### 6.5.10.1 Reinforcing by user defined reinforcement template

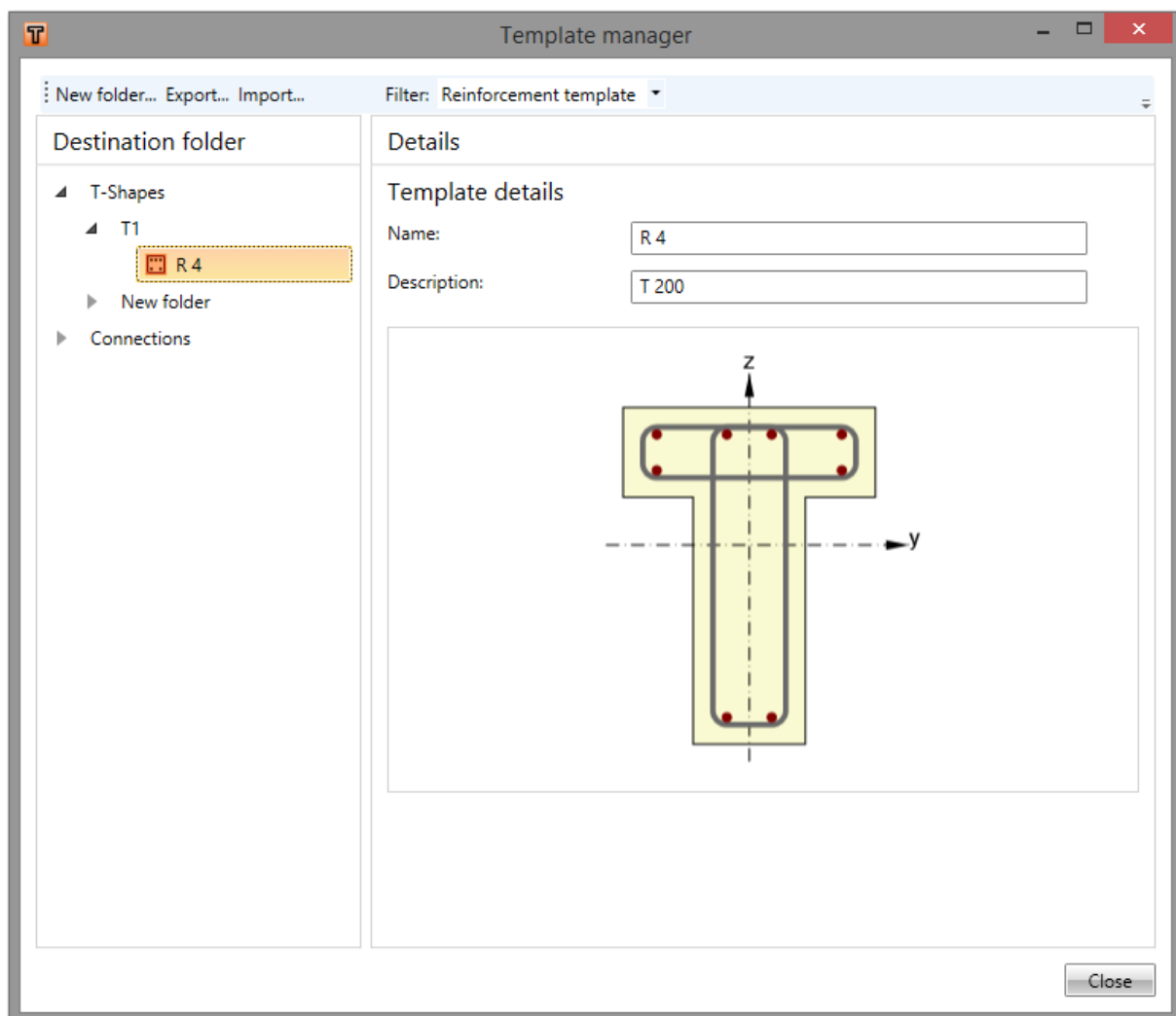
Dialog **Select template** appears after start of reinforcing by user defined reinforcement template.

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.



### 6.5.10.2 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.

## 6.6 Data for buckling effects calculation and deflection check

Click navigator command **Concrete design 1D > Buckling/Deflections** to input data for calculation of buckling effects and deflection check. According to the type of the design group either data for deflection check of beams or data for buckling effects calculation and deflection check of single span columns or data for buckling effects calculation and deflection check of multi-spans columns are defined in IDEA Designer.

### 6.6.1 Data for deflections check of beam

For the design group, which contains beams, the table with representative design member spans and a simplified picture of representative design member with supports is drawn in the Data window.



- **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.

The bearing schema is taken from the structural model.

Checkboxes in the column **Supported** can be used to switch particular supports on/off to obtain calculation model for deflection calculation, which differs from the calculation model of the whole structure. Switching the support in the design member on/off causes the change of the number of spans for deflection calculations and thus the values of limit deflection for particular spans change.

- **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**.

#### *6.6.1.1 Deflection check settings for the current span*

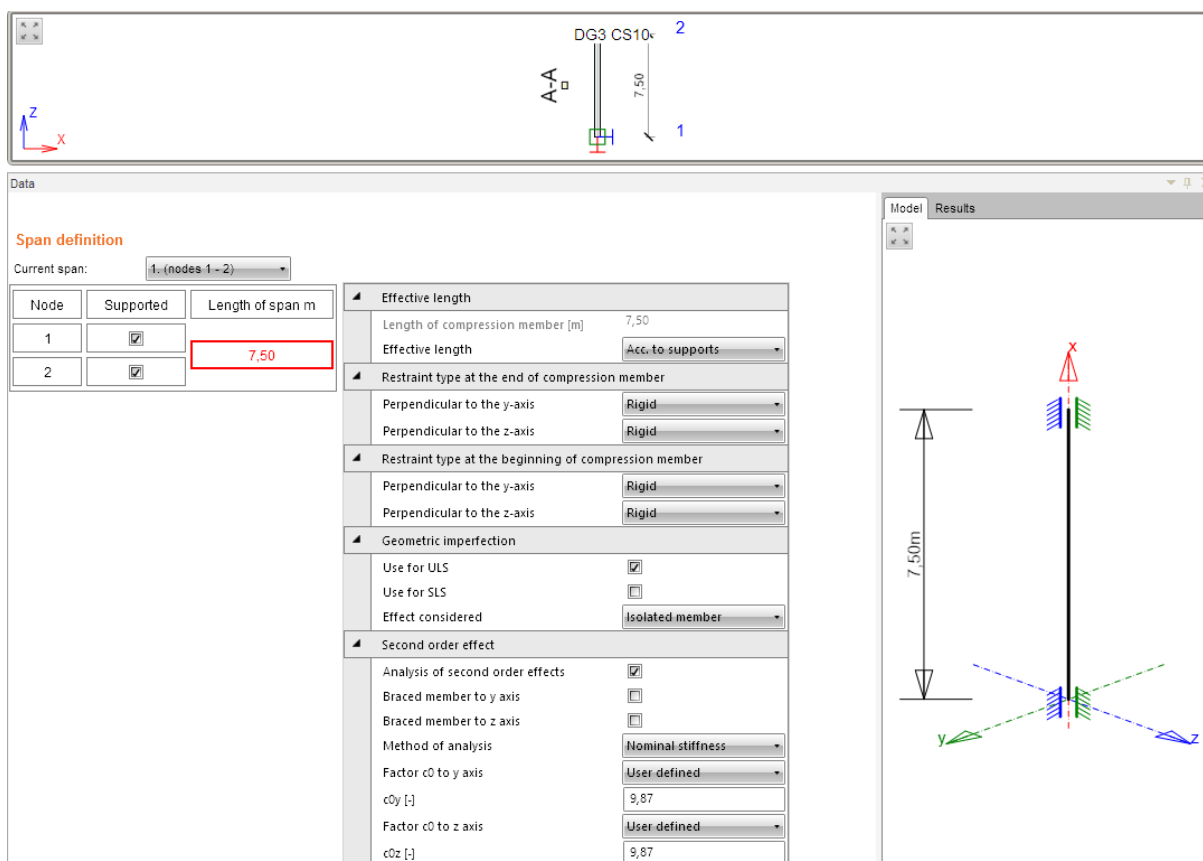
For the current span and current deflection direction (plane) following parameters can be set in **Deflection limits** properties group:

- **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).

### 6.6.2 Data for buckling effects calculation and deflection check for one span columns

Following tabs are displayed in the Data window for the design group, which contains one-span columns:

- tab **Buckling** to define parameters to determine the effective lengths and parameters of geometric imperfections and second order effects.
- tab **Deflection** to define parameters of limit deflection checks.



- **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**.

#### 6.6.2.1 Buckling effects calculation data

Data to determine effective lengths, geometrical imperfections and second order effects are defined on **Buckling** tab.

Properties group **Effective length**:

- **Length of compression member** – member length is displayed.
- **Effective length** – select mode to determine the effective length:
  - **User input** – effective lengths are defined by user.
    - **Effective length y** – input the effective length for buckling perpendicular to y-axis.
    - **Effective length z** – input the effective length for buckling perpendicular to z-axis.
  - **Acc. to supports** – effective lengths are determined automatically according to column supports.

Properties group **Restraint type at the end of compression member**:

- **Perpendicular to y-axis** – select restraint type at the end of compression member against buckling perpendicular to y-axis.
  - **Perpendicular to z-axis** – select restraint type at the end of compression member against buckling perpendicular to z-axis.
- Properties group **Restraint type at the beginning of compression member:**

- **Perpendicular to y-axis** – select restraint type at the beginning of compression member against buckling perpendicular to y-axis.
- **Perpendicular to z-axis** – select restraint type at the beginning of compression member against buckling perpendicular to z-axis.

Properties group **Geometric imperfections:**

- **Use for ULS** – switch on/off taking into account the geometric imperfections for ULS checks.
- **Use for SLS** – switch on/off taking into account the geometric imperfections for SLS checks.
- **Effect considered** – select type of effect considered for determination of imperfections according to 5.2 (6):
  - **Isolated member**
  - **Bracing system:**
    - **Total height of building** – input building total height above the level of moment restraint.
    - **Nr. of vertical members - my** – number of vertical members contributing to the horizontal force in ,y'-direction.
    - **Nr. of vertical members - mz** – number of vertical members contributing to the horizontal force in ,z'-direction.

Properties group **Second order effects:**

- **Analysis of second order effects** – switch on/off taking into account the second order effects when checking compressed members.
- **Braced member to y-axis** – switch on/off considering the existence of bracing system acc. to 5.8.3.1 (1) in direction perpendicular to y-axis.
- **Braced member to z-axis** – switch on/off considering the existence of bracing system acc. to 5.8.3.1 (1) in direction perpendicular to z-axis.
- **Method of analysis** – select method to determine second order effects according to 5.8.5:
  - **Nominal stiffness** – second order effects are determined using method based on nominal stiffness.
  - **Nominal curvature** – second order effects are determined using method based on nominal curvature.
- **Factor c0 to axis** – select method to determine the coefficient which depends on the distribution of first order moment to the corresponding axis 5.8.7.3(2):
  - **User defined** – value of coefficient is defined by user.
  - **Constant first order moment** – coefficient value is 8.
  - **Parabolic first order moment** - coefficient value is 9,6.
  - **Triangular first order moment** - coefficient value is 12.
- **Factor c to axis** – select method to determine the coefficient which depends on the distribution of curvature to the corresponding axis 5.8.8.2 (4):
  - **User defined** – value of coefficient is defined by user.
  - **Constant curvature distribution** – coefficient value is 8.



- **Sinusoidal curvature distribution** - coefficient value is 9,6.

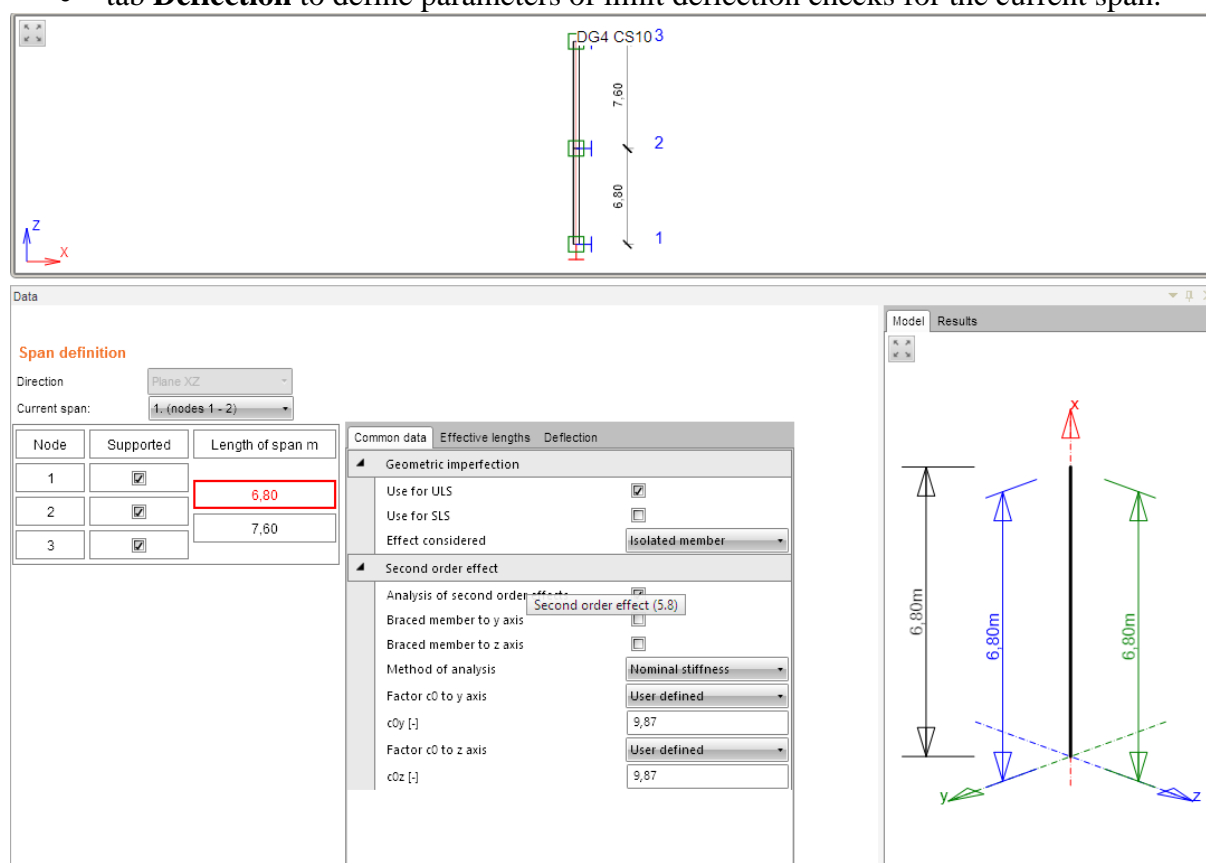
#### ***6.6.2.2 Deflection check settings***

The deflection check settings for the current span can be defined on **Deflection** tab - see **6.6.1.1 Deflection check settings for the current span**.

### 6.6.3 Data for buckling effects calculation and deflection check for multi-span columns

Following tabs are displayed in the Data window for the design group, which contains multi-span columns:

- tab **Common data** to define parameters to determine buckling effects common for all column spans.
- tab **Effective lengths** to define effective lengths for the current span.
- tab **Deflection** to define parameters of limit deflection checks for the current span.



The bearing schema is taken from the structural model.

Checkboxes in the column **Supported** can be used to switch particular supports on/off to obtain calculation model for deflection calculation, which differs from the calculation model of the whole structure. Switching the support in the design member on/off causes the change of the number of spans for deflection calculations and thus the values of limit deflection for particular spans change.

- **Direction** – select plane, for which the deflection check settings are defined:
  - **Plane XZ** – parameters of deflection check in global XZ plane are defined.
  - **Plane XY** – parameters of deflection check in global XY plane are defined.
- **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**.

#### 6.6.3.1 Common data for buckling effects calculation

Data for calculation of buckling effects common for all spans of the column are defined on **Common data** tab.

Properties group **Geometric imperfections**:

- **Use for ULS** – switch on/off taking into account the geometric imperfections for ULS checks.
- **Use for SLS** – switch on/off taking into account the geometric imperfections for SLS checks.
- **Effect considered** – select type of effect considered for determination of imperfections according to 5.2 (6):
  - **Isolated member**
  - **Bracing system:**
    - **Total height of building** – input building total height above the level of moment restraint.
    - **Nr. of vertical members - my** – number of vertical members contributing to the horizontal force in ,y'-direction.
    - **Nr. of vertical members - mz** – number of vertical members contributing to the horizontal force in ,z'-direction.

Properties group **Second order effects:**

- **Analysis of second order effects** – switch on/off taking into account the second order effects when checking compressed members.
- **Braced member to y-axis** – switch on/off considering the existence of bracing system acc. to 5.8.3.1 (1) in direction perpendicular to y-axis.
- **Braced member to z-axis** – switch on/off considering the existence of bracing system acc. to 5.8.3.1 (1) in direction perpendicular to z-axis.
- **Method of analysis** – select method to determine second order effects according to 5.8.5:
  - **Nominal stiffness** – second order effects are determined using method based on nominal stiffness.
  - **Nominal curvature** – second order effects are determined using method based on nominal curvature.
- **Factor c0 to axis** – select method to determine the coefficient which depends on the distribution of first order moment to the corresponding axis 5.8.7.3(2):
  - **User defined** – value of coefficient is defined by user.
  - **Constant first order moment** – coefficient value is 8.
  - **Parabolic first order moment** - coefficient value is 9,6.
  - **Triangular first order moment** - coefficient value is 12.
- **Factor c to axis** – select method to determine the coefficient which depends on the distribution of curvature to the corresponding axis 5.8.8.2 (4):
  - **User defined** – value of coefficient is defined by user.
  - **Constant curvature distribution** – coefficient value is 8.
  - **Sinusoidal curvature distribution** - coefficient value is 9,6.

### 6.6.3.2 Data to determine effective lengths

Data to determine effective lengths of the current span are defined on **Effective lengths** tab.

Properties group **Effective length:**

- **Length of compression member** – member length is displayed.
- **Effective length** – select mode to determine the effective length:
  - **User input** – effective lengths are defined by user.
    - **Effective length y** – input the effective length for buckling perpendicular to y-axis.

- **Effective length  $z$**  – input the effective length for buckling perpendicular to  $z$ -axis.

#### ***6.6.3.3 Deflection check settings***

The deflection check settings for the current span can be defined on **Deflection** tab - see **6.6.1.1 Deflection check settings for the current span**.

## 6.7 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 redistributions and reductions 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.

### 6.7.1 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

Redistribution and redistribution

Redistribution of moments ☒

Reduction of moments ☒

Reduction of shear force ☒

Limited interaction check ☐

Reduction of shear force

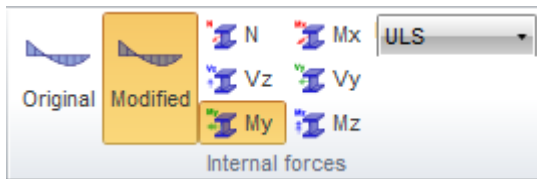
Type of effective depth d

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\*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).

### 6.7.2 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  $V_z$ .
- **Vy** - switches to draw the course shear force  $V_y$ .
- **Mx** – switches to draw the course of torsional moment  $M_x$ .
- **My** – switches to draw the course of bending moment  $M_y$ .
- **Mz** – switches to draw the course of bending moment  $M_z$ .
- **Result class** – select the current result class, for which the internal forces courses are drawn.

## 6.8 Detailed check



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

- **Design member** – if the option is selected, a separate section is generated for each zone on each design member in the design group.
- **Design group** – if the option is selected, sections are generated for zones on the representative design member. The internal forces from all design members in the design group are collected to the corresponding load extremes.

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 Designer and are respected in calculation of deflections.

## 6.9 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** is available for evaluation of section check results.

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

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

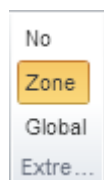
### 6.9.1 Ribbon group Concrete design

See **6.2 Settings for section checks and calculation of deflections** and **6.8 Detailed check**.

### 6.9.2 Ribbon group View settings and scale

See **6.4.3 Ribbon group View settings and scale**.

### 6.9.3 Ribbon group Extreme

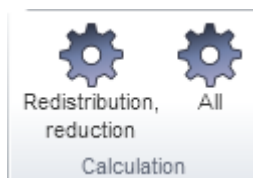


The drawing of check results labels can be modified using commands in this ribbon group.

- **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.

### 6.9.4 Ribbon group Recalculate

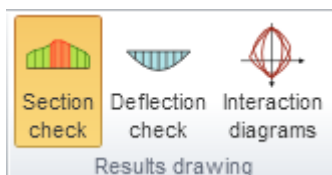




- **Redistribution, reduction** – runs calculation of redistributed and reduced internal forces on the current design member. This command is available, if parameters of redistribution and reduction parameters were changed.

- **All** – runs calculation of redistributed internal forces and 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.

### 6.9.5 Ribbon group Results drawing

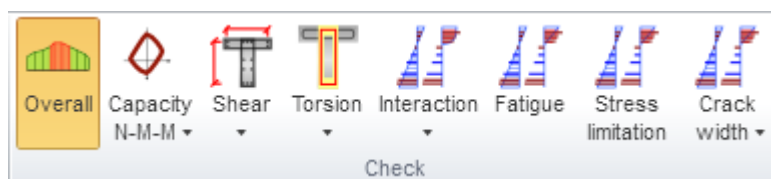


Use options in ribbon group **Results drawing** to set the mode of graphical evaluation of results:

- **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 subzone of the design member.

### 6.9.6 Ribbon group Check

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

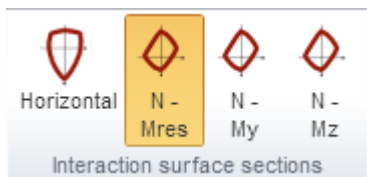


- **Overall** – switch to drawing of envelope of extremes of all performed checks.

- **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,  $w$ ,  $w_{lim}$ ,  $d_e$ ,  $d_{e,lim}$ .

## 6.9.7 Drawing of interaction diagrams

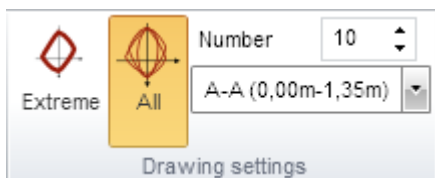
### 6.9.7.1 Ribbon group Interaction surface sections



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.

### 6.9.7.2 Ribbon group Drawing settings



- **Extreme** – switch to draw the extremal interaction diagram in the current position.

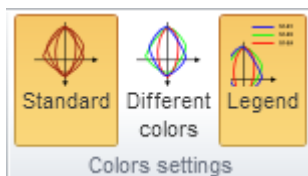
- **All** – switch to draw all interaction diagrams in the current position.

- **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.

### 6.9.7.3 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.

## 6.9.8 Drawing of deflections check results

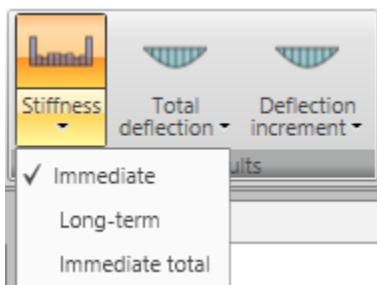
### 6.9.8.1 Ribbon group Settings



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** – switches to draw course of deflection in Z-axis direction of local coordinate system of member.
- **Y** – switches to draw course of deflection in Y-axis direction of local coordinate system of member

### 6.9.8.2 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 stiffnesses 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.

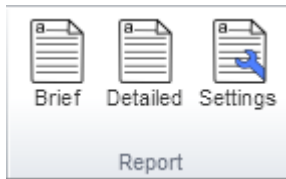
### 6.9.8.3 Ribbon group Stiffness



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

- **EAx** – switch to drawing of axial stiffness  $E A_x$ .
- **EIy** – switch to drawing of flexural stiffness  $E I_y$ .
- **EIz** – switch to drawing of flexural stiffness  $E I_z$ .

### 6.9.9 Check report



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

- **Brief** – 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.

### 6.9.9.1 Report settings

Detailed report settings

- ☒ Redistribution and reduction
  - ☒ Internal forces
    - ☐ All results classes
    - ☒ Selected results classes
    - ☐ SLS Characteristic
  - ☒ Draw picture
    - ☐ All values
    - ☒ Selected values
  - ☐ Vy
  - ☐ Vz
  - ☐ My
  - ☐ Mz
  - ☒ 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
  - ☐ Second order results
  - ☒ Explanation
  - ☒ 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

OK Cancel

To set the content of report click **Settings** in ribbon group **Report**.  
Particular dialog options:

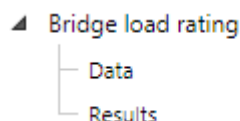
- **Redistributions and reductions** – switch on/off print of all outputs concerning calculation of redistributions and reductions of internal forces.
  - **Internal forces** – switch on/off print of tables of modified internal forces.
    - **All result classes** – switch to print tables of modified 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.
  - **Second order results** – switch on/off the print of the tables with second order calculation.
    - **Explanations** – switch on/off the print of the tables with explanations of second order calculation.
  - **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.
- **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 the print of the table with bill of material.
- **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.

## 7 Bridge load rating of concrete members

For the representative design member of the current design group, 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**.



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

### 7.1 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.

For the representative design member of the current design group the checks to be performed to determine the load rating and the positions, in which the checks are performed, have to be set.

Moreover, groups of load cases and combinations of load cases to determine the load rating have to be defined. But load groups and combinations settings are common for all design groups.

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

#### 7.1.1 Check settings

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

- **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 of the representative design member of the design group:

- **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 of the representative design member of the design group:

- **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.


### 7.1.2 Check positions

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


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"/>	
>	Section 2	2	3,00	6,20	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
	Section 3	3	0,75	9,95	<input 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.

### 7.1.3 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.

#### 7.1.3.1 Permanent load groups

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

	Name	$\gamma G_{sup}$ [-]	$\gamma G_{inf}$	$\xi$ [-]		
>	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.
- **$\gamma G_{sup}$**  – 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.
- **$\xi$**  – 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 **7.1.3.5 Load groups manager**.
- - delete the appropriate permanent load group.

### 7.1.3.2 Variable load groups

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



Check settings Check positions Permanent load groups Variable load groups Load cases Combinations									
Variable load groups								User bridge load group	
Name	Type	Bridge load group	$\gamma_q$ [-]	$\psi_0$ [-]	$\psi_1$ [-]	$\psi_2$ [-]		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.
- $\gamma_q$**  – input the value of partial load factor of variable load cases in ULS combinations.
- $\psi_0$**  – input the value of partial load factor of variable load cases in ULS and SLS Characteristic combinations.

- $\psi_1$  - input the value of partial load factor of variable load cases in SLS Frequent combinations.
- $\psi_2$  - 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 **7.1.3.5 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.

### 7.1.3.3 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 <b>Load cases</b> Combinations					
<b>Load cases</b> Load groups manager					
	Name	Load group	Type	Dynamic factor $\delta$	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 **7.1.3.5 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 is displayed for the load, which has been input in current load case.
- **m1** – for footbridges only. Input weight of uniformly distributed load in  $t/m^2$  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.

### 7.1.3.4 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 <b>Combinations</b>						
<b>Combinations</b>  						
	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 SLSF 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 SLSF 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 SLSF 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 SLSQ 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 SLSQ 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 SLSQ 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 **4.3.2 Combinations of load cases**.
-  - delete the appropriate combination.
- Description** – displays the content of combination rule.



### 7.1.3.5 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**.

**Load Group Manager**

**Load Groups**

- Not grouped cases
- LR LG1
  - SW (1)
  - R (2)
  - SWS (2)
  - R (4)
  - G (4)
- LR gr1a - TS
- LR gr1a - UDL
  - Q**
  - LR gr1a - Pedestr. + cycle track
  - LR gr1b - Single axle
  - LR gr2 - Horizontal forces
  - LR gr3 - Pedestrian loads
  - LR gr4 - Crowd loading
  - LR gr5 - Special vehicles
  - LR Fwk - Persistent
  - LR Fwk - Execution
  - LR F\*\*W - Design
  - LR Thermal - Tk
  - LR QSn,k - Execution
  - LR Construction - Qc

☒ Expand all items

New Delete

**Load Case Properties**

Name: Q

Assigned Load Group: LR gr1a - UDL

Dynamic factor  $\delta$ : 1,00

M1 [t]: 0,00

**Load Group Properties**

Name: LR gr1a - UDL

Type: Exclusive

Road bridge load group: gr1a - UDL

Partial load factor for variable load case in ULS combination ( $\gamma_q$ ): 1,35

Partial load factor for variable load case in ULS and SLS characteristic combination ( $\Psi_0$ ): 0,40

Partial load factor for variable load case in SLS frequent combination ( $\Psi_1$ ): 0,40

Partial load factor for variable load case in SLS quasi-permanent combination ( $\Psi_2$ ): 0,00

When extreme values in the combination are searched, the results of the load cases from :

- (a) permanent group are simply added up,
- (b) standard group are added to the positive extreme if they are positive and to the negative extreme if they are negative,
- (c) exclusive group are evaluated in the way that only one most positive is added to the positive extreme and only one most negative to the negative extreme,
- (d) fatigue group are considered as Qfat load in fatigue combinations,
- (e) accidental group are considered as Ad load in accidental combinations.

OK Cancel

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  $t/m^2$  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.
    - **$\xi$**  – 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:
    - **$\gamma_q$**  – input value of partial load factor for variable load cases in ULS combinations.
    - **$\psi_0$**  – input value of partial load factor for variable load cases in ULS and SLS Characteristic combinations.
    - **$\psi_1$**  – input value of partial load factor for variable load cases in SLS Frequent.
    - **$\psi_2$**  – 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 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  $Q_{fat}$ . 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.

#### 7.1.4 Ribbon group View settings and scale

See 6.4.3 Ribbon group View settings and scale

## 7.2 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.

### 7.2.1 Ribbon group View settings and scale

See 6.4.3 Ribbon group View settings and scale.

### 7.2.2 Ribbon group Extreme

See 6.9.3 Ribbon group Extreme.

### 7.2.3 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..

### 7.2.4 Ribbon group Check

See 6.9.6 Ribbon group Check.

## 8 Steel members design



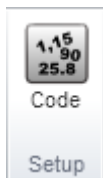
Use commands in navigator group **Steel design** to input design data, buckling lengths, to set design options and to perform and evaluate the check of steel members.

To be able to run the design of steel members, following conditions must be met:

- The frame 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.

### 8.1 Default check settings



Click **Code** in ribbon group **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 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

$\gamma_{M0}$	1
$\gamma_{M1}$	1
$\gamma_{M2}$	1,25
Use always eq. 6.2 in combined strength check	<input type="checkbox"/>
Max slenderness $\lambda$ 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 (i)
Do not take small moments $M_z$ into account in stability check if $M_{zEd}/M_{zRd}$ is less than limit	0,01
Use art. 6.3.3 also for unsymmetrical cross-section when limit for $M_{zEd}/M_{zRd}$ 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 c
Required time of fire resistance ( $t_{fi,req}$ ) [s]	900
Time interval of analysis - Unprotected members ( $\Delta t$ ) [d]	0,0
Time interval of analysis - Protected members ( $\Delta t$ ) [d]	0,0
Fire exposure	All sic
Type of protection	None
Temperature curve	Standard
Net heat flux - Configuration factor ( $\Phi$ )	1
Net heat flux - Surface emissivity of member ( $\epsilon_m$ )	0,7
Net heat flux - Emissivity of flame ( $\epsilon_f$ )	1
Fire protection material - Temperature independent specific heat ( $c_p$ ) [kJ/(kg.K)]	0,0011

Expand all Collapse all OK Cancel

**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 class 2 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  $\lambda$  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  $(\gamma_{M.NEd})/N_{cr}$**  – 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).
- **$\lambda_{LT0}$**  – 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  $M_z$  into account in stability check if  $M_{zEd}/M_{zRd}$  is lesser than limit** – input the limit value when the influence of  $M_z$  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  $M_{zEd}/M_{zRd}$  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  $\epsilon_m$  to determine the net heat flux.
- **Net heat flux – emissivity of flame** – input the emissivity of flame  $\epsilon_f$  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.

## 8.2 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 $M_{zEd}/M_{zRd}$ is exceeded	<input type="checkbox"/>
Do not take into account bending around minor axis in stability check of unsymmetrical cross-s	<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 ( $\alpha_c$ ) [W/(m <sup>2</sup> .K)]	25,0
Net heat flux - Configuration factor ( $\Phi$ )	1
Net heat flux - Surface emissivity of member ( $\epsilon_m$ )	0,7
Net heat flux - Emissivity of flame ( $\epsilon_f$ )	1
Fire protection material - Temperature independent specific heat ( $c_p$ ) [kJ/(kg.K)]	0,0011

#### 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 **8.1 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 **8.1 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 **8.1 Default check settings**. If the option is not selected, specific fire design check settings can be set for the current design group.

### 8.3 Design data

Click navigator command **Steel design > Design data** to input or edit the design data. 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 **Setup**, **LTB restraints**, **Check data** and **View settings** are available.

The screenshot displays the main window of IDEA Designer with a structural model of a beam labeled DM7. The beam is divided into three segments with lengths 2,65, 4,51, and 5,29, totaling 12,45. The segments are labeled M23, M24, and M25 respectively, each with a CSS 4(IPE240) section. The beam is supported at points 1, 2, 3, and 4. A coordinate system (X, Y, Z) is shown at the bottom left.

The Data window is open, showing the design member DM7. It contains two main tables: "Members of current design member" and "Restraints, check data".

ID	Member	Beginning [m]	Length [m]
23	M23	0,00	2,65
24	M24	2,65	4,51
25	M25	7,16	5,29

Type	Side	Beg. position [m]	End. position [m]
Distributed	Top	0,00	2,65

The right side of the Data window shows the "LTB Restraint" properties for the selected member. The Type is set to "Distributed", the Location is "Top", the Position [m] is 0,00, and the End position [m] is 2,65.

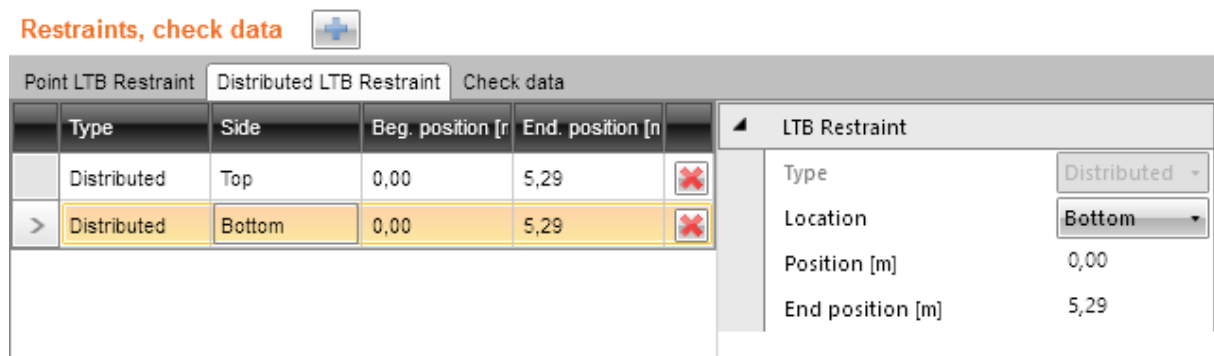
The design data are defined on single members of the current design member.

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 **Restraints, check data**.


Properties of the current design data can be edited in the right part of the table.

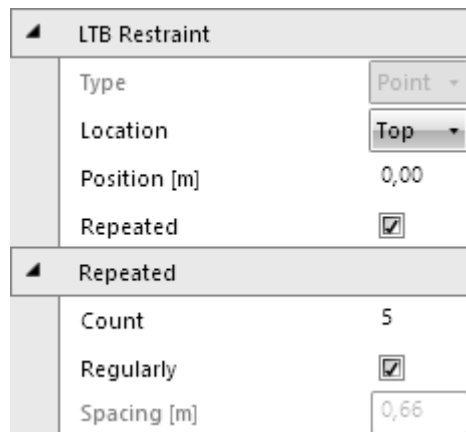


### 8.3.1 Ribbon group Setup

See 8.1 Default check settings.


### 8.3.2 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**.

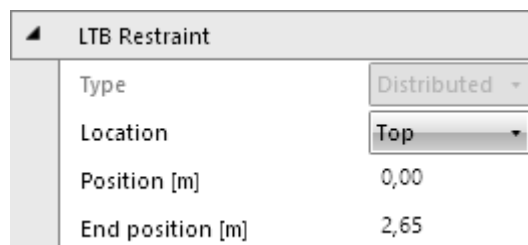



Point restraint properties:

- **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.

### 8.3.3 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**.


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.

### 8.3.4 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]	2,646			
Not checked field				
From beginning [m]	0,600			
From end [m]	0,400			

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.

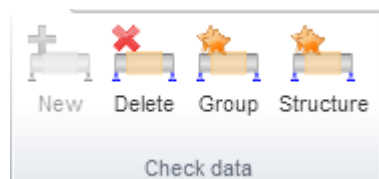
### 8.3.5 Ribbon group LTB restraints



Commands in ribbon group:

- **Point** – add new point LTB restraint on the current member.
- **Distributed** – add new distributed LTB restraint on the current member.
- **Group** – copy the current LTB restraint to all members of the current design group.
- **Structure** – copy the current LTB restraint to all members of the structure.

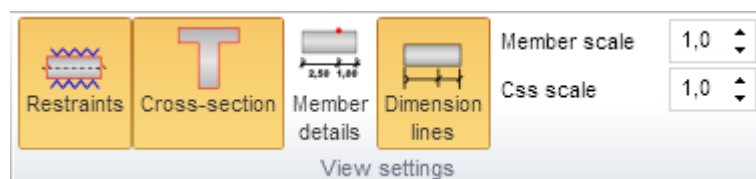
### 8.3.6 Ribbon group Check data



Commands in ribbon group:

- **New** – add new not checked field on the current design member.
- **Delete** – delete the current not checked field.
- **Group** – copy the current not checked field to all members of the current design group.
- **Structure** – copy the current not checked field to all members of the structure.

### 8.3.7 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.

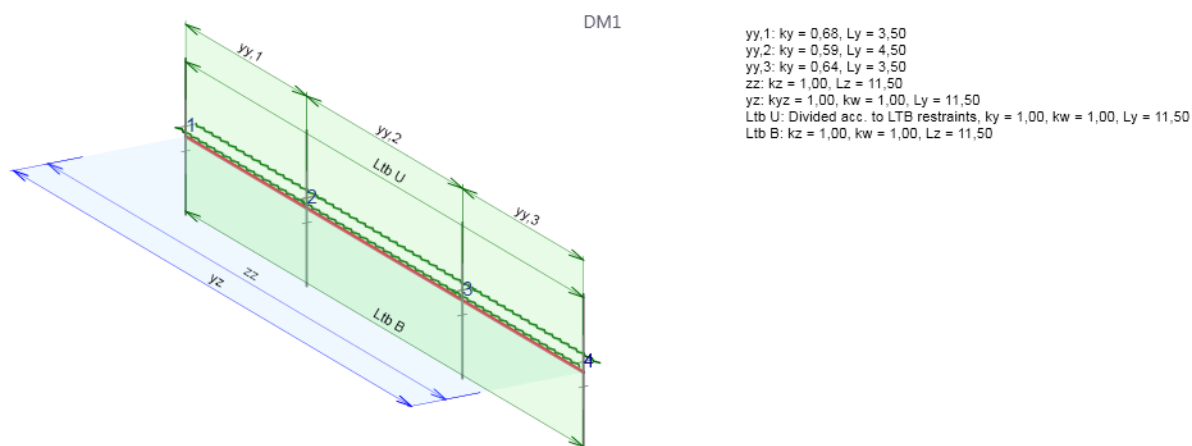
- **Member details** – switch on/off the detailed drawing of the current member of the design member.
- **Dimension lines** – switch on/off the drawing of the dimension lines of the current design member.
- **Member scale** – set the value of the exceeded scale to draw the members of the design member.
- **Css scale** – set the value of the exceeded scale to draw the cross-section picture above the design member.

## 8.4 Buckling lengths

Click navigator command **Steel design > Buckling lengths** to input buckling lengths coefficients required for the check of the buckling resistance.

Ribbon groups **Buckling lengths**, **3D view**, **Structure** and **Dimensions drawing** are available for this navigator command.

Buckling lengths parameters are defined for the current design member.



Design member: **DM1**

**Buckling coefficients definition**

Node	yy	$k_y$	Length	Ltb U	$k_z$	$k_w$	Mcr	Ltb B	$k_z$	$k_w$	Mcr	Def y	Def z	Node
1	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1
2	<input checked="" type="checkbox"/>	0,68	3,50	<input type="checkbox"/>				<input type="checkbox"/>				<input type="checkbox"/>	<input type="checkbox"/>	2
3	<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"/>	3
4	<input checked="" type="checkbox"/>	0,64	3,50	<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    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
  - **k<sub>w</sub>** - either input the value of coefficient or the buckling length
- for LT buckling the coefficients for top and bottom flange of the section:
  - **k<sub>z</sub>**
  - **k<sub>w</sub>**
  - **M<sub>cr</sub>**

Design member DM15 ◀ ▶

**Buckling coefficients definition**

Node	yy	k <sub>y</sub>	Length	Ltb U	k <sub>z</sub>	k <sub>w</sub>	M <sub>cr</sub>	Ltb B	k <sub>z</sub>	k <sub>w</sub>	M <sub>cr</sub>	Def y	Def z	Node
1	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1
2	<input checked="" type="checkbox"/>	0,68	3,50	<input type="checkbox"/>				<input type="checkbox"/>				<input type="checkbox"/>	<input checked="" type="checkbox"/>	2
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"/>	<input checked="" type="checkbox"/>	3
4	<input checked="" type="checkbox"/>	0,72	3,50	<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  
 M<sub>cr</sub> - Calculated    M<sub>cr</sub> - 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 M<sub>cr</sub> 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 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 **k<sub>y</sub>**, **k<sub>z</sub>** - 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 **k<sub>z</sub>** and **k<sub>w</sub>** - if the mode of input of coefficients is set to **User input – factor**, the values of coefficients k<sub>z</sub> and k<sub>w</sub> can be set.
- column **M<sub>cr</sub>** - if the mode of coefficients input for LT buckling is set to **M<sub>cr</sub> – input**, the values of critical moment M<sub>cr</sub> 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.

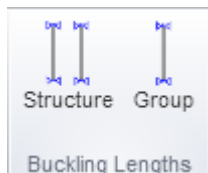
If the option **Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB** is selected, groups for input of buckling coefficients for flexural buckling zz and for



torsional buckling yz are not available. The calculation of flexural buckling zz and torsional buckling yz in this case takes the values of coefficients  $k_z$  and  $k_w$  specified for the check of LTB.

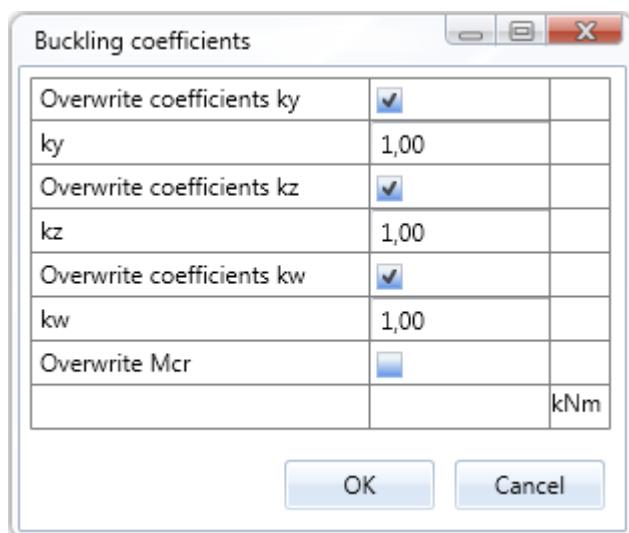
If there are LTB restraints on a member, they are taken into account when determining the buckling lengths and the coefficients  $k_z$  and  $k_w$  cannot be changed.

#### 8.4.1 Mass input of buckling lengths coefficients



Some buckling lengths coefficients can be input en mass – either to all members of the design group or to all members of the whole structure. Commands in ribbon group **Buckling lengths** can be used for the massive input:

- **Structure** – click to input the buckling lengths coefficients to all members of the structure.
- **Group** – click to input the buckling lengths coefficients to all members of the current design group.

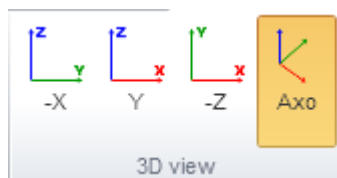


Options of dialog **Buckling coefficients**:

- **Overwrite coefficients ky** – if the option is selected, buckling coefficients  $k_y$  can be overwritten using the value entered in edit box **ky**.
- **Overwrite coefficients kz** – if the option is selected, buckling coefficients  $k_z$  can be overwritten using the value entered in edit box **kz**.
- **Overwrite coefficients kw** – if the option is selected, buckling coefficients  $k_w$  can be overwritten using the value entered in edit box **kw**.
- **Overwrite Mcr** – if the option is selected, critical moment  $M_{cr}$  can be

overwritten using the value entered in edit box **Mcr**.

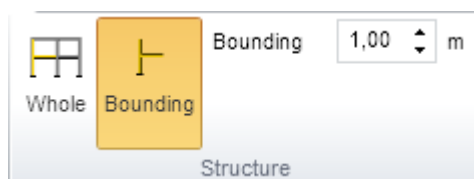
#### 8.4.2 Ribbon group 3D view



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.

#### 8.4.3 Ribbon group Structure

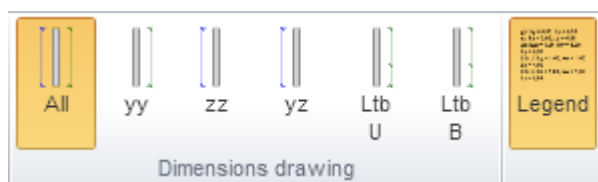


Commands in ribbon group:

- **Whole** – switch to display the whole structure.
- **Bounding** – switch to display the current design member and its neighbourhood.
- **Bounding** – input of the neighbourhood length

around the current design member.

### 8.4.4 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.

## 8.5 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 **Evaluation mode**, **Design member**, **Steel design**, **Extremes**, **Type of check** and **Type of output** are available when evaluating check results.

### 8.5.1 Ribbon group Evaluation mode



Use commands in ribbon group **Evaluation mode** to set the type of design entities, on which the check is evaluated:

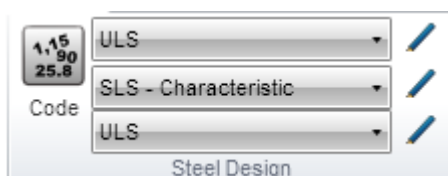
- **Design member** – switch to evaluation on the current design member.
- **Design group** – switch to evaluation on the current design group. A design member with the most extreme value of check is found in the design group. Results are evaluated on this design member.

### 8.5.2 Ribbon group Design member



The current design member, on which the results are evaluated, can be set in the list box **Design member**.

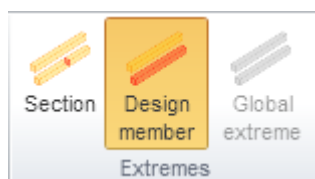
### 8.5.3 Ribbon group Steel design



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 8.1 Default check settings.
- **List of ULS result classes** - select the result class for section check and buckling check.
- **List of SLS result classes** - select the result class for deflection check.
- **List of ULS result classes** - select the result class for fire resistance check.

### 8.5.4 Ribbon group Evaluation

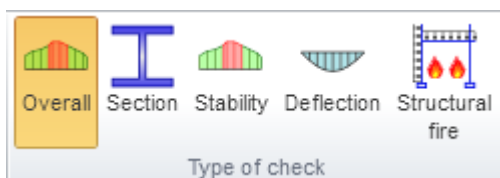


The mode for evaluation of extremes can be set using commands in ribbon group **Extremes**. Following evaluation modes can be set:

- If the **Evaluation mode** is set to **Design group**:
  - **Global** – extreme values of individual checks are searched from all design members in the current design group. One result is printed for each type of check for the current design group.
  - **Des. member** – extreme values of individual checks are searched on each design member. One result is printed for each type of check for each design member of the current design group.

- If the **Evaluation mode** is set to **Design member**:
  - **Des. member** – extreme values of individual checks are searched from all sections on the current design member. One result is printed for each type of check for the current design member.
  - **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.

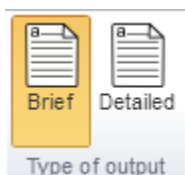
### 8.5.5 Ribbon group Type of check



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.

### 8.5.6 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 outputs only. .
- **Detailed** – switch to print tables with detailed outputs.

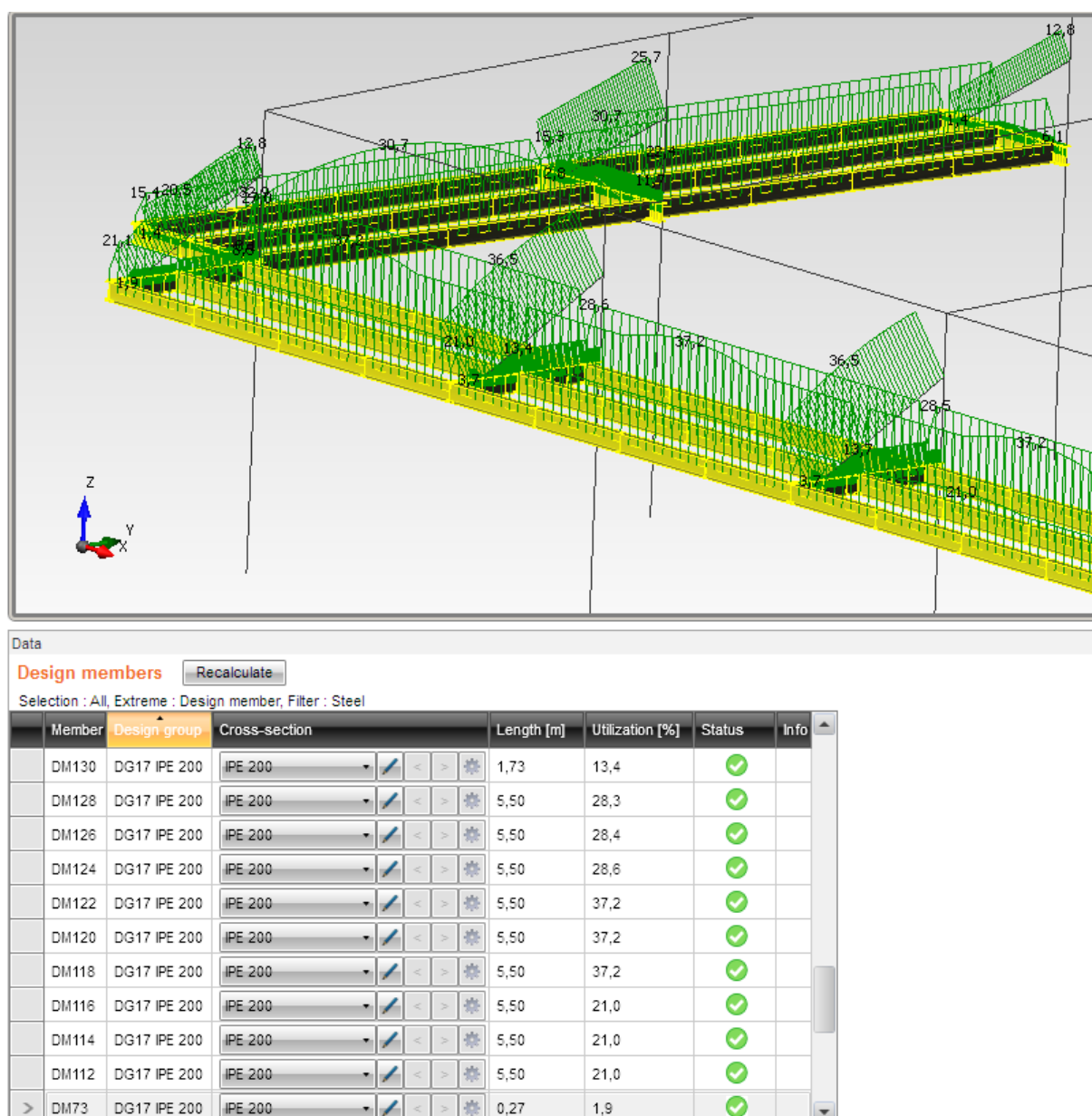
## 9 Checks summary, optimisation of steel cross-sections

Click navigator command **Design summary > Design members** to draw check results courses on the whole structure, to evaluate check results of more design groups and to optimize the steel cross-sections.

The structure and courses of calculated check results is drawn in the main window.

Table of check results is displayed in the data window.

Ribbon groups **Concrete design**, **Steel design**, **Check**, **Results drawing**, **View 3D**, **Visibility**, **Structure drawing** and **Structure labels** are available for this navigator command.



### 9.1 Evaluation on design members

According to the current evaluation settings, list of design members is displayed in the table **Design members**.

Depending on the evaluation settings, following results are displayed in the table:

- Check results for each design member of the current design group;
- Check results for the extreme design member of the current design group;
- Check results for the extreme design member of each design group;
- Check results for each design member of each design group;

Data

**Design members**

Selection : All, Extreme : Design member, Filter : Steel

	Member	Design group	Cross-section	Length [m]	Utilization [%]	Status	Info
	DM130	DG17 IPE 200	IPE 200	1,73	13,4	✓	
	DM128	DG17 IPE 200	IPE 200	5,50	28,3	✓	
	DM126	DG17 IPE 200	IPE 200	5,50	28,4	✓	
	DM124	DG17 IPE 200	IPE 200	5,50	28,6	✓	
	DM122	DG17 IPE 200	IPE 200	5,50	37,2	✓	
	DM120	DG17 IPE 200	IPE 200	5,50	37,2	✓	
	DM118	DG17 IPE 200	IPE 200	5,50	37,2	✓	
	DM116	DG17 IPE 200	IPE 200	5,50	21,0	✓	
	DM114	DG17 IPE 200	IPE 200	5,50	21,0	✓	
	DM112	DG17 IPE 200	IPE 200	5,50	21,0	✓	
>	DM73	DG17 IPE 200	IPE 200	0,27	1,9	✓	





Table **Design members** contains following columns:

- **Member** – displays the name of the design member.
- **Design group** – displays the name of the design group.
- **Cross-section** – displays the cross-section of the design member.
- **Length** – displays the length of design member.
- **Utilisation** – displays the maximal utilisation value from all checks performed on the design member.
- **Status** – displays the graphical representation of the check result on the design member.

### 9.1.1 Optimisation of steel cross-sections

The cross-section dimensions or the whole cross-section can be changed in the column **Cross-section** to reach the required utilisation value.

Note: the cross-section optimisation can be performed only for cross-sections defined in IDEA applications (Frame, Beam). For structures imported from external applications (Axis) the cross-sections has be replaced with the cross-section selected from IDEA database. Column Cross-section contains following options:

- **List of cross-section** – select a cross-section from the list of cross-sections in the project. The selected cross-section is assigned to all members of the current design member.
-  - click to edit the cross-section parameters. The change of dimensions or material affects all members (in all design groups), which the edited cross-section is assigned to.
-  - replaces the cross-section with the previous one (lesser) in the table of rolled sections or reduces the height of welded cross-section by 10 mm. The change of dimensions affects all members (in all design groups), which the edited cross-section is assigned to.
-  - replaces the cross-section with the next one (greater) in the table of rolled sections or increases the height of welded cross-section by 10 mm. The change of dimensions affects all members (in all design groups), which the edited cross-section is assigned to.
-  - starts the automatic optimisation of cross-section. The program searches in the table of rolled sections or changes height of the welded cross-section to find the optimal cross-section. The change of dimensions affects all members (in all design groups), which the edited cross-section is assigned to.

### 9.1.2 Ribbon group Concrete design

Use commands in ribbon group **Concrete design** to set the national code parameters and result classes for concrete check - see **6.2 Settings for section checks and calculation of deflections**.

### 9.1.3 Ribbon group Steel design

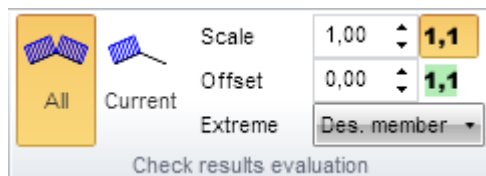
Use commands in ribbon group **Steel design** to set the national code parameters and result classes for steel check – see **8.5.3 Ribbon group Steel design**.

### 9.1.4 Ribbon group Check



- **Recalculate** – run the update of check results on the current design group or design members.



### 9.1.5 Ribbon group Check results evaluation



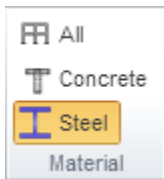
Use commands in ribbon group **Check results evaluation** to set the members to be evaluated:

- **All** – switch to evaluate check results on all design groups in the structure. If this mode is active, commands in ribbon group **Material** can be used to filter design groups or design members according to their materials.
- **Current** – switch to evaluate check results on the current design group.
- **Extreme** – set the extremes evaluation mode:
  - **Group** – the extreme check value is searched for the whole design group.
  - **Des. member** – the extreme check value is searched for each design member in the design group.



- **Material** – the extreme check value is searched for each material type in the structure. This mode is available only if the evaluation is set to all design groups on the structure.
-  - switch on/off drawing of value numbers for evaluated components.
-  - switch on/off drawing of background under value numbers.
- **Scale** – to increase or decrease scale of results drawing, enter a number into edit box or click arrows on the right side of edit box.
- **Offset** – to increase or decrease the distance between graphs of results and drawing of structure, enter a number into edit box or click arrows on the right side of edit box.

#### 9.1.6 Ribbon group Material



If the evaluation is set to all members of the structure, commands in ribbon group **Material** can be used to filter the results::

- **All** – switch to evaluate check results on all members regardless the material type.
- **Concrete** – switch to evaluate check results only on members of concrete material.
- **Steel** – switch to evaluate check results only on members of steel material.



## 9.2 Bill of material

To display the simple bill of material click navigator command **Design summary > Bill of material**.

Following tables are displayed in the main window:

- The weight and surface summary of steel members;
- The concrete volume and reinforcement weight summary of concrete members;
- The price table of steel, concrete, reinforcement steel and painting area;
- List of concrete design groups;
- List of lengths and weight for individual steel cross-sections.

## 10 Design and check of joints











Data to design steel joint in IDEA Connection program can be exported for selected joints of the structure.

Click navigator command **Connection design > Joints** to prepare data for steel joints design.



Table of joints for design is displayed in the main window.

3D drawing of the structure is displayed in the main window.

Ribbon groups **Joints** and **Steel design** are available.

Data						
Drag a column header and drop it here to group by that column						
	Node ▼	Name ▼	Members ▼	Support ▼	Design ▼	
	N82	N82	DM160, DM161, DM180, DM376	<input type="checkbox"/>	<input checked="" type="checkbox"/>	 
	N98	N98	DM177	<input type="checkbox"/>	<input checked="" type="checkbox"/>	 
	N99	N99	DM177, DM178, DM186, DM187	<input type="checkbox"/>	<input checked="" type="checkbox"/>	 
	N100	N100	DM178, DM179, DM192, DM193	<input type="checkbox"/>	<input checked="" type="checkbox"/>	 
	N101	N101	DM179, DM180, DM198, DM199	<input type="checkbox"/>	<input checked="" type="checkbox"/>	 

Columns in the table of structural joints:

- **Node** – number of node is displayed.
- **Name** – user defined joint name is displayed.
- **Members** – displays list of members connected in the appropriate node. List of members can be modified by user.
- **Support** – if the option is selected, a support was found in the appropriate node.
- **Design** – if the option is selected, the appropriate joint is exported to be designed in IDEA Connection.
-  - starts modification of the appropriate definition of joint for design.
-  - deletes the appropriate definition of joint for design.

### 10.1 Ribbon group Joints




- **New** – starts input of new joint definition for design in IDEA Connection.
- **Design** – starts export of data to IDEA Connection program for joints selected in the column **Design**. Geometry of members in joint, cross-sections of members and load cases are generated for each selected joint. Load cases are generated respecting the content of the current result class for steel code check selected in ribbon group **Steel design**.
- **Scale** – value of scale to draw nodes defined for joint design.

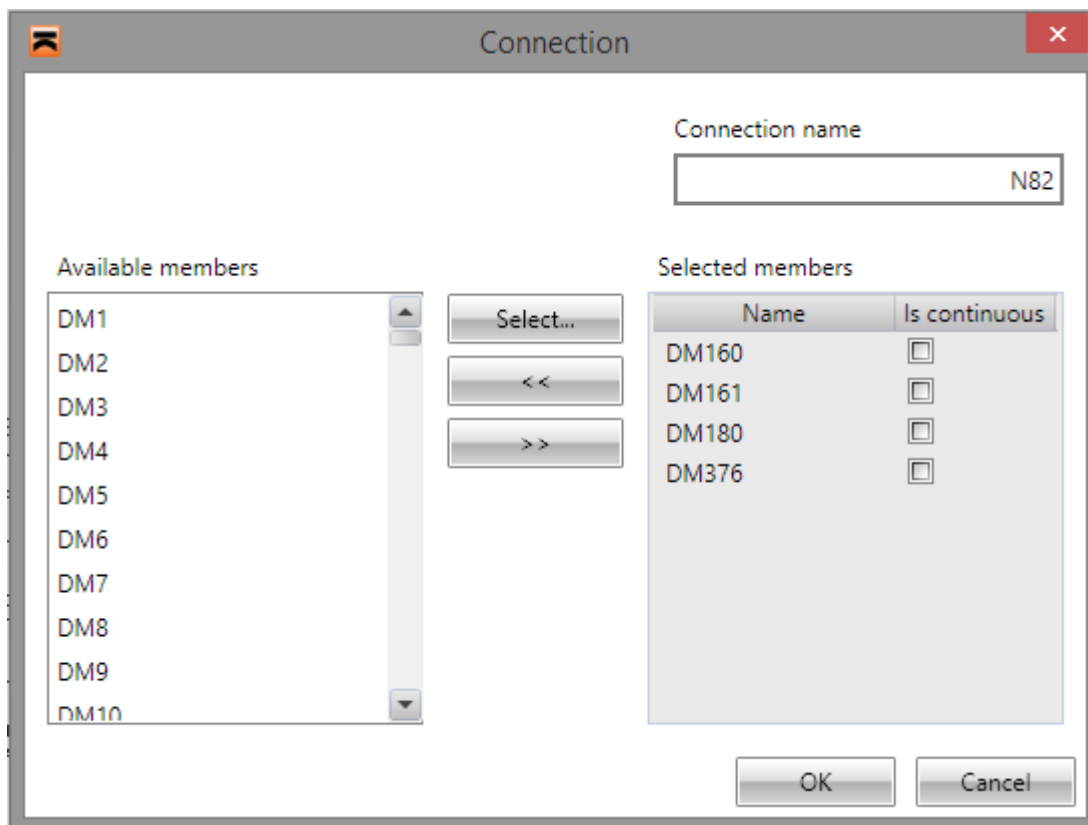
## 10.2 New joint for design

To define new joint(s) to be designed in IDEA Connection click **New** in ribbon group **Joints**.

Dialog for selection of items in 3D window appears – see **4.5.3.1 Selecting members in 3D** . After selecting nodes, for which the joints will be designed, a new row of joint definition is added into the table of joints in data window.

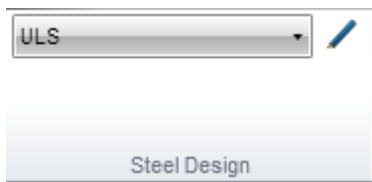
## 10.3 Modification of joint for design

To modify joint definition click  in the appropriate table row. Dialog of joint definition properties appears.




- **Connection name** – input user defined name of joint.
- **Selected members** – list contains members connected in the current joint. If **Is continuous** option is selected, the corresponding member is in IDEA Connection considered as continuous through the joint origin (internal forces are applied both on beginning and on end of member), otherwise then member is considered as ended in the joint origin.
- **Available members** – list contains all members of the structure.
- **Select** – start the interactive selection of structural members in 3D window – see **4.5.3.1 Selecting members in 3D window**.
- **>>** - add structural members, which are selected in the list **Available members**, to the modified joint.
- **<<** - remove structural members, which are selected in the list **Selected members**, from the modified joint.

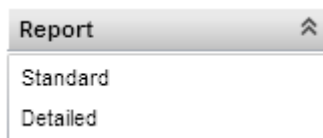
## 10.4 Ribbon group Steel design



Commands in ribbon group **Steel design** can be used to set or edit the result class for steel check at ultimate limit state.

- **List of ULS result classes** - select the current result class to generate the load cases of designed joints. Click  to modify the current result class.

## 11 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.

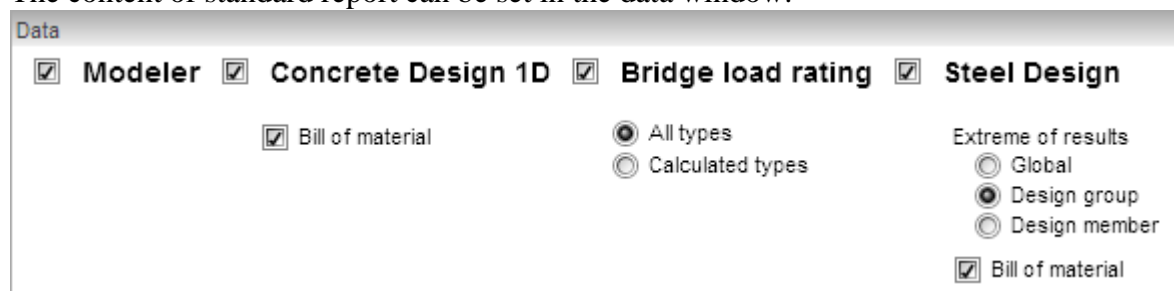
Use commands in ribbon group **Report** to set and print the report.

Ribbon group **Report view** is available for this navigator command.

### 11.1 Standard report

Click navigator command **Report > Brief** to generate the standard report.

The content of standard report can be set in the data window:



- **Modeler** – switch on/off print of the table with project data.
- **Concrete design 1D** switch on/off print of tables and pictures of check results of concrete design groups.
  - **Bill of material** – switch on/off print of table of bill of material of concrete members.
- **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).
- **Steel design** - switch on/off print of tables of steel check results.
  - **Global** – the extreme steel check values are searched from all appropriate design groups.
  - **Design group** – the extreme steel check values are searched for each appropriate design group.
  - **Design member** – the extreme steel check values are searched for each appropriate design member.
  - **Bill of material** – switch on/off print of table of bill of material of steel members.

## 11.2 Detailed report

Click navigator command **Report > Detailed** to generate the detailed report.

The content of detailed report can be set in the data window.

### 11.2.1 Input data

Options to set the content of input data detailed report:

☒ **Modeler**

- ☒ Project data
- ☐ Result classes, combinations, load groups
- ☒ Cross sections
- ☒ Design members
- ☒ Design groups
  - Drawings
    - ☒ None
    - ☐ Normal
    - ☐ Large
  - ☒ Use SVG
- ☒ Materials

- **Modeler** – switch on/off the print of tables with structure data and calculation results.
- **Project data** – switch on/off print of tables of project identification data.
- **Result classes, combinations, load groups** – switch on/off the print of tables of defined result classes, load combinations and groups of load cases..
- **Cross-sections** – switch on/off print of the tables of cross-section pictures and cross-sectional characteristics.
- **Design members** – switch on/off print of the table of design members.
- **Design groups** – switch on/off print of the table of design groups.
  - **Drawings** – switch on/off print of the design group pictures:
    - **None** – no design groups pictures are printed.
    - **Normal** – normal size design group pictures are printed.
    - **Large** – large size design group pictures are printed.
    - **Use SVG** – if the option is selected, the vector graphics is used to generate the pictures. If the option is not selected, the raster graphics is used to generate the pictures. The SVG pictures in the report can be viewed only if Internet Explorer 9 and higher is installed on the workstation.
- **Materials** – switch on/off the print of the table of used materials.

### 11.2.2 Concrete 1D design results

#### ☒ Concrete Design 1D

- ☒ Redistribution and reduction
  - ☒ Internal forces
    - ☐ All results classes
    - ☒ Selected results classes
  - ☒ Draw picture
    - ☐ All values
    - ☒ Selected values
    - ☐ Vy
    - ☐ Vz
    - ☐ My
    - ☐ Mz
  - ☒ 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
  - ☐ Second order results
  - ☒ Explanation
  - ☒ 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.
- **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.
  - **Second order results** – switch on/off the print of the tables with second order calculation.
    - **Explanations** – switch on/off the print of the tables with explanations of second order calculation.
  - **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.



### 11.2.3 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.

### 11.2.4 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

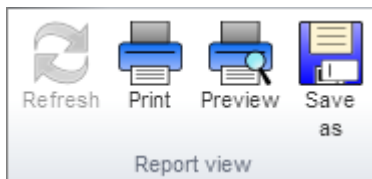
#### Extreme of results

- ☐ Global
- ☒ Design group
- ☐ Design member
- ☒ 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.
- **Global** – the extreme steel check values are searched from all appropriate design groups.
- **Design group** – the extreme steel check values are searched for each appropriate design group.
- **Design member** – the extreme steel check values are searched for each appropriate design member.
- **Bill of material** – switch on/off print of table of bill of material of steel members.

### 11.3 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.

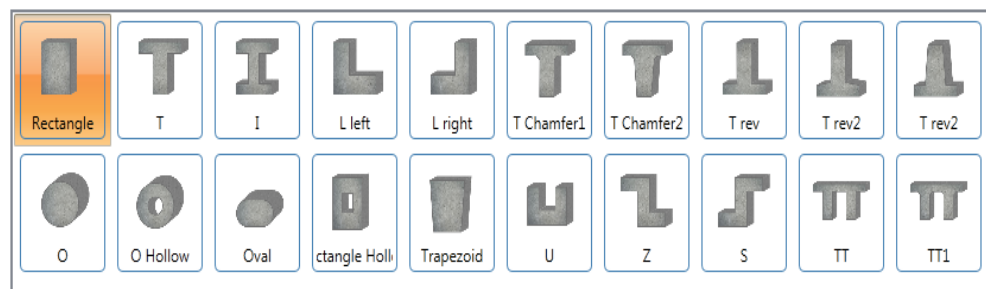
## 12 Cooperating programs

IDEA Designer can read data from several FEA programs used on the market. The interface is constantly developed and improved. Functionality and limitation of specific programs is described below.

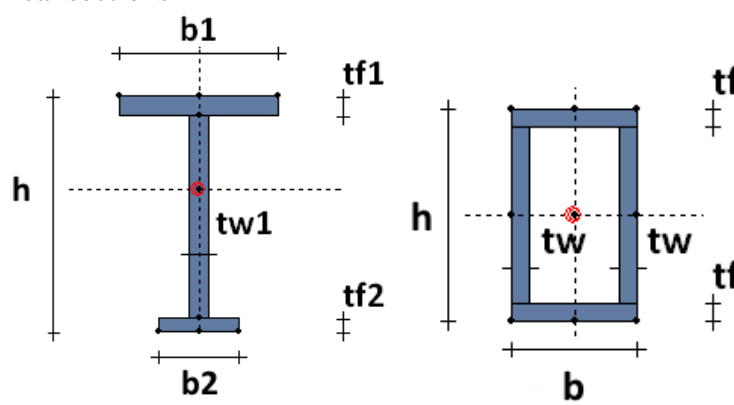
### 12.1 Program AxisVM

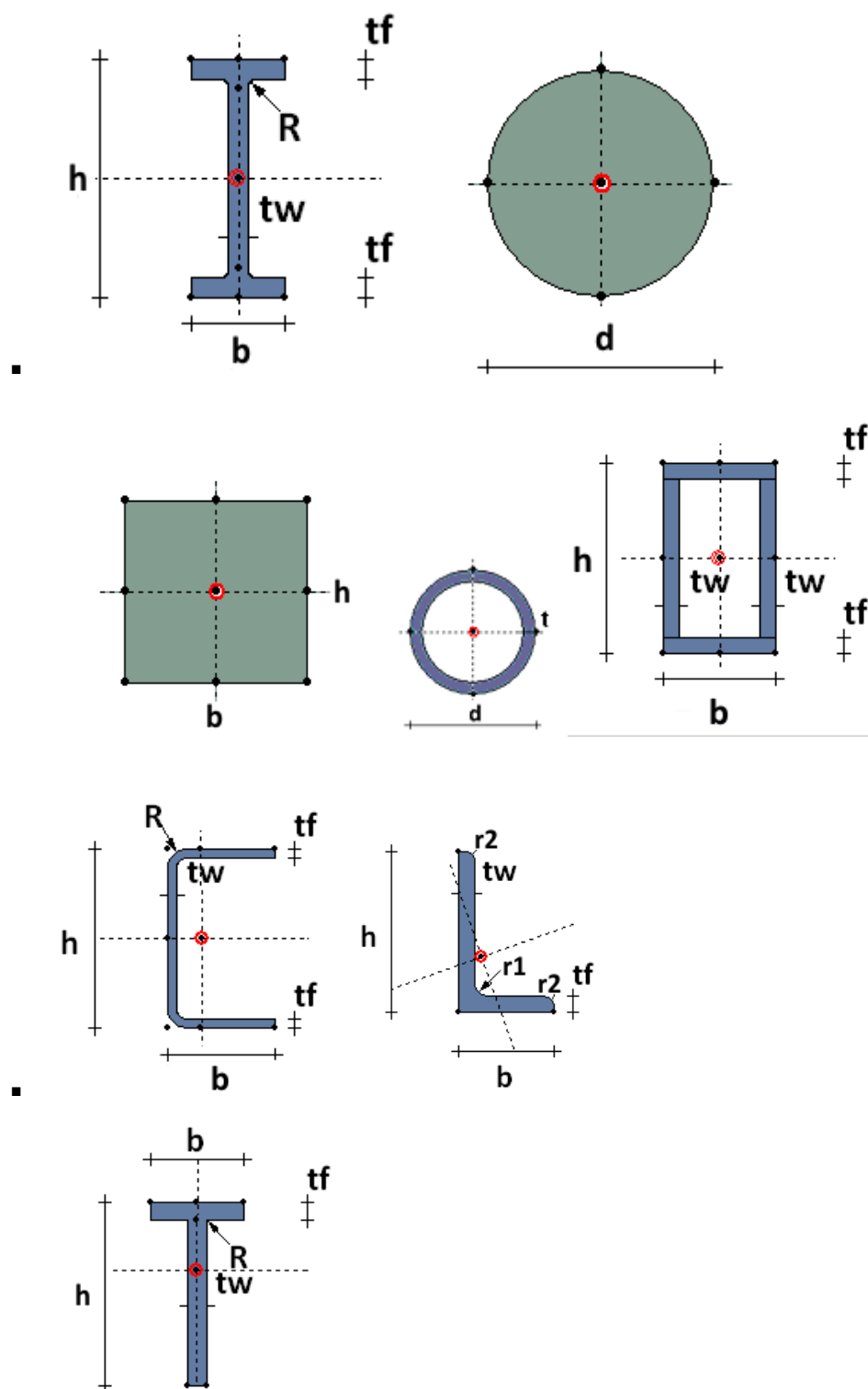
Following items are read from program AxisVM:

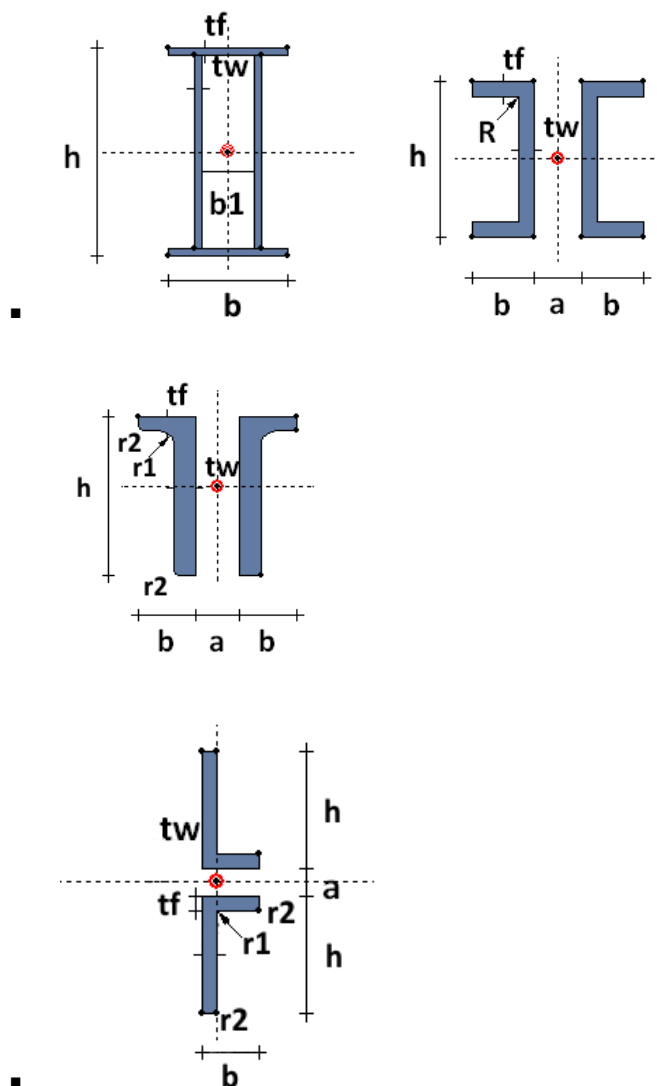
- Nodes
- Lines
  - straight
  - arcs
- Materials
  - Steel
  - Concrete EC, EC cz (C12/15, C16/20, C20/25, C25/30, C30/37, C35/45, C40/50, C45/55, C50/60, C55/67, C60/75, C70/85, C80/95, C90/105)
  - Timber
  - Other materials are imported as general
- Cross-sections
  - General CSS
  - IDEA RCS concrete sections



- Geometrical sections







- **Members 1D** - steel members - only prismatic. Concrete members 1D are imported including haunches, if the haunch has the same cross-section type at the beginning and at the end..
  - Trusses
  - Ribs
  - Members – no haunches
- **Members 2D (domains)** - slabs, walls
  - Outline – Lines, arcs
  - Outline – Lines, arcs
- **Nodal supports in GCS**
  - hinged
  - rigid
- **Combinations**
  - Description of load combinations – case + coefficient
- **Calculation model**
  - Nodes

- Mesh elements– triangles, rectangles
- Internal forces on members 1D -  $N, V_y, V_z, M_x, M_y, M_z$  in all sections
  - in principal axes
  - in LCS
- Deformations on members 1D -  $u_x, u_y, u_z, f_{ix}, f_{iy}, f_{iz}$  in all sections
  - in principal axes
  - in LCS
- Internal forces on members **2D** -  $m_x, m_y, m_{xy}, v_{xz}, v_{yz}, n_x, n_y, n_{xy}$  in all mesh elements

## 12.2 Program Esa Prima Win

Following items are read from program EPW:

- Nodes
- Materials
- Cross-sections
  - Concrete – Rectangle, T, I, Circle
  - Rolled
    - HD, HEA, HEB, HEC, HEM, HL, HX, I, IPE, IPN, J, RSJ, UB, UC
    - CH, KGU, PFC, PJR, RSC, U, UAP, UE, UNP, UPE
    - AC, AH, FQ, FR, K, MQ, MR, MSH, QRO, RQ, RR, RRO, SC, SH, SHS, SHS(cf), VHP
    - H, HA, HFLeq, HFLue, HS, L, LS, RSEA, RSUA
    - B, CHS, CHS(cf), MSRR, RO
    - FLA, FLB
  - Pairs
    - 2Uo, 2 Uc, 2 LT, 2 LU
  - Closed sections
    - BOX fl, BOX web, Box2I, Box2U, Box 2L, Box 4L
  - Sheet welded
    - Isym, I nonsym
- Members 1D
  - Trusses
  - Ribs
  - Members – no haunches
- Nodal supports
  - free
  - rigid
  - flexible
- Hinges
- Load cases
- Combinations - dangerous
- Internal forces on members 1D - N, Vy, Vz, Mx, My, Mz in all sections
  - in principal axes
- Deformations on members 1D - ux, uy, uz, fix, fiy, fiz Mz in all sections
  - in principal axes



## 12.3 Program SCIA Engineer

XML files generated from program SCIA Engineer are used to import data from SCIA Engineer.

To import data from SCIA Engineer it is necessary to have both files, \*.XML and \*.DEF, created by export from SCIA Engineer, in one directory.

Import is limited – only geometry of structure (members 1D, members 2D with straight edges) and internal forces on 1D members are imported. Internal forces on 2D elements are not imported.

Import from versions 2010.0 and higher is supported, import from versions 2007 and 2009 is limited.

XML document for export into IDEA Designer must contain following tables:

- Nodes
- Materials
- Cross-sections
- Members 1D
- Haunches (SCIA Engineer 2010 and higher)
- Arbitrary members (SCIA Engineer 2010 and higher)
- Beam openings
- 2D members (SCIA Engineer 2010 and higher)
- 2D member opening
- Load cases
- Load groups
- Combinations
- Internal forces on members

To export XML file XML document has to be created and tables of particular chapters in XML document has to be edited to follow rules described below.

Content of XML document (list of chapters) can be loaded from template files, which are included in installation package of IDEA Designer:

- \Templates\IDEA\_Import\_Designer\_2007.TDX (for SCIA Engineer 2007)
- \Templates\IDEA\_Import\_Designer\_2009.TDX (for SCIA Engineer 2009)
- \Templates\IDEA\_Import\_Designer\_2010.TDX (for SCIA Engineer 2010 and higher)

Edited table template files are stored in package:

- \Templates\otx\_2007.zip (for SCIA Engineer 2007)
- \Templates\otx\_2009.zip (for SCIA Engineer 2009)
- \Templates\otx\_2010.zip (for SCIA Engineer 2010 and higher)

The appropriate package of table template files (otx\_20xx.zip) has to be unpacked into the directory:

- xxx\DocumentTemplates\xml\, where xxx is User Settings Directory of SCIA Engineer application. The default full path of this directory is eq. for SCIA Engineer 2009:  
c:\Users\\_user\_name\_\ESA90\user\DocumentTemplates\XML\ on Win7 and Vista and  
c:\Documents and Settings\\_user\_name\_\ESA90\user\DocumentTemplates\XML\ on Win XP.

### 12.3.1 Nodes

Table of chapter Nodes must contain following columns:

- **Name**
- **Coord X**
- **Coord Y**
- **Coord Z**

### 12.3.2 Materials

Following materials are imported into IDEA Designer:

- All Steel EC materials
- Concrete materials EC (C12/15, C16/20, C20/25, C25/30, C30/37, C35/45, C40/50, C45/55, C50/60, C55/67, C60/75, C70/85, C80/95, C90/105)

Other materials are imported as general material.

Value of parameter '**Calculated dependent values**' is important during import of concrete material. It determines, whether material characteristics correspond to national code or material characteristic were edited by user.

If only code materials were used in imported project, table Materials must contain following columns:

- **Name**
- **Material type**
- **Calculated depended values**
- **Characteristic compressive cylinder strength fck(28)**

According those parameters is imported material compared with materials in IDEA Designer materials. If no material is found with matching name and fck(28), new one is created. But then table Materials must contain more columns:

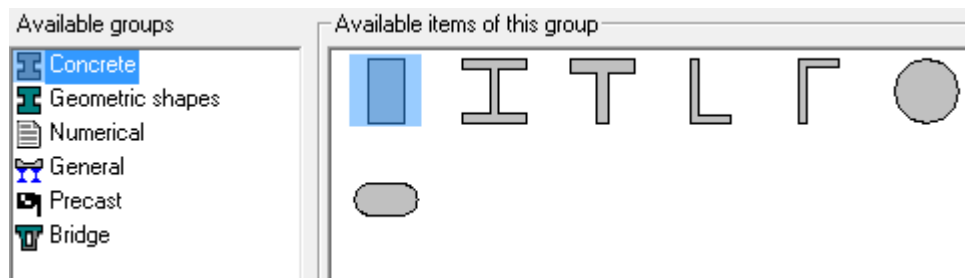
- **Thermal expansion**
- **Unit mass**
- **E modulus or Modul E** – acc. to version of SCIA Engineer
- **G modulus**
- **Poisson coeff.**
- **Stone diameter (dg)**
- **Cement class**
- **Characteristic compressive cylinder strength fck(28)**
- **Mean compressive strength fcm(28)**
- **Mean tensile strength fctm(28)**
- **fctk 0,05(28)**
- **fctk 0,95(28)**
- **Strain at reaching maximum strength eps c2**
- **Ultimate strain eps cu2**
- **Strain at reaching maximum strength eps c3**
- **Ultimate strain eps cu3**
- **n**
- **Type of aggregate**

- **Measured values of mean compressive strength (influence of ageing)**
- **Type of diagram**

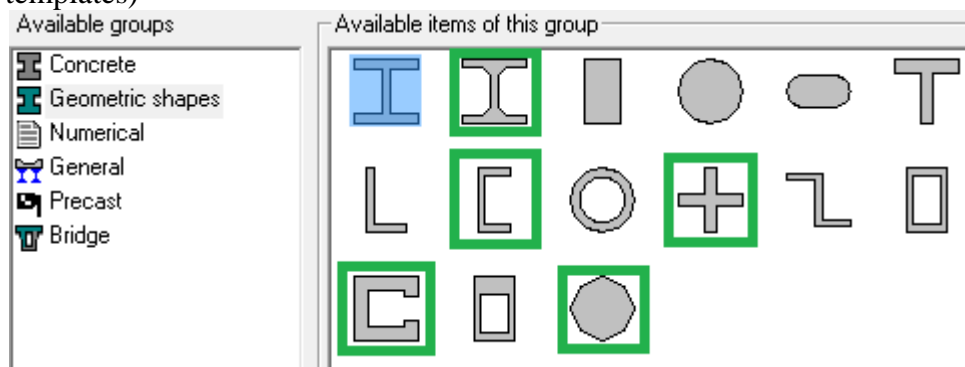
### 12.3.3 Cross-sections

Import of following cross-section types is supported:

a) all cross-section from **Concrete** group are converted to standard concrete sections in IDEA Designer/IDEA RCS (can be reinforced using reinforcement templates in IDEA RCS)



b) all cross-sections from **Geometric shapes** group can be imported into IDEA Designer/IDEA RCS. Cross-sections in green frames are converted to general cross-section (cannot be reinforced using reinforcement templates), other cross-sections are converted to standard concrete cross-sections (can be reinforced using reinforcement templates)

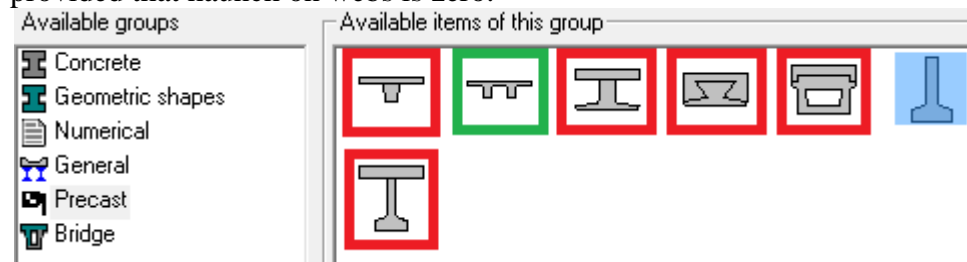


c) all cross-sections from **Geometric shapes** group can be imported into IDEA Designer/IDEA RCS. Cross-sections are imported as general cross-sections (defined by vertexes).

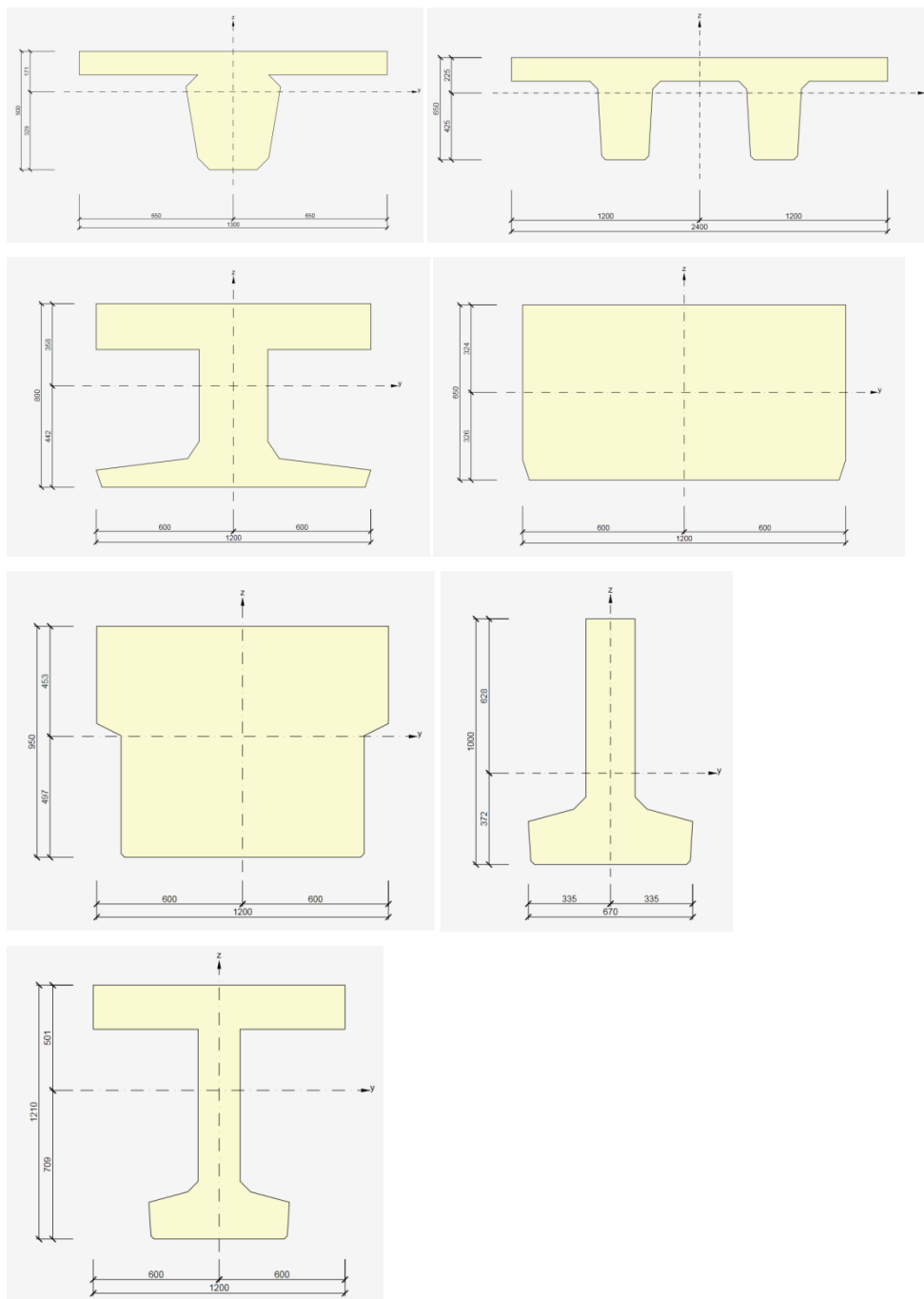
Cross-sections in red frames are converted to one component cross-sections, because current IDEA Designer/IDEA RCS does not support more components in cross-section. If components of imported cross-section have different materials, material from first component is used for imported cross-section.

Cross-section Double T shape (in green frame) is imported as standard Double T,

provided that haunch on webs is zero.



Shapes of particular precast bridge cross-sections after import into IDEA Designer/IDEA RCS:



c) all cross-sections from **Bridge** group can be imported into IDEA Designer/IDEA RCS.

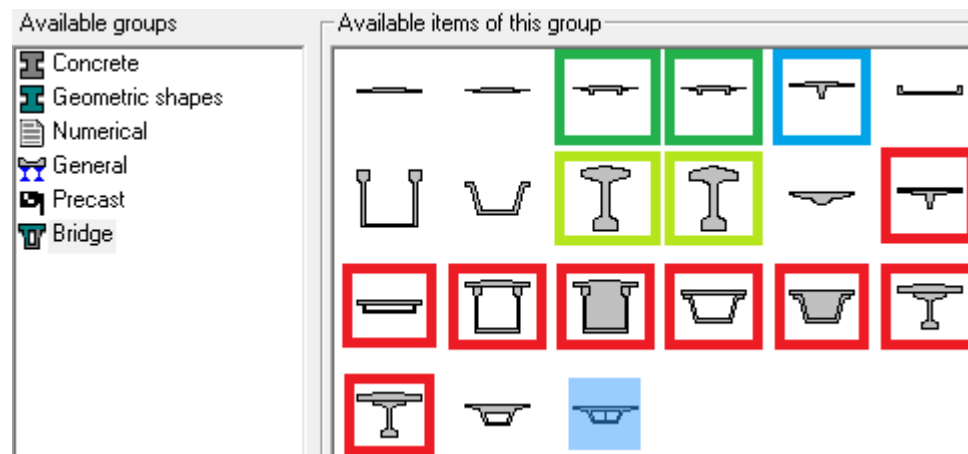
Cross-sections are imported as general cross-sections (defined by vertexes).

Cross-sections in red frames are converted to one component cross-sections, because current IDEA Designer/IDEA RCS does not support more components in cross-section. If components of imported cross-section have different materials, material from first component is used for imported cross-section.

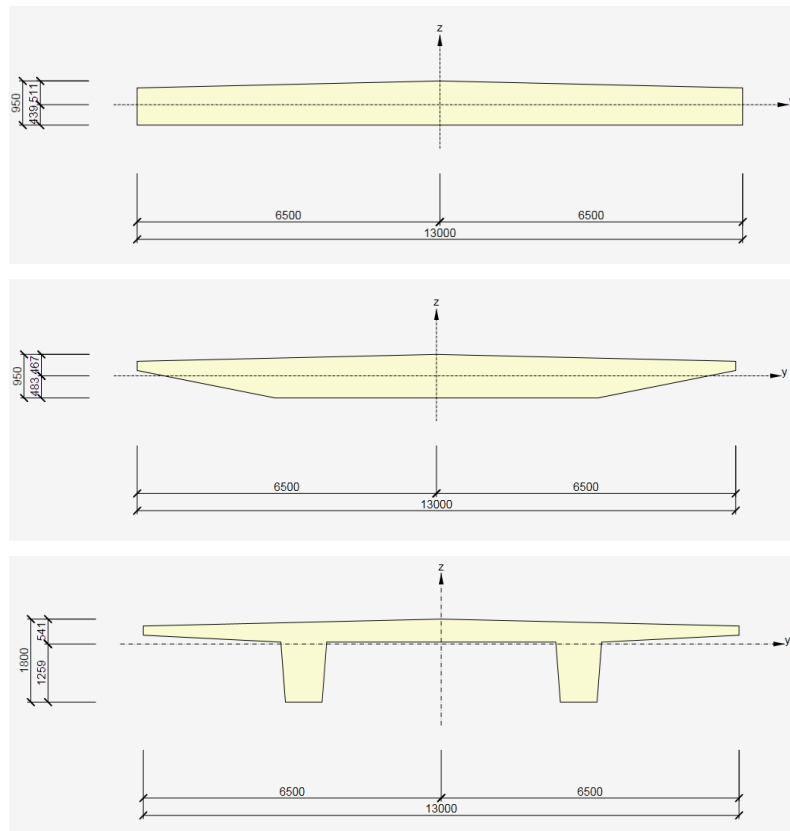
Cross-sections in dark green frames are imported as standard Double T, provided that haunches on cross-sections are zero.

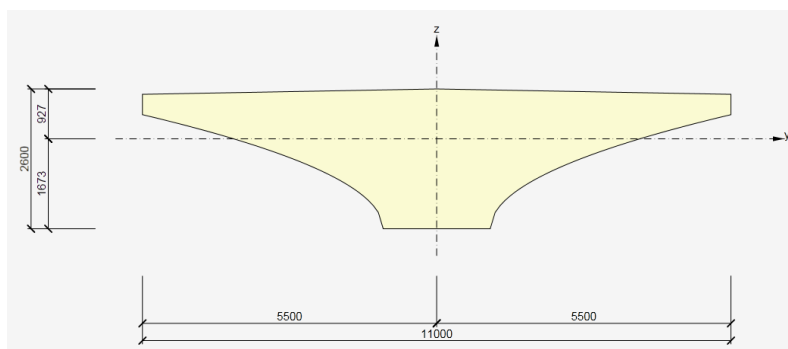
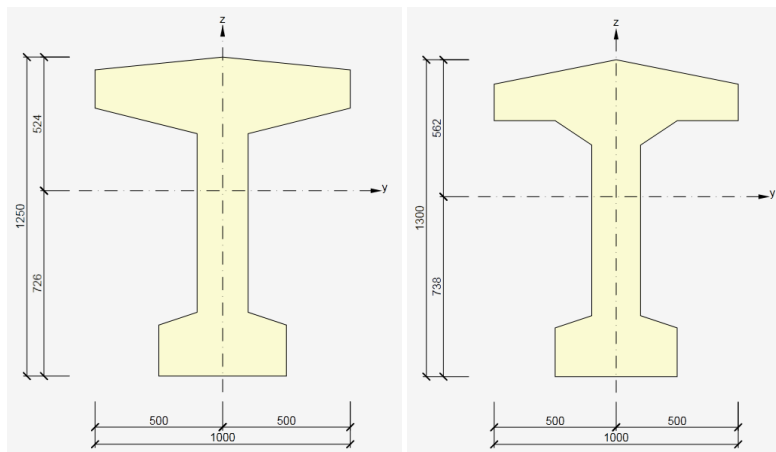
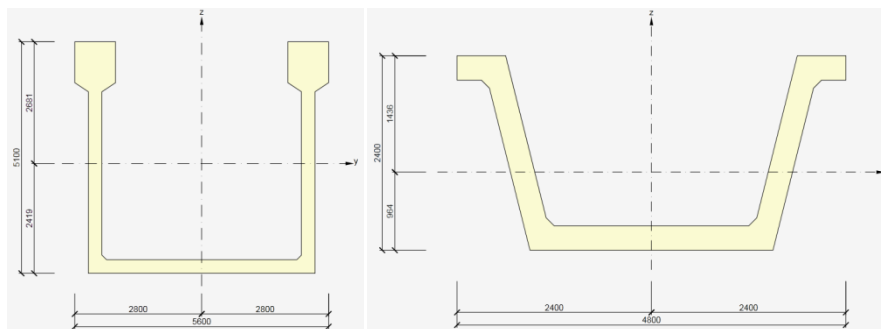
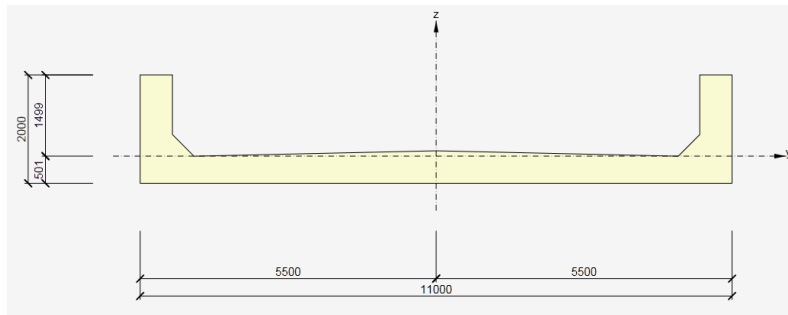
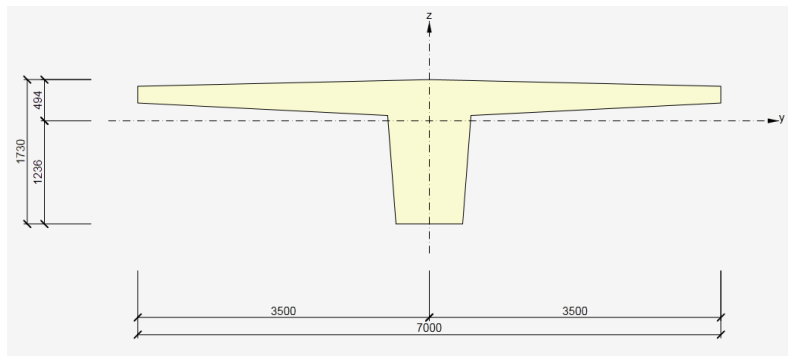
Cross-section T in blue frame is imported as standard T with haunches on flange, provided that haunches on flange of imported cross-section are non-zero.

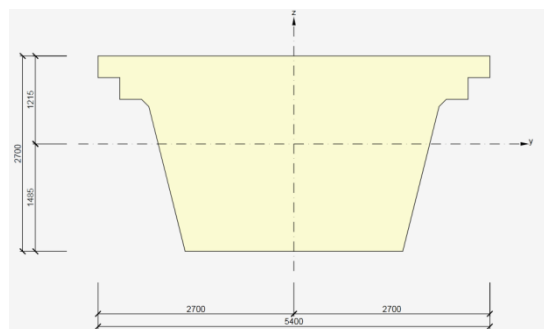
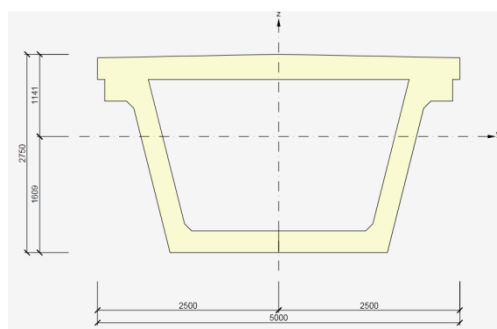
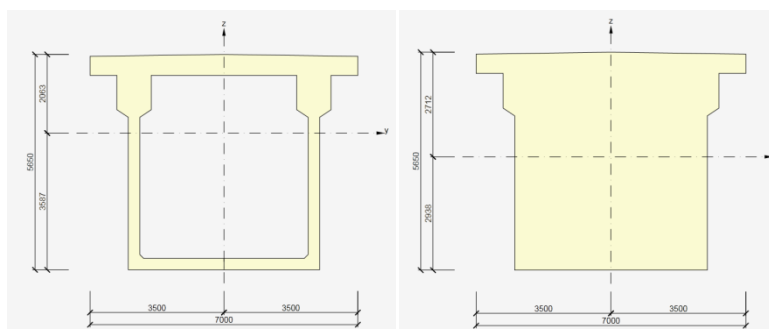
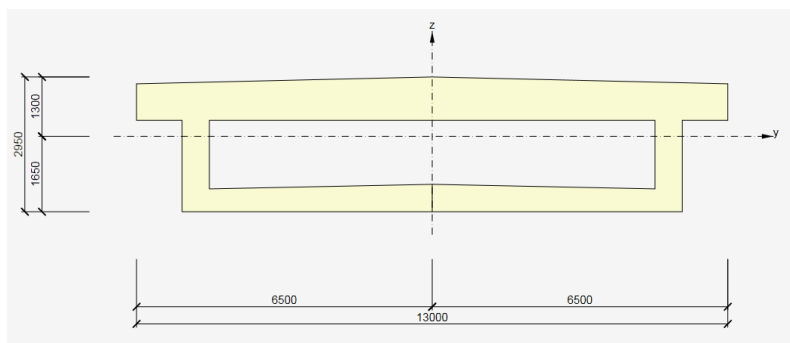
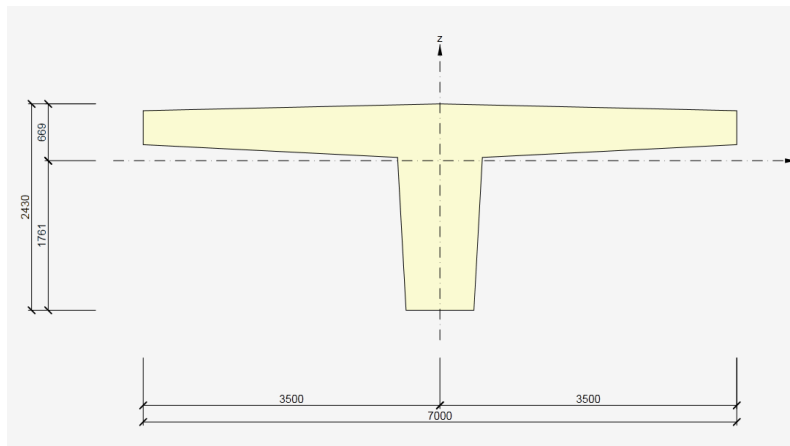
Cross-sections in light green frames are imported as standard I shape, provided that haunches are zero.

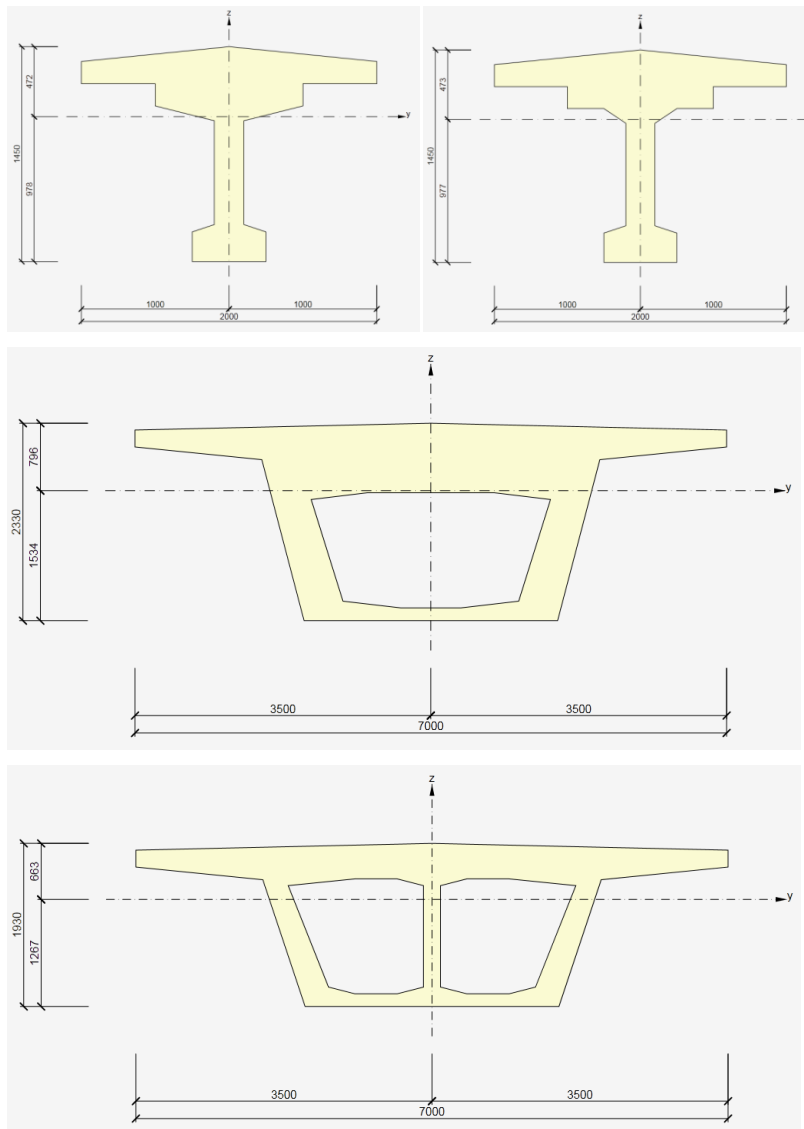


Shapes of particular bridge cross-sections after import into IDEA Designer/IDEA RCS:

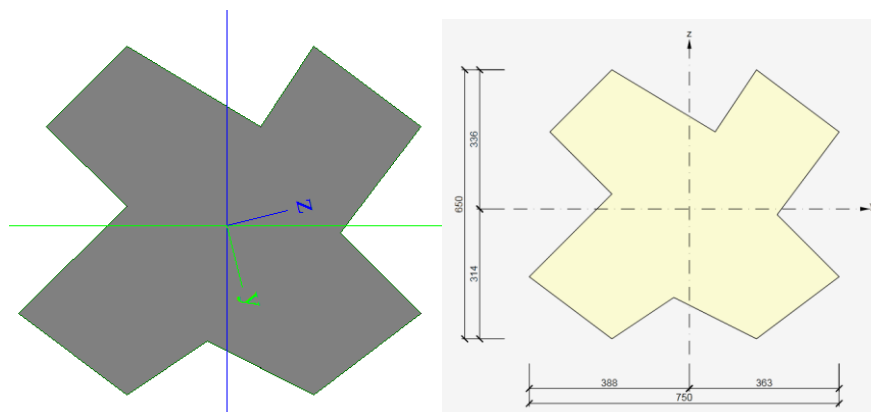




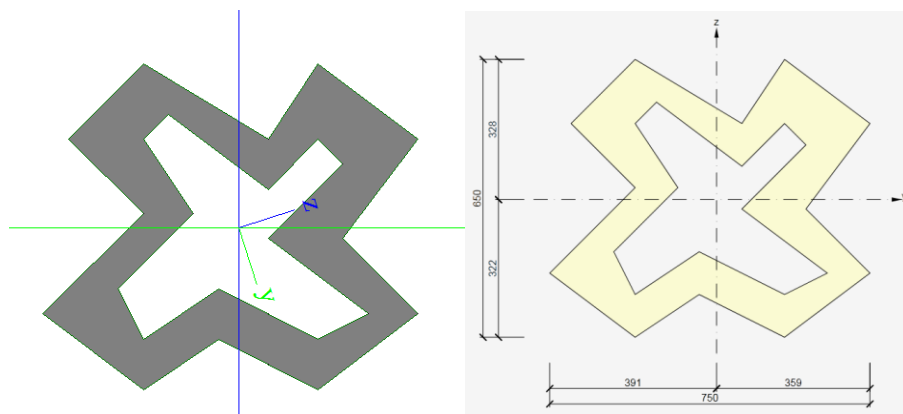




d) **general cross-sections** – import of cross-sections with one component only is supported. If general cross-section contains more components, only first one is imported. Number of openings is not limited.

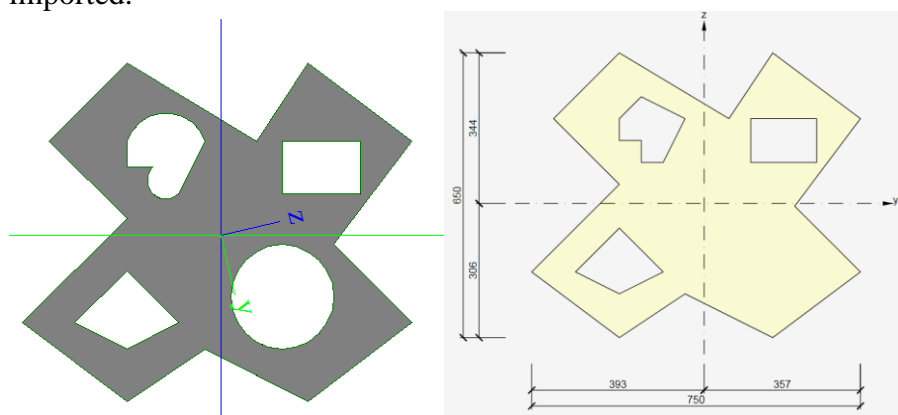






It is not possible to import cross-sections with curved edges, because curved edge is not described in XML file. Circular holes are not supported

Circular cross-section generated as general cross-section (consists of two arches) cannot be imported.



e) **numerical cross-sections** – import of numerical cross-sections is not supported.

Table of chapter **Cross-sections** must contain following columns:

- **Name**
- **Type**
- Items from folder **Parameters**
  - Name
  - Length
- **Catalog ID**
- **CatalogItem**
- **Material**
- Items from folder **Fibre**
  - y
  - z
  - Item
- Items from folder **Elements**
  - Type
  - Points
  - Materials

### 12.3.4 Members 1D

Table of chapter **Members 1D** must contain following columns:

- **Name**
- **Type**
- **Table of geometry**
- **Cross-section**
- **LCS rotation**
- **Alpha**

For members with type of Rib following columns are required:

- **Alignment**
- **Shapeofrib**
- **Effectivewidth**
- Items from folder **Reference table**

Columns for effective width for check (the same columns twice, because first two columns are for width input by value, second for input by multiple of slab height)

- **forcheck**
- **widthright**
- **forcheck**
- **widthright**

If structure contains ribs, cross-sections with T shape are created during import according to height of slab and effective width. Note that checkbox Ribs has to be checked in table with internal forces on members, otherwise only forces in members without integration from effective width are exported to XML file.

Member of type Rib is limited to rectangular cross-section in import. Such cross-section is imported as T shape, L shape or X shape.

If first member in SCIA Engineer project is not type of Rib, it is not possible to add folder Reference table to Members 1D table. Because of that data about corresponding slab are not exported into XML file, it is not possible to get height of slab and substitutive cross-section cannot be created in IDEA Designer.

In such case warning is displayed during import and substitutive cross-section is created using height of first slab in XML file or with default height 200 mm

### 12.3.5 Haunches

Haunches on concrete members are imported.

Table of chapter **Haunches** must contain following columns:

- **Name**

- **Reference Table**

### 12.3.6 Arbitrary beams

Arbitrary concrete beams are imported.

Table of chapter **Arbitrary beams** must contain following columns:

- **Name**
- **Reference Table**

### 12.3.7 Openings in beams

Import of openings in beams is not supported. If some of imported members have assigned an opening, it is imported using its original cross-section, which was input SCIA Engineer. Warning is displayed during import.

Table of chapter **Openings** must contain following columns:

- **Name**
- **Reference Table**

### 12.3.8 Members 2D

Only 2D members without curved lines can be imported.

Table of chapter **Members 2D** must contain following columns:

- **Name**
- **Thickness**
- **Material**
- **Type**

### 12.3.9 Openings in members 2D

Table **2D members openings** must contain columns:

- **2D member**
- **Table of geometry**
- **Node**
- **Edge**

### 12.3.10 Load cases

Table **Load cases** must contain columns:

- **Name**
- **Action Type**
- **Load type**

### 12.3.11 Groups of load cases

Table of chapter **Load groups** must contain following columns

- **Name**
- **Relation**
- **Load**

### 12.3.12 Combinations

Exploded linear combinations are imported.

Table of chapter **Combinations** must contain columns:

- **Name**
- **Description**
- **Type**
- **Load cases**

### 12.3.13 Internal forces on 1D members

To ensure that internal forces are exported for all load cases, it is necessary to indent chapter Results on 1D members under chapter Load cases. Set properties of results chapter:

- Selection – set **All** to export all members
- Type of loads – set to Load cases
- System – must be set to LCS
- If structure contains ribs, option Ribs has to be checked to take into account slab forces integrated from effective width.
- Extreme – set to Section or No
- Sections – set to All

Table of chapter **Internal forces on members** must contain columns:

- **Case**
- **Member**
- **Dx**
- **N**
- **Vy**
- **Vz**
- **Mx**
- **My**
- **Mz**

## 12.4 Program Midas Civil

Following data are imported from files \*.MID and \*.MCT, which were exported from Midas Civil:

### 12.4.1 Materials

Following materials are imported:

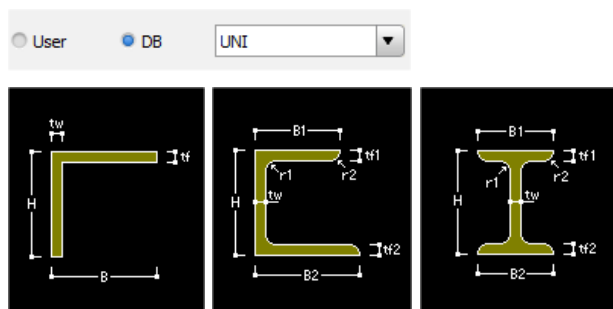
- Concrete EN: C12/15, C16/20, C20/25, C25/30, C30/37, C35/45, C40/50, C45/50, C50/60, C55/67, C60/75, C70/85, C80/95, C90/105
- Steel EN: S235, S275, S355, S420, S460

### 12.4.2 Steel cross-sections

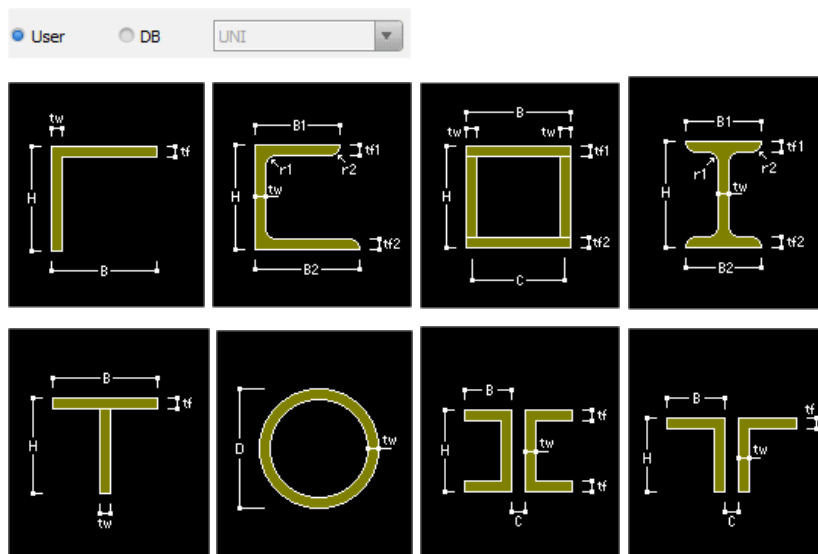
Following steel cross-sections are imported:

Tab **DB/USER**:

DB UNI- are imported as rolled sections. Cross-sections, which names corresponds with names of IDEA cross-section library, are imported. The conversion table file ConversionTables\UNI\_css.csv in the IDEA Designer installation directory contains the list of supported cross-section names.



USER – are imported as welded sections.

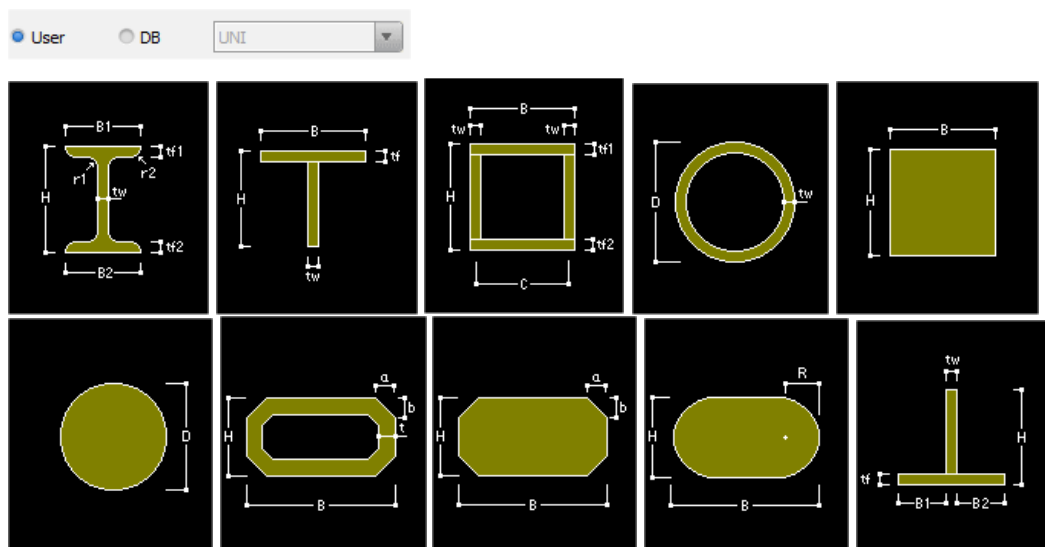


### 12.4.3 Concrete cross-sections

Following concrete cross-sections are imported:

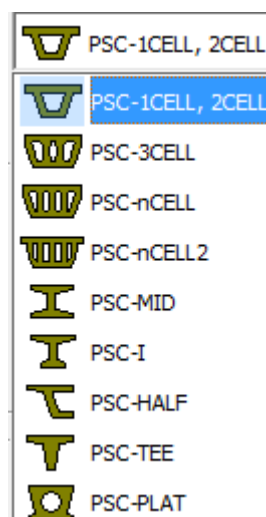
Tab **DB/USER**:

Depending on the shape, cross-sections DB/User are converted either to IDEA RCS sections or to the general sections.



Tab **PSC**:

All types of PSC sections are imported as general cross-section, which consist of the outline polygon and holes polygons.



### 12.4.4 Structural members

Following structural members are imported:

- 1D Members
  - Geometry – beginning and end node
  - Cross-sections – constant or variable
  - LCS orientation

- 2D members – are determined by the group of finite 2D elements with common outline and common properties:
  - Normal vector
  - Material
  - Thickness
- Prestressing tendons

#### 12.4.5 Internal forces

Following internal forces are imported:

- Internal forces on 2D members -  $m_x, m_y, m_{xy}, v_{xz}, v_{yz}, n_x, n_y, n_{xy}$  of all mesh elements.
- Internal forces on 1D members -  $N, V_y, V_z, M_x, M_y, M_z$  in all sections. Forces are imported to the principal axes of cross-section.

Internal forces of following load cases are imported::

- Static load case;
- Live loads – envelope combination;
- Settlements – envelope combination;
- Construction stages – individual actions are imported as increments to the previous step or stage. Thus the resulting action is a summation of all cases of the same type from previous steps and stages. Linear combinations are generated for this purpose.

#### 12.4.6 Combinations

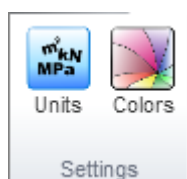
IDEA Designer imports all 3 basic types of combinations created in Midas Civil:

- **Add** – load is calculated for load case and coefficient. Result is summed according to the combination prescription respecting signs +/-.
- **Envelope** – selecting the type of load case determines the load group, which is respected in combination. Load groups are selected automatically when defining load cases. Each load case may have its own coefficient. Load group properties All/Add/One are respected when combining. Maximum and minimum for each effect is searched. Maximum may be less than zero, minimum may be greater than zero.
- **Combined types** – linear, envelope and code envelope combinations may contain other combinations in addition to the load cases. Each inserted combination has a coefficient. 4 variants are available:
  - $LIN (LIN0*c0 + LIN1*c1 + LIN2*c2)$  = linear combination of several other linear combinations. Effect of combinations are multiplied by corresponding coefficient and are summarized.  $max == min$ .
  - $LIN (LIN0*c0 + LIN1*c1 + ENV*c2)$  = linear combination of other linear and envelope combination. The result will have different  $max(LIN0*c0 + LIN1*c1 + ENV_{max}*c2)$  and  $min(LIN0*c0 + LIN1*c1 + ENV_{min}*c2)$ .
  - $ENV(LIN0*c0, LIN1*c1, LIN2*c2)$  = envelope combination of several other linear combinations. Each combination may have its coefficient. Properties of load groups are not respected when combining, all is taken as „All“. The result will have max and min.

- $ENV(LIN0*c0, LIN1*c1, ENV*c2)$  = envelope combination of linear and envelope combinations. Max and min is searched from all envelope and linear combinations.



## 13 Application setting



Units and colors can be set using commands in ribbon group **Settings** on ribbon tab **Home**.

### 13.1 Units setting

To set change units settings click **Units** in ribbon group **Settings** on the tab **Home**.

Unit type	Unit	Precision	Format
Length - Structure	m	2	D S A
Length - Cross section	mm	0	D S A
Angle	°	1	D S A
Force	kN	1	D S A
Moment	kNm	1	D S A
Stress	MPa	1	D S A
Temperature	°C	0	D S A
Time (long-term)	d	1	D S A
Coefficient	-	2	D S A
Relative Humidity	%	0	D S A
Time (short-term)	s	0	D S A

Magnitudes, for which the units can be set, are grouped into categories Main, Material and Results. The categories are displayed in the column on the left of the dialog. For the selected category the table of corresponding magnitudes is displayed. For each magnitude, which is listed in column **Unit type**, one of the available units can be set in the column **Unit**.

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

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

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

Current unit settings is not used automatically at next application startup, it must be stored into application configuration files clicking **Save**.

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

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

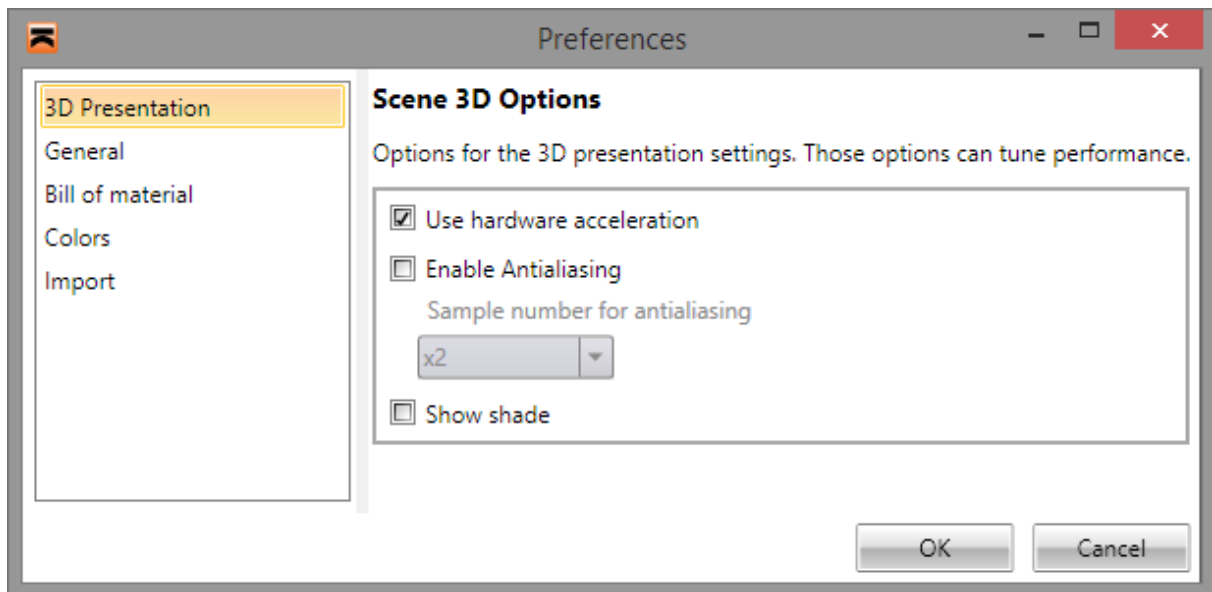
**Import** - reads the units configuration from a file.

**Export** - saves the current units settings to a file.

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

## 13.2 3D view settings

Click menu command **File > Preferences**, tab **3D presentation** to change the properties of 3D view.

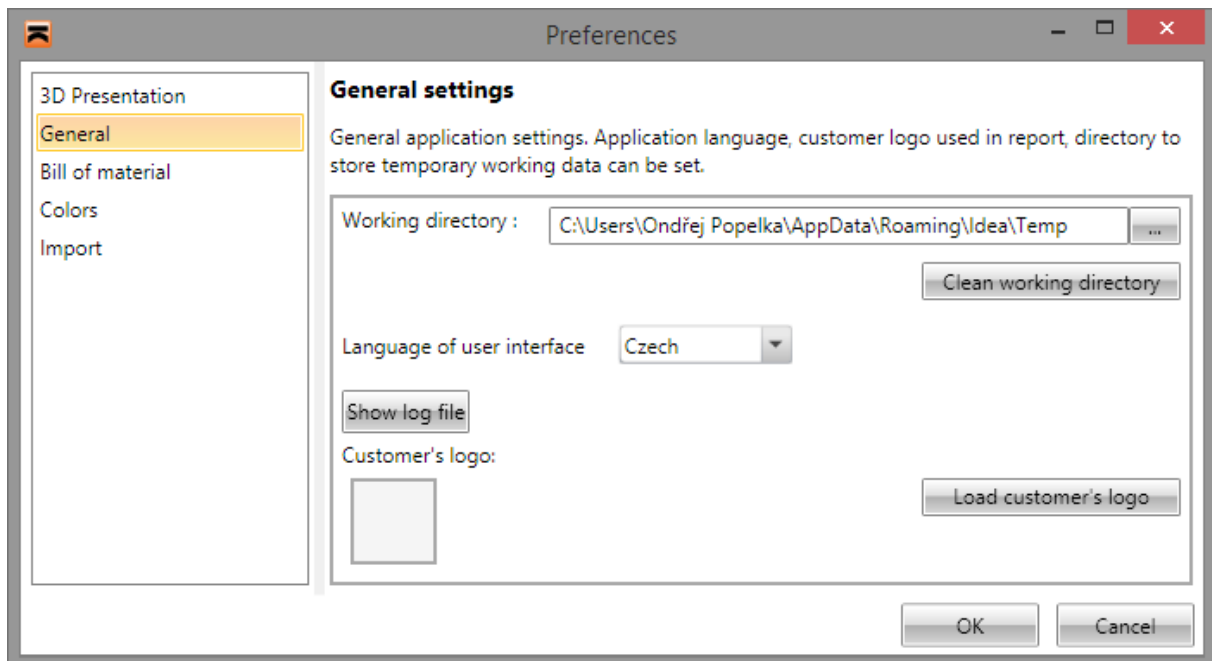


Tab **Scene 3D options**:

- **Use hardware acceleration** – switch on/off hardware acceleration of the graphics adapter. The default setting is on. The drawing speed is improved when proper graphics card is used.
- **Enable antialiasing** – switch on/off the algorithm to smooth the edges. The default setting is off. The drawing speed is decreased, if the antialiasing is on.
- **Show shades** – switch on/off the drawing of object shades. The default setting is off. The drawing speed is decreased, if the shades drawing is on.

### 13.3 General settings

Click menu command File > Preferences, tab General to set the temporary files directory, language of application or customer's logo.

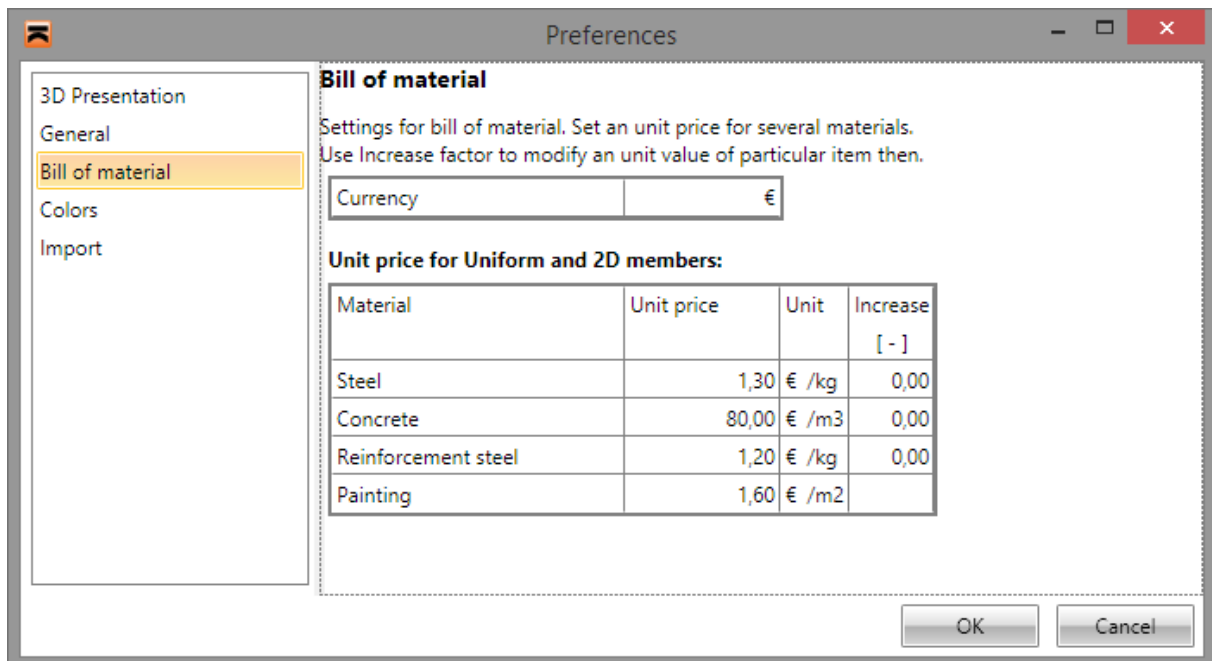


Tab General:

- **Working directory** – click ... to change temporary files directory. The change takes effect after next start of application.
- **Clean working directory** – delete the content of temporary directory.
- **Language of user interface** – select the language of application. The change takes effect after next start of application.
- **Load customer's logo** – select the customer's logo file. The logo is printed in the head of report pages.
- **Show log file** – display the log file of the application. The log file may contain records which can be helpful e.g. to analyse the application failure.

### 13.4 Bill of material

Click menu command **File > Preferences**, tab **Bill of material**, to set the currency and prices table for bill of material.



Tab **Bill of material**:

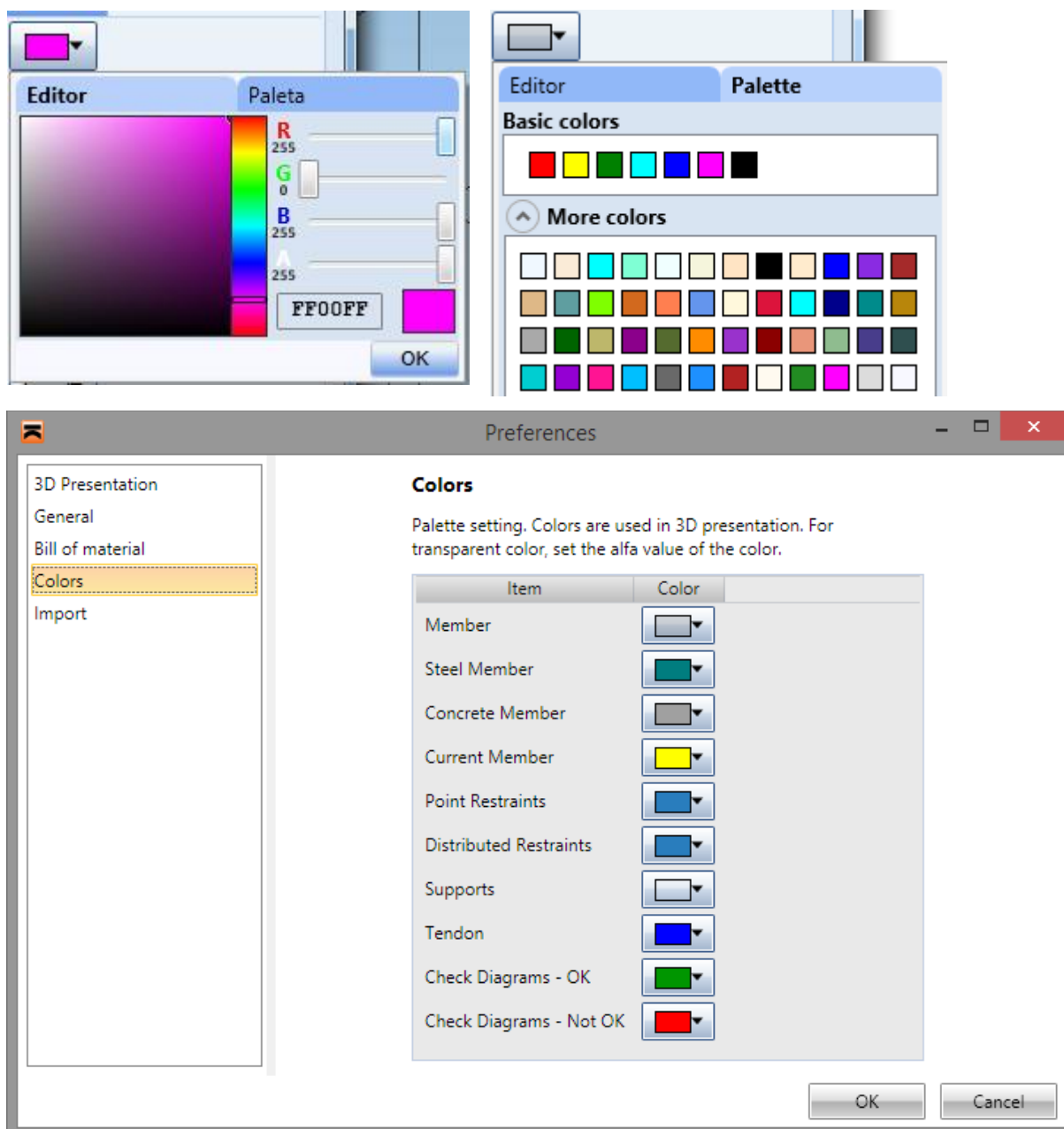
- **Currency** – input the text to describe the currency name.
- **Unit price** – input the price for the measurement unit of particular material.
- **Increase** – input the coefficient to calculate the increased weight. The increased weight is calculated as original weight \* (1+ increase coefficient).

## 13.5 Colours setting

Click menu command **File > Preferences**, tab **Colours** or **Colours** in ribbon group **Home** to set the colours for drawing of entities.

The colour can be set:

- On the tab **Editor** tab defining the colour channels R, G, B and opacity in channel A;
- On the tab **Palette** tab selecting the predefined colour.



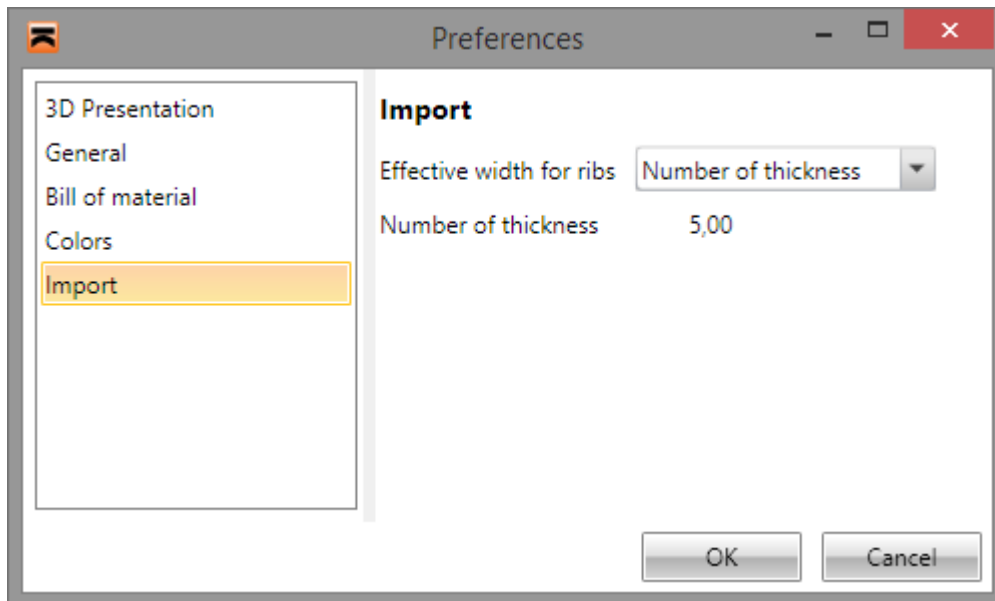
Tab **Colours**:

- **Member** – select the colour for drawing of general members in solids mode.
- **Steel member** – select the colour for drawing of steel members in solids mode.
- **Concrete member** – select the colour for drawing of concrete member in solids mode.
- **Current member** – select the colour for drawing of the currently selected member.
- **Point restraints** – select the colour for drawing of point LTB restraints.

- **Distributed restraints** – select the colour for drawing of distributed LTB restraints.
- **Supports** - select the colour for drawing of nodal supports.
- **Check diagram** – OK – select the colour for drawing of courses of passed checks.
- **Check diagram** – not OK – select the colour for drawing of courses of failed checks.

## 13.6 Import settings

Click menu command **File > Preferences**, tab **Import** to change the setting of conversion of slab ribs to equivalent cross-sections (T-shape, L-shape), when importing the structure from AxisVM.



Tab **Import**:

- **Effective width for ribs** – select the mode to specify the width of equivalent cross-section.
  - **Width** – the width of equivalent cross-section is identical for all ribs and is equal to the defined value **Width**.
  - **Number of thicknesses** – the width of equivalent cross-section is calculated for each rib as a multiple of the appropriate slab thickness and the defined value **Number of thicknesses**.