

TOPIC 4

CALCULATION AND ANALYSIS OF THE «SOIL-FOUNDATION-STRUCTURE» MODEL

For further calculation and analysis of the constructed design model, it is necessary to create calculated combinations of forces. Go to the "Project Management Screen" by clicking on the appropriate button  on the "Management" tab.

From the project tree, select  **Calculated combinations of forces and displacements** where we configure the calculated force combinations. Check the boxes to activate all loads. Set up mutual exclusion for wind loads. Since the type and type of loading, as well as the reliability factors are already set when creating loads, we do not need to configure them additionally. If any of these settings were omitted when creating loads, then the corresponding setting can be made in the dialog box "Calculated combinations of forces of forces".

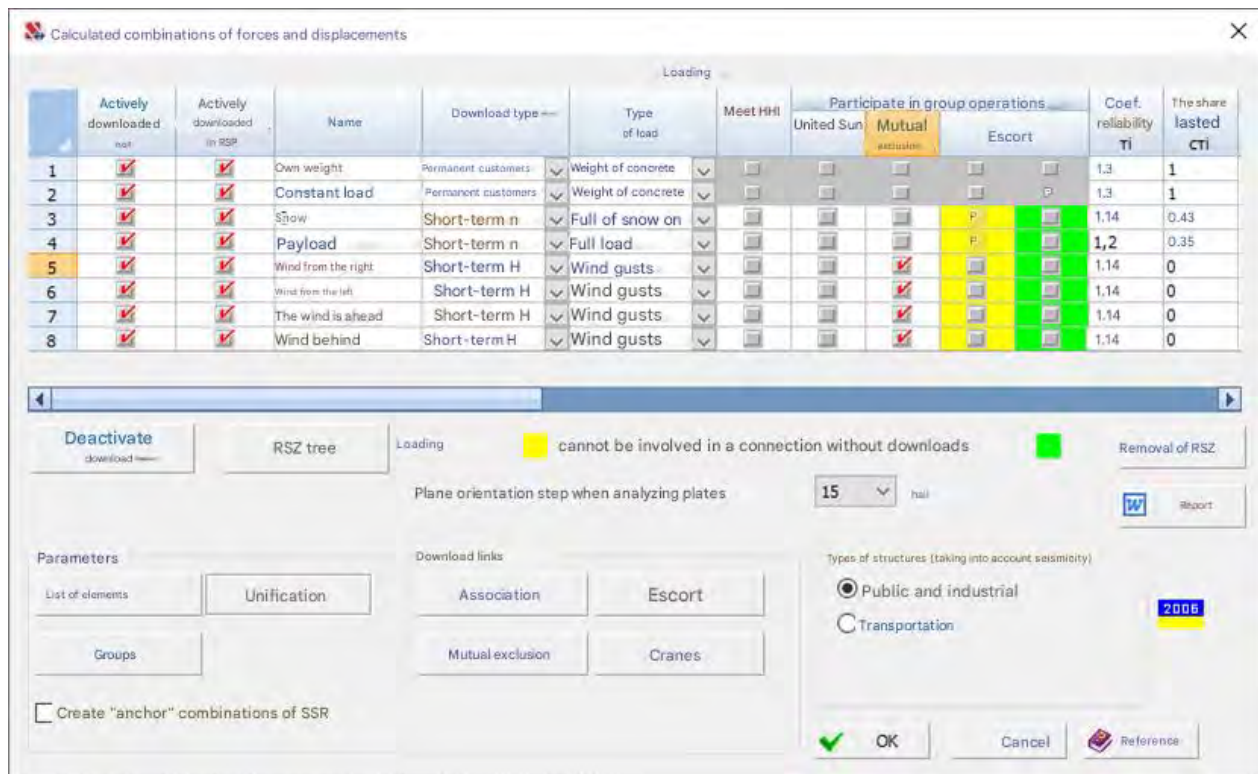


Figure 4.1 – Configuring the calculated force combinations in the corresponding dialog box

After the settings, click the OK button and click  **CALCULATION** the "Linear" calculation in the project tree. In the "Calculation Parameters" dialog box, leave everything unchanged and click the OK button.



Figure 4.2 – Calculation Parameters dialog box

We agree to save the project before the start of the calculation and wait for its completion.

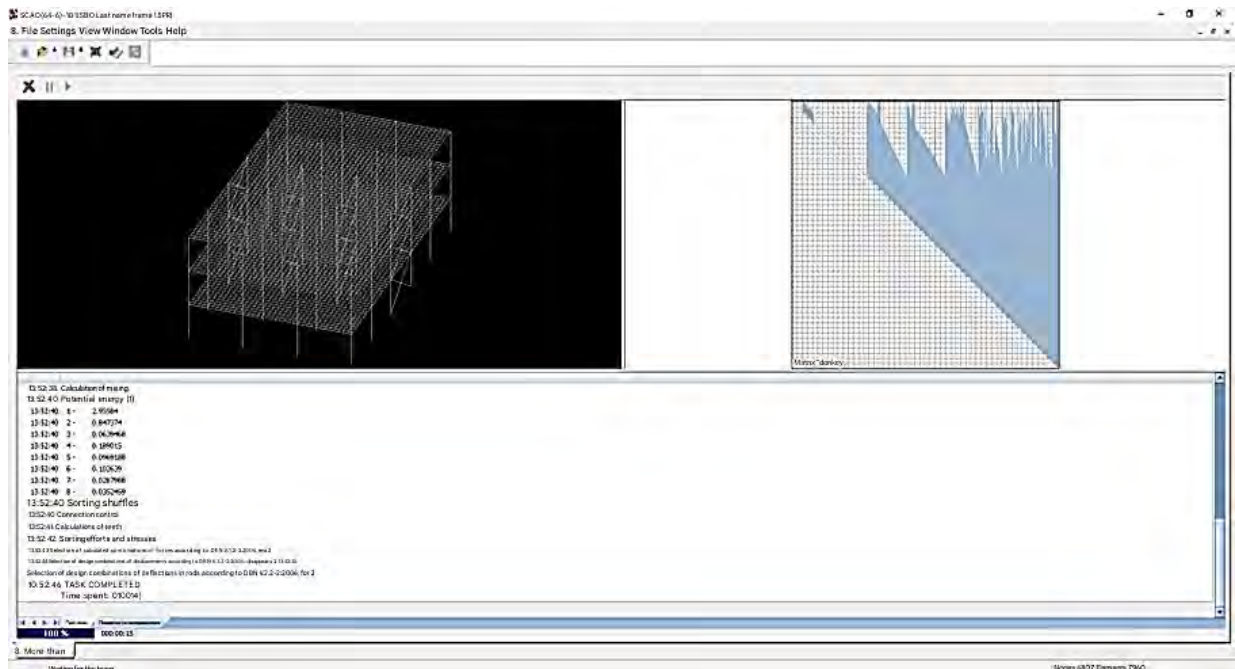


Figure 4.3 – Performing the calculation

After reaching 100% of the calculation progress and the inscription "TASK COMPLETED" in the calculation protocol, we analyze the protocol for errors. If there are critical errors, we return to working with the calculation scheme to eliminate them. If

there are no critical errors, close the calculation window by clicking on the button  and proceed to  the "Graphical analysis" of the calculation results.

For further selection of profiles of steel frame elements, they must be included in calculation groups and unification groups. Go to the group tab, and activate the 1st group "Columns of extreme frames" from the drop-down list.



Figure 4.4 – Activating a previously created group on the "Groups" tab

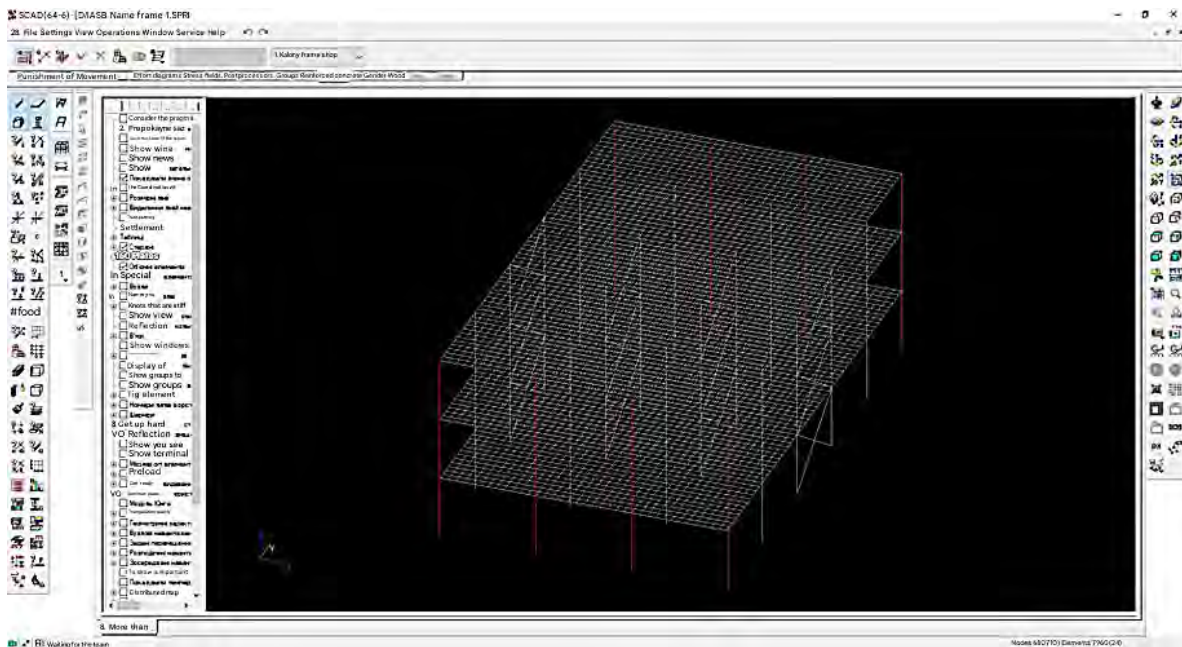


Figure 4.5 – Display of the active group on the design diagram

Go to the menu "Elements of steel structures"  on the "Groups" tab. To do this, collapse the menu "Working with groups..." by clicking on the corresponding  button. After displaying the menu list, click on the button .

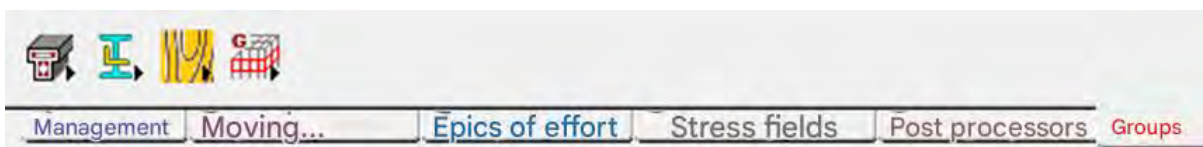


Figure 4.6 – List of the menu of the "Groups" tab



Expanding the menu "Elements of steel structures" click on the button  . "Assignment of groups of structural elements" and the confirmation button. In the "Groups of structural elements for checking sections" dialog box, assign the parameters of the created group of structural elements:

- type of structural group – rack;
- steel grade – C245
- design lengths 0.5 (XOZ), 0.7 (XOY) in accordance with DBN B.2.6-198:2014 (or determine using the CRYSTAL program);
- we select the maximum flexibility from the menu for the appropriate type of destination in the building frame;
- we select the coefficient of languages of work from the menu according to the purpose of the group in the frame of the building;
- reliability coefficients are set for the consequence class CC2, category A.

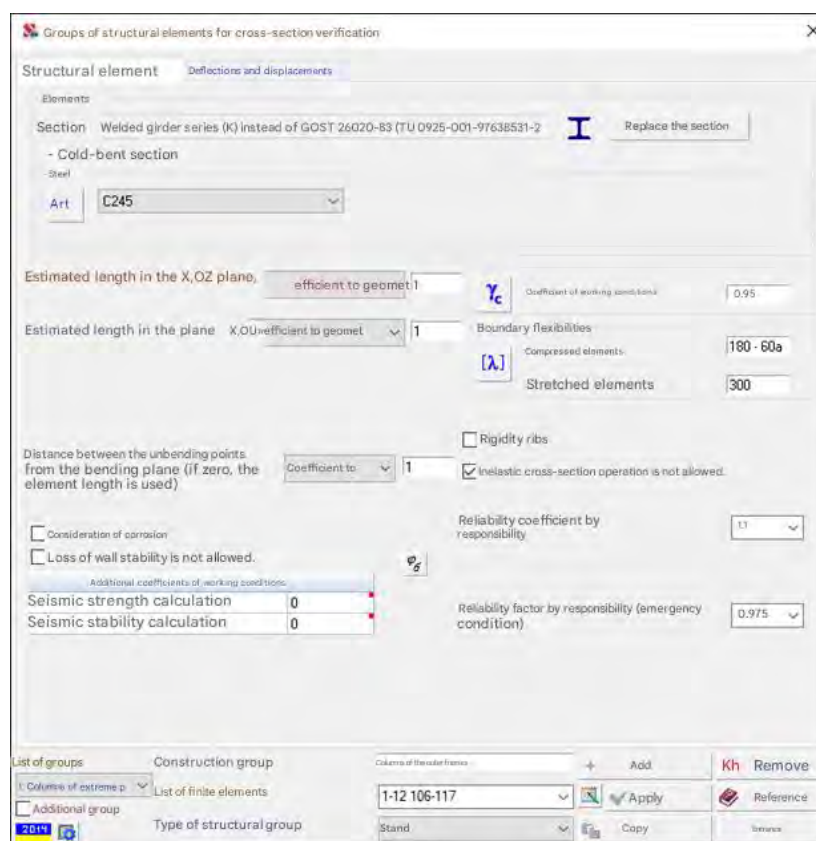


Figure 4.7 – Setting the parameters of a group of structural elements

After adjusting all the parameters, click the "Add" button  . Click the "Exit" button and agree to save the changed data.

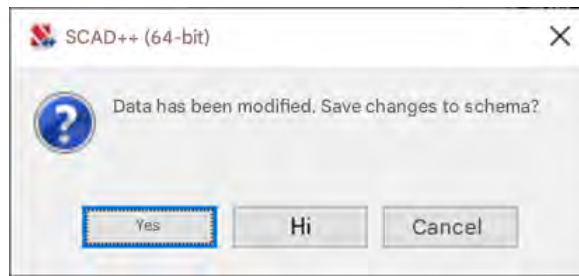


Figure 4.8 – Saving changed data

Return to the menu "Work with groups..." and activate the 2nd group "Columns of medium frames". Return to the menu "Elements of steel structures and configure and save the group similar to Figure 4.7. We assign the name "Columns of medium frames" to the new group.

In the menu "Working with groups..." choose the next group "Crossbars of extreme frames". For this group, we assign the following parameters:

- type of structural group – beam;
- steel grade – C245;
- we select the coefficient of languages of work from the menu according to the purpose of the group in the frame of the building;
- reliability coefficients are set for the consequence class CC2, category A.

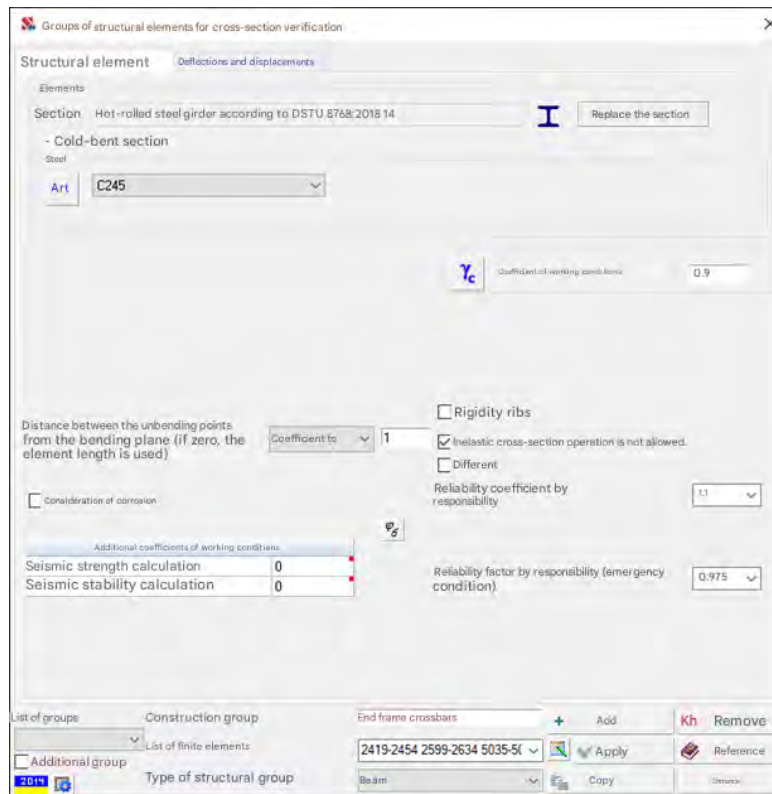


Figure 4.9 – Setting the parameters of a group of structural elements

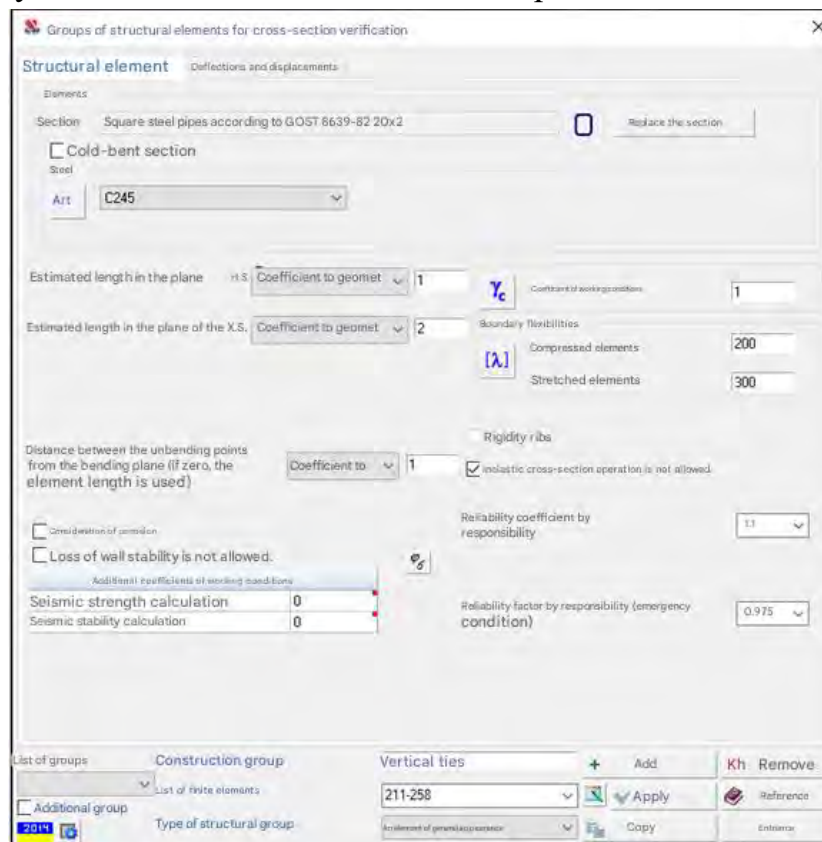
After adjusting all the parameters, click the "Add" button . Click the "Exit" button and agree to save the changed data.

Return to the menu "Work with groups..." and activate the 4th group "Crossbars of medium frames". Return to the menu "Elements of steel structures and configure and save the group similar to Figure 4.7. We assign the name "Crossbars of medium frames" to the new group.

We carry out similar operations for groups 5 and 6 "Longitudinal crossbars of extreme frames" and "Longitudinal crossbars of middle frames".

Activate group number 7 "Vertical knits" and assign the following parameters to it:

- type of structural group – an element of general appearance;
- steel grade – C245
- design lengths 1 (XOZ), 2 (XOY) in accordance with DBN B.2.6-198:2014 (or determine using the CRYSTAL program);
- we select the maximum flexibility from the menu for the appropriate type of destination in the building frame;
- we select the coefficient of languages of work from the menu according to the purpose of the group in the frame of the building;
- reliability coefficients are set for the consequence class CC2, category A.



The screenshot shows a software interface for configuring structural elements. The main window is titled "Groups of structural elements for cross-section verification". It has two tabs: "Structural element" (active) and "Deflections and displacements".

Structural element tab:

- Section:** Square steel pipes according to GOST 8639-82 20x2. There is a "Replace the section" button.
- Steel:** C245 (selected in a dropdown menu).
- Estimated length in the plane:** Coefficient to geomet: 1. There is a γ_c icon.
- Estimated length in the plane of the X.S.:** Coefficient to geomet: 2. There is a $[\lambda]$ icon.
- Distance between the unbending points from the bending plane (if zero, the element length is used):** Coefficient to: 1. There is a γ_e icon.
- Additional coefficients of working conditions:**
 - Seismic strength calculation: 0
 - Seismic stability calculation: 0
- Boundary flexibilities:**
 - Compressed elements: 200
 - Stretched elements: 300
- Rigidity ribs:** ☒ Inelastic cross-section operation is not allowed.
- Reliability coefficient by responsibility:** 1.3 (selected in a dropdown menu).
- Reliability factor by responsibility (emergency condition):** 0.975 (selected in a dropdown menu).

Bottom section:

- List of groups:** Construction group: 2004 (selected).
- List of finite elements:** 211-258 (selected in a dropdown menu).
- Type of structural group:** An element of general appearance (selected in a dropdown menu).
- Buttons:** Add, Remove, Apply, Copy, Reference, Exit.

Figure 4.10 – Setting the parameters of a group of structural elements

For the created groups of structural elements, assign unification groups, within which the selection of sections of steel elements of the same profile will be performed using the button of the same name,  call the dialog box "Groups of unification of the selection of sections".

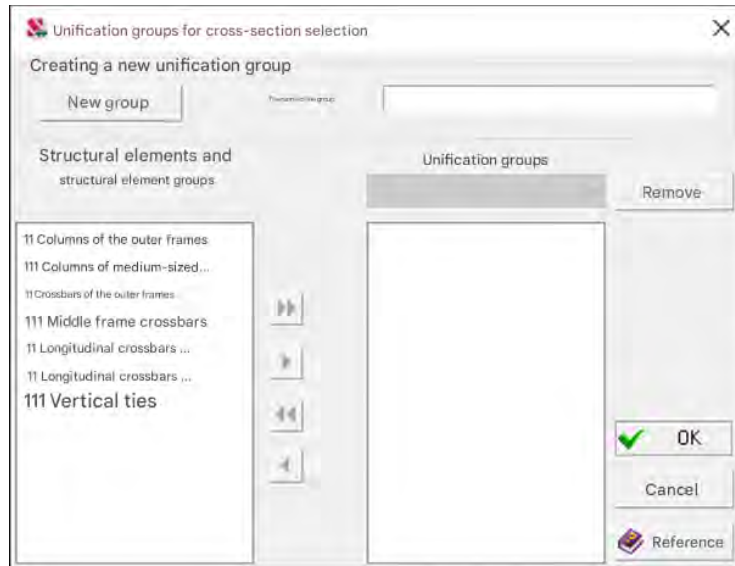


Figure 4.11 – Unification Groups dialog box to assign unification groups

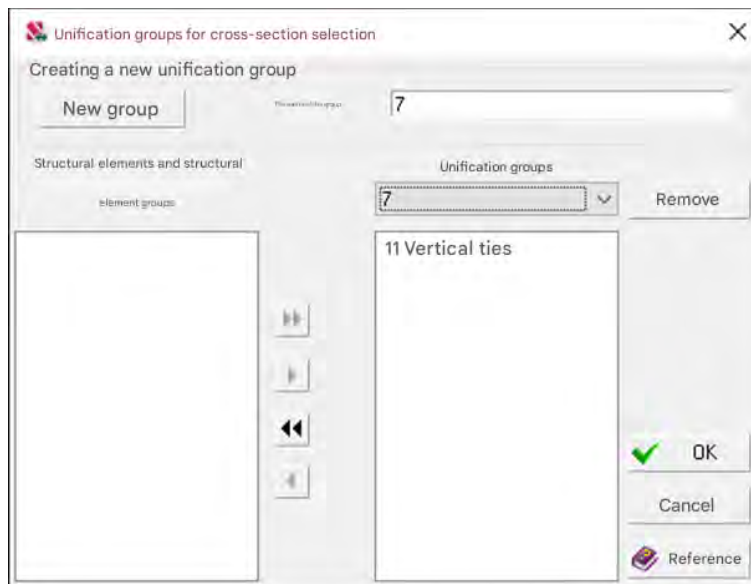


Figure 4.12 – Dialog box "Unification groups" with assigned 7 unification groups

For each group of structural elements, create a separate unification group and click the OK button.

Let's move on to working with reinforced concrete floor slabs and coatings. In the group menu, activate group number 8 "Reinforced concrete floors".

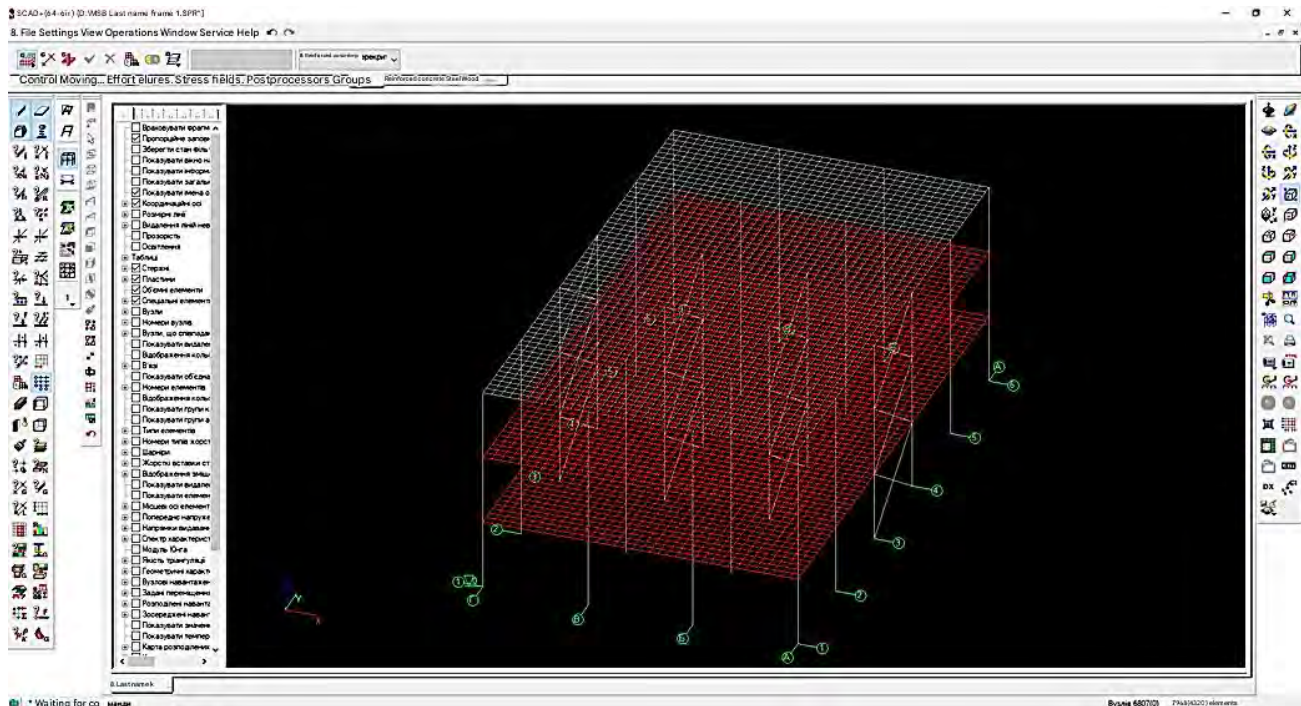


Figure 4.13 – Activated group reinforced concrete floors

Collapse the menu "Work with groups..." and expand the menu "Elements of reinforced concrete structures" on the "Groups" tab.

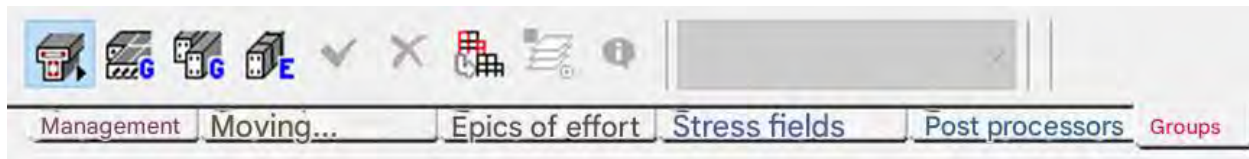


Figure 4.14 – Menu "Reinforced concrete structures"

Press the button plate element and the confirm button. We set the necessary parameters for the calculation of monolithic reinforced concrete floor slabs. Click the "Add" and "Exit" button.

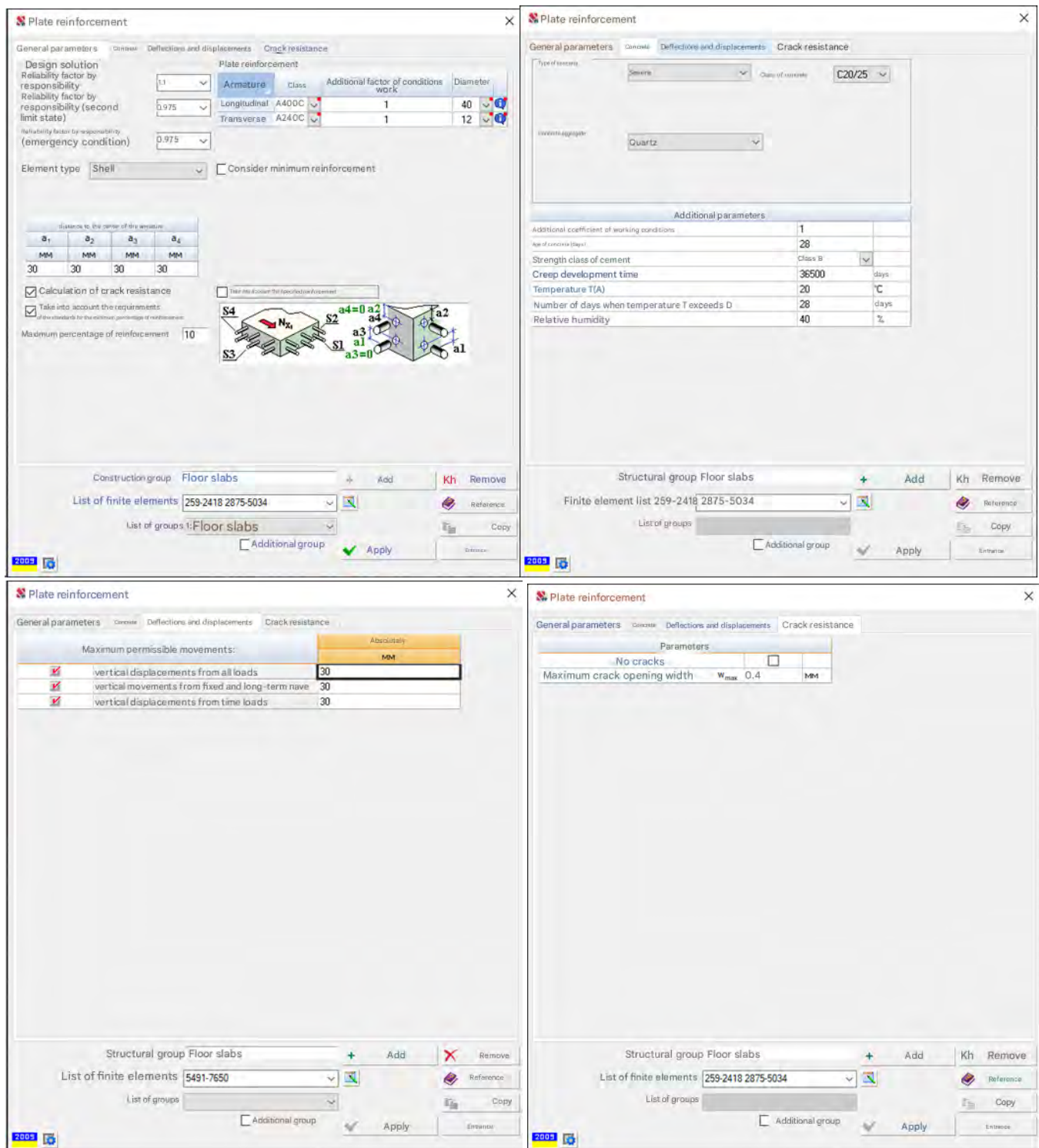


Figure 4.15 – Setting the parameters for the calculation of reinforced concrete floor slabs

We repeat the same operation for group number 9 "Reinforced concrete pavement". The calculation parameters are configured similarly to Figure 4.15. And we save the group.

To select sections of steel elements, go to the "Steel" tab. To get the results of the calculation of the previously assigned stiffnesses, press  the "Calculation" button and wait for it to end.

To visualize the results of the calculation of the set types of stiffness, press the corresponding button on the toolbar. Red on the diagram indicates the elements whose strength is insufficient, green indicates the strength of which is sufficient.

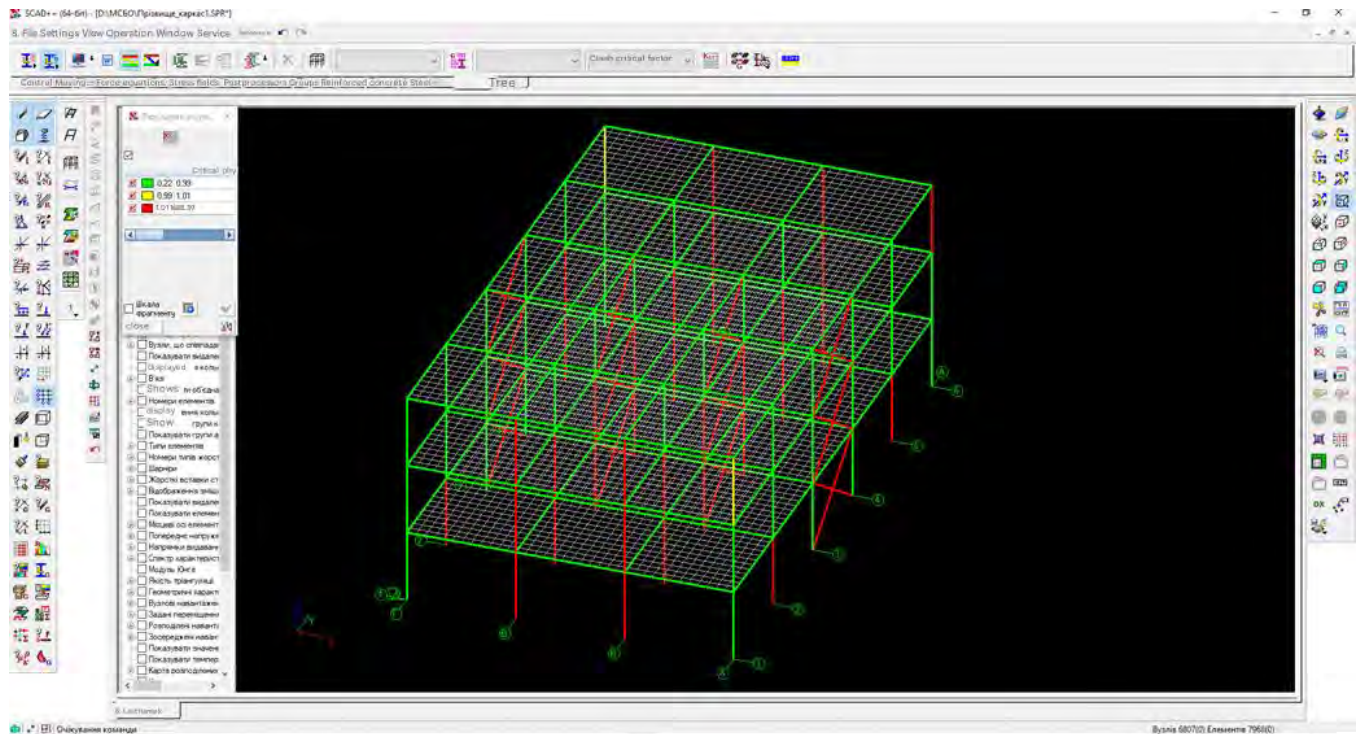


Figure 4.16 – Visualization of the results of the calculation of cross-sections of steel structures

To select the necessary cross-sections of steel structures, click  the "Selection" button. After receiving the selection results, check all the boxes and click the "Replace element stiffness" button and click OK. Since after replacing the stiffeners, there will be a redistribution of stresses in the frame elements, it is necessary to re-calculate the stresses. To do this, return to the "Control" tab and use the corresponding button to exit the "Project Control Screen". project management screen, start the calculation again and wait for it to end. Then we return to the "Graphic Analysis" tab on the "Steel" tab and perform the calculation of steel structures. If necessary, we perform a re-selection of steel elements and a full calculation with replaced stiffeners. Repeat the iterations until all the elements are green and the stiffness in each iteration does not change as a result of the selection.

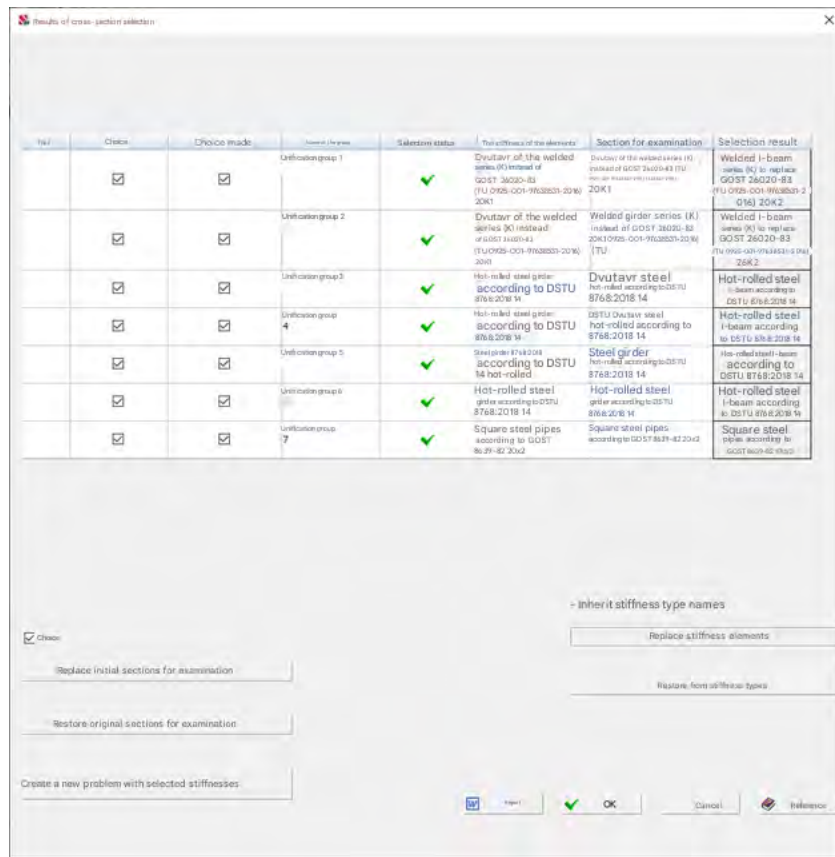


Figure 4.17 – Replacement of stiffeners based on the results of selection

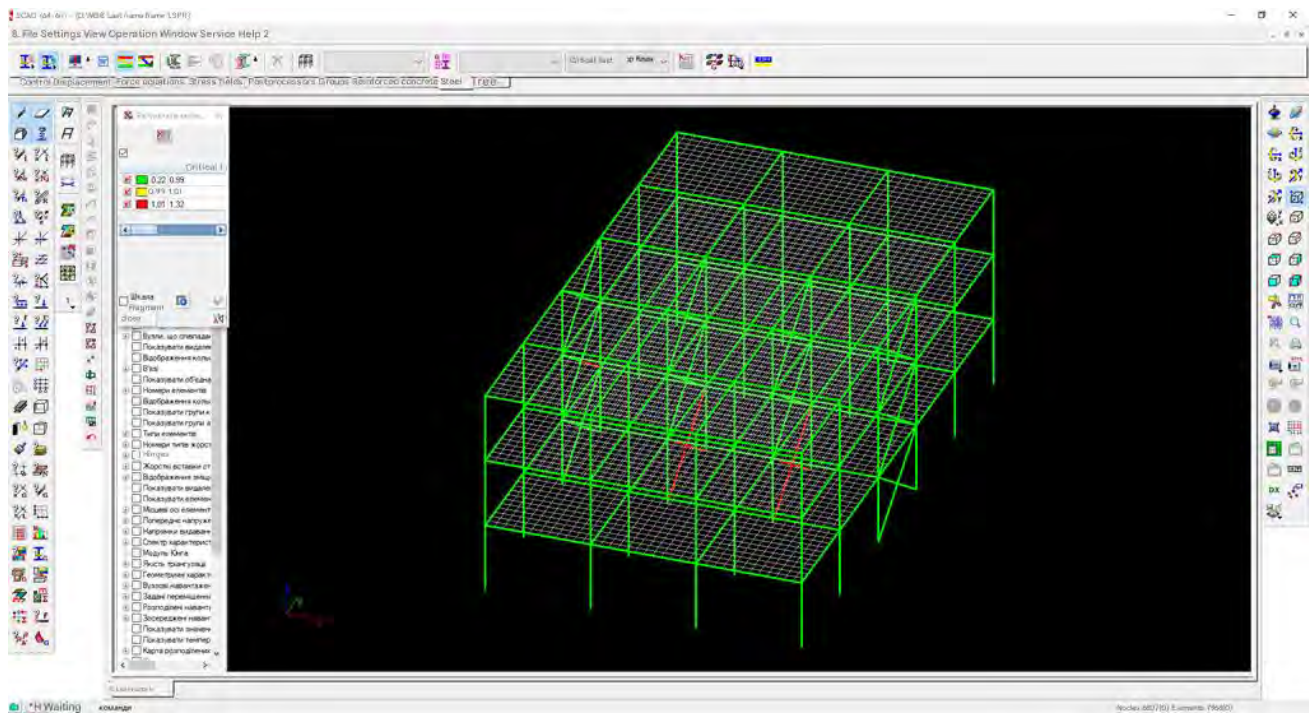


Figure 4.18 – Results of calculation of replaced cross-sections after stress recalculation

Since after recalculating the stresses with new stiffnesses, red elements with insufficient bearing capacity remain, we perform the second iteration of the selection.

Results of cross-section selection

Type	Choice	Choice made	Name of the group	Selection status	The stiffness of the elements	Section for examination	Selection result
	☒	☒	Unification group 1	☒	Dvutavr welded series (K) instead of GOST 26020-83 (TU 0925-001-97638531-2016) 20K2	Dvutavr of the welded series (K) instead of GOST 26020-83 (TU 0925-001-97638531-2016) 20K2	Welded I-beam series (K) to replace GOST 26020-83 (TU 0925-001-97638531-2016) 20K2
	☒	☒	Unification group 2	☒	Dvutavr welded series (K) instead of GOST 26020-83 (TU 0925-001-97638531-2016) 26K2	Dvutavr of the welded series (K) instead of GOST 26020-83 (TU 0925-001-97638531-2016) 26K2	Welded I-beam series (K) to replace GOST 26020-83 (TU 0925-001-97638531-2016) 26K2
	☒	☒	Unification group 3	☒	Steel girder hot-rolled according to DSTU 8768:2018 14	Steel girder hot-rolled according to DSTU 8768:2018 14	Hot-rolled steel I-beam according to DSTU 8768:2018 14
	☒	☒	Unification group 4	☒	Steel girder hot-rolled according to DSTU 8768:2018 14	Steel girder according to DSTU 8768:2018 14 hot-rolled	Hot-rolled steel I-beam according to DSTU 8768:2018 14
	☒	☒	Unification group 5	☒	Steel girder 8768:2018 according to DSTU 14 hot-rolled	Steel girder hot-rolled according to DSTU 8768:2018 14	Hot-rolled steel I-beam according to DSTU 8768:2018 14
	☒	☒	Unification group 6	☒	Steel girder 8768:2018 according to DSTU 14 hot-rolled	Steel girder 8768:2018 according to DSTU 14 hot-rolled	Hot-rolled steel I-beam according to DSTU 8768:2018 14
	☒	☒	Unification group 7	☒	Square steel pipes according to GOST 8639-82 100x3	Square steel pipes according to GOST 8639-82 100x3	Square steel pipes according to GOST 8639-82 100x4

☒ Choice

☒ Inherit stiffness type names

Replace initial sections for examination

Replace stiffness elements

Restore original sections for examination

Restore from stiffness types

Create a new problem with selected stiffnesses

Report OK Cancel Reference

Figure 4.19 – Results of the second iteration of the selection of sections

Based on the results of the selection, we replace the stiffeners and recalculate the stresses in the frame elements. Go to the project management screen, agree to replace the stiffeners with new selected ones, and start a linear calculation. At the end of the calculation process, return to graphical analysis and go to the steel tab and click calculation. We make sure that all elements have sufficient bearing capacity – the green color of the reflection on the design diagram. If not, we repeat the iterations of calculating stresses and selecting sections of steel frame elements for these stresses.

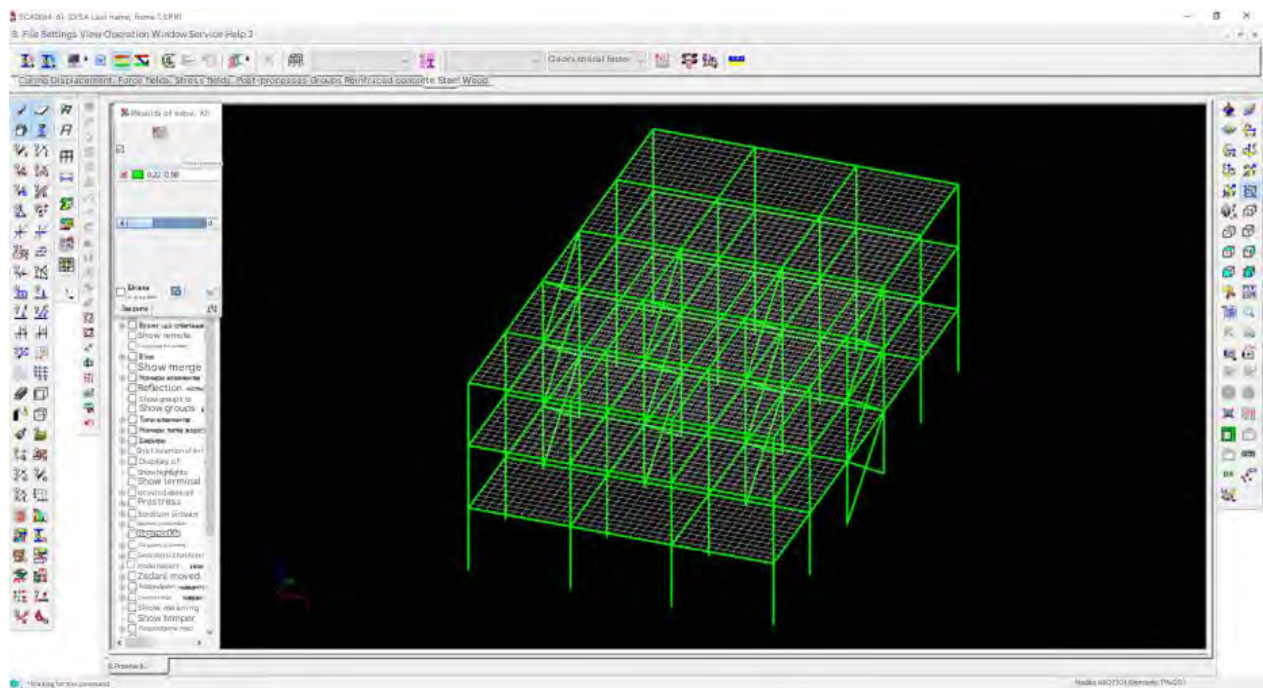


Figure 4.20 – Results of the final iteration of the selection of sections

To assess the bearing capacity of the specified parameters of reinforced concrete floor slabs and coatings, go to the reinforced concrete tab. Open the menu "Examination of reinforced concrete" and click  the "Calculation" button. We analyze the results of the calculation in the same way as the results of metal structures, green – the bearing capacity is sufficient, red – insufficient.

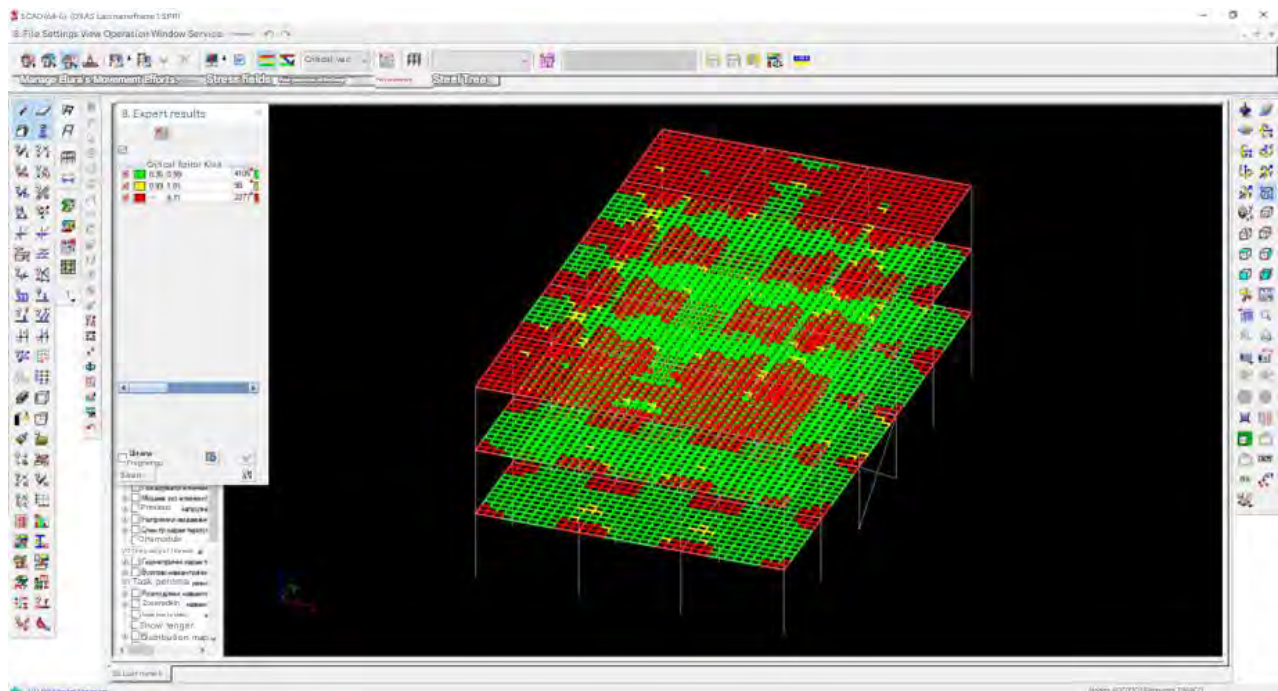


Figure 4.21 – Results of the calculation of the predefined reinforcement

Since the results of the calculation show us the insufficiency of the predetermined reinforcement, go to the "Reinforcement selection" menu and start the selection process  with the appropriate button.



Figure 4.22 – Menu "Selection of fittings"

We graphically evaluate the results of the selection of reinforcement and compare it with the predetermined one.

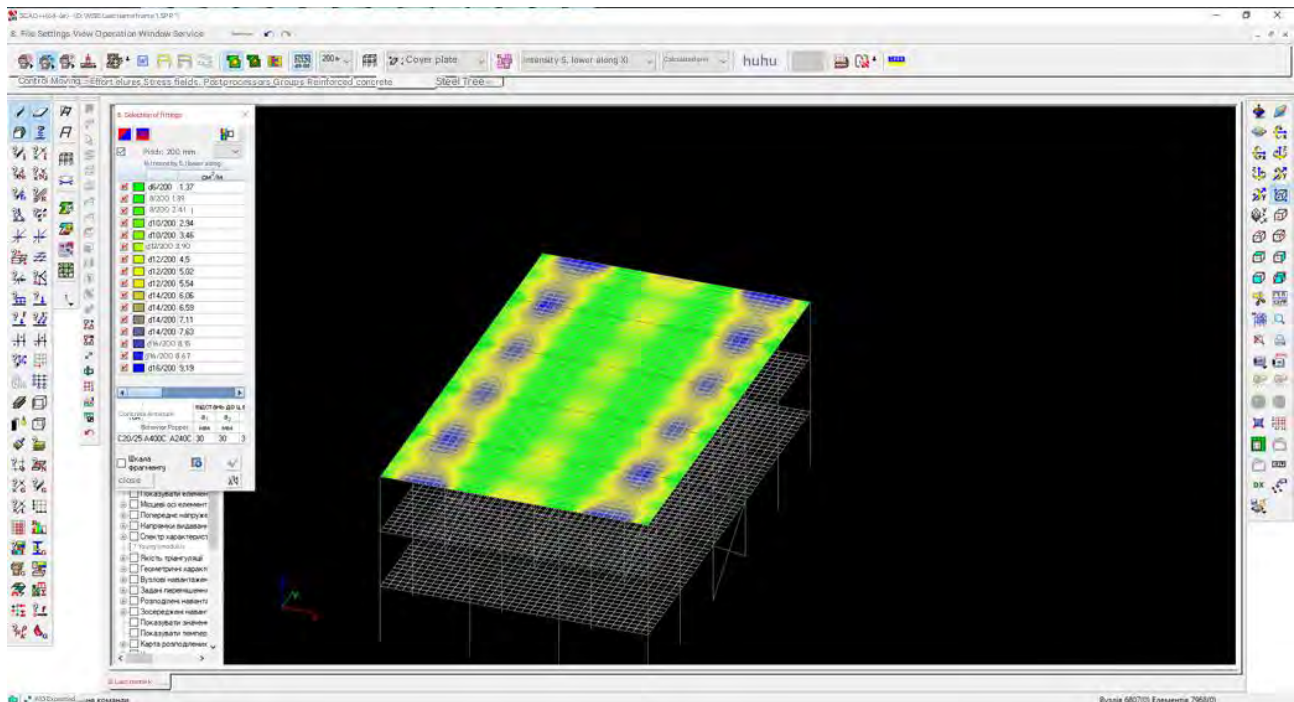


Figure 4.23 – Isopoles of reinforcement selection results

We state the insufficiency of the previously accepted fittings.

To change the reinforcement according to the selection results, click the  "Set reinforcement according to the selection results" button in the dialog box, click the OK button. Go to the "Reinforced concrete examination" menu and make sure that the reinforcement is sufficient according to the selection results.

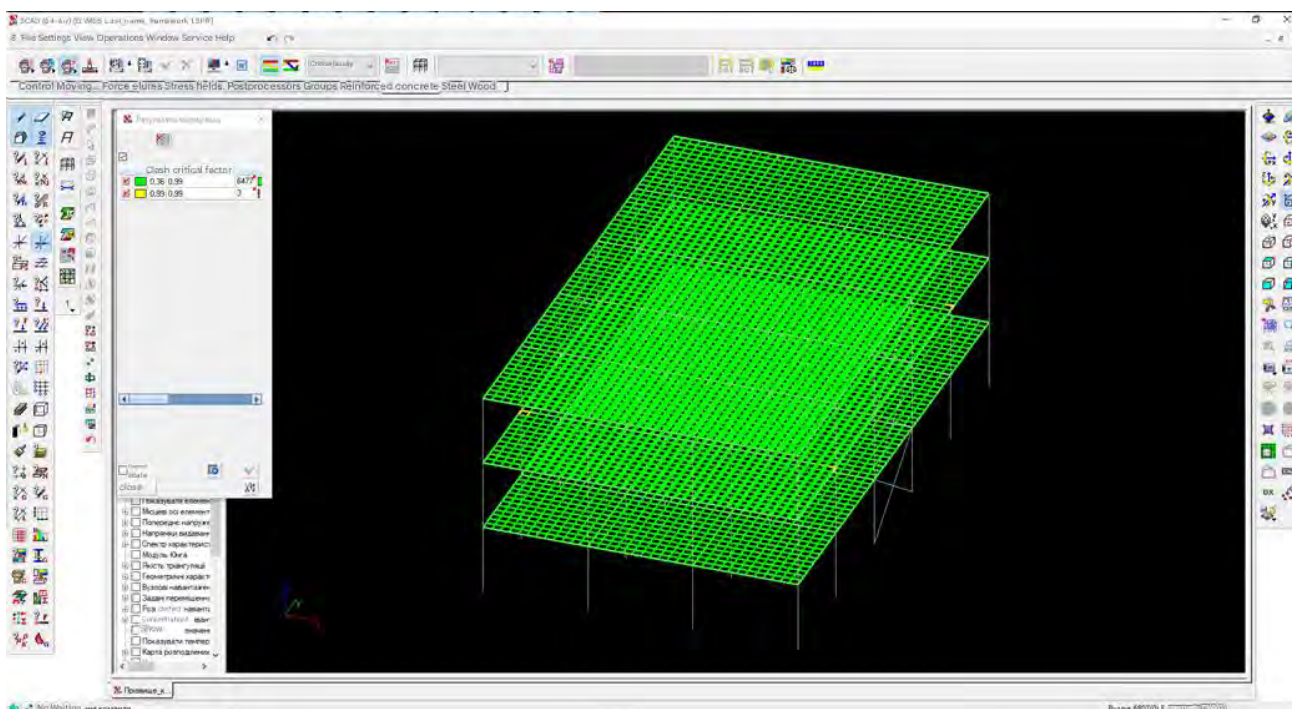


Figure 4.24 – Calculation of floor slabs and coatings with applied reinforcement based on the results of selection

We repeat the stress calculation taking into account the new stiffness of reinforced concrete elements on the project management screen. We make sure that there are no changes in the required rigidity of metal structures and reinforcement of reinforced concrete structures. If necessary, repeat the iteration of the selection-calculation.

Next, we combine the created scheme of the above-ground part of the building with the pre-created calculation scheme of the foundation slab. Through the "File – Open" menu, open the previously created file with the foundation slab.

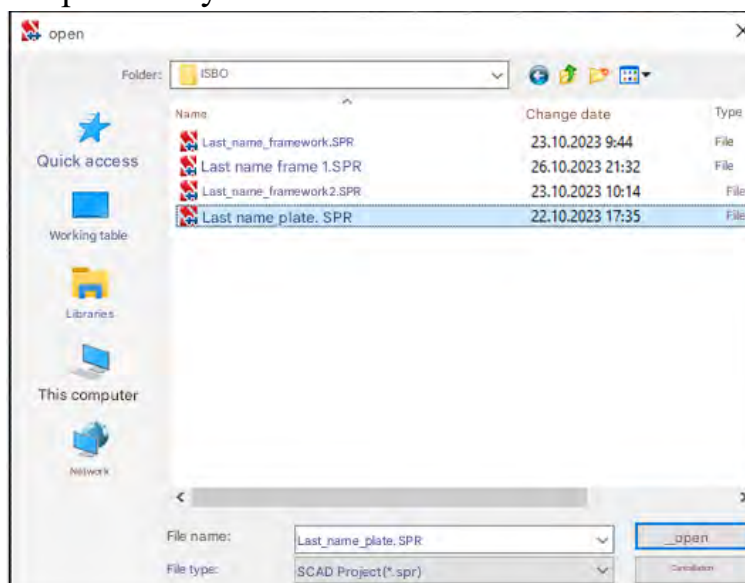


Figure 4.25 – Opening a stovetop file

As a result of this operation, we get two tabs with separate design models in the working field of the SCAD ++ program. Go to the design diagram of the above-ground part of the building through the project management screen. Then go to the "Diagram" tab and click  the "Assembly" button.



Figure 4.26 – Working field of the "Assembly" mode

Load the circuit that needs to be connected using the tools of the "Assembly" mode. Choose the scheme "Last name_plate.spr"

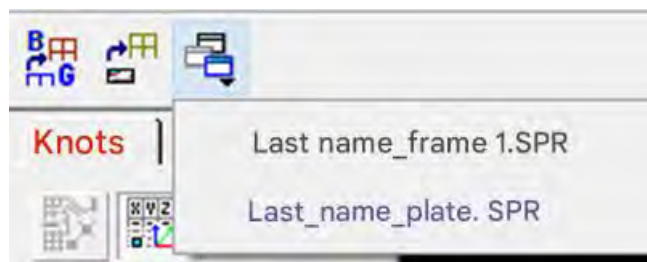


Figure 4.27 – Selection of the circuit for connection

After loading the subcircuit of the stove, click on the  "Select assembly method" button.

Choose the assembly method – binding to one node. Switching between subcircuits using the buttons,  select a node with coordinates 0.0.0 on each subcircuit (intersection of axes A–G) and press the "Confirm" button.

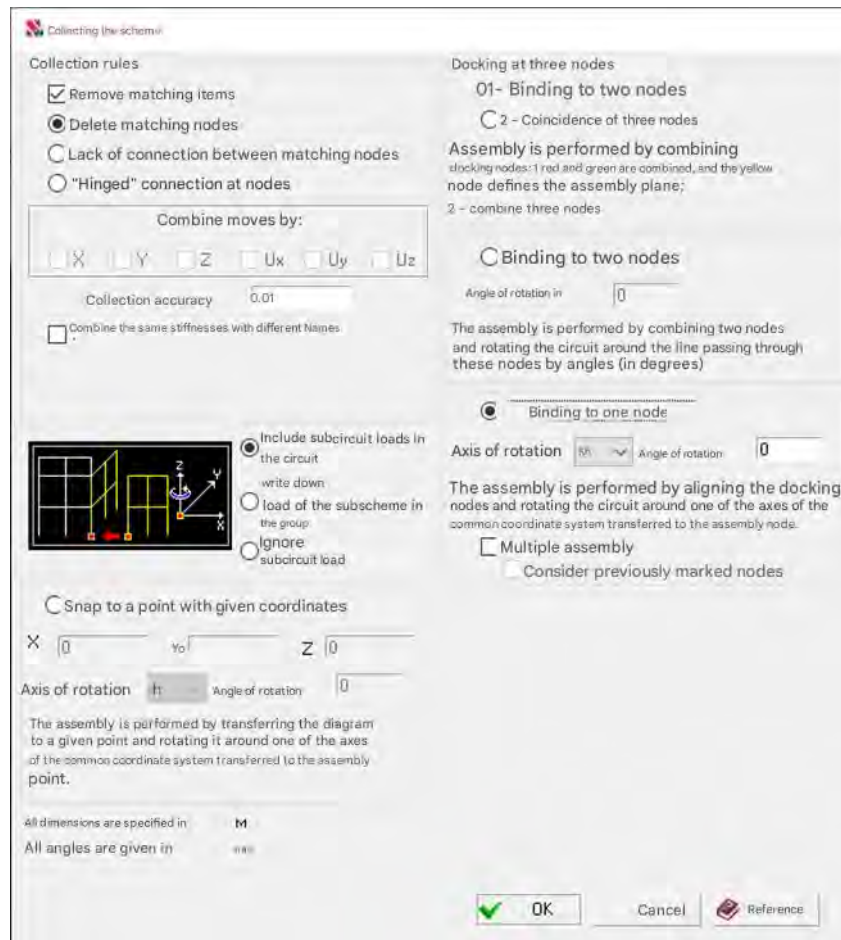


Figure 4.28 – Choosing the assembly method

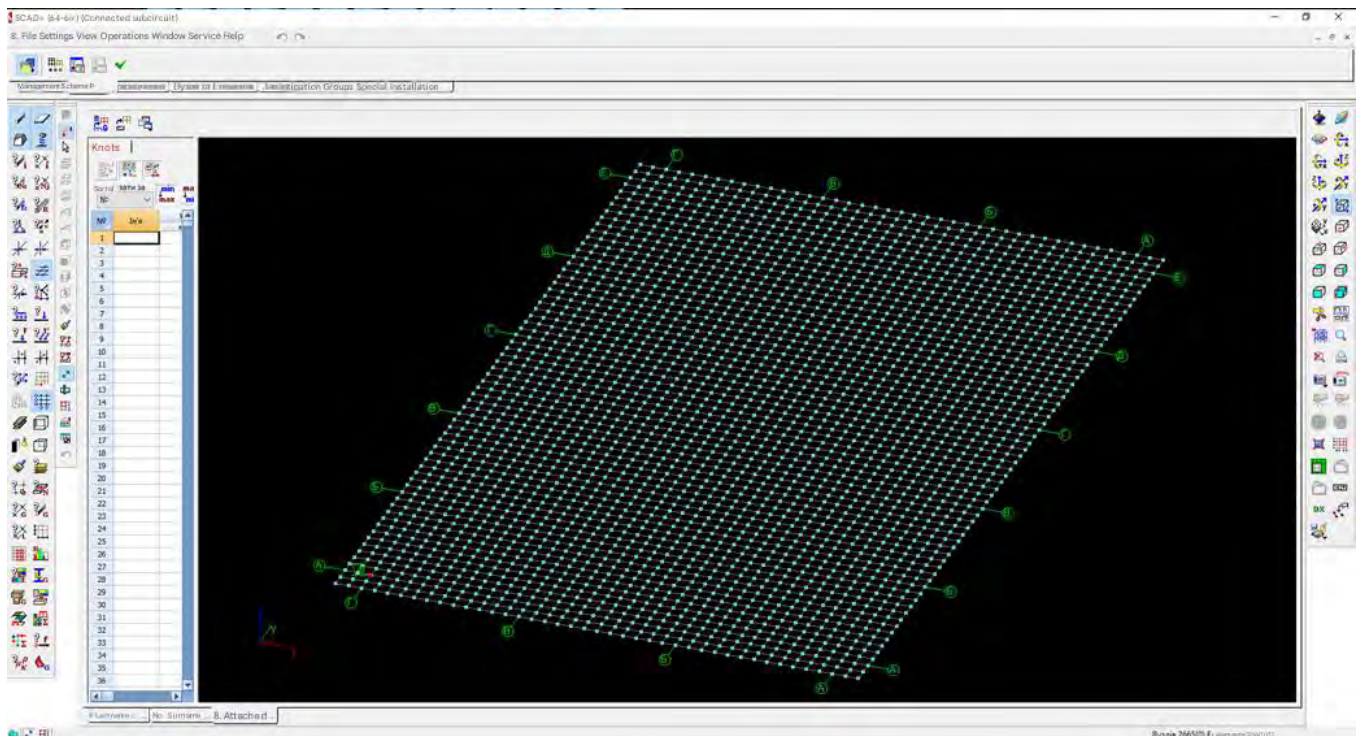


Figure 4.29 – Selecting the assembly point on the slab subcircuit

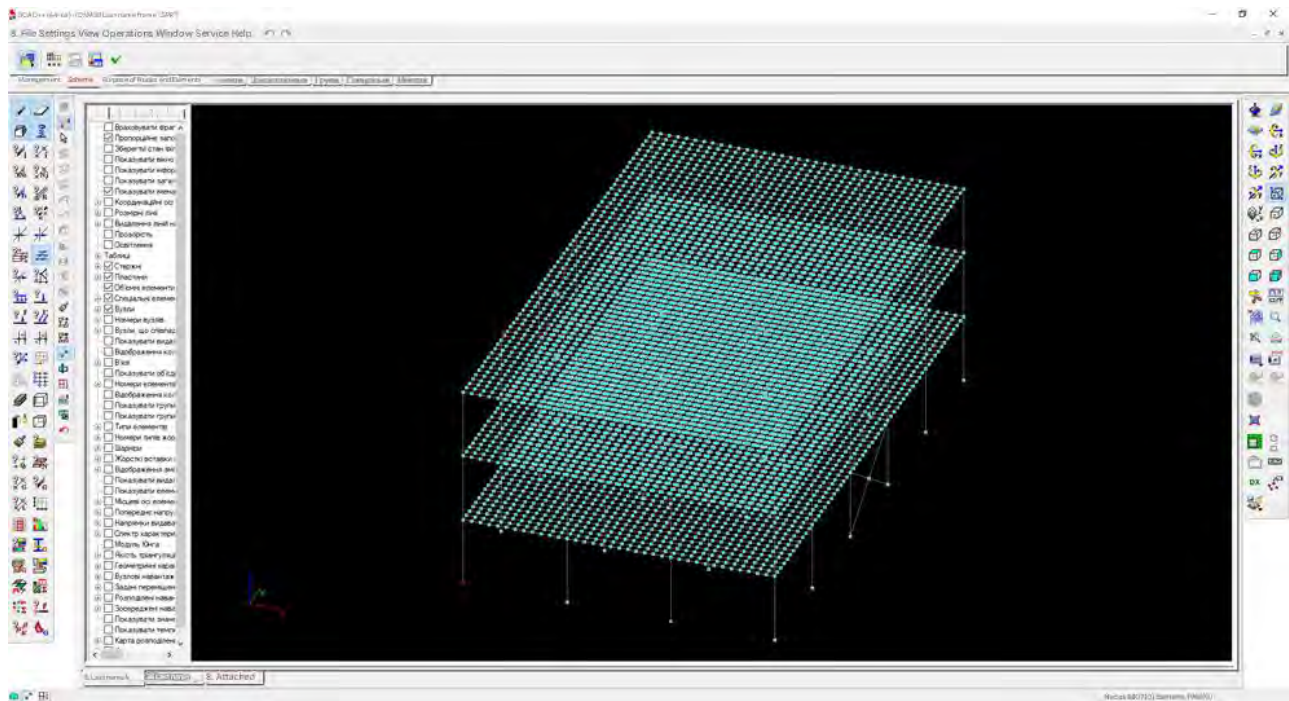


Figure 4.30 – Selecting an assembly point on the frame subcircuit

Make sure that the assembly is correct and click the confirm button.

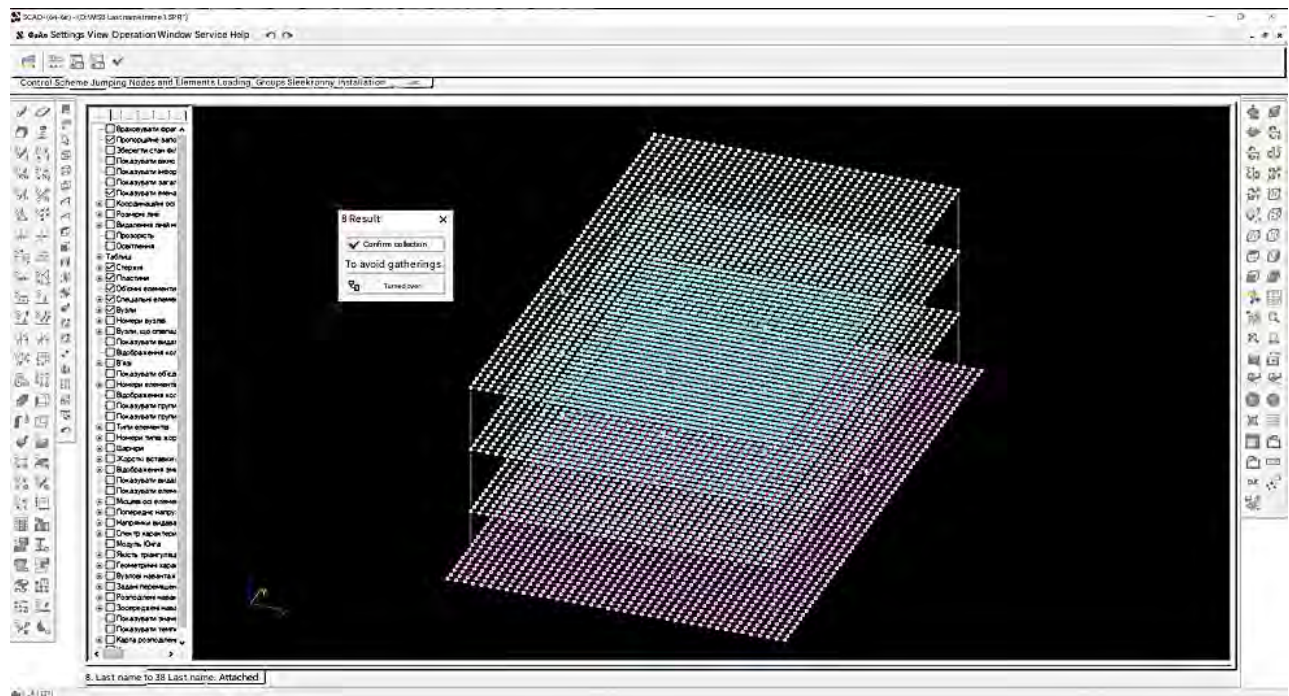


Figure 4.31 – Confirmation of assembly results

Save the build results under the name "Surname_build.spr" using the "File-save project as..." menu.

For the correct operation of the "Foundation-Foundation-Structure" system, we remove the pre-established adjoining conditions at the bottom of the columns.

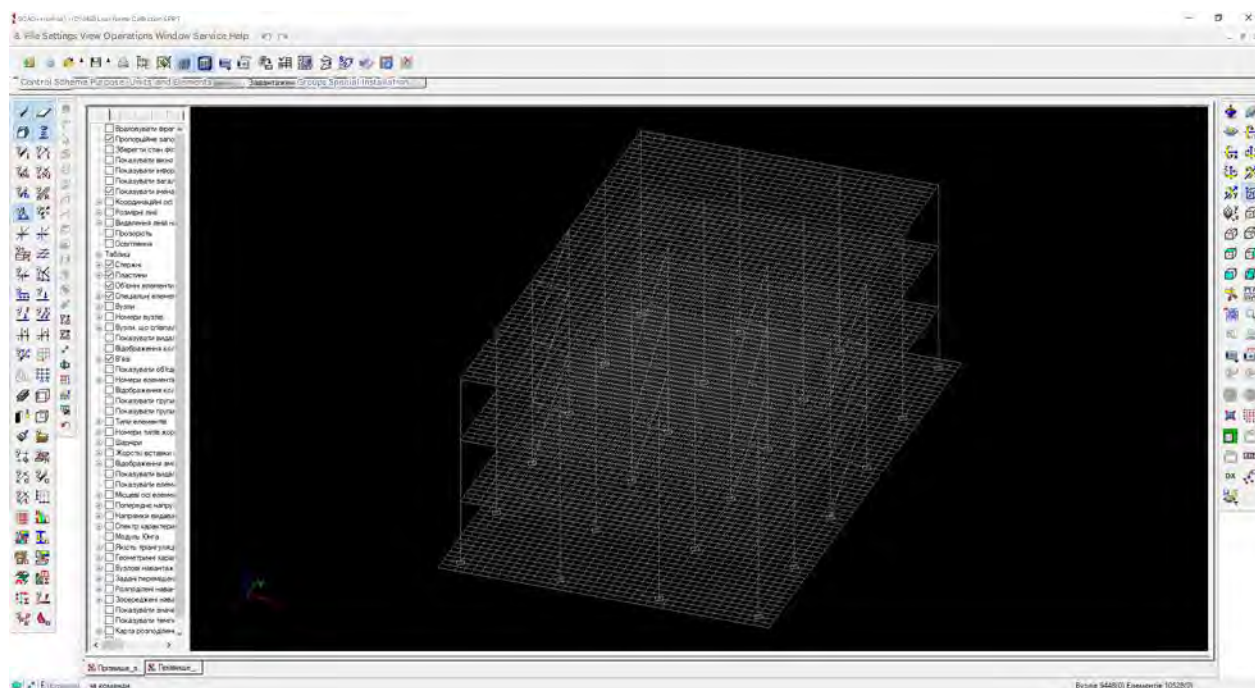


Figure 4.32 – Design scheme with extra knits

Using the "Node Information – Knits" tool, select the necessary nodes and turn off all pre-installed knits. And click the OK button.

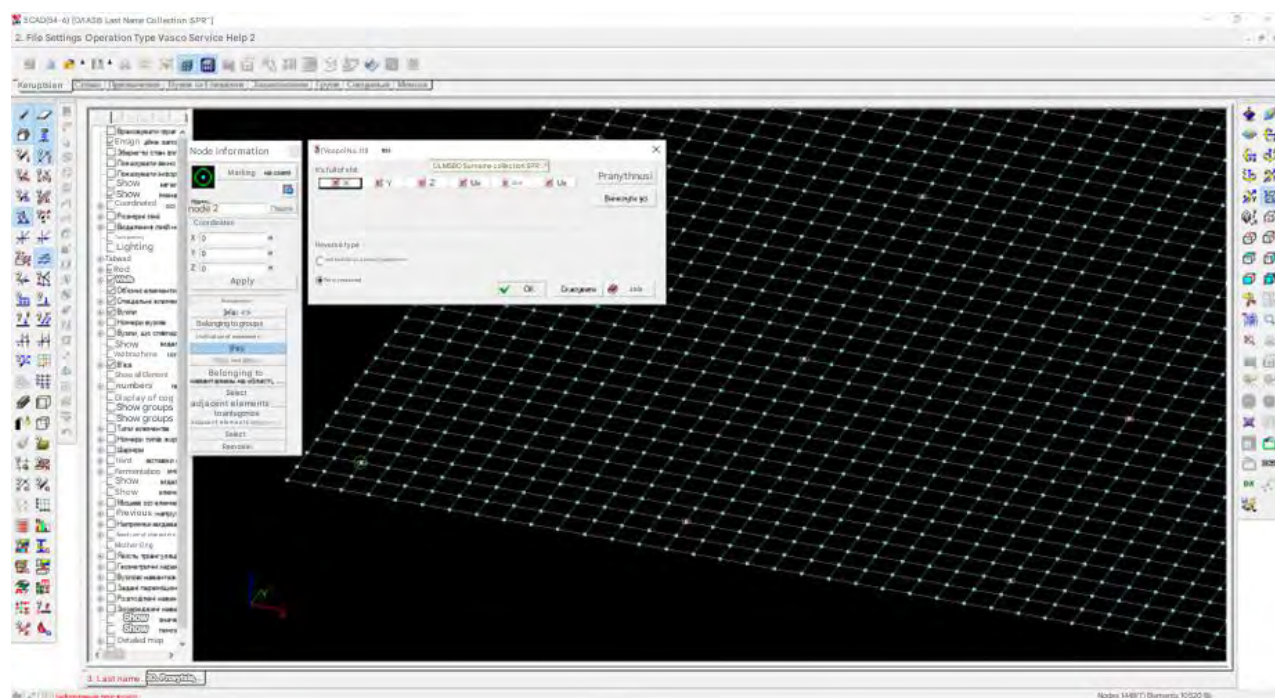


Figure 4.33 – Turning off elms using the Knot Information tool

We fix the corner nodes of the slab from horizontal displacement to avoid critical errors in the calculation.

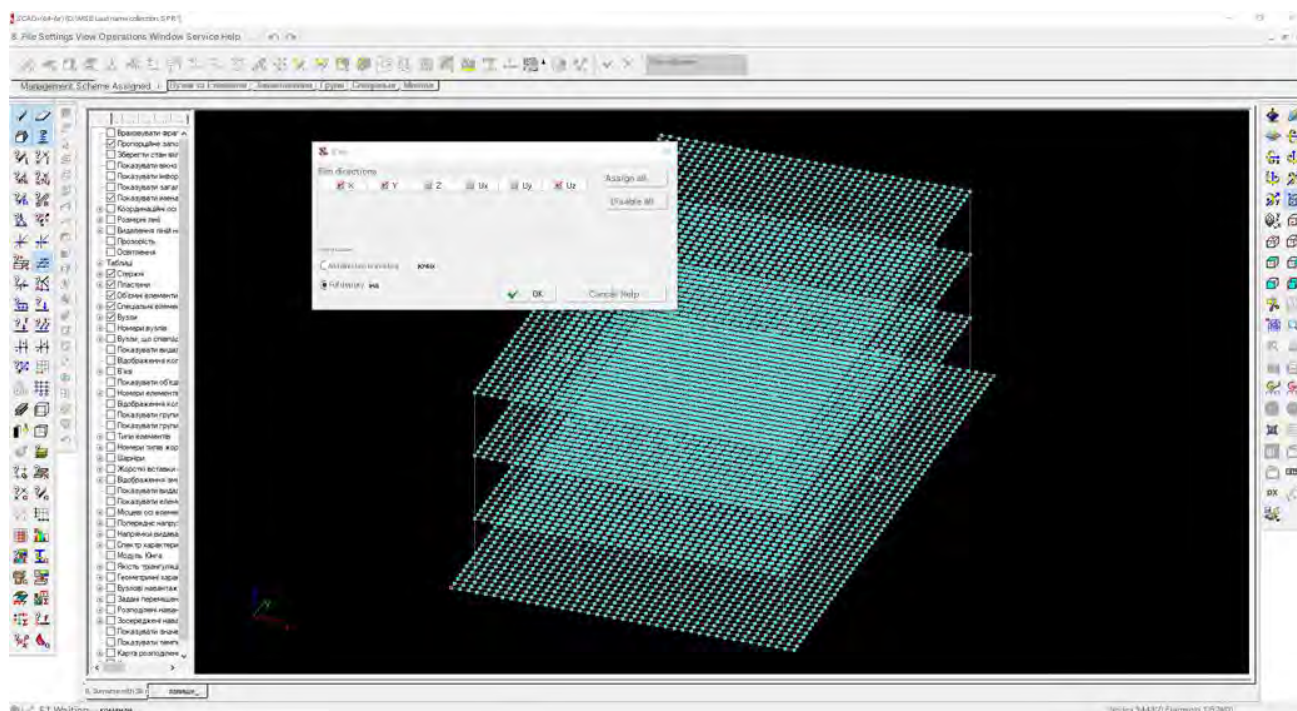


Figure 4.34 – Adding horizontal elms to the foundation slab

We perform the calculation. We analyze changes in the results of the calculation of metal and reinforced concrete structures in the "Graphic Analysis".

At this stage, it is necessary to perform additional iterations of the coefficients of the foundation slab bedding, taking into account the loads from the frame of the building.

On the project management screen, select "Load combinations".

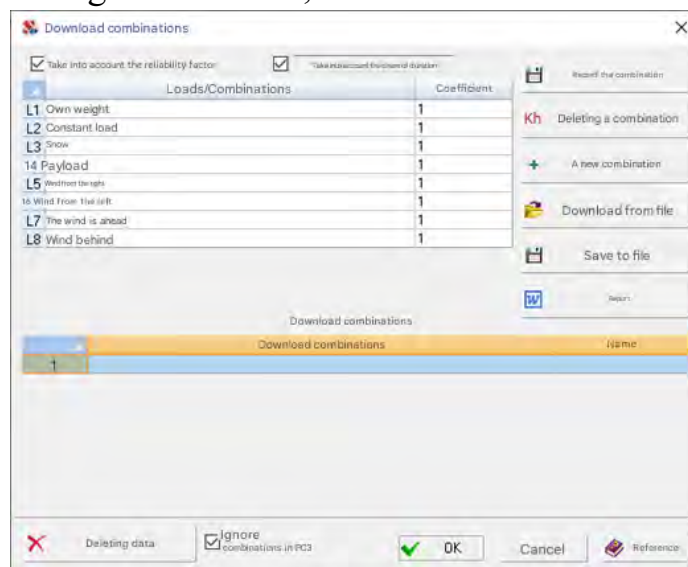


Figure 4.35 – Creating a combination of loads for the iterative calculation of bed coefficients

Click the "New combination" button and set the coefficients equal to 1, also check the boxes for taking into account the coefficients of reliability and the fraction of

duration and click the "Record combination" button. The loads will be brought to the standard ones that should be used to calculate the bed coefficients.

We perform a calculation to calculate the stresses from the created combination of forces.

We return to the calculation scheme, using the visualization panel we cut off the foundation slab. On the assignment tab, click the already familiar button "calculation of bed coefficients" and select all the elements of the foundation slab and click the "Confirm" button. To calculate the coefficients of bedding, we choose the created combination C1.

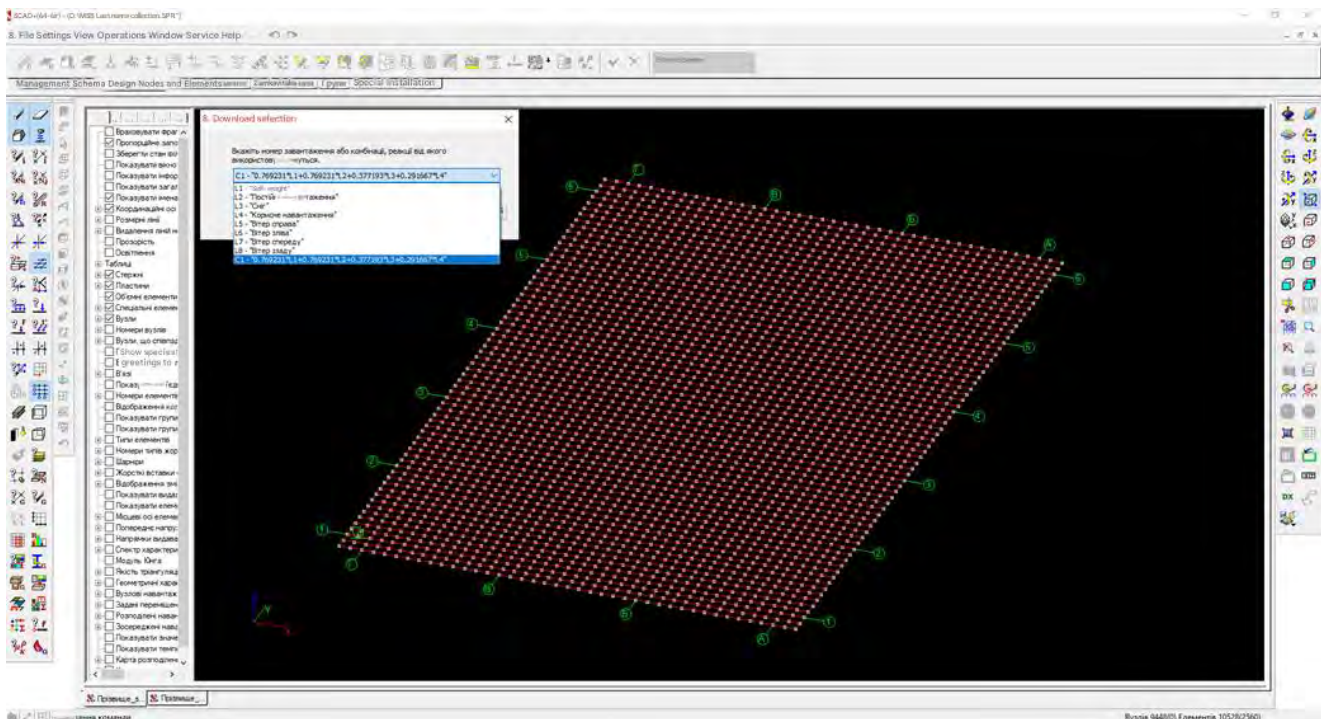


Figure 4.36 – Selection of foundation slab elements and combinations for calculating bedding coefficients

Select the previously created plane and click "Continue with the selected plane". In the CROSS window, immediately change the parameters of the grid for placing the foundation slab of the building and place the foundation slab.

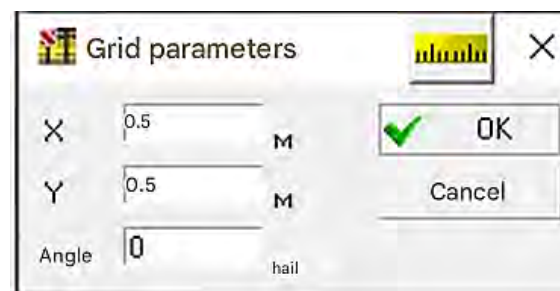


Figure 4.37 – Changing the pitch of the plane grid

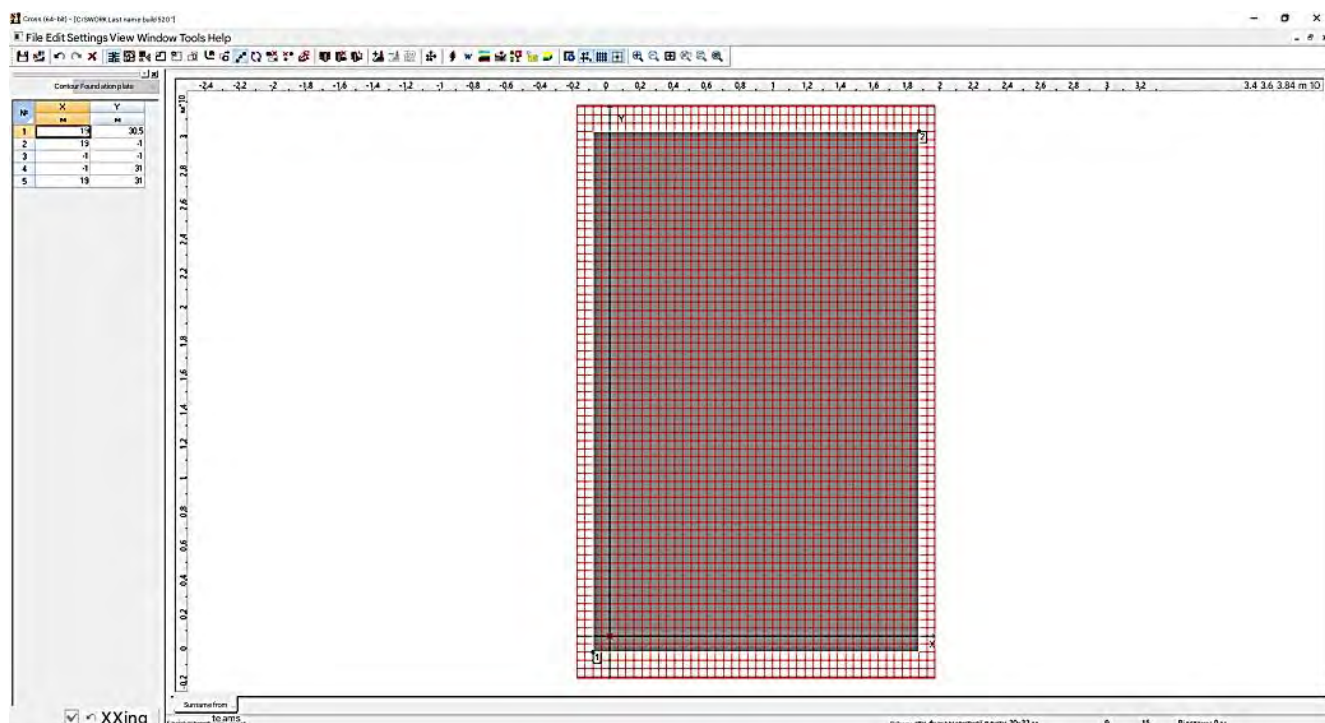


Figure 4.38 – Installing the foundation slab on the plane

Assign the mark of the slab sole, leave the load 0 to apply the load from the calculated combination with SCAD ++.

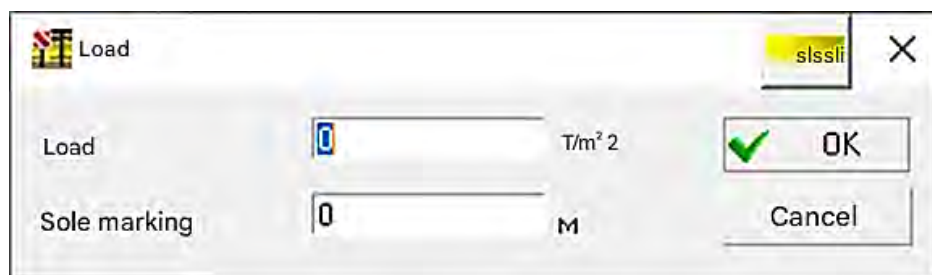


Figure 4.39 – Purpose of the foundation slab mark

Calculate the bedding coefficients and click the apply button. Change the gradation of the output of values to 10 and press the button C_2 to calculate the C_2 coefficient and click the "Apply" button. Click the save data for SCAD button. Close the CROSS window.

Calculation results

Minimum bed coefficient value	563,551	T/m ³ 3
Maximum bed coefficient value	2334,851	T/m ³ 3
Average bed ratio	685,892	T/m ³ 3
Standard deviation of the bed coefficient	0.008	T/m ³ 3
Maximum settlement	0.614	CM
Average settlement	0.493	CM
The mark of the compressible layer was determined at the point with coordinates	0x0	M
The lower mark of the compressible layer at a given point	-4,000	M
The thickness of the compressible layer at a given point	4,000	M
Tilt of the foundation slab	0,000	fra.1
Total load	2110,391	T

Gradations:

	Min T/m ³ 3	Max T/m ³ 3
1	563,551	740,681
2	740,681	917,811
3	917,811	1094,941
4	1094,941	1272,071
5	1272,071	1449,201
6	1449,201	1626,331
7	1626,331	1803,461
8	1803,461	1980,591
9	1980,591	2157,721

Figure 4.40 – Results of calculation of bed coefficients in CROSS

Assignment of elastic base coefficients

☐ Assigning an individual value to each element

☒ Assigning a given number of coefficients

☐ Assign C1-C, (according to clause 6.1.3 of SP 26.13330)

☒ Average values

Number of coefficients:

☒ The value of the coefficients ☐ Maximum values

	The beginning of the interval	Value T/m ³ 3
1	1,575,27	640,09
2	704.9	769.71
3	834.52	899.34
4	964.15	1028.96
5	1093.77	1158.59
6	1223.4	1288.21
7	1353.02	1417.84
8	1482.65	1547.46
9	1612.27	1677.09
10	1741.9	1871.53

Figure 4.41 – Transfer of calculation results to SCAD ++

We perform a linear calculation with new bed coefficients. And we perform the next iteration of the calculation of bed coefficients.

Assignment of elastic base coefficients

☐ Assigning an individual value to each element
☒ Assigning a given number of coefficients

☐ Assign $C_1 = \mu C_2$ (according to clause 6.1.3 of SP 26.13330)

☒ Average values
☐ Minimum values
☐ Maximum values

Number of coefficients:

☒ The value of the coefficients

Calculate

	The beginning of the interval	Value
		3 T/m ³
1	577.33	652.66
2	727.98	803.31
3	878.64	953.97
4	1029.29	1104.62
5	1179.95	1255.27
6	1330.6	1405.93
7	1481.26	1556.58
8	1631.91	1707.24
9	1782.56	1857.89
10	1933.22	2083.87

Figure 4.42 – Results of the next iteration of the calculation of bed coefficients

If the results of the next iteration do not exceed the results of the previous one by more than 10%, the selection of bed coefficients can be considered completed. If they exceed, the iteration "Linear calculation – calculation of coefficients" is repeated until the condition is met.

After the last iteration, we perform a linear calculation of the "foundation-foundation-structure" system and proceed to graphical analysis.

We check possible changes in the stress-strain state of the building frame and, if necessary, repeat the selection of sections of the steel frame elements.

We check possible changes in the design reinforcement of reinforced concrete floor slabs and coatings and, if necessary, repeat the selection of reinforcement.

In case of changes in the rigidity of the elements, we carry out a repeated linear calculation, calculation of bed coefficients, and calculation of the replaced sections of the frame elements.

