

Example 2. Import and analysis of the floor slab in the SLAB module

In this lesson you will learn how to:

- obtain model of the floor slab in the **SLAB** module of the program (through the import procedure). Import file with model of the floor slab as a map of (imported) displacements was generated in Example 1 in the **BUILDING** module according to FEA results of the structure;
- analyse the slab;
- evaluate modes for presentation of analysis results;
- explore the strategy and follow the sequence in slab design;
- generate the drawing and to learn the basic principles of working with the **DRAWING** of **SLAB** module;
- explore the options to export data to **LIRA-SAPR** program.




*In **BUILDING** module, it is possible to export the slab as a map of imported displacements or to export the FE model of the slab. In the latter case, the model is exported with ready-to-use solution (division into finite elements accepted in **BUILDING** module; displacements, stresses and forces computed in **BUILDING** module).*

Description:

File (*2_1.dai*) for import of slab as a map of imported displacements was generated in Example 1 according to FEA results of the structure (see directory *Port/Model1.chg*).



*Import files for design modules of **MONOMAKH-SAPR** program may be restored any time if you have the file with model of the structure. This file should be generated and saved previously in the **BUILDING** module of the program. To restore the import file, in the **BUILDING** module, on the **RESULTS** menu, click **Export data to MONOMAKH-SAPR modules** (button  on the toolbar).*

Model of the slab is presented in Fig.2.a.

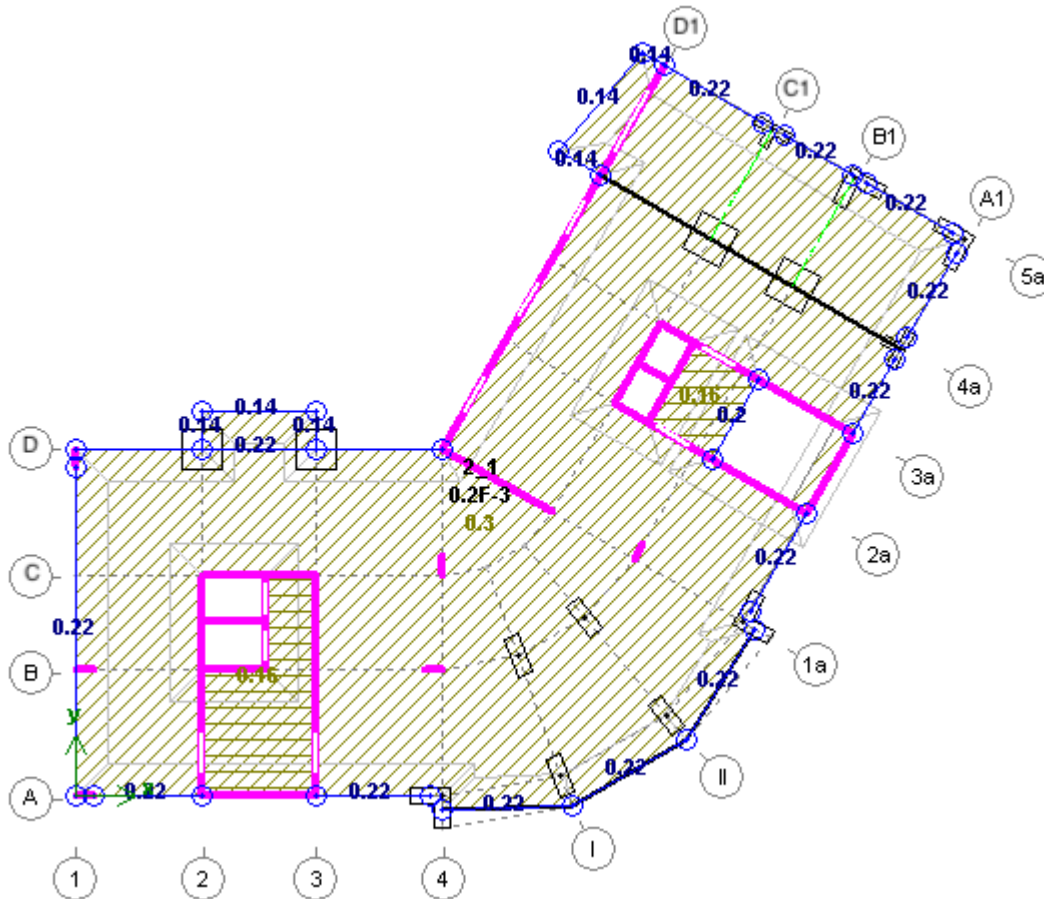


Figure 2.a. Slab No.2_1 in BUILDING module

Triangulation of slab for FEA in the BUILDING module is presented in Fig. 2.b.



To understand triangulation properly, lines of walls and nodes of columns are also presented in the figure.

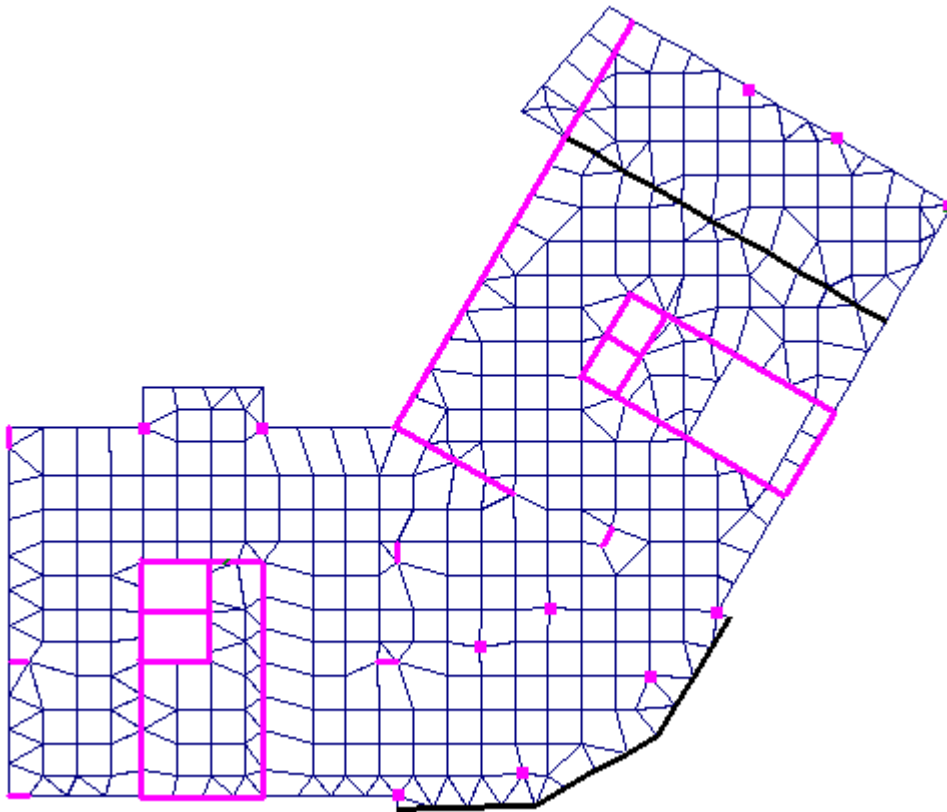



Figure 2.b. Triangulation of slab No.2_1 in BUILDING module

- ⇒ On the taskbar, click the **Start** button, then point to **Programs / MONOMAKH-SAPR 2016** folder and click **6. SLAB**.

Step 1. Create new problem in import mode

To import file generated in the **BUILDING** module:

- ⇒ On the FILE menu, click **Import** (button  on the toolbar).
- ⇒ In the **Open file** dialog box, define the following:
- path where the file *2_1.dai* was saved - *MONOMAKH-SAPR 2016 / Port / Model1* ;
 - file name *2_1.dai*.
- ⇒ Click **Open**.



*If the slab is exported as a map of imported displacements, elements that join the slab model are not exported to design module. Their presence at nodes of design model is simulated with group of initial (forced) displacements – linear translations and rotation angles. In this case it is possible to define triangulation step for slabs (for analysis in the **SLAB** module) different from the one defined in the **BUILDING** module. The smaller triangulation step is usually defined in design modules.*

Model of the slab with a map of displacements is presented in Fig. 2.1.1.

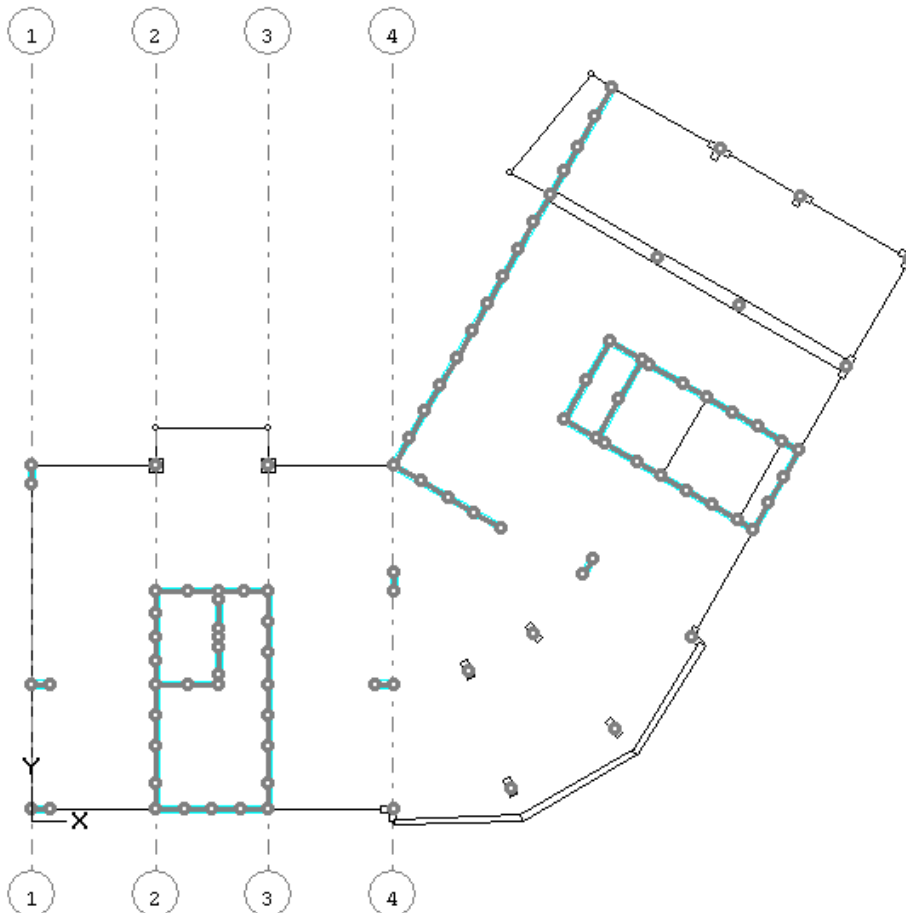


Figure 2.1.1. Slab No2_1

To check whether loads are imported, on the LOAD menu, click **Dead** (button  on the toolbar).

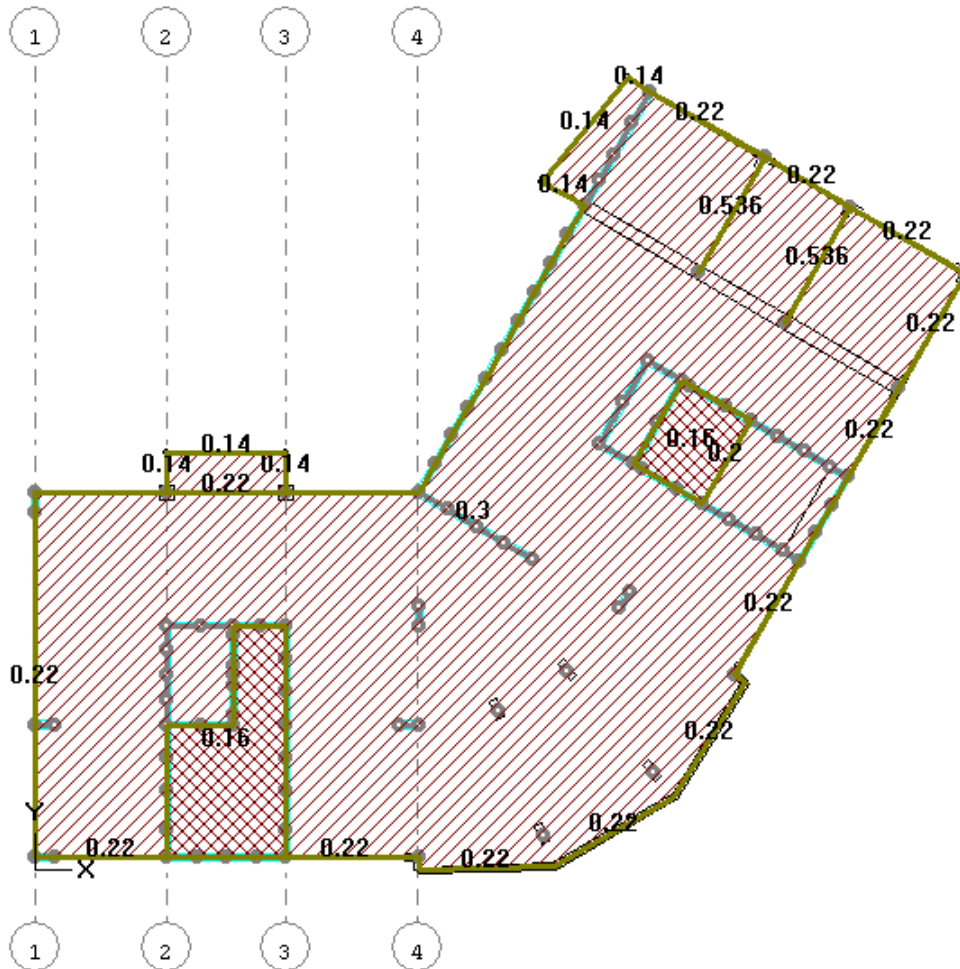




Figure 2.1.2. Slab No.2_1 with loads



Note that element 'partition' in the **SLAB** module is considered as linear load. Defined vertical loads for dead, live and short-term load cases are imported for floor slabs. Loads caused by slab behaviour in the whole model of structure are considered in a map of displacements.


- ⇒ To hide the load visualization, on the **LOAD** menu, click **Dead** (button  on the toolbar) once again in order to make this command not active.

To save the data about model:

- ⇒ On the **FILE** menu, click **Save** (button  on the toolbar).
- ⇒ In the **Save as** dialog box, define the following:
- file name **Slab1**;
 - directory **MONOMAKH-SAPR 2016** – directory specified as directory for problems when **MONOMAKH-SAPR** was installed.
- ⇒ Click **Save**.

Problem file *Slab1.plt* will be saved in the directory for problems in **MONOMAKH-SAPR**.

To re-open this file later:

- ⇒ To continue the work with the model, open the saved file *Slab1.plt*. To do this, on the FILE menu, click **Open** (button  on the toolbar).

Step 2. Check material properties

To check material properties:



The slab model is ready for analysis. If necessary, you could correct several parameters accepted by default.

- ⇒ On the MATERIALS menu, point to **Material properties by** and click **SNIP 2.03.01-84**.
- ⇒ In the dialog box, on the **Materials** tab (see Fig.2.2.1), evaluate the data:
 - parameters imported from the **BUILDING** module and additional parameters accepted by default in the **SLAB** module - all of them should remain.
- ⇒ Select the **Analysis parameters** tab (see Fig.2.2.2) and evaluate the data:
 - make sure that the **Analysis of transverse reinforcement per 1 sq.m.** check box is selected.
- ⇒ Click **OK**.

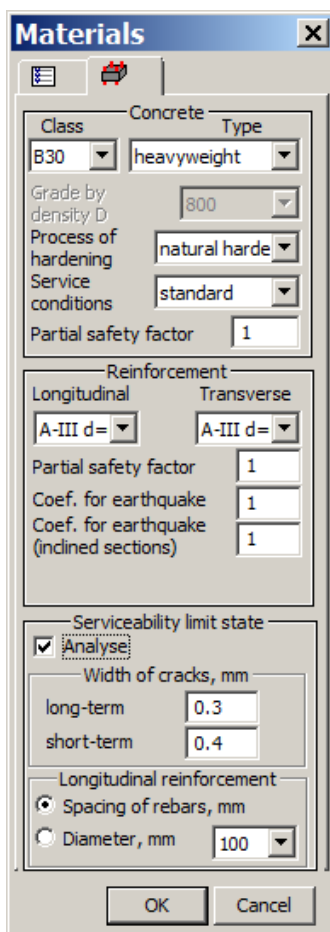


Figure 2.2.1. **Materials** dialog box

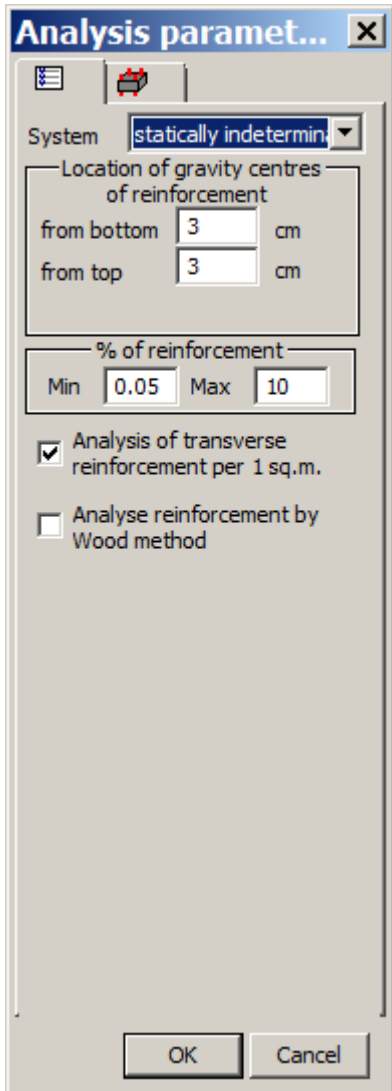



Figure 2.2.2. **Analysis parameters** dialog box

Step 3. Analysis of slab

To analyse the slab:



For analysis procedure, design FE model is generated automatically with account of the specified triangulation step. (Static) analysis is carried out by FEM (finite element method). The output data includes nodal displacements, stresses and forces in elements of the slab. Area of longitudinal and transverse reinforcement in slab is determined according to analysis results.

- ⇒ On the ANALYSIS menu, click **Analysis** (button  on the toolbar).
- ⇒ In the **Analysis** dialog box (see Fig.2.3.1) all parameters remain by default.



*Note that for FEA in the **BUILDING** module, triangulation step for this slab was defined as equal to 1m. Here, in the **SLAB** module, triangulation step is accepted by default as equal to 50cm.*

⇒ Click **OK**.

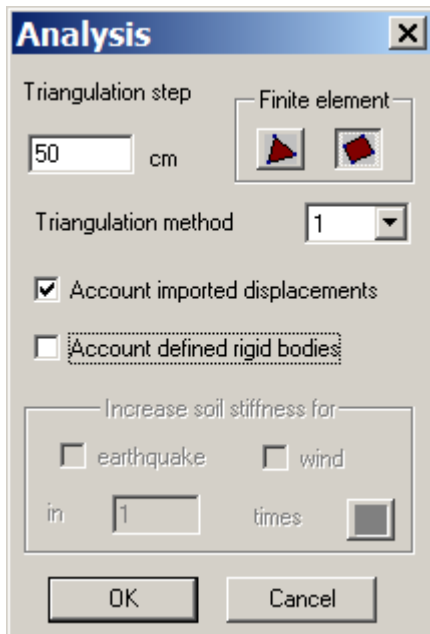


Figure 2.3.1. **Analysis** dialog box


In the Solver window you will see design model and analysis protocol. Wait when the analysis is complete.



*If you want to examine the slab behaviour without account of displacements of supports obtained in FEA in the **BUILDING** module, then it is possible to carry out analysis without account of map of imported displacements. To do this, in the **Analysis** dialog box, clear the **Account imported displacements** check box. It is also possible to export the slab model obtained in the **BUILDING** module according to results of preliminary analysis. In this case, type of restraints (fixed or flexible) in the support nodes may be assigned in the **SLAB** module. But keep in mind that when you import the slab model according to results of preliminary analysis, some effects concerning behaviour of fragment in the whole model of the structure are not considered.*

To save analysis results:



*To save the model, on the **FILE** menu, click **Save** (button  on the toolbar). When you save the model in this way, analysis results are also saved in *.plt file.*

To export to LIRA-SAPR:

- ⇒ To export design model to **LIRA-SAPR** program, on the **FILE** menu, click **Export to LIRA-SAPR**.
- ⇒ In the **Save as** dialog box, define the following:
 - path where the file will be saved (by default, directory **LData** in the base MONOMAKH-SAPR directory);
 - file name *Slab1.txt*.
- ⇒ Click **Save**.

To open the import file in LIRA-SAPR 2017:

Open the saved file *Slab1.txt* with design model in [LIRA-SAPR 2017](#).

- ⇒ In [LIRA-SAPR](#) program , on the Application menu, point to **Import data from** and click **Text files (*.txt)**.
- ⇒ In the **Open** dialog box, do the following:
 - open the directory **LData** in the base MONOMAKH-SAPR directory;
 - open file *Slab1.txt*.

The program will check the input data and then design model will be available in [VISOR-SAPR](#) window.



Design model is ready for analysis procedure.

- ⇒ In [LIRA-SAPR 2017](#), save the file as *Slab1.lir*.


Step 4. Evaluate analysis results

To preview contour plots of displacements:

- ⇒ On the RESULTS menu, click **Displacements** (button  on the toolbar).



*Contour plots of displacements along the Z-axis are generated as envelope lines. When you preview displacements, they are presented with design values. Please note the **difference** between max and min values of displacements. This difference may be caused either by uneven settlement of building supports or by max deflection of the slab in spans.*

- ⇒ On the RESULTS menu, click **Relative displacements** (button  on the toolbar).
- ⇒ In the **Value** dialog box (see Fig.2.4.1), define the following parameters:
 - select the **Account only nodes of supports** check box;
 - other parameters remain by default;
 - click **Select min deformations**.

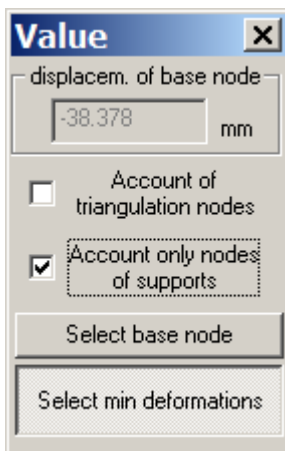


Figure 2.4.1. **Value** dialog box

Displacement contour plots along the Z-axis will be redrawn and values will be recomputed relative to the specified horizontal level – node of support with min displacement along the Z-axis (see Fig.2.4.2).

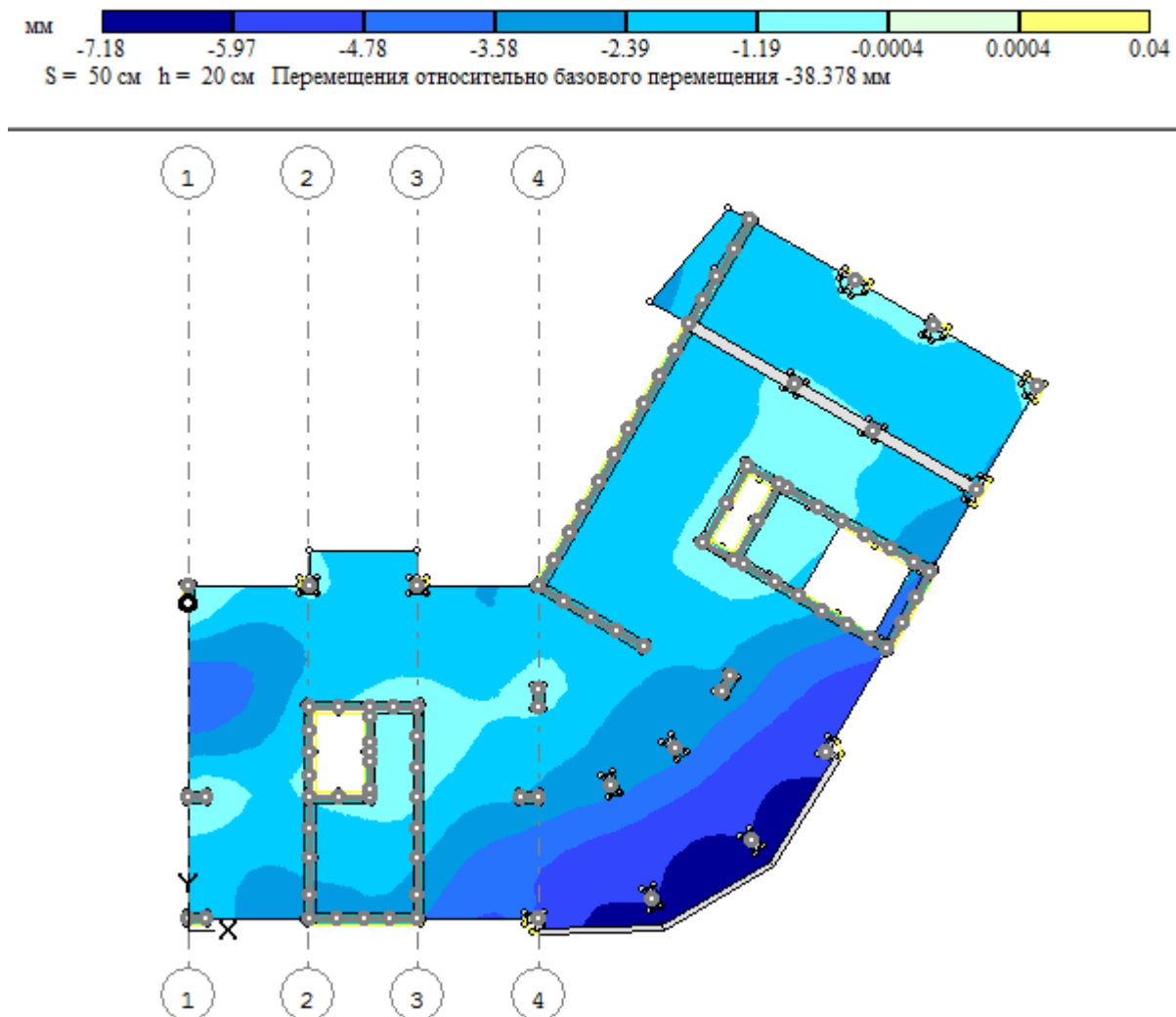






Figure 2.4.2 Displacement contour plot relative to the specified level



To print any image in the document window, on the FILE menu, click **Print** (button  on the toolbar). To preview how the image will look when you print it, on the FILE menu, click **Print preview**. When you print contour plots of displacements, forces and reinforcement, to make the image clear, it is possible to display contour lines on the screen.

- ⇒ To cancel visualization option, on the VIEW menu, point to **Display objects** and click **Non-transparent element** (button  on the VISUALIZATION toolbar). After this, the button should be not active.
- ⇒ To cancel visualization option, on the VIEW menu, point to **Display objects** and click **Imported displacements** (button  on the VISUALIZATION toolbar). After this, the button should be not active.
- ⇒ On the RESULTS menu, point to **Visualize results as** and click **Contour plots + Contour lines** (button  on the VISUALIZATION toolbar).

The model will be redrawn with the specified parameters for visualization as shown in Fig.2.4.3.

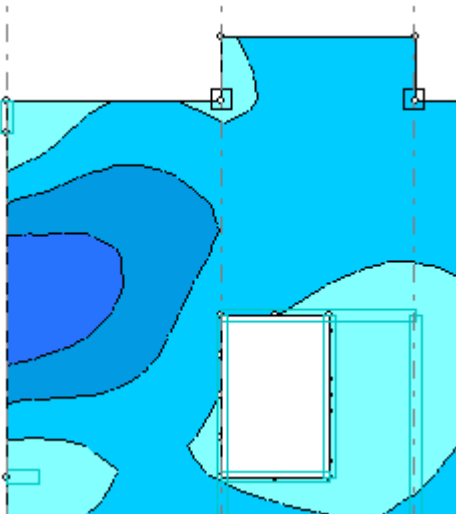



Figure 2.4.3 Contour plots and contour lines of displacements (fragment of the model)

To display diagram of values along the line segment:

- ⇒ On the RESULTS menu, click **Values along line segment** (button  on the toolbar).
- ⇒ Specify in sequence the lower and upper nodes of the slab contour along the axis 1 (that is, the line segment along the slab edge along the axis 1);
- ⇒ In the **Diagram of values along cutting plane** dialog box (see Fig.2.4.4), do the following:
 - click inside the diagram – there will be vertical line that is moved with the mouse pointer;
 - move this line to the point with coordinate $Y = 150$ cm – in the appropriate box you will see the value of displacement relative to the specified level (see Fig.2.4.2) at this point.
- ⇒ Click **OK**.

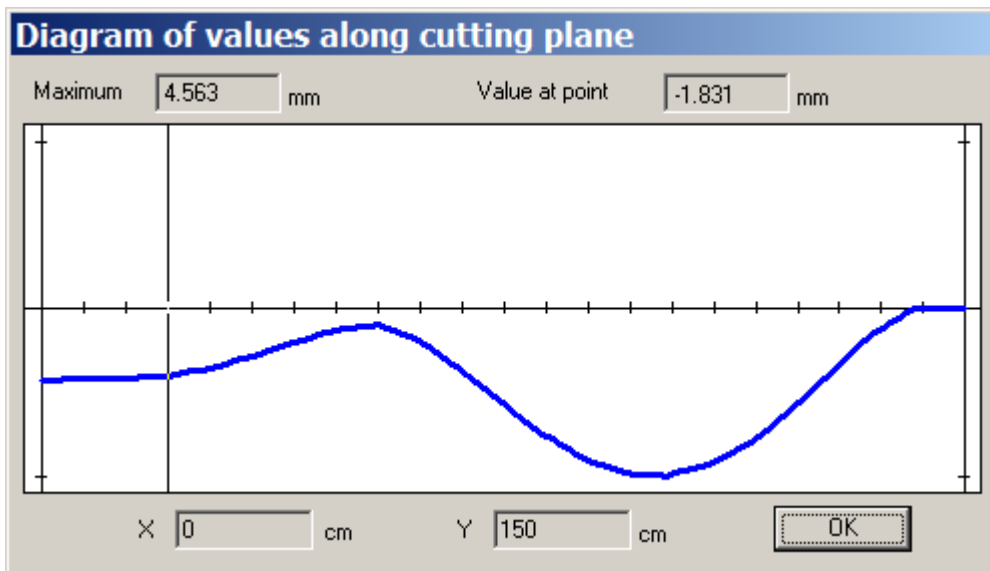





Figure 2.4.4 **Diagram of values (displacements) along cutting plane** dialog box

- ⇒ To set visualization option, on the VIEW menu, point to **Display objects** and click **Non-transparent element** (button  on the VISUALIZATION toolbar). After this, the button should become active.

- ⇒ To set visualization option, on the VIEW menu, point to **Display objects** and click **Imported displacements** (button  on the VISUALIZATION toolbar). After this, the button should become active.
- ⇒ On the RESULTS menu, point to **Visualize results as** and click **Contour plots** (button  on the VISUALIZATION toolbar).

To present contour plots of stresses and forces:

- ⇒ On the RESULTS menu, point to **Forces** and click **Moment Mx** (button  on the toolbar).

Contour plots of moments Mx will be presented for the dead load case (see Fig.2.4.5).

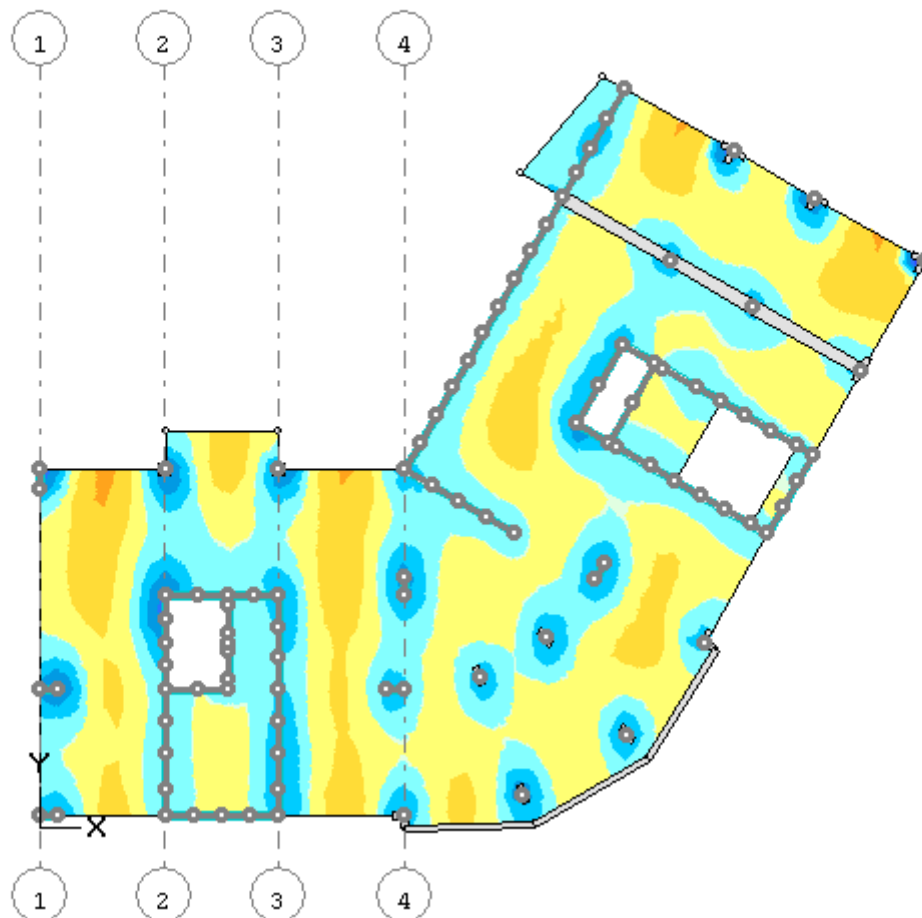



Figure 2.4.5 Contour plots of moments Mx



Contour plots of stresses and forces are generated separately by load cases. By default, design values of stresses and forces are displayed. To display normative values of stresses and forces, click appropriate button on the VISUALIZATION toolbar -  .

- ⇒ In the **Moment (Mx)** dialog box (see Fig.2.4.6), do the following:

- select live load case (Li) from the list (by default, dead load case is selected - De);

Contour plots of moments M_x will be displayed for the live load case.

- return to the dead load case – select the dead load case from the list (De).

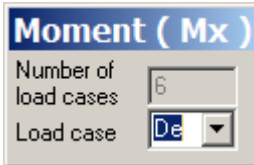








Figure 2.4.6 **Moment (Mx)** dialog box




Review and evaluate contour plots of stresses and forces in sequence:

- ⇒ On the RESULTS menu, point to **Forces** and click **Stress Nx** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Forces** and click **Stress Ny** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Forces** and click **Stress Txy** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Forces** and click **Moment My** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Forces** and click **Shear force Qx** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Forces** and click **Shear force Qy** (button  on the toolbar).

To present design reinforcement in graphical and numerical format:



*In the **SLAB** module, reinforcement in slab is considered separately – along the lower and upper edges of the slab. Transverse reinforcement and punching shear reinforcement are also considered separately.*

- ⇒ On the RESULTS menu, click **Top reinforcement** (button  on the toolbar).
- ⇒ By default, reinforcement is presented graphically, that is, the following commands are active: on the RESULTS menu, point to **Visualize reinforcement** and click **Along X** (button  on the toolbar is active) and **Along Y** (button  on the toolbar is active).

$S = 50 \text{ cm}$ $h = 20 \text{ cm}$ Top reinforcement. Maximum: $A_x = 9.3 \text{ cm}^2$, $A_y = 10 \text{ cm}^2$

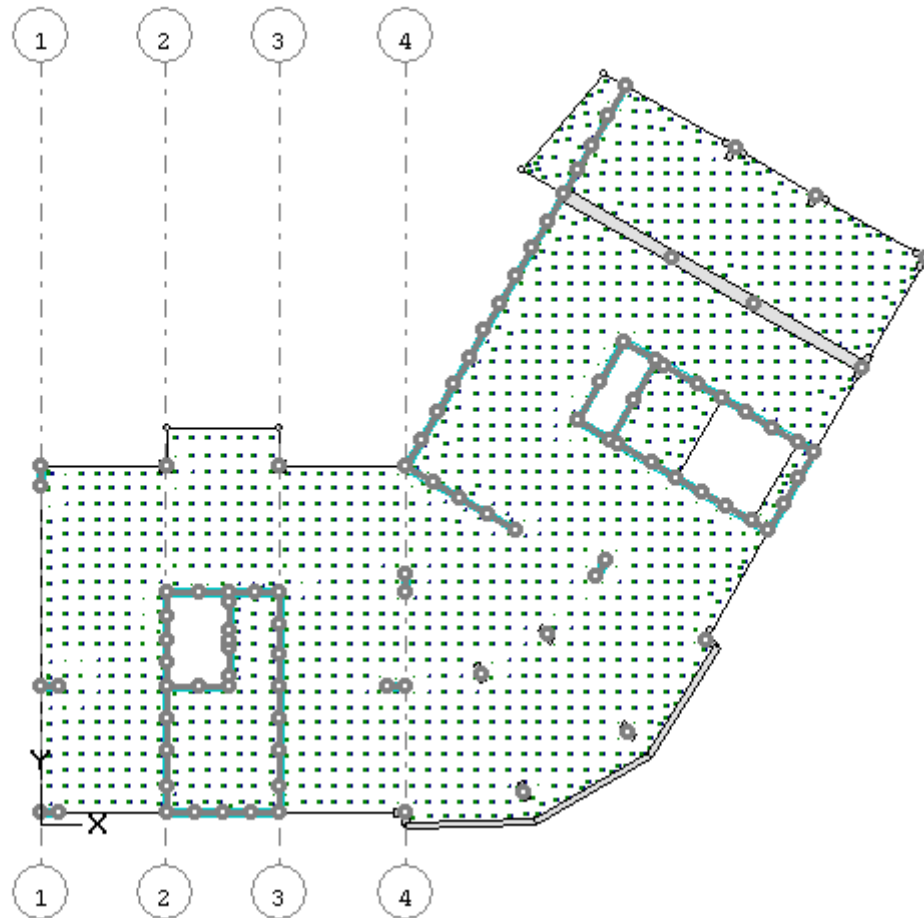






Figure 2.4.7 Graphical presentation of reinforcement along the upper edge of the slab

- ⇒ To set visualization option, on the VIEW menu, point to **Triangulation mesh** and click **Display** (button  on the VISUALIZATION toolbar). After this, the button should become active.
- ⇒ To increase the scale for model presentation, on the VIEW menu, point to **Image** and click **Zoom to selection** (button  on the toolbar).



At the centre of every element on design model of the slab the program displays lines of blue and green colours. These lines are drawn to the scale relative to max value of design area along the X and Y-axes. To present reinforcement numerically, on the RESULTS menu, point to **Visualize reinforcement** and click **Reinforcement area along X** (button  on the toolbar) and **Reinforcement area along Y** (button  on the toolbar).

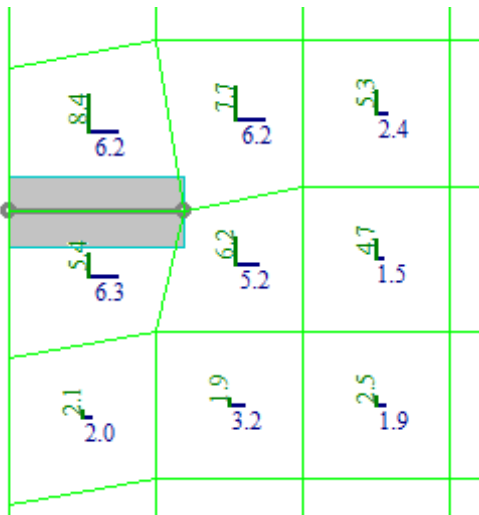




Figure 2.4.8 Reinforcement along the upper edge of the slab - in graphical and numerical formats (fragment)

- ⇒ To restore the scale of the image presentation, on the VIEW menu, point to **Image** and click **Fit in window** (button  on the toolbar).
- ⇒ To cancel visualization option, on the VIEW menu, point to **Triangulation mesh** and click **Display** (button  on the VISUALIZATION toolbar). After this, the button should be not active.

To present contour and mosaic plots of design reinforcement:



Contour and mosaic plots for design reinforcement of the slab are generated separately for directions X and Y.







- ⇒ By default, contour plots are presented – RESULTS menu / **Visualize results as / Contour plots** command is active (button  on the VISUALIZATION toolbar is active).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along X** (button  on the toolbar).



Figure 2.4.9 Contour plot of reinforcement Ax along the upper edge of the slab

- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along Y** (button  on the toolbar).
- ⇒ On the RESULTS menu, click **Bottom reinforcement** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along X** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along Y** (button  on the toolbar).

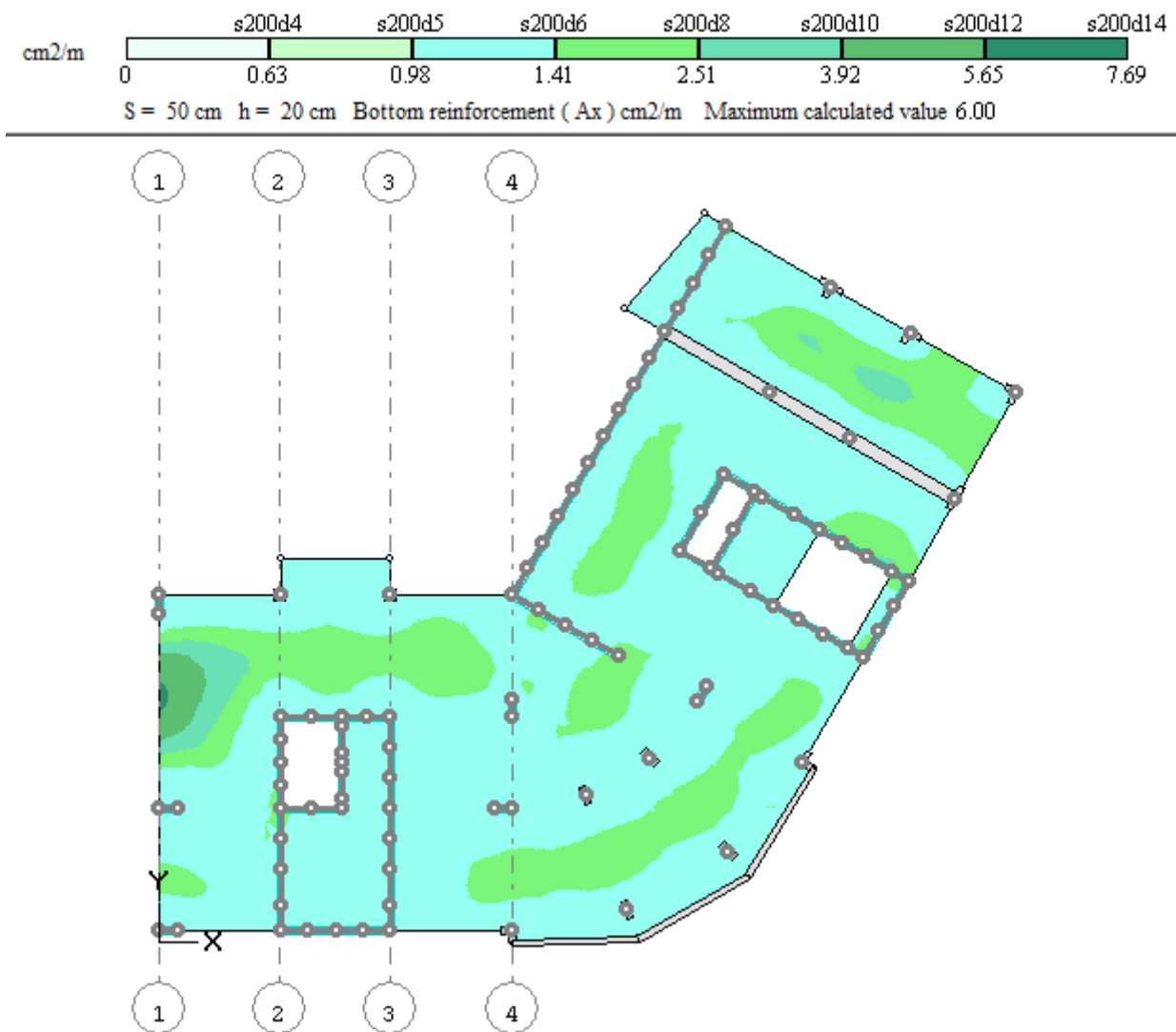



Figure 2.4.10 Contour plot of reinforcement Ay along the lower edge of the slab


⇒ On the RESULTS menu, point to **Visualize results as** and click **Mosaic plots** (button  on the VISUALIZATION toolbar).




Mosaic plots could help you diagnose FE of slab with max calculated value of design reinforcement.

⇒ On the RESULTS menu, click **Top reinforcement** (button  on the toolbar).

⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along X** (button  on the toolbar).

⇒ To cancel visualization option, on the VIEW menu, point to **Display objects** and click **Non-transparent element** (button  on the VISUALIZATION toolbar). After this, the button should be not active.

⇒ To cancel visualization option, on the VIEW menu, point to **Display objects** and click **Imported displacements** (button  on the VISUALIZATION toolbar). After this, the button should be not active.

The model will be redrawn with the specified parameters for visualization as presented in Fig.2.4.11.



To make a decision, it is necessary to check the values of design reinforcement as mosaic plot. Mosaic plots are generated according to calculated values of design reinforcement (with no smoothing) for every FE of slab.

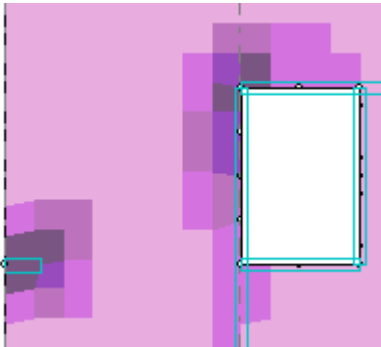







Figure 2.4.11 Mosaic plot of reinforcement Ax along the upper edge of the slab (fragment)

- ⇒ To restore visualization option, on the VIEW menu, point to **Display objects** and click **Non-transparent element** (button  on the VISUALIZATION toolbar). After this, the button should become active.
- ⇒ To restore visualization option, on the VIEW menu, point to **Display objects** and click **Imported displacements** (button  on the VISUALIZATION toolbar). After this, the button should become active.


To customize the colour palette for design reinforcement:




When nonuniform scale for reinforcement is customized, it is possible to define ranges either in numerical form or as formula. In this case, the value is defined as formula. For example, formula $s200d10$ denotes diameter of rebar $d = 10$ mm with spacing $s = 200$ mm and corresponds to area of reinforcement $3,925$ cm². Formula $5d10$ (or $k5d10$) denotes diameter of rebar $d = 10$ mm with spacing 5 rebars per 1 running metre and corresponds to area of reinforcement $3,925$ cm². It is possible to define expression like $s200d8 + s200d22$ or mixed expression like $s200d8 + 19.01$, where $19,01$ – area, cm².

- ⇒ On the RESULTS menu, point to **Visualize results as** and click **Contour plots + Contour lines** (button  on the VISUALIZATION toolbar).
- ⇒ On the RESULTS menu, click **Top reinforcement** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along X** (button  on the toolbar).

Contour and mosaic plots of design reinforcement Ax will be displayed along the upper edge of the slab.

- ⇒ On the RESULTS menu, click **Parameters of colour palette** (button  on the toolbar).
- ⇒ In the **Parameters of colour palette** dialog box (see Fig.2.4.12), do the following:
 - to select the cell with parameter $s200d4$, just click this cell (the cell will be selected with a frame);
 - to delete selected row, press DEL;
 - delete the next rows $s200d5$, $s200d6$;
 - click the cell with parameter $s200d10$, so that the cell will be selected with a frame. In the line above the table (where the cell parameter is duplicated) replace the value – define parameter $s200d8+s200d14$;

- delete the next rows *s200d12*, *s200d14*, *s200d16* (see Fig.2.4.13);
- click **Apply** ;

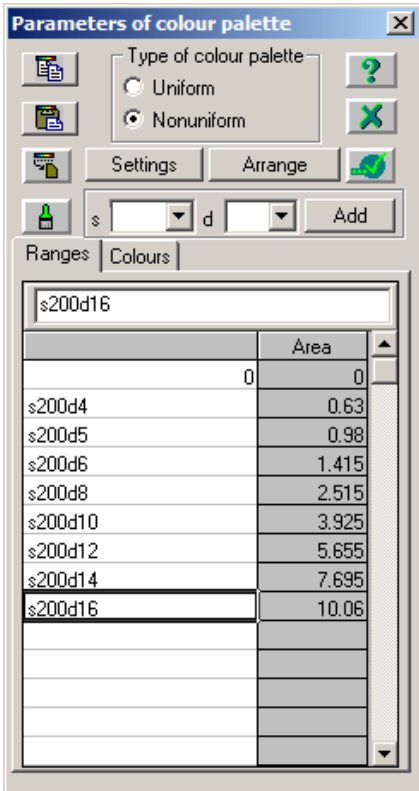


Figure 2.4.12 Parameters of colour palette dialog box

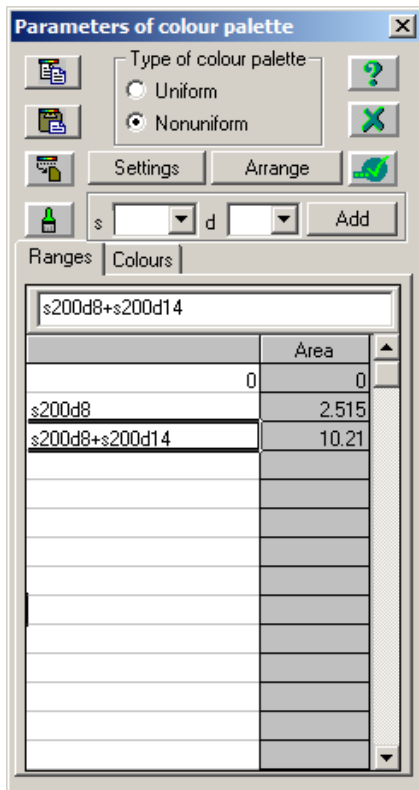


Figure 2.4.13 Parameters of colour palette dialog box (after edit options)

Contour plots will be redrawn according to new parameters of colour palette (see Fig.2.4.14).

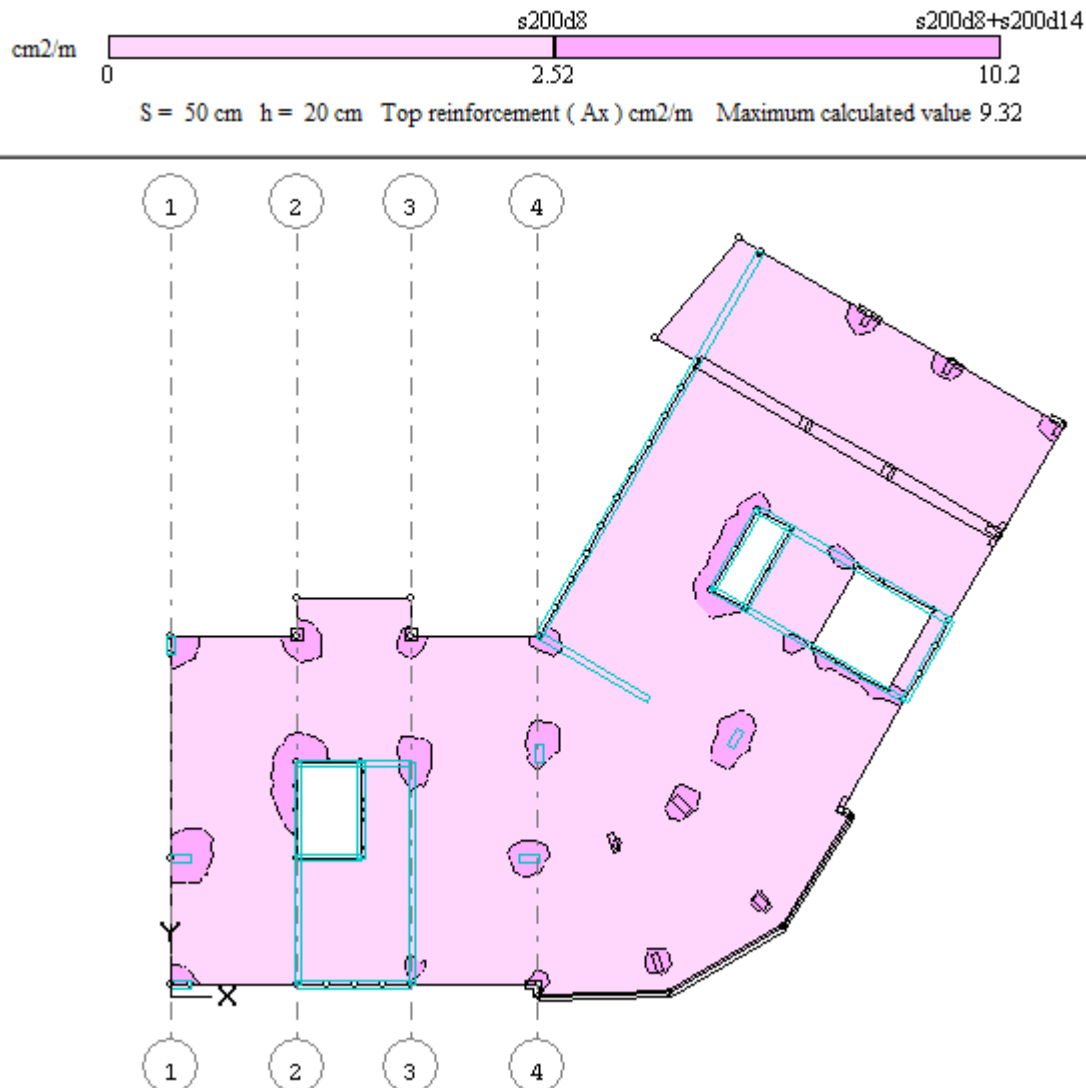



Figure 2.4.14 Contour plot of reinforcement Ax along the upper edge of the slab (after edit options)






Such presentation for contour plots of reinforcement (see Fig.2.4.14) is widely used in real design as it corresponds to accepted method for arrangement of slab reinforcement: rebars (meshes) of main reinforcement are arranged across the whole slab along the lower and upper edges of the slab, while rebars (meshes) of additional reinforcement are arranged at places where strengthening is required.



- to use the specified parameters of colour palette, select all rows with defined values in the table (to do this, hold down the CTRL key, click the first row and then the last row; the three rows in the table will be selected with a frame);
- to save the table with defined parameters of colour palette to the Clipboard, click the **Copy to Clipboard** button  in the dialog box.



You could also save the Clipboard contents to the text file on disk. Then edit the table with parameters of colour palette there, copy it to the Clipboard and paste it to the table in the **Parameters of colour palette** dialog box.

- ⇒ On the RESULTS menu, point to **Visualize results as** and click **Mosaic plots** (button  on the VISUALIZATION toolbar).
- ⇒ On the RESULTS menu, click **Top reinforcement** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along Y** (button  on the toolbar).

Mosaic plots of design reinforcement A_y will be displayed along the upper edge of the slab.

- ⇒ On the RESULTS menu, click **Parameters of colour palette** (button  on the toolbar).
- ⇒ In the **Parameters of colour palette** dialog box, do the following:
 - select all values in two columns for colour palette of reinforcement and delete them (to do this, press DEL);
 - click the first cell;
 - to paste the table values for ranges from the Clipboard, click the **Paste from Clipboard** button  in the dialog box;



If the Clipboard contains another data at the moment, then customize the colour palette once again as described above (see Fig.2.4.13).



*To return to initial settings for colour palette, then click **Settings** and in the **Settings for palette** dialog box define initial spacing for rebars 200 mm and click **OK**.*

- click **Apply** .

Mosaic plots will be redrawn according to new parameters of colour palette.

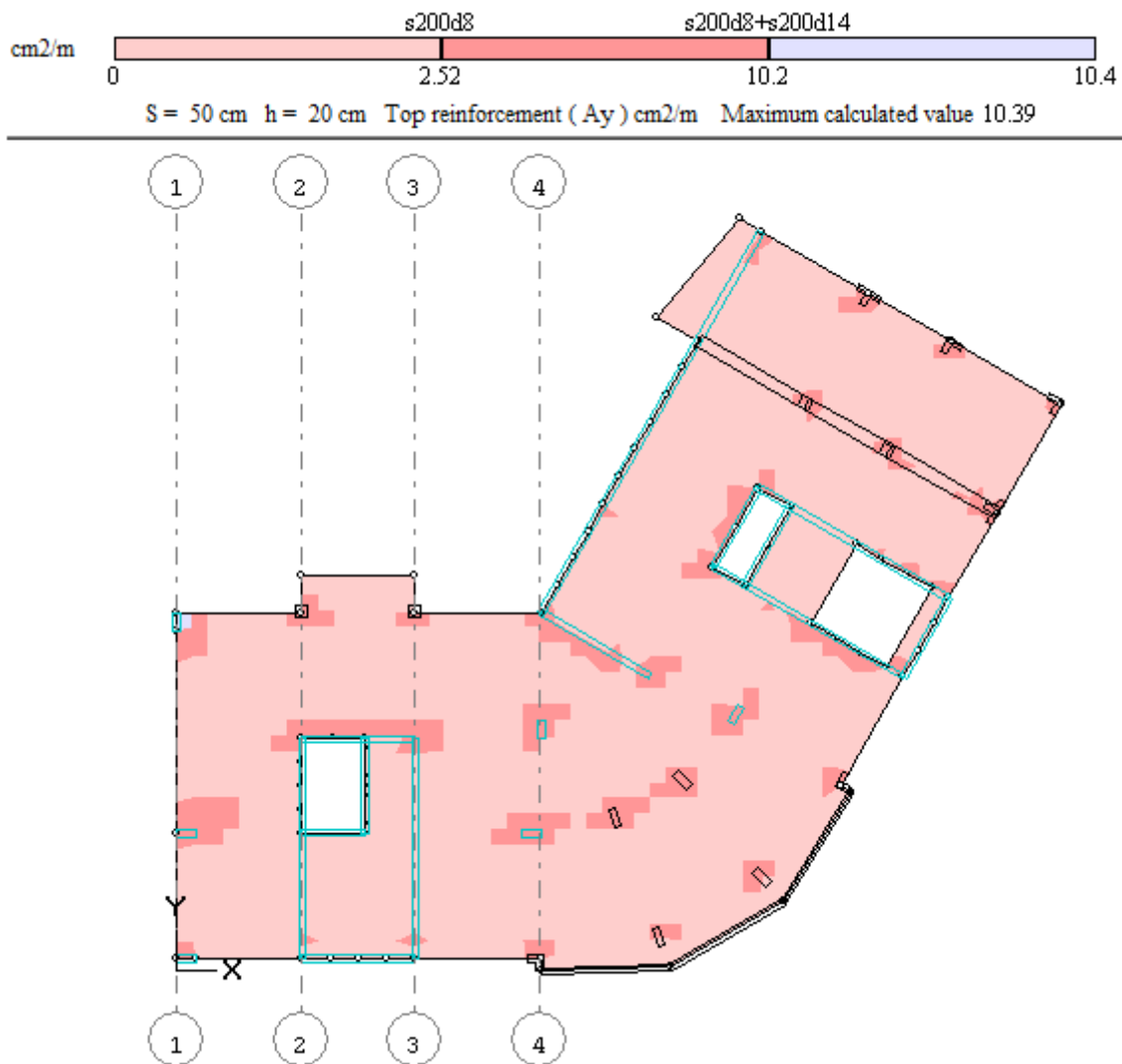


Figure 2.4.15 Mosaic plot of reinforcement A_y along the upper edge of the slab (after edit options)



On the model you should evaluate location and dimensions of the range that has no description by formula (see Fig.2.4.15 and Fig.2.4.16). If required, modify parameters of colour palette in the table. In this case you could increase diameter of additional layer of reinforcement in order to make it greater than max calculated value – define one more row s200d8+s200d16 in the table in the **Parameters of colour palette** dialog box.

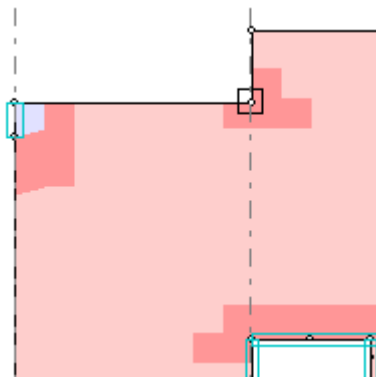






Figure 2.4.16 Mosaic plot of reinforcement A_y along the upper edge of the slab (fragment)



To present mosaic plots of design transverse reinforcement:



To display transverse reinforcement (either graphically or numerically or as contour/mosaic plots), select the type of reinforcement. To do this, on the **RESULTS** menu, click **Transverse reinforcement** (button  on the toolbar).

- ⇒ On the **RESULTS** menu, point to **Visualize results as** and click **Mosaic plots** (button  on the **VISUALIZATION** toolbar).
- ⇒ On the **RESULTS** menu, click **Transverse reinforcement** (button  on the toolbar).
- ⇒ On the **RESULTS** menu, point to **Visualize reinforcement** and click **Contour plots along X** (button  on the toolbar).

Mosaic plots of design transverse reinforcement will be presented per 1m^2 according to the **Analysis of transverse reinforcement per 1 sq.m.** check box option selected in the **Analysis parameters** dialog box (see Fig.2.2.2 earlier).

- ⇒ On the **RESULTS** menu, click **Parameters of colour palette** (button  on the toolbar).
- ⇒ In the **Parameters of colour palette** dialog box, do the following:
 - click **Settings**;
 - in the **Settings for palette** dialog box, do the following:
 - in the **Present** area, click **Numerically**;
 - click **OK**;
 - define number of colours as equal to 6 (see Fig.2.4.17);
 - click **Apply** .
 - ⇒ Close the **Parameters of colour palette** dialog box.

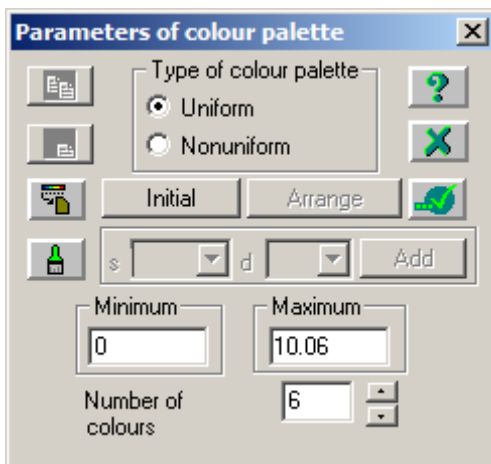


Figure 2.4.17 **Parameters of colour palette** dialog box (after edit options)

Mosaic plots will be redrawn according to new parameters of colour palette (see Fig.2.4.18).

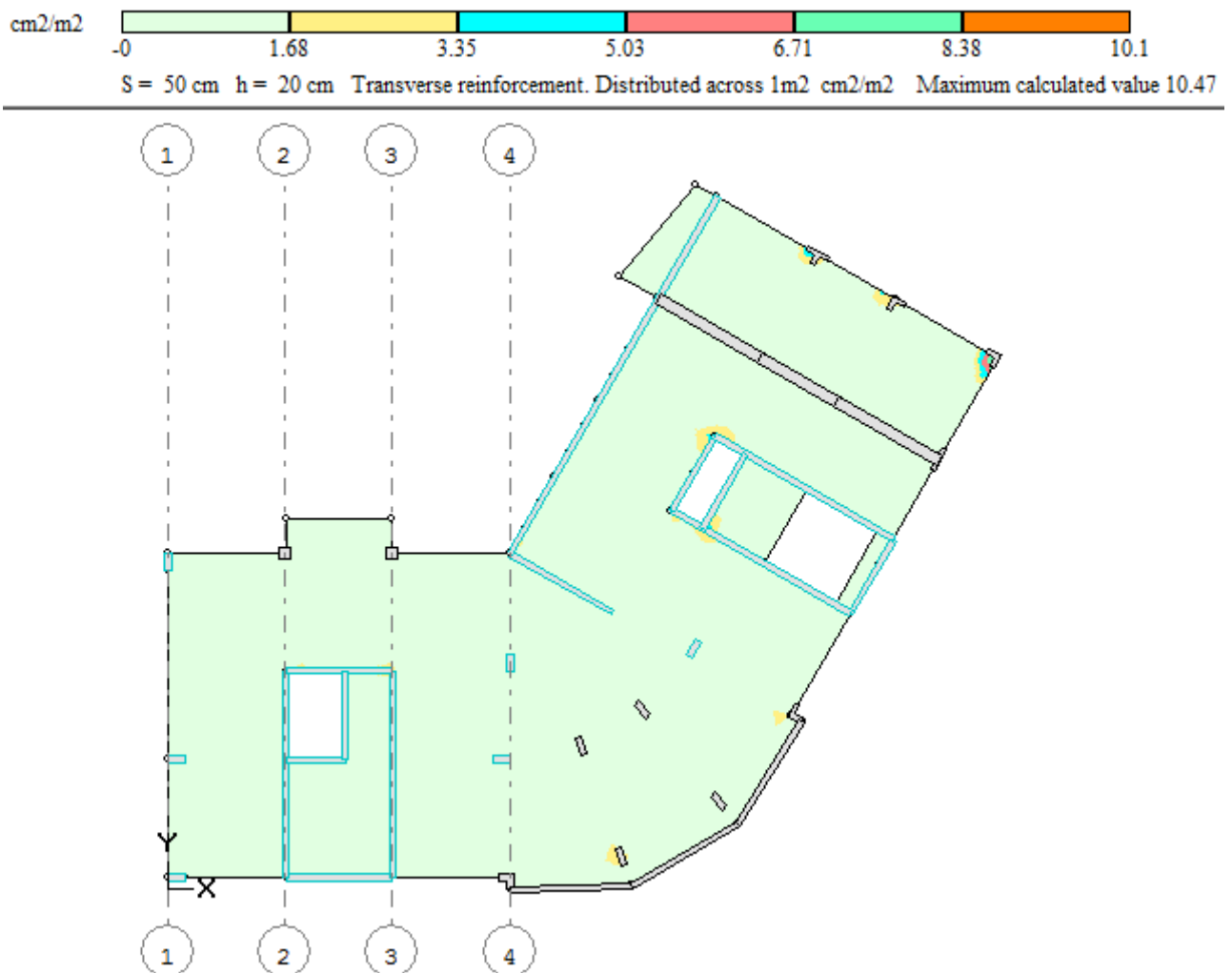





Figure 2.4.18 Mosaic plot of transverse reinforcement per 1m^2

- ⇒ To hide presentation of transverse reinforcement, on the RESULTS menu, click **Transverse reinforcement** (button  on the toolbar) once again in order to make this command not active.

Step 5. Punching shear analysis



Punching shear load is computed during main analysis as envelope design reactions in columns and walls R_z for all load cases. But punching shear analysis is not included into main analysis procedure, it should be carried out separately. In punching shear analysis, punching shear contours are generated automatically. Section shape for columns (walls) and slab thickness at the place of support are considered when contours are generated. Contours are automatically 'cut' if they are located at the edge of floor slab. If there are closely located openings, punching shear contour may be modified.

- ⇒ To cancel visualization option, on the VIEW menu, point to **Display objects** and click **Imported displacements** (button  on the VISUALIZATION toolbar). After this, the button should be not active.
- ⇒ To display contour plots of reactions R_z , on the RESULTS menu, point to **Forces** and click **Reaction R_z** (button  on the toolbar).



Contour (mosaic) plots of reactions are generated by load cases. Design values of reactions are displayed.

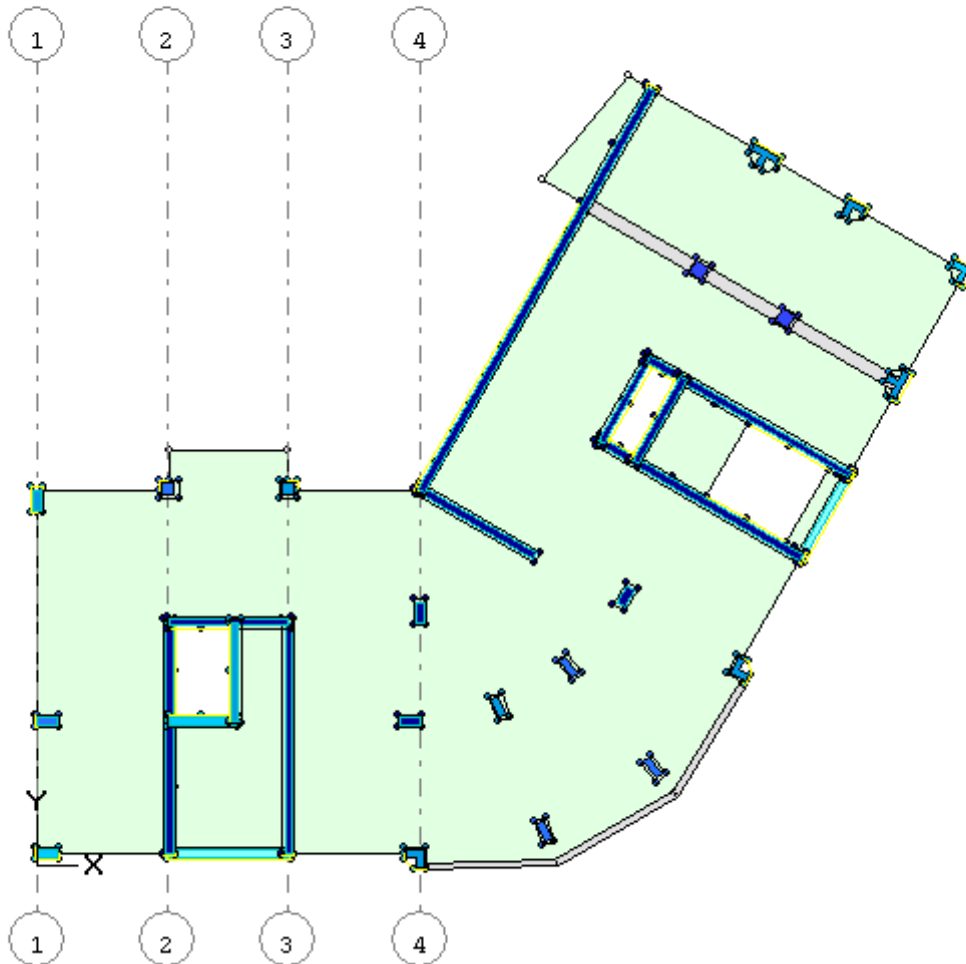
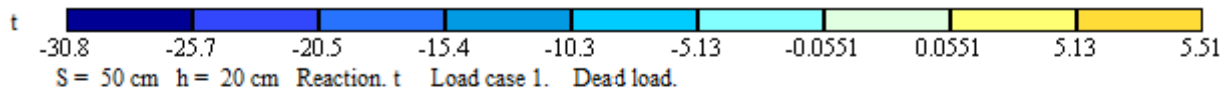


Figure 2.5.1 Mosaic plot of reactions R_z and punching shear contours

⇒ On the ANALYSIS menu, click **Punching shear analysis** (button  on the toolbar).





If the **Refresh punching shear contour** option is selected in the **Punching shear analysis** dialog box, then automatically generated contours will be considered in analysis.



To display area of transverse reinforcement it is advisable to define colour palette in numerical presentation (see settings for colour palette of reinforcement).

⇒ On the RESULTS menu, click **Transverse reinforcement** (button  on the toolbar).

Mosaic plot of transverse punching shear reinforcement will be presented.

- ⇒ To set visualization option, on the RESULTS menu, point to **Visualize reinforcement** and click **Area of punching shear reinforcement** (button  on the VISUALIZATION toolbar).
- ⇒ To set visualization option, on the RESULTS menu, point to **Visualize reinforcement** and click **Bearing capacity coefficient** (button  on the VISUALIZATION toolbar).

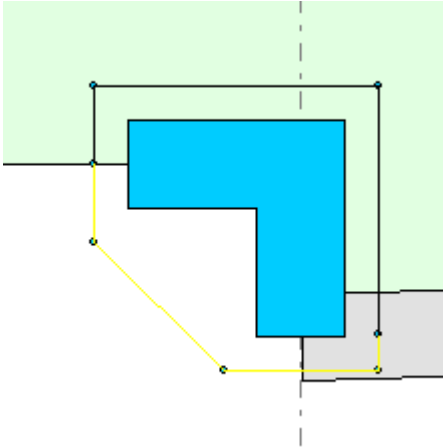


Figure 2.5.2 Mosaic plot of reactions R_z and punching shear contour (fragment)

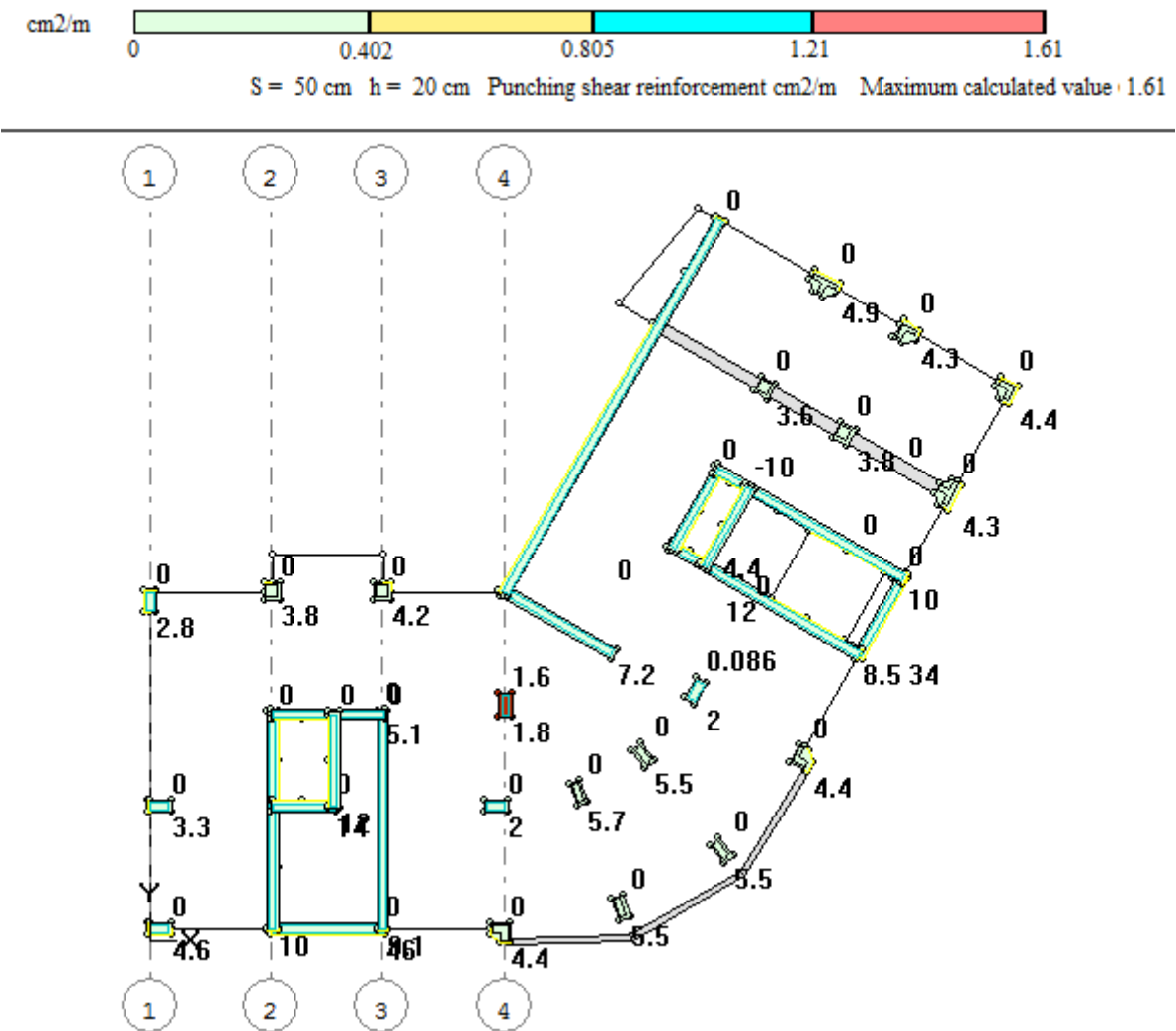



Figure 2.5.3 Area of transverse punching shear reinforcement and bearing safety coefficient

i If the bearing safety coefficient in punching shear is less than 1.0, it means that the program could not determine reinforcement and the strength of slab in punching shear is not adequate. In this case it is necessary to increase slab thickness or class of concrete. If the bearing safety coefficient is greater than 2.0, then transverse reinforcement is not necessary. Bearing safety coefficient is displayed at the bottom to the right of punching shear contour (see Fig.2.5.4). Transverse punching shear reinforcement is presented in cm² per the area limited by punching shear contour. Area is displayed at the top to the right of punching contour (see Fig.2.5.4). Envelope reaction of punching shear is displayed at the bottom to the left of punching contour. To display this value, on the RESULTS menu, point to **Visualize reinforcement** and click **Reaction of punching shear** (button  on the VISUALIZATION toolbar).

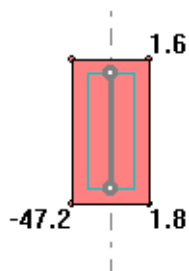




Figure 2.5.4 Results of punching shear analysis (fragment)

- ⇒ To cancel visualization option, on the VIEW menu, point to **Display objects** and click **Punching contour** (button  on the VISUALIZATION toolbar).

Step 6. Review and evaluate the report file

- ⇒ On the RESULTS menu, point to **Report** and click **Save HTML and open** (button  on the toolbar).



Save and open commands are presented as one button  on the toolbar. To reveal all buttons, click this button with the pointer and hold it down.

- ⇒ In the **Parameters of report** dialog box (see Fig.2.6.1), remain all parameters as defined by default.
 ⇒ Click **OK**.

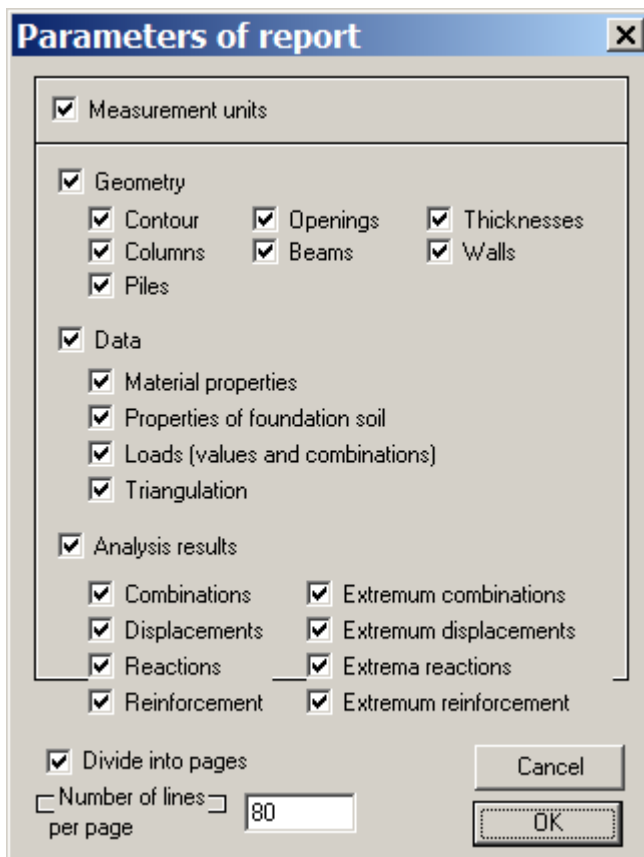


Figure 2.6.1 **Parameters of report** dialog box

Report file *Slab1.htm* will be saved to the disk in the **Notes** subdirectory in the base **MONOMAKH-SAPR** directory. This file will be automatically opened in Internet Explorer window (or any similar program).



Report file contains several tables. It is possible to preview and print them.

- ⇒ In the open HTML file, do the following:
- click **Punching** link – table with analysis results for punching shear will be displayed (see Fig.2.6.2);

⇒ When you preview and print the report file, close it in Internet Explorer.

PUNCHING										
No.	X	Y	Column type	B ovr.	H ovr.	Perimeter	H slab	Reaction	Reinforcement	Safety factor
1	1399.4	442.2	rectangle	24.0	60.0	234.1	17.0	-15.3	0.0	5.7
2	1610.0	563.8	rectangle	24.0	60.0	232.8	17.0	-15.9	0.0	5.5
3	1536.2	66.3	rectangle	24.0	60.0	234.1	17.0	-15.9	0.0	5.5

Figure 2.6.2 Tables from HTML file of the report (fragment)

Step 7. Design of slab

To design slab:







Slab is designed in the interactive mode. For the specified zones, the program determines spacing and diameter of longitudinal rebars along the upper and lower edges of the slab. Remaining reinforcement is checked. Remaining area of reinforcement is determined for every FE of slab as difference between design area and area provided by meshes and rebars arranged at certain zones. Transverse reinforcement is not arranged.

To present max areas of reinforcement at certain zone:



Max values of design reinforcement A_x , A_y at certain zone are displayed in order to estimate values of design reinforcement and to recompute values of design reinforcement in radial directions.

- ⇒ On the RESULTS menu, click **Bottom reinforcement** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Along X** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Along Y** (button  on the toolbar).
- ⇒ On the DESIGN menu, point to **Select max A_x , A_y** and click **Define contour** (button  on the toolbar).
- ⇒ In the dialog box, define the following parameters:
 - divide zone into cells 1 x 3.
- ⇒ Specify (on the model) node of intersection between lines 1 and A according to plan of the structure.
- ⇒ Specify (on the model) node of intersection between lines 2 and A.
- ⇒ Specify (on the model) node of intersection between lines 2 and D.
- ⇒ Specify (on the model) node of intersection between lines 1 and D.

Contour with three cells will be displayed. In every cell, max area of reinforcement will be presented graphically.



The program takes into account all areas computed by design combinations of forces at centres of finite elements (of slab) located at certain cell. At the centre of every cell you will see max areas A_x , A_y . Direction of reinforcement is shown by lines that connect midsides of cells.

- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Reinforcement area along X** (button ).

- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Reinforcement area along Y** (button ).

In addition, values for area of reinforcement will be displayed (see Fig.2.7.1).

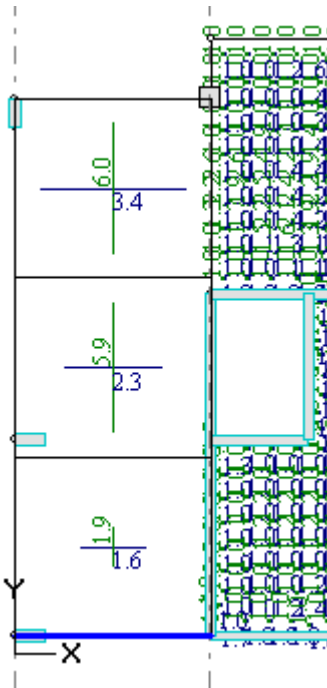


Figure 2.7.1 Max reinforcement at certain zone presented graphically and numerically (fragment)

Assign one more zone for presentation max area. The **Define contour** command (DESIGN menu / **Select max Ax, Ay**) is still active:

- ⇒ In the dialog box (that is still open), define the following parameters:
- divide zone into cells 1 x 1.



If figures for areas of reinforcement in FE of slab do not enable you to select nodes on the model, you could temporarily hide numerical presentation of reinforcement.

- ⇒ Specify (on the model) node of intersection between lines I and A according to plan of the structure.
- ⇒ Specify (on the model) node of intersection between lines II and A.
- ⇒ Specify (on the model) node of intersection between lines II and B.
- ⇒ Specify (on the model) node of intersection between lines I and B.

Contour with one cell will be presented. In this cell you will see max value for area of reinforcement along axes different from global coordinate axes of the model X and Y.



When contour plots of design reinforcement are displayed it is helpful to estimate values of max reinforcement.

- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Contour plots along Y** (button  on the toolbar).

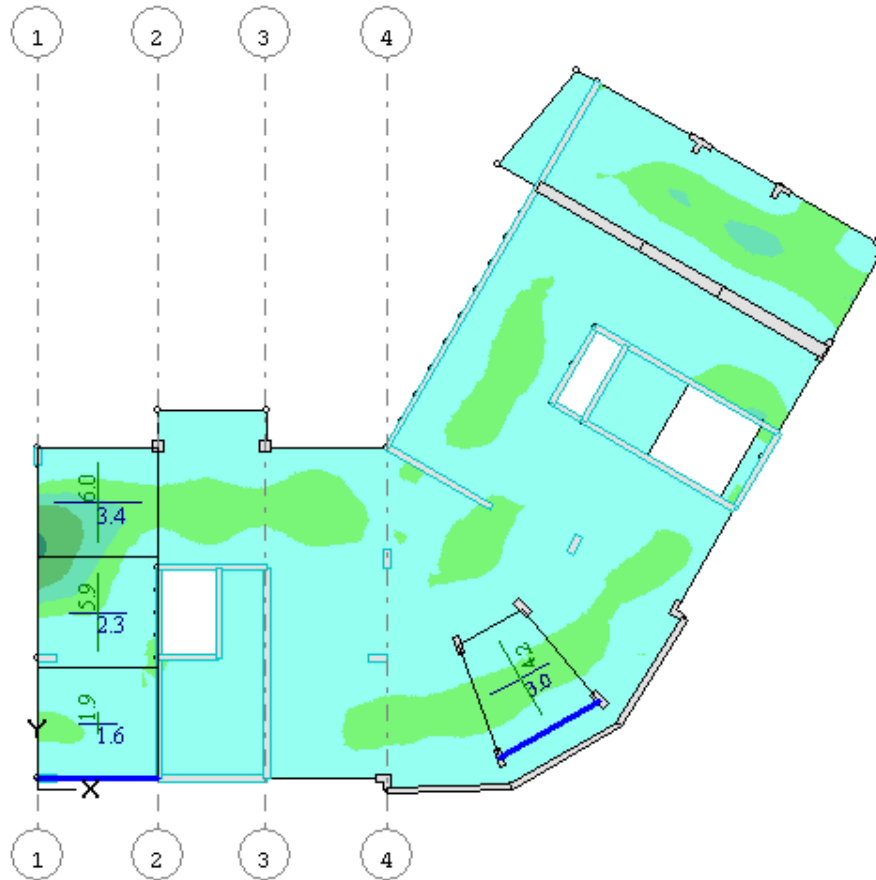
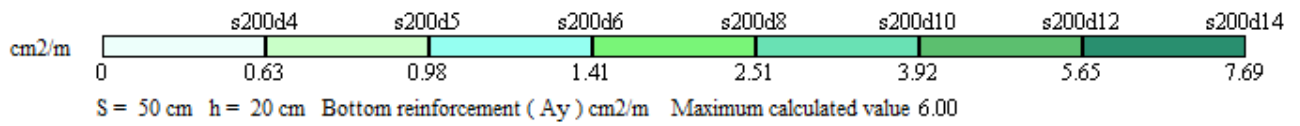



Figure 2.7.2 Max reinforcement at certain zones on contour plot of reinforcement Ay


⇒ To cancel visualization option, on the Design menu, point to **Select max Ax, Ay** and click **Display contour** (button  on the toolbar).

To arrange rebars of main reinforcement at certain zone:



Defined zone of reinforcement with rebars may be identified either as main reinforcement or additional one. As a rule, zone defined for the whole contour of slab should be identified as main reinforcement. Zones of additional reinforcement are defined separately so that remaining reinforcement is reduced to zero.

Assign the zone with pattern of rebars (main reinforcement) to the whole contour of slab:

- ⇒ On the DESIGN menu, point to **Pattern for wire mesh and rebars** and click **Rectangular contour** (button  on the toolbar).
- ⇒ In the **Assign reinforcement zone** dialog box (see Fig.2.7.3), do the following:
 - select from the list diameter of rebars arranged orthogonally to the base line 8 mm;
 - select from the list diameter of rebars arranged along the base line 8 mm;

- other parameters remain by default;
- click **Whole contour**.

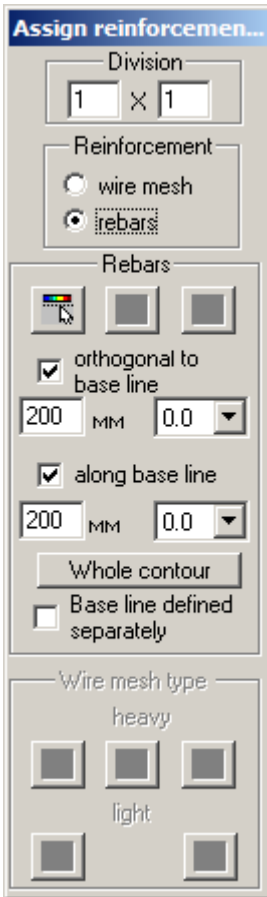


Figure 2.7.3 **Assign reinforcement zone** dialog box

- ⇒ In the **Assign wire mesh and rebars** dialog box (see Fig.2.7.4), do the following:
 - make sure that **Main** option is selected;
 - click **Assign** for the 1st layer of reinforcement;
 - other parameters remain by default.
- ⇒ Click **OK**.

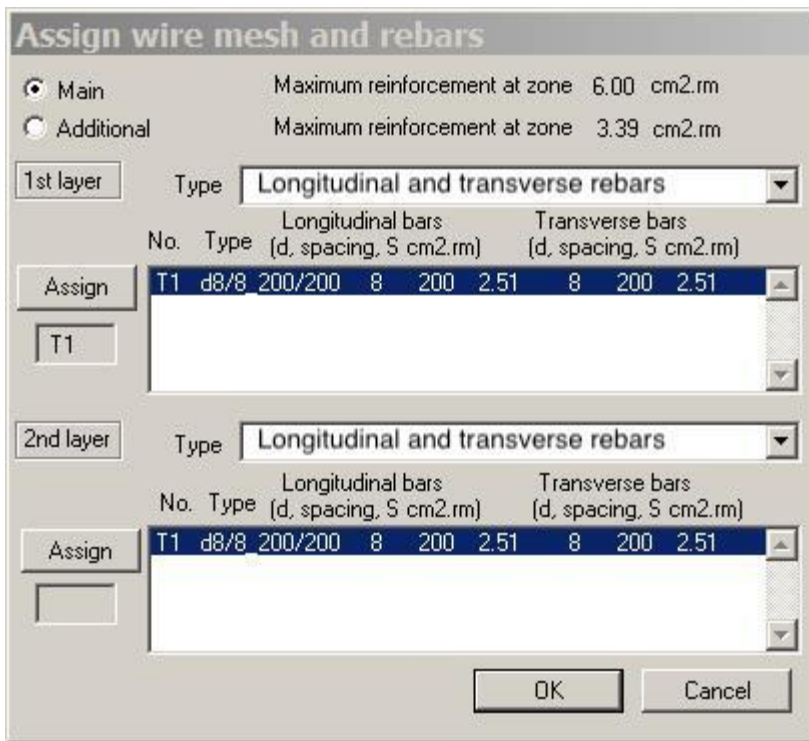


Figure 2.7.4 Assign wire mesh and rebars dialog box

Pattern of rebars *s200d8* along X and *s200d8* along Y (main reinforcement) is defined to the whole contour of the slab.

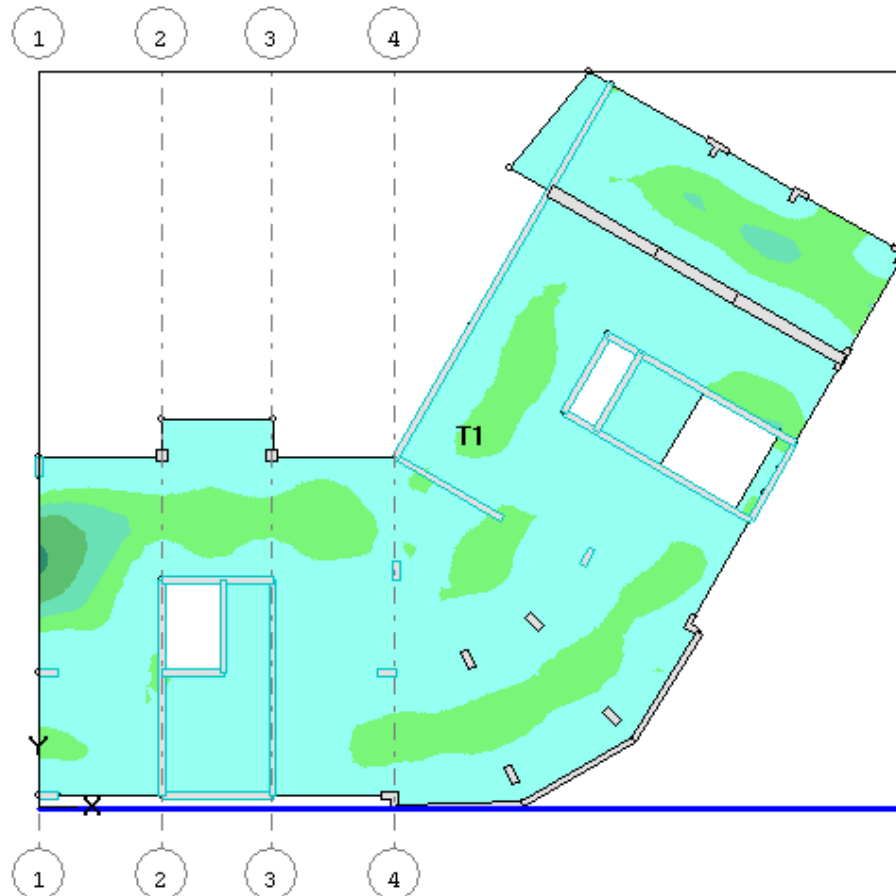
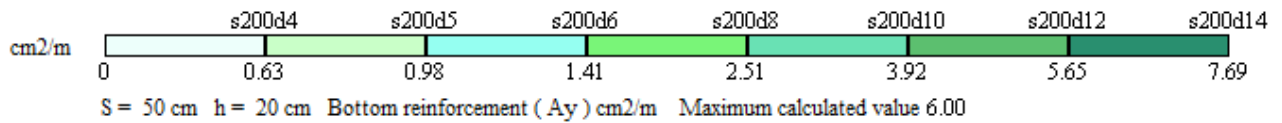


Figure 2.7.5 Pattern of rebars (main reinforcement) with contour plots of reinforcement A_y (on the background)






Base line defines orientation of rebars within the pattern: longitudinal rebars will be arranged orthogonally to this line while transverse rebars - along this line. When you arrange rebars, the base line may be defined arbitrarily relative to the zone location. Base line will be coloured blue on the model.

To check remaining reinforcement:



Remaining reinforcement is presented when areas of reinforcement A_x, A_y are displayed either graphically or numerically for selected type of reinforcement – along the upper or lower edge of the slab (top or bottom reinforcement).

- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Along X** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Along Y** (button  on the toolbar).
- ⇒ On the RESULTS menu, point to **Visualize reinforcement** and click **Remaining reinforcement** (button  on the toolbar).

$S = 50 \text{ cm}$ $h = 20 \text{ cm}$ Bottom reinforcement. Maximum: $A_x = 3.4 \text{ cm}^2$, $A_y = 6 \text{ cm}^2$,

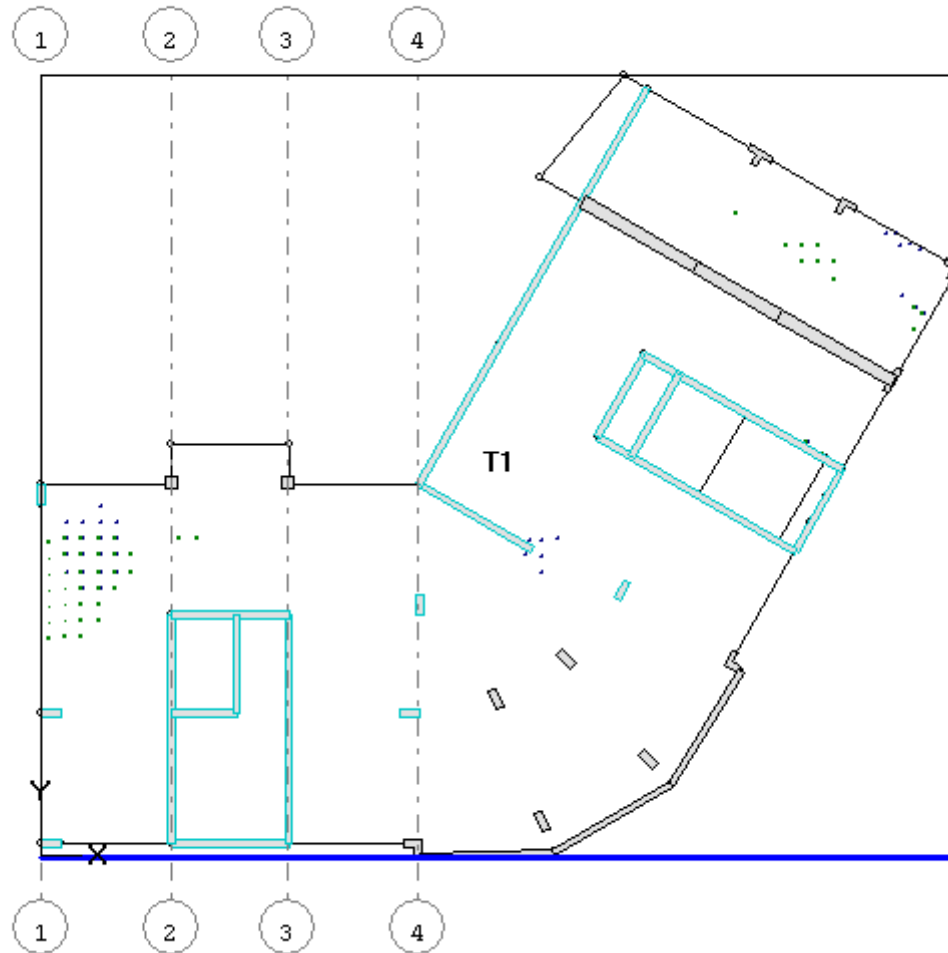



Figure 2.7.6 Remaining reinforcement along the lower edge of the slab

To arrange rebars of additional reinforcement at the specified zones:

Assign the zone with pattern of rebars (additional reinforcement):

- ⇒ On the DESIGN menu, point to **Pattern for wire mesh and rebars** and click **Rectangular contour** (button  on the toolbar).
- ⇒ In the **Assign reinforcement zone** dialog box, do the following:
 - select from the list diameter of rebars arranged orthogonally to the base line 0 mm;
 - select from the list diameter of rebars arranged along the base line 0 mm;
 - other parameters remain by default.



If only spacing of rebars is defined, then required diameter will be selected according to required (in this case, remaining) area of reinforcement.

- ⇒ Specify (on the model) node of intersection between lines 1 and B according to plan of the structure.
- ⇒ Specify (on the model) node of intersection between lines 2 and B.

- ⇒ Specify (on the model) node of intersection between lines 2 and D.
- ⇒ In the **Assign wire mesh and rebars** dialog box (see Fig.2.7.7), do the following:
 - select option **Additional**;
 - click **Assign** for the 1st layer of reinforcement;
 - other parameters remain by default.
- ⇒ Click **OK**.

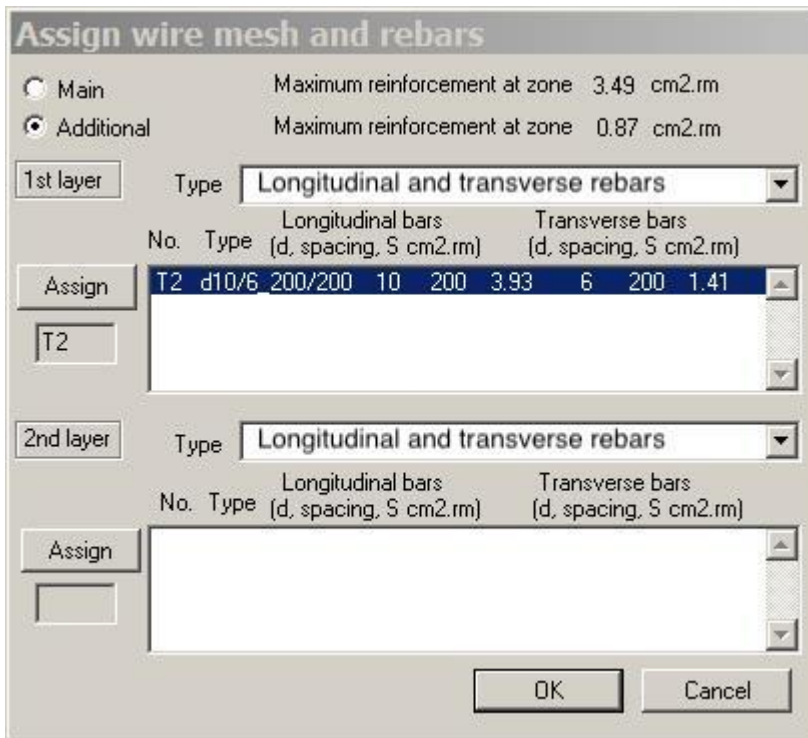


Figure 2.7.7 **Assign wire mesh and rebars** dialog box

Pattern of rebars (additional reinforcement) is defined.

$S = 50 \text{ cm}$ $h = 20 \text{ cm}$ Bottom reinforcement. Maximum: $A_x=3.4 \text{ cm}^2$, $A_y=6 \text{ cm}^2$,

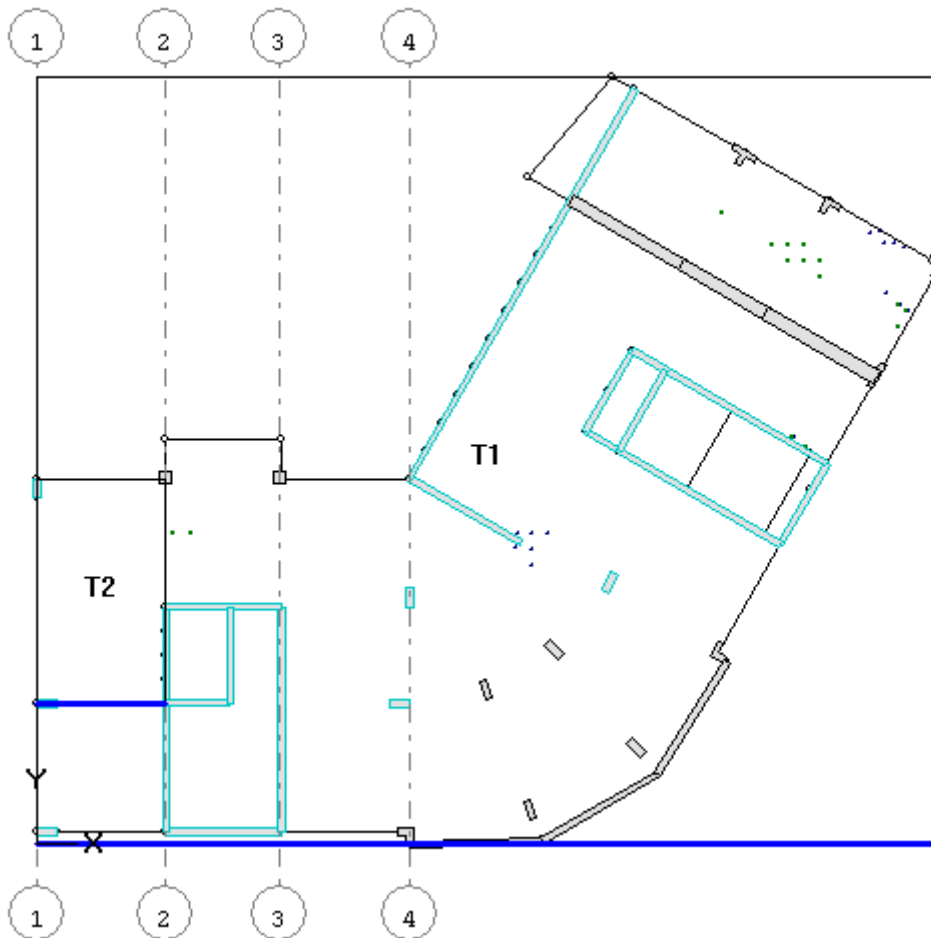






Figure 2.7.8 Patterns of rebars (main and additional reinforcement) with check for remaining reinforcement






To present parameters of the specified pattern zones for reinforcement, on the **DESIGN** menu, click **Edit pattern zone** (button  on the toolbar). To delete the specified pattern zones, on the **DESIGN** menu, click **Delete pattern zone** (button  on the toolbar). When this command is active, just specify the pattern zone on the model. To display defined pattern zones for reinforcement, on the **DESIGN** menu, click **Pattern zones for main wire mesh and rebars** (button  on the toolbar) and **Pattern zones for additional wire mesh and rebars** (button  on the toolbar).

To arrange wire meshes within the specified zone:



To define dimensions for wire mesh, on the **OPTIONS** menu, click **Mesh dimensions for design**.

- ⇒ On the **RESULTS** menu, click **Top reinforcement** (button  on the toolbar).
- ⇒ On the **DESIGN** menu, point to **Pattern for wire mesh and rebars** and click **Rectangular contour** (button  on the toolbar).
- ⇒ In the **Assign reinforcement zone** dialog box (see Fig.2.7.9), do the following:

- in the **Reinforcement** area, select **Wire mesh** option (by default, **Rebars** option is selected);
- divide zone into cells 3 x 3;
- select the **Wire mesh type** as **heavy** and click **Reinforcement in both directions** .

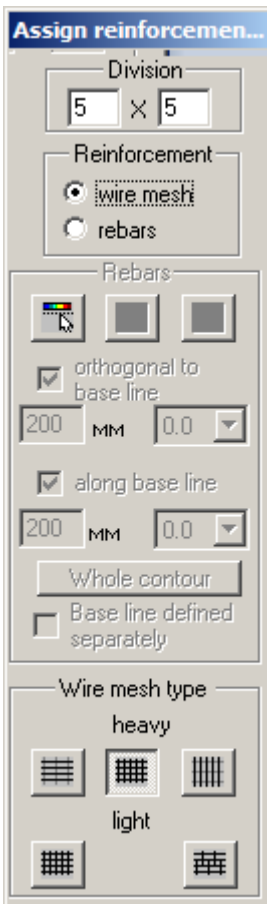


Figure 2.7.9 **Assign reinforcement zone** dialog box



It is less helpful to assign wire meshes on the specified zones as you could not pre-define diameters of rebars and that's why it is rather difficult to divide reinforcement into the main and additional one.

- ⇒ Specify (on the model) node of intersection between lines 1 and A according to plan of the structure.
- ⇒ Specify (on the model) node of intersection between lines 4 and A.
- ⇒ Specify (on the model) node of intersection between lines 4 and D.
- ⇒ In the **Assign wire mesh and rebars** dialog box (see Fig.2.7.10), do the following:
 - from the first list select the wire mesh *s200d12 – s200d12*;
 - click **Assign** for the 1st layer of reinforcement;
 - from the second list select the wire mesh *s200d12 – s200d8*;
 - click **Assign** for the 2nd layer of reinforcement;
 - other parameters remain by default.
- ⇒ Click **OK**.

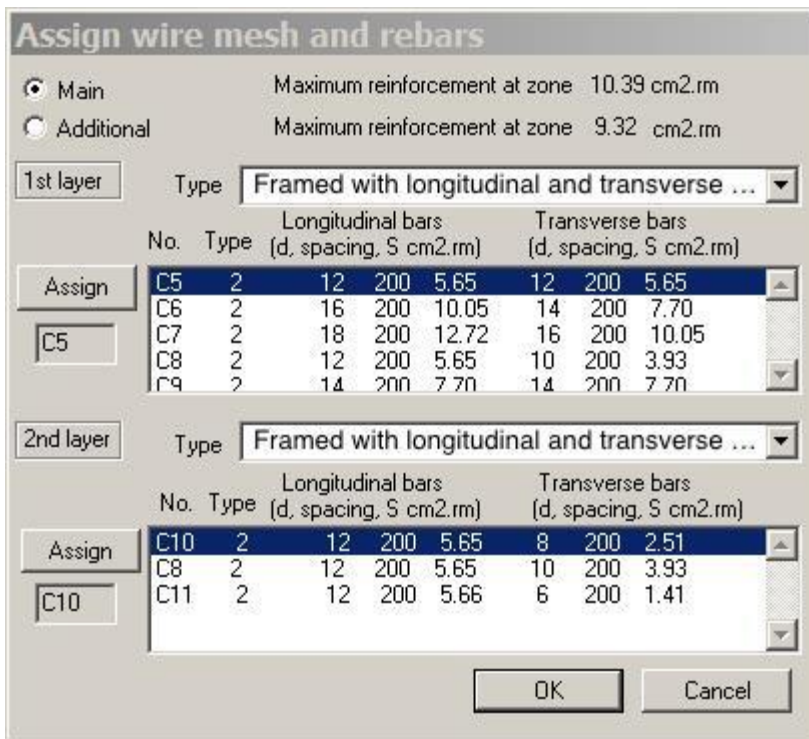


Figure 2.7.10 Assign wire mesh and rebars dialog box



Zone with double-layer reinforcement is indicated on the model with inclined hatching.

Two patterns of reinforcement will be defined.

$S = 50 \text{ cm}$ $h = 20 \text{ cm}$ Top reinforcement. Maximum: $A_x = 9.3 \text{ cm}^2$, $A_y = 10 \text{ cm}^2$

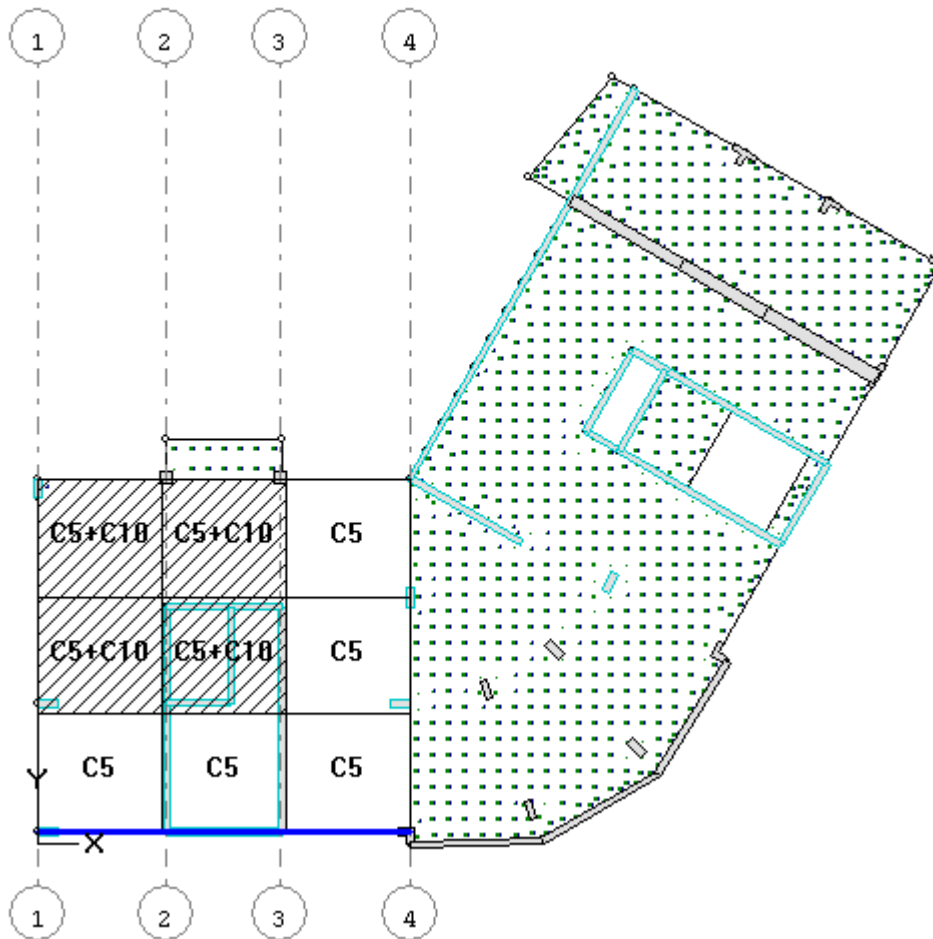



Figure 2.7.11 Patterns of reinforcement with check of remaining reinforcement


To save results of design procedure:



On the **FILE** menu, click **Save** (button  on the toolbar). In this case, data about design procedure will be also saved to the *.plt file.

Step 8. Drawing of slab

Drawing of slab reinforced with rebars

⇒ On the **RESULTS** menu, point to **Drawing** and click **Drawing of rebars** (button  on the toolbar).



Commands for drawing are presented as one button on the toolbar. To reveal all buttons, click this button with the pointer and hold it down.



The **DRAWING OF SLAB** module will be started (in version - reinforcement as rebars).



The drawing is supposed to contain several fragments: pattern for rebars and wire meshes, specifications, title block, etc. Every fragment has its own borders on the sheet of drawing. The scale of the fragment presentation (except fragments with tables or texts) is determined with dimensions of such borders. It is possible to modify dimensions of fragment borders, to move, delete and add new fragments from the current list of fragments. You could also modify the sheet size, colour and dimensions of separate elements of drawing (e.g. height of characters), modify location of leader lines, etc.

Pattern of rebars (main reinforcement)



The pattern of rebars in main reinforcement along the lower edge of the slab is displayed by default (buttons  and  on the toolbar are active). Please note that by default complete specification and list of steel consumption are displayed, that is, you will see data about all defined rebars in both main and additional reinforcement. If required, it is possible to hide the data about rebars that are not visible on the drawing at the moment.





- ⇒ On the VIEW menu, click **Specification for all reinforcement** (button  on the toolbar – as result, the button should become not active).

Data in the specification table and in list of steel consumption will be modified.

To edit the drawing:

- ⇒ On the OPTIONS menu, click **Edit drawing** (button  on the toolbar).

To edit location of rebars, link-lines of leaders and leaders themselves:

- ⇒ On the OPTIONS menu, click **Move presentation of reinforcement** (button  on the toolbar).
- ⇒ Drag any rebar from the edge of the slab.
- ⇒ In the same way, drag the rest of rebars from the slab edges so that pattern of the main reinforcement becomes more visible.
- ⇒ On the OPTIONS menu, click **Move link-line of reinforcement leader** (button  on the toolbar).
- ⇒ Drag any arrowhead along the rebar.
- ⇒ In the same way, drag the rest of arrowheads on link-lines of leaders in order to align these lines.
- ⇒ On the OPTIONS menu, click **Move leader** (button  on the toolbar).
- ⇒ Drag the beginning point of the leader line along the link-line of leader.
- ⇒ Drag the leader head line to any point.
- ⇒ In the same way, drag the rest of leaders.
- ⇒ To cancel the mode, on the OPTIONS menu, click **Edit drawing** (button  on the toolbar) once again to make this command (mode) not active.

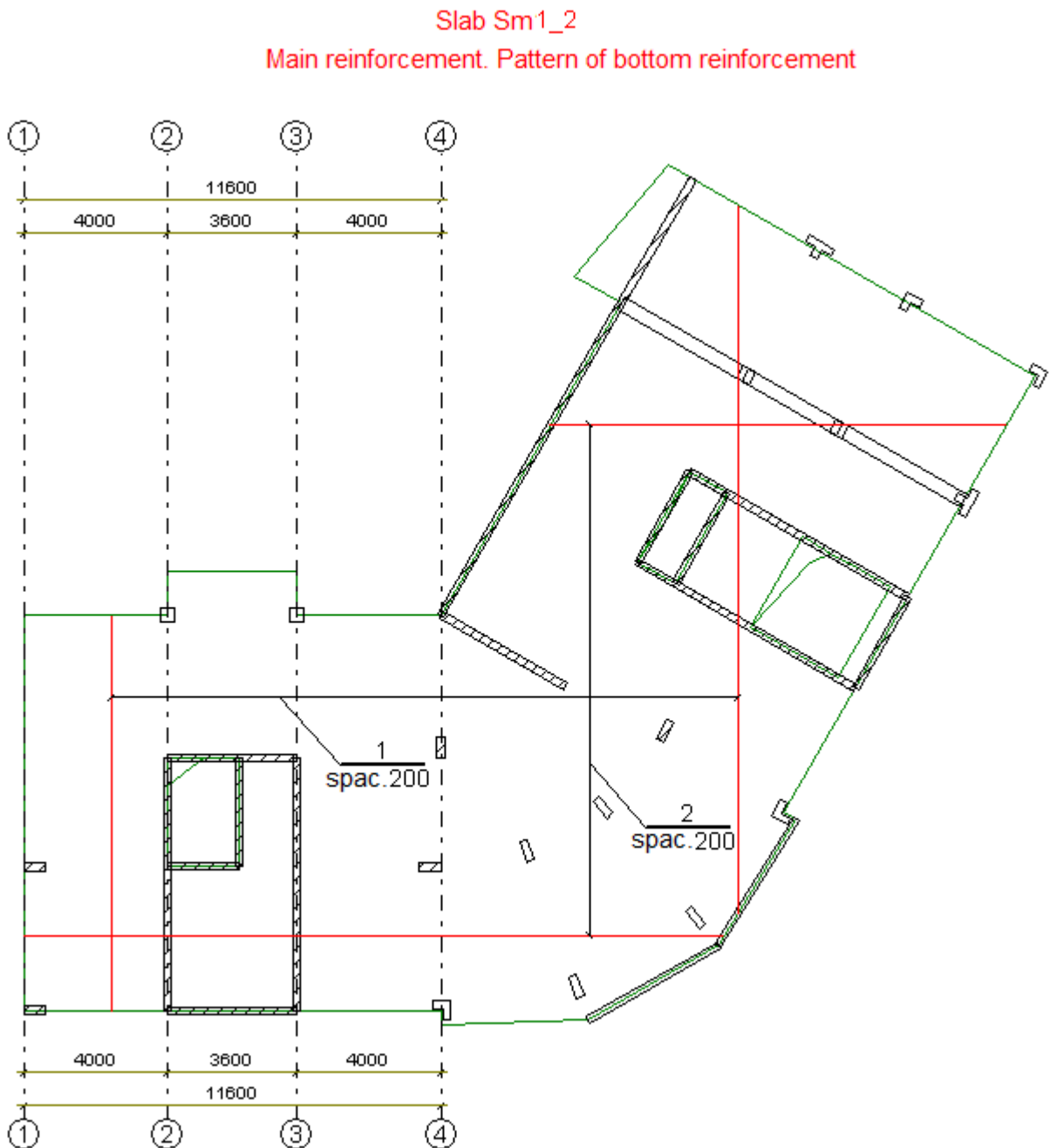




Figure 2.8.1 Pattern of rebars in main reinforcement after edit option (fragment)

To edit the scale and move fragments of the drawing:

- ⇒ On the FRAGMENT menu, click **Fragment borders** (button  on the toolbar). The pointer becomes a .


The borders will be displayed around the fragments of the drawing.


To edit the scale for presentation the cross-section of the slab:

- ⇒ Drag the fragment border and modify the dimension.


The fragment image will be inscribed into new dimensions of the fragment border.



To move the fragment, drag the fragment  to the new place. The fragment will be moved together with the fragment border.

- ⇒ To cancel the mode, on the FRAGMENT menu, click **Fragment borders** (button  on the toolbar) once again in order to make this command (mode) not active.

Parameters of specification:

- ⇒ To increase the scale of model presentation, on the VIEW menu, point to **Image** and click **Zoom to selection** (button  on the toolbar).

Specification of slab Sm1_2


No.	Description	Name	Q-ty	Unit mass,kg	Notes
		<u>Assembly units</u>			
		<u>Components</u>			
1		∅8 AIII Lnm=1982		0.4	
2		∅8 AIII Lnm=1980		0.4	
		<u>Materials</u>			
		Class of concrete B30			68.9 m3

Class of reinforcement AIII GOST 5781-82


Figure 2.8.2 Specification (fragment)



As rebars in the main reinforcement have the length that is greater than ultimate length of rebars 11m, then in the **Name** columns of the specification you will see length of rebars per running metre. Length of rebars per running metre is presented with account of overlap of rebars (total length of rebars is multiplied by overlap coefficient). The weight of 1 r.m. of rebar is presented in the **Unit mass** column. To modify description of rebars in specification, on the OPTIONS menu, click **Parameters of specification**.

- ⇒ On the OPTIONS menu, click **Parameters of specification** (button  on the toolbar).
- ⇒ In the **Parameters of specification** dialog box (see Fig.2.8.3), do the following:
- click **Help**;

The Help system for the DRAWING OF SLAB module will be displayed.

- close the Help system window.
- ⇒ Click **OK**.
- ⇒ To restore the presentation scale, on the VIEW menu, point to **Image** and click **Fit in window** (button  on the toolbar).

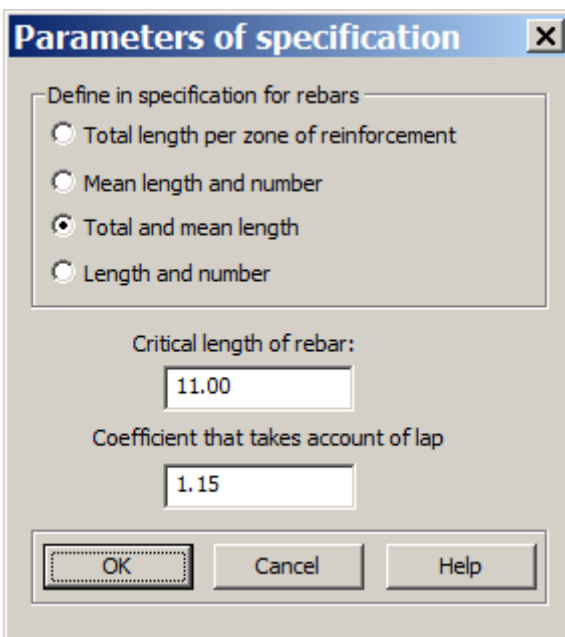



Figure 2.8.3 **Parameters of specification** dialog box

To fill in the title block:

- ⇒ On the SHEET menu, click **Title block** (button  on the toolbar).
- ⇒ In the **Contents of the title block for drawings and documents** dialog box (see Fig.2.8.4), do the following:
 - in the **Names of images on the drawing** box, type **Slab Sm1_2** (by default, Sm1_2);
 - in the **Names of images on the drawing** box, to move the pointer to the next row, press CTRL+ENTER and type **Main reinforcement along the lower edge**.

Contents of the title block for drawings and documents [X]

1. Code of the document:

2. Company name:

3. Name of the structure:

4. Names of images on the drawing:

5. Name of the article:

7. Sheet No.:

8. Total number of sheets:

9. Name or index mark of design company:

6. Design stage:

Designed by:

Job title:	Surname:
<input type="text"/>	<input type="text"/>
<input type="text"/>	<input type="text"/>
<input type="text"/>	<input type="text"/>
<input type="text"/>	<input type="text"/>
<input type="text"/>	<input type="text"/>
<input type="text"/>	<input type="text"/>

Save Title Block...
Download Title Block...
OK
Cancel

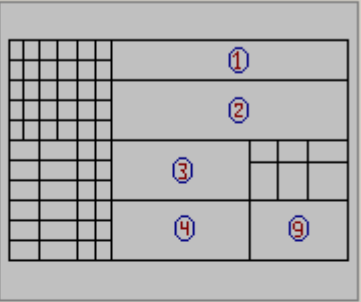


Figure 2.8.4 Contents of the title block for drawings and documents dialog box




Ready title block may be saved to a certain file on the disk. To do this, click **Save title block**. Later on, to insert the saved title block text to the dialog box, click **Download title block**.

⇒ Click **OK**.

The title block will be modified.

To print the drawing:



It is possible to modify style and settings for colours and dimensions for elements of the drawing. To do this, on the **OPTIONS** menu, click **Colours and dimensions** (button  on the toolbar).

⇒ On the **OPTIONS** menu, point to **Colour and dimension settings for** and click **Printer** (button  on the toolbar).

⇒ On the **FILE** menu, click **Print** (button  on the toolbar).

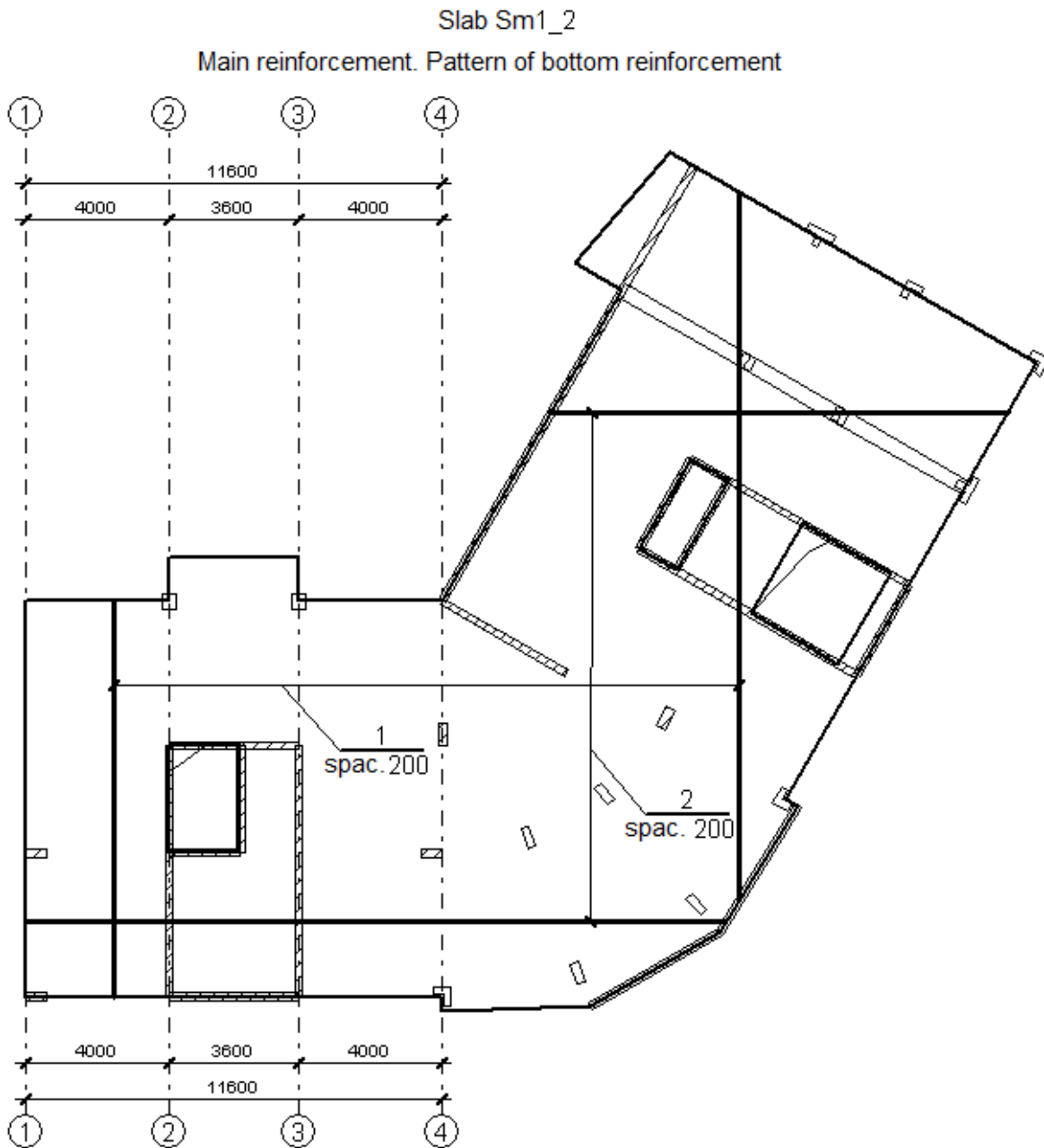



Figure 2.8.5 Pattern of rebars in main reinforcement - style for printer (fragment)



To save the drawing:

- ⇒ To save the modified drawing, on the FILE menu, click **Save** (button  on the toolbar).
- ⇒ In the **Save as** dialog box, define the for file *Slab1.pls* in the directory for problems of MONOMAKH-SAPR and click **Save**.

To import dxf-file of drawing:





*If it is required to work with the drawing further and it is not possible to do it in the DRAWING OF SLAB module, you could generate the *.dxf file for import.*

- ⇒ On the **OPTIONS** menu, point to **Colour and dimension settings for** and click **DXF-file** (button  on the toolbar).
- ⇒ Make sure that on the **FRAGMENT** menu, the **Scale fragments for DXF** command is active (button  on the toolbar is active).
- ⇒ To check scales for fragments, on the **FRAGMENT** menu, the **Define scale for fragments in DXF**.
- ⇒ In the **Scale fragments of the drawing for DXF file**, do the following:
 - in the **Fragments** list, select **Reinforcement pattern for slab** (in this case there is only one row in the list);



If necessary, it is possible to modify the scale for presentation (define it as the new one).

- other parameters remain by default.
- ⇒ Click **OK**.
- ⇒ On the **FILE** menu, click **Save drawing as DXF** (button  on the toolbar).
- ⇒ In the **Data for DXF file** dialog box, click **Next**:
- ⇒ In the **Save file DXF** dialog box, define the following:
 - directory where the file should be saved to, for example, directory for problems of MONOMAKH-SAPR.
 - file name *Slab1.dxf*;
- ⇒ Click **Save**.
- ⇒ On the **OPTIONS** menu, point to **Colour and dimension settings for** and click **Screen** (button  on the toolbar).

To open dxf-file in AutoCAD:

- ⇒ Open file with the drawing in AutoCAD program.
- ⇒ On the taskbar, click the **Start** button, point to **All programs / Autodesk / AutoCAD 2013 / AutoCAD 2013**.
- ⇒ To open the file, on the **FILE** menu, click **Open**.
- ⇒ In the **Select file** dialog box, do the following:
 - in the **file type** list, select **DXF (*.dxf)**;
 - open the directory with problems in MONOMAKH-SAPR;
 - open file *Slab1.dxf*.
- ⇒ When you complete the work in AutoCAD, save the file as *Slab1.dwg*.


Reinforcement patters for rebars of additional reinforcement:


- ⇒ On the **VIEW** menu, click **Additional reinforcement** (button  on the toolbar).




At the new sheet, perform all commands necessary to obtain the quality drawing: to modify location of leader lines, to set parameters of specification, to fill in the title block, to save the drawing and to print it).

Reinforcement patterns for rebars in main and additional reinforcement along the upper edge of the slab:

*If there are defined patterns for main and additional reinforcement along the upper edge of the slab, it is possible to obtain the drawings in the same way. On the VIEW menu, click **Top reinforcement** (button  on the toolbar).*


- ⇒ When you complete the work in the DRAWING OF SLAB module (version: reinforcement by rebars), on the FILE menu, click **Exit** (button  on the toolbar) or simply close the application.

Drawing of slab reinforced with wire meshes:

- ⇒ Return to the **SLAB** module.
- ⇒ To make the drawing of the slab, on the RESULTS menu, point to **Drawing** and click **Drawing of wire mesh** (button  on the toolbar).

The DRAWING OF SLAB module will be started (in version - reinforcement with wire mesh).





This version is less helpful as there is no automatic division of models into main and additional reinforcement. But it is the only option to obtain the drawing with reinforcement as wire meshes. The pattern of wire meshes along the lower edge of the slab is displayed by default (button  on the toolbar is active). In the specification and list of steel consumption you will see data about all defined rebars and wire meshes that are visible on the drawing at the moment.

- ⇒ On the VIEW menu, click **Top reinforcement** (button  on the toolbar).



As a rule, the border for pattern of wire meshes is not enough. You could increase the fragment border for the pattern with wire meshes.



To delete, move and modify dimensions of the drawing fragments:

- ⇒ On the FRAGMENT menu, click **Fragment borders** (button  on the toolbar).
- ⇒ On the FRAGMENT menu, click **Select** (button  on the toolbar).
- ⇒ Select on the model the fragment of the slab drawing and fragment of the slab section.

Borders of selected fragments will be coloured red.

- ⇒ On the FRAGMENT menu, click **Delete** (button  on the toolbar).

Selected fragments will be deleted.

- ⇒ To cancel the mode, on the FRAGMENT menu, click **Select** (button  on the toolbar) once again in order to make this command (mode) not active.
- ⇒ Drag the fragment with pattern of wire mesh to the upper part of the sheet
- ⇒ Increase dimensions for the fragment with pattern of wire mesh – just drag the fragment border.
- ⇒ To cancel the mode, on the FRAGMENT menu, click **Fragment borders** (button  on the toolbar) once again in order to make this command (mode) not active.

To present certain zones with the pattern of wire mesh:



*The overlap of wire mesh patterns (which is typical for double-layer reinforcement) also makes the drawing unreadable. In this case, it is recommended to generate several sheets with separate presentation of zones. In the same way, it is possible to generate sheets with 'manual' subdivision into main and additional reinforcement, if such subdivision was supposed when defining the wire mesh pattern in the **SLAB** program.*

⇒ On the **VIEW** menu, click **Select zones of reinforcement** (button  on the toolbar).

Instead of reinforcement pattern, numbers of zones will be displayed on fragments (see Fig.2.8.6).

⇒ In the **Select zone** dialog box (see Fig.2.8.7), do the following:

- select the row **Zone 1** in the list;

⇒ Click **OK**.

Pattern of top mesh reinforcement and rebars

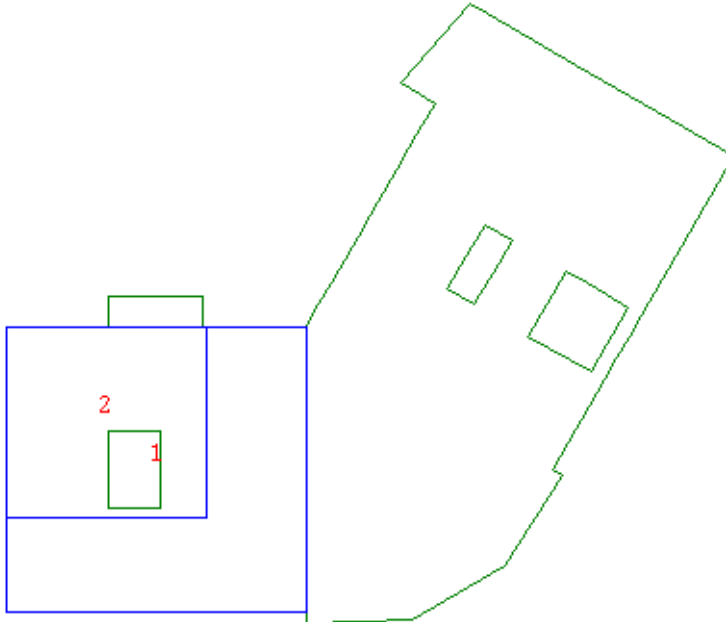


Figure 2.8.6 Pattern with wire meshes with number of zones (fragment)

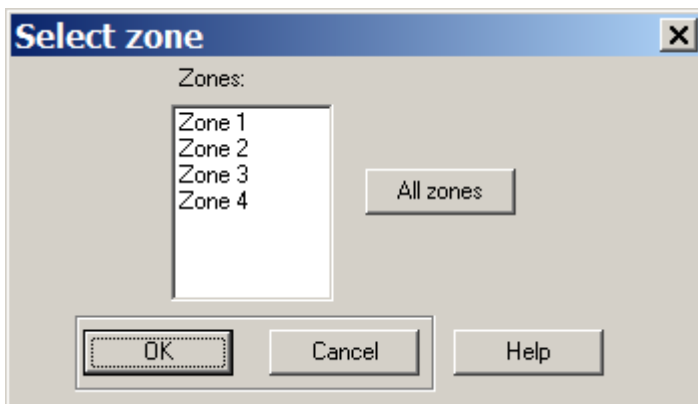


Figure 2.8.7 **Select zone** dialog box

- ⇒ On the VIEW menu, click **Select zones of reinforcement** (button  on the toolbar) once again to make this command not active.

Pattern of wire meshes will be displayed only at selected zones (see Fig.2.8.8).



In the same way it is possible to generate the sheet with other patterns of wire meshes (see Fig.2.8.9).

Pattern of top mesh reinforcement

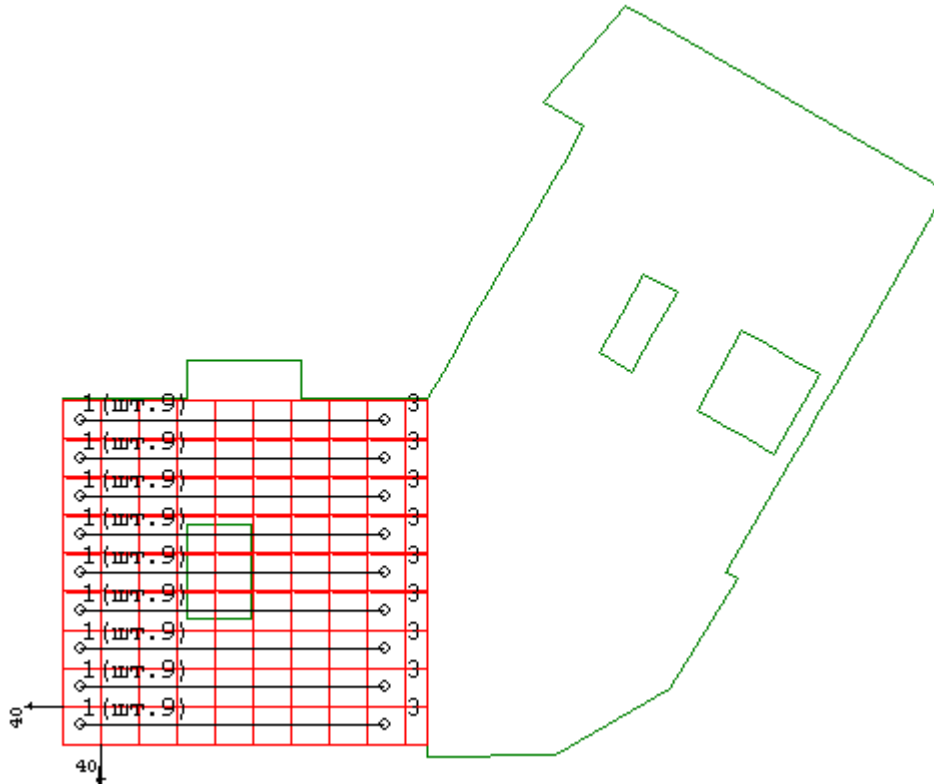


Figure 2.8.8 Pattern of mesh reinforcement along the upper edge of the slab, zone 1 (fragment)

Pattern of top mesh reinforcement

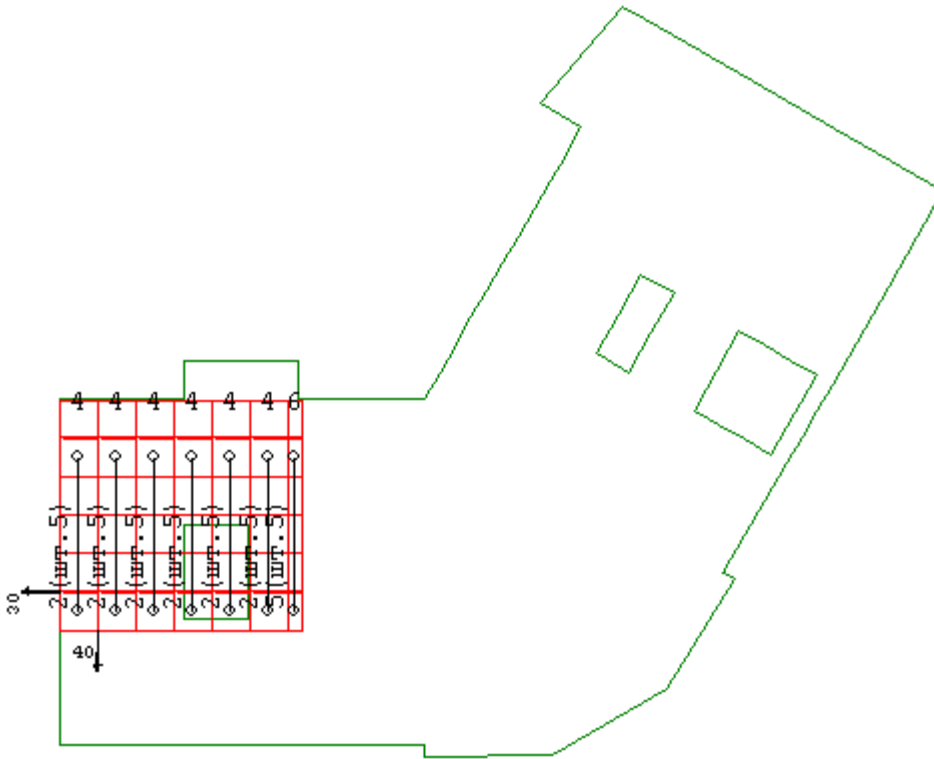



Figure 2.8.9 Pattern of mesh reinforcement along the upper edge of the slab, zone 2 (fragment)

⇒ When you complete the work in the DRAWING OF SLAB module (version: reinforcement with wire mesh), on the FILE menu, click **Exit** (button  on the toolbar) or simply close the application.