

MINISTRY OF EDUCATION AND SCIENCE OF UKRAINE

**O. M. BEKETOV NATIONAL UNIVERSITY
of URBAN ECONOMY in KHARKIV**

METHODOLOGICAL RECOMMENDATIONS

to conduct practical classes and
independent work
in the academic discipline

«MODELING OF A SOIL-STRUCTURE INTERACTION SYSTEM»

*(for applicants for the second (master's) level of higher education of all forms of study
in the specialty G19 – Building and Civil Engineering)*

**Kharkiv
O. M. Beketov NUUE
2025**

Methodological recommendations to conduct practical classes and independent work in the academic discipline «MODELING OF A SOIL-STRUCTURE INTERACTION SYSTEM» *(for applicants for the second (master's) level of higher education of all forms of study in the specialty G19 – Building and Civil Engineering)* / O. M. Beketov National University of Urban Economy in Kharkiv ; comp. : V. A. Aleksandrovykh, O. V. Havryliuk. – Kharkiv : O. M. Beketov NUUE, 2025. – 138 p.

Compilers: Cand. Tech. Sci., Assoc. Prof. V. A. Aleksandrovykh,
Senior Lecturer and PhD student O. V. Havryliuk

Reviewer

Samorodov O. V., Doctor of Technical Sciences, Professor, Professor of the Department of Geotechnics, Underground Structures and Hydrotechnical Construction, O. M. Beketov National University of Urban Economy in Kharkiv

Recommended by the Department of Geotechnics, Underground Structures and Hydraulic Engineering, Protocol № 12 on June 23, 2025

CONTENT

INTRODUCTION.....	4
TOPIC 1 CONSTRUCTION AND ANALYSIS OF THE SOIL BASE MODEL IN THE FORM OF A LINEARLY DEFORMABLE LAYER OF FINITE WIDTH.....	5
TOPIC 2 CONSTRUCTION OF A SOIL MODEL USING THE SUBGRADE REACTION MODULUS.....	15
TOPIC 3 CONSTRUCTION OF THE «SOIL-FOUNDATION-STRUCTURE» MODEL.....	34
TOPIC 4 CALCULATION AND ANALYSIS OF THE «SOIL-FOUNDATION-STRUCTURE» MODEL.....	86
TOPIC 5 ANALYSIS OF THE MUTUAL INFLUENCE OF ADJACENT BUILDING FOUNDATIONS.....	112
TOPIC 6 ANALYSIS OF STRESS DISTRIBUTION IN A LINEARLY DEFORMABLE SOIL LAYER MODEL.....	127
LIST OF RECOMMENDED SOURCES.....	136

INTRODUCTION

The purpose of teaching the discipline "Modeling of a Soil-Structure Interaction System" is to train construction specialists who have skills in calculating the stress-strain state of the structure and its base as a single system in accordance with the current regulatory documents in construction, using software based on the finite element method. Training of a future specialist in such methods of work is an extremely important part of professional training and should be built on the basis of modern software complexes used in professional practice.

Practical work is carried out using the SCAD software package. The SCAD software package is widely used in professional engineering practice, so its study contributes to the formation of modern engineering thinking in students and increases their competitiveness in the labor market. Within the framework of practical classes, the basic principles of building design models, the use of loads, the assignment of boundary conditions, the analysis of the results obtained, the assessment of the stress-strain state of structures and design decision-making are considered.

The guidelines include step-by-step instructions and recommendations for building schemes, performing calculations, interpreting results and making project decisions, as well as tasks for independent work designed to activate the research component of students' education. The material is presented taking into account the level of training of students and provides for the performance of individual tasks during practical classes and independent work of applicants. Initial data for individual tasks are given during the first lesson by name (indicating the name of the applicant on the sheet with the initial data).

The methodological recommendations are compiled in accordance with the normative provisions for conducting practical classes and organizing independent work at O. M. Beketov Kharkiv National University of Urban Economy and are intended for applicants of the second (master's) level of higher education of all forms of study in the specialty G19 – Construction and Civil Engineering.

TOPIC 1

CONSTRUCTION AND ANALYSIS OF THE SOIL BASE MODEL IN THE FORM OF A LINEARLY DEFORMABLE LAYER OF FINITE WIDTH

There are analytical spatial models of a continuous linear-deformed environment (half-space or layer), which were developed by well-known scientists and were adopted in regulatory documents for calculating the deformations of foundation foundations and remain in the state construction norms of Ukraine. However, today, due to the powerful development of information technologies, the most widespread model of the soil base is the model of a continuous layer of finite width (Fig. 1.1), which, in addition to limiting vertical deformations at a certain depth H , also has limitations on horizontal deformations at a certain distance from the load in plan ($L_x \times L_y$). For calculating foundations that interact with a soil foundation under plane strain conditions, this model can also be a plane model, as a special case. When modeling and performing numerical calculations of the "soil-foundation-structure" (SFS) system in powerful computational complexes such as SOFiSTiK, ABAQUS, Plaxis, SCAD, Lira, etc., the soil foundation model in the form of a continuous layer of finite width is typically used. In this approach, any patterns of soil deformation under loads can be specified, including time-dependent behavior. [1].

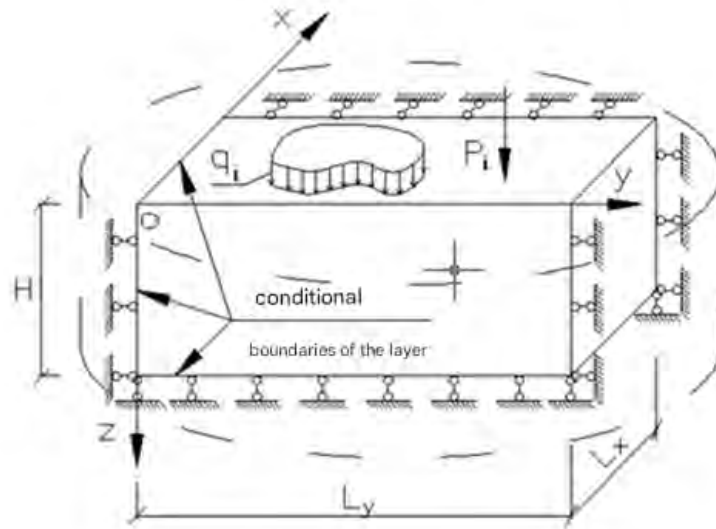


Figure 1.1 – Model of the soil base in the form of a continuous layer of finite width

It is well known that in order to obtain reliable and reliable results for calculating the stress-strain state of foundation structures in the "SFS" system, it is necessary to

choose such a model of the soil base with appropriate parameters, which is close to the real soil environment according to two criteria: dissolving capacity and deformability of the foundations of structures. In practice, the model of a linear-deformed layer of finite width is the most common, since it requires only the task of deformation characteristics of the soil (modulus of total deformation E and Poisson's ratio ν). However, a significant resolution of the model causes the occurrence of excessive reactions at the edges of foundation structures and, accordingly, an increase in moment forces from the total bending of foundations, especially large-sized ones.

At present, there is no single approach to the dissolving capacity of soil foundations, therefore, the study of the influence of boundary conditions in the plan (according to the angle of distribution α) of the model of a continuous linear-deformed layer of finite width on the stress-strain state of foundation structures is an urgent task for adequate modeling of the "SFS" system.

Creating a flat model of the soil base.

With the help of the SCAD software package, we will create a model of the soil base in a flat problem statement

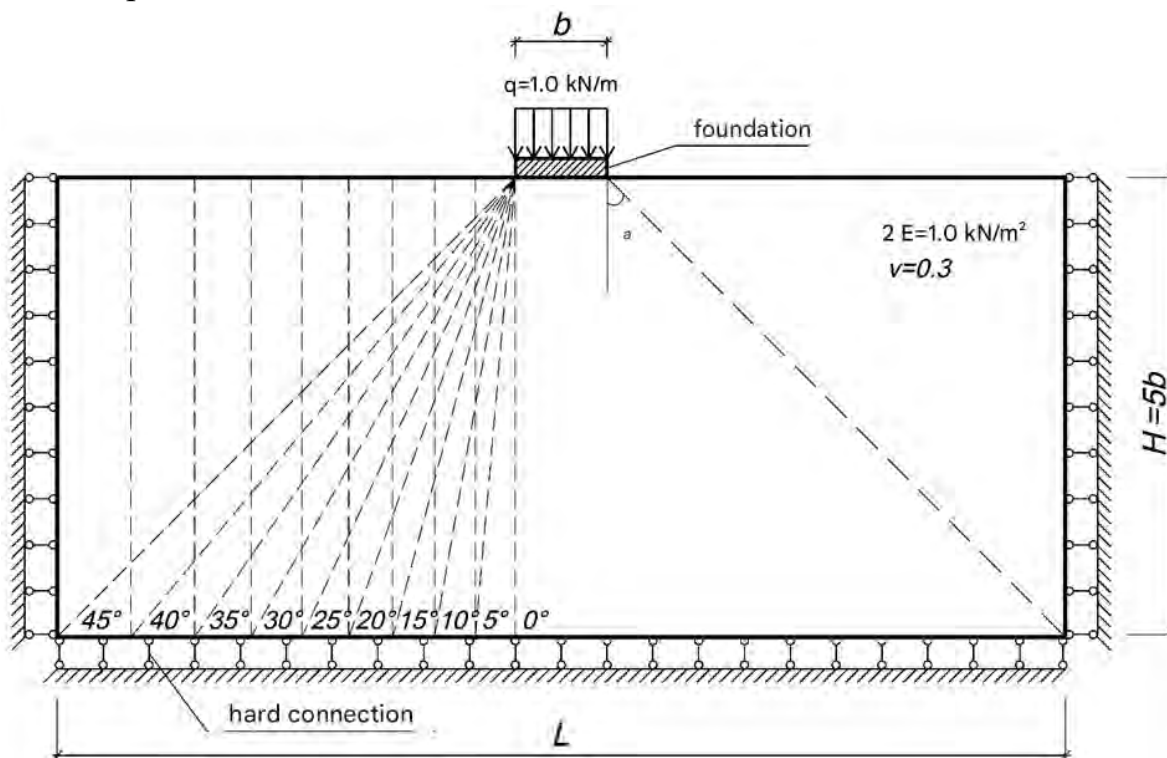


Figure 1.2 – Diagram for creating a flat model of the soil base

Create a new project in the SCAD++ application. In the name field, enter your last name, select the type of scheme "2 – Flat frame" and click OK (Fig. 1.3).

Download the stencil diagram at the link <https://tinyurl.com/y33p2f8a> and open it in AutoCAD (Fig. 1.4).

Project identification

Name:

Organization:

Object:

OK Reference Cancel

Reliability coefficient by responsibility:

Reliability factor for responsibility (second limit state):

Scheme type:

Location in space	Movement in a node in the direction of the common coordinate system	Characteristics of constructions
XOZ plane	Linear along X, Z; UY rotation around oci Y	Flat frame

Figure 1.3 – Dialog box for creating a new project

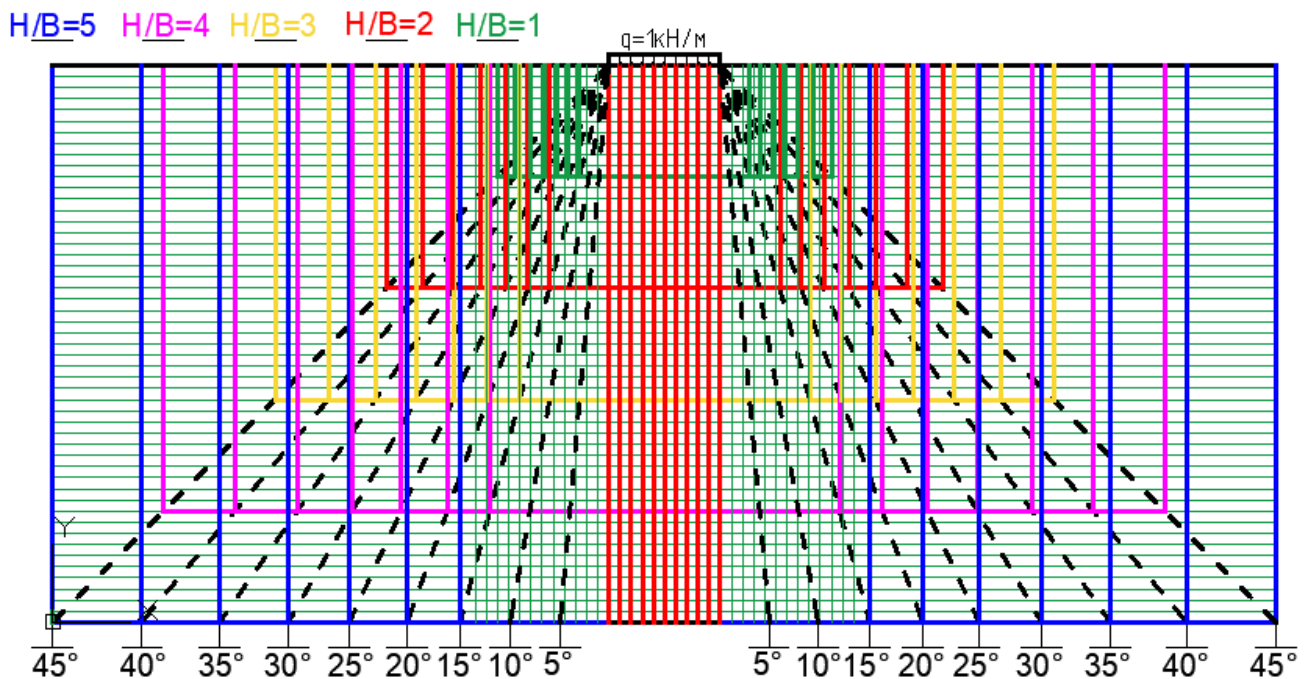


Figure 1.4 – Scheme of a stencil for the formation of a design scheme

Adjust the stencil scheme according to your individual task. Remove additional symbols from the diagram and save it under the name "Last Name_HB = _Y =.dwg", where HB = the ratio of the depth of the soil model to the width of the foundation, Y = the angle of distribution in the soil mass according to the individual task. Use the File-Import-DWG import function to import the saved file.

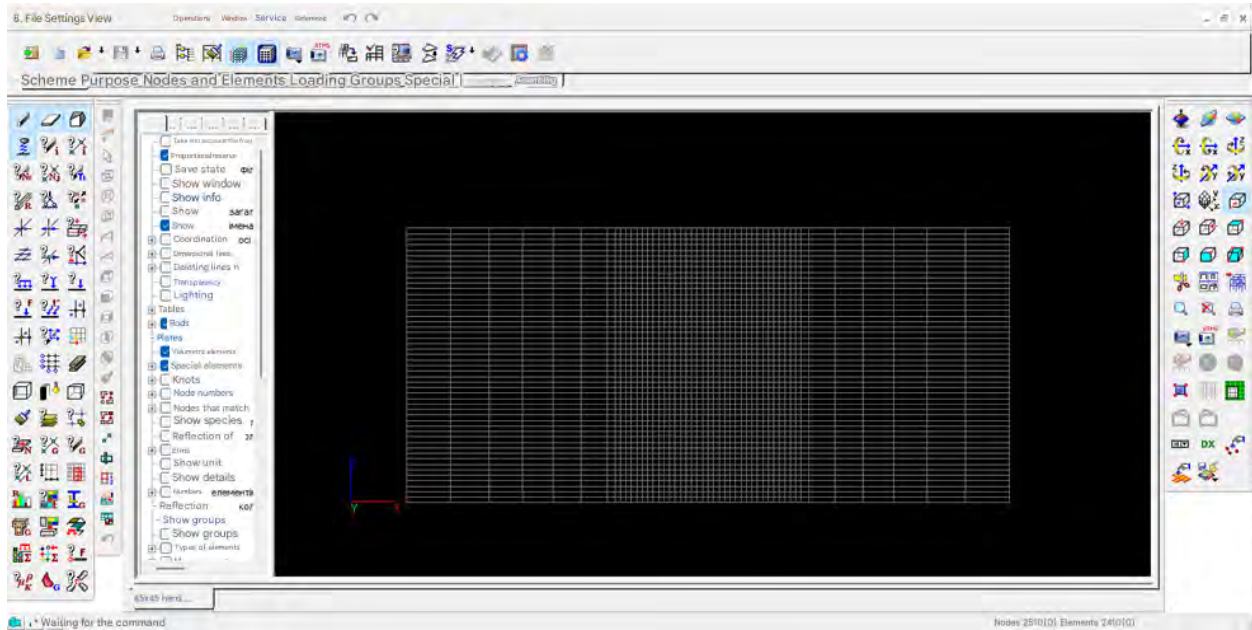


Figure 1.5 – Result of importing the calculation scheme

Next, we will create a load model from a flat foundation of unit length. To do this, we build a flat rod in the middle of the calculated soil model, for ease of use, we set a slight excess of the coordinates "Z" above the surface of the soil model (for example, 1.0 m). We divide the unit length foundation model into 10 equal parts using the rod split function on the nodes and elements tab (Fig. 1.6).

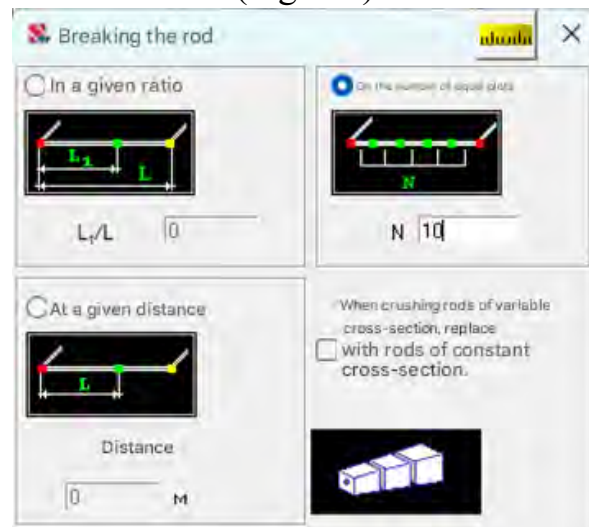


Figure 1.6 – Rod Partition Dialog

Combine the displacements of the corresponding nodes of the foundation model with the soil base model using the function of combining displacements on the "Purpose" tab (Fig. 1.7). Having set the directions of joining X and Z, press the OK button and select the corresponding nodes in pairs and press the confirm or "Enter" button.

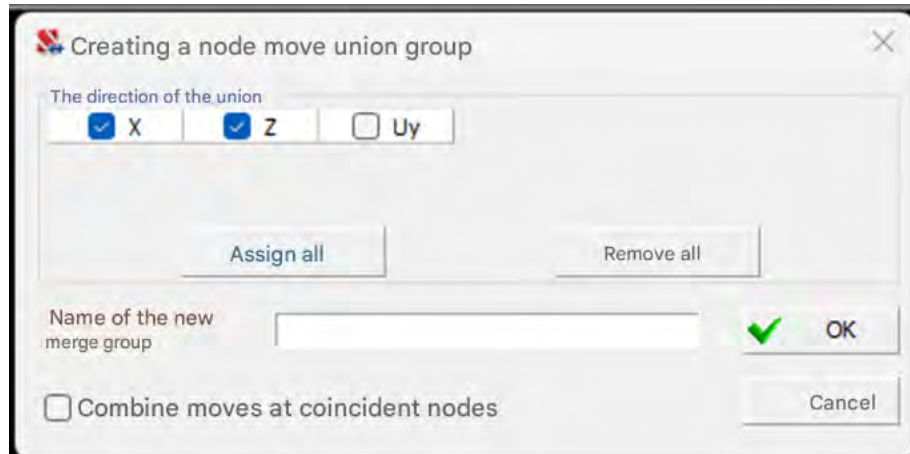


Figure 1.7 – Movement Merge Dialog Box

Check the correctness of the merge using the function of displaying the merge of movements in the filter panel. The corresponding nodes in the diagram must have the same color and union designation in the specified directions. Further, the movement of the nodes of the foundation model will occur together with the corresponding nodes of the surface of the soil model (as if the foundation lies on its surface).

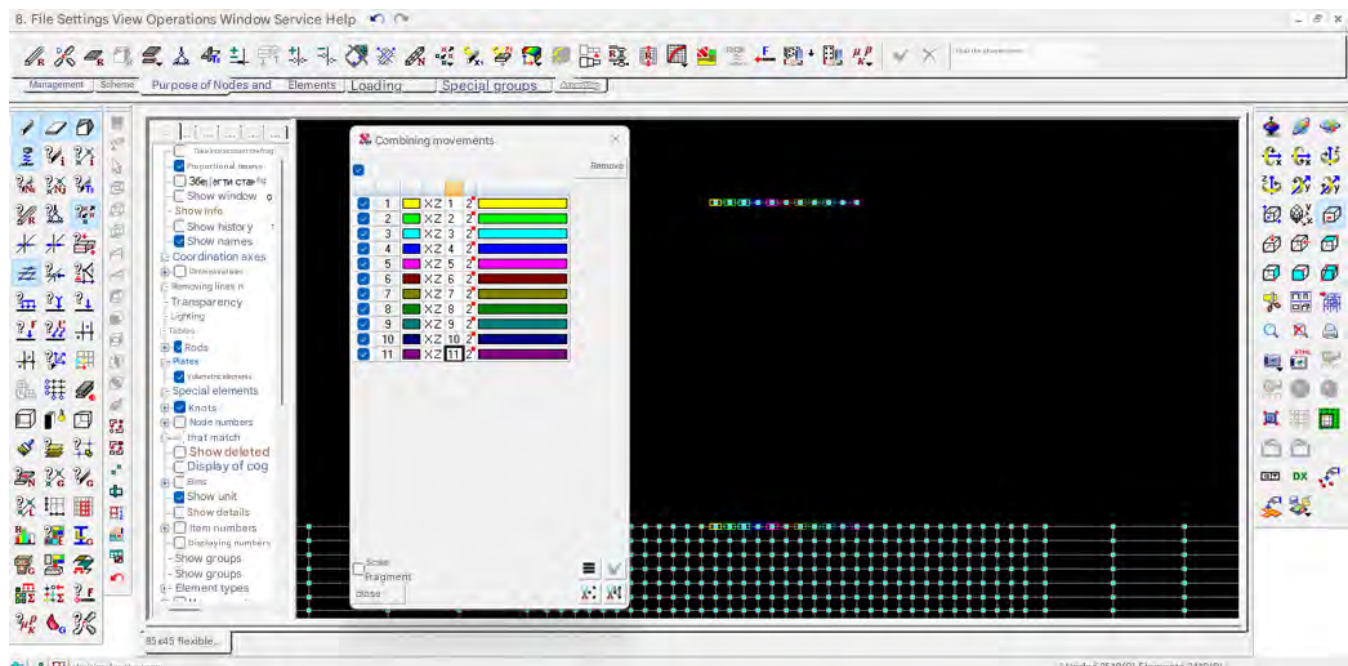


Figure 1.8 – Displaying the "Merge Displacements" filter

We assign soil models and foundation models to stiffness. To do this, go to the "Assignment" tab and click the button for assigning "Stiffness to plates" and "Stiffness to rods" to perform the appropriate actions.

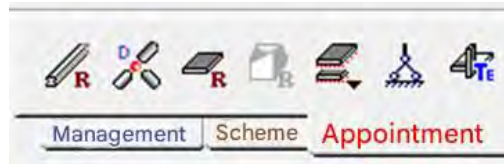


Figure 1.9 – Buttons for assigning stiffeners to plates and rods on the corresponding tab

We assign stiffness to the soil mass and foundation according to the individual task using the appropriate dialog boxes with parameters according to the individual task.

Rigidities of the plates

Isotropy

Material: Others...

Bulk weight: 18 kN/m³

Parameters:

Modulus of elasticity: 10 MPa

Poisson's ratio: 0.3

Linear expansion coefficient: 1.e-005 1/°C

Plate thickness: 1 m

Damping parameter (in fractions of critical): 0

Reduction factors: ?

Types of stiffness:

Thickness	The name of the stiffness type
1	1 Basis

Physical nonlinearity: ☐ Arbitrary mate ☐ Ferroconcrete

Isotropy: ☒ Orthotropy: ☐ Constructive orthotropy: ☐ Flat-tension state: ☐ Flat deformation: ☒

The name of the stiffness type: Basis

☐ Use description as name

Replace and exit Replace and continue OK Cancel Reference

Figure 1.10 – Soil Base Model Stiffness Assignment Dialog Box

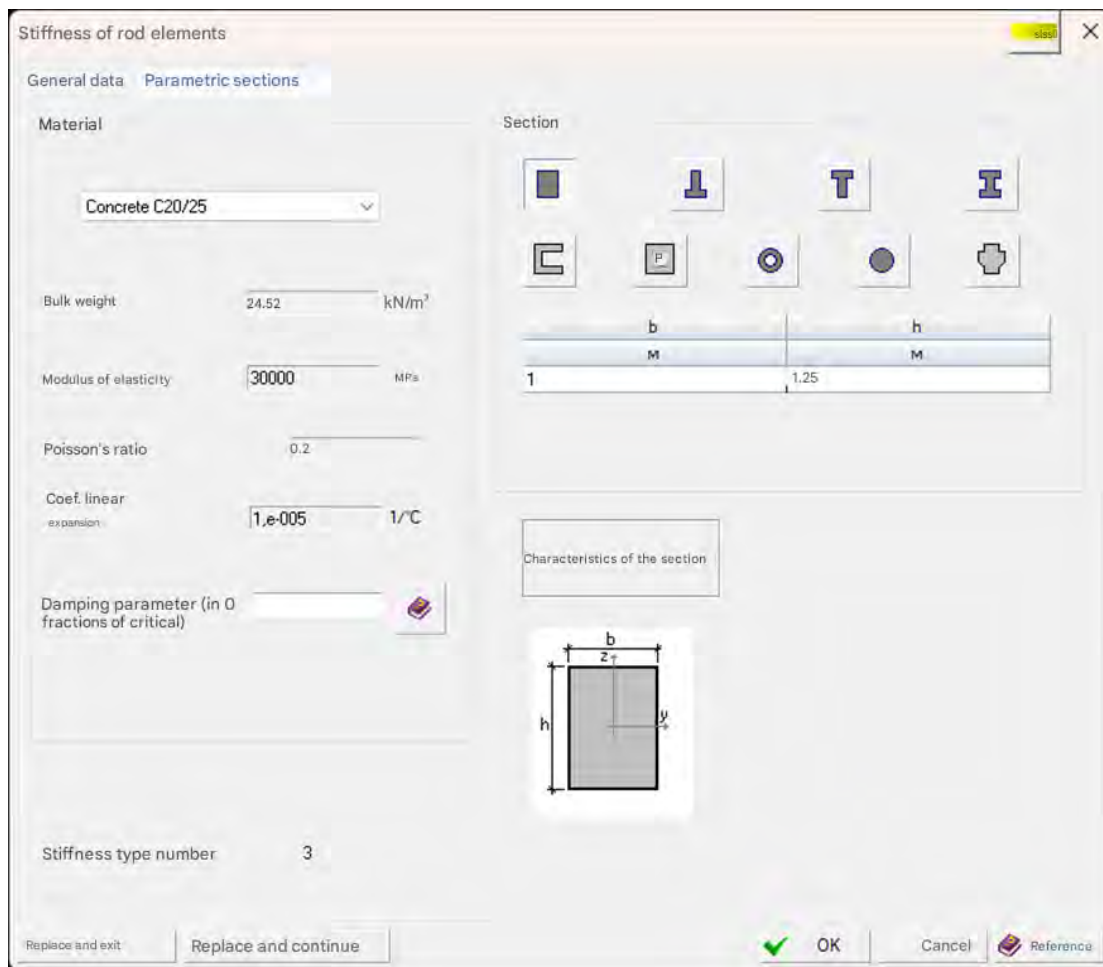


Figure 1.11 – Stiffness assignment dialog box of the foundation model

In accordance with the task, assign the load to the foundation using the button for setting loads on the core elements on the "Loading" tab.

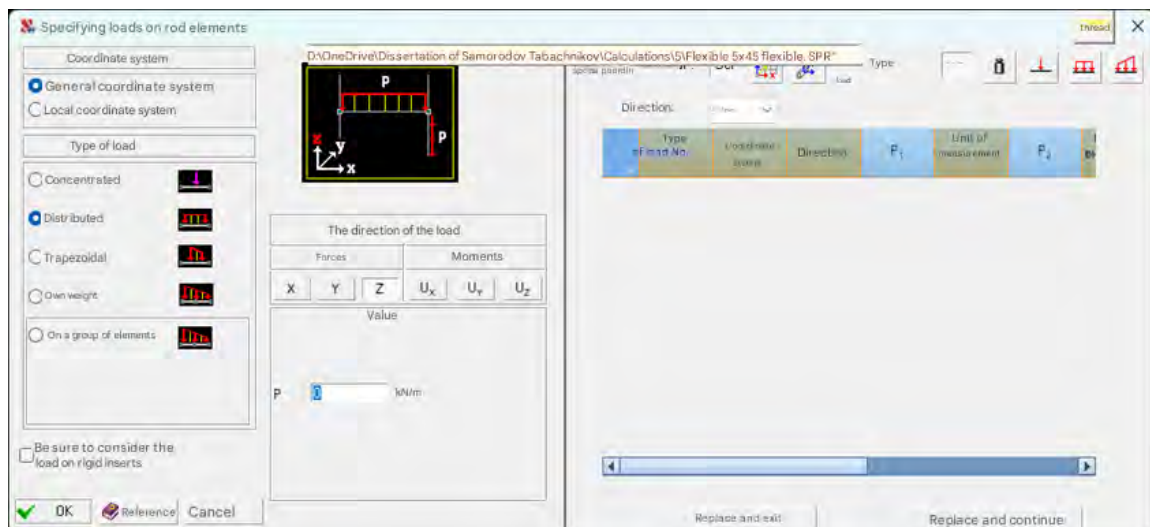


Figure 1.12 – Dialog box for setting downloads

Next, assign boundary conditions to the soil model (anchoring it in space) using the button for assigning elms in nodes on the assignment tab. For the nodes of the lower face of the model, assign fastening in the Z direction, for the nodes of the side faces – in the X direction.



Figure 1.13 – Load Setting Dialog Box

We check the correct installation of elms using the button for displaying elms in the filter panel.

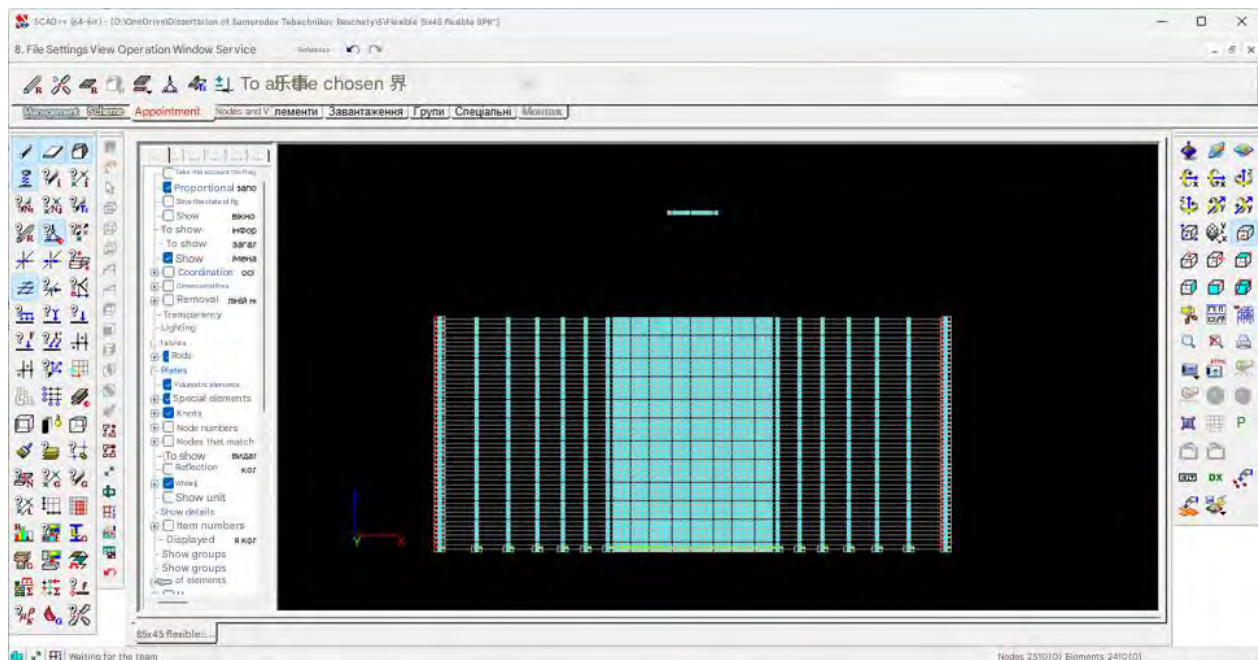


Figure 1.14 – Display of the design scheme with assigned boundary conditions

Go to the project management screen and perform a linear calculation. If everything is done correctly, then the linear calculation will be carried out without errors, after its completion we go to the "Graphical Analysis" module.

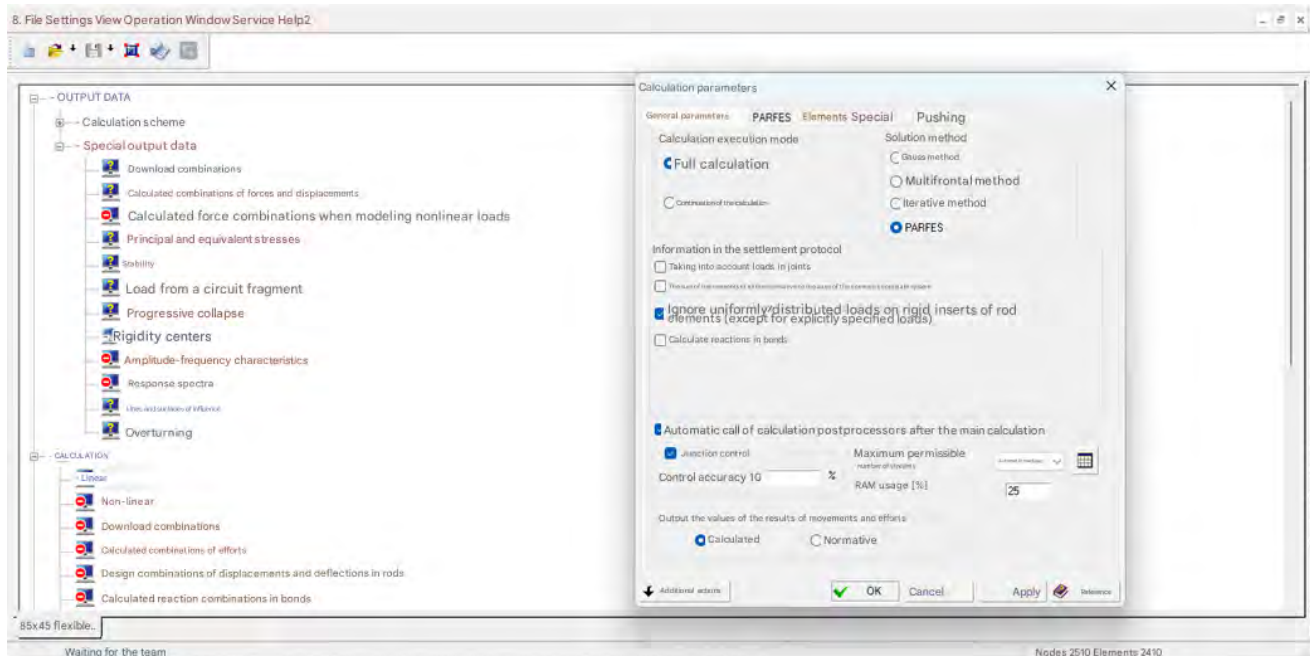


Figure 1.15 – Project management screen and running linear calculation

On the "Displacements" tab, we analyze the resulting displacements and calculate the value of the average subsidence of the foundation model S_{ser} .

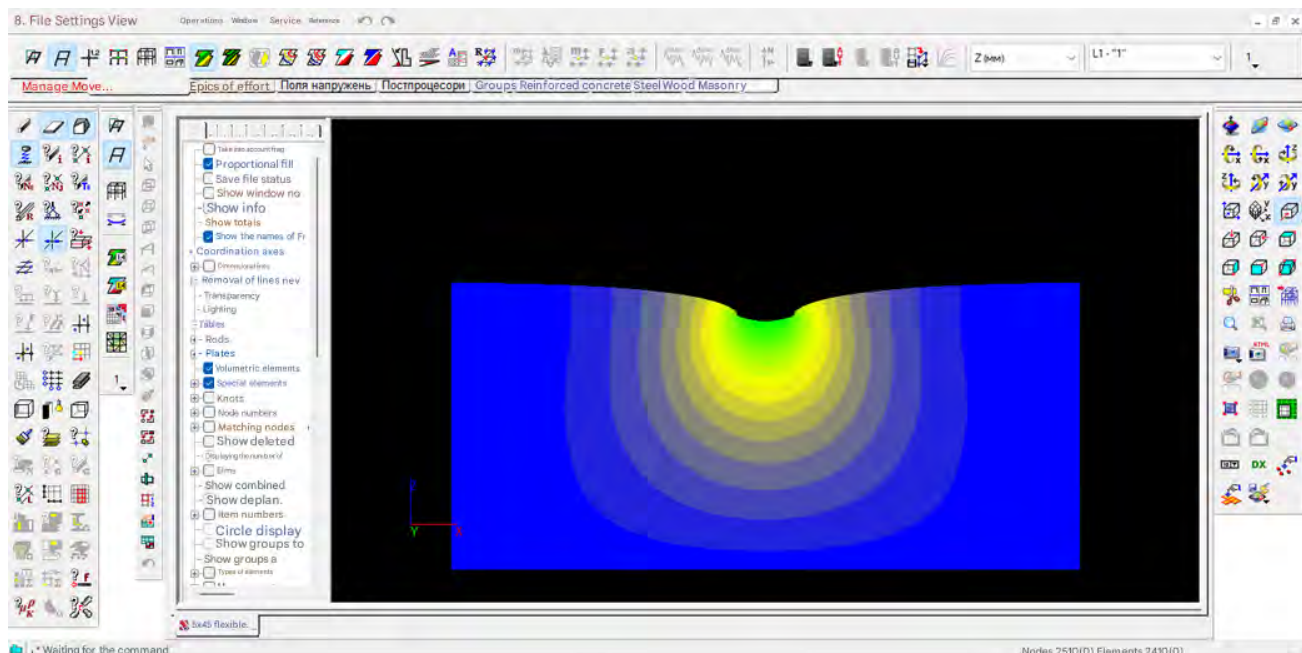


Figure 1.16 – Soil Model Deformation Analysis Screen

Go to the tab "Plots of Effort" , turn on the display of the plot of moments for the foundation and fix the value of the maximum moment in the middle of the span.

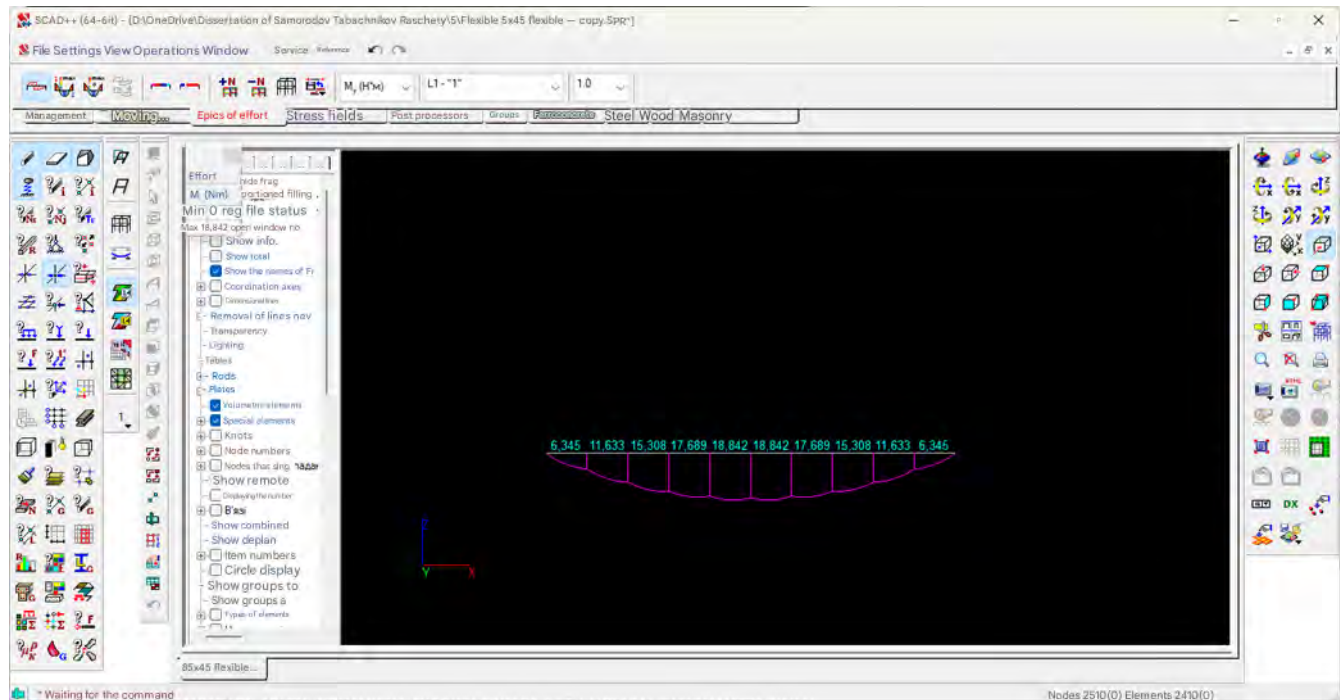


Figure 1.17 – Bending moment analysis screen in the foundation model

We repeat the entire described procedure, creating a calculation model of the soil with a distribution angle of 45° , 25° and 0° . We take the H/B ratio in accordance with the individual task.

Table 1.1 – Results of the analysis of the influence of boundary conditions of the model of a soil base of finite width.

Schema code	Average subsidence, mm	Maximum torque, kN/m
Surname HB=5 Y=45		
Surname HB=5 Y=25		
Surname HB=5 Y=0		

Tasks for independent work on topic 1. Perform the calculation of the subsidence of the foundation according to the model of a linear-deformed medium (in the program "REQUEST") using identical parameters of dimensions, soil characteristics and load. Compare the results of the calculation with the results of subsidence determination for different distribution angles from Table 1.1.

TOPIC 2

CONSTRUCTION OF A SOIL MODEL USING THE SUBGRADE REACTION MODULUS

The calculation of the coefficients of the bed under the foundation slab is carried out by the "Cross" program on the basis of geological survey data, presented in the form of well characteristics. The initial data for the calculation includes information about the construction site, which indicates the dimensions of the site, the position of the wells and their characteristics, the position and geometry of the buildings and structures existing on the site, as well as other data necessary to calculate the bed coefficients. This information must be prepared in advance in the Cross program and recorded in the form of a file with the .crs extension. With each subsequent access to the program, the information can be used by calling the desired file.

To determine the coefficients of the bedding, it is most often necessary to perform several iterations, using, for each subsequent iteration, the values of the pressure on the ground under the foundation slab, obtained in the previous step. As a pressure on the soil, the value R_z is used – the reaction of the soil. Before the first calculation of the coefficients is performed, bedding (usually the same for all elements of the slab) are assigned by the user.

Creating a model of a foundation slab.

Create a new project in the SCAD++ application. In the First Name field, enter your last name and click OK.

The screenshot shows the 'New project' dialog box with the following fields and options:

- Name:** A text field with the placeholder 'Name'.
- Organization:** A text field.
- Object:** A text field.
- Standard:** A radio button that is selected.
- Variation of models:** A radio button.
- Assembling:** A radio button.
- Forum:** A radio button.
- Design standards:** A button.
- Reliability coefficient by responsibility:** A dropdown menu with '1' selected.
- Reliability factor for responsibility (second limit state):** A dropdown menu with '1' selected.
- Accuracy of matching node estimation:** A text field with '0.01' and a unit selector 'M'.
- Scheme type:** A dropdown menu with '5-General system' selected.
- I am located in space:** A text field.
- Movement in a node in the direction of the common coordinate system:** A text field.
- Characteristics of constructions:** A text field.
- Any:** A text field.
- Linear along X, Y, Z; rotations UX, UY and UZ around the axes X, Y, Z:** A text field.
- General type scheme feature:** A text field.

Buttons: OK, Cancel, Reference.

Figure 2.1 – Dialog box for creating a new project

In the new window, enter the file name of the new project and its location in the "Name_Plate" format. Click the Save button.

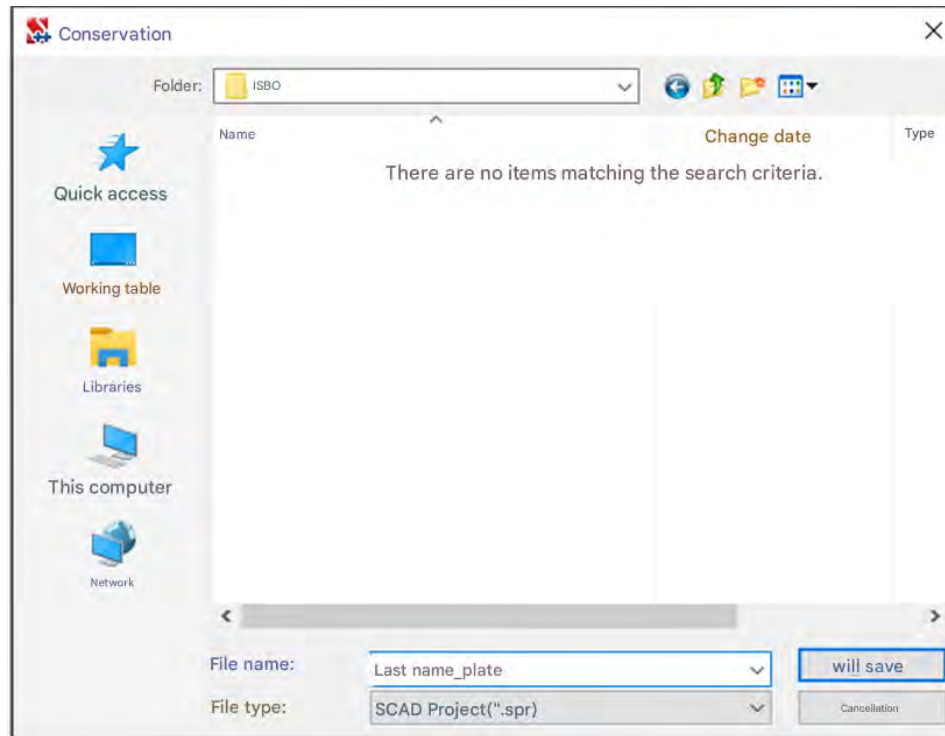


Figure 2.2 – Dialog box for choosing where to save a new project file

In the project tree, go to the calculation diagram tab.

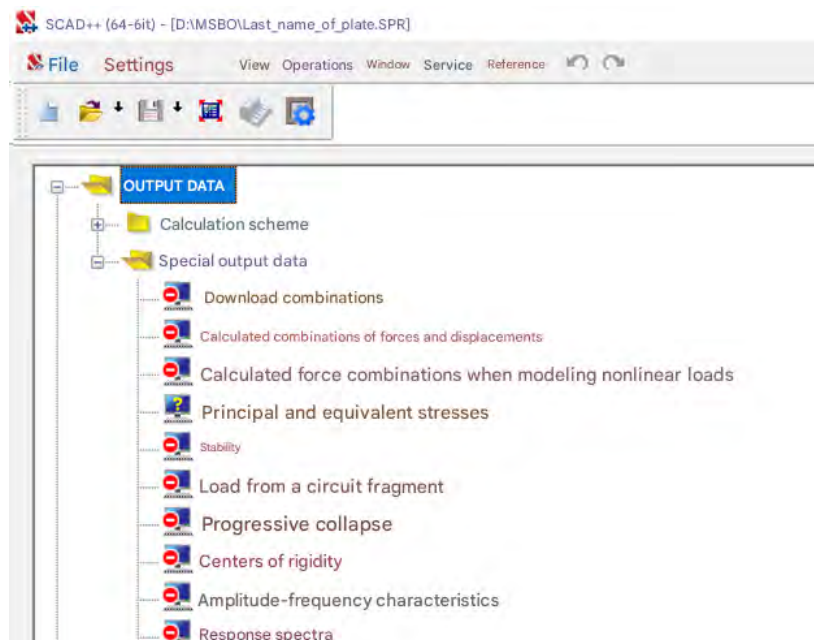


Figure 2.3 – Project tree dialog box

In the main working window of the program, create a grid of stakeout axes according to the individual one. To do this, click the "Set Layout Axis Grid" button on the "Diagram" tab.

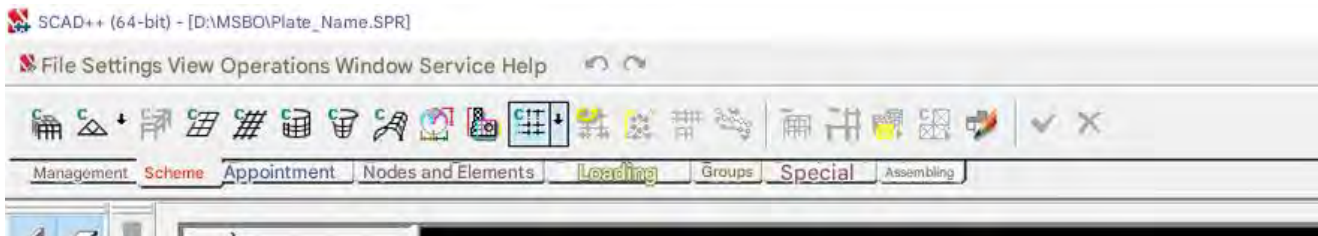


Figure 2.4 – Invoking the Set Partition Grid dialog box

In the Set Split Grid dialog box, enter the pitch of the axes in the longitudinal and transverse direction according to your individual task and click OK.

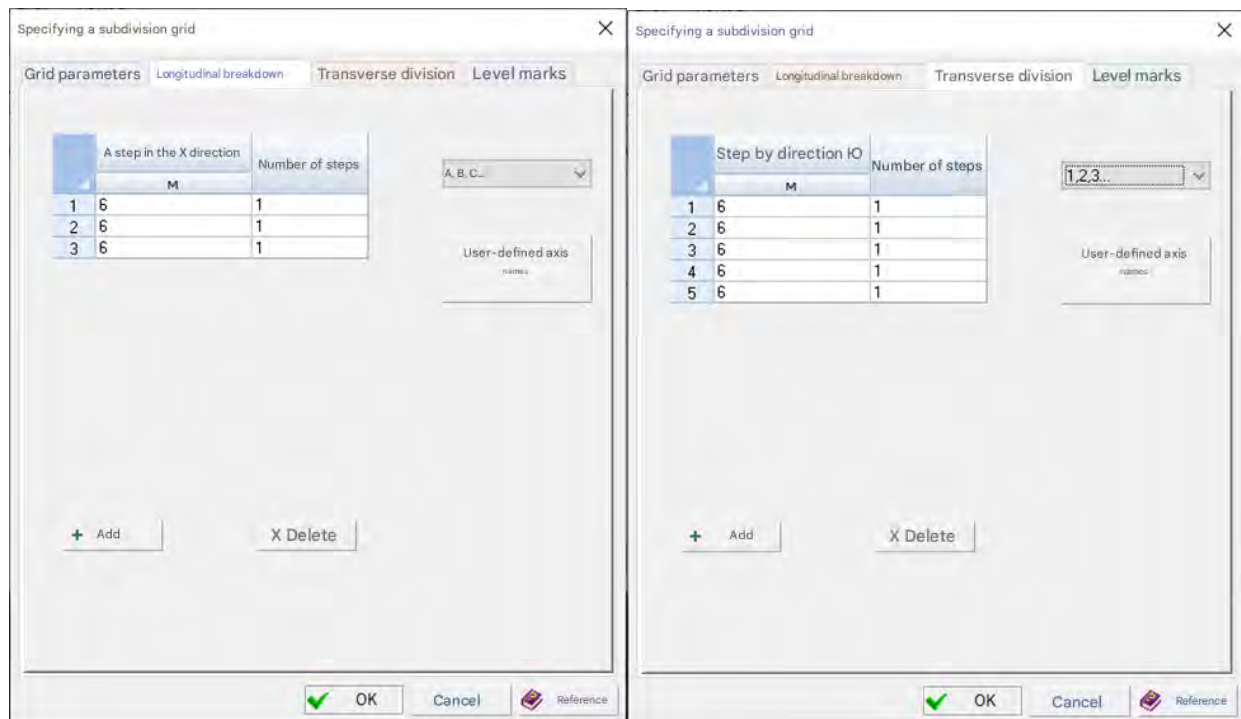


Figure 2.5 – Set Partition Grid Dialog Box

To display the created split grid, use the appropriate tool in the filter panel.



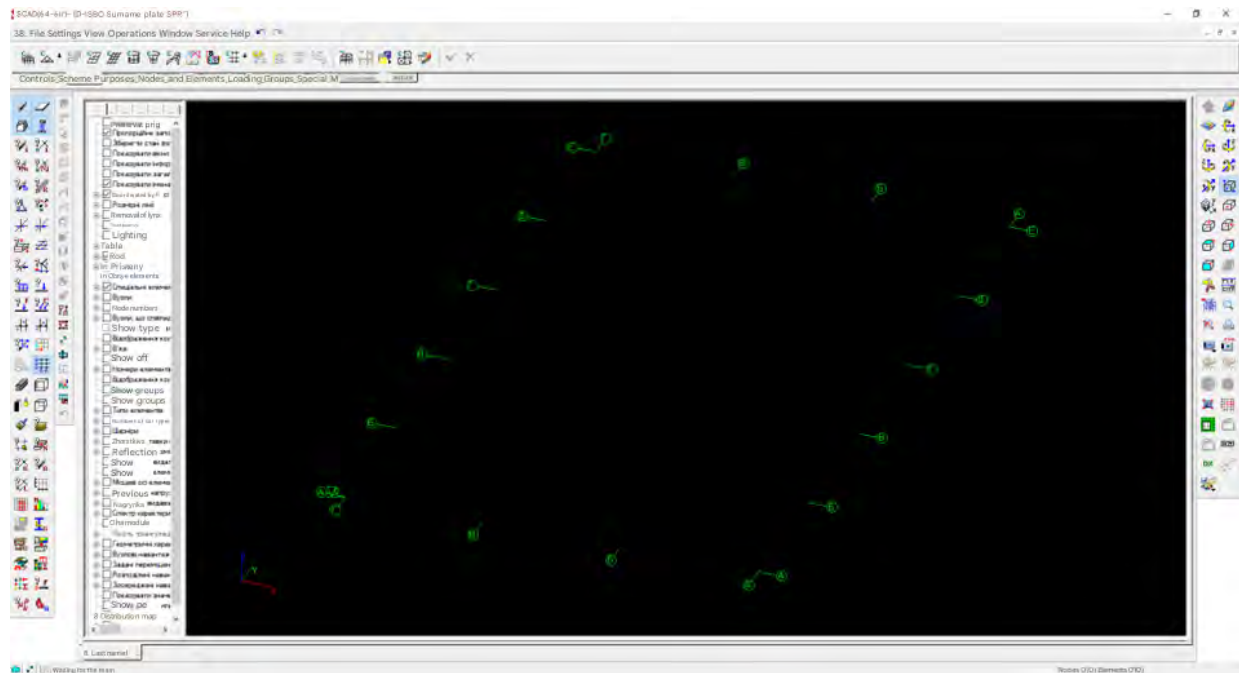


Figure 2.6 – Displaying the Layout Grid in the Program Work Field

Create the contour nodes of the foundation slab of your building according to the obstacle based on the condition that the slab is 1 m larger than the contour of the building in the axes. To do this, use the Node Input tool on the Nodes and Elements tab. Lead 4 nodes with X and Y coordinates 1 m larger than the extreme coordinates of the corner points of the stakeout axes.

Figure 2.7 – Node Input dialog box

Triangulate the created path with rectangular plate finite elements measuring 0.5×0.5 m. on the "Scheme" tab.

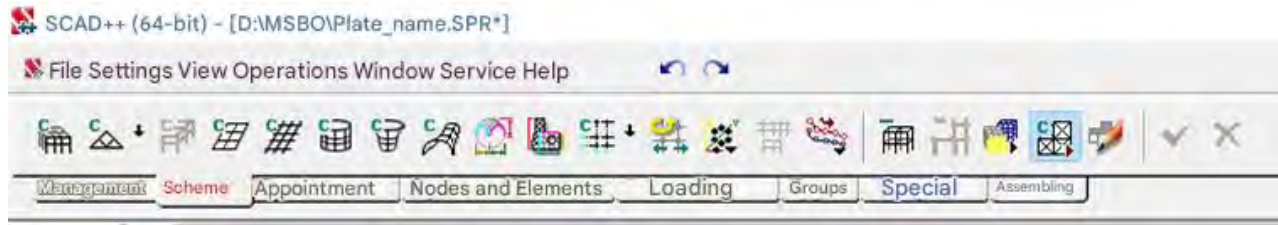


Figure 2.8 – Invoking the triangulation tool in the toolbar

Click the Set Kennel button on the Triangulation toolbar. Traverse the path through the 4 pre-created nodes, finishing the traversal at the first point from which you started.



Figure 2.9 – Triangulation toolbar

After selecting the last point of the outline, the outline of the upcoming triangulation will appear on the screen.

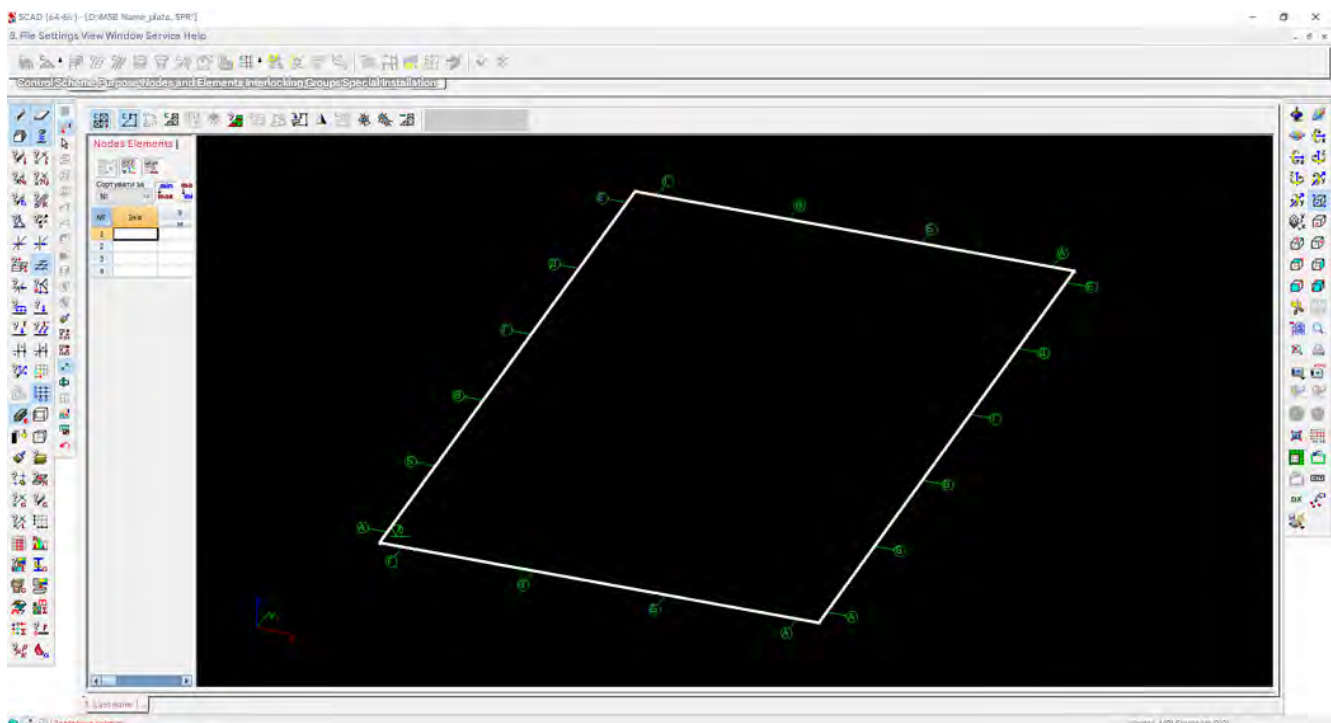


Figure 2.10 – The result of creating a path

After creating the triangulation outline, click  the "Mesh Generation..." button to assign triangulation parameters.

In the new Automatic Triangulation dialog box, set the following options:

- creating an orthogonal stick with a given maximum element size;
- combine three-node elements into four-node ones;
- triangulation step – 0.5m;
- and click OK.

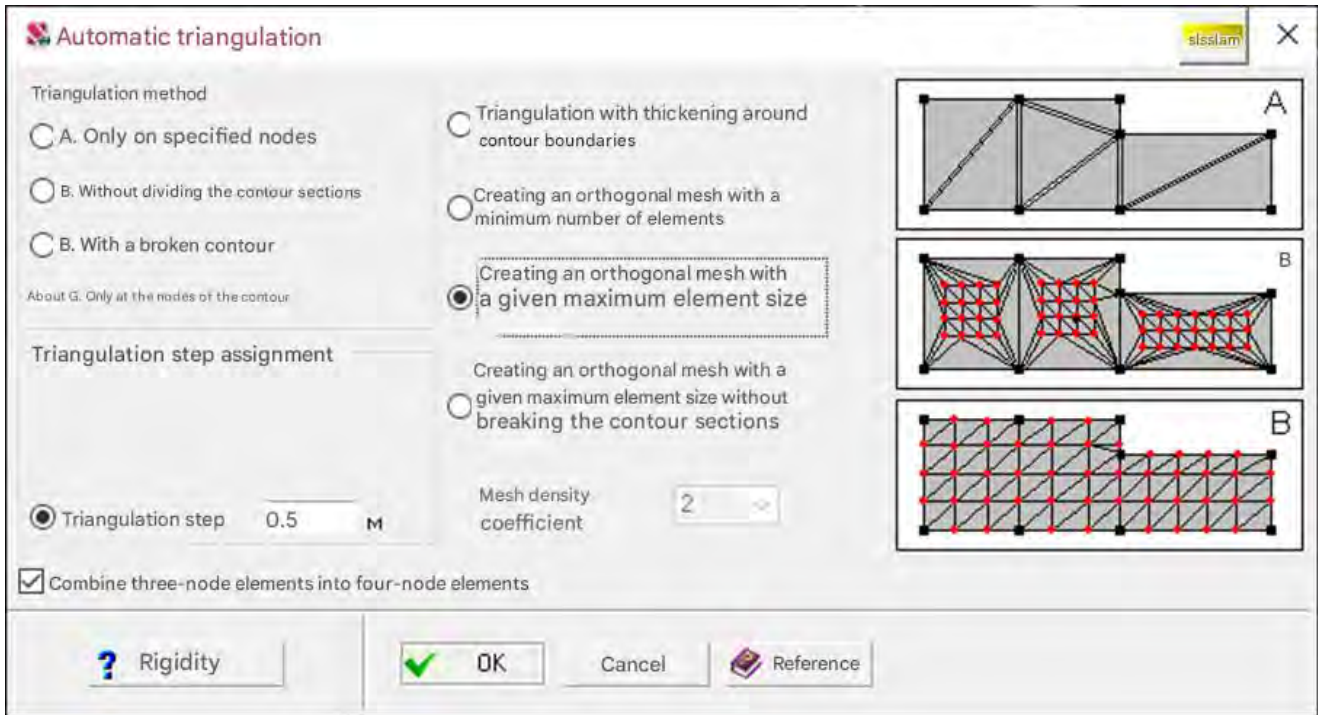


Figure 2.11 – Automatic Triangulation dialog box

Apply the results of the triangulation using the  "Triangulation Placement..." button

Exit triangulation mode by pressing  the button.

After creating the triangulation grid, the created elements must be assigned stiffnesses that correspond to the material of the monolithic reinforced concrete foundation slab. Go to the "Assignment" tab and click the "Assign Stiffeners to Plates" button

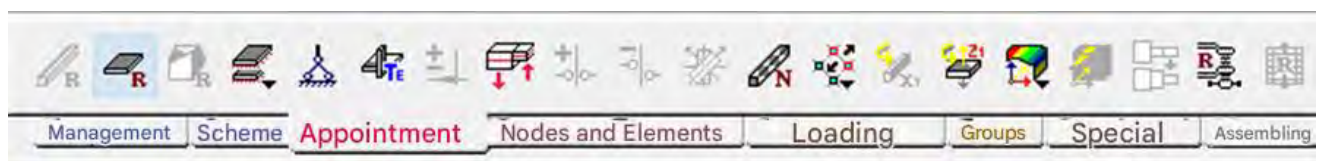


Figure 2.12 – Assigning stiffeners to plates on the assignment tab

In the Plate Hardness dialog box, set the following options:

- material – heavy concrete of class C16/20;
- slab thickness – 0.5 m.

Assign a stiffness type name – Foundation Slab.

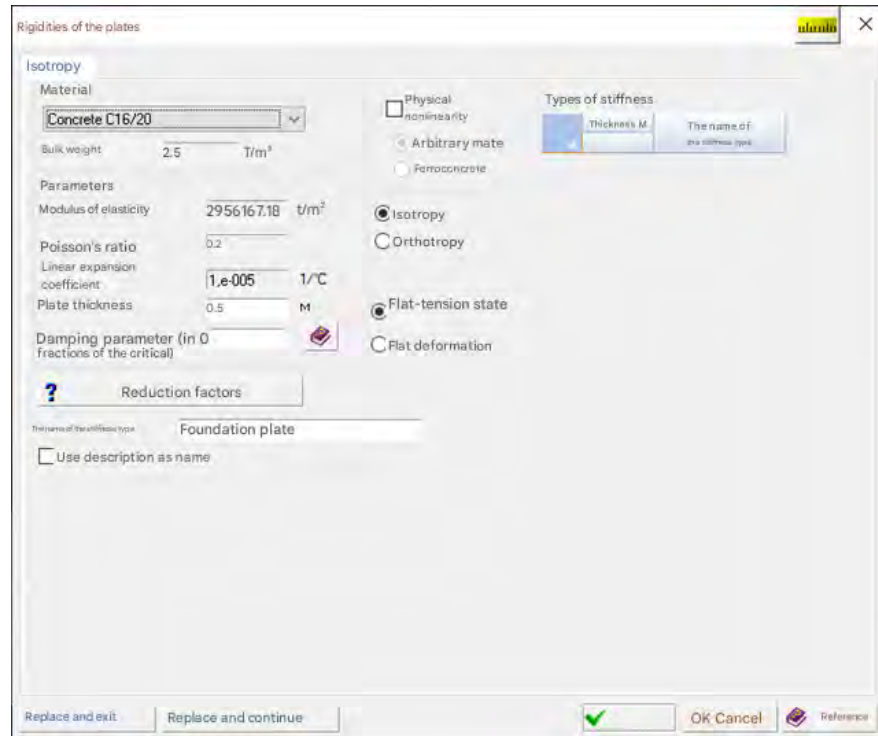


Figure 2.13 – Plate Stiffness Dialog Box

Select all created plate finite elements with a rectangle and press the confirm button.

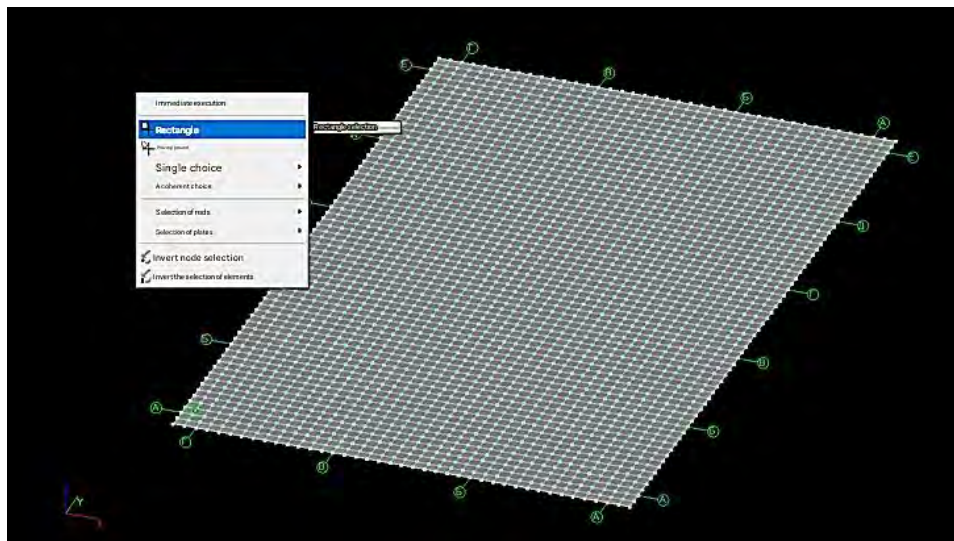


Figure 2.14 – Selection of plates for assigning stiffness in the working field of the program

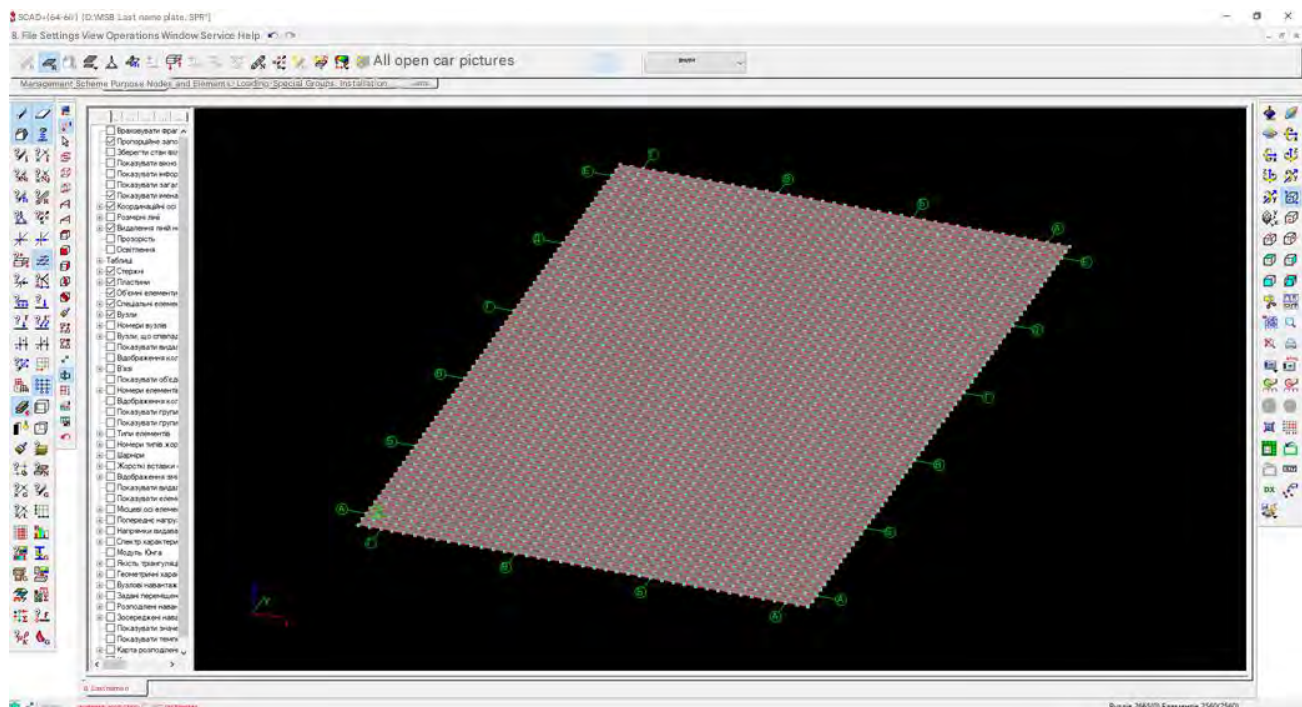


Figure 2.15 – Confirmation of the application of stiffeners to the foundation slab

Right-click on the "Remove Invisible Path Lines" button in the filter panel and check the box for "Show plate thickness" to control the application of the stiffness type.

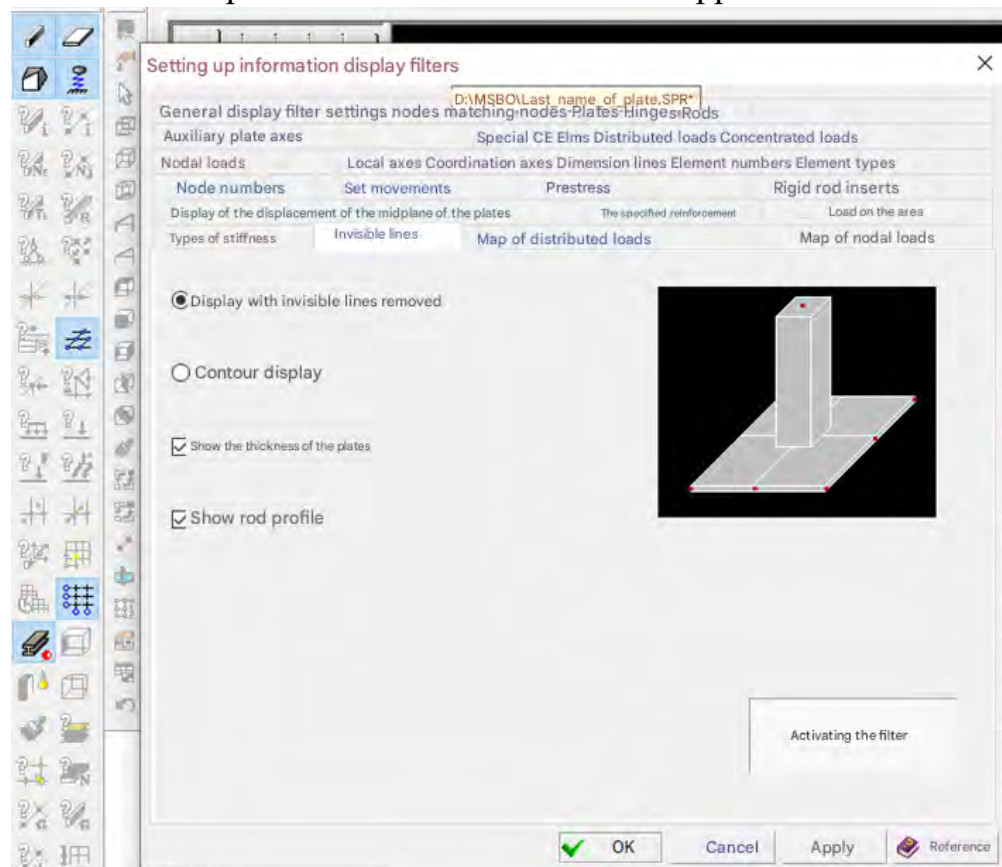


Figure 2.16 – Activating the plate thickness display filter

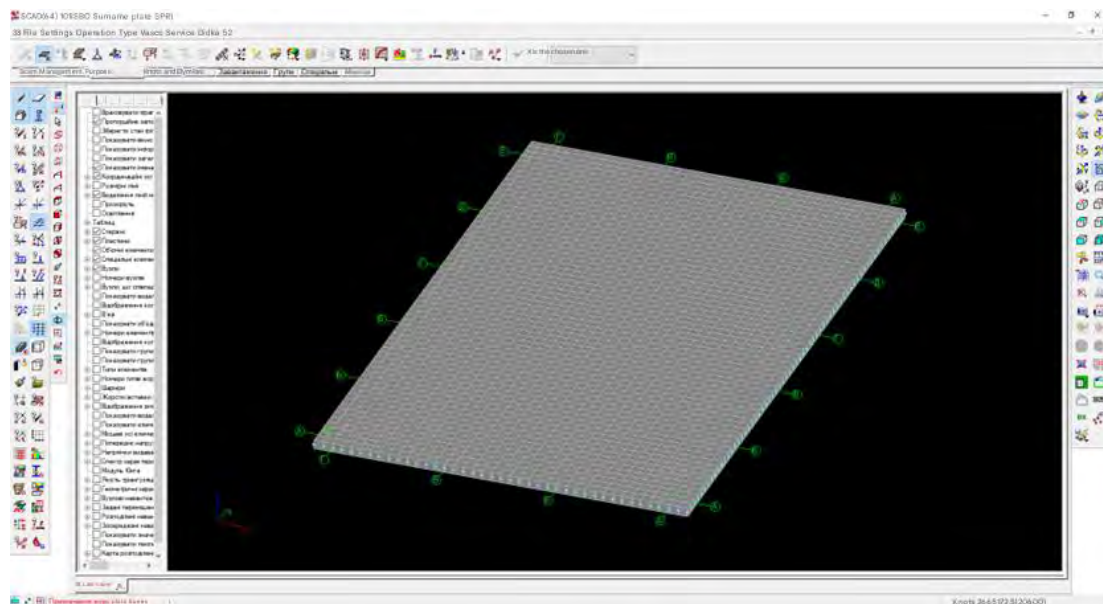


Figure 2.17 – Display of the created foundation slab with the activated filter for displaying the thickness of the plates

For the created foundation slab, specify the reinforcement using the tool "Setting the Reinforcement Scheme..." on the "Assignment" tab.

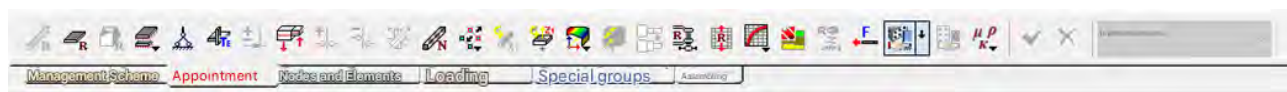


Figure 2.18 – Tool "Setting the reinforcement scheme..." on the "Assignments" tab

In the new dialog box, assign working reinforcement $\varnothing 12$ in increments of $200 \text{ mm} \times 200 \text{ mm}$, and transverse reinforcement $\varnothing 10$ in increments of $600 \text{ mm} \times 600 \text{ mm}$, assign a name to the created reinforcement and click OK.

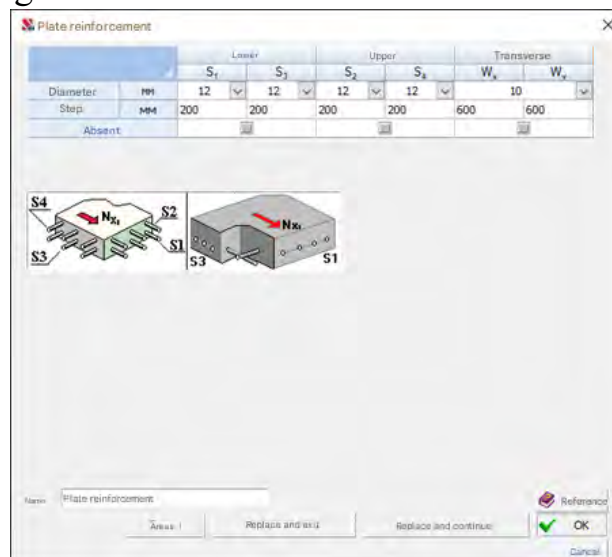


Figure 2.19 – Purpose of plate reinforcement

Select all finite elements of the created slab and press the confirm button to assign the specified reinforcement.

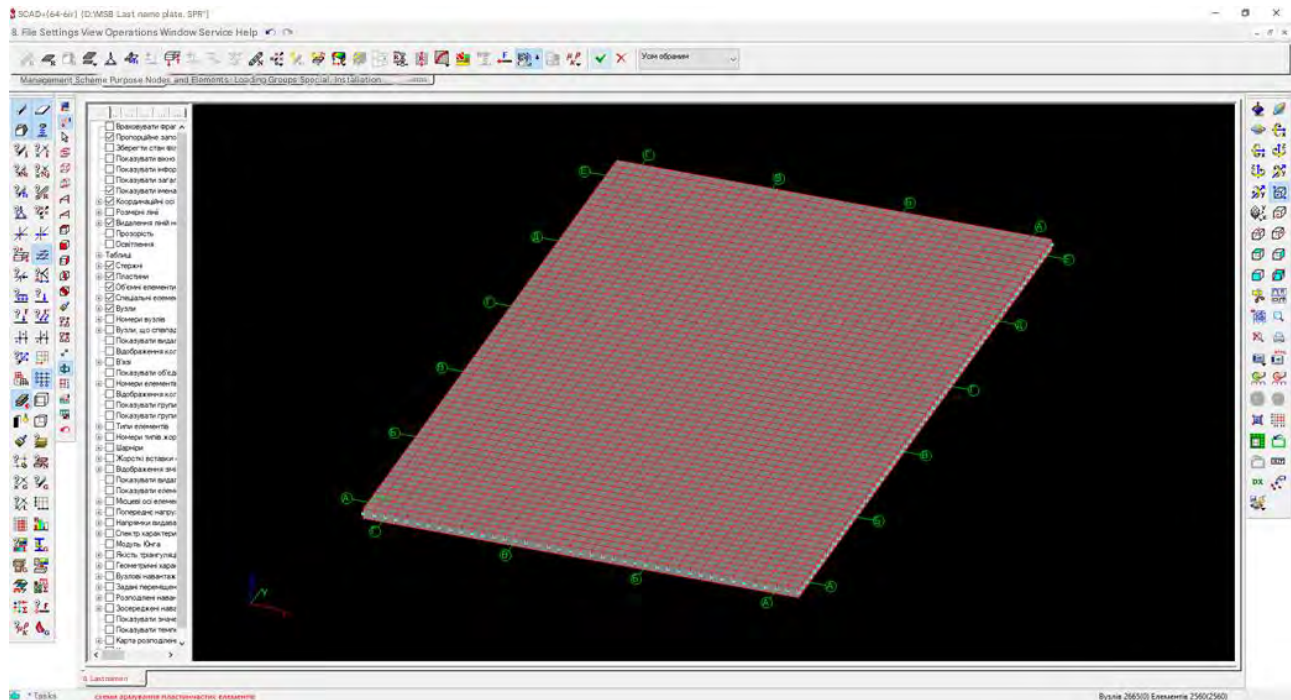
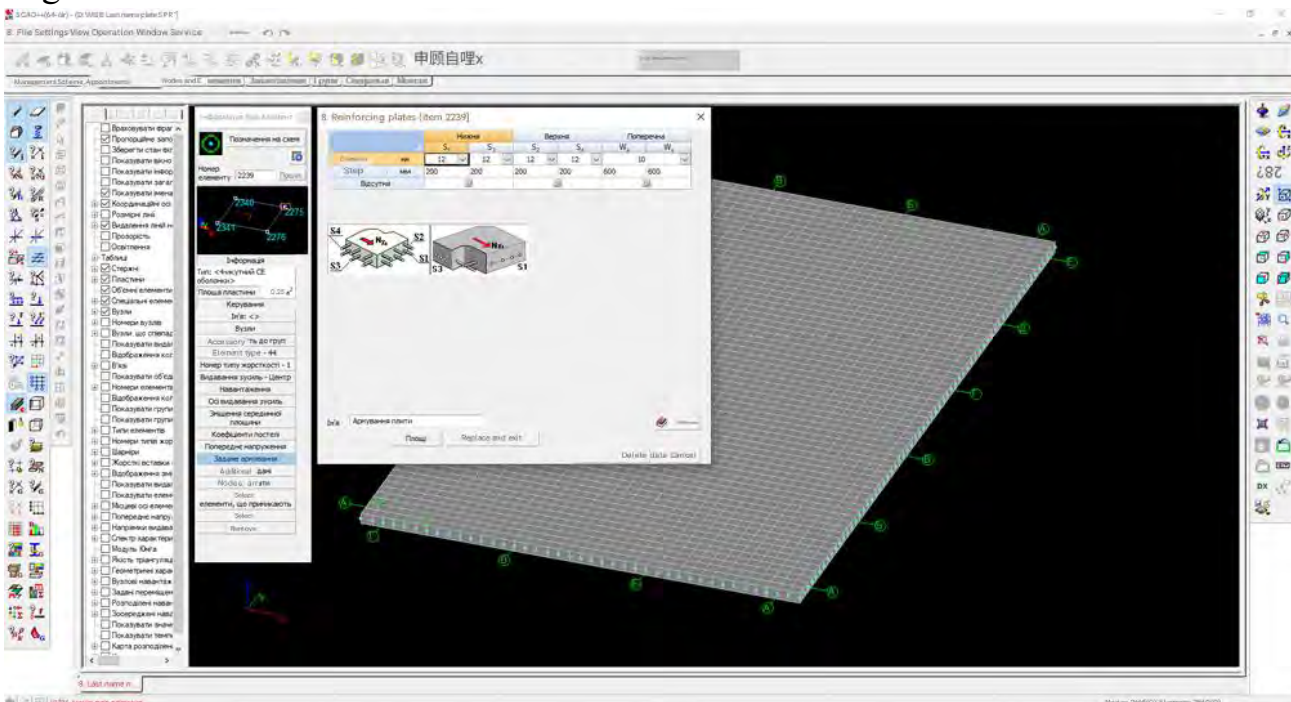


Figure 2.20 – Application of the specified reinforcement to the foundation slab model

Use the Element Information tool in the filter panel to control reinforcement assignment and stiffness.



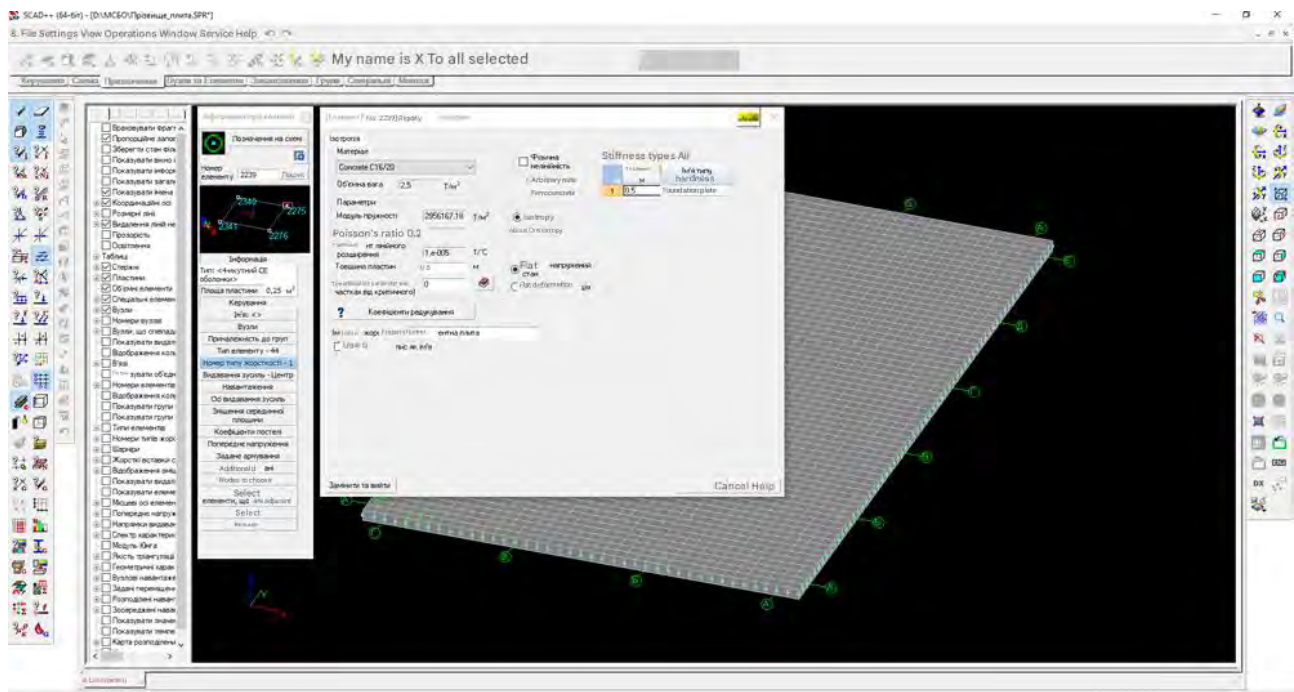


Figure 2.22 – Information about the stiffness of the selected finite element

After setting the material parameters and reinforcing the created foundation slab, create a load of its own weight. To do this, go to the "Downloads" tab and click on the "Own Weight" button.

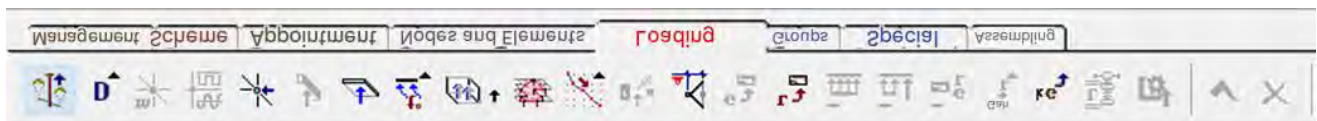


Figure 2.23 – "Dead weight" button on the "Downloads" tab

In the "Dead weight" dialog box, set the load reliability factor from the dead weight of reinforced concrete structures manufactured on the construction site in accordance with DBN B.1.2-2:2006 Loads and impacts.



Figure 2.24 – Dead weight dialog box

Make sure that the load is applied from your own weight by activating the appropriate options in the filter panel  .

In the workspace of the application, the corresponding display of distributed loads and their values will be displayed.

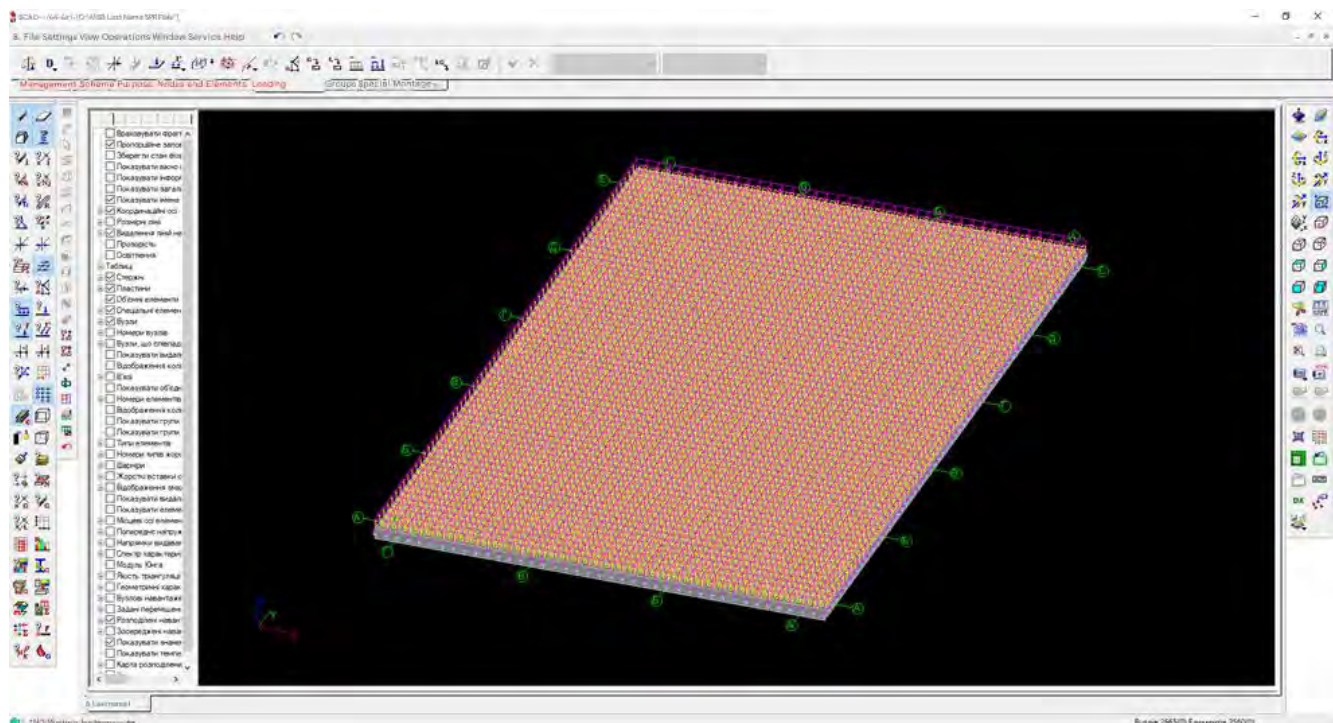


Figure 2.25 – Display of the dead weight of the foundation slab

Save the created download by clicking on the  "Save Download" button.

In the Save Loads dialog box, type a name for the load "Slab Dead Weight" and select the type and type of load. After that, click the "Save as New" button then OK.

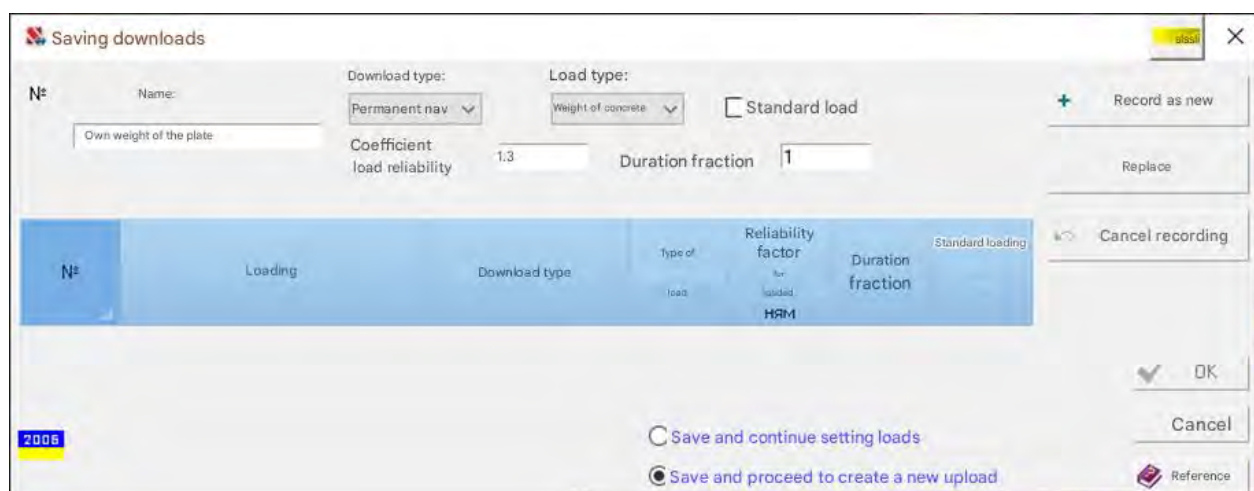


Figure 2.26 – Save Downloads dialog box

The next step is to assign to the plate the previous coefficients of the elastic base, obtained from the results of the previous task in the REQUEST program. The assignment of the previous coefficients is necessary for the first iteration of the calculation.

Go to the "Assignment" tab and click the button  to assign elastic base coefficients.

In the Assign Elastic Base Plates dialog box, enter the values for the first iteration that are obtained in the REQUEST program.

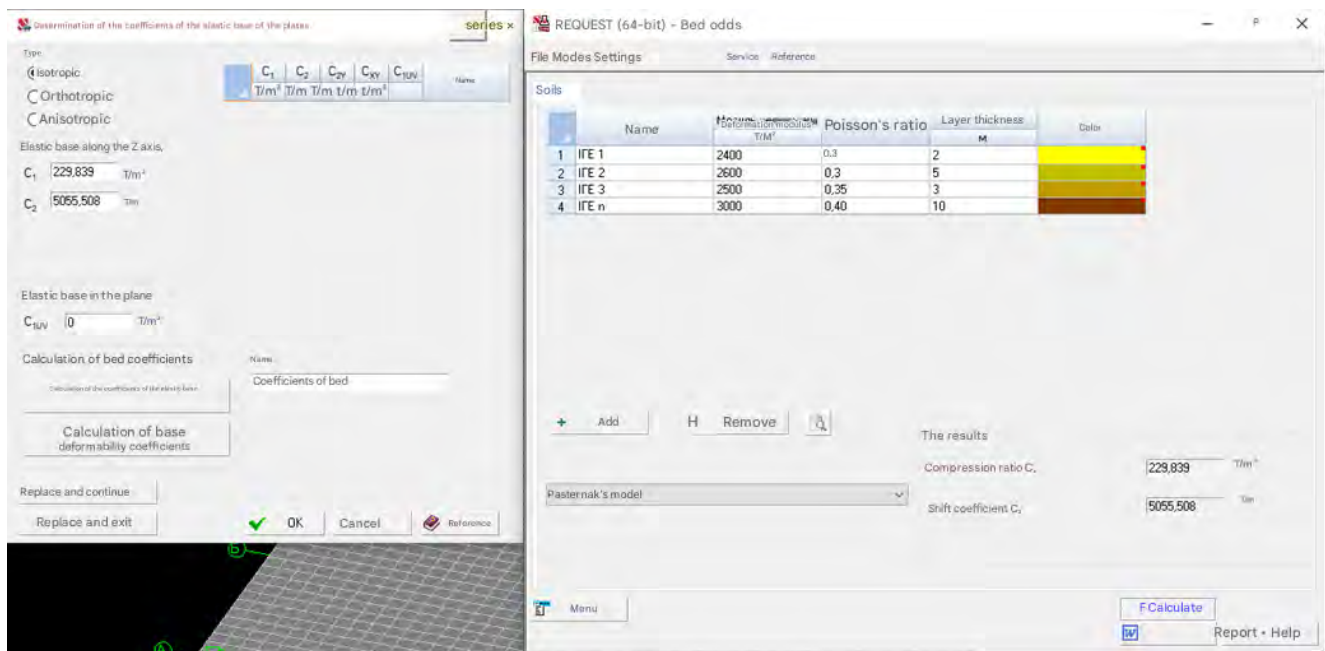
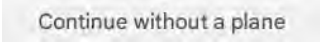


Figure 2.27 – Dialog box "Assignment of elastic base coefficients of plates" and results of calculation of previous bedding coefficients in the REQUEST program

Click OK in the dialog box and select all the finite elements of the foundation slab, and then click the confirm button.

Next, export the resulting finite-element model to the CROSS subroutine of iterative calculation of bed coefficients. Click the  "Calculation of elastic base coefficients" button on the "Purpose" tab and select all finite elements of the foundation slab. Click the confirmation button.

In the dialog box, click the button . When activated, the CROSS subprogram will once again ask about the availability of a prepared site, again confirm its absence and proceed to its creation in a new window of the CROSS program.

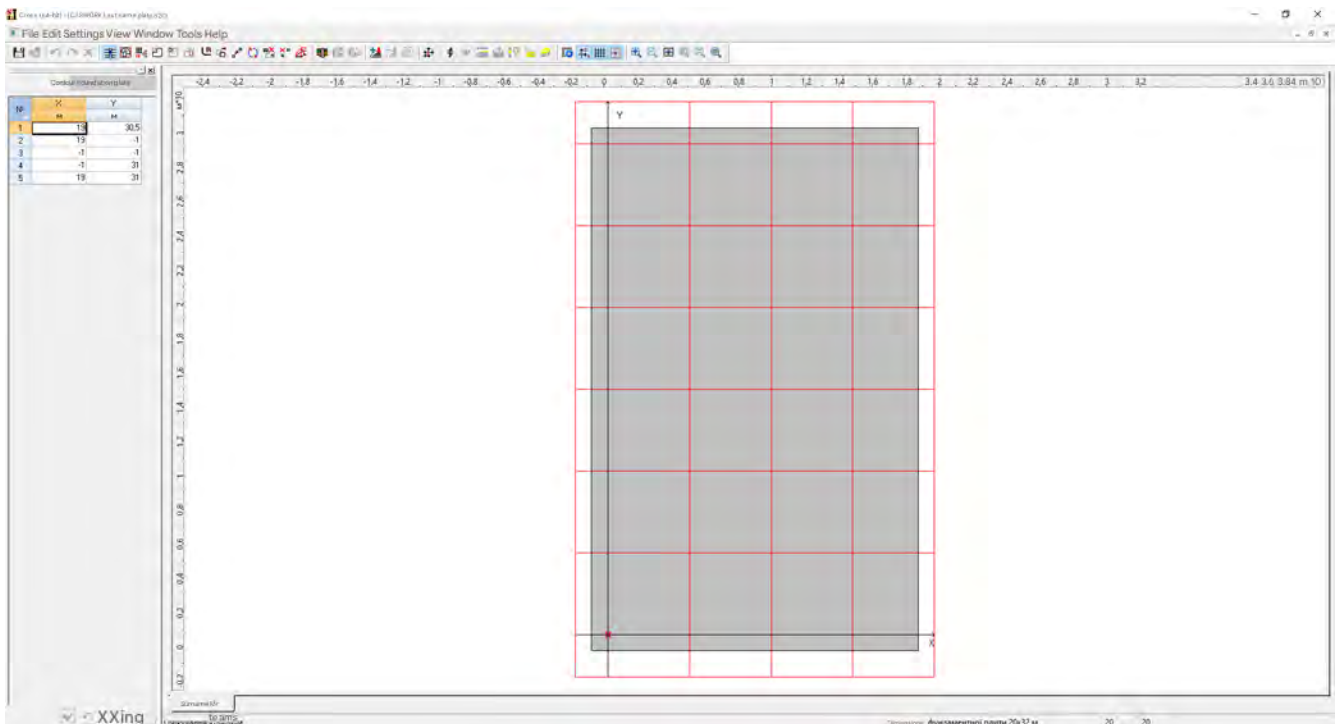


Figure 2.28 – Workspace of the CROSS program with imported foundation slab from the SCAD++ program

Adjust the grid settings by setting the step to 0.5 m using the appropriate button  and the program dialog box.

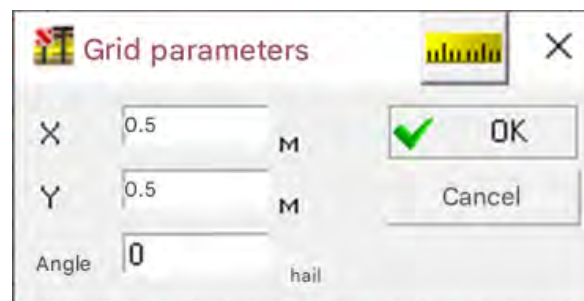


Figure 2.29 – Grid Options dialog box of CROSS

Next, you need to set the soil conditions according to your task. Using the  "Add Wells" tool on the toolbar of the KROOS program, add two wells at the diagonal corners of the building.



Figure 2.30 – Toolbar of the CROSS program

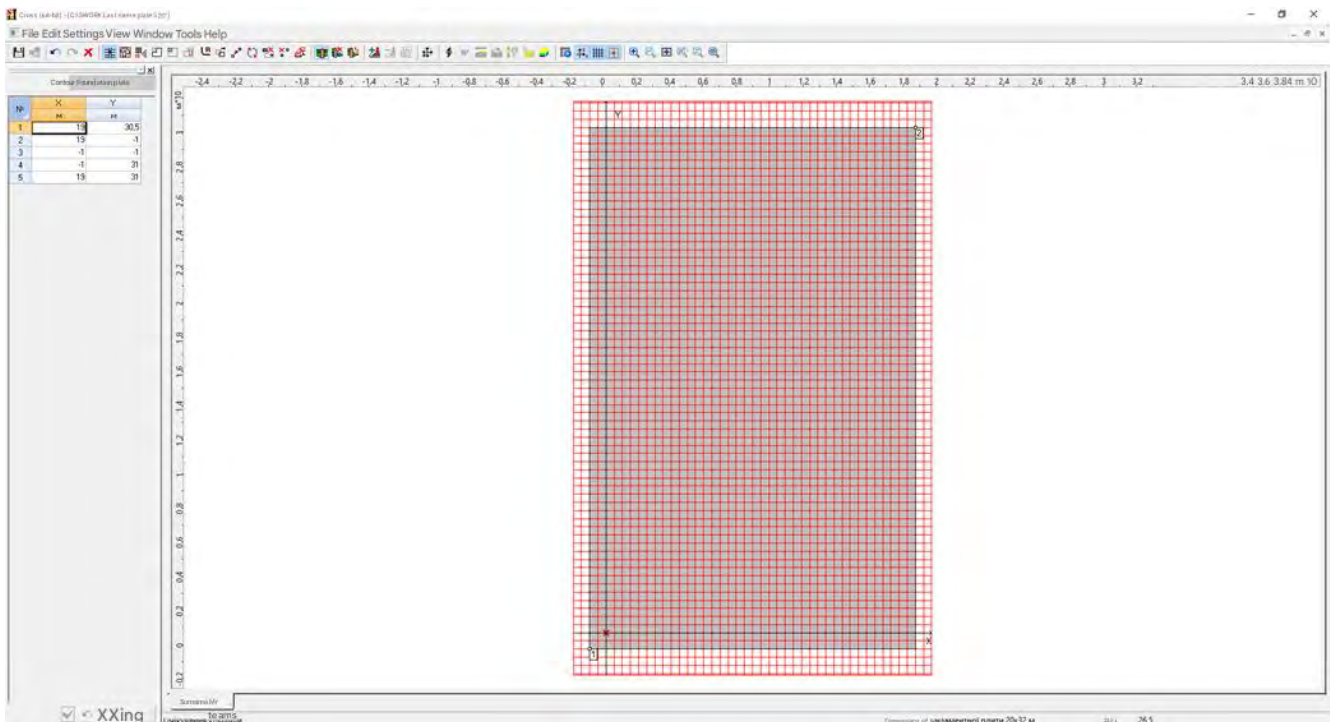


Figure 2.31 – Placement of wells on site

Next, click  the "Well Parameters" button to create soil conditions in accordance with your task.

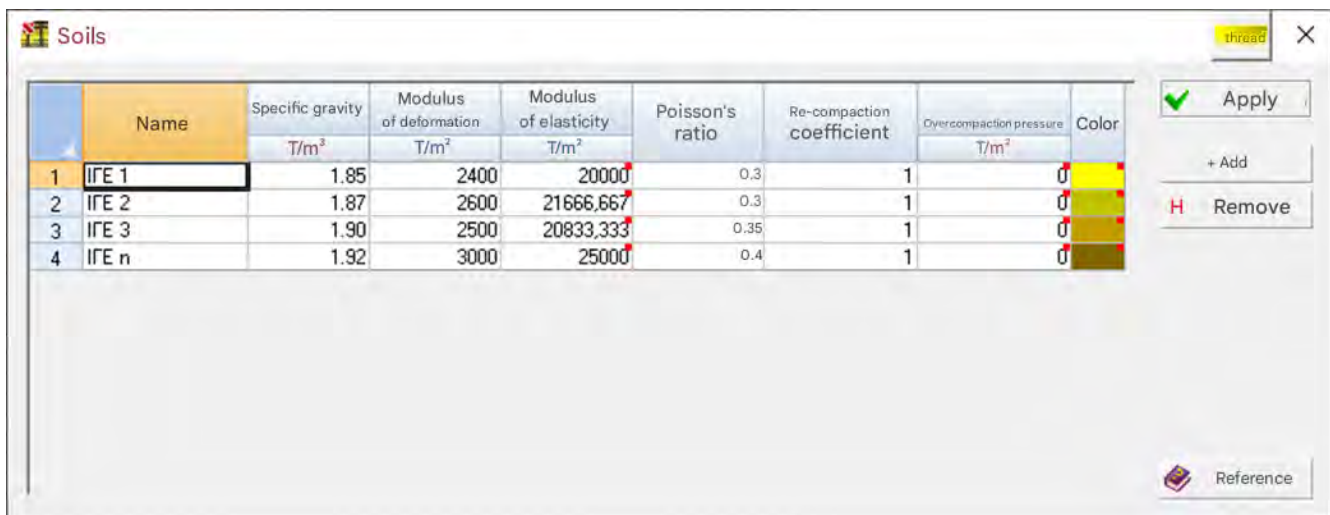


Figure 2.32 – Entering information about soil conditions on the soil tab

After entering all the necessary data, click the "Apply" button and return to the "Well Parameters" dialog box. In the "Well Parameters" dialog box, transfer the created engineering and geological elements to the geological column and specify the top marks

of each IGE in accordance with the task. Make sure that the geological column is created correctly by clicking  the "Preview" button to visualize the geological column.

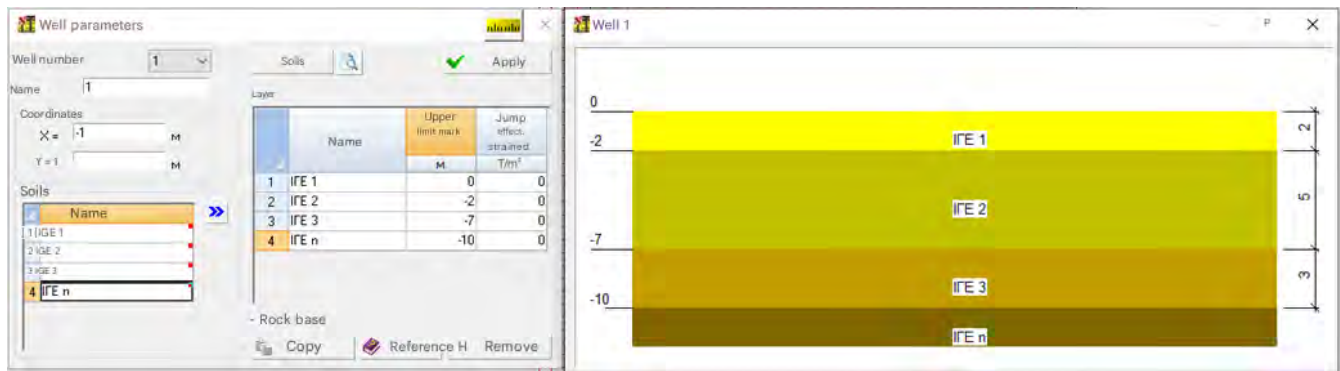


Figure 2.33 – Creation of a geological column for well 1

From the drop-down list, select well No. 2 and repeat the same operation to create a geological column. After that, click the "Apply" button.

Click the "Volumetric display" button to control the correctness of the construction of the geological structure of the site.

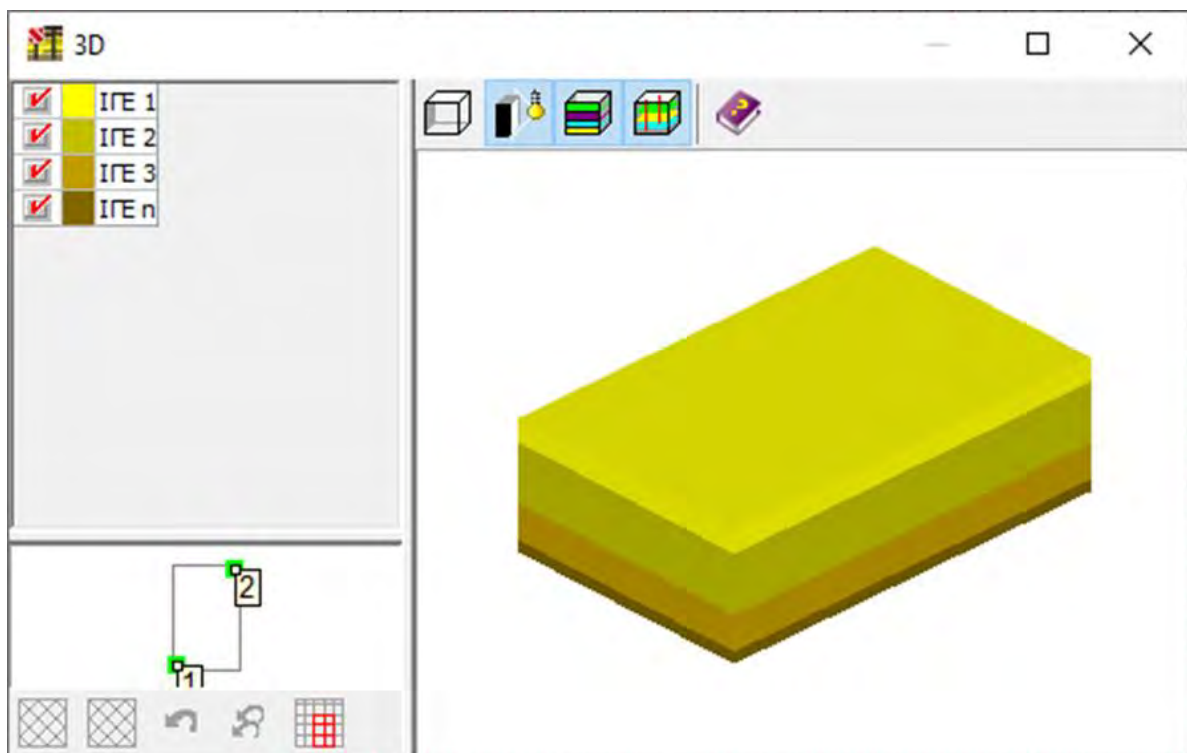


Figure 2.34 – Volumetric display of the geological structure of the site

Set the mark of the slab sole and the load for the first iteration (which is equal to the load from the own weight of the foundation slab) using the menu called by the button  , confirm with the OK button.



Figure 2.35 – Load dialog box

Move the calculation point of the compressible thickness to the center of the plate using the "Edit" menu and click  the "Calculate" button.

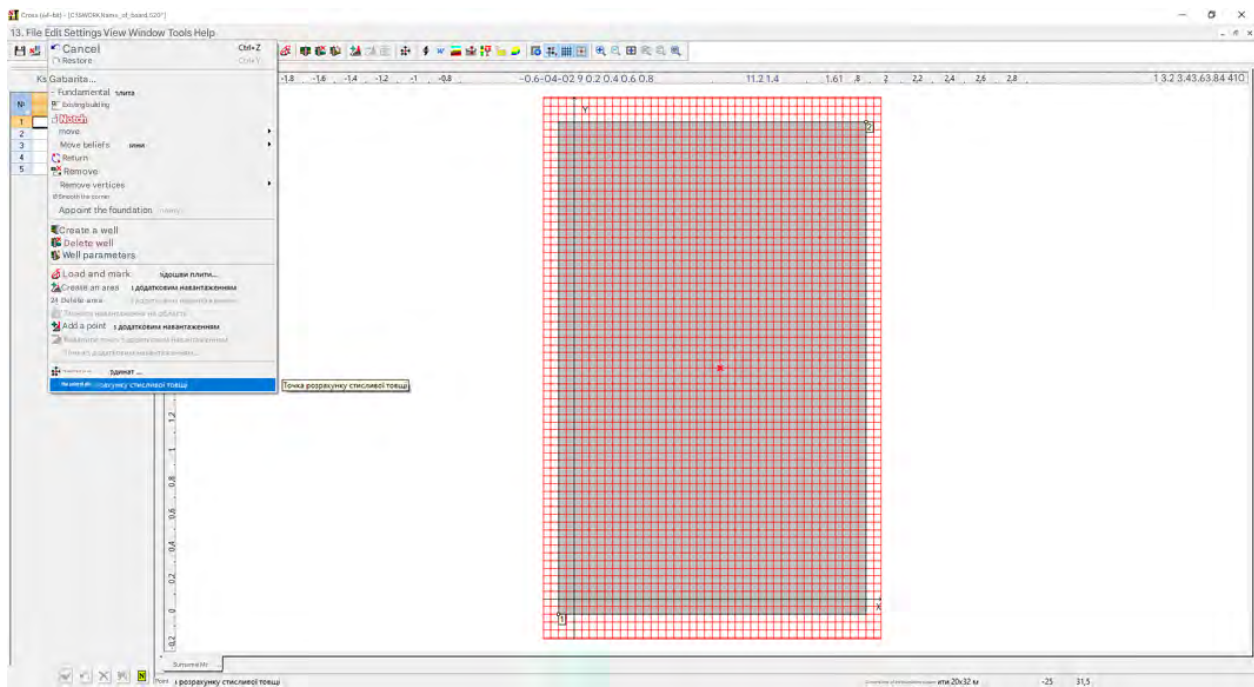
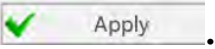


Figure 2.36 – Setting the compressible thickness calculation point

In the window with the calculation results, click the button .

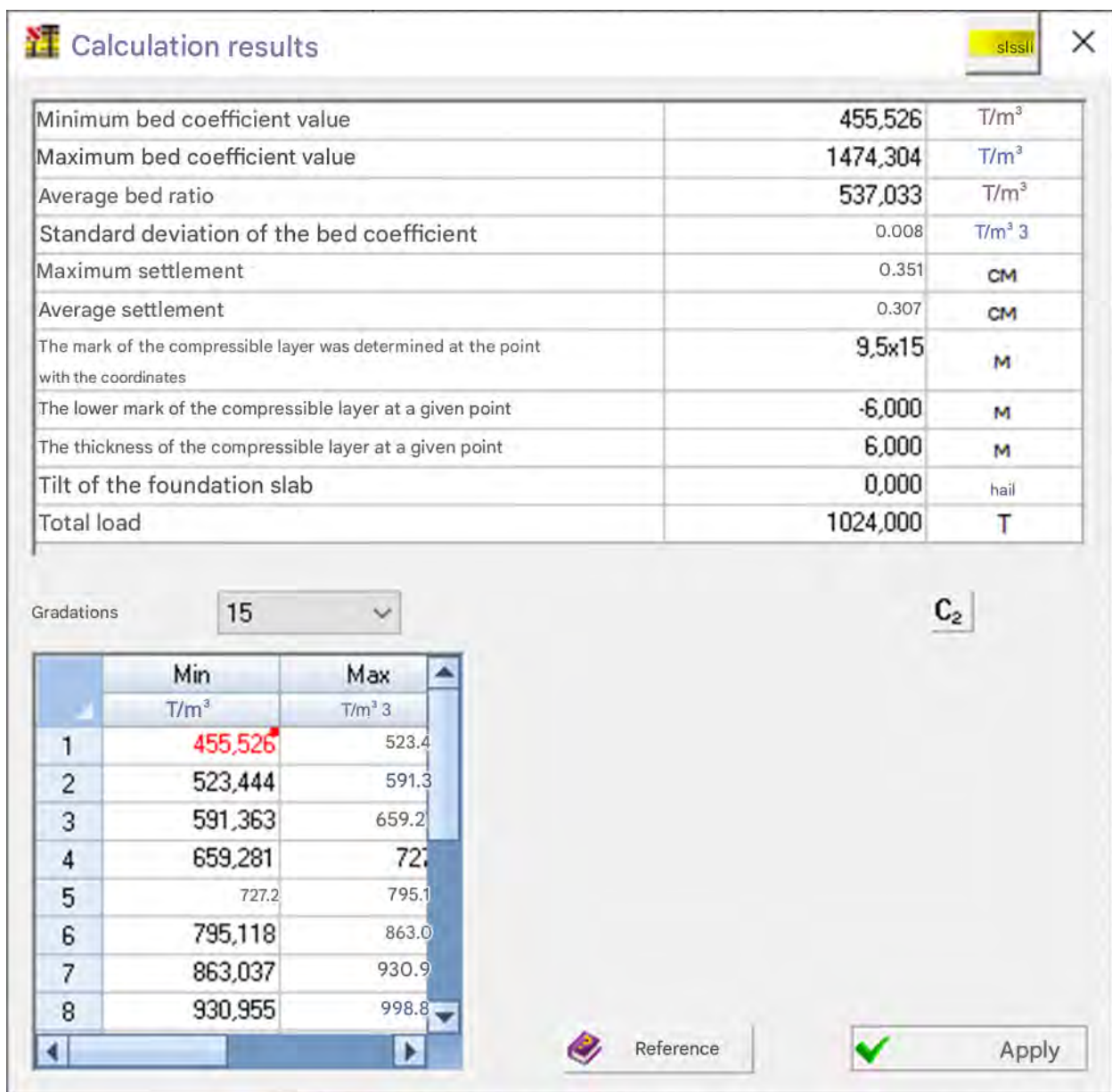


Figure 2.37 – Calculation Results Dialog Box

Click  the "Save data for SCAD" button and close the CROSS window. The received data from the CROSS module is transferred to SCAD++, where the dialog box "Assignment of elastic base coefficients" appears. Rearrange the box to "Assign a specified number of coefficients". Set the value of the number of coefficients to 10 and click the "Calculate" button. After that, click the OK button.

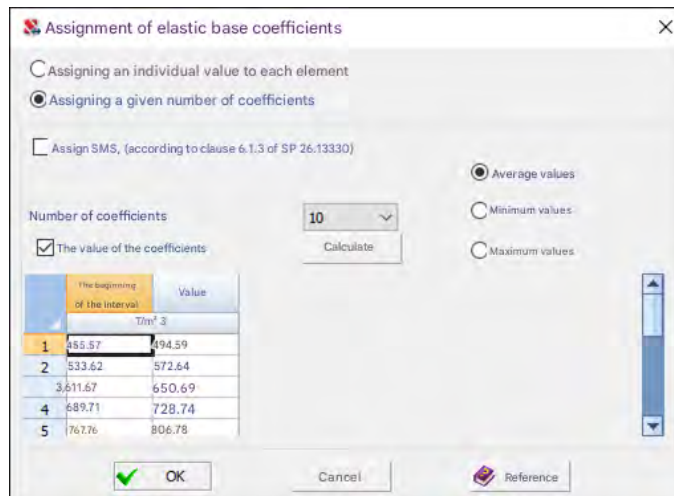


Figure 2.38 – Assignment of elastic base coefficients dialog box

Make sure that the assignment of bed coefficients is correct using the "Display Hardness Types" tool of the filter panel, setting it to display the bed coefficient C1 in the dialog menu by right-clicking on the corresponding button of the filter panel.

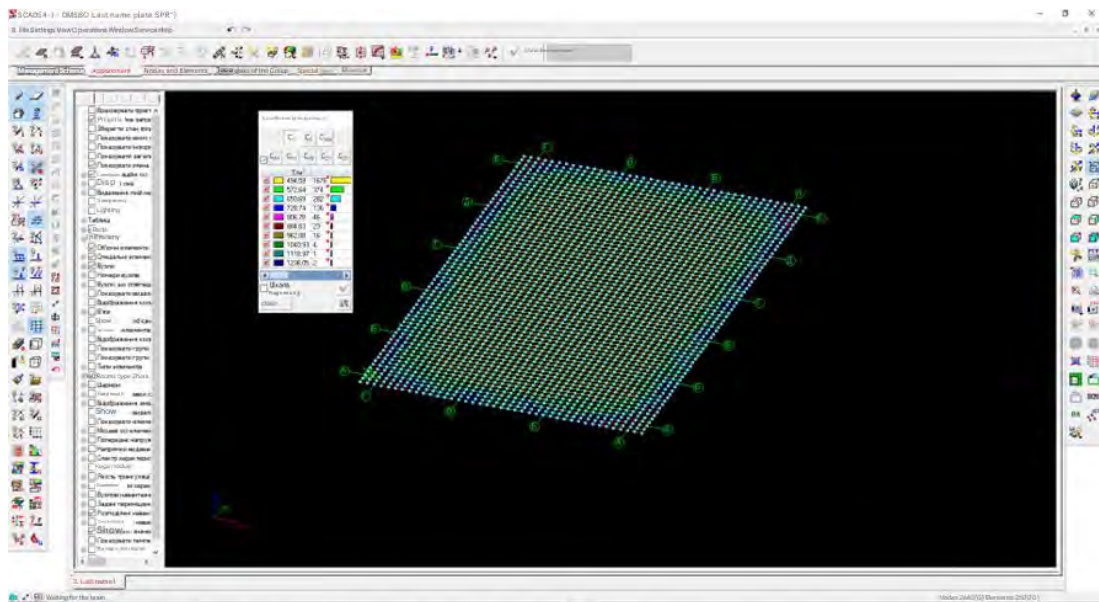


Figure 2.39 – Isopoles displaying the assigned coefficient C1 on the foundation slab

We save the project. We continue the further iteration of the bedding coefficients after modeling the above-ground part of the building and its assembly with the foundation slab.

Tasks for independent work on topic 2. Calculate the coefficients of bedding in the "REQUEST" program by applying identical parameters of soil conditions. Explain the results obtained.

TOPIC 3

CONSTRUCTION OF THE «SOIL-FOUNDATION-STRUCTURE» MODEL

Creation of a calculation scheme of the above-ground part of the building

To create a new project, left-click on the corresponding button  in the File menu or on the toolbar. The "New Project" dialog box will appear on the screen. Enter your last name in the field Name, control the selection of design standards and click OK.

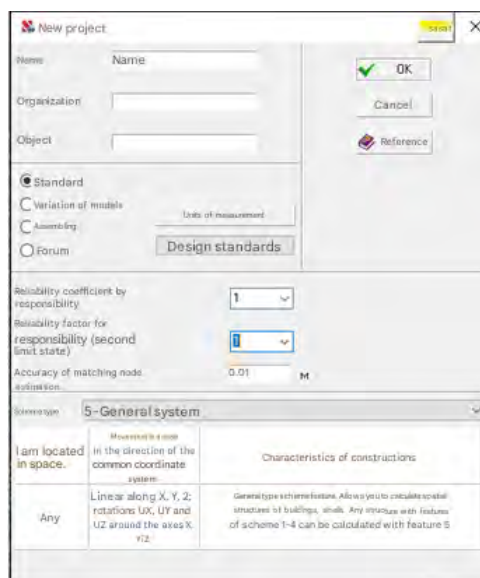


Figure 3.1 – New Project dialog box

In the new window, select the place to save the project and its name in the "Project_Surname" format.

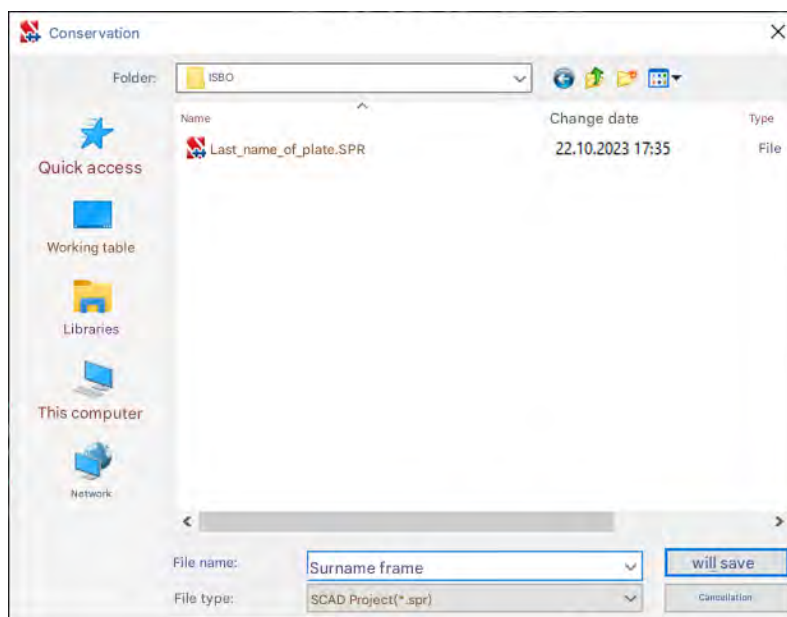


Figure 3.2 – Save dialog box

Next, we create a design scheme of the frame of the above-ground part of the building in accordance with the individual interference.

In the project tree, go to the calculation diagram tab.

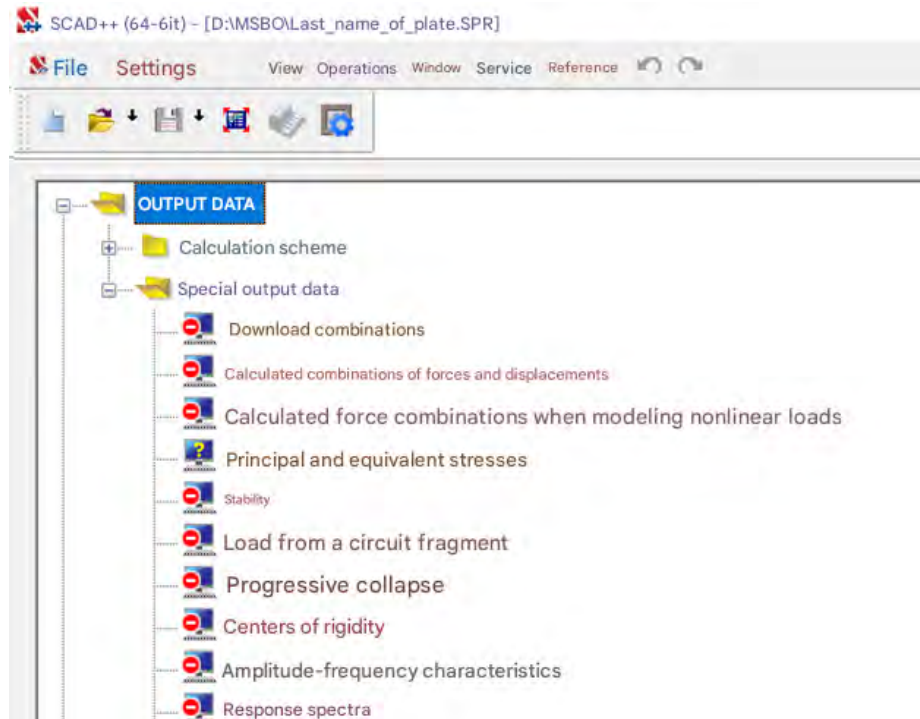


Figure 3.3 – Project tree dialog box

In the main working window of the program, create a grid of stakeout axes according to the individual task (the grid of stakeout axes should be identical to the grid of axes created when modeling a slab according to your option). To do this, click the "Set Layout Axis Grid" button on the "Diagram" tab.



Figure 3.4 – Invoking the Set Partition Grid dialog box

In the Set Split Grid dialog box, enter the pitch of the axes in the longitudinal and transverse direction according to your individual task and click OK.

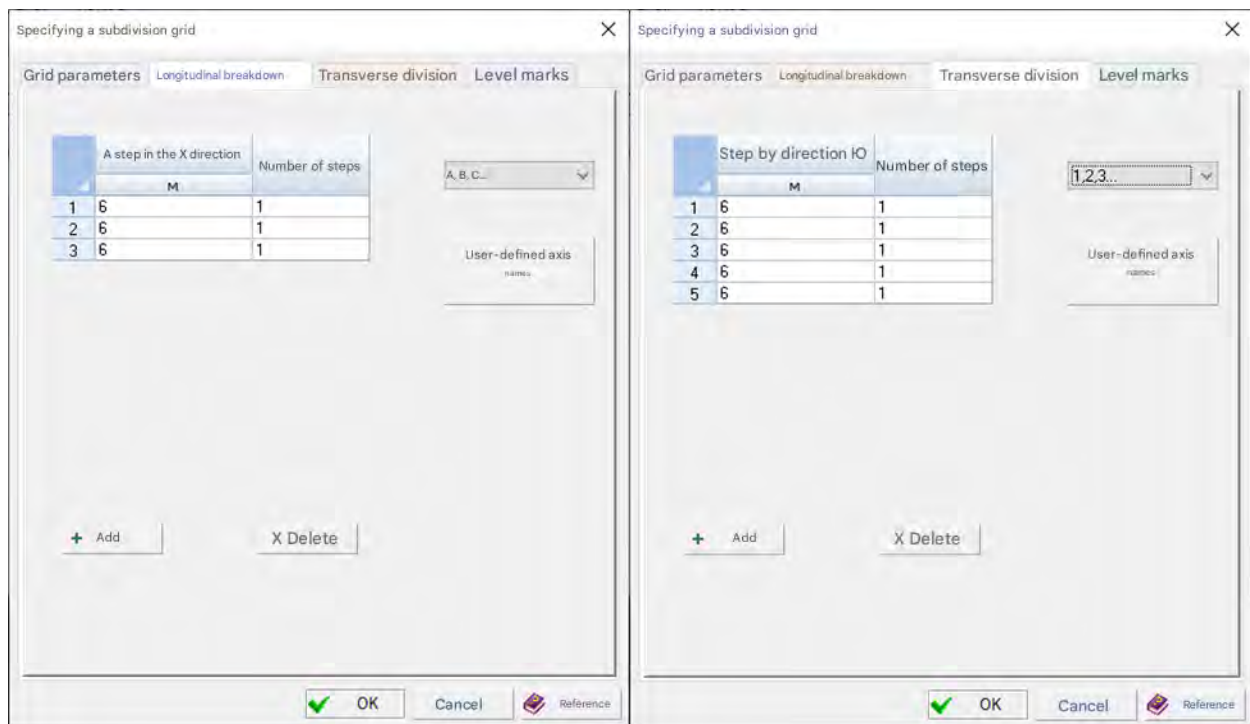


Figure 3.5 – Set Partition Grid Dialog Box

To display the created split grid, use the appropriate tool in the filter panel.

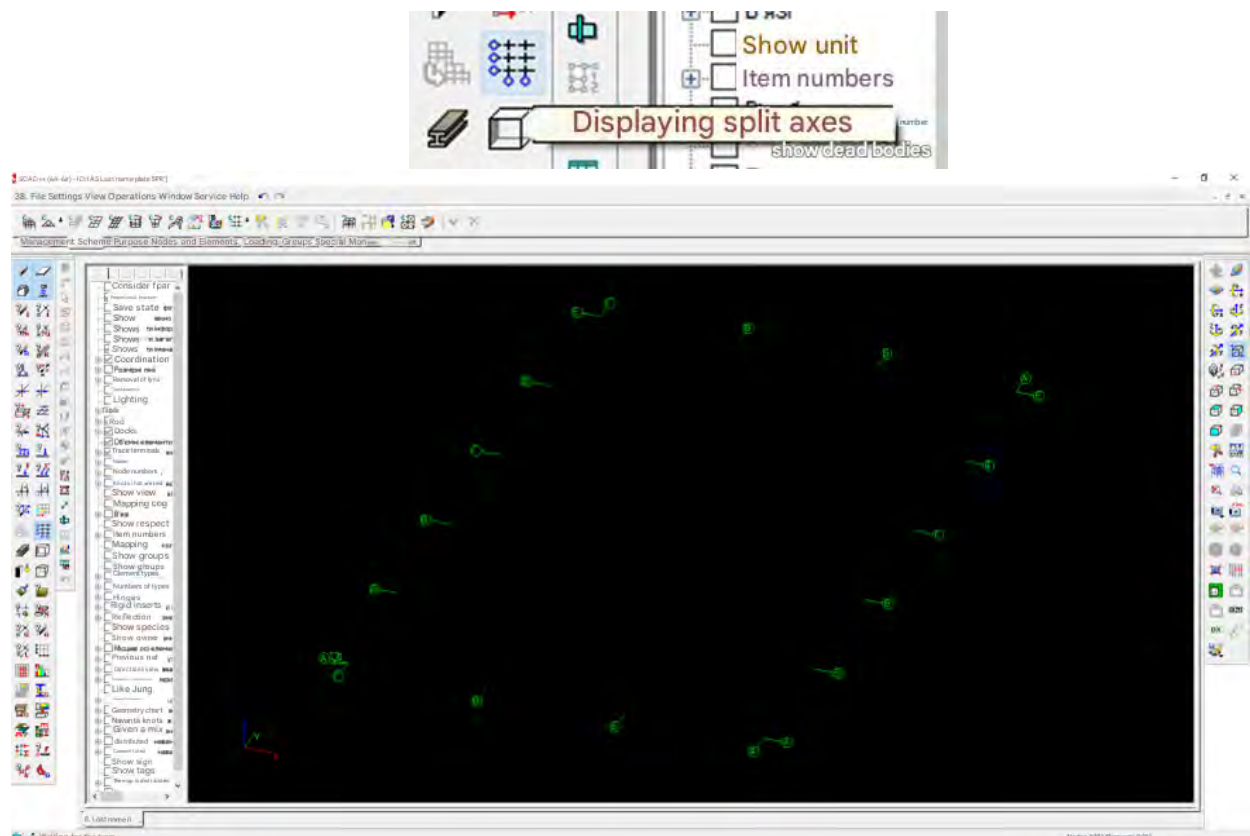


Figure 3.6 – Displaying the Layout Grid in the Application Work Field

The design scheme is a spatial metal frame, which is designed according to a frame-elm scheme to ensure spatial rigidity and stability.

First, enter the coordinates of the points of the nodes of the transverse frame along axis 1. Click the "Node Input" button on the "Nodes and Elements" tab. The coordinates of the nodes are determined in accordance with the individual task.

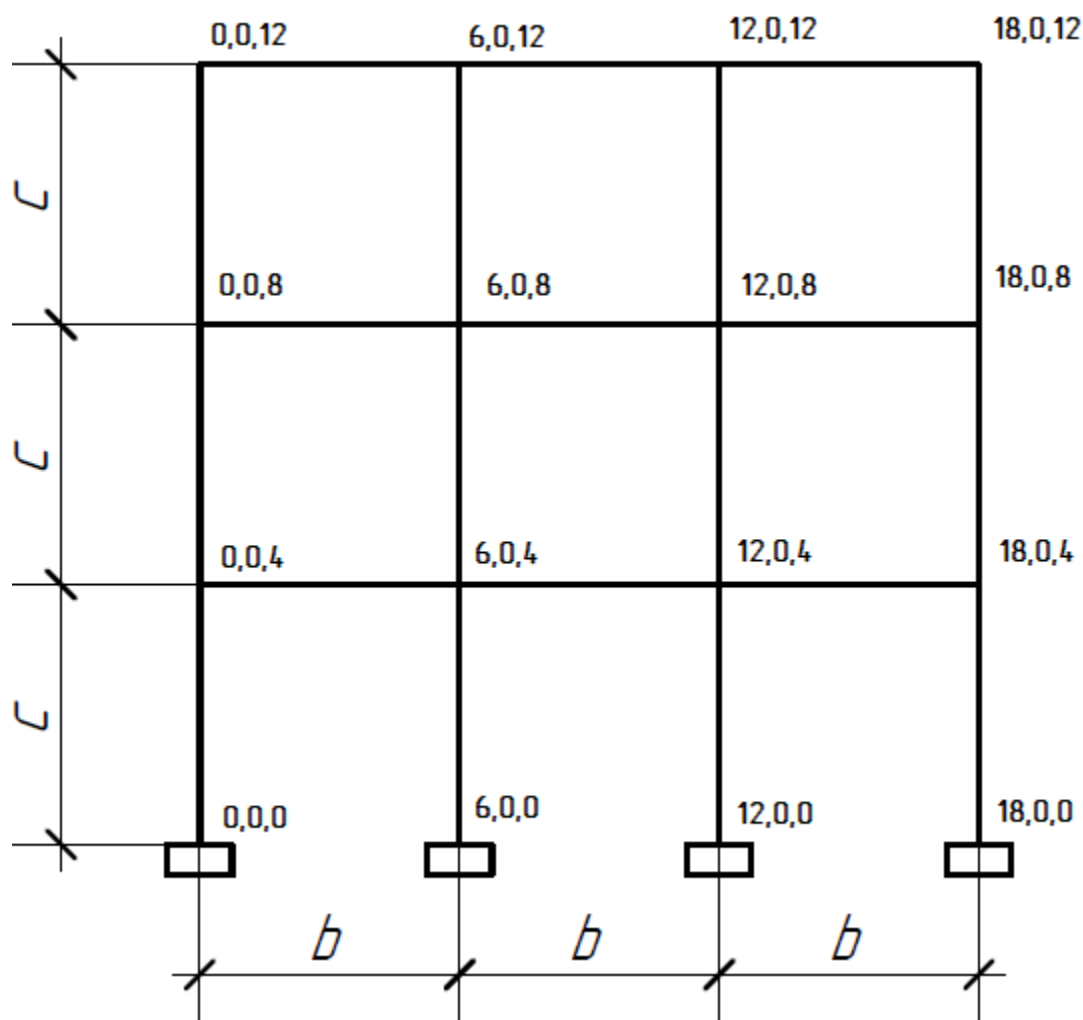


Figure 3.7 – Determination of the coordinates of the frame nodes along axis 1



Figure 3.8 – "Enter nodes" button on the "Nodes and elements" tab

Introduction of nodes

X M

Y M

Z M

☐ Repeat

ΔX M

ΔY M

ΔZ M

N

☐ Automatic transfer of the origin to the last entered node

☐ Changing the input direction

☒ Around the X axis Angle of rotation

☐ Around the Y axis half

☐ Around the Z axis

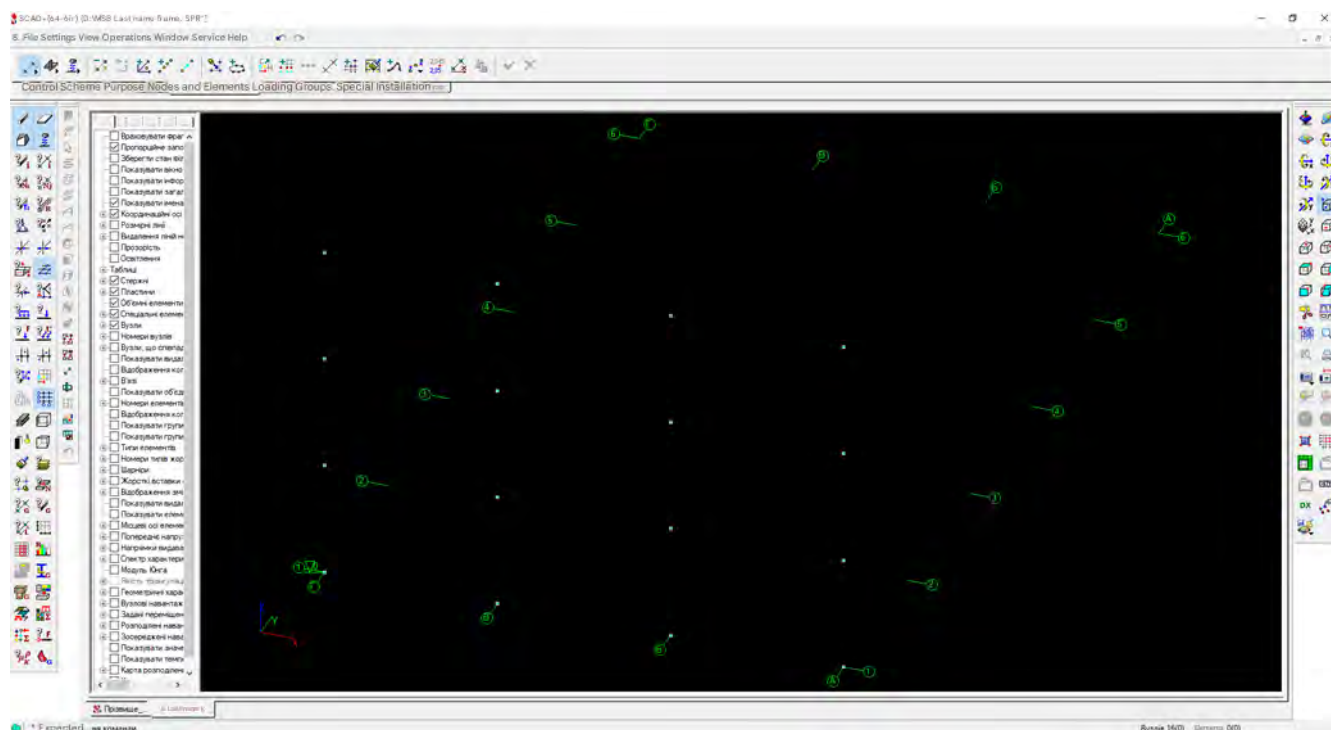
? Elms

+ Add

close

Reference

Check the correct input of nodes in the working field of the program.



38

Let's connect the created nodes with rods that model crossbars and frame columns using the Add Bars tool on the "Nodes and Elements" tab.



Figure 3.11 – "Add Rods" button on the "Nodes and Elements" tab

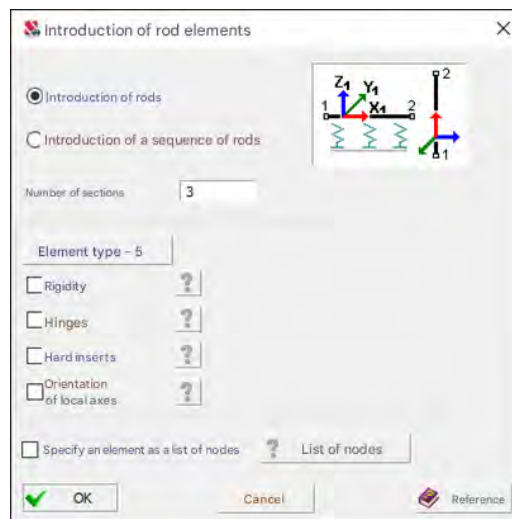


Figure 3.12 – Input of Core Elements dialog box

In the "Input of Core Elements" dialog box, click the OK button and in the working field of the program, connect the corresponding nodes with the rods, observing the rule of combining "bottom-up" and "left-right"

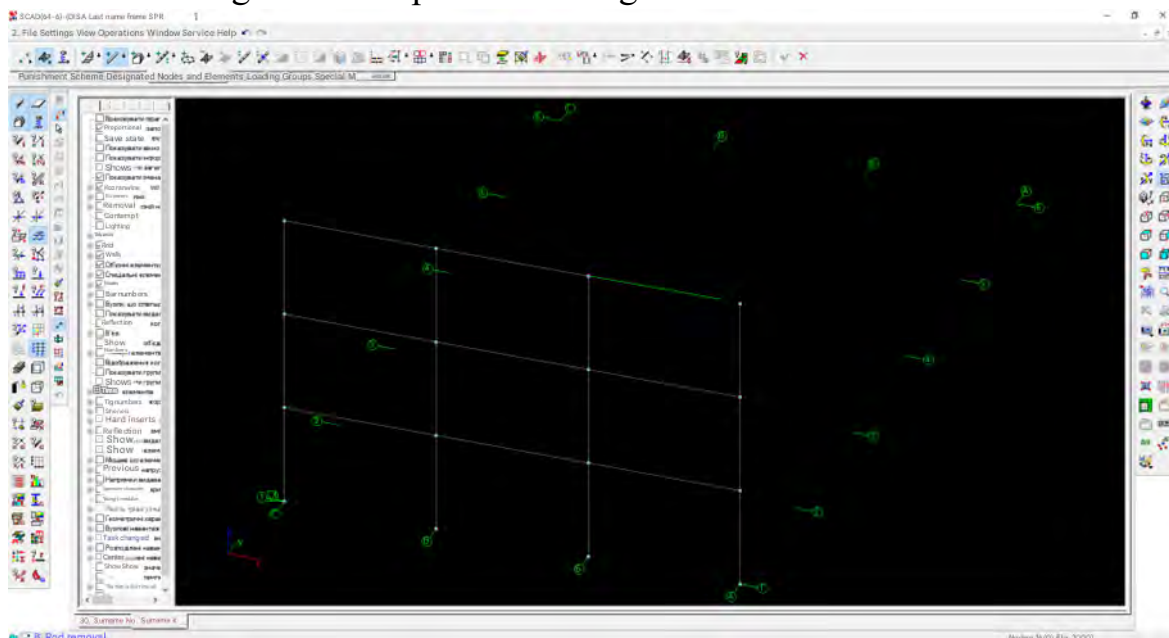


Figure 3.13 – Connection of frame assemblies with rods

After creating the frame along the 1 axis, we assign the preliminary rigidities to the frame elements that are necessary for the first iteration of the calculation.

Go to the assignment tab and click the "Assign stiffeners to bars" button. In the "Stiffness of bar elements" dialog box, check the box on "rolled metal profiles" and go to the tab of the same name.

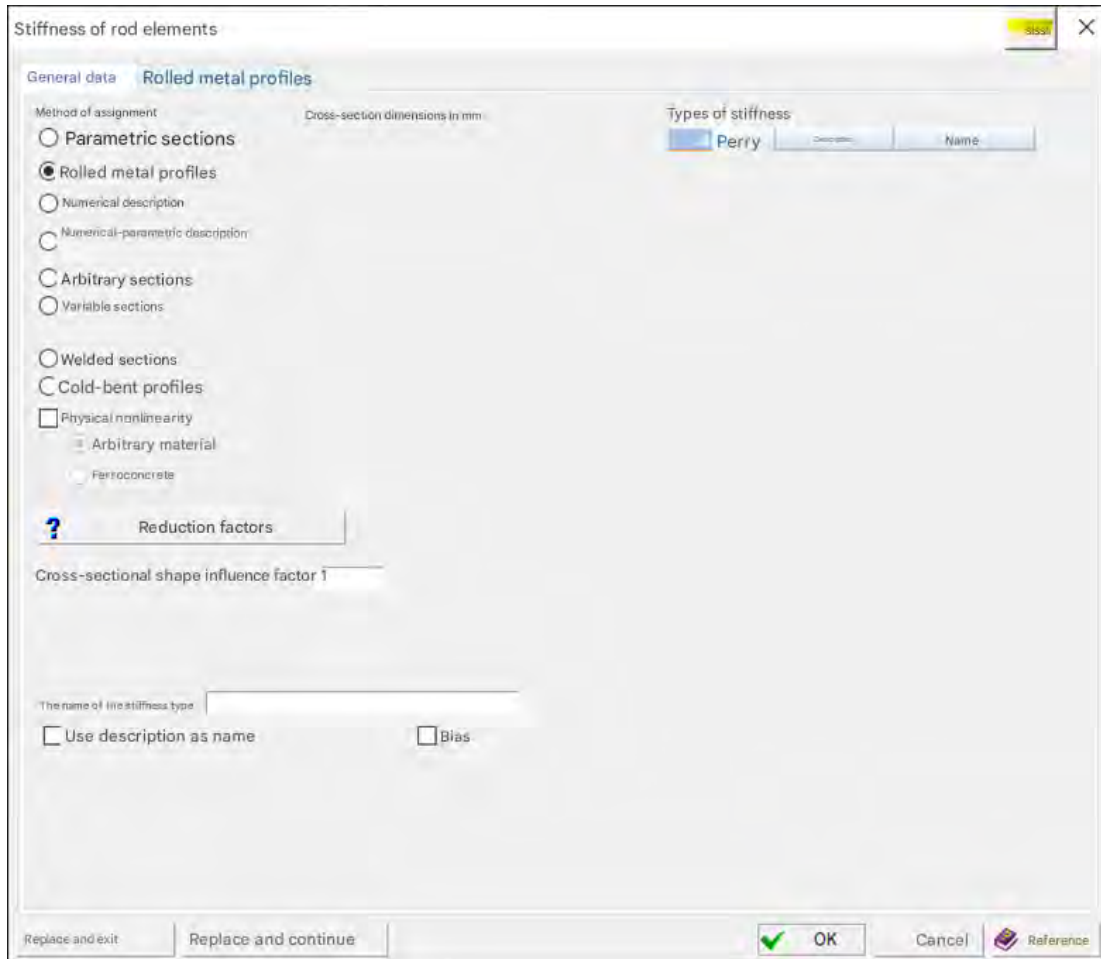


Figure 3.14 – Bar Stiffness Dialog Box

On the "Rolled metal profiles" tab, select the required assortment for which the selection will be carried out. For vertical frame elements that model frame columns, we choose any profile from the assortment of welded I-beams of the series (K) to replace GOST 26020-83. For horizontal frame elements that simulate frame crossbars, we choose hot-rolled steel I-beam according to DSTU 8768:2018. After selecting the appropriate item from the assortment, click the OK button and select all the relevant elements on the diagram and press the confirm button. Repeat the operation for the rods that model the columns and for the rods that model the crossbars.

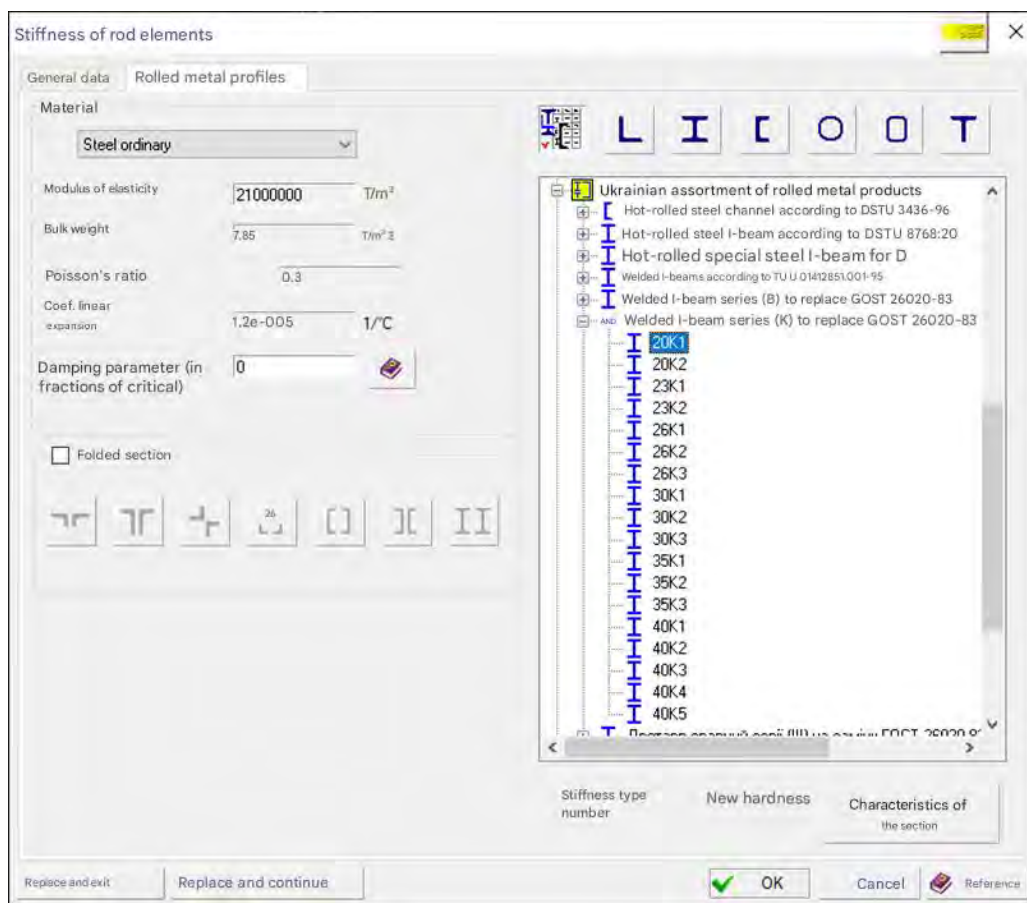


Figure 3.15 – Selection of assortment for the selection of column rigidities

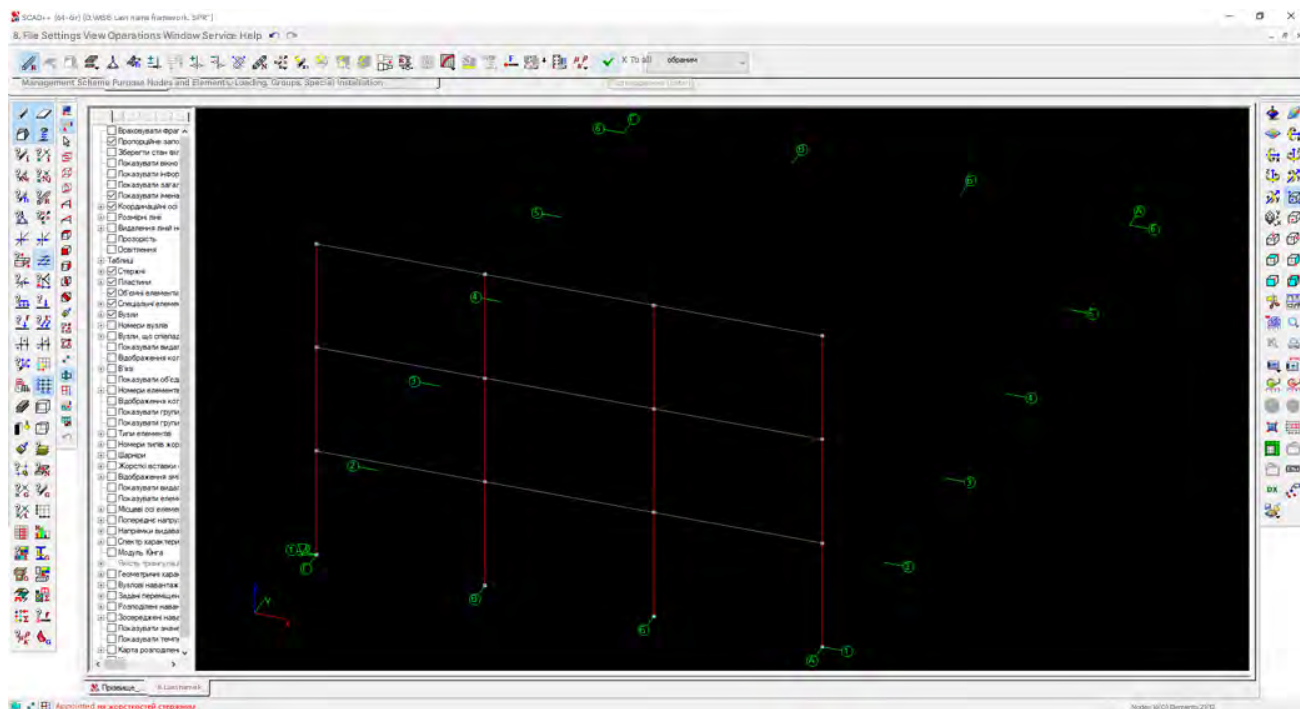


Figure 3.16 – Assignment of the selected stiffness for columns

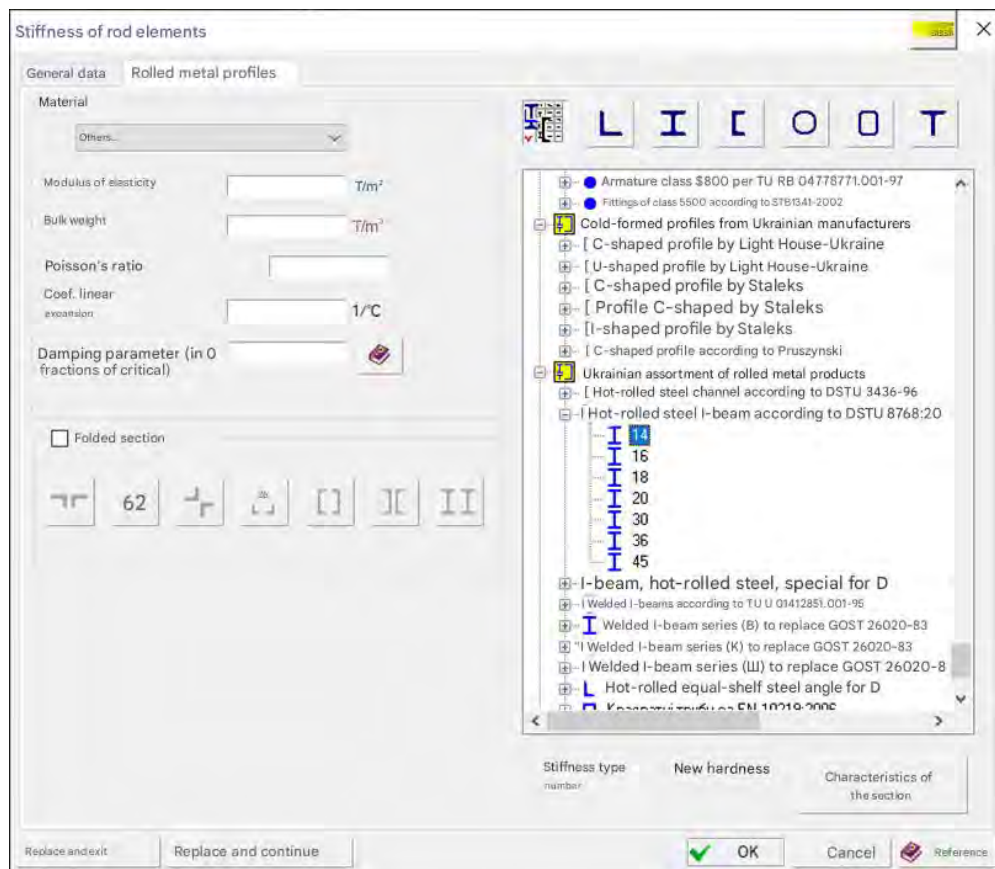


Figure 3.17 – Selection of assortment for the selection of crossbar stiffeners

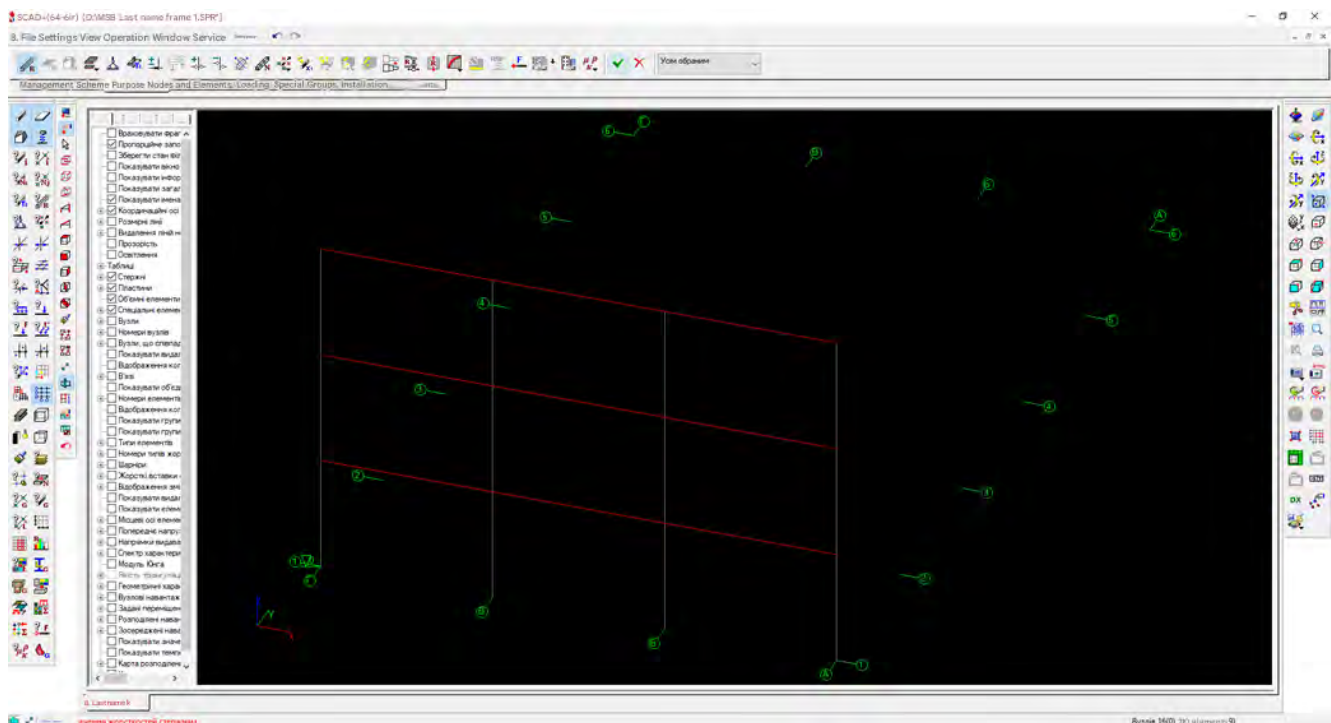


Figure 3.18 – Assignment of the selected stiffness for crossbars

Verify that the stiffeners are assigned correctly by enabling the display of stiffness type numbers and  the display of rod profiles in the filter panel.

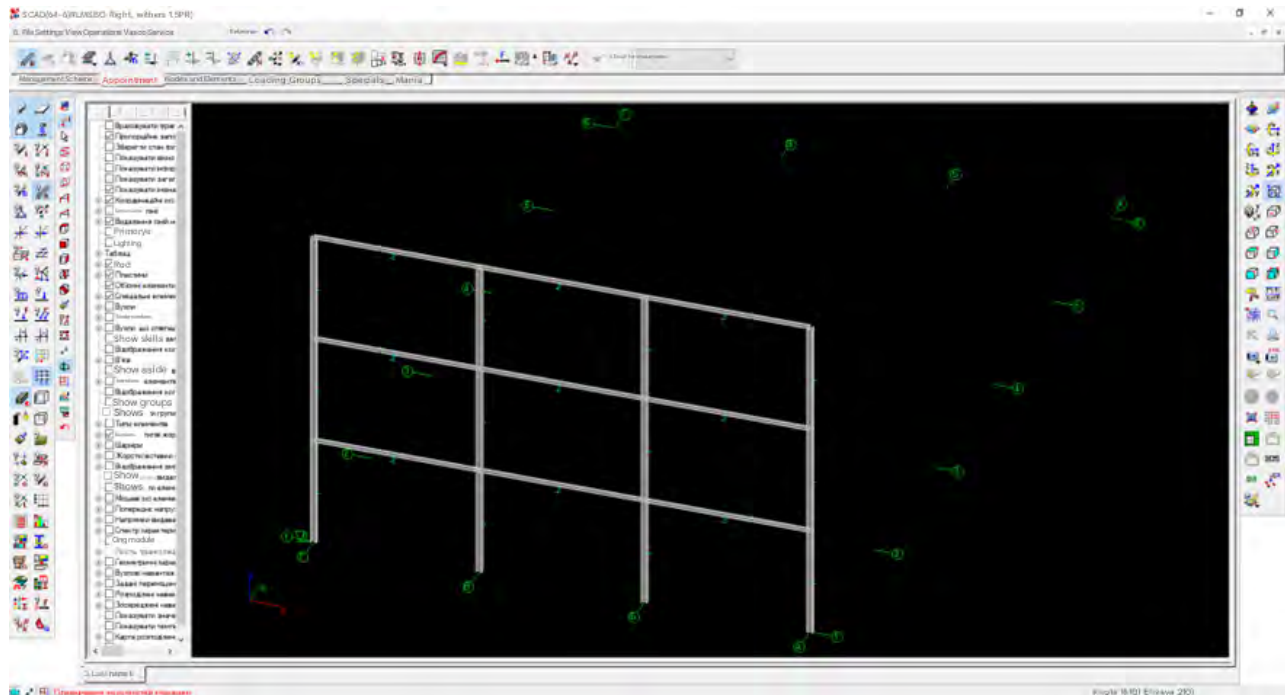


Figure 3.19 – Control of the assignment of stiffeners by enabling the corresponding functions in the filter panel

Next, we assign the preliminary conditions for supporting the created frame using the  tool "Arrangement of elms in knots".



Figure 3.20 – Tool "Arrangement of elms in knots" on the "Purpose" tab

In the dialog box, check all types of elms.

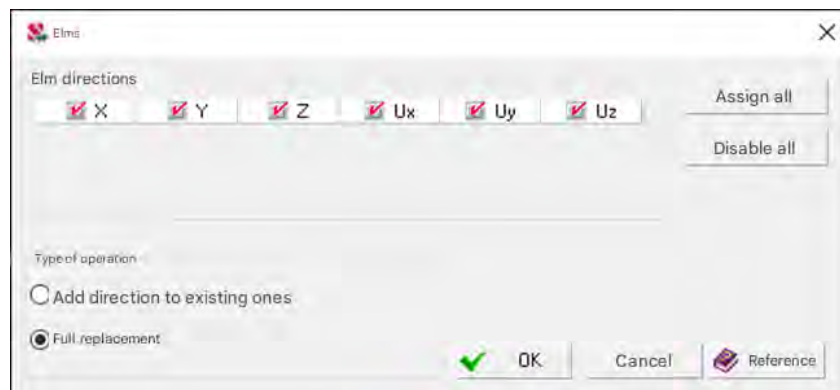


Figure 3.21 – Knitting dialog box

Press the OK button and select all frame nodes at 0.000 and press the confirm button.

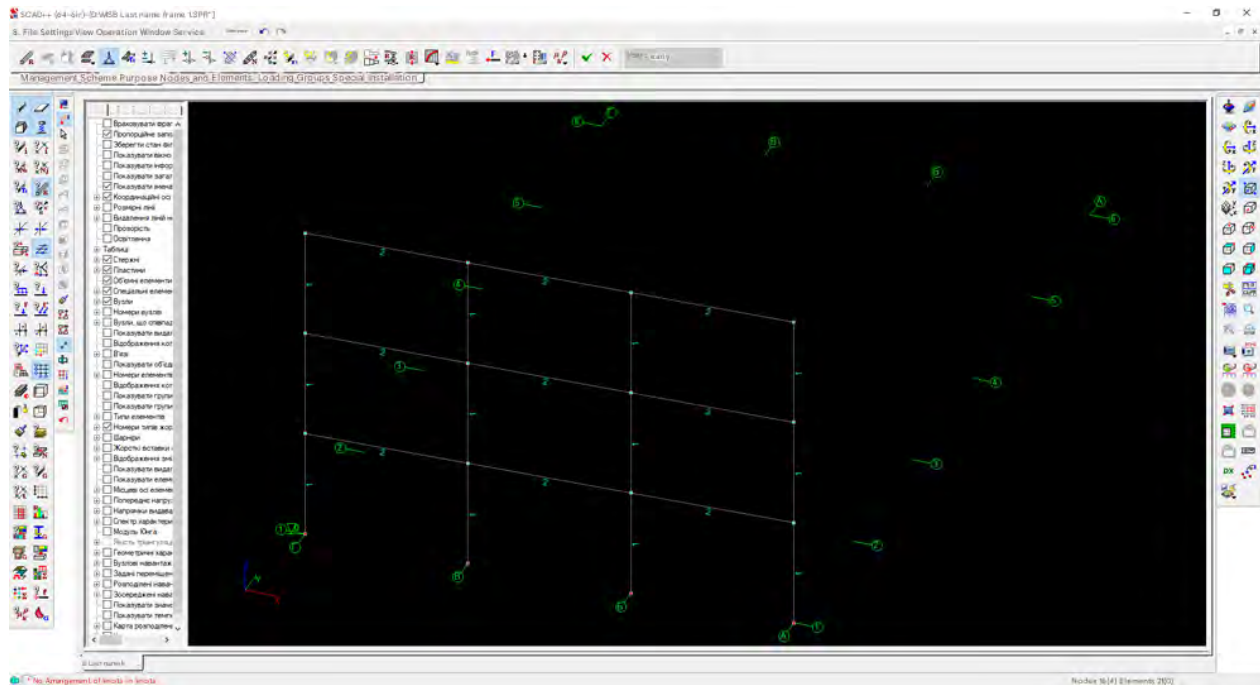


Figure 3.22 – Assignment of frame node junctions to ref. 0,000

To control the correctness of the assignment of elms, click the "Knitting" button on the filter panel

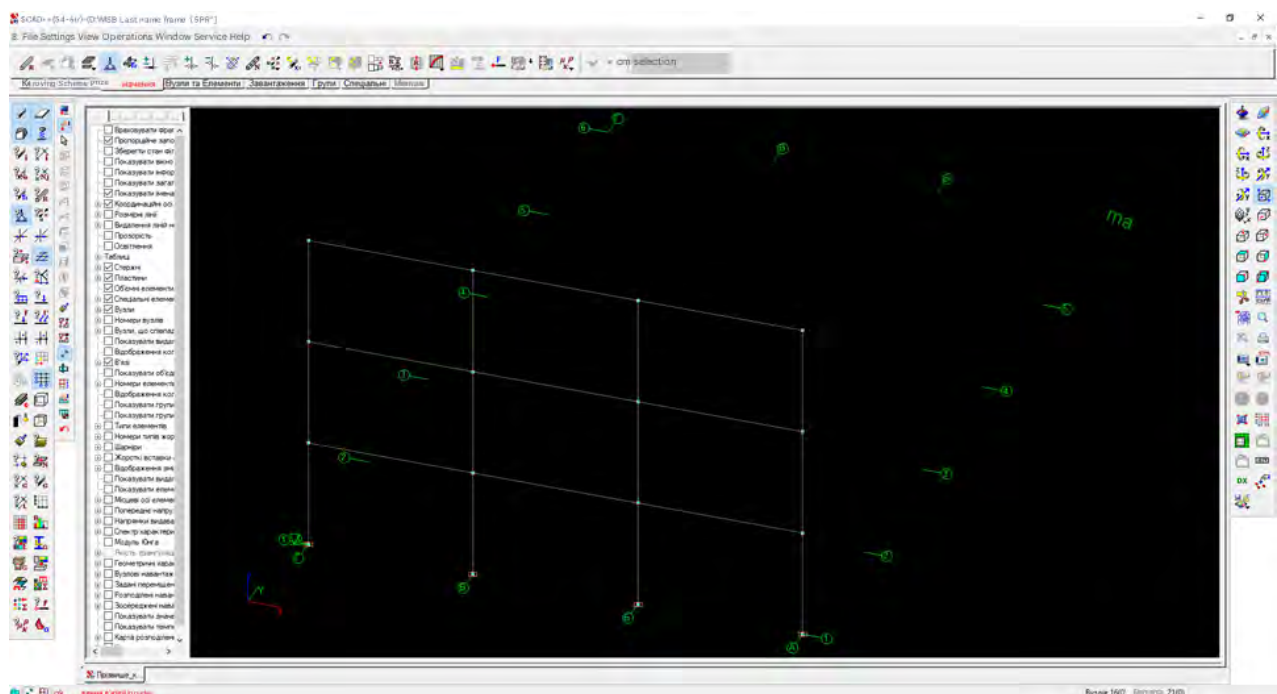


Figure 3.23 – Displaying elms in the working field of the program

Next, create a spatial design diagram by copying the frame along the 1 axis. To do this, use the tool for copying a fragment of the diagram on the diagram tab. To activate the tool, select the elements of the diagram that need to be copied. Go to the "Groups" tab and click  the "Mark elements" button.

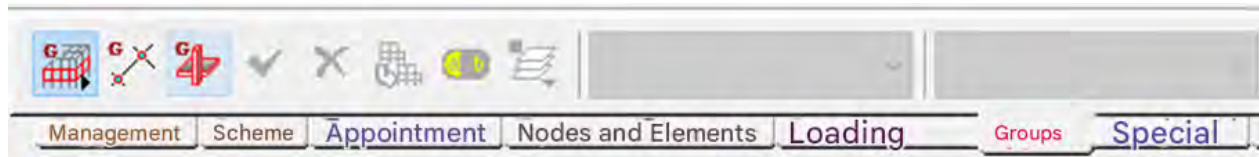


Figure 3.24 – "Mark items" button on the "Groups" tab

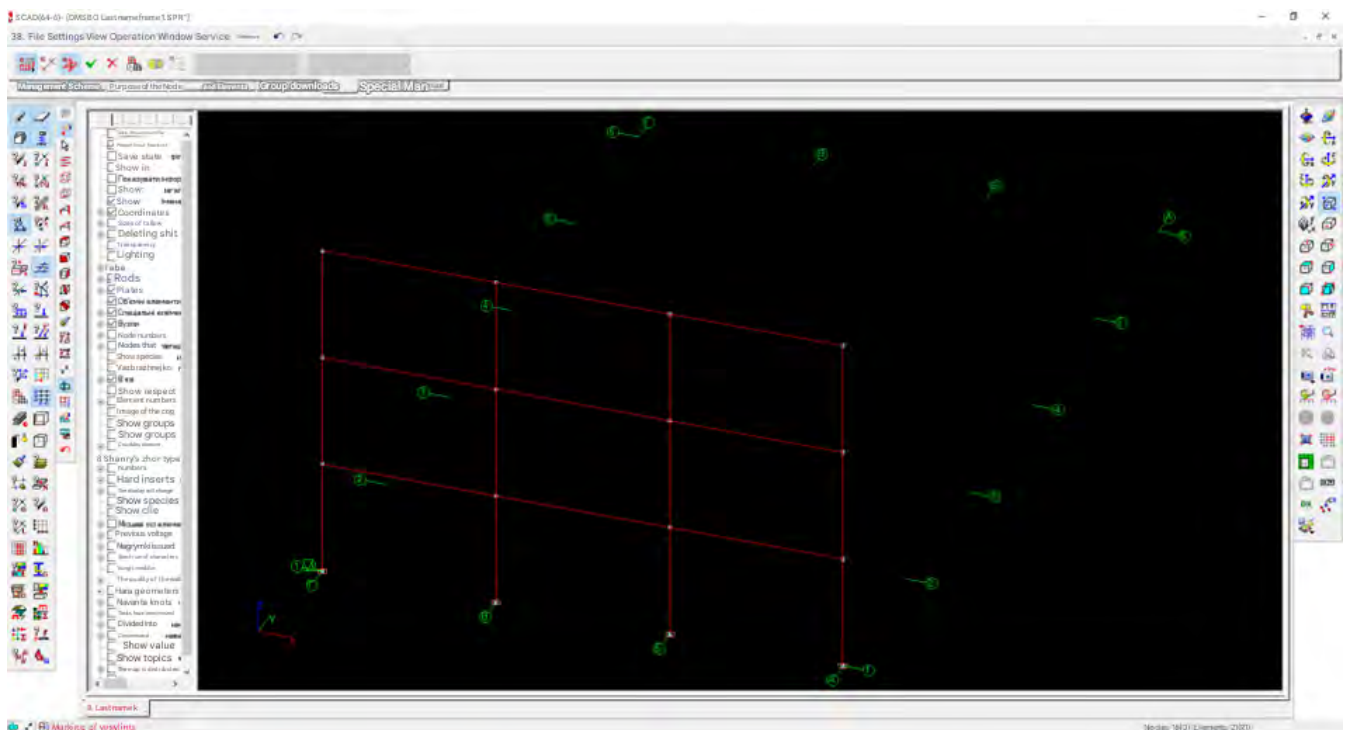


Figure 3.25 – The result of selecting frame elements in the working field of the program

After selecting the elements to be copied, the button  for copying a fragment of the diagram on the "Diagram" tab becomes active.

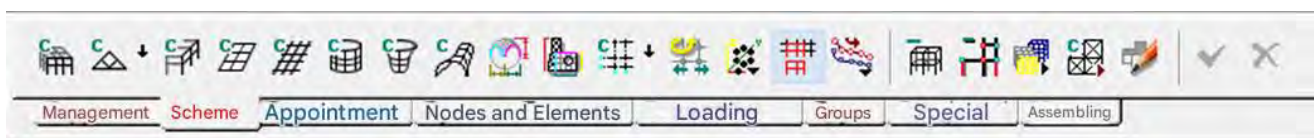


Figure 3.26 – Diagram toolbar

In the "Copy Diagram Fragment" dialog box, set the following parameters:

- copying in the Y direction;
- step – in accordance with the task;
- quantity – according to the task.

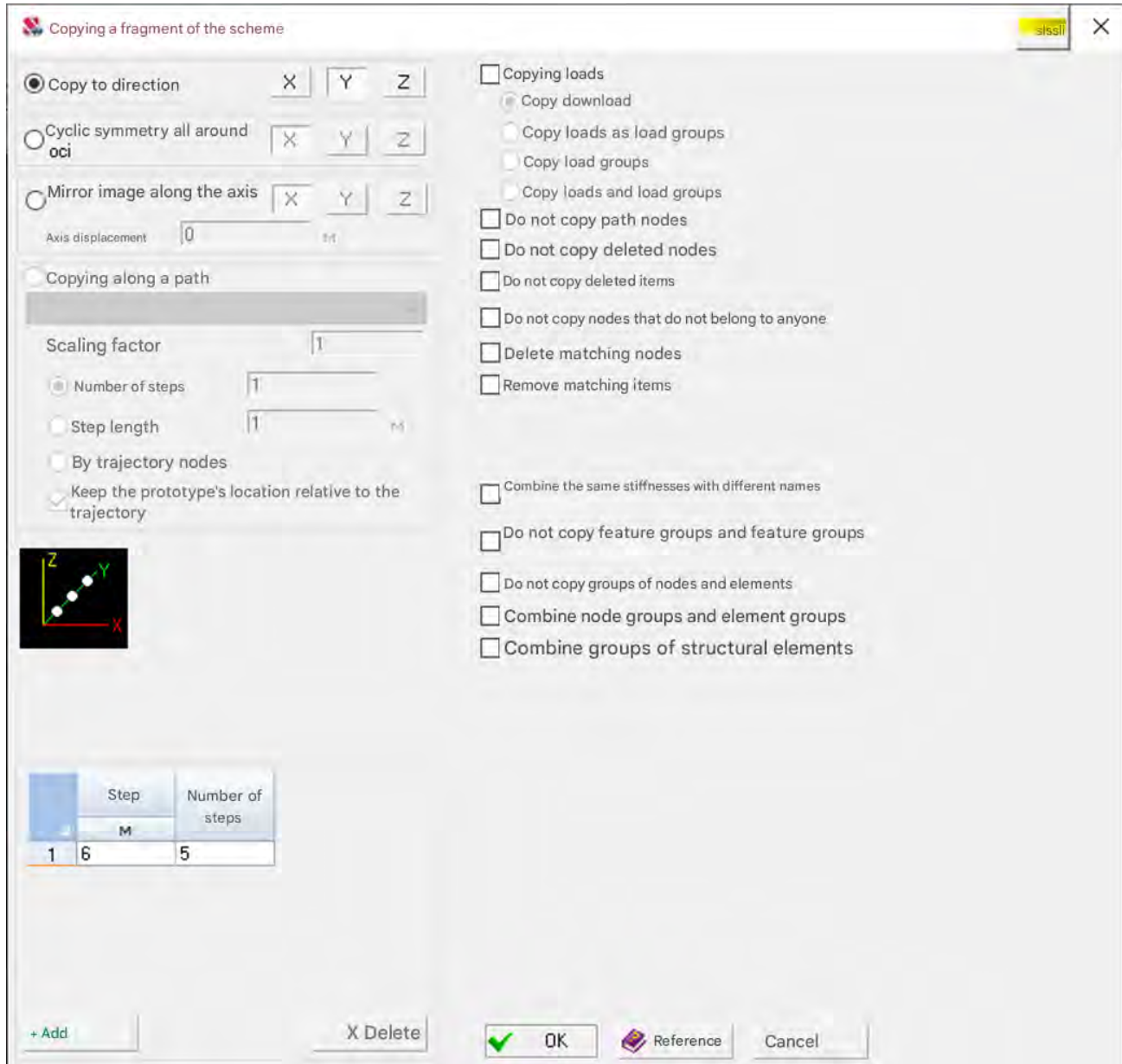


Figure 3.27 – Copy Schema Fragment Dialog Box

After setting all the necessary parameters, click the OK button and make sure that the copying is correct.

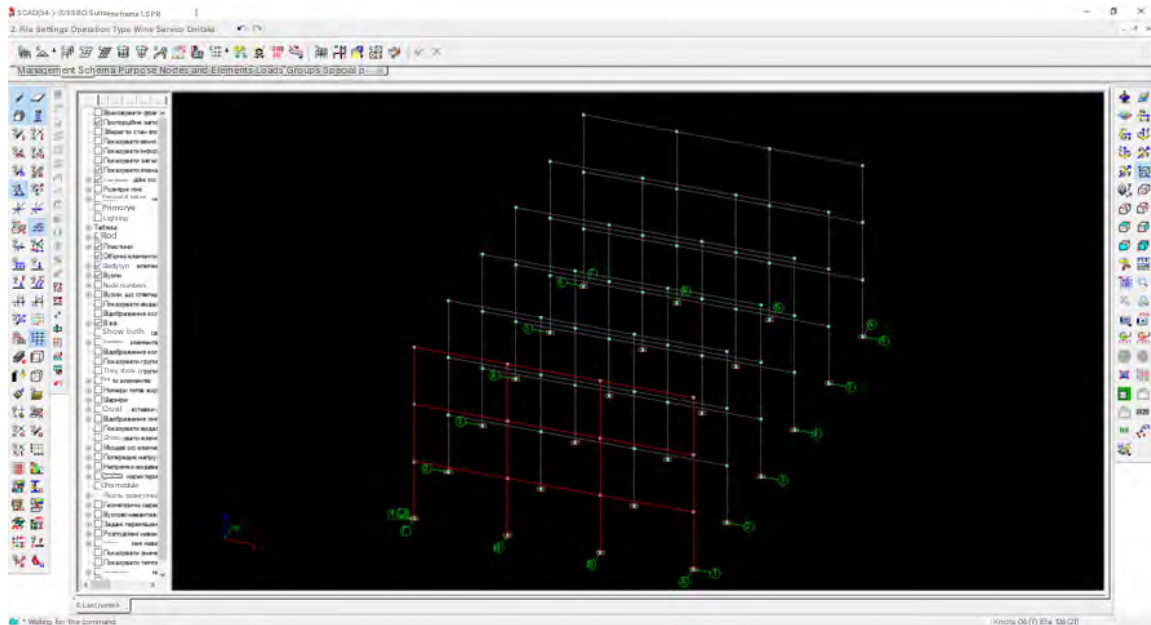


Figure 3.28 – The result of copying in the working window of the program

Remove the selection of the frame along the 1 axis using the button  "Reset all marks on the group tab. We connect adjacent frames with horizontal rods to form a spatial scheme. For the convenience of combining adjacent frames, use the  "Cross Section by Plane HOY" tool in the "Visualization Panel". We combine adjacent frames at the level of each overlap and cover with rod elements in the same way as the previous steps for creating a frame.

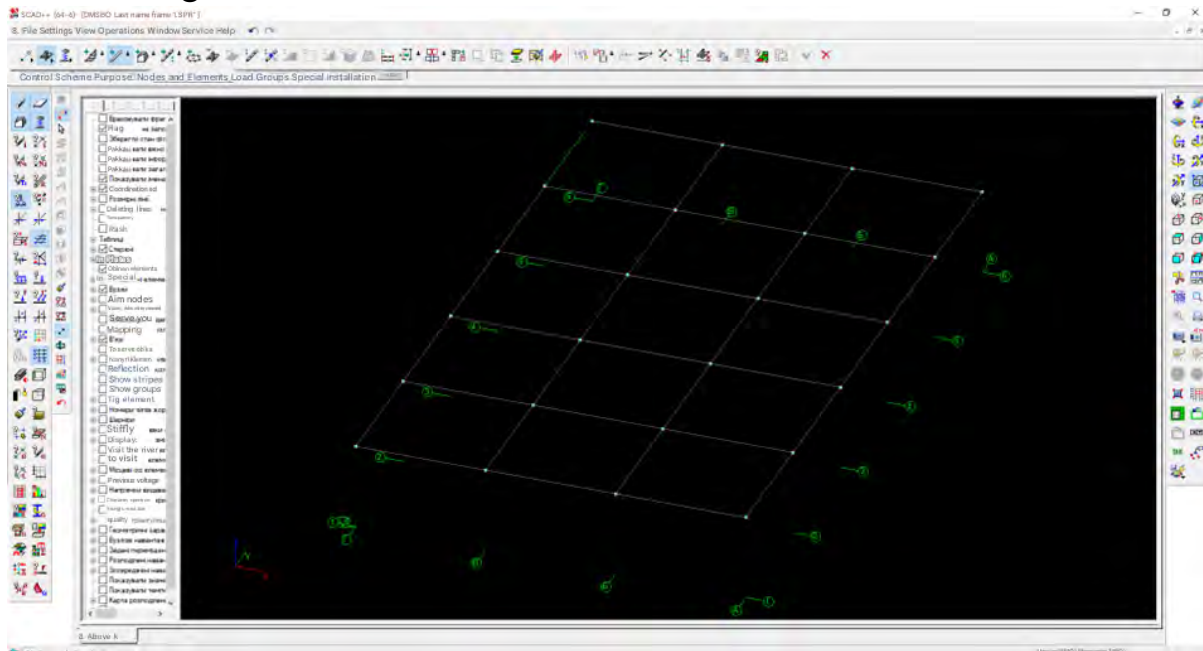


Figure 3.29 – Connecting adjacent frames with horizontal rods

To return to the original display of the diagram, use the  "Initial Diagram Mapping" button on the "Visualization Panel". Repeat the combination operation at each level of overlaps and coverage. After the combination is completed, return to the initial display of the diagram using the corresponding button.

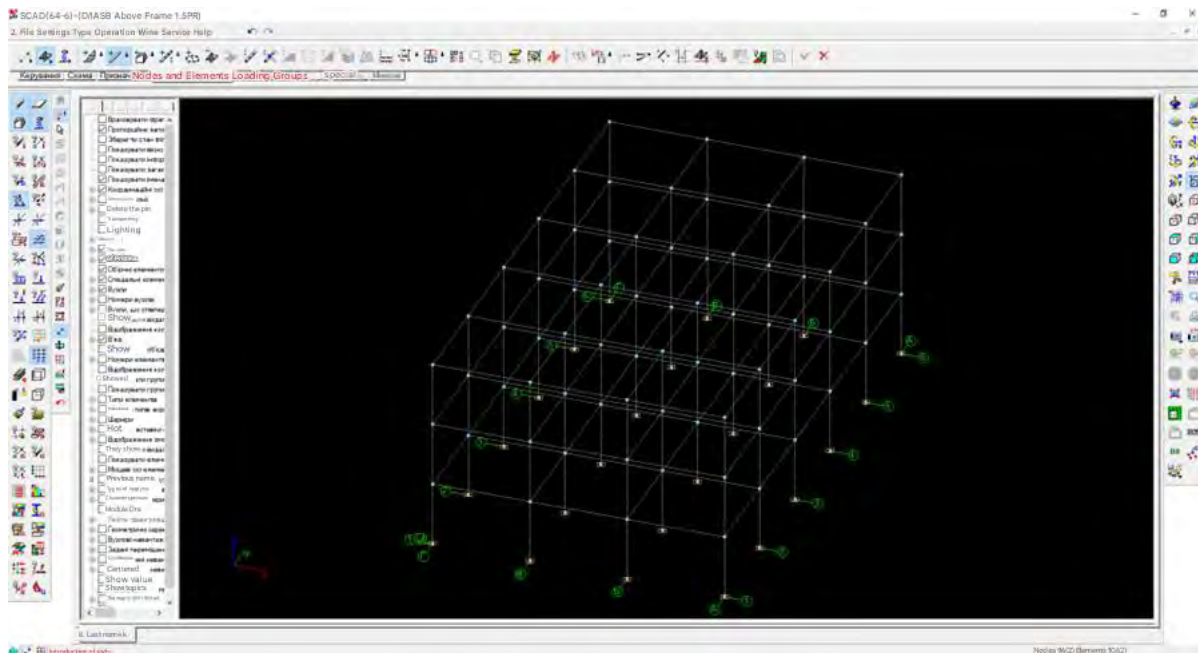


Figure 3.30 – Spatial display of the design scheme

We assign rigidities to new horizontal rods in the same way as the previous steps. We choose from the assortment hot-rolled steel I-beam according to DSTU 8768:2018 as a preliminary type of rigidity for horizontal elements that combine adjacent frames.

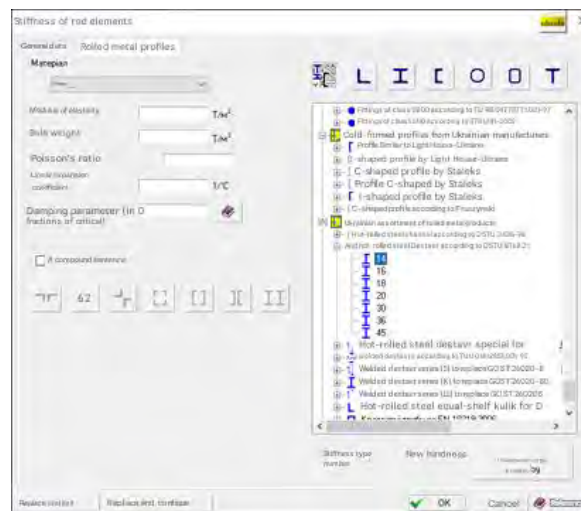


Figure 3.31 – Selection of assortment for the selection of stiffeners of longitudinal beams

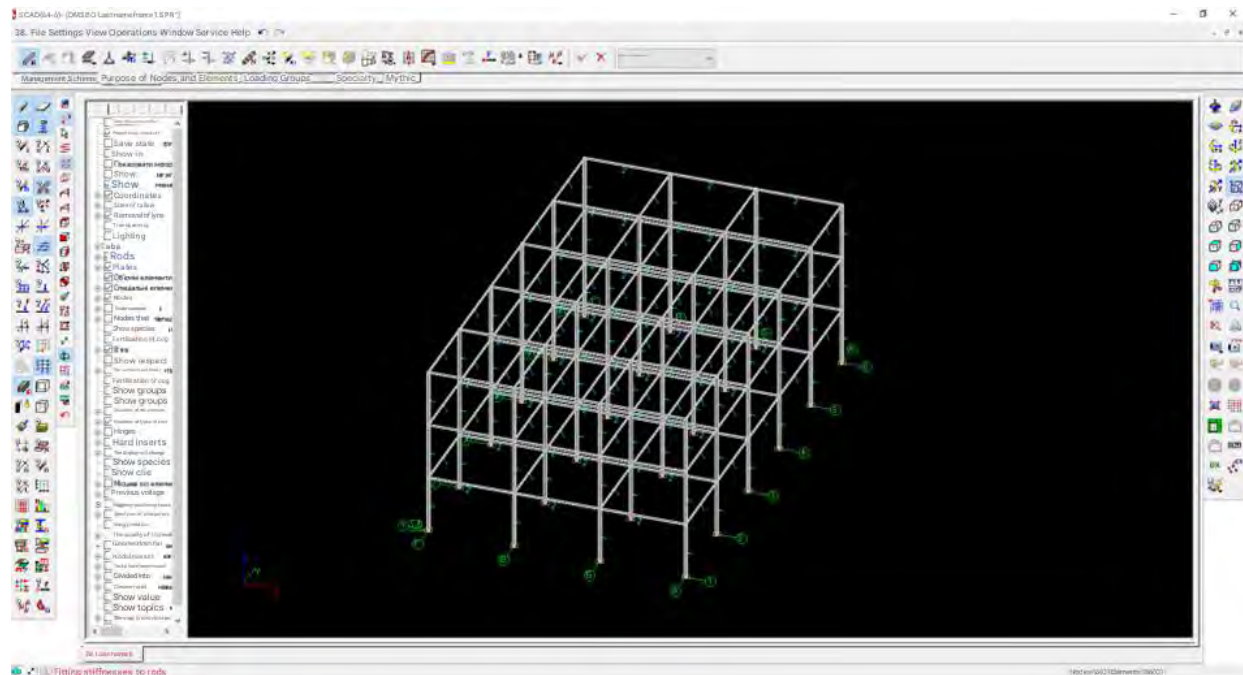


Figure 3.32 – Display of the circuit with assigned stiffeners

The next step in building the design scheme is the construction of vertical knits to give the frame the features of a frame-elm structural scheme in accordance with the task. Vertical knits are taken in a cross form and installed in each longitudinal frame in axes 3-4. Cross knits are built using the tool for adding rods. For ease of construction, use the  tool "Section by plane HOZ" on the "Visualization Panel".

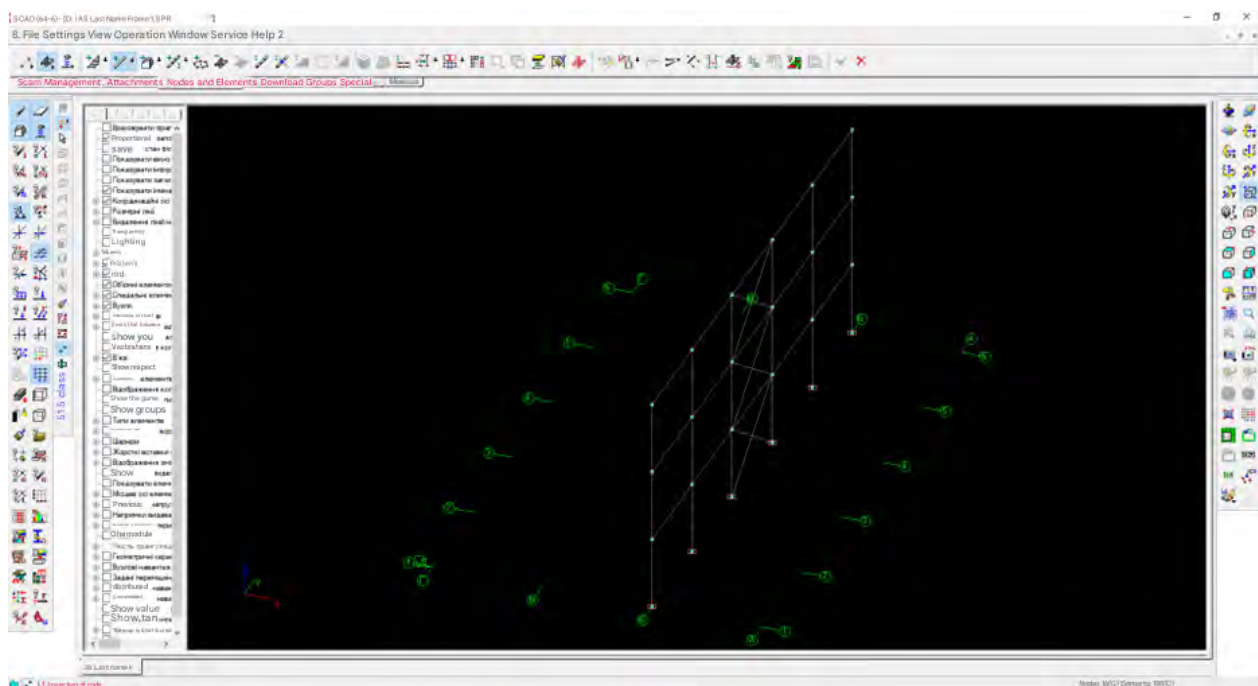


Figure 3.33 – Construction of vertical elms at the section along the B axis

To return to the original display of the diagram, use  the "Initial Diagram Display" button on the "Visualization Panel". Repeat the combination operation on the longitudinal frames along each letter axis.

We assign rigidity to new elements in the same way as the previous actions. As a preliminary rigidity for elms, we assign square steel pipes according to GOST 8639-82. We select the material from the drop-down list – ordinary steel.

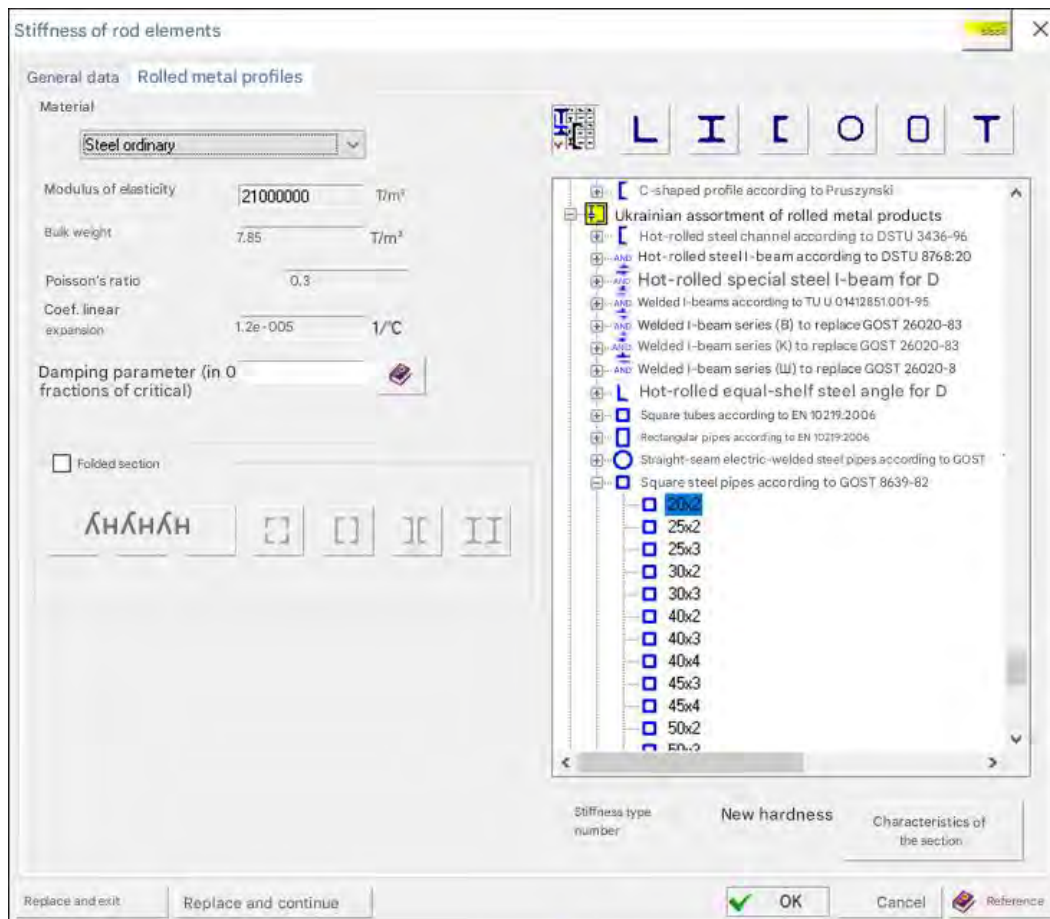


Figure 3.34 – Selection of assortment for assigning stiffness to vertical knits

After selecting the profile to assign pre-rigidity to vertical knits, click the OK button and select all vertical knits and press the confirm operation button.

For the correct calculation of frame lames, we perform crushing of the rod elements of the elms at the intersection point using the  "Bar Crushing at the Intersection Point" tool on the toolbar in the "Nodes and Elements" tab. After selecting the appropriate tool, select all the elements of the vertical elms and click the confirmation button. We ensure the correctness of the construction by activating the corresponding filters on the filter panel.

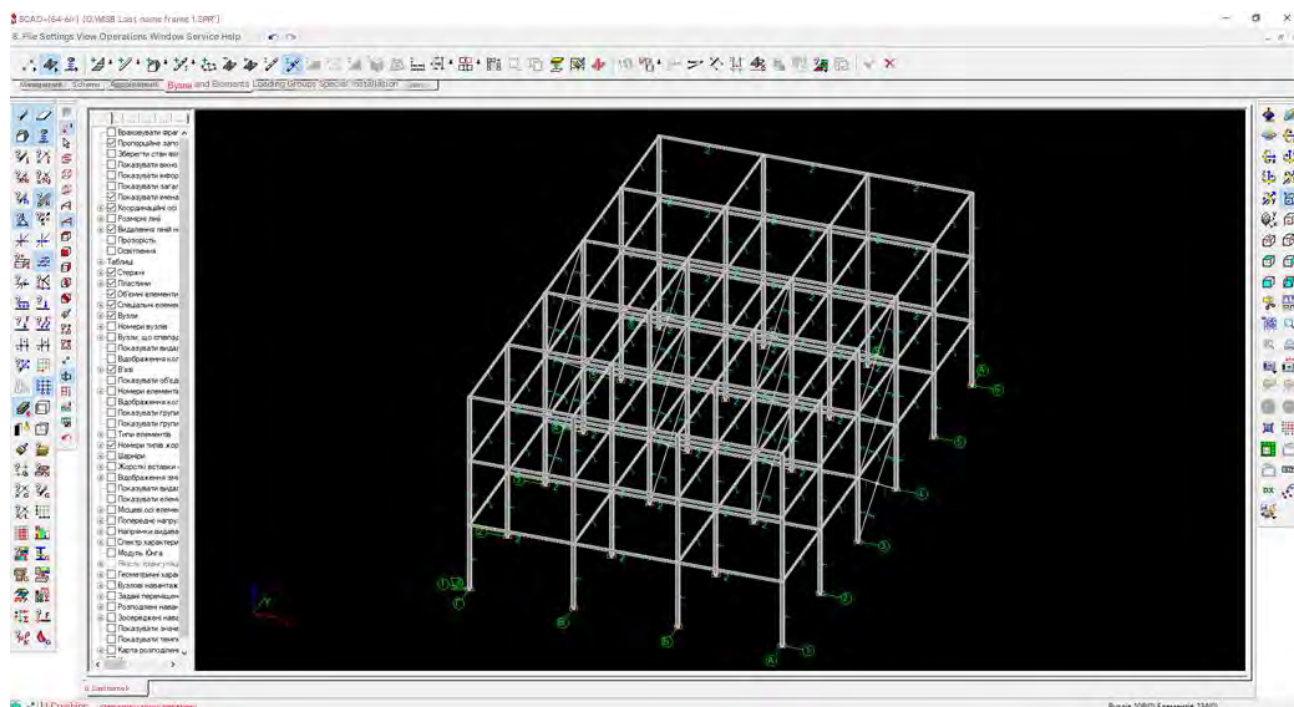
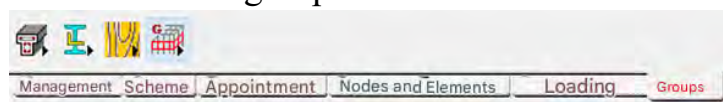


Figure 3.35 – Design diagram of the steel frame

For the convenience of further work and, if necessary, making modifications to the design model, its elements must be grouped according to design features.

You group items on the Groups tab using the



Work with Groups list tools.

When performing this task, it is recommended to distinguish the following groups of structural elements:

- columns of extreme frames;
- columns of medium frames;
- crossbars of the extreme frames;
- crossbars of medium frames;
- longitudinal crossbars of the extreme frames;
- longitudinal crossbars of middle frames;
- vertical knitting.

Expand the "Work with groups" list and use the element marking tool to select the elements that will be assigned to the first group.

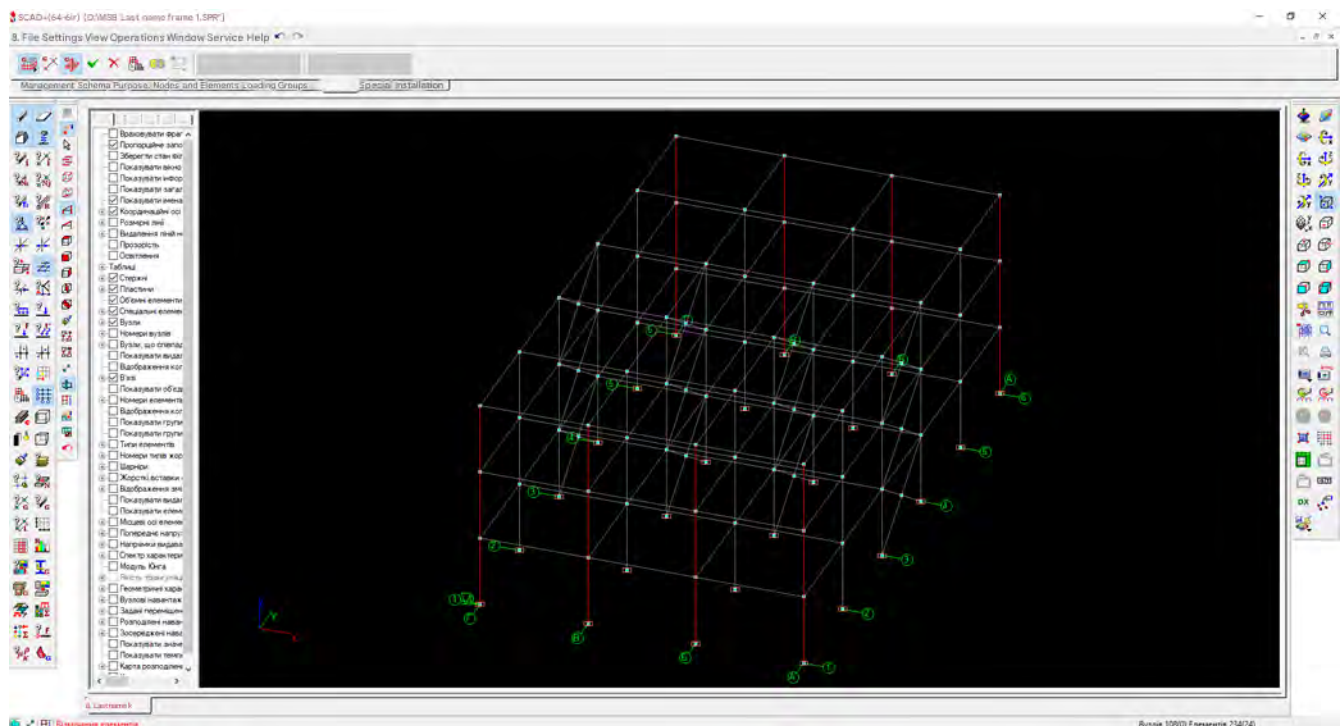


Figure 3.36 – Marking the elements of the group "columns of extreme frames"

After selecting all the necessary elements that should be included in the group, click  the "Save Group" button. In the dialog box, enter the name of the corresponding group and click on the button  Add a group, then on the OK button.

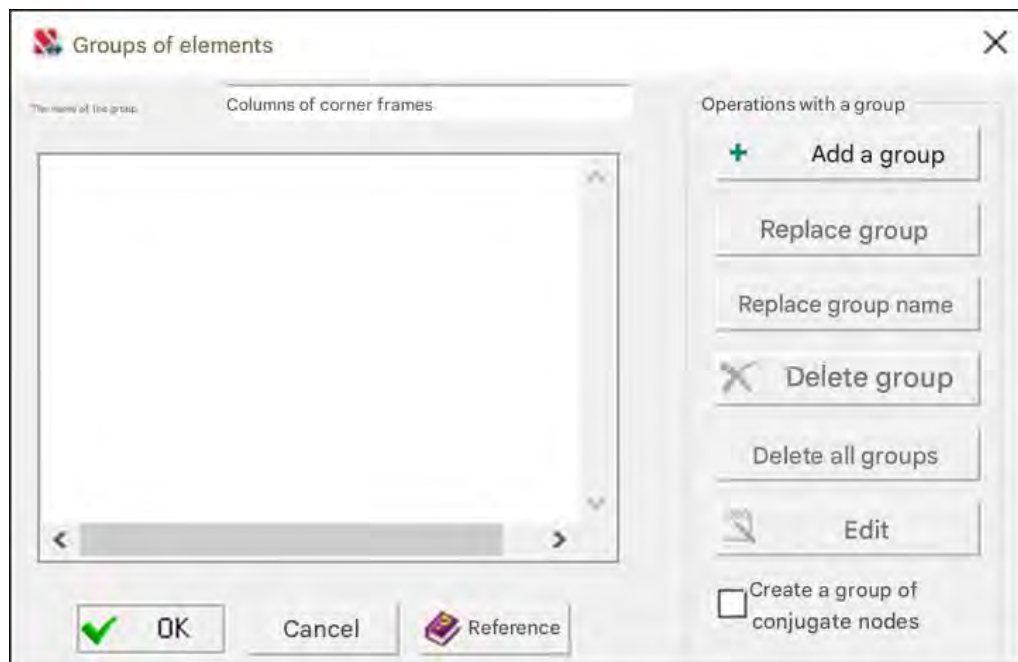


Figure 3.37 – Adding a group group "outermost frame columns" in the Item Groups dialog box

In the same way, we select the elements that should be included in the second group – "columns of medium frames".

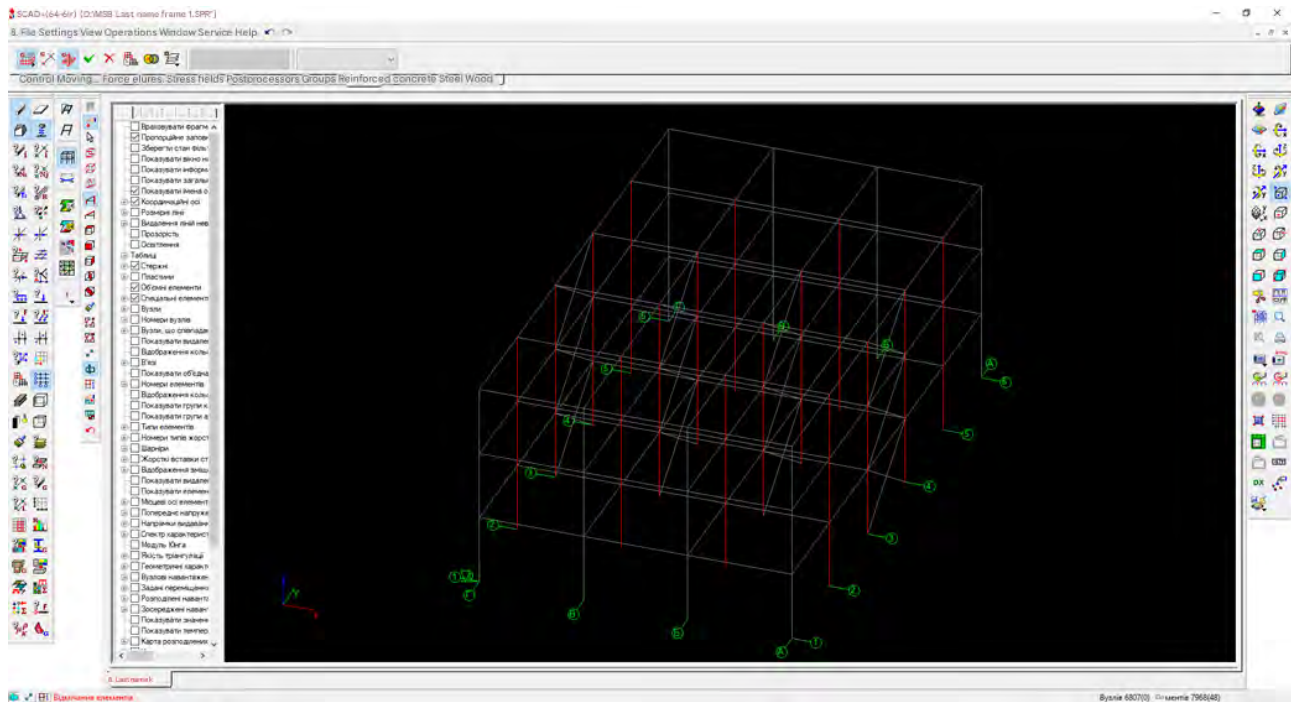


Figure 3.38 – Marking the elements of the group "columns of middle frames"

After selecting all the necessary elements that should be included in the group, click  the "Save Group" button. In the dialog box, enter the name of the corresponding group and click on the button  Add a group, then on the OK button.

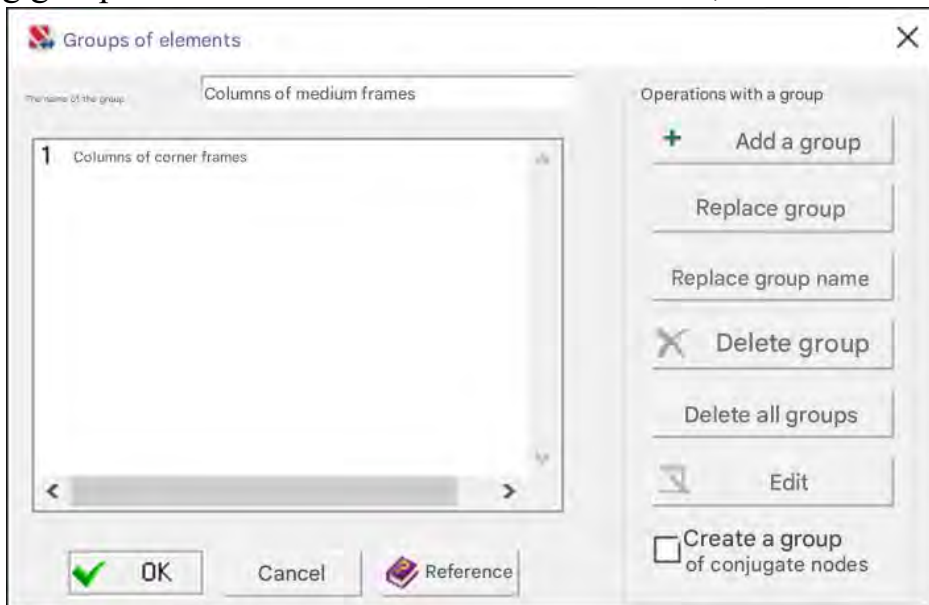


Figure 3.39 – Add a group group "columns of middle frames" in the "Groups of Elements" dialog box

Create the rest of the groups in the same way. Alternately select the elements of the corresponding groups and save them through the "Groups of Elements" dialog box.

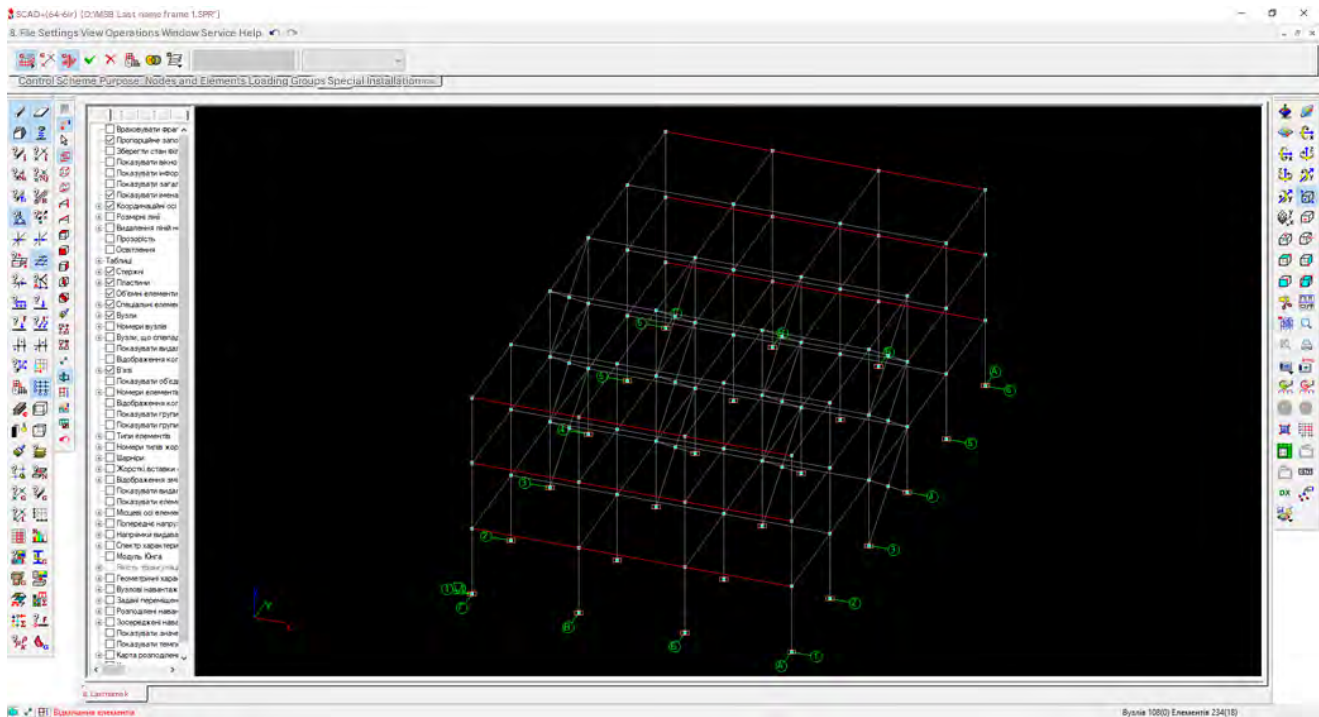


Figure 3.40 – Marking the elements of the group "crossbars of the extreme frames"

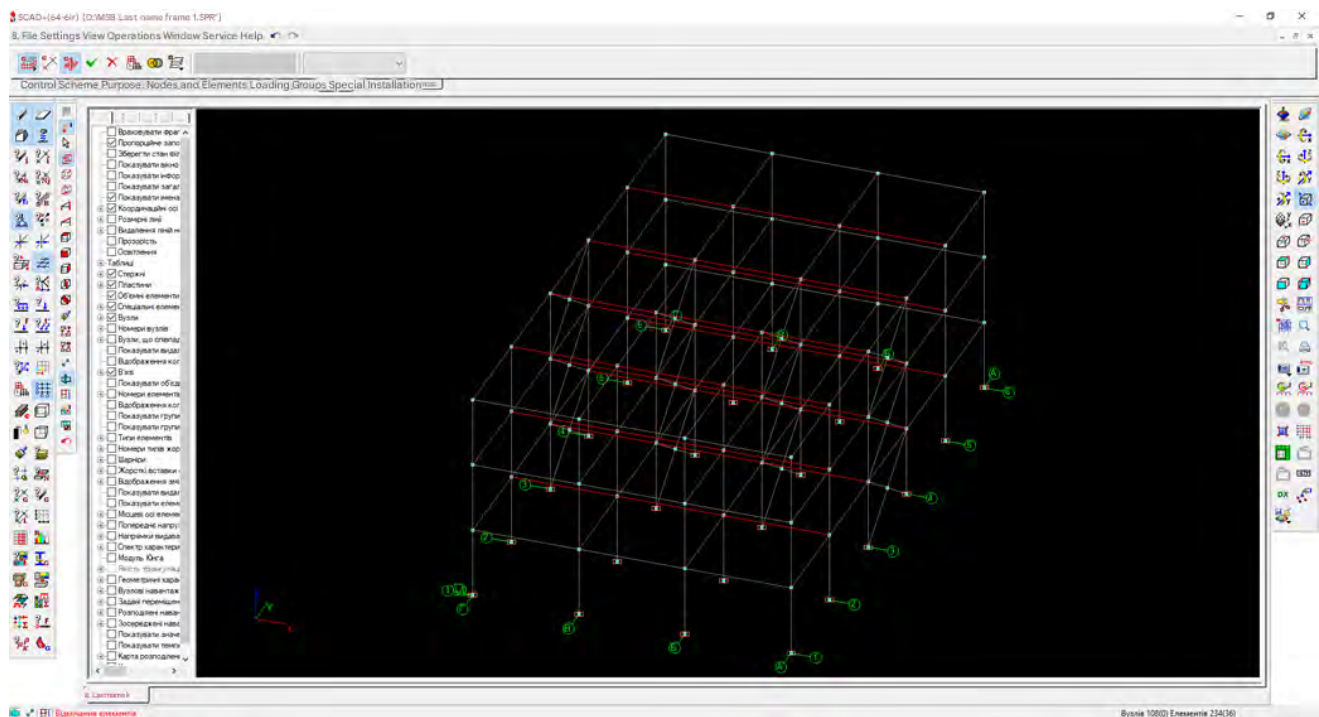


Figure 3.41 – Marking the elements of the group "crossbars of middle frames"

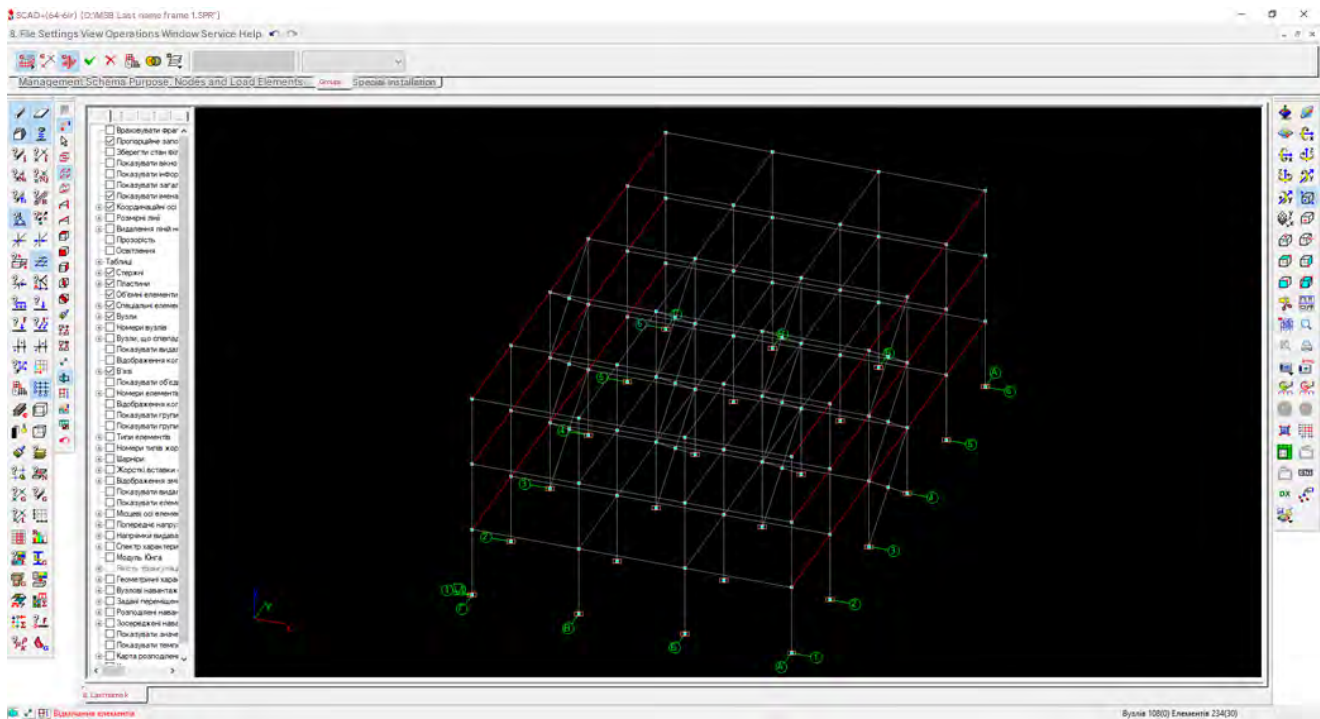


Figure 3.42 – Marking the elements of the group "longitudinal crossbars of extreme frames"

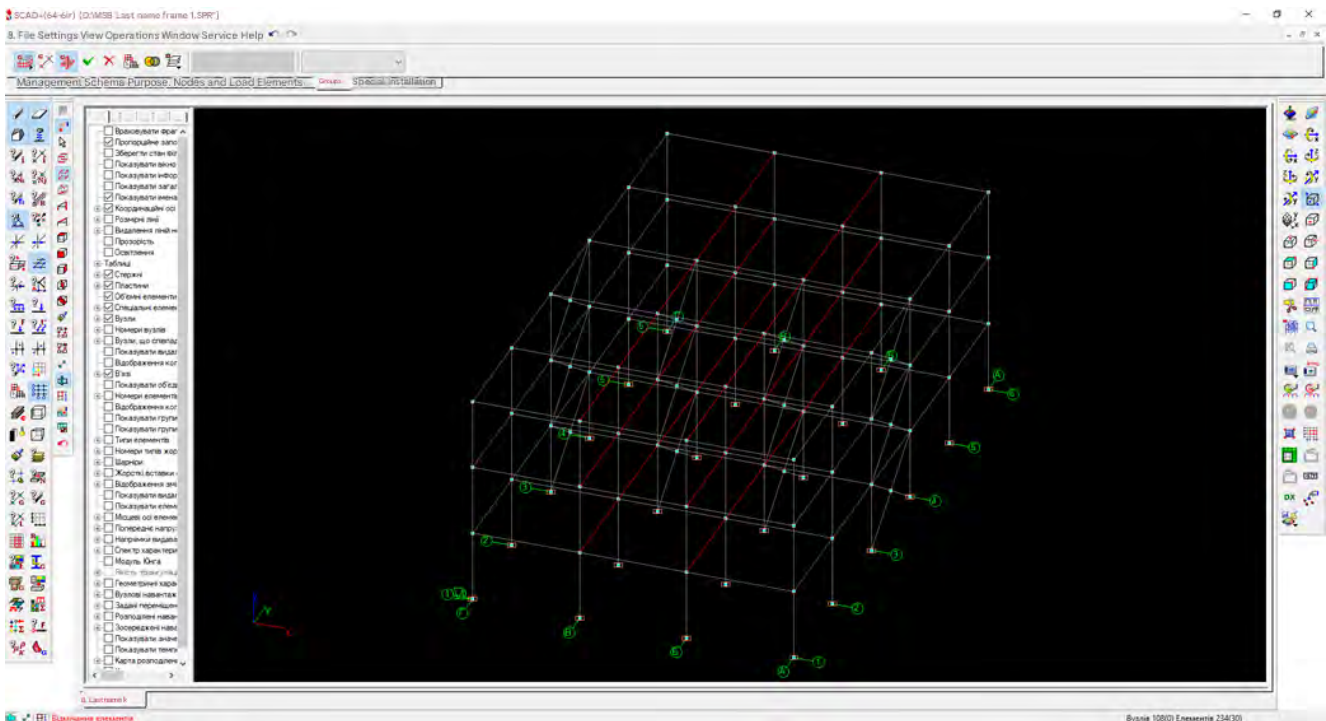


Figure 3.43 – Marking the elements of the group "longitudinal crossbars of middle frames"

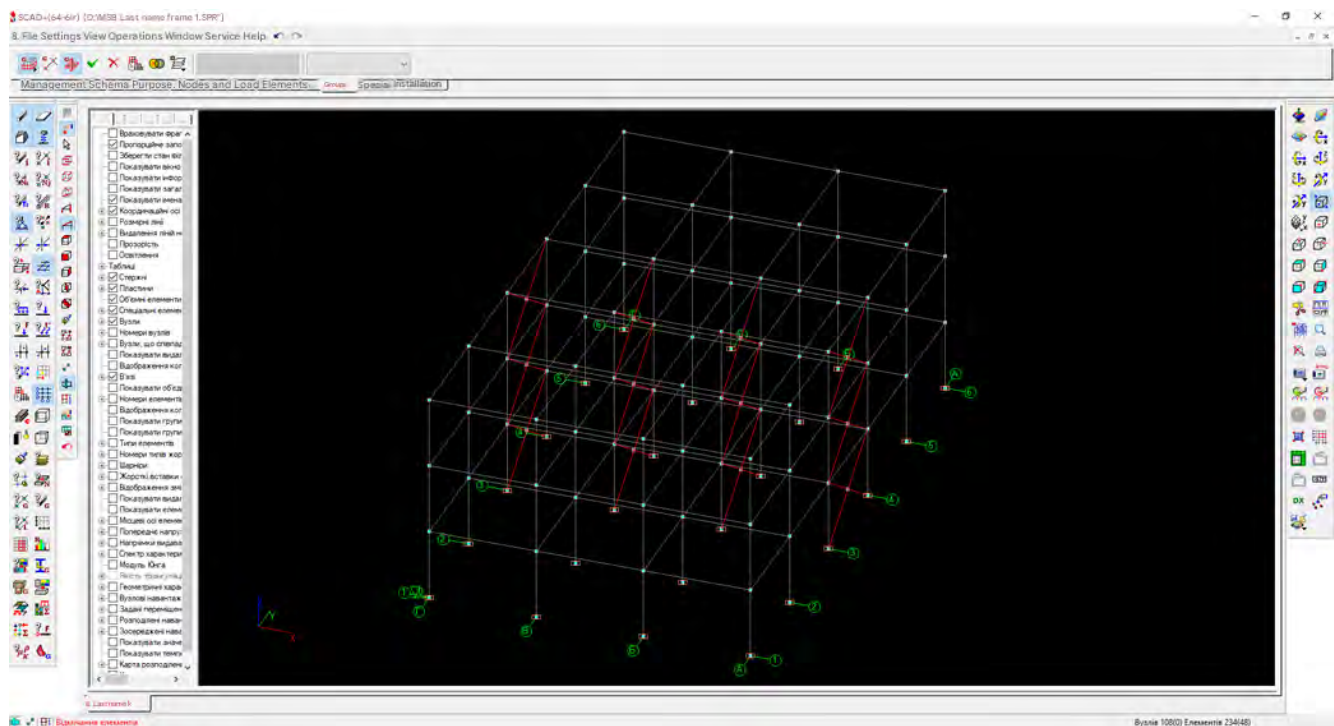


Figure 3.44 – Marking the elements of the group "vertical knits"

As a result, we should get 7 groups of elements, which are displayed in the "groups of elements" dialog box and in the drop-down list of "groups of elements".

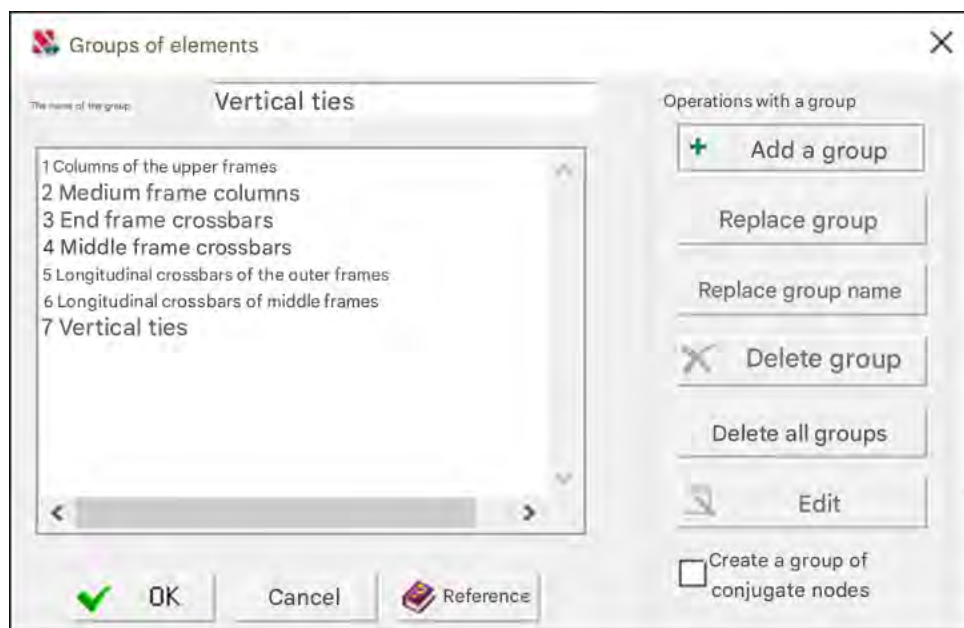


Figure 3.45 – Item Groups dialog box with created groups

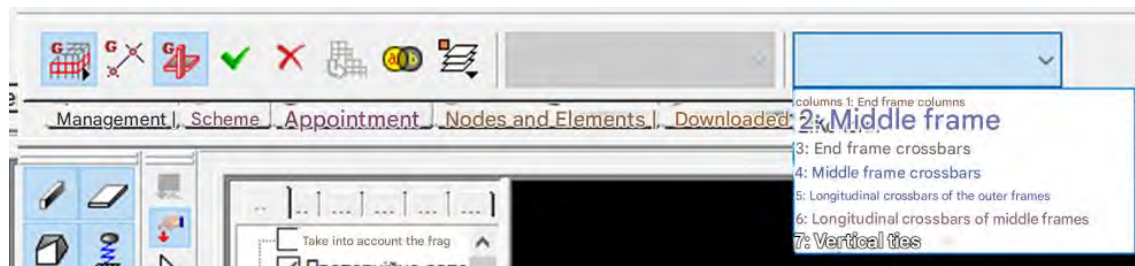


Figure 3.46 – Item Groups drop-down list

In further work, it is convenient to select and make active (selected) the elements of the frame that were assigned to the corresponding group from the drop-down list.

Creation of monolithic reinforced concrete floors.

With the help of the tool "Triangulation Mesh Generation..." Create the contours of reinforced concrete floors on each floor and covering. For ease of construction, use the XOY Plane Section tool in the render panel.

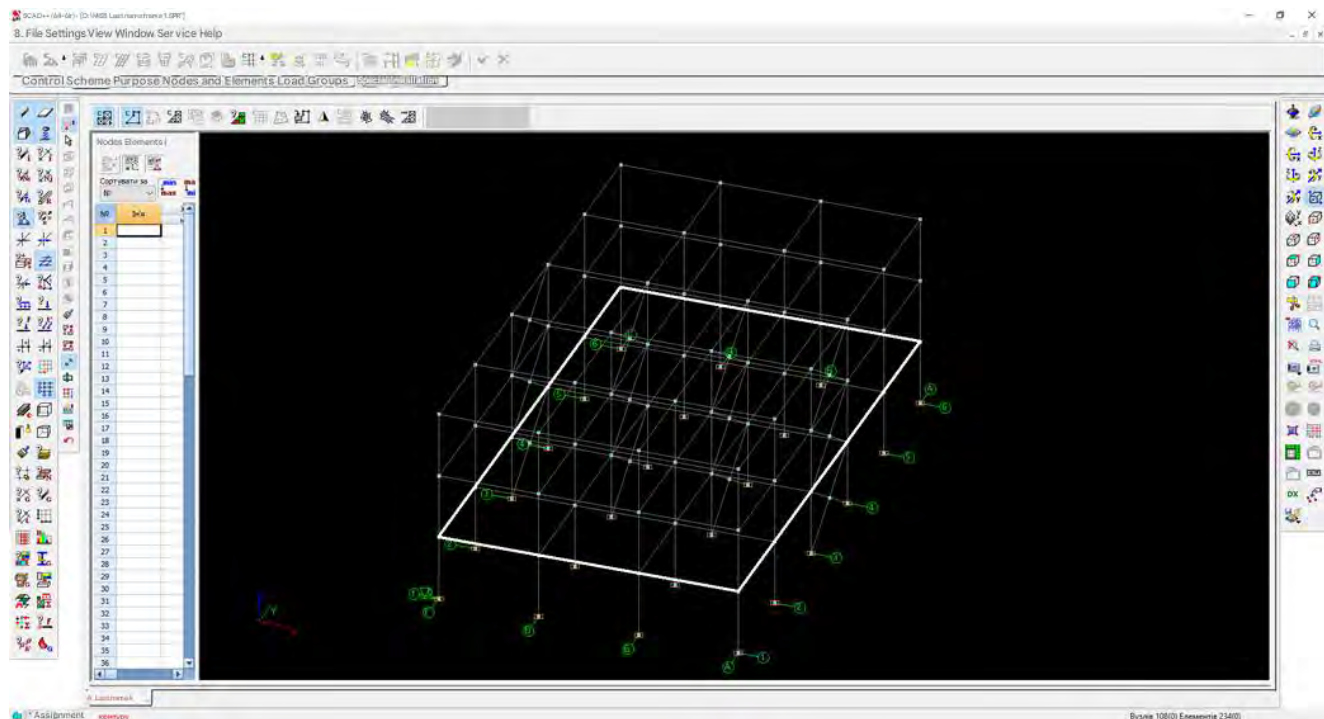


Figure 3.47 – Creating contours of floor floor triangulation on the design diagram

After creating the path, click the "Generate Mesh..." button and set the parameters shown in Figure 3.48:

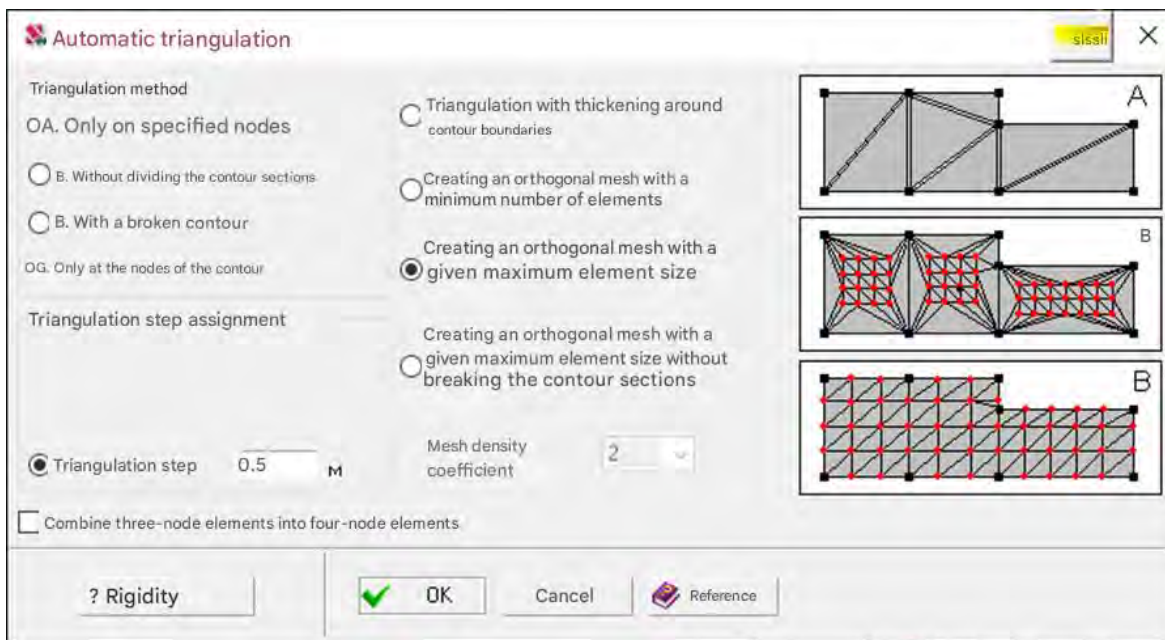


Figure 3.48 – Creating floor triangulation contours on the design diagram

Assign rigidity to the elements to be created: concrete of class C20/25, slab thickness 0.2 m.

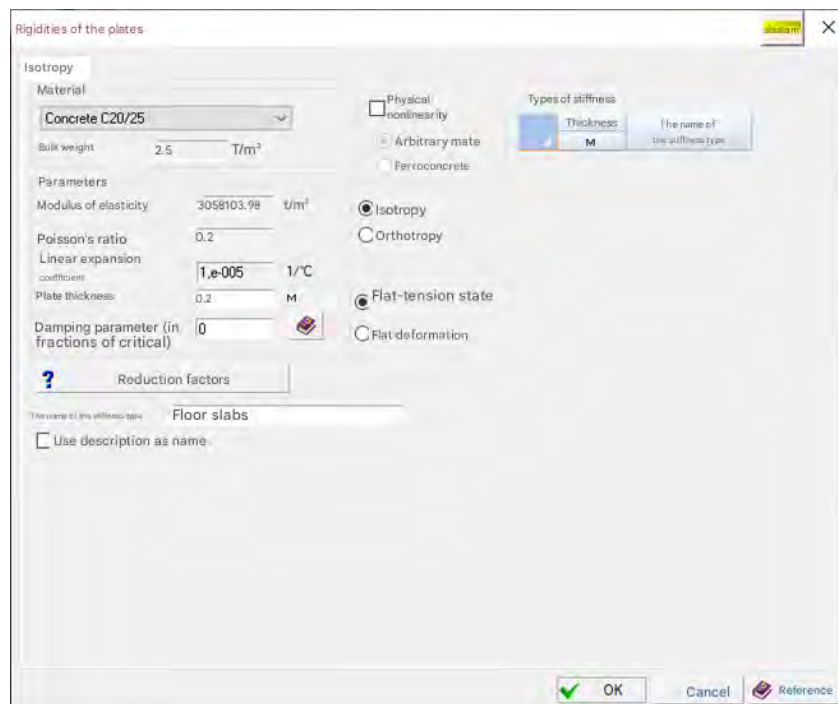


Figure 3.49 – Creation of floor triangulation contours on the design diagram

Click OK, after displaying the triangulation results, click the "Place triangulation results in place in the diagram" button.

Repeat the operation for all slabs and coverings

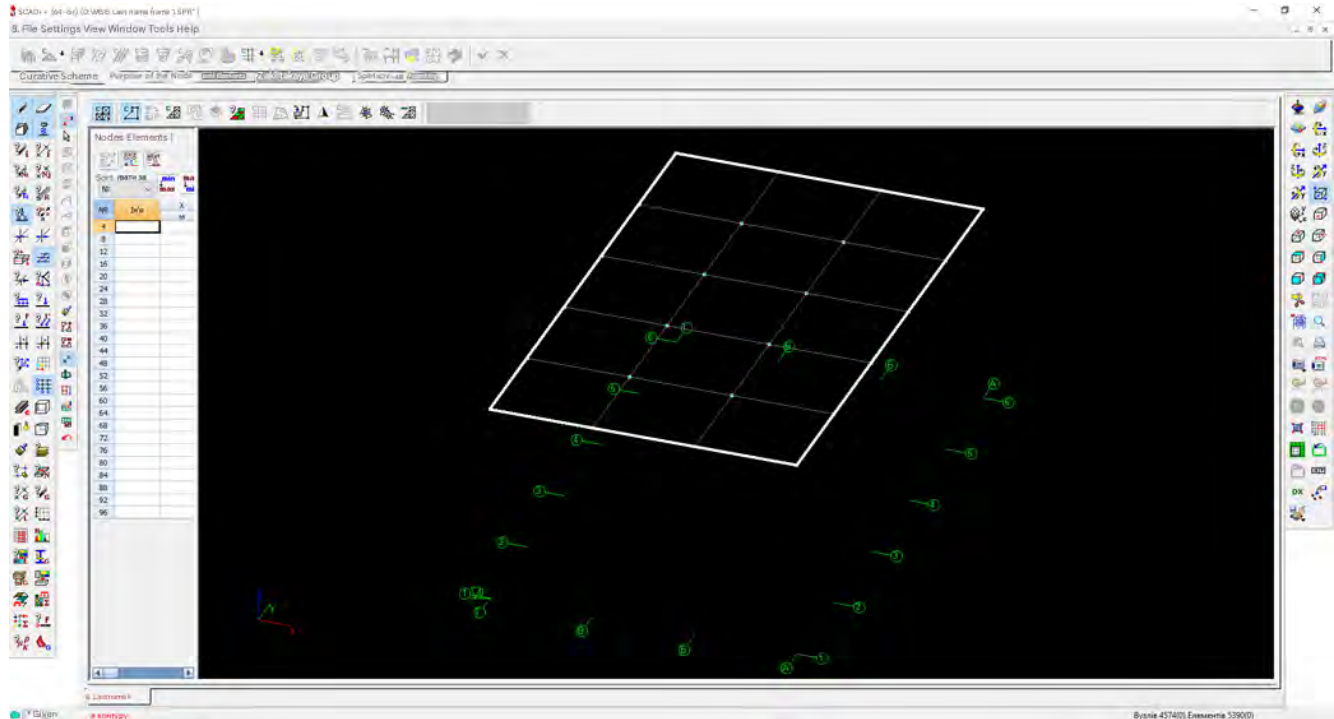


Figure 3.50 – Creating slab triangulation contours on the design diagram using the "XOY Plane Section" tool

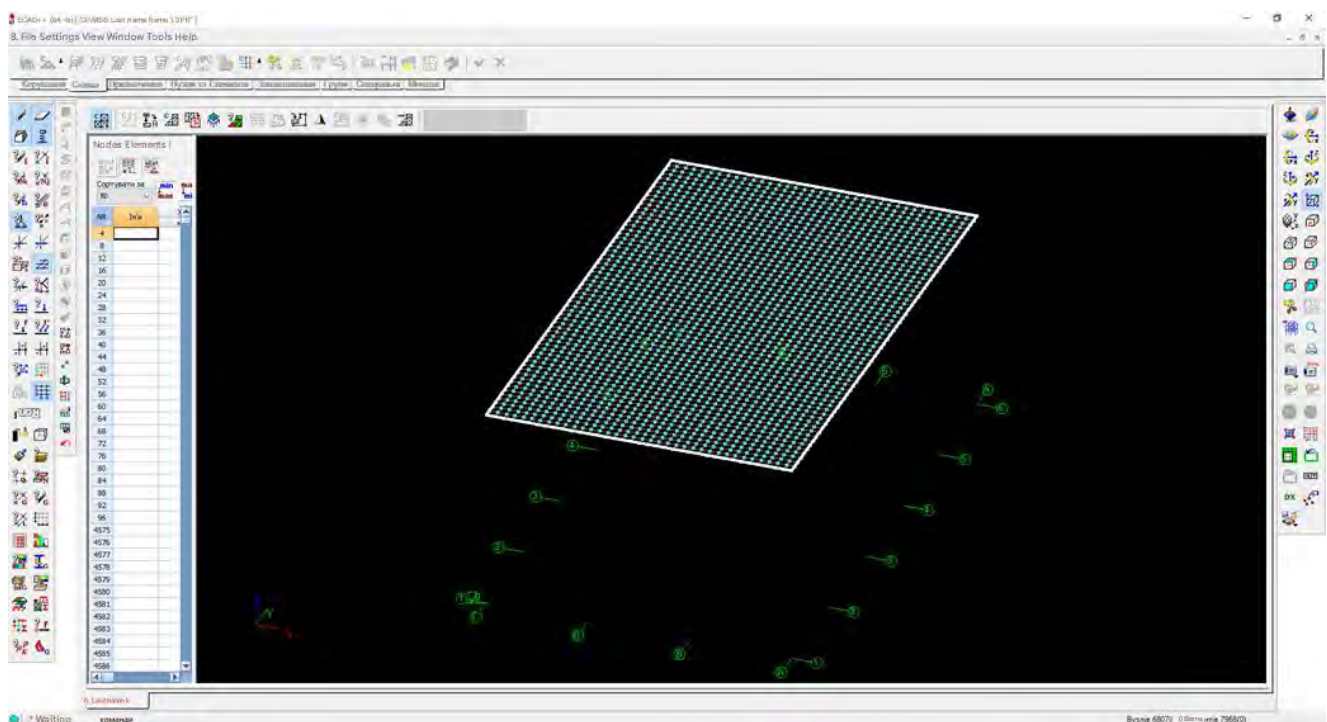


Figure 3.51 – Results of floor triangulation

Using the tools of the filter panel, make sure that the design scheme is created correctly.

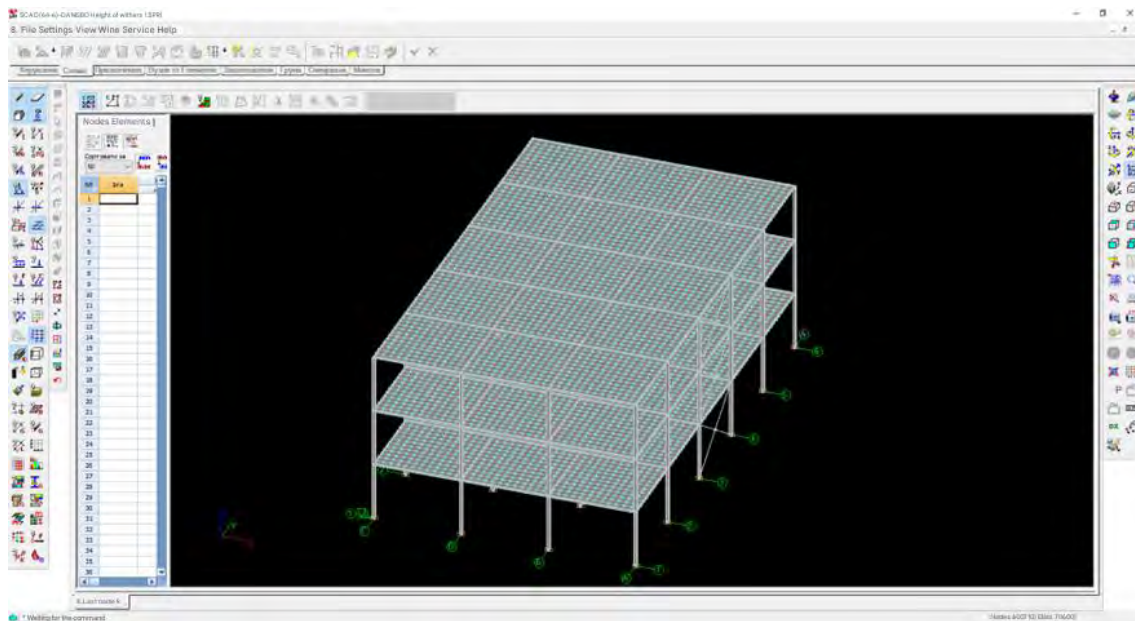


Figure 3.52 – Results of creating a design scheme with the display of the intended profiles of the steel frame and reinforced concrete floors and coatings

Exit triangulation mode by pressing the 

Let's add the created overlaps and coverages to the new groups.

Using the filter panel, we will disable the display of rod elements by clicking the  button.

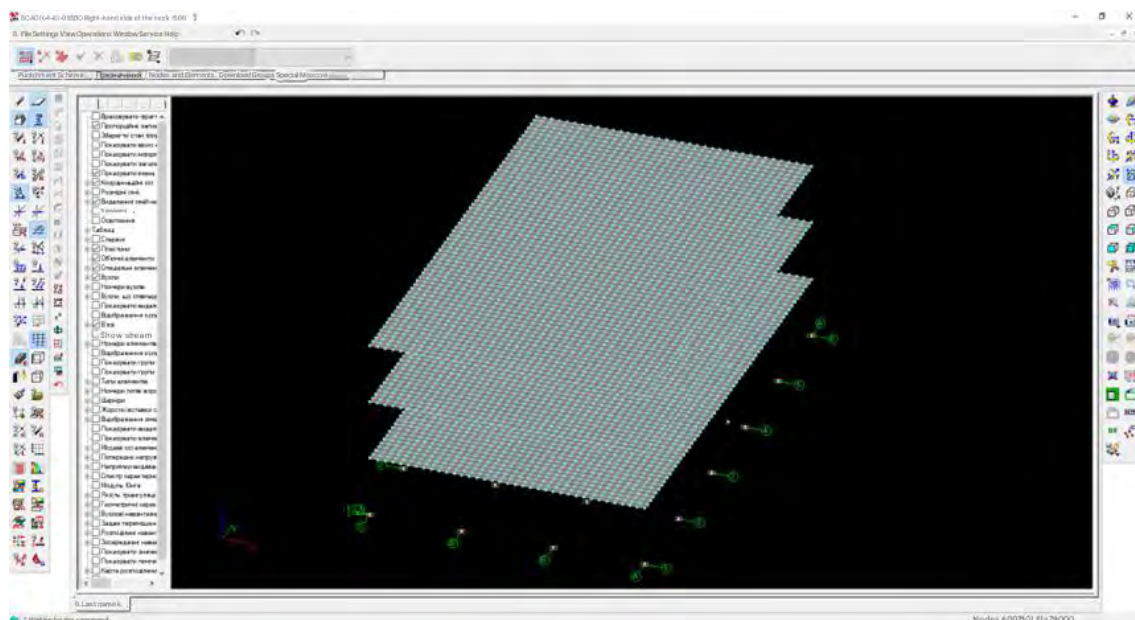


Figure 3.53 – Design scheme with switched off rod elements

For ease of use, rotate the design scheme by holding down Shift and the mouse wheel or using the rotation tools in the "Render Panel".

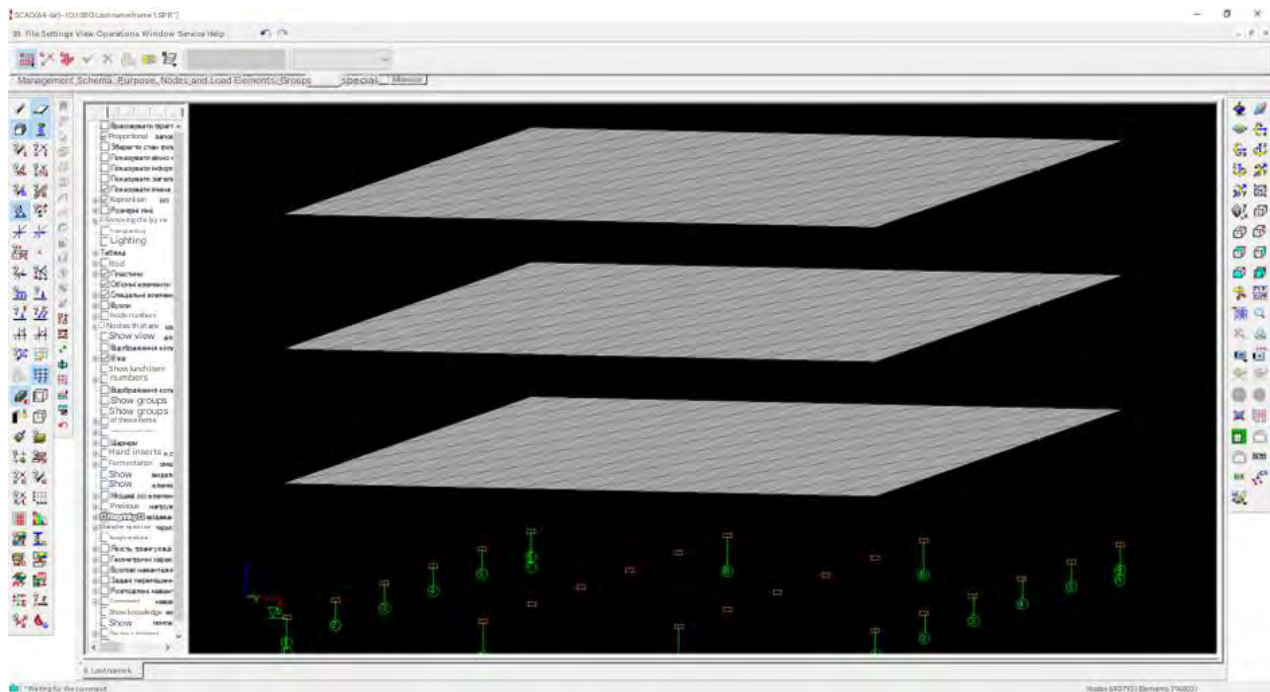


Figure 3.54 – Floor discs and coatings rotated for ease of operation

Go to the groups tab and activate the  "Mark Items" tool.

Right-click and set the selection to a rectangle. Select all the overlaps on the design diagram.

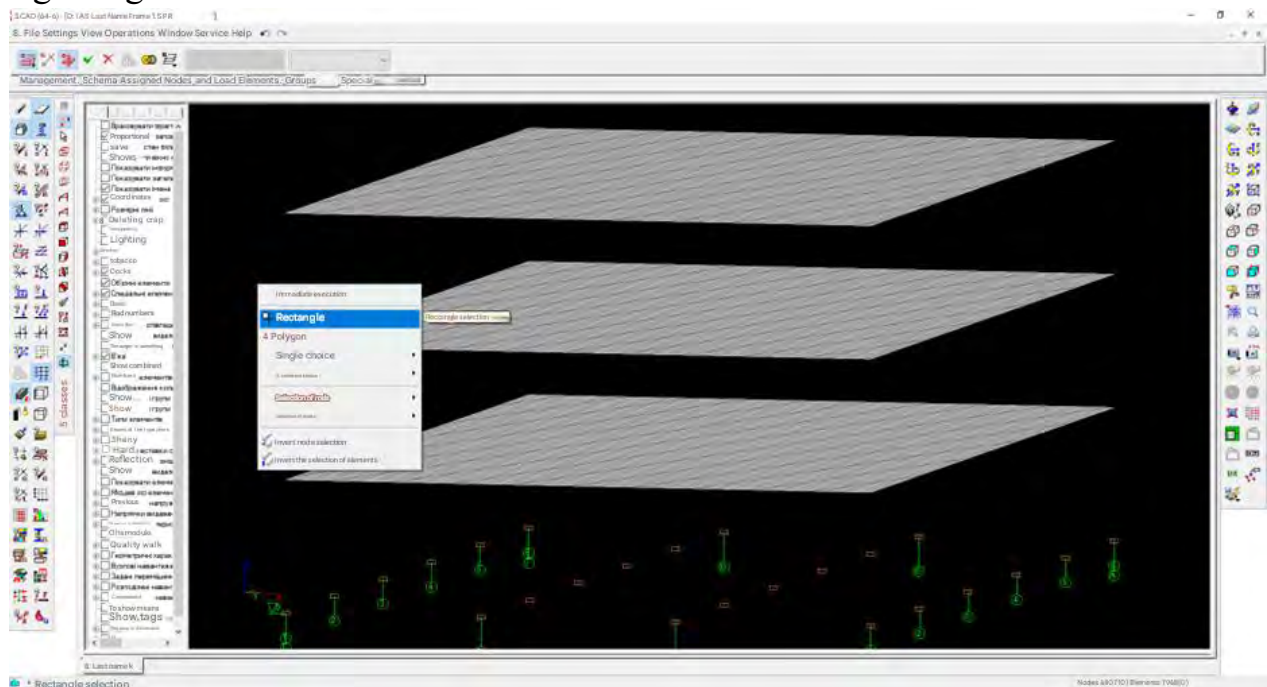


Figure 3.55 – Activating the Rectangle Select tool

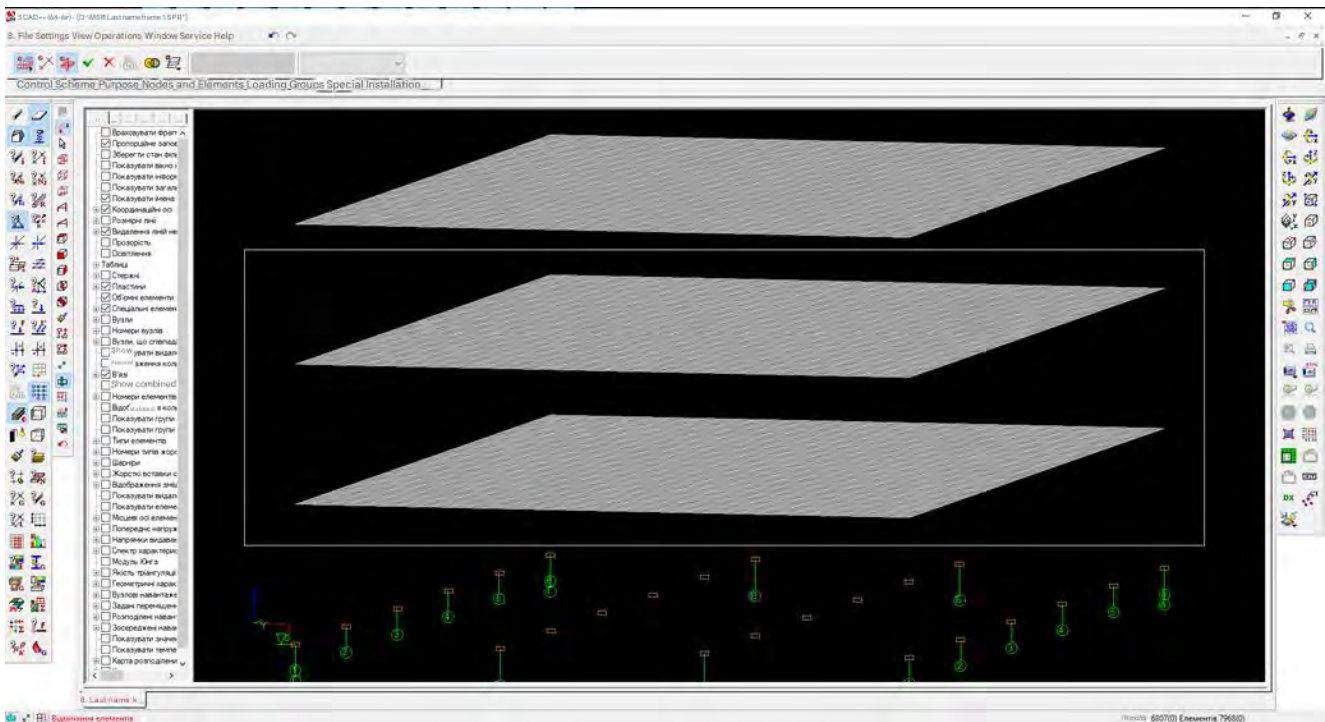


Figure 3.56 – Using the Rectangle Selection tool

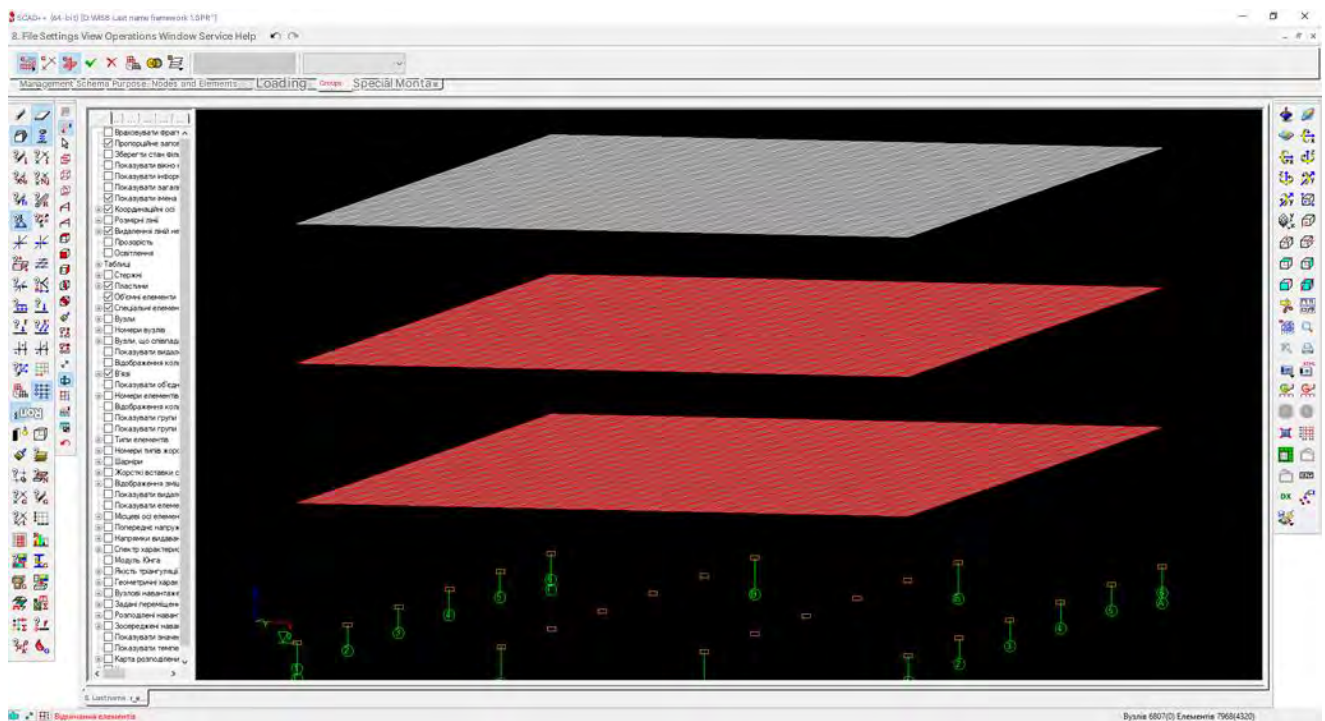


Figure 3.57 – Results of using the Rectangle Selection tool

After selecting the overlap elements, click the "Save Group" button. In the Element Groups dialog box, add and save a group called Reinforced Concrete Floors.

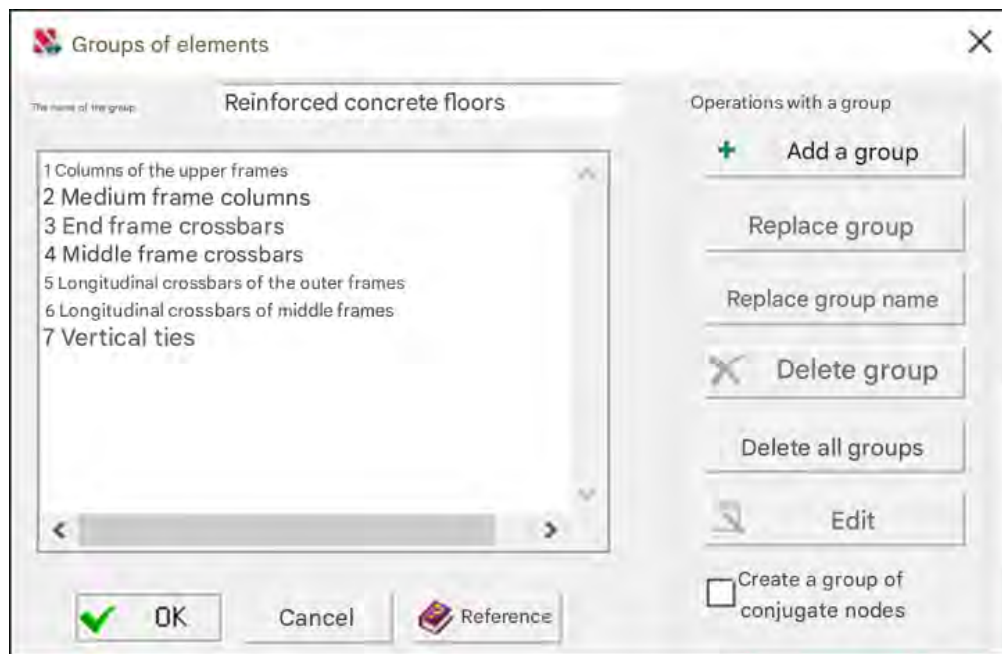


Figure 3.58 – Creating the "Reinforced Concrete Floors" group in the "Groups of Elements" dialog box

Similarly, create the "Reinforced concrete pavement" group.

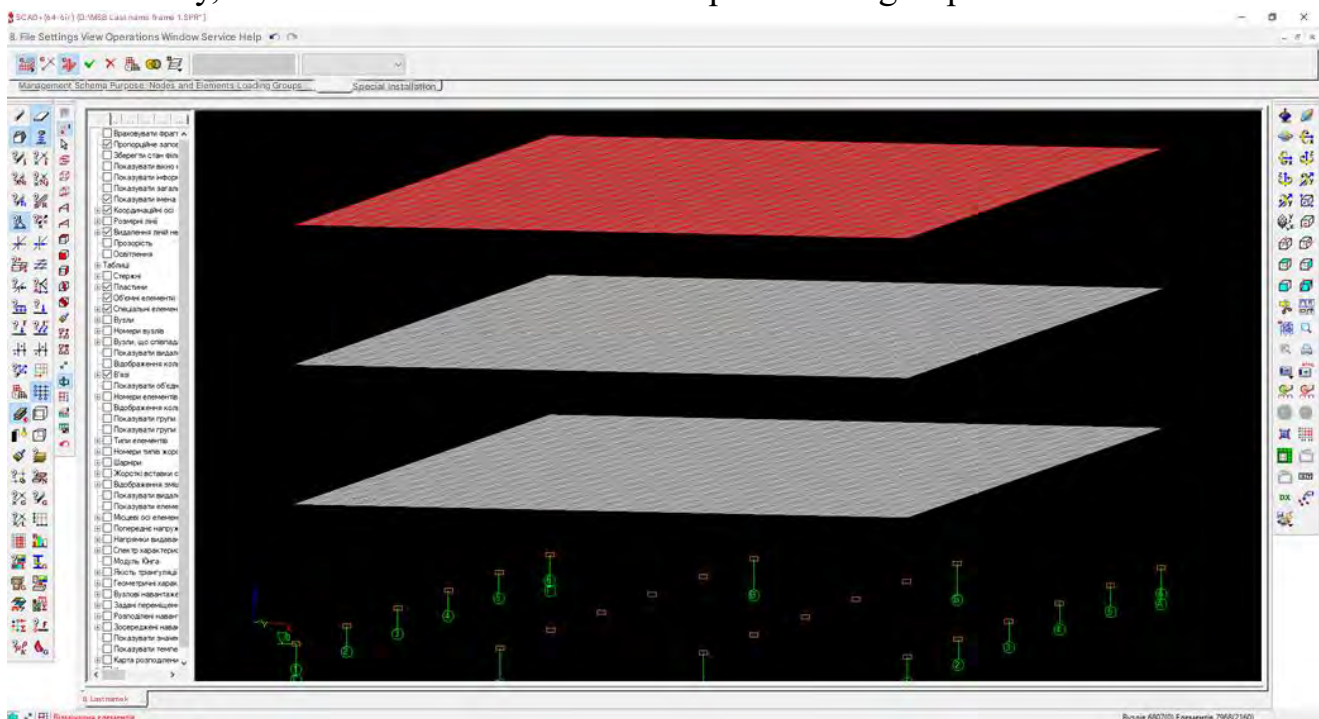


Figure 3.59 – Selected elements of the reinforced concrete coating to create the appropriate group

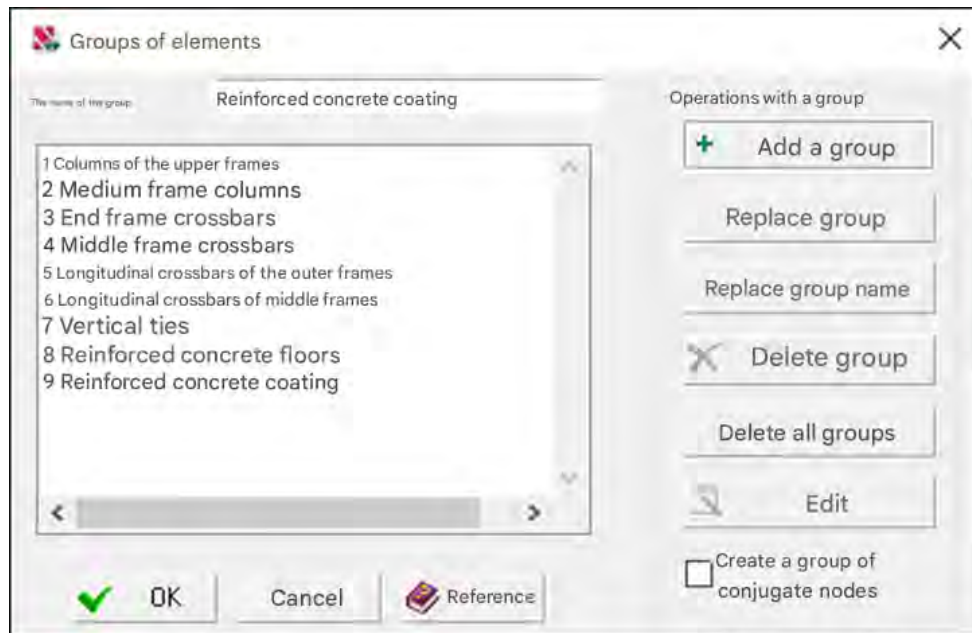


Figure 3.60 – List of created groups in the Item Groups dialog box

Enable the display of all elements of the diagram in the filter panel and the initial display of the diagram in the render panel.

From the drop-down list of groups, select the group "Reinforced concrete floors".

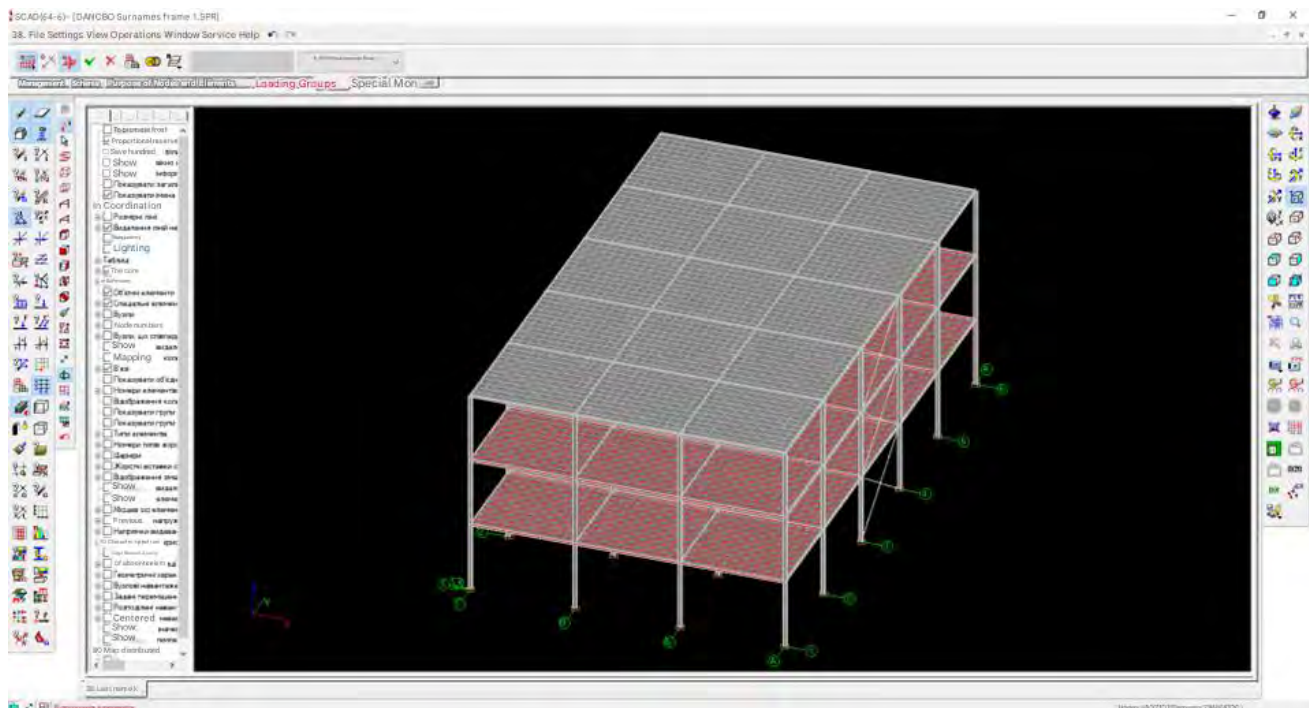


Figure 3.61 – Activated group "Reinforced concrete floors" on the design diagram

Click the "Assignment" tab and activate the "Set Plate Element Reinforcement" tool.

Set the following reinforcement parameters:

- lower fittings $\varnothing 12$ cells 200×200 ;
- upper reinforcement $\varnothing 12$ cells 200×200 ;
- transverse reinforcement $\varnothing 10$ step 400×400 .

Click OK. Then press the confirmation button  to apply reinforcement to the selected floors.

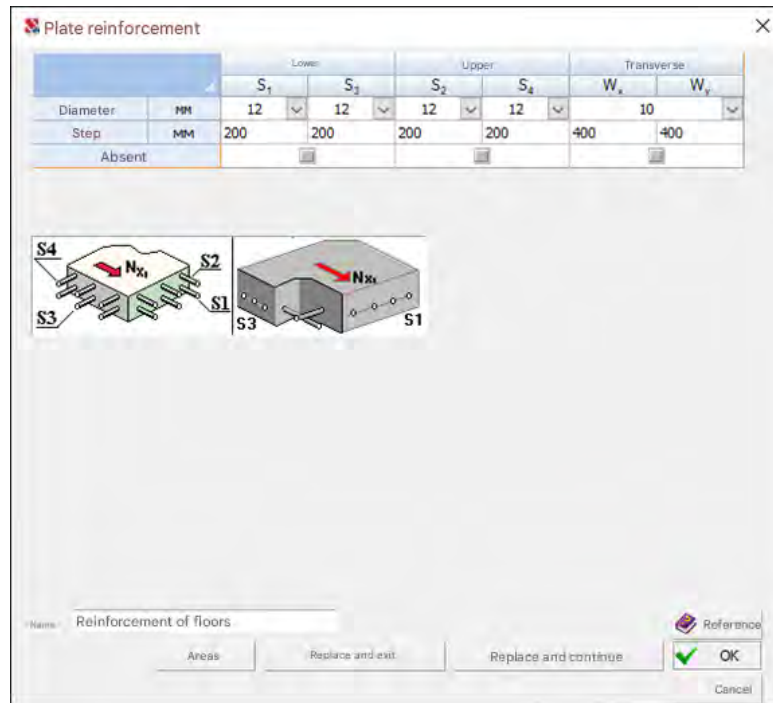


Figure 3.62 – Purpose of reinforcement for reinforced concrete floor slabs

Select the "Reinforced concrete pavement" group from the drop-down list.

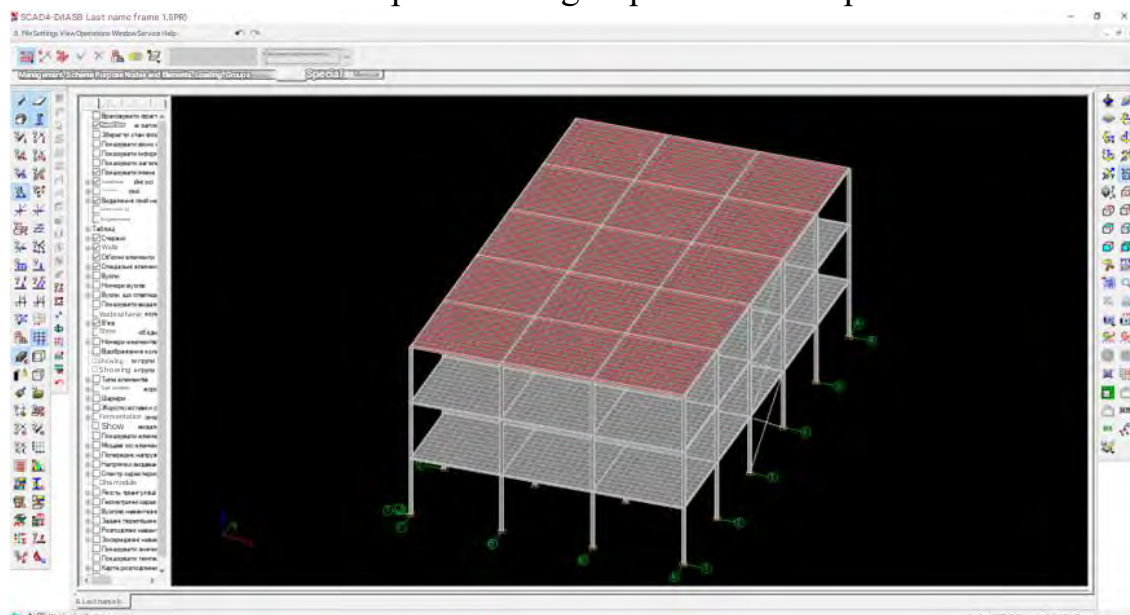


Figure 3.63 – Activated group "Reinforced concrete pavement" on the design diagram

Similar to the reinforcement of floors, specify the reinforcement for the reinforced concrete slab of the coating. Click OK. Then the confirmation button.

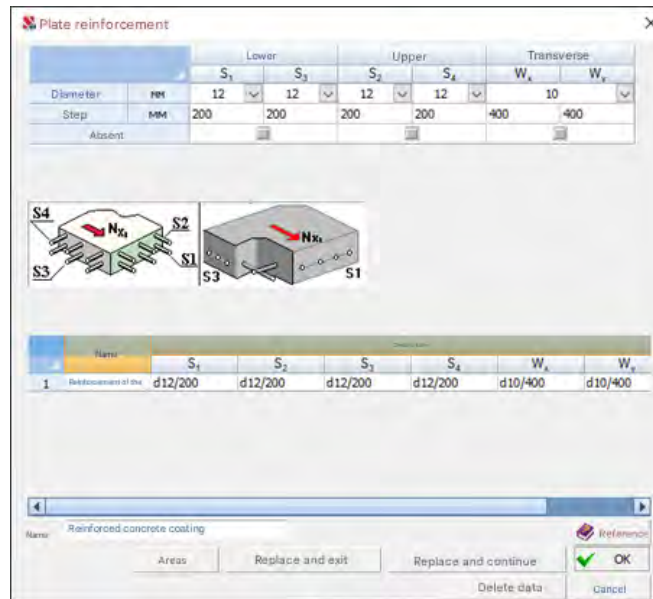


Figure 3.64 – Purpose of reinforcement for reinforced concrete coating slab

Loading the calculation scheme

To perform loads of the design scheme, we use DBN B.1.2-2:2006 Loads and Impacts, or the WEST program, which implements the provisions of the specified state building codes. Next, we consider the use of the WEST program, which greatly simplifies the work with the provisions of the relevant building codes.

The first load that should be applied to the design scheme is the load from the dead weight of the simulated elements of the building frame in the SCAD ++ program. Go to the "Loading" tab and activate the  "Dead weight" tool. The coefficient of inclusion of the dead weight in the dialog box is set according to DBN B.1.2-2:2006 or according to the corresponding page γ_f of the WEST program.



Figure 3.65 – Dead weight dialog box

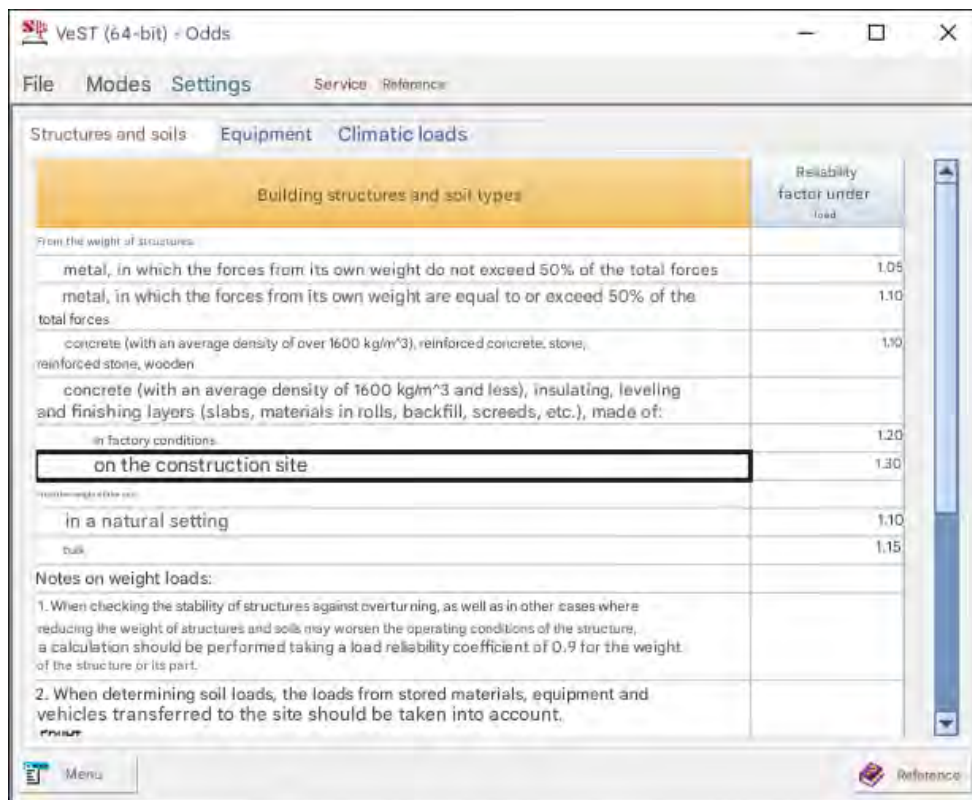


Figure 3.66 – Page coefficients of the WEST program

Enable the display of applied loads on the filter panel to control the correctness of the application of loads.

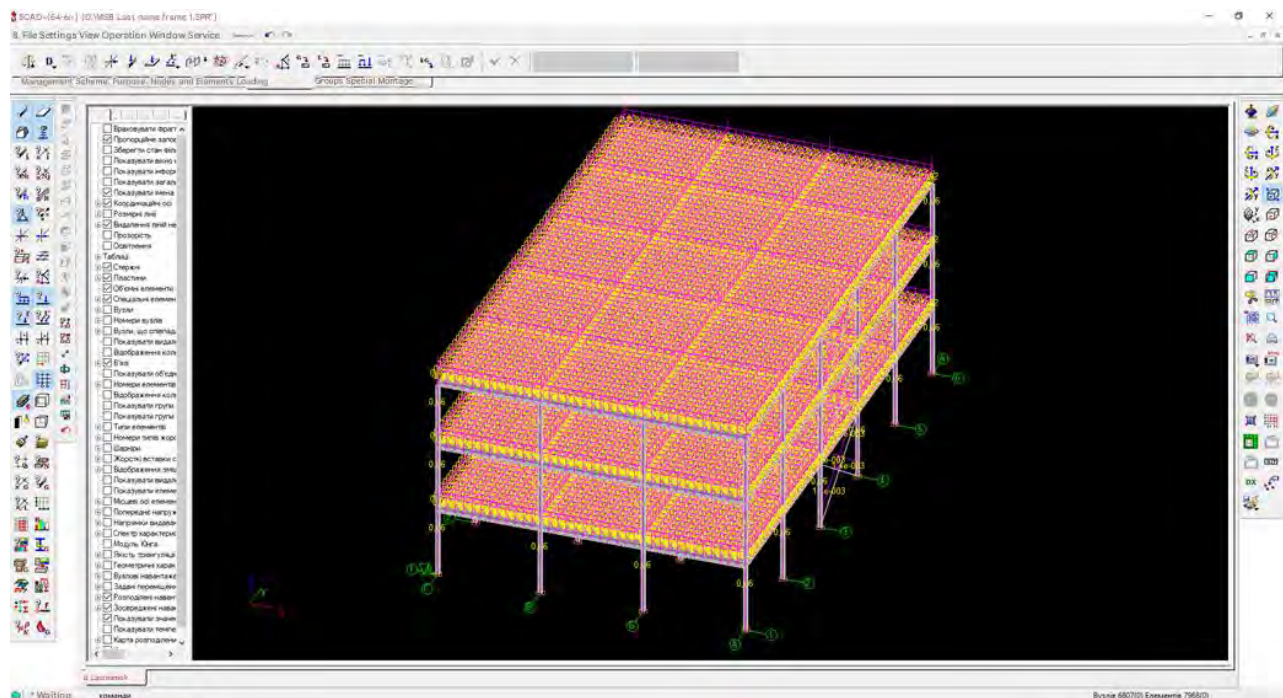


Figure 3.67 – Display of dead weight loads on the design diagram

To save the applied load, press the corresponding button  on the toolbar.

In the load saving dialog box, assign a name to the load "Dead weight", and select the load type and load type from the drop-down lists. After that, click "Save as new" and the OK button.

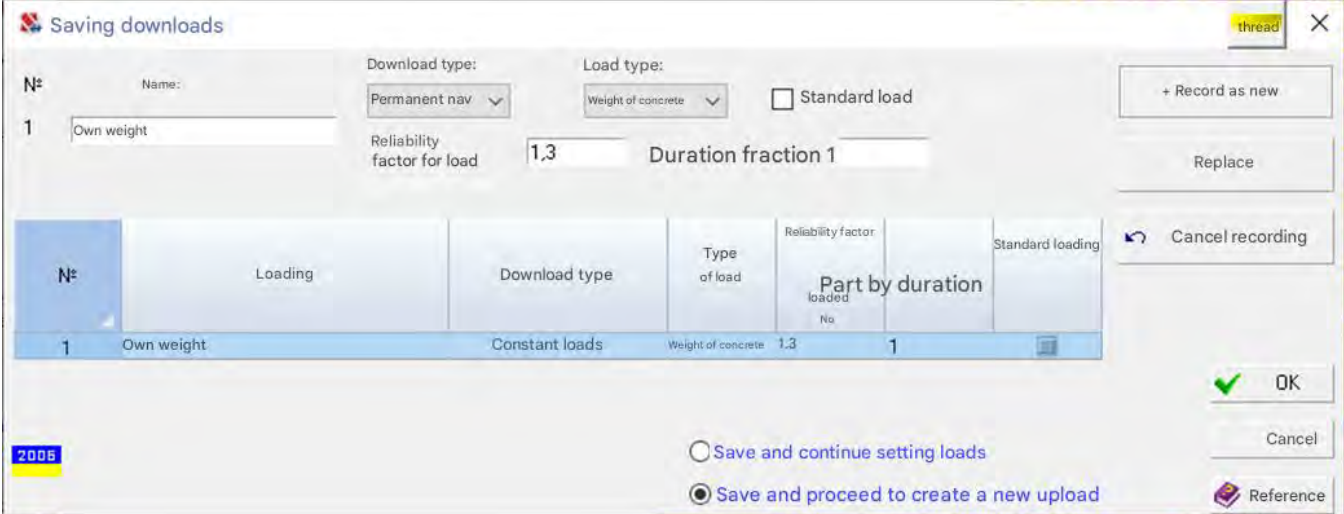


Figure 3.68 – Saving the load "Dead weight" in the "Saving loads" dialog box

The next load is constant from the structure of floors and coverings without taking into account the already applied own weight of structures. To calculate the constant load, we will use the WEST program. Activate the page  in the WEST program. This page allows you to collect constant loads per 1 m² of the horizontal plane of the floor and/or covering of the building.

To calculate the maximum loads from the materials of the floor structure of the floor of the building, we take the following initial data:

- floor material – linoleum 5 mm;
- cement-sand leveling screed – 50 mm;
- waterproofing – PVC film.

This list is not exhaustive and is used in this educational task. When designing real objects, the floor structure is set by the architect of the project. Select the accepted floor construction materials on the "Dead weight" page of the WEST program. Adjust the thickness of the materials in accordance with the values taken above and click the "Calculate" button.

The resulting limit value based on the results of calculating the dead weight of the floor structure of the floors is applied to the simulated floors in the SCAD ++ program.

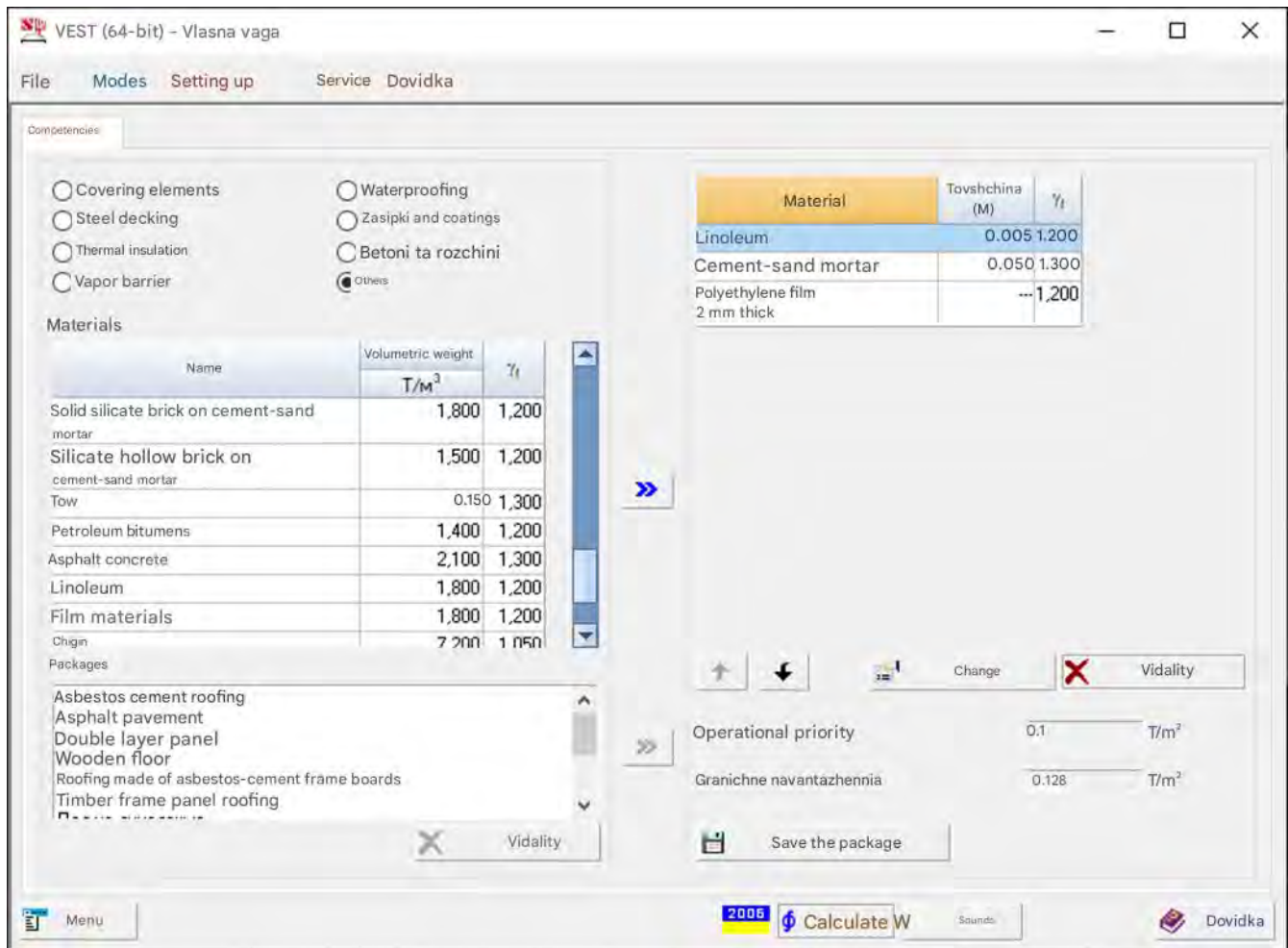


Figure 3.69 – Results of calculation of constant loads from the floor structure in the WEST program

Let's move on to the workspace of the SCAD ++ application. To apply the calculated load in the WEST program, use the "Plate loads" tool on the "Loading" tab.

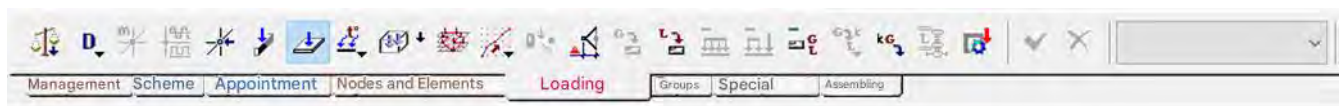


Figure 3.70 – Activating the "Plate Load" tool on the "Loading" tab

To select the floors on which you want to apply the load, you can use the XOY Plane Section tool in the "Visualization Panel" or select the previously created group "Reinforced Concrete Floors" in the drop-down list on the "Groups" tab

Apply the load calculated in the WEST program to all floors of the building. Select the type of distributed load, and enter its value in the "Load Value" window

according to the previous calculation. Click OK in the dialog box, and click the Confirm button on the toolbar.

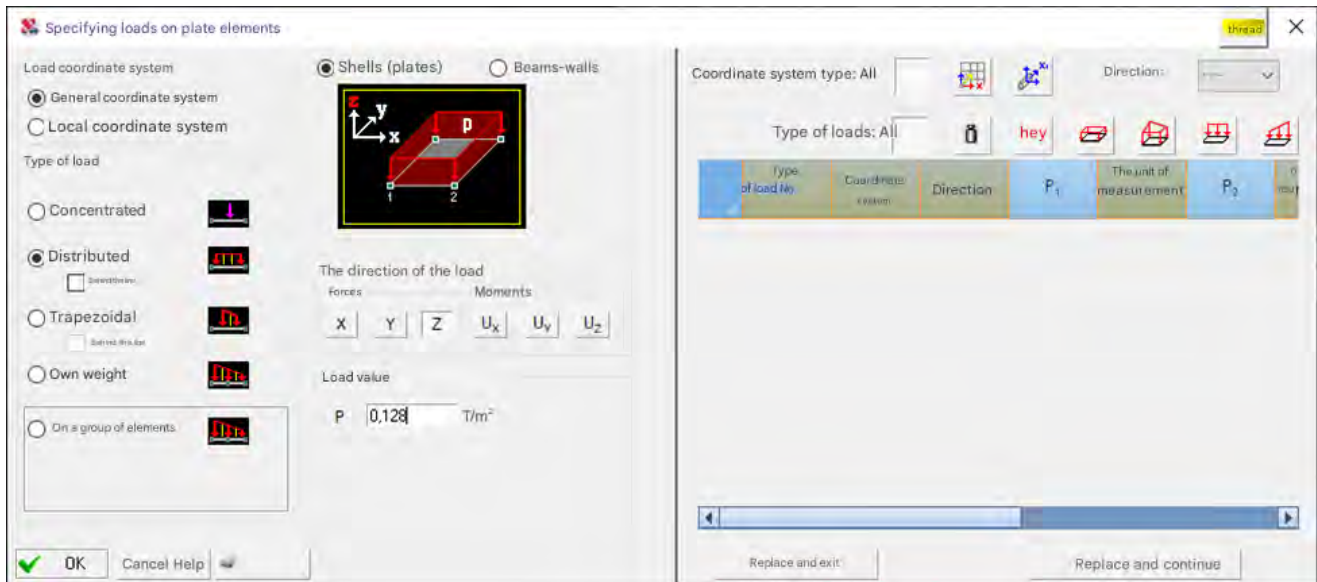


Figure 3.71 – Plate Load Dialog Box

Using the filter panel, make sure that the loads are applied correctly.

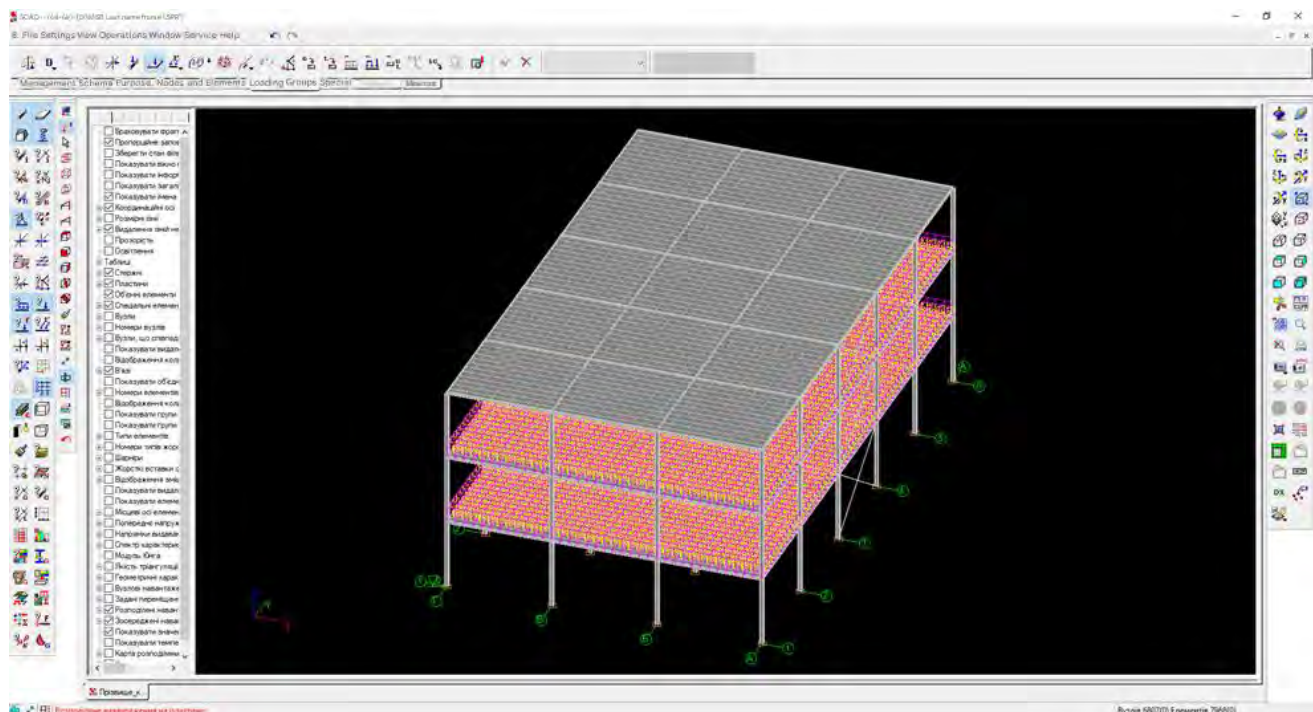


Figure 3.72 – Display of constant loads on the floor

To calculate the maximum loads from the materials of the roof structure of the building, we take the following initial data:

- roofing material with coarse-grained sprinkles;
- roofing material lining;
- cement-sand screed – 50mm;
- waterproofing – PVC film;
- thermal insulation – mineral wool plates – 300 mm;
- vapor barrier – PVC film.

Enter the initial data into the dialog box on the "Dead weight" page of the WEST program. Click the calculate button.

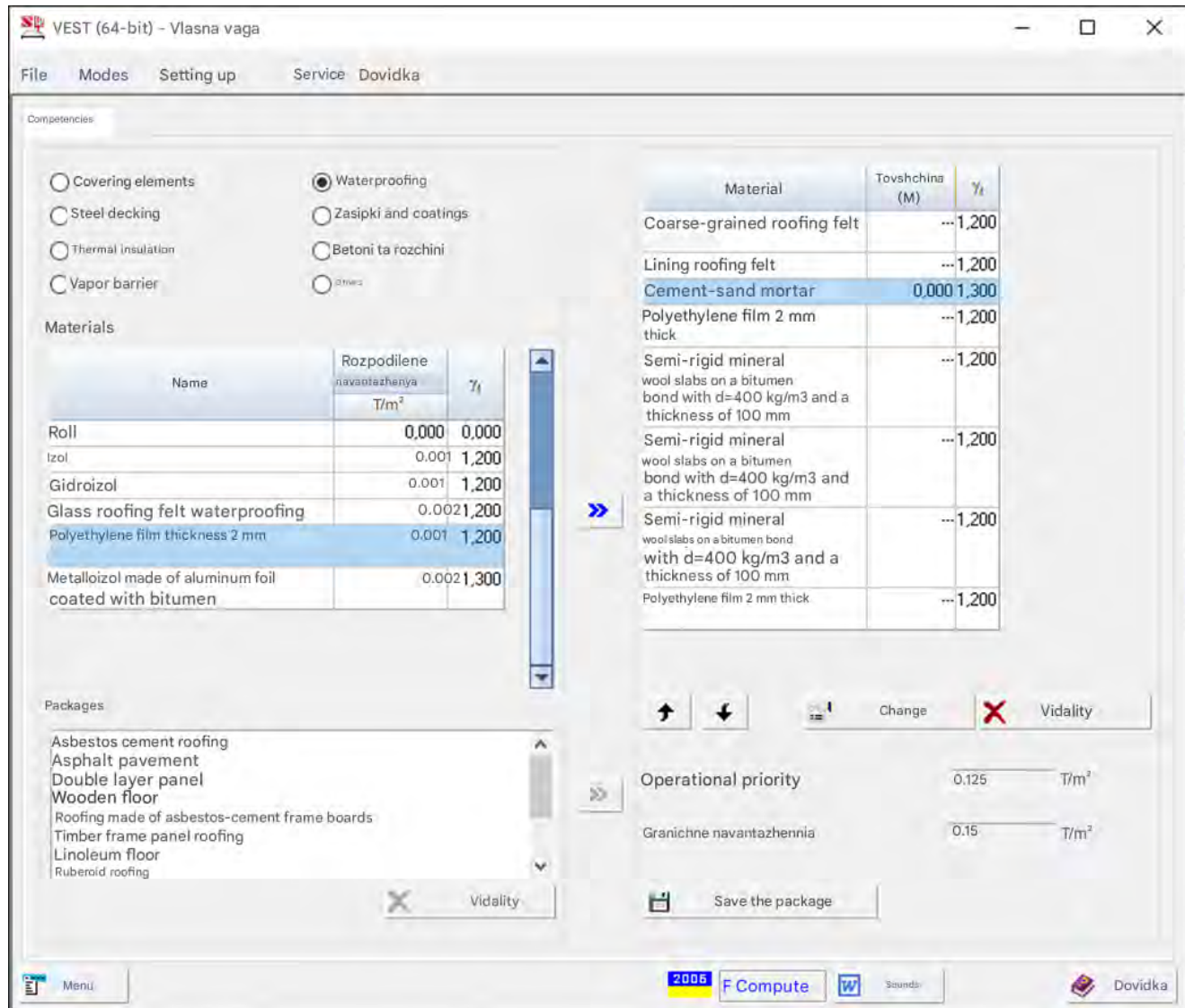


Figure 3.73 – Results of calculation of constant loads from the roof structure in the WEST program

To select a coating for applying a load, you can use the XOY Plane Intersection tool in the Render Panel or select the previously created Reinforced Concrete Pavement group from the drop-down list on the Groups tab.

Apply the load calculated in the WEST program to the building coverage. Select the type of distributed load, and enter its value in the "Load Value" window according to the previous calculation. Click OK in the dialog box, and click the Confirm button on the toolbar.

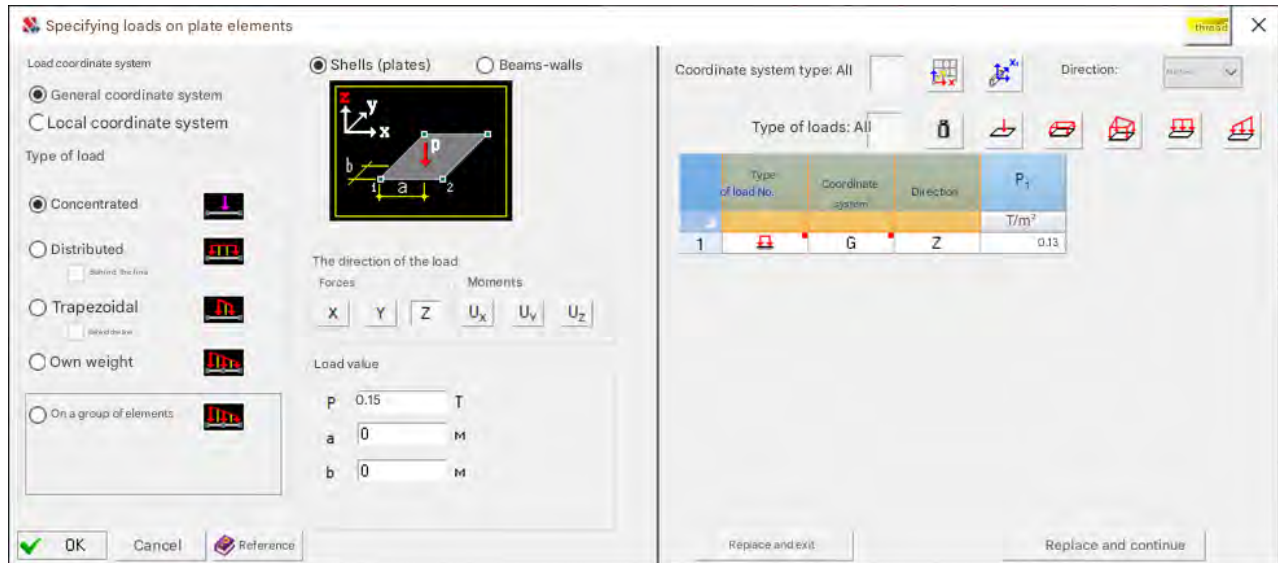


Figure 3.74 – Plate Load Dialog Box

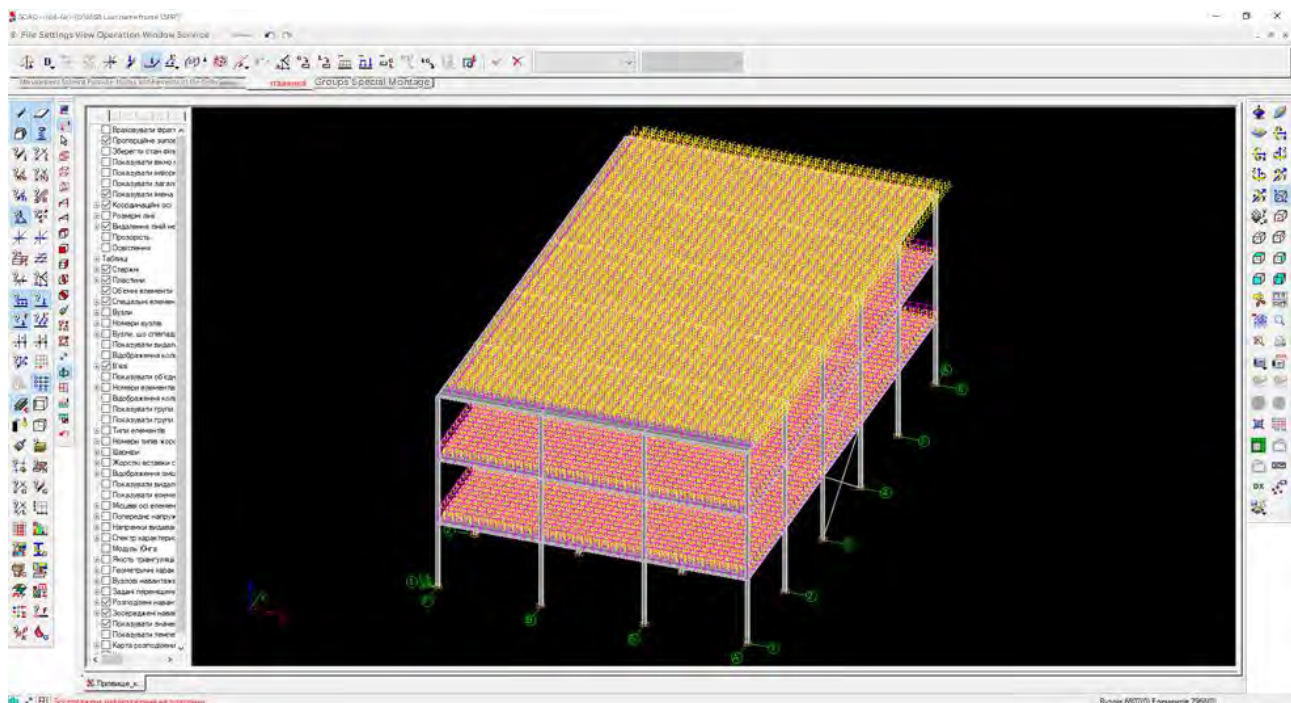


Figure 3.75 – Display of constant loads on floors and coatings

To save the applied load, press the corresponding button  on the toolbar.

In the load saving dialog box, assign a name to the load "Dead weight", and select the load type and load type from the drop-down lists. After that, click "Save as new" and the OK button.

Nº	Loading	Download type	Type of load	Reliability factor	Standard loading
1	Own weight	Constant loads	Weight of concrete 1.3	1	
2	Constant load	Constant loads	Weight of concrete 1.3	1	

Figure 3.76 – Saving the load "Constant load" in the dialog box "Saving loads"

The next load is the snow load. To calculate it, we will also use the WEST program. Go to the tab  in the WEST program. In the dialog box of the "Snow" tab, enter the necessary initial data in accordance with the design task and click the "Calculate" button.

City	Characteristic value of snow load (Pa)
Ternopil	1390
Kharkiv region	
Kharkiv	1600
Luhansk	1460
Kupiansk	1460
Luhansk	1490
Luhansk	1570
May Day	1510
Cherkiv	1600
Kherson region	
Kherson	760
Kakhovka	840
Novo Kakhovka	820
Khmelnytskyi region	
Chernivtsy	1340
Kamianets-Podilskyi	1270
Perishin	1330
Slavuta	1350
Cherkassy	1370

Figure 3.77 – Input of initial data for the calculation of snow load in the WEST program

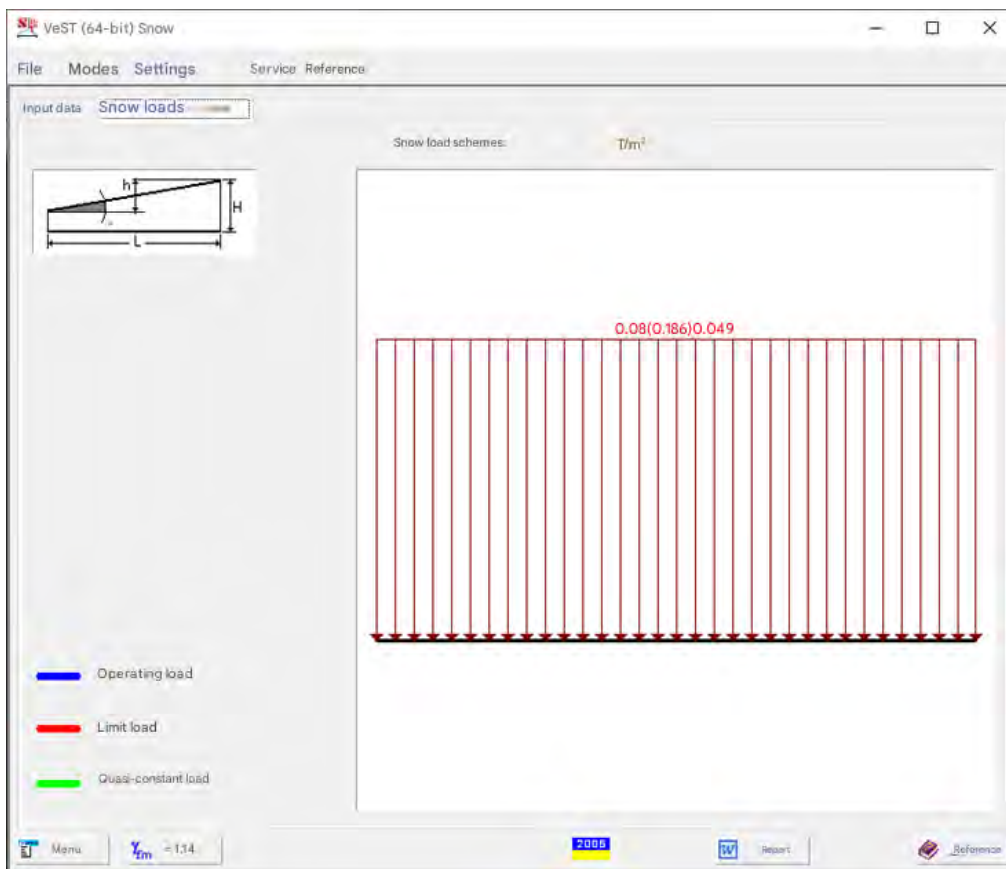


Figure 3.78 – Results of calculation of snow loads in the WEST program

If it is necessary to select a snow area (there is no settlement in the table), it can be found on the  WEST program tab. This tab helps to determine the zoning of the area in accordance with DBN B.1.2-2:2006 for further use in calculations.

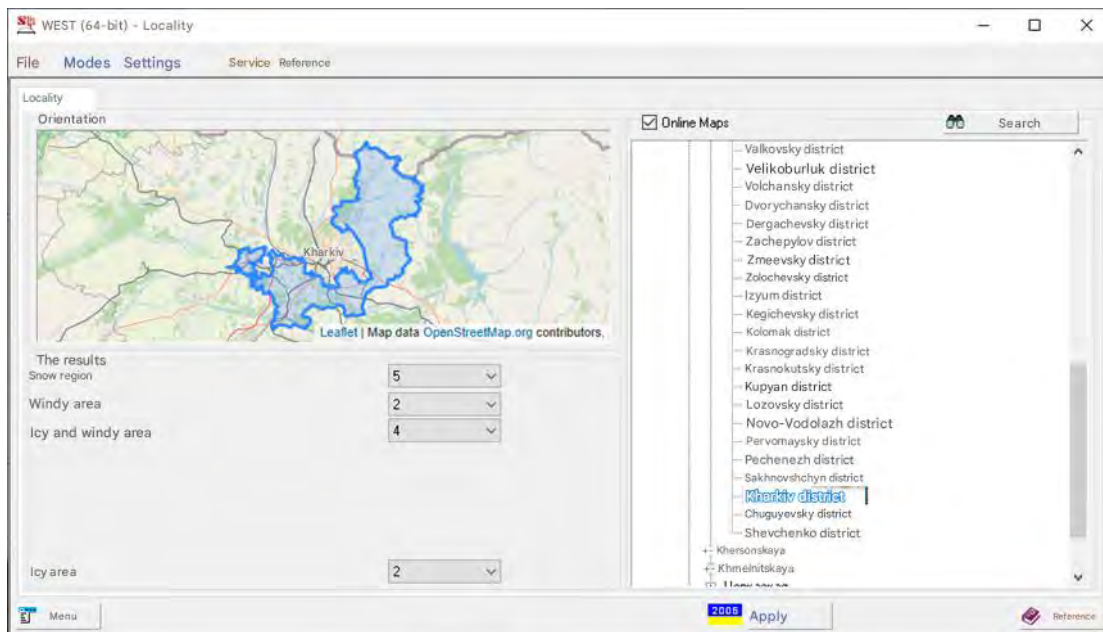


Figure 3.79 – Determining the zoning of the area using the WEST program

The calculated limit value of the snow load is applied to the roof of the building in the same way as the previous loads. Select the "Plate load" tool and apply the load obtained in the WEST program to the coating of the design scheme of the building.

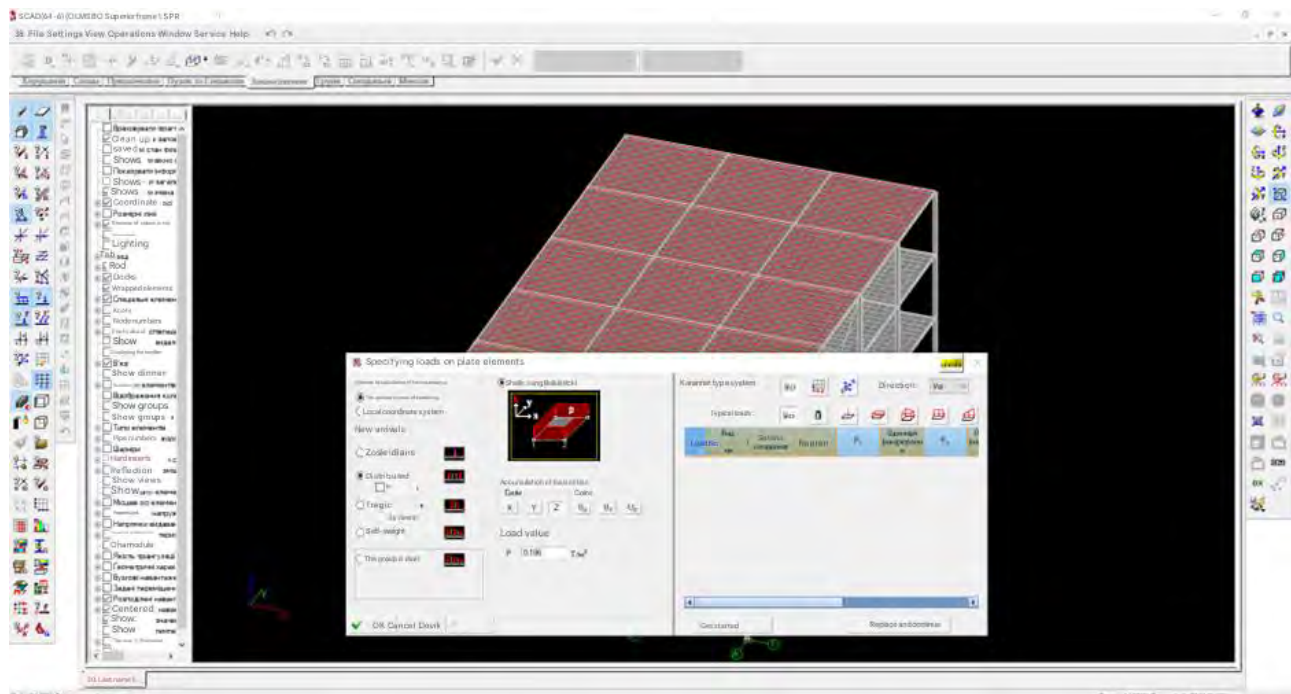


Figure 3.80 – Applying a snow load to the roof of the building

Press the OK button, then the confirm button (after selecting all the finite coating elements). We make sure that the snow load is applied correctly.

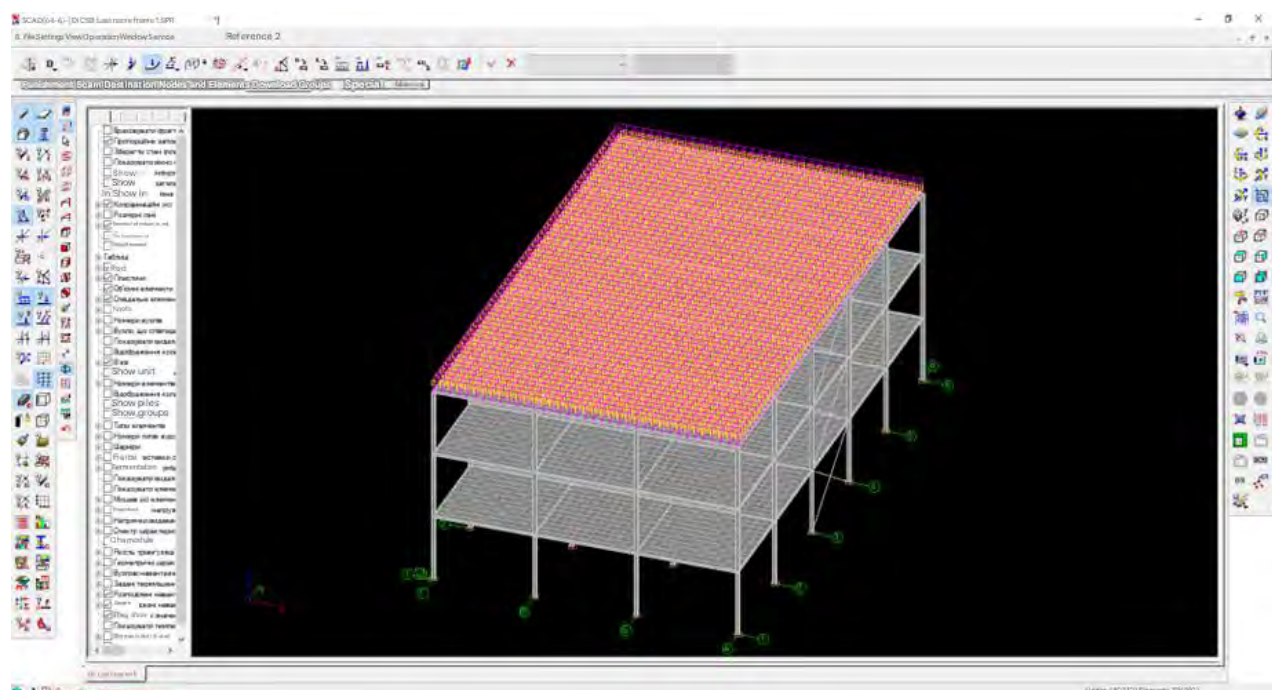


Figure 3.81 – Display of the snow load on the pavement

Save the load. Set the load name to "Snow", Load type – short-term. Type of load – full snow loads. Reliability factor – corresponds to the one calculated in the WEST program (1.14 for 100 years of the average recurrence period). The share of the duration is calculated as the ratio of "Operating load" to "Maximum load" received from the WEST program. Click the "Save as new" button, then the OK button.

Nº	Loading	Download type	Load Type	Reliability factor for load γ_{lim}	Duration fraction	Standard loading
1	Own weight	Constant loads	Weight of concrete	1.3	1	
2	Constant load	Constant loads	Weight of concrete	1.3	1	

Figure 3.82 – Saving the load "Snow" in the "Saving loads" dialog box

The next load is the payload on the floor. To set the maximum payload on the floor in accordance with the current building codes, we will refer again to the WEST program. In the dialog box , select pt. 2 which corresponds to the individual task and click the Calculate button.

Temporary loads

Building and imperfections

1. Apartments in residential buildings; bedrooms in preschools and boarding schools; living quarters in rest homes and boarding houses, hostels and hotels; wards in hospitals and sanatoriums; terraces
2. Office premises of administrative, engineering, technical, and scientific personnel of organizations and institutions; classrooms of educational institutions; household premises (changing rooms; showers; washrooms; toilets) of industrial enterprises and public buildings and structures
3. Offices and laboratories of healthcare institutions; laboratories of educational and scientific institutions; computer rooms; kitchens of public buildings; technical floors; basements
4. Halls:
 - a) meeting rooms
- 6) lunch (in cafes, restaurants, canteens)
- c) gatherings and gatherings of catch and contest sports

Load: operational (selected) 0.2 T/m²

quasi-permanent 0.085 T/m²

Cargo space Total number of overlaps M²

Result

Operating load 0.2 T/m² Limit load 0.24 T/m²

Quasi-constant load 0.085 T/m²

Figure 3.83 – Calculation of payloads in the WEST program

The calculated payload limit value is applied to the ceilings of the building in the same way as the previous loads. Select the "Plate load" tool and apply the load obtained in the WEST program to the ceilings of the design scheme of the building.

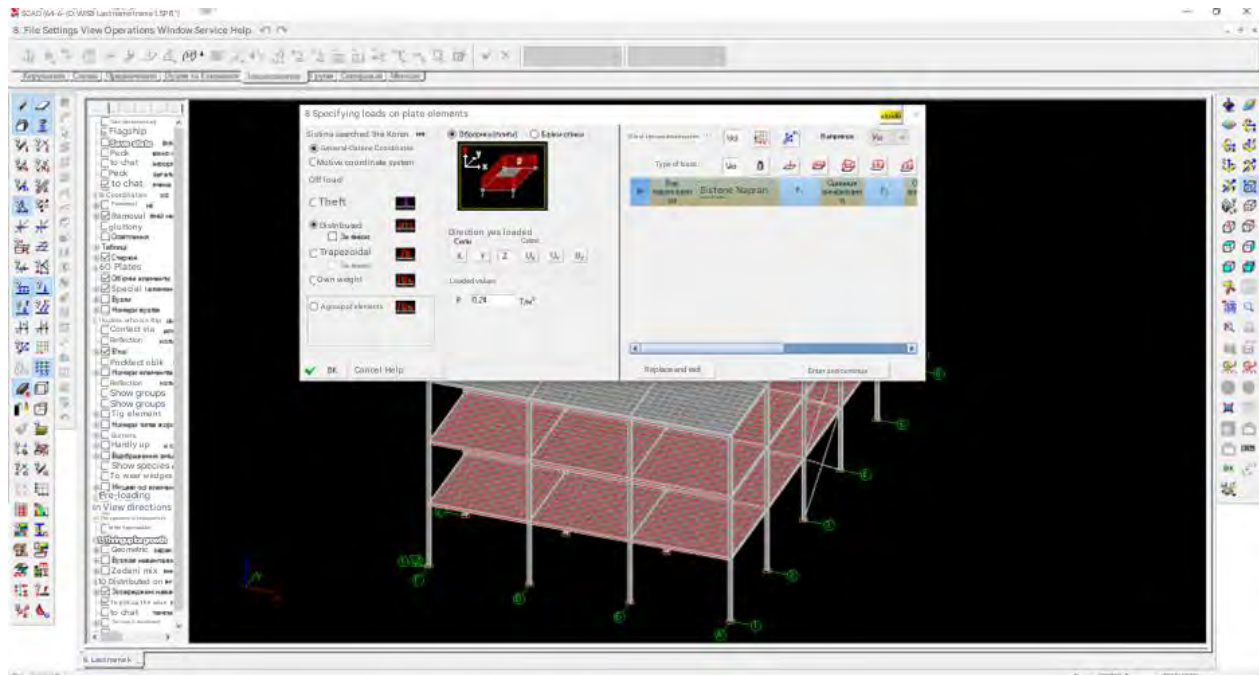


Figure 3.84 – Applying a payload to the ceilings of a building

Click the OK button, then the confirm button (after selecting all the finite elements of the overlap). We make sure that the payload is applied correctly.

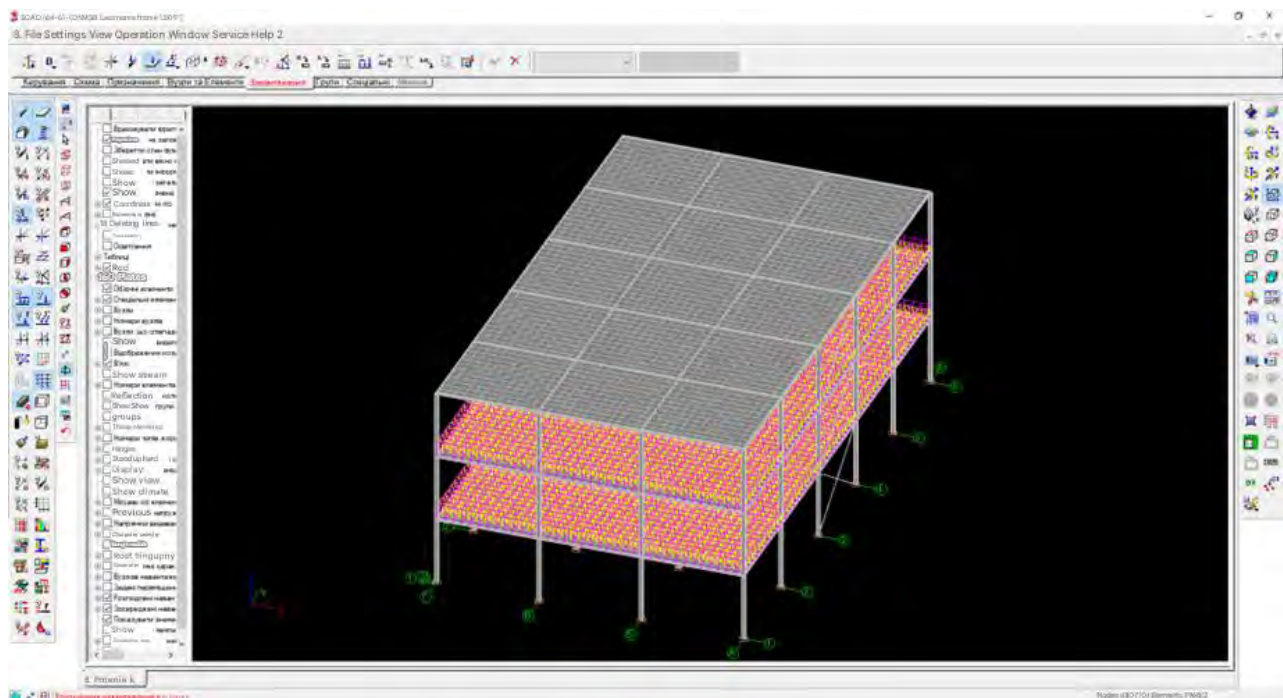


Figure 3.85 – Display of the payload on the ceiling

We save the load using the algorithm discussed earlier. We use the name – payload. The type of load is short-term. The type of load is "Full loads... with a standard value greater than 2.0 kPa". The fraction of the duration is set as the ratio of the "Operating load" to the "Maximum load" obtained from the WEST program. Click the "Save as new" button, then the OK button.

Nº	Loading	Download type	Type of load	Reliability factor for load Nyam	Duration fraction	Norm download
2	Constant load	Constant loads	Weight of concrete	1.3	1	
3	Snow	Short-term loads	Full of snow	1.14	0.43	
4	Payload	Short-term loads	Full navan	1.2	0.35	

Figure 3.86 – Saving the Payload load in the Save Loads dialog box

The application of wind loads has certain peculiarities compared to previous loadings. Firstly, it should be kept in mind that wind load can act from any side of the building, but only from one side at a time. Secondly, considering the aerodynamic features of the building, it can have specific values on the end faces of the building as well as on surfaces perpendicular or inclined to them. And thirdly, depending on the height of the building, the type of terrain, and the altitude above sea level, the values of the wind load can differ significantly from one another even within the same wind region. It should also be noted immediately that the wind load distribution diagram does not follow a linear law of distribution with the height of the building, so determining the values of wind loads and applying them to the calculation model is a rather complex, comprehensive task, which we will consider in detail during practical classroom sessions. In these methodological guidelines, the solution to this problem will be provided with certain simplifications (which will contribute to the margin of safety) while maintaining the correctness of the calculation algorithm and the application of wind loads to the calculation model. So, let's return to the VEST program and move to the tab .

Set the necessary parameters for the calculation of the wind load:

- choose the city of location of the construction object – Kharkiv;
- type of terrain – IV;
- altitude above sea level – 0,16 km
- load reliability coefficient – 1,14.

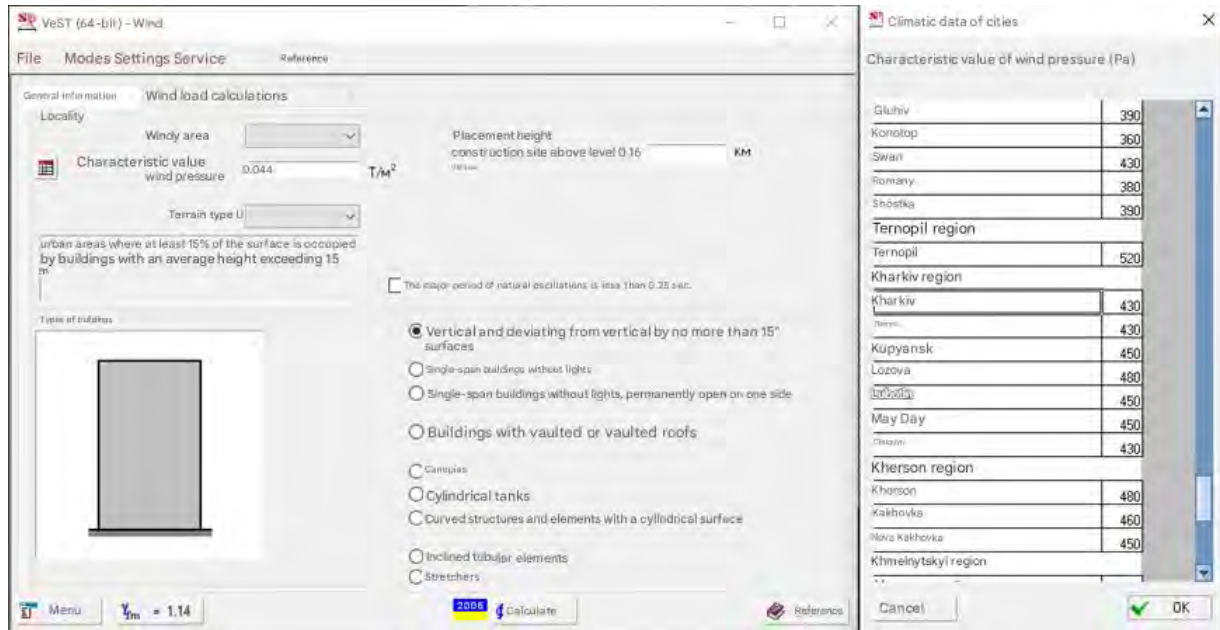


Figure 3.87 – Setting the parameters for calculating the wind load

After configuring all the parameters, press the button **2006** . In the next dialog box, enter the height of the building according to the individual task.

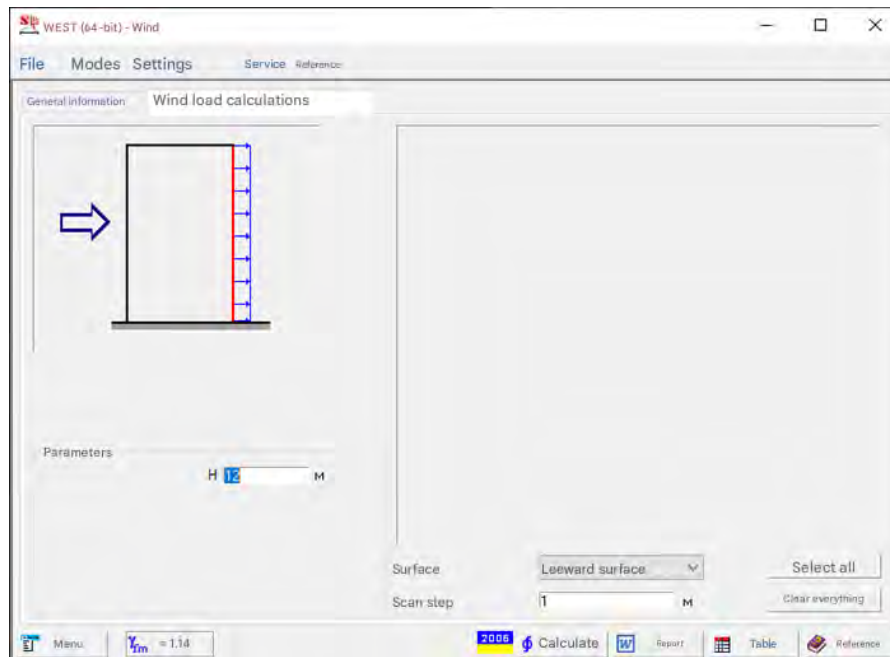


Figure 3.88 – Setting the parameters for calculating the wind load

After adjusting the parameters of the height of the building and the scanning step (in our case, 1 m is optimal), select the surface to calculate and press the button  .

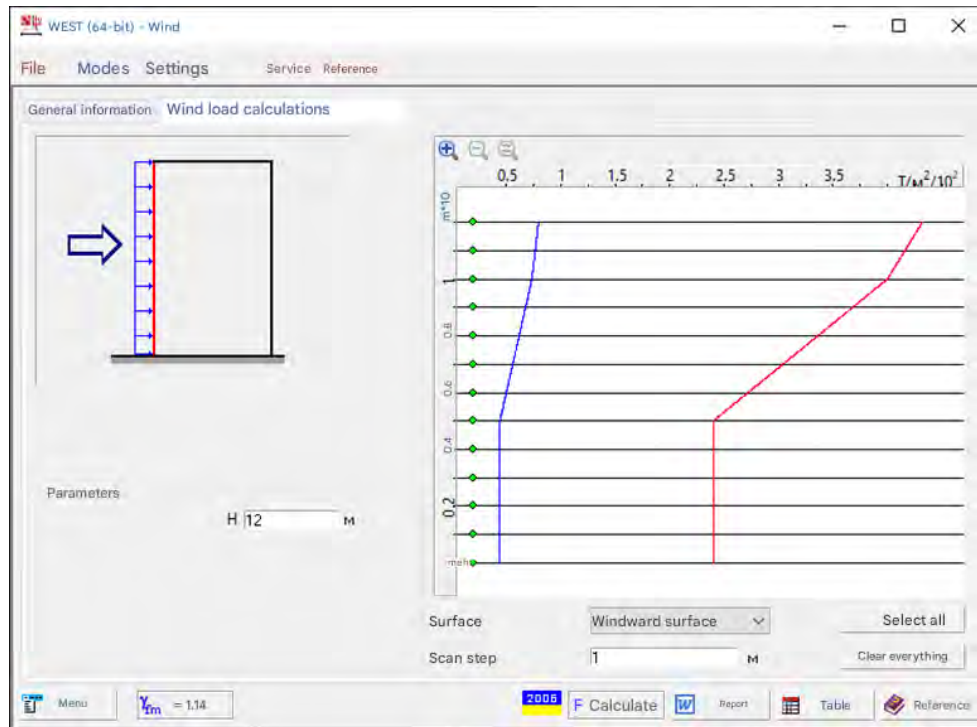


Figure 3.89 – Results of calculation of wind loads on the windward side

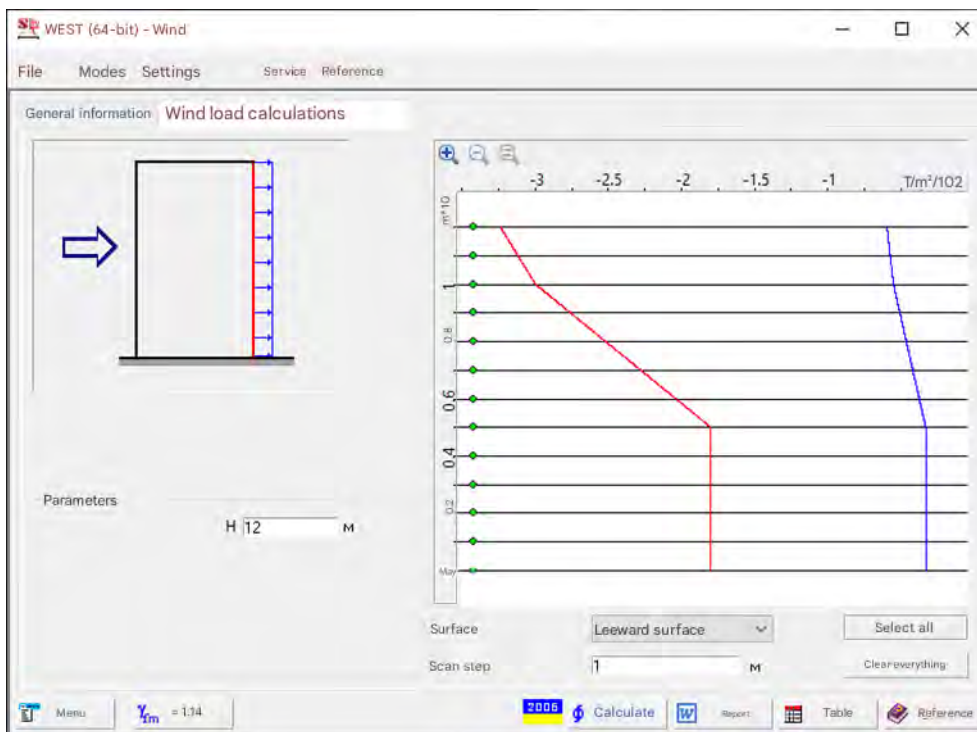


Figure 3.90 – Results of calculation of wind loads on the leeward side

To simplify the modeling and analysis of the design scheme of the building, we conditionally do not create a wall fencing of the building when building a design model in SCAD ++. Therefore, we apply wind loads directly to the columns of the building, converting the load per 1 m² of the wall fence area into a load per 1 m.p. of column length. To do this, we determine the product of the wind load per 1 m² at a certain elevation of the building to the loading area (which is equal to the pitch of the columns). The obtained values are applied to the columns using the  "rod load" tool.

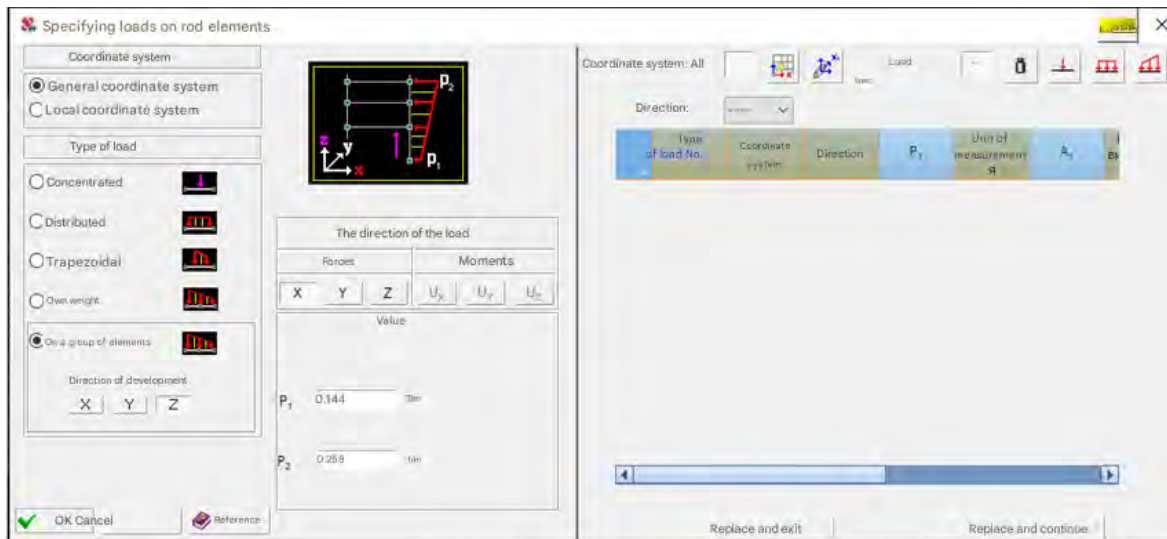


Figure 3.91 – Application of wind loads "Wind on the right" from the windward side (middle columns)

For the outermost columns, we use half of the load on the middle columns.

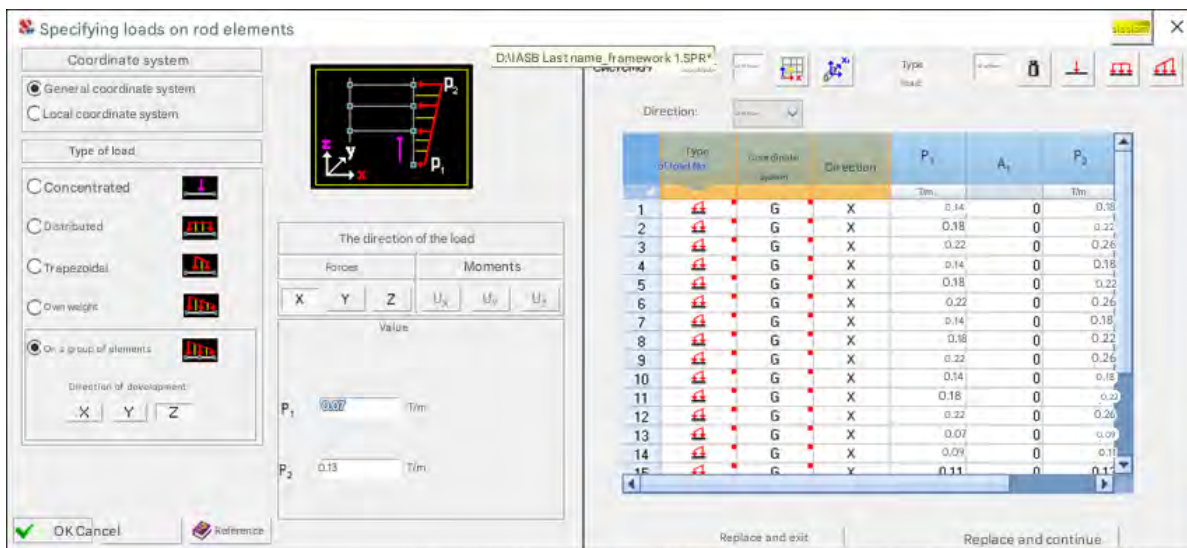


Figure 3.92 – Application of wind loads "Wind on the right" from the windward side (extreme columns)

From the load diagram for the leeward surface, we obtain the value of the wind load for the leeward side of the building.

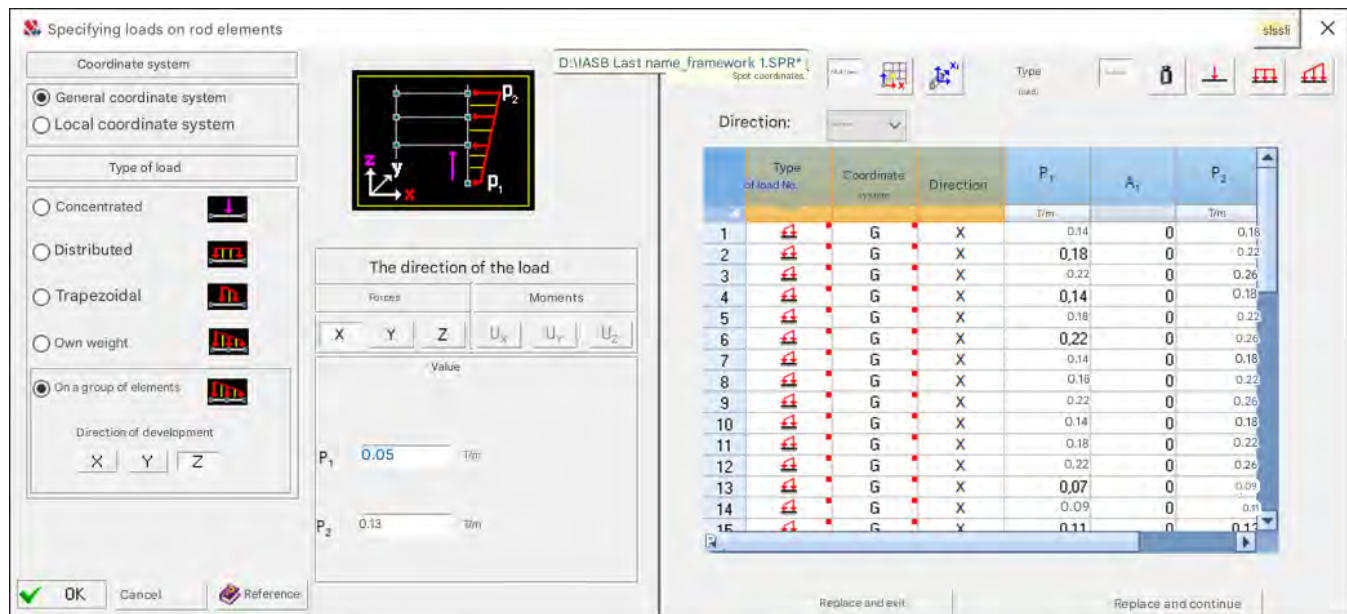


Figure 3.93 – Application of wind loads "Wind on the right" from the leeward side (middle columns)

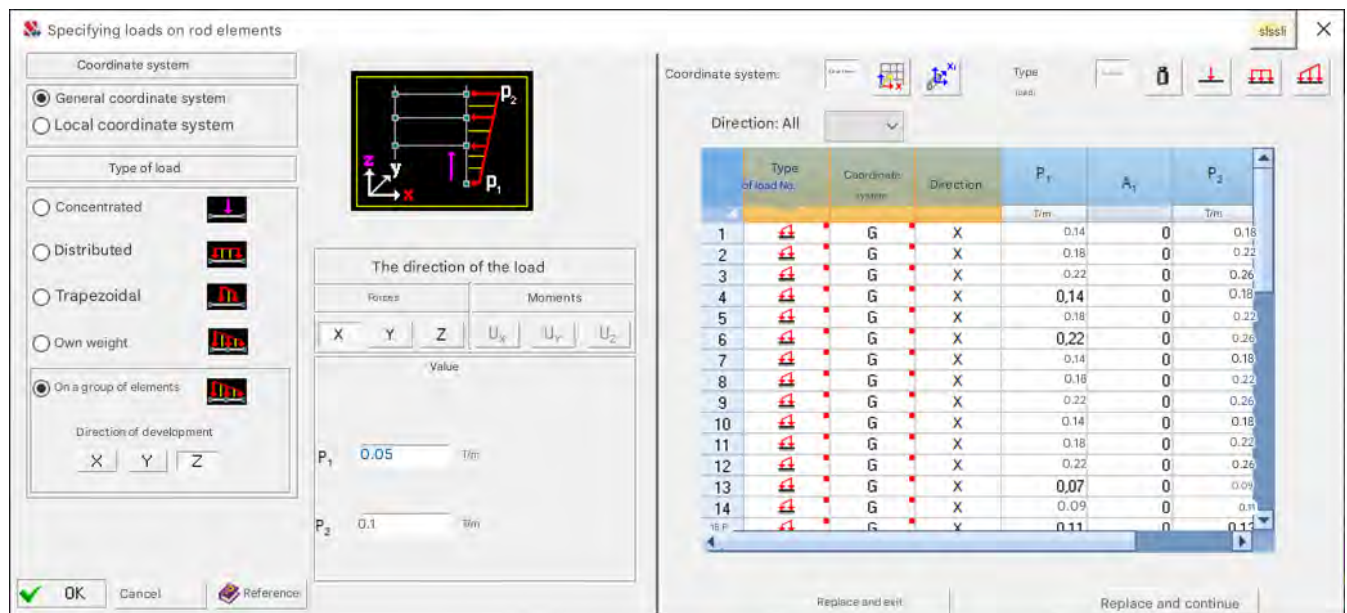


Figure 3.94 – Application of wind loads "Wind on the right" from the leeward side (extreme columns)

With the help of filter panel tools and visual analysis of the model, we make sure that the wind loads are set correctly.

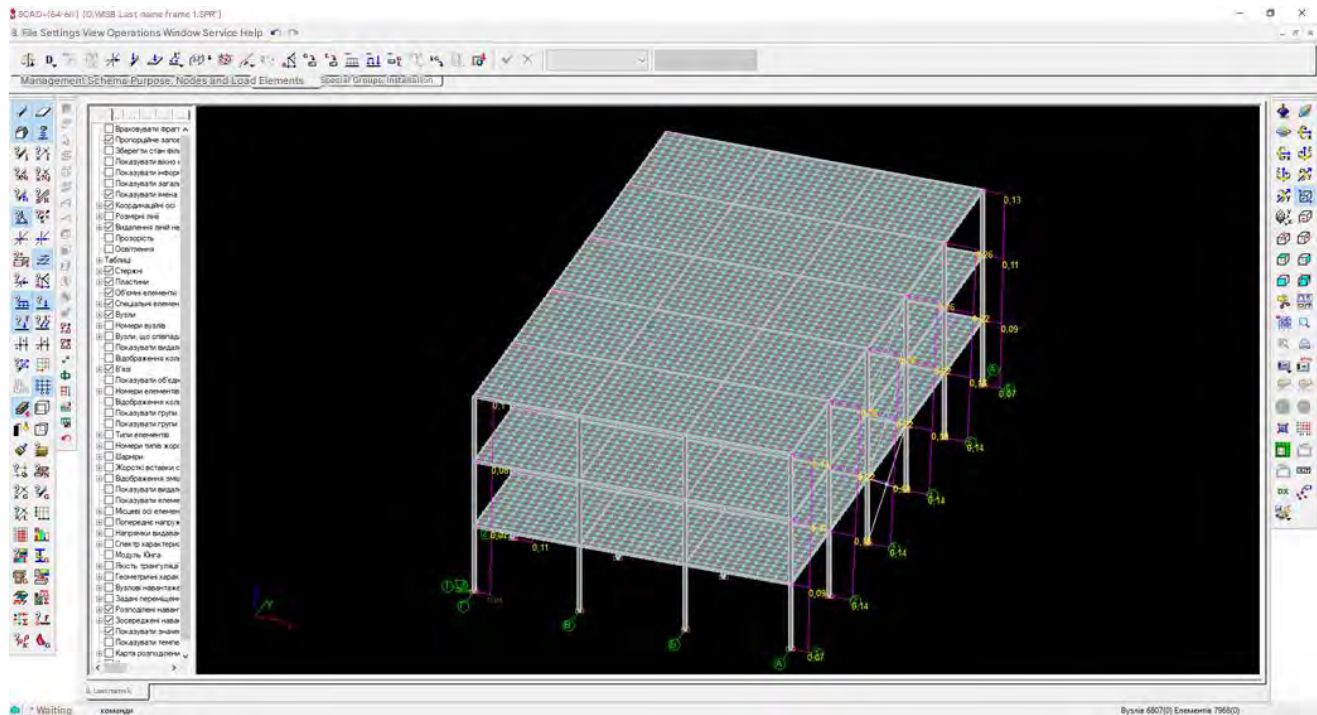


Figure 3.95 – Wind loads "Wind on the right" are applied to the design model of the building

We save the wind load called "Wind on the right" by choosing the appropriate type and type of load. The load reliability factor is set to 1.14, the fraction of the duration is left equal to 0.

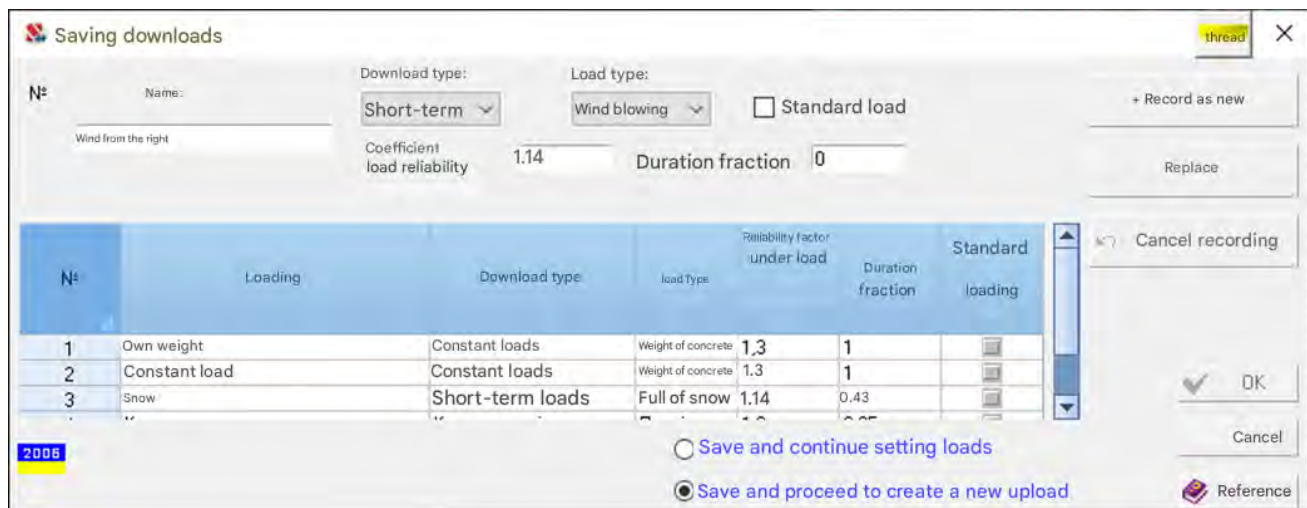


Figure 3.96 – Preservation of wind load "Wind on the right"

Click the "Save as new" button, then the OK button.

Similarly, we apply the wind loads "Wind on the left", "Wind behind" and "Wind in front", changing the direction of action and the sign of the applied loads and save them under the appropriate names.

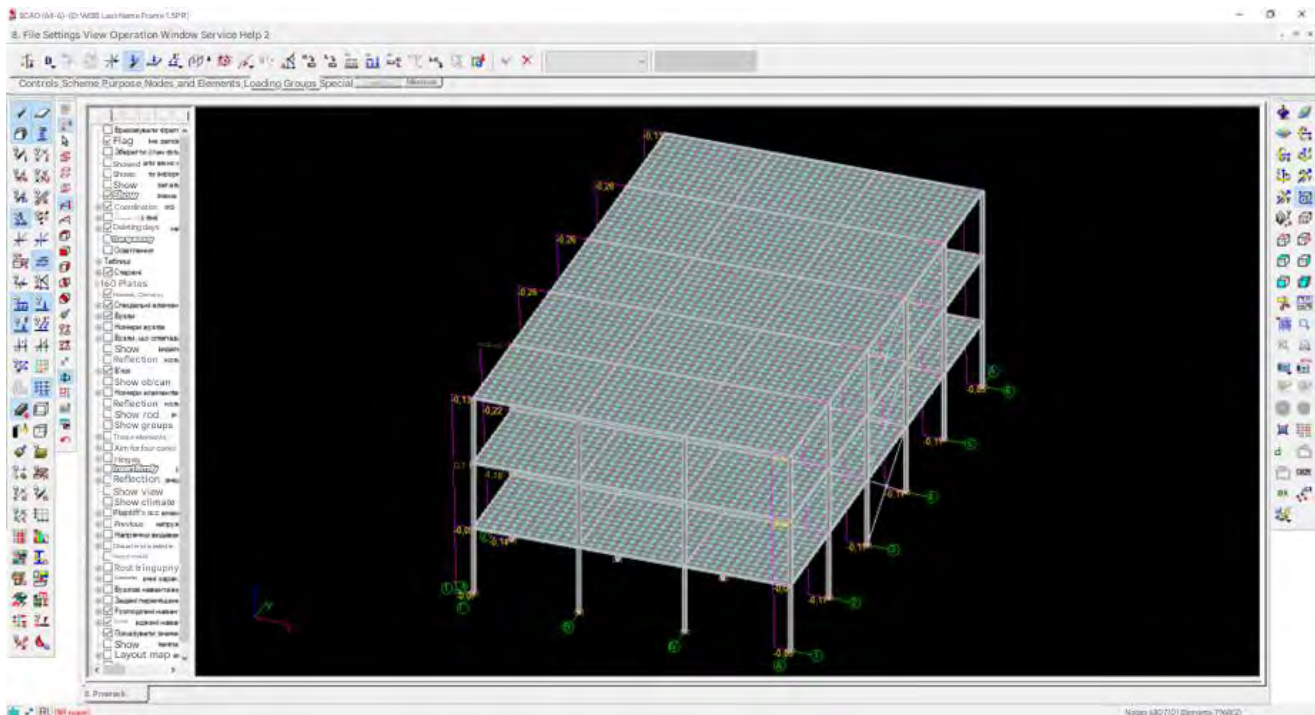


Figure 3.97 – Wind loads "Wind on the left" are applied to the design model of the building

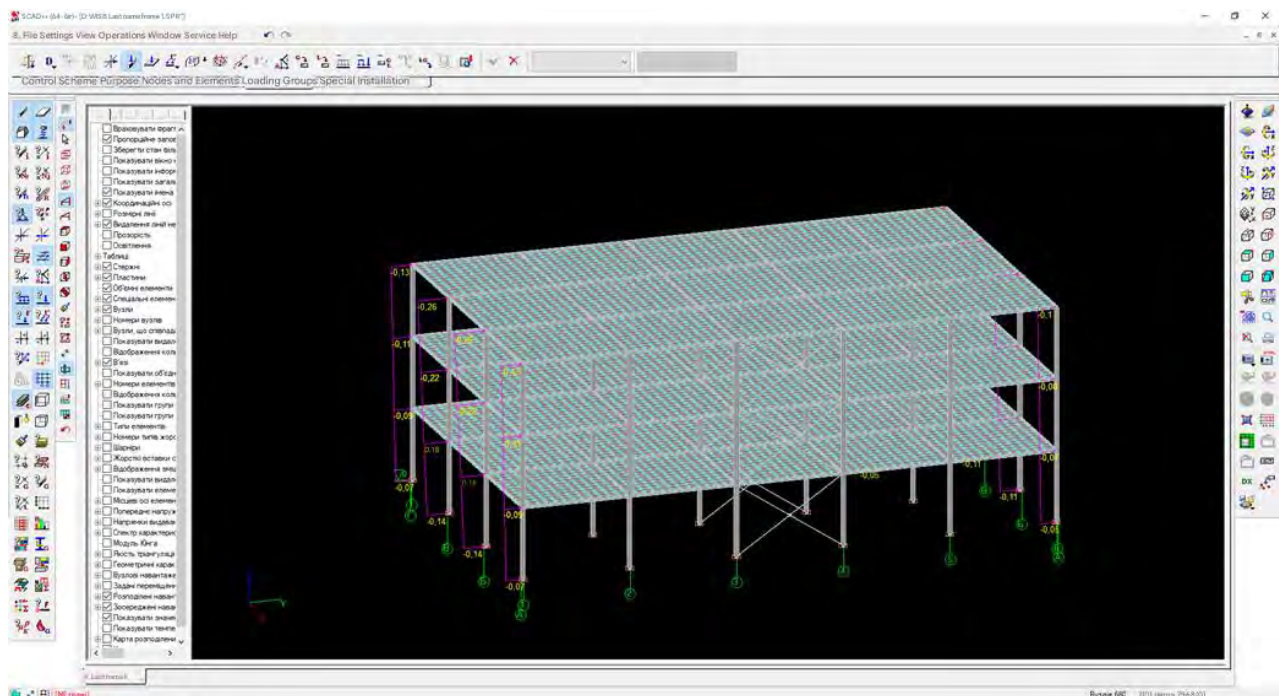


Figure 3.98 – Wind Ahead Wind Loads Applied to the Design Model of the Building

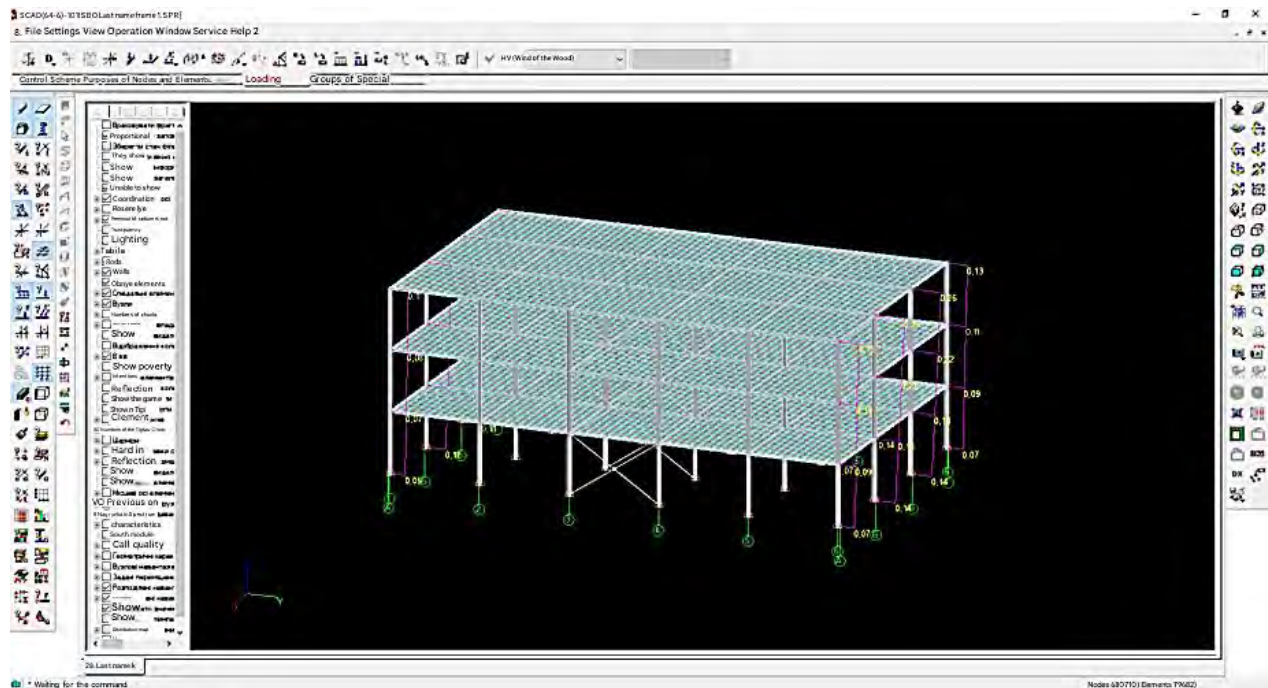


Figure 3.99 – Wind loads from behind are applied to the design model of the building

Tasks for independent work on the topic 3. On the assignment tab: using the stiffening assignment mode, change the type of frame core elements from metal to reinforced concrete with a cross section of 400x400mm, assign the material – concrete of class C20/25, assign a specified reinforcement for 4 rods Ø16A400s.

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".

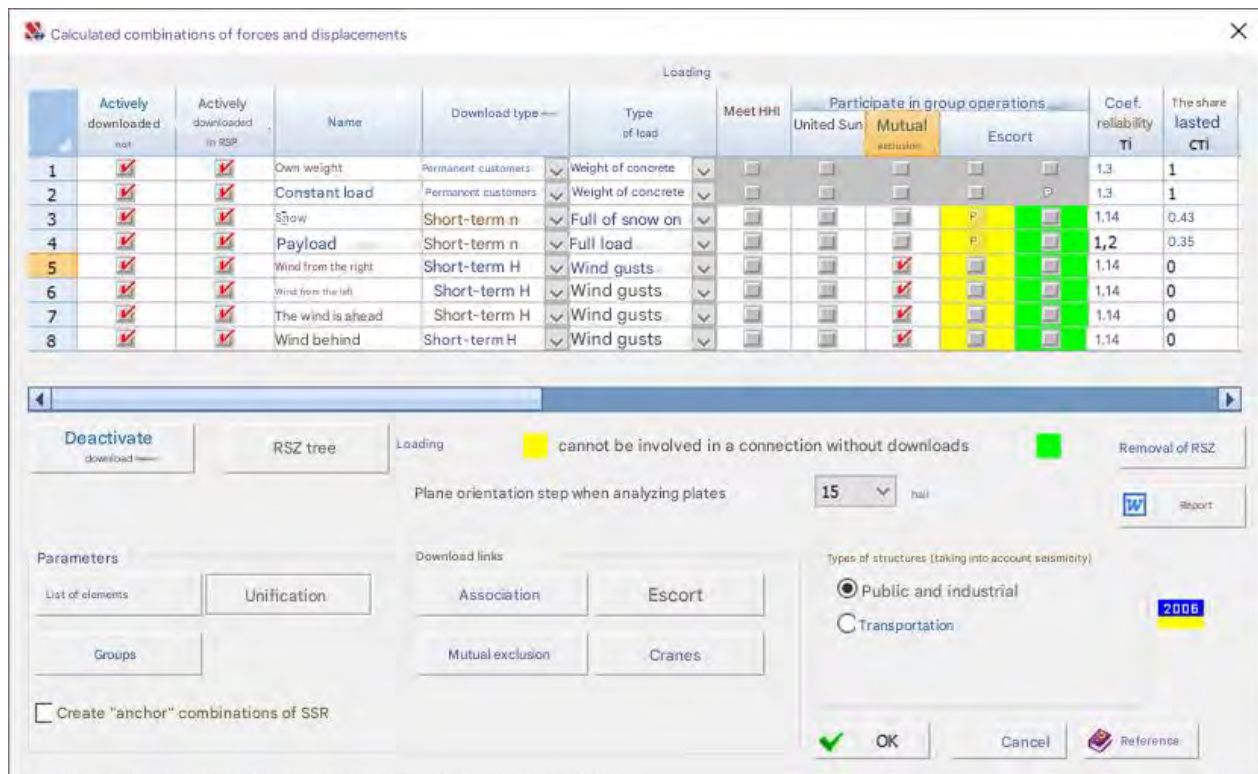


Figure 4.1 – Configuring the calculated force combinations in the corresponding dialog box

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.

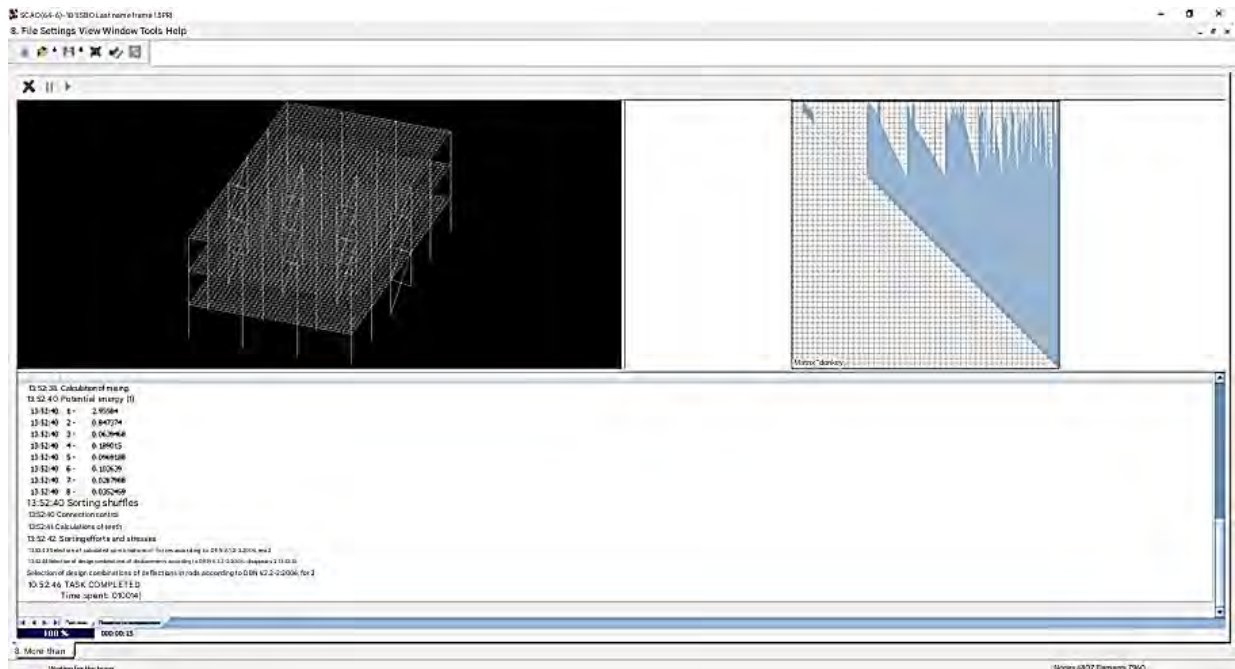


Figure 4.3 – Performing the calculation

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

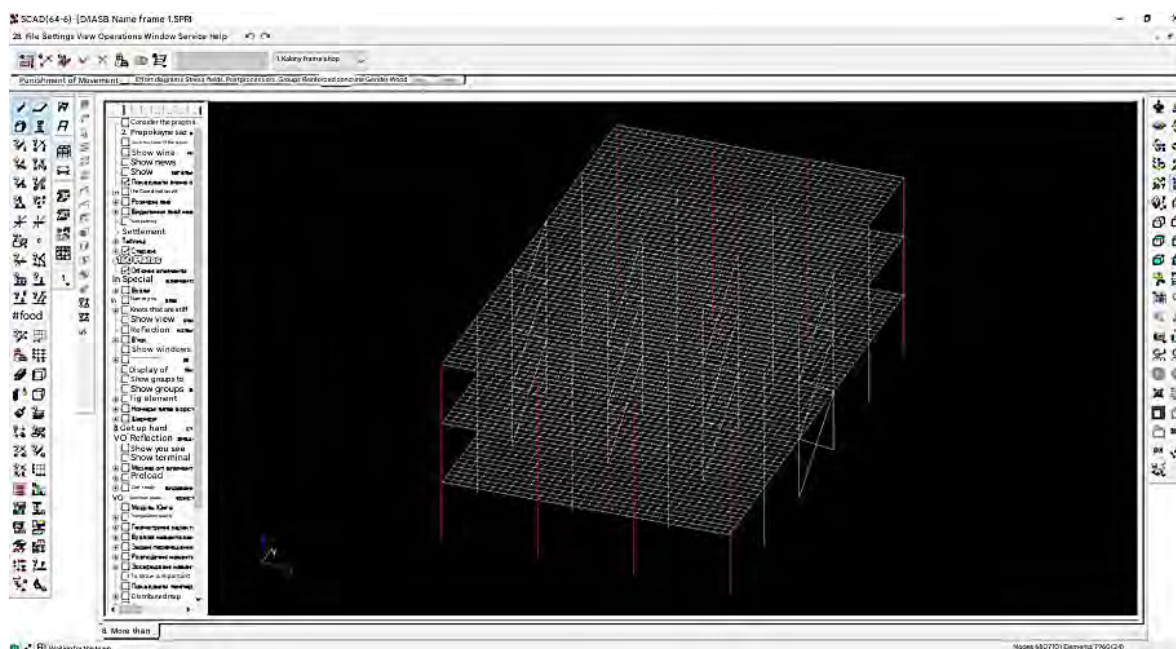


Figure 4.5 – Display of the active group on the design diagram

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 .

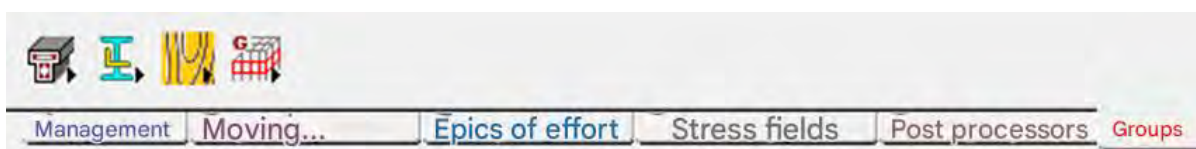


Figure 4.6 – List of the menu of the "Groups" tab



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.

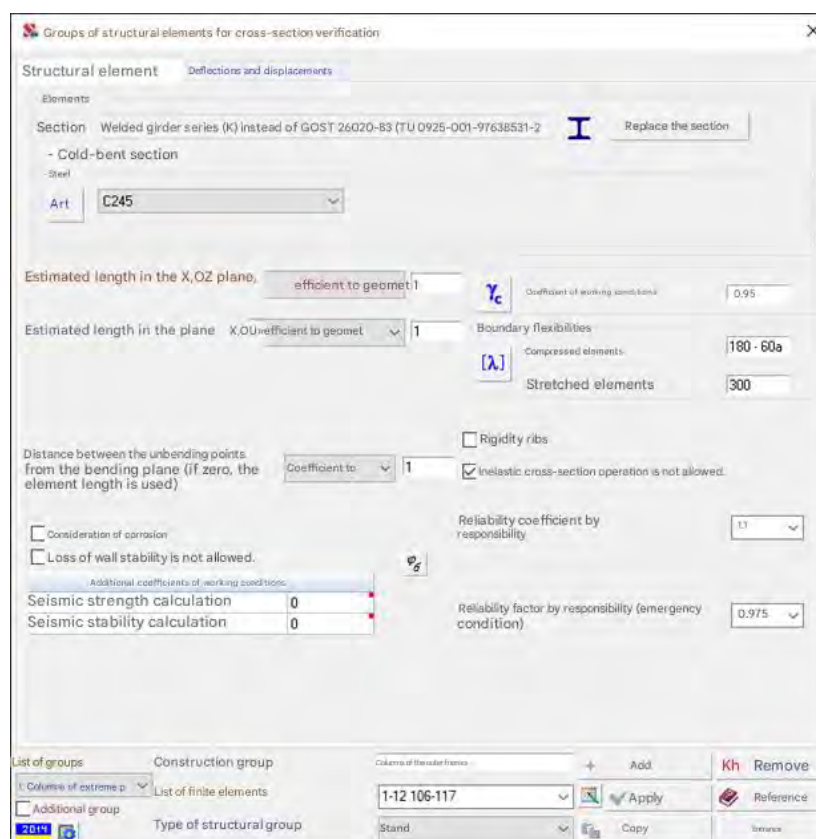


Figure 4.7 – Setting the parameters of a group of structural elements

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

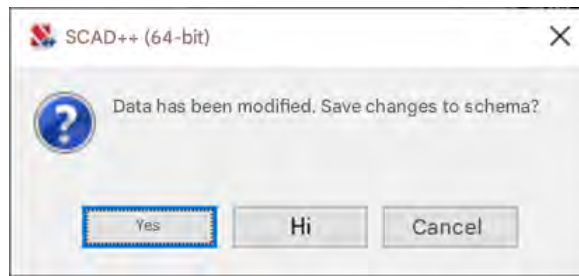


Figure 4.8 – Saving changed data

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.

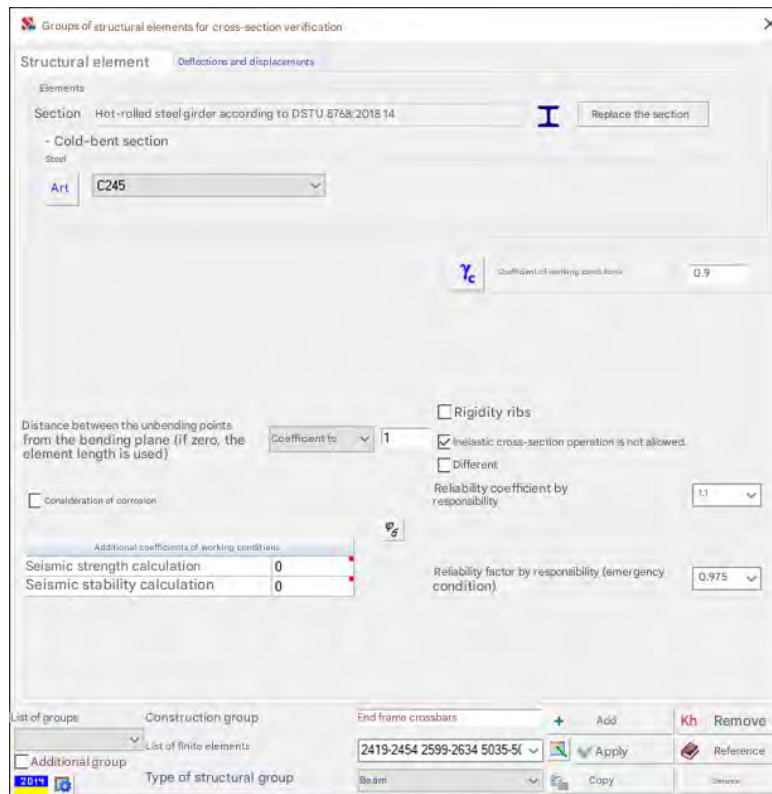


Figure 4.9 – Setting the parameters of a group of structural elements

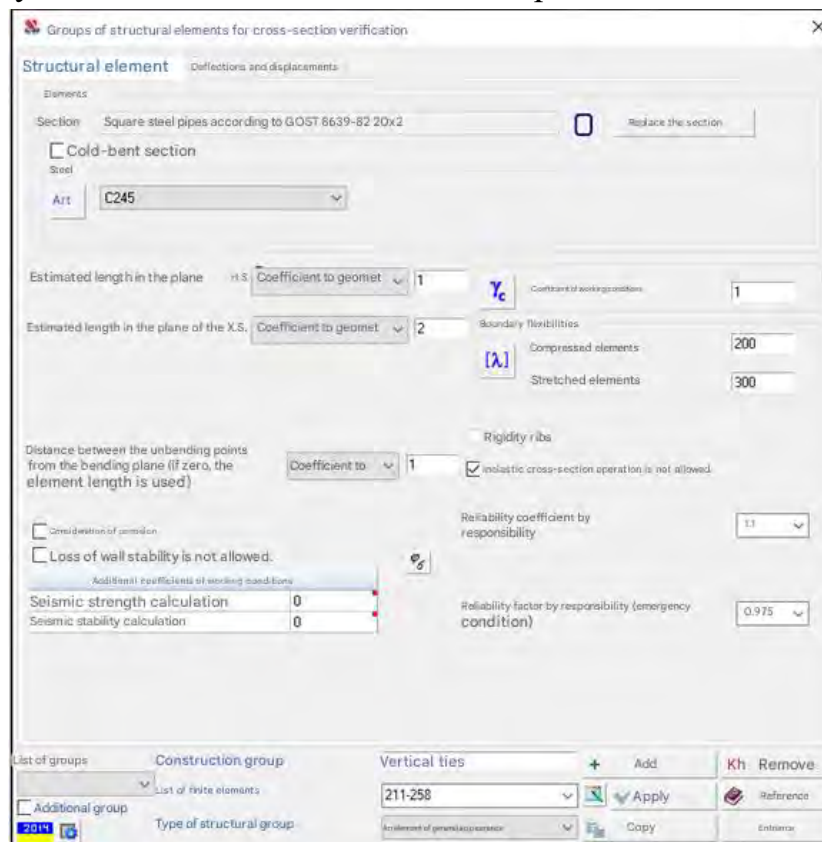
After adjusting all the parameters, click the "Add" button . Click the "Exit" button and agree to save the changed data.

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.



Groups of structural elements for cross-section verification

Structural element Deflections and displacements

Elements

Section: Square steel pipes according to GOST 8639-82 20x2  Replace the section

☐ Cold-bent section

Steel

Art: C245

Estimated length in the plane: Coefficient to geomet: 1  Coefficient of working conditions: 1

Estimated length in the plane of the X.S.: Coefficient to geomet: 2  Boundary flexibilities

Compressed elements: 200

Stretched elements: 300

Distance between the unbending points from the bending plane (if zero, the element length is used): Coefficient to: 1  Rigidly ribs

☒ Elastic cross-section operation is not allowed.

☐ Consideration of corrosion.

☐ Loss of wall stability is not allowed.

Additional coefficients of working conditions

Seismic strength calculation: 0

Seismic stability calculation: 0

Reliability coefficient by responsibility: 1.3

Reliability factor by responsibility (emergency condition): 0.975

List of groups: Construction group: Vertical ties: 211-258  Add  Remove

☐ Additional group: List of finite elements:  Apply  Reference

Type of structural group:  Copy  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".

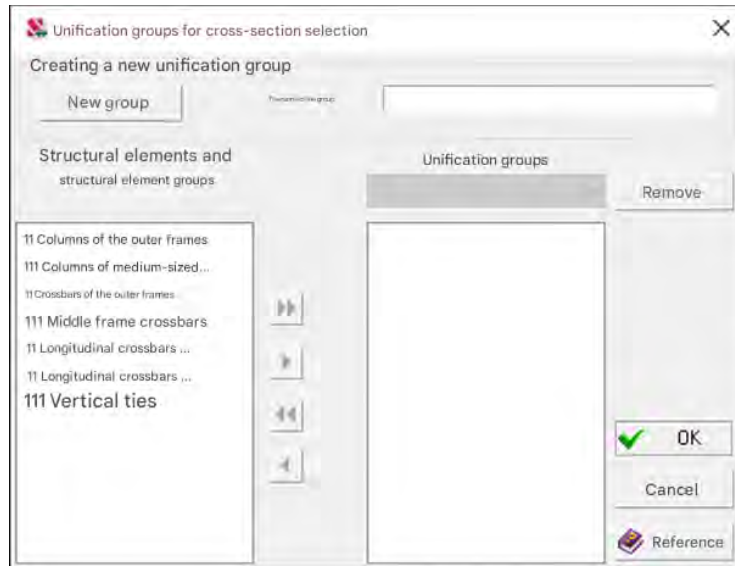


Figure 4.11 – Unification Groups dialog box to assign unification groups

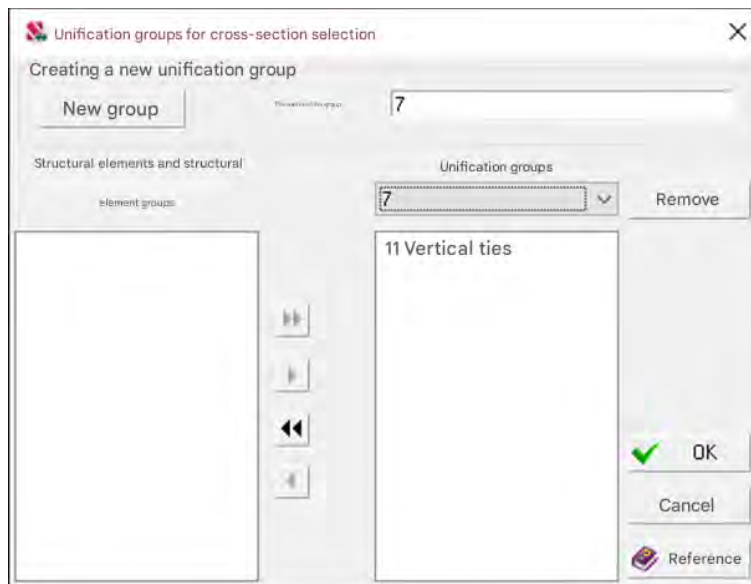


Figure 4.12 – Dialog box "Unification groups" with assigned 7 unification groups

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".

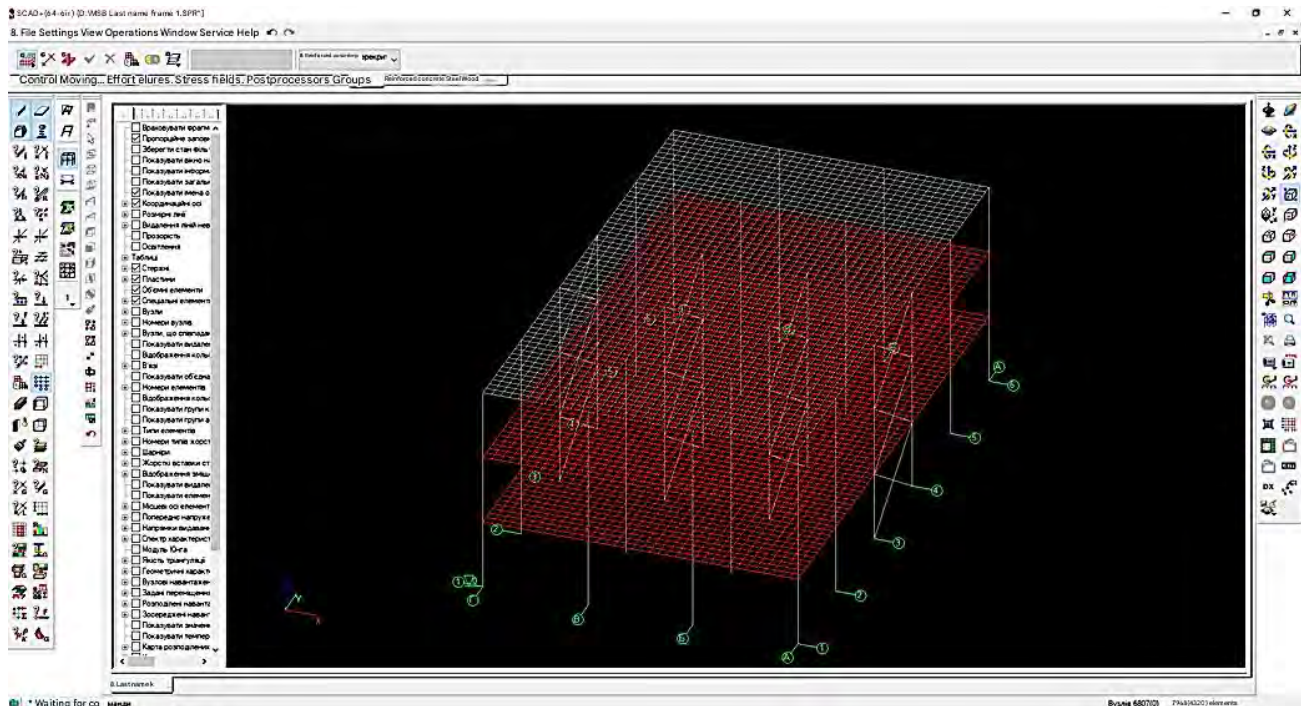


Figure 4.13 – Activated group reinforced concrete floors

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

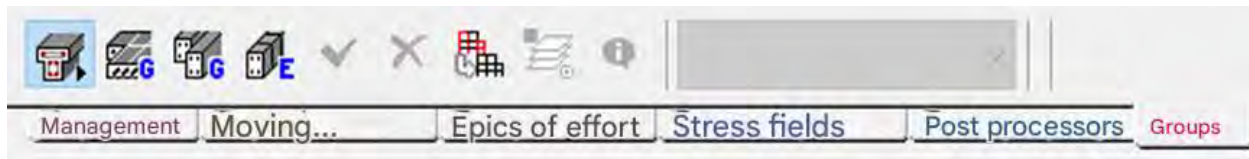


Figure 4.14 – Menu "Reinforced concrete structures"

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.

Plate reinforcement - General parameters

Design solution: 1.1

Reliability factor by responsibility: 0.975

Reliability factor by responsibility (second limit state): 0.975

Reliability factor by responsibility (emergency condition): 0.975

Element type: Shell ☐ Consider minimum reinforcement

Plate reinforcement

Acuminate	Class	Additional factor of conditions work	Diameter
Longitudinal	A400C	1	40
Transverse	A240C	1	12

Distances to the center of the structure:

a ₁	a ₂	a ₃	a ₄
30	30	30	30

☒ Calculation of crack resistance

☒ Take into account the requirements of the standards for the minimum percentage of reinforcement

Maximum percentage of reinforcement: 10

Plate reinforcement - Deflections and displacements

Type of loading: Static

Class of concrete: C20/25

Concrete aggregate: Quartz

Additional parameters

Additional coefficient of working conditions	1
Age of concrete (days)	28
Strength class of cement	Class B
Creep development time	36500 days
Temperature T(A)	20 °C
Number of days when temperature T exceeds D	28 days
Relative humidity	40 %

Plate reinforcement - Crack resistance

Construction group: Floor slabs

List of finite elements: 259-2418 2875-5034

List of groups: Floor slabs

Plate reinforcement - Parameters

Maximum permissible movements:

	Absolutely	MM
☒ vertical displacements from all loads	30	
☒ vertical movements from fixed and long-term rive	30	
☒ vertical displacements from time loads	30	

Structural group: Floor slabs

List of finite elements: 5491-7650

List of groups:

Plate reinforcement - Parameters

Parameters

No cracks: ☐

Maximum crack opening width: w_{max} 0.4

Structural group: Floor slabs

List of finite elements: 259-2418 2875-5034

List of groups:

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.

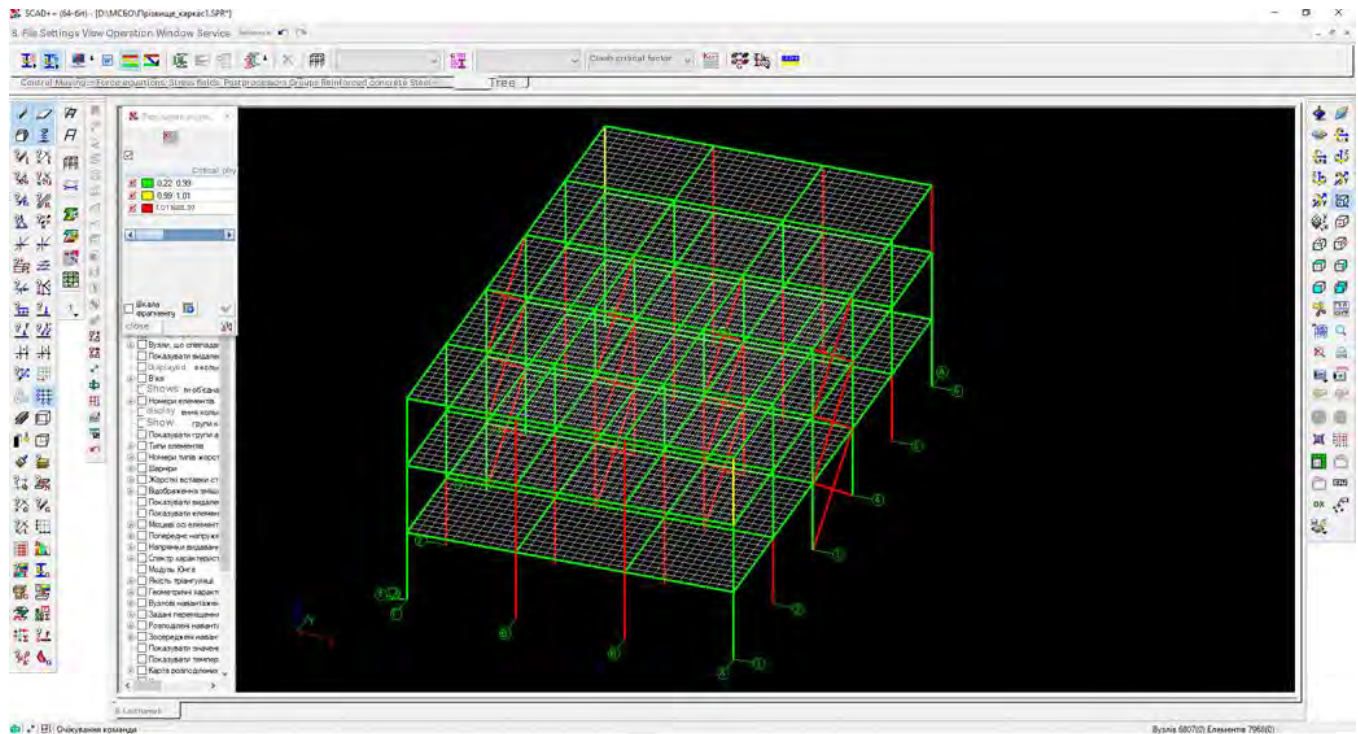


Figure 4.16 – Visualization of the results of the calculation of cross-sections of steel structures

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.

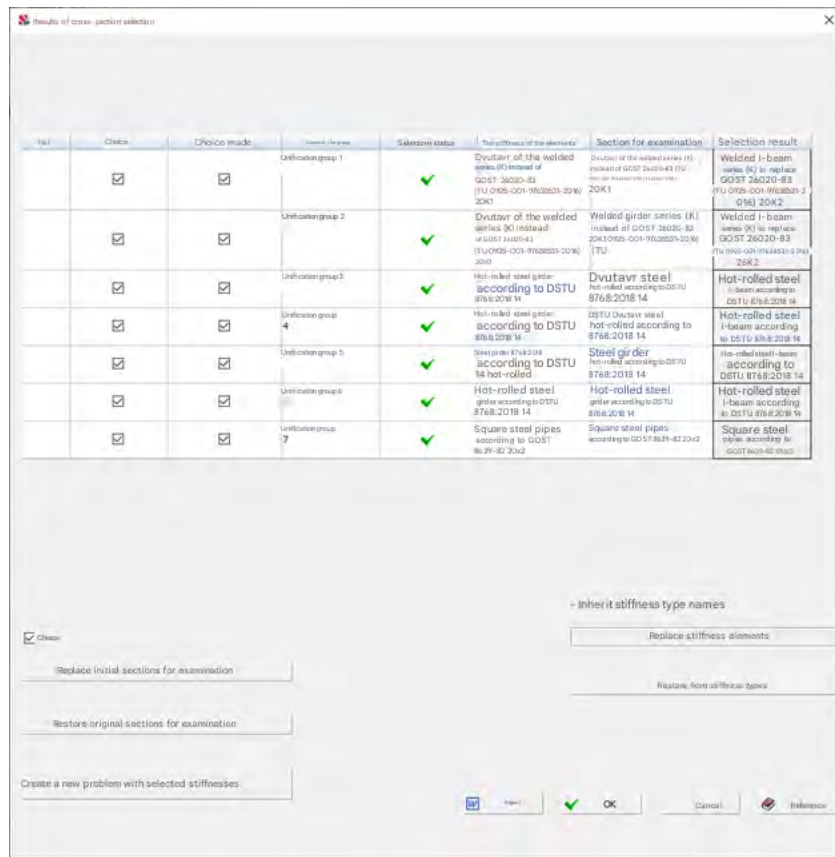


Figure 4.17 – Replacement of stiffeners based on the results of selection

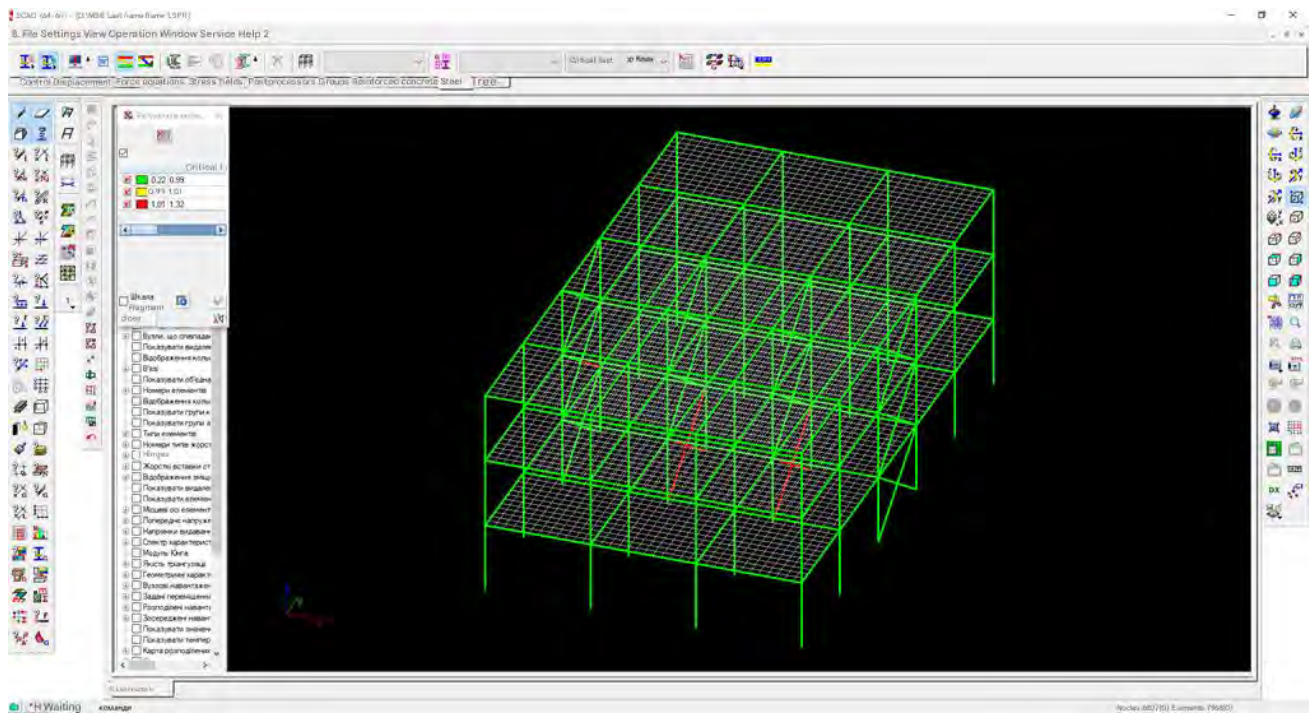


Figure 4.18 – Results of calculation of replaced cross-sections after stress recalculation

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 hot-rolled according to DSTU 8768:2018 14	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.

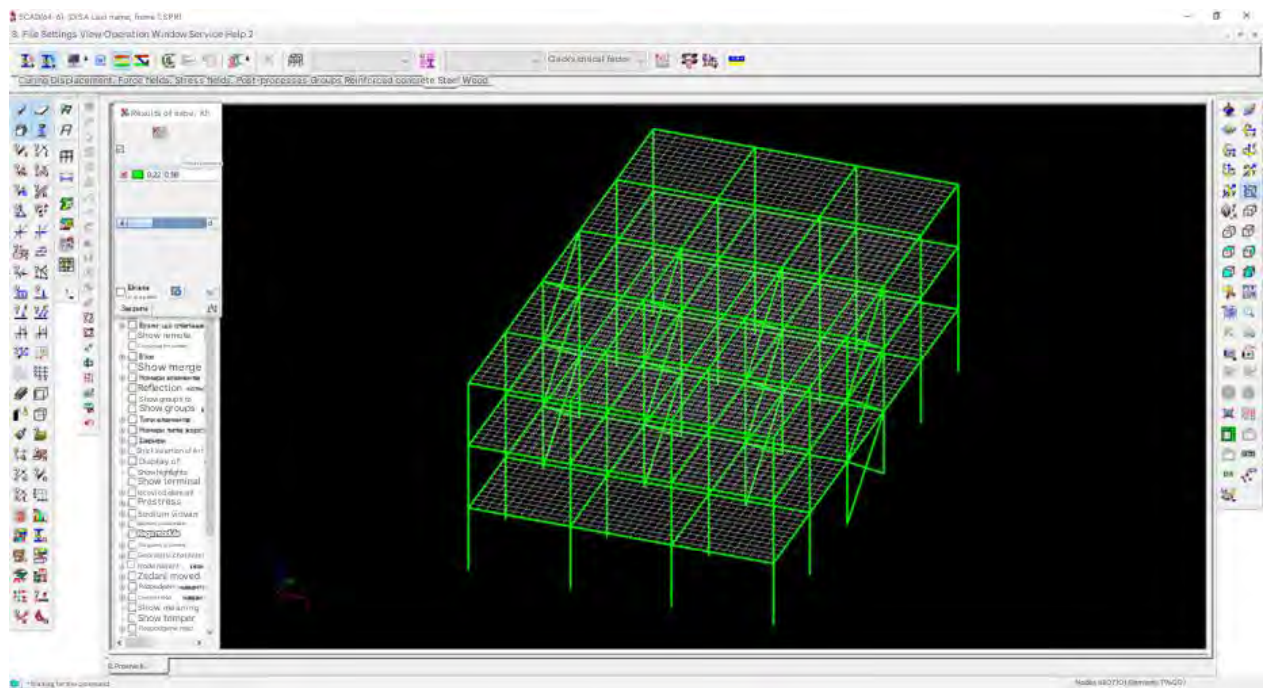


Figure 4.20 – Results of the final iteration of the selection of sections

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.

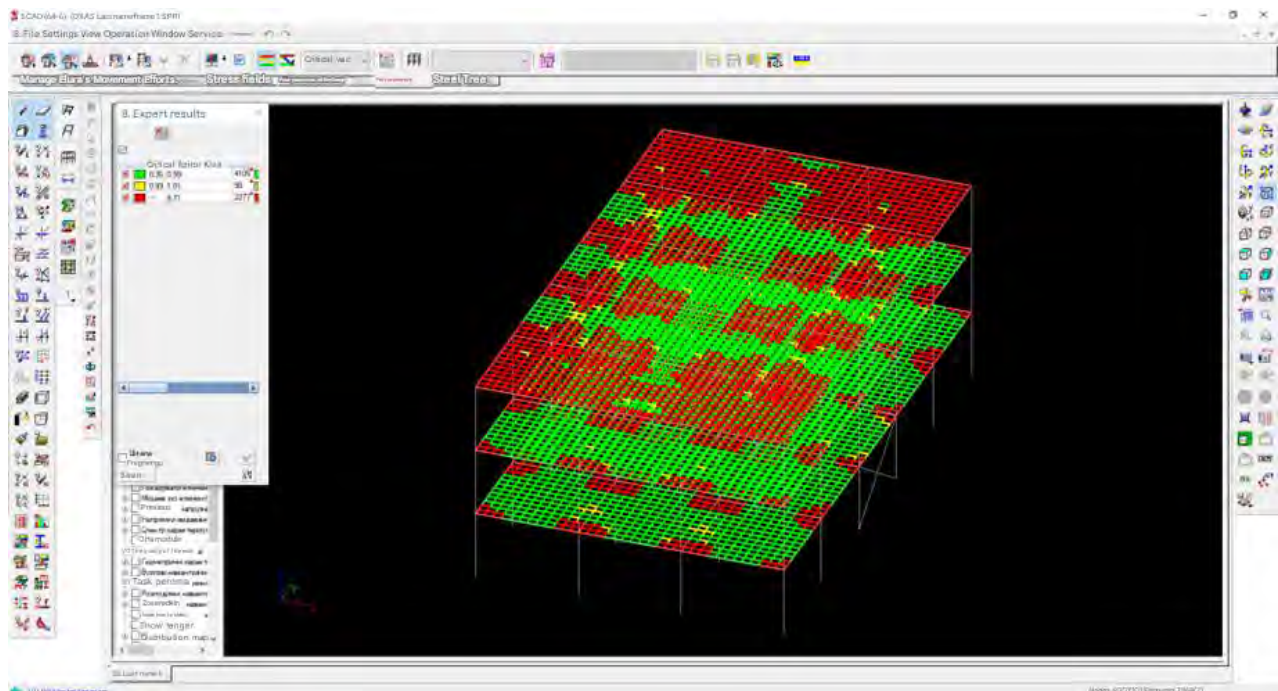


Figure 4.21 – Results of the calculation of the predefined reinforcement

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.

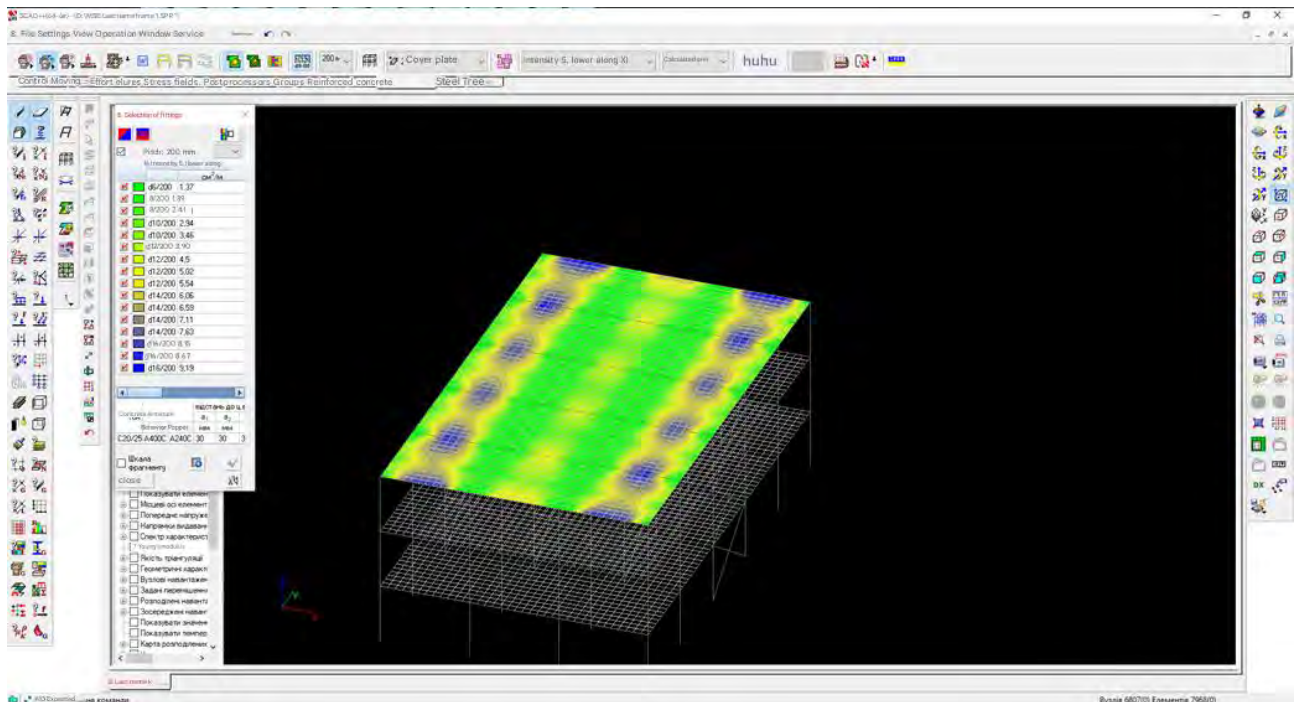


Figure 4.23 – Isopoles of reinforcement selection results

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.

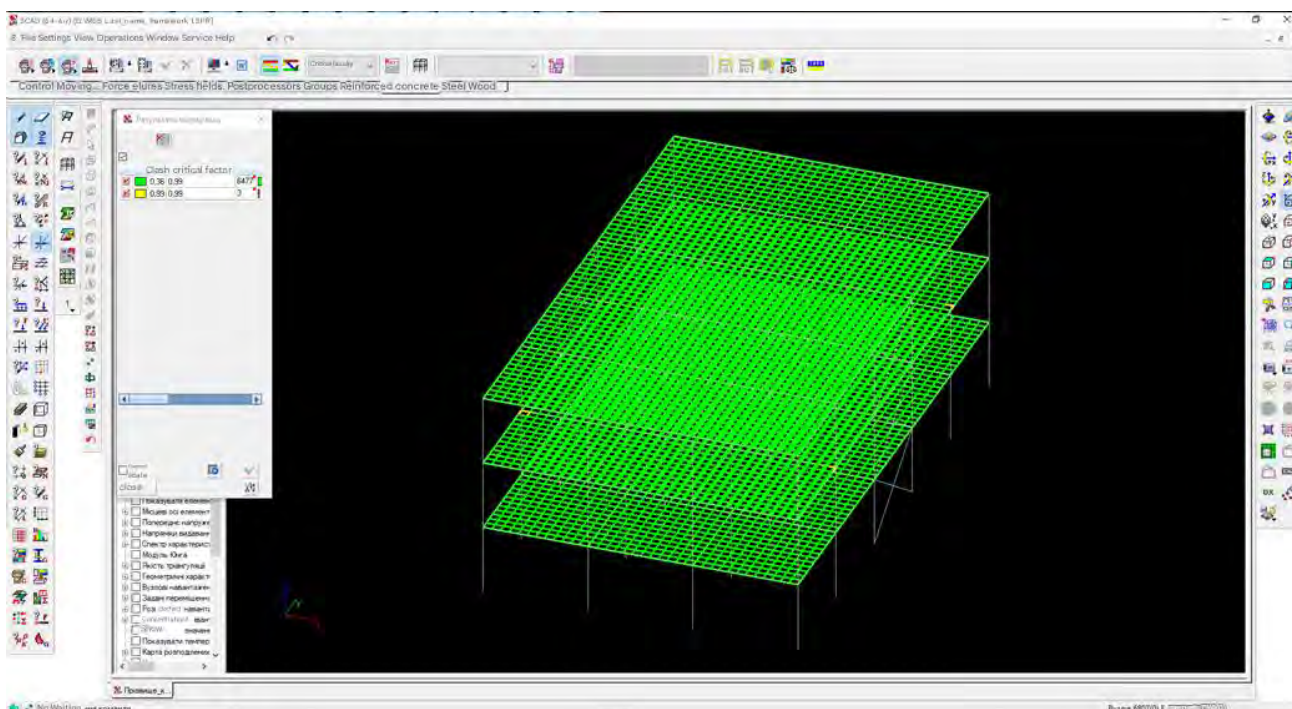


Figure 4.24 – Calculation of floor slabs and coatings with applied reinforcement based on the results of selection

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.

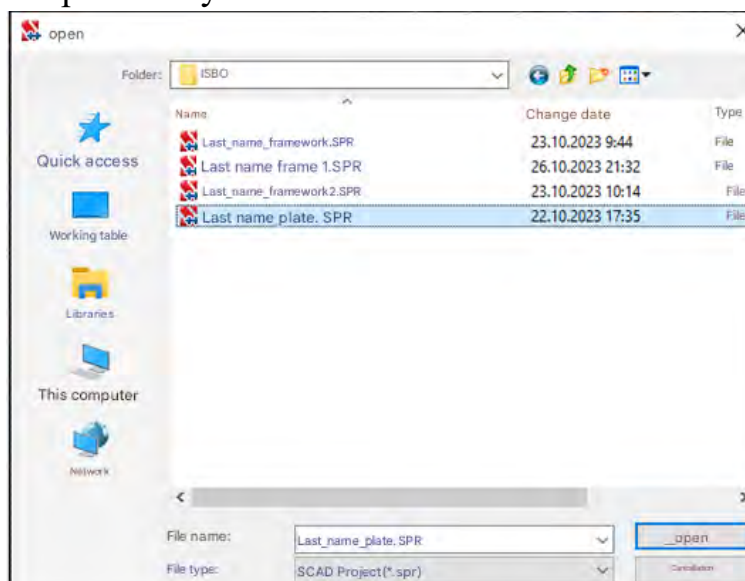


Figure 4.25 – Opening a stovetop file

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"

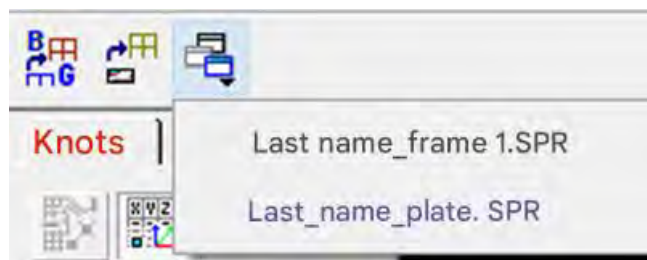


Figure 4.27 – Selection of the circuit for connection

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.

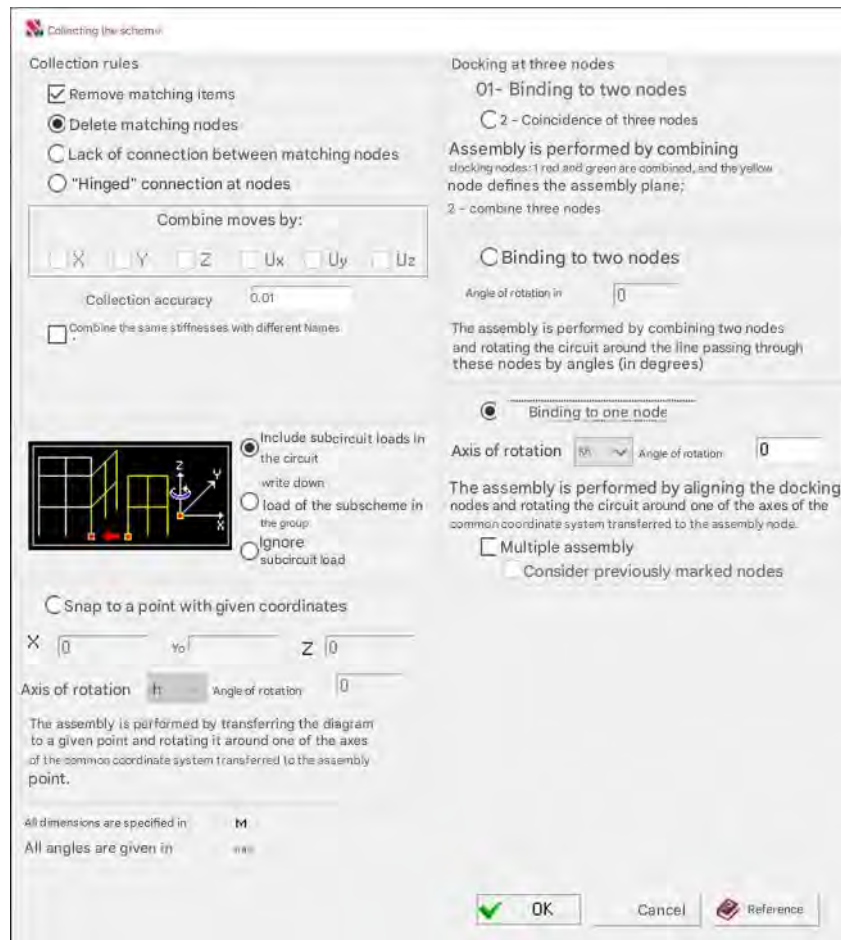


Figure 4.28 – Choosing the assembly method

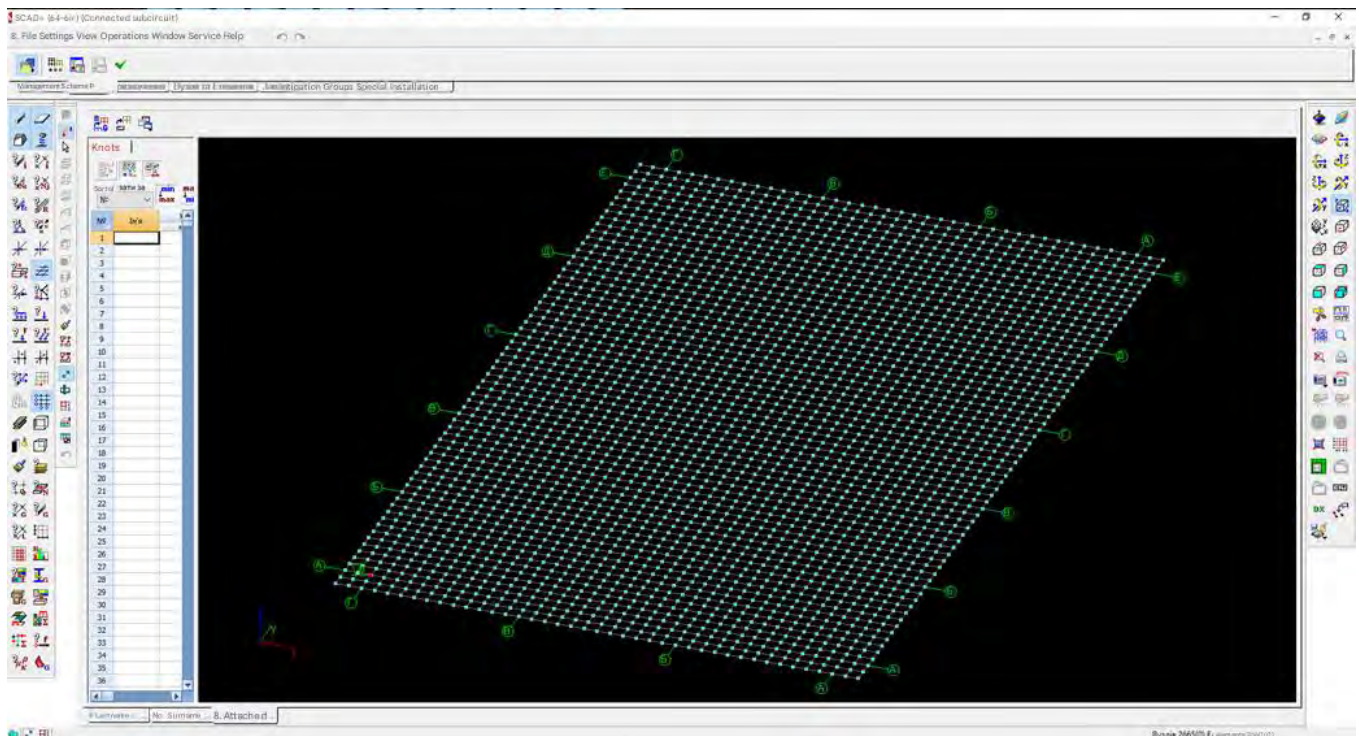


Figure 4.29 – Selecting the assembly point on the slab subcircuit

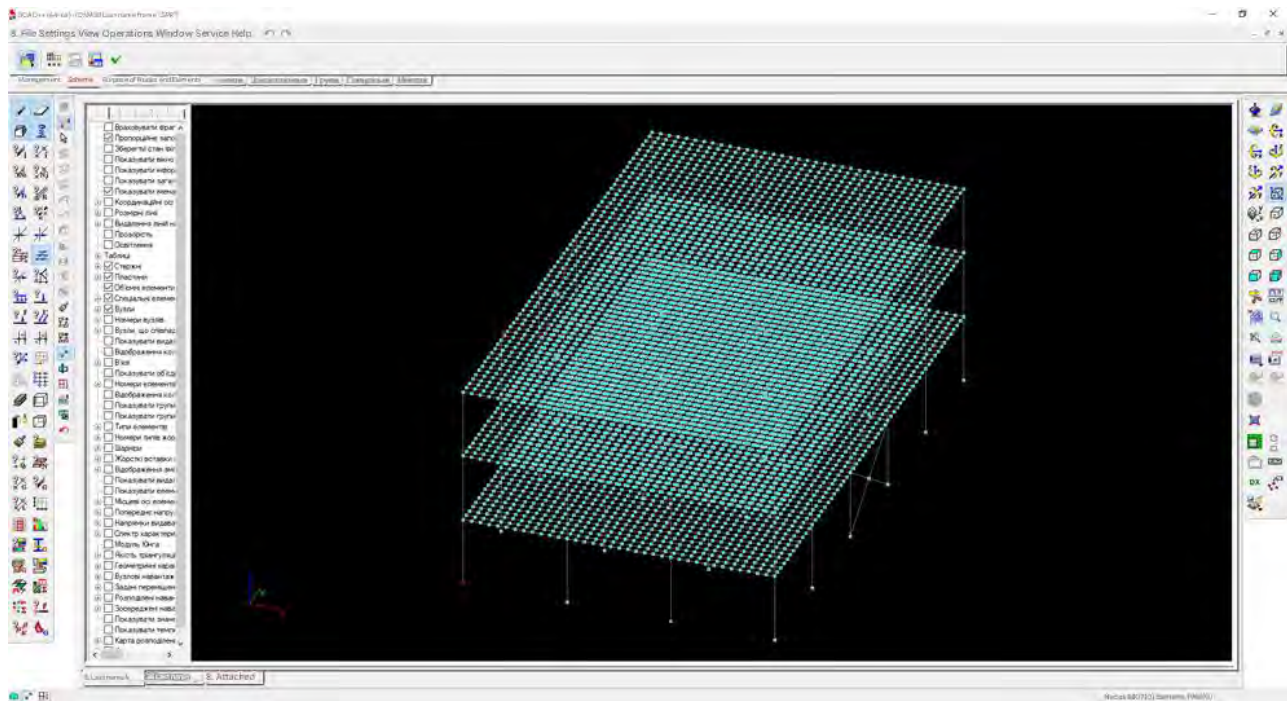


Figure 4.30 – Selecting an assembly point on the frame subcircuit

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

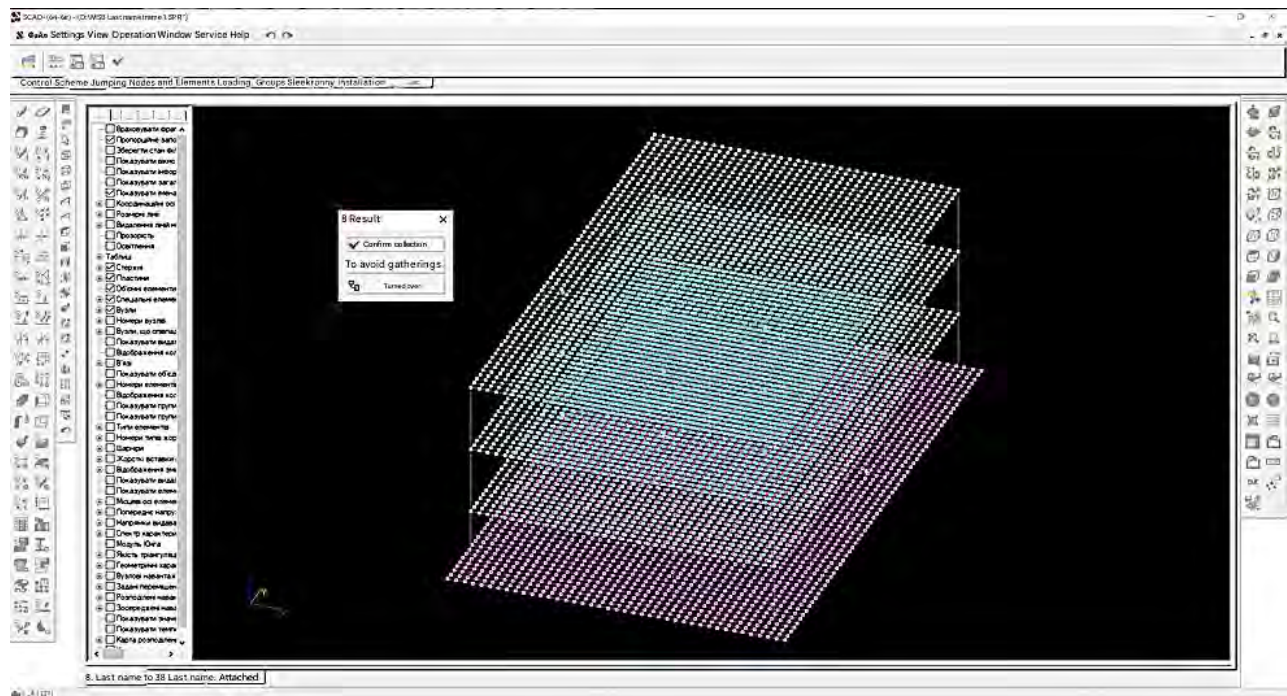


Figure 4.31 – Confirmation of assembly results

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.

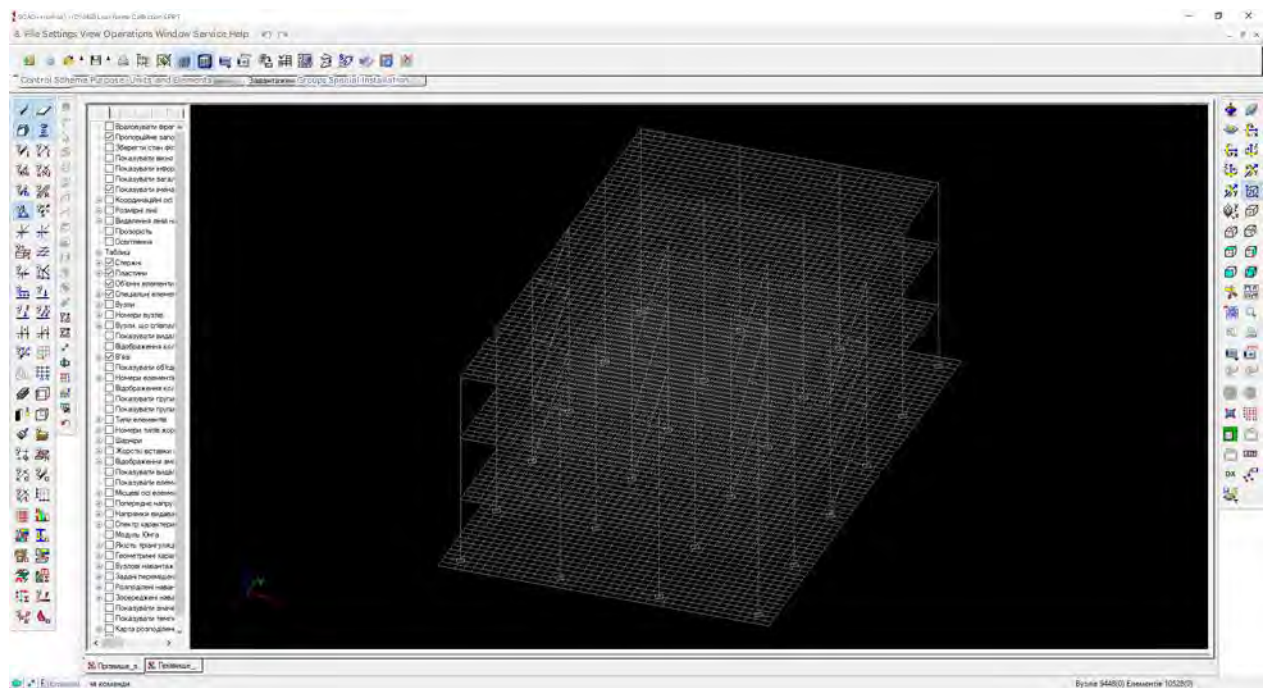


Figure 4.32 – Design scheme with extra knits

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

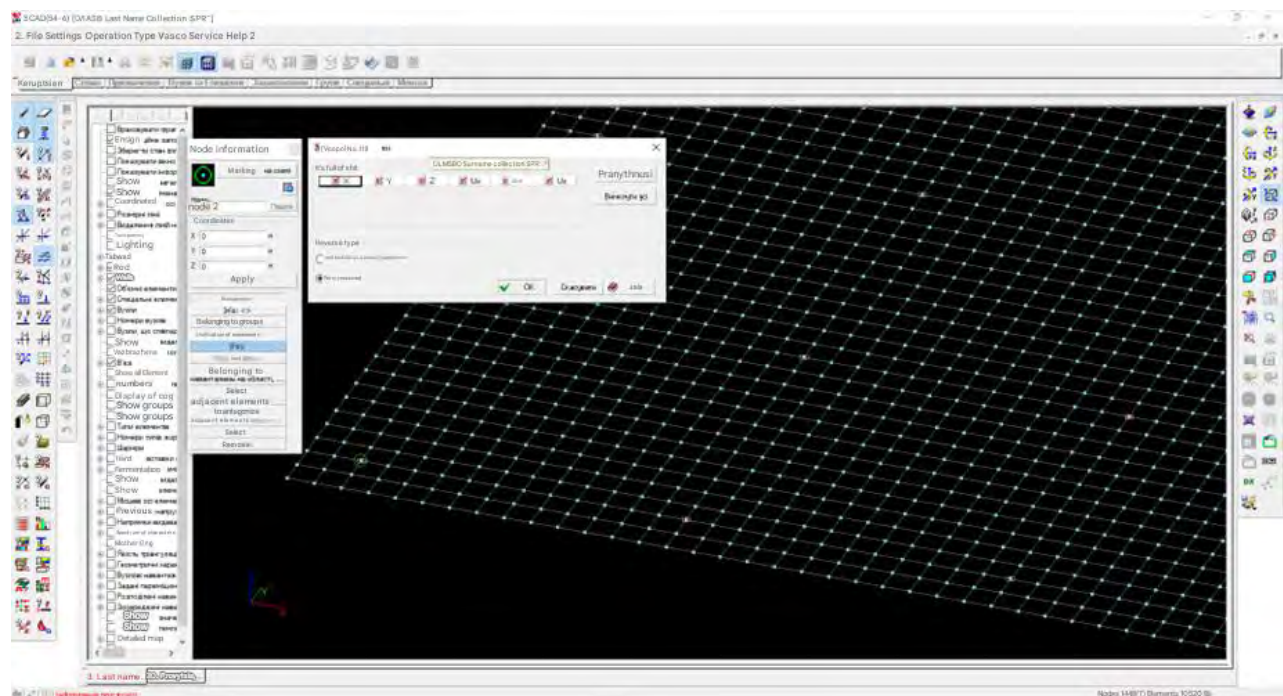


Figure 4.33 – Turning off elms using the Knot Information tool

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

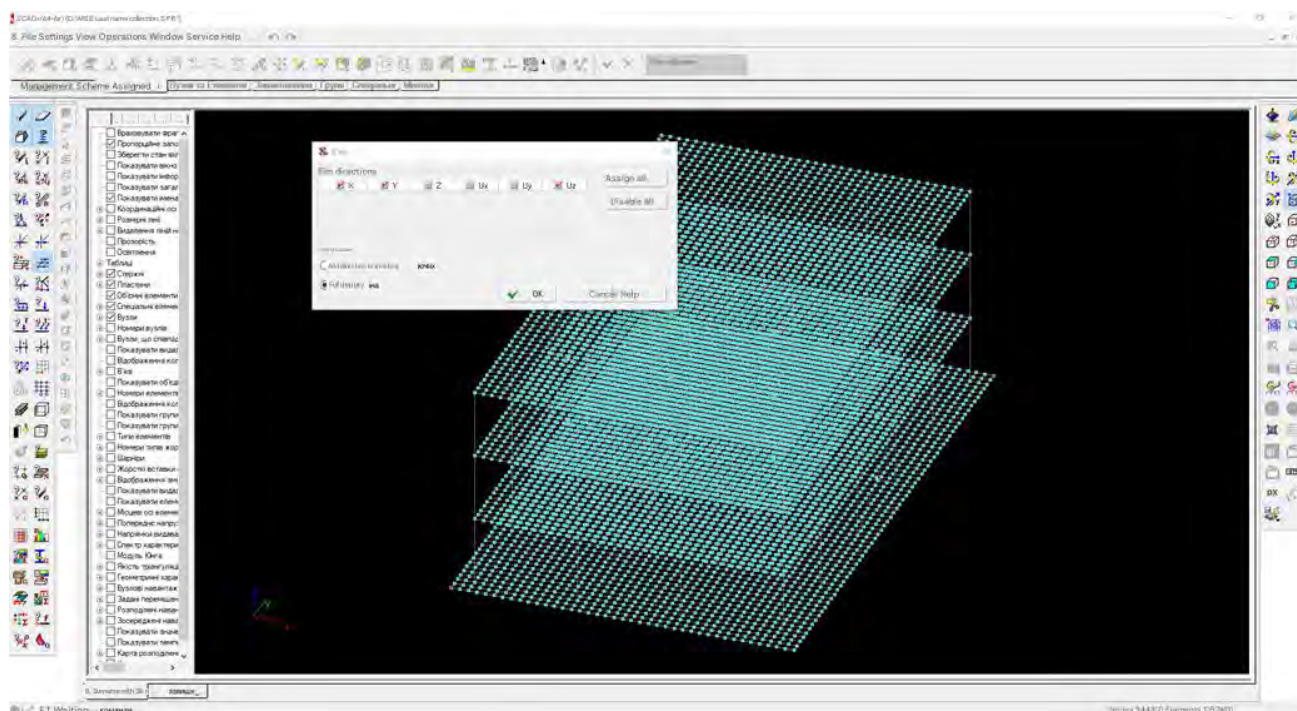


Figure 4.34 – Adding horizontal elms to the foundation slab

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".

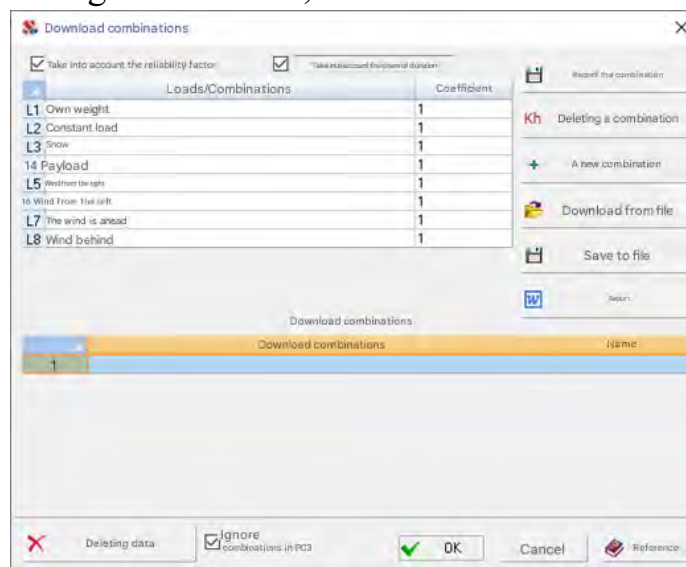


Figure 4.35 – Creating a combination of loads for the iterative calculation of bed coefficients

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.

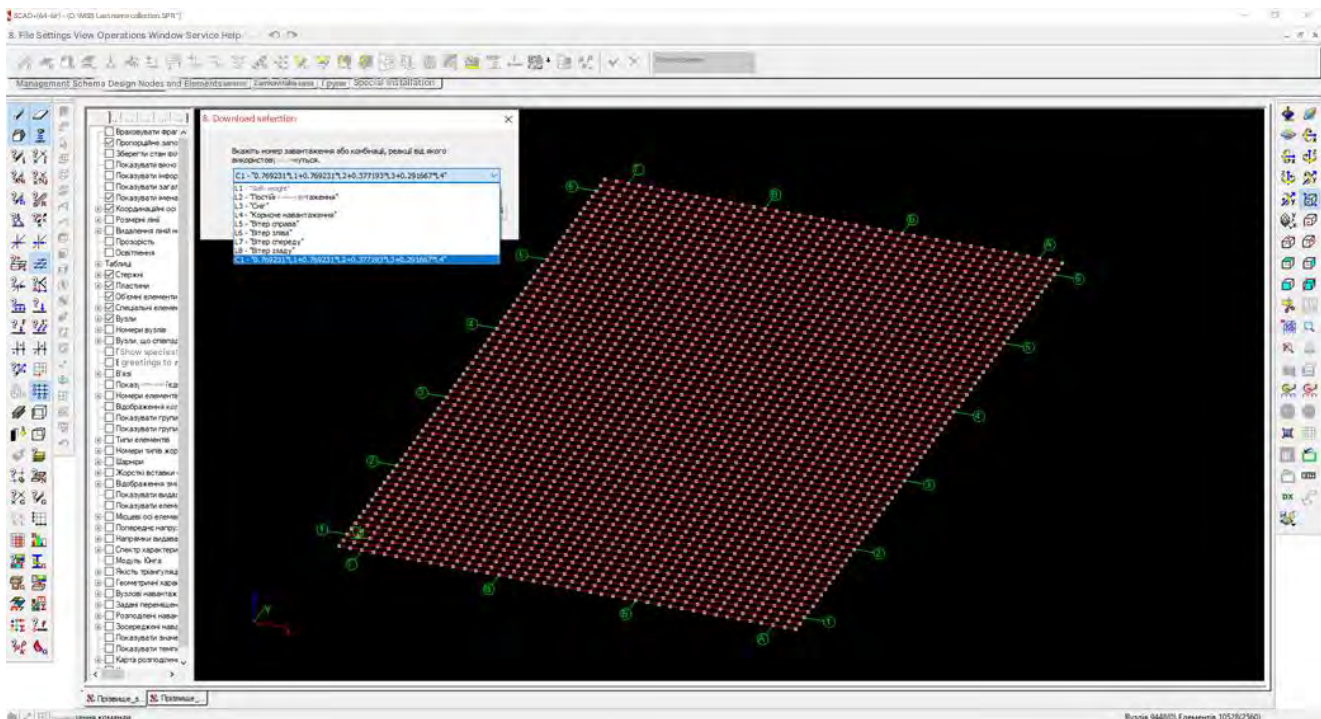


Figure 4.36 – Selection of foundation slab elements and combinations for calculating bedding coefficients

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.

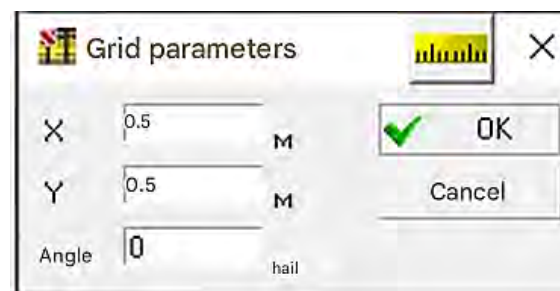


Figure 4.37 – Changing the pitch of the plane grid

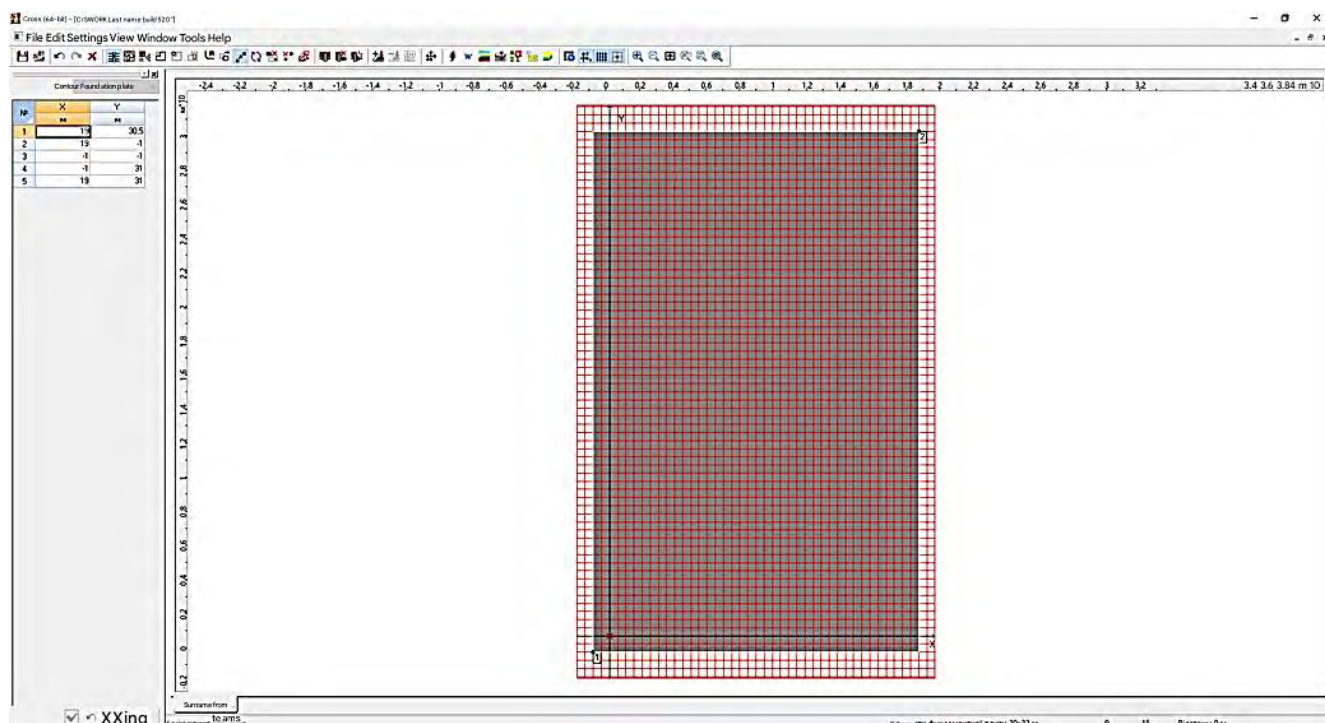


Figure 4.38 – Installing the foundation slab on the plane

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

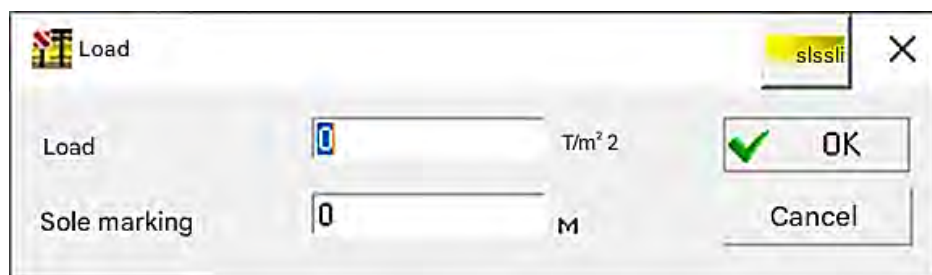


Figure 4.39 – Purpose of the foundation slab mark

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	2008.54

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.

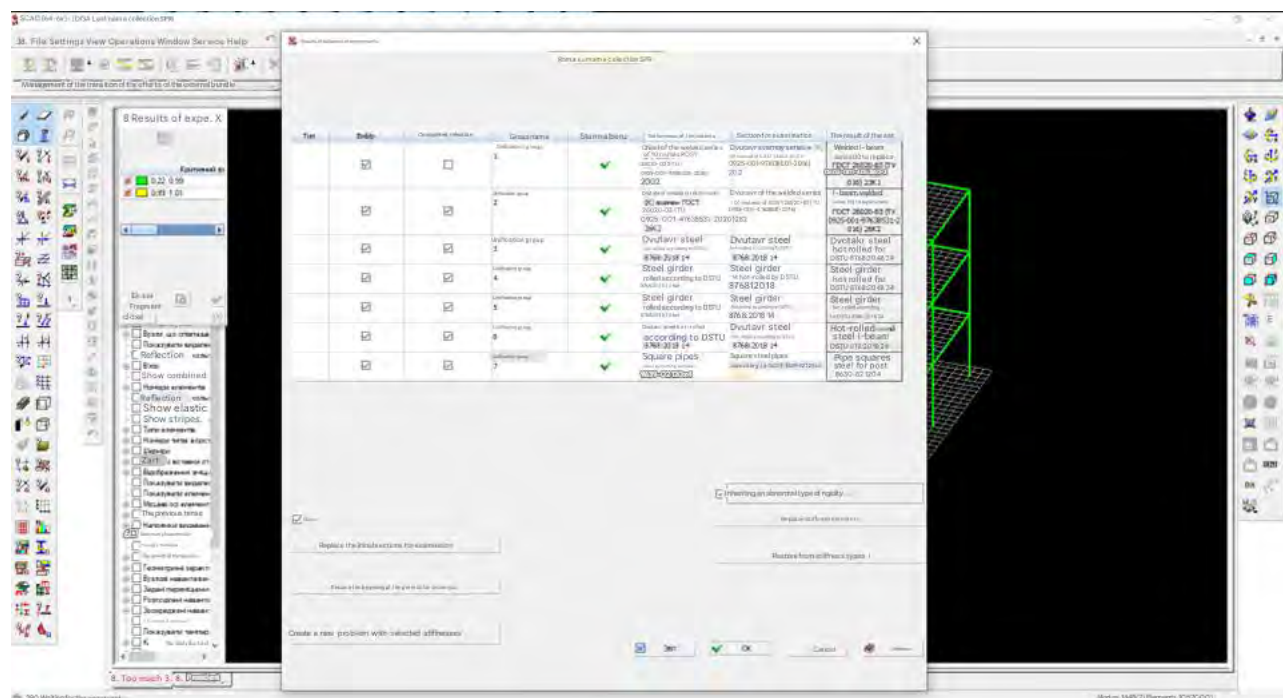


Figure 4.43 – Results of calculation and selection of steel elements after the calculation of the "Foundation-Foundation-Structure" system.

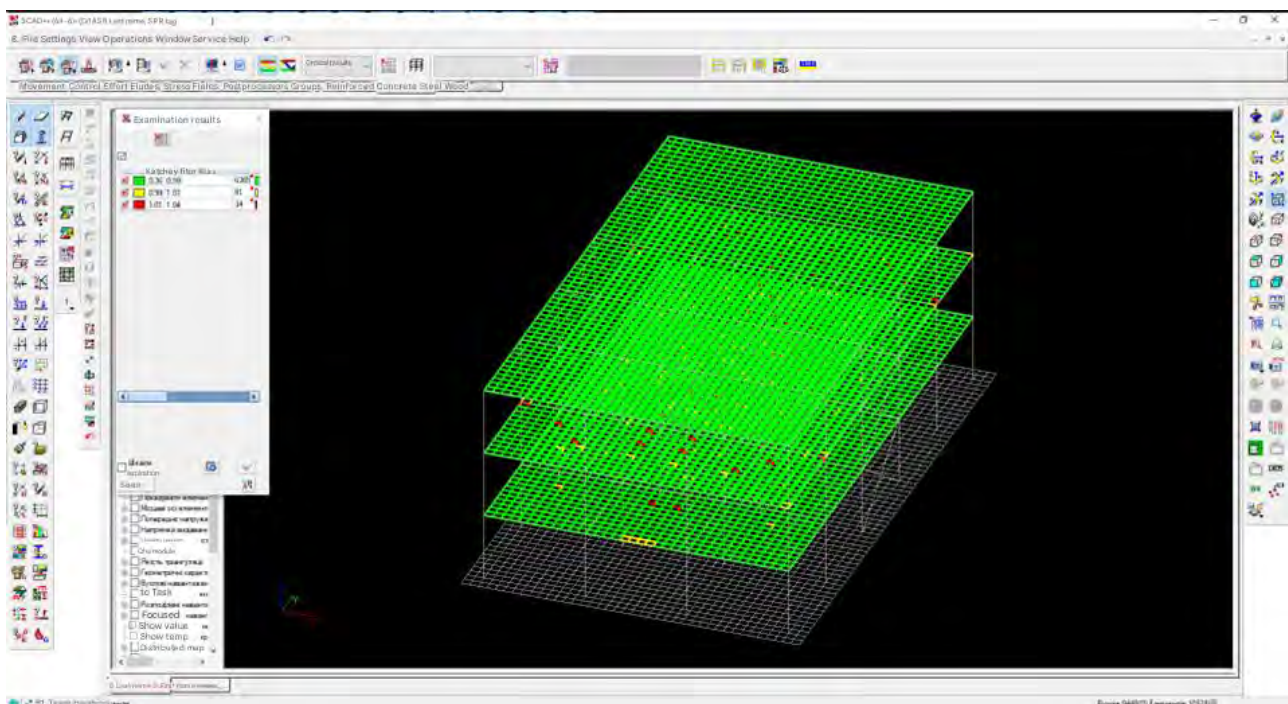


Figure 4.44 – Results of the examination of floor slabs and coatings after the calculation of the "Foundation-Foundation-Structure" system

Create a group and apply the parameters to calculate the foundation slab in a similar way to floor and coating slabs. Make a selection of reinforcement.

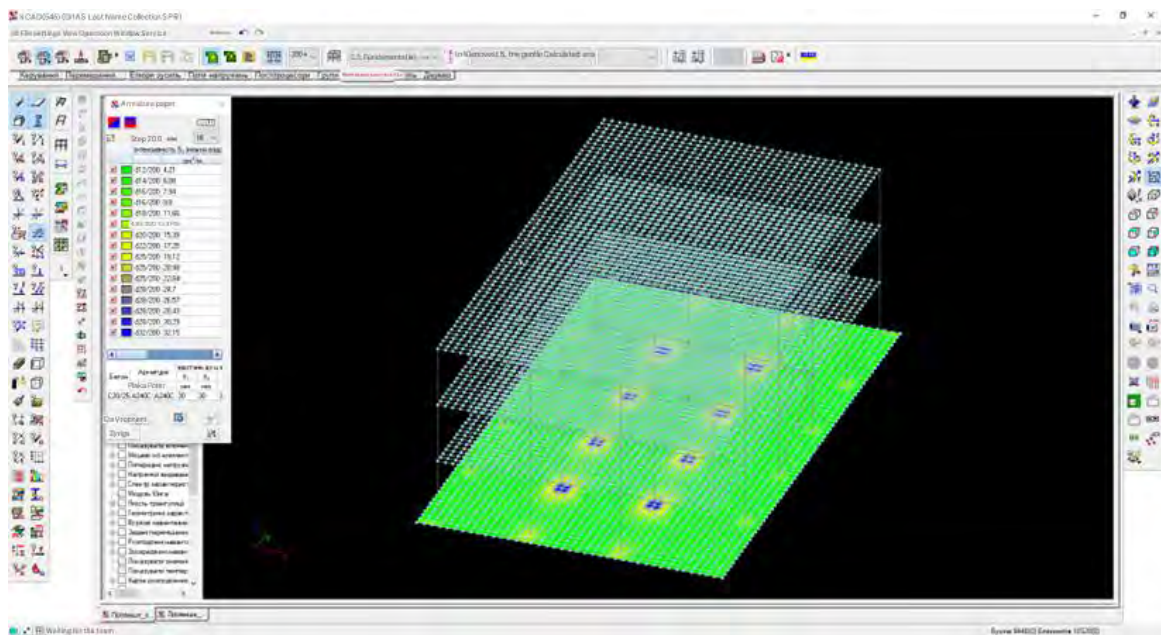


Figure 4.45 – Results of the selection of reinforcement for the foundation slab

Apply reinforcement based on the results of the selection using the button .

Exit to the project management screen, replace the stiffness with those obtained as a result of selection. Perform a linear calculation of the bed coefficients. If necessary, perform the following iteration.

Assign the coefficients of the C2 bedding obtained from the CROSS in the "Assign elastic base coefficients" dialog box. Leave the C1 coefficients obtained from the CROSS unchanged.

Perform a linear calculation. Analyze the results of the calculation of the cross-sections of steel and reinforced concrete elements. If necessary, perform selection and subsequent iteration until the desired result is obtained.

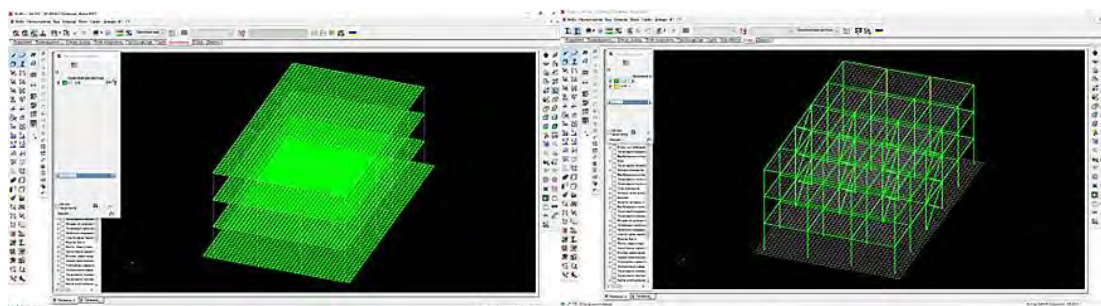


Figure 4.46 – Final iteration. All elements meet the requirements

Tasks for independent work on topic 4.

Compare the results of the selection of metal and reinforced concrete elements of the frame installed on a rigid base and taking into account joint work with the soil massif. Explain the results obtained.

TOPIC 5

ANALYSIS OF THE MUTUAL INFLUENCE OF ADJACENT BUILDING FOUNDATIONS

In the CROSS program, we will increase the site in the direction of Y by 30 m.

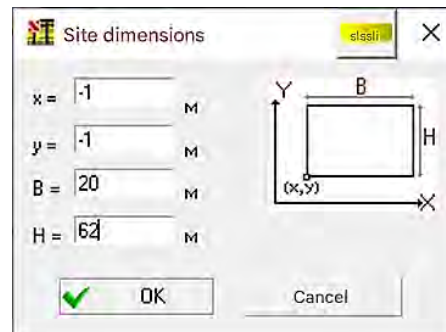


Figure 5.1 – Increasing the site by 30 m

Use the Existing Building tool to draw an outline of an adjacent existing building next to the building you are designing.

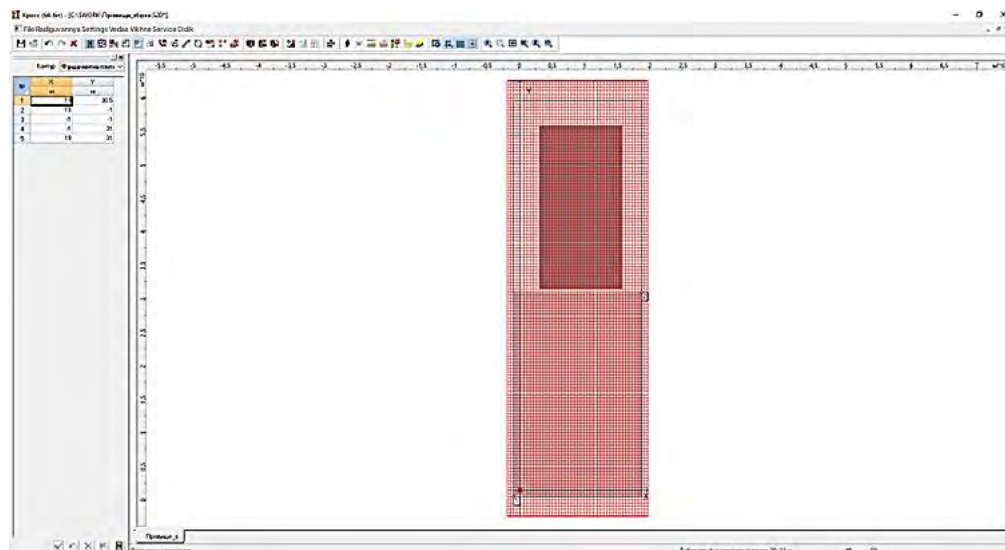


Figure 5.2 – Entering the outline of an adjacent existing building

Assign the load from the neighboring building and the mark of the sole of its foundation.

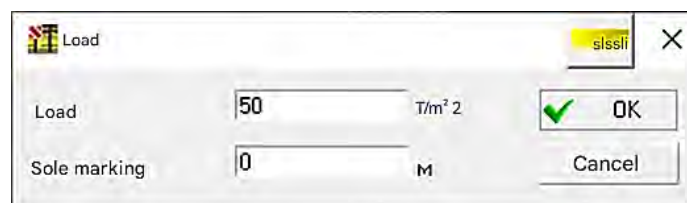


Figure 5.3 – Assignment of load from a neighboring building

Perform the calculation and analyze the bedding coefficients, taking into account the presence of a neighboring building.

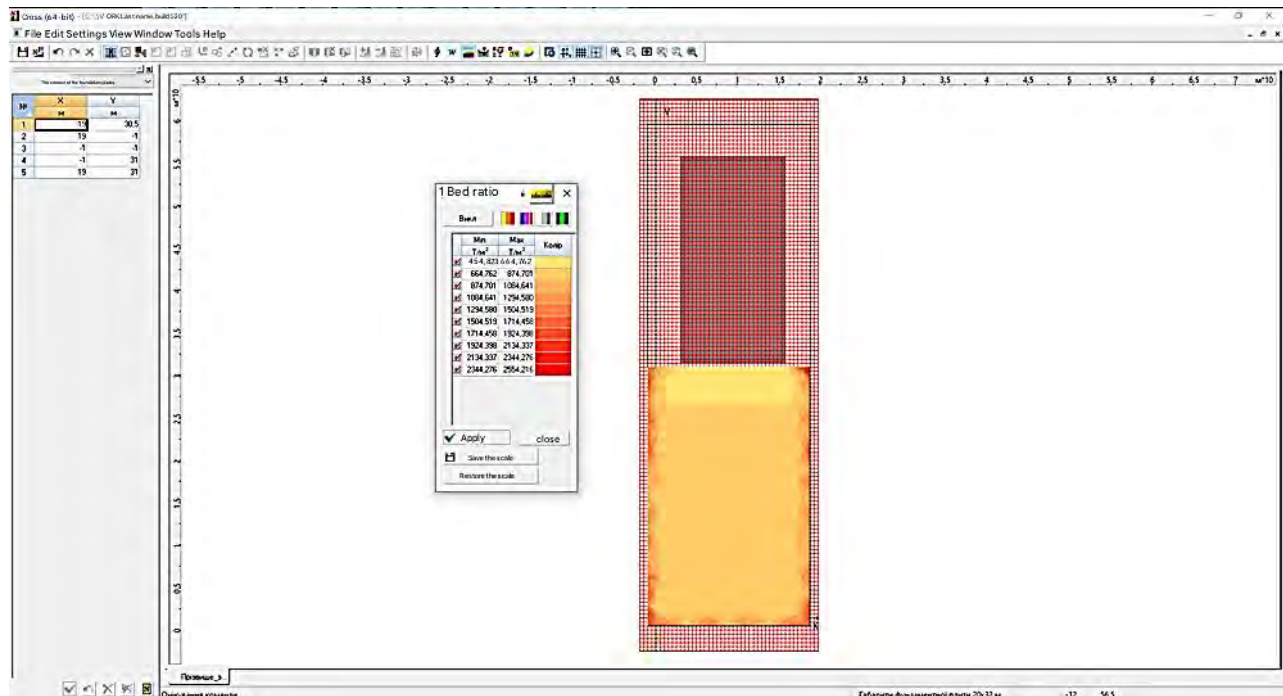


Figure 5.4 – Isopoles of bed coefficients taking into account the presence of a neighboring building

Import the calculated bedding coefficients into SCAD++ and make sure that they are applied correctly using the filter panel.

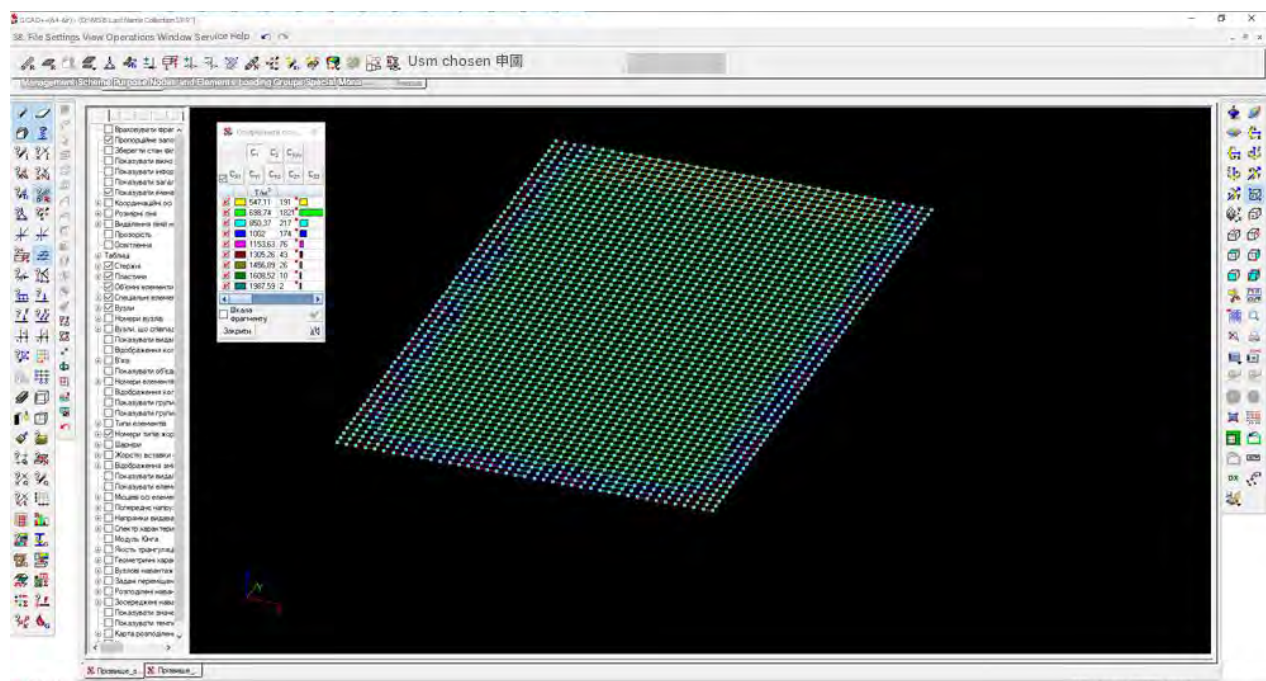


Figure 5.5 – Display of calculated bedding coefficients in SCAD++

Assign the coefficients of the C2 bedding obtained from the CROSS.

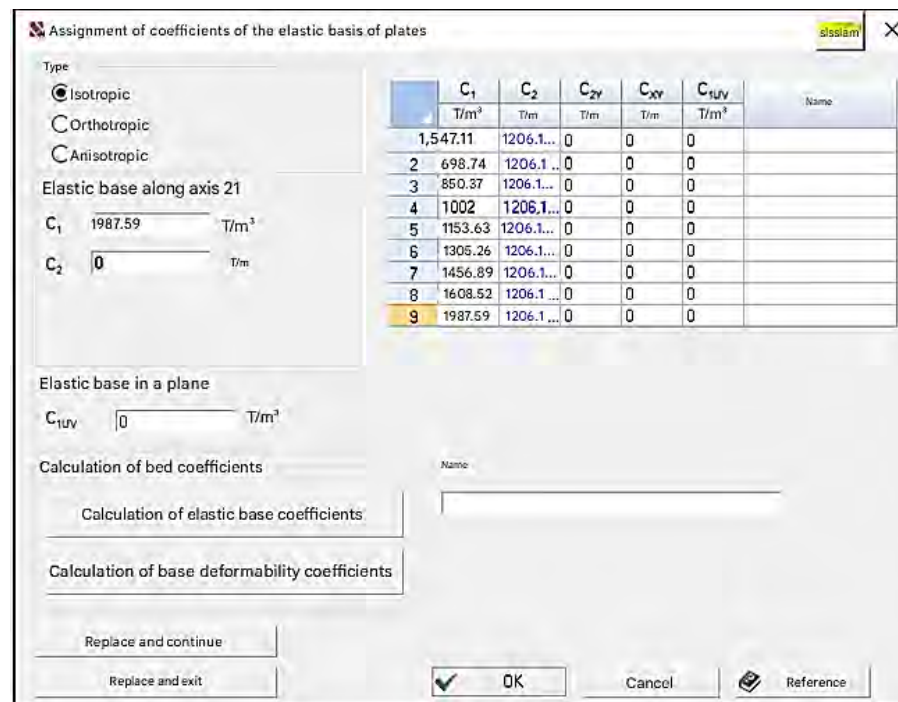


Figure 5.6 – Assignment of bed coefficients C2

Perform a linear calculation.

Go to the graphical analysis tab and analyze the changes in the stress-strain state of the building elements and the need to make adjustments to the accepted sections and reinforcement based on the results of the impact of the neighboring building.

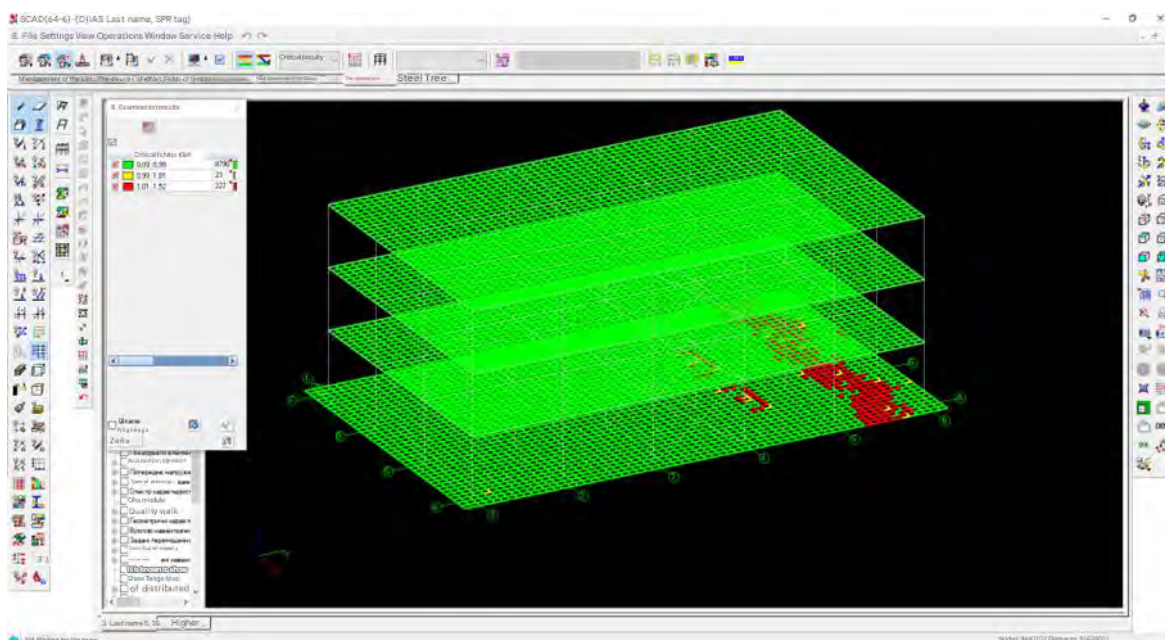


Figure 5.7 – Results of calculating the strength of the foundation slab, taking into account the presence of a neighboring building

Analyze the deformed state of the building on the "Displacement" tab from the applied combination of forces, taking into account the presence of a neighboring building.

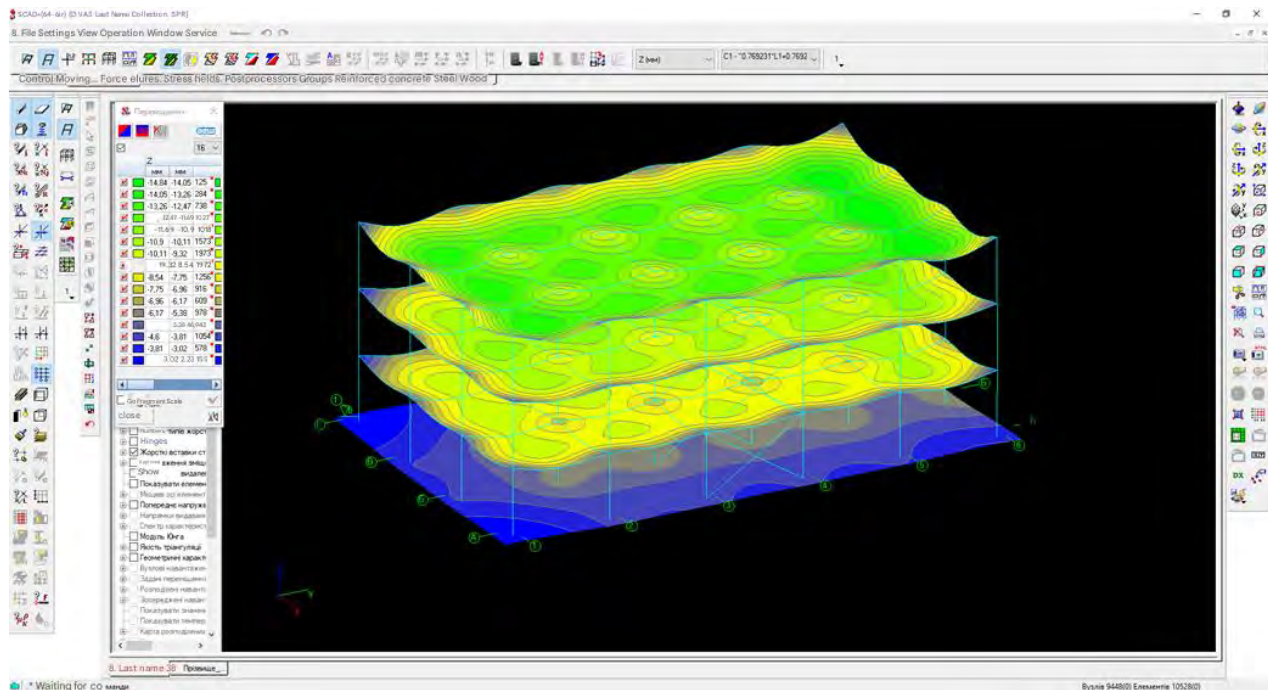


Figure 5.8 – Deformed condition of the building as a whole, taking into account the neighboring building

Separately, pay attention to the deformed state of the foundation slab.

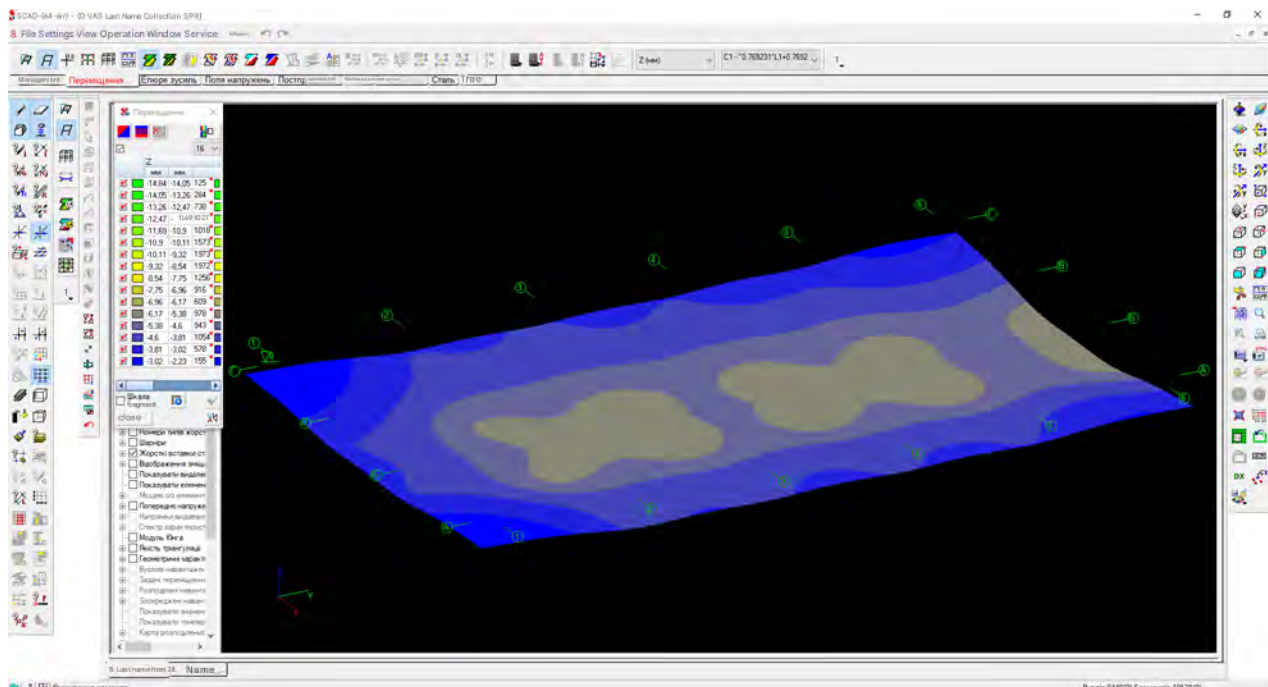


Figure 5.9 – Deformed state of the foundation slab, taking into account the neighboring building

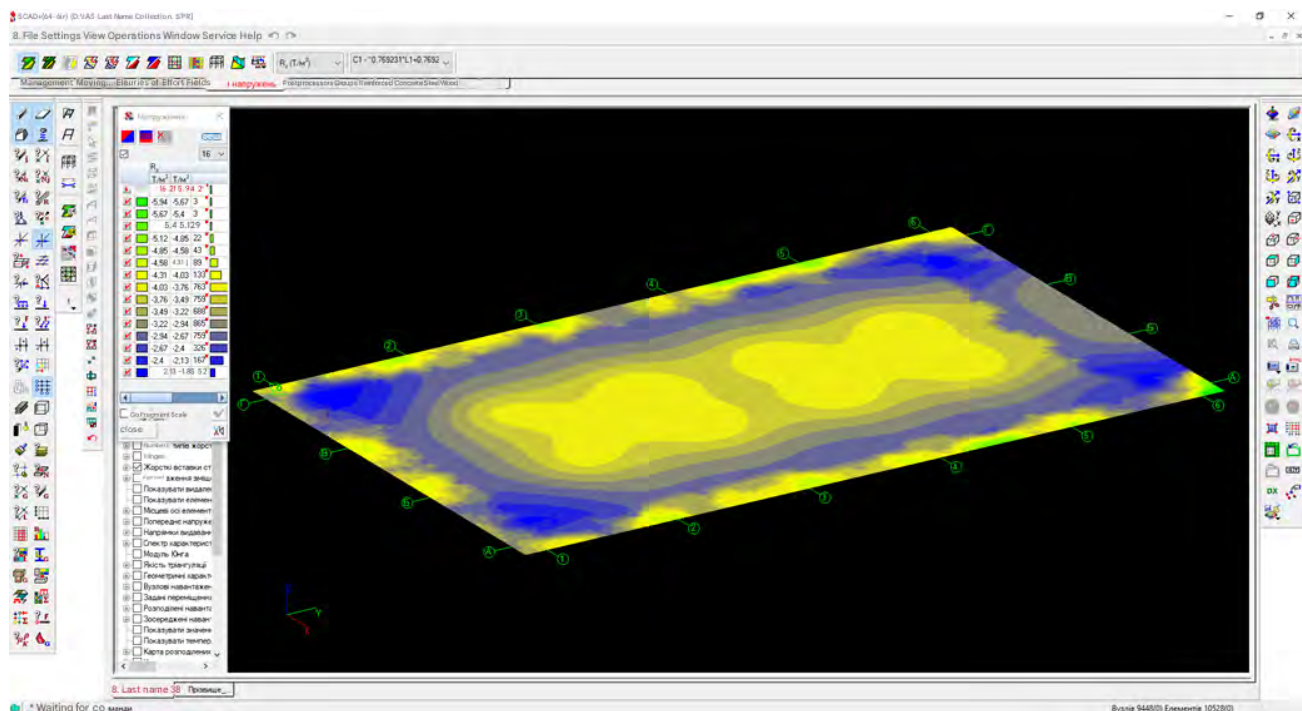


Figure 5.10 – Isopole stresses Rz of the foundation slab, taking into account the adjacent building

Compare the isopoles of the deformed state and stresses Rz of the foundation slab, taking into account the neighboring building and without it. Draw conclusions.

Perform the necessary selection and calculation iterations to meet the requirements of regulatory documents.

Modeling of the soil base with volumetric elements.

Return to the calculation diagram through the project management screen.

Remove the bedding coefficients previously assigned to the foundation plate plates using the bedding coefficient assignment function, assigning their value equal to 0 for all plates of the foundation slab model

Figure 5.11 – Removing previously assigned bedding coefficients

Remove the previously assigned fasteners of the two nodes of the foundation slab using the "Knitting Node Information" tool. Turn off all the elms on the corresponding nodes and save the changes.

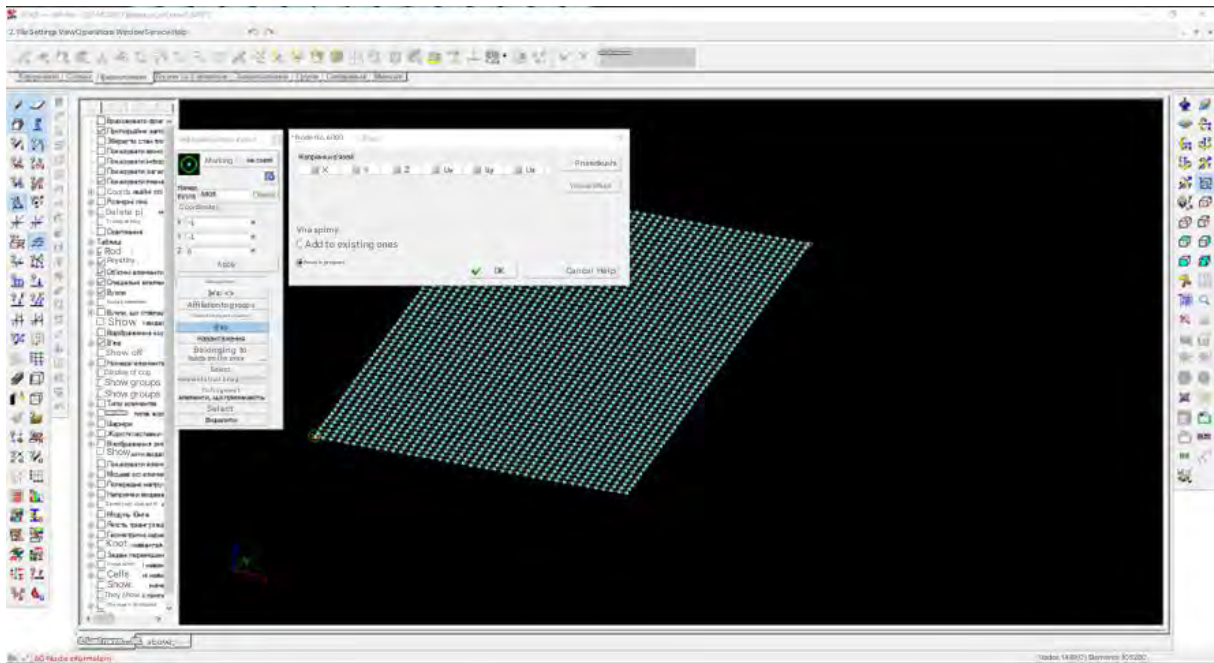


Figure 5.12 – Removing previously assigned bedding coefficients

Select the entire slab using the Mark Items tool on the Groups tab.

On the Diagram tab, select the Copy Diagram tool. Select "Copy visible part of the diagram" from the drop-down menu.

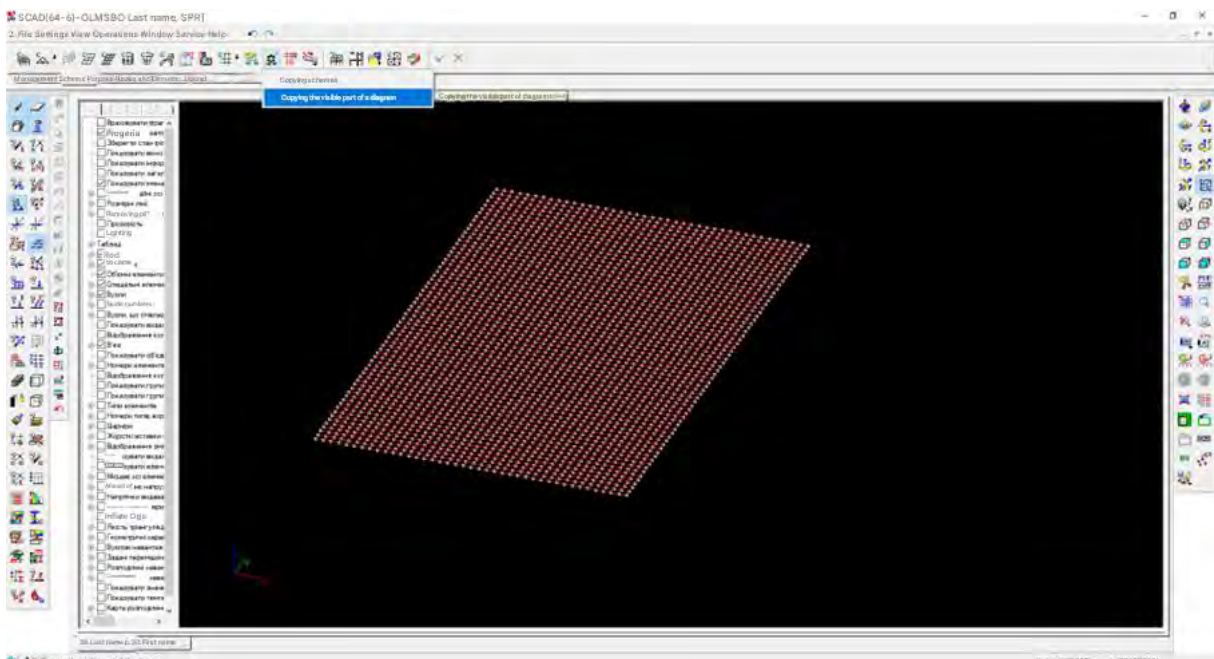


Figure 5.13 – Preparation for the creation of volumetric elements

Configure the copy settings as shown in Figure 5.14 and click OK.



Figure 5.14 – Plate copy options for creating volumetric elements

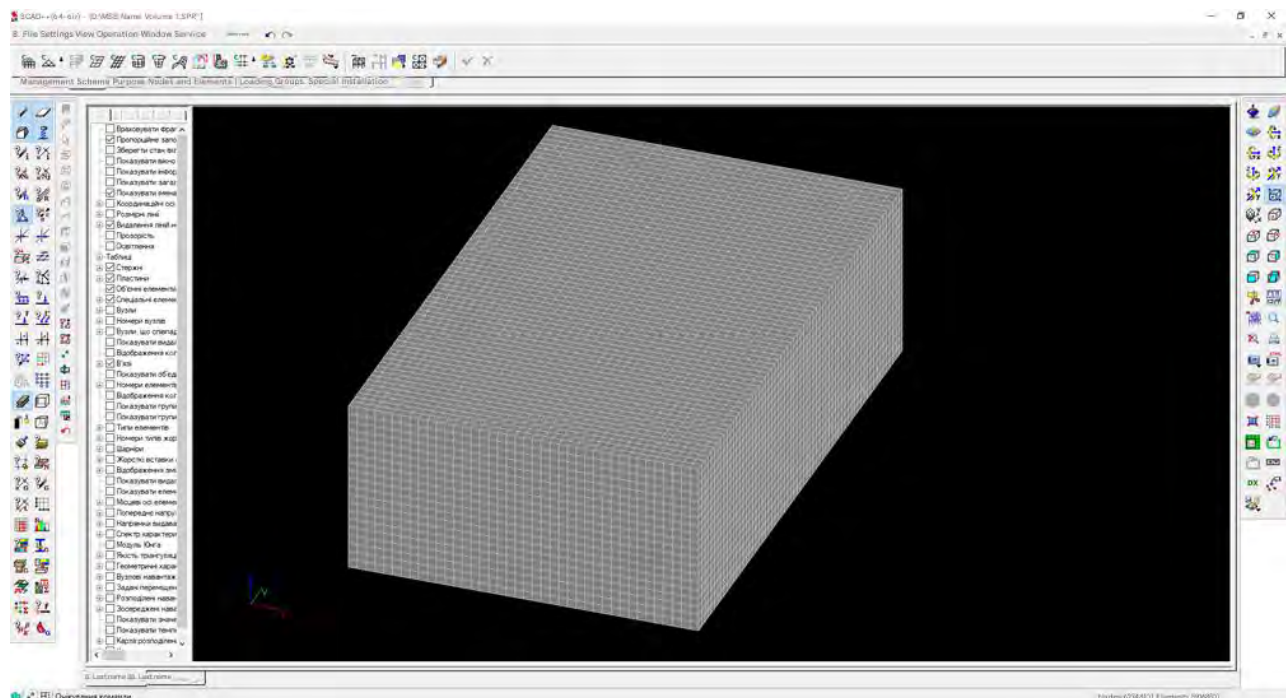


Figure 5.15 – Created soil mass with volumetric elements

Enable the initial display of the design diagram and the display of element profiles in the filter panel.

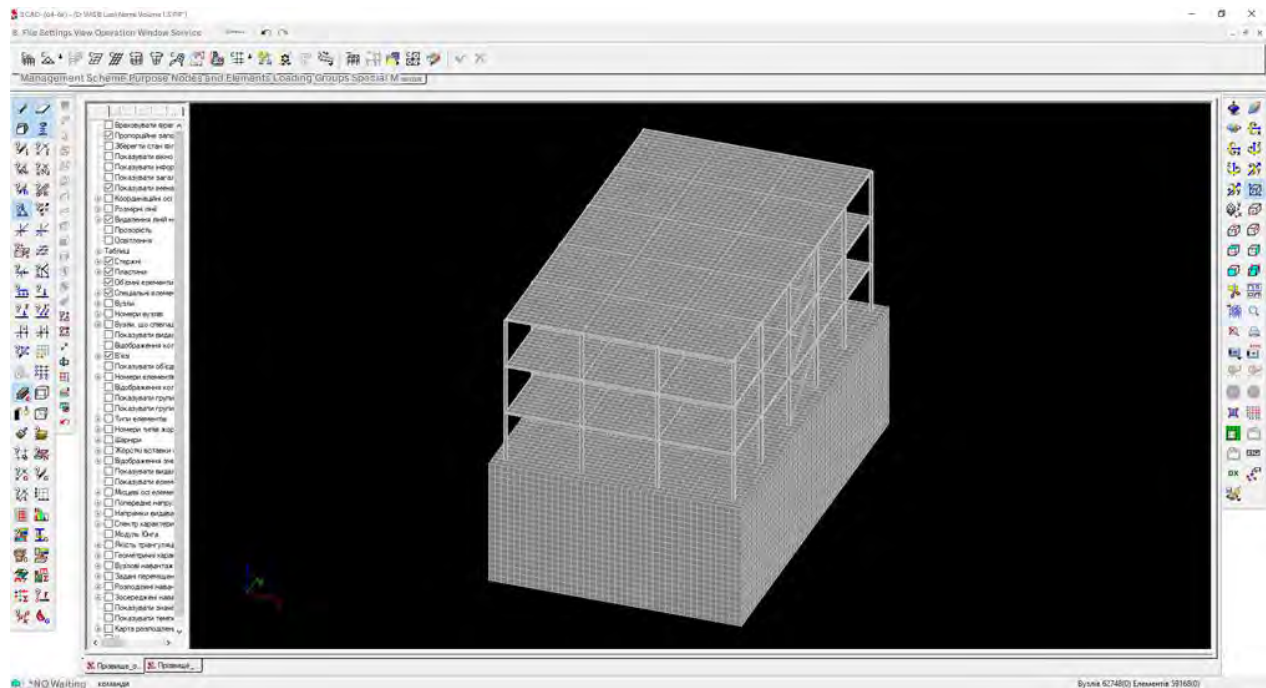


Figure 5.16 – Initial display of the design scheme

Fragment the top edge of the soil mass using the render panel and enable axis mapping.

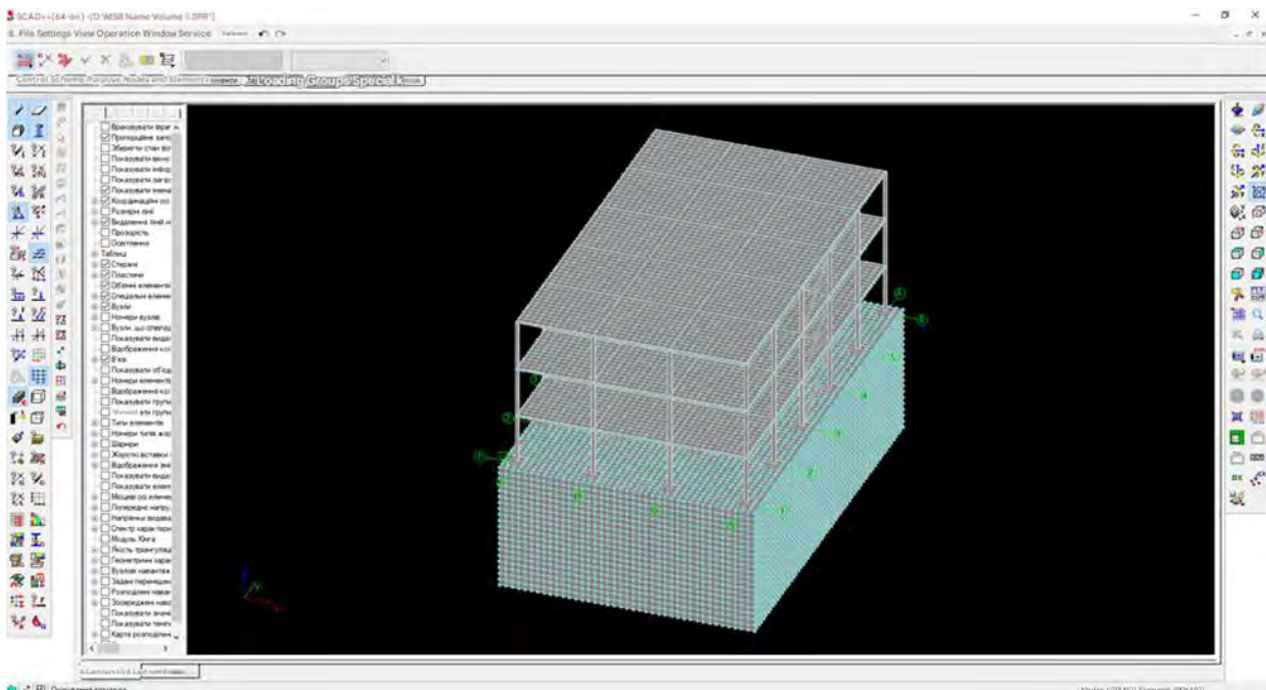


Figure 5.17 – Fragmentation of the upper edge of the soil mass

Using the Plate Element Input tool, create models of individual foundations for columns measuring 1 m × 1 m with the rigidity of a foundation slab.

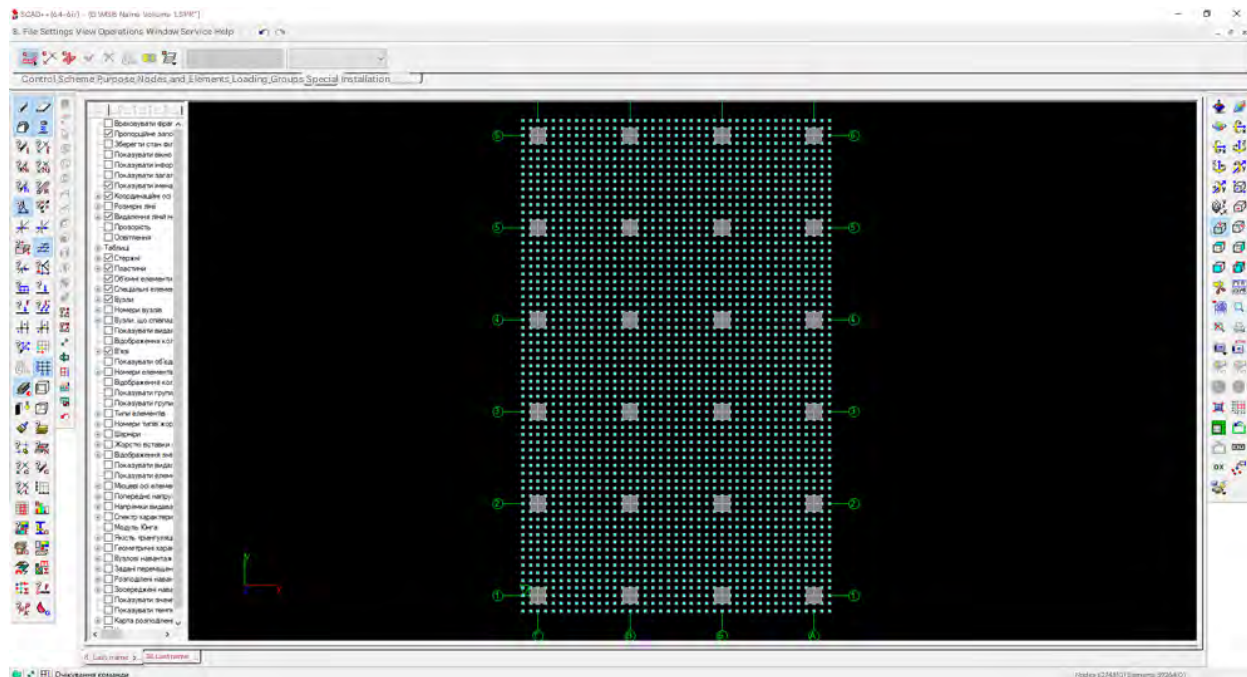


Figure 5.18 – Creating separate foundations for columns

Return to the display of the original scheme to control the correctness of creating foundations for columns.

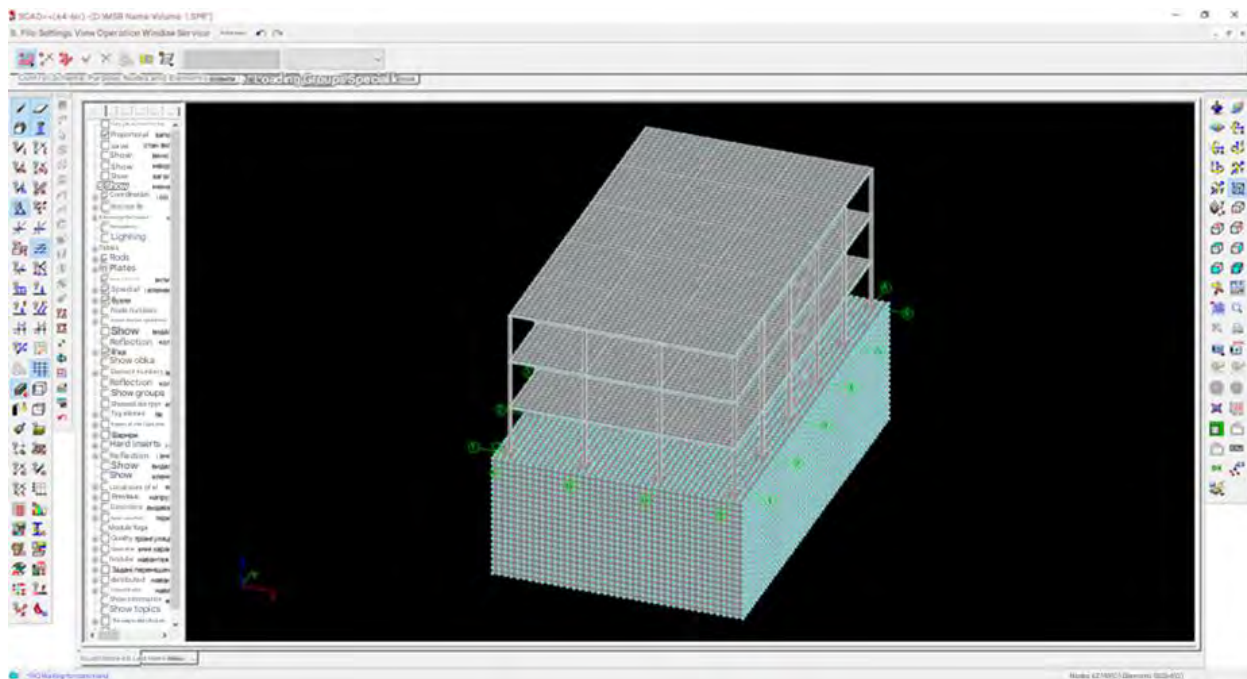


Figure 5.19 – Initial display of the scheme with foundations for columns

Apply the stiffness settings to the created volumetric elements. We take the volumetric weight, modulus of elasticity and Poisson's ratio from the CROSS program for IGE 1. The rest of the soil layers are conditionally ignored to simplify the calculation scheme.

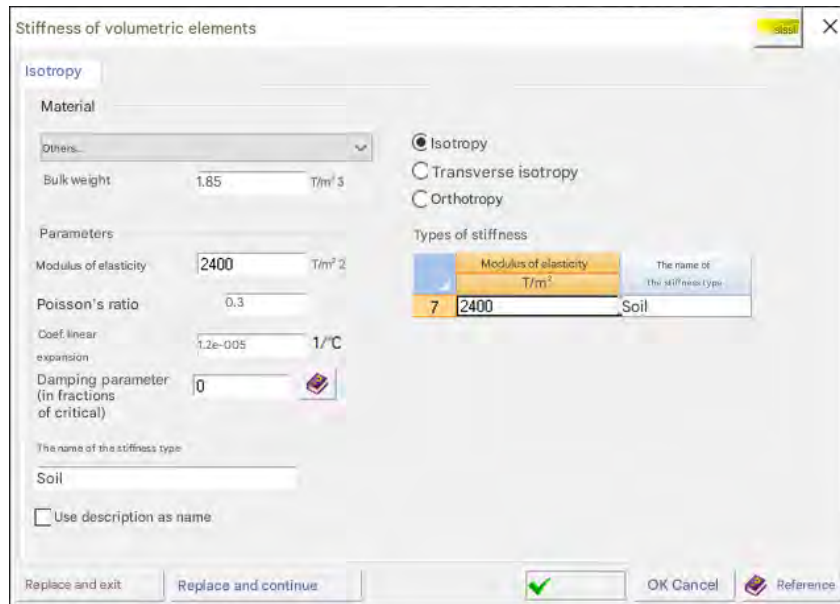


Figure 5.20 – Assignment of stiffness parameters for the soil model

Select all the volumetric elements and press the confirm button.

Next, we set the conditions for the adjoining of the soil massif.

We cut off the lower horizontal plane of the soil mass and assign fixing to all nodes of this plane along the Z axis.

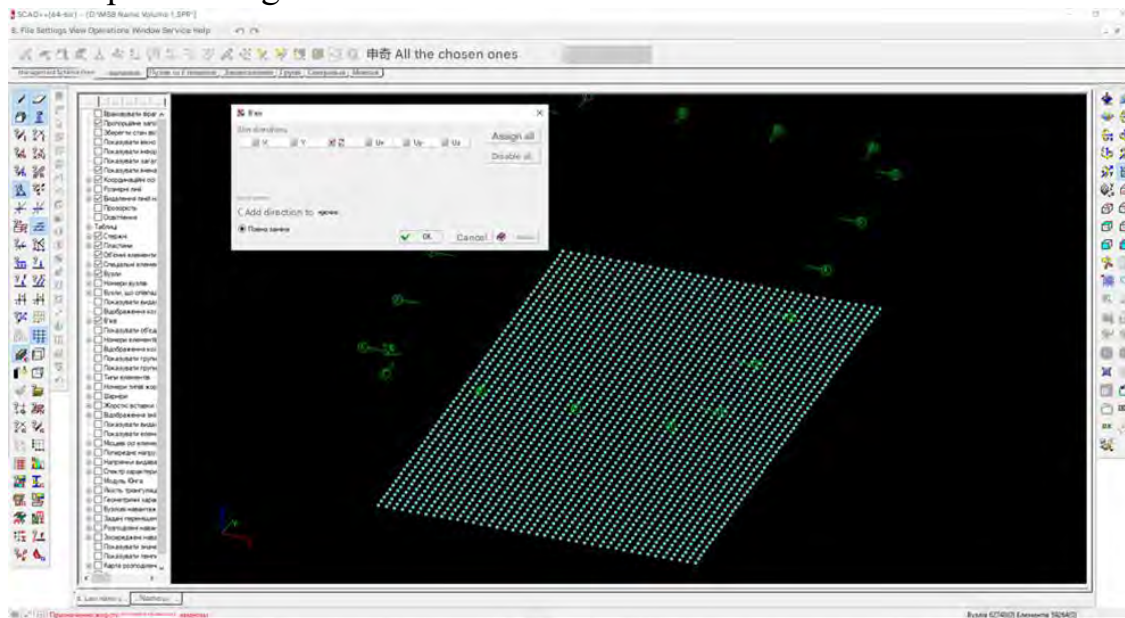


Figure 5.21 – Assignment of boundary conditions of the soil mass

We return to the original display of the diagram and assign fixings for the left and right faces along the X axis, and for the front and rear faces along the Y axis, fragmenting them alternately. When assigning boundary conditions to side faces, rearrange the checkbox to "Add direction to existing".

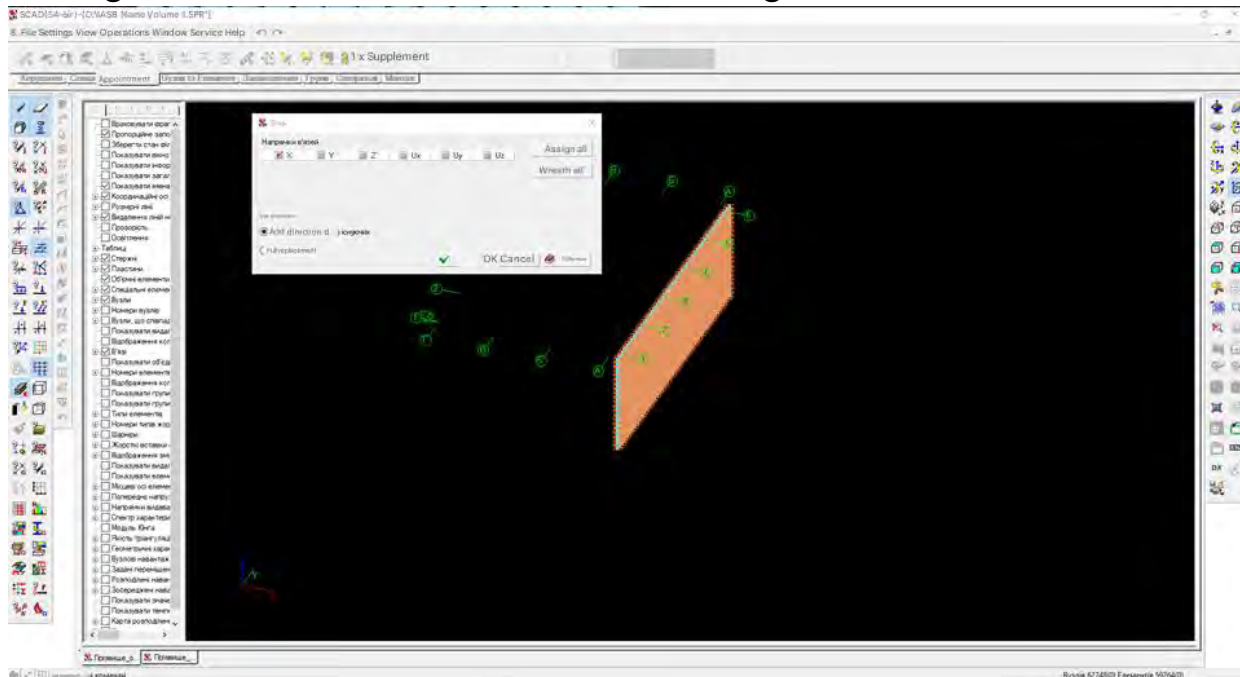


Figure 5.22 – Assignment of boundary conditions of the soil mass

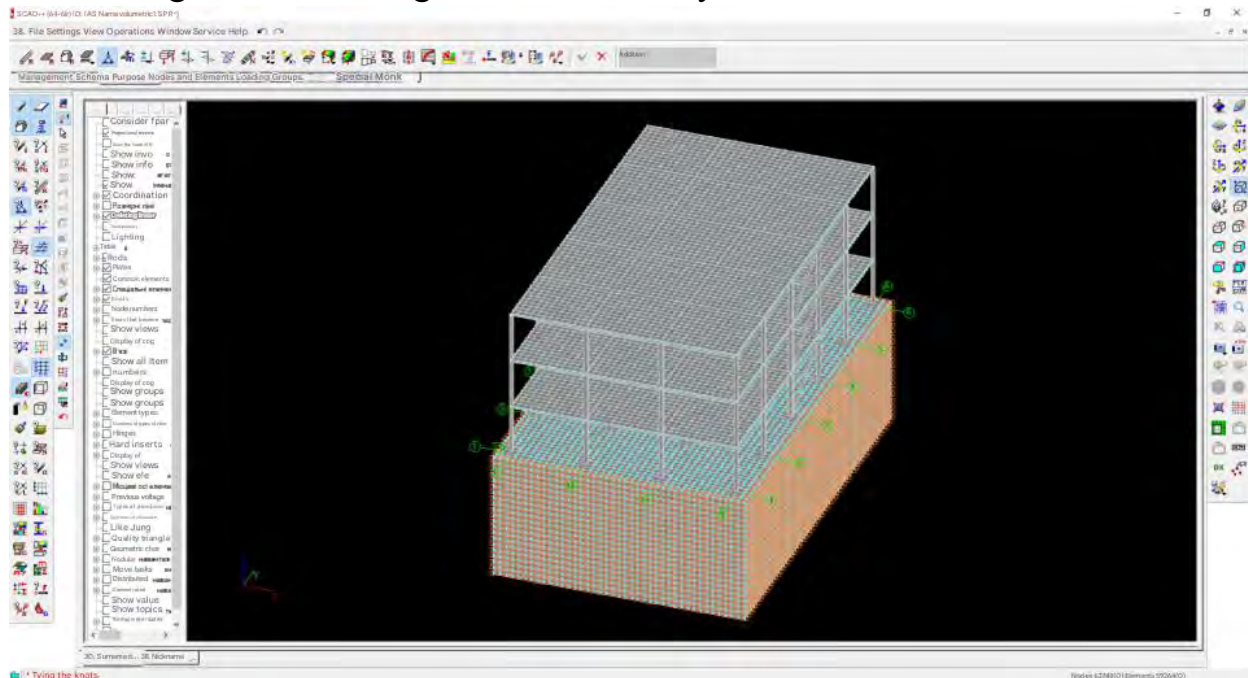


Figure 5.23 – Initial display of the circuit with assigned boundary conditions

We perform a linear calculation. But let's move on to graphical analysis.

We analyze possible changes in the selection of steel and reinforced concrete elements of the building frame.

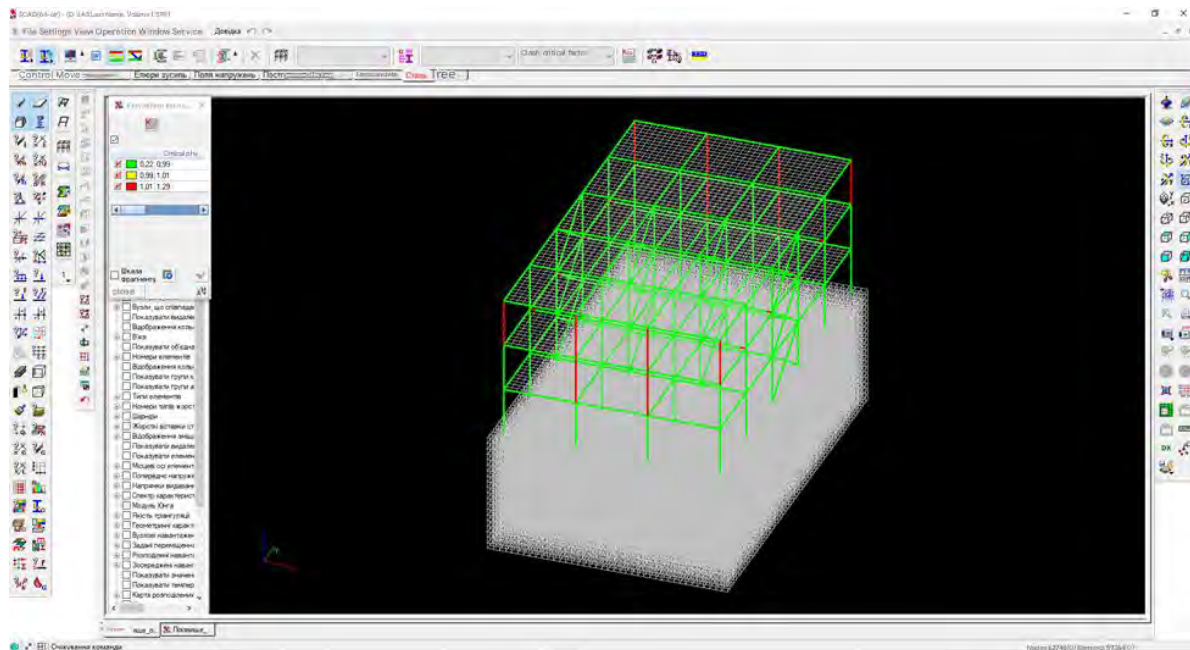


Figure 5.24 – Results of the calculation of the bearing capacity of the previously selected elements of the steel frame of the building, taking into account the joint operation of the "building-foundation" system

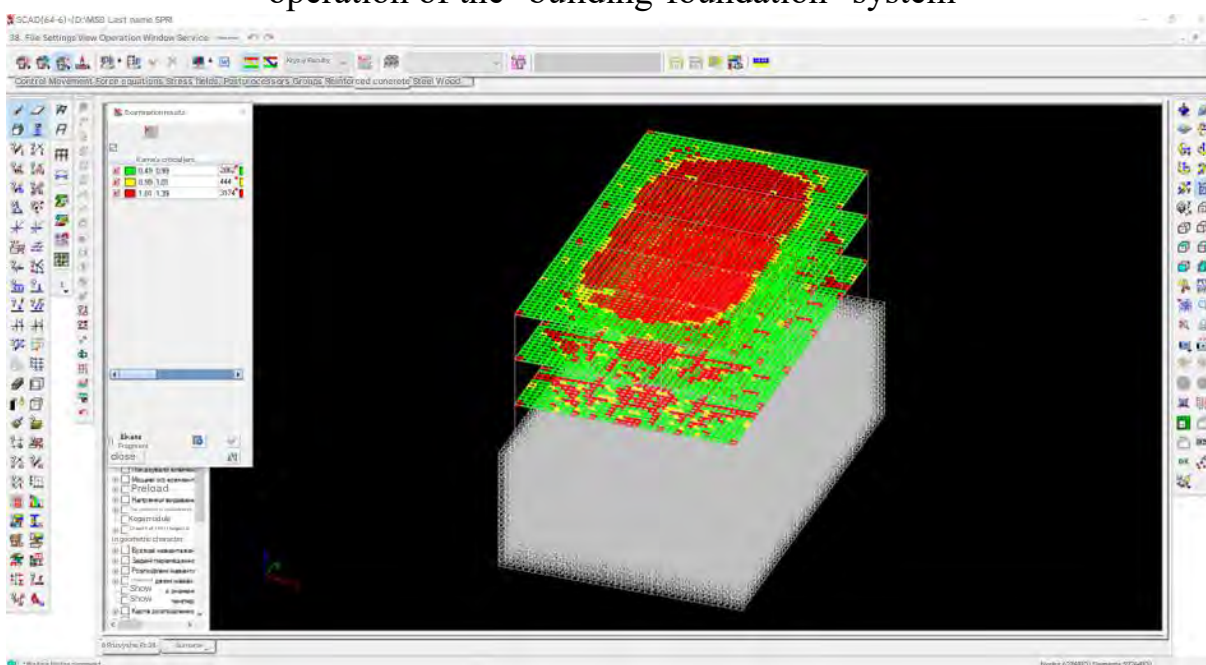


Figure 5.25 – Results of calculating the load-bearing capacity of reinforced concrete floors and building coverings, taking into account the joint operation of the "building-foundation" system

Analyze the results of the calculation of displacements on the "Displacements" tab for the building as a whole and separately for the foundations of the building.

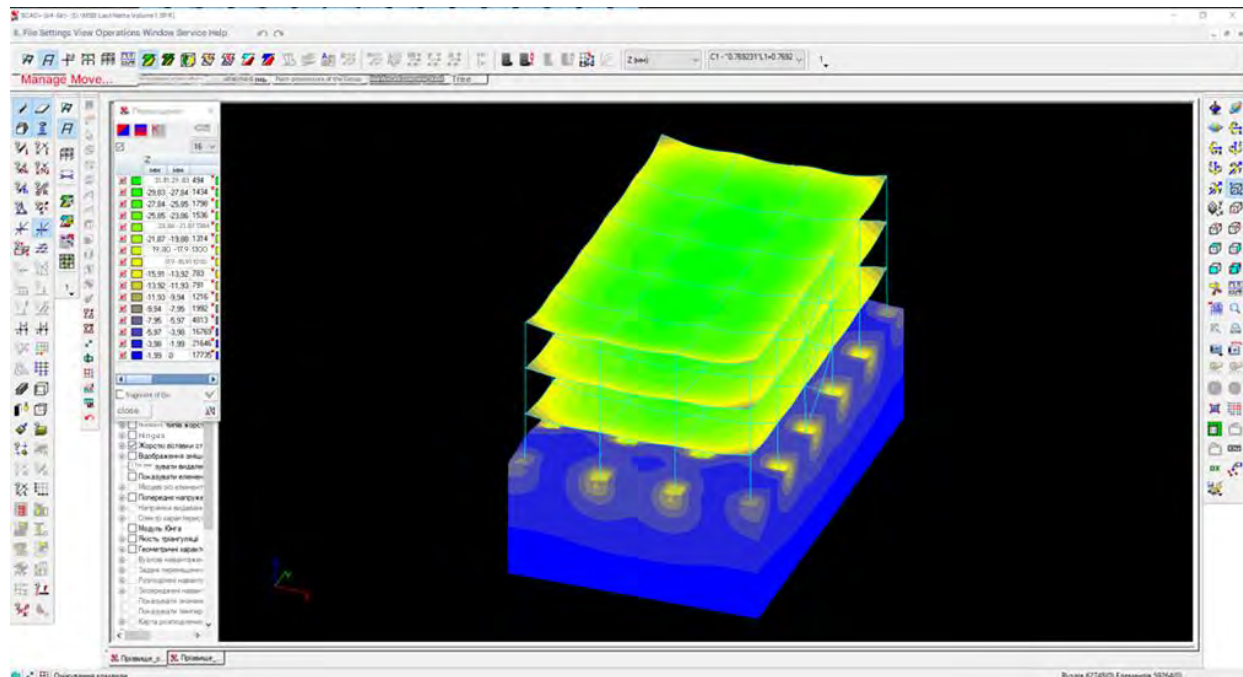


Figure 5.26 – Isopoles of deformations of the "foundation-foundation-structure" system from the combination of forces C1

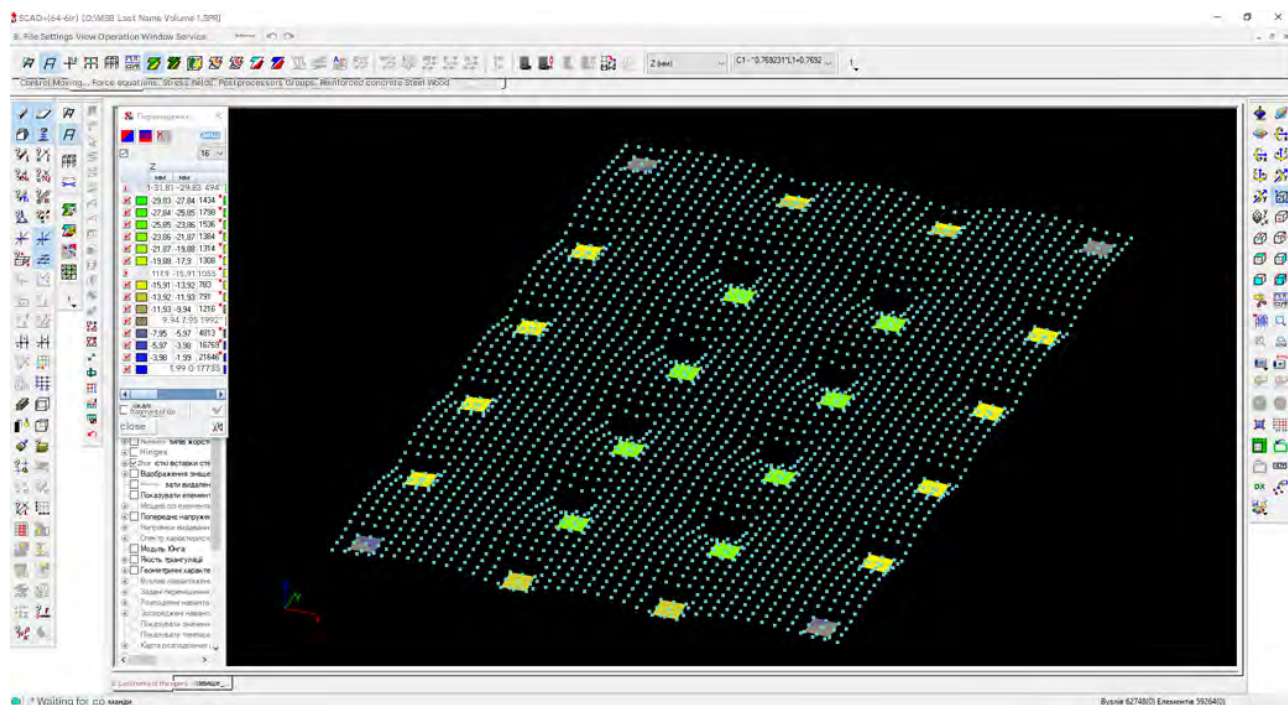
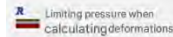


Figure 5.27 – Isopoles of deformations of individual foundations for columns from the combination of forces C1

Calculate the unevenness of deformation of adjacent foundations using the formula, $\frac{\Delta S}{L}$ compare it with the permissible according to DBN B2.1.10-2018. Help on deformation limit values can be found in the REQUEST program.

Make a conclusion about the compliance of the maximum absolute S and the relative difference of subsidence (unevenness) with the requirements of DBN B2.1.10-2018.

Draw a conclusion about the effect of uneven subsidence on the redistribution of forces in the building frame by analyzing Figures 5.24–5.27.

Using the REQUEST program, the page  and the formula for calculating the area of the foundation $A = \frac{N}{R}$; by iteration, calculate the correct area of the foundation for each column. Get the force N from the design scheme using the tool "Element Information – DCW". Set the soil conditions in accordance with the task (previously specified in CROSS and REQUEST when calculating the bed coefficients).

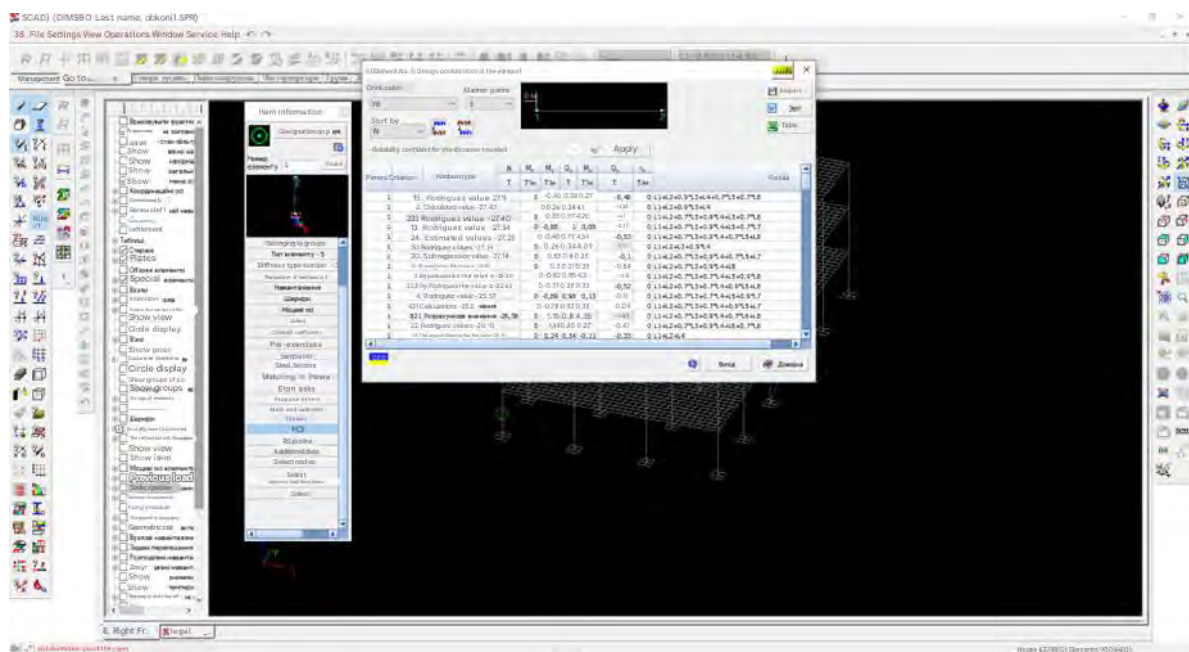


Figure 5.28 – Display of the column DCS to determine the loads on the foundation

Change the dimensions of the foundations in the calculation scheme according to the calculation results. Perform a linear calculation and analyze the results similarly to Figures 5.25–5.27. Achieve the transition of the sections of steel structures assigned according to the previous calculations and reinforcement of reinforced concrete floors and coatings without a new selection of elements by leveling uneven subsidence by correctly calculating and assigning the dimensions of individual foundations for columns.

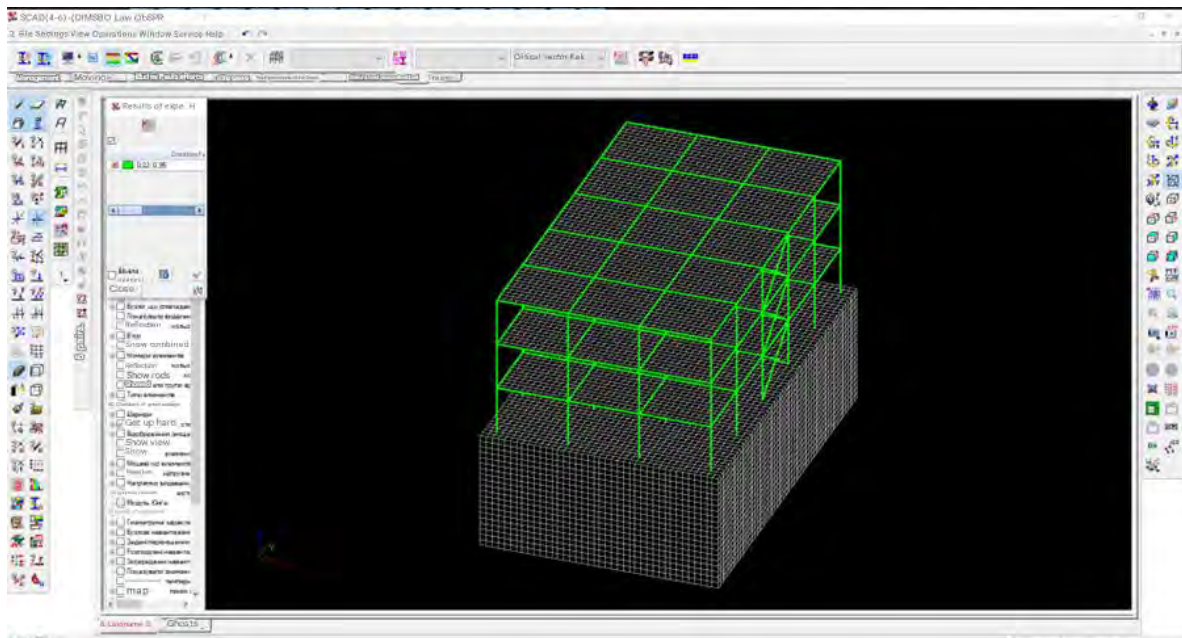


Figure 5.29 – Normalization of the frame operation due to the correct assignment of the dimensions of individual foundations for columns

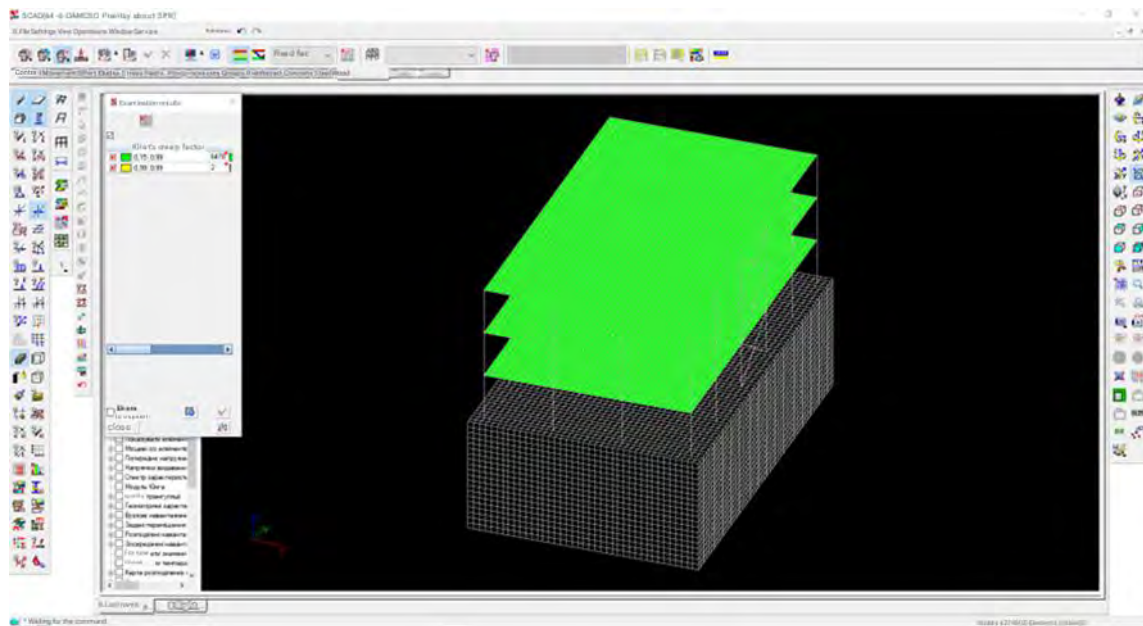


Figure 5.30 – Normalization of the frame operation due to the correct assignment of the dimensions of individual foundations for columns

Tasks for independent work on topic 5.

Perform the calculation of the stress-strain state and the selection of the necessary sections and reinforcement for the previously preserved version with a fully reinforced concrete frame. Compare the results of the calculation of the stress-strain state (in particular, the subsidence of foundations) with the results for a mixed frame. Explain the results obtained.

TOPIC 6

ANALYSIS OF STRESS DISTRIBUTION IN A LINEARLY DEFORMABLE SOIL LAYER MODEL

For a more visual analysis of the impact of the resolution of the soil model, we will replace the type of foundation of the building with a slab foundation. First, we will remove the previously created models of individual foundations for columns. For the convenience of working in the filter panel, we will disable the display of volumetric elements on the working field.

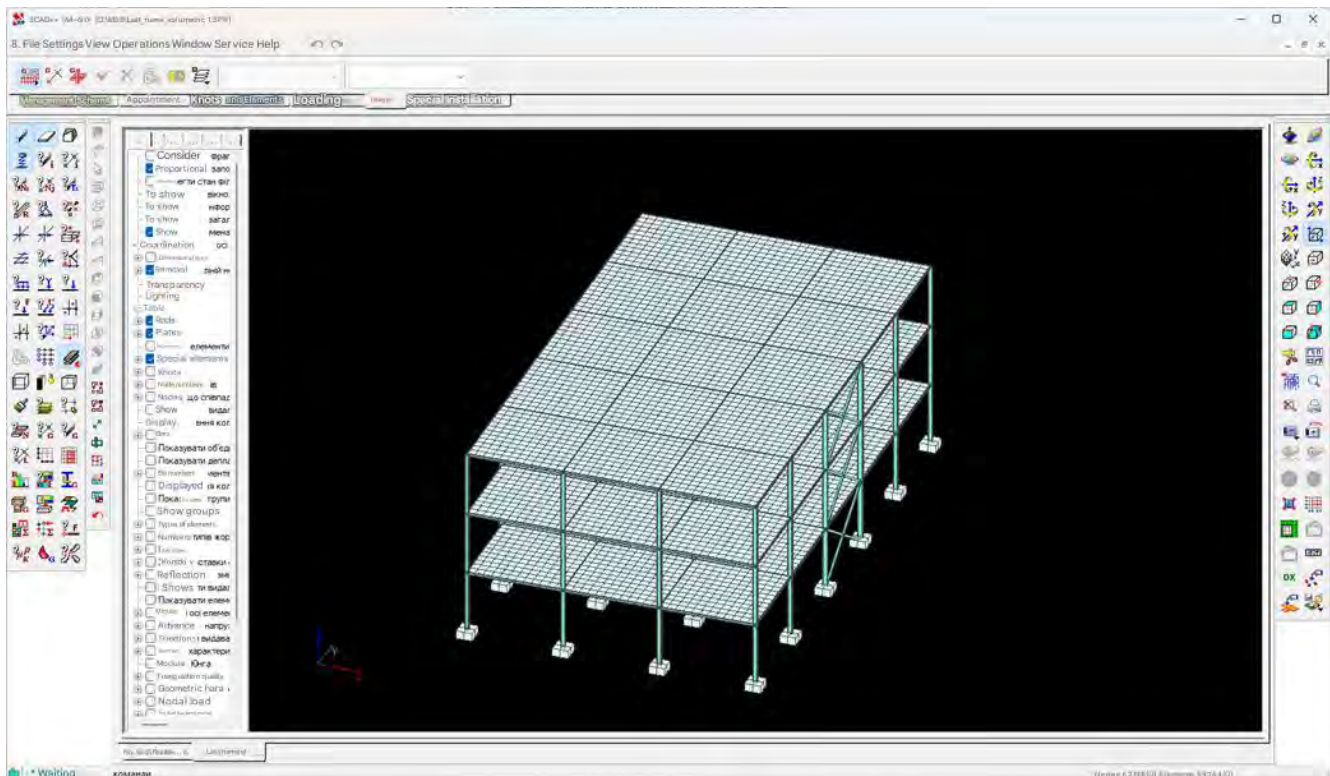


Figure 6.1 – Design scheme with the display of volumetric elements (soil mass) disabled

By returning the diagram to a convenient position for selecting foundations using the "Remove Elements" tool, select and delete the models of the foundations of the building (Fig. 6.2). After selecting the corresponding elements, press the confirm or "Enter" button on the keyboard. Return to the original display of the diagram (using the visualization panel) and enable the display of volumetric elements (soil mass) in the filter panel. For the convenience of work, we perform a cross-section with the plane of the HOU along the upper plane of the soil mass (using the function already known to us on the visualization panel). Go to the "Nodes and Elements" tab and run the "Add Plates" tool.

To ensure the joint operation of the "foundation-foundation-structure" system, we crush the foundation slab, taking into account the intermediate nodes (bases of columns and nodes of the soil mass model) using the appropriate tool on the "Nodes and Elements" tab. Click the corresponding button  on the toolbar and select the previously created plate and confirm the operation.

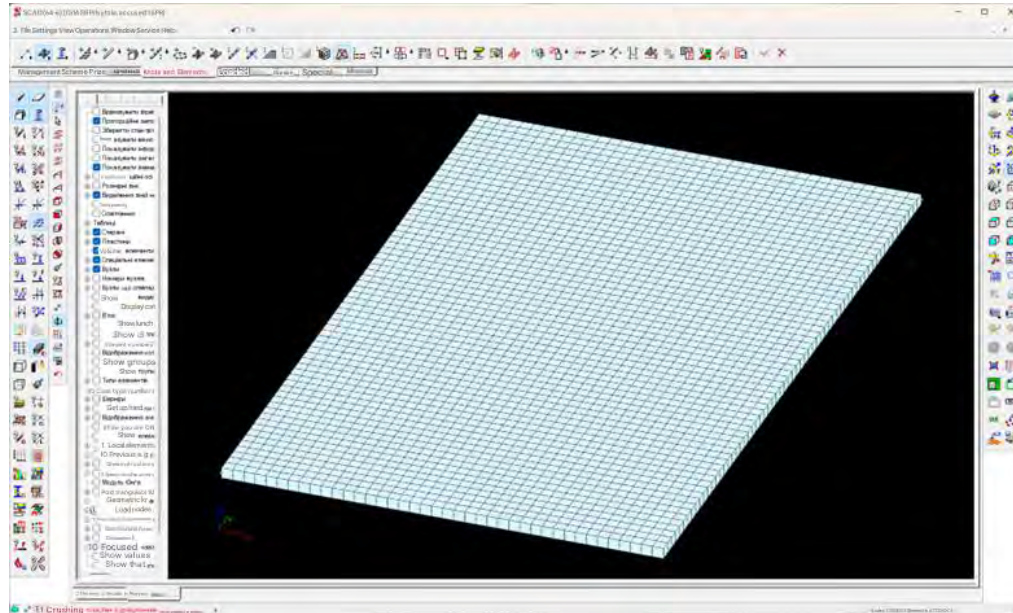


Figure 6.4 – The result of crushing the foundation slab model for linking with other elements of the "foundation-foundation-structure" system model

Using the visualization panel, we return to the original display of the design scheme. Go to the project management screen and perform the calculation.

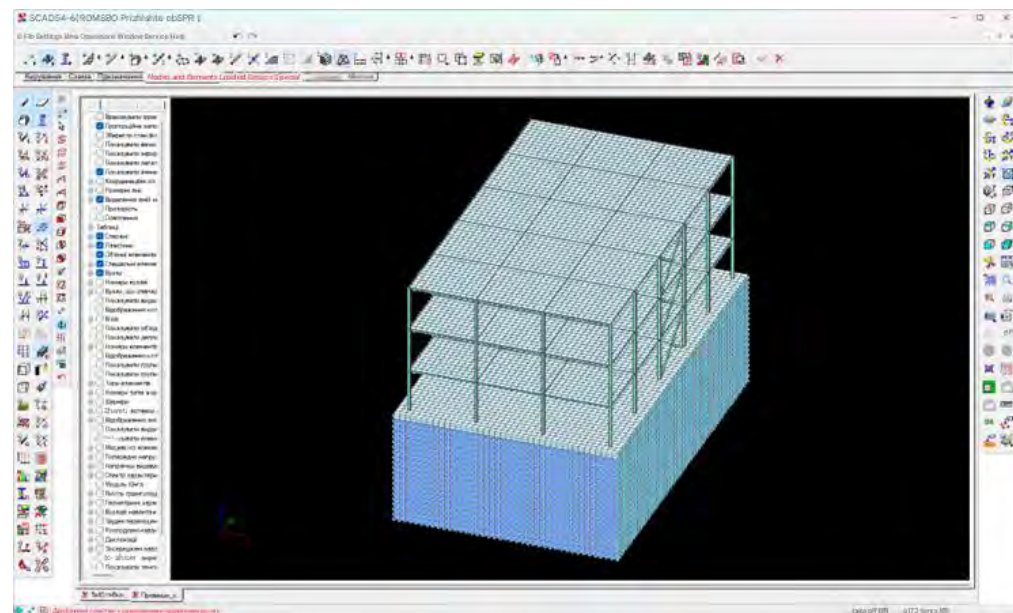


Figure 6.5 – Full display of the design scheme with a modified foundation

We go to the mode of graphical analysis of the results to the "Displacement" tab, where we record the magnitude and nature of the subsidence of the slab foundation of the building from the combination of forces created for the calculation of the slab foundation using the coefficients of the bedding. For the convenience of analysis, we perform a cross-section with the plane of the HOU along the plane of the slab. We determine the average subsidence S_{ser} .

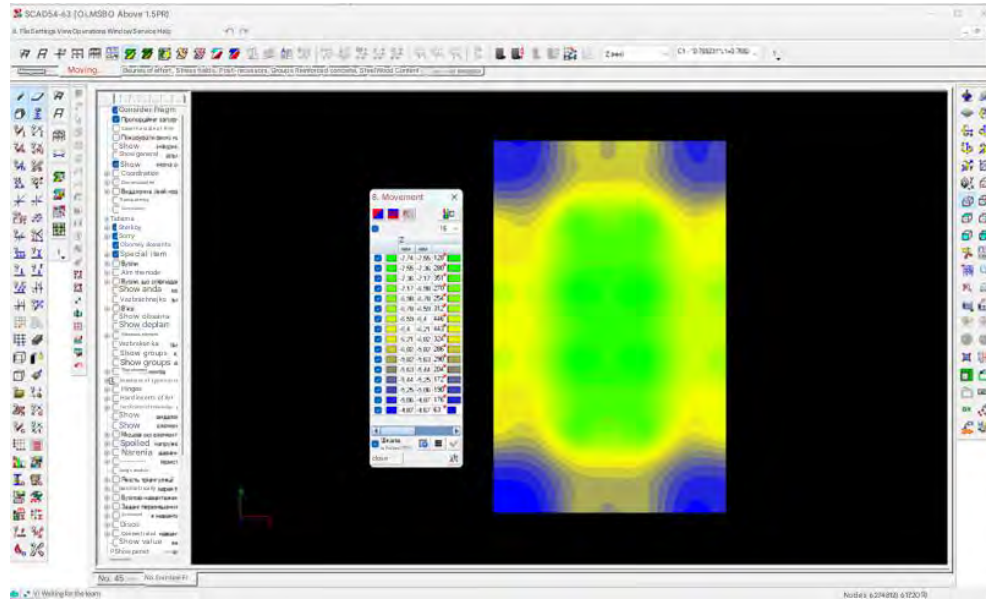


Figure 6.6 – Analysis of the subsidence of the foundation slab on the soil model represented by volumetric elements

Go to the "Stress fields" tab and determine the maximum values of the moments M_x and M_y in the slab foundation for this design scheme.

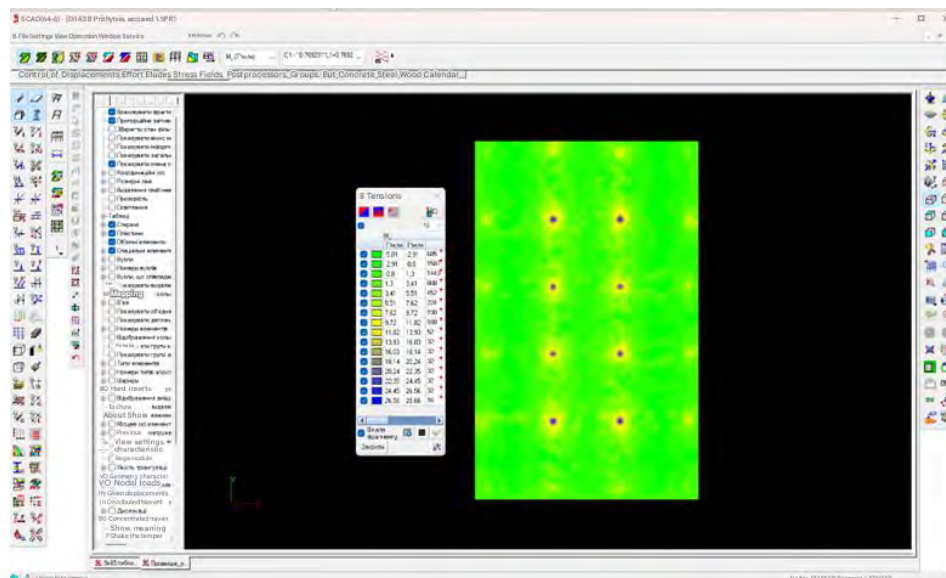


Figure 6.7 – Isopole of moments M_x in a slab foundation

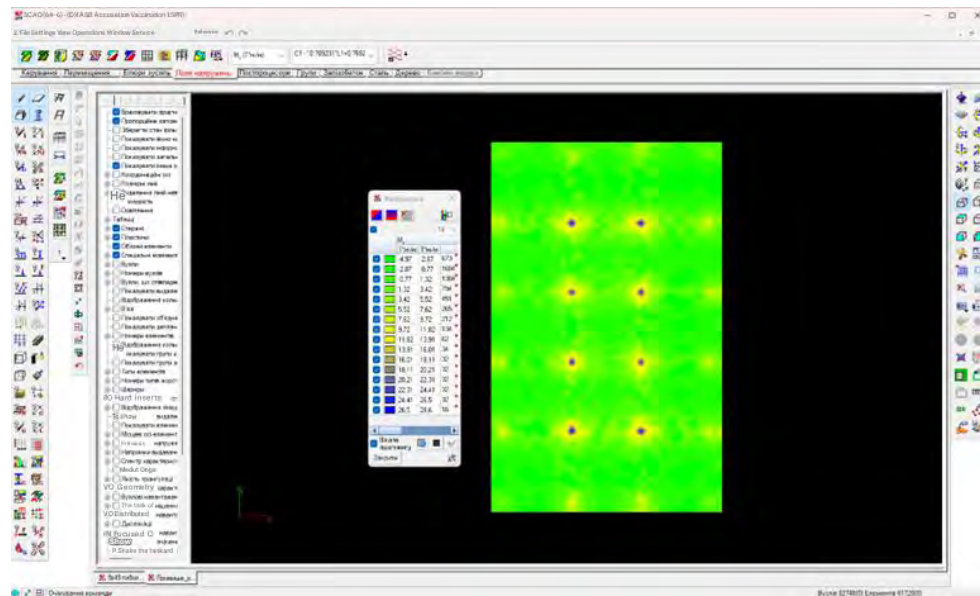


Figure 6.8 – Isopoles of moments M_y in a slab foundation

Write the results in Table 6.1. Save the calculation option under the first cipher of Table 6.1. Copy the file and save it under the second cipher of the same table. Then work with it, modifying the calculation scheme for the distribution angle $\alpha = 25^\circ$ (choose the optimal distribution angle according to [2]).

Table 6.1 – Results of the analysis of the influence of boundary conditions of the model of the soil basis of finite resolution.

Schema code	Average subsidence, mm	Maximum torques +/-, kN/m
Last name HB=5 a=0		
Last name HB=5 a=25		
Last name HB=5 a=45		

Through the project management screen, we return to the mode of building the design scheme and extend the boundaries of the soil base model to fulfill the angle condition from the distribution $\alpha = 25^\circ$. The first step is to remove the previously set boundary conditions (elms in the nodes of the soil model). Uncheck all the boxes in the elm dialog box and, having selected the entire soil massif, click the confirm button.

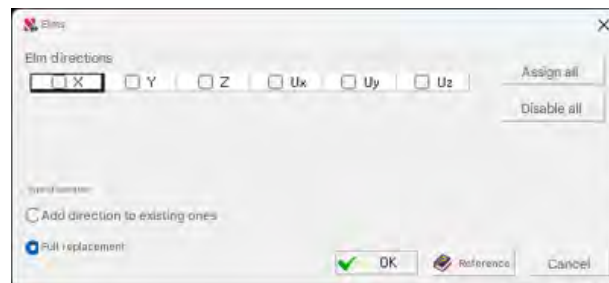


Figure 6.9 – Disabling Elms in the Dialog Box

For further convenience of working with the soil mass model, turn off the display of rods and plates on the filter panel.

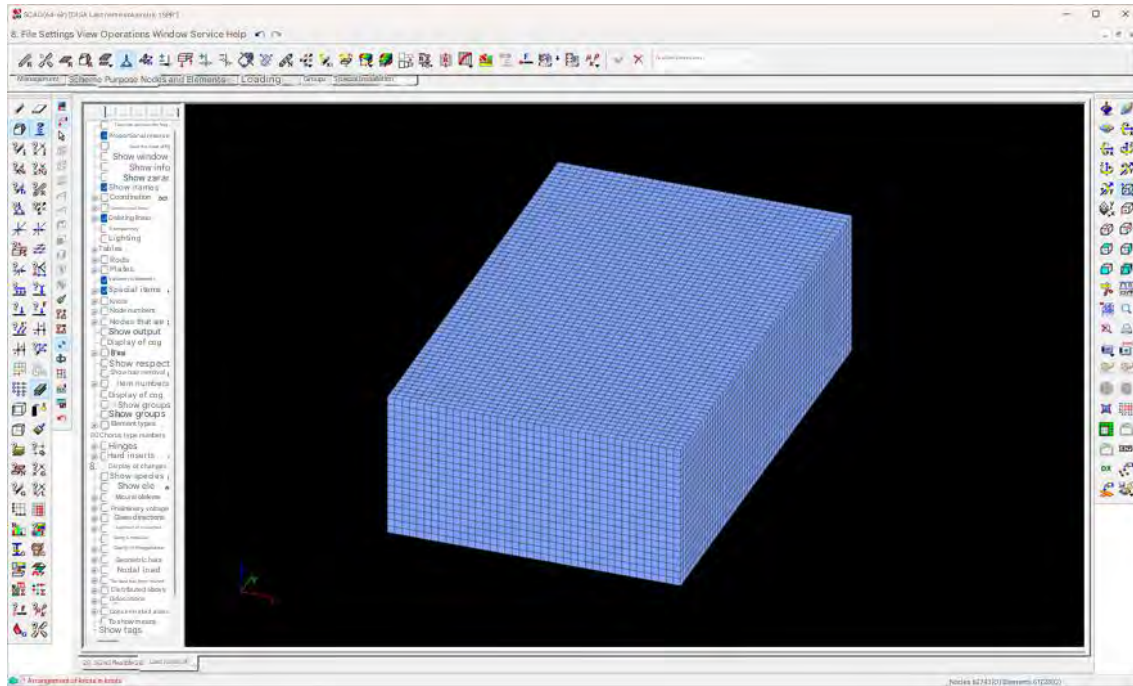


Figure 6.10 – Soil model on the working field of the program

We make a projection of the soil model onto the plane of the HOU and turn on the display of the coordination axes. We calculate the required size of the expansion of the soil model in the plan in accordance with its geometry with the task of achieving a distribution angle α of 25° using geometric construction, or trigonometric dependencies, for example, as in Figure 6.11.

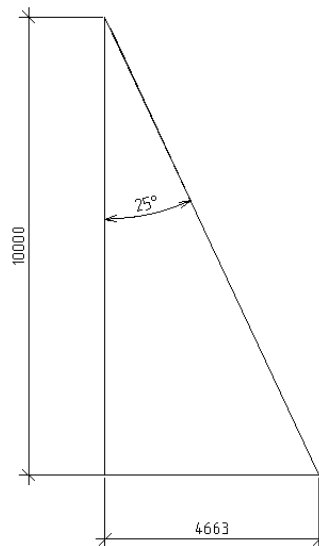


Figure 6.11 – Scheme for calculating the geometry of the soil massif.

We round the obtained value in accordance with the value of the triangulation step of your scheme and copy or complete the required number of elements of the soil massif. We return to the initial display of the scheme, check the result and set the boundary conditions (elms in the nodes on the edges of the soil massif) similar to the actions discussed in Topic 5.

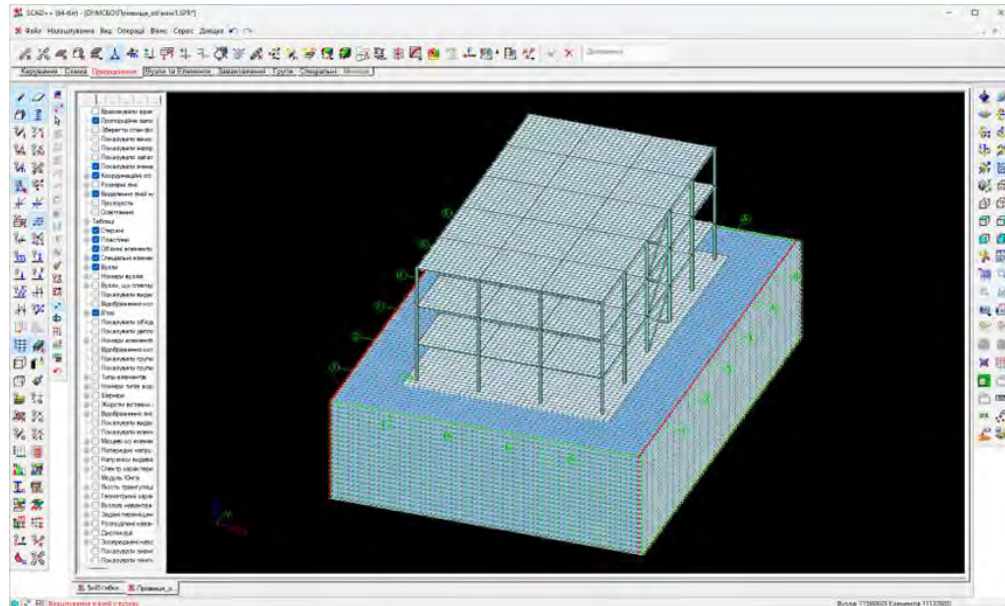


Figure 6.12 – The calculation scheme is prepared for calculation

On the project management screen, perform a linear calculation and switch to the graphical analysis mode.

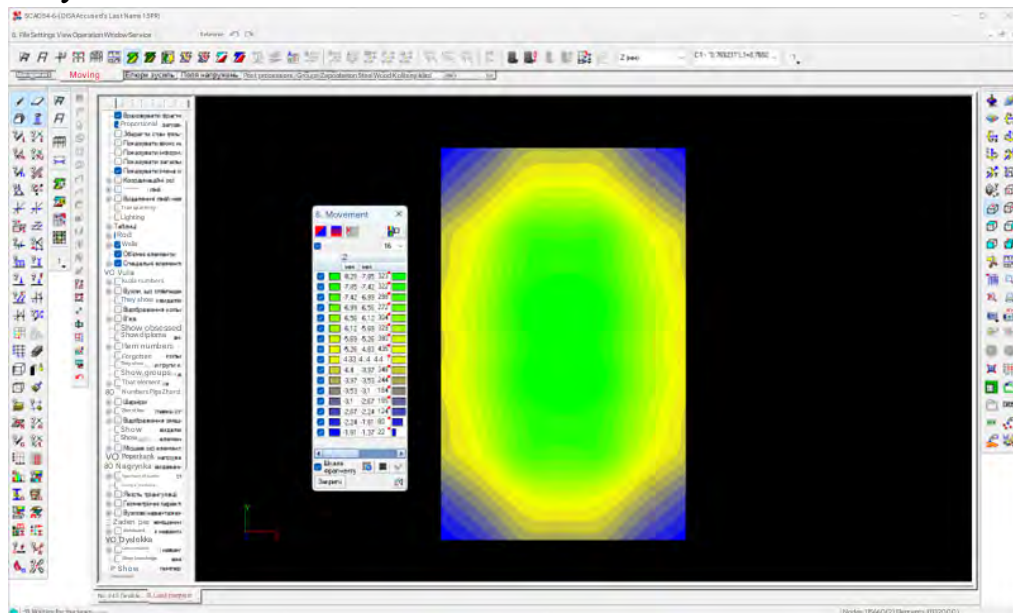


Figure 6.13 – Isopoles of displacements (subsidence) of the foundation slab

The screenshot shows the SAP2000 software interface. The main window displays a stress field plot for a reinforced concrete slab. The plot is color-coded, with a central blue region indicating high stress. A 'Stress' dialog box is open, showing the 'Tension' tab. The dialog box contains a table of stress values for various elements. The table has two columns: 'Tension' and 'Tension (ksi)'. The values are as follows:

Tension	Tension (ksi)
1.52	1.52
1.52	1.52
1.52	1.52
1.52	1.52
1.52	1.52
1.52	1.52
1.52	1.52
1.52	1.52
1.52	1.52

The interface also includes a menu bar, a toolbar, and a command line.

The screenshot shows the ANSYS Workbench environment. The main workspace displays a 3D model of a mechanical part, likely a bracket or a plate, with a green mesh. A 'Tension' plot is overlaid on the model, indicating the distribution of tensile stress. A 'Tension' table is open, showing the following data:

Element	Min	Max	Avg
1	1.7	0.4	1.72
2	0.4	0.4	0.4
3	0.4	0.4	0.4
4	0.4	0.4	0.4
5	0.4	0.4	0.4
6	0.4	0.4	0.4
7	0.4	0.4	0.4
8	0.4	0.4	0.4
9	0.4	0.4	0.4
10	0.4	0.4	0.4
11	0.4	0.4	0.4
12	0.4	0.4	0.4
13	0.4	0.4	0.4
14	0.4	0.4	0.4
15	0.4	0.4	0.4
16	0.4	0.4	0.4
17	0.4	0.4	0.4
18	0.4	0.4	0.4
19	0.4	0.4	0.4
20	0.4	0.4	0.4
21	0.4	0.4	0.4
22	0.4	0.4	0.4
23	0.4	0.4	0.4
24	0.4	0.4	0.4
25	0.4	0.4	0.4
26	0.4	0.4	0.4
27	0.4	0.4	0.4
28	0.4	0.4	0.4
29	0.4	0.4	0.4
30	0.4	0.4	0.4
31	0.4	0.4	0.4
32	0.4	0.4	0.4
33	0.4	0.4	0.4
34	0.4	0.4	0.4
35	0.4	0.4	0.4
36	0.4	0.4	0.4
37	0.4	0.4	0.4
38	0.4	0.4	0.4
39	0.4	0.4	0.4
40	0.4	0.4	0.4
41	0.4	0.4	0.4
42	0.4	0.4	0.4
43	0.4	0.4	0.4
44	0.4	0.4	0.4
45	0.4	0.4	0.4
46	0.4	0.4	0.4
47	0.4	0.4	0.4
48	0.4	0.4	0.4
49	0.4	0.4	0.4
50	0.4	0.4	0.4
51	0.4	0.4	0.4
52	0.4	0.4	0.4
53	0.4	0.4	0.4
54	0.4	0.4	0.4
55	0.4	0.4	0.4
56	0.4	0.4	0.4
57	0.4	0.4	0.4
58	0.4	0.4	0.4
59	0.4	0.4	0.4
60	0.4	0.4	0.4
61	0.4	0.4	0.4
62	0.4	0.4	0.4
63	0.4	0.4	0.4
64	0.4	0.4	0.4
65	0.4	0.4	0.4
66	0.4	0.4	0.4
67	0.4	0.4	0.4
68	0.4	0.4	0.4
69	0.4	0.4	0.4
70	0.4	0.4	0.4
71	0.4	0.4	0.4
72	0.4	0.4	0.4
73	0.4	0.4	0.4
74	0.4	0.4	0.4
75	0.4	0.4	0.4
76	0.4	0.4	0.4
77	0.4	0.4	0.4
78	0.4	0.4	0.4
79	0.4	0.4	0.4
80	0.4	0.4	0.4
81	0.4	0.4	0.4
82	0.4	0.4	0.4
83	0.4	0.4	0.4
84	0.4	0.4	0.4
85	0.4	0.4	0.4
86	0.4	0.4	0.4
87	0.4	0.4	0.4
88	0.4	0.4	0.4
89	0.4	0.4	0.4
90	0.4	0.4	0.4
91	0.4	0.4	0.4
92	0.4	0.4	0.4

134

The results are entered in Table 6.1.

We save the next version of the calculation file and perform the described similar procedures, changing the distribution angle to $\alpha=45^\circ$. We repeat the calculation, enter the results of the calculation of subsidence and bending moments in the plate in Table 6.1. We analyze the results obtained, draw conclusions.

Tasks for independent work on topic 6.

Perform the calculation of the subsidence of the foundation according to the model of a linear-deformed medium (in the program "REQUEST") using identical parameters of dimensions, soil characteristics and load. Compare the results of the calculation with the results of determining the subsidence for different distribution angles from Table 6.1.

To perform an examination of reinforced concrete and metal elements of the building frame for three options for the resolution of the soil base. Analyze the critical factors of the elements in all three versions of topic 6 and in the version from topic 5. If necessary, select reinforcement, change the grade of concrete, sections of metal elements, etc. Explain the results obtained.

LIST OF RECOMMENDED SOURCES

1. Investigation of the influence of boundary conditions on the deformability of a model of soil base of limited width in the form of a linearly deformable elastic half-plane [Electronic resource] / Aleksandrovykh V. et al. // IOP Conference Series : Earth and Environmental Science. – IOP Publishing, 2024. – Vol. 1376. – №. 1. – P. 012017. – Electronic text data. – Regime of access: <https://doi.org/10.1088/1755-1315/1376/1/012017>, free (date of access: 10.06.2025). – Header from the screen.

2. Fialko S. Time history analysis of buildings and structures design models in SCAD software on multicore computers [Electronic resource] / Fialko S. // ECMS 2024 : proceedings of the 38th ECMS International Conference on Modelling and Simulation, June 4th-June 7th, 2024 Cracow, Poland / Grzonka Daniel et al., Communications of the ECMS. – Caserta, ECMS, 2024. – Vol. 38. – № 1. – P. 187–193. – Electronic text data. – Regime of access: <https://scs-europe.net/conf/ecms2024/>, free (date of access 10.06.2025). – Header from the screen.

3. Fialko S. Parallel finite element solver for multi-core computers with shared memory [Electronic resource] / Fialko S. // Computers & Mathematics with Applications. – 2021. – Vol. 94. – P. 1–14. – Electronic text data. – Regime of access: <https://doi.org/10.1016/j.camwa.2021.04.013>, free (date of access 10.06.2025). – Header from the screen.

4. Завацький С. В. Автоматизований аналіз напружено-деформованого стану конструкцій в програмному комплексі Structure Cad Office : Навчально-методичний посібник до виконання розрахунково-графічних робіт по курсам: «Будівельна механіка», «Металеві конструкції», «Залізобетонні конструкції» та «Конструкції з деревини» [Електрон. ресурс] / С. В. Завацький, М. М. Корзаченко. – Електрон. текст. дані. – Чернігів : ЧНТУ 2017. – 184 с. – Режим доступу: <http://ir.stu.cn.ua/bitstream/handle/123456789/18140/Посібник%20СКАД%20Завацький%20С.В.%20Корзаченко%20М.М.2018%20ред1.pdf?sequence=1&isAllowed=y>, вільний (дата звернення 01.06.2025). – Назва з екрана.

5. ДБН В.2.1-10-2018. Основи і фундаменти будівель і споруд. Основні положення. – Чинний від 2019–01–01. – Київ : Мінрегіонбуд України, 2018. – 36 с. – Існує електрон. версія. (Режим доступу: https://e-construction.gov.ua/files/new_doc/3019954595758606072/2023-01-20/1a3452ff-56ec-492e-a881-416dcba1604d.pdf, вільний).

6. ДБН В.1.1-45:2017. Будівлі і споруди в складних інженерно-геологічних умовах. Загальні положення. – Чинний від 2017–10–01. – Київ : Мінрегіонбуд, 2017. – 29 с. – Існує електрон. версія. (Режим доступу: https://e-construction.gov.ua/files/new_doc/3019941257964160320/2023-01-20/73c4ae2b-f6c2-4c1f-a53e-87cf6470ae5e.pdf, вільний).

7. ДБН В.1.2-5:2007. Науково-технічний супровід будівельних об'єктів. – Чинний від 2008–01–01. – Київ : Мінрегіонбуд, 2007. – 16 с. – Існує електрон.

версія. (Режим доступу: https://e-construction.gov.ua/files/new_doc/3074778358307882794/2023-04-06/d169c363-9531-4a87-8b10-1e706a6d2221.pdf, вільний).

8. ДБН В.1.2-14:2018. Система забезпечення надійності та безпеки будівельних об'єктів. Загальні принципи забезпечення надійності та конструктивної безпеки будівель і споруд. – Чинний від 2019–01–01. – Київ : Мінрегіонбуд, 2018. – 30 с. – Існує електрон. версія. (Режим доступу: https://e-construction.gov.ua/files/new_doc/3022057264165946908/2023-01-24/70ffd643-c8d7-47d7-92d9-47d0bf543e46.pdf, вільний).

Електронне навчальне видання

Методичні рекомендації
до проведення практичних занять
та організації самостійної роботи
з навчальної дисципліни

«МОДЕЛЮВАННЯ СИСТЕМИ «БУДІВЛЯ-ОСНОВА»

*(для здобувачів другого (магістерського) рівня вищої освіти всіх форм
навчання зі спеціальності G19 – Будівництво та цивільна інженерія)*

(Англ. мовою)

Укладачі : **АЛЕКСАНДРОВИЧ** Вадим Анатолійович,
ГАВРИЛЮК Ольга Володимирівна

Відповідальний за випуск *В. А. Александрович*
За авторською редакцією
Комп'ютерне верстання *В. А. Александрович*

План 2025, поз. 584М

Підп. до друку 10.09.2025. Формат 60 × 84/16.
Ум. друк. арк. 8,0.

Видавець і виготовлювач:
Харківський національний університет
міського господарства імені О. М. Бекетова,
вул. Чорноглазівська, 17, Харків, 61002.
Електронна адреса: office@kname.edu.ua
Свідоцтво суб'єкта видавничої справи:
ДК № 8386 від 14.07.2025.