IDEA Frame 8

User guide

Content

1.1 Program requirements 7
1.2 Installation guidelines 7
2 Basic Terms 8
3 User interface 9
3.1 Control of view in main window 9
3.1.1 DXF export settings 10
3.2 Units setting 11
4 Working with project 12
4.1 Structure drawing setting 12
4.1.1 Ribbon group View settings 12
4.1.2 Ribbon group CSS drawing 12
5 Input of the structure 13
5.1 Project data 13
5.2 Input of cross-sections 15
5.3 Materials 17
5.4 Geometry 19
5.4.1 Nodes 19
5.4.2 Members 21
5.4.3 Haunches 22
5.5 Blocks 24
5.5.1 Grid 24
5.5.2 Portal frame 25
5.5.3 Block from DXF 27
5.6 Loads 29
5.6.1 Groups of load cases 29
5.6.2 Load cases 33
5.6.3 Nodal loads 34
5.6.4 Point forces 35
5.6.5 Point moments 37
5.6.6 Uniform loads 38
5.6.7 Line load 40
5.6.8 User defined internal forces 42
5.6.9 Combinations 47
5.6.10 Manager of load cases combinations 49
6 Results 51
6.1 Results evaluation setup 51

6.2 Reactions in supports 52
6.2.1 Ribbon group Results 52
6.2.2 Ribbon group System of reactions 52
6.2.3 Ribbon group Extreme 53
6.3 Deformations 54
6.3.1 Ribbon group Results 54
6.3.2 Ribbon group Selection 54
6.3.3 Ribbon group Extreme 55
6.3.4 Ribbon group Deformations 55
6.4 Internal forces 56
6.4.1 Ribbon group Results 56
6.4.2 Ribbon group Selection 56
6.4.3 Ribbon group Extreme 56
6.4.4 Ribbon group Transformations 57
6.4.5 Ribbon group Internal forces 57
6.4.6 Ribbon group Fatigue evaluation 57
7 Design and check of structural items 58
7.1 Design of reinforced concrete members 58
7.2 Check of steel members 58
7.3 Prestressed design members 59
8 Design members and design groups 60
8.1 Design members 60
8.2 Design groups 62
9 Design of concrete members 63
9.1 Check of the design group 63
9.2 Settings for section checks and calculation of deflections 64
9.2.1 Code and calculation settings 64
9.2.2 Setting of result class for calculation of deflections 65
9.2.3 Setting of result classes for section checks 65
9.2.4 Editing result class 66
9.3 Design member data 68
9.4 Reinforcement zones 70
9.4.1 Check positions for check of pre-stressed members 71
9.4.2 Zone templates 73
9.4.3 Ribbon group View settings and scale 73
9.4.4 Ribbon group Detailed view 74
9.4.5 Ribbon group Internal forces 74

9.5 Editor of reinforcement 75
9.5.1 Editing cover 76
9.5.2 Input of reinforcement by template 77
9.5.3 Input of one-way slabs reinforcement by a template 79
9.5.4 Shear reinforcement 80
9.5.5 Longitudinal reinforcement 89
9.5.6 User settings of reinforced cross-section 97
9.5.7 Deleting reinforcement 99
9.5.8 Import and export of reinforced cross-section 99
9.5.9 View settings of reinforced cross-section 99
9.5.10 User defined reinforcement templates 100
9.6 Data for buckling effects calculation and deflection check 104
9.6.1 Data for deflection check of beam 104
9.6.2 Data for buckling effects calculation and deflection check for one span columns 106
9.6.1 Data for buckling effects calculation and deflection check for multi-span columns 109
9.7 Reduction and redistribution of internal forces 111
9.7.1 Supports definition for calculation of redistributions and reductions 111
9.7.2 Ribbon group Internal forces 112
9.8 Detailed check 113
9.9 Results evaluation 114
9.9.1 Ribbon group Concrete design 114
9.9.2 Ribbon group View settings and scale 114
9.9.3 Ribbon group Extreme 114
9.9.4 Ribbon group Calculation 114
9.9.5 Ribbon group Results drawing 115
9.9.6 Ribbon group Stage 115
9.9.7 Drawing of section check results courses 115
9.9.8 Drawing of interaction diagrams 116
9.9.9 Drawing of deflections check results 117
9.9.10 Check report 118
10 Bridge load rating of concrete members 122
10.1 Input data of bridge load rating 122
10.1.1 Check settings 122
10.1.2 Check positions 123
10.1.3 Load cases and combinations for bridge load rating 125
10.1.4 Ribbon group View settings and scale 132

10.2 Results of bridge load rating calculation 133
10.2.1 Ribbon group View settings and scale 133
10.2.2 Ribbon group Extreme 133
10.2.3 Ribbon group LR type 133
10.2.4 Ribbon group Check 133
11 Steel members design 134
11.1 Default check settings 134
11.2 Check settings for the current design group 138
11.3 Design data 141
11.3.1 Ribbon group Project setup 141
11.3.2 Point LTB restraint 142
11.3.3 Distributed LTB restraint 142
11.3.4 Not checked field 143
11.3.5 Ribbon group LTB restraints 143
11.3.6 Ribbon group Check data 143
11.3.7 Ribbon group View settings 143
11.4 Buckling lengths 145
11.4.1 Mass input of buckling lengths coefficients 147
11.4.2 Ribbon group 3D View 147
11.4.3 Ribbon group Structure 148
11.4.4 Ribbon group Dimensions drawing 148
11.5 Check results evaluation 149
11.5.1 Ribbon group Evaluation mode 149
11.5.2 Ribbon group Design member 149
11.5.3 Ribbon group Steel design 149
11.5.4 Ribbon group Extremes 149
11.5.5 Ribbon group Type of check 150
11.5.6 Ribbon group Type of output 150
12 Design summary, optimisation of steel cross-sections 151
12.1 Evaluation on design members 151
12.1.1 Optimisation of steel cross-sections 152
12.1.2 Ribbon group Concrete design 153
12.1.3 Ribbon group Steel design 153
12.1.4 Ribbon group Recalculate 153
12.1.5 Ribbon group Check results evaluation 153
12.1.6 Ribbon group Material 153
12.2 Bill of material 154

13 Report 155
13.1 Standard report 155
13.1.1 Input data 155
13.1.2 Calculation results 155
13.1.3 Bridge load rating results 156
13.1.4 Concrete 1D design results 156
13.1.5 Steel design results 156
13.2 Detailed report 157
13.2.1 Input data 157
13.2.2 Calculation results 158
13.2.3 Concrete 1D design results 159
13.2.4 Bridge load rating results 161
13.2.5 Steel design results 162
13.3 Ribbon group Report view 163

1.1 Program requirements

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

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

1.2 Installation guidelines

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

2 Basic Terms

IDEA Frame is a simple program for the analysis and design of plane frames and trusses. IDEA Frame is one from the group of programs developed especially for 2D FEA structural analysis. All these programs work with the same data model. This allows their direct connection with all IDEA design modules.

A frame or truss of any shape can be defined in IDEA Frame program. Frame can be supported in points. Loads are sorted into load cases. There can be nodal or point and line impulses on members. Analysis of internal forces and deformations is done by finite element method. Program offers internal forces N, V M and deformations ux, uz, fiy. Load cases can be combined.

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

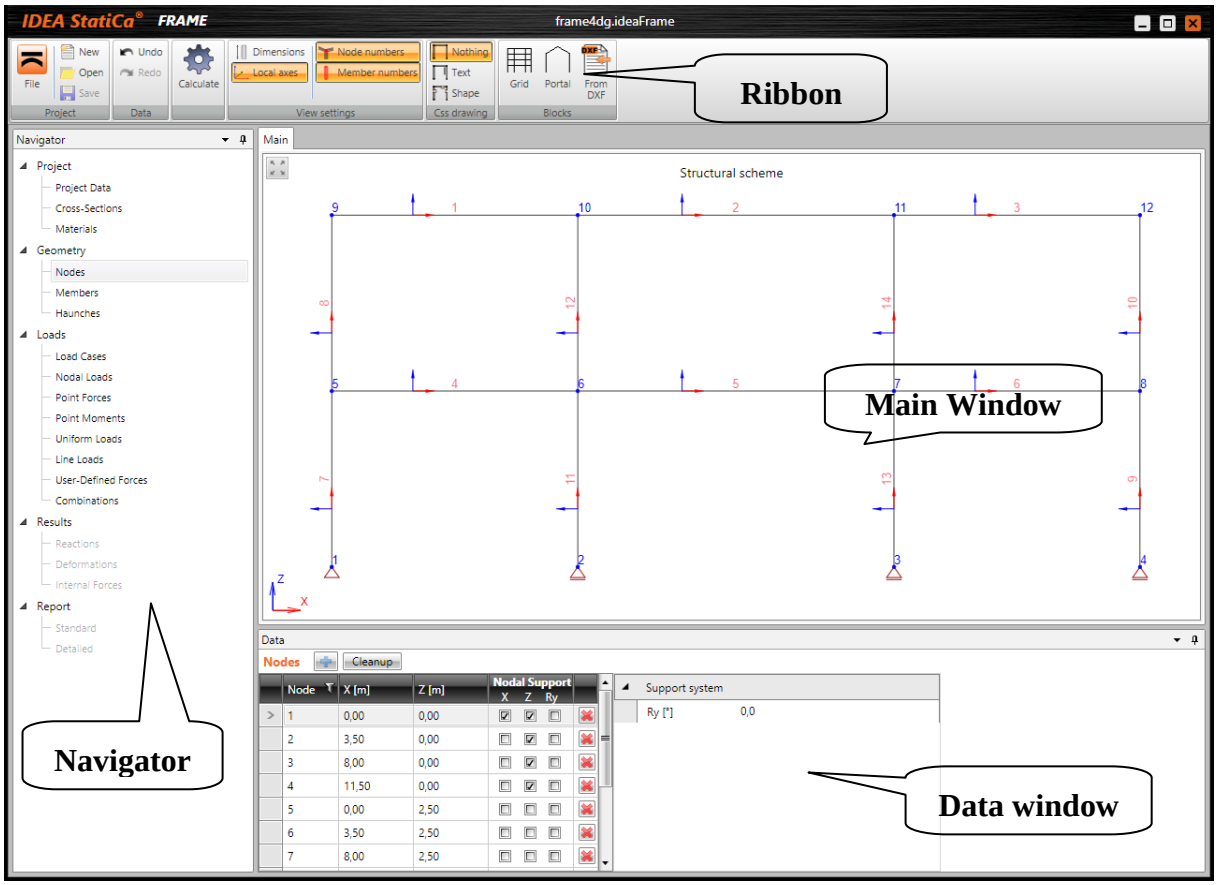
- IDEA RCS where the required reinforcement can be designed
- IDEA Steel where the steel members can be designed
- IDEA Tendon where prestressed members can be designed.

3 User interface

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

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

All other IDEA applications have the similar design.



3.1 Control of view in main window

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

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

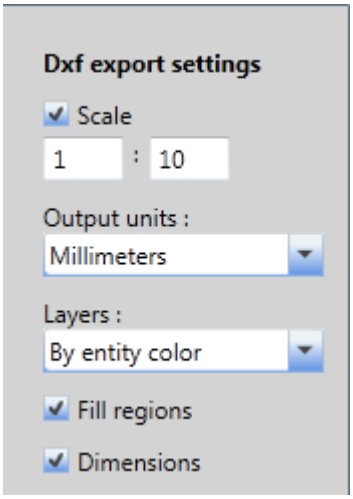
To set the required view using keyboard and mouse following combinations can be used:

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

Click 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

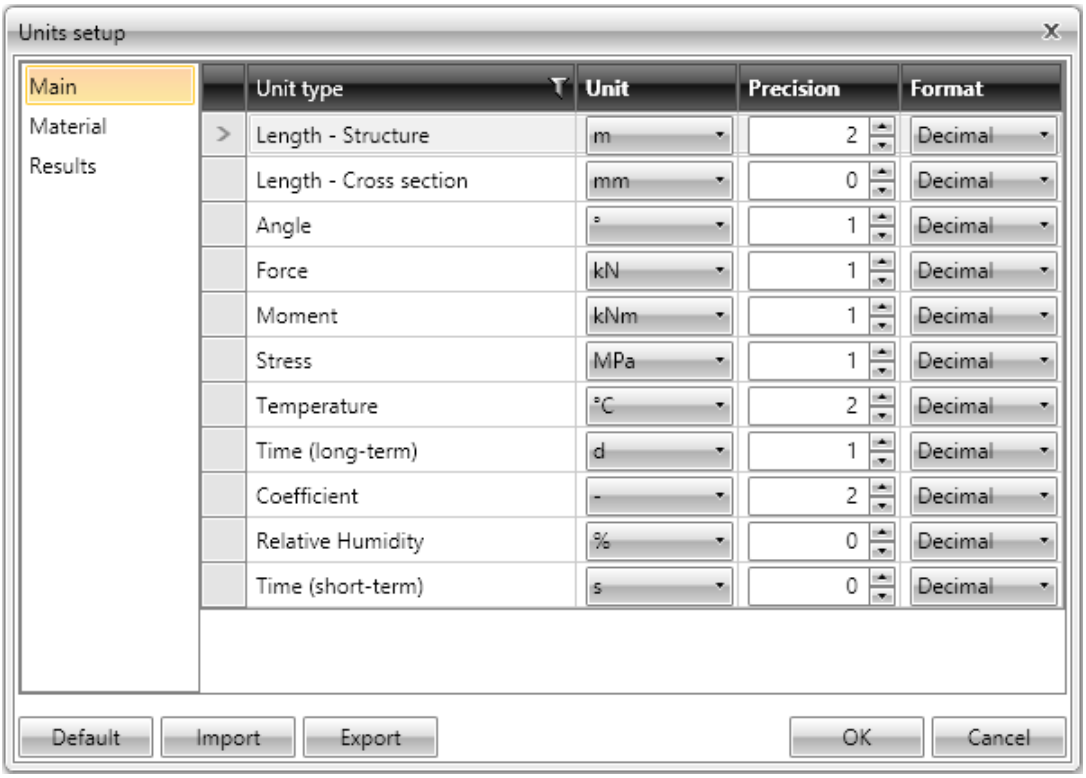


Following export parameters can be set in the Save as dialog when exporting the view to the 2D file:

- **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.

3.2 Units setting

To change units settings click command **Units** in menu **File**. The setting of units is valid for the instance (current run of the software) of the application. Settings are not automatically saved with the project.



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

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

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

- **Decimal** – display numbers in standard decimal format (“-ddd.ddd…”).
- **Scientific** – display numbers in exponential format ("-d.ddd…E+ddd").
- **Automatic** – according to length of resulting string it is automatically chosen whether to use decimal or exponential format. In this mode value specified in **Precision** column means number of significant digits in the resulting string.
- **Imperial** – display numbers in fractional format (only for imperial unit types).

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

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

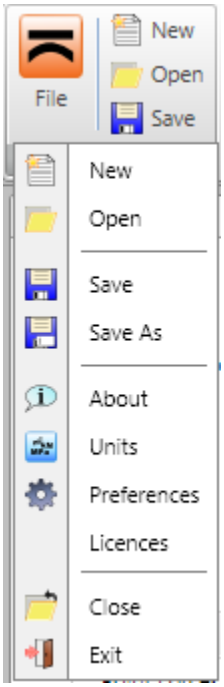
Import - reads the units configuration from a file.

Export - saves the current units settings to a file.

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

4 Working with project

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

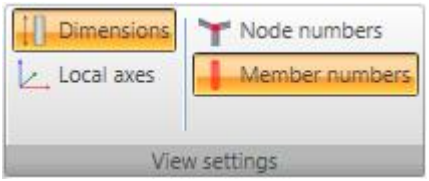


- **New** – create a new project.
- **Open** – open an existing project (files with extension *.ideaFrame or *.wsFrame).
- **Save** – save the current project into the data file.
- **Save as** – save the current project into the data file using a new file name.
- **About** – open the About application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

4.1 Structure drawing setting

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

4.1.1 Ribbon group View settings



Commands of ribbon group **View settings**:

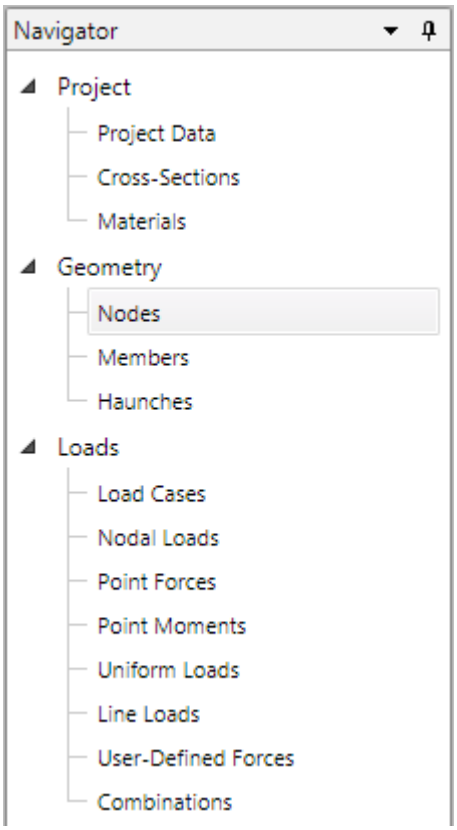
- **Dimensions** – switch on/off drawing of overall dimension lines in the picture of the structure.
- **Local Axis** – switch on/off drawing of local coordinate systems on members
- **Node numbers –** switch on/off the drawing of node numbers in the picture of structure
- **Member numbers -** switch on/off the drawing of member numbers in the picture of structure

4.1.2 Ribbon group CSS drawing

There are 3 modes of displaying cross-sections in the drawing of the structure:

- **Nothing** – cross-sections are not displayed
- **Text** – labels of cross-sections are displayed in the picture of the structure
- **Shape** – shapes of cross-sections are displayed in the picture of the structure

5 Input of the structure

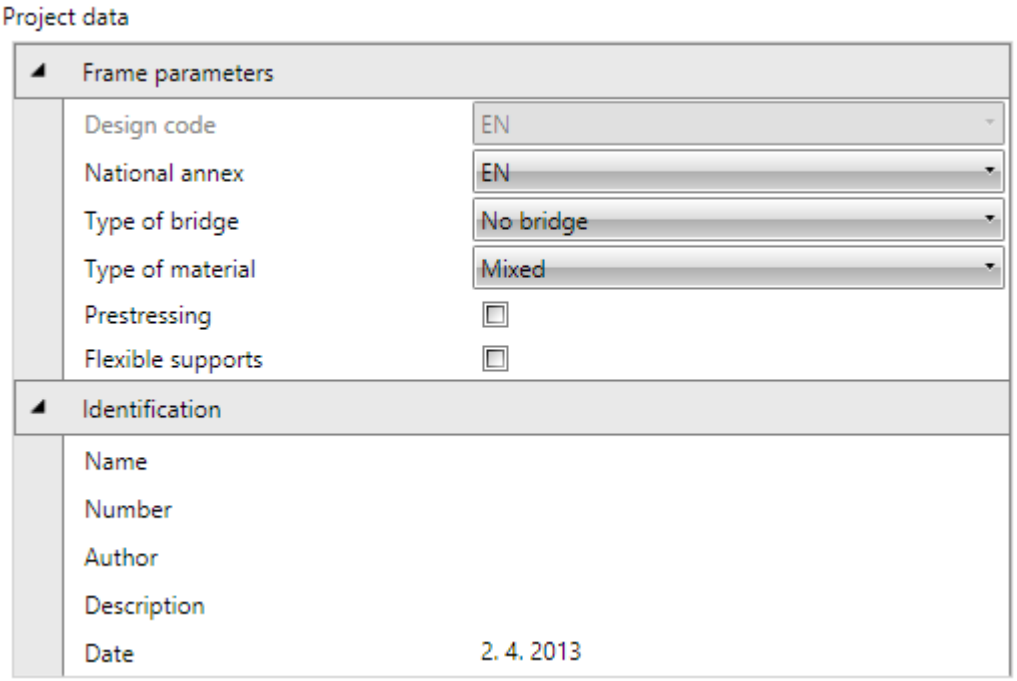


Appropriate structural data can be input by specific navigator commands. Input of data is sorted into following groups of commands: **Project**, **Geometry** and **Loads**. User simply goes through navigator from top to bottom.

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

5.1 Project data

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

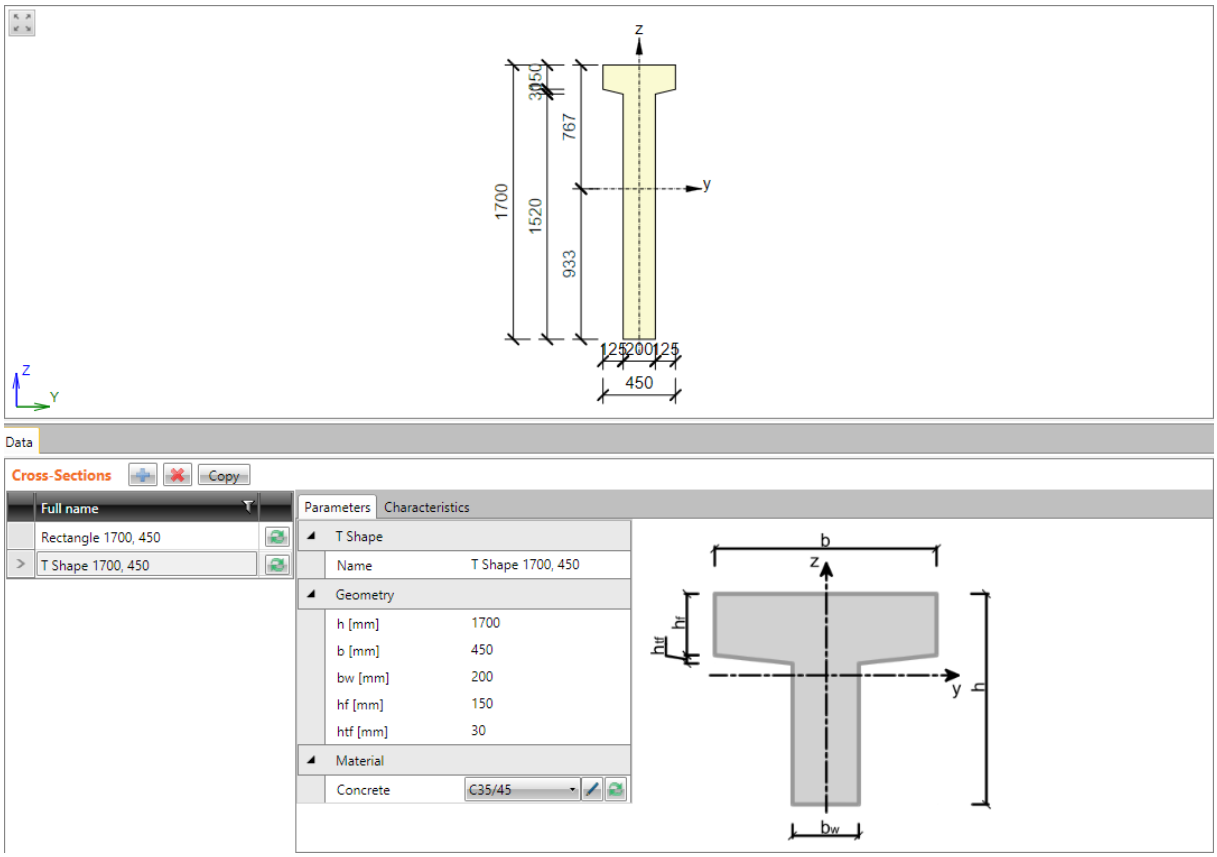


- **Design code** – set the national code of the project. Once the first cross-section is defined, the design code cannot be changed.

- **National annex** – set the national annex for concrete members designed according to EC code.
- **Type of bridge** – select the type of bridge for generation of variable load cases groups.
- **Type of material** – user can set if the structure is from concrete, steel or both materials. This settings influences available materials and cross-sections.
- **Prestressing** – if this option is checked, tendons can be designed and checked in module IDEA Tendon.
- **Flexible supports** – if the option is selected, translational and rotational stiffness of supports can be defined.
- **Name** – input of the project name.
- **Number** – input of the project identification number.
- **Author** – input name of the project author name.
- **Description** – input of additional information about the structure.
- **Date** – date of calculation.

5.2 Input of cross-sections

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



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

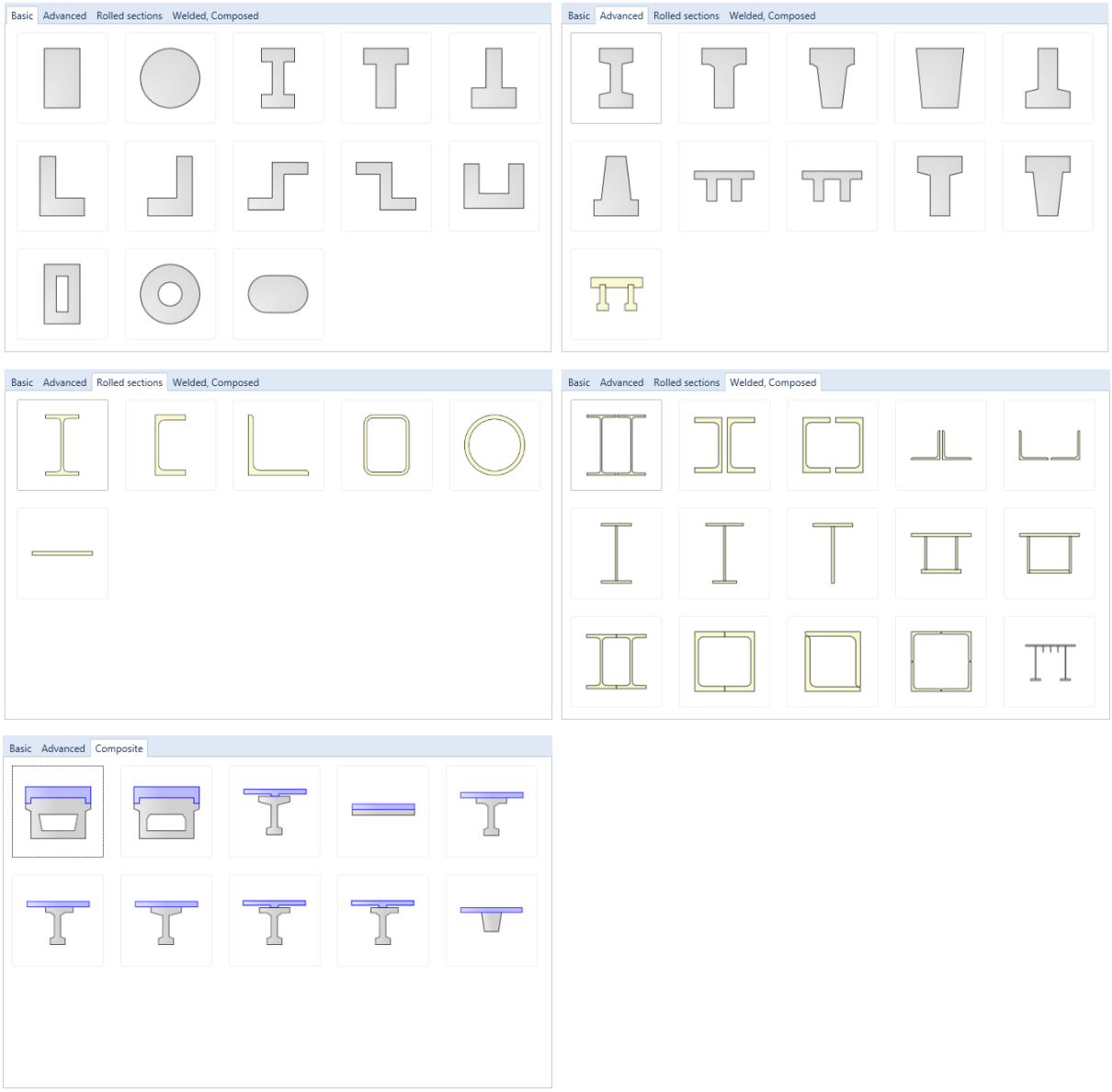
Click  above the table to delete the current cross-section. Cross-section, which is used anywhere in the structure, cannot be deleted.

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

Dialog **Cross-Section Navigator** appears with 4 groups of available cross-sections:

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

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

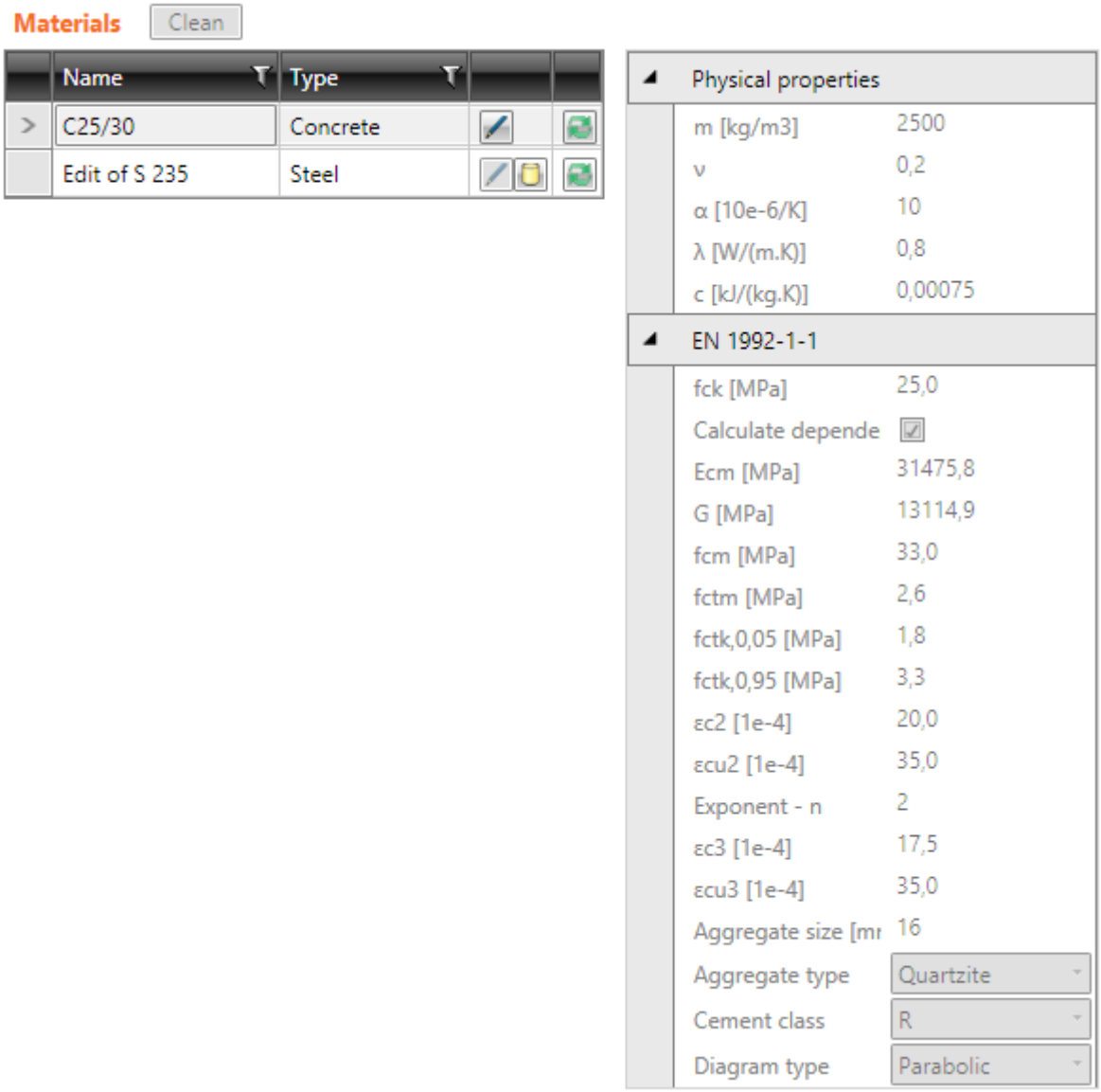


There is a list of already defined cross-sections in the left part of data window. Following buttons are available for each cross-section:

-  - input a new cross-section. Current cross-section is replaced by a new one.
- Properties of current cross-section can be edited on the tab **Parameters** in the right part of data table. Table of cross-sectional characteristics is displayed on the tab **Characteristics**.

5.3 Materials

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



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

- - click the edit button to convert the material to the editable material. The name of material changes and the particular material characteristics can be edited. The change affects all cross-sections, which have assigned the edited material.
- - click the button to display the dialog with the list of all materials available in the system material library. When a material from the library is selected, it replaces the edited material. The change affects all cross-sections, which have assigned the edited material.
- - save current (modified) material to the selected or new materials table into the users database.

- **Clean** – the button is available, if there is a material in the project, which is not assigned to any cross-section. Click the button to delete not used materials from the project.

5.4 Geometry

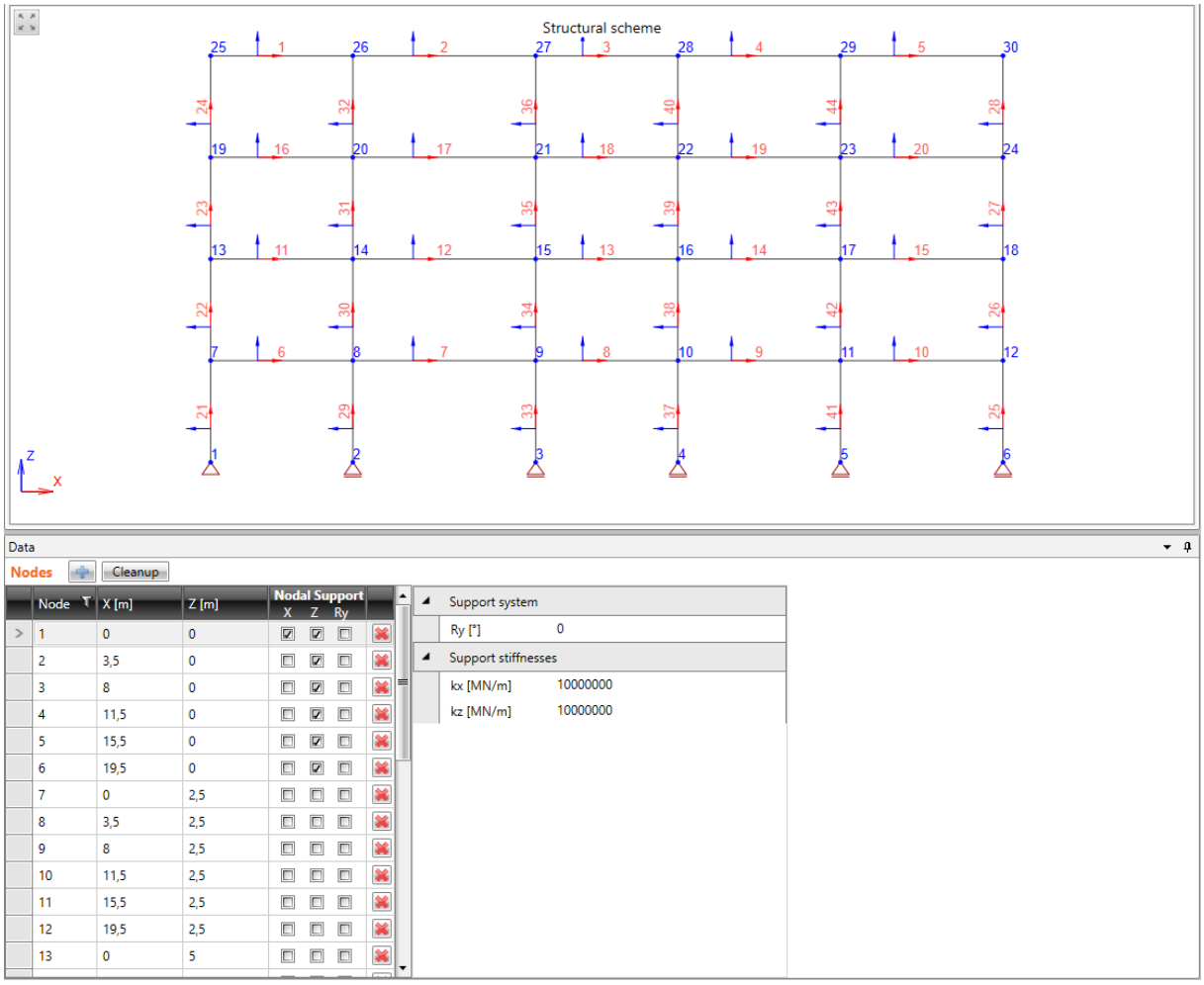
Particular commands of navigator group **Geometry** are used for input of frame shape and for supports.

Geometry of frame can be also defined by blocks – grid of nodes or portal frames.

5.4.1 Nodes

Click navigator command **Geometry > Nodes** to input nodes.

Nodes can be edited in the table in the Data window. Each node represents one line of a table. Numbers of nodes are generated automatically and cannot be changed.



Click  above the table to add new node.

Click **Cleanup** to delete all free nodes (not connected to any member) from the structure.

Individual columns in the table **Nodes**:

- **Node** –index of node is displayed.
- **X[m],**
- **Z[m]** – input the coordinate of node in the direction of appropriate global axe.
- **X** – switch on/off the support resisting translations in the direction of global X axis.
- **Z** – switch on/off the support resisting translations in the direction of global Z axis.
- **Ry** – switch on/off the support resisting rotations about the global Y axis.

-  - delete the appropriate node.

Properties group **Support system** – if there is a support in the current node, the rotation of support can be defined:

- **Ry** – input rotation angle about Y-axis of global coordinate system.

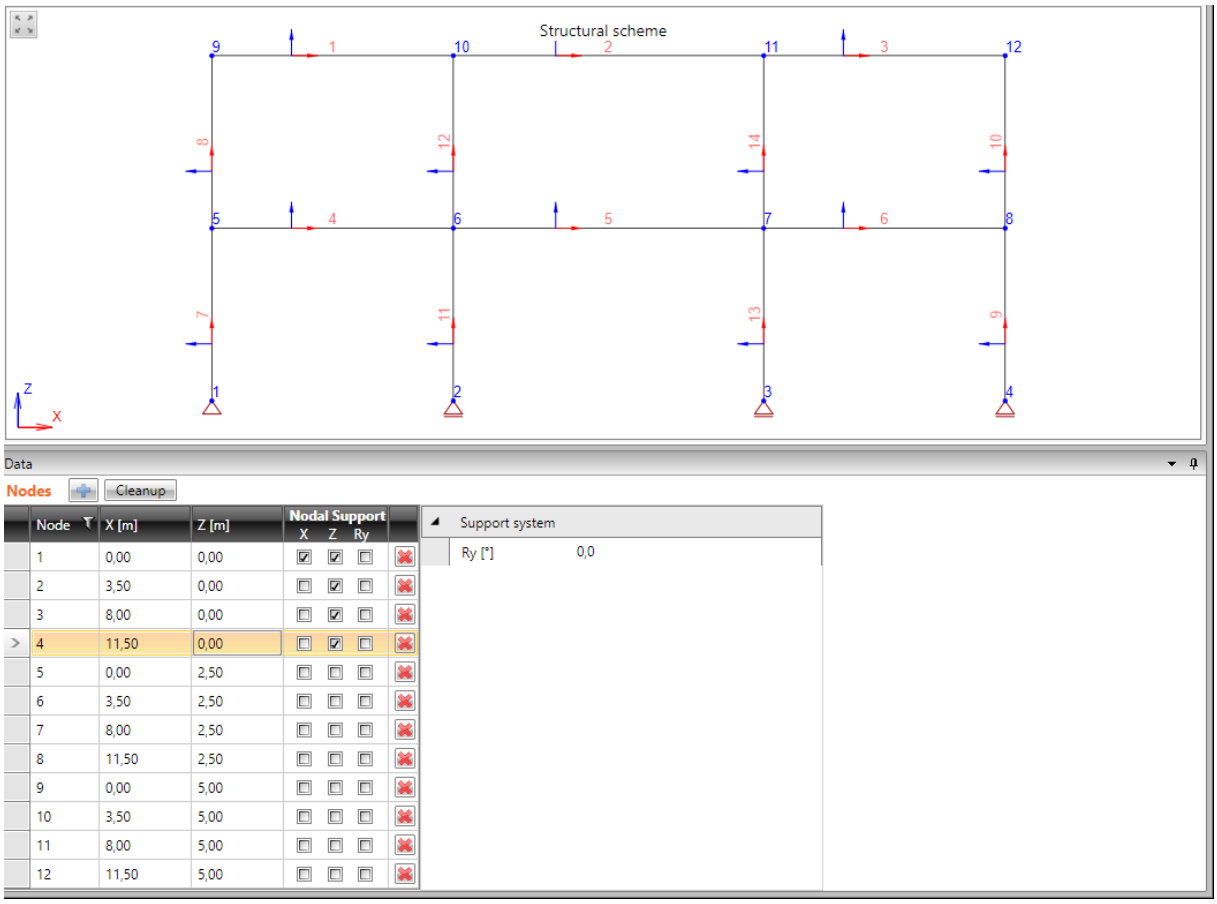
Properties group **Support stiffnesses** – definition of stiffnesses of supports. For flexible supports, the value of stiffness 1e7 MN/m, or MN/rad means the rigid support in the appropriate direction.

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

5.4.2 Members

Click navigator command **Geometry > Members** to input members.

Members are defined by end nodes. Each member has assigned a cross-section.



Click  above table to add a new member.

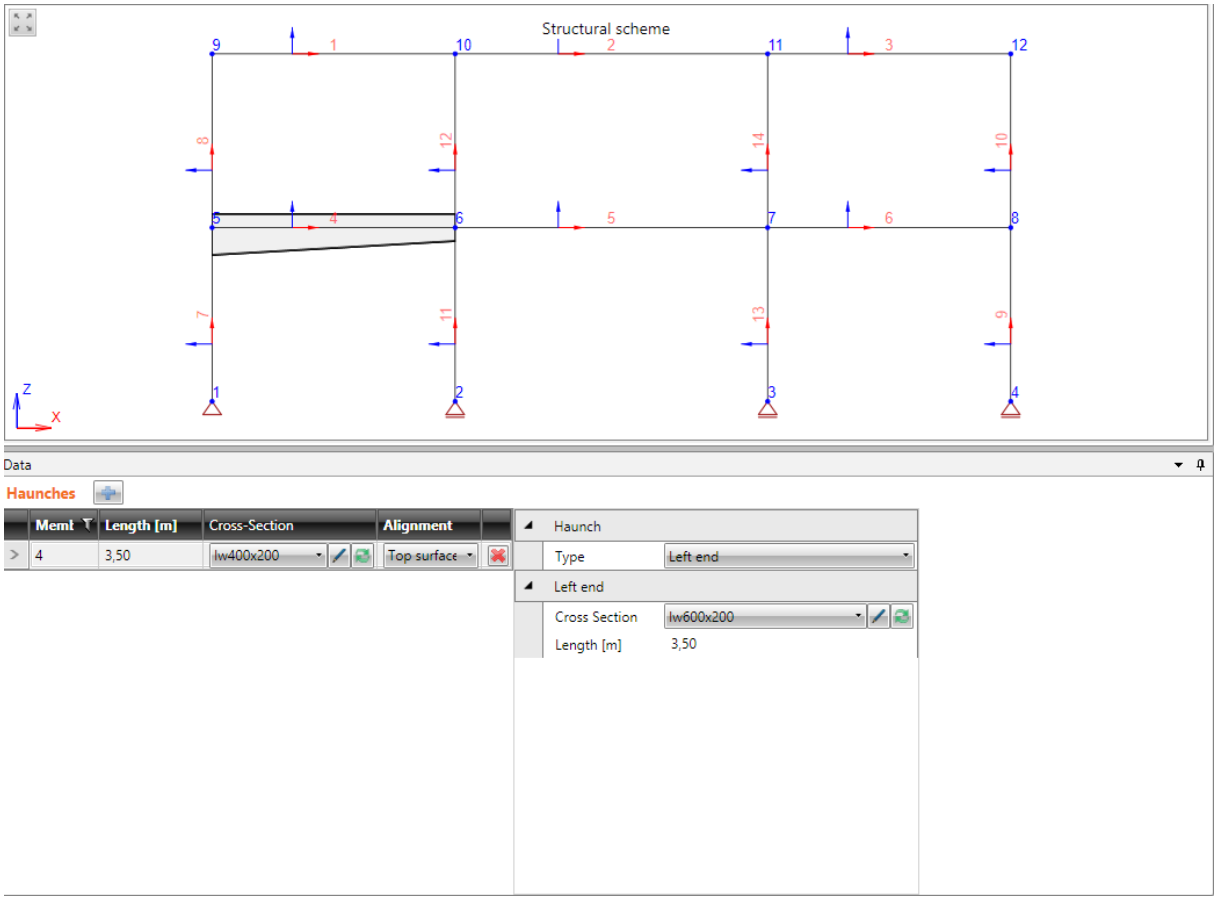
Table of members contains following columns:

- **Begin Node** – input the number of the begin node of the member.
- **End Node** – input the number of the end node of the member.
- **Cross-section** – assign the appropriate cross-section to the member. Cross-section can be selected in a list of all available cross-sections. Click  to change cross-section parameters of the member. Click  to add new cross-section to project. The new cross-section is assigned to the appropriate member
- **Length** – display the length of a member.
- **Begin Hinge** – switch on/off a My-hinge at the beginning of member.
- **End Hinge** – switch on/off a My-hinge at the end of member.
-  - delete the appropriate member.

5.4.3 Haunches

A haunch can be defined on any particular member of the frame, but the member must have a cross-section, which can create haunch. The haunch can be defined on the beginning or on the end or on both ends of the member. The haunch is defined by length and by two cross-sections – by the original cross-section of the member and by the cross-section assigned to the haunch on the appropriate member end.

Click navigator command **Geometry > Haunches** to input and edit haunches on members.



There is a list of members in the left part of the table **Haunches**. There is a table with haunch properties of the current member in the right part of the table.

Click  above the table to add a new haunch.

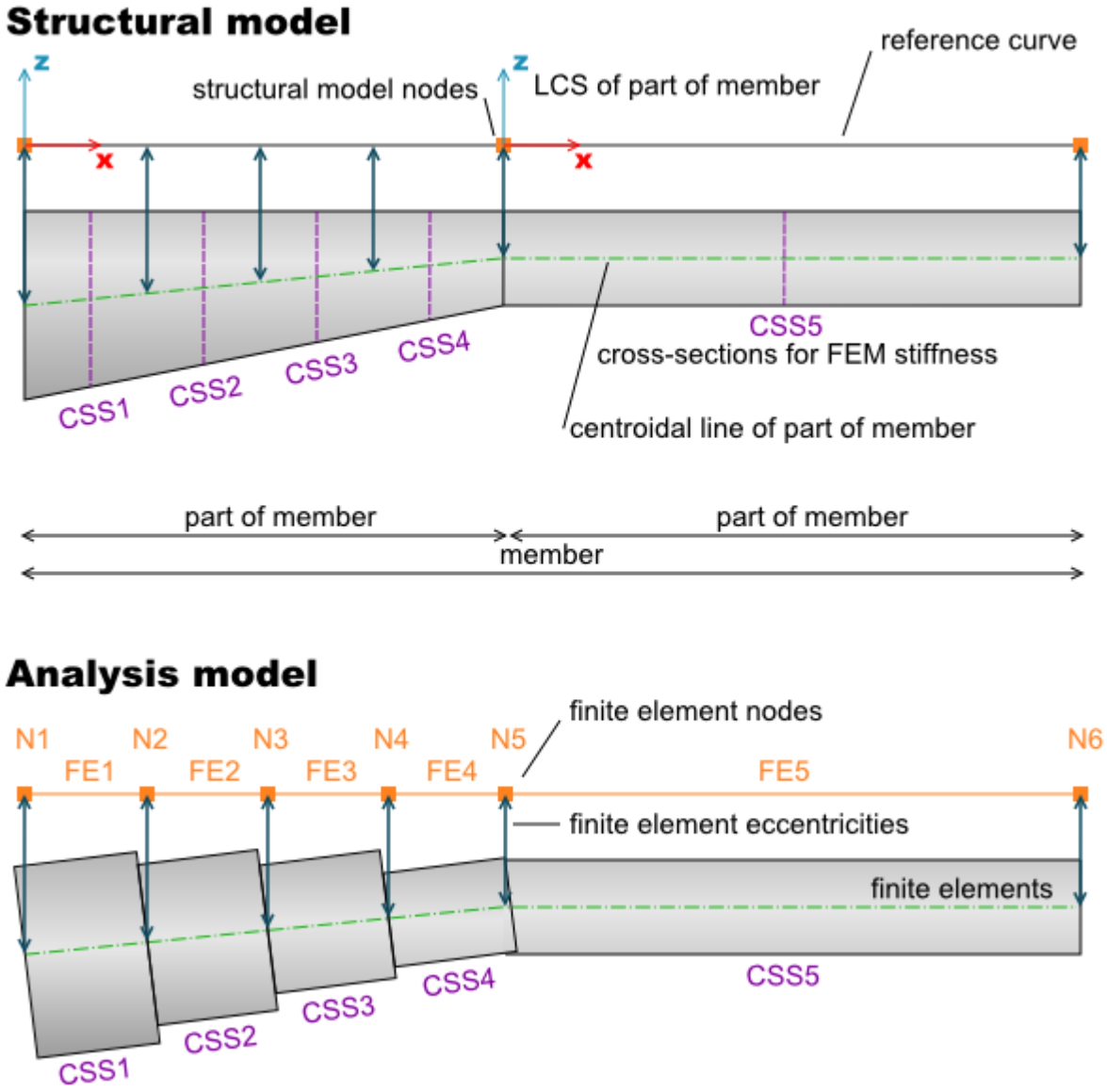
Columns in **Haunches** table:

- **Member** – input the number of member, on which the haunch is defined.
- **Length** – display the length of a member.
- **Cross-section** – assign the appropriate cross-section to the member. Cross-section can be selected in a list of all available cross-sections. Click  to change cross-section parameters of the member. Click  to add new cross-section to project. The new cross-section is assigned to the appropriate member
- **Alignment** – set the alignment of the haunch related to the original cross-section of the member. The haunch can be aligned either to the top surface, or to the centre line or to the bottom edge of the cross-section.

Properties of the haunch are defined in the right part of the table:

- **Type** – select the position of the haunch on member.
  - **None** – there is no haunch defined on the member
  - **Symmetrical** – haunches are defined on both ends of member and both haunches have the same properties
  - **Left end** – haunch is defined on the begin of the member
  - **Right end** – haunch is defined on the end of the member
  - **Both ends** – haunches are defined on both ends of the member and both haunches can have different properties.
- **Left (Right) end** – haunch properties on the appropriate end of member
  - **Cross-Section** – select the cross-section of the haunch in the list. The list contains only cross-sections which can create haunch on the current member. Click edit button to change properties of the haunch cross-section.

    Click button to input new cross-section of haunch. This cross-section is of the same type as the cross-section of the current member.



Picture 5-1 – transformation of the structural model to the analysis model (finite elements for the analysis)

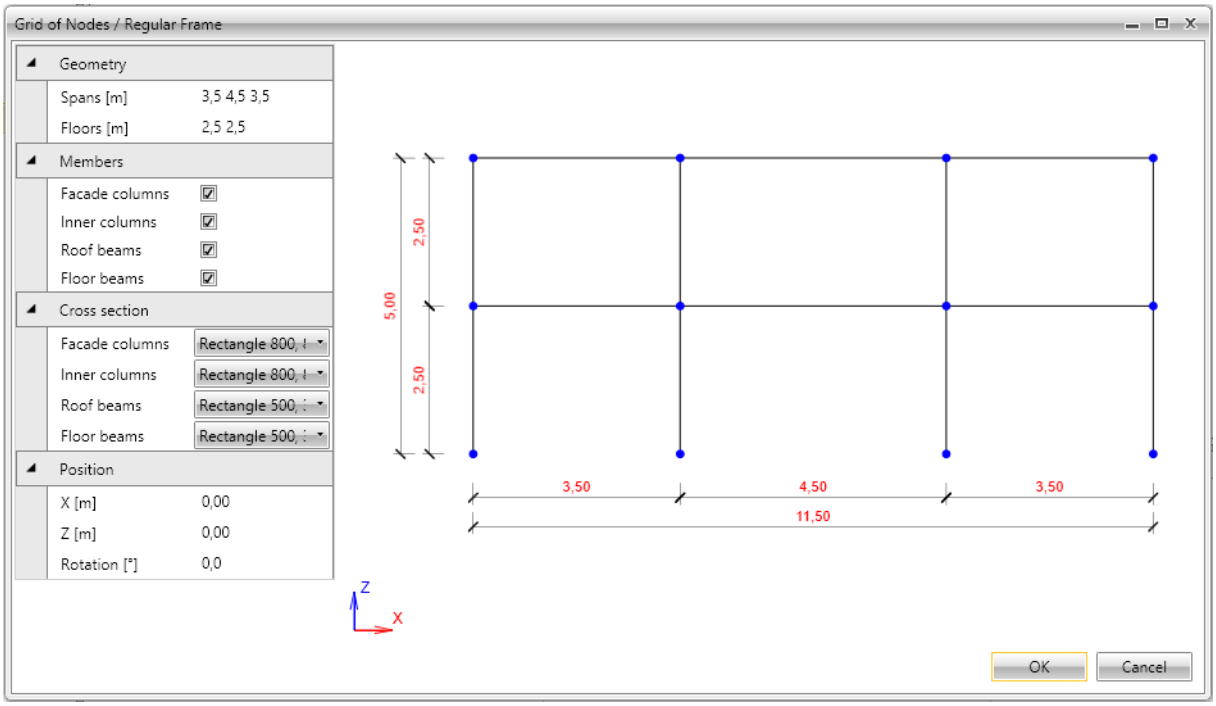
5.5 Blocks

Blocks can be used for faster input of typical structures. The selection of blocks will be enlarged in future. The shape of a block is defined by several parameters and then the block is located into required position in the structure.

To input a block click appropriate button in ribbon group **Block**.

5.5.1 Grid

Block **Grid** is used for the input of regularly located nodes. Also beams and columns can be generated.



Individual inputs of dialog **Grid of Nodes**:

Group **Geometry**:

- **Spans** – input of a list of spans (distance between neighbor columns). Individual values of span are divided by spaces.
- **Floors** – input of a list of floor heights (horizontal beams). Individual values of heights are divided by spaces.

Group **Members**:

- **Facade columns** – edge columns are generated if this option is checked.
- **Inner columns** – inner columns are generated if this option is checked.
- **Roof beams** – beams in roof level ere generated if this option is checked.
- **Floor beams** – beams in all floor levels are generated if this option is checked.

Group **Cross-section**:

- **Facade columns** – select cross-section for all facade columns. Cross-section can be selected from a list of all available cross-sections.
- **Inner columns** – select cross-section for all inner columns. Cross-section can be selected from a list of all available cross-sections.

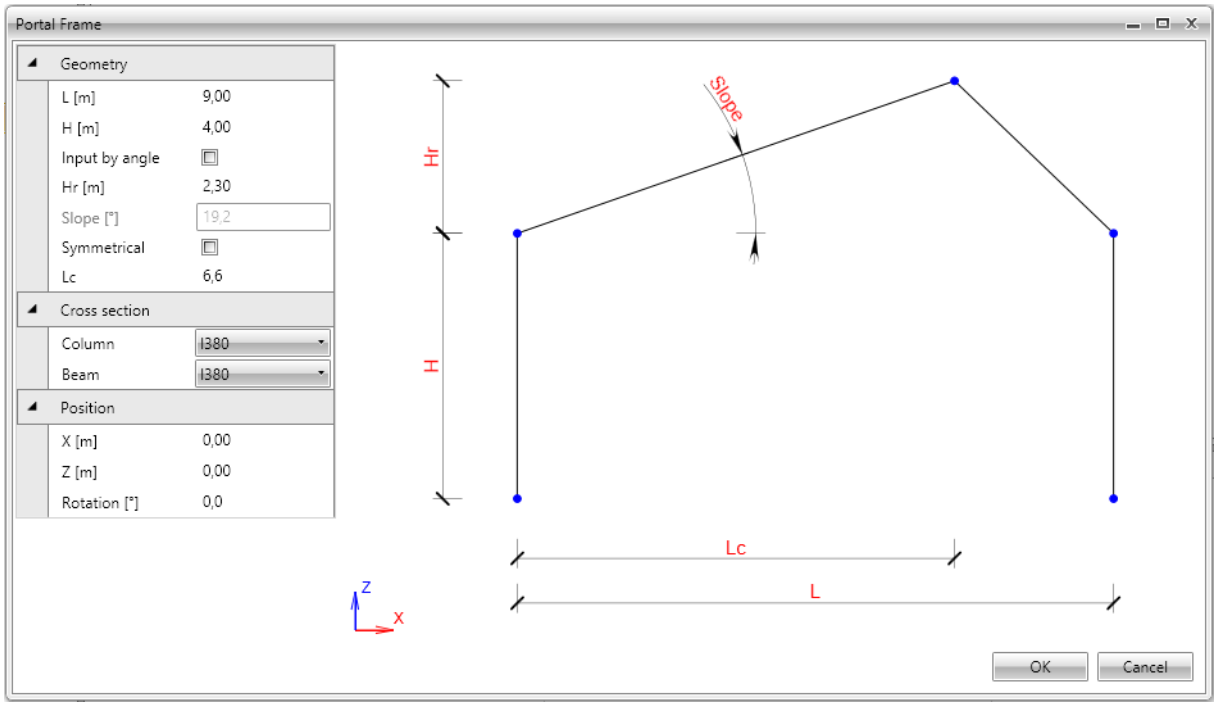
- **Roof beams** – select cross-section for all roof beams. Cross-section can be selected from a list of all available cross-sections.
- **Floor beams** – select cross-section for all floor beams. Cross-section can be selected from a list of all available cross-sections.

Group **Position** – definition of position of block inserted into the structure:

- **X**, **Y** – input of insert point coordinates relative to the origin [0;0] of the structure.
- **Rotation** – input of rotation value of the block around the axis perpendicular to the plane of the structure.

5.5.2 Portal frame

Portal frame is a frequently used structure. It can be symmetrical or non-symmetrical.



Group **Geometry**:

- **L [m]** –input of a list of spans (distance between neighbor columns). Individual values of span are divided by spaces.
- **H [m] -** input of a list of heights. Individual values of heights are divided by spaces.
- **Input by angle** – if this option is checked, user can input the slope of the roof. The height is calculated.
- **Hr[m] –** input of the height of the roof top above the top of the column. This value can be input only if the option **Input by angle** is not checked.
- **Slope [°]** – input of slope of roof. This value can be input only if the option **Input by angle** is checked.
- **Symmetrical** – if this option is checked, the shape of frame is symmetrical to the vertical axis.
- **Lc [m]** – input of the half of span. This value can be input only if the option **Symmetrical** is checked.

Group **Cross-section**:

- **Column** – select cross-section for all columns. Cross-section can be selected from a list of all available cross-sections.
- **Beam** – select cross-section for all beams. Cross-section can be selected from a list of all available cross-sections.

Group **Position** – definition of position of block inserted into the structure:

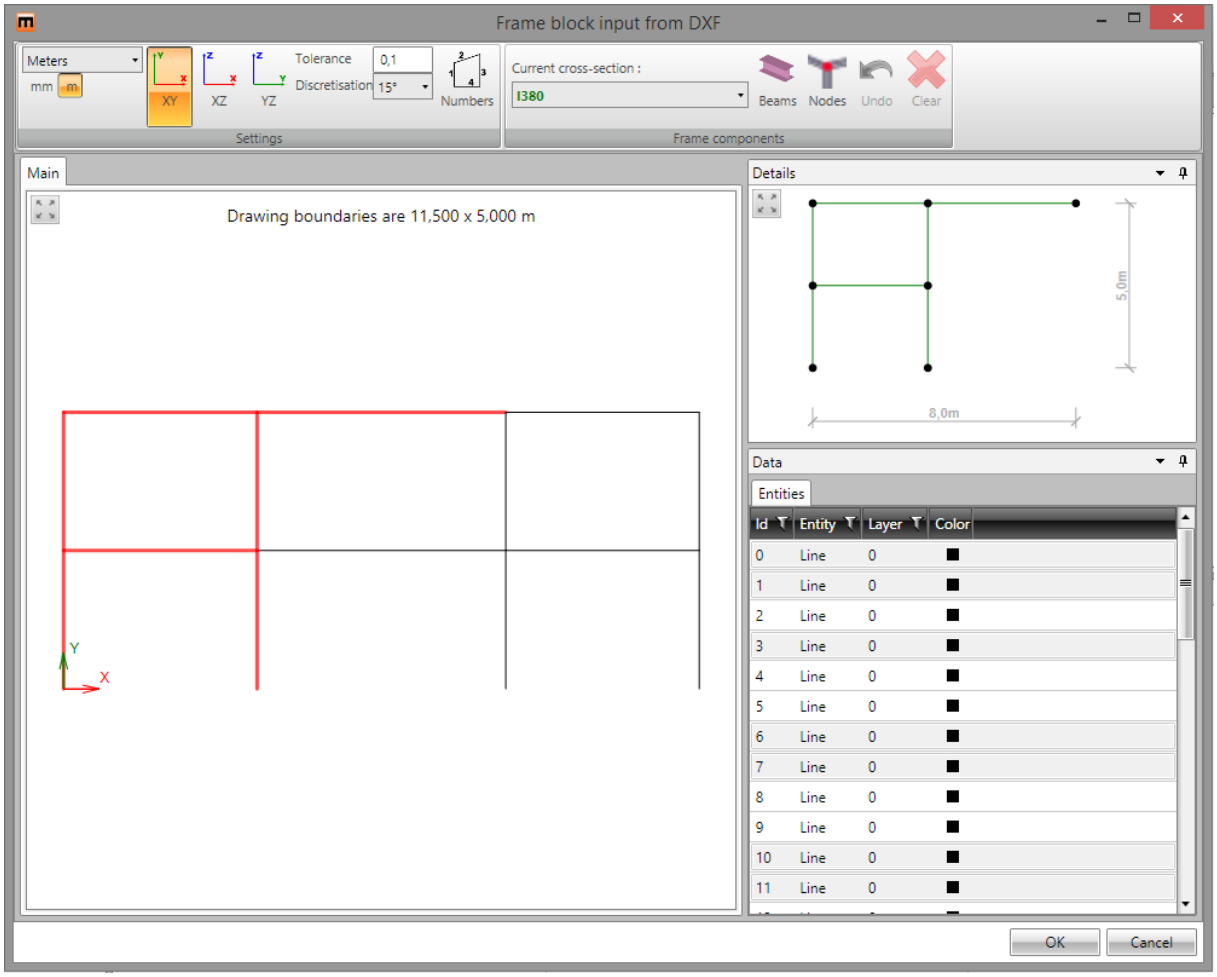
- **X**, **Y** – input of insert point coordinates relative to the origin [0;0] of the structure.
- **Rotation** – input of rotation value of block around the axis perpendicular to the plane of the structure.

5.5.3 Block from DXF

Frame members and nodes can be imported from DXF file.

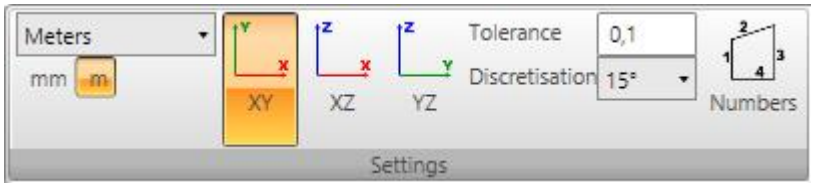
Import of following entities from DXF file is supported: LINE, POLYLINE, SPLINE, ARC, CIRCLE, TEXT. Blocks are not imported. Blocks must be exploded to single entities before import.

To create a new part of frame from DXF file click **From DXF** in ribbon group **Blocks**. Entities, which will be converted to the part of frame, must be selected.



The content of loaded DXF file appears in the dialog **Frame block input from DXF**. Ribbon groups Settings and Frame components are available.

5.5.3.1 Ribbon group Settings

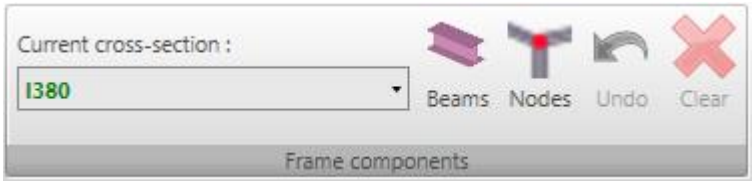


Options in ribbon group **Settings**:

- **Units** – select the length units, which correspond to units of the DXF file.
- **XY** – switch to draw the projection of loaded DXF file to the XY plane of the global coordinate system.

- **XZ** – switch to draw the projection of loaded DXF file to the XZ plane of the global coordinate system.
- **YZ** – switch to draw the projection of loaded DXF file to the YZ plane of the global coordinate system.
- **Tolerance** – the maximal admissible distance between two points to consider the points as identical (to be able to connect the following-up lines).
- **Discretisation** – the angle to convert the arcs to polygons.
- **Numbers** – switch on/off drawing of numbers of entities.

5.5.3.2 Ribbon group Frame components



Options in ribbon group **Frame components**:

- **Current cross-section** – select the current cross-section, which will be assigned to the newly created frame members.
- **Members** – create frame members from the selected entities. The created frame members are displayed in the **Details** window.
- **Nodes** – create frame nodes from the end points of selected entities. The created frame nodes are displayed in the Details window.

5.5.3.3 Creating frame members from loaded entities

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

Entities can be selected in the **Entities** table in the data window – each line corresponds to one entity of DXF file.

After finishing the selection, click **Beams** or **Nodes** in ribbon group **Frame components**. The created frame components are drawn in the **Details** window.

Click **OK** to insert the new entities to into the existing frame.

5.6 Loads

Use commands in navigator group **Loads** to input of load cases, nodal loads, point loads, line loads and load combinations.

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

5.6.1 Groups of load cases

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

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

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

5.6.1.1 Groups of permanent loads

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

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

- **Name** – input name of load group.
- **γ G,sup** – input of partial factor for permanent unfavourable load cases in ULS combinations.
- **γ G,inf** – input of partial factor for permanent favourable load cases in ULS combinations.
- **ξ** – input of reduction factor of unfavourable permanent loads.

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

5.6.1.2 Groups of variable loads

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

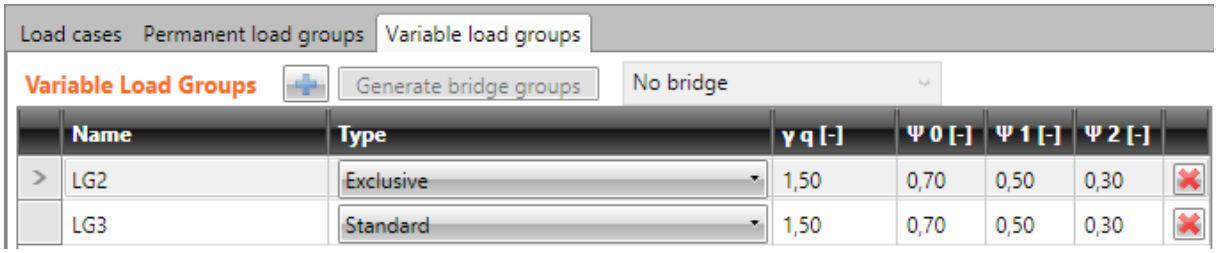


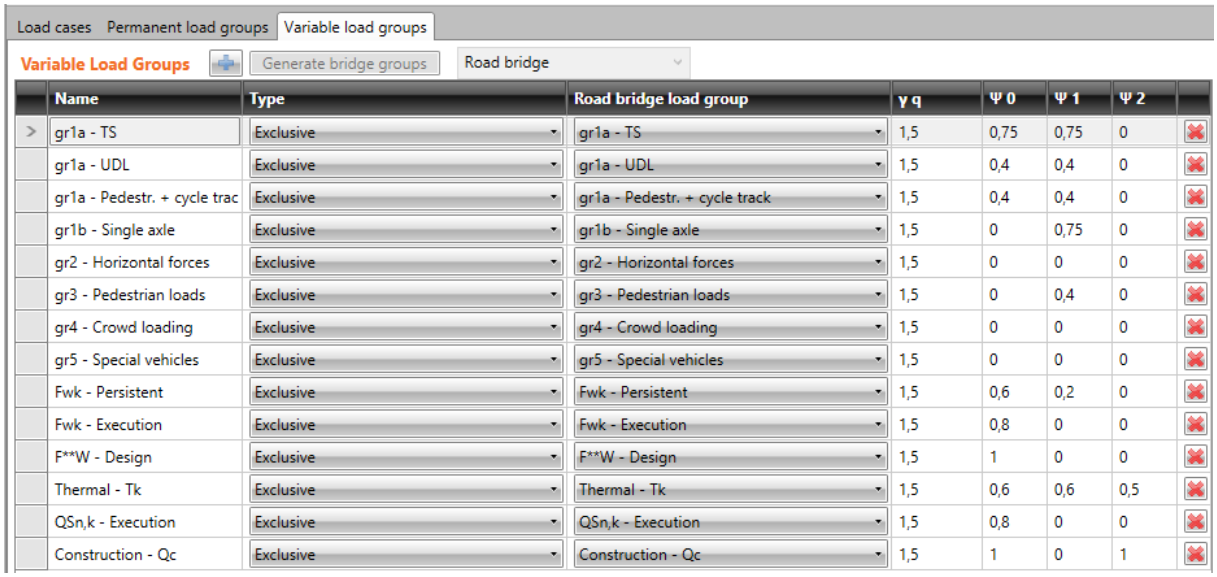
Table **Variable load groups** contains following columns:

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

  - **Exclusive** – load cases from the group are considered as additional load in ULS and SLS combinations. Only one load case from the group can act in single critical combination.
  - **Accidental** – load cases from the group are considered as additional load in ULS and SLS combinations. In the ULS Accidental combination the load cases are considered as design value of an accidental action 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.
- **γq** – input the value of partial load factor of variable load cases in ULS combinations.
- **ψ0** – input the value of partial load factor of variable load cases in ULS and SLS Characteristic combinations.
- **ψ1**- input the value of partial load factor of variable load cases in SLS Frequent combinations.
- **ψ2** - input the value of partial load factor of variable load cases in SLS Quasi-permanent combinations.

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

5.6.1.3 Groups of variable loads for bridges



Name	Type	Road bridge load group	γ q	Ψ 0	Ψ 1	Ψ 2
gr1a - TS	Exclusive	gr1a - TS	1,5	0,75	0,75	0
gr1a - UDL	Exclusive	gr1a - UDL	1,5	0,4	0,4	0
gr1a - Pedestr. + cycle trac	Exclusive	gr1a - Pedestr. + cycle track	1,5	0,4	0,4	0
gr1b - Single axle	Exclusive	gr1b - Single axle	1,5	0	0,75	0
gr2 - Horizontal forces	Exclusive	gr2 - Horizontal forces	1,5	0	0	0
gr3 - Pedestrian loads	Exclusive	gr3 - Pedestrian loads	1,5	0	0,4	0
gr4 - Crowd loading	Exclusive	gr4 - Crowd loading	1,5	0	0	0
gr5 - Special vehicles	Exclusive	gr5 - Special vehicles	1,5	0	0	0
Fwk - Persistent	Exclusive	Fwk - Persistent	1,5	0,6	0,2	0
Fwk - Execution	Exclusive	Fwk - Execution	1,5	0,8	0	0
F**W - Design	Exclusive	F**W - Design	1,5	1	0	0
Thermal - Tk	Exclusive	Thermal - Tk	1,5	0,6	0,6	0,5
QSn,k - Execution	Exclusive	QSn,k - Execution	1,5	0,8	0	0
Construction - Qc	Exclusive	Construction - Qc	1,5	1	0	1

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

- **Road bridge** – variable loads groups for road bridges according to table A2.1 of appendix A2 EN 1900 will be defined/generated.
- **Footbridge** – variable loads groups for footbridges according to table A2.2 of appendix A2 EN 1900 will be defined/generated.
- **Railway bridge** - variable loads groups for road bridges according to table A2.3 of appendix A2 EN 1900 will be defined/generated.
- **No bridge** – ordinary (not for bridges) variable loads groups will be defined.

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

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

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

Footbridges:

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

Road bridges:

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

Railway bridges:

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

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

-  - add new user defined bridge load group.

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

- **Name** – input name of load group.

- **Traffic** – if selected, loads in the group are considered as the traffic loads when evaluating combinations.

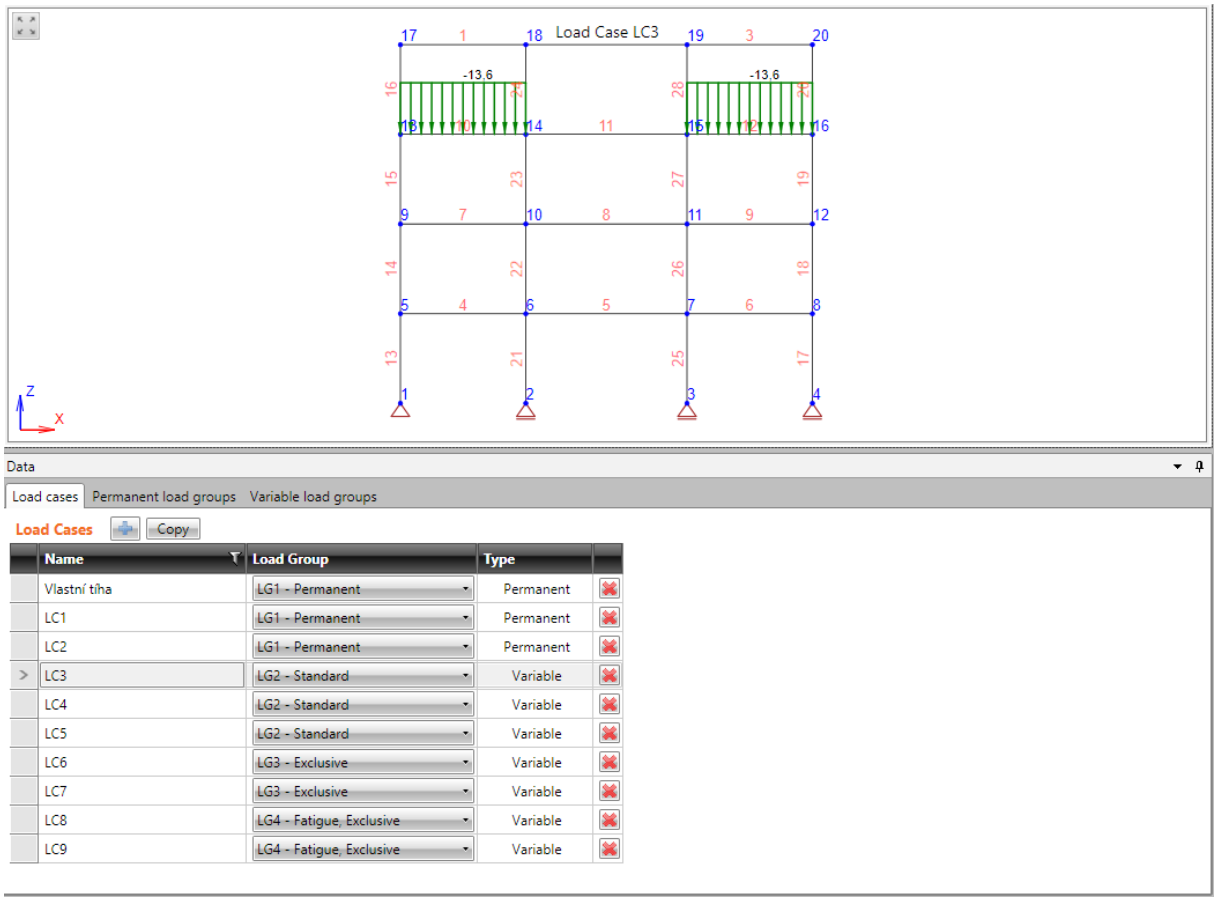
5.6.2 Load cases

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

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

If **Prestressing** is active in project, one permanent load case named "**Prestress**" is generated automatically. This load case is used for transfer of equivalent load actions caused by prestressing tendons into the static model of the structure. Load case "**Prestress**" cannot be deleted.

Load case of self-weight is generated automatically.



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

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

If there is no load case of self-weight, click **Self weight** above the table to add load case of self-weight.

Table **Load Cases** contains following columns:

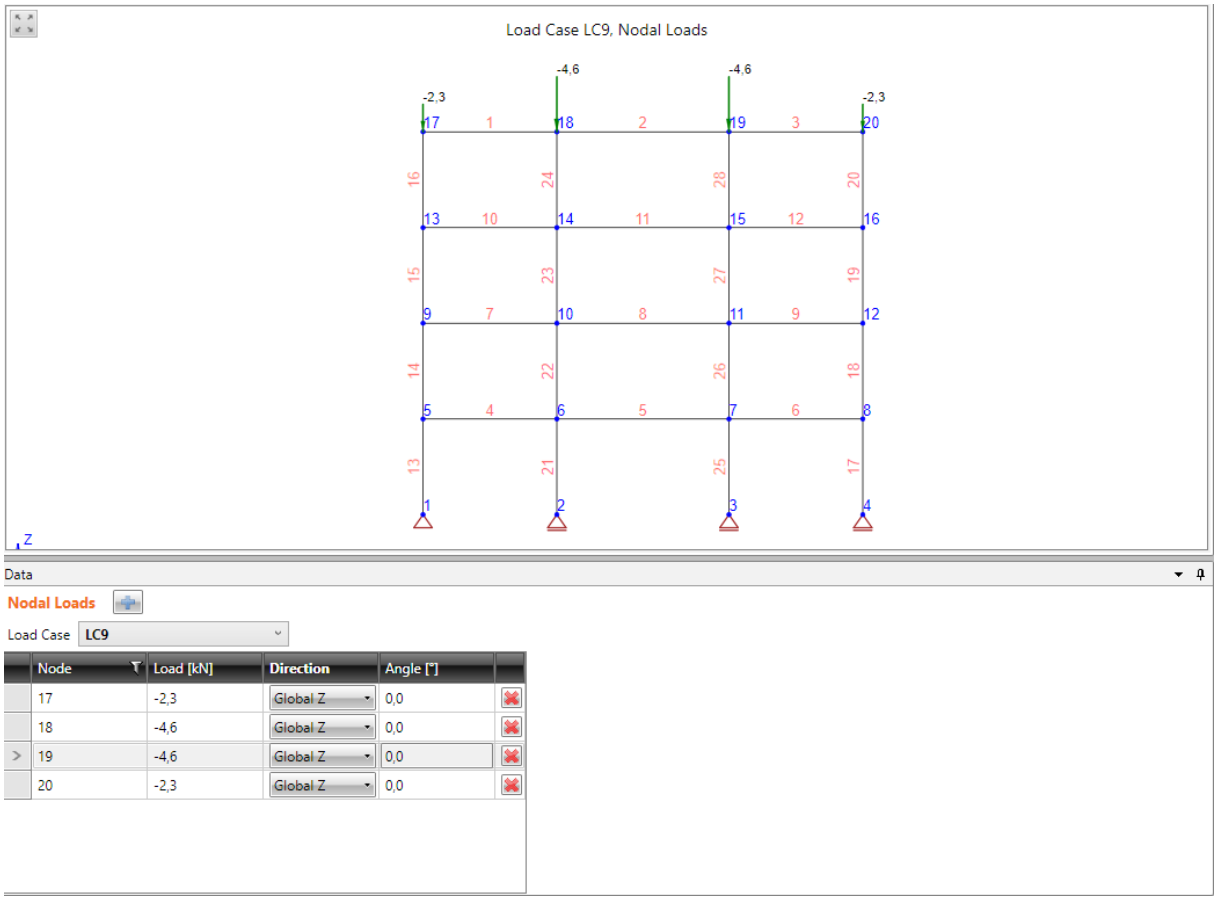
- **Name** – enter name of load case.
- **Load group** – select the load group to be assigned to the load case.
- **Type** - click the cell to change the type of load.
-  - delete the appropriate load case.

5.6.3 Nodal loads

Click navigator command **Loads > Nodal Loads** to start input of nodal loads.

Nodal load can be defined in any node of a frame. The force can act in direction of global axis X or Z. Also inclination can be defined..

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



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

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

Table **Nodal Loads** contains following columns:

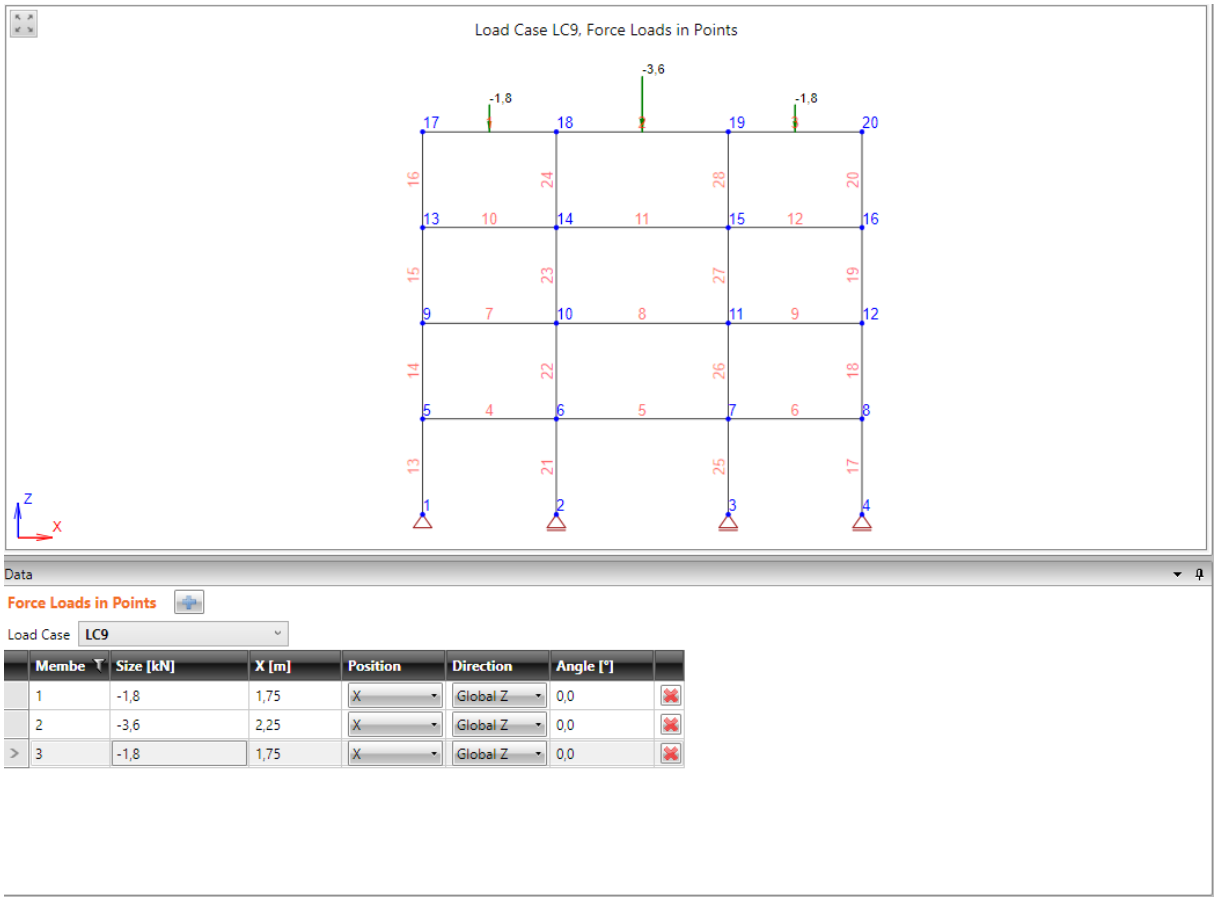
- **Node** –input the number of node loaded by nodal load.
- **Load** –input the value of nodal load.
- **Direction** – select the axis of global coordinate system, in which direction the load is acting.
- **Angle** – input value of load inclination to defined direction.
-  - delete the appropriate nodal load.

5.6.4 Point forces

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

Point force can be defined on any member of a frame. The force can act in direction of X or Z-axis of global coordinate system or of local coordinate system of member. Also inclination can be defined.

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



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

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

Table **Point loads** contains following columns:

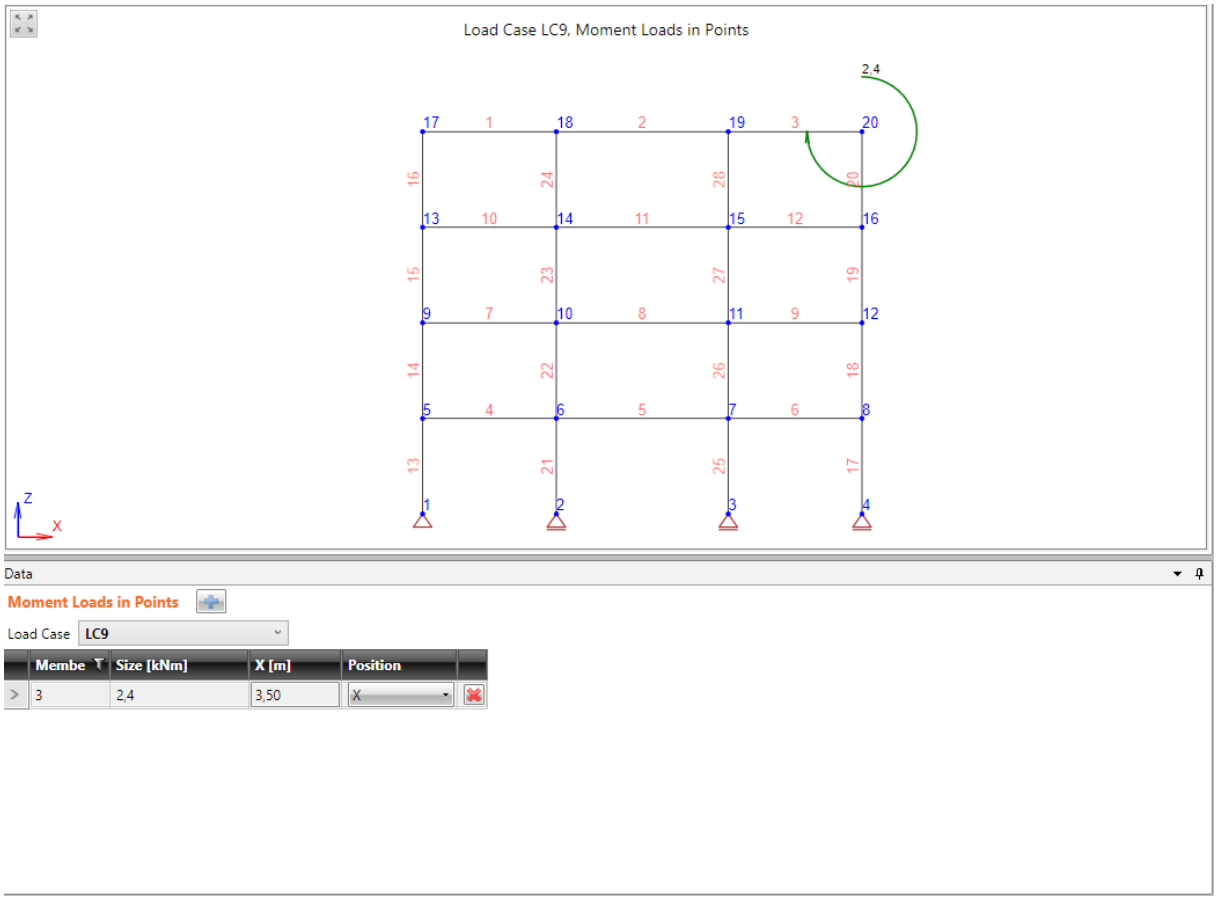
- **Member –** input the number of member loaded by point force.
- **Size** – input the value of point force.
- **X [m]** – input the position of load from the beginning of the member. The value is taken into account only if the value in column **Position** is set to **X**.
- **Position** – select the mode to define the force position. Following modes are available:
  - **X –** one force on the member in the distance defined in the X-column.
  - **P1/2 –** one force in the mid of the member.
  - **P1/3** – two forces in the thirds of the member.
  - **P1/4** – three forces in the quarters of the member.
  - **P1/5** – four forces in the fifths of the member.
- **Direction** - select the axis of coordinate system in which point force is defined. Following directions are available:

  - **Global Z** – the load acts in the direction of Z-axis of global coordinate system.
  - **Global X** – the load acts in the direction of X-axis of global coordinate system.
  - **Local z** – the load acts in the direction of z-axis of the member local coordinate system.
  - **Local x** – the load acts in the direction of x-axis of the member local coordinate system.
- **Angle** – input the value of point force inclination to specified direction.
-  - delete the appropriate point load.

5.6.5 Point moments

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

Point moments can be defined on any member of a frame. Moment acts around the Y axis of global coordinate system.



Current load case is set in the list **Load cases**. Point moments already defined in this load case are displayed in the table.

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

Table **Moment loads in point**s contains following columns:

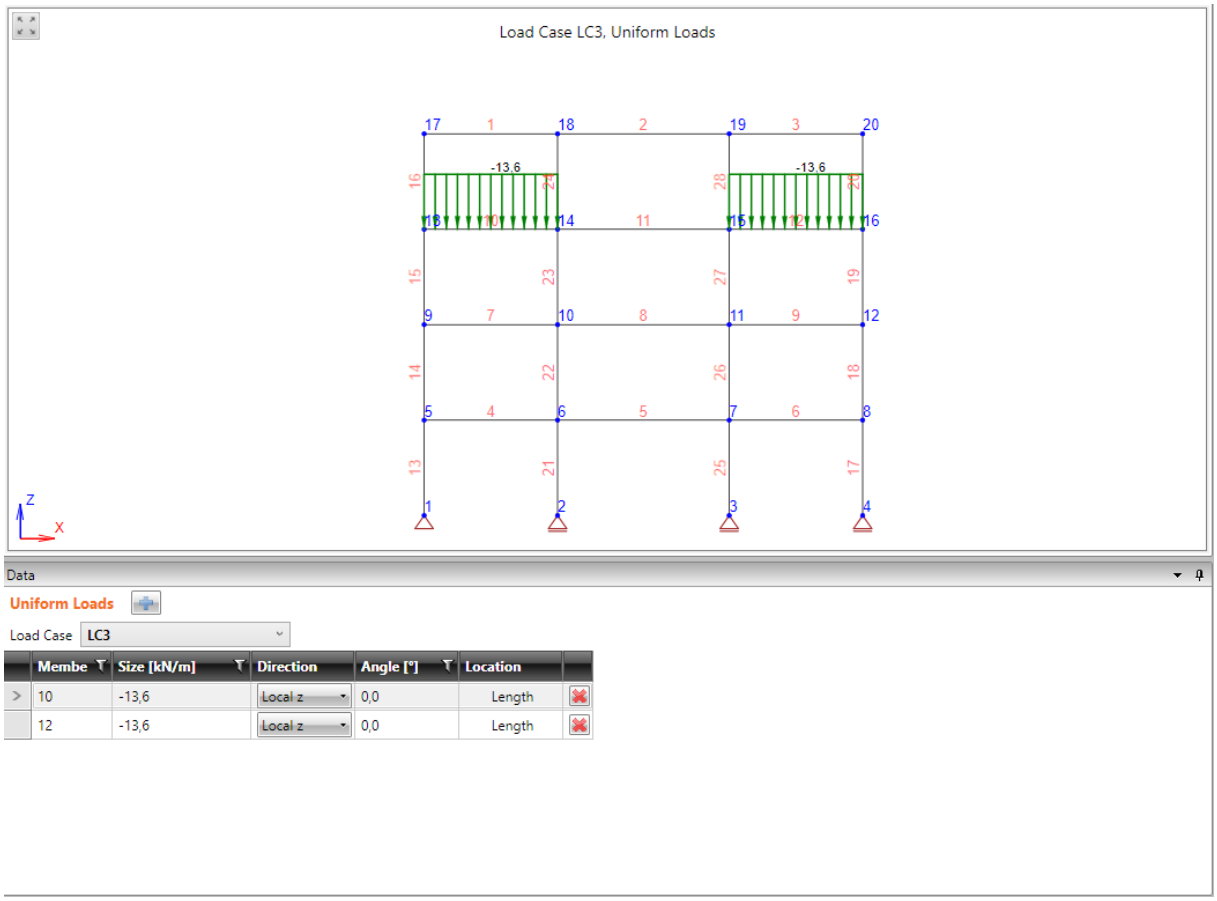
- **Member –** input the number of member loaded by point moment load.
- **Size** – input the value of point moment load.
- **X [m]** – input the position of moment load from the beginning of the member. The value is taken into account only if the value in column **Position** is set to **X**.
- **Position** – select the mode to define the moment position. Following modes are available:
  - **X –** one moment on the member in the distance defined in the X-column.
  - **P1/2 –** one moment in the mid of the member.
  - **P1/3** – two moments in the thirds of the member.
  - **P1/4** – three moments in the quarters of the member.
  - **P1/5** – four moments in the fifths of the member.
-  - delete the appropriate moment load.

5.6.6 Uniform loads

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

Uniform load can be defined on any member of a frame. The load can act in direction of axis X or Z of global coordinate system or of local coordinate system of member. Also inclination can be defined.

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



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

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

Table **Uniform Loads** contains following columns:

- **Member** – input the number of member loaded by uniform load.
- **Size** – input the value of uniform load.
- **Direction** - select the axis of coordinate system in which distributed load is defined. Following directions are available:
  - **Global Z** – the load acts in the direction of Z-axis of global coordinate system.
  - **Global X** – the load acts in the direction of X-axis of global coordinate system.
  - **Local z** – the load acts in the direction of z-axis of the member local coordinate system.
  - **Local x** – the load acts in the direction of x-axis of the member local coordinate system.
- **Angle** – input the value of distributed load inclination to specified direction.
- **Location** – select the location mode of the load. Following modes are available:

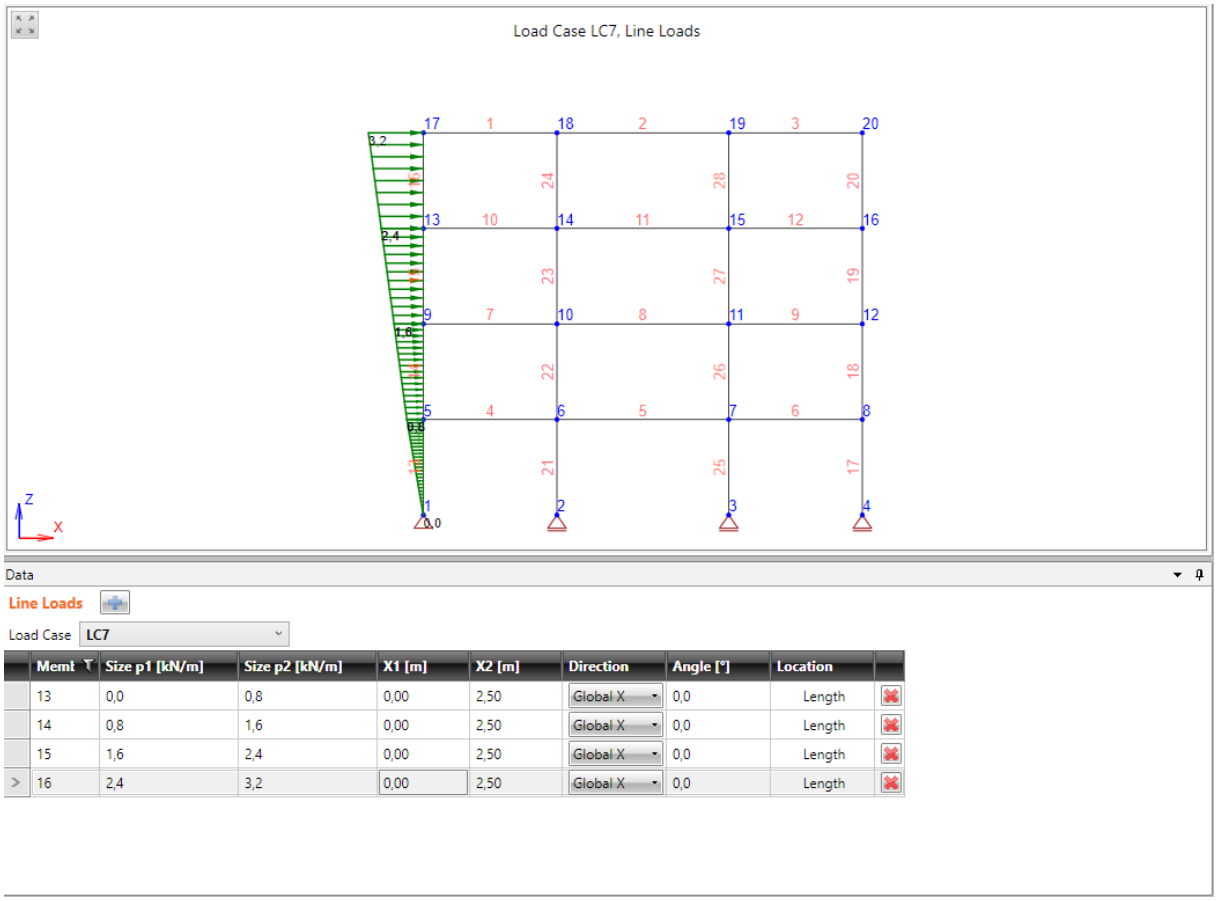
  - **Length** – the load is placed on the whole length of the member.
  - **Projection** – the load is placed on the projection of member on the corresponding axis. This mode can be used only for loads acting in global coordinate system.
-  - delete the appropriate uniform load.

5.6.7 Line load

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

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

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



Current load case is set in the list **Load cases**. Line loads already defined in this load case are shown in table.

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

Table **Line loads** contains following columns:

- **Member** – input the number of member loaded by uniform load.
- **Size p1** – input the value of line load at the beginning (in position **X1**)
- **Size p2** – input the value of line load at the end (in position **X2**)
- **Direction** - select the axis of coordinate system in which distributed load is defined. Following directions are available:
  - **Global Z** – the load acts in the direction of Z-axis of global coordinate system.
  - **Global X** – the load acts in the direction of X-axis of global coordinate system.
  - **Local z** – the load acts in the direction of z-axis of the member local coordinate system.
  - **Local x** – the load acts in the direction of x-axis of the member local coordinate system.

- **Angle** – input the value of distributed load inclination to specified direction.
- **Location** – select the location mode of the load. Following modes are available:
  - **Length** – the load is placed on the whole length of the member.
  - **Projection** – the load is placed on the projection of member on the corresponding axis. This mode can be used only for loads acting in global coordinate system.
-  - delete the appropriate line load.

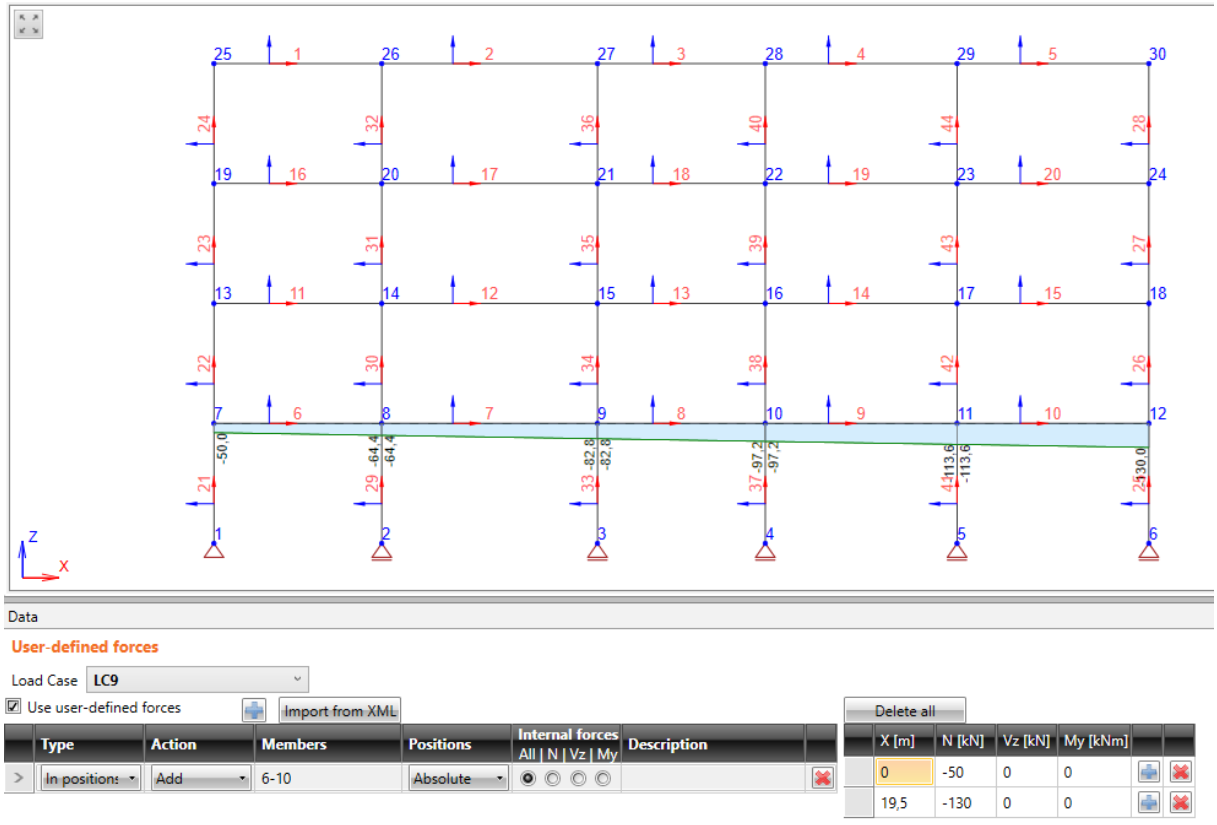
5.6.8 User defined internal forces

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

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

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

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



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

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

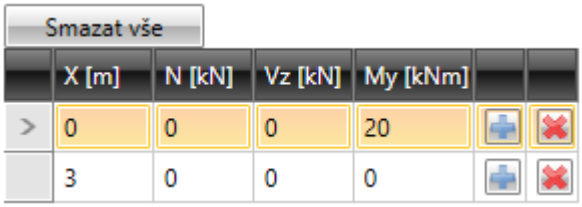
If the check box **Use user defined forces** is not selected, the user defined forces of appropriate load case are not taken into account during the calculation event though they are defined.

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

- **Type** – select mode of user defined force input:
  - **In position** – user defined forces are defined by values of internal force components in specified positions. The values of user defined forces are interpolated linearly between specified positions. Positions and corresponding force values are defined in the next table.
  - **By curve** - user defined forces are defined by curve between specified positions. Course curves are defined in the next table.

- **Action** – select the interaction between user defined internal forces and calculated internal forces caused by other loads in the same load case:
  - **Add** – user defined internal forces are added to calculated internal forces from other loads in the appropriate load case.
  - **Replace** – user defined internal forces replace the calculated internal forces.
- **Members** – input list of numbers of consecutive members to apply the user defined internal forces on.
- **Position** – select evaluation mode of specified positions of user defined forces:
  - **Absolute** – position coordinates are specified in absolute values from the beginning of first selected member.
  - **Relative** – coordinates are specified relative to the total length of selected members.
- **Internal forces** – select components of internal force to be defined:
  - **All** – values in positions or curves of all internal forces components can be defined in the next table.
  - **N** – values in positions or curves of normal force N can be defined in the next table.
  - **Vz** – values in positions or curves of shear force Vz can be defined in the next table.
  - **My** – values in positions or curves of bending moment My can be defined in the next table.
-  - delete the appropriate user defined force.

5.6.8.1 User defined forces in positions



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

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

Columns in table:

- **X** – input position to define values of internal forces. According to the setting in **Position** column the value is evaluated as absolute to the beginning of first selected member or relative to all selected members.
- **N** – input value of normal force N in appropriate position.
- **Vz** – input value of shear force Vz in appropriate position.
- **My** – input value of bending moment My in appropriate position.
-  - add new position for input of user defined internal forces.
-  - delete existing position with values of user defined internal forces.

5.6.8.2 User defined internal forces by curves

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

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

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

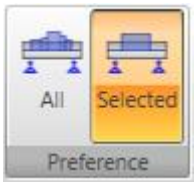
- **Not set** – course of internal force component is not defined.
- **Constant** – the course of internal force component is constant and is defined by following values:
  - **Begin** – input position of beginning point of curve.
  - **End** – input position of end point of curve.
  - **Value – begin**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

5.6.8.3 Ribbon group User defined forces

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

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

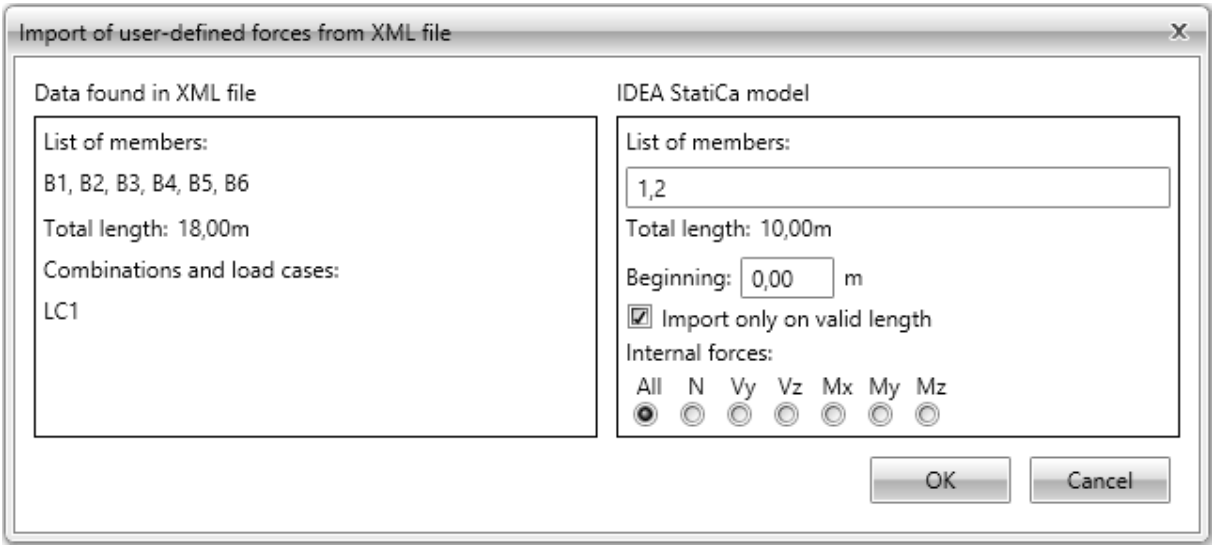
5.6.8.4 Ribbon group Preference



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

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

5.6.8.5 Import of internal forces from XML file



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

Members, sections on members and internal forces in sections are read from XML file. Single members are lined up and the resulting sequence of positions with internal forces is applied on selected members in IDEA model. The shift of imported sequence related to the beginning of first selected member in IDEA model can be specified.

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

Group **Data found in XML file**:

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

Group **IDEA StatiCa model**:

- **List of members** – input list of consecutive members in IDEA model to apply the imported internal forces on.
- **Total length** – the overall length of selected members is displayed.
- **Beginning** – input the offset between beginning point of imported internal forces and beginning of the first selected member in IDEA StatiCa.

- **Import only on valid length** – if the option is selected, only positions in range from defined beginning to the end point of last selected member in IDEA model are imported. Positions out of this range are excluded from import.
- **Internal forces** – select components of internal forces to be imported.

5.6.9 Combinations

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

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

One of following combination types can be assigned to each combination:

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

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

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

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

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

Click  above the table of combinations to input new combination.

Table **Combinations** contains following columns:

- **Name** – input the name of combination.
- **Type** – select the type of current combination.
- **Evaluation** – select the evaluation mode of combination.
-  - launch Combinations manager to edit combinations – see **5.6.10 Manager of load cases combinations**.

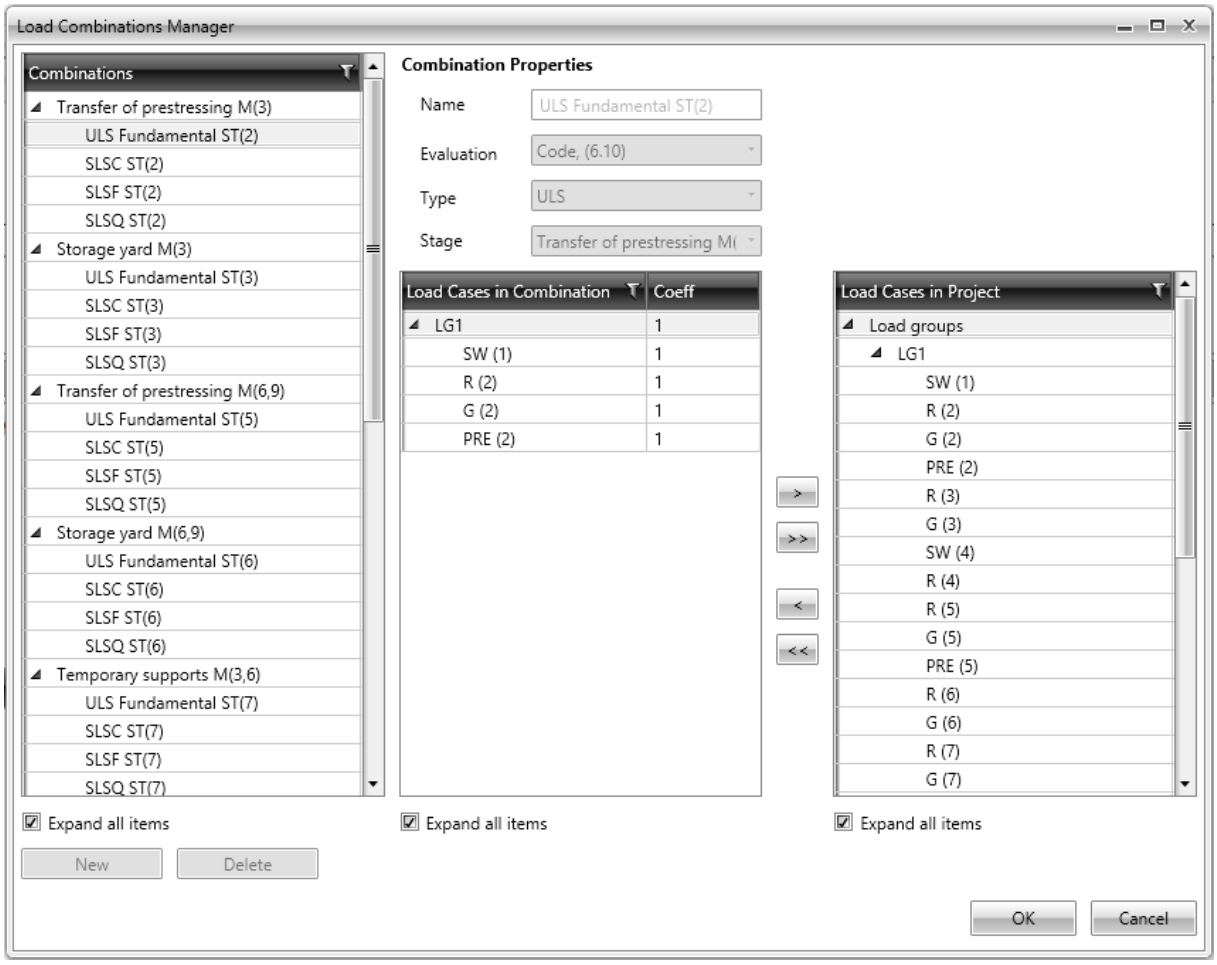
-  - delete the appropriate combination.

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

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

5.6.10 Manager of load cases combinations

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



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

One of following types can be assigned to each combination:

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

Following evaluation mode can be set for the combination:

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

- **For SLS Frequent combination** formula 6.15b;
- **For SLS Quasi-permanent combination** formula 6.16b;
- For **ULS Accidental combination** formula 6.11b;
- For **ULS Fatigue combination** formula 6.69 EN 1992-1-1.

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

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

Options of **Combinations manager** dialog:

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

Combination properties:

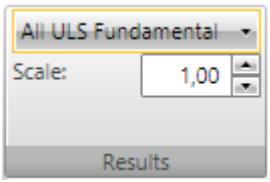
- **Name** – input name of the current combination.
- **Evaluation** – select the evaluation mode of the current combination.
- **Type** – select the type of the current combination.
- **Stage** – construction stage, to which is the current combination assigned.
- **Load cases in combination** – load cases 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.

6 Results

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

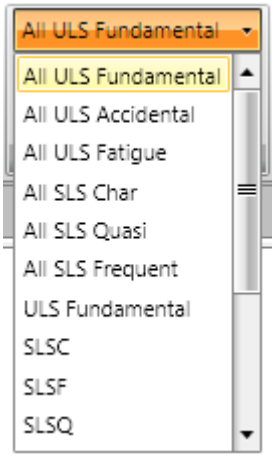
After the analysis finishes, all navigator commands in the **Results** group are available.

6.1 Results evaluation setup



Evaluation of reactions, deformations and internal forces is performed for the current load case, combination or result class.

Current load case, combination or result class can be selected in the list in ribbon group **Results**.

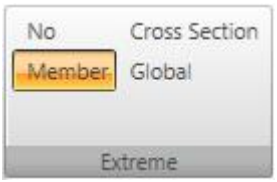


Following result classes are generated automatically:

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

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

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



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

6.2 Reactions in supports

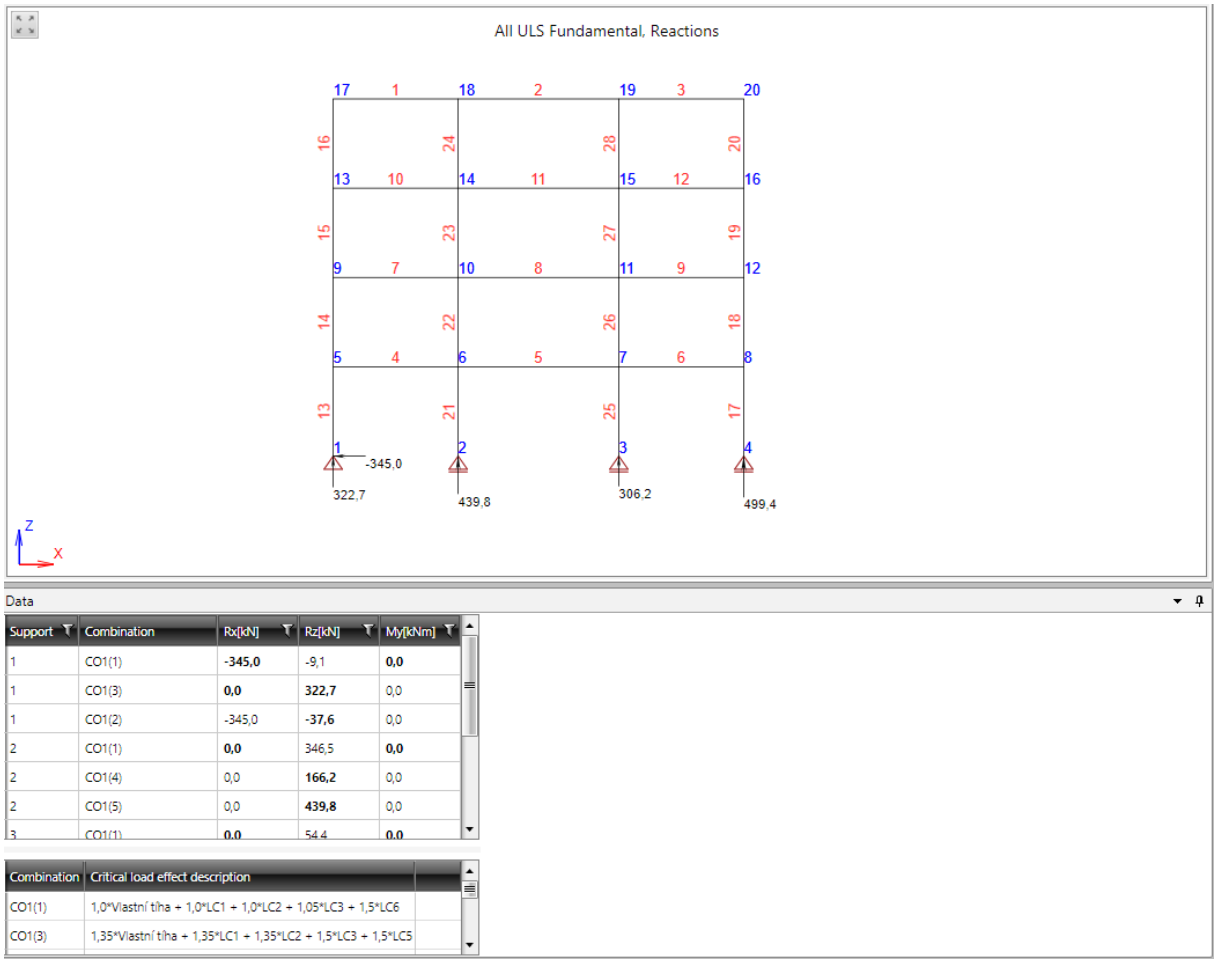
Click navigator command **Results** > **Reactions** to evaluate reactions in supports.

Calculated reactions are evaluated:

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

Reactions are evaluated for current load case or combination.

Ribbon groups **Results**, **System of reactions** and **Extreme** are available when evaluating reactions in supports.



6.2.1 Ribbon group Results

See **6.1 Results evaluation setup**.

6.2.2 Ribbon group System of reactions

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

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

6.2.3 Ribbon group Extreme

See **6.1 Results evaluation setup**.

6.3 Deformations

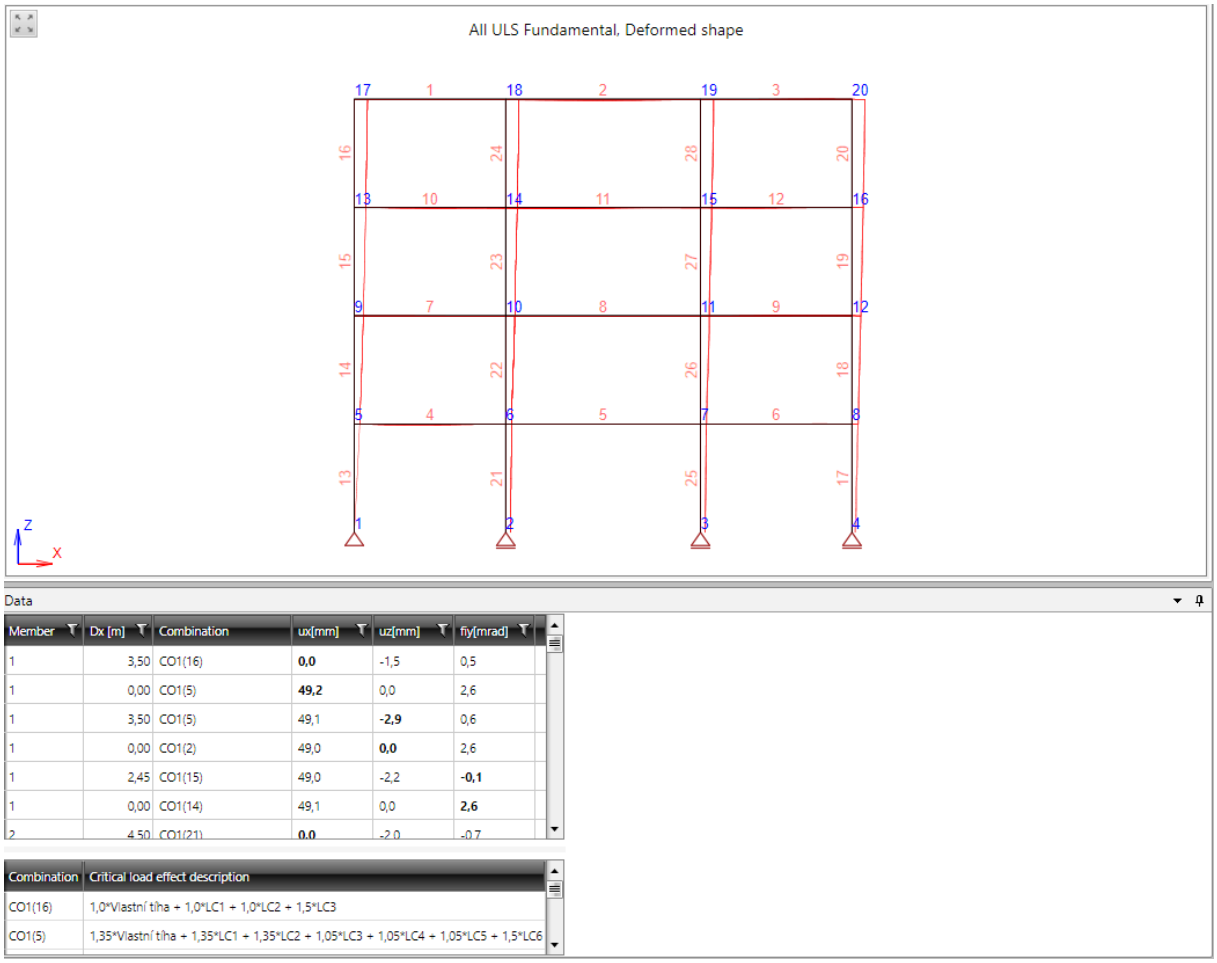
Click navigator command **Results** > **Deformations** to evaluate deformations of the frame.

Calculated deformations are evaluated:

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

Deformations are evaluated for current load case or combination.

Ribbon groups **Results, Selection, Extreme** and **Deformations** are available.



6.3.1 Ribbon group Results

See **6.1 Results evaluation setup**.

6.3.2 Ribbon group Selection

Use this ribbon group to evaluate results on individual member only.

- **Single select** – switch on/off the mode of evaluation of one member only. Graphical and text evaluation is then performed only for one selected member.
  Number of current member can be selected or entered in the list under this button.

6.3.3 Ribbon group Extreme

See **6.1 Results evaluation setup**

6.3.4 Ribbon group Deformations

Particular options of ribbon group **Deformations**:

- **ux** – switch to graphical evaluation of deformation in direction of local member axis x.
- **uz** – switch to graphical evaluation of deformation in direction of local member axis z.
- **fiy** – switch to graphical evaluation of rotation around local member axis y.
- **Deformed frame** – switch on graphical evaluation of deformation of the whole structure.

6.4 Internal forces

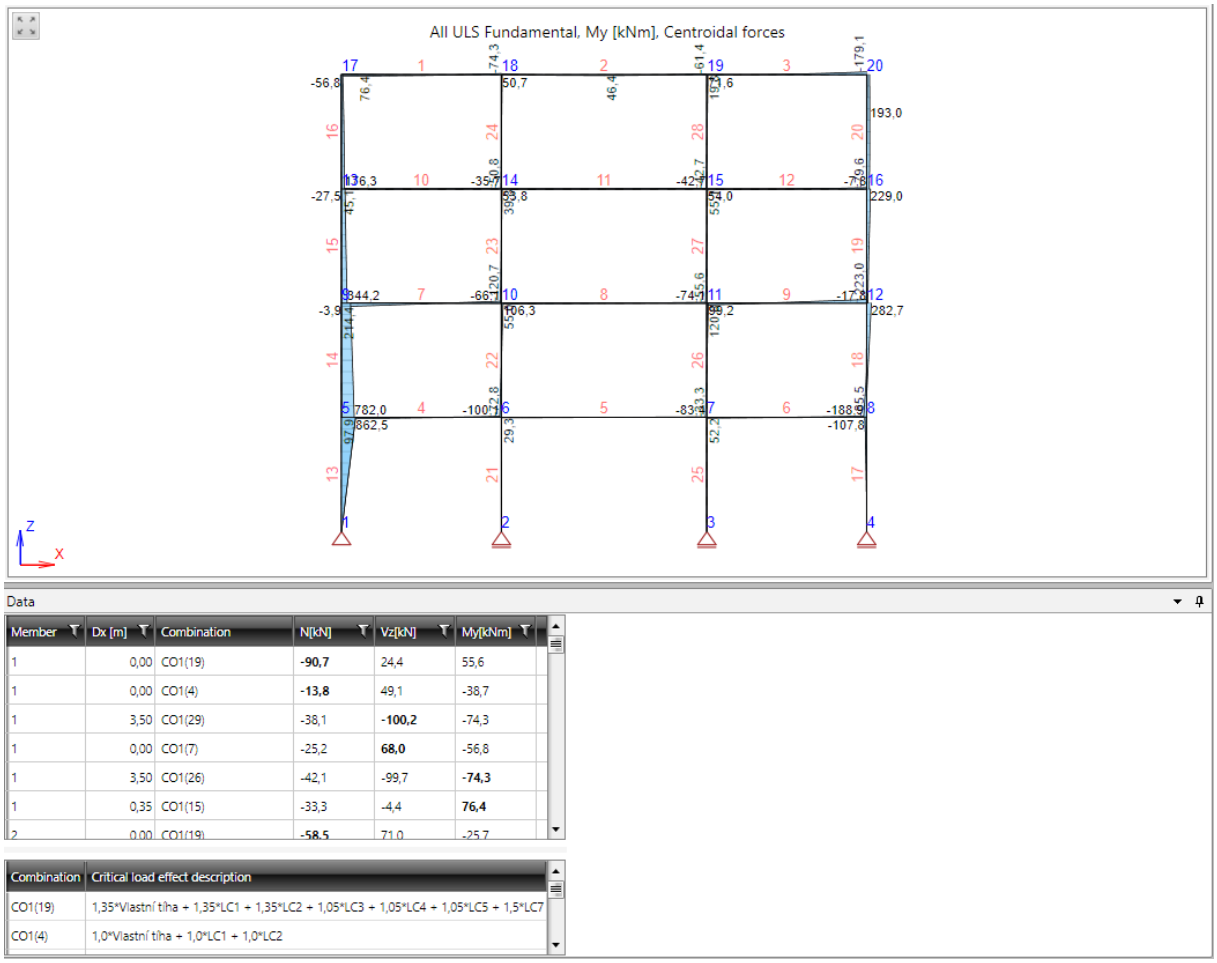
Click navigator command **Results** > **Internal forces** to evaluate internal forces of the frame.

Calculated internal forces are evaluated:

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

Internal forces are evaluated for current load case or combination.

Ribbon groups **Results, Selection, Extreme, Transformation, Internal forces** and **Fatigue evaluation** are available.



6.4.1 Ribbon group Results

See **6.1 Results evaluation setup**.

6.4.2 Ribbon group Selection

See **6.3.2 Ribbon group Selection**.

6.4.3 Ribbon group Extreme

See **6.1 Results evaluation setup**.

6.4.4 Ribbon group Transformations

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

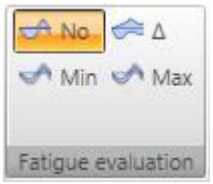
- **Centroidal** – switch to evaluate internal forces in sections perpendicular to the centroidal axis of members.
- **Design** – switch to evaluate internal forces in sections perpendicular to the reference line.

6.4.5 Ribbon group Internal forces

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

- **N** – switch to the graphical evaluation of normal force N.
- **Vz** – switch to the graphical evaluation of shear force Vz.
- **My** – switch to the graphical evaluation of bending moment My.

6.4.6 Ribbon group Fatigue evaluation



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

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

7 Design and check of structural items

IDEA Frame is an application for static analysis of the structure. Linear deformations, internal forces and reactions in supports caused by acting load are results of this analysis.

The design of concrete members in IDEA Frame consist of few steps

- Definition of the design members. Design member contains one or group of consecutive structural model members. Consecutive members must have common node of structural model and must have the same orientation – end point of one member is beginning point of the following member. Design member is analysed as the whole.
- Assigning design members to design groups. Design groups contain design members, which are designed in similar way.

Following rules are applied when working with designer members:

- each structural member must be assigned to a design member;
- structural members in the design member must be of 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 total length;
    - Have the same element type (rib, column, beam);
    - Number of members in design member is the same.

7.1 Design of reinforced concrete members

Following steps have to be processed for the current design group, or rather for the representative design member of the design group:

- Design parameters setting;
- Definition of reinforcement zones;
- Input of reinforcement to the reinforcement zones. The reinforcement is defined on the representative design member of the design group.
- Adaptation of supports for the calculation of deflections
  Check of reinforced cross-sections and calculation of deflections and evaluation of result along the representative design member. The calculation of deflections is performed on each design member in the design group. The deflections are evaluated as an envelope from all design members in the design group.

7.2 Check of steel members

For the design members, which belongs to the design group of steel members, following steps are performed:

- Design parameters settings;;
- Input of design data – LTB supports, fields to be checked;
- Input of buckling parameters on members of the design member;
- Check and evaluation of the section resistance, buckling resistance and deflections of the structure.

7.3 Prestressed design members

For prestressed structures the whole analysis model is transferred to IDEA Tendon. Design of tendons is performed in IDEA Tendon and equivalent load forces are transferred back to load case for prestressing.

Click **Tendon design** in ribbon to start IDEA Tendon module to design prestressing tendons into the design member.

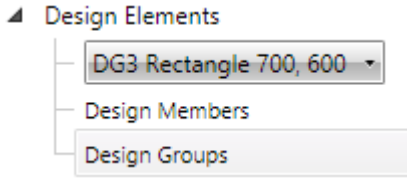
Input of tendons is described in separate user guide IDEA Tendon.

Reinforcing of prestressed design member by concrete reinforcement and its check is performed in IDEA Frame - similar to non-prestressed design members.

Following preconditions have to be fulfilled to be able to start the design of tendons:

- The option Prestressing is selected in the Project data;
- Load cases for transfer of loads caused by prestressing exist (it should be generated automatically after the prestressing support is turned on);
- ULS and SLS load cases combinations are defined (characteristic, quasi-permanent, frequent);
- The results of linear calculation are available.

8 Design members and design groups



The design members have to be defined to be able to check the structure. The design members, which are expected to be checked in the similar way, should be assigned to one design group. Use commands in navigator group **Design elements** to create and edit design members and design groups.

By default, following rules are applied when creating design members and design groups on the new structure:

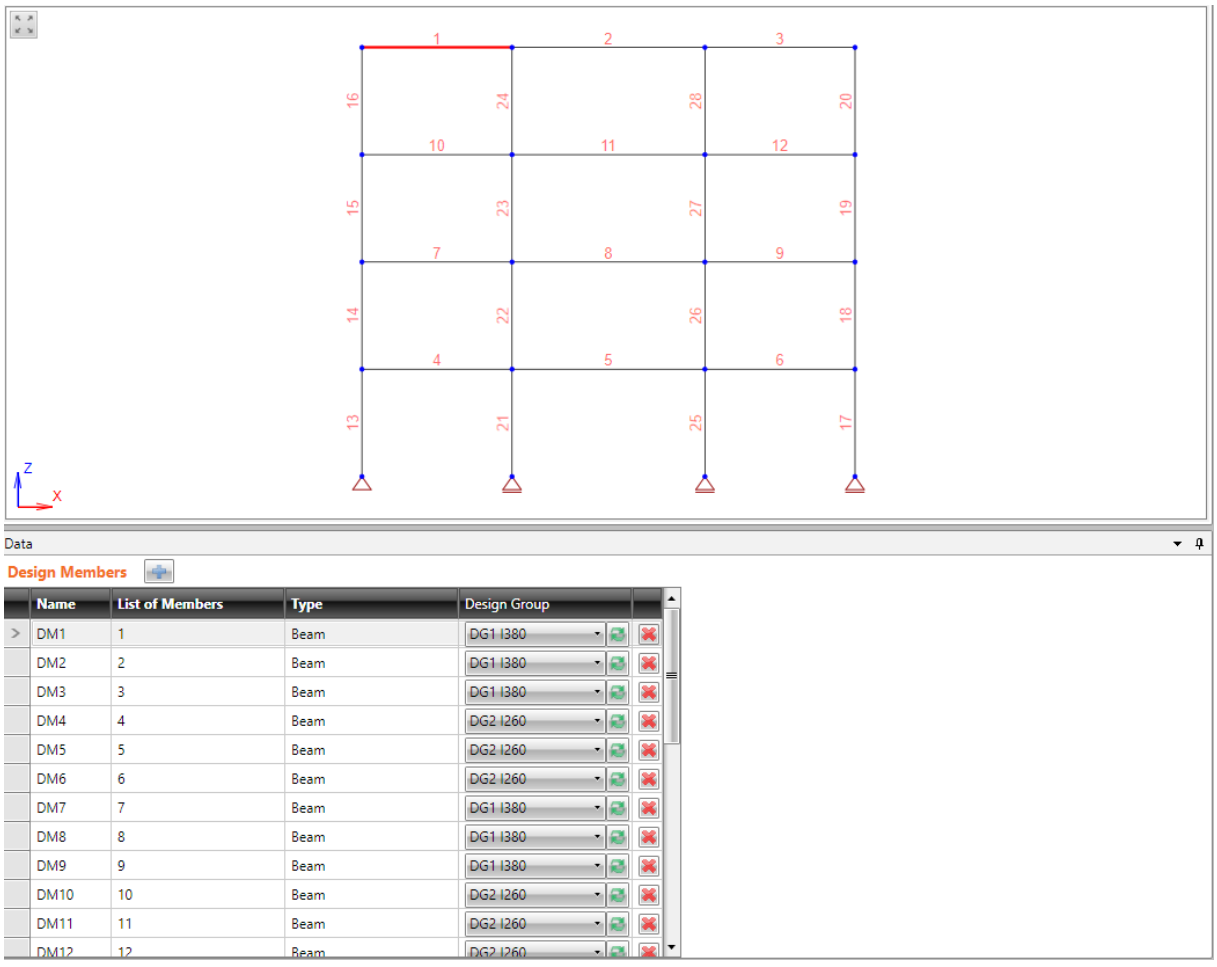
- One design member is generated for each structural member.
- Design members with the same cross-section and the same length are assigned to one design member.

8.1 Design members

Click navigator command **Design elements > Design members** to create or edit design members.

The structure is drawn in the main window and the current design member is drawn in bold.

The table of already defined design members is displayed in the data window.



Click  above the table to add new design member.

Table contains following columns:

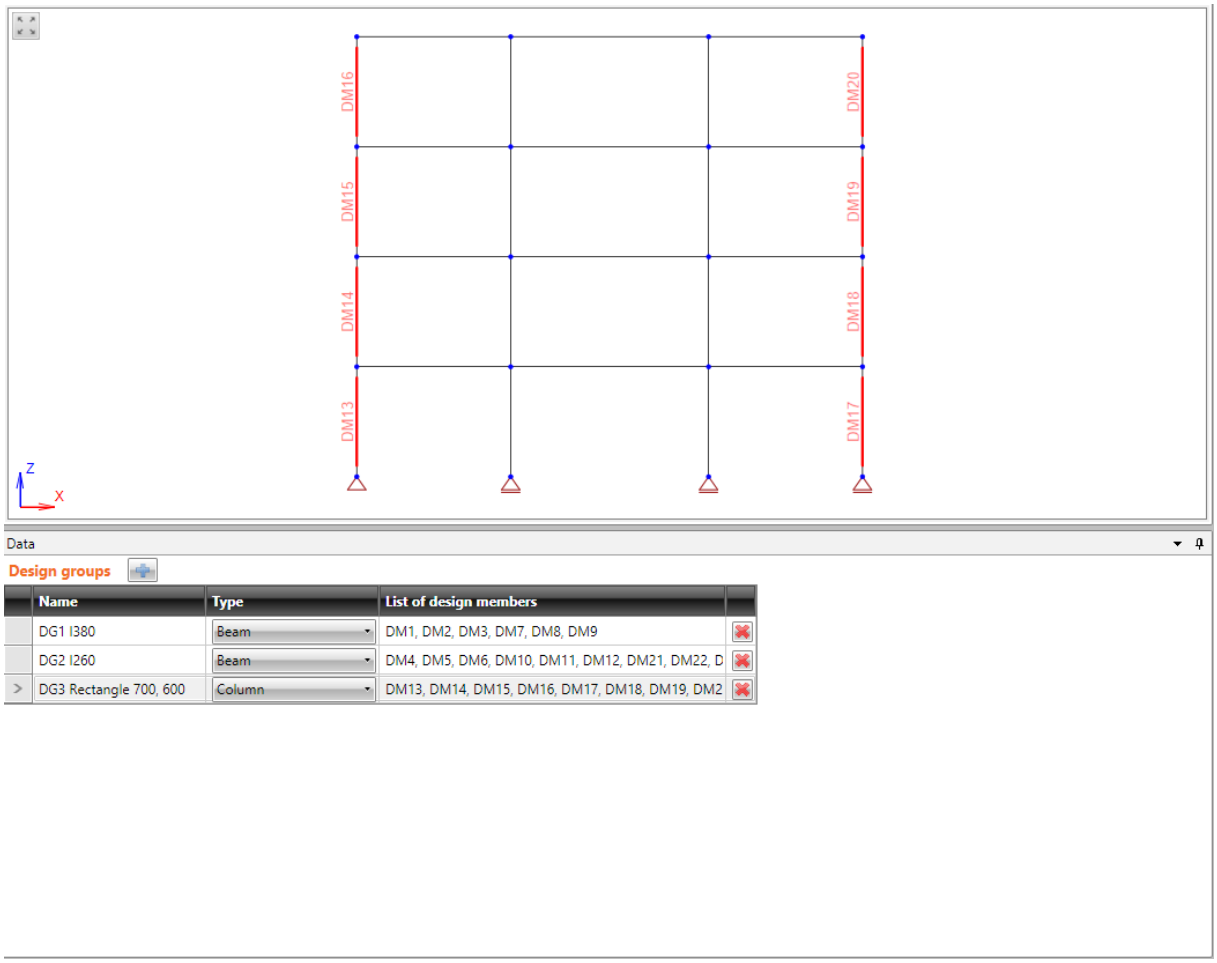
- **Name** – input name of design member.
- **List of members** – input numbers of structural members, which the current design member consists of. The list of members can be edited using numbers separated with commas or the sequence defined by two numbers separated by dash. Structural member can be assigned only to one design member only, thus if the structural member is re-assigned to the other design member, it is removed from the original design member. If the original design member does not contain any structural member, it is removed automatically.
- **Design group** – design group, to which is assigned the current design member. The list contains design groups, which are suitable for the current design member. Click  to create new design group and assign the current design member to newly created design group.
-  - delete the appropriate design member.

8.2 Design groups

Click **Design elements** > **Design groups** to create or edit design groups.

The structure is drawn in the main window and all design members in the current design group are drawn in bold.

The table of already defined design groups is displayed in the data window.



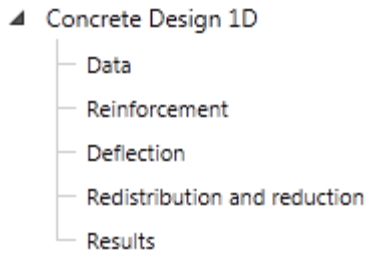
Click  above the table to add new design group.

Table **Design groups** contains following columns:

- **Name** – input the name of design group.
- **Type** – select the type of design group from the list. The possible types are **Column** or **Beam**. The type affects the type of calculations and checks available for design group.
- **List of design members** – display list of design members, assigned to the design group. The list cannot be edited. The design member can be assigned to other design group when editing design member properties only
-  - delete the design group. If the design group is not empty, the affected design members are removed from the design group (are not assigned to any design group).

9 Design of concrete members

9.1 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 representative design member of the current design group is used to input the reinforcement and data for check. The current design group is set in the list in the navigator group **Design elements**.

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 components of the whole zone are found from all design members in the design group.

To be able to design the reinforcement of design group several preconditions have to be fulfilled:

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

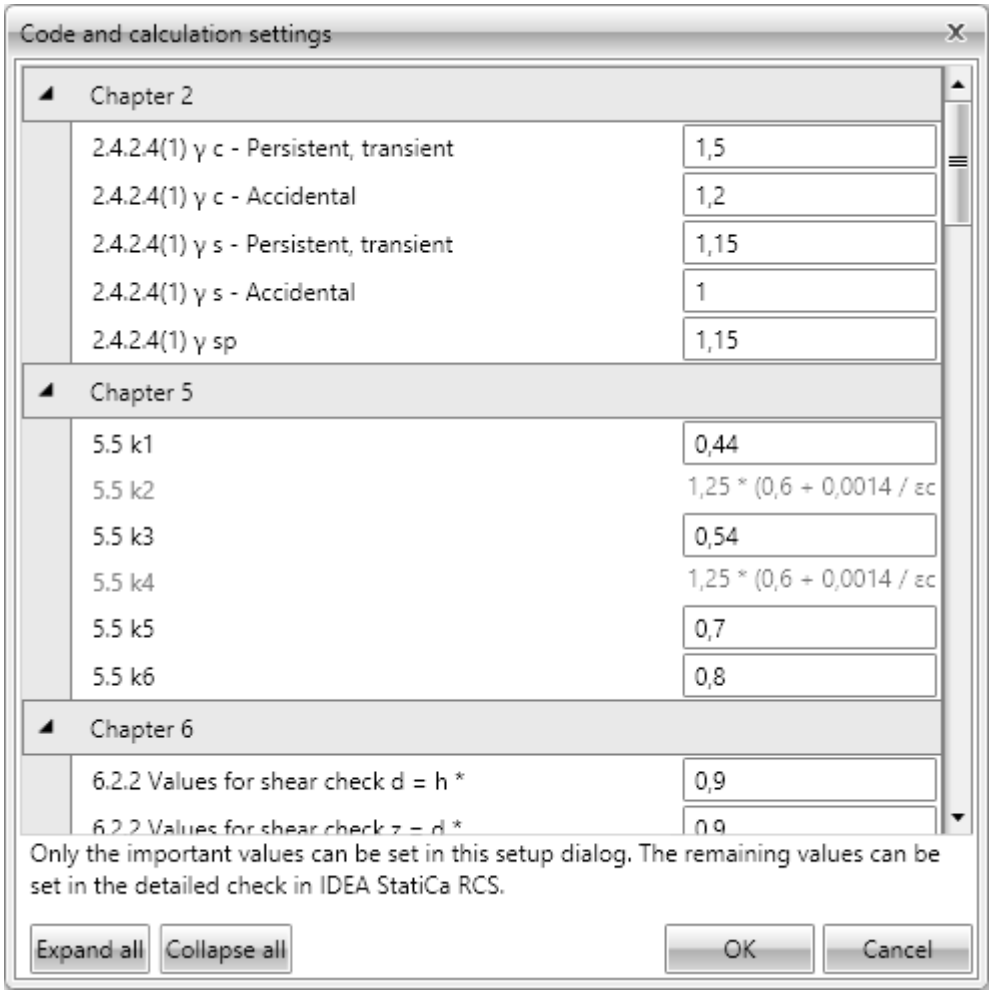
9.2 Settings for section checks and calculation of deflections

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

- **Code** – code and calculation settings – see **9.2.1 Code and calculation settings**.
- **Deflection settings** – settings of parameters of loads, which are used for calculation of deflections – see **9.2.2 Setting of result class for calculation of deflections**.
- **Section check RC** – settings of result classes for check of reinforce sections – see **9.2.3 Setting of result classes for section checks**.

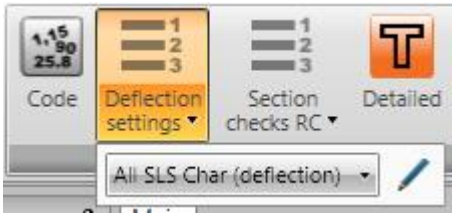
9.2.1 Code and calculation settings

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



9.2.2 Setting of result class for calculation of deflections

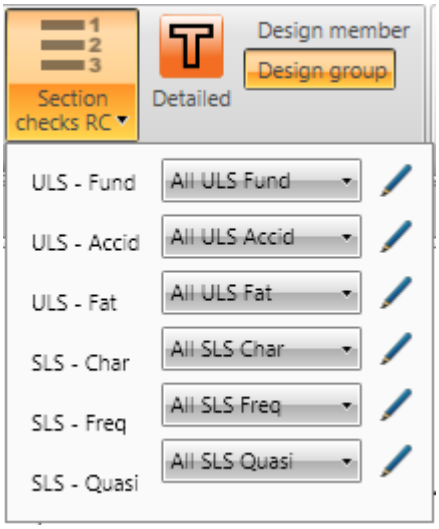
The deflections are calculated for all combinations assigned to the result class which is set to be used to calculation of deflections. The default result class for calculation of deflections is named **All SLS (Deflections)**. Combinations in this result class are considered as characteristic combinations.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



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

- **Result class** – select the result class for calculation of deflections. Click edit button to edit the content of the selected result class **9.2.4 Editing result class** .

9.2.3 Setting of result classes for section checks



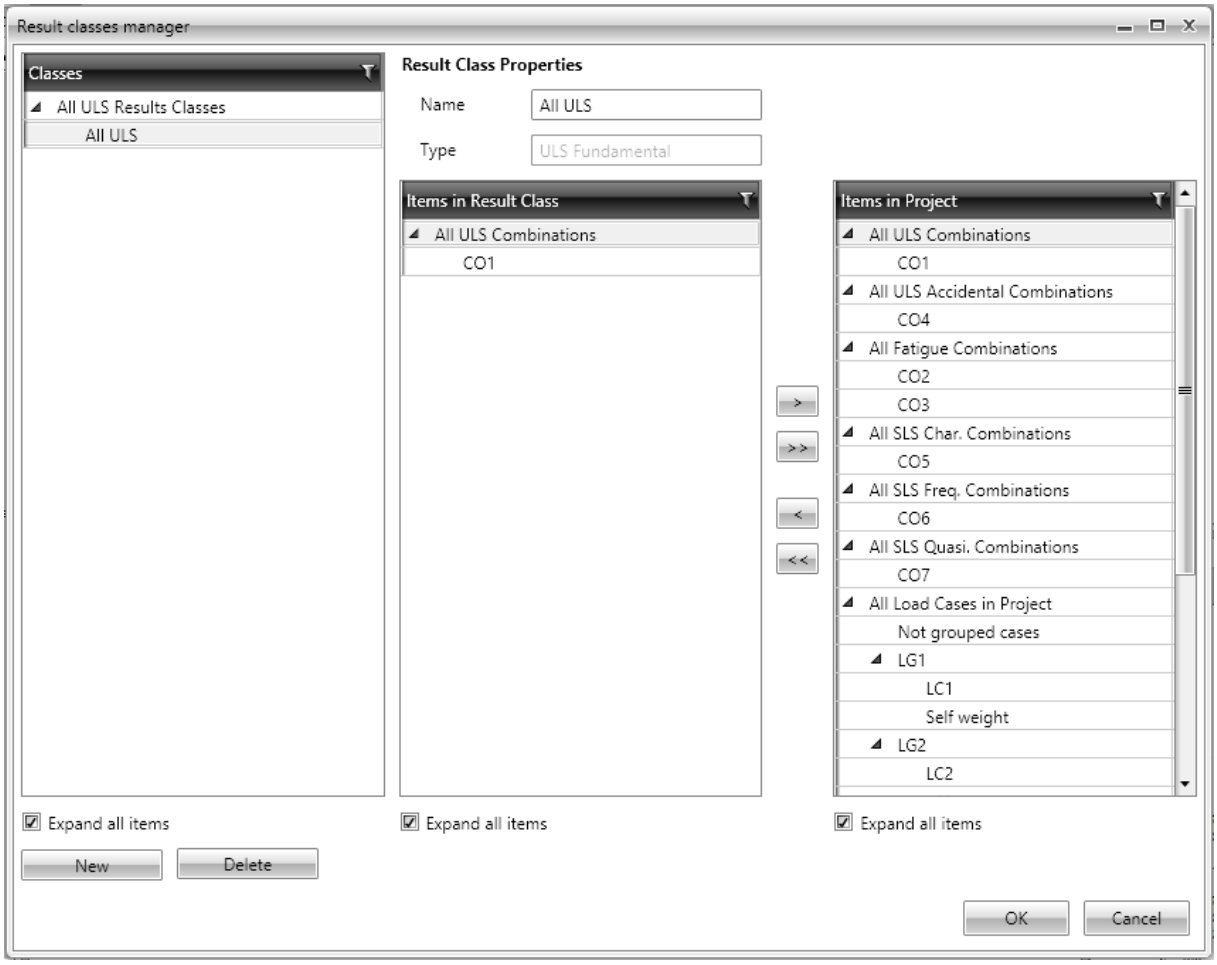
Result classes for sections check of current design member have to be defined before performing sections check.

Click **Sections check RC** in ribbon group **Concrete design** to set result classes. 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.

9.2.4 Editing result class

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



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

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

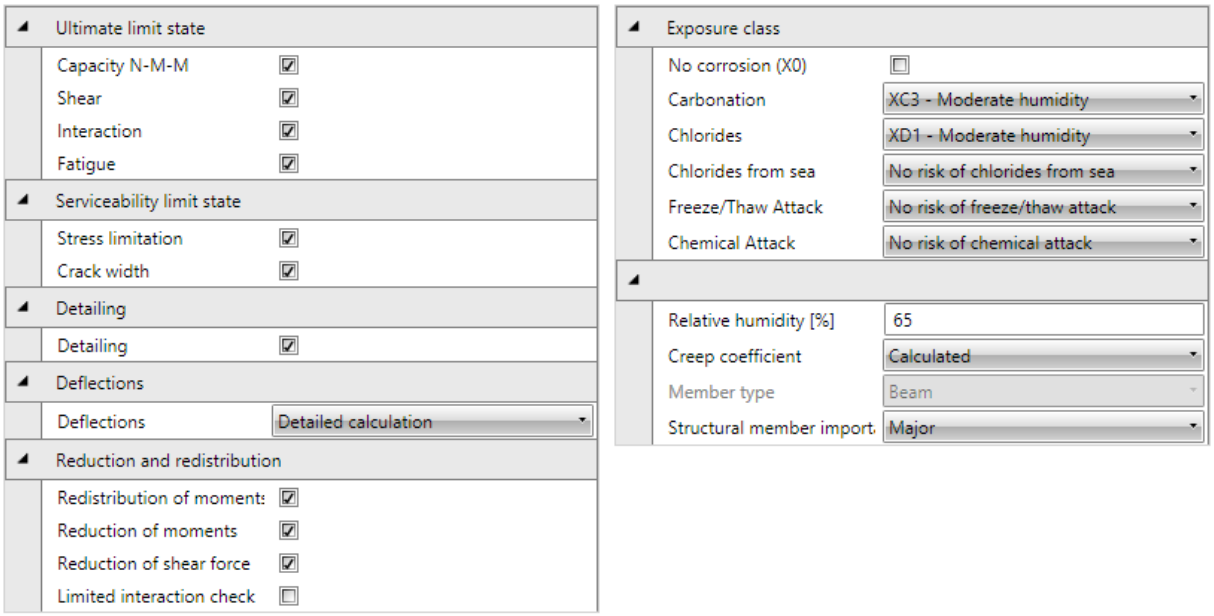
Combination properties:

- **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.

9.3 Design member data

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



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.
- **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 **Φinf** can be defined by user.
- **Structural member importance** – select type of structural member importance according to 6.2.1(4).

9.4 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 **9.4.2 Zone templates**.

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

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

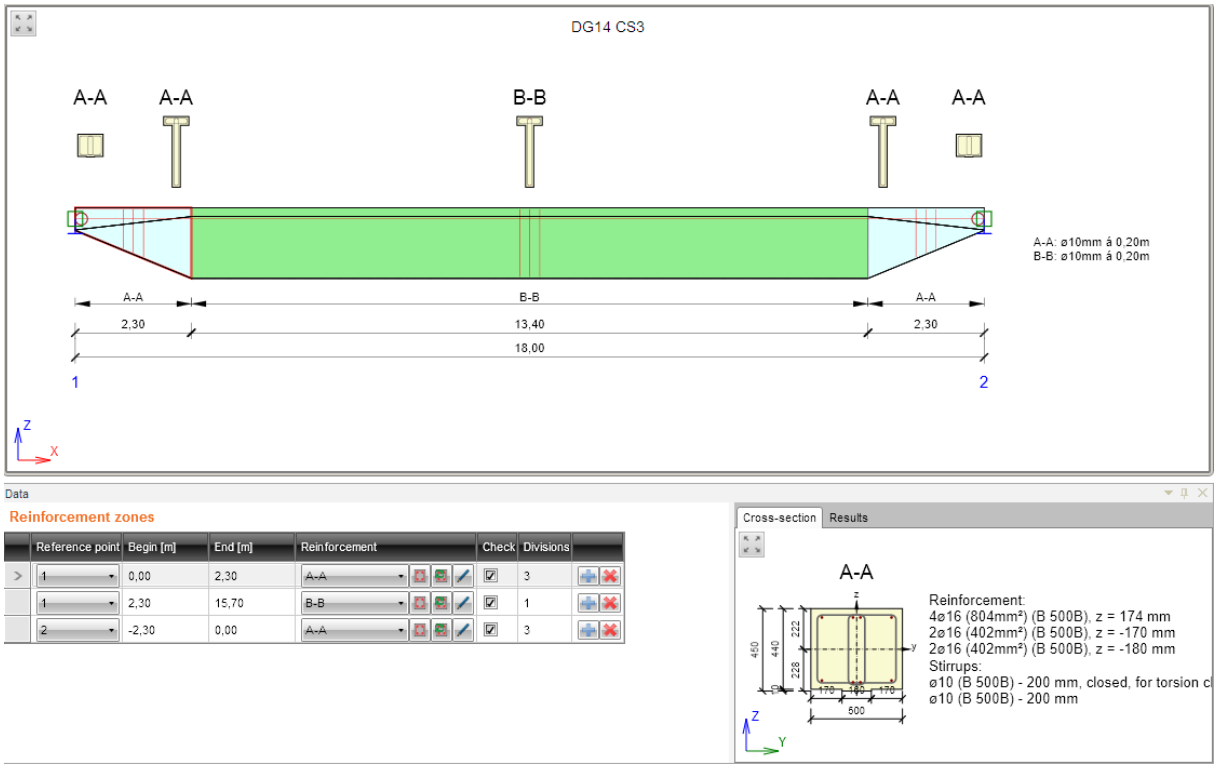


Table **Reinforcement zones** contains following columns:

- **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 **9.5 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.

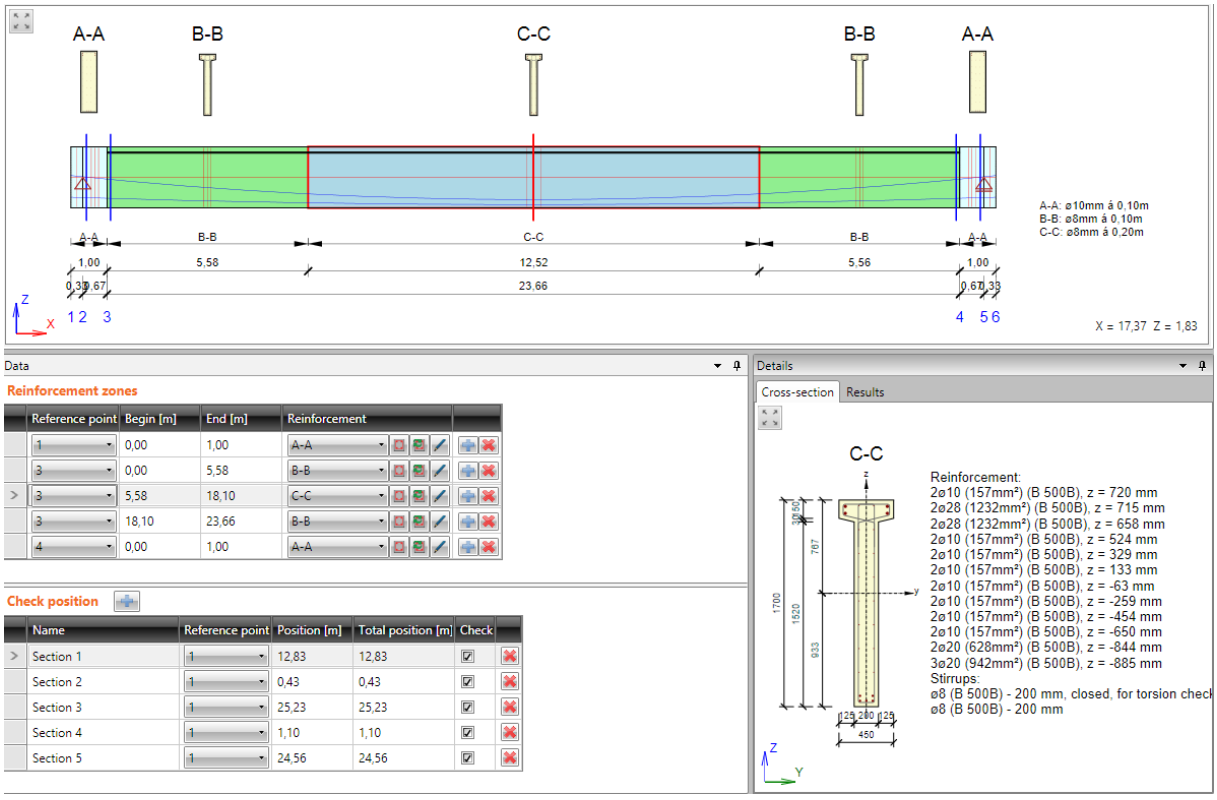
- **Check** – option, if the zone is checked or not. If the option is turned off, then no sections from this zone are generated in IDEA RCS.
- **Division** – input number of subzones, to which the current zone will be split. The sections for check are generated for each subzone.
-  - insert new zone. 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. Than the reinforcement is interpolated from the governing cross-section to the rest of haunch cross-sections.

9.4.1 Check positions for check of pre-stressed members



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:

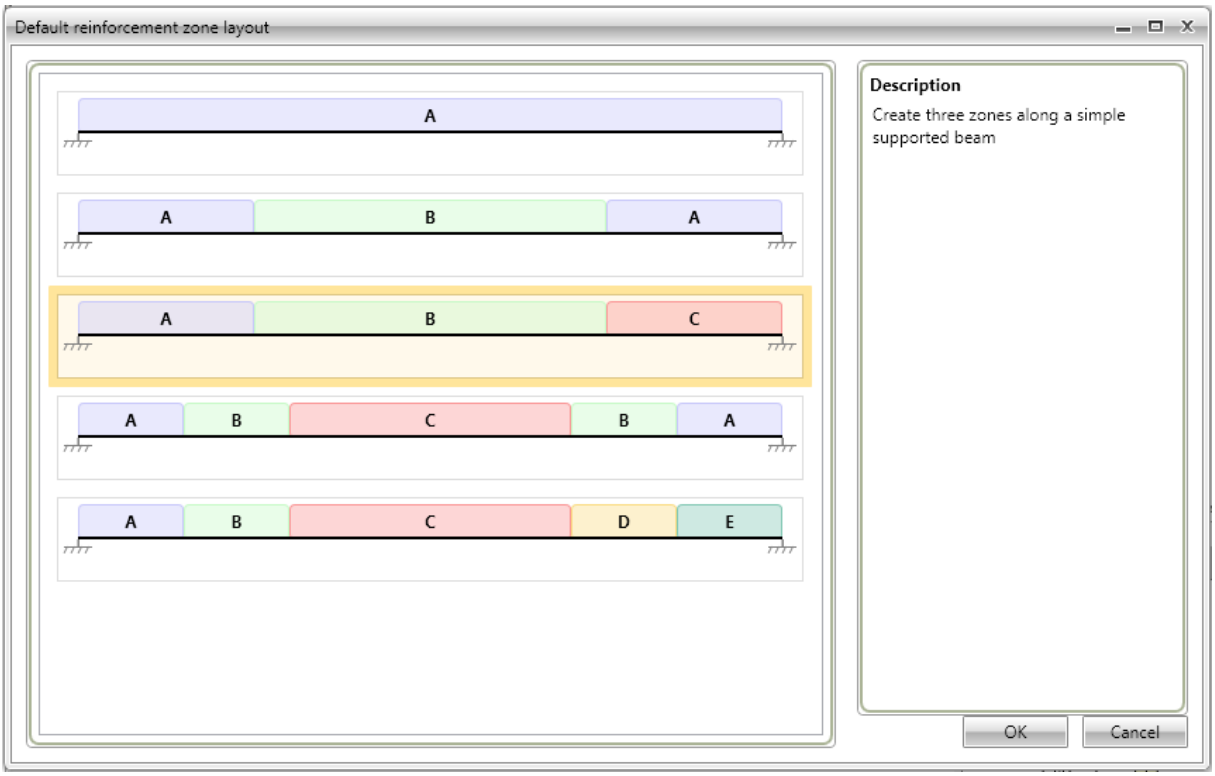
- **Name** – input the name of position. The name is used to generate the name of section in the IDEA RCS module.
- **Reference point** – select the point of the design member, which the distance of section is related to.
- **Position** – input the distance between the check position and the reference 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

9.4.2 Zone templates

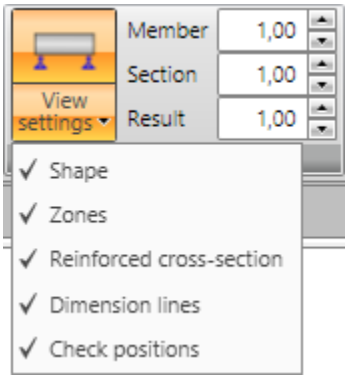
Zone templates can be used to generate zones along the design member. To generate zones according to templates click **Zone templates**.

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.



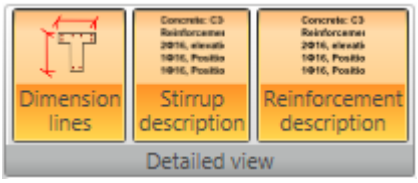
9.4.3 Ribbon group View settings and scale



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

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

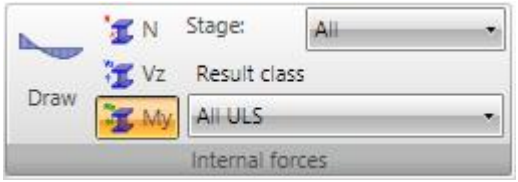
9.4.4 Ribbon group Detailed view



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.

9.4.5 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.
- **Vz** – switch to drawing of shear force Vz.
- **My** – switch to drawing of bending moment My.
- **Result class** – select the current result class, for which the internal forces courses are drawn.
- **Stage** – select the current construction stage to draw the internal forces. This setting is available only for prestressed design members.
- **Result class** – select the current result class to draw the internal forces.

9.4.6 Ribbon group Bill of material

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

9.5 Editor of reinforcement

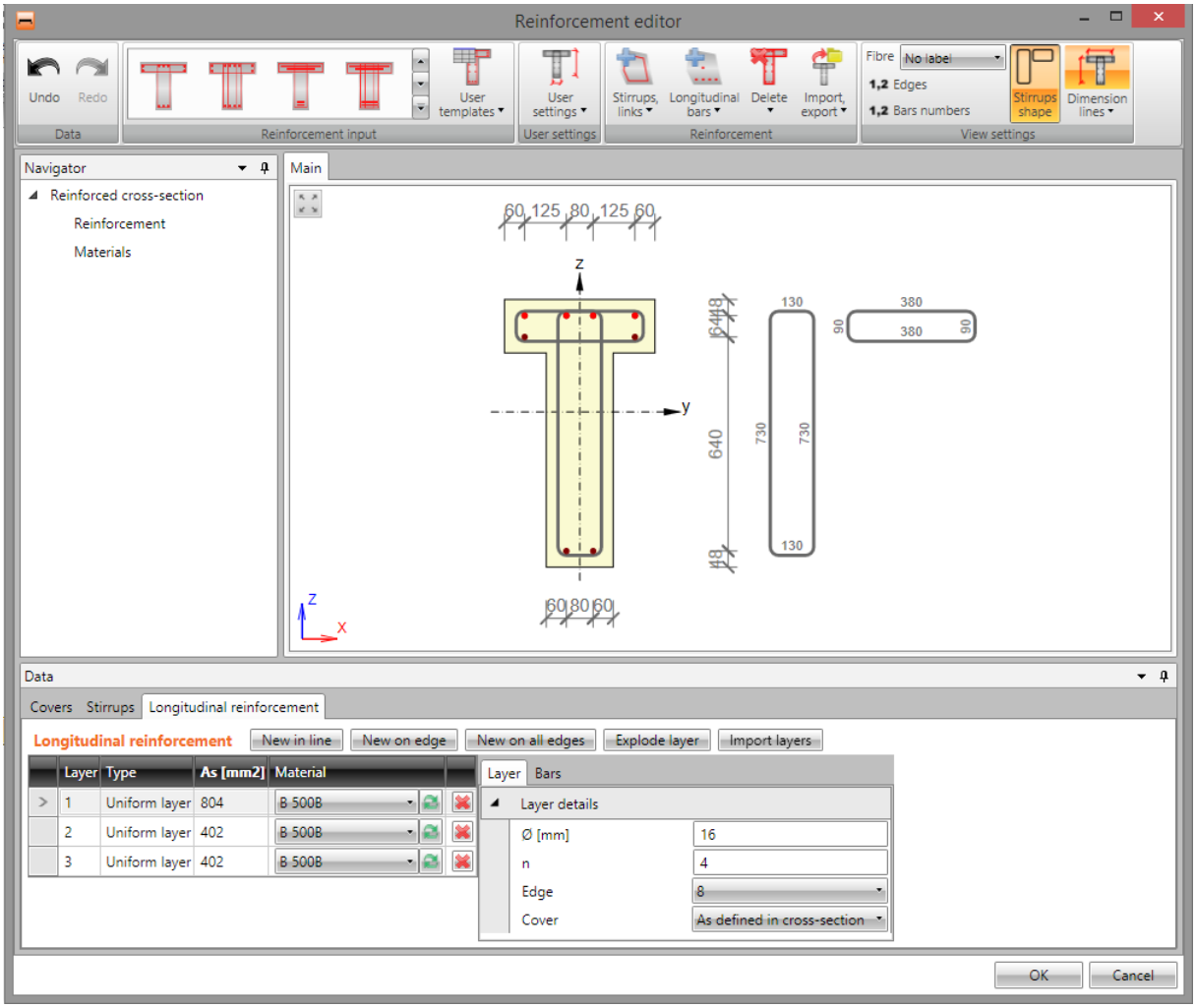
Input of current reinforcement template can be started

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

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.



9.5.1 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**.

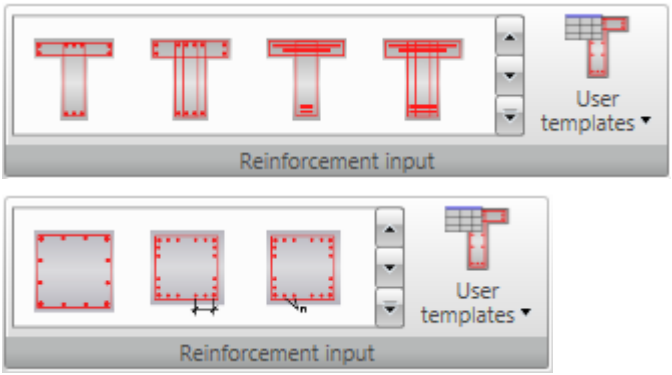
Cover can be defined:

- 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

9.5.2 Input of reinforcement by template

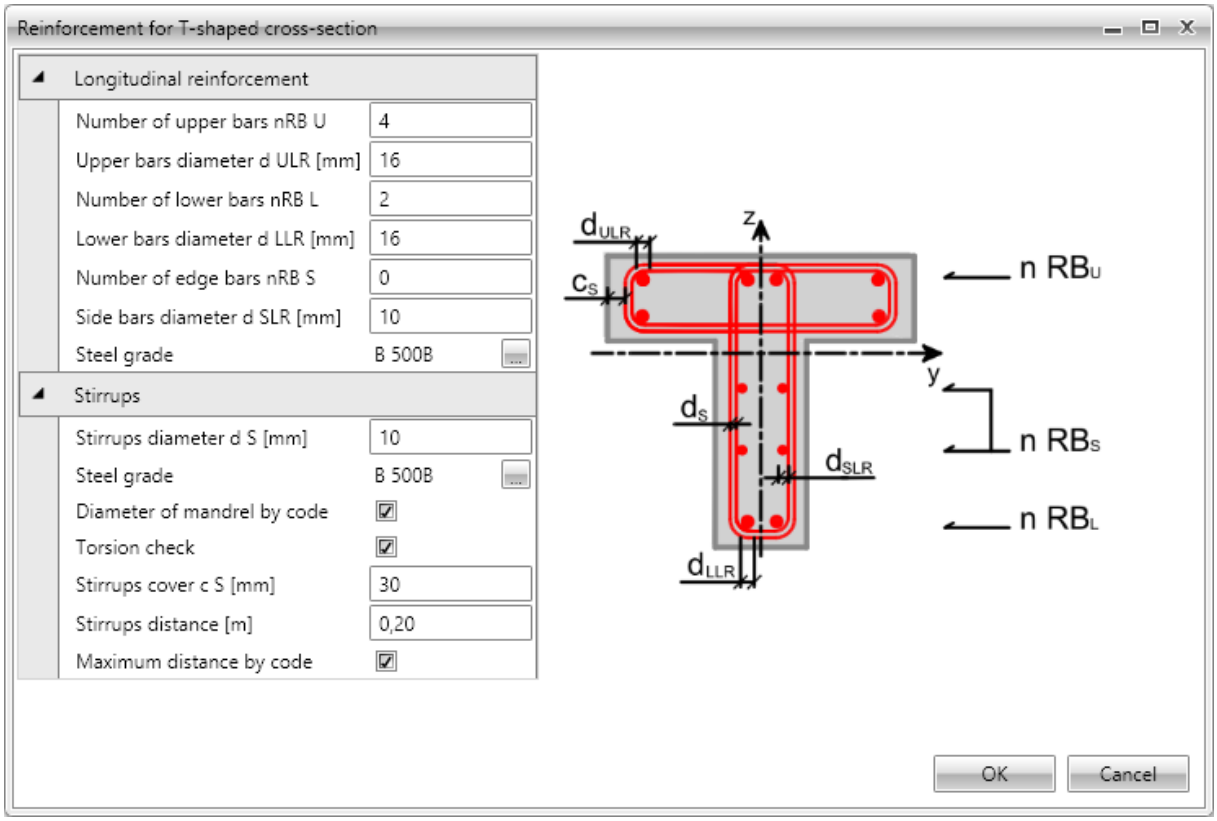


Reinforcement templates are available for some pre-defined sectional shapes. Reinforcement templates available for the current cross-section are displayed in ribbon group **Reinforcement**.

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 **9.5.10 User defined reinforcement templates**.

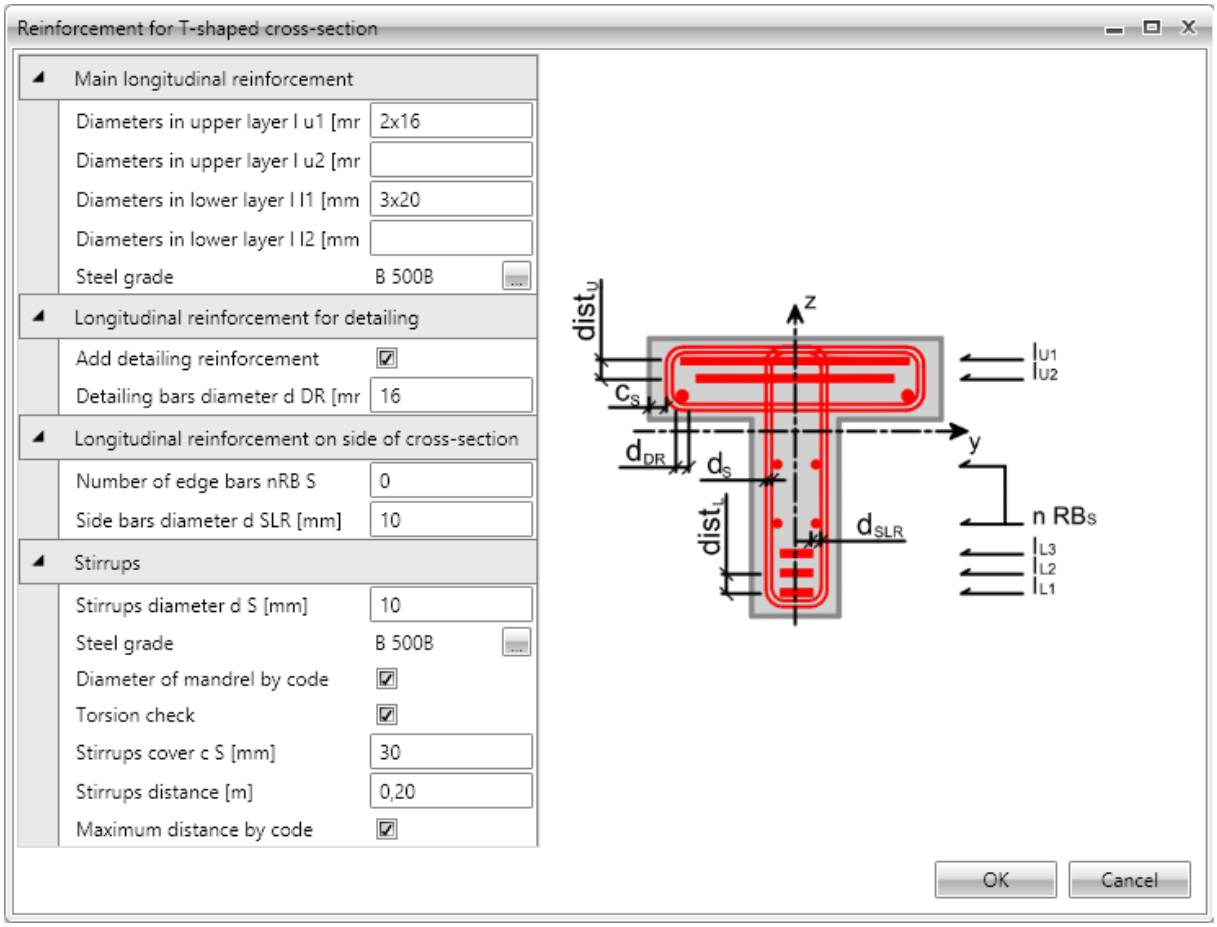
Reinforcement template parameters for T-shaped cross-section (including the reinforcement design):



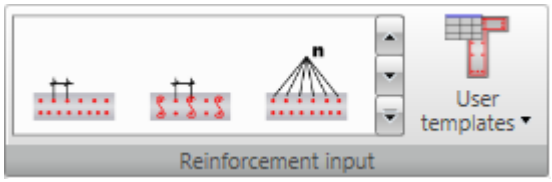
Click **OK** to add the reinforcement into the cross-section.

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‘.

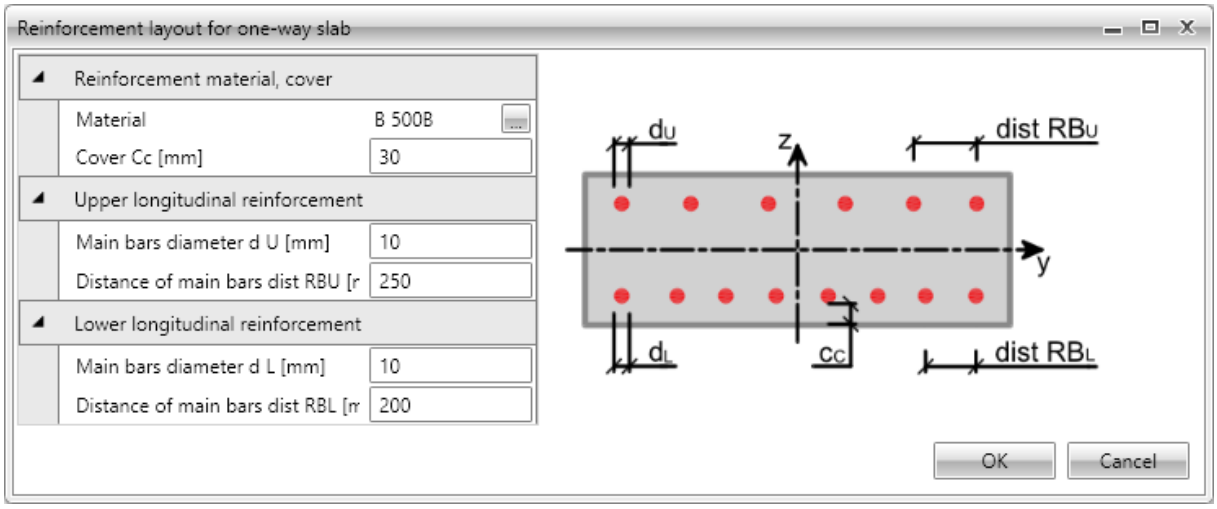


9.5.3 Input of one-way slabs reinforcement by a template



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 **9.5.10 User defined reinforcement templates**.

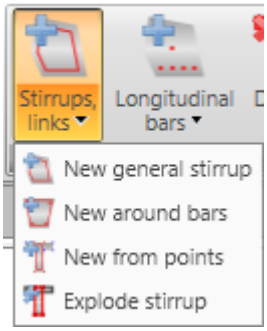


For one way slabs the distance between bars or bars number, bars diameter, bars material and concrete cover are defined in the template dialog.

9.5.4 Shear reinforcement

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

9.5.4.1 Stirrups



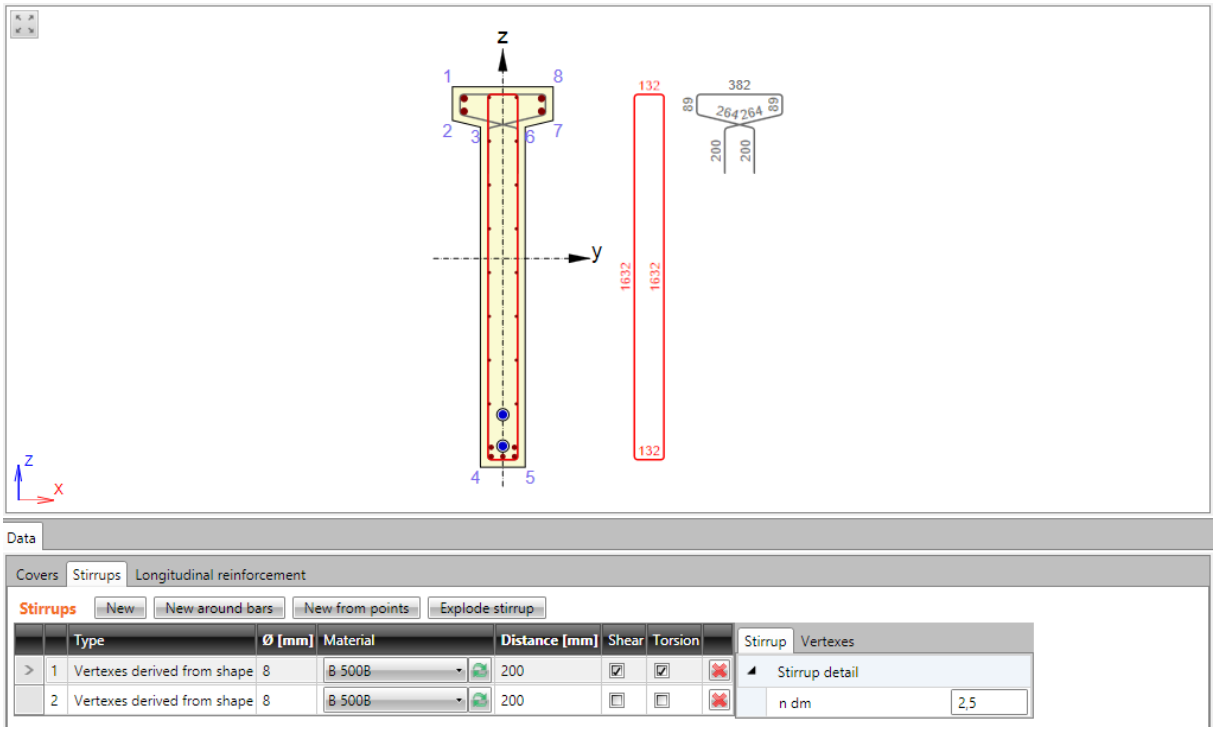
Drop-down button **Stirrups, links** in ribbon group **Reinforcement** collects commands for stirrups operations:

- **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
> 1	Vertexes derived from shape	8	B 500B	200	☑	☑
2	Vertexes derived from shape	8	B 500B	200	☐	☐

Stirrup | Vertexes

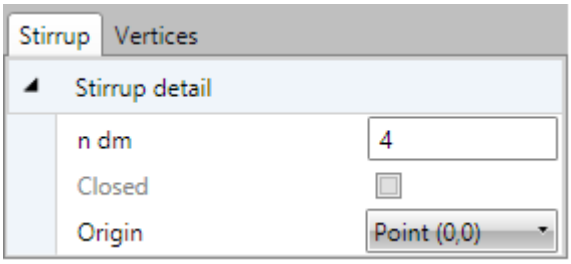
Stirrup detail

n dm 2,5

9.5.4.1.1. General stirrups

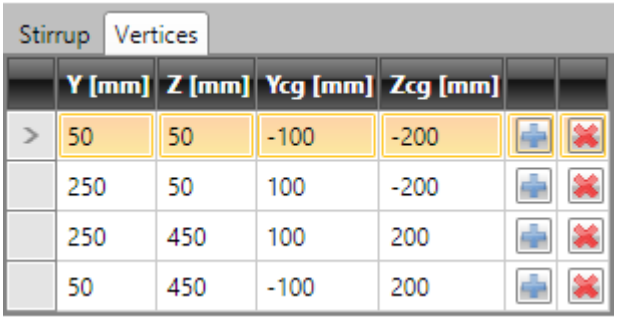
The general stirrup shape is defined by coordinates of the stirrup vertices. A vertex is the intersection of two stirrup branches axes.

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 it 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.



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.

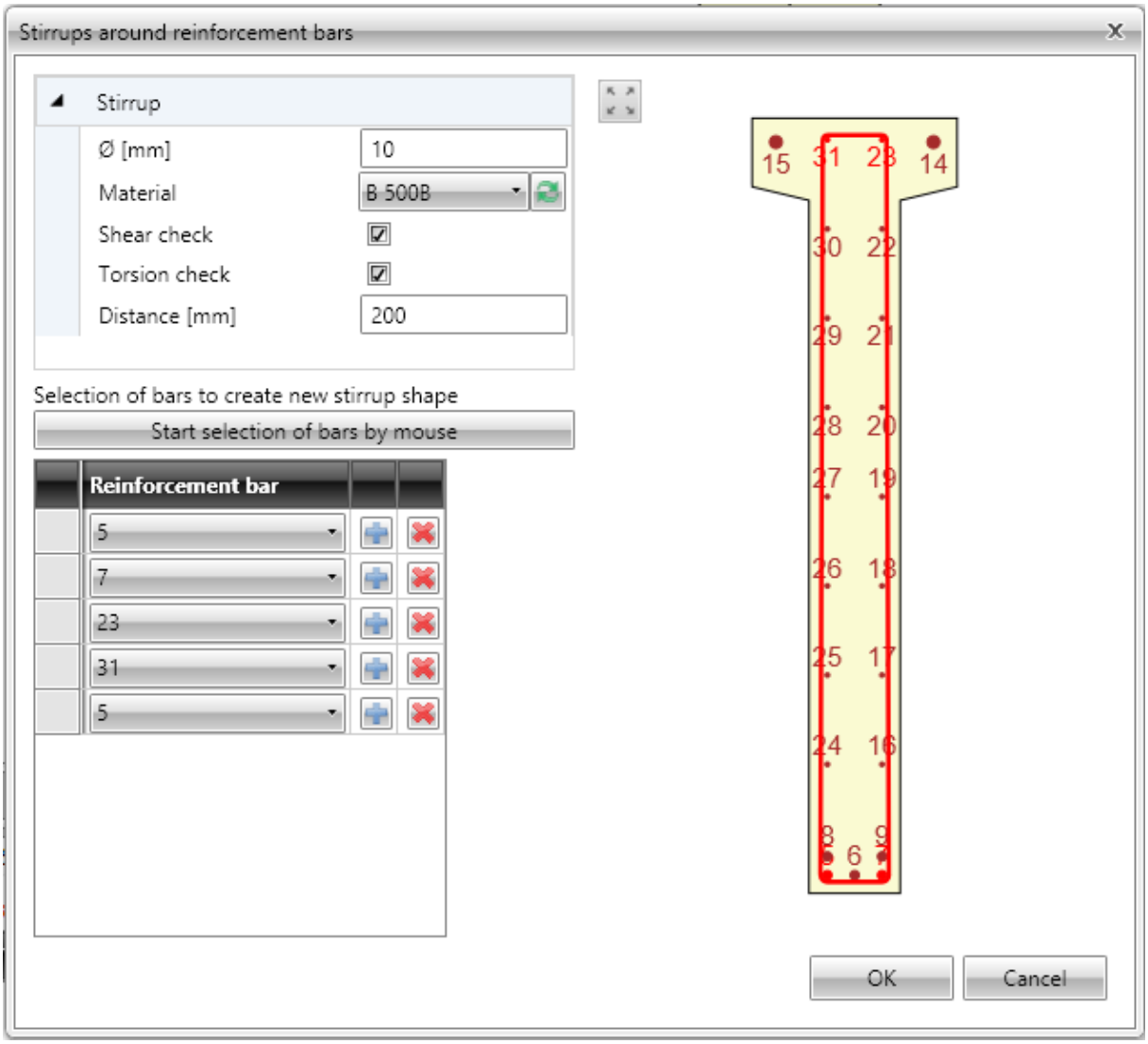
9.5.4.1.2. Stirrups around bars of longitudinal reinforcement

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.

9.5.4.1.3. Stirrup by cross-section vertices

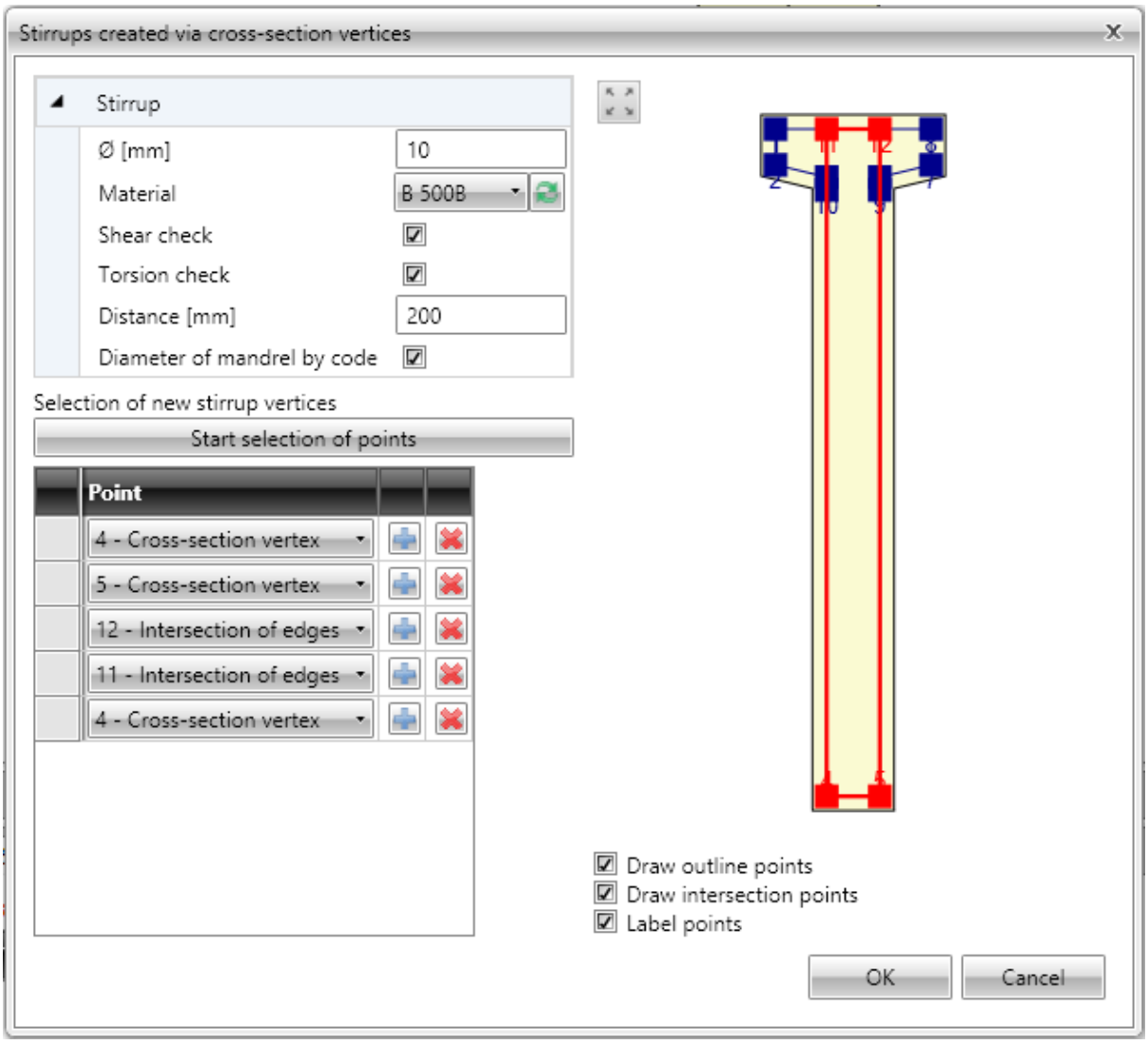
To input stirrup around main reinforcement click **Stirrups, links > New from points** in ribbon group **Reinforcement** or click **New from points** above the table **Stirrups**.

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.

9.5.4.1.4. Exploding stirrups

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

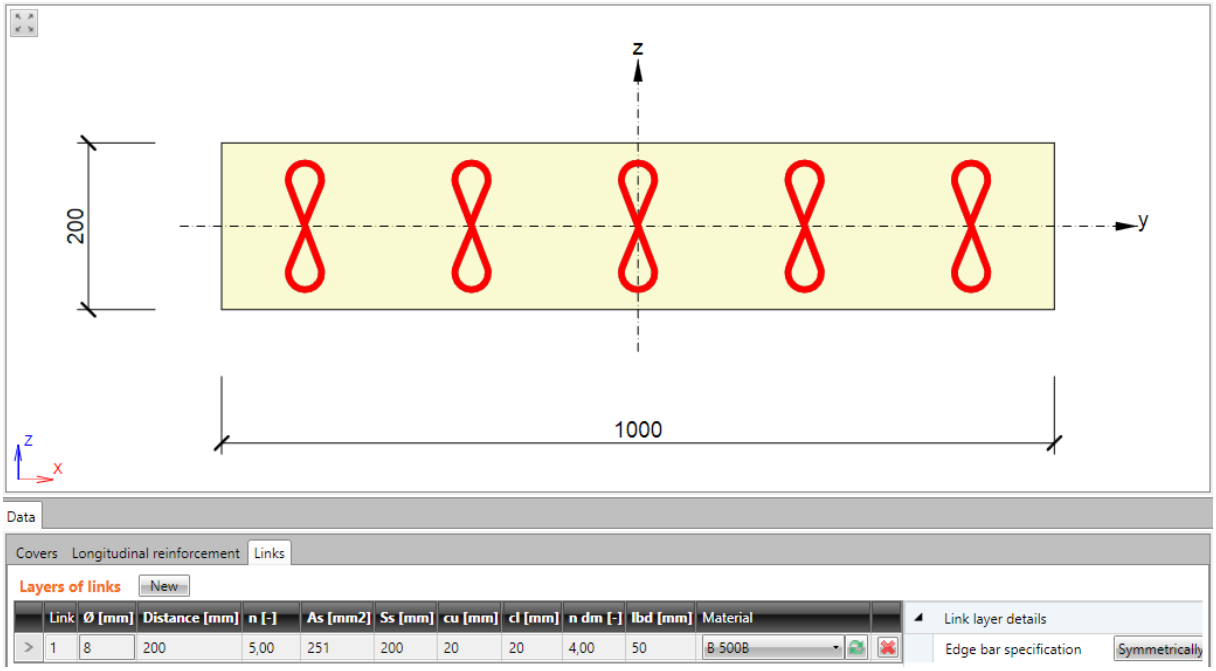
9.5.4.2 Links

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.

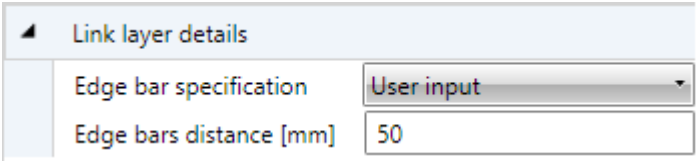
Columns in **Link layers** table:

- **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.



9.5.4.2.1. 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**.



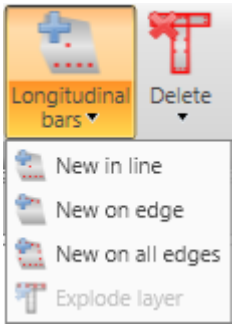
Properties group **Link layer details**:

- **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.

9.5.5 Longitudinal reinforcement

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



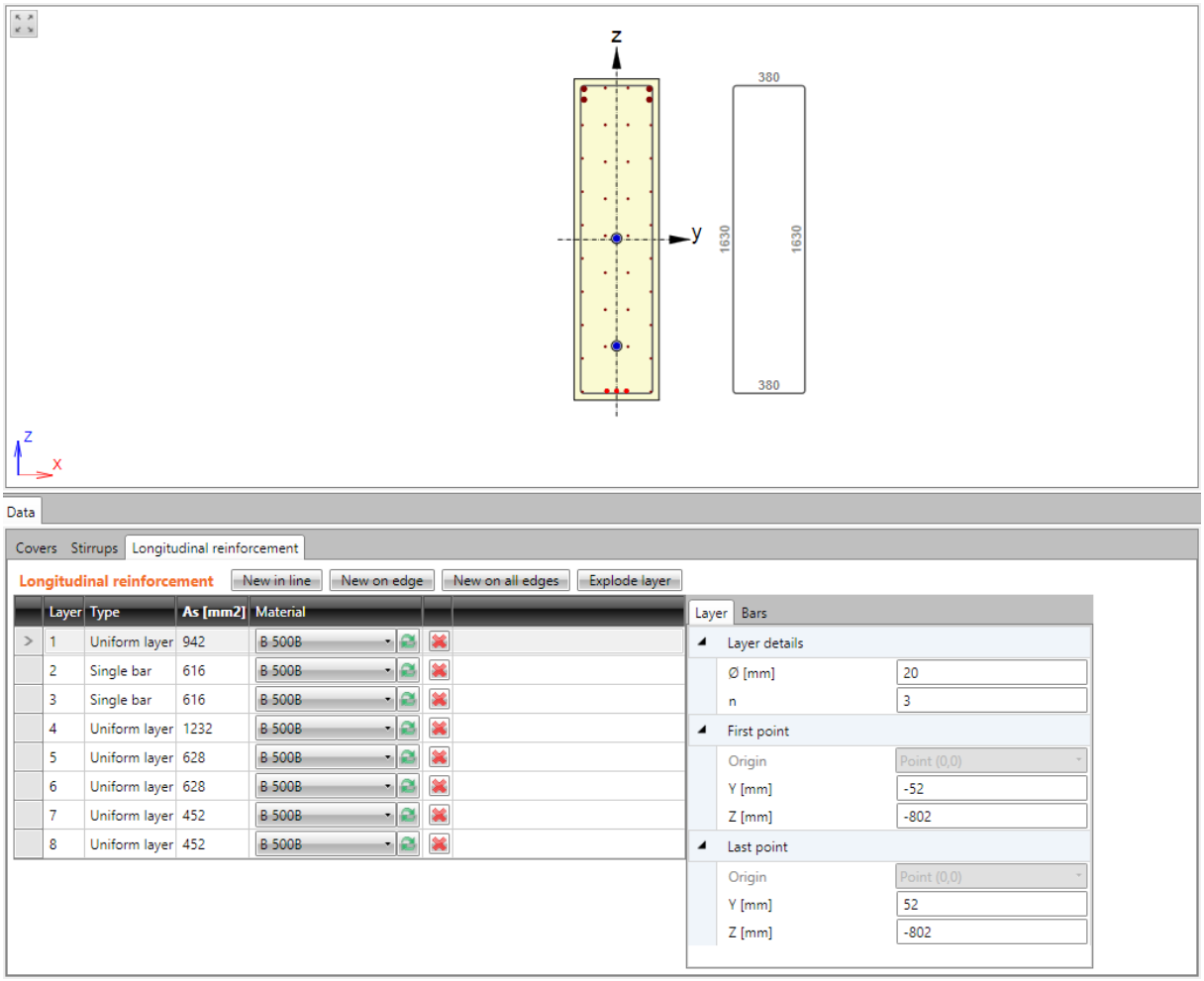
- **New in line** - adds a new layer of longitudinal reinforcement defined by coordinates of edge bars.
- **New on edge** - adds a new layer of longitudinal reinforcement related to cross-section edge.
- **New on all edges** - adds new layers of longitudinal reinforcement on all edges of cross-section.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

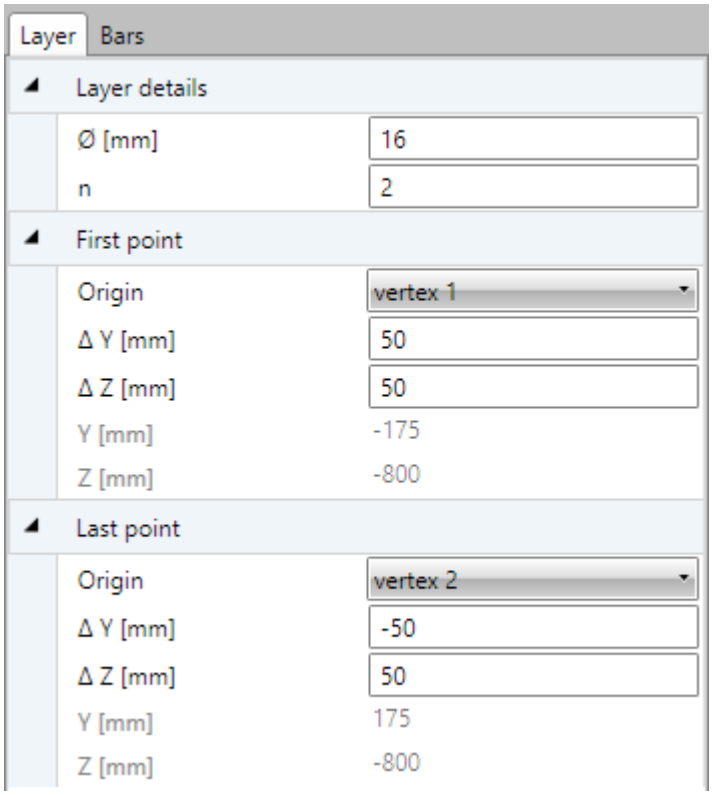


Columns in **Tendons** table:

- **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.

9.5.5.1 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**.



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**:

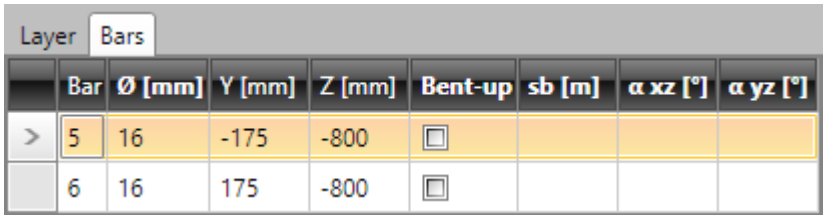
- **Origin** – select origin, to which coordinates of first bar in layer are related. Position of bar can be related to point [0;0] (center of gravity) or to selected cross-section vertex.
- **ΔY,**
- **Δ Z** – input distance of first bar in layer to the selected origin in direction of the corresponding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

9.5.5.1.1. Layer details

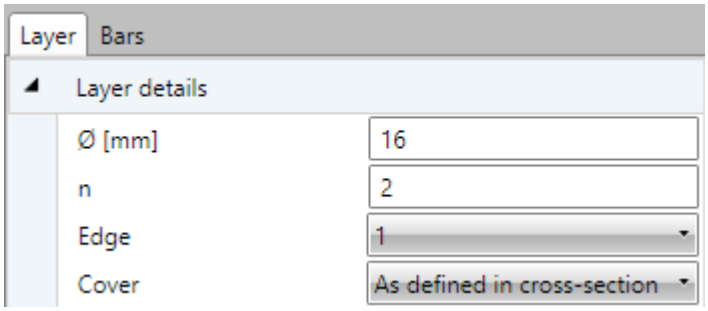


Properties of individual bars in current layer are displayed on tab **Bars**. Columns in the table:

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

9.5.5.2 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**.



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

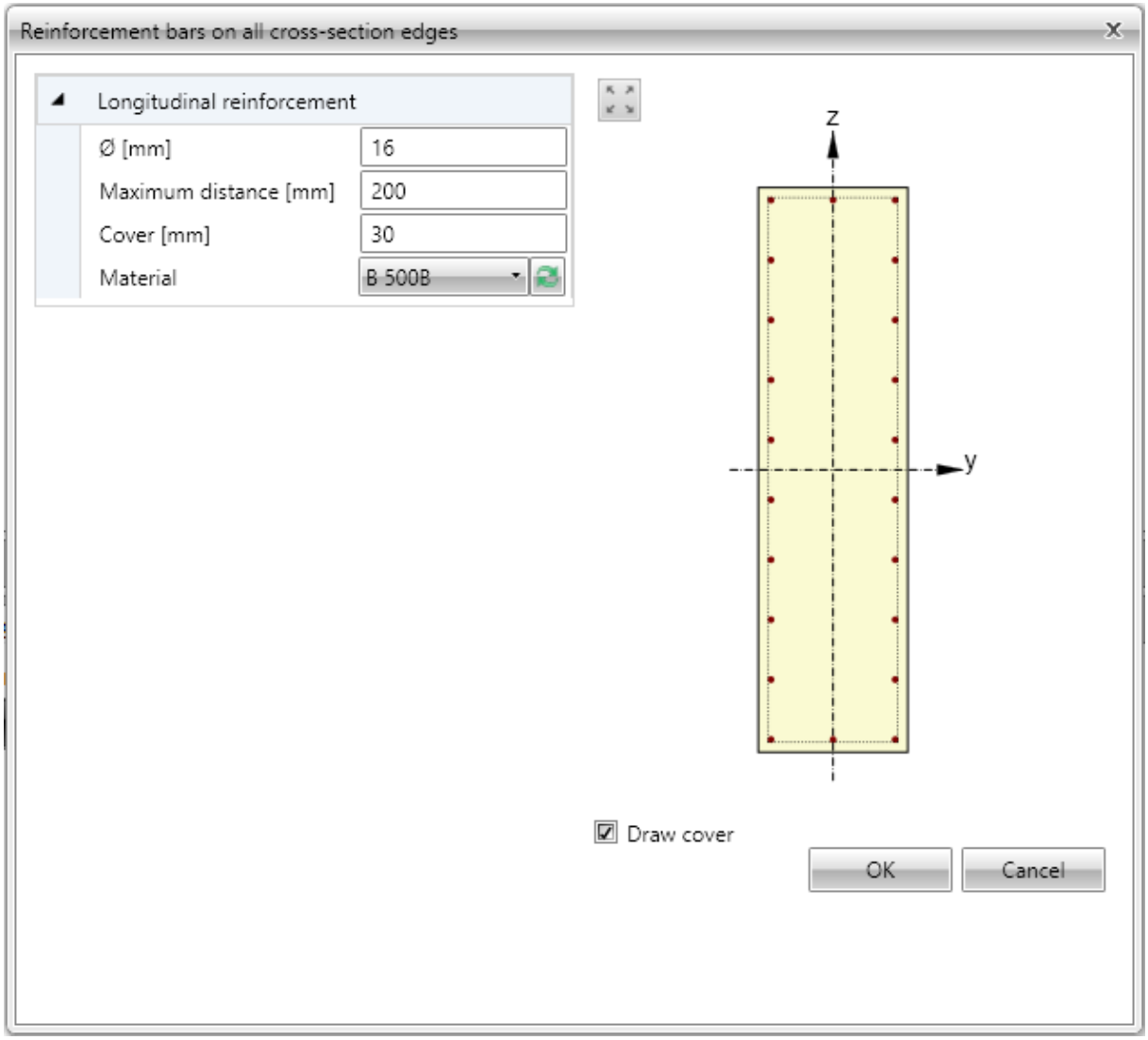
Properties group **Layer details:**

- **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 **9.5.5.1.1 Layer details**.

9.5.5.3 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.

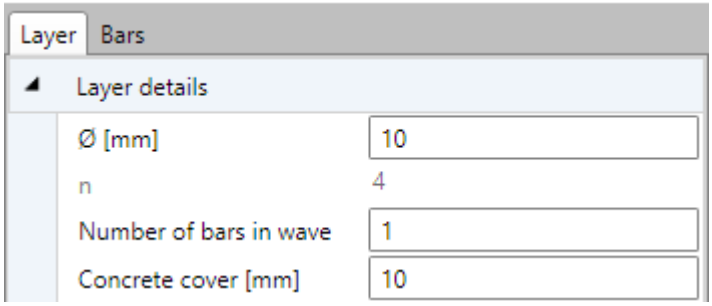
Individual dialog options:

- **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 **9.5.5.1.1 Layer details**.

9.5.5.4 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**.



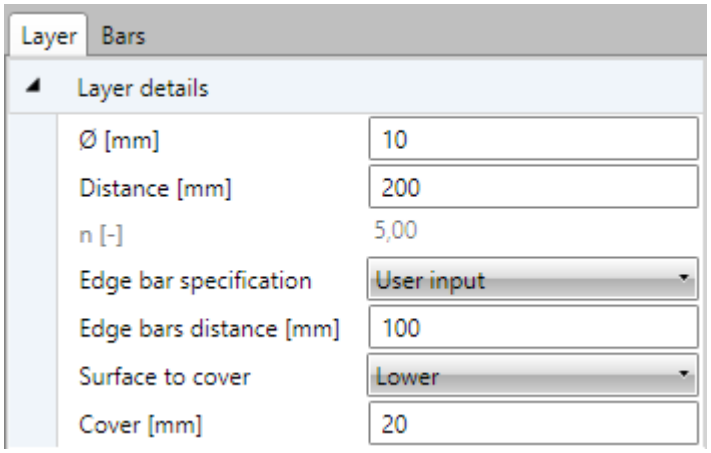
Properties group **Layer details**:

- **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 **9.5.5.1.1 Layer details**.

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

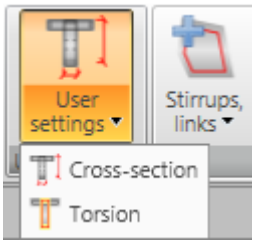
Properties group **Layer details**:

- **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 **9.5.5.1.1 Layer details**.

9.5.6 User settings of reinforced cross-section



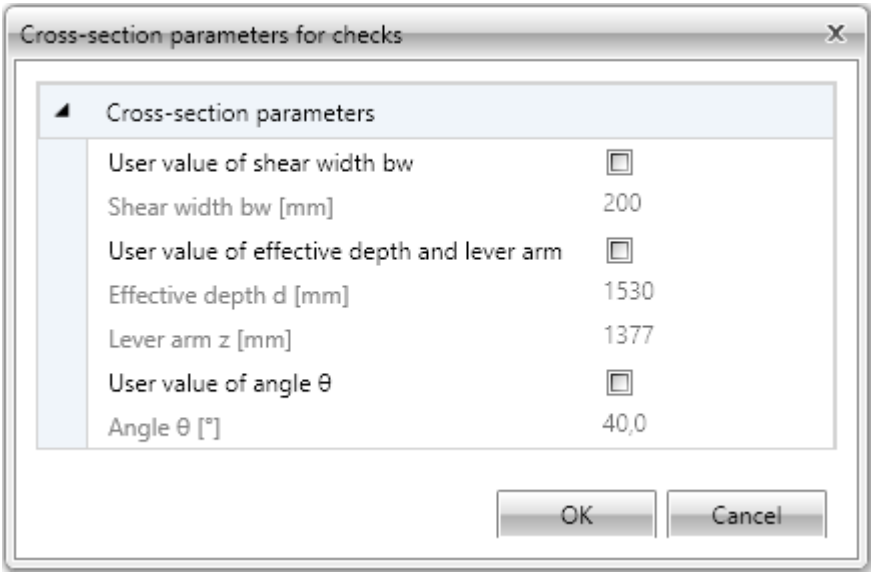
Ribbon group **User setting** contains commads to modify some shear and torsion calculation parameters of reinforced cross-section:

- **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.

9.5.6.1 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**.

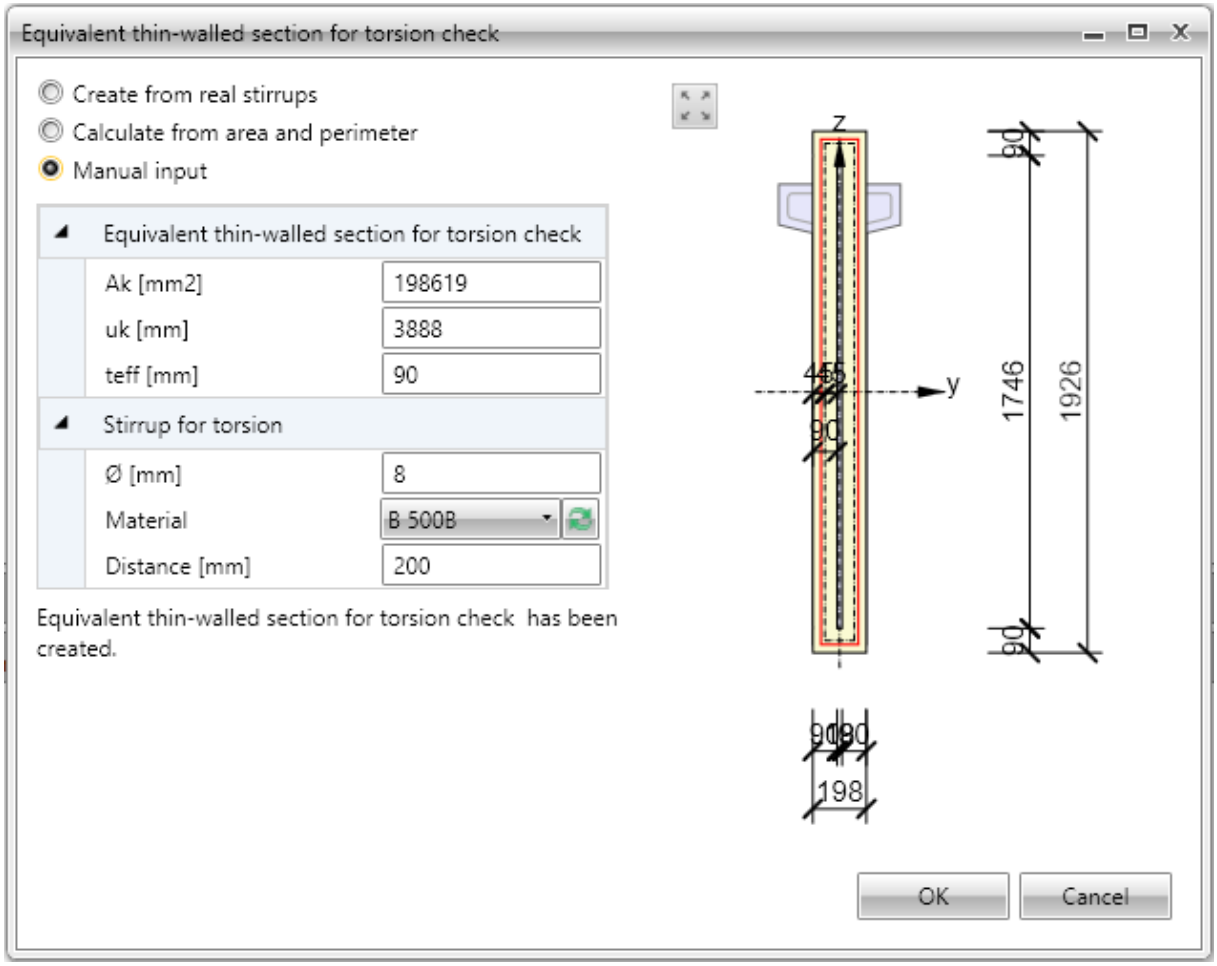


Calculated values of effective cross-section dimensions and overridable values of shear check from code and calculation settings are displayed in dialog. To enable input of user defined values check particular checkbox in first column.

9.5.6.2 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.



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.

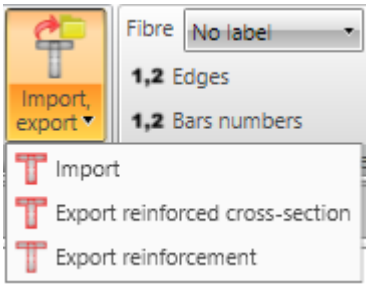
9.5.7 Deleting reinforcement

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

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

9.5.8 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.

9.5.9 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.

9.5.10 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:

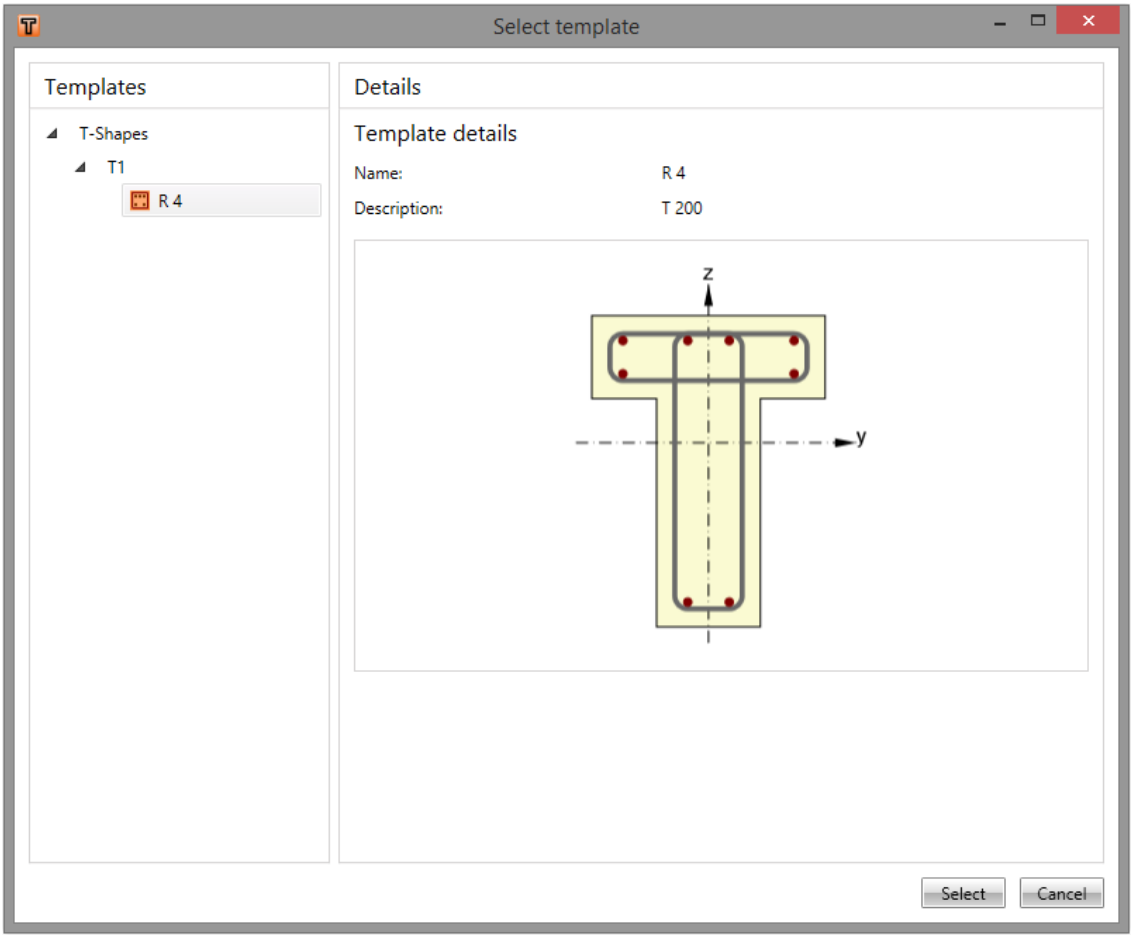
-  – starts input of reinforcement using the user defined reinforcement template – see **9.5.10.1 Reinforcing by user defined reinforcement template**.
-  – stores the current reinforcement into the database of user reinforcement templates. Dialog **Add template** appears. The target folder must be selected 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 **9.5.10.2Templates manager**.

9.5.10.1 Reinforcing by user defined reinforcement template

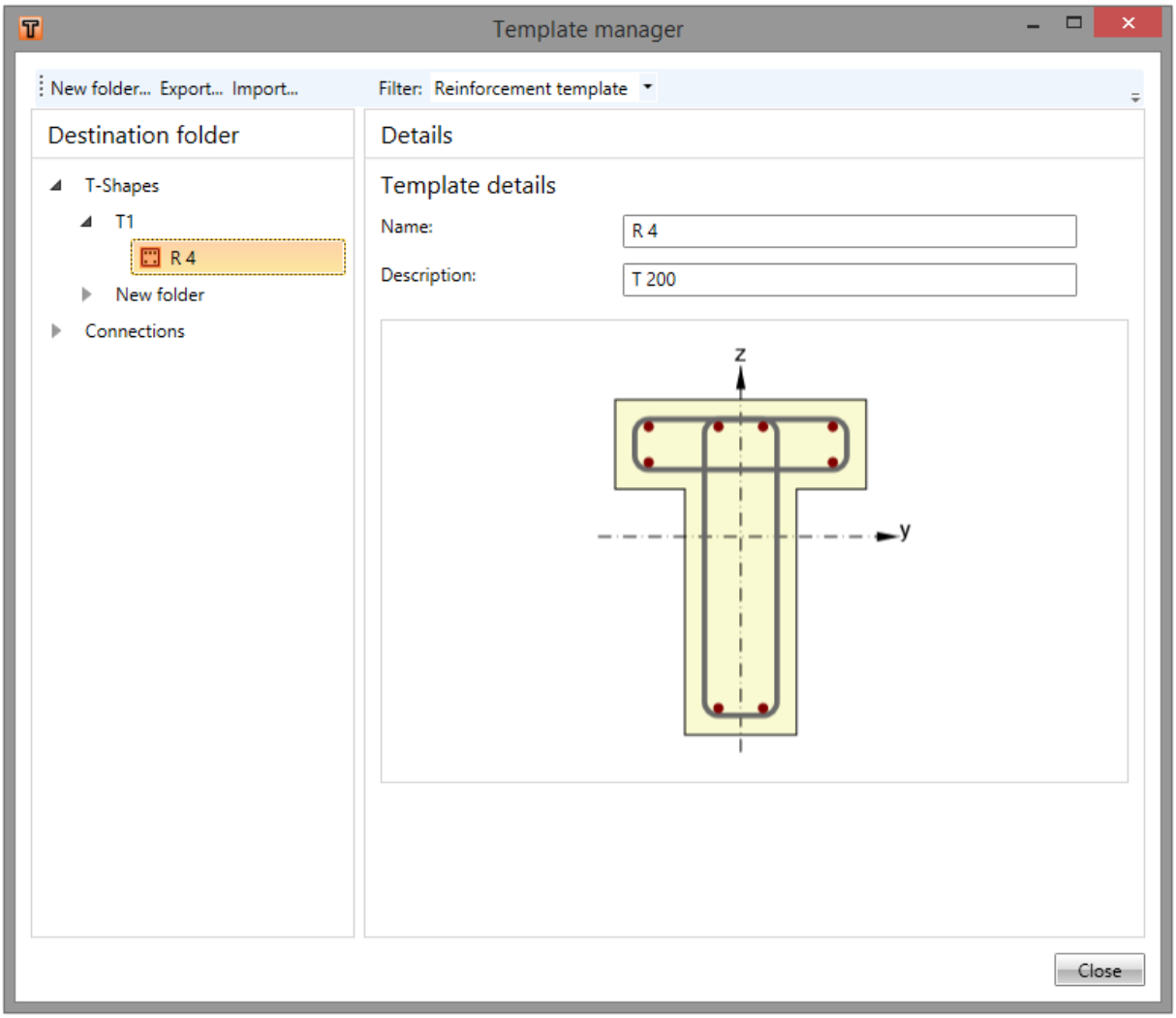
Dialog **Select template** appears after start of reinforcing by user defined reinforcement template.

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.


Select template
Templates
T-Shapes
T1
R 4
Details
Template details
Name:
R 4
Description:
T 200
z
y
Select
Cancel

9.5.10.2 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.

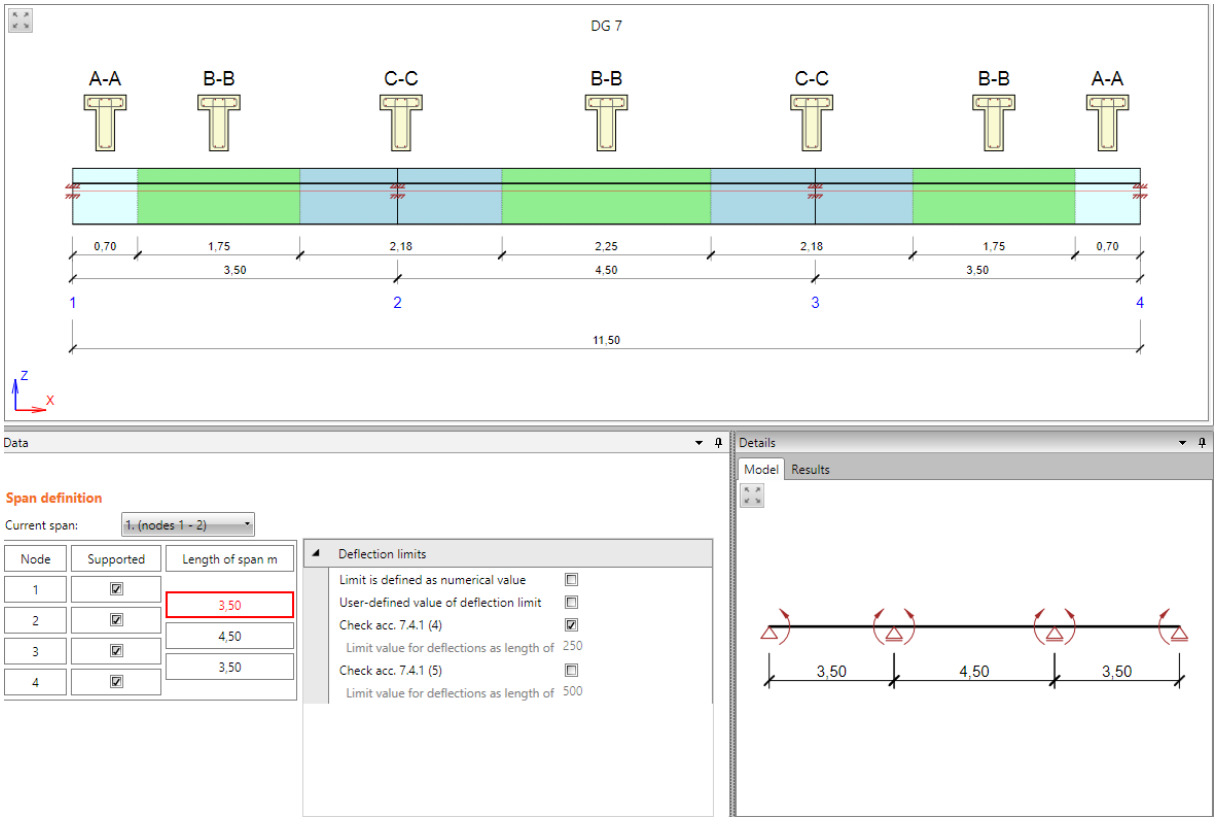
Import templates – by command **Import…** in the main menu. Templates from the selected file with extension *.EXP are imported into the database of templates.

9.6 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 effect calculation and deflection check of multi-spans columns are defined in IDEA Frame.

9.6.1 Data for deflection 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.

- **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**.

9.6.1.1 Deflection check settings for the current span

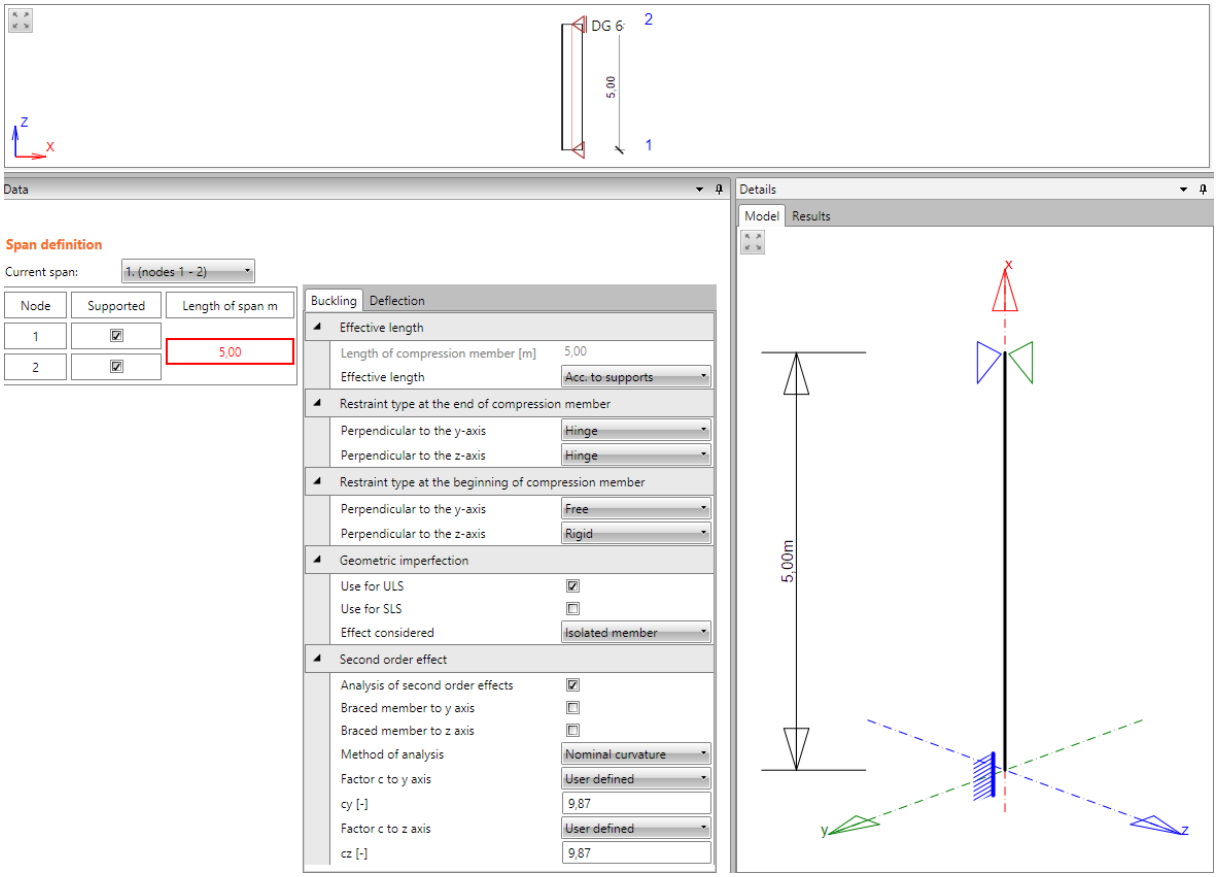
For the current span following parameters can be set in **Deflection limits** properties group:

- **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).

9.6.2 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**.

9.6.2.1 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.

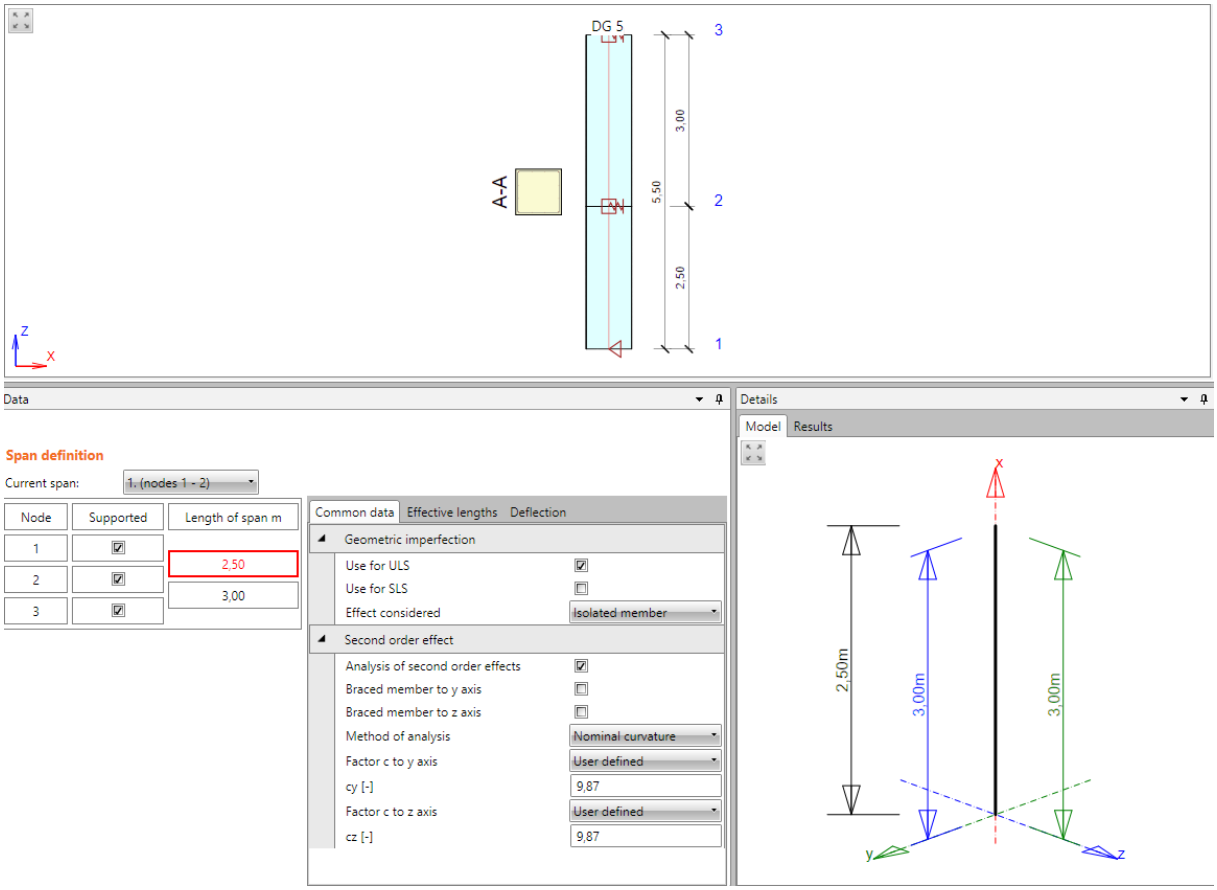
9.6.2.2 Deflection check settings

The deflection check settings for the current span can be defined on **Deflection** tab - see **9.6.1.1 Deflection check settings for the current span**.

9.6.1 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.



- **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**.

9.6.1.1 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.

9.6.1.2 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.

9.6.1.3 Deflection check settings

The deflection check settings for the current span can be defined on **Deflection** tab - see **9.6.1.1 Deflection check settings for the current span**.

9.7 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.

9.7.1 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**.

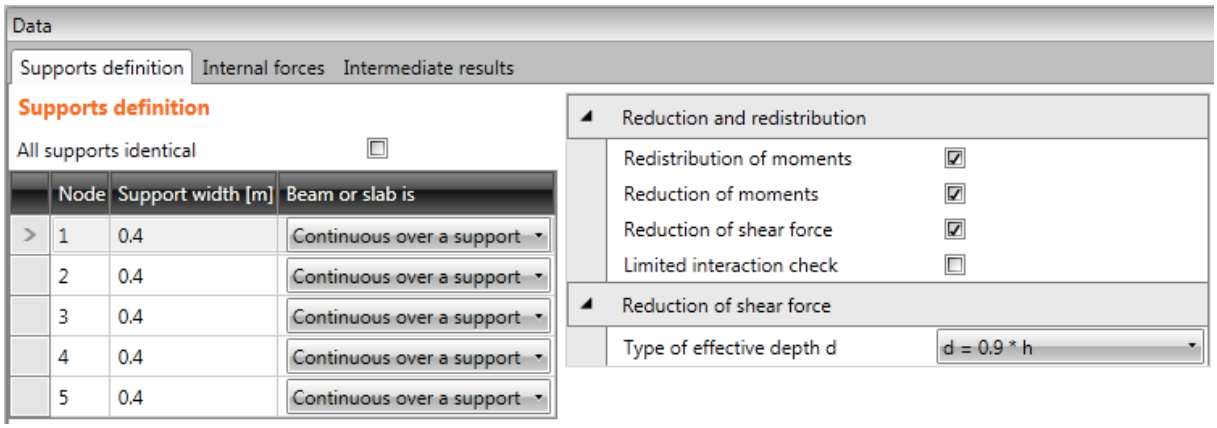
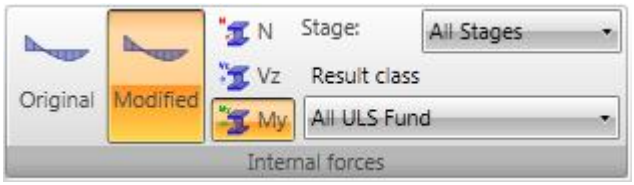


Table **Supports definition**:

- **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).

9.7.2 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 Vz.
- **My** – switches to draw the course of bending moment My.

9.8 Detailed check

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

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

- **Design member** – if the option is selected, separate sections are generated for each design member in the design group.
- **Design group** – if the option is selected, sections are generated for the representative design member only. The internal forces are merged from all design members in the design group to the load extremes at generated sections.

9.9 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.
  - **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 groups **Stage** and **Check** are available for evaluation of section check results.

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

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

9.9.1 Ribbon group Concrete design

See **9.2 Settings for section checks and calculation of deflections** and **9.8 Detailed check**.

9.9.2 Ribbon group View settings and scale

See **9.4.3 Ribbon group View settings and scale**.

9.9.3 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.

9.9.4 Ribbon group Calculation

- **Redistribution, reduction** – run calculation of redistributed and reduced internal forces on the current design member. The command is

available after change of parameters of redistribution and reduction calculation.

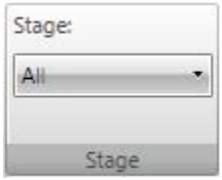
- **All** – run calculation of checks results of the current design member. This command is available, if the check results were deleted e.g. after the calculation settings were changed.

9.9.5 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.

9.9.6 Ribbon group Stage



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

9.9.7 Drawing of section check results courses

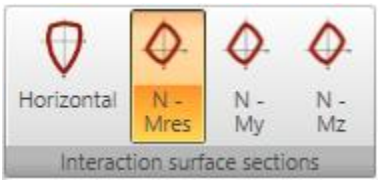
9.9.7.1 Ribbon group Check

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

- **Overall** – switch to drawing of envelope of extremes of all performed checks.
- **Capacity N-M-M** – switch to drawing of course of one of available components of capacity check – check value, bending moments of resistance, axial force of resistance.
- **Shear** – switch to drawing of course of one of available components of shear check – check value, $V_{Rd,c}$, $V_{Rd,max}$, $V_{Rd,s}$.
- **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}, de, de_{lim}.

9.9.8 Drawing of interaction diagrams

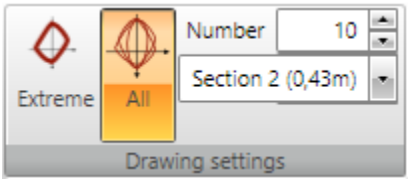
9.9.8.1 Ribbon group Interaction surface sections



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

- **Horizontal** – switch to draw the horizontal section of intersection surface through the point Ned, 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,MEd,z) parallel with the plane N-My.
- **N–Mz** – switch to draw the vertical section of intersection surface through the point (0,0,MEd,y) parallel with the plane N-Mz.

9.9.8.2 Ribbon group Drawing settings



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

9.9.8.3 Ribbon group Colors settings

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

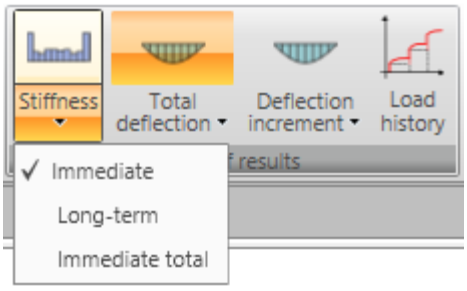
- **Standard** – switch to draw all interaction diagrams in one color – default color for drawing of interaction diagrams.
- **Different colors** – switch to draw each interaction diagram in different color.
- **Legend** – switch on/off drawing of legend describing the points which represents the design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

9.9.9 Drawing of deflections check results

9.9.9.1 Ribbon group Combination

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

9.9.9.2 Ribbon group Type of results



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

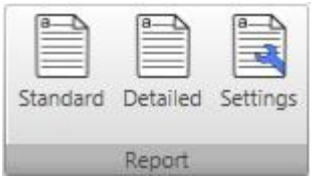
- **Stiffness** – switch to drawing of calculated stiffnes for the current combination along the design member:
  - **Immediate** – switch to drawing of stiffness for calculation of immediate effects of long-term load components for the current combination.
  - **Long-term** – switch to drawing of stiffness for calculation of long-term effect of long-term load components for the current combination.
  - **Immediate total** – switch to drawing of stiffness for calculation of immediate effect of total load for the current combination
- **Total deflection** – switch to drawing of calculated total deflections for the current combination along the design member:
  - **Linear** – switch to drawing of deflections from linear calculation for the current combination.
  - **Immediate** – switch to drawing of immediate deflections (calculated from short-term stiffness) from total load for the current combination.
  - **Long-term** – switch to drawing of long-term deflections (calculated including the effects of creep) from the long-term loads for the current combination.
  - **Total** – switch to drawing of total deflections (calculated including the effects of creep) for the current combination.
  - **Limit** – switch to drawing of limit deflections.
- **Deflection increment** – switch to drawing of calculated deflection increments for the current combination along the design member:
  - **Increment** – switch to drawing of deflection increments for the current combination.
  - **Limit** – switch to drawing of limit of deflection increments.
- **Load history** – switch to drawing of load history of current combination for deflection check. Load history diagram shows load cases and their contribution to deflection calculation and the effective time of their application.

9.9.9.3 Ribbon group Stiffness

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

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

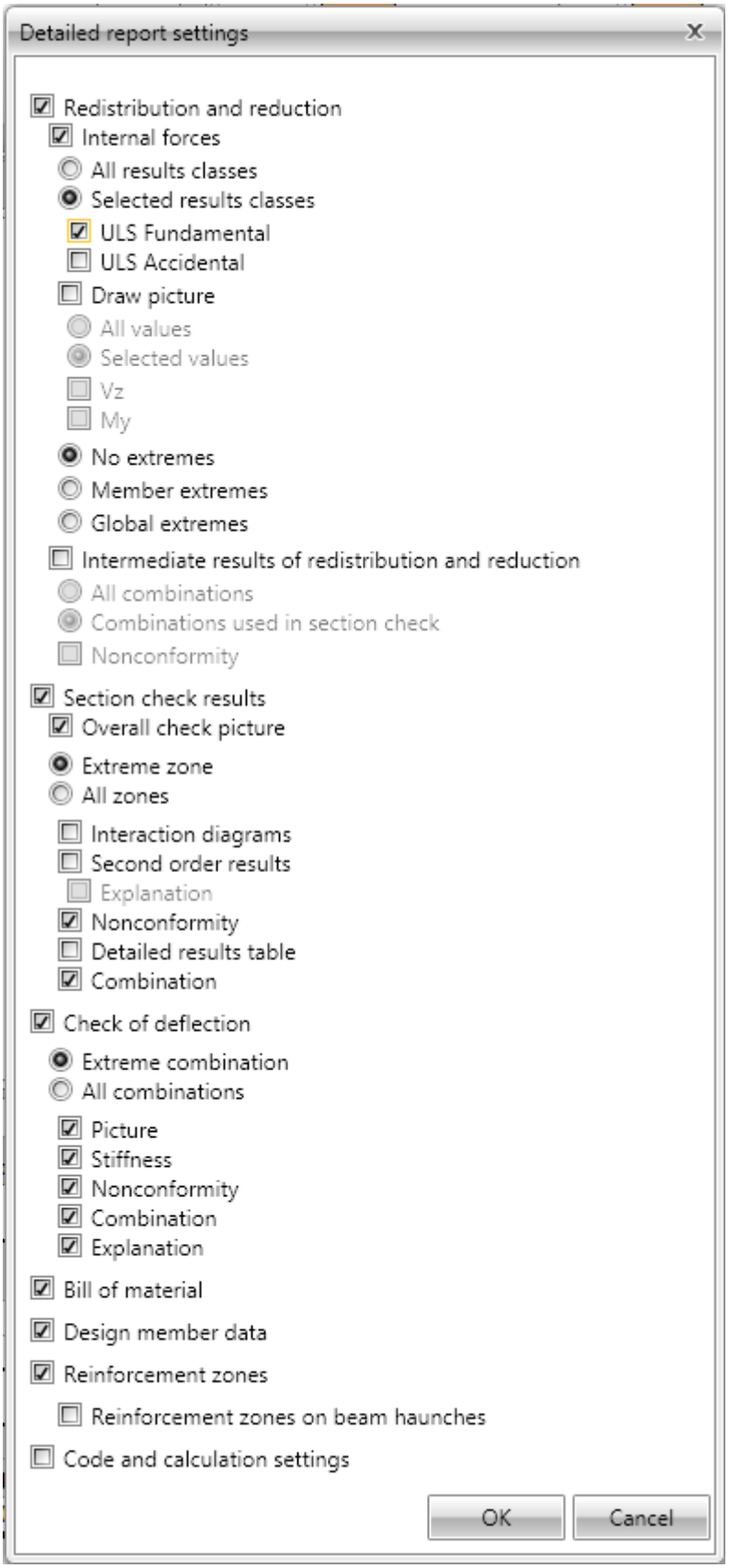
9.9.10 Check report



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

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

9.9.10.1 Report settings



To set the content of detailed 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.

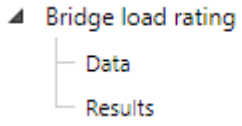
  - **Extreme combinations** – if the option is selected, results of deflection check will be printed for the combination, which caused the extreme check value**.**
  - **All combinations** – if the option is selected, results of deflection check will be printed for all combinations.
  - **Picture** – 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.

10 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 raging 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.

10.1 Input data of bridge load rating

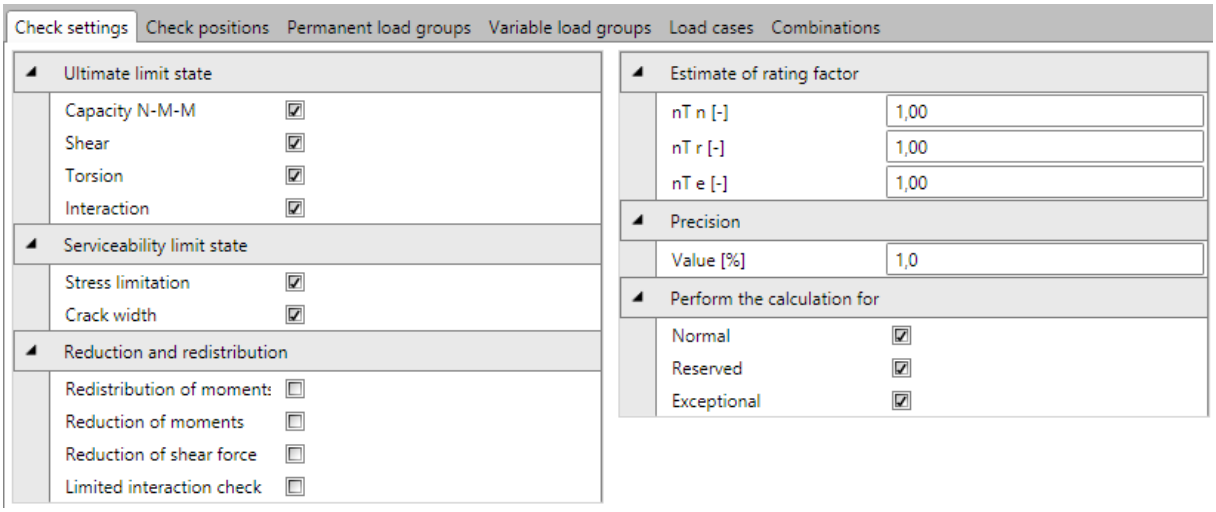
Click navigator command **Bridge load rating > Data** to start to define input data of bridge load rating calculation. Individual tabs to define the input data are displayed in the Data window.

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.

10.1.1 Check settings



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.

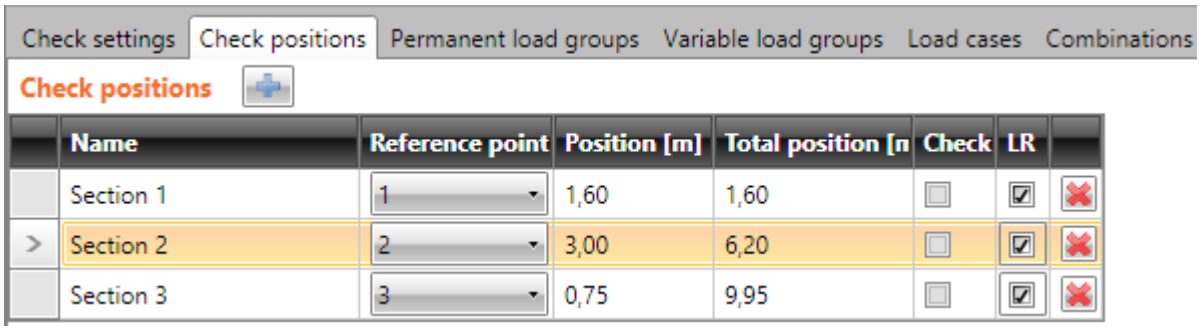
Properties group **Perform the calculation for**

- **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.

10.1.2 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.



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:

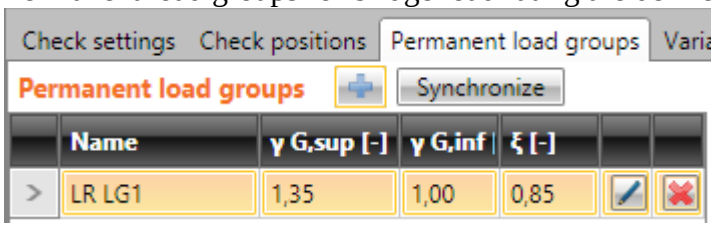
- **Name** – input the name of position. The name is used to generate the name of section in the IDEA RCS module.
- **Reference point** – select the point of the design member, which the distance of section is related to.
- **Position** – input the distance between the check position and the reference 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.

10.1.3 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.

10.1.3.1 Permanent load groups

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



Commands above the table:

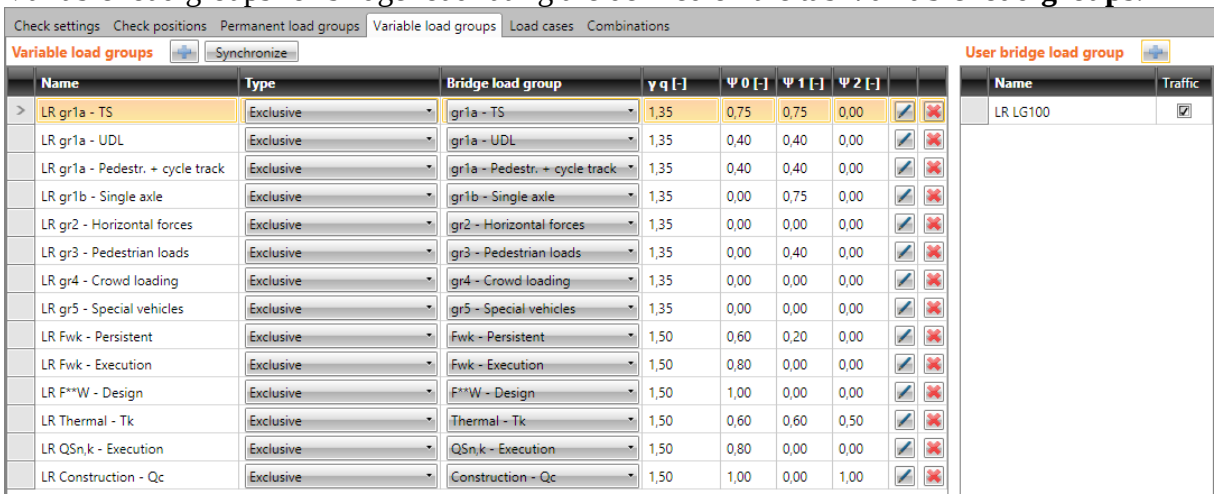
-  - add new group of permanent loads for bridge load rating.
- **Synchronize** – updates permanent load groups for bridge load rating according to permanent load groups for check of design loads.

Columns in **Permanent load groups** table:

- **Name** – input name of load group.
- **γ G,sup** – input of partial factor for permanent unfavourable load cases in ULS combinations.
- **γ G,inf** – input of partial factor for permanent favourable load cases in ULS combinations.
- ξ – input of reduction factor of unfavourable permanent loads.
-  - launch **Load groups manager** to assign load cases to load groups and to modify properties of load groups (also multiple) – see **10.1.3.5 Load groups manager**.
-  - delete the appropriate permanent load group.

10.1.3.2 Variable load groups

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



Name	Type	Bridge load group	γ q [-]	Ψ 0 [-]	Ψ 1 [-]	Ψ 2 [-]
LR gr1a - TS	Exclusive	gr1a - TS	1,35	0,75	0,75	0,00
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

Name	Traffic
LR LG100	☑

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

-  - 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.
- **γq** – input the value of partial load factor of variable load cases in ULS combinations.
- **ψ0** – input the value of partial load factor of variable load cases in ULS and SLS Characteristic combinations.

- **ψ1**- input the value of partial load factor of variable load cases in SLS Frequent combinations.
- **ψ2** - 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 **10.1.3.5 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**:

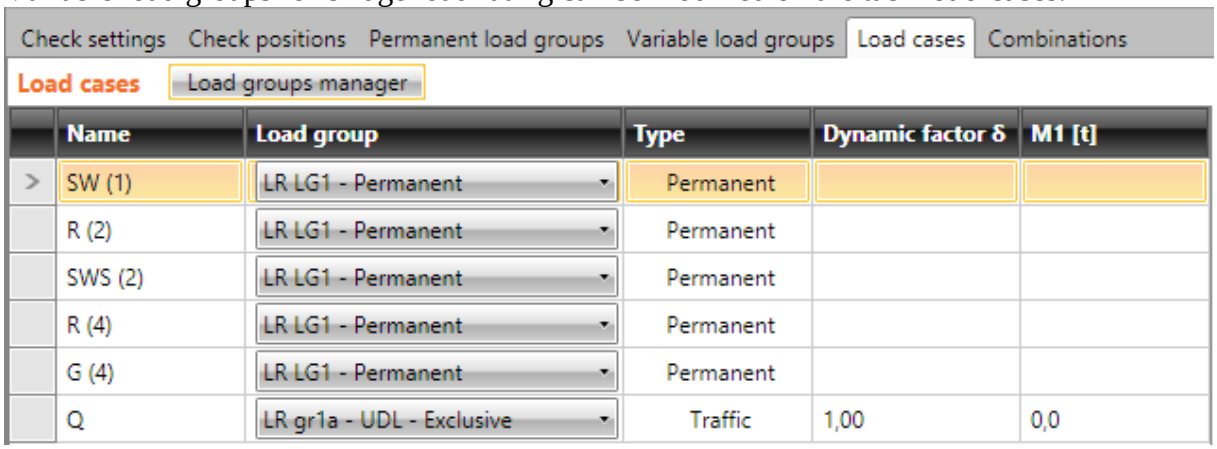
-  - add new user defined bridge load group.

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.

10.1.3.3 Load cases

Variable load groups for bridge load rating can be modified on the tab **Load cases**.



Name	Load group	Type	Dynamic factor δ	M1 [t]
SW (1)	LR LG1 - Permanent	Permanent		
R (2)	LR LG1 - Permanent	Permanent		
SWS (2)	LR LG1 - Permanent	Permanent		
R (4)	LR LG1 - Permanent	Permanent		
G (4)	LR LG1 - Permanent	Permanent		
Q	LR gr1a - UDL - Exclusive	Traffic	1,00	0,0

Commands above the table:

- **Load groups manager** - launch **Load groups manager** to assign load cases to load groups and to modify properties of load groups (also multiple) – see **10.1.3.5 Load groups manager**.

Columns in **Load cases** table:

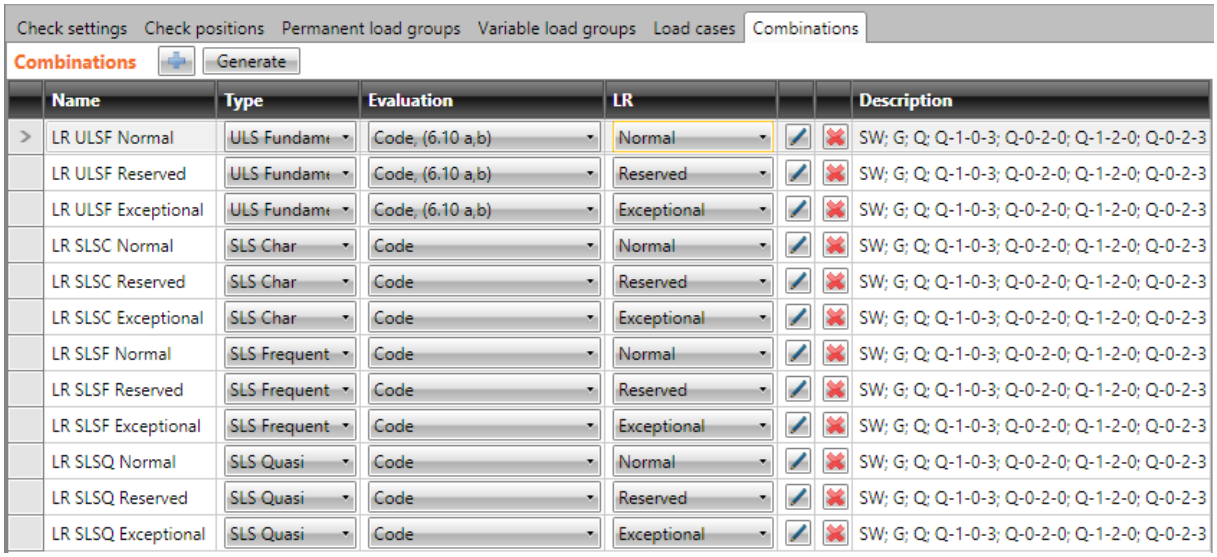
- **Name** –name of load case is displayed.
- **Load group** – select the load group to assign the load case to.
- **Type** – displays type of load – **Permanent** or **Traffic** for variable loads for bridge structures.
- **Dynamic factor** – input value of dynamic factor applied on variable load cases 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 is calculated, only rating factor is 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.
-  - delete the appropriate load case.

10.1.3.4 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.



	Name	Type	Evaluation	LR			Description
>	LR ULSF Normal	ULS Fundam	Code, (6.10 a,b)	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR ULSF Reserved	ULS Fundam	Code, (6.10 a,b)	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR ULSF Exceptional	ULS Fundam	Code, (6.10 a,b)	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSC Normal	SLS Char	Code	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSC Reserved	SLS Char	Code	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSC Exceptional	SLS Char	Code	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSF Normal	SLS Frequent	Code	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSF Reserved	SLS Frequent	Code	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSF Exceptional	SLS Frequent	Code	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSQ Normal	SLS Quasi	Code	Normal			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSQ Reserved	SLS Quasi	Code	Reserved			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3
	LR SLSQ Exceptional	SLS Quasi	Code	Exceptional			SW; G; Q; Q-1-0-3; Q-0-2-0; Q-1-2-0; Q-0-2-3

Commands above the table:

-  - add a new combination for bridge load rating.
- **Generate** – generates missing combinations required to perform checks selected to determine load rating of design member.

Columns in the **Combinations** table:

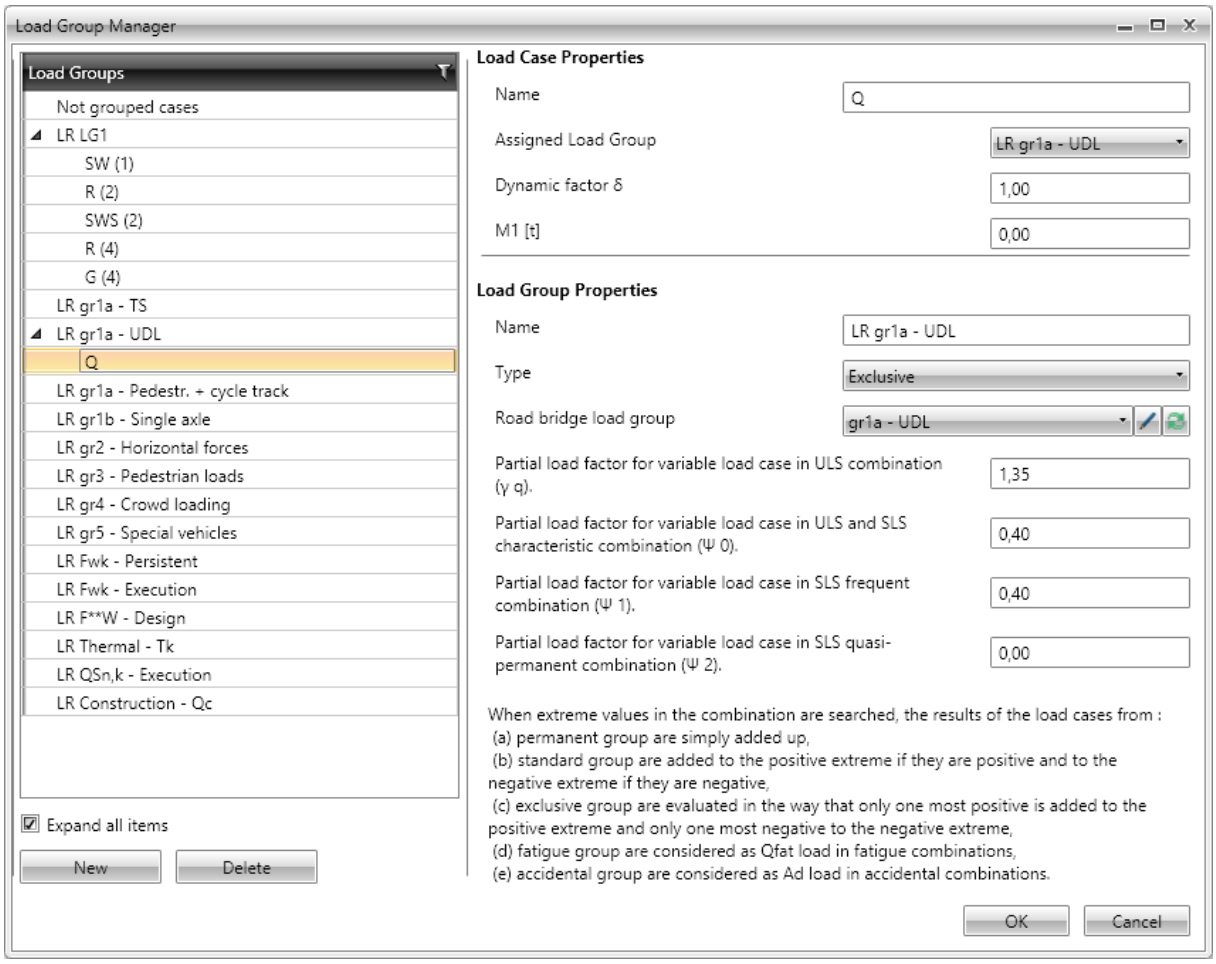
- **Name** – input the name of combination.
- **Type** – select the type of current combination.
- **Evaluation** – select the evaluation mode of combination.
- **LR** – select type of traffic load (load rating) assigned to the combination:
  - **Normal -** maximum weight of single truck, which can access the bridge with no traffic restrictions, in arbitrary number and with no restrictions for pedestrians and cyclists.
  - **Reserved -** maximum weight of single truck, which can access the bridge as the only vehicle with no other traffic restrictions for pedestrians and cyclists.
  - **Exceptional** - maximum weight of single truck, which can access the bridge as the only vehicle with no other traffic of pedestrians and cyclists, and subject to further restrictive measures.
-  - launch the **Combinations manager** to modify the combination rules – see **5.6.10 Manager of load cases combinations**.
-  - delete the appropriate combination.
- **Description** – displays the content of combination rule.

10.1.3.5 Load groups manager

Each load case is assigned to a load cases group.

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

To view or modify load cases groups for bridge load rating determination click **Load groups manager** in ribbon group above the table **Load cases**.



A type of group and values of partial load factors can be set for load group

Particular options of **Load groups manager** dialog:

- **Load groups** – load groups with associated load cases are displayed in the tree view.
  If a load case is selected in the tree view, properties of load case and properties of the appropriate load group are displayed in the right part of dialog.
  If a load group is selected in the tree view, properties of the appropriate load group are displayed in the right part of dialog.
  Load cases can be moved between load groups in the tree view using drag and drop (single or multiple).
- **New** – add a new load cases group.
- **Delete** – delete the current group of load cases.
- **Expand all items** – expand/collapse all items in the tree view.

Load case properties:

- **Name** – input name of the load case.

- **Assigned load group** – select load group, which the current load case is assigned to. Load cases can be moved between load groups in the tree view using drag and drop.
- **Dynamic factor** – input value of dynamic factor applied on variable load cases for traffic loads. Factor takes into account dynamic impact of moving vehicles in current load case.
- **M1** – for road bridges only. Total weight of one vehicle, which corresponds with type of rating, and which was specified in current load case in the model for structural analysis. If the value is zero, no weight limit in tons will is calculated, only rating factor is 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:
    - **γ qu, sup** – input value of partial coefficient for permanent unfavourable loads in ULS combinations.
    - **γ qu, inf** – input value of partial coefficient for permanent favourable loads in ULS combinations.
    - **ξ** – input value of reduction coefficient for unfavourable permanent actions.
  - **Standard** – load group for standard variable load cases. Effects of load cases in standard group are added up to the positive extreme, if they are positive, and are added up to the negative extreme, if they are negative. Following partial load factors can be set for the variable load cases group:
    - **γ q** – input value of partial load factor for variable load cases in ULS combinations.
    - **ψ0** – input value of partial load factor for variable load cases in ULS and SLS Characteristic combinations.
    - **ψ1** – input value of partial load factor for variable load cases in SLS Frequent.
    - **ψ2** – 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 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 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.

10.1.4 Ribbon group View settings and scale

See **9.4.3 Ribbon group View settings and scale**.

10.2 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.

10.2.1 Ribbon group View settings and scale

See **9.4.3 Ribbon group View settings and scale**.

10.2.2 Ribbon group Extreme

See **9.9.3 Ribbon group Extreme**.

10.2.3 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.

10.2.4 Ribbon group Check

See **9.9.7.1 Ribbon group Check**.

11 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.

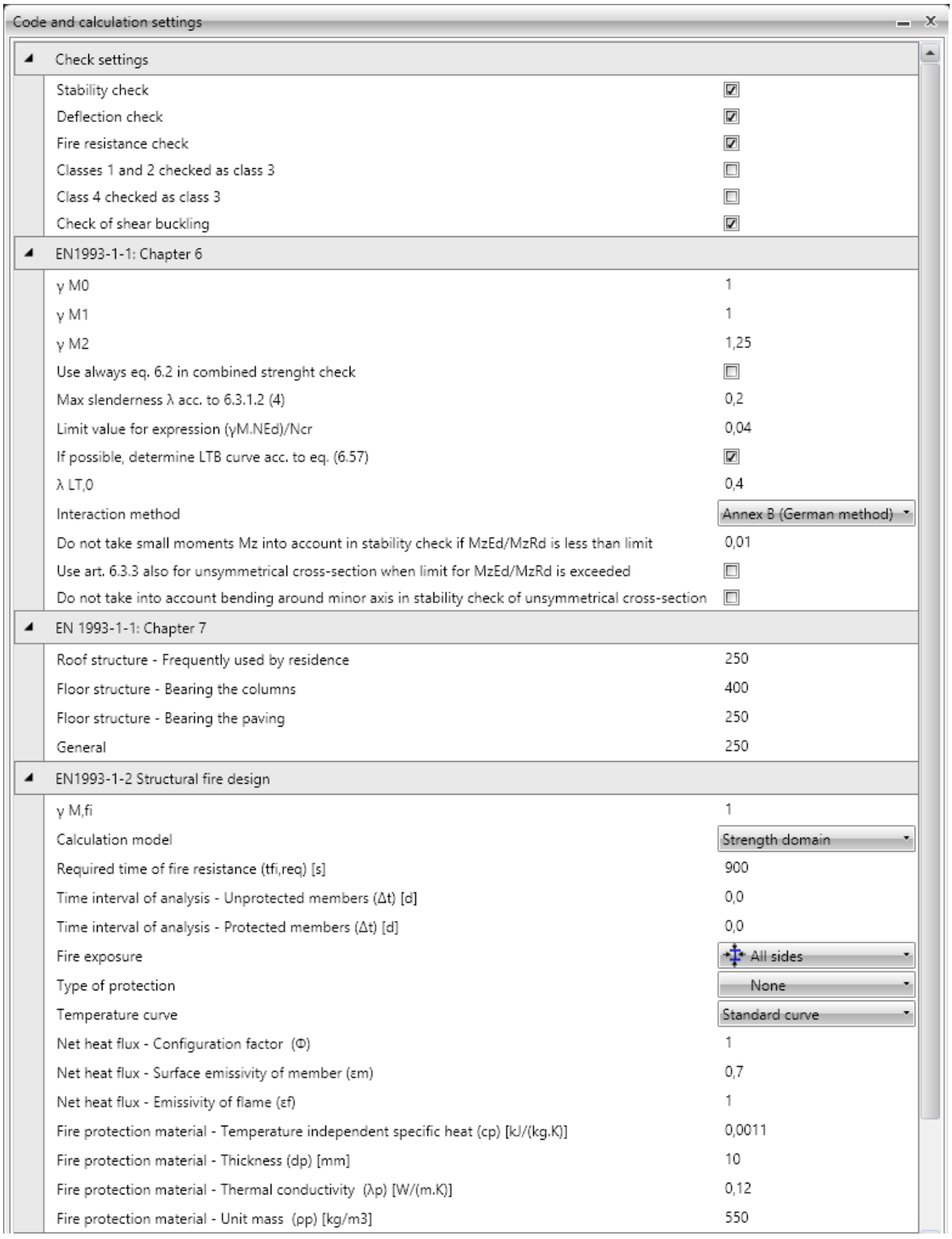
To be able to run the design of steel members, following conditions must be met:

- 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.

11.1 Default check settings

Click **Code** in ribbon group **Project setup** to change the default check settings (common for all design groups) and the national code settings.



Code and calculation settings dialog options:

Group **Check settings**:

- **Stability check** – switch on/off the execution of stability check. If the option is not selected, the stability check of designed members is not performed, only the section check is performed.

- **Deflection check** – switch on/off the execution of deflection check. If the option is selected, the deflection check of designed members is performed.
- **Fire resistance check** – switch on/off the execution of check of structural fire design according to EN1993-1-2 is performed. If the option is selected, the fire resistance check is performed.
- **Classes 1 and 2 checked as class 3** – switch on/off the execution of plastic check. If the option is selected, cross-sections classified as class 1 or class2 are checked using articles for class 3.
- **Class 4 cross-checked as class 3 -** if the option is selected, cross-sections classified as class 4 are checked using articles for class 3. The check of class 4 cross-sections is not supported. If the option is not selected and the cross-section is classified as class 4, the check value of cross-section is set to 500%.
- **Check of shear buckling** – if the option is selected, the limit slenderness of cross-section web is checked. If the limit slenderness is checked and is exceeded, the check value of cross-section is set to 500% – the check according to EN1993-1-5 should be performed in such case, but it is not supported.

Group **EN1993-1-1 Chapter 6**:

- **γM0** – input value of partial factor for resistance of cross-sections whatever the class is.
- **γM1** – input value of partial factor for resistance of members to instability assessed by member check.
- **γM2** – 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.
- **Shear buckling limit is not tested** – if this option is selected, the shear-buckling limit check is disabled. If web-slenderness exceeds the limit in 6.2.6(6) then shear buckling should be checked acc. to EN 1993-1-5.
- **Max slenderness λ acc. to 6.3.1.2 (4)** – input the limit value of relative slenderness to neglect the buckling resistance check according to 6.3.1.2 (4).
- **Limit value for expression (γM.NEd)/Ncr** – input the limit value of formula to neglect the buckling resistance check according to (6.3.1.2(4)).
- **If possible, determine LTB curve acc. to eq. (6.57)** – if the option is selected and it is possible to determine LTB curve according to (6.57), the determined LTB curves are used. If the curves cannot be determined according to (6.57) or if the option is not selected, the buckling curves for LTB are determined according to (6.56).
- **λ 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 Mz into account in stability check if MzEd/MzRd is less than limit** – input the limit value when the influence of Mz can be neglected for checks of unsymmetrical cross-section under compression and bending checked acc. to art. 6.3.4 or modified method 6.3.3 for mono-symmetrical cross-section where only in plain bending can be used. When limit is exceeded the check fails and error is written in report if the other options do not cover this case.

- **Use art. 6.3.3 also for unsymmetrical cross-section when limit for MzEd/MzRd is exceeded** – if the option is selected, members with unsymmetrical cross-section are checked acc. to art. 6.3.3 when art. 6.3.4 or alternative method for mono-symmetrical cross-section cannot be used. The utilisation reserve to cover possible inaccuracy must be ensured.
- **Do not take into account bending around minor axis in stability check of unsymmetrical cross-section –** if the option is selected, bending around minor axis is neglected. This option allows to use art. 6.3.4 or alternative method 6.3.3 for mono-symmetrical cross-section always. Then utilisation reserve to cover possible inaccuracy must be ensured.

Group **Chapter 7** – input the value of limit deflections for particular types of structural elements related to the length of the member (1/n).

Group **EN1993-1-2 – Structural fire design**

- **γM,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 εm to determine the net heat flux.
- **Net heat flux –emissivity of flame** – input the emissivity of flame εf to determine the net heat flux.
- **Fire protection material – temperature independent specific heat** – input the specific heat of applied fire protection material.

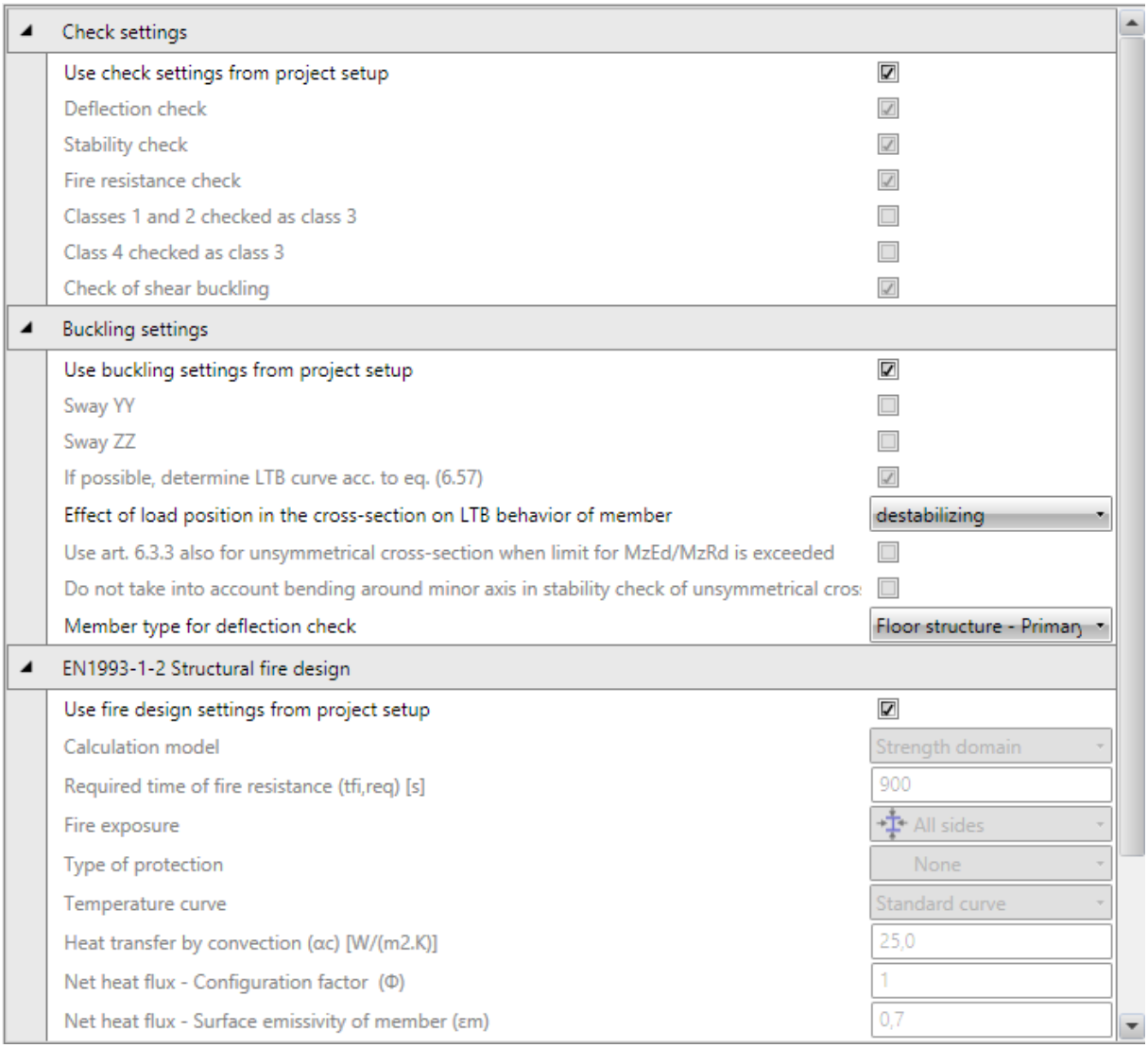
- **Fire protection material – thickness** – input the thickness of applied fire protection material.
- **Fire protection material – thermal conductivity** – input the thermal conductivity of applied fire protection material.
- **Fire protection material – unit mass** – input the unit mass of applied 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.

11.2 Check settings for the current design group

Click navigator command **Steel design > Options** to change the check settings for the current design group.



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 **11.1 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 **11.1 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**:

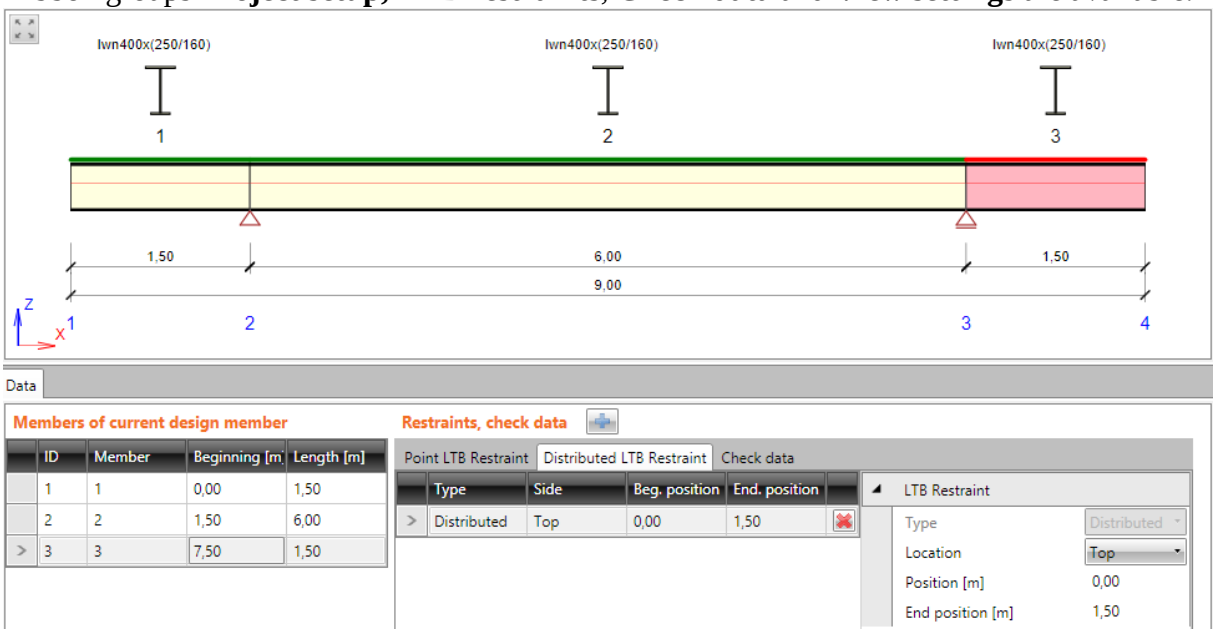
- **Use fire design from project setup** – if this option is selected, the fire design check settings are taken from the default project check settings – see **11.1 Default check settings**. If the option is not selected, specific fire design check settings can be set for the current design group.

11.3 Design data

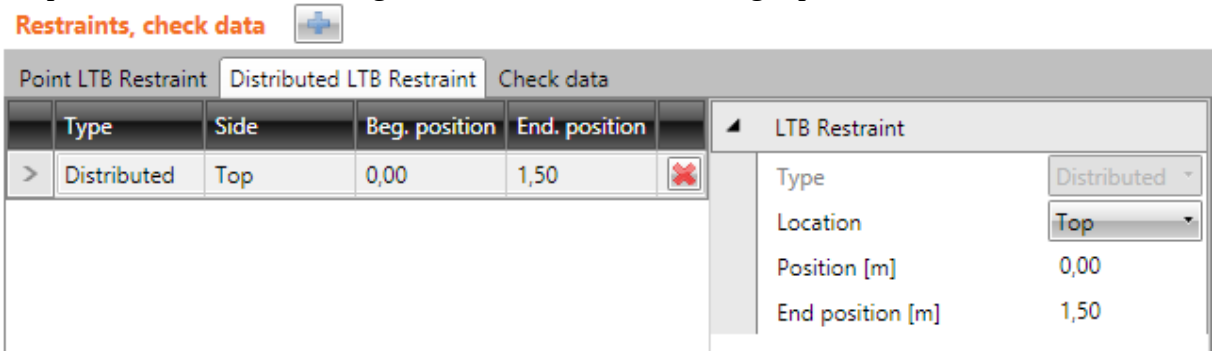
Click navigator command **Steel design > Design data** to input or edit the design data.
Following design data can be defined on the members of the design member:

- 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 **Project setup, LTB restraints**, **Check data** and **View settings** are available.



The design data are defined on single members of the current design member.
The current design member can be selected in the combo box **Design member** above the tables.
The list of members of the current design member is displayed in the table **Members of current design member**.
For the selected member, the defined design data are displayed in the table **Restraints, check data**.
Properties of the current design data can be edited in the right part of the table.

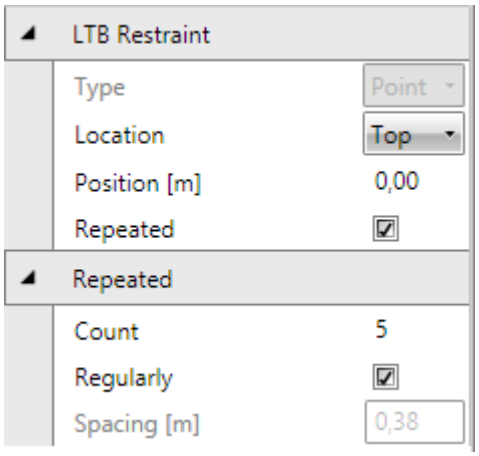


11.3.1 Ribbon group Project setup

See **11.1 Default check settings**.

11.3.2 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**.

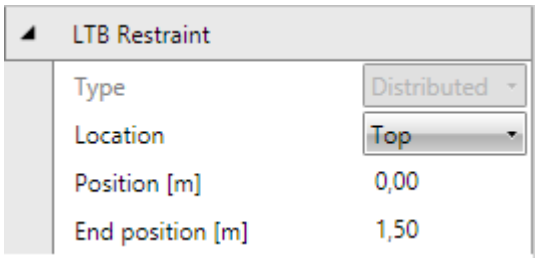


Point restraint properties:

- **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.

11.3.3 Distributed LTB restraint



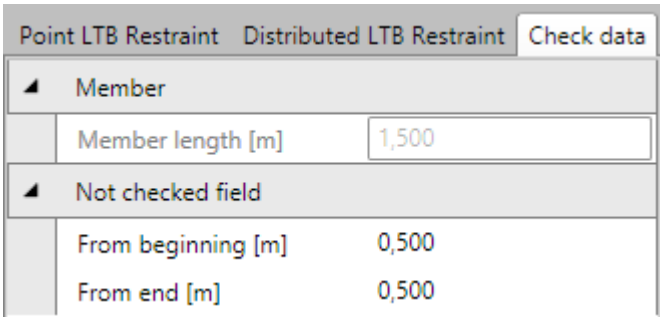
To add new distributed LTB restraint on the current member click  above the design data table (the tab **Distributed LTB restraint** must be active) or click **Distributed** in the ribbon group **LTB restraints**.

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.

11.3.4 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**.

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.

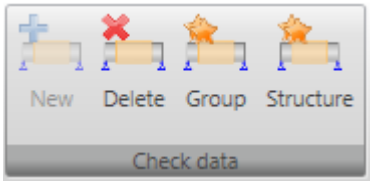
Click **Delete** in the ribbon group **Check data** to delete the not checked field.

11.3.5 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.

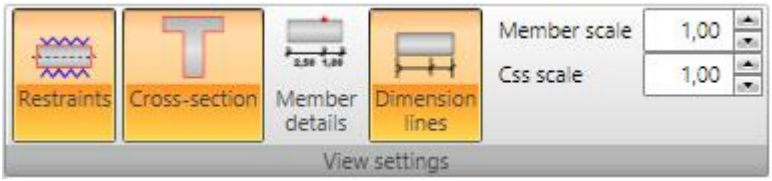
11.3.6 Ribbon group Check data



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.

11.3.7 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.

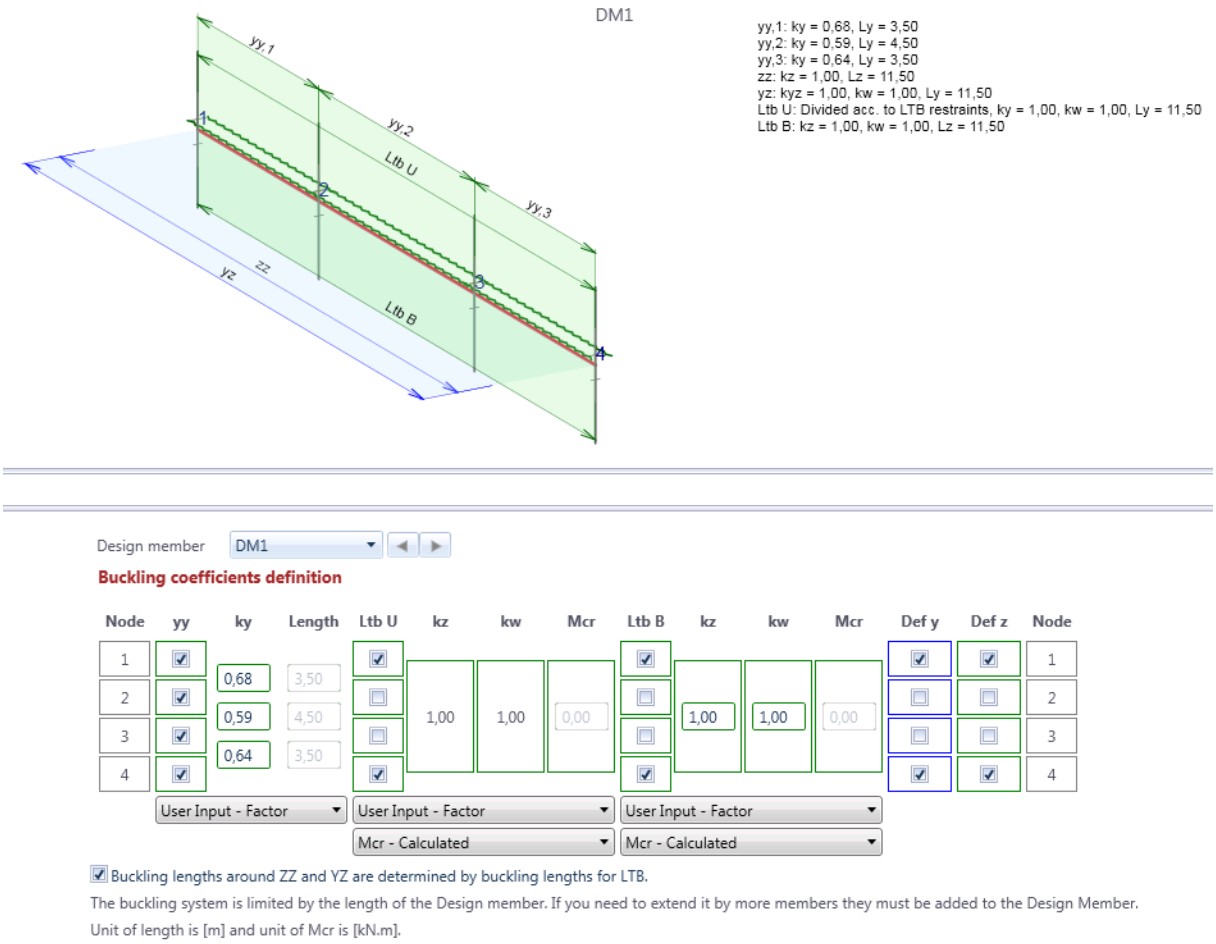
- **Dimension lines** – switch on/off the drawing of the dimension lines of the current design member.
- **Member scale** – set the value of the exceeded scale to draw the members of the design member.
- **Css scale** – set the value of the exceeded scale to draw the cross-section picture above the design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

11.4 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.



The current design member is draw in 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.

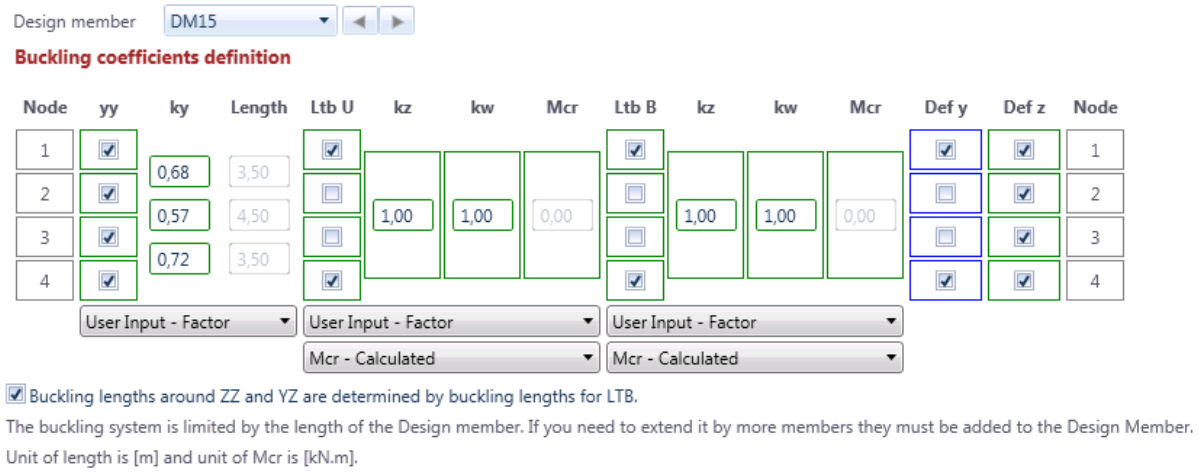
The table to define the system lengths and to input bucking lengths coefficients is displayed in the data window.

It is possible to configure the system length for flexural buckling yy, for flexural buckling zz, for torsional buckling yz, for lateral torsional buckling of upper flange, for lateral torsional buckling of lower flange and system lengths for limit deflections Defy and Defz.

Following coefficients can be assigned to system lengths:

- For flexural buckling:
  - **yy** for flexural buckling around the axis yy (either use the calculated value of the coefficient or input the value of coefficient or input the value of buckling length).

  - **zz** for flexural buckling around the axis zz (either use the calculated value of the coefficient or input the value of coefficient or enter the value of buckling length)
- for torsional buckling
  - **kw** - either input the value of coefficient or the buckling length
- for LT buckling the coefficients for top and bottom flange of the section:
  - **kz**
  - **kw**
  - **Mcr**



The current design member can be selected in the combo box **Design member** above the table.
In the table in the data window data about the buckling are entered. The first and last column of the table contains numbers of nodes which represent the nodes of individual members of the current design member.

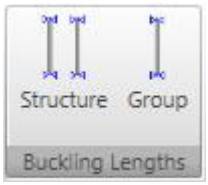
For each type of buckling to the table consists of three or more columns:

- column of checkboxes – selecting the checkboxes at individual nodes determines nodes, between which the system length is measured. The value of the buckling length of the member for flexural and torsional buckling is then calculated as the product of the coefficient of buckling length and the system length of the member. The course and shape of the moment curves is evaluated along the whole system length.
- column of buckling length coefficients **ky**, **kz** - values of coefficient for individual spans. If the mode of input is set to **Calculated**, the calculated values of buckling coefficient are displayed. In case of **User input – factor** the user defined value of buckling length coefficient can be entered. Option **Calculated** is only available for flexural plane buckling yy and zz and only for projects imported from the Ida Nexis.
- column of specified buckling lengths **Length** - if the mode of input is set to **Input – length**, the total value of the buckling length can be entered in this column.
- columns for input of coefficients **kz** and **kw**- if the mode of input of coefficients is set to **User input – factor**, the values of coefficients kz and kw can be set.
- column **Mcr**- if the mode of coefficients input for LT buckling is set to **Mcr – input,** the values of critical moment Mcr can be set..

Use the checkboxes in columns **Defy**, **Defz** to set the system length for calculation of deflections in the similar way as for buckling assessment.
If the option **Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB** is selected, groups for input of buckling coefficients for flexural buckling zz and for

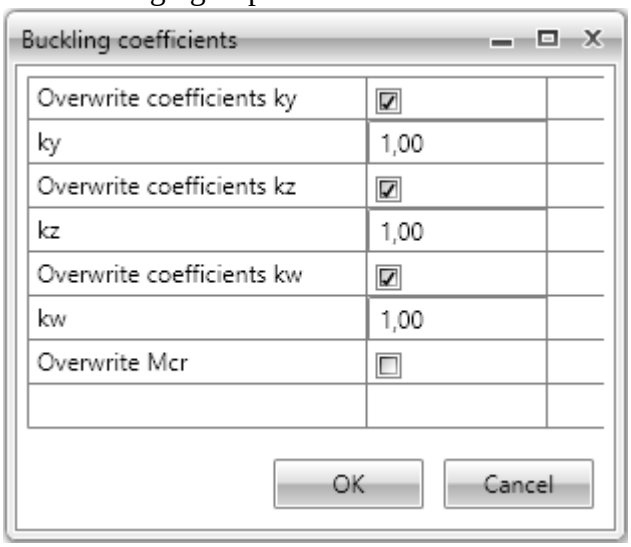
torsional buckling yz are not available. The calculation of flexural buckling zz and torsional buckling yz in this case takes the values of coefficients kz and kw specified for the check of LTB.
If there are LTB restraints on a member, they are taken into account when determining the buckling lengths and the coefficients kz and kw cannot be changed.

11.4.1 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:

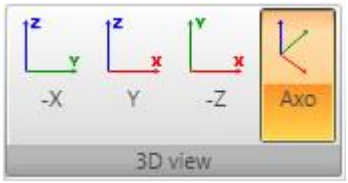
- **Structure** – click to input the buckling lengths coefficients to all members of the structure.
- **Group** – click to input the buckling lengths coefficients to all members of the current design group.



Options of dialog **Buckling coefficients**:

- **Overwrite coefficients ky** – if the option is selected, buckling coefficients ky can be overwritten using the value entered in edit box **ky**.
- **Overwrite coefficients kz** – if the option is selected, buckling coefficients kz can be overwritten using the value entered in edit box **kz**.
- **Overwrite coefficients kw** – if the option is selected, buckling coefficients kw can be overwritten using the value entered in edit box **kw**.
- **Overwrite Mcr** – if the option is selected, critical moment Mcr can be overwritten using the value entered in edit box **Mcr**.

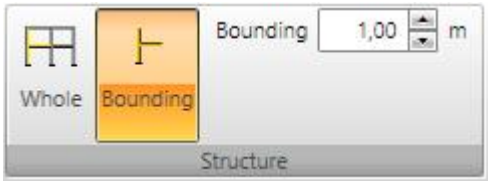
11.4.2 Ribbon group 3D View



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.

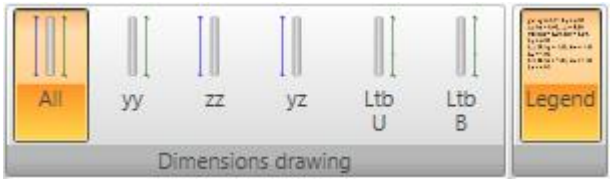
11.4.3 Ribbon group Structure



Commands in ribbon group:

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

11.4.4 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, dole** - 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.

11.5 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.

11.5.1 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.

11.5.2 Ribbon group Design member

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

11.5.3 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 **11.1 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.

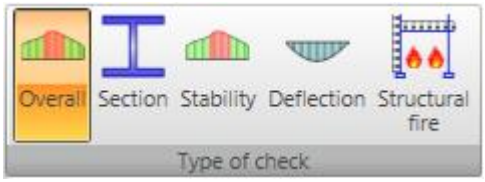
11.5.4 Ribbon group Extremes

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 **Des. 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.

11.5.5 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.

11.5.6 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 summary results only. .
- **Detailed** – switch to print tables with detailed outputs.

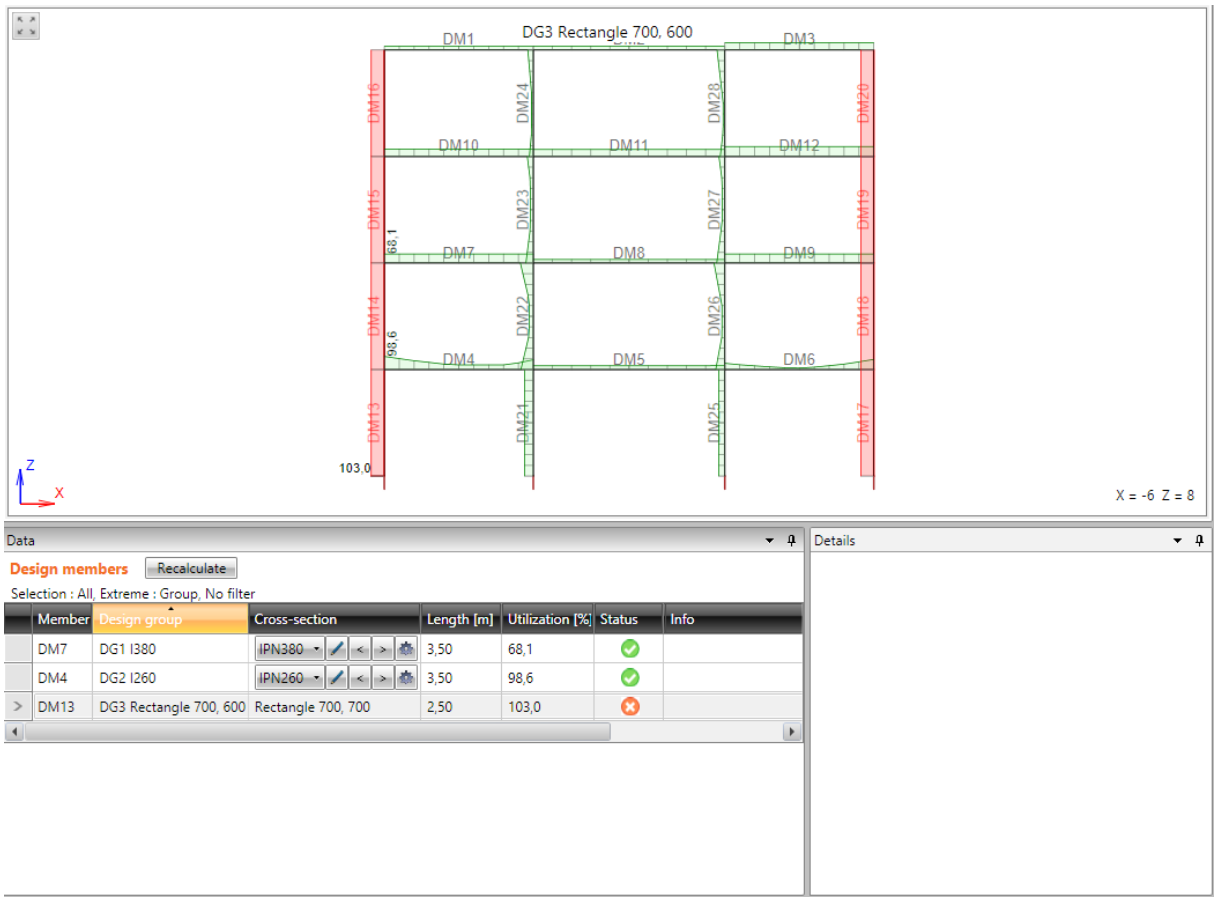
12 Design summary, optimisation of steel cross-sections

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**, **Recalculate**, **Check results evaluation** and **Material** are available for this navigator command.



12.1 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;

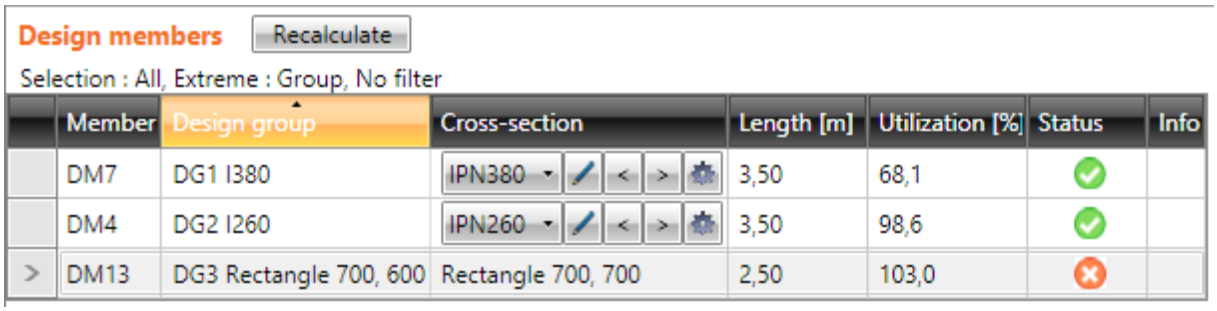


Table **Design members** contains following columns:

- **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.

12.1.1 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.

12.1.2 Ribbon group Concrete design

Use commands in ribbon group **Concrete design** to set the national code parameters and result classes for concrete check – see **9.2 Settings for section checks and calculation of deflections**.

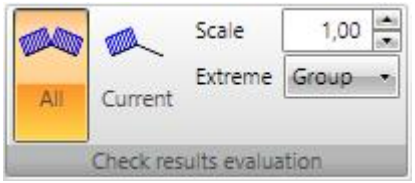
12.1.3 Ribbon group Steel design

Use commands in ribbon group **Steel design** to set the national code parameters and result classes for steel check – see **11.5.3 Ribbon group Steel design**.

12.1.4 Ribbon group Recalculate

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

12.1.5 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.
- **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.

12.1.6 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.

12.2 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.

13 Report

Input data and calculation results can be printed in report. Report can contain texts, tables and pictures. Structure of protocol is fixed, it is only possible to set, which tables and which pictures should be generated.

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

Click **Setup** to set the content of the report. In dialog **Report setup** particular items can be checked to be printed in the report.

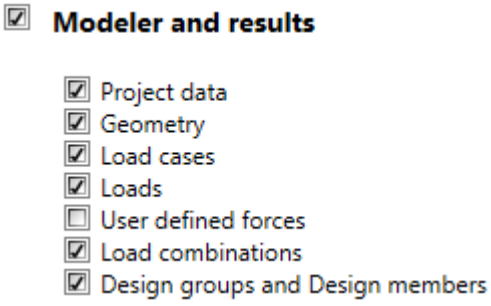
13.1 Standard report

To generate the standard report click navigator command **Report > Standard**.

The content of standard report can be modified in the Data window.

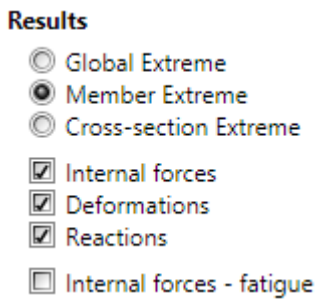
13.1.1 Input data

Options to set the content of input data standard report:



- **Modeler and results** – switch on/off print of tables of frame input data and linear calculation results.
- **Project data** – switch on/off print of tables of project identification data, cross-sections and materials.
- **Structure** – switch on/off print of tables of nodes, members and print of the picture of the structure.
- **Load cases** – switch on/off print of tables of load cases and groups of variable loads.
- **Loads** – switch on/off print of tables of load actions in individual load cases.
- **User defined forces** – switch on/off print of tables of basic information about user defined internal forces.
- **Load combinations** – switch on/off print of tables of load cases combinations definitions.
- **Design groups and Design members** – switch on/off print of tables of design groups and design members.

13.1.2 Calculation results

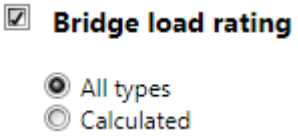


Options to set the content of calculation results standard report:

- **Global extreme** – if the option is selected, tables of global extremes of evaluated magnitudes will be printed to the report.
- **Member extreme** – if the option is selected, the tables of extreme values of evaluated magnitudes on each member will be printed to the report.
- **Cross-section** – extreme values of evaluated components are found per each cross-section of the frame.
- **Internal forces** – switch on/off print of tables of internal forces on members.
- **Deformations** – switch on/off print of tables of deformations on members.
- **Reactions** – switch on/off print of tables of reactions in supports.

- **Internal forces – fatigue** – switch on/off print of tables of minimal and maximal values and amplitude of internal forces.

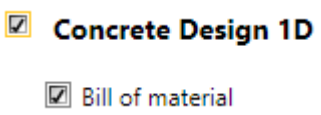
13.1.3 Bridge load rating results



Options to set the content of bridge load rating results standard report:

- **Bridge load rating** – switch on/off print of tables and pictures of bridge load rating calculation results.
- **All types** – switch to print results for all types of bridge load rating.
- **Calculated types** – switch to print results only for calculated types of bridge load rating (according to defined combinations for bridge load rating.

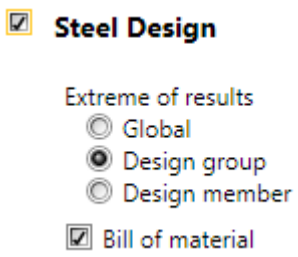
13.1.4 Concrete 1D design results



Options to set the content of concrete 1D check results standard report:

- **Concrete design 1D** – switch on/off print of tables and pictures of check results of concrete design groups.
  - **Bill of material** – switch on/off print of table of bill of material of concrete members.

13.1.5 Steel design results



Options to set the content of steel check results standard report:

- **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.

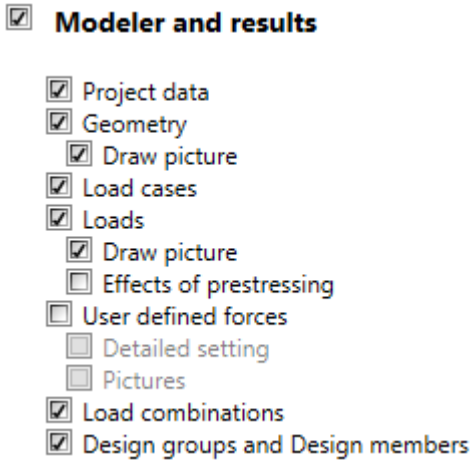
13.2 Detailed report

To generate the detailed report click navigator command **Report** > **Detailed**.

The content of detailed report can be modified in the Data window.

13.2.1 Input data

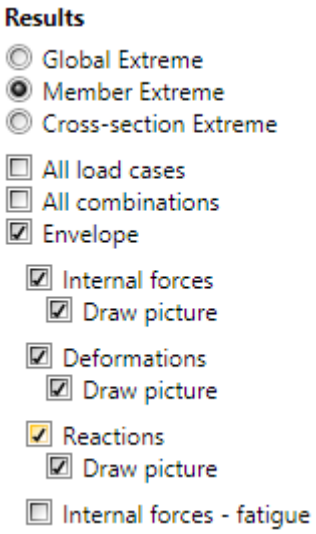
Options to set the content input data detailed report:



- **Modeler and results** – switch on/off print of tables of frame input data and linear calculation results.
- **Project data** – switch on/off print of tables of project identification data, cross-sections and materials.
- **Geometry** – switch on/off print of tables of nodes and members
  - **Draw picture** – switch on/off print of the picture of the structure.
- **Load cases** – switch on/off print of tables of load cases and groups of variable loads.
- **Loads** – switch on/off print of tables of load actions in individual load cases.
  - **Draw picture** – switch on/off print of the pictures of loads in individual load cases.
  - **Effects of prestressing** – switch on/off print of the table with loads caused by prestressing elements.
- **User defined forces** – switch on/off print of tables and pictures of user defined internal forces.
  - **Detailed setting** – switch on/off print of detailed tables of input data of user defined internal forces in individual load cases.
  - **Pictures** – switch on/off print of pictures of user defined internal forces in individual load cases.
- **Load combinations** – switch on/off print of tables of load cases combinations definitions.
- **Design groups and Design members** – switch on/off print of tables of design groups and design members.

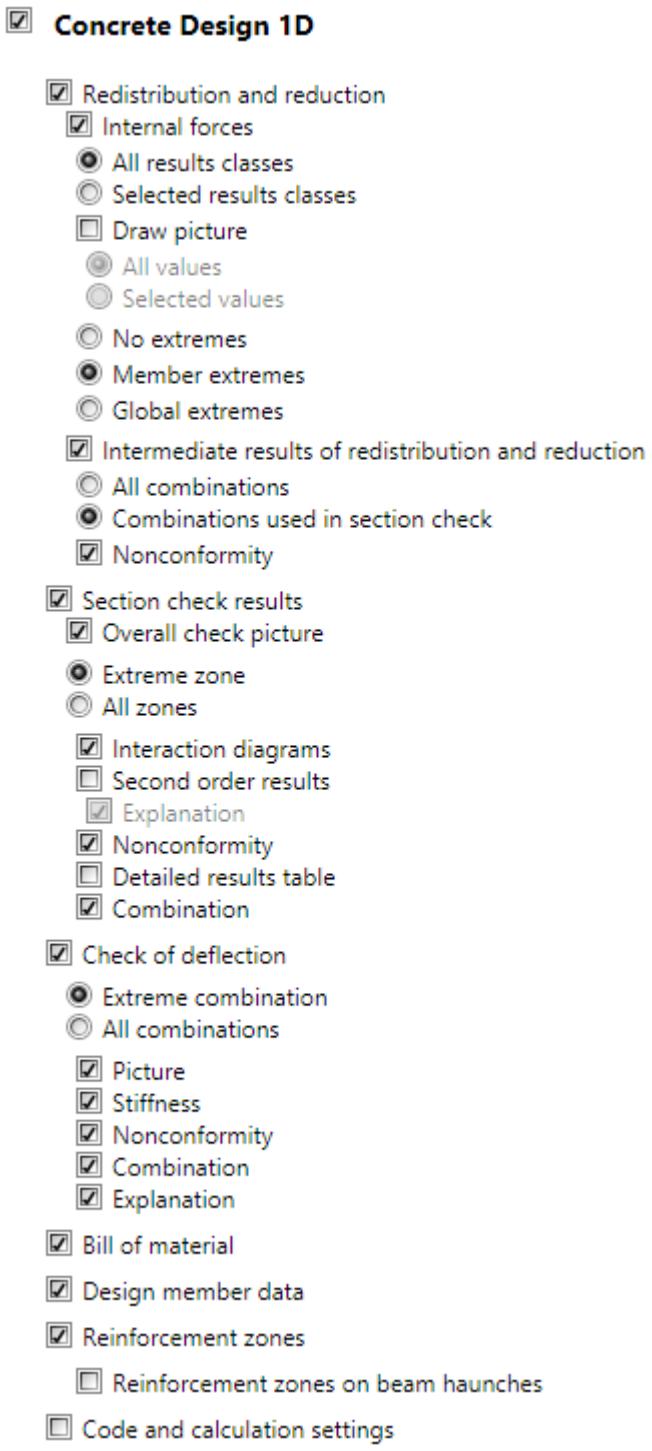
13.2.2 Calculation results

Options to set the content of calculation results detailed report:



- **Global extreme** – if the option is selected, tables of global extremes of evaluated magnitudes will be printed to the report.
- **Member extreme** – if the option is selected, the tables of extreme values of evaluated magnitudes on each member will be printed to the report.
- **Cross-section** – extreme values of evaluated components are found per each cross-section of the beam.
- **All load cases** – if the option is selected, the tables and pictures of results are printed for all defined load cases.
- **All combinations** – if the option is selected, the tables and pictures of results are printed for all defined combinations.
- **Envelope** – if the option is selected, the tables and pictures of results are printed for an envelope of combinations.
- **Internal forces** – switch on/off print of tables of internal forces on members.
  - **Draw picture** – switch on/off print of pictures of internal forces on members.
- **Deformations** – switch on/off print of tables of deformations on members.
  - **Draw picture** – switch on/off print of pictures of deformations on members.
- **Reactions** – switch on/off print of tables of reactions in supports.
  - **Draw picture** – switch on/off print of picture of reactions in supports.
- **Internal forces** – fatigue – switch on/off print of tables of minimal and maximal values and amplitude of internal forces.

13.2.3 Concrete 1D design results



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.

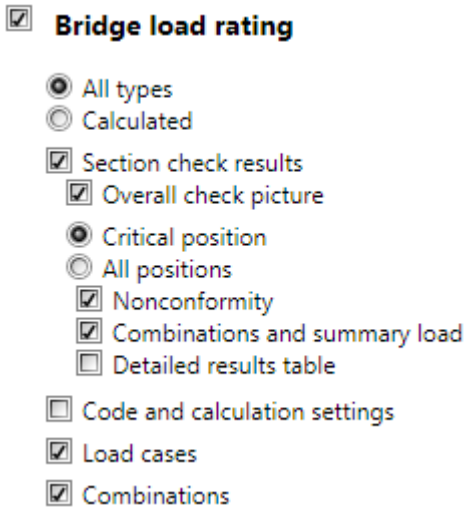
        - **Combinations used in section check** – switch to print tables of intermediate results only for combinations, which were used during concrete sections check.
        - **Nonconformity** – switch on/off the print of the table with nonconformities of reduction and redistribution calculation.

- **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.
    - **Explanations** - switch on/off the print of the table with explanations.

- **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.

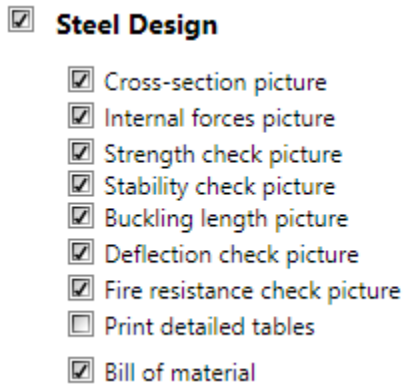
13.2.4 Bridge load rating results



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.

13.2.5 Steel design results



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.

13.3 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.