

IDEA CSS 8

General cross-section

User guide

Content

1.1 Program requirements	3
1.2 Installation guidelines	3
2 Basic Terms.....	4
3 User interface	5
3.1 Control of view in the Main window	5
3.1.1 DXF export settings	6
3.2 Units setting.....	7
4 Working with project	8
4.1 New project	8
5 Input of cross-sections and analysis of cross-section characteristics	9
5.1 Project data.....	9
5.2 Input of cross-sections.....	10
5.2.1 Components of general cross-section.....	11
5.2.2 Input of shape of general cross-section	11
5.2.3 Ribbon group Cross-section component	12
5.2.4 Editing cross-section component	20
5.2.5 Ribbon group Drawing settings.....	21
5.2.6 Ribbon group Colours	21
5.2.7 Ribbon group Collisions.....	21
5.2.8 Ribbon group Mesh.....	21
5.2.9 Points for stress analysis	22
5.2.10 Cross-section characteristics	24
6 Check of cross-sections	26
6.1 Design matrix	26
6.2 Check requests.....	28
6.3 Check results	29
6.3.1 Ribbon group Results	29
6.3.2 Ribbon group Drawing settings.....	30
7 Report	31
7.1 Brief report	31
7.2 Detailed report.....	32
7.3 Ribbon group Report view	33

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

2 Basic Terms

IDEA CSS is a program for input of general cross-section, analysis of cross-section characteristics and analysis of stresses caused by defined internal forces. IDEA CSS 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 cross-sections of general shape, which consist of one or more components. Cross-section characteristics and courses of stresses caused by unit loads are calculated for defined cross-sections.

It is possible to input design matrixes. Each design matrix can contain one or more sextuplets of internal forces.

Design matrixes and cross-sections can be combined into check requirements. Stresses are calculated for defined check requirements.

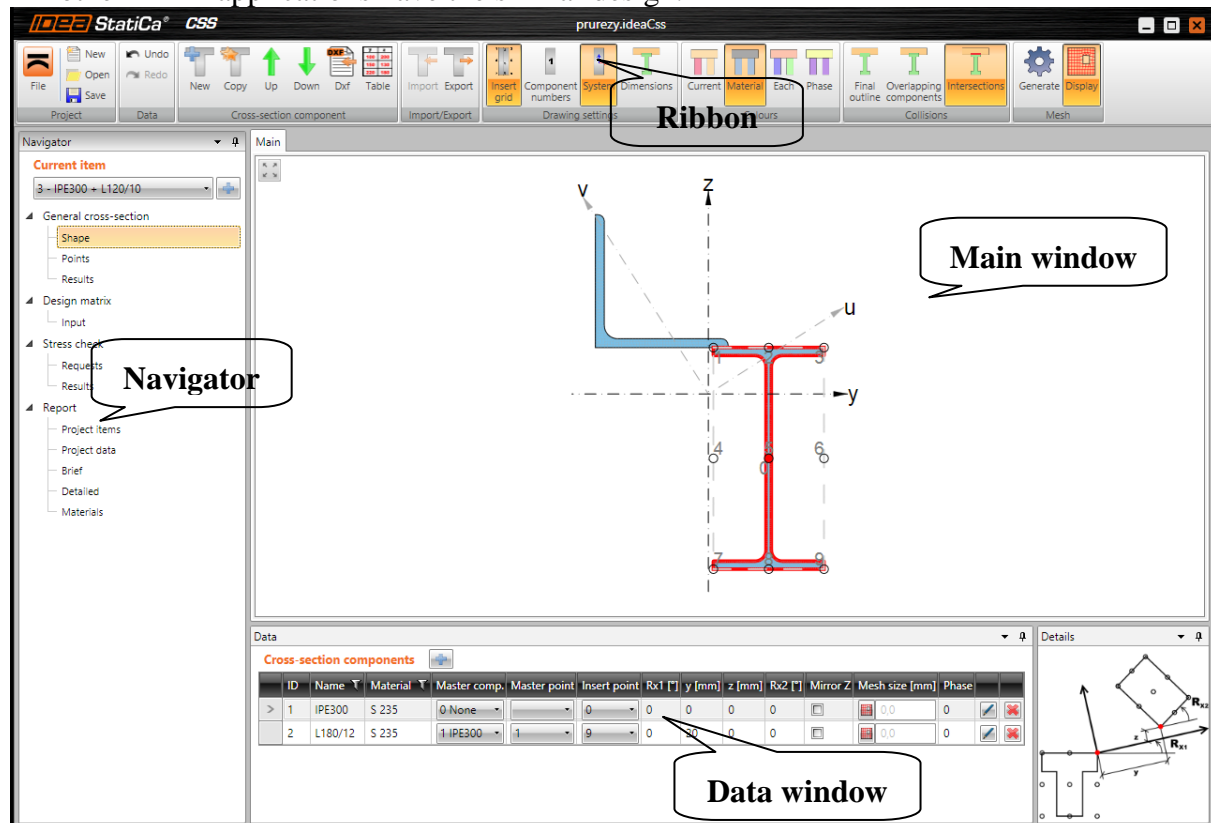
The program general cross-section can be used as plugin to define general cross-sections in modules IDEA Beam, IDEA Frame and .

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:

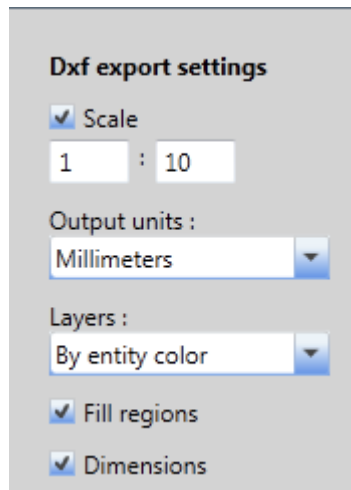
- Click and hold mid mouse button – moving the mouse pans the view.
- Roll with mid mouse button – moving the mouse increases/decreases the view.
- Push CTRL+SHIFT and hold mid mouse button – moving the mouse defines the window for zoom.

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



Dxf export settings

☒ Scale
1 : 10

Output units :
Millimeters

Layers :
By entity color

☒ Fill regions

☒ Dimensions

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

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

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

Default - metric Default - imperial Import Export OK Cancel

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.

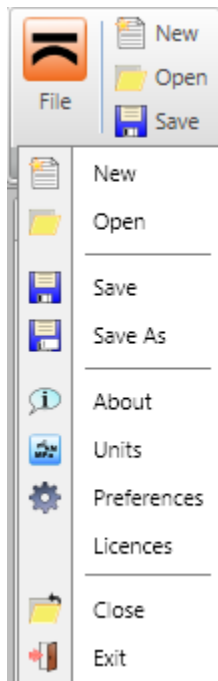
Import - reads the units configuration from a file.

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

Commands to work with project data are collected in ribbon group **Data**:

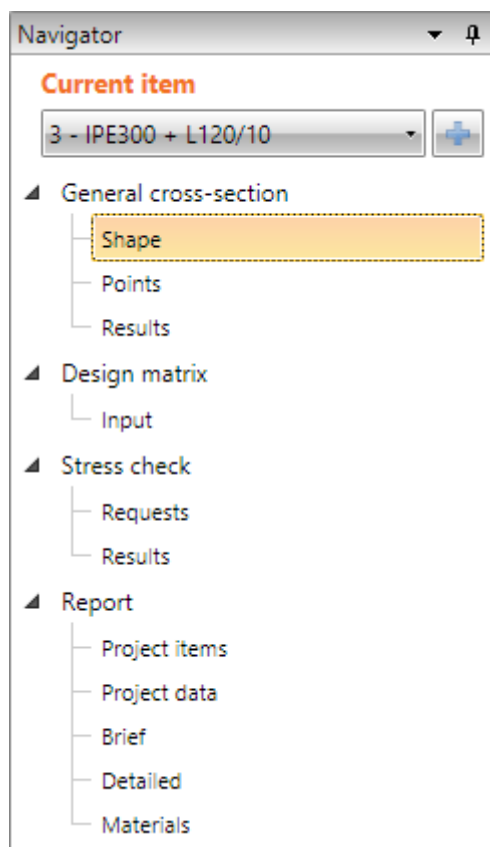


- **New** – create a new project.
- **Open** – open an existing project (files with extension *.ideaCSS or *.wsCSS).
- **Save** – save the current project into the data file.
- **Save as** – save the current project into the data file using a new file name.
- **About** – open the About application dialog.
- **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 a current project.

4.1 New project

To create a new project click **New** in ribbon group **Project**. First general cross-section is created automatically in new project.

5 Input of cross-sections and analysis of cross-section characteristics



Partial general cross-section data are defined using appropriate navigator commands. Cross-section shape has to be defined to calculate cross-section characteristics. Design matrix and Check requirements have to be defined to calculate the stress in cross-section.

5.1 Project data

To display a table with basic data and identification data of the current project click navigator command **Report > Project data**.

Project parameters	
Code	EN
Material type	Mixed
Identification	
Name	
Number	
Author	
Description	
Date	28.1.2013

- **Code** – select the national code. The selected code affects the available materials.
- **Material type** – select the project type from the point of view of the materials. Either steel, or concrete or mixed cross-sections can be defined in the project.
- **Name** – input of the project name.
- **Number** – input of the project identification number.
- **Author** – input name of the project author name.
- **Description** – input of additional information about the project.
- **Date** – date of calculation.

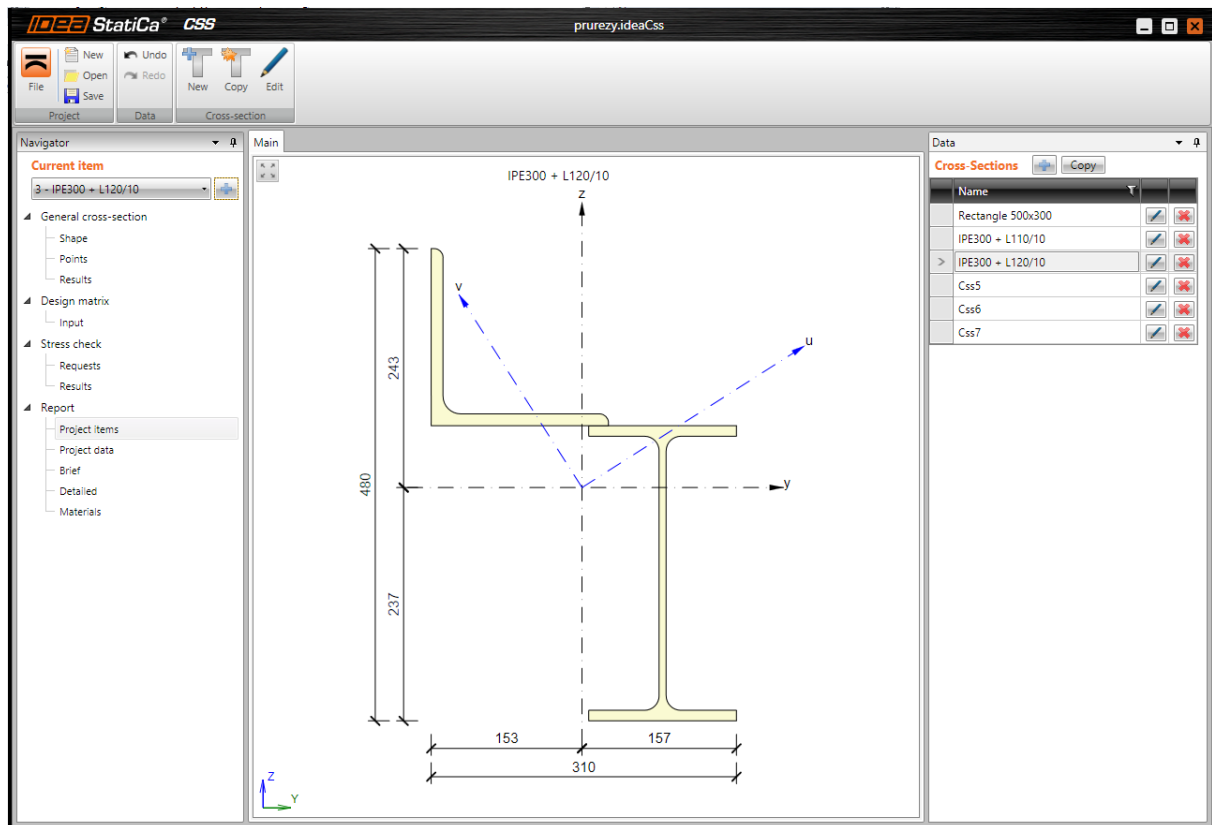
5.2 Input of cross-sections



One project of IDEA CSS application can contain more general cross-sections. To start input of cross-sections click navigator command **Report > Project items**.


The current cross-section is drawn in the Main window.

Table with the list of existing general cross-sections is displayed in the Data window.


Ribbon group **Cross-section** is available for this navigator command.



To add a new cross-section click either  above the table or **New** in ribbon group **Cross-section** or click  near the list of cross-sections in project in the top of navigator.

To edit the current cross-section click navigator command General cross-section > Shape or click edit button  in the appropriate table row or **Edit** in ribbon group **Cross-section**.

To copy an existing cross-section click either **Copy** above the table or **Copy** in ribbon group **Cross-section**.

To delete an existing cross-section click either  in the appropriate table row.

5.2.1 Components of general cross-section

The general cross-section geometry contains components of cross-section.

Components of cross-section can be defined:

- Selecting a component shape from the database of concrete cross-sections;
- Selecting a component shape from the database of steel cross-sections;
- Importing the shape from DXF file;
- Defining the shape using a table editor.

Each cross-section component has a local coordinate system. The local coordinate system of the component is placed to the origin of global coordinate system by default.

If the cross-section component depends on some master component, the local coordinate system of slave component is placed to the Master point of the Master component.

Each cross-section component has an **Insert point**. The **Insert point** is coincident with the origin of local coordinate system of the component. When the Insert point is changed, the component moves to keep the new **Insert point** in the origin of local coordinate system.

The **Insert point** of first cross-section component is placed to the origin of global coordinate system of the cross-section.

The Insert grid is generated for each cross-section component. This grid contains points in important points of the cross-section. The **Insert point** can be set to any of point of the **Insert grid**.

Dependencies can be defined between components of cross-section. One component can be set as a master component for another component. The dependency is defined by two points. A **Master point**, which is one of the points of the insert grid of the master component, is set for the dependent component. Then the dependent component moves to keep coincident the **Insert point** of the dependent component with the **Master point** on the master component..

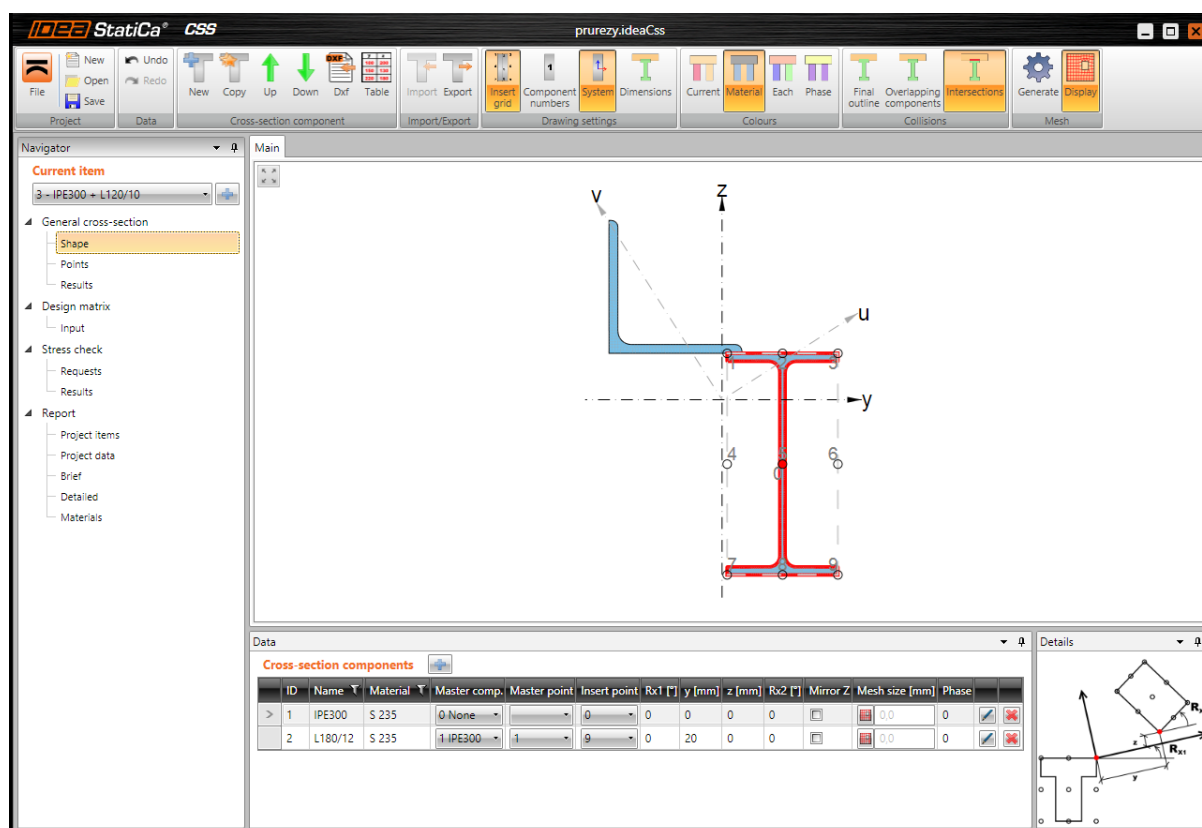
5.2.2 Input of shape of general cross-section

To start input of general cross-section geometry click navigator command **General cross-section > Shape**. The current section, which has been selected in the navigator table **Project > Cross-sections**, is edited.

Components of cross-section are drawn in the main window.

The table **Cross-section components** with properties of cross-section components is displayed in the **Data window**.

Ribbon groups **Cross-section component**, **Drawing settings**, **Colours**, **Collisions** and **Mesh** are available for this navigator command.



5.2.3 Ribbon group Cross-section component



Commands in ribbon group **Cross-section component**:

- **New** – create new cross-section component using shapes, which are available in the libraries of concrete and steel cross-sections – see **5.2.3.1 New cross-section component**.
- **Copy** – copy the current cross-section component.
- **Up** – move the current cross-section component up in the table of cross-section components.
- **Down** – move the current cross-section component down in the table of cross-section components. The order of components affects the method of collision resolving and the conversion of components to ideal cross-section, if the components are of different materials.
- **DXF** – create new cross-section component using import of shape from DXF file – see **5.2.3.2 New component from DXF**.
- **Table** – create new cross-section component of polygonal shape using a table – see **5.2.3.3 New component using table editor**.

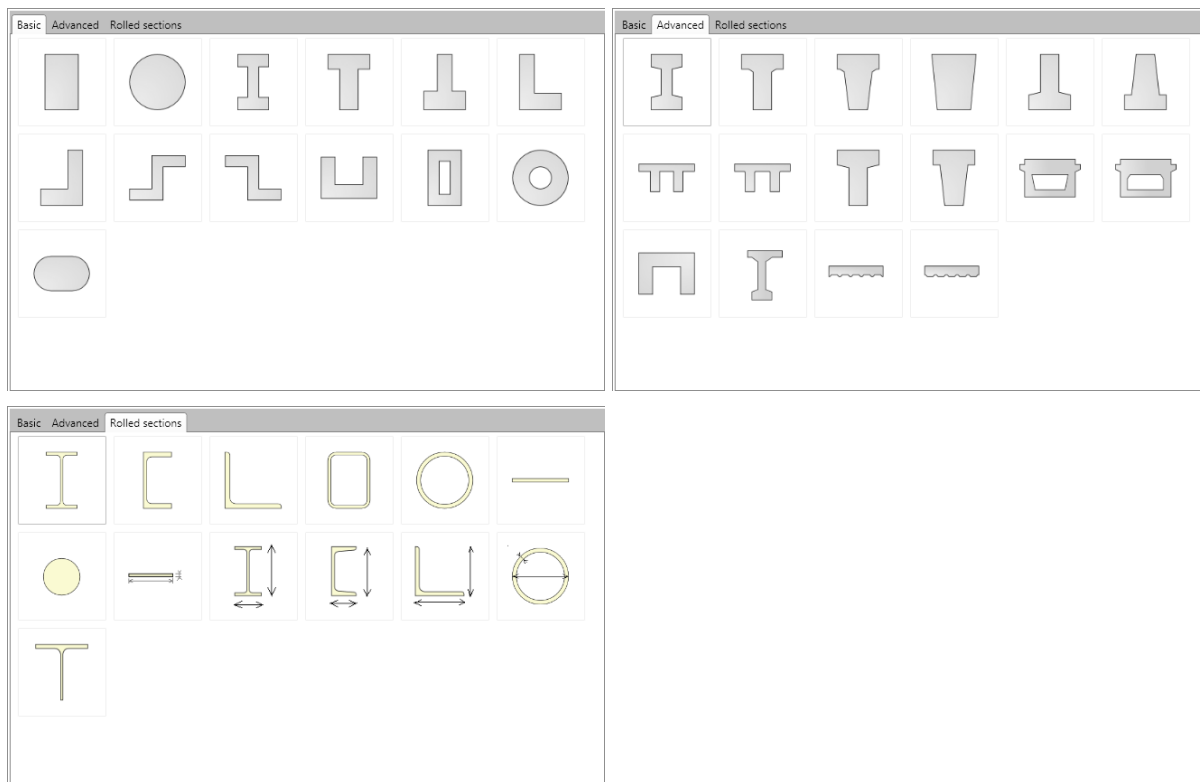
5.2.3.1 New cross-section component

To add new cross-section component click **New** in ribbon group **Cross-section component**.

The dialog **Cross-section navigator** appears, in which available cross-sections sorted into three groups are displayed:

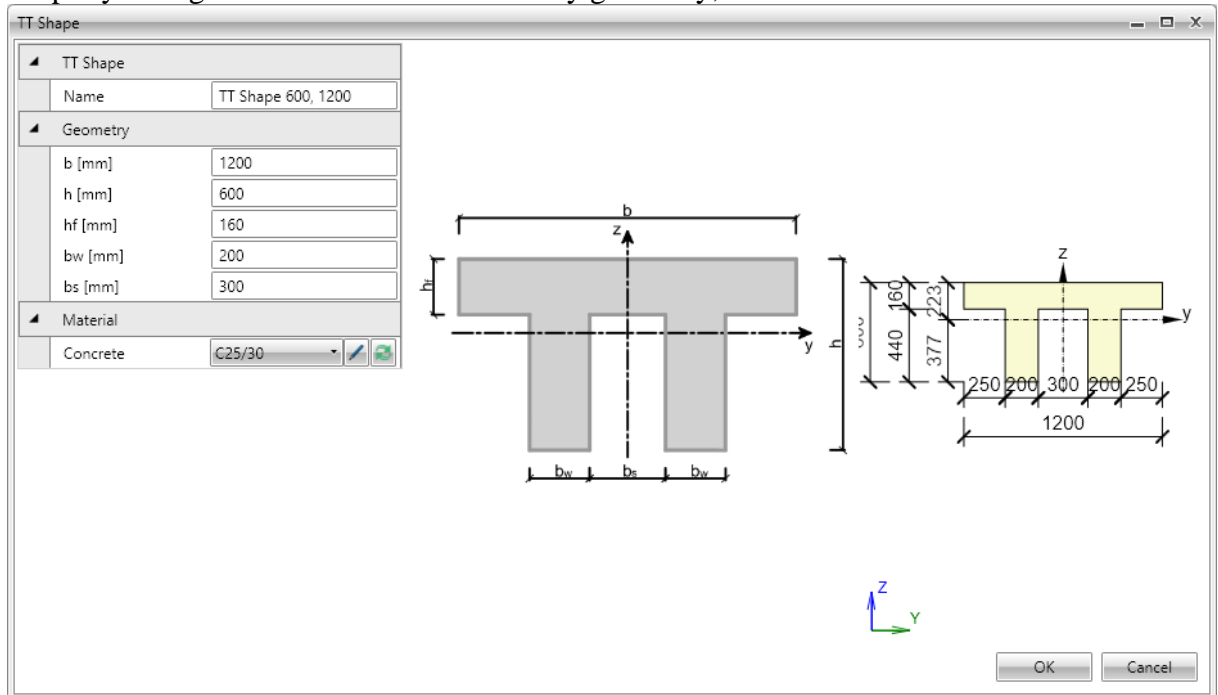
- **Basic** – standard shapes of concrete cross-sections.
- **Advanced** – shapes of concrete cross-sections used in bridges building.
- **Rolled sections** – steel rolled cross-sections I, L, U, T, plate, circular, square and rectangular hollow section.

Click the required shape of cross-section to input a new cross-section component.

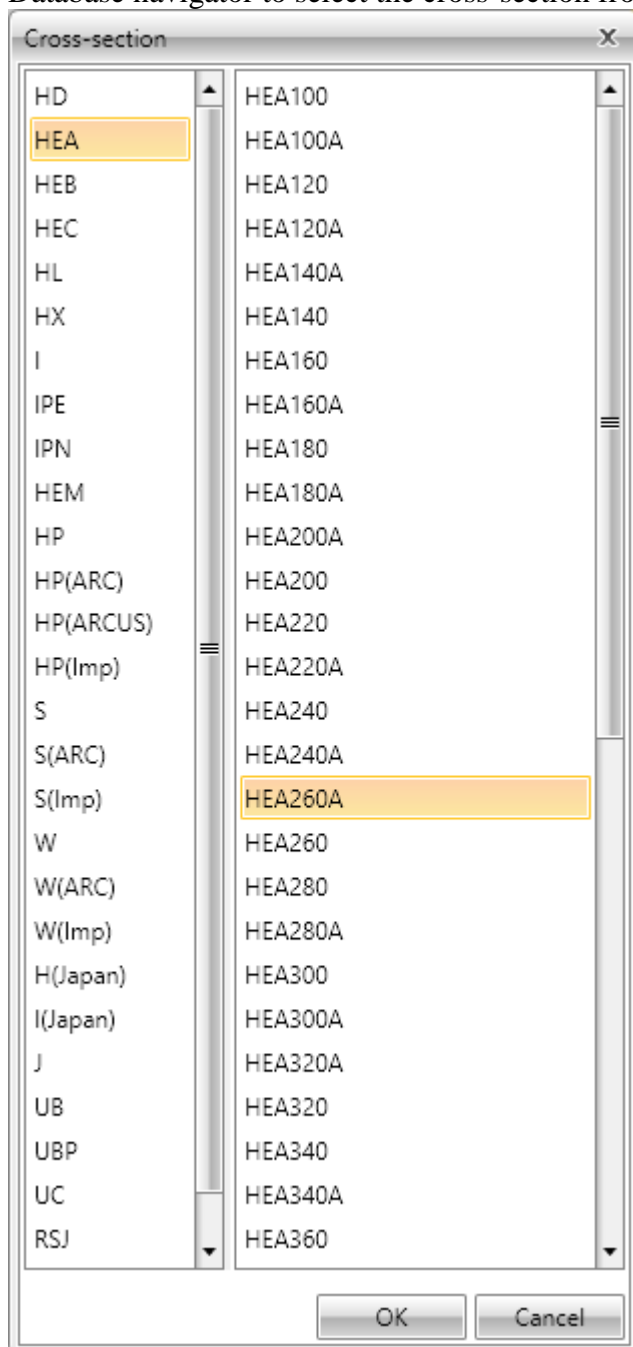


Depending on the required shape one of following dialogs appear:

- Property dialog for cross-sections defined by geometry;



- Database navigator to select the cross-section from the database.

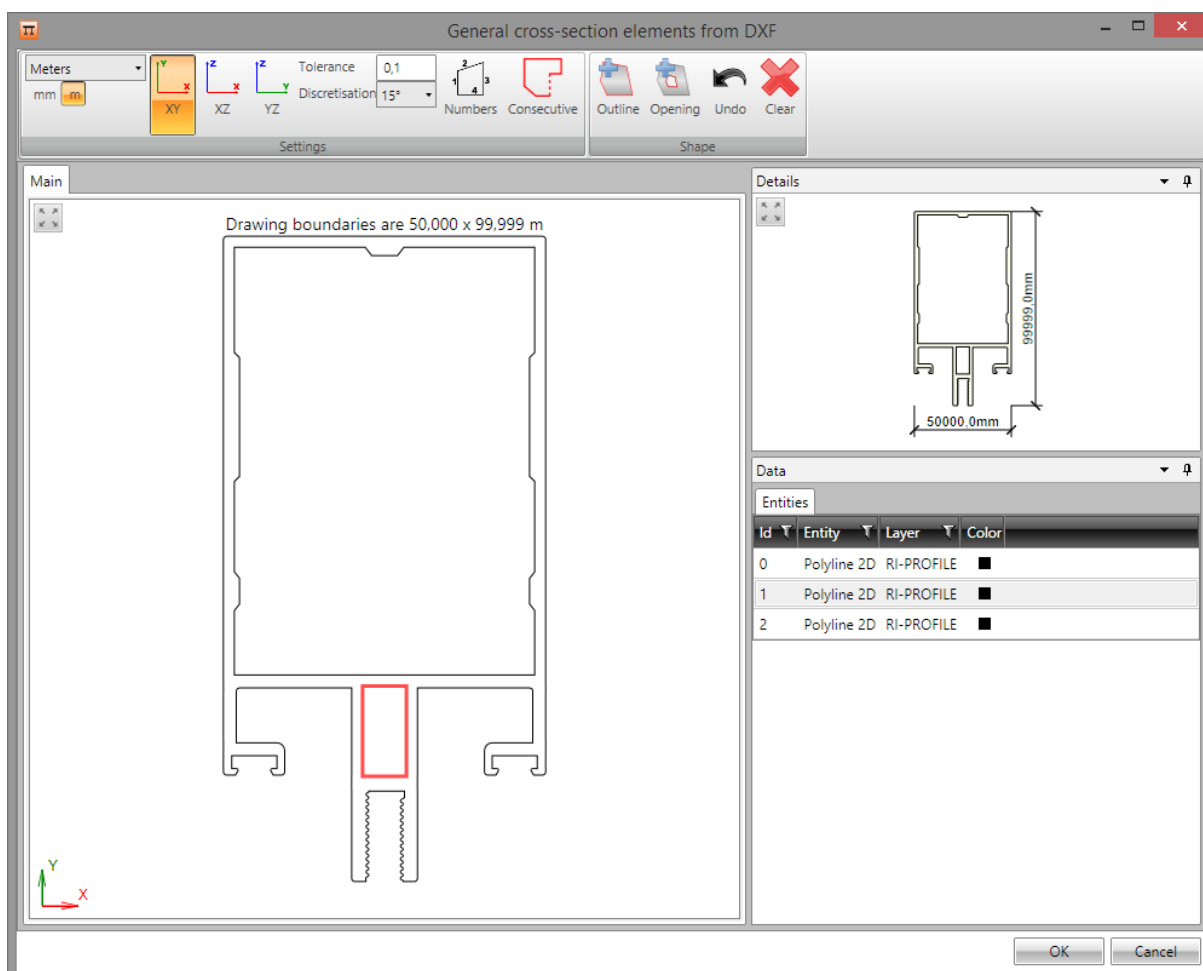


After entering the shape properties or choosing the cross-section from database the new component is added to the general cross-section shape.

5.2.3.2 New component from DXF file

To create new cross-section component by the import from DXF file click **DXF** in ribbon group **Cross-section component**.

Following entities can be imported from the DXF file: LINE, POLYLINE, SPLINE, ARC, CIRCLE, TEXT. Blocks are not imported, it is necessary to explode them to individual entities before importing.



The content of imported DXF file is displayed in the dialog **General cross-section elements from DXF**.

The proper units, which correspond with units of DXF file, must be set in ribbon group **Settings**. Otherwise the dimensions of cross-section can be wrong.

The lines, which create the shape of one part of the cross-section, should be selected in the main window. The selected group of lines should be continuous and should create a closed polygon.

Lines, which create tendon polygon, must be selected in the main window. Lines can be selected like standard irregular selections in Windows applications – hold CTRL and select single lines. To select lines, which are continuous to the selected line, click **Consecutive** in the ribbon group **Settings**.

The selected group of lines must be converted to:

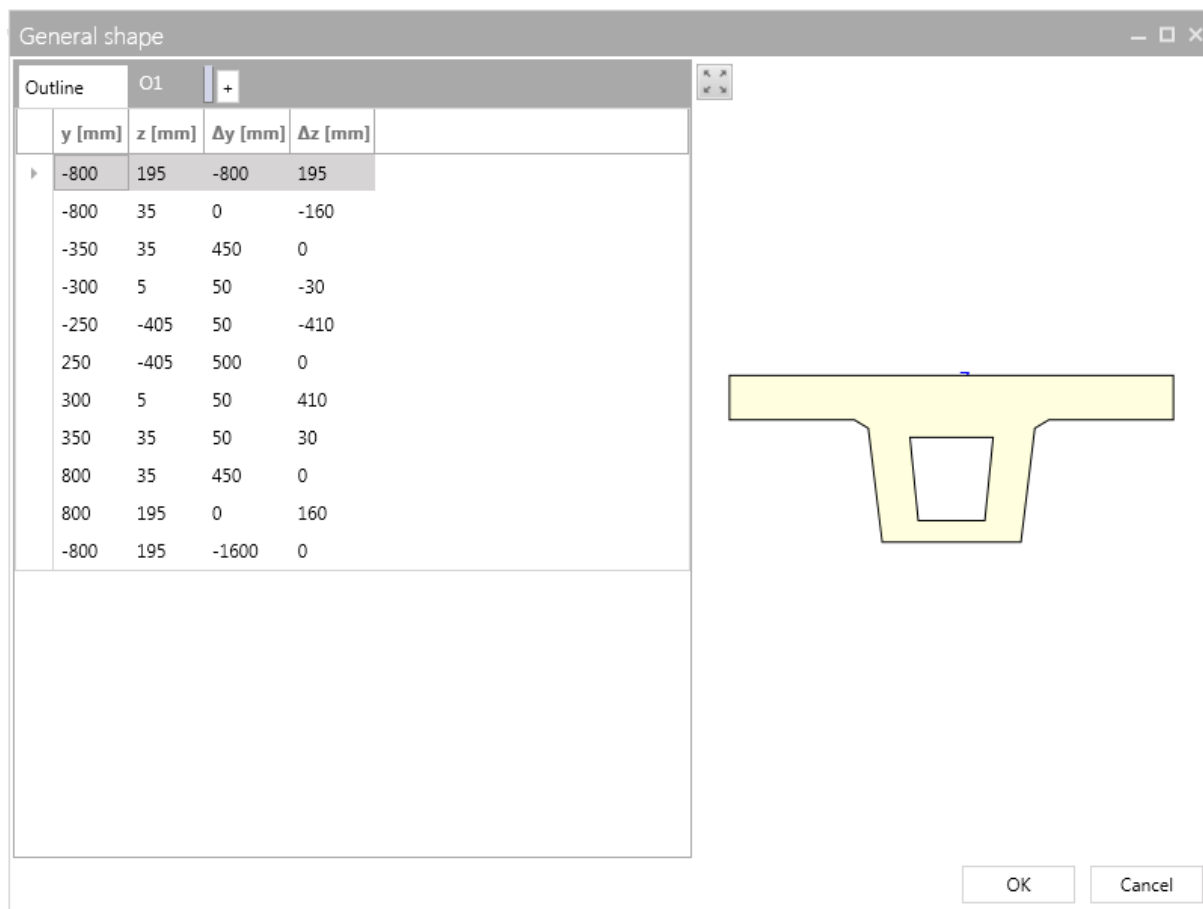
- The cross-section outline. To convert the lines click **Outline** in the ribbon group **Cross-section component**.

- The cross-section opening. To convert the lines click **Opening** in the ribbon group **Cross-section component**.

The converted outlines and openings are drawn in the **Detail window**. Click **Clear** in ribbon group **Cross-section component** to delete the converted parts of the cross-section.

5.2.3.3 New component using table editor

To create new cross-section component using the table editor click **Table** in ribbon group **Cross-section component**.



Dialog **General shape** contains one tab **Outline** to define the coordinates of vertexes of the cross-section outline and can contain one or more tabs **O1 - On** to define the coordinates of vertexes of individual openings.

To add a new opening click right the tabs.

To delete the opening click on the tab of appropriate opening.

The tables of coordinates can be copied using a clipboard from e.g. Microsoft Excel tables.

5.2.3.4 Copy the cross-section component


To copy the current cross-section component click **Copy** in the ribbon group **Cross-section components**.









5.2.3.5 Order of components

To change the order of cross-section components click **Up** or **Down** in the ribbon group **Cross-section components**. The order of components affects the method of collision resolving. If the cross-section contains components with different materials, the material of the first component on the table is used for calculation of the characteristics of ideal cross-section.


5.2.3.6 Cross-section component properties



The positions of components and relations between components can be set in the table **Cross-section components**.

Cross-section components 


ID	Name	Material	Master comp.	Master point	Insert point	Rx1 [°]	y [mm]	z [mm]	Rx2 [°]	Mirror Z	Mesh size [mm]	Phase		
> 1	IPE300	S 235	0 None		0	0	0	0	0		 0,0	0		
2	L180/12	S 235	1 IPE300	1	9	0	20	0	0		 0,0	0		

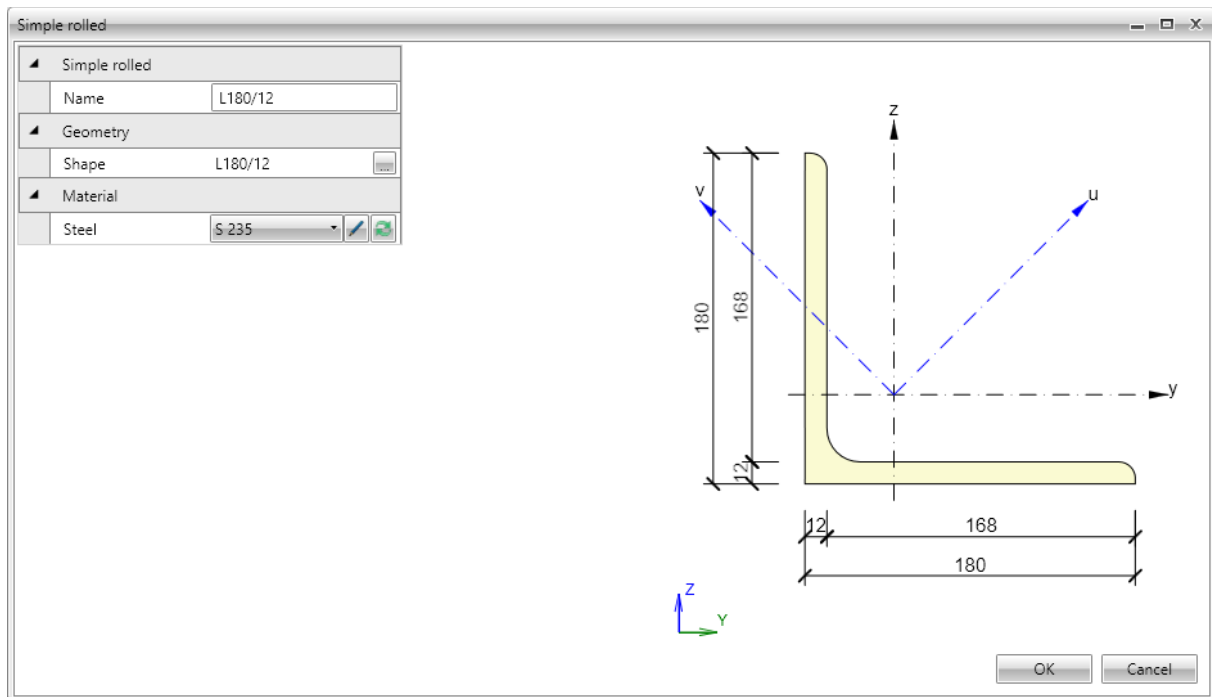
Columns in the table **Cross-section components**:

- **ID** – the identification number of the component.
- **Name** – input the name of the component.
- **Material** – the material of the component. Material can be changed when editing the component.
- **Master component** – select a master component from the list of existing components. The current component moves to keep its **Insert point** coincident to the **Master point** of the **Master component**. The local coordinate system of the dependent component moves to the **Master point**.
- **Master point** – select a point from the list of points on the Master component. The current component moves to keep its **Insert point** coincident to the **Master point**.
- **Insert point** – select a point from the list of points on the current component. The current component moves to keep the Insert point in the origin of local coordinate system of the component.
- **Rx1** – input the value of rotation of the local coordinate system of the component. The current component is rotated too.
- **y** – input the **Insert point** offset in direction of y-axis of local coordinate system of the component.
- **z** – input the **Insert point** offset in direction of z-axis of local coordinate system of the component.
- **Rx2** – input the rotation of component about the Insert point.
- **Mirror Z** – switch on/off the mirror of component about the centroidal reference z-axis of the current component.
- **Mesh size** – user defined size of mesh finite element. The mesh element size is calculated automatically by default. If necessary, click  to enable user defined value of mesh size for the current component.
- **Phase** – input number of construction stage, in which the cross-section component is built.

-  -start shape and material editing of the current component.
-  - delete the current cross-section component.

5.2.4 Editing cross-section component

To start editing of cross-section material and shape click edit button  in the appropriate row in the table **Cross-section component**.



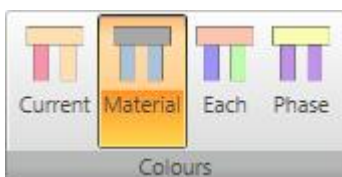
5.2.5 Ribbon group Drawing settings



Commands in ribbon group Drawing settings:

- **Insert grid** – switch on/off the drawing of the insert grid on the current cross-section component.
- **Component numbers** – switch on/off drawing of numbers of cross-section components.
- **System** – switch on/off drawing of reference coordinate system and principal coordinate system of the general cross-section.
- **Dimensions** – switch on/off drawing of the basic dimension lines of the general cross-section.

5.2.6 Ribbon group Colours



Use commands in ribbon group **Colours** to set the drawing mode of component colours:

- **Current** – switch to draw the current cross-section component in a different (red) colour.
- **Material** – switch to draw components of the same material in one colour.
- **Each** – switch to draw each component in different colour.

5.2.7 Ribbon group Collisions



Use commands in ribbon group Collisions to set the drawing mode of collisions:

- **Final outline** – switch to draw the final outline of individual components after cutting off the collision parts. The order of components in the **Cross-section components** table is taken into account during the calculation of the final outline. The final outline is used to generate the finite elements mesh and the cross-section characteristics.
- **Overlapping components** – switch to draw the original outline of cross-section components. The components are drawn overlapped and the order of components in the **Cross-section components** table is taken into account to calculate the overlaps.
- **Intersections** – switch to highlight the overlapping areas between the cross-section components. The order of components in the **Cross-section components** table is taken into account to calculate the intersections.

5.2.8 Ribbon group Mesh



Commands in ribbon group **Mesh**:

- **Generate** – run finite elements mesh generator on the final outline of the cross-section.
- **Display** – switch on/off drawing of the generated finite elements mesh.

5.2.9 Points for stress analysis

Axial and shear stress can be checked in points for analysis of stress. The point can be of two types:

- **Point** – individual point defined by coordinates. All stresses are calculated exactly in the point.
- **Section** – is defined by the point coordinates and by the slope of the line in this point. Following values are taken from the section when calculating the stresses:
 - The unit axial stress is taken in the section point.
 - The unit shear stress caused by torsion is taken as the maximal shear stress value along the whole section.
 - The unit shear stress caused by shear force is taken as the average shear stress value along the whole section.

For the sections from database the points for stress calculation are generated automatically in important points of the cross-section. Stress points have to be added for cross-sections of general shape.

The position of point for stress calculation is related to the defined cross-section component and the master point on this component.

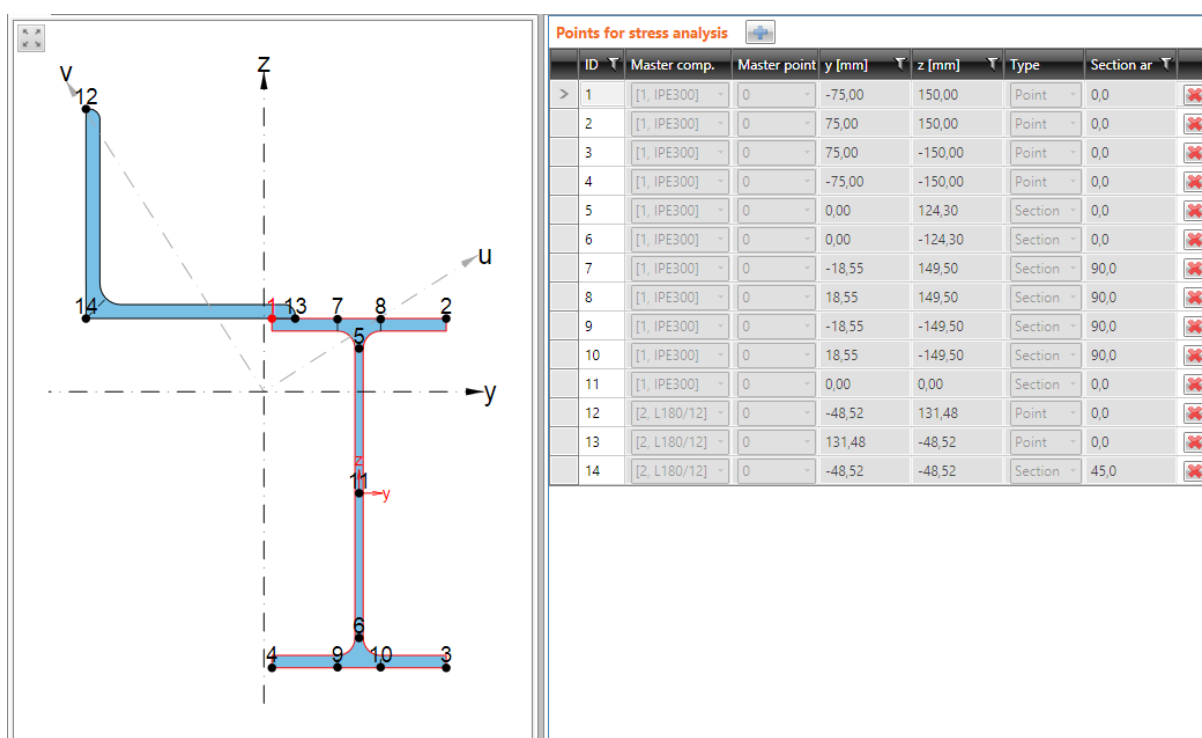
If there are no points for stress calculation on the cross-section, stress check cannot be performed on the cross-section.

To input the points for stress calculation on the cross-section click navigator command **General cross-section > Points**.

The general cross-section with defined points for stress calculation is drawn in the main window.

The table of defined points for stress calculation is drawn in the data window.

Ribbon groups **Stress point**, **Drawing settings** and **Mesh** are available for this navigator command.



To add a new point for stress analysis click **New** in ribbon group **Stress point**.

Columns in the table **Points for stress analysis**:

- **ID** – the number of the stress point.
- **Master comp.** – select the cross-section component, which the stress point is related to.
- **Master point** – select the point of Master component, which the point coordinates are related to.
- **y** – input the distance between stress point and Master point in direction of y-axis of the local coordinate system of the Master component.
- **z** – input the distance between stress point and Master point in direction of z-axis of the local coordinate system of the Master component.
- **Type** – choose the stress point type – either **Point** or **Section**.
- **Section angle** – for the stress point of **Section** type, input the angle of the section, in local coordinate system of the master component.

5.2.9.1 Ribbon group Stress point



Click **New** in ribbon group **Stress point** to add new point for stress analysis into the cross-section.

5.2.9.2 Ribbon group Drawing settings

See 5.2.5 Ribbon group Drawing settings.

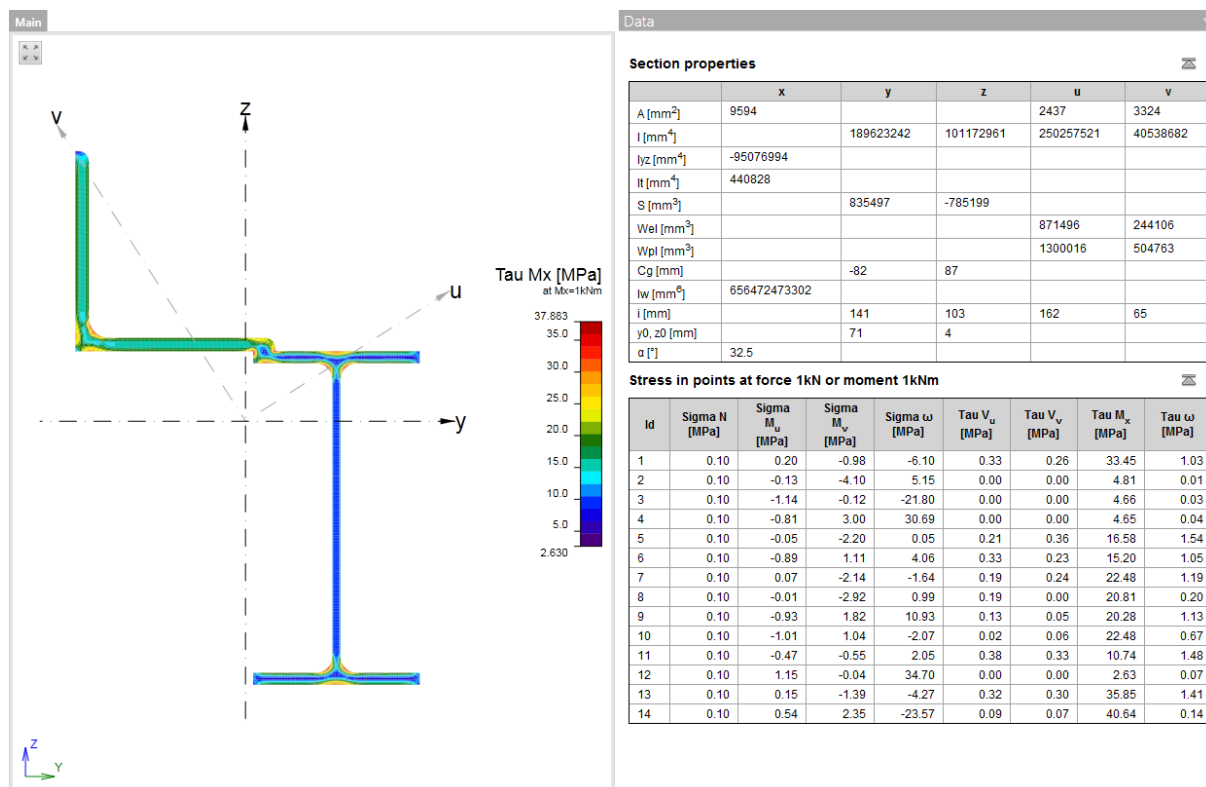
5.2.10 Cross-section characteristics

To evaluate the results of the cross-section characteristics analysis click navigator command **General cross-section > Results**.

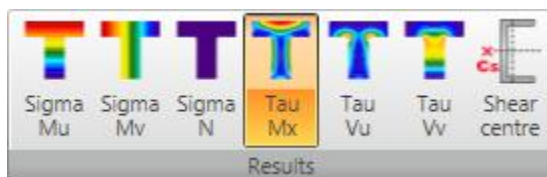
The courses of evaluated stresses are drawn in the Main window.

The table of cross-section characteristics and the table of unit stresses are displayed in the data window.

Ribbon groups **Results**, **Advanced results** and **Drawing setup** are available for this navigator command.



5.2.10.1 Ribbon group Results



Use commands in ribbon group **Results** to set the component of basic results to be drawn:

- **Sigma Mu** – switch to draw the course of normal stress caused by unit bending moment about the main principal axis u .
- **Sigma Mv** – switch to draw the course of normal stress caused by unit bending moment about the main principal axis v .
- **Sigma N** – switch to draw the course of normal stress caused by unit normal force N .
- **Tau My** – switch to draw the course of shear stress caused by unit torsional moment M_x .
- **Tau Vu** – switch to draw the course of shear stress caused by unit shear force V_u .
- **Tau Vv** – switch to draw the course of shear stress caused by unit shear force V_v .
- **Shear centre** – switch to draw the shear centre of the cross-section.

5.2.10.2 Ribbon group Advanced results



Use commands in ribbon group **Advanced results** to set the component of other results to be drawn:

- **Tau_u Mx** – switch to draw the course of shear stresses parallel to the first principal axis u caused by unit torsional moment Mx.
- **Tau_v Mx** – switch to draw the course of shear stresses parallel to the second principal axis v caused by unit torsional moment Mx.
- **Tau_u Vu** – switch to draw the course of shear stresses parallel to the first principal axis u caused by unit shear force Vu.
- **Tau_v Vu** – switch to draw the course of shear stresses parallel to the second principal axis v caused by unit shear force Vu.
- **Tau_u Vv** – switch to draw the course of shear stresses parallel to the first principal axis u caused by unit shear force Vv.
- **Tau_v Vv** – switch to draw the course of shear stresses parallel to the second principal axis v caused by unit shear force Vv.
- **Sigma ω** – switch to draw the course of stresses caused by unit bi-moment B
- **Tau ω** – switch to draw the course of stresses caused by the unit warping torsion of cross-section.

5.2.10.3 Ribbon group Drawing settings



- **Mesh** – switch on/off the drawing of the generated finite elements mesh.
- **Extreme** – switch on/off the drawing of extreme value of evaluated magnitude..
- **Arrows** – switch on/off the drawing of stress course directions.
- **Stress points** – switch on/off the drawing of points for stress analysis.
- **Stress points values** – switch on/off the drawing of values of evaluated magnitude in the points of stress analysis.

6 Check of cross-sections

The defined general cross-sections can be checked using defined internal forces components. Internal forces components are defined using so called Design matrixes. Each Design matrix can contain one or more rows with internal forces components.

The cross-section check is performed for defined **Requests for check**. One request consists of selected cross-section and selected design matrix. Moreover, in the request for check, it is possible to define coefficients to multiply internal forces components of the design matrix.

6.1 Design matrix

To define the design matrixes click navigator command **Design matrix > Input**.

Tables **Design matrixes** and **Internal forces** are displayed in the Main window. The internal forces components defined in the selected design matrix are displayed in the table **Internal forces**.

Design matrix

Copy

	Name	Type	System	
>	DM1	ULS	u,v y,z principal axis	
	DM2	SLS Char	u,v y,z principal axis	


Internal forces

	#	Name	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]	
>	1		0,0	0,0	0,0	0,0	30,0	0,0	
	2		0,0	0,0	0,0	30,0	0,0	0,0	
	3		120,0	0,0	0,0	30,0	0,0	0,0	
	4		-120,0	0,0	0,0	30,0	0,0	0,0	
	5		-120,0	0,0	0,0	30,0	0,0	30,0	

To add new design matrix click  above the table **Design matrix**.

To copy the current design matrix click **Copy** above the table **Design matrix**.

The table **Design matrix** contains following columns:

- **Name** – input the name of design matrix.
- **Type** – choose the type of combination for the current design matrix. The type of combination is considered as informative only (to determine the origin of the internal forces) and does not influence the calculation of stresses on the cross-section.
- **System** – choose the coordinate system, in which act the defined internal forces:
 - **u,v** – the defined internal forces act in main principal coordinate system u, v;
 - **y,z** - the defined internal forces act in reference coordinate system y, z.
-  - delete the appropriate design matrix.



To add new row of internal forces into the current design matrix click  above the table **Internal forces**.

Table **Internal forces** contains following columns:




- **Name** – input the name of internal forces row.
- **N** – input the value of normal force N.
- **Vy**,
- **Vz** – input the values of shear forces Vy or Vz.
- **Mx** – input the value of torsional moment Mx.
- **My**,
- **Mz** – input the values of bending moments My or Mz.
-  - delete the appropriate row of internal forces.

The table of internal forces can be copied using a clipboard from e.g. Microsoft Excel tables.

6.2 Check requests


To define the check requests click navigator command **Stress check > Requests**.

The table **Stress checks** is displayed in the main window.

Stress checks 											
	Id	Name	Cross-section	Design matrix	Coeff. N	Coeff. Vy	Coeff. Vz	Coeff. Mx	Coeff. My	Coeff. Mz	
	1	Check 1	IPE300 + L120/10 (Obecný)	DM1	1	1	1	1	1	1	
▶	2	Check 2	IPE300 + L120/10 (Obecný)	DM1	1	1	1	1	1.35	1	

To add new check request click  above the table **Stress checks**.

Table **Stress check** contains following columns:

- **Name** – input the name of check.
- **Cross-section** – select one from defined cross-section, which is checked it the current check.
- **Design matrix** – select one from defined design matrixes to be used in the current check.
- **Coeff. N** – input the value of coefficient to multiply the value of normal force **N** in all rows of the design matrix.
- **Coeff. Vy**,
- **Coeff. Vz** – input the values of coefficients to multiply the values of shear forces **Vy**, **Vz** in all rows of the design matrix.
- **Coeff. Mx** – input the value of coefficient to multiply the value of torsional moment **Mx** in all rows of the design matrix.
- **Coeff. My**,
- **Coeff. Mz** – input the values of coefficients to multiply the values of bending moments **My**, **Mz** in all rows of the design matrix.
-  - delete the appropriate check request.

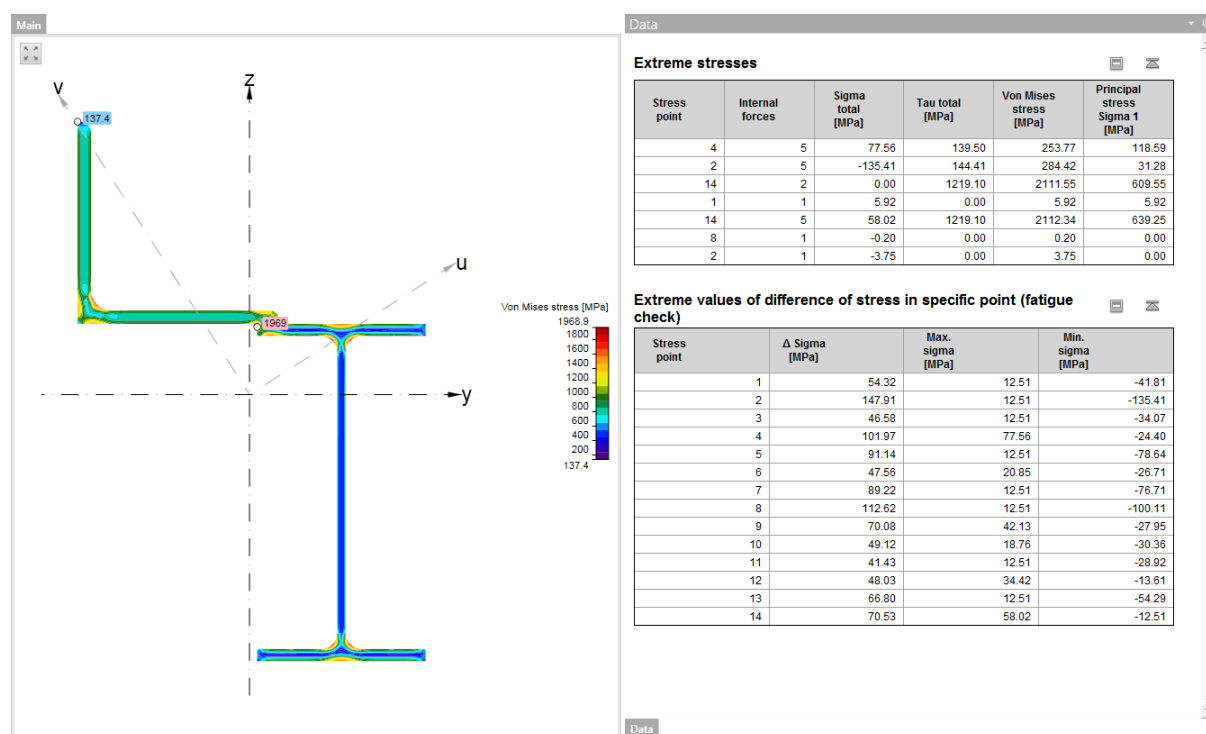
6.3 Check results

To evaluate the results of stress checks click navigator command **Stress check > Results**

The course of evaluated result component is drawn in the main window.

Table with the extreme values of stress check is displayed in the data window.

The ribbon group **Results** and **Drawing settings** are available for this navigator command.



6.3.1 Ribbon group Results

Use commands in ribbon group **Results** to set the evaluation of cross-sections checks:

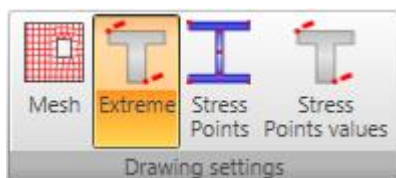


- **Requested check** – select one of defined checks to be evaluated.
- **Evaluation** – select mode of evaluation. Following modes are available:
 - **Envelope max** – switch to draw the envelope of maximal values of evaluated stress magnitude. The maximal value of evaluated magnitude from all rows in design matrix in each mesh finite element is found.
 - **Envelope min** – switch to draw the envelope of minimal values of evaluated stress magnitude. The minimal value of evaluated magnitude from all rows in design matrix in each mesh finite element is found.
 - **One row** – switch to draw course of evaluated stress magnitude for one row of the design matrix. The evaluated row can be set in the combo box **Matrix row**.
 - **Max row** – a row, which causes the maximal value of evaluated stress magnitude, is found in the design matrix. The course is drawn for this row. The number of this row is displayed in the combo box **Matrix row** and the row cannot be changed.

- **Min row** – a row, which causes the minimal value of evaluated stress magnitude, is found in the design matrix. The course is drawn for this row. The number of this row is displayed in the combo box **Matrix row** and the row cannot be changed.
- **Delta** – switch to draw the difference between the maximal and minimal value of evaluated stress magnitude, calculated from all rows of the design matrix..
- **Matrix row** – select a row of design matrix to display the course of evaluated stress magnitude. This option is not available for some **Evaluation** modes.
- **Sigma total** – switch to display the course of total normal stress caused by normal force and bending moments.
- **Tau total** – switch to display the course of total shear stress caused by shear forces, torsion and bimoment.
- **Von Mises** – switch to display the course of Von Mises stress.
- **Sigma 1** – switch to display the course of principal tensile axial stress.
- **Sigma Mu** – switch to display the course of normal stress caused by moment about the main principal axis u.
- **Sigma Mv** – switch to display the course of normal stress caused by moment about the main principal axis v
- **Sigma N** – switch to display the course of normal stress caused by normal force N.
- **Tau Mx** – switch to display the course of shear stress caused by torsional moment Mx.
- **Tau Fu** – – switch to display the course of shear stress caused by shear force Fu.
- **Tau Fv** – – switch to display the course of shear stress caused by shear force Fv.

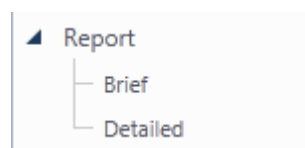
6.3.2 Ribbon group Drawing settings

Commands to set drawing options:



- **Mesh** – switch on/off the drawing of generated finite element mesh.
- **Extreme** – switch on/off the drawing of extreme value of evaluated magnitude.
- **Stress points** – switch on/off the drawing of the points for stress calculation.
- **Stress point values** – switch on/off the drawing of stress values in points for stress calculation.

7 Report



The input data, calculation results, data for stress check and stress check results can be printed to an output report. The report can contain texts, tables and pictures. The structure of report is pre-defined and the individual chapters and pictures can be switched

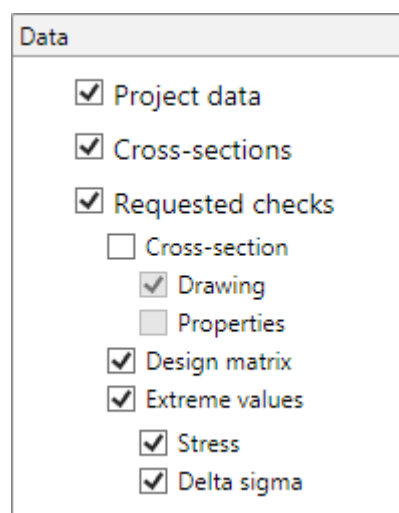
on/off.

Use commands in navigator **Report** to generate the report.

Ribbon group **Report view** is available for those navigator commands.

7.1 Brief report

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



The content of brief protocol can be set in the Data window:

- **Project data** – switch on/off printing of the table with basic and project identification data.
- **Cross-sections** – switch on/off printing of cross-sections data.
 - **Drawings** - switch on/off printing of cross-sections pictures.
 - **Properties** - switch on/off printing of tables with cross-section characteristics.
- **Requested checks** – switch on/off printing of tables and pictures for defined check requests..
 - **Cross-sections** – switch on/off printing of cross-sections data.
 - **Drawings** - switch on/off printing of cross-sections pictures.
 - **Properties** - switch on/off printing of tables with cross-section characteristics.
 - **Design matrix** – switch on/off printing of tables of design matrix.
 - **Extreme values** switch on/off printing of tables with resulting stresses:
 - **Stress** – switch on/off printing of table with extreme values of basic stresses.
 - **Delta sigma** – switch on/off printing of table with calculated amplitude of normal stress.

7.2 Detailed report

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

Data

- ☒ Project data
- ☒ Cross-sections
 - ☒ Components
 - ☒ Drawing
 - ☒ Properties
 - ☒ Stress points definition
 - ☒ Stress points results
- ☒ Materials
- ☒ Design matrixes
- ☒ Requested checks
 - ☐ Cross-section
 - ☒ Drawing
 - ☐ Properties
 - ☒ Design matrix
 - ☒ Extreme values
 - ☒ Stress
 - ☒ Delta sigma
 - ☒ Drawings

The content of brief protocol can be set in the Data window:

- **Project data** – switch on/off printing of the table with basic and project identification data.
- **Cross-sections** – switch on/off printing of cross-sections data:
 - **Components** - switch on/off printing of table with cross-section components.
 - **Drawings** - switch on/off printing of cross-sections pictures.
 - **Properties** - switch on/off printing of tables with cross-section characteristics.
 - **Stress points definition** – switch on/off printing of stress point coordinates tables.
 - **Stress points results** – switch on/off printing of tables with stresses caused by unit loads.
- **Materials** - switch on/off printing of tables with used materials.
- **Design matrixes** – switch on/off printing of tables with content of design matrixes.
- **Requested checks** – switch on/off printing of tables and pictures for each defined check request.
 - **Cross-sections** – switch on/off printing of cross-sections data.
 - **Drawings** - switch on/off printing of cross-sections pictures.
 - **Properties** - switch on/off printing of tables with cross-section characteristics.
 - **Design matrix** – switch on/off printing of tables of design matrix.
 - **Extreme values** switch on/off printing of tables with resulting stresses:
 - **Stress** – switch on/off printing of table with extreme values of basic stresses.
 - **Delta sigma** – switch on/off printing of table with calculated amplitude of normal stress.
 - **Drawings** - switch on/off printing of pictures with stress calculation results.

7.3 Ribbon group Report view



To print and export the report use commands in ribbon group **Report view**.

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