Example 26. Analysis of section of the composite (steel & RC) floor slab with monolithic slab by profiled steel sheeting (in Cross-section Design Toolkit module)

In this lesson you will learn how to:
– determine geometric properties for the section; they are required to define stiffness properties for bars in the meshed model.

Description:
Section of the composite (steel & RC) floor slab with monolithic slab by profiled steel sheeting (see Fig. 26.1):
– RC slab: material for slab - concrete B 30; embedded reinforcement: GOST 5781-82, class – A1, Ø20mm;
– steel sheet cold-formed sections with trapezoidal corrugations: height h=116mm, B1=187mm, t=1.2mm; material – galvanized steel;
– beam: asymmetric I-section (top flange – 150x12; bottom flange – 630x10; web – 320x16); material – rolled steel.

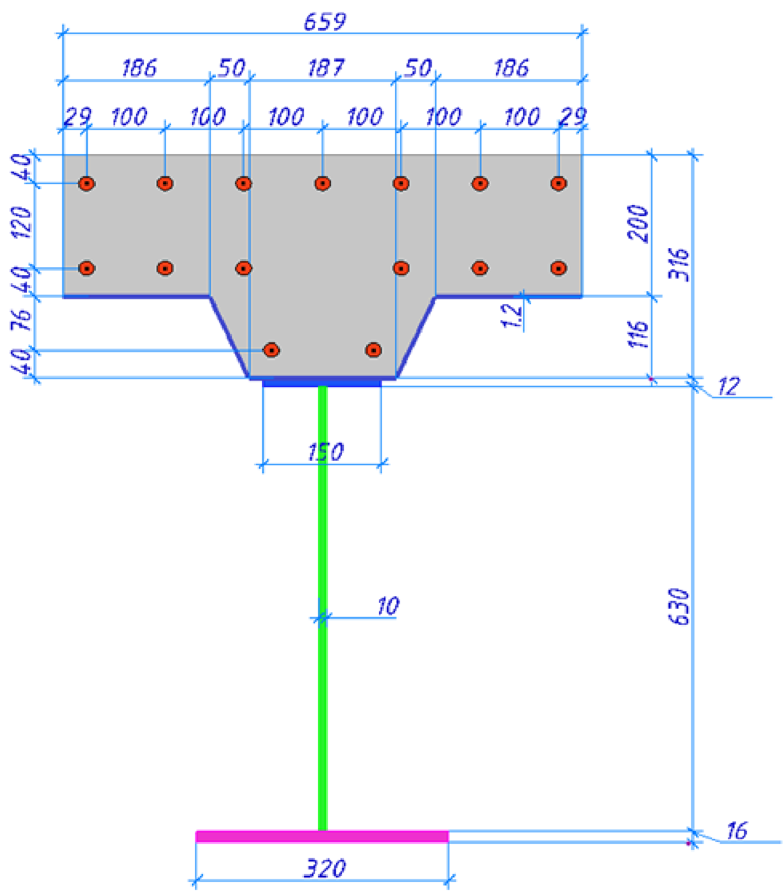


Figure 26.1. Cross-section of the composite (RC & steel) floor slab

- On the taskbar, click the **Start** button, then point to the folder that contains **LIRA-SAPR 2020** and click **Cross-section Design Toolkit**.

The Cross-section Design Toolkit module is a special graphic environment. It contains tools that enable the user to generate mono- and multi-material arbitrary sections. This module enables the user to do the following:

– generate geometry of arbitrary multi-material solid, thin-walled and combined sections of bars with complex and simple, nonstandard and standard shapes;
– define physical and mechanical properties of materials available in the section in order to determine stresses that nonlinearly depend on strains;
– compute stiffness parameters of the whole section and its components and export these parameters to VISOR-SAPR module;
– determine stress-strain state of the whole section in defined forces or forces imported from VISOR-SAPR module;
– define stress-strain diagrams for concrete of different grades and reinforcement according to SP 63.13330.2012, SNIP 2.03.01-84 and Eurocode 2;*
– define stress-strain diagrams for flats and steel shapes;
– define stress-strain diagrams in tabular format;
– visualize stress-strain state as mosaic plots, contour plots and diagrams for stress;
– animate stresses in step-type increase of defined forces.
Interface of this module is realized on the basis of SAPFIR tools.

The Cross-section Design Toolkit module may be also activated from other modules:

– from SAPFIR-Structures module – To generate the new built-up section, on the **Create** ribbon tab, on the **Sketching tools** panel, click the **CS Design Toolkit** button .

– from VISOR-SAPR module – to generate the new built-up section, on the **Create and edit** ribbon tab, on the **Stiffness and restraints** panel, click the **Cross-section Design toolkit** button .

Step 1. Creating new project

- To create the new project, on the **Application** menu, click **New** (button on the Quick Access Toolbar).

To define the project name:

- To save the project data, on the **Cross-section Design Toolkit** menu (Application menu), click **Save as** (button on the toolbar).
- In the **Save as** dialog box specify the following data:
  - file name – **Example26**;
  - location (folder) where you want to save this file.
- Click **Save**.

Step 2. Creating contour of multi-material section in the 'Cross-section Design Toolkit' module

- On the **Cross-section Design Toolkit** ribbon tab, on the **Section tools** panel, click the **Create section** button . The properties for generation of section contours will be displayed in the **Properties** dialog box.
- In the **Properties for generation: Contour of section** dialog box, select the **Material** row and click the **Browse** button .
- In the **Materials** dialog box, select the **General** tab and select material **Concrete B30** (see Fig. 26.2).
- In the **Materials** dialog box, click **OK** (defined material will be displayed in the **Material** row as the current material).
- In the **Properties for generation: Contour of section** dialog box, define the following data:

- Triangulation step – 20 (if this parameter is equal to zero, then triangulation step will be considered as equal to **division step by default** defined in the **Analysis settings** dialog box).

➪ Click **Apply to object** button ✔ .

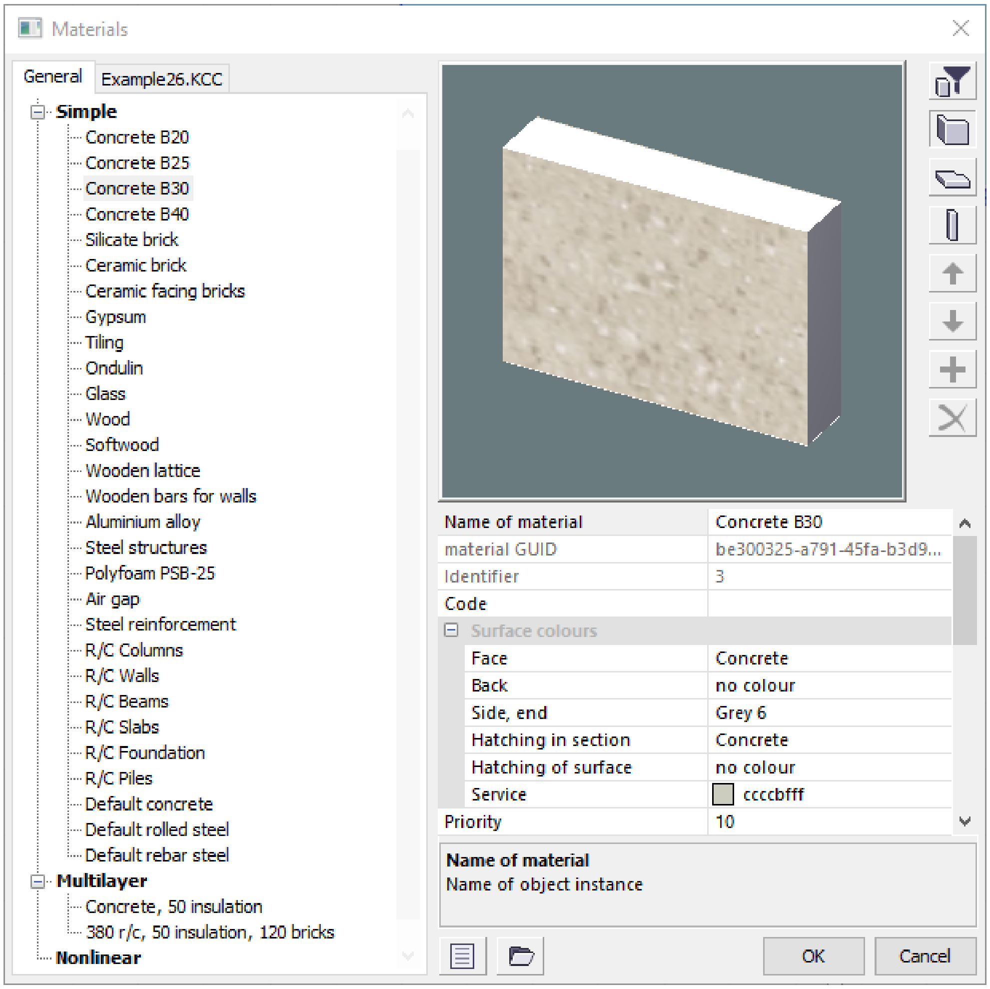


Figure 26.2. **Materials** dialog box

➪ On the **Create section** Options Bar, click the **Section** button Section .

➪ In the **Contours** dialog box (see Fig.26.3), unfold the **Standard** block, select the **T-section(S2)** section and define the following parameters:

- b=187 mm;
- h=316 mm;
- b1=660 mm;
- h1=200 mm.

➪ Click **OK**.

➪ Locate the section in the graphic area.

➪ To complete the generation procedure, press **Esc**.

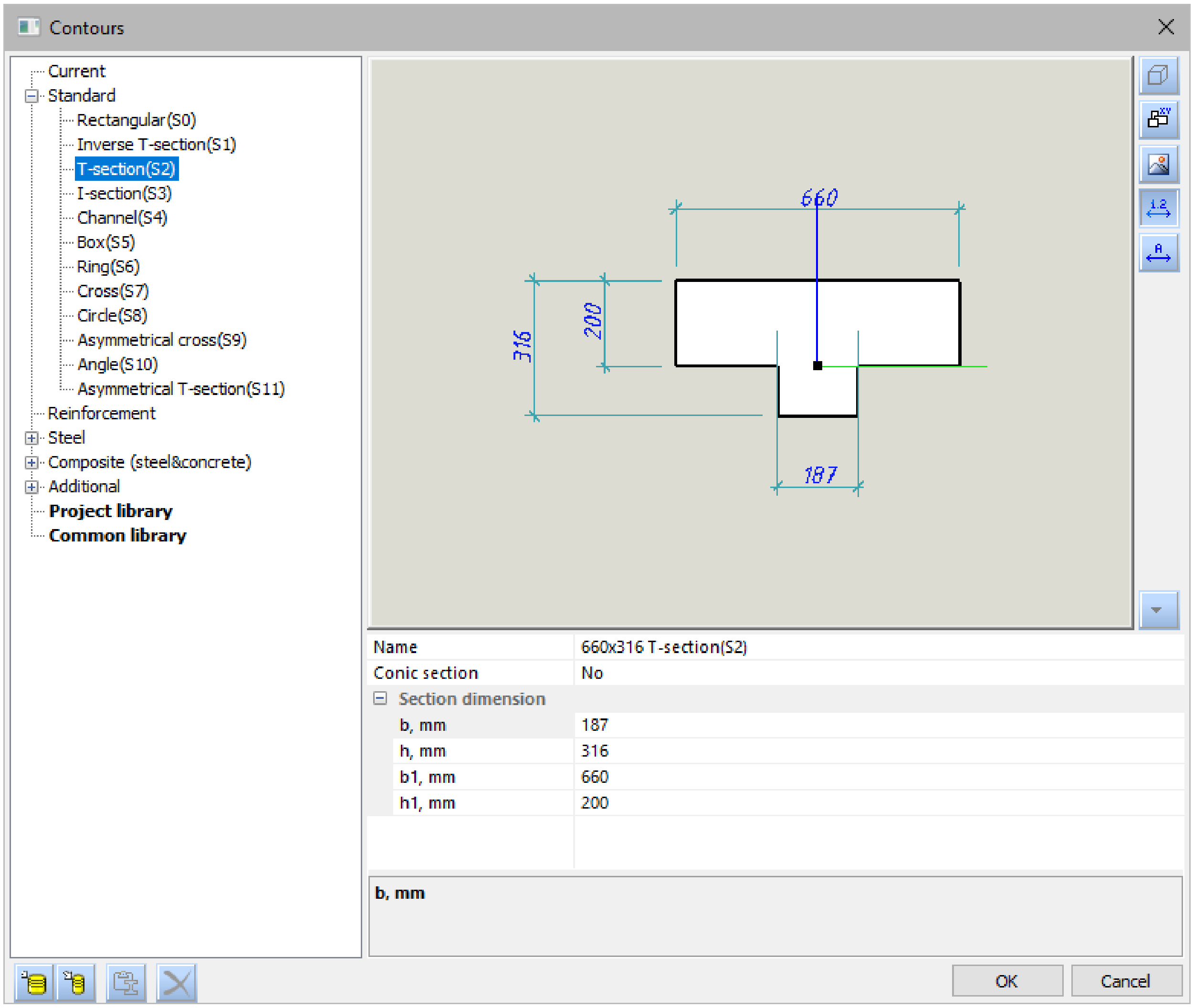


Figure 26.3. **Contours** dialog box

To edit the contour for the RC slab section:

- To place the local coordinate system at point 3, use the **UCS to point** command on the shortcut menu (see Fig. 26.4).

To place locator (that is used to generate and edit elements of the model), the following commands are provided:
F3 - to place locator to the nearest point on the model.
F4 - to place locator to the origin of coordinates. Locator is placed to the origin (if the UCS is visible in the graphic area).

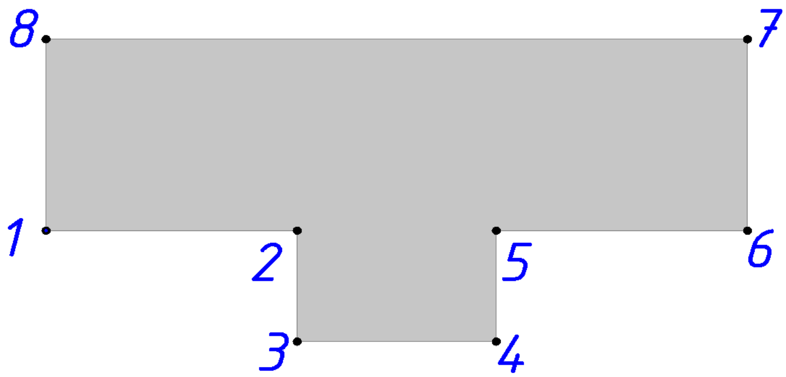
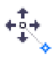


Figure 26.4. Contour of the slab section

- Select the contour of the section.
- On the **Cross-section Design Toolkit** ribbon tab, on the **Modify** panel, click the **Move vertex** button  .
- Drag the point 2 to the left.

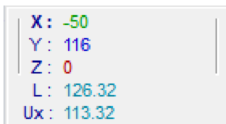
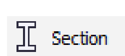


Figure 26.5. **Coordinates** window

- Press **X** on the keyboard. In the **Coordinates** window the X-coordinate becomes active. Define the coordinate as equal to -50mm (see Fig.26.5). Make sure that Y-coordinate is equal to 116mm.
- To confirm the specified value, press **Enter**.
- Drag the point 5 to the right.
- Press **X** on the keyboard. In the **Coordinates** window the X-coordinate becomes active. Define the coordinate as equal to 237mm. Make sure that Y-coordinate is equal to 116mm.
- To confirm the specified value, press **Enter**.
- To unselect the slab contour, press **Esc**.

To define coordinates from the keyboard, use the following shortcut keys to activate appropriate boxes in the ***Coordinates*** *window:*
X – to define the X-coordinate;
Y – to define the Y-coordinate;
Z – to define the Z-coordinate;
L – to define the length (indent from the last generated point);
U – to define the angle from the X-axis.
Up arrow key and Down arrow key on the numeric keypad - to switch between appropriate boxes in the ***Coordinates*** *window.*

<u>To define the point inclusion:</u>

- On the **Cross-section Design Toolkit** ribbon tab, on the **Section tools** panel, click the **Point inclusion** button  .
- On the **Point inclusion** Options Bar, define the following data:
  - method for generation – Point.
- On the **Point inclusion** Options Bar, click the **Section** button 

.
- In the **Contours** dialog box, define the following data:

- in the list of section types, select **Reinforcement**;
- in the right part of the dialog box, select the **Section dimensions** row, click the **Browse** button .

- In the **Select reinforcement** dialog box, define the following data:
  - Building code – GOST 5781-82;
  - Class – A1;
  - Diameter – 20;
  - Type – plain.
  - Click **OK** .
- In the **Contours** dialog box, click **OK**.
- Define coordinates where embedded reinforcement will be located:
  - the first inclusion - (X=29mm, Y= 40mm);
  - to confirm the data, press **Enter**;
  - the second inclusion - (X=157mm, Y= 40mm);
  - to confirm the data, press **Enter**.
- To place the local coordinate system at point 1, use the **UCS to point** command on the shortcut menu (see Fig. 26.4).
- Define coordinates where embedded reinforcement will be located:
  - the third inclusion - (X=29mm, Y= 40mm);
  - to confirm the data, press **Enter**;
  - the fourth inclusion - (X=29mm, Y= 160mm);
  - to confirm the data, press **Enter**.

- To exit from the **Point inclusion** mode, press **Esc**.
- To select the 3rd and the 4th inclusion, hold down the **Shift** key.
- On the **Edit** ribbon tab, on the **Modify** panel, point to the **Move** drop-down list and click the **Move by coordinates** button .

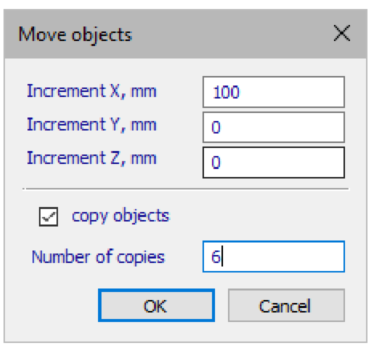


Figure 26.6. **Move objects** dialog box

- In the **Move objects** dialog box (see Fig. 26.6), define the following data:
  - increment X, mm – 100;
  - select the **copy objects** check box;
  - number of copies – 6.
- Click **OK**.
- To unselect the moved inclusions, press **Esc**.
- To delete the 5th inclusion (see Fig. 26.7), select it with the pointer and click **Delete** on the keyboard.

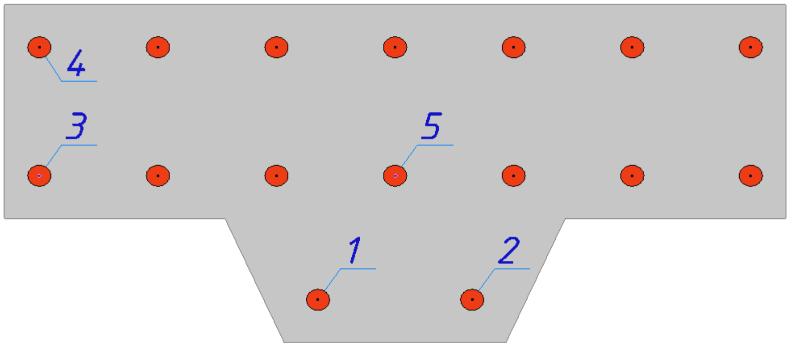


Figure 26.7. Section contour with inclusions (embedded reinforcement)

To create the profiled steel sheeting:

- On the **Cross-section Design Toolkit** ribbon tab, on the **Section tools** panel, click the **Strip** button .
- In the **Properties for generation: Strip** window, define the following data:
  - Line weight – Thick 09;
  - Thickness, mm – 1.2.
- Click **Apply to object** button .
- On the **Strip** Options Bar, define the following data:
  - method for generation - Line segment;
  - clear the **Close** check box;
  - make sure that the **Chain** check box is selected.
- Generate the profiled steel sheeting in this way: draw a line from initial point No.1 up to point No.2 (see Fig. 26.4), then specify points No.3,4,5,6 in sequence (when it is generated in the 'chain' mode, the end point of the previous segment will be used as initial point of the next segment).
- To confirm the data, press **Enter**.
- To exit from the **Strip** generation mode, press **Esc**.

To create the asymmetric I-section:

- On the **Cross-section Design Toolkit** ribbon tab, on the **Section tools** panel, click the **Create section** button . In the **Properties for generation** window you will see properties for generation of contour section.
- In the **Properties for generation: Contour of section** window, define the following data:
  - triangulation step - 4.
- Click **Apply to object** button .
- On the **Create section** Options Bar, click the **Section** button Section .
- In the **Contours** dialog box, unfold the **Steel** list, select the section **Asymmetric I-section**;
- In the right part of the dialog box, select the **Section components** row and click the **Browse** button ... .
- In the **Steel cross-section** dialog box (see Fig. 26.8), select the top flange of the I-section and define the following:
  - in the **Profile** drop-down list, select the steel table - **Prokat listovoj goryachekatanyj tolshhinoj 2.5...25 mm < list2-25_most.srt>**;
  - as there is no sheet with appropriate dimensions in the drop-down list, click the **Add new profile to steel table** button .

- In the **Add row to steel table** dialog box, define the following data:
- H = 150 mm;
- Tw = 12 mm;
- click the **Add** button (when the row is added, the steel table will be sorted by area of profile).
- in the **Steel** drop-down list, select **Stali po SP 16.13330. 2011, list i fason <SpListProf.steels.srt>**;
- in the next list, select the steel grade - C245.

➪ for the web:

- in the drop-down list select dimensions for the steel sheet - 630 x 10.

➪ for the bottom flange:

- in the drop-down list select dimensions for the steel sheet - 320 x 16.

➪ Click **OK**.

➪ In the **Contours** dialog box, click **OK**.

To compute the section that is not available in the steel table, it is also possible to use the SRS-SAPR module (Steel Rolled Shapes). In this module you could preview and edit available steel tables as well as create new steel tables. When you define geometric dimensions according to the profile layout, the geometric properties for the new profile will be computed. In this module (with FEM analysis) it is possible to determine twisting moment, warping stiffness, shear areas along the principal axes of the section, location of the shear centre and torsion centre.

➪ On the **Create section** Options Bar, define the following data:

- point to the **Snap** button ;
- from the drop-down list, select the base point of section snap as **Top centre** .

➪ Place the asymmetric I-section between points 3 and 4 (see Fig. 26.4).

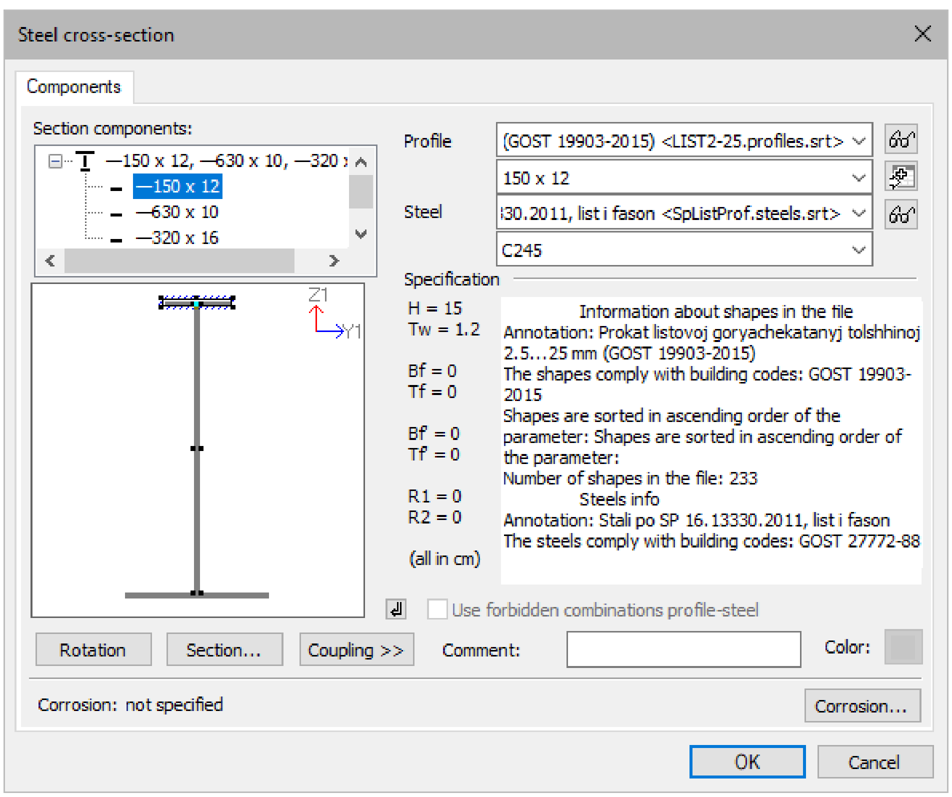


Figure 26.8. **Steel cross-section** dialog box

The image of the asymmetric section is moved together with 3D locator. To define appropriate location, click the 3D locator.
To snap to the centre of the section edge, this centre is marked with the pink triangle. To define location of the section, just click the left mouse button.

- To exit from the **Asymmetric I-section** generation mode, press **Esc**.
- Now geometry is defined and it is possible to compute the section properties.

To determine the stress-strain state of the section from axial, flexural, torsional and shear forces as well as bimoments, then specify appropriate set of forces in the ***Forces*** *dialog box (see the* ***Cross-section Design Toolkit*** *ribbon tab,* ***Forces*** *panel). When analysis is complete, you could visualize (as contour/mosaic plots) principal, axial, shear stresses as well as equivalent stresses according to the specified criterion of rupture. The stress-strain state is visualized together with the colour palette that describes the ranges of values. If there are two materials that are widely different in modulus of elasticity, you could work with two colour palettes.*

Step 3. Carrying out analysis of the section

- To define the triangulation step, on the **Cross-section Design Toolkit** ribbon tab, on the **Analysis** panel, click the **Settings** button .
- In the **Analysis settings** dialog box, define the following data:
  - division step by default - 1 mm.
- Click **OK**.

If another step is not mentioned for element of the section, then element contour will be divided with the step accepted by default.

- On the **Cross-section Design Toolkit** ribbon tab, on the **Analysis** panel, click the **Analysis** button .

Step 4. Preview and evaluation of analysis results

- On the **Cross-section Design Toolkit** ribbon tab, on the **Results** panel, click the **Rotate section** button .
- To preview the table of properties for the section as well as reference data for section components, on the **Cross-section Design Toolkit** ribbon tab, on the **Results** panel, click the **Properties** button EI (see Fig. 26.9).
- After complete analysis it is possible to show or hide the triangulation mesh for the section. To do this, on the **Cross-section Design Toolkit** ribbon tab, on the **Results** panel, use the **Mesh** button .
- To show or hide principal axes of the section, ellipse of inertia, core of the section, location of shear centre and torsion centre, intersections of neutral axes, on the **Cross-section Design Toolkit** ribbon tab, on the **Results** panel, click the **Axes and core of section** button .

Cross-section properties

Copy | Copy all

Annotation	Value	Units	Name
Geometric properties of whole cross-section			
Xo	0	mm	X-coordinate of gravity centre in the current coordinate system
Yo	0	mm	Y-coordinate of gravity centre in the current coordinate system
ф	0.01	°	Rotation of principal Y1-axis of the section relative to x-axis of current coordinate system
Ry	314.07	mm	Gyration radius relative to principal Y1-axis
Rz	152.81	mm	Gyration radius relative to principal Z1-axis
Pext	3809.45	mm	Perimeter of outer contours
Pint	0	mm	Perimeter of inner contours
Ro	3.064	t/m³	Average density of cross section
g	0.546	tf/m	Average unit weight
Y-	70.72	mm	Core distance in negative direction of principal Y1-axis
Y+	70.78	mm	Core distance in positive direction of principal Y1-axis
Z-	318.84	mm	Core distance in negative direction of principal Z1-axis
Z+	148.39	mm	Core distance in positive direction of principal Z1-axis
Torsional properties			
Yt	17.13	mm	Y1 coordinate of torsion centre in coordinate system of principal axes Y1oZ1
Zt	110.99	mm	Z1 coordinate of torsion centre in coordinate system of principal axes Y1oZ1
Shear properties			
Ys	16.62	mm	Y1-coordinate of shear centre in coordinate system of principal axes Y1oZ1
Zs	118.15	mm	Z1-coordinate of shear centre in coordinate system of principal axes Y1oZ1
Stiffness properties			
EA	9.1786e5	tf	Axial stiffness
EIu	90539	tf·m²	Bending stiffness relative to central U-axis
EIv	21432	tf·m²	Bending stiffness relative to central V-axis
EIuv	-17.23	tf·m²	Centrifugal stiffness relative to central axes UV
EIy	90539	tf·m²	Bending stiffness relative to principal Y1-axis
EIz	21432	tf·m²	Bending stiffness relative to principal Z1-axis
ESy	1.1431e5	tf·m	Product of static moment of half-section and its elasticity modulus relative to principal Y1-axis
ESz	54274	tf·m	Product of static moment of half-section and its elasticity modulus relative to principal Z1-axis
GJt	4248.9	tf·m²	Torsional stiffness - product of shear modulus and torsion moment of inertia
EIw	676.157354	tf·m⁴	Warping stiffness - product of elasticity modulus and warping moment
GFy	2.3998e5	tf	Shear stiffness Y1 - product of shear modulus and shear area relative to principal Y1-axis
GFz	21461	tf	Shear stiffness Z1 - product of shear modulus and shear area relative to principal Z1-axis

Figure 26.9. **Cross-section properties** dialog box

Step 5. Generating file for LIRA-SAPR

- On the **Cross-section Design Toolkit** ribbon tab, on the **Save** panel, click the **Save section as CSDT file** button .
- In the **Save as** dialog box, with the file type **Cross-section Design Toolkit (*.KCC)**, defined file name **Example26** and the path where the file should be saved, click **Save**.

Generated arbitrary multi-material cross-section of the bar may be exported to VISOR-SAPR module and assigned to a bar element. It is also possible to import to the Cross-section Design Toolkit module the cross-section and forces obtained in analysis. To import forces for several elements, use appropriate table in the Report Book.

