

Example 23. Analysis of multi-storey building with a braced frame and design of monolithic RC slab in SAPFIR-Structures and SAPFIR-RC modules

In this lesson you will learn how to:

- generate the architectural and analytical model for the multi-storey building in [SAPFIR](#) module;
- create assemblage tables in [SAPFIR](#) module;
- generate the FE meshed model for multi-storey building in [SAPFIR-Structures](#) module in order to transfer it further to [VISOR-SAPR](#) module;
- import the meshed model to [VISOR-SAPR](#) module;
- use design options;
- analyse reinforcement for elements of the braced frame in the multi-storey building;
- import analysis results for reinforcement from the [VISOR-SAPR](#) module to [SAPFIR-RC](#) module;
- design the monolithic RC floor slab in [SAPFIR-RC](#) module.

Description:

Plan of the 1st storey and sectional elevation are presented in Fig. 23.1.a, 23.1.b.

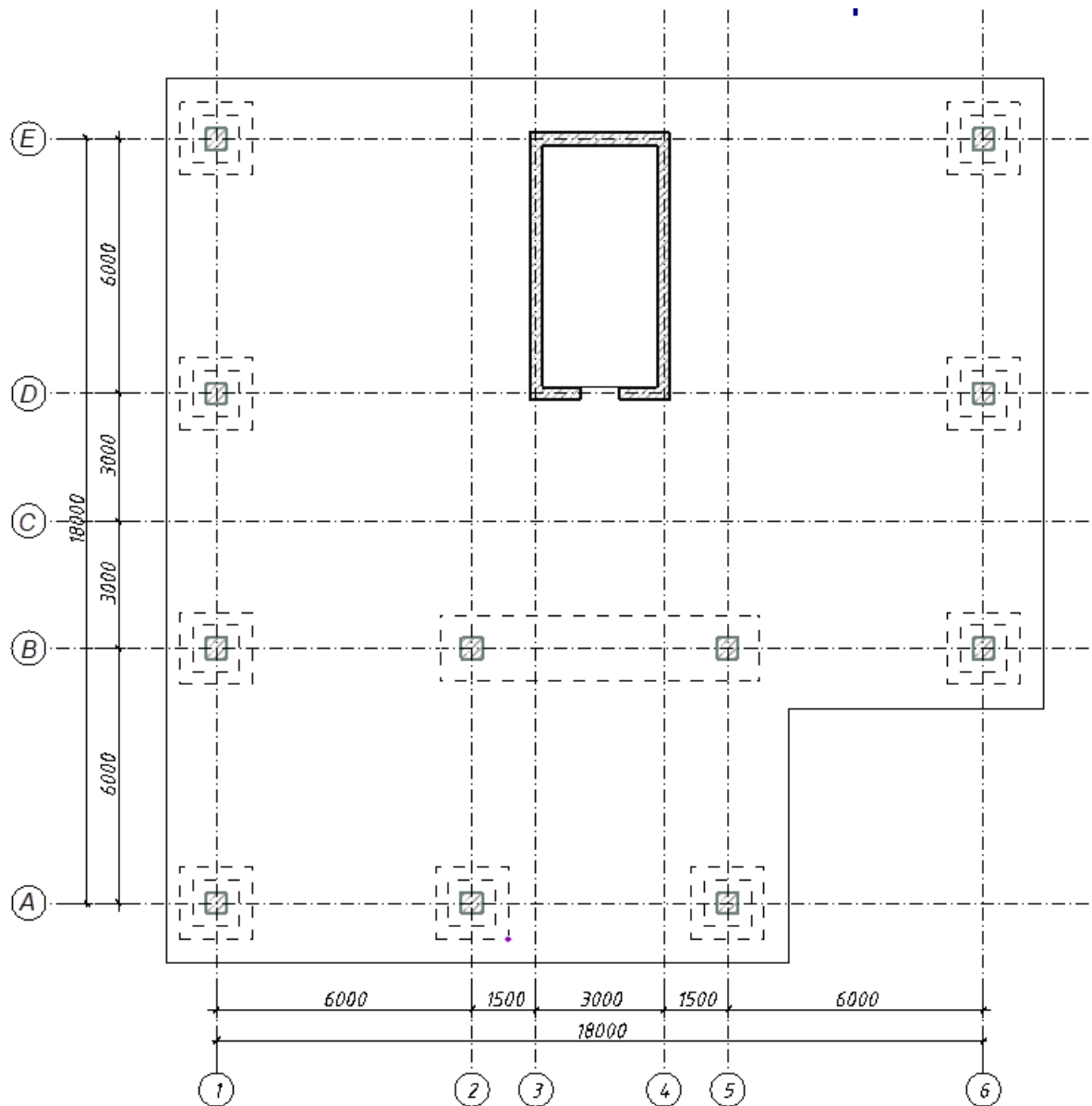


Figure 23.1.a. Plan of the 1st storey

Height of standard storey - 4 m. Number of storeys - 5. Floor level of the 1st storey 0.000.

Building code for analysis of elements – SP 63.13330.2012/2018.

Material for elements: columns, capitals – concrete B30; walls, floor slabs, foundation slab – concrete B25.

Column section 0.6x0.8 m.

Capital: two drop panels, $b \times h = 0.3 \times 0.2$ m.

Thickness of floor slab – 0.2 m. Drop panel in a slab – 0.2 m. Thickness of foundation slab - 0.6 m. Wall thickness – 0.2 m.

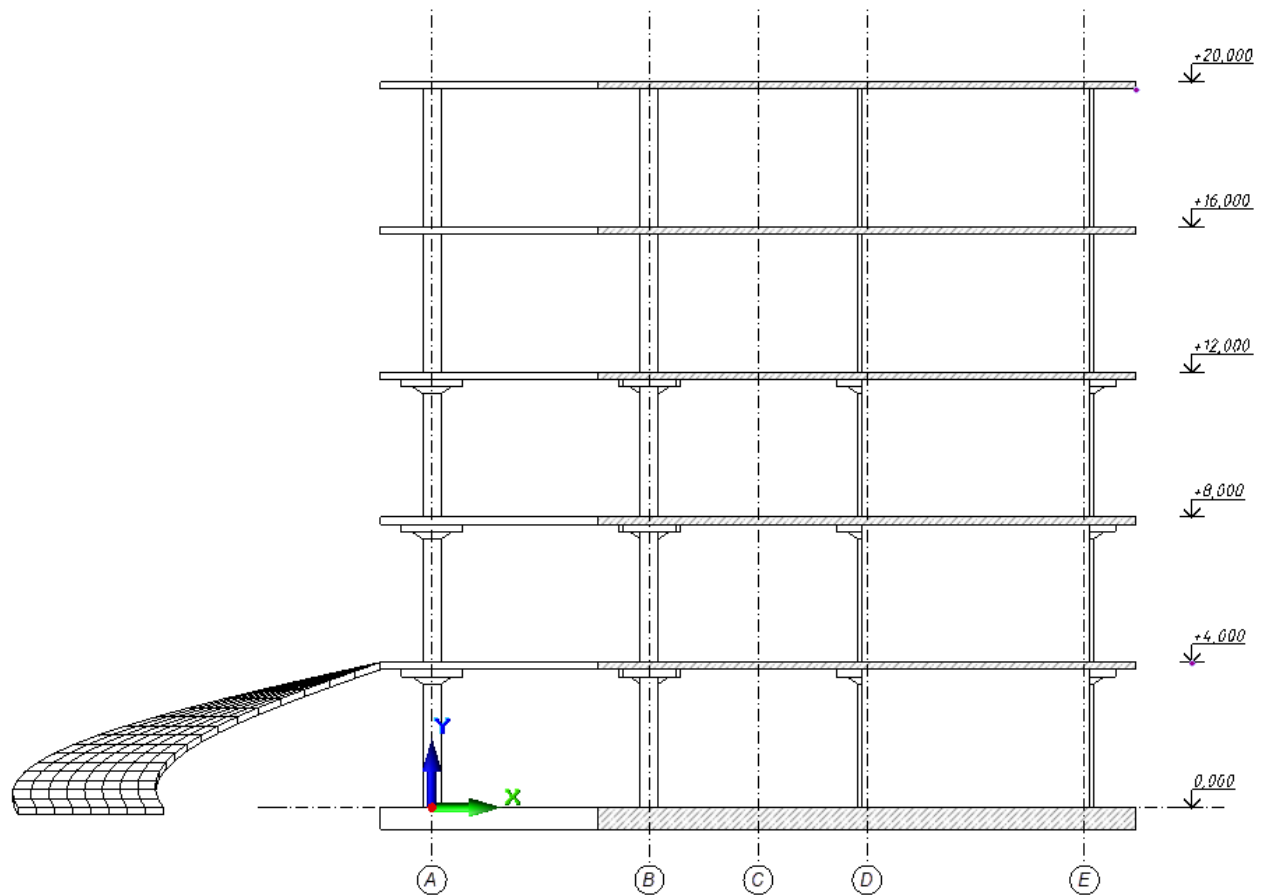


Figure 23.1.b. Sectional elevation

Loads:

– Load case 1

load from enveloping walls – dead uniformly distributed along the line $g_1 = 1.6 \text{ tf/m}$, applied to floor slabs at all storeys;

load from partition – dead uniformly distributed along the line $g_2 = 1.6 \text{ tf/m}$, applied to floor slabs at all storeys;

load from floor structure – dead uniformly distributed across the area $g_3 = 0.3 \text{ tf/m}^2$, applied to floor slabs at all storeys;

load from roof structure – dead uniformly distributed across the area $g_4 = 0.1 \text{ tf/m}^2$, applied to floor slab.


– Load case 2

live load from floor slab $g_5 = 0.5 \text{ tf/m}^2$;


surface load on floor slabs $g_6 = 2.0 \text{ tf/m}^2$.

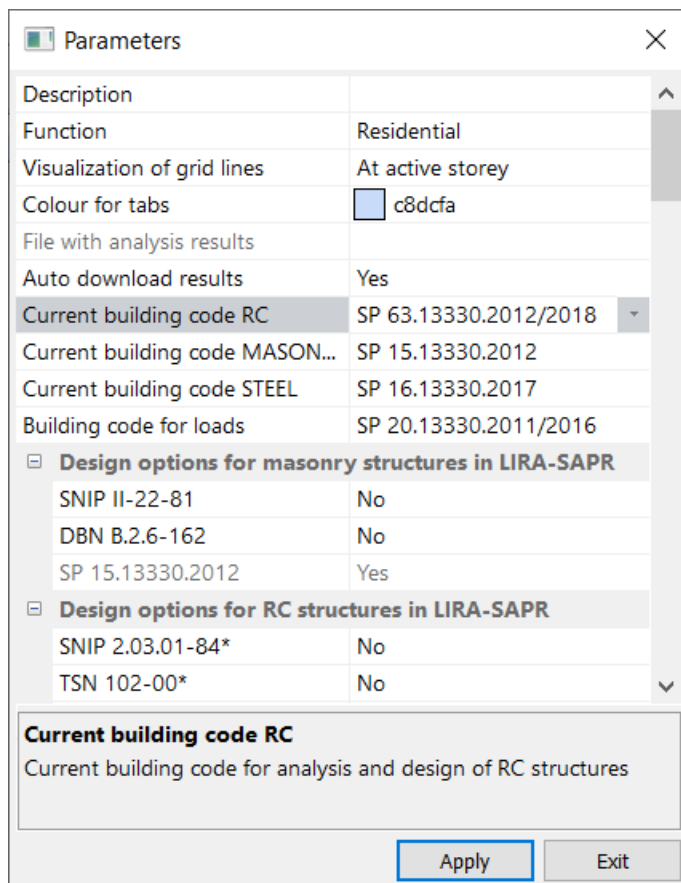
- ⇒ On the taskbar, click the **Start** button, and then point to **All Programs**. Point to the folder that contains **LIRA SAPR / LIRA-SAPR 2020** and then click **SAPFIR-3D 2020**.

Step 1. Creating new project and defining its properties


- ⇒ To create new project, on the **SAPFIR-3D** menu (Application menu), click **New** (button  on the toolbar).

To define the project properties:

- ⇒ On the **Create** ribbon tab, on the **Project** panel, click **Project properties** .
- ⇒ In the **Parameters** dialog box, make sure that SP 63.13330.2012/2018 is defined as the current building code for analysis of RC structures. Other parameters remain by default (see Fig. 23.2).
- ⇒ Click **Apply**.



The Parameters dialog box is shown with the following settings:


Parameters	
Description	
Function	Residential
Visualization of grid lines	At active storey
Colour for tabs	 c8dcfa
File with analysis results	
Auto download results	Yes
Current building code RC	SP 63.13330.2012/2018
Current building code MASON...	SP 15.13330.2012
Current building code STEEL	SP 16.13330.2017
Building code for loads	SP 20.13330.2011/2016
Design options for masonry structures in LIRA-SAPR	
SNIP II-22-81	No
DBN B.2.6-162	No
SP 15.13330.2012	Yes
Design options for RC structures in LIRA-SAPR	
SNIP 2.03.01-84*	No
TSN 102-00*	No

Current building code RC
Current building code for analysis and design of RC structures

Buttons: Apply, Exit

Figure 23.2. **Parameters** dialog box

To visualize the workspace:

- ⇒ To define the settings for visualization of the workspace, on the **View** ribbon tab, on the **Settings** panel, click **Visualization** .
- ⇒ In the **Visualization options** dialog box (see Fig. 23.3), define the following data:
 - in the **Metric grid** area, select the **Only in the 1st quadrant** option;
 - define the number of cells – 20
- ⇒ To close the dialog box and apply modifications, click **OK**.

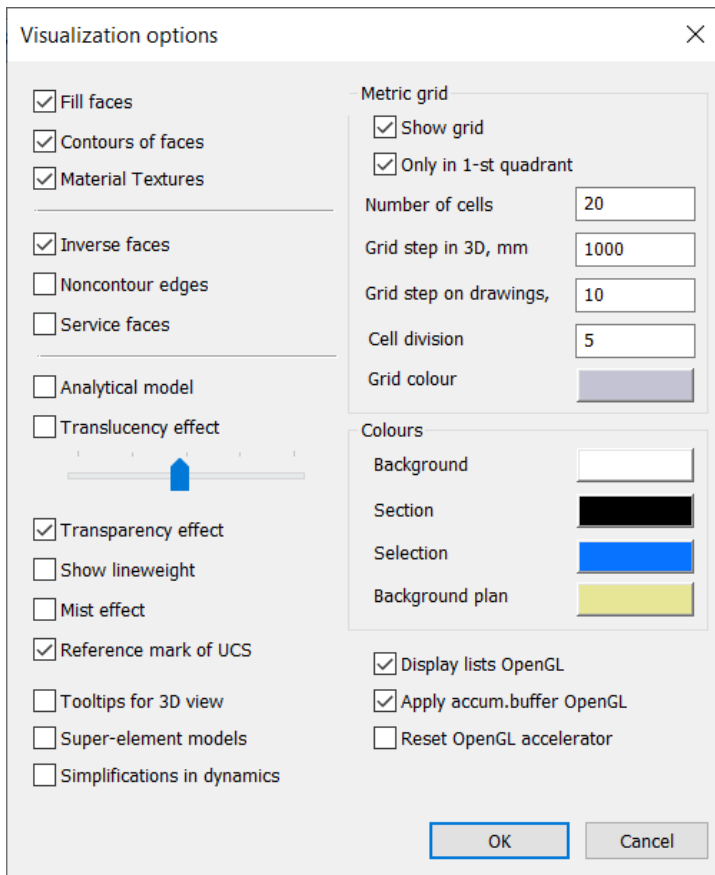





Figure 23.3. Visualization options dialog box

To define the project name:

- ⇒ To save the project data, on the **SAPFIR-3D** menu (Application menu), click **Save** (button  on the toolbar).
- ⇒ In the **Save as** dialog box specify the following data:
 - file name – Example23;
 - location (folder) where you want to save this file.
- ⇒ Click **Save**.

Step 2. Modifying the properties for the storey


The building and the storey are generated automatically when the first object is generated in the graphic window.

- ⇒ In the **Structure** window (in the right part of the screen), select the  **Storey No.1** row. The properties of this (selected) storey will be displayed in the **Properties** window.
- ⇒ In the **Properties** window, define the following data:
 - storey height, mm - 4000.
 - other parameters remain by default.
- ⇒ To apply modifications to the object, click the **Apply to object**  button in the **Properties** window or press **Enter**.

Step 3. Defining grid lines

To define rectangular grid in grid lines 1–6 and A–E:

- ➔ On the **Create** ribbon tab, on the **Sketching tools** panel, select the **Grid**  command in the appropriate drop-down list.

- ➔ Click the **Parameters**  button on the **Grid lines** Options Bar.

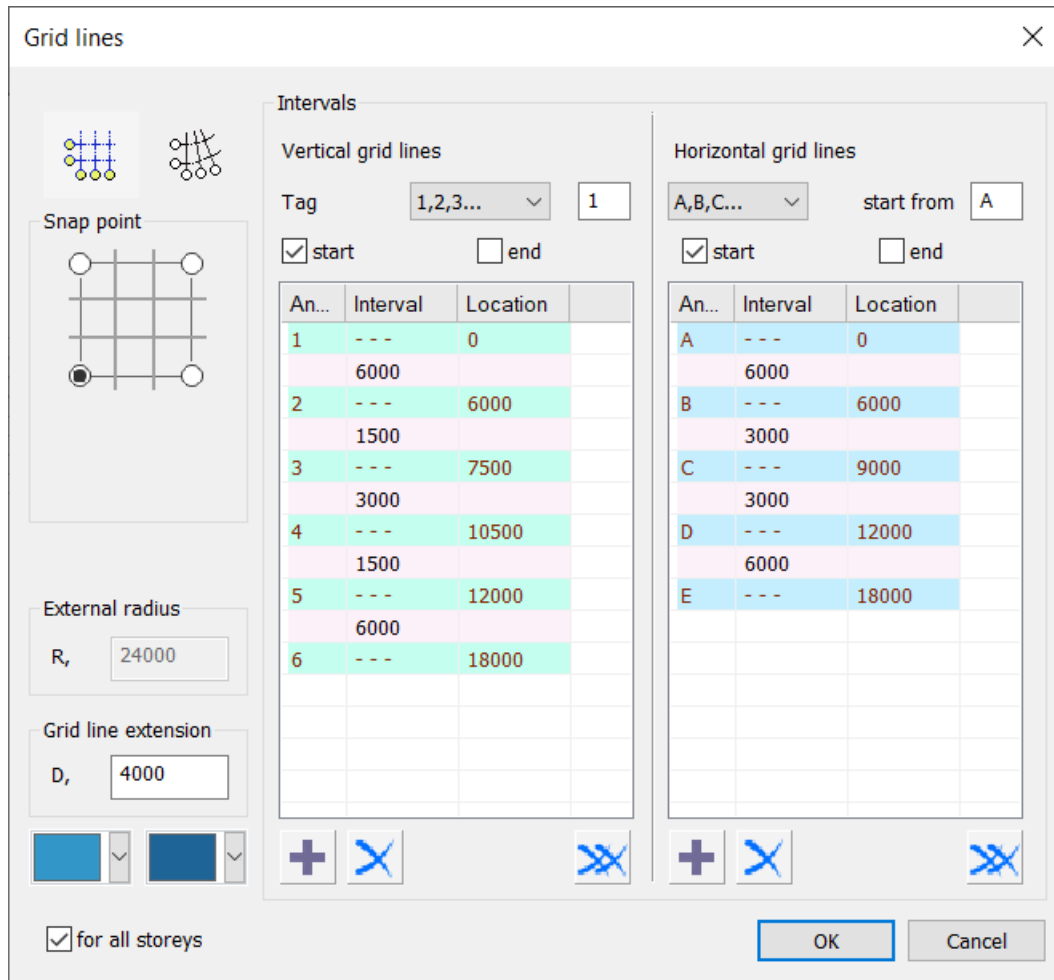




Figure 23.4. **Grid lines** dialog box



- ➔ In the **Grid lines** dialog box (see Fig. 23.4), define the following data:
 - select the grid type – **Rectangular grid** (by default, the left lower corner is defined as the snap point); coordinates of the snap point – X, mm =0, Y, mm =0;
 - extension for grid lines D, mm = 4000;
 - click the **Add interval** button  for the vertical grid lines;
 - select the value in the **Interval** column and replace it with 6000;
 - in the same way, add several intervals between vertical grid lines (vertical grid lines are tagged with Arabic numerals 1,2,3...) with values 1500, 3000, 1500, 6000 mm. Resultant location of grid lines are determined automatically;
 - for the horizontal grid lines, select the type for the tag as A, B, C...
 - define the intervals between horizontal grid lines – 6000, 3000, 3000, 6000 mm. Resultant location of grid lines are determined automatically.
- ➔ Click **OK**. (In the graphic window you will see rectangular grid with appropriate grid lines).

To define (automatically) dimensions for grid lines:

- ⇒ Select the grid with the pointer (the grid will be coloured blue).
- ⇒ On the **Grid lines** Options bar, click the **Specify dimensions** button .
- ⇒ To unselect the grid lines, press **Esc**.

Step 4. Defining columns

To define columns:

- ⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, point to the **Column** drop-down list and click the **Column** button . In the **Properties** window you will see the properties for column generation.
- ⇒ On the **Column** Options Bar, click the **Section** button  **Section**.
- ⇒ In the **Parameters of cross-section** dialog box (see Fig.23.5), define the following data:
 - select the type of sections - **Rectangular(S0)**;
 - parameter b, mm = 600;
 - parameter h, mm = 800.
- ⇒ Click **OK**.

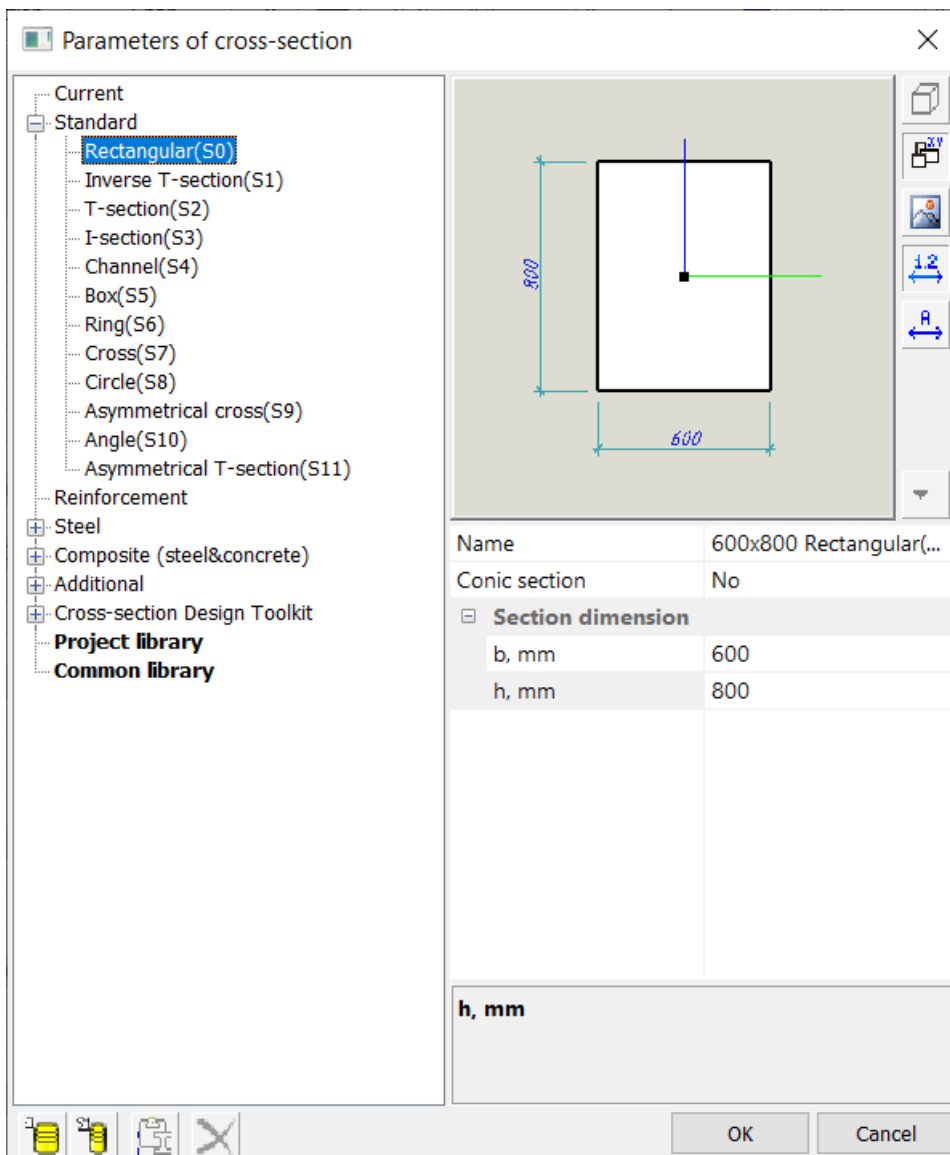
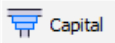





Figure 23.5. Parameters of cross-section dialog box

- ⇒ In the **Properties for generation: Column** window, in the **PRB parameters** list, define the **Generate PRB** option as yes to automatically generate the perfectly rigid body (PRB) in the slab along the contour of the column section.
- ⇒ Click the **Apply to object** ✓ in the **Properties** window or press **Enter**.
- ⇒ Locate the columns at intersections of grid lines B-2 and B-5 (see Fig. 23.7).

To define columns with capitals:

- ⇒ On the **Column Options Bar**, click the **Capital** button .
- ⇒ In the **Capital** dialog box (see Fig. 23.6), click the **Add drop panel** button  to define the drop panel for the capital. Then select the row and define the following data:
 - h, mm = 200;
 - bx, mm = 300.
 - by, mm = 300.
 - select the **Slope** check box.

- ⇒ Click the **Add drop panel** button  once again to define the second drop panel for the capital. Then select the row and define the following data:
- h , mm = 200;
 - bx , mm = 300
 - by , mm = 300.
- ⇒ To visualize the view of created capital in analytics, select the **Analytical model** option.
- ⇒ Click **Apply** .

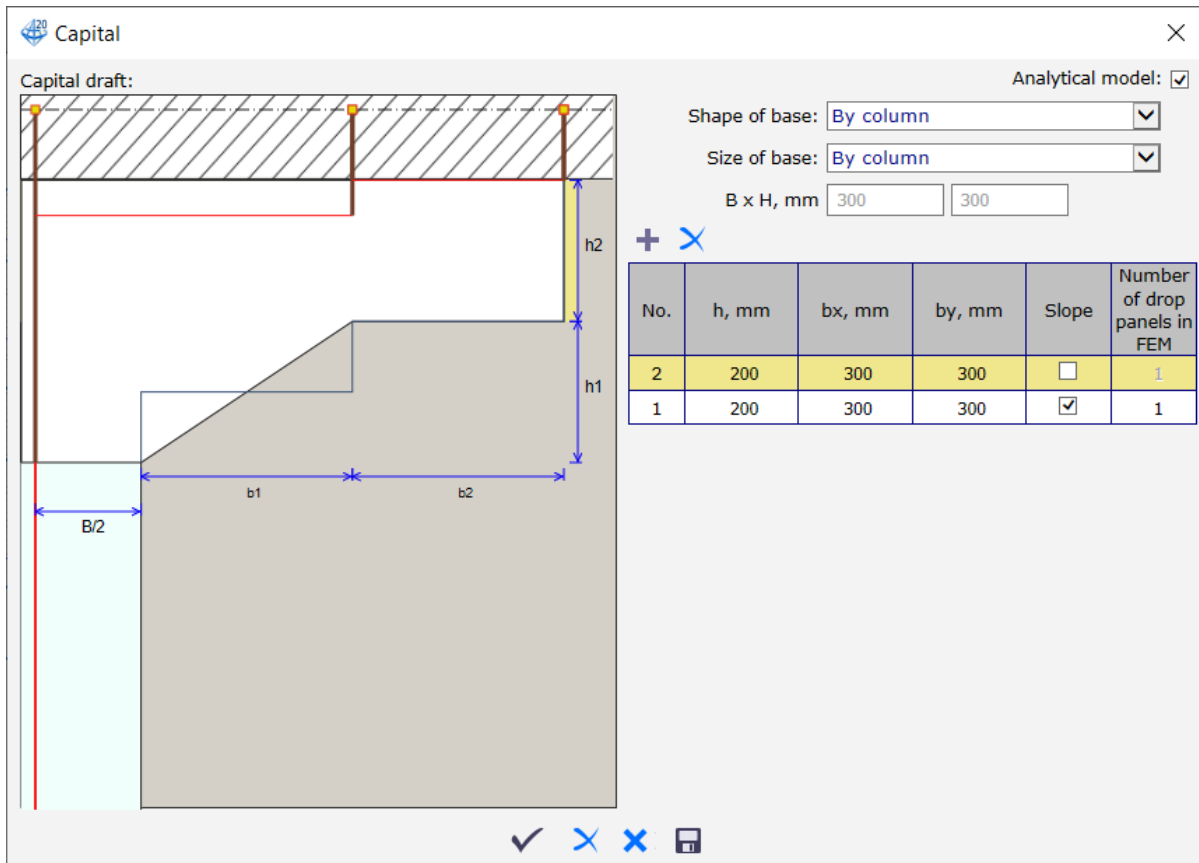


Figure 23.6. **Capital** dialog box

- ⇒ Locate the columns as indicated in Fig. 23.7.



To rotate the model to select appropriate perspective both before and during the generation, hold down the right mouse button.
To navigate in the graphic window of the project, hold down the wheel mouse button.
To move the object near, rotate the wheel mouse button.

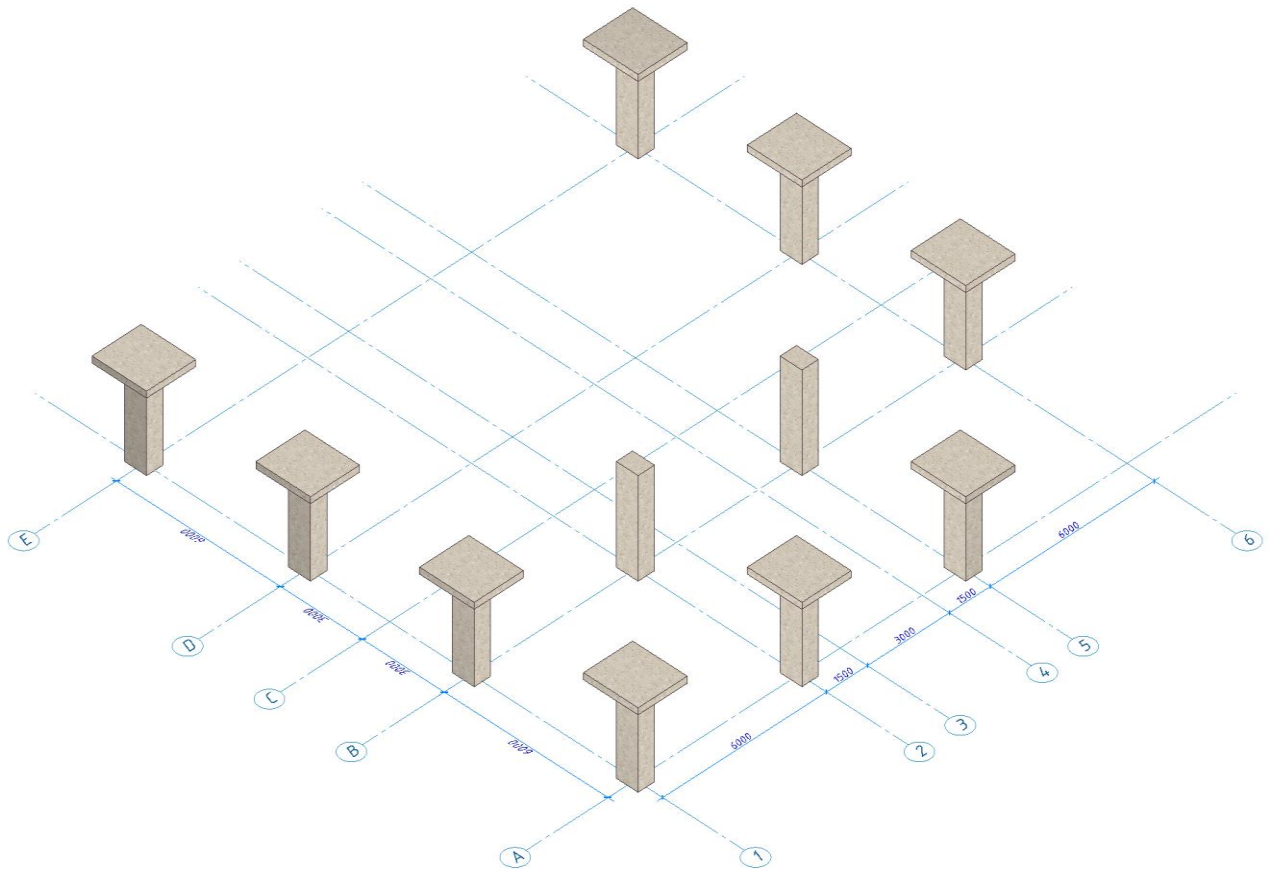






Figure 23.7. Layout of columns

⇒ To complete the generation of columns, press **Esc**.

Step 5. Defining walls

To define walls for the staircase:

- ⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, point to the **Wall** drop-down list and click the **Load-bearing wall** button .
- ⇒ In the **Properties for generation: Wall** window, define the **Generate PRB** option as yes.
- ⇒ Click the **Apply to object**  in the **Properties** window.
- ⇒ On the **Wall Options Bar**, define the following data:
 - method for generation - Rectangle ;
 -  thickness – 200;
 - snap – axis at centreline of the wall;
 - base level – 0;
 - level of wall top – 0 from storey top.
- ⇒ To generate the stiffness core, specify in sequence two points 3E and 4D (see Fig. 23.8).

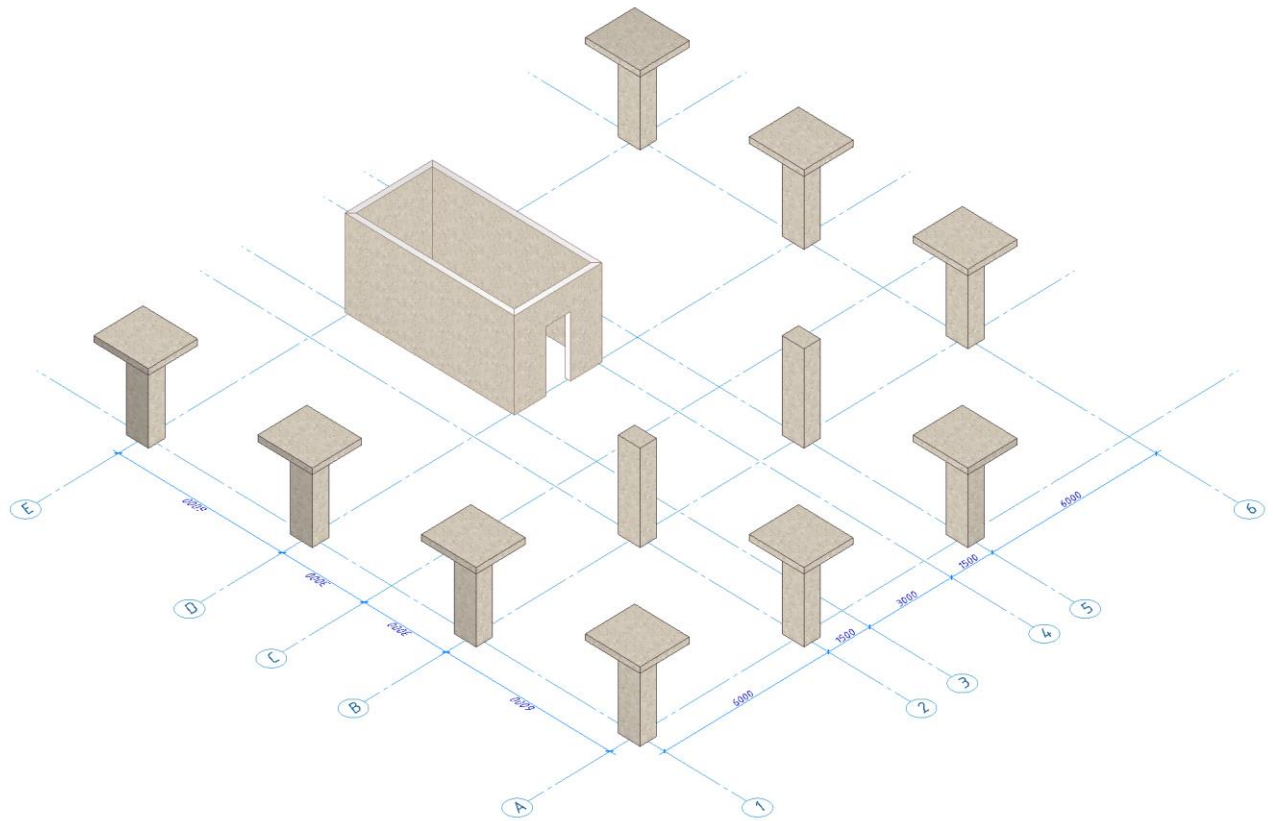




Figure 23.8. Layout for columns and walls

Step 6. Defining door

To define door:

- ⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, click the **Door** button .
- ⇒ On the **Door Options Bar**, click the **Parameters** button .
- ⇒ In the **Parameters of doors** dialog box (see Fig. 23.9), define the following data:
 - open the **Rectangular** list and select the door type – **Rectangular opening**.
- ⇒ Click **OK**.

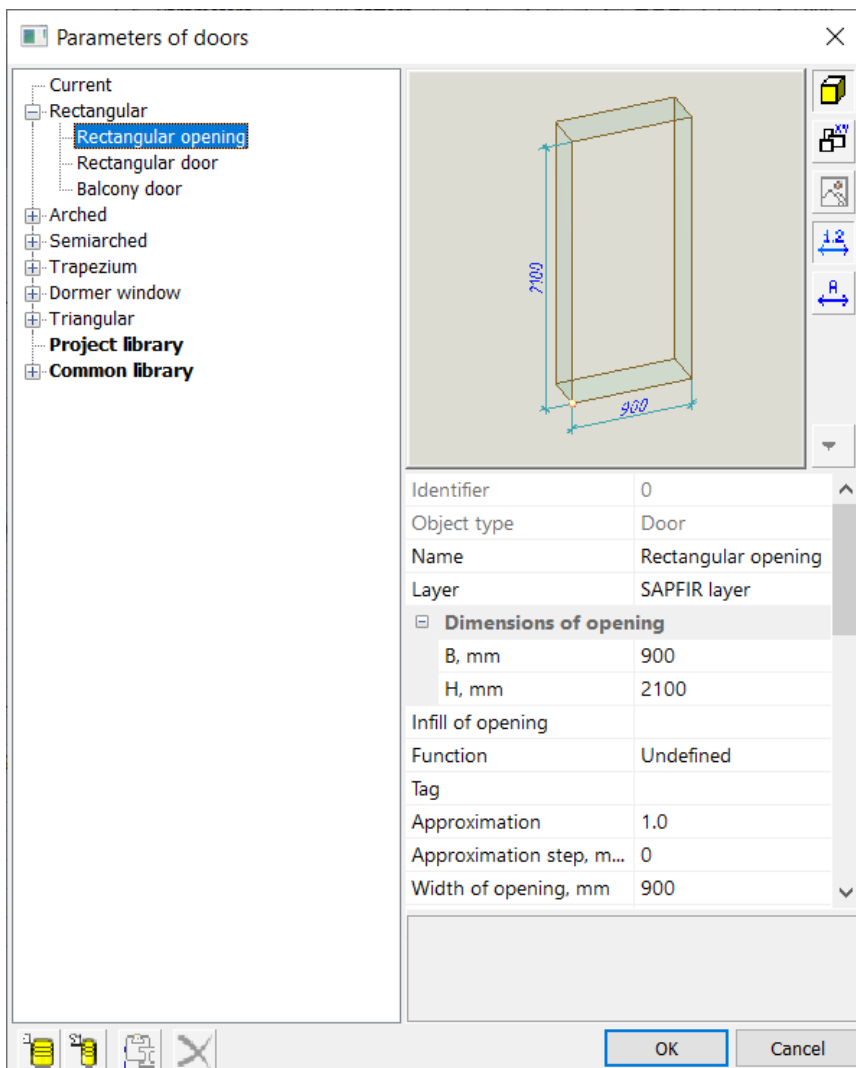




Figure 23.9. Parameters of doors dialog box

⇒ On the **Door** Options Bar, define the following data:

- span for the door – centre;
-  - door width, mm – 900;
-  - door height, mm – 2100.

⇒ Place the opening to the centre of the wall (for the snap option, the wall centre will be denoted with the pink triangle) located along the D line (see Fig. 23.8).


⇒ To complete the generation of door, press **Esc**.






The frame image of the door opening is moved together with the 3D locator. To define appropriate location, use the 3D locator. To fix the location of the door opening, just click at appropriate location.

Step 7. Defining and modifying the floor slab

To define floor slab:

⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, point to the **Slab** drop-down list and click the **Slab** button .

⇒ On the **Slab** Options Bar, define the following data:

- method for generation –  **Rectangle**;
- thickness  – 200;
- level –  0 from storey bottom;

⇒ To start generation, specify in sequence 2 points (beginning and end of the diagonal in rectangle):

- press **X** on the keyboard (now you could edit the X-coordinate in the coordinate window) and specify the value (-1170);
- use the *Down arrow* key on the numeric keypad and define the Y-coordinate with the value (-1420);
- to confirm the data, press **Enter** (the first point of the slab diagonal is defined and the slab will be generated further dynamically);
- on the keyboard, type 19420 (the value will be placed to the Y-coordinate, as the previous defined coordinate);
- use the *Up arrow* key on the numeric keypad and define the X-coordinate with the value 19420;
- to confirm the data, press **Enter**;
- to complete the generation of the slab, 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.*

To move dimension line:

- ⇒ Select the dimension line along the X-axis (the program automatically opens the **Edit** ribbon tab).
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, make sure that the **Move** option is active (the **Move** button  should be selected).
- ⇒ Specify one of the check points on dimension line (at the beginning or at the end) as the base point and drag the dimension line. Release the mouse button.
- ⇒ Click at the place where dimension line should be located.
- ⇒ To unselect dimension line, press **Esc**.
- ⇒ Repeat the same procedure for the dimension line along the Y-axis.

To modify the contour of the floor slab:

- ⇒ Select the floor slab (the program automatically opens the **Edit** ribbon tab).
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, click the **Insert vertex** button .
- ⇒ Rotate the model (just hold down the right mouse button) so that to bring to front the right angle of the slab (intersection of grid lines 6 and A).
- ⇒ Drag the slab edge that is parallel to the X-axis (the nearest one to the A grid line) towards yourself (the slab contour will be broken).
- ⇒ To define the X-coordinate for the 1st vertex, press **X** on the keyboard and in the **Coordinates** window define the value X=13420. Then use the *Down arrow* key on numeric keypad and define the Y-coordinate with value Y= - 1420.

- ⇒ To confirm the data, press **Enter**.
- ⇒ Between the vertex 1 and the right angle of the slab, insert the vertex 2 in the similar way. Coordinates for the vertex 2 are $X=13420$, $Y=4580$ (see Fig. 23.10).
- ⇒ To insert the vertex 3 - move the right angle of the slab to the vertex 3 with coordinates $X=19420$, $Y=4580$ (the procedure is the same).
- ⇒ To make the **Insert vertex** command not active, press **Esc**.
- ⇒ To unselect the slab, press **Esc**.

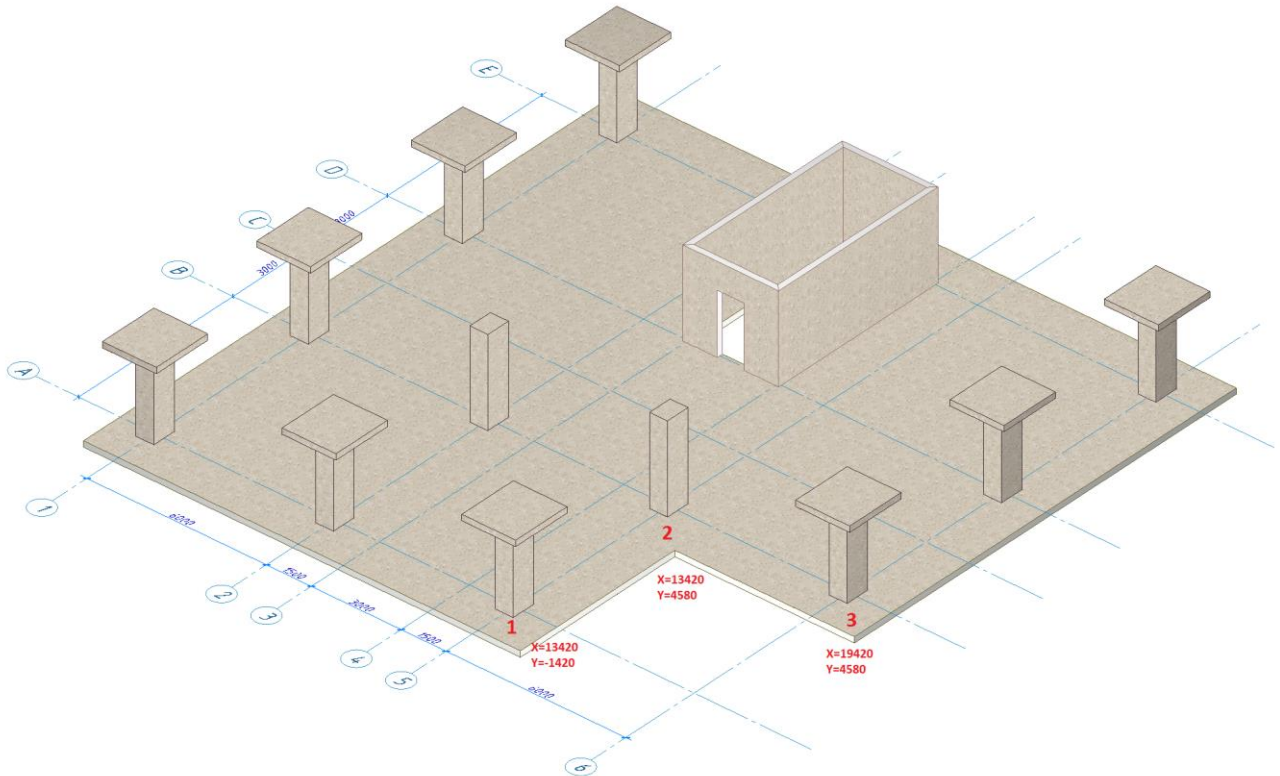





Figure 23.10. Modified contour of the floor slab

To create opening in the floor slab:

- ⇒ To present the model in the projection to the horizontal plane XOY, on the **Projections** toolbar, click the **Top view** button .
- ⇒ Select the floor slab.
- ⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, click the **Opening** button .
- ⇒ On the **Opening** Options Bar, select the method for generation as **Rectangle** .
- ⇒ To define the rectangular opening for the staircase in the stiffness core, specify in sequence two points - vertices of rectangle along diagonal (see Fig. 23.11).
- ⇒ To unselect the slab, press **Esc**.

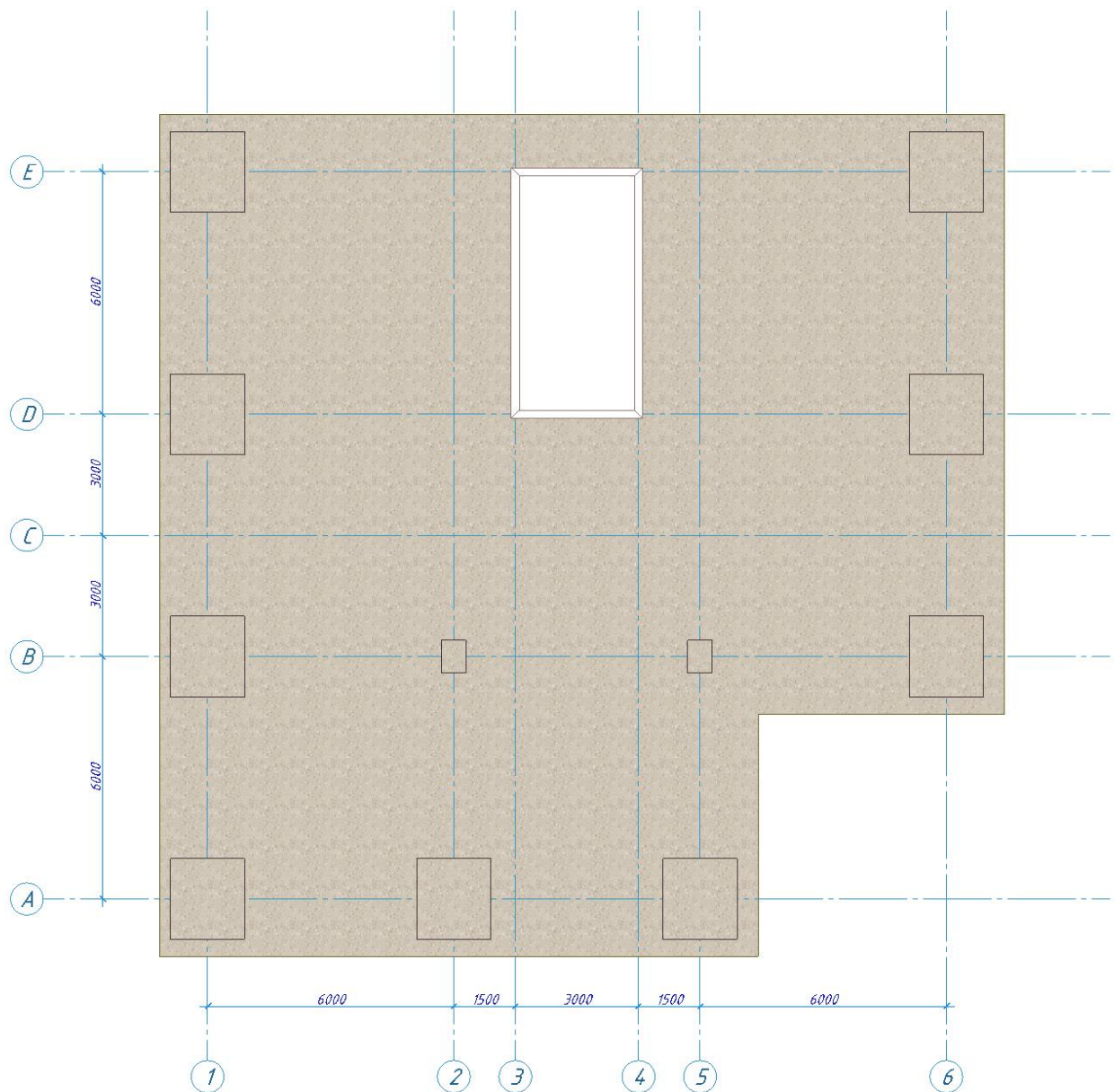


Figure 23.11. Location of the opening in the floor slab on the top view

To create zone of another thickness in the floor slab:



⇒ To present the model in the isometric projection, on the **Projections** toolbar, click the **Isometry** button



⇒ Select the floor slab.

⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, click the **± ΔH** button .

⇒ On the **± ΔH** Options Bar, define the following data:

- side  - back;
- method for generation  - **Rectangle**;
- depth, mm – (-200).

⇒ Define coordinates for the drop panel:

- the first point – (X=5250, Y=5250);

- the second point – (X=12750, Y=6750) (see Fig. 23.12)
- ⇒ To unselect the slab, press **Esc**.

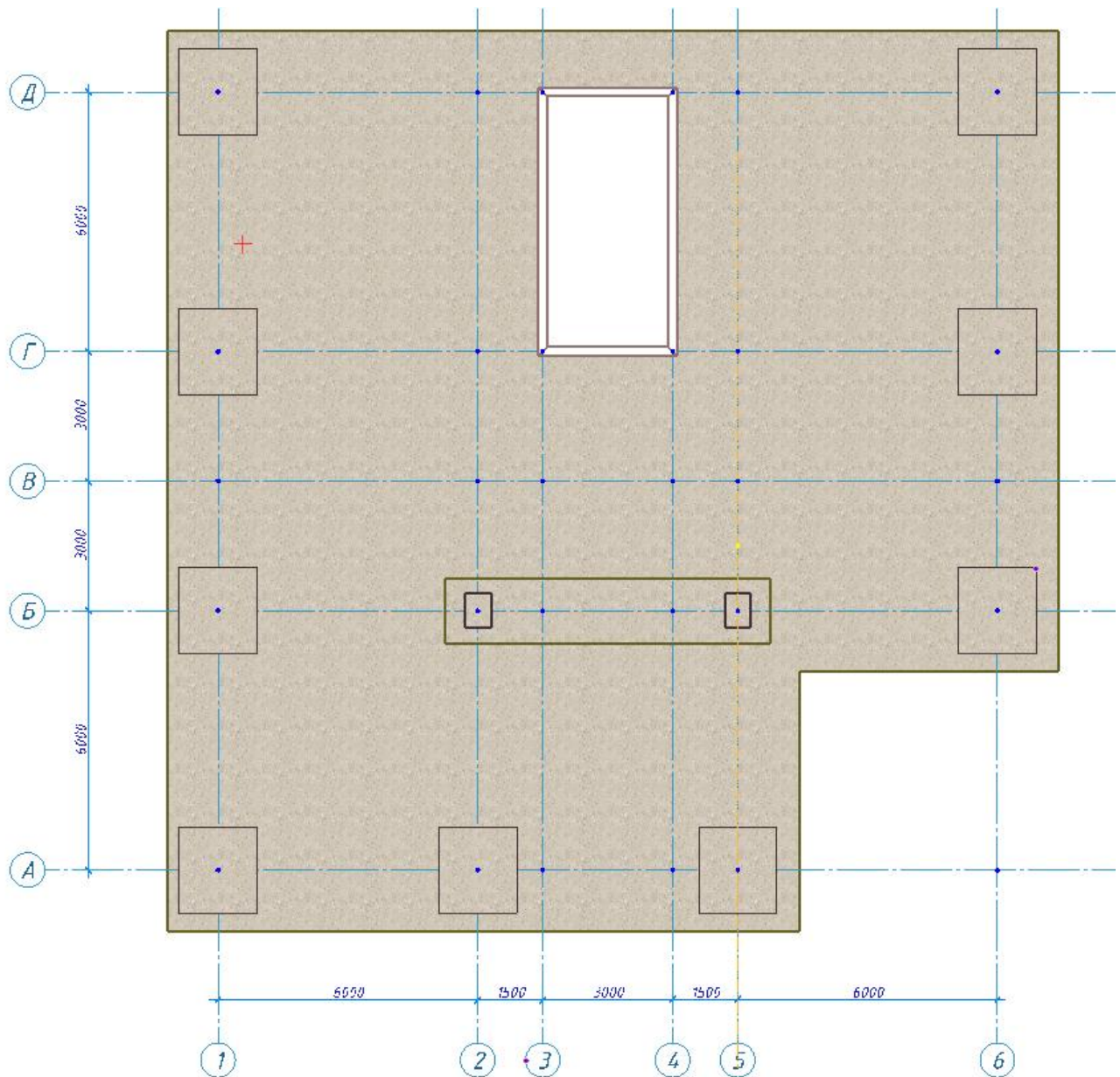




Figure 23.12. Location of the drop panel in the floor slab on the top view

Step 8. Copying storeys

To copy storeys:


- ⇒ On the **Create** ribbon tab, on the **Project** panel, click the **Storey** button .
- ⇒ In the **Create new storey** dialog box (see Fig. 23.13), define the following data:
 - number – 4;
 - storey height, mm – 4000;
 - select the **copy elements** option;
 - click the **Filter** button  and clear the **Designation** option in the filter for objects;

- in the **Filter for objects** dialog box, click **OK**;
- in the **Create new storey** dialog box, click **OK**.


Figure 23.13. **Create new storey** dialog box

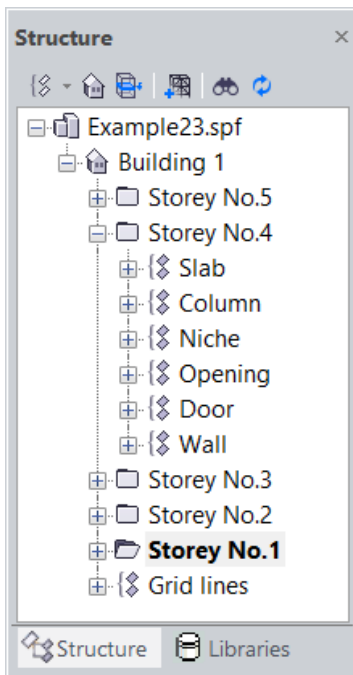







The program automatically checks the model for any error before the storeys are copied. In case of any error, you will see the warning message. It is recommended to correct all errors.

- ⇒ To present the whole model, on the **Projections** toolbar, click the **Fit in window** button  or double-click the wheel mouse button in the graphic area of the screen.

To modify the 4th and the 5th storeys:


- ⇒ In the **Structure** window (see Fig. 23.14), select the **Storey No.4** row  **Storey No.4** and unfold the list of elements of the 4th storey.



Figure 23.14. **Structure** window

- ⇒ Right-click the **Column** row.
- ⇒ To select all columns of the 4th storey, on the shortcut menu, click **Select**.
- ⇒ On the **Edit** ribbon tab, on the **Select objects** panel, click the **Select up** button  to select the columns of the 5th storey.
- ⇒ In the **Properties for 22 objects** window, define the following:
 - click the **Browse** button in the **Capital** row;
 - in the **Capital** dialog box, click **Delete** button.
- ⇒ To unselect columns, press **Esc**.
- ⇒ Rotate the camera (hold down the right mouse button) so that the drop panel in the floor slab of the 5th storey will be visible.
- ⇒ To select the drop panel in the floor slab of the 5th storey, click its edge.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, click the **Delete** button .
- ⇒ In the **Structure** window, make sure that **Storey No.5** is defined as the current one  **Storey No.5** (current storey is indicated with bold type).
- ⇒ Select the floor slab of the 5th storey.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, point to the **Copy** drop-down list and click the **Copy** button .
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, point to the **Paste** drop-down list and click the **Paste** button  to paste the copy of the slab to the 5th current storey.









Now there are two slab models at the same level within the same storey, one of them remains selected. As the element of the slab type is selected, then the **Slab Options Bar** is displayed.

- ⇒ On the **Slab Options Bar**, define the level  - 0 **from storey top**.
- ⇒ To unselect the slab, press **Esc**.

- ⇒ Use the wheel mouse button to bring the opening for the staircase (in the upper floor slab) near. To select the opening, just click its edge.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, click the **Delete** button .
- ⇒ To present the whole model, on the **Projections** toolbar, click the **Fit in window** button  or double-click the wheel mouse button in the graphic area of the screen.

To modify the foundation slab:

- ⇒ In the **Structure** window, right-click the **Storey No.1** row .
- ⇒ On the shortcut menu, click the **Set as current storey** command (or double-click the row ).
- ⇒ Click the **Active storey** button  on the **Visualization** toolbar (by default, it is located at the bottom of the screen).
- ⇒ Use the wheel mouse button to bring the slab near (at the bottom level of the storey 1).
- ⇒ To select the drop panel in the slab and the opening for the staircase, hold down the **Shift** key.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, click the **Delete** button .
- ⇒ Select the slab.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, point to **Convert objects to** list and click the **Foundation slabs** button .
- ⇒ On the **Slab Options Bar**, define the **thickness, mm**  - 600.
- ⇒ In the **Properties** window, under the **Boundary conditions**, define the following data:
 - C1, tf/m3 - 1000;
 - in the **Restraints** row, click the **Browse** button;
 - in the **Restraints** dialog box (see Fig. 23.15), select the check boxes for X and Y;
 - click **OK**.

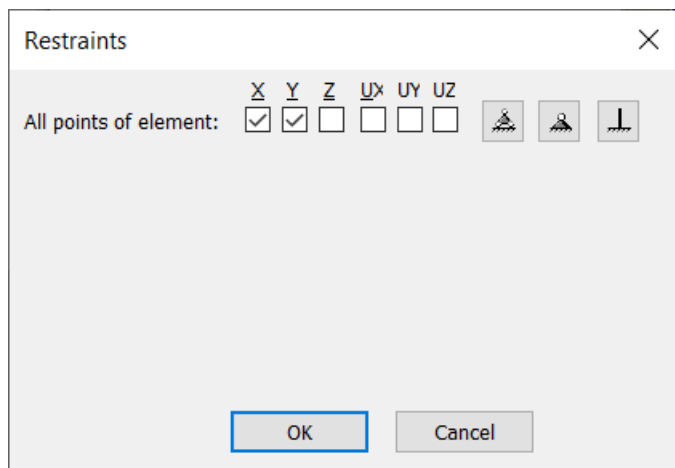


Figure 23.15. **Restraints** dialog box

- ⇒ Click the **Apply to object**  in the **Properties** window.
- ⇒ To unselect the foundation slab, press **Esc**.

Step 9. Simulating ramp

To modify location of UCS in the space:

- ⇒ Right-click in the graphic area of the screen and on the shortcut menu, click **Rotate UCS**.
- ⇒ In the **Enter rotation angle** dialog box, define rotation angle of the UCS as equal to 270 degrees.
- ⇒ To confirm the data and close the dialog box, click **OK**.
- ⇒ To move the UCS to the left bottom corner of the slab on the top view (see Fig. 23.16), place the locator over this point, right-click this point and on the shortcut menu, click the **To Point** command (**Ctrl+period** (.) on the numeric keypad).

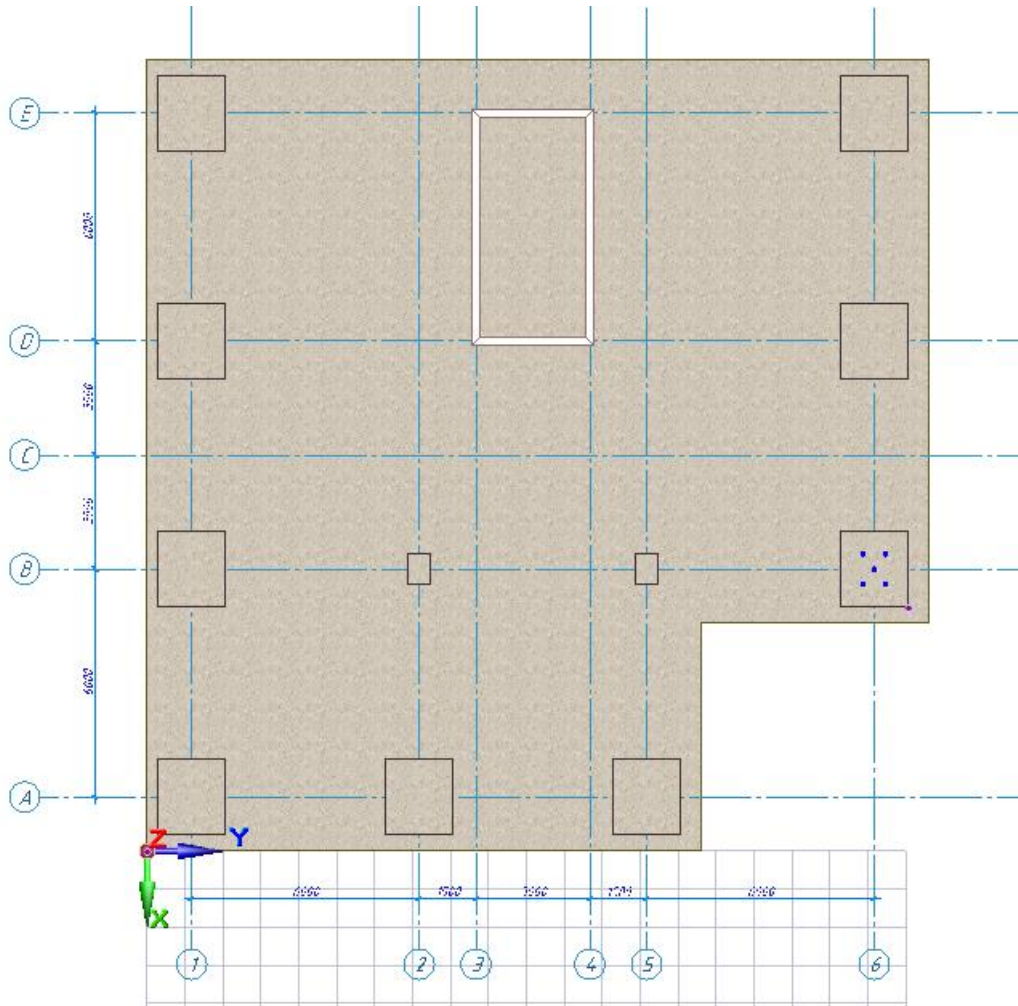








Figure 23.16. Location of the UCS reference mark

To generate the trajectory and generatrices for the ramp:

- ⇒ On the **Create** ribbon tab, on the **Sketching tools** panel, click the **Line** button .
- ⇒ On the **Line** Options Bar, define the following data:
 - method for generation  - line segment;
 - make sure that the **Chain** and **Close** options are clear;
 - line weight  - Basic6;
 - colour – red.
- ⇒ To define the 1st generatrix, use these points:

- initial point (X=0, Y=2000);
 - final point (X=0, Y=6000).
- ⇒ To define the 2nd generatrix, use these points:
- initial point (X=10000, Y=14000);
 - final point (X=6000, Y=14000).
- ⇒ On the **Line** Options Bar, define the following data:
- hold down the **Arc** button  up to the drop-down list appears on the screen;
 - from the drop-down list, select the method for generation  - **Arc P1 P3 P2**.
- ⇒ To generate the arc trajectory, use the following points:
- the 1st point of the trajectory (X=0, Y=2000) - initial point of the 1st generatrix;
 - the 2nd point of the trajectory (X=10000, Y=14000) - initial point of the 2nd generatrix;
 - the 3rd point of the trajectory (X=8000, Y=6000).
- ⇒ To complete the generation of lines, press **Esc** two times.
- ⇒ Select the 1st generatrix.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, point to the **Move** list and click the **Move by coordinates** button .
- ⇒ In the **Move objects** dialog box (see Fig. 23.17), define the following data:
- increment Z, mm – 4000.
- ⇒ Click **OK**.
- ⇒ To unselect the moved line, press **Esc**.

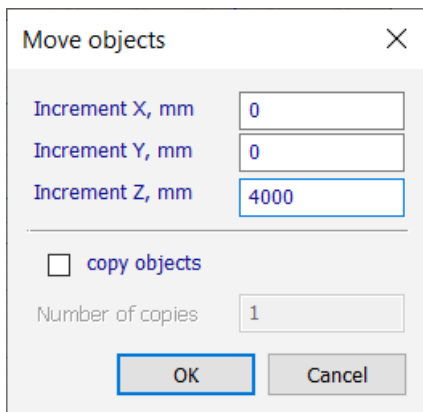






Figure 23.17. **Move objects** dialog box

To generate the ramp:

- ⇒ Select in sequence the trajectory, the 1st and the 2nd generatrix (the order is important). To do this, just hold down the **Shift** key.
- ⇒ On the **Create** ribbon tab, on the **Shapes** panel, point to the **3D by line** drop-down list and click the **1 trajectory 2 generatrices** button .
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, click the **Delete** button .
- ⇒ Select the generated shape.
- ⇒ In the **Properties** window, define the following data:
- material – RC slabs;
 - interpretation – Bearing;
 - number of parts in generatrix – 8;

- shell thickness, mm – 180;
 - normal relocation of surface, mm - -90.
- Click the **Apply to object**  button.
- To unselect the ramp, press **Esc**.
- Right-click in graphic window and on the shortcut menu, select UCS **To origin (0,0,0)**.
- Click the **Active storey** button  on the **Visualization** toolbar (by default, it is located at the bottom of the screen) to cancel the fragmentation of the storey.
- The building with the ramp should look like this (see Fig. 23.18).

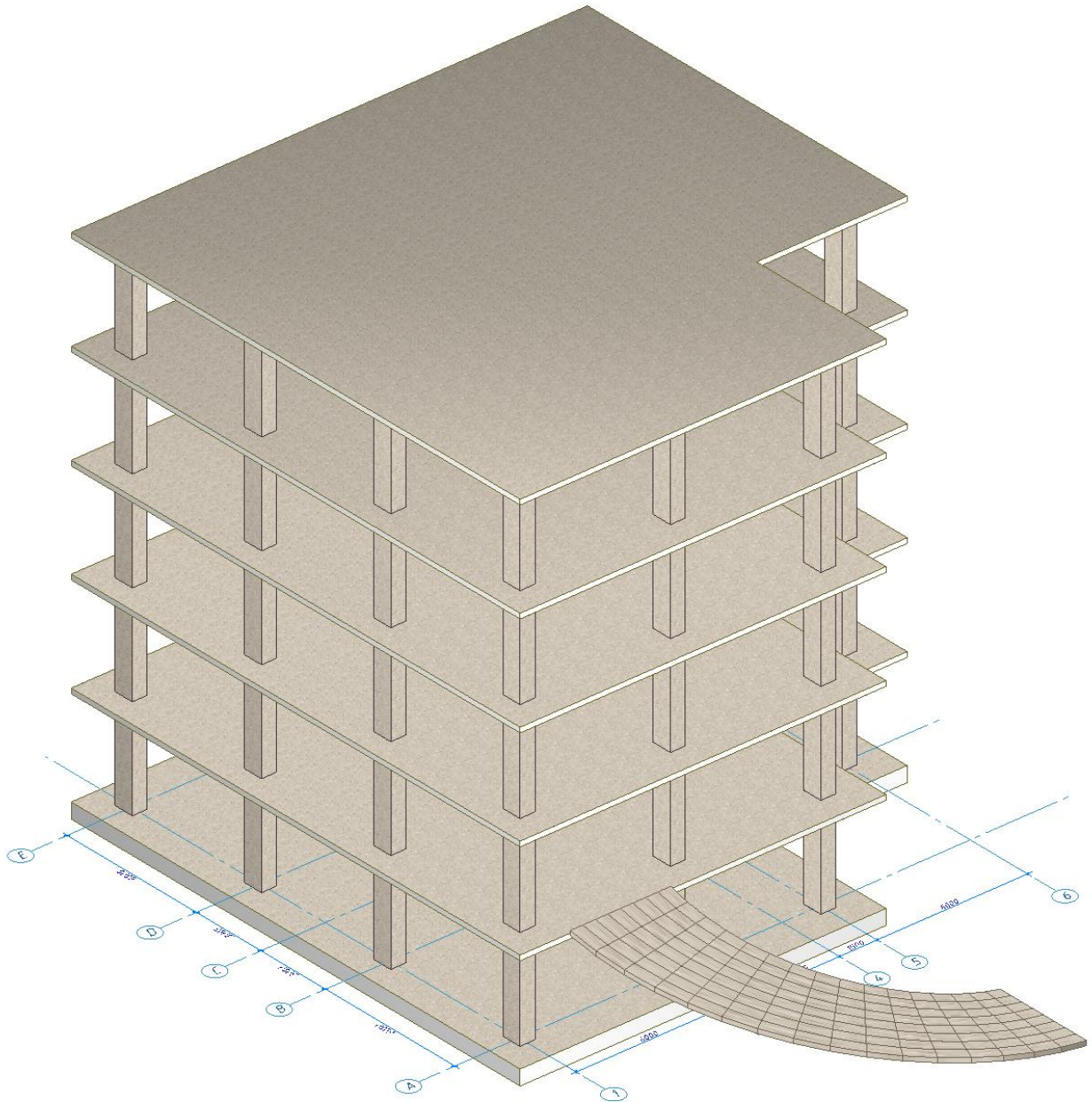




Figure 23.18. Building with ramp

Step 10. Simulation for erection of the structure (ASSEMBLAGE)

To generate assemblage events automatically:

- ⇒ On the **Create** ribbon tab, on the **Auto generation** panel, click the **Assemblage stages** button .
- ⇒ In the **Assemblage stages** dialog box (see Fig. 23.19), in the **Service** area, click the **Auto** button.
- ⇒ In the **Automatic generation of events** dialog box, select method for generation **By types of elements**.
- ⇒ Clear the **Stages by storeys** option.
- ⇒ Click the **Order of element types for generation of actions** button  to preview the order in which the elements should be erected.
- ⇒ Then click the **Generate** button (in the **Assemblage stages** dialog box you will see the chain of events - the order of erection for the structure).

To define assemblage stages manually:

- ⇒ To change the order of erection for the structure, in the **Assemblage stages** dialog box, drag the event **4,Other (Storey No.1)** to the right to reverse it with the event **5,Slab (bottom) (Storey No.2)**.
- ⇒ Select the event **5,Other (Storey No.1)** and select the **Stage** option for this event.
- ⇒ In the similar way, select the **Stage** option for the following events: **8,Slab (bottom) (Storey No.3)**; **11,Slab (bottom) (Storey No.)**; **14,Slab (bottom) (Storey No.5)**; **17,Slab (top) (Storey No.5)** (see Fig. 23.19).

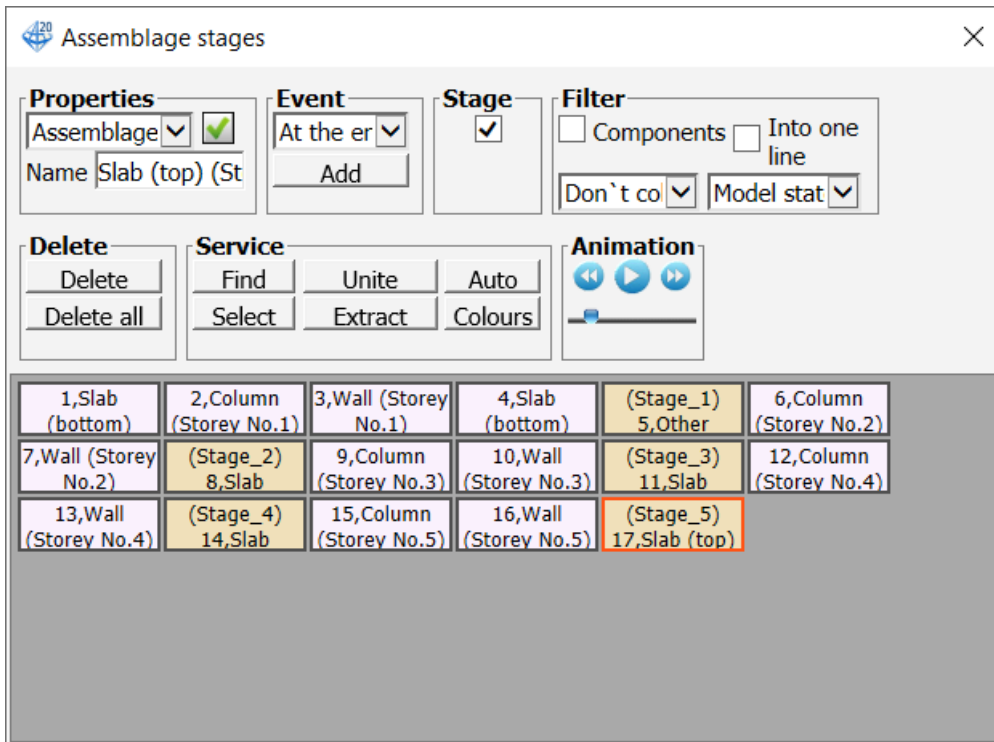


Figure 23.19. **Assemblage stages** dialog box

To preview animation of the erection for the structure:

- ⇒ In the **Assemblage stages** dialog box, select the first event of assemblage.
- ⇒ In the **Filter** area, from the appropriate drop-down list, select the **Model state at the moment of active event** option to visualize the objects included into the current event.
- ⇒ In the **Animation** area, move the slider slightly to the right to delay the rendering of the event.

- ➔ Click the **Play assemblage events** button to visualize the animation of events and stages (see Fig. 23.20; 23.21).
- ➔ Close the **Assemblage stages** dialog box.

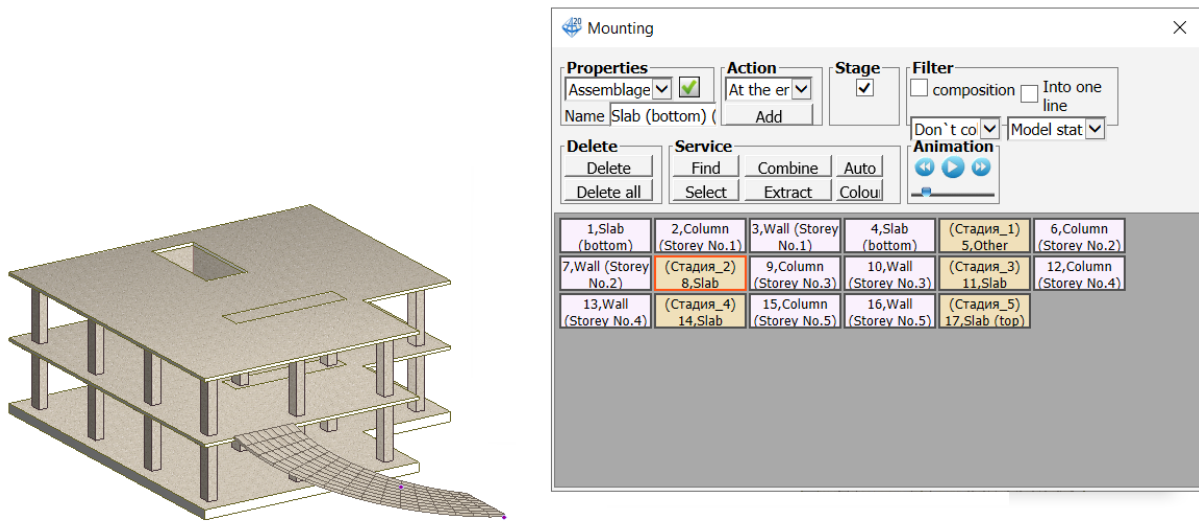


Figure 23.20. The second stage of assemblage

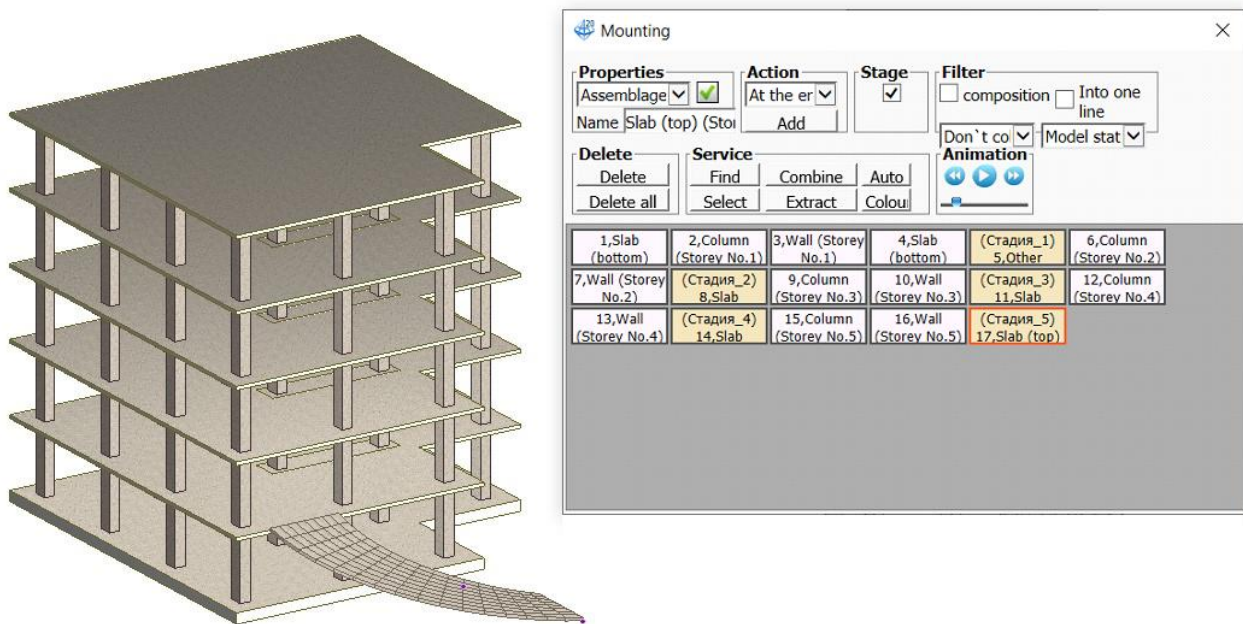


Figure 23.21. The last stage of assemblage






Step 11. Creating load cases and applying loads

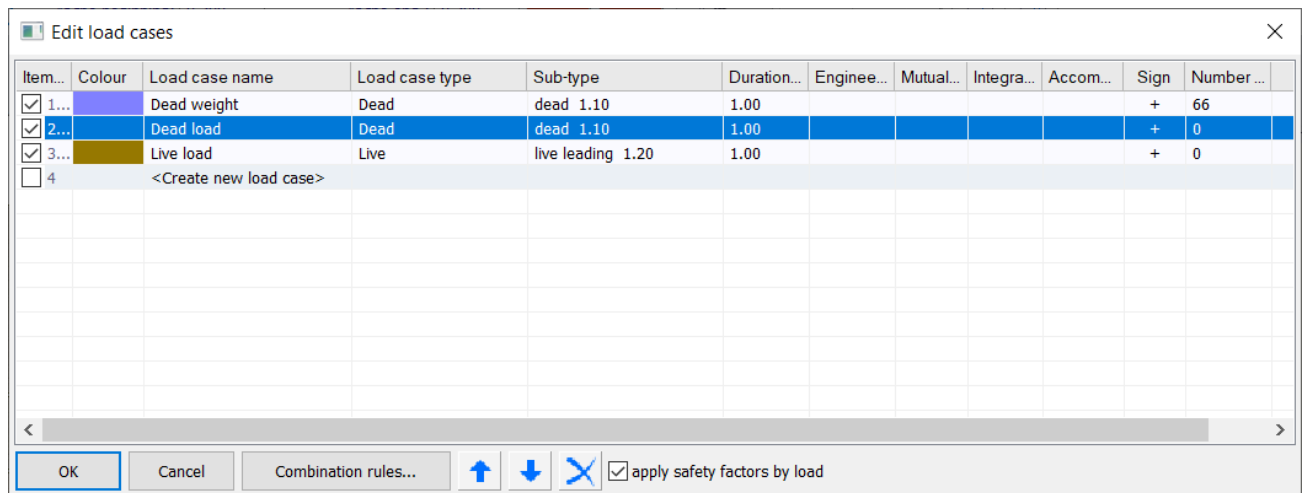
Load case No.1 (Dead weight)







In SAPFIR-3D program, the dead weight is generated automatically according to materials and sections defined for the objects.

Load case No.2 (Dead load)

- ⇒ On the **Create** ribbon tab, on the **Loads** panel, point to the **Loads** drop-down box and click the **Linear load** button .
- ⇒ On the **Load Options Bar**, define the following data:
 - method for generation  – Axial line;
 - for the load case option, click the **Browse** button .
 - in the **Edit load cases** dialog box, double-click the **Dead loads on slabs** row and rename it to the **Dead load**;
 - define the **Load case type** for it as **Dead**;
 - in the same way, rename the **Live loads on slabs** row to the **Live load**;
 - define the **Load case type** for it as **Live**;
 - hold down the **Ctrl** key and select the rows **Short-term loads on slabs** and **Load case other**;
 - click the **Delete** button .
 - select the **Dead load** row (make it as current one) (see Fig. 23.22);
 - click **OK**.
 - on the **Load Options Bar**, define the load values:
 - at the beginning – 1.6tf/m^2 ;
 - at the end – 1.6tf/m^2 ;
 - level  – from storey bottom.

Figure 23.22. **Edit load cases** dialog box

- ⇒ Click the edge of foundation slab to 'capture' its axial line (the light green contour is displayed along perimeter of the slab).
- ⇒ Press **Enter** to confirm the selected axial line.
- ⇒ On the **Load Options Bar**, select method of generation as **Line segment** .
- ⇒ Define linear load with the following coordinates:
 - the first point (X=3000, Y=7500);
 - the second point (X=3000, Y=13500);
 - to confirm defined coordinates, press **Enter**.

- ⇒ On the **Create** ribbon tab, on the **Loads** panel, point to the **Surface loads** drop-down box and click the **Surface load** button .
- ⇒ On the **Load Options Bar**, define the following data:
 - method for generation  – Axial line;
 - at the beginning – $0.3tf/m^2$;
 - at the end – $0.3tf/m^2$;
 - level  – from storey bottom.
- ⇒ Click the edge of foundation slab to 'capture' its axial line (the light green contour is displayed along perimeter of the slab). Press **Enter** to confirm the selected axial line.
- ⇒ To complete the generation of load, press **Esc**.

To edit contour of linear load:

- ⇒ Select linear load applied along the edge of foundation.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, click the **Equidistance** button .
- ⇒ In the **Equidistant relocation** dialog box (see Fig. 23.23), define the indent value – **-100mm**.
- ⇒ Click **OK**.
- ⇒ To complete the generation of load, press **Esc**.

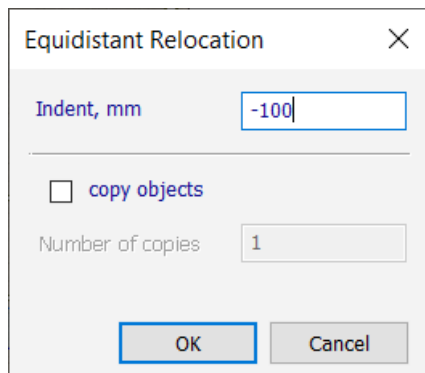







Figure 23.23. **Equidistant relocation** dialog box

Load case No.3 (Live load)

- ⇒ On the **Create** ribbon tab, on the **Loads** panel, point to the **Surface loads** drop-down box and click the **Surface load** button .
- ⇒ On the **Load Options Bar**, define the following data:
 - load case – Live load;
 - at the beginning – $0.5tf/m^2$;
 - at the end – $0.5tf/m^2$;
 - colour – green;
 - level  - from storey bottom.
- ⇒ Click the edge of foundation slab to 'capture' its axial line (the light green contour is displayed along perimeter of the slab). Press **Enter** to confirm the selected axial line.
- ⇒ On the **Load Options Bar**, define the following data:
 - method for generation  – Rectangle;
 - at the beginning – $2tf/m^2$;

- at the end – $2tf/m^2$;
 - colour – red;
 - level  - from storey bottom.
- ⇒ Generate the surface load by the following points:
- 1st point of diagonal (X=13000, Y=11000);
 - 2nd point of diagonal (X=15500, Y=15000);
 - to confirm defined coordinates, press **Enter**.
- ⇒ To complete the generation of load, press **Esc**.

To multicopy loads by storeys:

- ⇒ On the **Visualization** toolbar, click the **Filter for elements** button .
- ⇒ In the **Filter for elements** dialog box (see Fig. 23.24), select the **Load** check box.
- ⇒ Click the **Select** button.

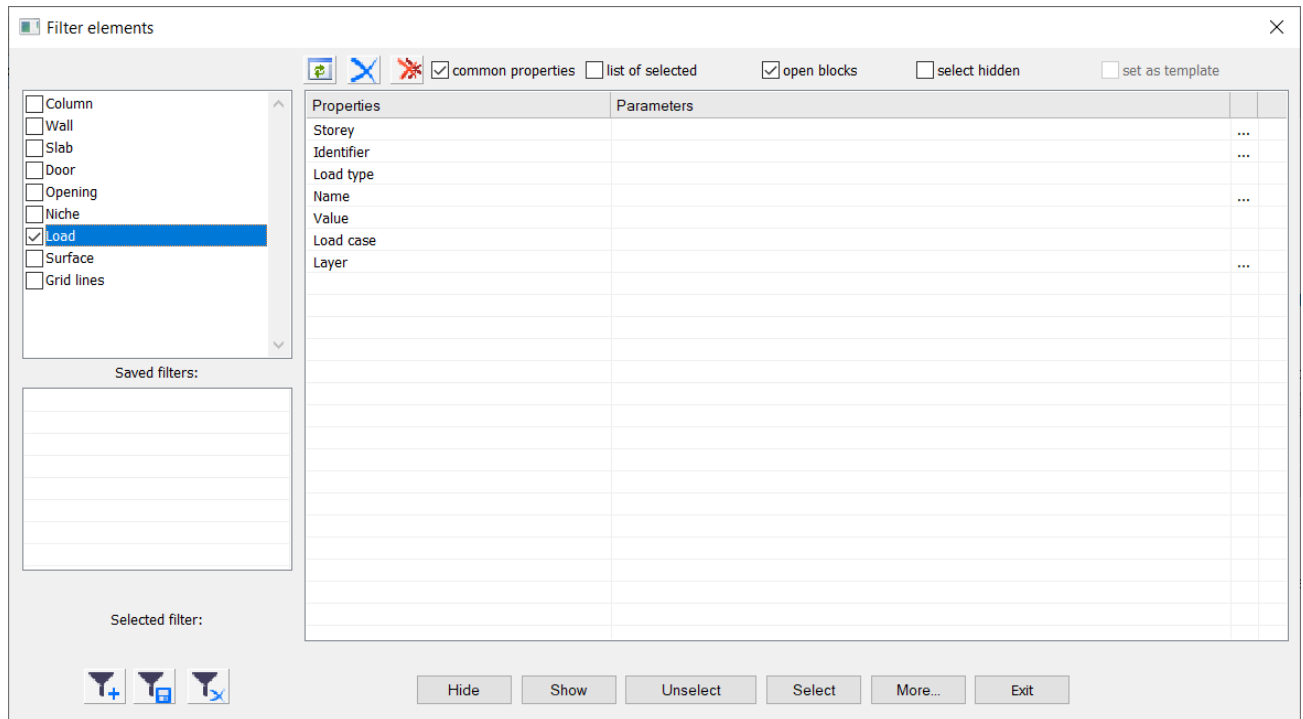



Figure 23.24. **Filter for elements** dialog box

- ⇒ To close the dialog box, use appropriate button in the upper right corner of the dialog box.
- ⇒ On the **Edit** ribbon tab, on the **Modify** panel, point to the **Paste** list and click the **Paste to selected storeys** button .
- ⇒ In the **Select storeys** dialog box (see Fig. 23.25), make sure that storeys 2, 3, 4 and 5 are selected.
- ⇒ Click **OK**.
- ⇒ To unselect the load, press **Esc**.

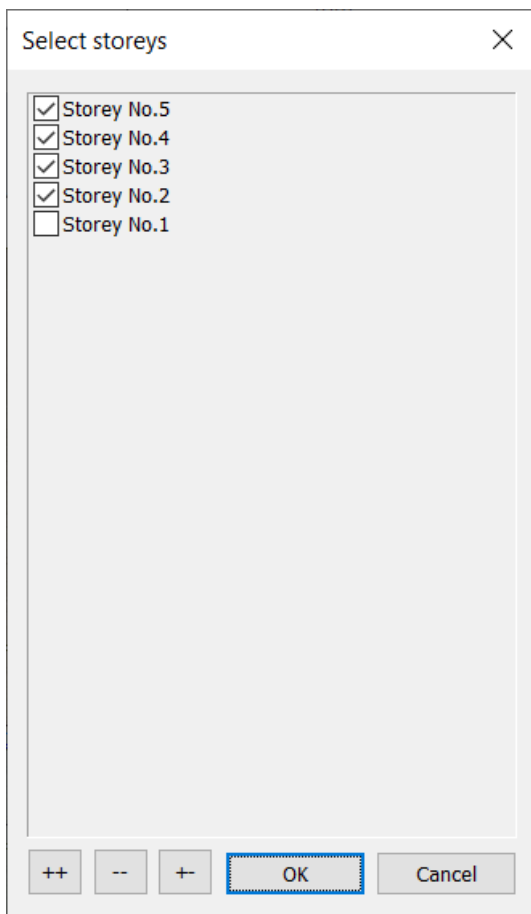







Figure 23.25. **Select storeys** dialog box

To define the load from the roof:

- ⇒ In the **Structure** window, double-click the Storey No.5 row  **Storey No.5** to set the storey as the current one.
- ⇒ On the **Create** ribbon tab, on the **Loads** panel, point to the **Surface loads** drop-down box and click the **Surface load** button .
- ⇒ On the **Load Options Bar**, define the following data:
 - method for generation  – Axial line;
 - load case – Dead load;
 - at the beginning – 0.1tf/m^2 ;
 - at the end – 0.1tf/m^2 ;
 - colour – blue;
 - level  - from storey top.
- ⇒ Click the edge of roof slab to 'capture' its axial line (the light green contour is displayed along perimeter of the slab). Press **Enter** to confirm the selected axial line.
- ⇒ To complete the generation of load, press **Esc**.





To visualize the load separately for every load case, on the Visualization toolbar, click the **Filter by load cases**  button. To change the current load case, use appropriate drop-down list.

Step 12. Editing material properties for elements of the model



When you generate objects in **SAPFIR**, materials are assigned by default. For walls it is RC walls, for columns – RC columns, for slabs – RC slabs, for foundation slabs RC foundation slab. Each of these materials already has design parameters accepted by default (type of reinforcement, design parameters of concrete and design parameters of reinforcement). If required, in **SAPFIR** program it is possible to create new design parameters for material or edit default materials. When the meshed model is transferred to **VISOR-SAPR** module, you could also edit materials with **VISOR-SAPR** tools.

- ⇒ On the **View** ribbon tab, on the **Settings** panel, click the **Materials** button .
- ⇒ In the **Materials** dialog box (see Fig.23.26), select **RC Columns** from the list and in the right part of the dialog box click the **Browse** button  in the building code row.

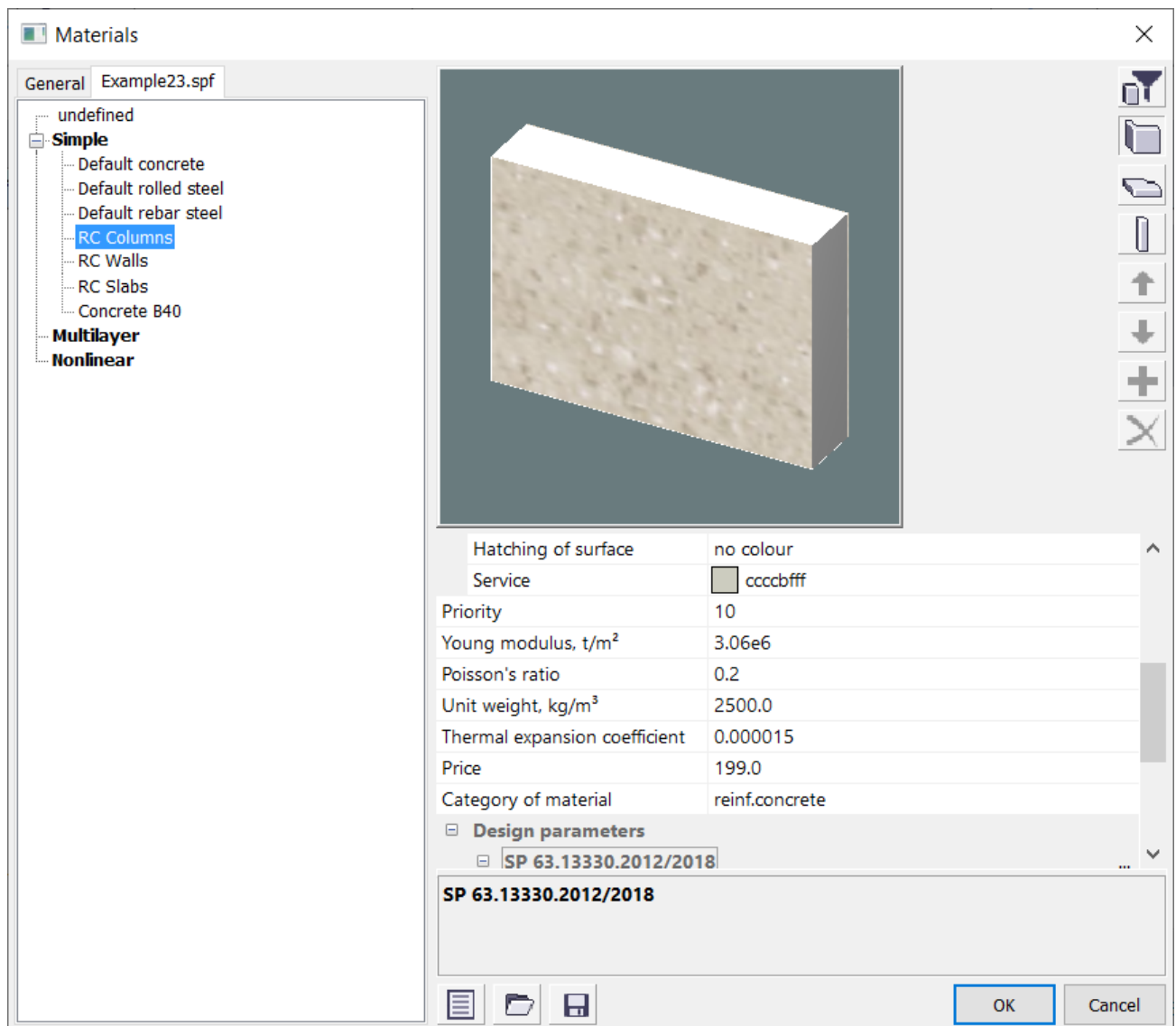


Figure 23.26. **Materials** dialog box

- ⇒ In the **Material properties for analysis of RC structures** dialog box (see Fig. 23.27), for the type of reinforcement **Bar**, select the first row **Column** and in the right part of the dialog box define the following parameters for columns:
 - in the **Analysis type** list, select **Column regular**;


- in the **Analysis by serviceability limit states (SLS)** area, select the **Diameter of rebars** option and define diameter of rebars equal to **25 mm** in the drop-down list;
 - in the **Length of element, Effective lengths** area, edit effective length factors for $LY = 0.7$, $LZ = 0.7$;
 - other parameters should remain by default.
- ⇒ In the **Type: Plate** table, select the first row **Wall**. In the right part of the dialog box, in the **Distance to gravity centre of reinforcement** area, define the values for A1X and A2X – 4cm, for A1Y and A2Y – 3cm. Other parameters should remain by default.
- ⇒ Define the same parameters for the type of reinforcement **Slab**.
- ⇒ Properties for concrete and reinforcement remain by default.
- ⇒ Click **OK** .
- ⇒ Then, in the **Materials** dialog box, for the **RC Columns** material define the following parameters:
- in the **General parameters** drop-down list, select **(1)Column Column regular**;
 - in the **Concrete** drop-down list, select **(1)Vert B25**;
 - in the **Reinforcement** drop-down list, select **(1)Vert A400.A400.A240**.

Figure 23.27. Material properties for analysis of RC structures dialog box

- ⇒ In the **Materials** dialog box, for the **RC Walls** material define the following parameters:
- in the **General parameters** drop-down list, select **(2)Wall SHELL (Flexure, Compression/Tension)**;
 - in the **Concrete** drop-down list, select **(1)Vert B25**;
 - in the **Reinforcement** drop-down list, select **(1)Vert A400.A400.A240**.
- ⇒ In the **Materials** dialog box, for the **RC Slabs** material define the following parameters:
- in the **General parameters** drop-down list, select **(3)Slab SHELL (Flexure, Compression/Tension)**;
 - in the **Concrete** drop-down list, select **(2)Horiz B20**;
 - in the **Reinforcement** drop-down list, select **(2)Horiz A400.A400.A240**.
- ⇒ In the **Materials** dialog box, click **OK**.

Step 13. Creating meshed model in SAPFIR-Structures module

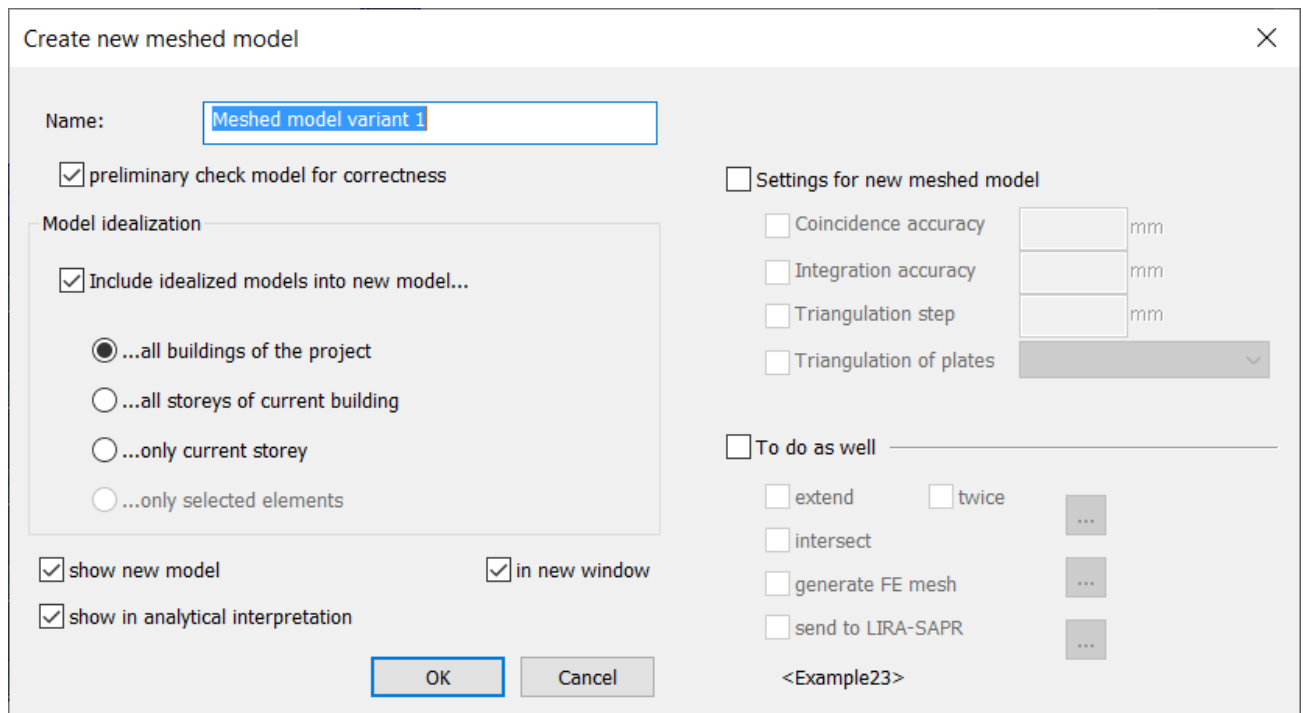
To create the meshed model:

- ⇒ On the **Analytics** ribbon tab, click the **Meshed model** button .



The **Meshed model** button switches the program from the mode where you generate the meshed model to the tools of the meshed model and vice versa.

- ⇒ In the **Create new meshed model** dialog box (see Fig. 23.28), click **OK** (the new tab will be displayed on the screen, the tab name is *Example23.spf:Meshed model*).



The dialog box titled "Create new meshed model" contains the following fields and options:

- Name:** A text field containing "Meshed model variant 1".
- ☒ preliminary check model for correctness
- Model idealization**
 - ☒ Include idealized models into new model...
 - ☒ ...all buildings of the project
 - ☐ ...all storeys of current building
 - ☐ ...only current storey
 - ☐ ...only selected elements
- ☒ show new model ☒ in new window
- ☒ show in analytical interpretation
- ☐ Settings for new meshed model
 - ☐ Coincidence accuracy mm
 - ☐ Integration accuracy mm
 - ☐ Triangulation step mm
 - ☐ Triangulation of plates
- ☐ To do as well
 - ☐ extend ☐ twice
 - ☐ intersect
 - ☐ generate FE mesh
 - ☐ send to LIRA-SAPR


Buttons: **OK**, **Cancel**. Footer: <Example23>

Figure 23.28. **Create new meshed model** dialog box



The program automatically checks the model for any error before the meshed model is generated. In case of any error, you will see the warning message. It is recommended to correct all errors.

To edit properties of the meshed model:

- ⇒ On the **Analytics** ribbon tab, on the **Meshed model: generation** panel, click the **Parameters of the meshed model** button .
- ⇒ In the **Parameters** dialog box (see Fig. 23.29), in the **Settings for intersections** row, select the **Real volumes** from the drop-down list.
- ⇒ Click **OK**.



By default, SAPFIR program accepts analytics of slabs by the slab top. In case you defined analytics for slabs At the middle of the slab, in Properties of the meshed model you should modify the search parameter *L*. *L* search, mm is the distance along or against the load vector; the load may be moved for this distance to search for the elements that it is applied to. By default, *L* search is equal to 250mm. For

this parameter it is necessary to define the value that is greater than half thickness of the thickest slab, as a rule it is greater than the half thickness of the foundation slab. Let's suppose that in our model, foundation slab has thickness 600mm, half its thickness is 300mm. Parameter L to search for load may be defined as equal to 320...350mm.

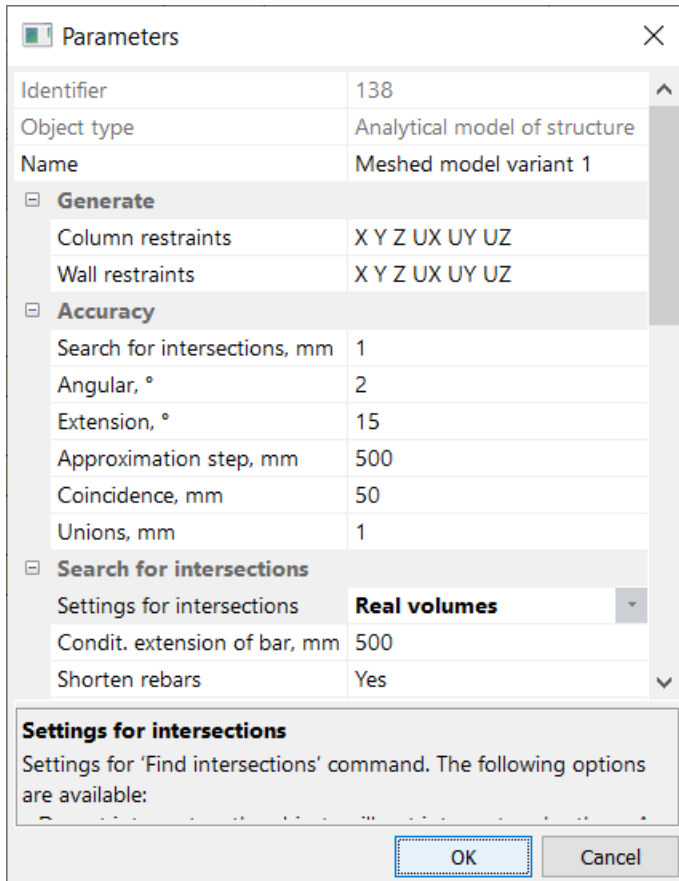



Figure 23.29. Parameters dialog box

To idealize the model:

- ⇒ On the **Analytics** ribbon tab, on the **Meshed model: triangulation** panel, point to the **Extend** drop-down list and click the **Extend twice** button  in order to search for intersections correctly and to eliminate minor architectural inaccuracies.
- ⇒ In the SAPFIR message box (see Fig. 23.30), click **Yes**.

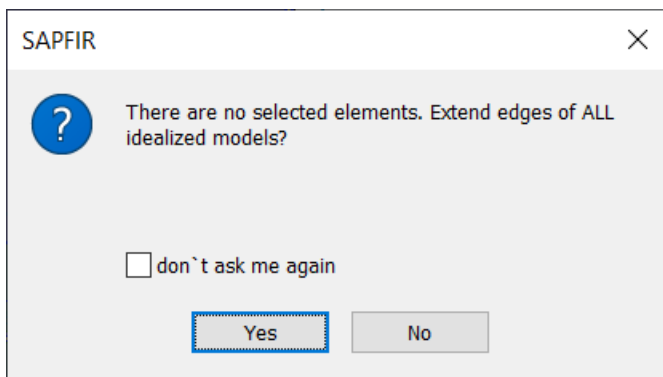



Figure 23.30. SAPFIR message box

- ⇒ On the **Analytics** ribbon tab, on the **Meshed model: triangulation** panel, point to the **Intersection** drop-down list and click the **Generate intersections** button .
- ⇒ In the SAPFIR message box (see Fig. 23.31), click **Yes**.

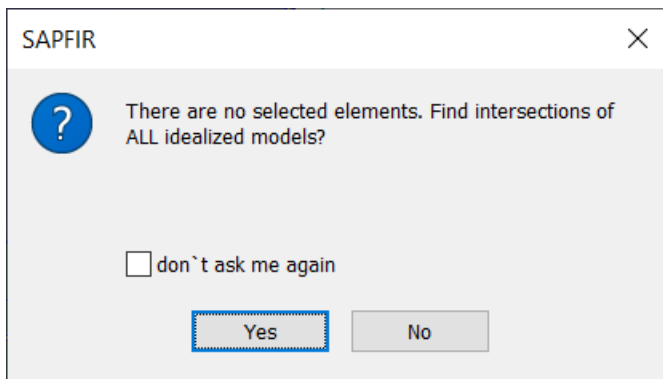


Figure 23.31. SAPFIR message box

- ⇒ Meshed model with generated intersections will look like the image below (see Fig. 23.32).

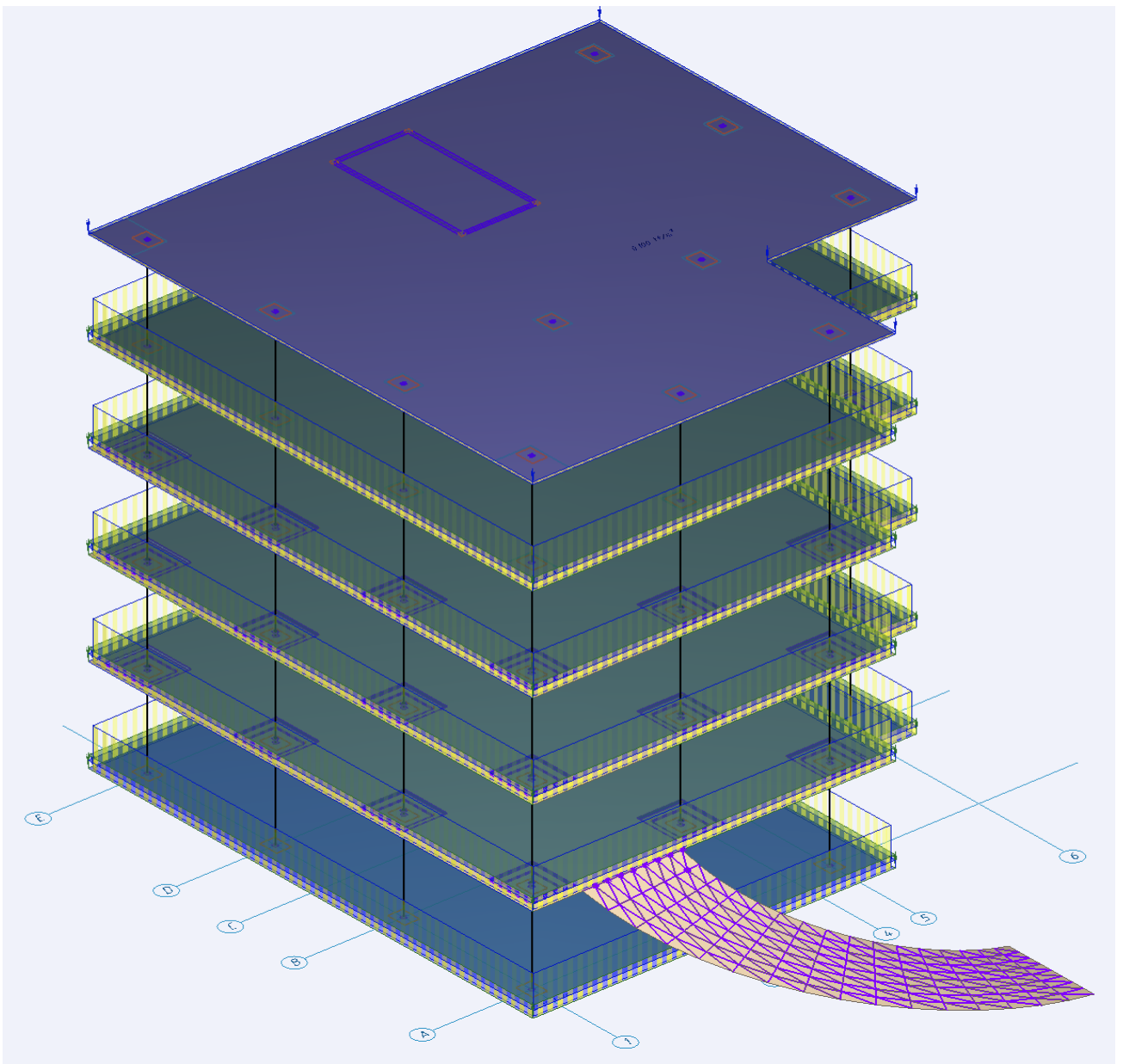






Figure 23.32. Meshed model with generated intersections

To triangulate the model:

- ⇒ On the **Visualization** toolbar, click the **Filter by load cases** button .
- ⇒ In the **Load cases** drop-down list, select the **Dead weight** as the current load case in order to temporarily hide the visualization of loads.
- ⇒ On the **Analytics** ribbon tab, on the **Meshed model: triangulation** panel, click the **Options** button .
- ⇒ In the **Triangulation settings** dialog box (see Fig.23.33), modify **Step, m** as equal to 0.6.
- ⇒ Click the **Apply** button.
- ⇒ On the **Projections** toolbar, click the **Left view** button .
- ⇒ With selection window, select (from the left to the right) all elements of the model except the ramp.
- ⇒ On the **Analytics** ribbon tab, on the **Meshed model: triangulation** panel, point to the **FE mesh** drop-down list and click the **Generate FE mesh** button  to divide into FE.

⇒ To unselect the elements, press **Esc**.

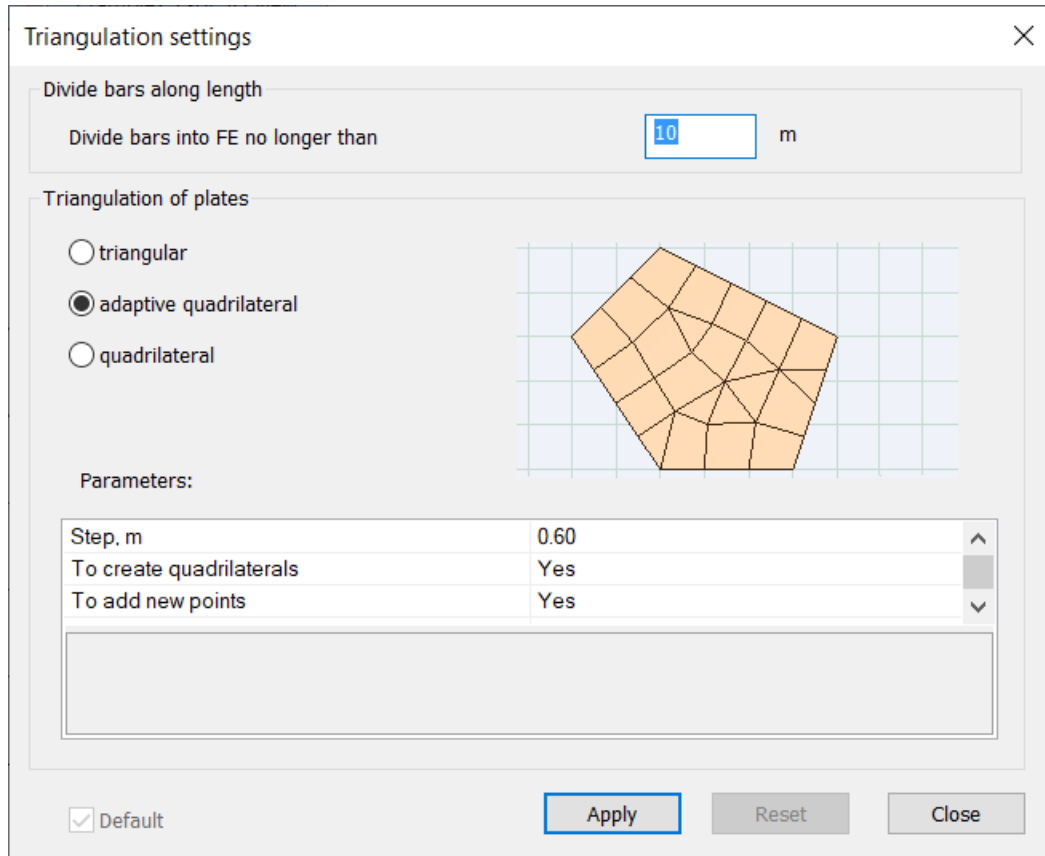




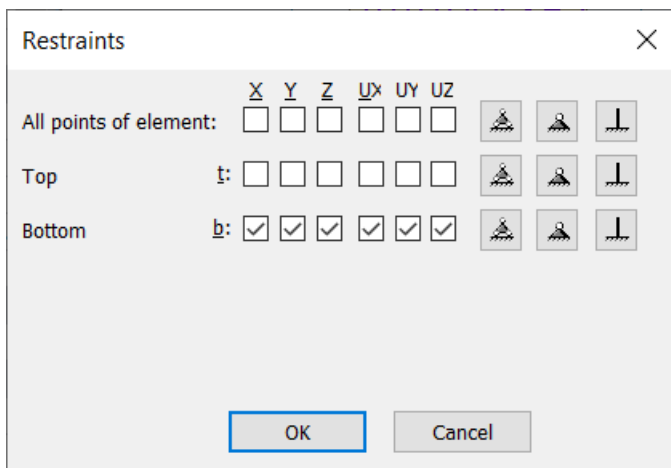
Figure 23.33. **Triangulation settings** dialog box



When the 3D plane is generated, the program automatically divides the plane. Then it is not necessary to triangulate the ramp.


To define the boundary conditions to elements of the ramp:

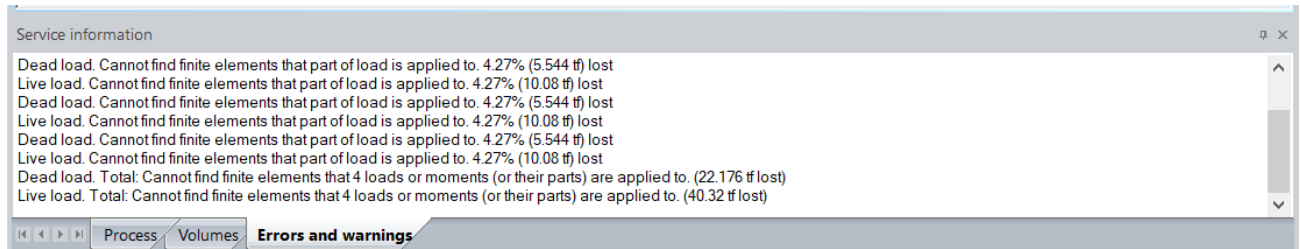
- ⇒ To switch to the XOY-projection, on the **Projections** toolbar, click the **Top view** button .
- ⇒ With selection window, select (from the right to the left) the lower row of elements in the ramp.
- ⇒ In the **Properties of 16 objects** window, in the **Restraints** row click the **Browse** button.
- ⇒ In the **Restraints** dialog box (see Fig. 23.34), define the following data:
 - for the **Bottom** group, click the  button;
 - click **OK**.

Figure 23.34. **Restraints** dialog box

⇒ To unselect the elements of the ramp, press **Esc**.

Step 14. Create and open file in LIRA-SAPR

- ⇒ To open the FE model in **LIRA-SAPR** program, on the **Analytics** tab, on the **Analysis in LIRA-SAPR** panel, point to the appropriate drop-down list and click the **Open in LIRA-SAPR** button .
- ⇒ In the **Service information** window (see Fig. 23.35), there is the message: 'Cannot find finite elements that part of load is applied to. 4.27% lost.' (the lost part of load is located above the opening of the staircase).


Figure 23.35. **Service information** window

The *.s2l file will be generated in the folder where Example23.spf is located. This *.s2l file will be opened in the VISOR-SAPR module.

Step 15. Unifying local axes of plates



SAPFIR module automatically unifies axes for floor slabs and diaphragms.

- ⇒ To check direction of local axes, on the **Select** toolbar (by default, it is displayed at the bottom of the screen), click the **Flags of drawing** button .
- ⇒ In the **Display** dialog box (see Fig.23.36), select the **Unified local axes of plates and solids (for results)** check box on the **Elements** tab (the first tab).
- ⇒ For all horizontal FE and elements of the ramp, local axes should be directed according to the global coordinate system. In elements of walls, local Y-axis (blue arrow) should be directed upward.

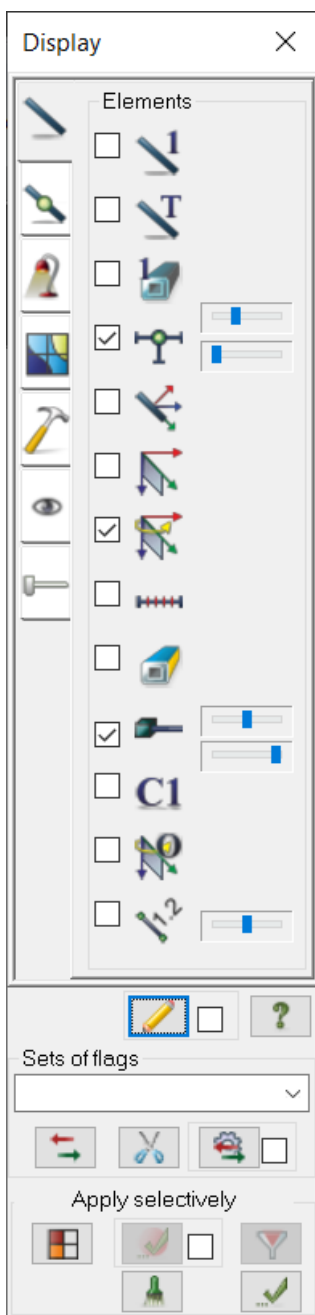




Figure 23.36. Display dialog box

Step 16. Editing assemblage table

- ⇒ On the **Analysis** ribbon tab, on the **Assemblage** panel, click **Stages** button .
- ⇒ In the **Model nonlinear load cases of structure** dialog box (see Fig.23.37), select the **Additional load cases** tab.
- ⇒ In the **History** area of the dialog box, select the row for the assemblage stage 5.
- ⇒ In the **Coefficients for account of additional load cases** table, for the 5.Stage 5, define coefficient for the 6th load case as equal to 1 and coefficient for the 7th load case as equal to 1.
- ⇒ Click **OK** .

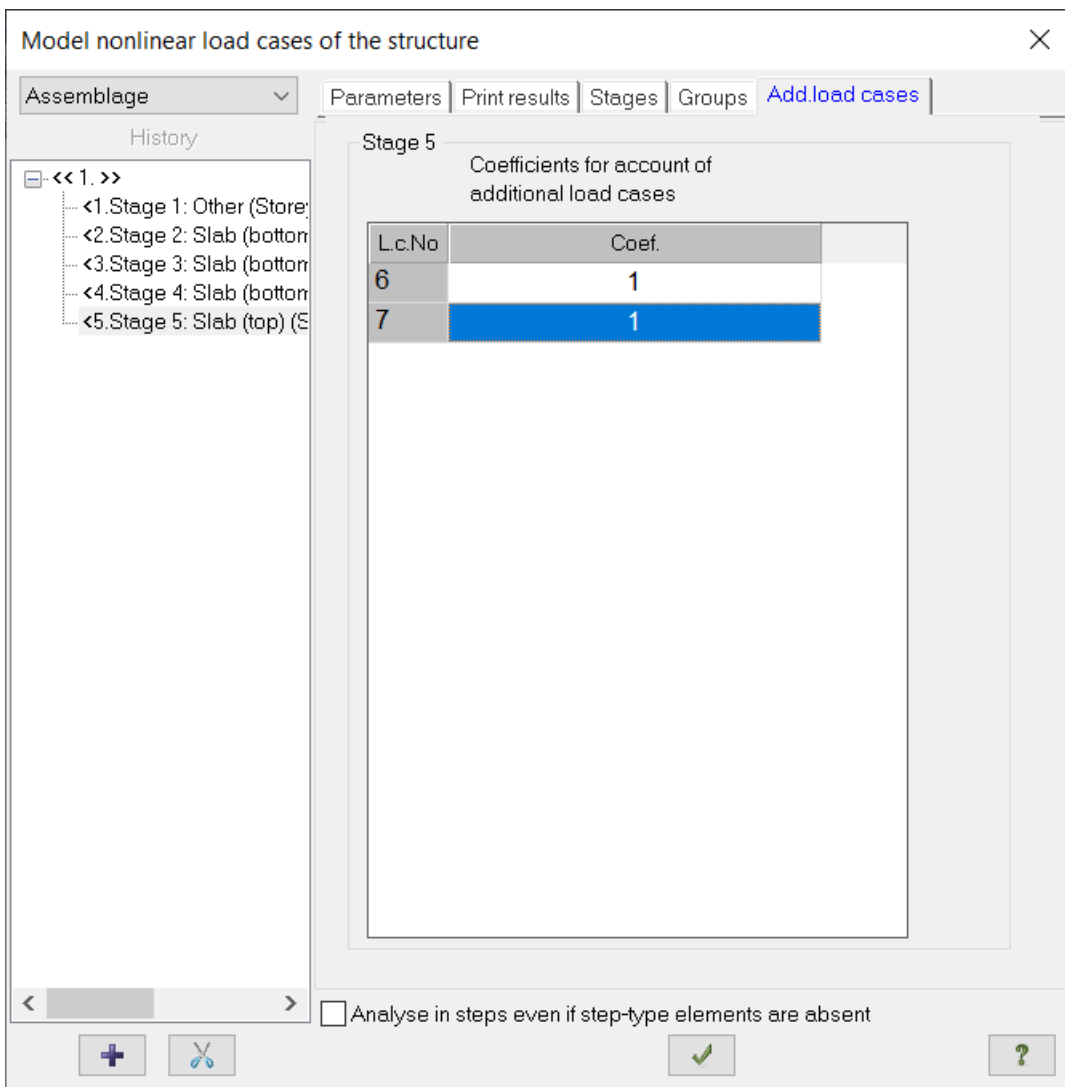


Figure 23.37. Model nonlinear load cases of structure dialog box

Step 17. Generating DCL table

⇒ On the **Analysis** ribbon tab, select the **More calculations** panel and click **DCL** button .



The DCL table is generated automatically from SAPFIR module, that is, all parameters may be defined in SAPFIR and they will be saved when transferred to VISOR-SAPR module. Note that load cases No. 6 and 7 become inactive as we consider them at the last stage of assemblage (load case 5).

⇒ Close the **Design combinations of loads** dialog box (see Fig. 23.38). Use appropriate button at the upper right corner of the dialog box.

Design combinations of loads

No. of DCL table: 1

Name of DCL table: Import from SAPFIR:SP 20.13330.2016 (RF) (

SP 20.13330.2011/2016

☐ Dynamics by absolute value

☐ Characteristic DCL

Safety factor for:

ULS: 1

SLS: 1

☐ Ignore earthquake loads in SLS

☒ Ignore specific loads in SLS

Load case No.	Name	Type	Sign variable	Mutually exclusive	Load factor	Duration co
1	Stage 1: Other (Storey No.1)	Dead (P)	+	1	1.1	1.0
2	Stage 2: Slab (bottom) (Storey No.3)	Dead (P)	+	1	1.1	1.0
3	Stage 3: Slab (bottom) (Storey No.4)	Dead (P)	+	1	1.1	1.0
4	Stage 4: Slab (bottom) (Storey No.5)	Dead (P)	+	1	1.1	1.0
5	Stage 5: Slab (top) (Storey No.5) (φи	Dead (P)	+	1	1.1	1.0
6	Dead load	Inactive (N/a)	+		1.1	1.0
7	Live load	Inactive (N/a)	+		1.2	1.0

Fundamental combination

Specific combination


$$p^d + \psi_{11} \cdot p_{11}^d + \sum_{i=2}^{nl} \psi_{1i} \cdot p_{1i}^d + \psi_{t1} \cdot p_{t1}^d + \psi_{t2} \cdot p_{t2}^d + \sum_{j=3}^{nt} \psi_{tj} \cdot p_{tj}^d$$

Coefficients

Add

Figure 23.38. Design combinations of loads dialog box


Step 18. Carrying out complete analysis of the model

- ⇒ To carry out complete analysis of the model, on the **Analysis** ribbon tab, select the **Analysis** panel and in the **Analyse** drop-down list, click **Complete analysis** .




Step 19. Review and evaluation of static analysis results




When analysis procedure is complete, to review and evaluate results of static and dynamic analyses, select the **Results** ribbon tab.


- ⇒ In the mode of analysis results visualization, by default design model is presented with account of nodal displacements for the first stage of assemblage. To display the model without nodal displacements, on the **Results** ribbon tab, on the **Deformations** panel, click **Initial model** button .

To present diagrams of internal forces:


- ⇒ To display diagram **M_y**, on the **Results** tab, select **Forces in bars** panel and click **Moment diagrams (M_y)** button .
- ⇒ To display diagram **Q_z**, on the **Results** tab, select **Forces in bars** panel and click **Diagrams Q_z** button .
- ⇒ To display diagram **N**, on the **Results** tab, select **Forces in bars** panel and click **Axial force diagrams (N)** button .

- ⇒ To display mosaic plots **N**, on the **Results** tab, select **Forces in bars** panel and click **Mosaic plot of forces in bars** command  in the **Force diagrams/Mosaic plots** drop-down list.




To change number of active load case:

- ⇒ On the status bar (displayed at the bottom of the screen), in the **Load case No.** list, select No. **2** and click **Apply** .


To present displacement contour plots:

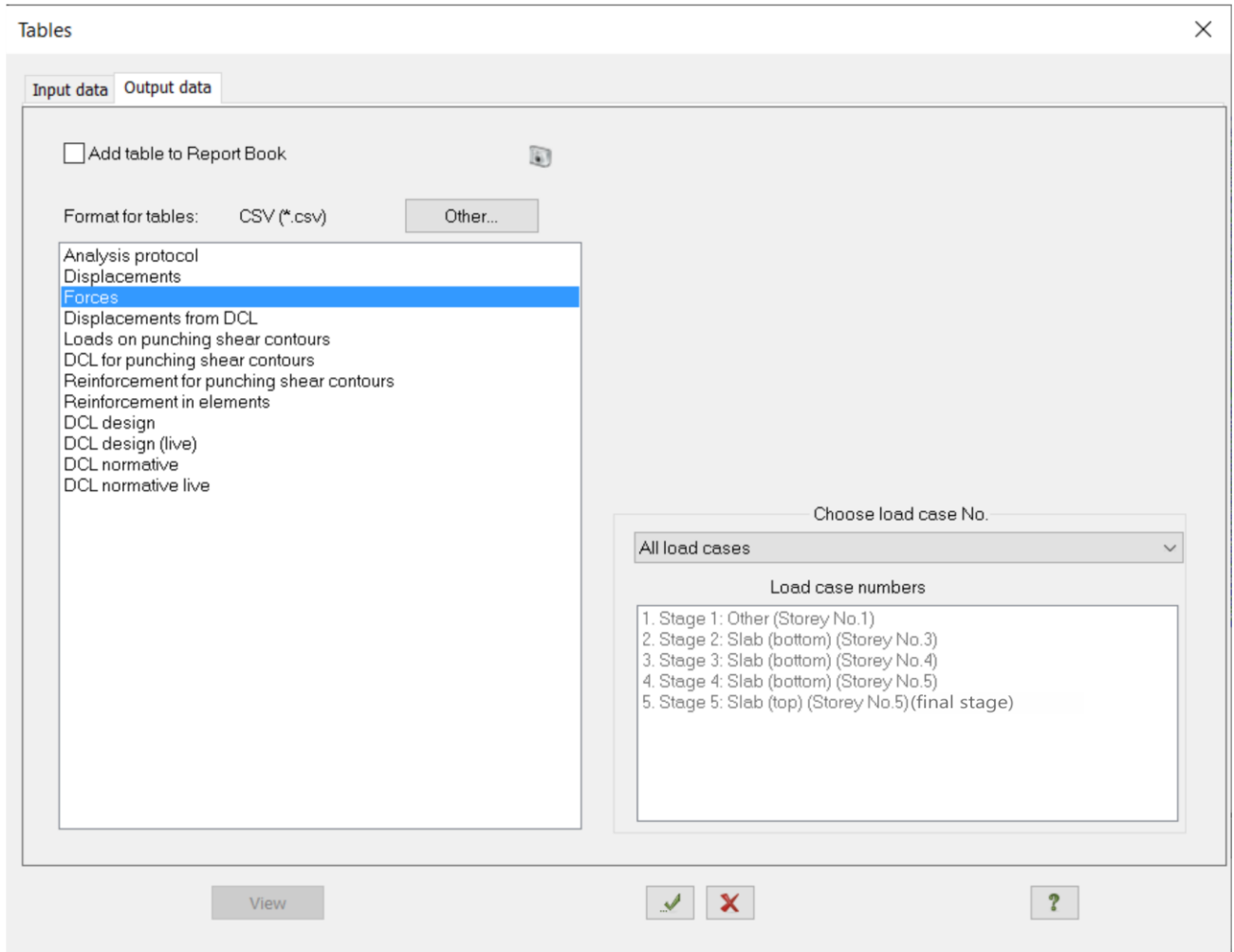
- ⇒ To present contour plot of displacements along the Z-axis, on the **Results** ribbon tab, on the **Deformations** panel, select the **Displacement contour plot in global coordinate system** command  in the **Displacement mosaic/contour plot** drop-down list.

To present stress mosaic plots:

- ⇒ To present stress mosaic plot for M_x , on the **Results** ribbon tab, on the **Stress in plates and solids** panel, select the **Stress mosaic plot** command  in the **Stress mosaic/contour plots** drop-down list.
- ⇒ Then click **Stress M_x** button  on the same panel.
- ⇒ To present stress mosaic plot for N_x , click **Stress N_x** button  on the same panel.

To generate and review tables of analysis results:

- ⇒ To present table with force values in elements of the model, on the **Results** ribbon tab, select **Tables** panel and click **Standard tables**  in the **Documents** drop-down list.
- ⇒ In the **Tables** dialog box (see Fig.23.39), select **Forces** in the list.
- ⇒ Click **Apply**. To generate table in HTML format, in the **Standard tables** dialog box, click **Other**. Then in the **Table format** dialog box, select appropriate option and click **OK**. To generate table in [Document Maker \(DOC-SAPR module\)](#), select RPT format.

Figure 23.39. **Tables** dialog box



⇒ Close the table and then close the **Tables** dialog box.





Step 20. Review and evaluate results from analysis of reinforcement






*When analysis procedure is complete, to review and evaluate results from analysis of reinforcement, select the **RC** ribbon tab.*

To present results from analysis of reinforcement:

- ⇒ To present information about determined reinforcement in a certain FE, on the **Select** toolbar, click **Information about nodes and elements** button  and specify with a pointer any plate element on the model.
- ⇒ In the dialog box that appears on the screen, the **Information about reinforcement** tab is automatically presented. This dialog box contains complete information about selected element, including results for reinforcement.
- ⇒ To close the dialog box, click **Close** button.
- ⇒ To switch to the mode for presentation of symmetric reinforcement in rebars, on the **Design** ribbon tab, select **RC: Bars** panel and click **Symmetric reinforcement** command  in the **Reinforcement** drop-down list.

- ⇒ To display mosaic plot for area of longitudinal reinforcement in the lower left corner of the section AU1, click the **Corner reinforcement AU1** button  (on the **Design** ribbon tab, the **RC: Bars** panel).
- ⇒ To display mosaic plot for area of longitudinal reinforcement in the lower right corner of the section AU2, click the **Corner reinforcement AU2** button  (on the **Design** ribbon tab, the **RC: Bars** panel).
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the X1-axis, click the **Lower reinforcement in plates along X1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).
- ⇒ To display mosaic plot for area of lower reinforcement in plates along the Y1-axis, click the **Lower reinforcement in plates along Y1** button  (on the **Design** ribbon tab, the **RC: Plates** panel).

To generate and review table with analysis results for reinforcement:

- ⇒ On the **Select** toolbar, click **Select vertical bars** button . Select all elements of columns with the pointer.
- ⇒ On the **Design** ribbon tab, select the **Tables** panel and click **Analysis results tables for RC** command  in the **Documents** drop-down list.
- ⇒ In the **Tables** dialog box (see Fig.23.40), the following data is mentioned by default: **Reinforcement in elements** option is selected in the list, the **in bars** option is selected under the **Reinforcement**.
- ⇒ To generate the table with analysis results for reinforcement in bars, click **Apply** .

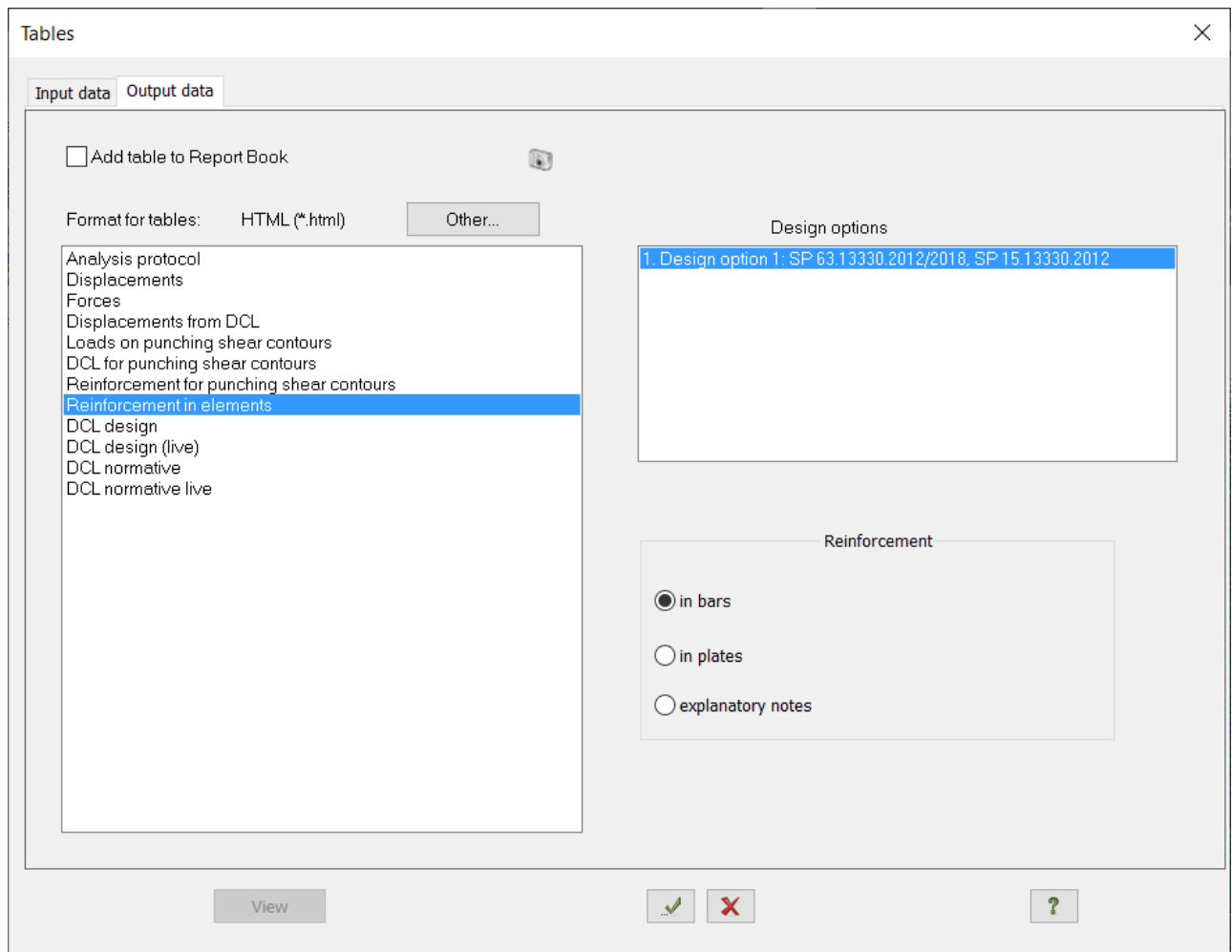



Figure 23.40. **Tables** dialog box

Step 21. Exporting analysis results for reinforcement in floor slabs to SAPFIR module

- ⇒ To export results for reinforcement in floor slabs from **VISOR-SAPR** module, on the **LIRA-SAPR menu** (Application menu), click the **Export reinforcement results to SAPFIR** command .
- ⇒ In the **Export reinforcement results to SAPFIR** dialog box (see Fig. 23.41), define the following data:
 - location where you want to save this file;
 - make sure that **whole design model** option is selected.
- ⇒ Click **OK**.

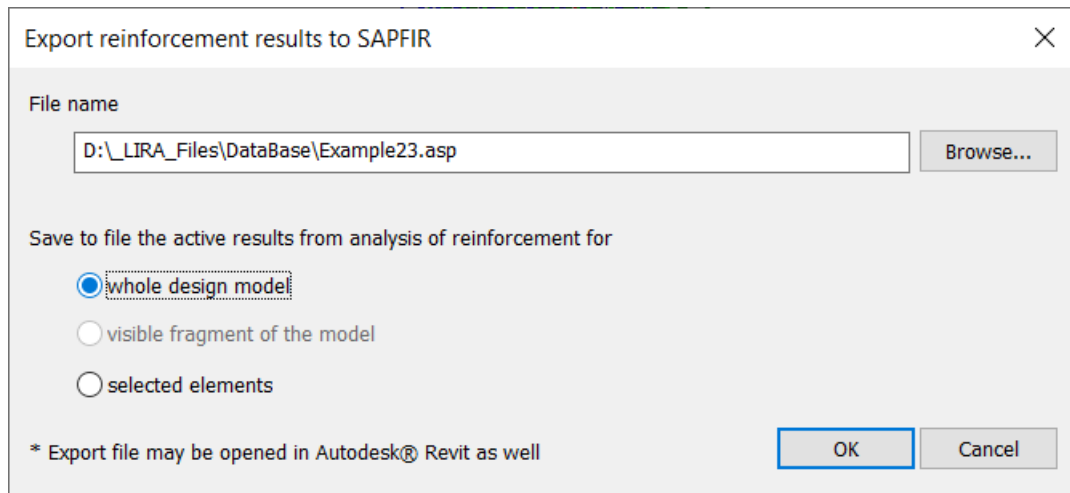




Figure 23.41. **Export reinforcement results to SAPFIR** dialog box



It is recommended to save the file with results (.asp) generated in **LIRA-SAPR** to the same folder where initial file of the model (*.spf) is located. Both files should be named as 'Example23'. Then in **SAPFIR** module, the results will be available automatically when you click the **Show results for reinforcement** button  (the **Auto download results** option in **Project properties**).*

Step 22. Importing analysis results for reinforcement to SAPFIR-RC module

To import results of FEM analysis:

- ⇒ Select the **Example23.spf: 3D view** tab as the current view.
- ⇒ On the **Visualization** toolbar, click the **Filter by object types** button .
- ⇒ In the **Filter for object visualization** dialog box (see Fig. 23.42), clear the **Load** check box.
- ⇒ Click **OK**.

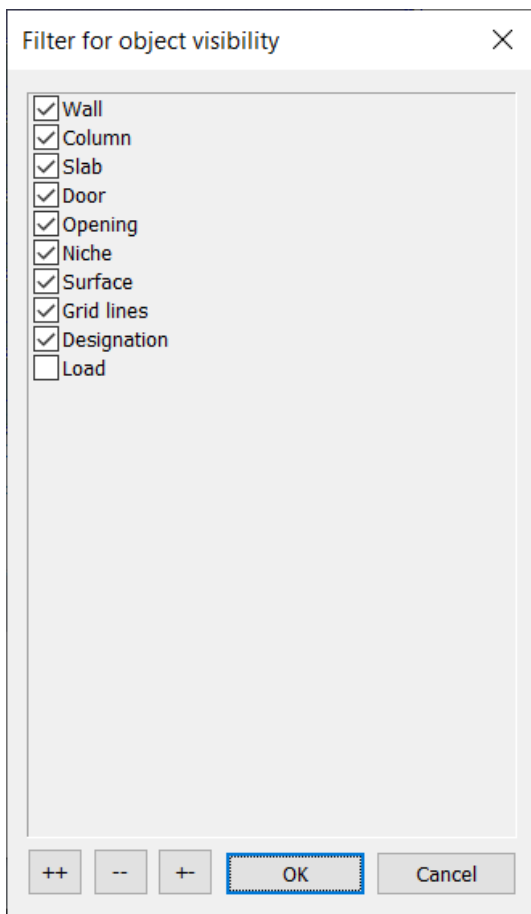



Figure 23.42. **Filter for object visualization** dialog box

- ⇒ To display analysis results for reinforcement, on the **Reinforcement** ribbon tab, on the **Results** panel, click the **Show results for reinforcement** button .
- ⇒ During the import of analysis results for reinforcement, in the **Service information** window, on the **Process** tab (see Fig. 23.43), the following data is mentioned:
 - Building code: SP 63.13330.2012/2018;
 - Analysis: DCL;
 - Variant No.0 ID0: Variant 1: SP 63.13330.2012/2018, SP 15.13330.2012
 - 9328 FE (from 9584) are geometrically related to reinforced elements;
 - 55 columns and beams in project should be reinforced.

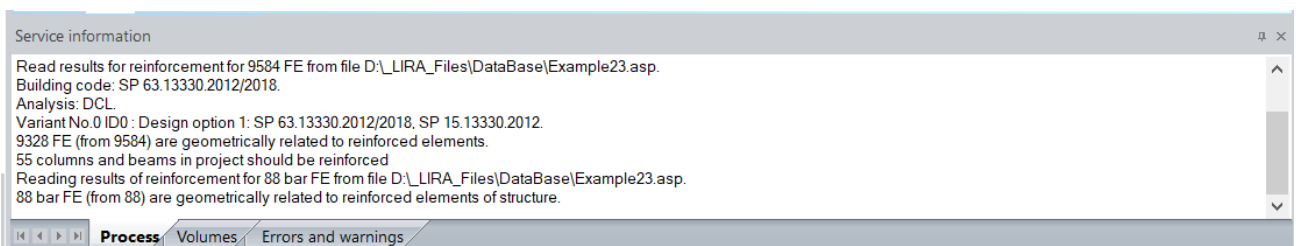



Figure 23.43. **Service information** window

- ⇒ Select (with the pointer) the floor slab between the 4th and the 5th storeys.
- ⇒ On the **Reinforcement** ribbon tab, on the **Main reinforcement** panel, click the **Arrange reinforcement** button .

⇒ In the **SAPFIR 10.0** window (see Fig. 23.44), click **Yes**.

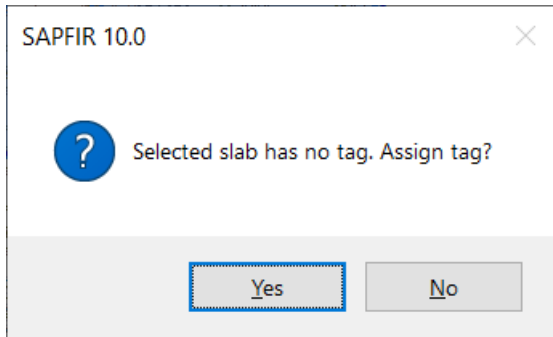


Figure 23.44. **SAPFIR 10.0** window

⇒ Then in the **Tags for elements of structure** dialog box (see Fig. 23.45), accept the suggested tag Sm-1 and click **OK**.

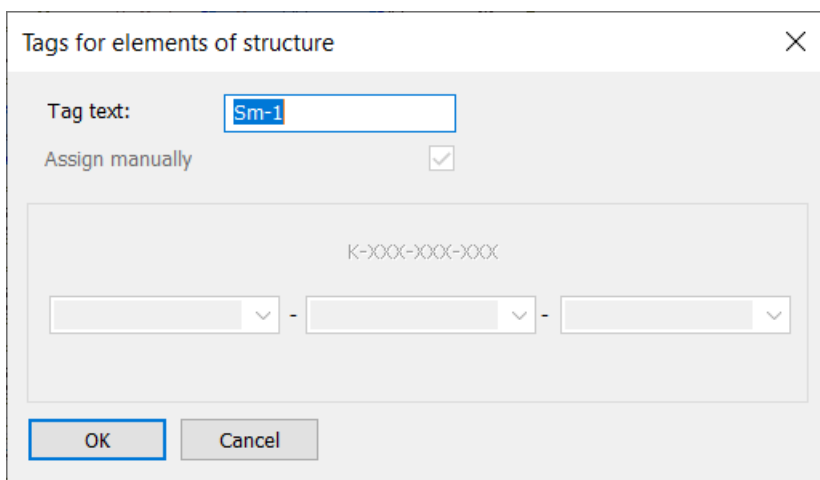


Figure 23.45. **Tags for elements of structure** dialog box

⇒ In the **SAPFIR** window (see Fig. 23.46), click **Yes**.

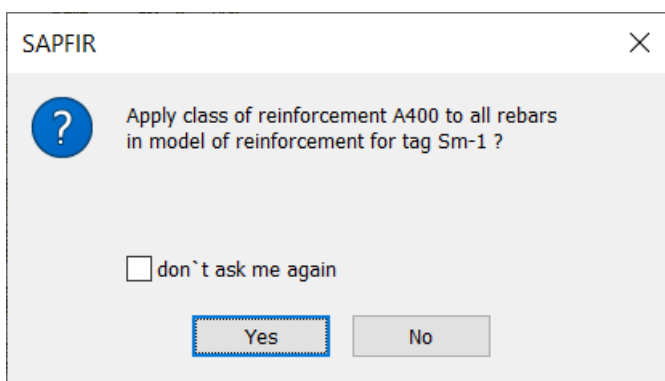



Figure 23.46. **SAPFIR** window



*If analysis results for reinforcement are not saved to the same folder where the model is saved, then it is necessary to download analysis results for reinforcement. Indicate the path to appropriate file with results and click **Open**.*

- ⇒ The new tab **Example23:Sm-1** will be presented on the screen. There you will see the formwork drawing of the floor slab. The top view will be displayed.

To define the settings for reinforcement colour palette:

- ⇒ On the **Reinforcement** ribbon tab, on the **Settings** panel, point to the **Reinforcement palette** drop-down list and click the **Settings for colour palette of reinforcement** button .
- ⇒ In the **Customize reinforcement palette** dialog box (see Fig. 23.47), define the following data:
- right-click anywhere within the dialog box and select the **DT(M), DAKK colours** command;
 - select the **for all** check box;
 - click **OK**.

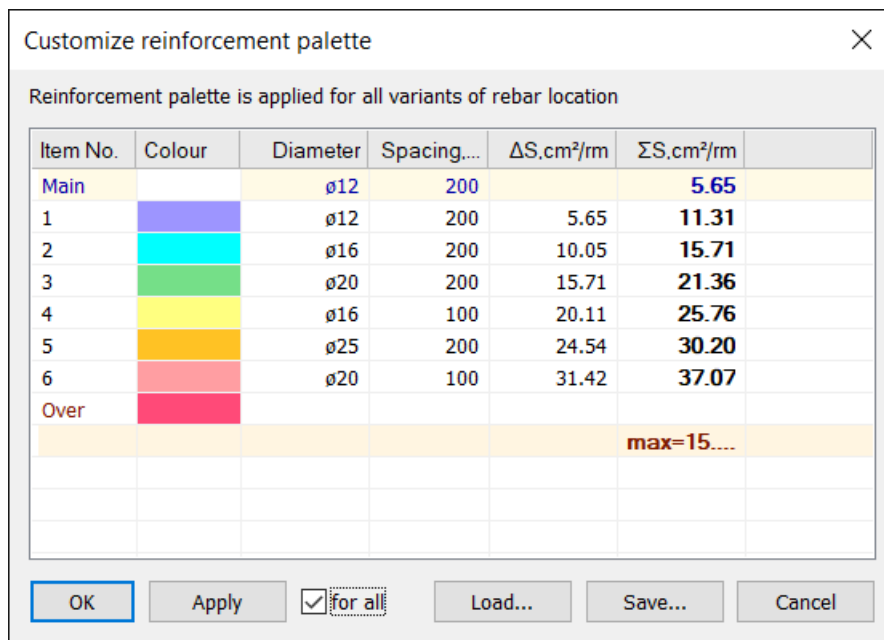





Figure 23.47. **Customize reinforcement palette** dialog box



*In the **Customize reinforcement palette** dialog box, it is possible to vary diameters and spacing for the main and additional reinforcement, create new combinations for diameter and spacing of reinforcement and delete the available ones. Customized colour palette may be saved to file and you could download it to other projects.*

- ⇒ Select the grid.
- ⇒ On the Options Bar, click the **Specify dimensions** button .
- ⇒ To unselect the grid lines, press **Esc**.
- ⇒ To modify location of dimension line, on the **Reinforcement** tab, on the **Modify** panel, use the **Move** command .
- ⇒ To display the upper reinforcement along Y, on the **Reinforcement** tab, on the **Slab** panel, click appropriate button .
- ⇒ Formwork drawing of the floor slab with analysis results for reinforcement as mosaic plot will look like (see Fig. 23.48):

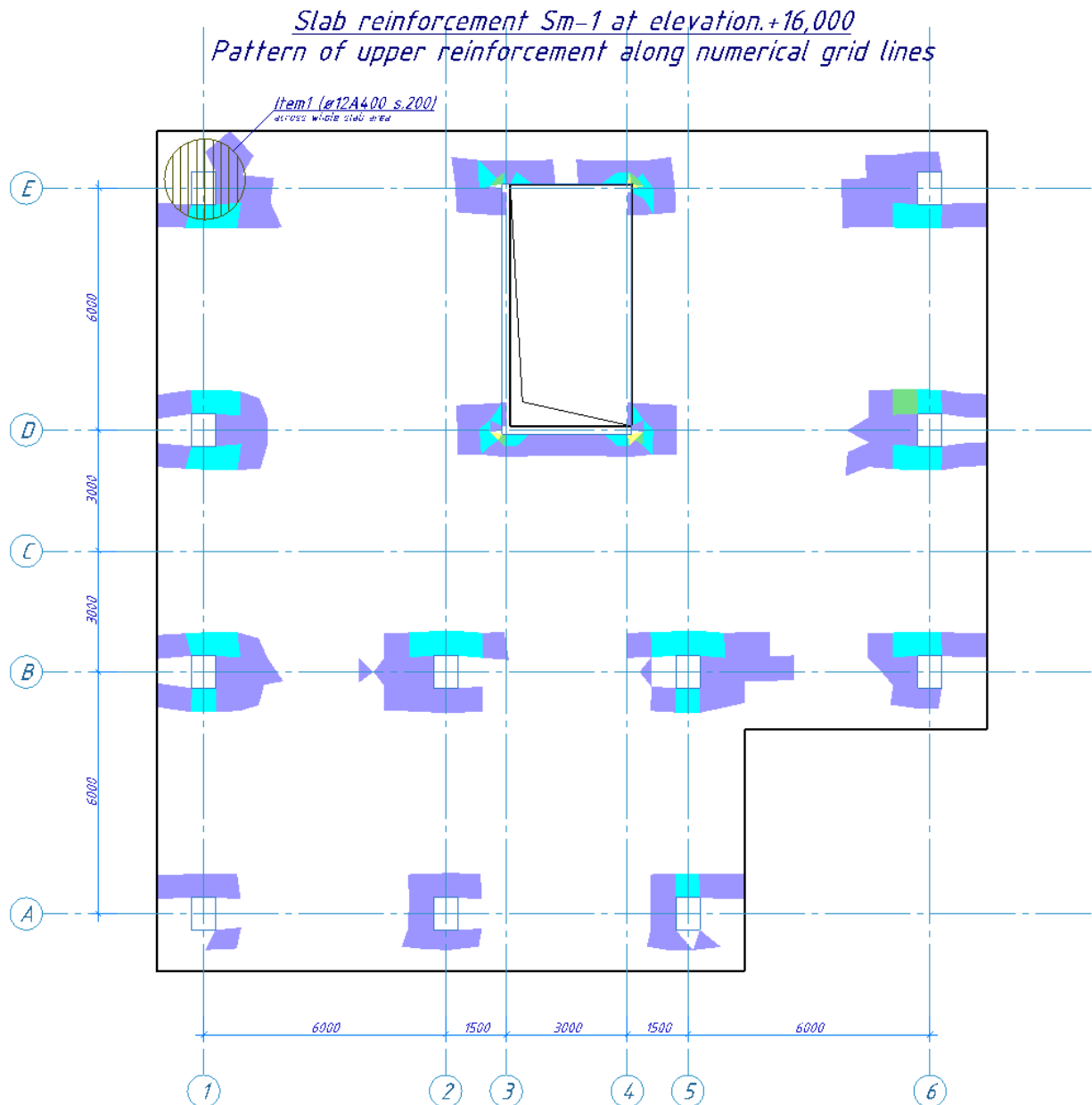



Figure 23.48. Mosaic plot for areas of reinforcement at the upper edge of slab along the Y-axis.

Step 23. Arranging zones of additional reinforcement on the model

To compute the length of anchorage:



Anchorage may be computed only for building codes SNIP 2.03.01-84* and DSTU 3760-98. For example, we will compute the anchorage according to SNIP 2.03.01-84*.

- ⇒ On the **Reinforcement** tab, on the **Settings** panel, click the **Anchorage and lap** button .
- ⇒ In the **Anchorage and overlapping for nonprestressed reinforcement** dialog box (see Fig. 23.49), define the following data:
 - from the drop-down list select concrete class B20;
 - from the drop-down list select diameter of reinforcement 12mm;

- click the **Calculate** button.
- ⇒ To close the dialog box, click **OK**.

Anchorage and overlapping for nonprestressed reinforcement

Code: SNIP 2.03.01-84*

Concrete type: Heavyweight and fine-aggregate

☐ Fine-aggregate concrete of group B

Concrete class: B20 Rb: 11.5 MPa

Reinforcement: A-III d=6...40 Rs: 365 MPa

Diameter: 12 mm Shape of rebars: Variable

Ratio of reinforcement area required by calculation to area actually placed:

☒ As req./As act. 1

☐ As req. 2.54469 cm²

☐ As act. 2.54469 cm²

Service conditions for nonprestressed reinforcement:

☒ Tensile reinforcement embedded (concrete in tension)

☐ Compressive/tensile reinforcement embedded (concrete in compression)

☐ Lap splices of rebars (concrete in tension)

☐ Lap splices of rebars (concrete in compression)

ω_{an} 0.7

$\Delta\lambda_{an}$ 11

At least:

λ_{an} 20

l_{an} 250 mm

$\lambda_{an}d$ 240 mm

10d 120 mm

By formula:

λ_{an} 33.2174

l_{an} 398.609 mm

Calculate

Anchorage length:

l_{an} 399 mm

OK

Figure 23.49. **Anchorage and overlapping for nonprestressed reinforcement** dialog box




By default, all pattern zones of the slab have initial value of the anchorage length according to the specified diameter. The anchorage length is taken from the **Reinforcement** table (**Settings** menu / **Reinforcement**). This table is generated according to SNIP 2.03.01-84* for concrete class B25 and reinforcement class A-III. It is possible to define your own set of rebars and settings for this set, and save them to a file. User-defined set may be applied later in other projects.



To arrange rectangular zones of additional reinforcement:

- ⇒ On the **Reinforcement** ribbon tab, on the **Main reinforcement** panel, point to the **Slab** drop-down list and click the **Rectangular zone** button .
- ⇒ On the **Rectangular zone** Options Bar, define the following data:
 - methods for generation **Define pattern with diagonal** and **Define reinforcement zone** ;
 - from the drop-down list select **Ø12 spacing 200mm**;

- make the **Get from mosaic or contour plot**  option not active.






When the **Get from mosaic or contour plot** mode  is active, the **SAPFIR-RC** module will automatically obtain from mosaic/contour plot of reinforcement the intensity of reinforcement (diameter and spacing) required by analysis to arrange at this zone of reinforcement.

- ⇒ In the **Properties for generation: Zone of slab reinforcement** window, define the **anchorage length** as 400mm.
- ⇒ Click the **Apply to object**  in the **Properties** window or press **Enter**.
- ⇒ Drag several rectangles of reinforcement above the supports so that the whole zone of additional reinforcement will be included. To define location for the rectangular zone of additional reinforcement, specify two vertices for diagonal of the rectangle (see Fig. 23.50).
- ⇒ On the **Reinforcement** ribbon tab, on the **Results** panel, point to the appropriate drop-down list and click the **Lack of reinforcement** button .



When the **Lack of reinforcement** visualization mode is active, as soon as the zone is overlapped with additional reinforcement, such zone will not be displayed on the formwork drawing any more.

- ⇒ On the **Rectangular zone** Options Bar, define the following data:
 - select method for generation as **Define pattern from centre**  ;
 - make the **Get from mosaic or contour plot**  option as active.
- ⇒ Once more, drag several rectangles of reinforcement above the supports so that the whole zone of additional reinforcement will be included. To define location for the rectangular zone of additional reinforcement, specify the centre of the reinforcement zone (in our case it will be the central point of the column) and half of diagonal of rectangle. The pattern zone will be symmetric from the centre (see Fig. 23.50).
- ⇒ To exit the mode of additional reinforcement, press **Esc**.
- ⇒ Select the reinforcement zone where the edges extend beyond the contour of the floor slab.
- ⇒ On the **Rectangular zone** Options Bar, click the **Cut by place** button  to indicate this zone with * and define that it will be cut on the building site.
- ⇒ To unselect the zone of reinforcement, press **Esc**.

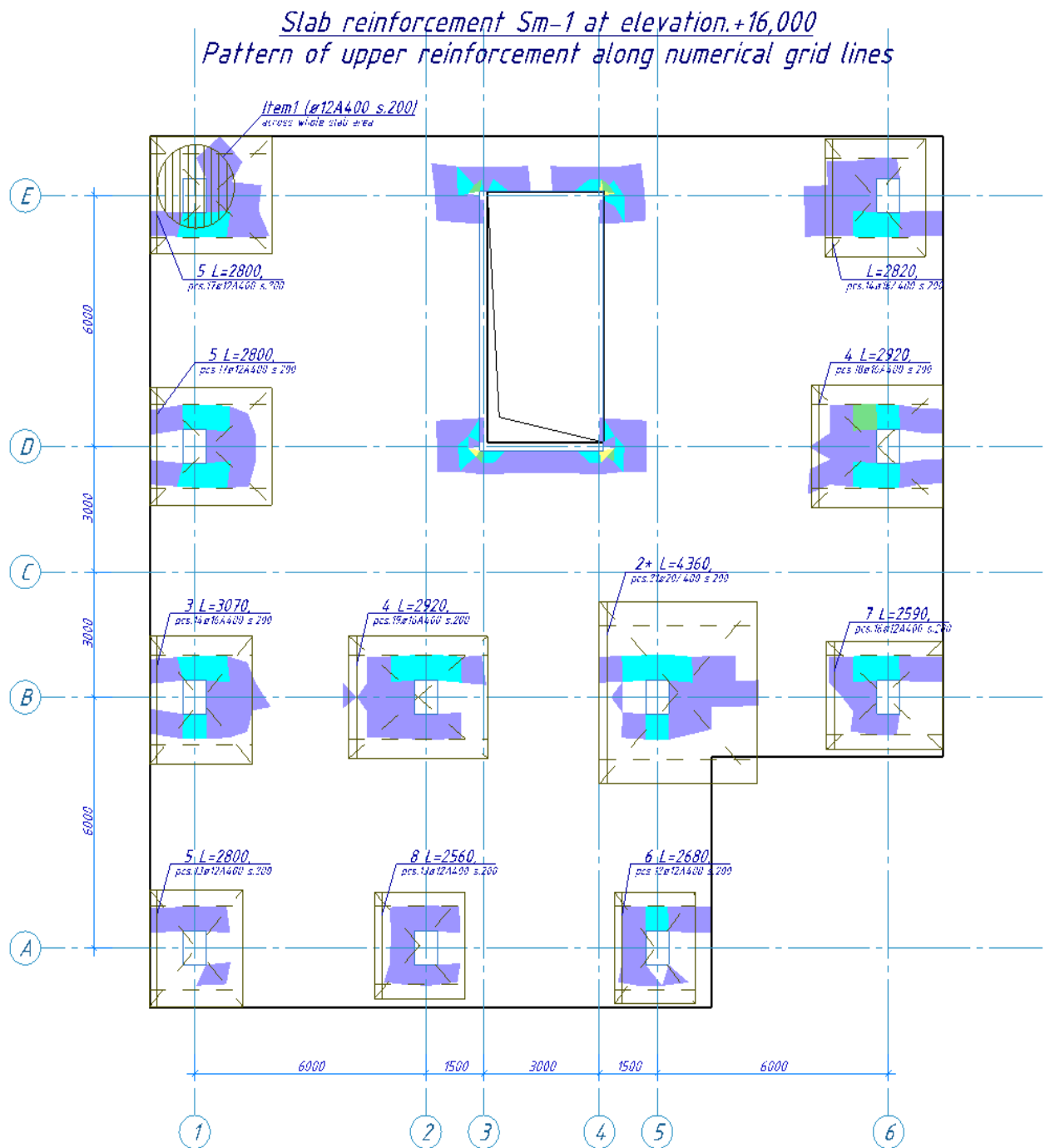





Figure 23.50. Pattern of upper reinforcement along the Y-axis

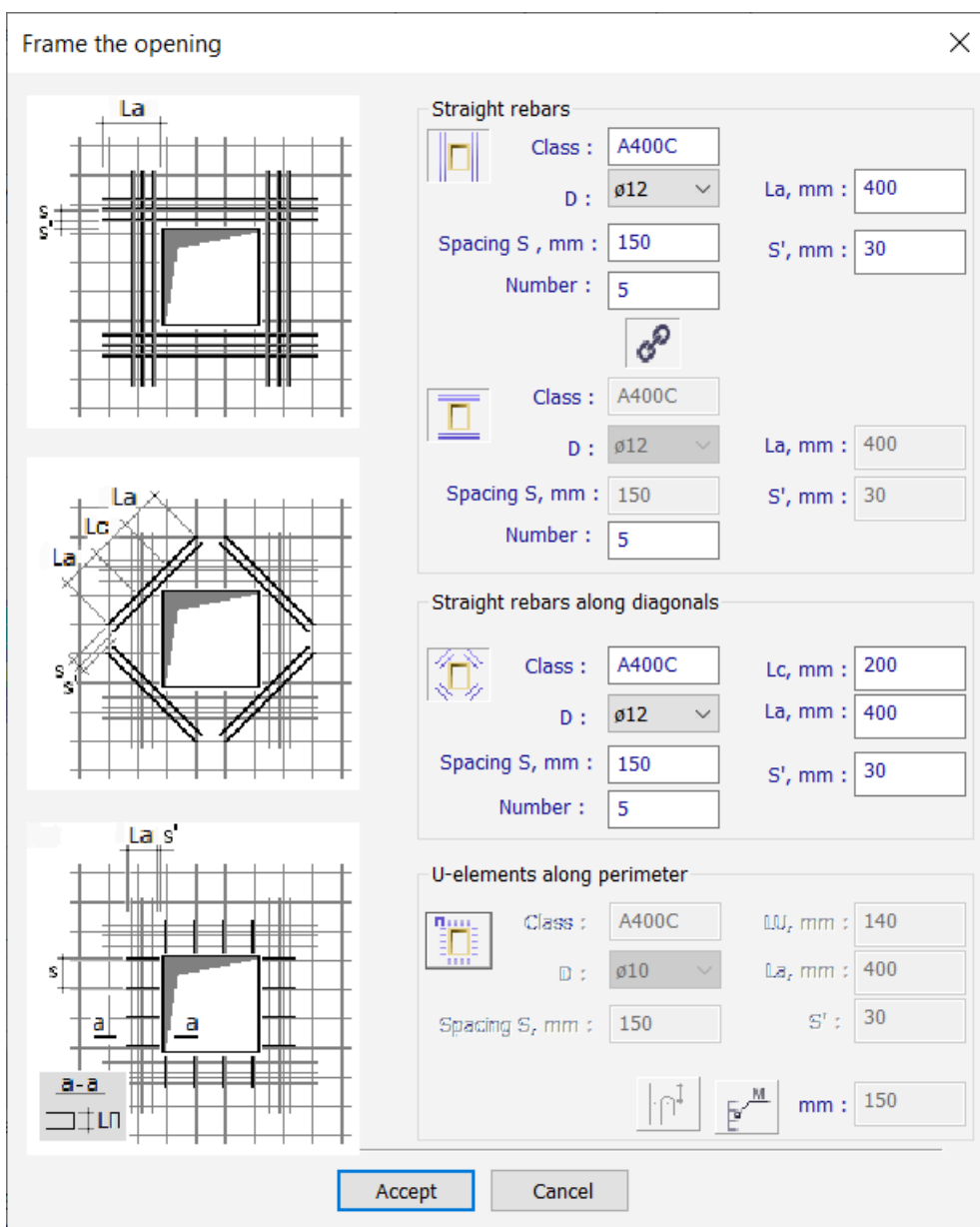
- ⇒ In the same way, create the reinforcement zones for upper reinforcement along the X-axis and for lower reinforcement along the X-axis and Y-axis.

Step 24. Framing the opening in the floor slab

To frame the opening:

- ⇒ Select the opening in the floor slab by clicking the edge of the opening.

- ⇒ On the **Reinforcement** ribbon tab, on the **Additional reinforcement** panel, point to the **Framing** drop-down list and click **Frame the opening** .
- ⇒ In the **Frame the opening** dialog box (see Fig. 23.51), define the following data:
 - D, mm – 12;
 - in the **Straight rebars** area, **spacing S**, mm – 150;
 - number – 5;
 - click the **Identical parameters** button  ;
 - D, mm – 12;
 - in the **Straight rebars along diagonals** area, **spacing S**, mm – 150;
 - number – 5;
 - click the **U-elements** button  to cancel arrangement of U-elements.
- ⇒ Click **Accept**.



Frame the opening

Straight rebars

Class : A400C
 D : $\varnothing 12$
 Spacing S, mm : 150
 Number : 5
 La, mm : 400
 S', mm : 30

Straight rebars along diagonals

Class : A400C
 D : $\varnothing 12$
 Spacing S, mm : 150
 Number : 5
 Lc, mm : 200
 La, mm : 400
 S', mm : 30

U-elements along perimeter

Class : A400C
 D : $\varnothing 10$
 Spacing S, mm : 150
 L_U, mm : 140
 L_S, mm : 400
 S' : 30
 mm : 150



Accept Cancel

Figure 23.51. **Frame the opening** dialog box




*If rebars for the frame of the opening are defined, then in the **Lack of reinforcement** mode, mosaic plot of reinforcement is still visible near the opening.*

To create leaders with tags for the reinforcement of the frame:

- ⇒ Select vertical rebars of the framing.
- ⇒ On the **Rebar** Options Bar, click the **Tag-leader** button .
- ⇒ Create leaders for the rest of rebars in the framing (see Fig. 23.50).
- ⇒ To edit location of the leaders, on the **Reinforcement** ribbon tab, on the **Modify** panel, use the **Move vertex** command .

Step 25. Generating schedule of reinforcement and sheet of drawing

To unify zones of additional reinforcement:

- ⇒ On the **Reinforcement** ribbon tab, on the **Documentation** panel, point to the **Schedule** drop-down list and click the **Schedule: Reinforcement** button .
- ⇒ In the **Rebar Schedule. Sm-1** dialog box (see Fig. 23.52), select the item (row) that should be merged with the close value and right-click this row.
- ⇒ On the shortcut menu, select the **Merge with previous** command (when this command is complete, in the **Δm, kg** column you will see the value for the overexpenditure of reinforcement because of unification).



*As the pattern for zones of additional reinforcement was produced not exactly to dimensions and depends on the specified module for length of rebar and spacing of reinforcement, then number and length of rebars in the Schedule of reinforcement table may differ from the values displayed in the Figure below. It is possible to merge items for the zones where it is rational depending of the length of rebars. To modify the module for length of rebar as well as other default parameters, on the **SETTINGS** menu, select **SAPFIR preferences / Reinforcement**.*

Rebar List. Sm-1						
Item ...	Reference d...	Name	Number	Mass	Unif.Δm,...	Note
1	GOST 1088...	ø12A400, ΣL=7765 ...	-	6893.7	-	Include overexpenditu...
2	GOST 1088...	ø20A400, L=4360	21 pcs.	225.8	-	
3	GOST 1088...	ø16A400, L=3070	14 pcs.	67.8	-	
4	GOST 1088...	ø16A400, L=2920	37 pcs.	170.5	8.8	2 zone
5	GOST 1088...	ø16A400, L=2820	14 pcs.	62.3	2.2	
6	GOST 1088...	ø12A400C, L=6800	20 pcs.	120.7	-	
7	GOST 1088...	ø12A400C, L=3800	20 pcs.	67.5	53.3	
8	GOST 1088...	ø12A400, L=2800	47 pcs.	116.8	41.7	3 zone
9	GOST 1088...	ø12A400, L=2680	12 pcs.	28.6	1.3	
10	GOST 1088...	ø12A400, L=2590	16 pcs.	36.8	1.3	
11	GOST 1088...	ø12A400, L=2560	13 pcs.	29.5	0.3	
12	GOST 1088...	ø12A400C, L=1000	40 pcs.	35.5	55.4	
SP11	GOST 1088...	ø10A400, L=1210	586 pcs.	437.2	-	105mm height
Sm-1		B20	75.02 ...			
Total:				8292.8		on the average 110.5 ...

Figure 23.52. Rebar list. Sm-1 dialog box

- ⇒ To merge three and more items, and if it is necessary to define the length of unified rebars, select three rows in the table and click the **Unify** button.
- ⇒ In the **Unify length of rebars** dialog box, define the new value for the length (by default, this value will be equal to the greatest of the items that are unified).



The reference document may be modified directly in the **Rebar schedule** table (see appropriate column). Right-click the cell and select the **Reference document** command on the shortcut menu. To define the reference document accepted by default, on the **View** ribbon tab, on the **Settings** panel, click the **SAPFIR Preferences** button (see the **Reinforcement** tab, **Pattern of rebars / Reference document, default**). Click **Apply** in the dialog box to apply modifications to the following project.

- ⇒ To apply modifications and close the **Rebar Schedule. Sm-1** dialog box, click **OK**.

To define dimensions:



It is recommended that you should define patterns for zones of additional reinforcement, unify reinforcement, and only then indicate dimensions automatically.

- ⇒ On the **Reinforcement** ribbon tab, on the **Annotation** panel, click the **Indicate dimensions**

automatically button

- ⇒ For all pattern zones in the slab, the program automatically indicates overall dimensions of the zone and its snap to the grid lines.

- ⇒ To edit location of dimensions, on the **Reinforcement** ribbon tab, on the **Modify** panel, use the **Move**

vertex command

Slab reinforcement Sm-1 at elevation.+16,000
Pattern of upper reinforcement along numerical grid lines

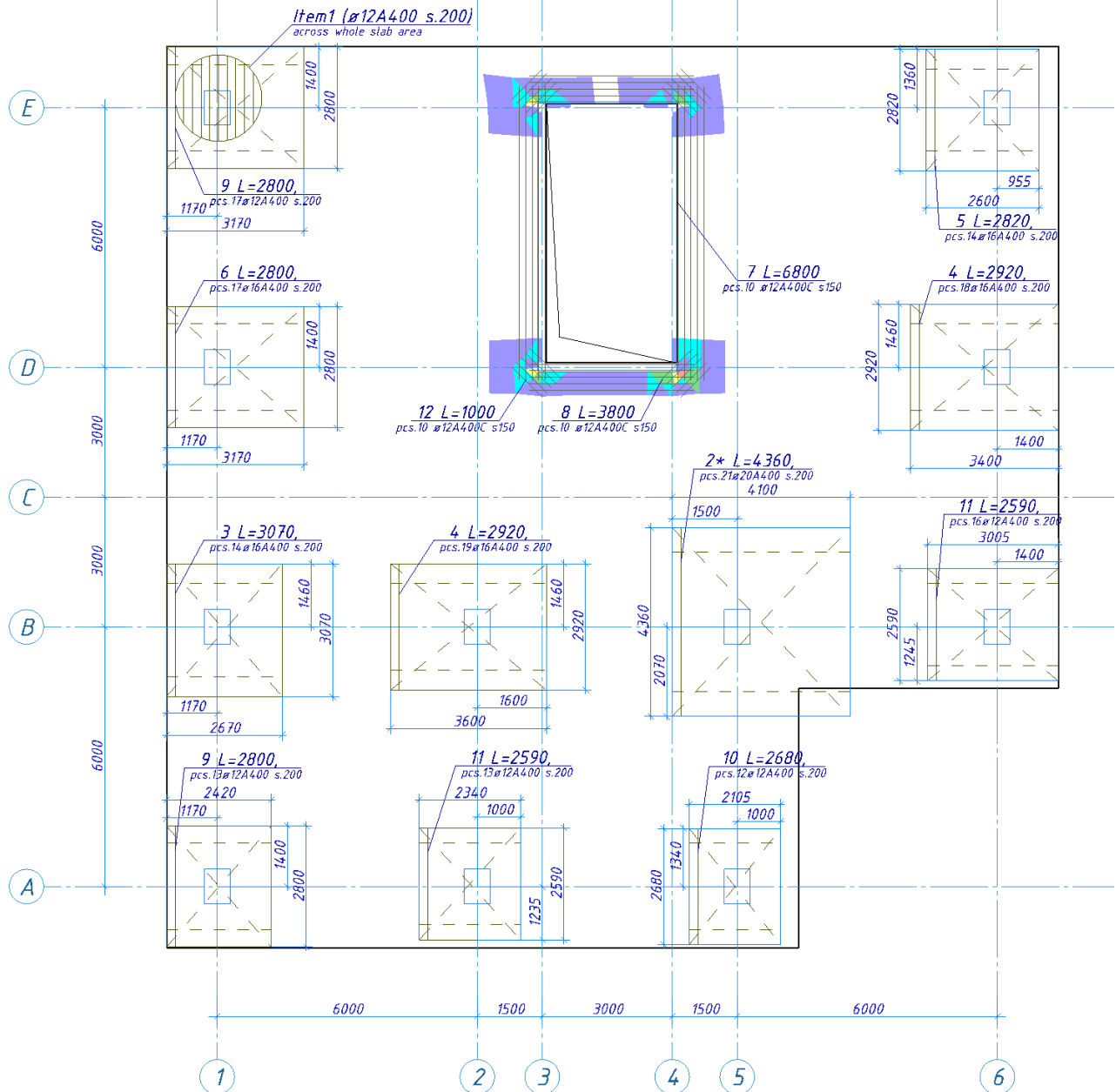



Figure 23.53. Pattern of upper reinforcement along the Y-axis with snap of zones to the grid lines

To generate the sheet of drawing:

- ⇒ On the **Reinforcement** ribbon tab, on the **Documentation** panel, point to the **Schedule** drop-down list and click the **Schedule: Reinforcement** button .
- ⇒ In the **Rebar Schedule. Sm-1** dialog box (see Fig. 23.52), click the **Place on drawing** button.
- ⇒ In the **Draw schedules and rebar lists** dialog box (see Fig.23.54), define the **Name** as **Reinforcement in floor slab Sm-1 at +16.000**.

Draw schedules and rebar lists

Slab Sm-1

Sheet

☒ new ☐ prepared ☐ active

Name :
Reinforcement in floor slab Sm-1 at +16.000

Format : A2 **Title block :** Shape 4 **Template name :**
Orientation : ☐ portrait ☒ landscape

Snap of the tables
Table snap - upper right corner. Table height - max height.

	Schedule :	List of components :	List of steel consumption :
horizontal :	589 mm	394 mm	399 mm
vertical :	415 mm	415 mm	68 mm
table height :	350 mm	410 mm	
	generate <input checked="" type="checkbox"/>	generate <input checked="" type="checkbox"/>	generate <input checked="" type="checkbox"/>

Add auto notes ☒

CSV ☒ ☐

Figure 23.54. Draw schedules and rebar lists dialog box

- ⇒ Click the **Draw selected tables and notes** button ☒ ; these items will be generated and placed on the drawing.
- ⇒ The new tab **Example23.spf: Reinforcement in floor slab Sm-1 at +16.000** will be displayed in the working window of the program. The rebar schedule, list of steel consumption, list of components and notes will be presented there.

To place the reinforcement pattern on the sheet of drawing:

- ⇒ To place the reinforcement pattern for the floor slab on the sheet of drawing, in the **Views** service window, open the **Assemblies** tree-like list.
- ⇒ In the **Assemblies** list, right-click the **Sm-1** row.
- ⇒ On the shortcut menu, click **Place on drawing**.
- ⇒ The formwork drawing of the slab with the reinforcement pattern will be displayed on the sheet of drawing.
- ⇒ Select the reinforcement pattern.
- ⇒ To move the model, simply drag it.
- ⇒ To indicate appropriate location, click the mouse button (see Fig. 23.55).
- ⇒ To unselect the model, press **Esc**.

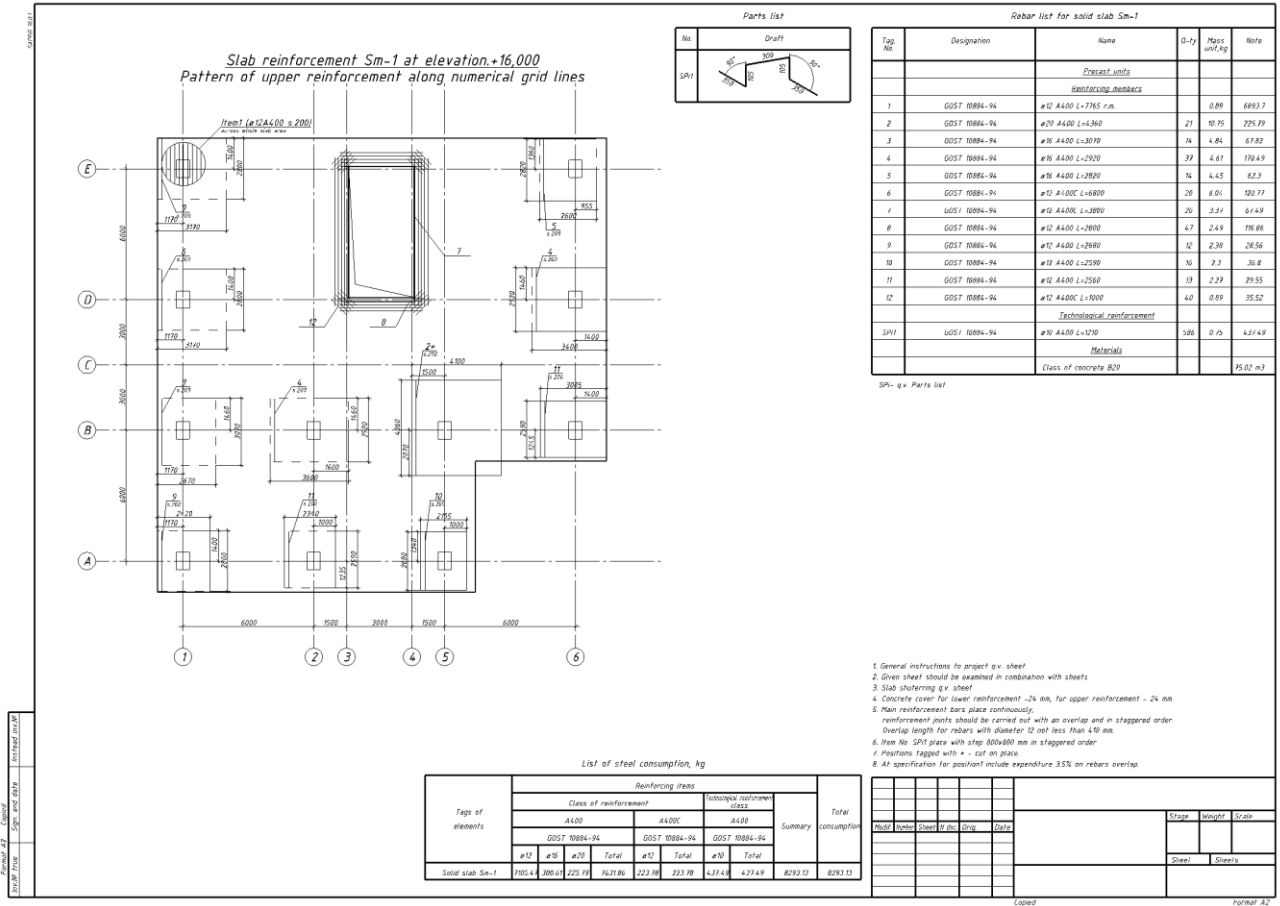


Figure 23.55. Drawing for the reinforcement pattern in the floor slab at +16.000

- ➔ In the same way, generate the drawings with reinforcement for the upper reinforcement along the X-axis, lower reinforcement along the X-axis and the Y-axis in Sm-1.



Rebar schedule is provided for the whole slab Sm-1, all pattern zones of reinforcement are considered.