

IDEA Column 8

User guide

Content

1.1 Program requirements	5
1.2 Installation guidelines	5
2 Basic Terms.....	6
3 User interface	7
3.1 Control of view in the Main window	7
3.1.1 DXF export settings	8
3.2 Units setting.....	9
4 Working with project	11
5 Input of the structure	12
5.1 Project data.....	12
5.2 Creating a model	14
5.2.1 Ribbon group Cross-section	16
5.2.2 Ribbon group Scale	17
5.2.3 Ribbon group View	17
5.2.4 Ribbon group Projection	17
5.2.5 Ribbon group Axonometry.....	17
5.3 Loads	18
5.3.1 Load items	19
5.4 Combinations	21
6 Results	23
6.1 Reactions	24
6.1.1 Ribbon group View settings	24
6.1.2 Ribbon group Drawing.....	24
6.1.3 Ribbon group Results	25
6.2 Deformations	26
6.2.1 Ribbon group View settings	26
6.2.2 Ribbon group Drawing.....	26
6.2.3 Ribbon group Results	26
6.2.4 Ribbon group Selection.....	27
6.2.5 Ribbon group Extreme	27
6.2.6 Ribbon group Deformations.....	27
6.3 Internal forces.....	28
6.3.1 Ribbon group View settings	28
6.3.2 Ribbon group Drawing.....	28
6.3.3 Ribbon group Results	28
6.3.4 Ribbon group Selection.....	28

6.3.5 Ribbon group Extreme	29
6.3.6 Ribbon group Internal forces.....	29
7 Design and check of structural items	30
8 Check of concrete members	31
8.1 Section check and deflection check settings	32
8.1.1 Code and calculation settings	32
8.1.2 Setting of result class for calculation of deflections	32
8.1.3 Setting of result classes for section checks	33
8.1.4 Editing result class	34
8.2 Design member data.....	36
8.3 Reinforcement zones	38
8.3.1 Ribbon group View settings and scale	40
8.3.2 Ribbon group Detailed view	40
8.3.3 Ribbon group Internal forces.....	40
8.4 Editor of reinforcement	41
8.4.1 Editing cover	42
8.4.2 Input of reinforcement by template	43
8.4.3 Shear reinforcement	45
8.4.4 Longitudinal reinforcement.....	52
8.4.5 User settings of reinforced cross-section	60
8.4.6 Deleting reinforcement.....	62
8.4.7 Import and export of reinforced cross-section	62
8.4.8 View settings of reinforced cross-section	62
8.4.9 User defined reinforcement templates.....	63
8.5 Buckling data and deflection check data.....	67
8.5.1 Data for buckling effects calculation	67
8.5.2 Data for deflection check	68
8.6 Detailed check.....	70
8.7 Results evaluation	71
8.7.1 Ribbon group Concrete design.....	71
8.7.2 Ribbon group View settings and scale	71
8.7.3 Ribbon group Extreme	71
8.7.4 Ribbon group Calculation	71
8.7.5 Ribbon group Results drawing.....	72
8.7.6 Drawing of section check results courses	72
8.7.1 Drawing of interaction diagrams.....	72
8.7.2 Drawing of deflections check results	73

8.7.3 Check report	75
9 Steel members design.....	77
9.1 Default check settings	77
9.2 Check settings for the current design group.....	81
9.3 Design data.....	84
9.3.1 Ribbon group Project setup	84
9.3.2 Point LTB restraint.....	85
9.3.3 Distributed LTB restraint	85
9.3.4 Not checked field	85
9.3.5 Ribbon group LTB restraints.....	86
9.3.6 Ribbon group Check data.....	86
9.3.7 Ribbon group View settings	86
9.4 Buckling lengths.....	87
9.4.1 Ribbon group 3D view	89
9.4.2 Ribbon group Dimensions drawing.....	89
9.5 Check results evaluation.....	90
9.5.1 Ribbon group Steel design	90
9.5.2 Ribbon group Extremes.....	90
9.5.3 Ribbon group Type of check	90
9.5.4 Ribbon group Type of output	91
10 Calculation report.....	92
10.1 Brief report	92
10.2 Standard report	92
10.3 Detailed report.....	93
10.3.1 Input data.....	93
10.3.2 Calculation results	93
10.3.3 Concrete 1D design results.....	94
10.3.4 Steel design results	95
10.4 Ribbon group Report view	96

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

2 Basic Terms

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

It is possible to input an one-span column.

Supports can be defined in column bottom and column top nodes. Stiffness can be defined in supports.

Loads are sorted into load cases. There can be point forces and point moments in the column top node or corbel node and distributed load on wall panel. Load cases may be combined.

Analysis of internal forces and deformations is done by finite element method. Program calculates internal forces N , V_y , V_z , M_y , M_z and deformations u_x , u_y , u_z , f_{ix} , f_{iy} , f_{iz} .

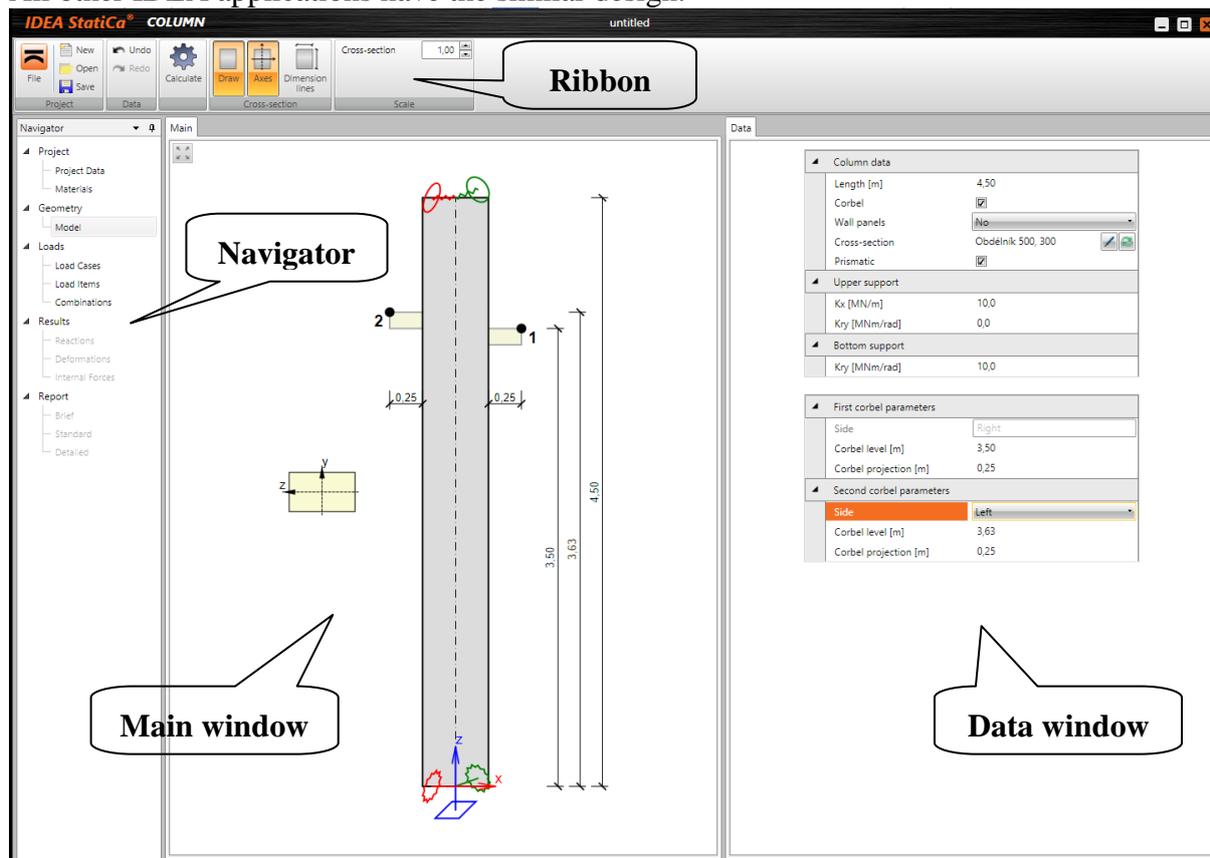
Results of IDEA Column are used in module IDEA RCS to design and check reinforcement or in module IDEA Steel to check steel members taking into account effects of buckling and lateral torsional buckling.

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 the Main window

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



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

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

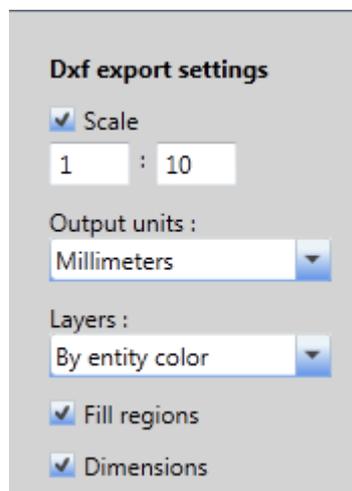
- Click and hold mid mouse button – moving the mouse pans the view.
- Roll with mid mouse button – moving the mouse increases/decreases the view.
- Push CTRL+SHIFT and hold mid mouse button – moving the mouse defines the window for zoom.

Click 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

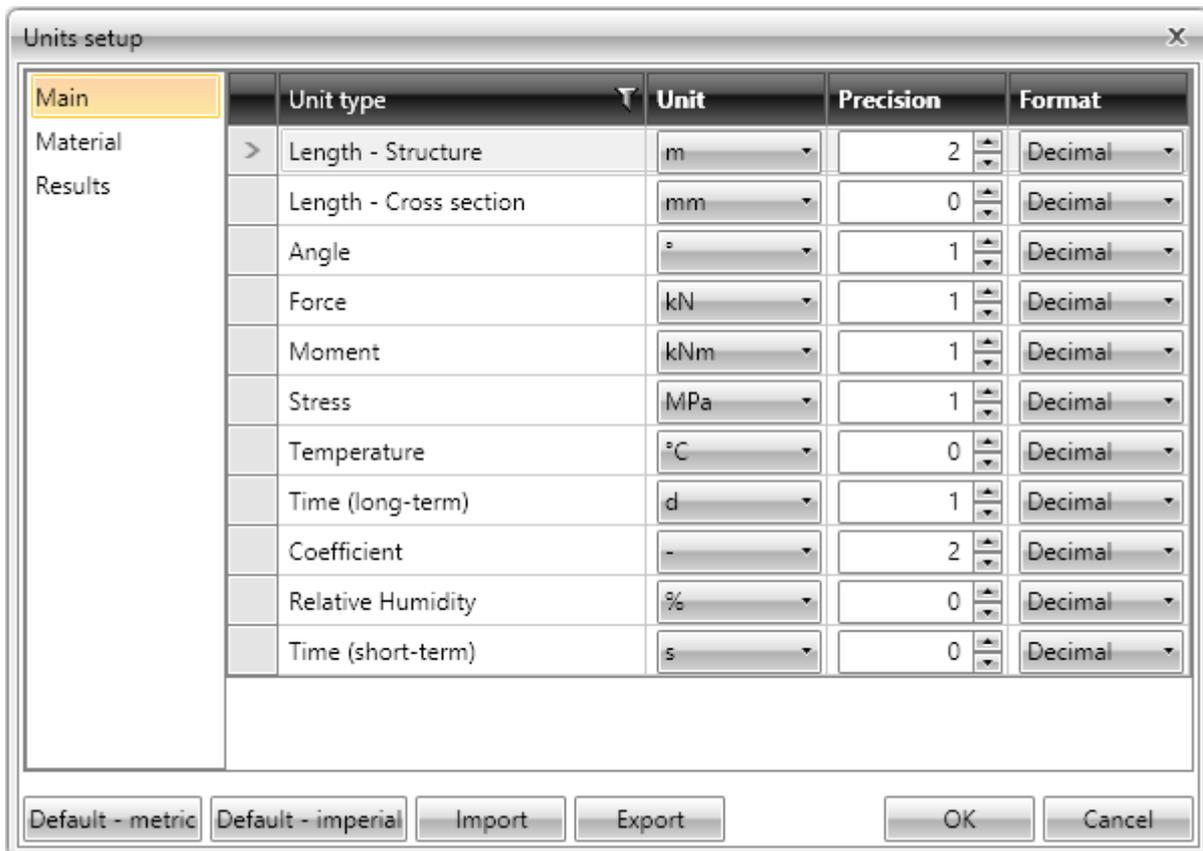


Following export parameters can be set in the Save as dialog when exporting the view to the 2D file:

- **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**.



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.

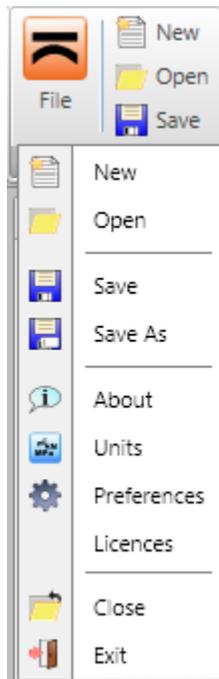
Import - reads the units configuration from a file.

Export - saves the current units settings to a file.

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

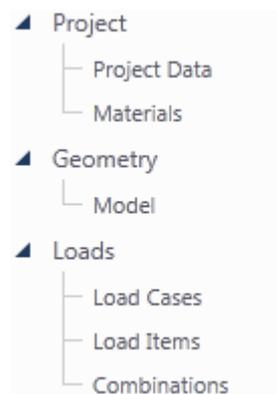
4 Working with project

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



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

5 Input of the structure



Appropriate structural data can be input using specific navigator commands. Commands for input of data are grouped into the 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.

Project parameters	
Code	EN
National annex	EN
Type	Concrete
Loads	One plane
Flexible supports	<input checked="" type="checkbox"/>
Identification	
Project	
Identification	
Number	
Author	
Description	
Date	12. 8. 2015

Properties group **Project parameters**

- **Code** – choose the national code of the project. The code cannot be changed after the project is calculated.
- **National annex** – choose national annex for the check of reinforced sections.
- **Type** – select the column material. The whole column is made of one material type. After the column is calculated, the material type cannot be changed.
- **Loads** – select the loads acting directions. Following directions can be chosen:
 - **One plane** – the column is planar, supports and loads are defined in the XZ plane of global coordinate system. The corresponding input and results evaluation methods are restricted respectively.
 - **Two planes** – the column is spatial, supports and loads are defined in planes XZ and YZ of global coordinate system.
- **Flexible supports** – if the option is selected, the values of rotational or translational stiffness can be defined for column supports.

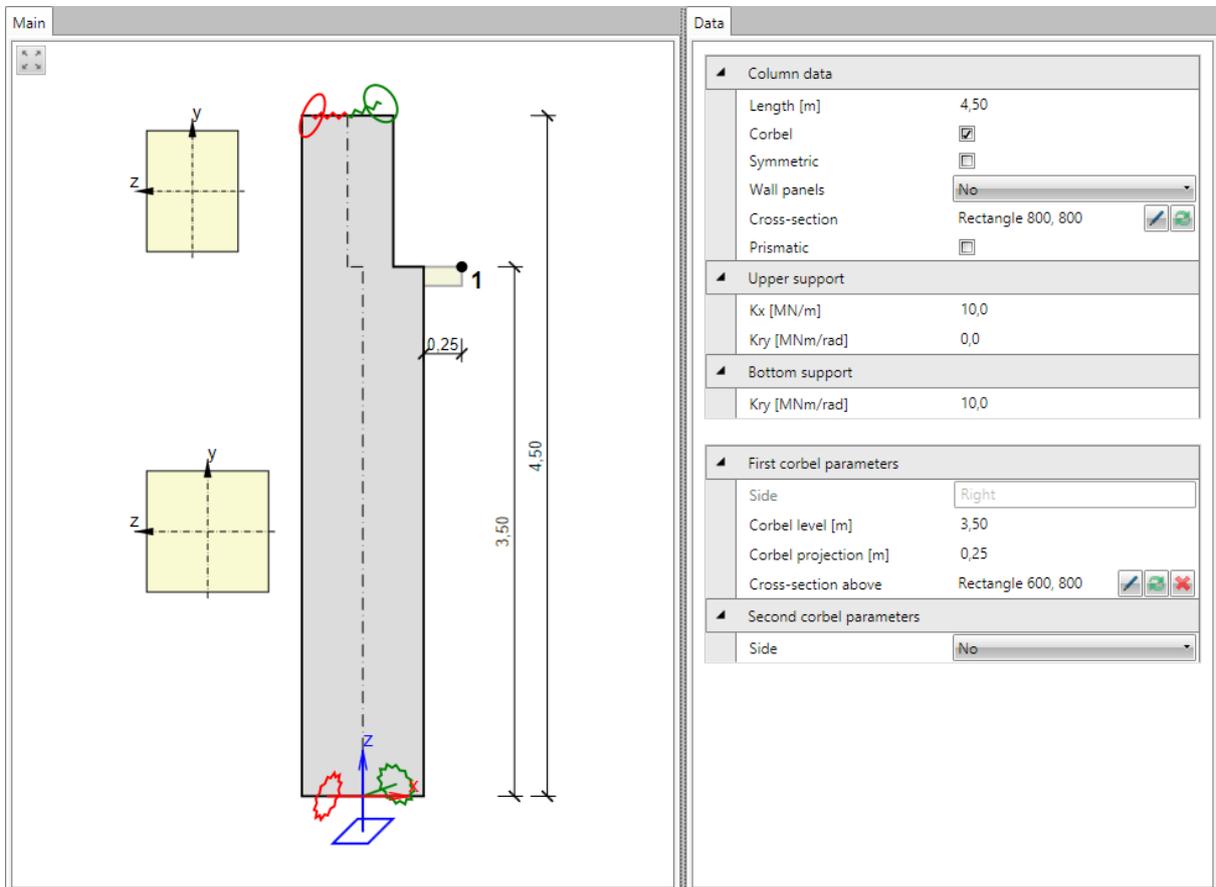
Properties group **Identification**

- **Project** – input of the project name.
- **Identification** – input the identification text.
- **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 Creating a model

Click navigator command **Geometry > Model** to create the model of the column. Length of column, cross-section, bearing conditions, corbels and wall panels are defined.

Ribbon groups **Cross-section**, **Scale**, **View**, **Projection** and **Axonometry** are available.



Column model properties can be edited using property tables in the data window. Basic column data can be changed in the property group **Column data**:

- **Length** – input the total length of column between the supports
- **Corbel** – if the checkbox is on, the first column corbel can be defined. The first corbel is always placed on the right column face (in XZ plane of the global coordinate system).
- **Wall panels** – choose the placement of wall panel. Wall panel can be loaded by wind or facade weight load. Wall panel can be placed either on the left or on the right column face or the wall panel can be disabled. The wall panel is over the whole length of column on the face without corbel or from the highest corbel to the column top on the face with corbel.
- **Symmetrical** – if the checkbox is on, the column is considered to be symmetrical. It means that all cross-sections are aligned to have centres of gravity of on the common axis. If the column is not symmetrical, the cross-sections are aligned to the left edge, thus the loads in column top and wall panels cause additional moments due to eccentricity.
- **Cross-section** – definition of cross-section of the column bottom part. Click edit button  to change properties of the current cross-section. Click  to input new cross-section of the bottom part of the column.

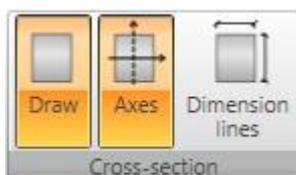
- **Prismatic** – if the checkbox is on, the column is considered to be prismatic. It means that there is only one cross-section over the whole length of the column. If the column is no prismatic, different cross-sections can be defined above each corbel.
- **Upper support** – set the support conditions in the node of column top. If the flexible support is defined, the stiffness value $1e7$ MN/m (or MN/rad) represents the rigid support. According to the column type the following top support properties can be defined:
 - **Planar column without flexible supports:**
 - **Free** – both translation and rotation are free in the column top.
 - **Hinge** – the column top support resists the translation in the X-axis direction of the global coordinate system.
 - **Fixed** – the column top support resists the translation in the X-axis direction and rotation about the Y-axis of the global coordinate system.
 - **Planar column with flexible supports:**
 - **Kx** – input the translational stiffness in the X-axis direction of the global coordinate system.
 - **Kry** – input the rotational stiffness about the Y-axis direction of the global coordinate system.
 - **Spatial column without flexible supports:**
 - **Plane XZ**
 - **Free** – the support in XZ plane resists neither translation, nor rotation.
 - **Hinge** – the support resists the translation in the X-axis direction of the global coordinate system.
 - **Fixed** – the support resists the translation in the X-axis direction and rotation about Y-axis of the global coordinate system.
 - **Plane YZ**
 - **Free** - the support in YZ plane resists neither translation, nor rotation.
 - **Hinge** – the support resists the translation in the Y-axis direction of the global coordinate system.
 - **Fixed** - the support resists the translation in the Y-axis direction and rotation about X-axis of the global coordinate system.
 - **Spatial column with flexible supports:**
 - **Kx** - input the translational stiffness in the X-axis direction of the global coordinate system.
 - **Ky** - input the translational stiffness in the Y-axis direction of the global coordinate system.
 - **Krx** - – input the rotational stiffness about the X-axis direction of the global coordinate system.
 - **Kry** - – input the rotational stiffness about the Y-axis direction of the global coordinate system.
- **Bottom support** – set the support conditions in the node of column base. If the flexible support is defined, the stiffness value $1e7$ MN/m (or MN/rad) represents the rigid support. According to the column type the following base support properties can be defined:
 - **Planar column without flexible supports:**
 - **Hinge** – the column base support resists the translations in the X and Z-axes directions of the global coordinate system.

- **Fixed** – the column top support resists the translations in the X and Z-axes directions and rotation about the Y-axis of the global coordinate system.
 - **Planar column with flexible supports:**
 - **Kry** – input the rotational stiffness about the Y-axis direction of the global coordinate system.
 - **Spatial column without flexible supports:**
 - **Plane XZ**
 - **Hinge** – the support resists the translations in the X and Z-axes directions of the global coordinate system.
 - **Fixed** – the support resists the translations in the X and Z-axes direction and rotation about Y-axis of the global coordinate system.
 - **Plane YZ**
 - **Hinge** – the support resists the translations in the Y and Z-axes directions of the global coordinate system.
 - **Fixed** - the support resists the translations in the Y and Z-axes directions and rotation about X-axis of the global coordinate system.
 - **Spatial column with flexible supports:**
 - **Krx** - – input the rotational stiffness about the X-axis direction of the global coordinate system.
 - **Kry** - – input the rotational stiffness about the Y-axis direction of the global coordinate system.

Properties of corbels can be defined in property groups **First corbel parameters** and **Second corbel parameters**:

- **Side** – choose the side to place the corbel. The side orientation is referred to the projection of column to the XZ plane of the global coordinate system (view in the Y-axis direction of the global coordinate system). First corbel is always placed to the right face of the column. Second corbel can be placed either to the left or to the right column face or can be disabled.
- **Corbel level** – input distance between the top of the corbel and the column base.
- **Corbel projection** – input the length of projection of the corbel.
- **Cross-section above** – definition of the cross-section above the corbel. Click the edit button  to change properties of the current cross-section above the corbel. Click  to add a new cross-section above the corbel. Click  to delete the cross-section above the corbel. The cross-section is replaced by the cross-section of the bottom part of the column.

5.2.1 Ribbon group Cross-section

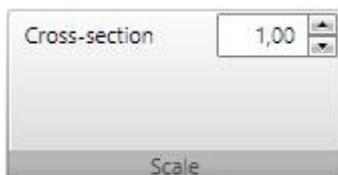


Commands in the ribbon group **Cross-section** are available, if the column is displayed in 2D.

- **Draw** – switch on/off drawing of cross-sections besides the individual column parts.
- **Axes** – switch on/off drawing of axes in the pictures of cross-sections.

- **Dimension lines** – switch on/off drawing of dimension lines in the pictures of cross-sections.

5.2.2 Ribbon group Scale



- **Cross-section** – set the scale value for drawing of cross-section picture.

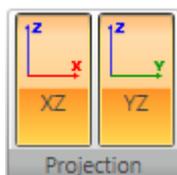
5.2.3 Ribbon group View



This ribbon group is available for columns loaded in two planes.

- **2D** – switch to drawing of column in projections in planes XY and YZ.
- **Axo** – switch to drawing of column in axonometric view.

5.2.4 Ribbon group Projection



This ribbon group is available for 2D view of columns loaded in two planes.

- **XZ** – switch on/off drawing of column projection in XZ plane of global coordinate system.
- **YZ** – switch on/off drawing of column projection in YZ plane of global coordinate system.

5.2.5 Ribbon group Axonometry



This ribbon group is available for axonometric view of columns loaded in two planes.

- **Wired** – switch to drawing of wired model of the column.
- **Solid** – switch to drawing of solids representation of the column members.

5.3 Loads

Nine predefined load cases can be used to define the load actions on the column.

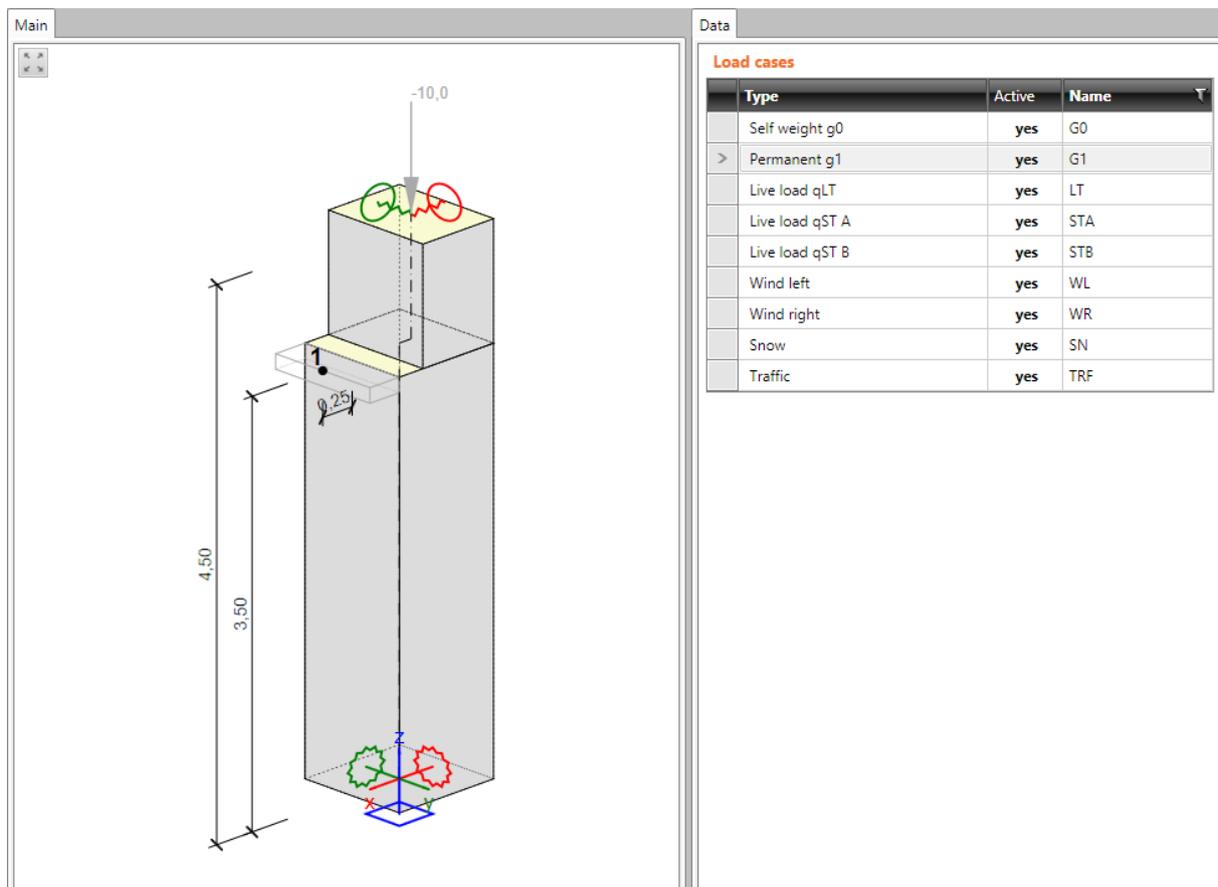
Load cases can be neither added, nor deleted. Individual load cases can be activated or deactivated. Deactivated load cases are not taken into account during the analysis, results of such load cases are not available.

To edit load cases click navigator command **Loads > Load cases**.

The column and load items of the current load case are drawn in the main window.

Table of load cases is displayed in the data window.

Individual load types are available depending of the load case type, number of corbels, support conditions etc.



To activate or deactivate the load case click appropriate button in **Active** column.

The name of load case can be changed in column **Name**.

5.3.1 Load items

To define load items in individual load cases click navigator command **Loads > Load items..**

The column with loads in the current load case is drawn in the main window.

The table with the current load case properties is displayed in the data window.

Current load case

Description Permanent loads, which can be applied on the upper end of the column, on corbels and on wall panels.

Load case	
Load type	Permanent g1
Top	
Vertical [kN]	-10,0
Horizontal Hx [kN]	0,0
Horizontal Hy [kN]	0,0
Moment Mx [kNm]	0,0
Moment My [kNm]	0,0
Corbel	
Vertical on first [kN]	0,0
Horizontal Hx on first [kN]	0,0
Wall	
Vertical [kN/m]	0,0
Bottom	
Moment Mx [kNm]	0,0
Moment My [kNm]	0,0

Select the current load case in the list **Current load case**. Properties of the current load case are displayed in the table.

The load on column top can be specified in the property group **Top**:

- **Vertical** – input of the vertical force acting in the top column node.
- **Horizontal Hx** – input of the horizontal force acting in the top column node in the X-axis direction of global coordinate system.
- **Horizontal Hy** – input of the horizontal force acting in the top column node in the Y-axis direction of global coordinate system.
- **Moment Mx** – input of the moment load acting about the X-axis of the global coordinate system in the top column node.
- **Moment My** – input of the moment load acting about the Y-axis of the global coordinate system in the top column node.

The load on column bottom can be specified in the property group **Bottom**:

- **Moment M_x** – input of the moment load acting about the X-axis of the global coordinate system in the bottom column node.
- **Moment M_y** – input of the moment load acting about the Y-axis of the global coordinate system in the bottom column node.

The load on corbel can be specified in the property group **Corbel** (if corbels are defined on column):

- **Vertical H_x on first** – input of the vertical force acting in the end node of the first corbel in the X-axis direction of the global coordinate system.
- **Horizontal on first** – input of the horizontal force acting in the end node of the first corbel.
- **Vertical on second** – input of the vertical force acting in the end node of the second corbel.
- **Horizontal H_x on second** – input of the horizontal force acting in the end node of the second corbel in the X-axis direction of the global coordinate system.

The load on wall panel can be specified in the property group **Wall** (if wall is defined on the column):

- **Vertical** – input of the vertical continuous load acting along the wall panel (weight of the facade). The load acts in the axis of the appropriate column part. The vertical load can be defined in permanent and long term variable load cases.
- **Horizontal H_x** – input of the horizontal continuous load acting along the wall panel in the X-axis direction of the global coordinate system (wind on the facade). The load acts in the axis of the appropriate column part. The horizontal load can be defined in load cases Wind left and Wind right.

Load caused by car crash can be defined in the property group **Impact**. This property group is available only for load case of type **Traffic**.

- **Horizontal H_x** – input value of horizontal force caused by the impact in the X-axis direction of the global coordinate system.
- **Horizontal H_y** – input value of horizontal force caused by the impact in the Y-axis direction of the global coordinate system.
- **Position** – input the distance between the point of impact and the column base.

5.4 Combinations

Click navigator command **Loads > Combinations** to define combinations of load actions..

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

Each combination can be of type:

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

Following type of evaluation can be set for the combination:

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

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

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

The column with loads acting in current combination is drawn in the main window.

Tables of load and combination coefficients and table of combinations are displayed in the data window.

The partial load factors of permanent loads and reduction factor of permanent loads can be modified in the table **Permanent loads**. Click **Restore** above the table to set the default values of permanent loads factors.

The partial load factors of variable loads and combination factors for individual load types can be modified in the table **Variable loads**. Click **Restore** above the table to set the default values of factors for variable loads.

Permanent load cases

	$\gamma_{G,sup}$	$\gamma_{G,inf}$	ξ
>	1,35	1	0,85

Variable load cases

	Type of loads	γ_q	ψ_0	ψ_1	ψ_2
>	Long-term	1,5	0,7	0,5	0,3
	Short-term	1,5	0,7	0,5	0,3
	Wind	1,5	0,7	0,5	0,3
	Snow	1,5	0,7	0,5	0,3

Combinations

	Name	Type	Evaluation		G0	G1	LT	STA	STB
	ULS	ULS Fundar	Code (6.10)		1	1	1	1	1
	SLS Char	SLS Char	Code		1	1	1	1	1
	SLS Freq	SLS Freque	Code		1	1	1	1	1
>	SLS Quasi	SLS Quasi	Code		1	1	1	1	1

Click above the table with combinations to add new combination.

The table Combinations contains following columns:

- **Name** – input the name of combination.
- **Type** – select the type of combination.
- **Evaluation** – select the evaluation mode of combination
- - delete the combination.

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

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

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

6 Results



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

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

Ribbon groups **View settings**, **Drawing**, **Results** and **Extreme** are available, when evaluating any type of calculation results.

6.1 Reactions

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. The table with content of critical combinations is displayed, if the evaluation is performed for combination.

Reactions are evaluated for current load case or combination.

Ribbon groups **View settings**, **Drawing** and **Results** are available when evaluating reactions in supports.

The screenshot displays the 'Main' window on the left and the 'Data' window on the right. The 'Main' window shows two diagrams: 'XZ' and 'YZ'. The 'XZ' diagram shows a column with a top node (2) and a bottom node (1). Reactions at node 2 are: $R_x = -10.9$ kN, $R_y = 3.2$ kN, and $R_z = 0$ kN. Reactions at node 1 are: $R_x = -2.9$ kN, $R_y = -2.4$ kN, and $R_z = 185.2$ kN. The 'YZ' diagram shows a column with a top node (2) and a bottom node (1). Reactions at node 2 are: $R_x = 0$ kN, $R_y = -7.2$ kN, and $R_z = 0$ kN. Reactions at node 1 are: $R_x = 0.6$ kN, $R_y = -0.3$ kN, and $R_z = 0$ kN.

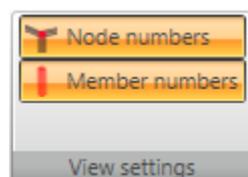
The 'Data' window contains a table with the following data:

Support	Combination	R_x [kN]	R_y [kN]	R_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]
2	ULS(4)	-10,9	0,0	0,0	0,0	0,0	0,0
2	ULS(5)	1,8	-5,0	0,0	0,0	0,0	0,0
2	ULS(1)	-10,0	-7,2	0,0	0,0	0,0	0,0
1	ULS(2)	-0,5	0,0	76,7	0,0	0,1	0,0
1	ULS(3)	-2,8	-0,3	185,2	1,6	-1,5	0,0
1	ULS(1)	-2,9	-0,3	184,1	1,6	-2,2	0,0
1	ULS(4)	-2,0	0,0	121,0	0,0	-2,4	0,0
1	ULS(5)	-1,8	-0,2	157,0	1,1	0,3	0,0

Below the table is a section for critical load effect descriptions:

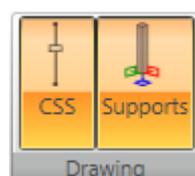
Combination	Critical load effect description
ULS(4)	$1,0^*G0 + 1,0^*G1 + 1,05^*LT + 1,5^*STA + 1,05^*WL$
ULS(5)	$1,35^*G0 + 1,35^*G1 + 1,05^*STB + 1,05^*WR + 1,5^*SN$
ULS(1)	$1,35^*G0 + 1,35^*G1 + 1,05^*LT + 1,5^*STA + 1,5^*STB + 1,05^*WL + 1,05^*SN$
ULS(2)	$1,0^*G0 + 1,0^*G1$
ULS(3)	$1,35^*G0 + 1,35^*G1 + 1,05^*LT + 1,5^*STA + 1,5^*STB + 1,05^*WR + 1,05^*SN$

6.1.1 Ribbon group View settings



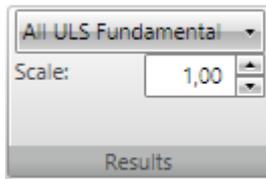
- **Node numbers** – switch on/off drawing of numbers of nodes.
- **Member numbers** – switch on/off drawing of numbers of members.

6.1.2 Ribbon group Drawing



- **CSS** – switch on/off drawing of the cross-section picture besides the column.
- **Supports** – switch on/off drawing of supports in the column top and column bottom.

6.1.3 Ribbon group Results



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

6.2 Deformations

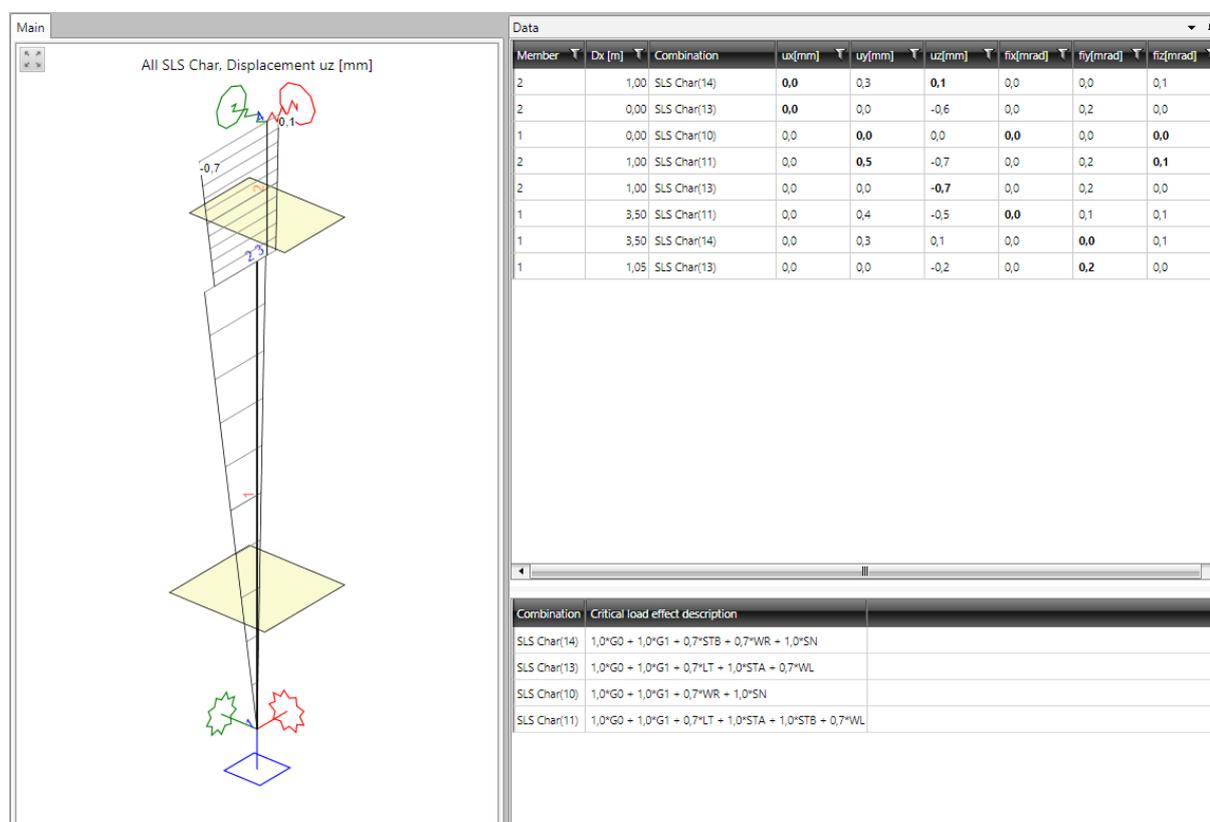
Click navigator command **Results > Deformations** to evaluate column deformations.

Calculated deformations are evaluated:

- Graphically – diagrams of selected component of deformation are displayed in the Main window.
- Textually – extreme values of deformations and rotations are displayed in the table in the Data window. The table with content of critical combinations is displayed, if the evaluation is performed for combination.

Deformations are evaluated for current load case or combination.

Ribbon groups **View settings**, **Drawing**, **Results**, **Extreme**, **Selection** and **Deformations** are available when evaluating deformations.



6.2.1 Ribbon group View settings

See 6.1.1 Ribbon group View settings.

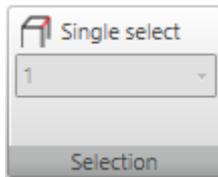
6.2.2 Ribbon group Drawing

See 6.1.2 Ribbon group Drawing.

6.2.3 Ribbon group Results

See 6.1.3 Ribbon group Results.

6.2.4 Ribbon group Selection

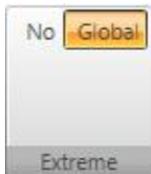


Use this ribbon group to evaluate results on individual member only.

- **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.

6.2.5 Ribbon group Extreme



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.
- **Global** – extreme values of evaluated components are found from all members/ supported nodes of the member.

6.2.6 Ribbon group Deformations



Particular options of ribbon group **Deformations**:

- **ux** – switch on graphical evaluation of deformation in direction of local member axis x.
- **uy** – switch on graphical evaluation of deformation in direction of local member axis y.
- **uz** – switch on graphical evaluation of deformation in direction of local member axis z.
- **fix** – switch on graphical evaluation of rotation about local member axis x.
- **fiy** – switch on graphical evaluation of rotation about local member axis y.
- **fiz** – switch on graphical evaluation of rotation about local member axis z.

6.3 Internal forces

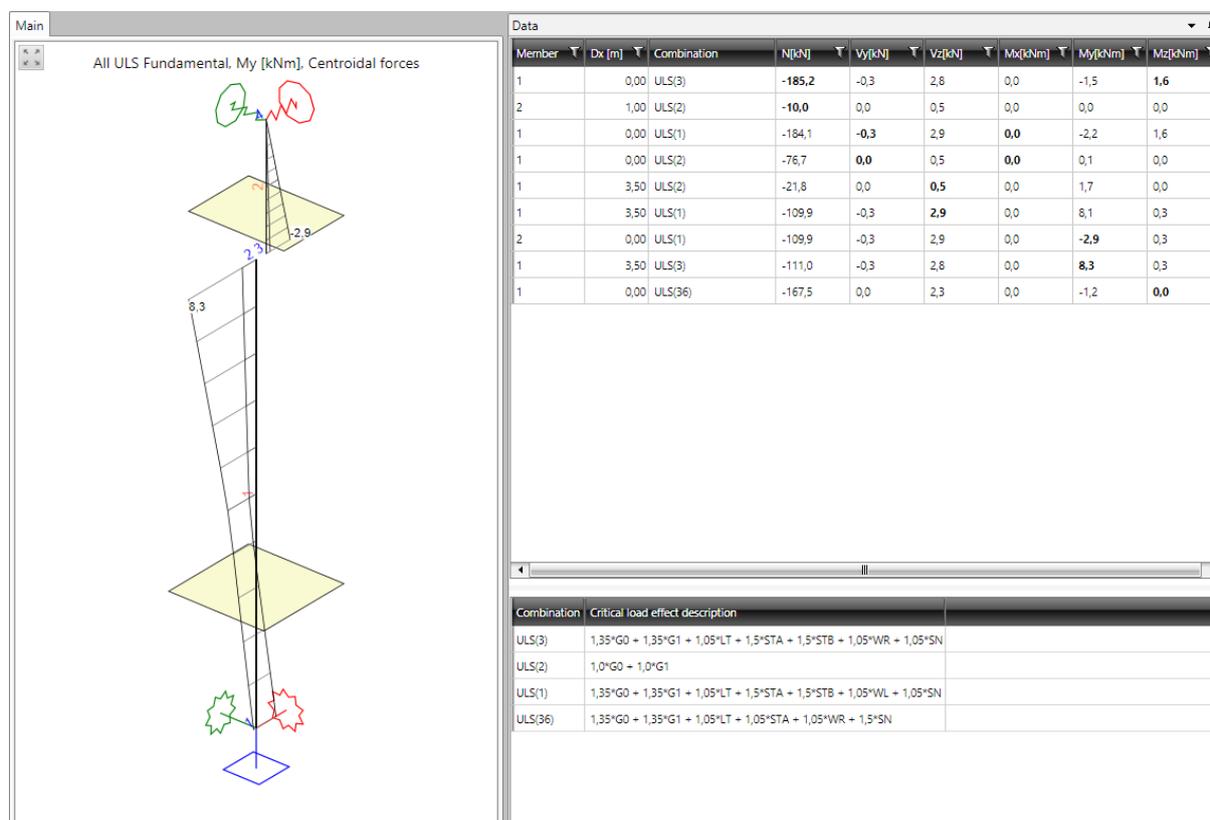
Click navigator command **Results** > **Internal forces** to evaluate column internal forces.

Calculated internal forces are evaluated:

- Graphically – diagrams of selected component of internal forces are displayed in the Main window.
- Textually – extreme values of internal forces are displayed in the table in Data window. The table with content of critical combinations is displayed, if the evaluation is performed for combination.

Internal forces are evaluated for current load case or combination.

Ribbon groups **View settings**, **Drawing**, **Results**, **Selection**, **Extreme** and **Internal forces** are available when evaluating internal forces.



6.3.1 Ribbon group View settings

See [6.1.1 Ribbon group View settings](#).

6.3.2 Ribbon group Drawing

See [6.1.2 Ribbon group Drawing](#).

6.3.3 Ribbon group Results

See [6.1.3 Ribbon group Results](#).

6.3.4 Ribbon group Selection

See [6.2.4 Ribbon group Selection](#).

6.3.5 Ribbon group Extreme

See 6.2.5 Ribbon group Extreme.

6.3.6 Ribbon group Internal forces



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

- **N** – switch to draw course of normal force N.
- **Vy** – switch to draw course of shear force Vy.
- **Vz** – switch to draw course of shear force Vz.
- **Mx** – switch to draw course of torsional moment Mx.
- **My** – switch to draw course of bending moment My.
- **Mz** – switch to draw course of bending moment Mz.

7 Design and check of structural items

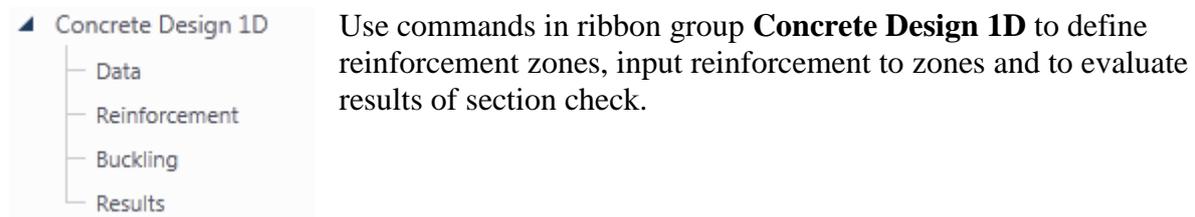
IDEA Column is an application for static analysis of the structure. Reactions in support, linear deformations and internal forces caused by acting load are results of this analysis.

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

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

8 Check of concrete members

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



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

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

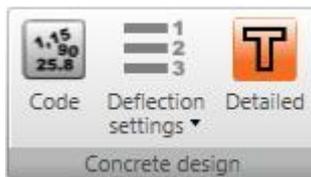
Extreme internal forces from the whole zone are found for the design.

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

- Column is of type Concrete Column.
- There are combinations of ULS, SLS Characteristic and SLS quasi-permanent.
- The project is calculated – internal forces are available.

8.1 Section check and deflection check settings

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



- **Code** – code and calculation settings – see **8.1.1 Code and calculation settings**
- **Deflection settings** – settings of parameters of loads, which are used for calculation of deflections – see **8.1.2 Setting of result class for calculation of deflections**.

8.1.1 Code and calculation settings

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

Section	Parameter	Value
Chapter 2	2.4.2.4(1) γ_c - Persistent, transient	1,5
	2.4.2.4(1) γ_c - Accidental	1,2
	2.4.2.4(1) γ_s - Persistent, transient	1,15
	2.4.2.4(1) γ_s - Accidental	1
Chapter 5	5.5 k1	0,44
	5.5 k2	$1,25 * (0,6 + 0,0014 / \epsilon_c)$
	5.5 k3	0,54
	5.5 k4	$1,25 * (0,6 + 0,0014 / \epsilon_c)$
	5.5 k5	0,7
	5.5 k6	0,8
Chapter 6	6.2.2 Values for shear check $d = h *$	0,9
	6.2.2 Values for shear check $z = d *$	0,9
	6.2.2(1) C_{rdc}	0,18

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

Buttons: Expand all, Collapse all, OK, Cancel

In the dialog **Code and calculation settings** setup values of coefficients referring to the current national code and national annex and general calculation settings can be changed.

8.1.2 Setting of result class for calculation of deflections

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

loads in variable load cases value of coefficient ψ_2 is used, which is taken from the load group, in which the load cases are assigned.



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

- **Result class** – select the result class for calculation of deflections. Click edit button  to edit the content of the selected result class – see **8.1.4 Editing**

result class.

8.1.3 Setting of result classes for section checks

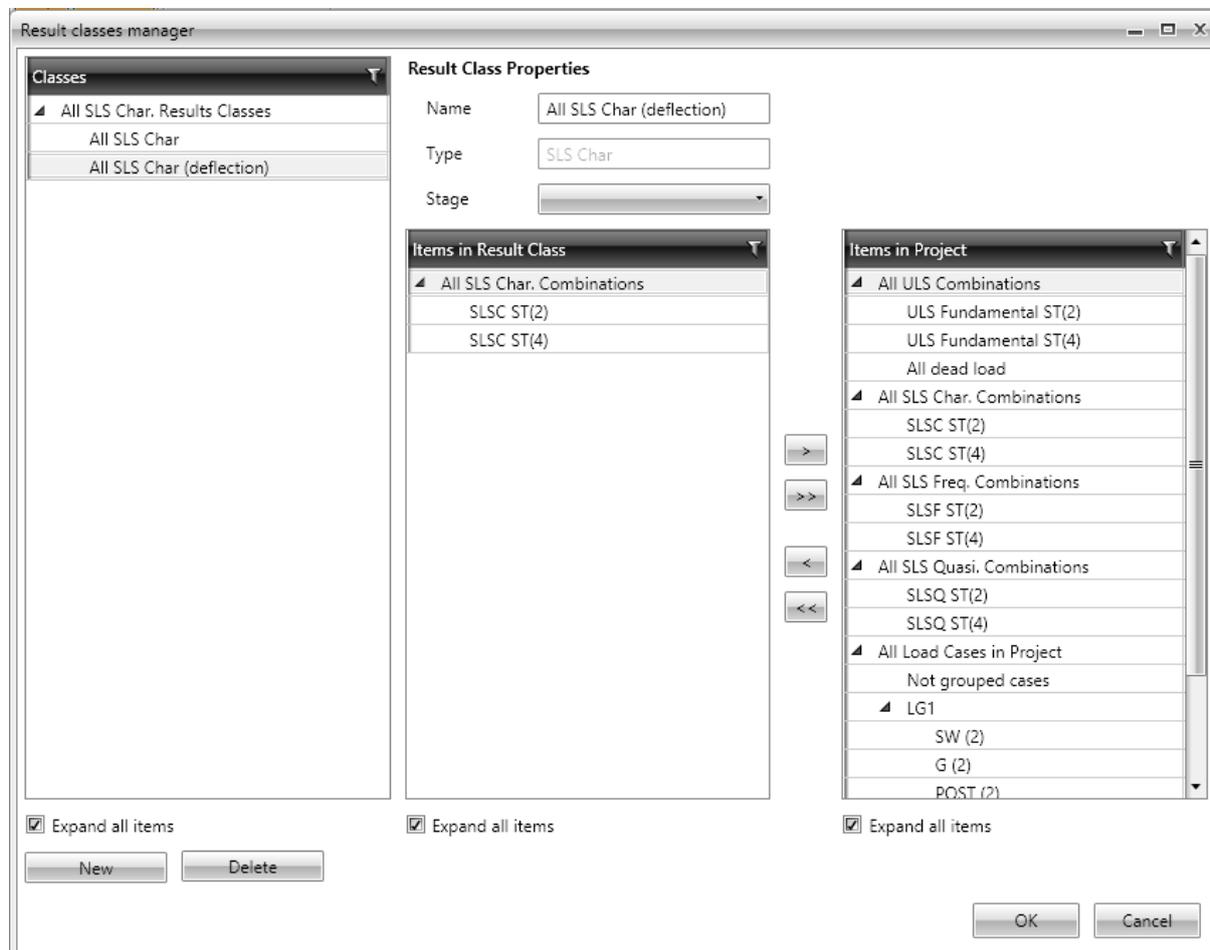
Result classes are required to be able to perform the sections checks of current design member.

The result classes are generated automatically in the background. All combinations of corresponding type are assigned to appropriate result classes. The content of result classes cannot be edited. Following combinations of load actions are used for check of concrete sections in IDEA RCS:

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

8.1.4 Editing result class

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



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

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

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.

8.2 Design member data

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

Individual concrete checks, which should be performed on current design member, can be selected in the first table.

▲ Ultimate limit state	
Capacity N-M-M	<input checked="" type="checkbox"/>
Shear	<input checked="" type="checkbox"/>
Torsion	<input checked="" type="checkbox"/>
Interaction	<input checked="" type="checkbox"/>
▲ Serviceability limit state	
Stress limitation	<input checked="" type="checkbox"/>
Crack width	<input checked="" type="checkbox"/>
▲ Detailing	
Detailing	<input checked="" type="checkbox"/>
▲ Deflections	
Deflections	Detailed calculation

Properties group **Ultimate limit state**:

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

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.
 - **Detailed calculation** – detailed calculation of deflections is performed, limit deflections can be checked.

Exposure classes and design member properties can be set in the second table.

Exposure class	
No corrosion (X0)	<input type="checkbox"/>
Carbonation	XC3 - Moderate humidity
Chlorides	XD1 - Moderate humidity
Chlorides from sea	No risk of chlorides from sea
Freeze/Thaw Attack	No risk of freeze/thaw attack
Chemical Attack	No risk of chemical attack
Relative humidity [%]	
	65
Creep coefficient	Calculated
Member type	Column
Structural member import.	Major
Second order moment	Real

Properties group **Exposure class**

- **No corrosion** – switch on/off exposure class with no risk of corrosion or attack X0.
- **Carbonation** – select exposure class for corrosion caused by carbonation.
- **Chlorides** – select exposure class for corrosion caused by chlorides.
- **Chlorides from sea** – select exposure class for corrosion caused by chlorides from sea.
- **Freeze/Thaw attack** – select exposure class for corrosion caused by freeze/thaw cycles.
- **Chemical attack** – select exposure class for corrosion caused by chemically aggressive environment.

- **Relative humidity** – input value of relative humidity.
- **Creep coefficient** – select mode to determine the creep coefficient value:
 - **Calculated** – creep coefficient value is calculated automatically.
 - **User input** – value of creep coefficient Φ_{inf} can be defined by user.
- **Structural member importance** – select type of structural member importance according to 6.2.1(4).

8.3 Reinforcement zones

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

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

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

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

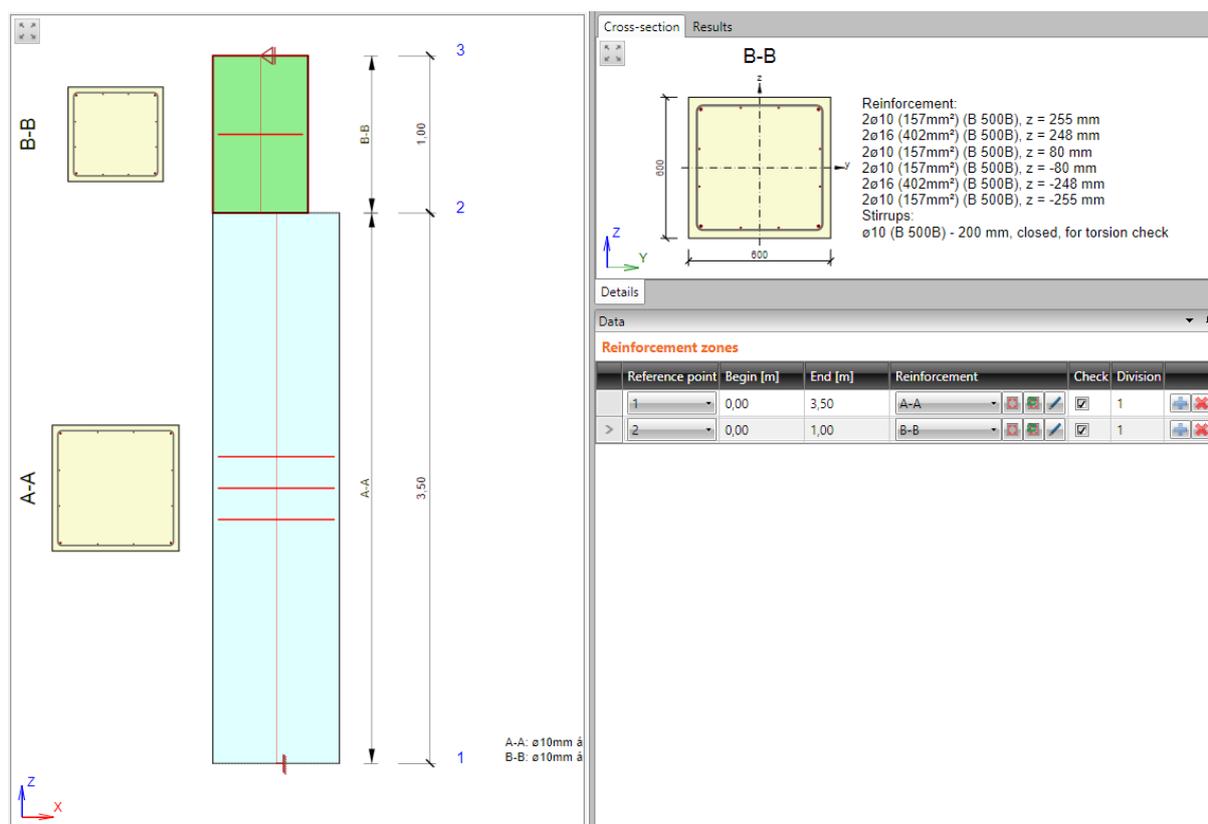


Table **Reinforcement zones** contains following columns:

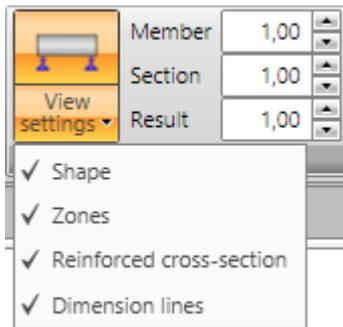
- **Reference point** – set the number of node. The coordinates defined in columns **Begin** and **End** are related to this point.
- **Begin** – position of zone beginning measured from reference point.
- **End** - position of zone beginning measured from reference point.
- **Reinforcement** – select the reinforcement template associated to the zone:
 -  - launches **Reinforcement editor** to input and modify the reinforcement in the current reinforcement template – see **8.4 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 data 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.

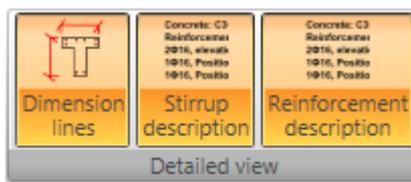
8.3.1 Ribbon group View settings and scale



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

- **Shape** – turn on/off the drawing of outer outlines of design member, respecting the assigned cross-sections. Otherwise only axial scheme is drawn.
- **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.
- **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...)

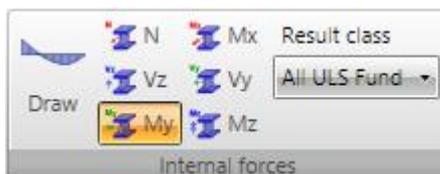
8.3.2 Ribbon group Detailed view



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

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

8.3.3 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 draw the course of axial force N.
- **Vy** – switch to draw the course of shear force Vz.
- **Vz** – switch to draw the course of shear force Vz.
- **Mx** – switch to draw the course of torsional moment Mx.
- **My** – switch to draw the course of bending moment My.
- **Mz** – switch to draw the course of bending moment Mz.

8.4 Editor of reinforcement

Input of current reinforcement template can be started

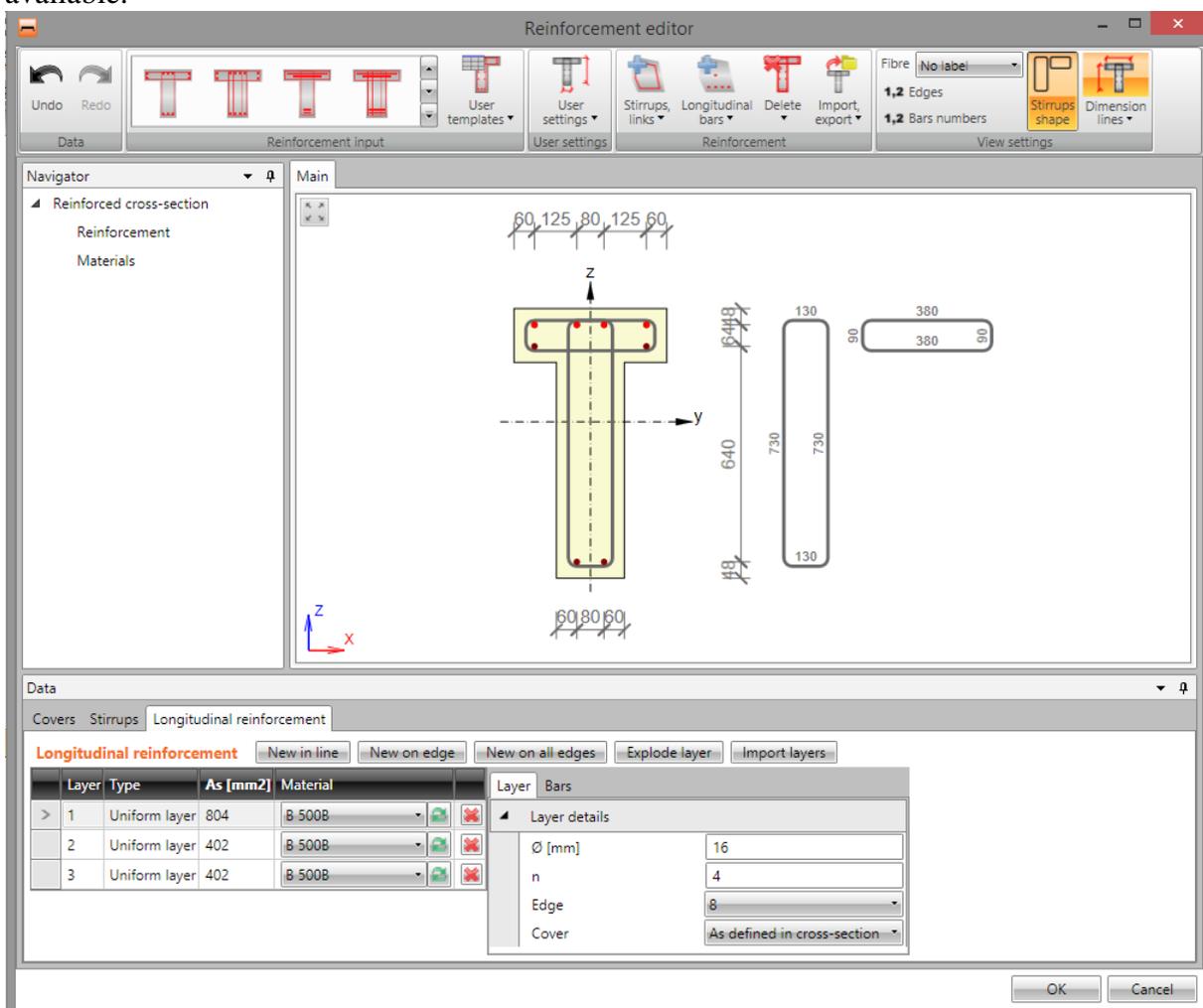
- Clicking the edit button  in the **Reinforcement** column in the zones table in the Data window
- Clicking the picture of section above the zone in the Main window.

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

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

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

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



8.4.1 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:

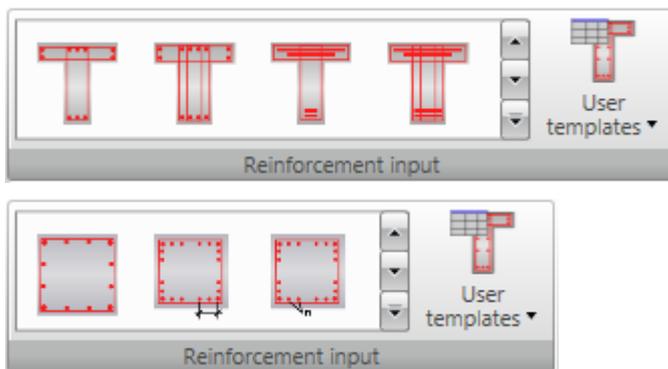
- for individual cross-section edges, if option **All edges** is selected.

- for individual cross-section surfaces, if option **All edges**

Covers All edges is not selected.

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

8.4.2 Input of reinforcement by template

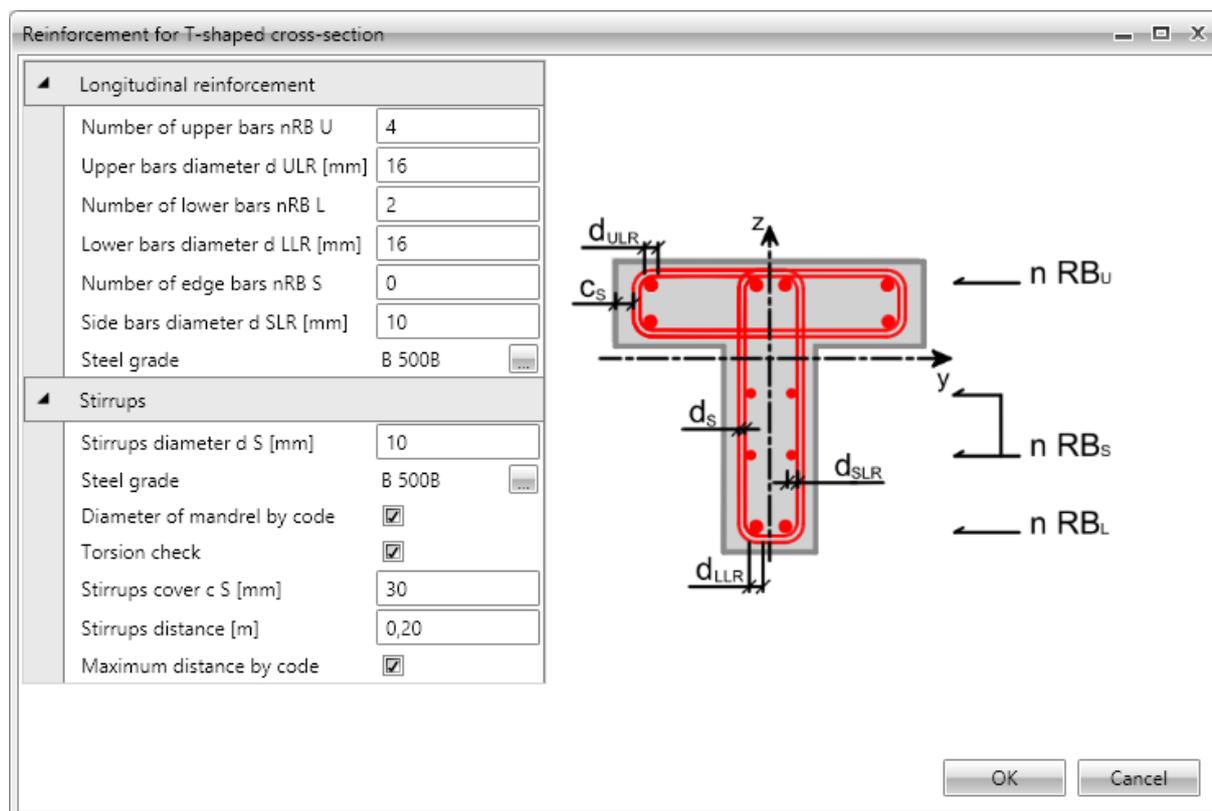


Reinforcement templates are available for some pre-defined sectional shapes. Reinforcement templates available for the current cross-section are displayed in ribbon group **Reinforcement**.

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

- **User templates** – input cross-section reinforcement using user defined templates of reinforcement – see **8.4.9 User defined reinforcement templates**.

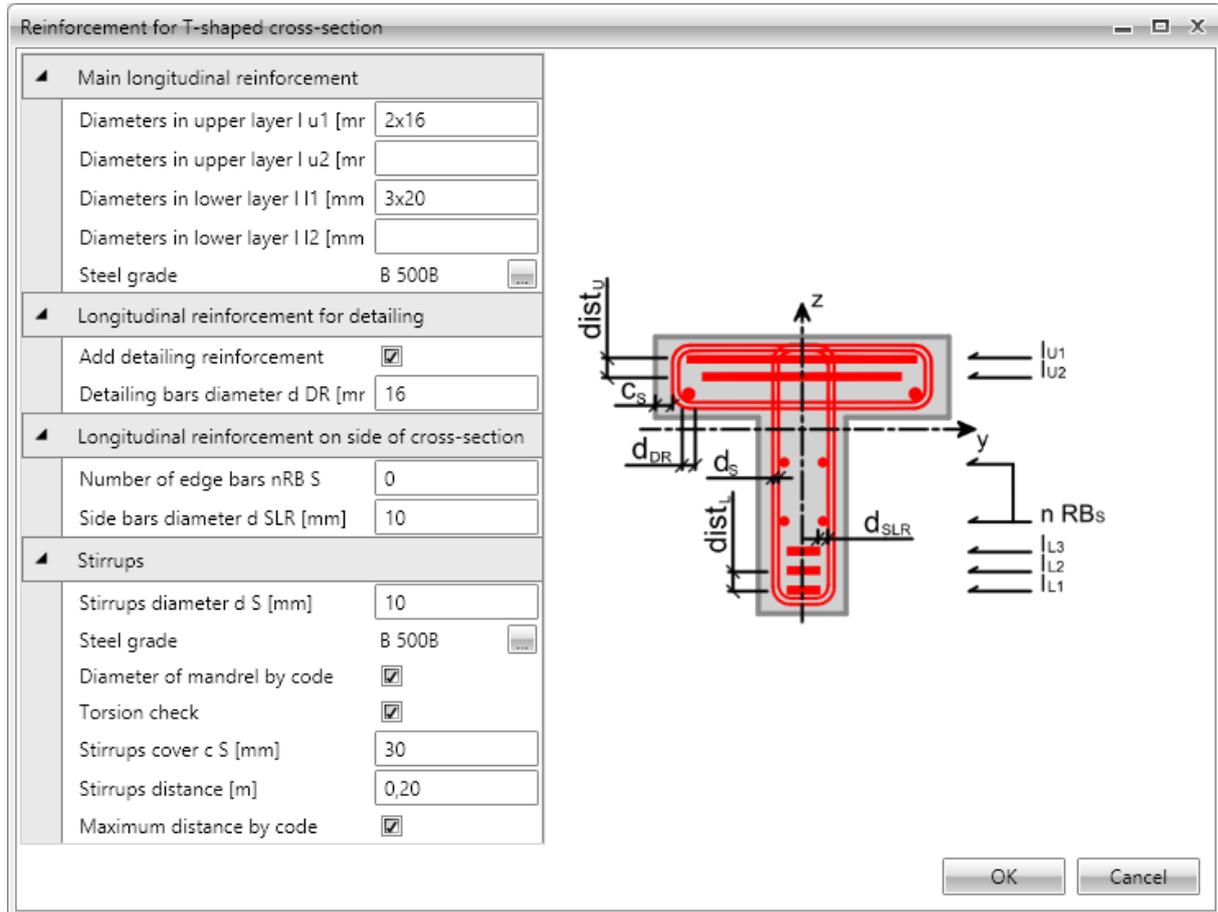
Reinforcement template parameters for T-shaped cross-section (including the reinforcement design):



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.

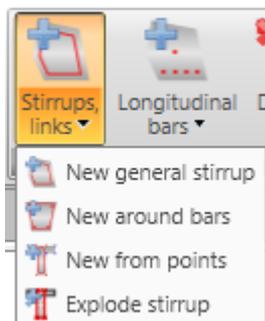
The reinforcement layer is defined by string, which describes diameters of individual bars in the layer. Individual diameters are separated by space, characters 'x' or '*' can be used to define multiple diameters, e. g. ,20 16 16 20' or ,20 2*16 20'.



8.4.3 Shear reinforcement

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

8.4.3.1 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.

The screenshot displays a 3D model of a column with a T-shaped top. The top flange has vertices labeled 1 through 8. A coordinate system with x, y, and z axes is shown. A red rectangular box highlights a stirrup with a height of 1632 mm and a width of 132 mm. To the right, a detailed view of the stirrup shows its dimensions: a total height of 382 mm, a width of 264 mm, and a hook length of 89 mm. Below the model is a 'Data' panel with tabs for 'Covers', 'Stirrups', and 'Longitudinal reinforcement'. The 'Stirrups' tab is active, showing a table with columns for Type, Ø [mm], Material, Distance [mm], Shear, and Torsion. Two rows of data are visible, both for a diameter of 8 mm and a distance of 200 mm. A 'Stirrup detail' window is open, showing a value of 2.5 dm.

Type	Ø [mm]	Material	Distance [mm]	Shear	Torsion	Stirrup	Vertexes
> 1	8	B 500B	200	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	Stirrup detail
2	8	B 500B	200	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	n dm <input type="text" value="2.5"/>

8.4.3.1.1. General stirrups

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

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

Properties group **Stirrup detail**:

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

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

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

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

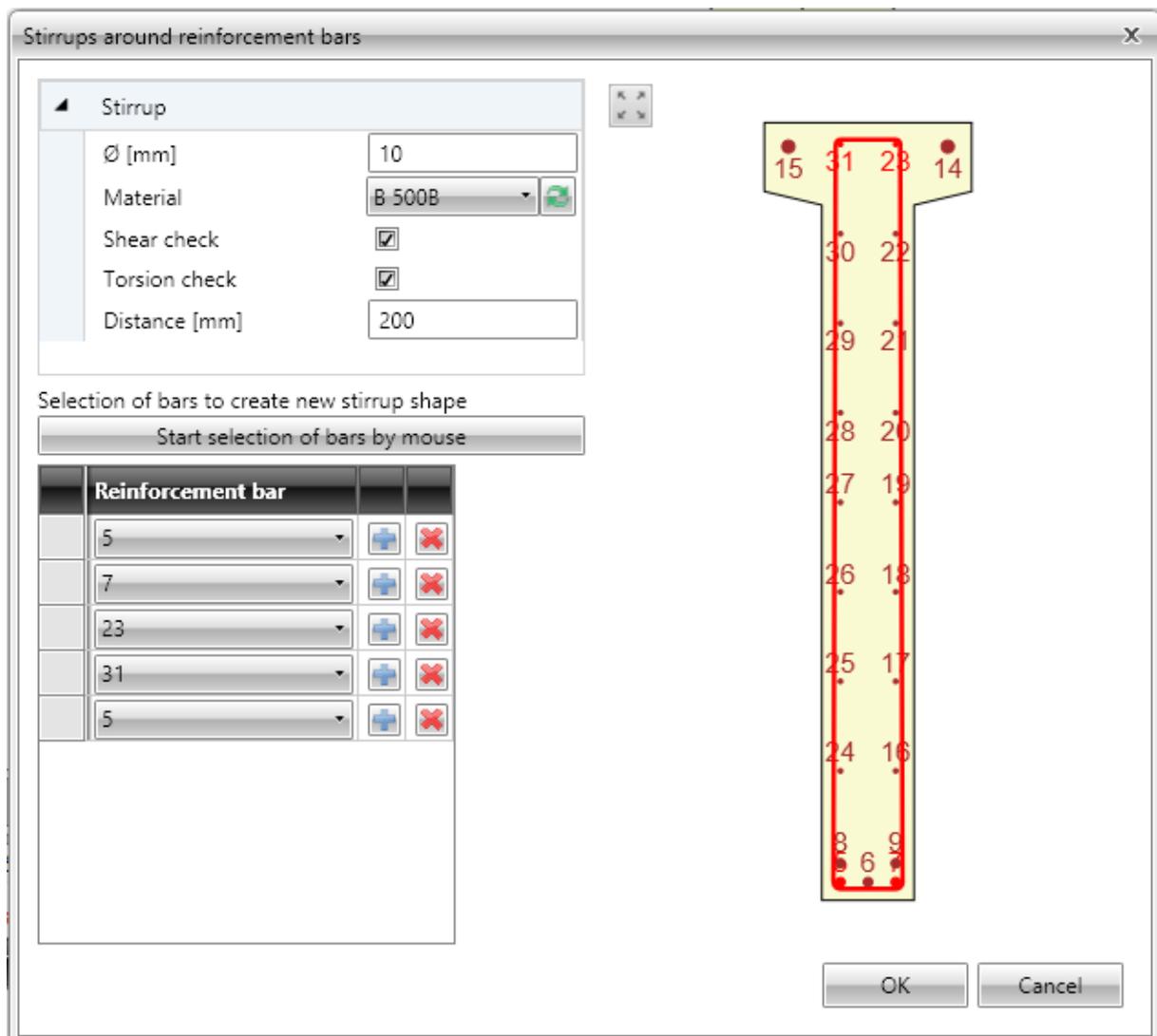
8.4.3.1.2. Stirrups around bars of longitudinal reinforcement

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.

8.4.3.1.3. Stirrup by cross-section vertices

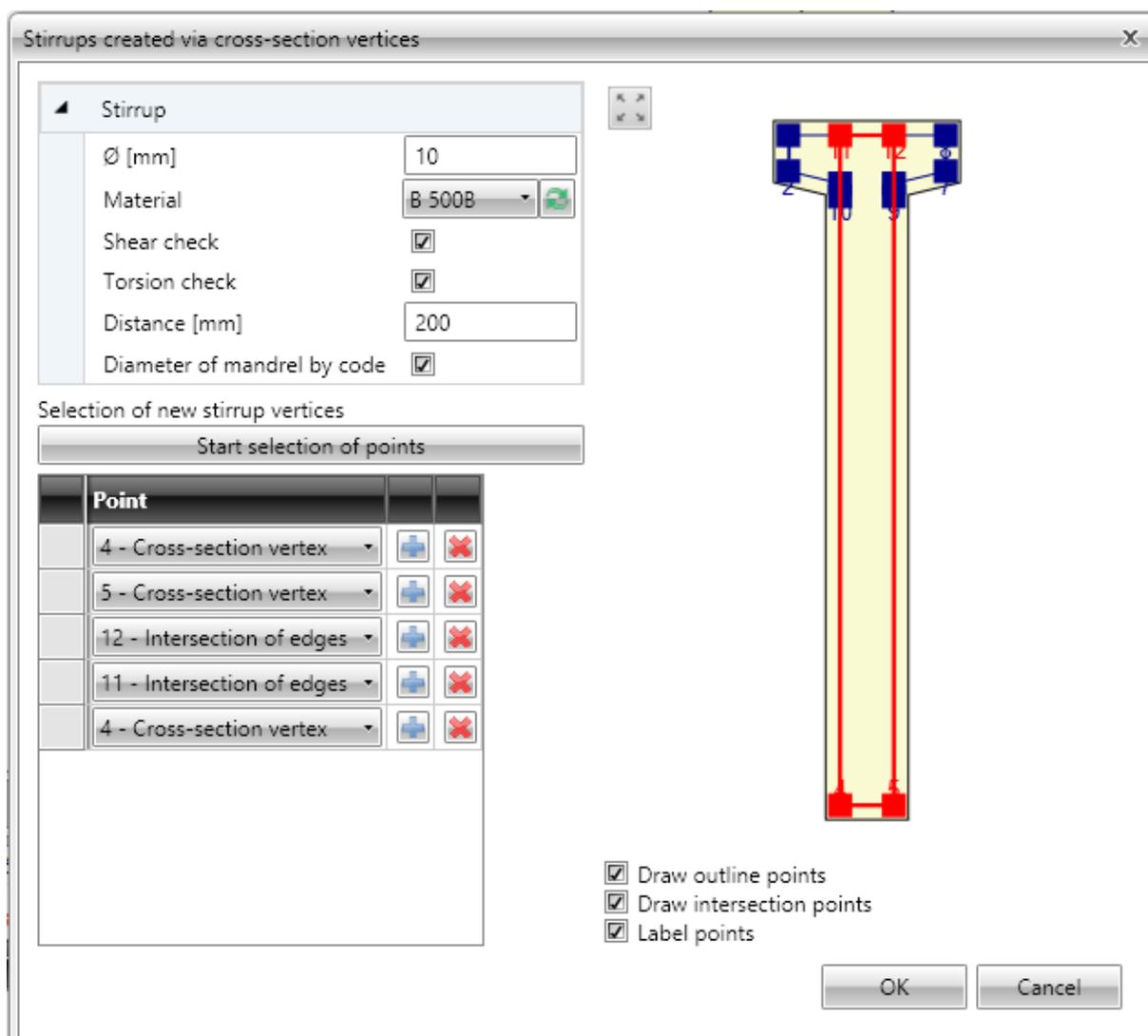
To input stirrup around main reinforcement click **Stirrups, links** > **New from points** in ribbon group **Reinforcement** or click **New from points** above the table **Stirrups**.

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.

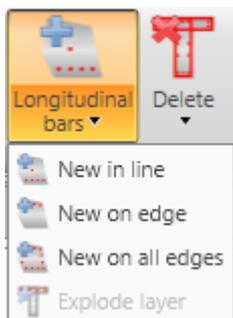
- **Shear check** – if selected, stirrup is taken into account for shear check.
- **Torsion check** – if selected, stirrup is taken into account for torsion check.
- **Distance** – input value of longitudinal distance between stirrups.
- **Diameter of mandrel by code** – switch on/off automatic determination 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.

8.4.3.1.4. Exploding stirrups

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

8.4.4 Longitudinal reinforcement

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



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

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

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

Bar diameter and material can be assigned to individual layers.

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

The screenshot shows the 'Longitudinal reinforcement' configuration window. It includes a table with 8 layers and a configuration panel on the right.

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

Configuration panel (Layer details):

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

Columns in **Tendons** table:

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

8.4.4.1 Layer of reinforcement by coordinates

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

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

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

Properties group **Layer details**:

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

Properties group **First point**:

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

Properties group **Last point**:

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

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

8.4.4.1.1. Layer details

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

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

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

8.4.4.2 Layer of reinforcement on edge

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

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

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

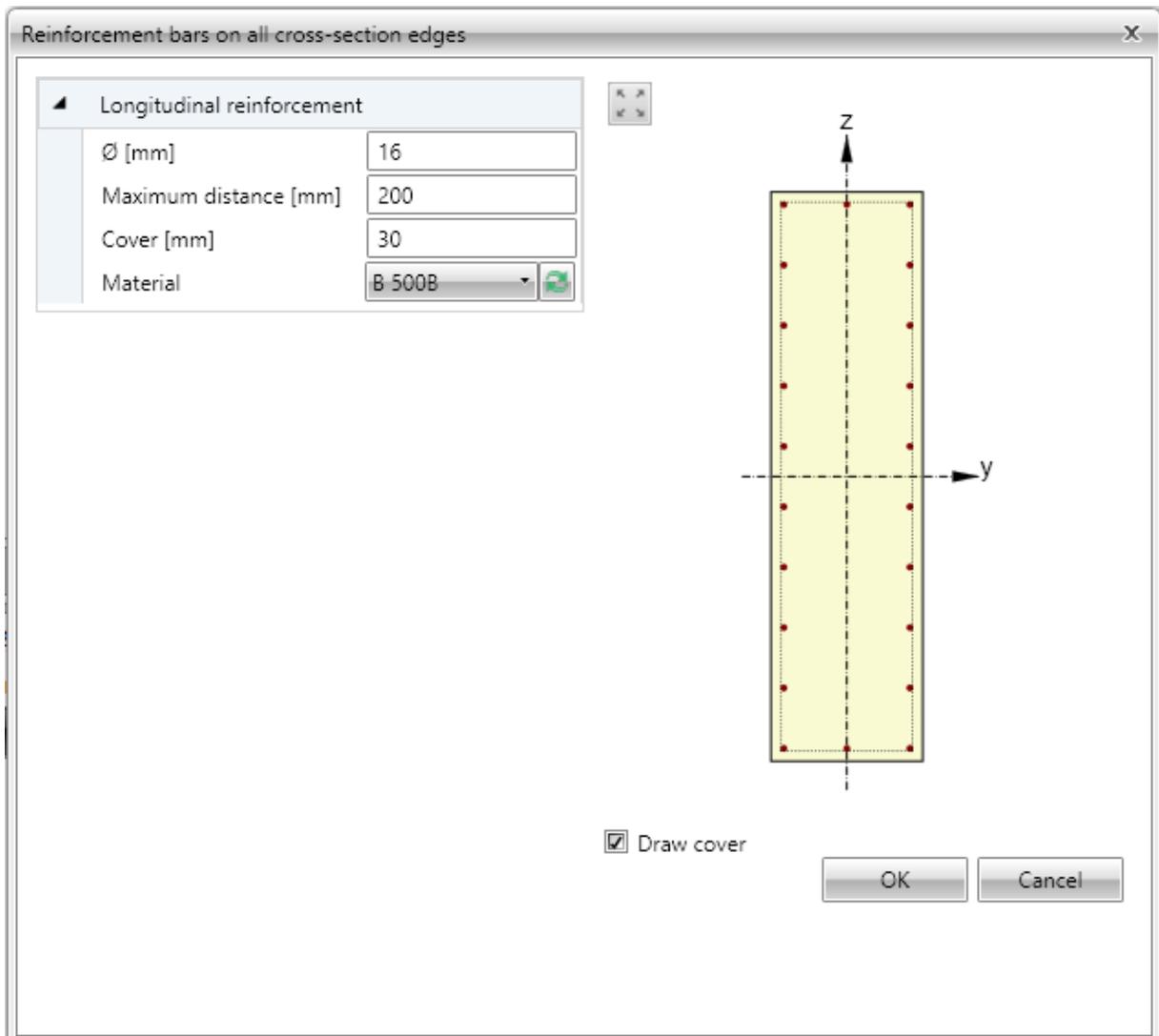
Properties group **Layer details**:

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

Properties of individual bars of the layer are displayed on tab **Bars** – see **8.4.4.1.1 Layer details**.

8.4.4.3 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 **8.4.4.1.1 Layer details**.

8.4.4.4 Layer of reinforcement to wave of trapezoidal sheet

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

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

Properties group **Layer details**:

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

Properties of individual bars of selected layer are displayed on tab **Bars** - see **8.4.4.1.1 Layer details**.

8.4.4.5 Layer of reinforcement by spacing

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

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

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

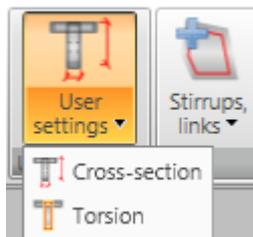
Properties group **Layer details**:

- **Ø** – input the diameter of bar in the layer.
- **Distance** – input the distance between axes of adjacent bars.
- **n** – the calculated number of bars per meter in layer is displayed.
- **Edge bar specification** – select the mode to determine the position of the edge bar. One of following modes can be selected:

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

Properties of individual bars of selected layer are displayed on tab **Bars** - see **8.4.4.1.1 Layer details**.

8.4.5 User settings of reinforced cross-section



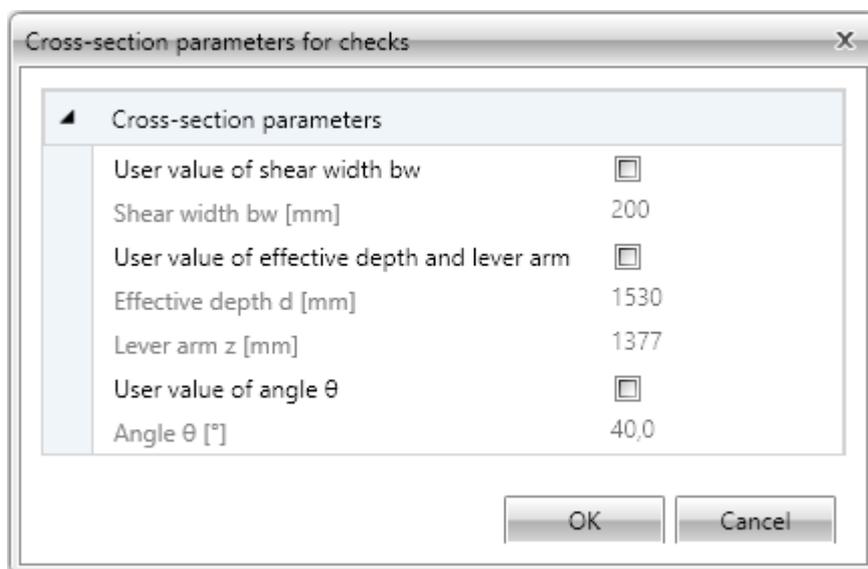
Ribbon group **User setting** contains commands to modify some shear and torsion calculation parameters of reinforced cross-section:

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

8.4.5.1 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**.

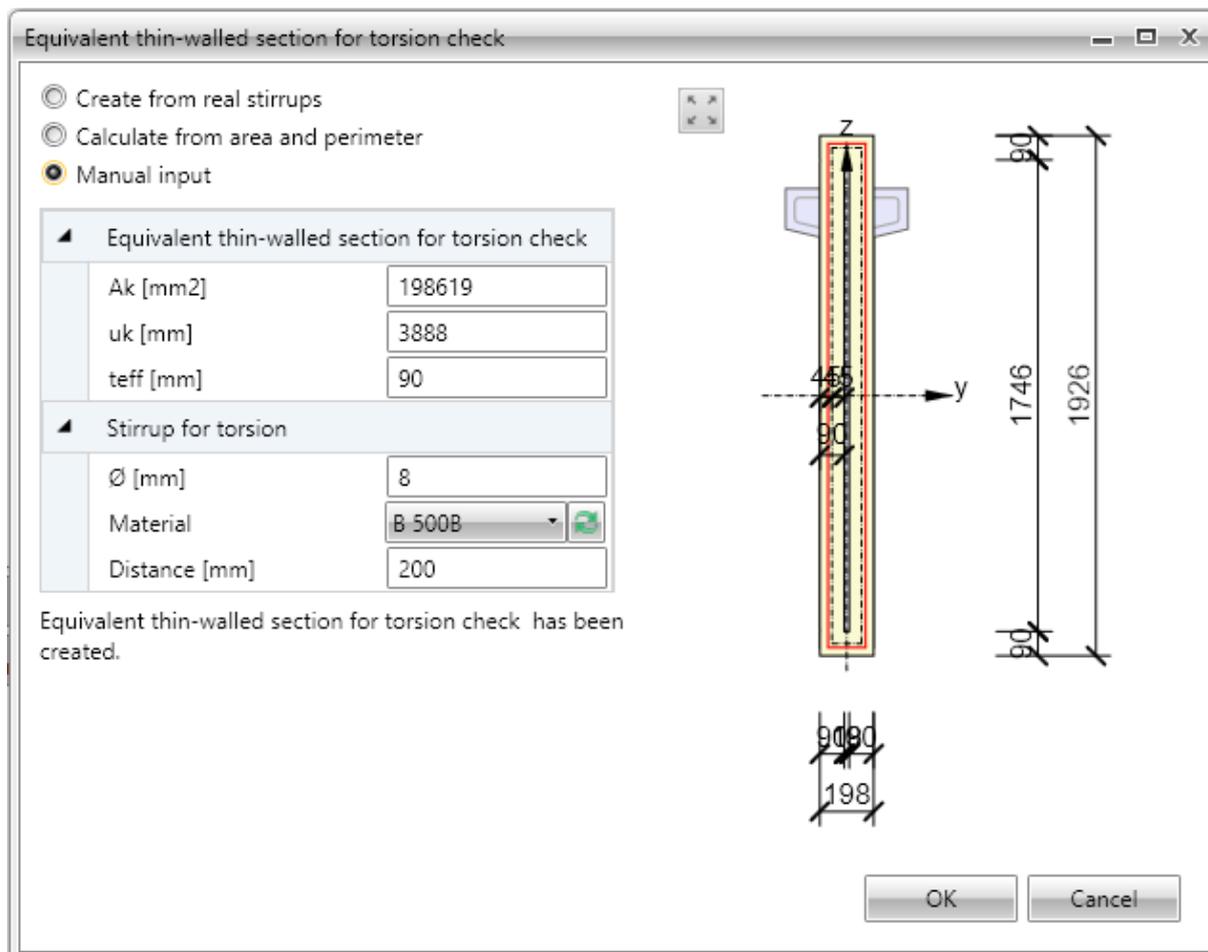


Calculated values of effective cross-section dimensions and overridable values of shear check from code and calculation settings are displayed in dialog. To enable input of user defined values check particular checkbox in first column.

8.4.5.2 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.

8.4.6 Deleting reinforcement

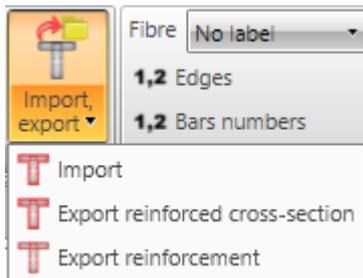


Use commands in ribbon group **Delete** to delete reinforcement from cross-section:

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

8.4.7 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.

8.4.8 View settings of reinforced cross-section



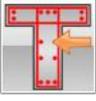
Ribbon group **View settings** can be used to set drawing options of reinforced cross-section:

- **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.

8.4.9 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:

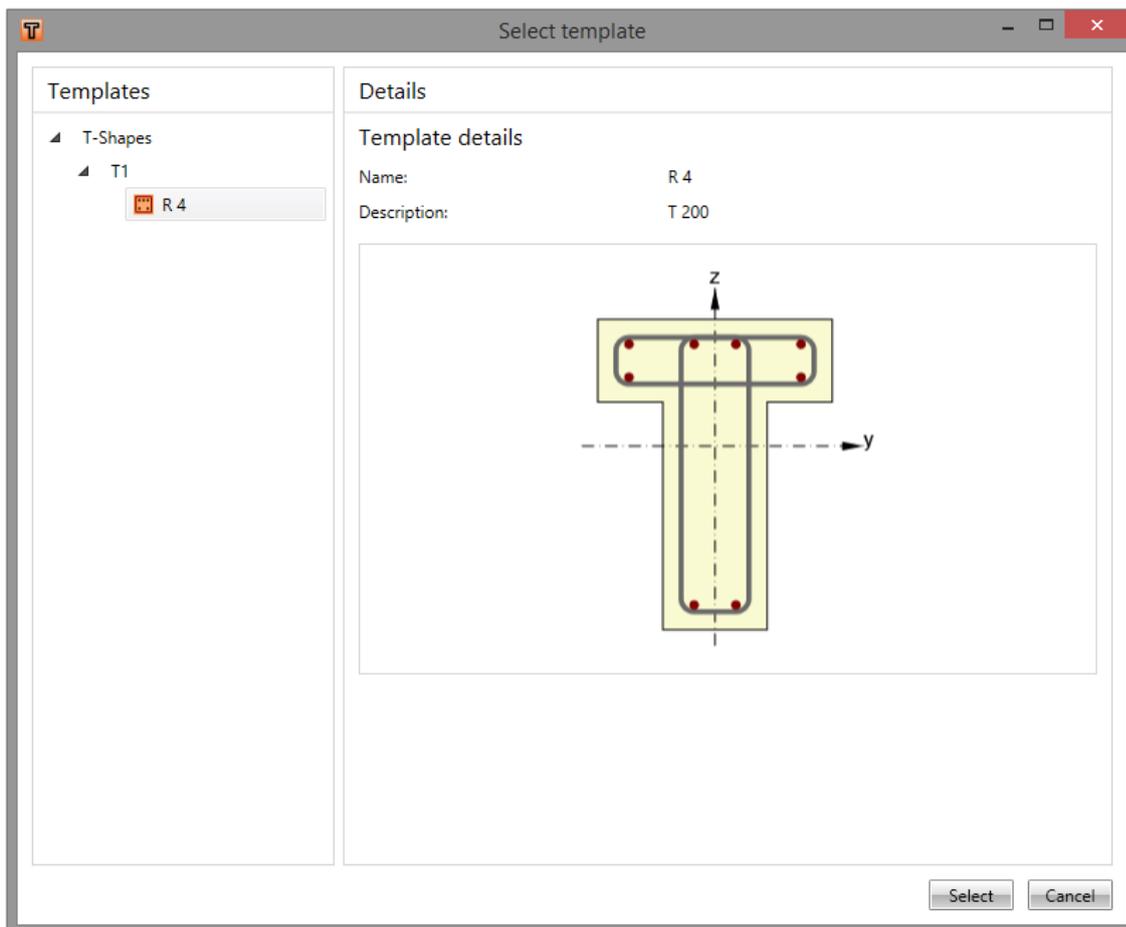
-  – starts input of reinforcement using the user defined reinforcement template – see **8.4.9.1 Reinforcing by user defined reinforcement template**.
-  – stores the current reinforcement into the database of user reinforcement templates. Dialog **Add template** appears. The target folder must be selected 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 **8.4.9.2 Templates manager**.

8.4.9.1 Reinforcing by user defined reinforcement template

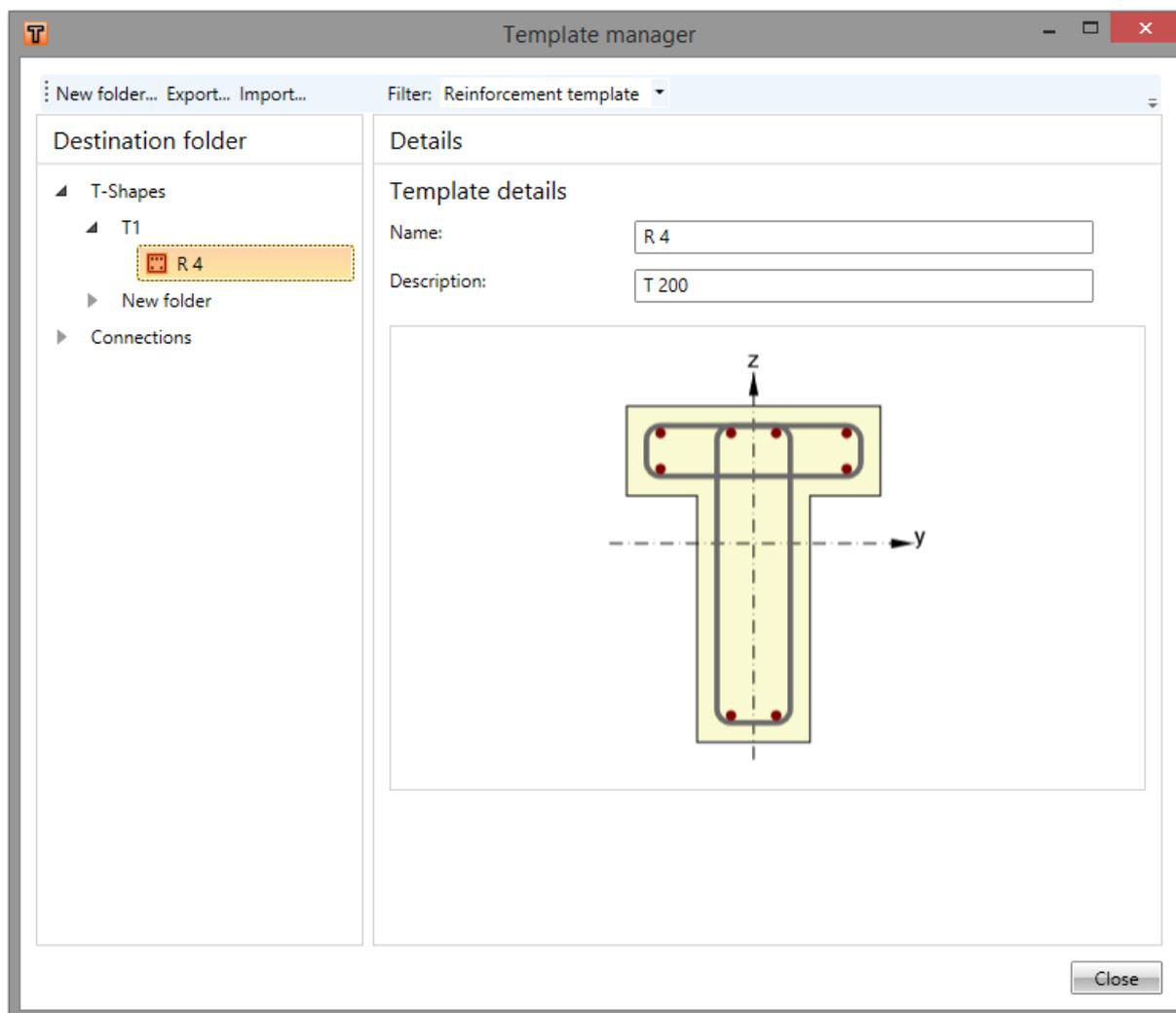
Dialog **Select template** appears after start of reinforcing by user defined reinforcement template.

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.



8.4.9.2 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.

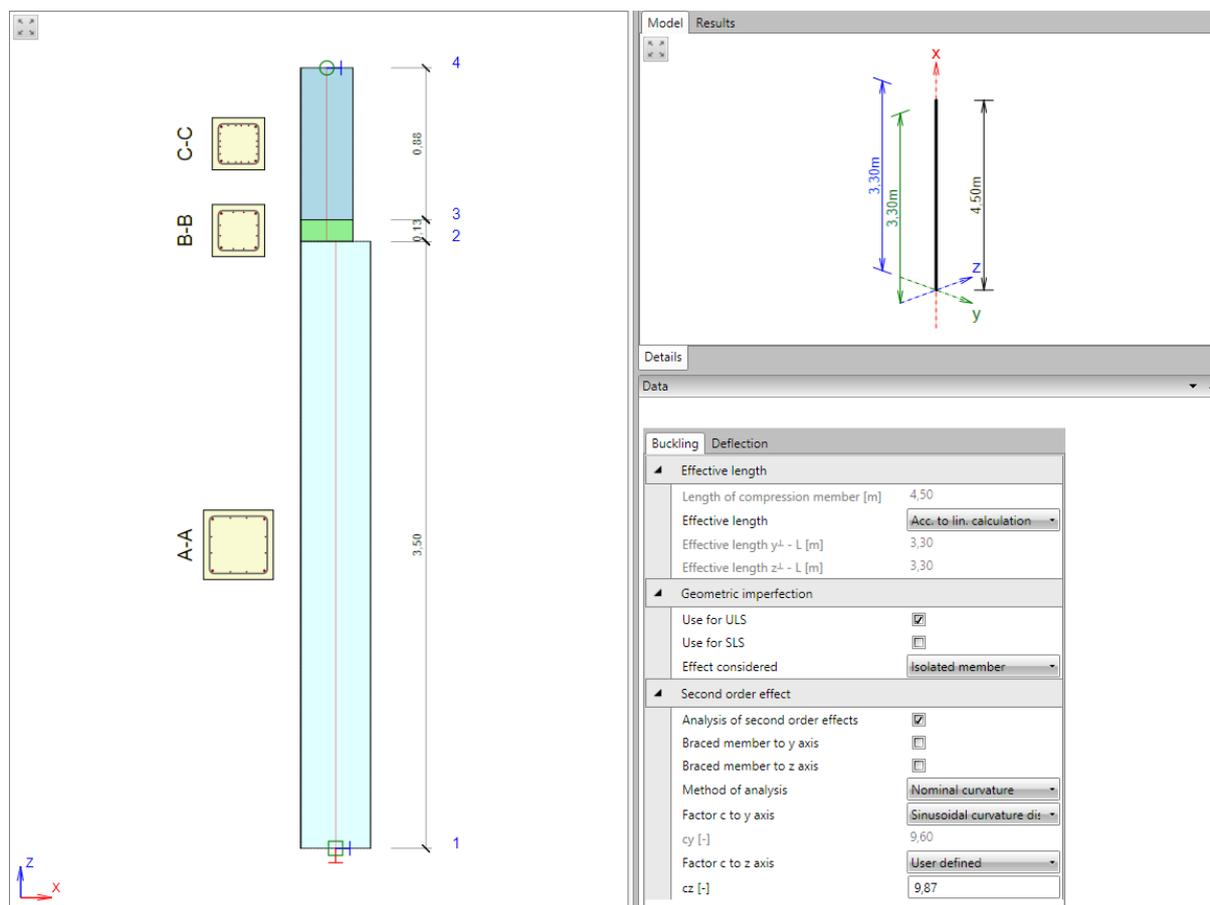
Import templates – by command **Import...** in the main menu. Templates from the selected file with extension *.EXP are imported into the database of templates.

8.5 Buckling data and deflection check data

Click navigator command **Concrete design 1D > Buckling / Deflection** to define the data for calculation of buckling effects and deflection check.

Following tabs are displayed in the Data window:

- Tab **Buckling** to define parameters for determination of effective lengths and second order effect.
- Tab **Deflections** to define parameters for check of limit deflections.



8.5.1 Data for buckling effects calculation

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.
 - **Acc. to supports** – effective lengths are determined automatically according to column supports.
 - **Acc. to lin. calculation** – effective lengths are determined using approximate formulas from deformations caused by special additional load cases.
 - **Effective length y** – input/print the effective length for buckling perpendicular to y-axis.
 - **Effective length z** – input/print the effective length for 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**

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 c_0 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.

8.5.2 Data for deflection check

Data for limit deflection check are defined on **Deflection** tab:

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

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

- **Limit is defined as numerical value** – if selected, the values of limit deflection are defined as absolute value (in length units). Otherwise the limit deflection value is calculated as ratio of current span length.
- **User-defined value of limit deflection** – if selected, the ratio of span length to define the limit deflection can be defined by user.
- **Check acc. to 7.4.1 (4)** – switch on/off the deflection check with respect to possible impairment of the appearance and general utility of the structure acc. 7.4.1 (4).

- **Limit value for deflection as length of span /** - display or input the ratio of span length to determine the limit value of deflection.
- **Numerical value of deflection limit** – user-defined absolute value of deflection limit.
- **Check acc. to 7.4.1 (5)** – switch on/off the deflection check with respect to possible impairment of the appearance and general utility of the structure.
 - **Limit value for deflection as length of span /** - display or input the ratio of span length to determine the limit value of deflection.
 - **Numerical value of deflection limit** – user-defined absolute value of deflection limit.

8.6 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 Column.

8.7 Results evaluation

Click navigator command **Concrete design 1D > Results** to start the checks and 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.

Either the courses of summary check or particular section check along the design member can be evaluated graphically or the interaction diagrams for individual zones of the design member can be drawn.

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

Ribbon group **Check** is available when evaluating courses of section check results along the design member.

Ribbon groups **Interaction surface sections**, **Drawing settings** and **Colours settings** are available when drawing interaction diagrams.

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

8.7.1 Ribbon group Concrete design

See **8.1 Section check and deflection check settings** and **8.6 Detailed check**.

8.7.2 Ribbon group View settings and scale

See **8.3.1 Ribbon group View settings and scale**.

8.7.3 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.

8.7.4 Ribbon group Calculation



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

8.7.5 Ribbon group Results drawing



The mode of graphical evaluation can be changed using commands in this group.

- **Section check** – switch to draw courses of check results along the design member.
- **Deflection check** – switch to evaluation of deflection check and calculated stiffnesses.
- **Interaction diagrams** – switch to draw interaction diagrams for selected zone or subzone of the design member.

8.7.6 Drawing of section check results courses

8.7.6.1 Ribbon group Check

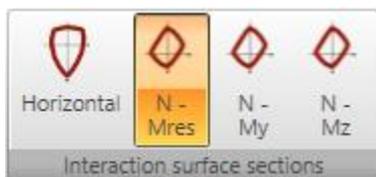


Use options in ribbon group **Check** to set the evaluation of particular check result along the design member.

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

8.7.1 Drawing of interaction diagrams

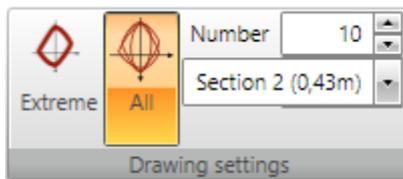
8.7.1.1 Ribbon group Interaction surface sections



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

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

8.7.1.2 Ribbon group Drawing settings



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

8.7.1.3 Ribbon group Colors settings

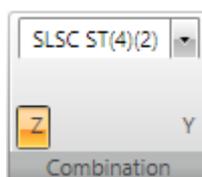


Use commands in this ribbon group to set the drawing colors of interaction diagrams..

- **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.

8.7.2 Drawing of deflections check results

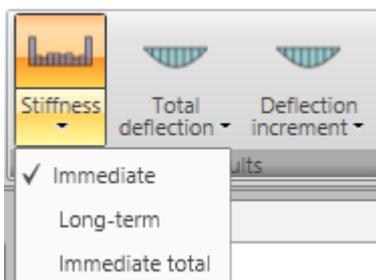
8.7.2.1 Ribbon group Combination



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

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

8.7.2.2 Ribbon group Type of results

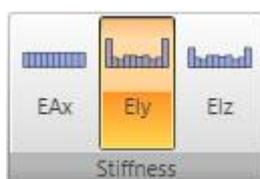


Use the options in ribbon group **Type of results** to set the results drawing mode:

- **Stiffness** – switch to drawing of calculated stiffnesses for the current combination along the design member:
 - **Immediate** – switch to drawing of stiffnesses for calculation of immediate effects of long-term load components for the current combination.
 - **Long-term** – switch to drawing of stiffnesses for calculation of long-term effect of long-term load components for the current combination..
 - **Immediate total** – switch to drawing of stiffnesses 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 on/off drawing of deflections from linear calculation for the current combination.

- **Immediate** – switch on/of drawing of immediate deflections (calculated from short-term stiffnesses) from total load for the current combination.
- **Long-term** – switch on/off drawing of long-term deflections (calculated including the effects of creep) from the long-term loads for the current combination.
- **Total** – switch on/off drawing of total deflections (calculated including the effects of creep) for the current combination.
- **Limit** – switch on/off drawing of limit deflections.
- **Deflection increment** - switch to drawing of calculated deflection increments for the current combination along the design member:
 - **Increment** - switch on/off drawing of deflection increment for the current combination.
 - **Limit** - switch on/off drawing of limit deflection increment.

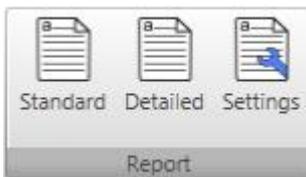
8.7.2.3 Ribbon group Stiffness



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
- **EIz** – switch to drawing of flexural stiffness EIz

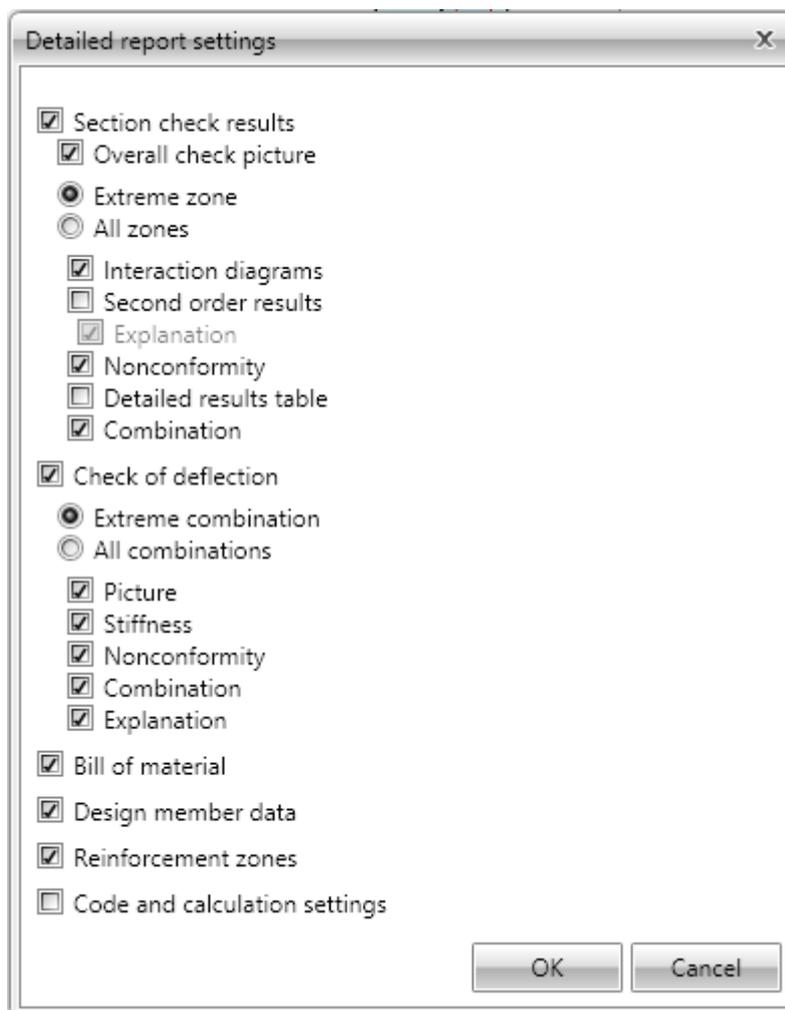
8.7.3 Check report



To generate and print the calculation report of the current design group 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.

8.7.3.1 Report settings



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

Particular dialog options:

- **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 the 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 of critical combinations.

- **Check of deflection** – switch on/off the print of all results of deflection calculations.
 - **Extreme combinations** – if the option is selected, results of deflection check will be printed for the combination, which caused the extreme check value.
 - **All combinations** – if the option is selected, results of deflection check will be printed for all combinations.
 - **Picture** – switch on/off the print of the picture with the courses of deflections.
 - **Stiffness** – switch on/off the print of the tables with stiffnesses.
 - **Nonconformity** - switch on/off the print of the table with nonconformities.
 - **Combination** – switch on/off the print of the table with description of combinations for calculation of deflections.
 - **Explanation** - switch on/off the print of the table with nonconformities.

- **Bill of material** – switch on/off 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.
- **Code and calculation setup** – switch on/off the print of table with national code and calculation coefficients values.

9 Steel members design

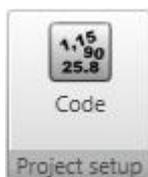
▲ Steel Design	Use commands in navigator group Steel design to input design data, buckling lengths, to set design options and to perform and evaluate the check of steel members.
Options	
Design Data	
Buckling Lengths	To be able to run the design of steel members, following conditions must be met:
Results	

- The column contains members with steel cross-sections.
- USL and SLS (characteristic) combinations are defined.
- Design members and design groups of steel members are defined.
- The results of linear calculation are available.

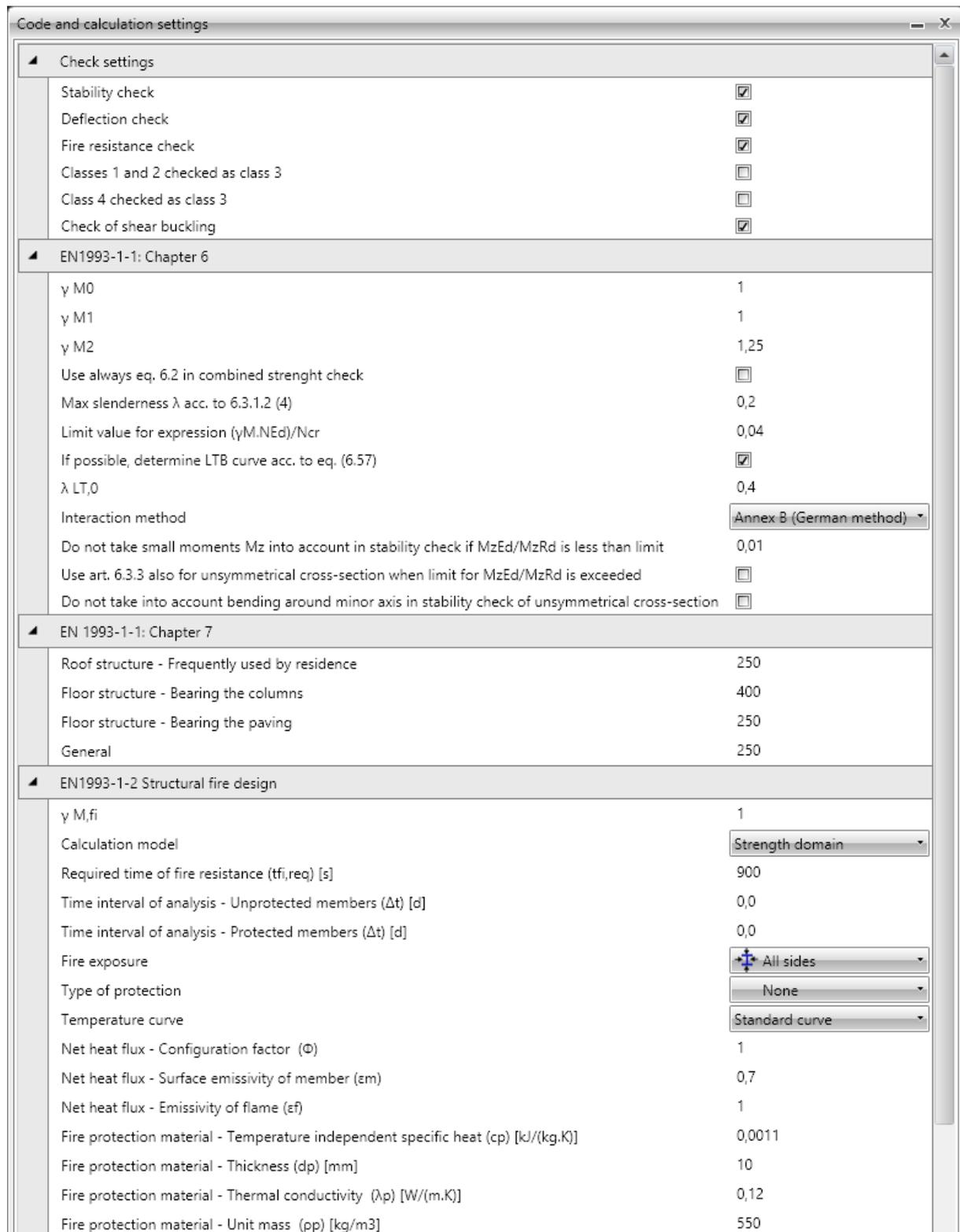
The design data are defined for the current design group, which can be set in the navigator **Design elements**. The detailed results evaluation can be performed for the current design group too.

Only one design member is automatically created in IDEA Column. This design member is automatically assigned to a design group and is set as the current design member.

9.1 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 class 2 are checked using articles for class 3.
- **Class 4 cross-checked as class 3** - if the option is selected, cross-sections classified as class 4 are checked using articles for class 3. The check of class 4 cross-sections is not supported. If the option is not selected and the cross-section is classified as class 4, the check value of cross-section is set to 500%.
- **Check of shear buckling** – if the option is selected, the limit slenderness of cross-section web is checked. If the limit slenderness is checked and is exceeded, the check value of cross-section is set to 500% – the check according to EN1993-1-5 should be performed in such case, but it is not supported.

Group EN1993-1-1: Chapter 6:

- **γ_{M0}** – input value of partial factor for resistance of cross-sections whatever the class is.
- **γ_{M1}** – input value of partial factor for resistance of members to instability assessed by member check.
- **γ_{M2}** – input value of partial factor for resistance of cross-sections in tension to fracture.
- **Use always eq. 6.2 in combined strength check** – if the option is selected, the equation 6.2 is used when analysing the section resistance. Otherwise the equation 6.41 is used.
- **Max slenderness λ acc. to 6.3.1.2 (4)** – input the limit value of relative slenderness to neglect the buckling resistance check according to 6.3.1.2 (4).
- **Limit value for expression $(\gamma_{M.NEd})/N_{cr}$** – input the limit value of formula to neglect the buckling resistance check according to (6.3.1.2(4)).
- **If possible, determine LTB curve acc. to eq. (6.57)** – if the option is selected and it is possible to determine LTB curve according to (6.57), the determined LTB curves are used. If the curves cannot be determined according to (6.57) or if the option is not selected, the buckling curves for LTB are determined according to (6.56).
- **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 M_z into account in stability check if M_{zEd}/M_{zRd} is less than limit** – input the limit value when the influence of M_z can be neglected for checks of unsymmetrical cross-section under compression and bending checked acc. to art. 6.3.4 or modified method 6.3.3 for mono-symmetrical cross-section where only in plain bending can be used. When limit is exceeded the check fails and error is written in report if the other options do not cover this case.
- **Use art. 6.3.3 also for unsymmetrical cross-section when limit for M_{zEd}/M_{zRd} is exceeded** – if the option is selected, members with unsymmetrical cross-section are checked acc. to art. 6.3.3 when art. 6.3.4 or alternative method for mono-symmetrical

cross-section cannot be used. The utilisation reserve to cover possible inaccuracy must be ensured.

- **Do not take into account bending around minor axis in stability check of unsymmetrical cross-section** – if the option is selected, bending around minor axis is neglected. This option allows to use art. 6.3.4 or alternative method 6.3.3 for mono-symmetrical cross-section always. Then utilisation reserve to cover possible inaccuracy must be ensured.

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

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

- $\gamma_{M,fi}$ – input the partial factor for the relevant material property, for the fire situation.
- **Calculation model** – select the type of calculation model of structural fire design. The design can be performed using either **Strength domain** or **Temperature domain**.
- **Method for critical temperature calculation** – select the method for calculation of critical temperature.
- **Required time of fire resistance** – input the time for which the structure should resist the fire.
- **Time interval of analysis – unprotected members** – input the time interval for calculation of temperature increment on unprotected members.
- **Time interval of analysis – protected members** – input the time interval for calculation of temperature increment on protected members.
- **Fire exposure** – select the mode of cross-section fire exposure. Following modes are available:
 - **All sides** – the cross-section is exposed to the fire on all sides.
 - **Three sides** – the cross-section is exposed to the fire on three sides (one side is protected).
- **Type of protection** – select the fire-protection type of the cross-section:
 - **None** – the member is not protected against the fire.
 - **Board** – the member is protected by fire protection boards.
 - **Spray** – the member is protected by fire protection spray.
- **Temperature curve** – select the temperature curve to calculate the temperature in time. Following curves are available:
 - **Standard curve**
 - **External fire curve**
 - **Hydrocarbon curve**
- **Net heat flux – configuration factor** – input the configuration factor to determine the net heat flux.
- **Net heat flux – surface emissivity of member** – input the member surface emissivity ϵ_m to determine the net heat flux.
- **Net heat flux – emissivity of flame** – input the emissivity of flame ϵ_f to determine the net heat flux.
- **Fire protection material – temperature independent specific heat** – input the specific heat of applied fire protection material.
- **Fire protection material – thickness** – input the thickness of applied fire protection material.

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

Fire protection material – unit mass – input the unit mass of applied fire protection material.

Group **General**:

- **Sway YY** – enable/disable sway buckling mode for buckling about y axis considered in stability check.
- **Sway ZZ** – enable/disable sway buckling mode for buckling about z axis considered in stability check.
- **Maximal buckling length coefficient** – input the maximal value of buckling coefficient for coefficients determined from calculation.
- **LT buckling system is equal as ZZ and YZ buckling system** – this option is used when creating a new design group. If the option is selected, the common buckling system is used for LTB, ZZ and YZ buckling.

9.2 Check settings for the current design group

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

Check settings	
Use check settings from project setup	<input checked="" type="checkbox"/>
Deflection check	<input checked="" type="checkbox"/>
Stability check	<input checked="" type="checkbox"/>
Fire resistance check	<input checked="" type="checkbox"/>
Classes 1 and 2 checked as class 3	<input type="checkbox"/>
Class 4 checked as class 3	<input type="checkbox"/>
Check of shear buckling	<input checked="" type="checkbox"/>
Buckling settings	
Use buckling settings from project setup	<input checked="" type="checkbox"/>
Sway YY	<input type="checkbox"/>
Sway ZZ	<input type="checkbox"/>
If possible, determine LTB curve acc. to eq. (6.57)	<input checked="" type="checkbox"/>
Effect of load position in the cross-section on LTB behavior of member	destabilizing
Use art. 6.3.3 also for unsymmetrical cross-section when limit for $MzEd/MzRd$ is exceeded	<input type="checkbox"/>
Do not take into account bending around minor axis in stability check of unsymmetrical cross-section	<input type="checkbox"/>
Member type for deflection check	Floor structure - Primary
EN1993-1-2 Structural fire design	
Use fire design settings from project setup	<input checked="" type="checkbox"/>
Calculation model	Strength domain
Required time of fire resistance ($t_{fi,req}$) [s]	900
Fire exposure	All sides
Type of protection	None
Temperature curve	Standard curve
Heat transfer by convection (α_c) [W/(m ² .K)]	25,0
Net heat flux - Configuration factor (Φ)	1
Net heat flux - Surface emissivity of member (ϵ_m)	0,7

Group **Check settings**:

- **Use check settings from project setup** – if this option is selected, the cross-section resistance check settings are taken from the default project check settings – see **9.1 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 **9.1 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 **9.1 Default check settings**. If the option is not selected, specific fire design check settings can be set for the current design group.

9.3 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 screenshot shows a 3D model of a beam with three members labeled 1, 2, and 3, all of type Iwn400x(250/160). Member 1 is 1.50m long, member 2 is 6.00m long, and member 3 is 1.50m long. The total length is 9.00m. Below the model is a 'Data' window with two tables:

Members of current design member			
ID	Member	Beginning [m]	Length [m]
1	1	0,00	1,50
2	2	1,50	6,00
3	3	7,50	1,50

Restrains, check data				
Point LTB Restraint		Distributed LTB Restraint		Check data
Type	Side	Beg. position	End. position	
> Distributed	Top	0,00	1,50	✖

On the right side of the 'Restrains, check data' window, there is a detailed view for the selected 'Distributed LTB Restraint':

LTB Restraint	
Type	Distributed
Location	Top
Position [m]	0,00
End position [m]	1,50

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

The current design member can be selected in the combo box **Design member** above the tables.

The list of members of the current design member is displayed in the table **Members of current design member**.

For the selected member, the defined design data are displayed in the table **Restrains, check data**.

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

This is a close-up of the 'Restrains, check data' window. It shows the 'Distributed LTB Restraint' table and its detailed properties:

Restrains, check data				
Point LTB Restraint		Distributed LTB Restraint		Check data
Type	Side	Beg. position	End. position	
> Distributed	Top	0,00	1,50	✖

LTB Restraint	
Type	Distributed
Location	Top
Position [m]	0,00
End position [m]	1,50

9.3.1 Ribbon group Project setup

See **9.1 Default check settings**.

9.3.2 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**.

LTB Restraint	
Type	Point
Location	Top
Position [m]	0,00
Repeated	<input checked="" type="checkbox"/>
Repeated	
Count	5
Regularly	<input checked="" type="checkbox"/>
Spacing [m]	0,38

Point restraint properties:

- **Location** – select the location of restraint. Restraint can be defined either on the top flange or on the bottom flange or on both flanges of the cross-section.
- **Position** – input the distance of point restraint related to the beginning of the member.
- **Repeated** – if the option is selected, the restraint contains more points.
- **Count** – input the count of the points in the repeated restraint.
- **Regularly** – if the option is selected, the points of repeated restraint are distributed regularly between the **Position** and the end of the current member.
- **Spacing** – input the spacing between points of the irregular repeated restraint.

Click  in the appropriate row of point restraints table to delete the point restraint.

9.3.3 Distributed LTB restraint

LTB Restraint	
Type	Distributed
Location	Top
Position [m]	0,00
End position [m]	1,50

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.

9.3.4 Not checked field

Point LTB Restraint		Distributed LTB Restraint		Check data
Member				
Member length [m]	1,500			
Not checked field				
From beginning [m]	0,500			
From end [m]	0,500			

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**.

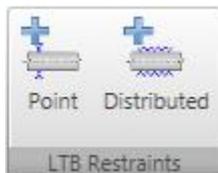
Properties of not checked field:

- **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.

9.3.5 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.

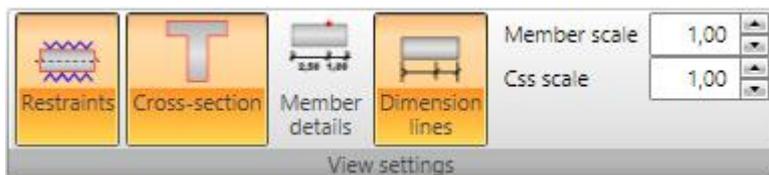
9.3.6 Ribbon group Check data



Commands in ribbon group:

- **New** – add new not checked field on the current design member.
- **Delete** – delete the current not checked field.

9.3.7 Ribbon group View settings



Use commands in ribbon group **View settings** to set the view options of the design member in uncoiled view:

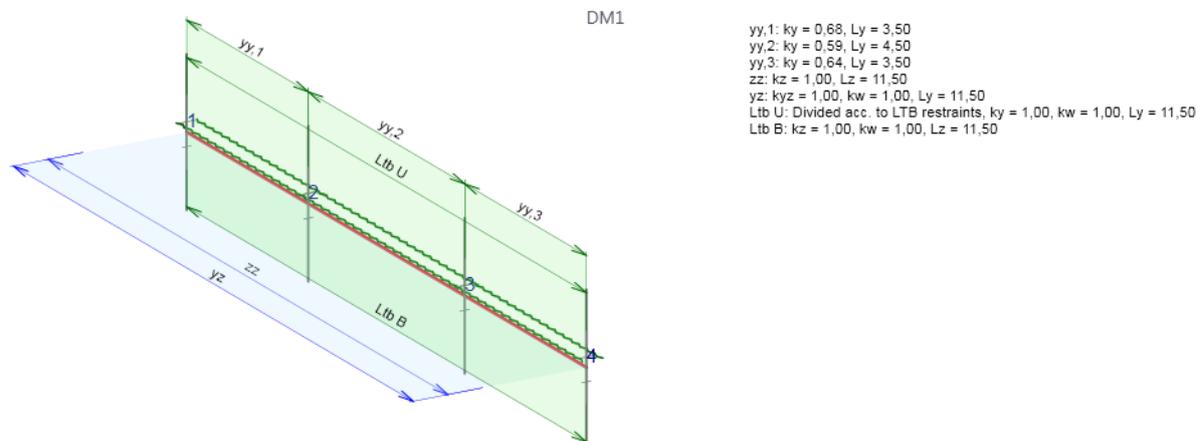
- **Restraints** – switch on/off the drawing of LTB restraints.
- **Cross-section** – switch on/off the drawing of the cross-section picture above the current design member.
- **Member details** – switch on/off the detailed drawing of the current member of the design member.
- **Dimension lines** – switch 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.

9.4 Buckling lengths

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

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

Buckling lengths parameters are defined for the current design member.



Design member

Buckling coefficients definition

Node	yy	ky	Length	Ltb U	kz	kw	Mcr	Ltb B	kz	kw	Mcr	Def y	Def z	Node
1	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1
2	<input checked="" type="checkbox"/>	0,68	3,50	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	<input type="checkbox"/>	2
3	<input checked="" type="checkbox"/>	0,59	4,50	<input type="checkbox"/>				<input type="checkbox"/>				<input type="checkbox"/>	<input type="checkbox"/>	3
4	<input checked="" type="checkbox"/>	0,64	3,50	<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	4

User Input - Factor User Input - Factor User Input - Factor

Mcr - Calculated Mcr - Calculated

Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB.

The buckling system is limited by the length of the Design member. If you need to extend it by more members they must be added to the Design Member.

Unit of length is [m] and unit of Mcr is [kN.m].

The current design member is drawn in the main window. The picture of the design member contains drawing of defined system lengths for individual buckling modes. Values of buckling coefficients are drawn too.

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

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

Following coefficients can be assigned to system lengths:

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

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

Design member ◀ ▶

Buckling coefficients definition

Node	yy	ky	Length	Ltb U	kz	kw	Mcr	Ltb B	kz	kw	Mcr	Def y	Def z	Node
1	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1
2	<input checked="" type="checkbox"/>	0,68	3,50	<input type="checkbox"/>				<input type="checkbox"/>				<input type="checkbox"/>	<input checked="" type="checkbox"/>	2
3	<input checked="" type="checkbox"/>	0,57	4,50	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	1,00	1,00	0,00	<input type="checkbox"/>	<input checked="" type="checkbox"/>	3
4	<input checked="" type="checkbox"/>	0,72	3,50	<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>				<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	4

Buckling lengths around ZZ and YZ are determined by buckling lengths for LTB.

The buckling system is limited by the length of the Design member. If you need to extend it by more members they must be added to the Design Member. Unit of length is [m] and unit of Mcr is [kN.m].

The current design member 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 Nexus.
- 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..

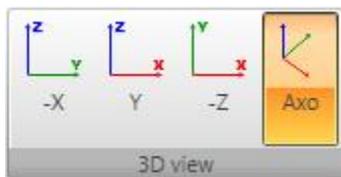
Use the checkboxes in columns **Defy**, **Defz** to set the system length for calculation of deflections in the similar way as for buckling assessment.

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 k_z and k_w specified for the check of LTB.

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

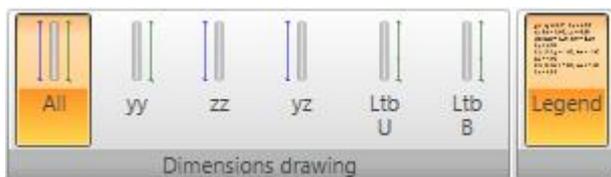
9.4.1 Ribbon group 3D view



Commands in ribbon group **3D view**:

- **-X** – switch the view direction against the global X axis direction.
- **Y** – switch the view direction to the global Y axis direction.
- **-Z** – switch the view direction against the global Z axis direction.
- **Axo** – switch to the axonometric view.

9.4.2 Ribbon group Dimensions drawing



Use commands in this group to set the drawing of dimension lines of system lengths:

- **All** – switch to draw dimension lines of system lengths for all types of buckling.
- **yy** – switch to draw dimension lines of system length of buckling yy .
- **zz** – switch to draw dimension lines of system length of buckling zz .
- **yz** – switch to draw dimension lines of system length of torsional buckling.
- **Ltb, U** – switch to draw dimension lines of system length of lateral torsional buckling at the upper flange.
- **Ltb, B** – switch to draw dimension lines of system length of lateral torsional buckling at the bottom flange.
- **Legend** – switch on/off the drawing of descriptions of system and buckling lengths.

9.5 Check results evaluation

Click navigator command **Steel design > Results** to perform the check and to evaluate the check results.

The graphical course of check results along the current design member is drawn in the Main window.

Tables with text presentation of the check results are displayed in the Data window.

Table with check results overview is displayed in the Details view.

Ribbon groups **Steel design**, **Extremes**, **Type of check** and **Type of output** are available when evaluating check results.

9.5.1 Ribbon group Steel design



Use commands in ribbon group **Steel design** to set or edit the result class for steel check or to set the national code coefficients for all design groups.

- **Code** - see **9.1 Default check settings**.
- **List of ULS result classes** - select the result class for section check and buckling check. Click edit button  to modify the content of result class.
- **List of SLS result classes** - select the result class for deflection check. Click edit button  to modify the content of result class.
- **List of ULS result classes** - select the result class for fire resistance check. Click edit button  to modify the content of result class.

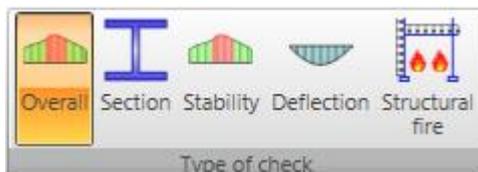
9.5.2 Ribbon group Extremes



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

- **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.
- **Global** – extreme values of individual checks are searched from all design members. One result is printed for each type of check.

9.5.3 Ribbon group Type of check

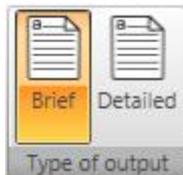


Use commands in ribbon group **Type of check** to set the check type to be evaluated:

- **Overall** – switch to evaluation of overall results of all checks. Courses are drawn and tables are printed for the main checks – section resistance check, buckling resistance check and deflection check (if the appropriate checks are set to be performed).
- **Section** – switch to evaluation of results of section resistance check. The course of resistance check is drawn and the tables with results of all partial section resistance checks are printed.

- **Stability** – switch to evaluation of results of buckling resistance check. The course of buckling resistance check is drawn and the tables with results of all partial buckling resistance checks are printed.
- **Deflections** – switch to evaluation of results of the deflection check. The course of deflection check is drawn and the tables with results of deflection check are printed.
- **Structural fire** - switch to evaluation of results of the structural fire design. The course of structural fire design is drawn and the tables with results of structural fire design are printed.

9.5.4 Ribbon group Type of output



Use commands in ribbon group **Type of output** to set the range of printed results:

- **Brief** – switch to print table with brief summary results only.
- **Detailed** – switch to print tables with detailed outputs.

10 Calculation report



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

Use commands in navigator group **Report** to generate reports for the whole structure.

Use commands in ribbon group **Report** view to print and export the report

10.1 Brief report

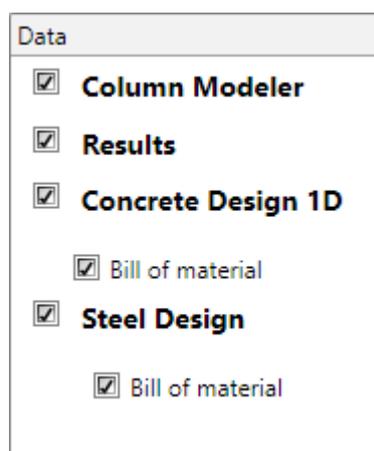
Click navigator command **Report > Brief** to generate the brief report.

The content of brief report is fixed and cannot be changed.

10.2 Standard report

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

The content of standard report can be defined in the Data window:



- **Column modeler** – switch on/off print of the tables of column input data
- **Results** – switch on/off print of the tables and the pictures of internal forces, deformations and reactions
- **Concrete design 1D** – switch on/off print of the tables and the pictures of result of concrete column check.
 - **Bill of material** – switch on/off print of the tables of bill of material of concrete column.
- **Steel design** – switch on/off print of the tables and the pictures of result of steel column check.
 - **Bill of material** – switch on/off print of the tables of bill of material of steel column.

10.3 Detailed report

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

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

10.3.1 Input data

- Column Modeler** Options to set the content of input data report:
 - Project data
 - Column data
 - Materials
 - Cross-sections
 - Load cases
 - Input loads
 - Pictures of loads
 - Combinations
 - **Column modeler** – switch on/off print of the tables of column input data.
 - **Project data** – switch on/off print of the table of project data.
 - **Column data** – switch on/off print of the table and the picture of column geometry.
 - **Material** – switch on/off print of the table of used materials.
 - **Cross-sections** – switch on/off print of the table and the pictures of column cross-sections.
 - **Load cases** – switch on/off print of the table of load cases.
 - **Input loads** – switch on/off print of the tables of defined loads in individual load cases.
 - – **Pictures of loads** - switch on/off print of the pictures of defined loads in individual load cases.
 - **Combinations** – switch on/off print of the table of defined combinations.

10.3.2 Calculation results

- Results** Options to set the content of calculation results report:
 - Internal forces
 - Pictures
 - Deformations
 - Reactions
 - **Results** – switch on/off print of the tables and the pictures of results of static analysis.
 - **Internal forces** – switch on/off print of the tables of internal forces caused by combinations.
 - Pictures – switch on/off print of the pictures of internal forces caused by combinations.
 - **Deformations** – switch on/off print of the table of deformations caused by combinations.
 - **Reactions** – switch on/off print of table of reactions in supports caused by combinations.

10.3.3 Concrete 1D design results

Concrete Design 1D

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

- Section check results
 - Overall check picture
 - Extreme zone
 - All zones
 - Interaction diagrams
 - Second order results
 - Explanation
 - Nonconformity
 - Detailed results table
 - Combination
- Check of deflection
 - Extreme combination
 - All combinations
 - Picture
 - Stiffness
 - Nonconformity
 - Combination
 - Explanation
- Bill of material
- Design member data
- Reinforcement zones
- Code and calculation settings

- **Concrete design 1D** – switch on/off print of the table with overall check result and the picture of reinforcement scheme.
 - **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 print of the 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.

- **Combinations** – switch on/off the print of the table with detailed results of section check.
- **Combinations** – switch on/off the print of the table of critical combinations.
- **Check of deflection** – switch on/off the print of all results of deflection calculations.
 - **Extreme combinations** – if the option is selected, results of deflection check will be printed for the combination, which caused the extreme check value.
 - **All combinations** – if the option is selected, results of deflection check will be printed for all combinations.
 - **Picture** – switch on/off the print of the picture with the courses of deflections.
 - **Stiffness** – switch on/off the print of the tables with stiffnesses.
 - **Nonconformity** - switch on/off the print of the table with nonconformities.
 - **Combination** – switch on/off the print of the table with description of combinations for calculation of deflections.
 - **Explanation** - switch on/off the print of the table with nonconformities.
- **Bill of material** – switch on/off print of the table with bill of reinforcement and concrete.

- **Design member data** – switch on/off print of the tables with calculation settings of individual design members.
- **Reinforced zones**– switch on/off print of the tables and the pictures of reinforced zones and the reinforcement along the design member.
- **Code and calculation setup** – switch on/off the print of table with national code and calculation coefficients values.

10.3.4 Steel design results

Steel Design

- Cross-section picture
- Internal forces picture
- Strength check picture
- Stability check picture
- Buckling length picture
- Deflection check picture
- Fire resistance check picture
- Print detailed tables
- Bill of material

Options to set the content of steel check results detailed report

- **Steel design** - switch on/off print of tables of steel check results.
 - **Cross-section picture** – switch on/off the print of table with cross-sections picture and cross-section characteristics.
 - **Internal forces picture** - switch on/off the print of pictures with course of internal forces.
 - **Strength check picture** – switch on/off the print of pictures with course of section resistance check.
- **Stability check picture** – switch on/off the print of pictures with course of buckling resistance check.
- **Buckling lengths picture** – switch on/off the print of pictures of buckling lengths.
- **Deflection check picture** – switch on/off the print of pictures with course of deflection check.
- **Fire resistance check picture** – switch on/off the print of pictures with course of fire resistance check.
- **Print detailed tables** – if the option is selected, the detailed tables of check with intermediate results are printed. If the option is not selected, only the resulting utilisations for particular checks are printed.
- **Global** – the extreme steel check values are searched from all appropriate design groups.
- **Design group** – the extreme steel check values are searched for each appropriate design group.
- **Design member** – the extreme steel check values are searched for each appropriate design member.
- **Bill of material** – switch on/off print of table of bill of material of steel members.

10.4 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.