

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.

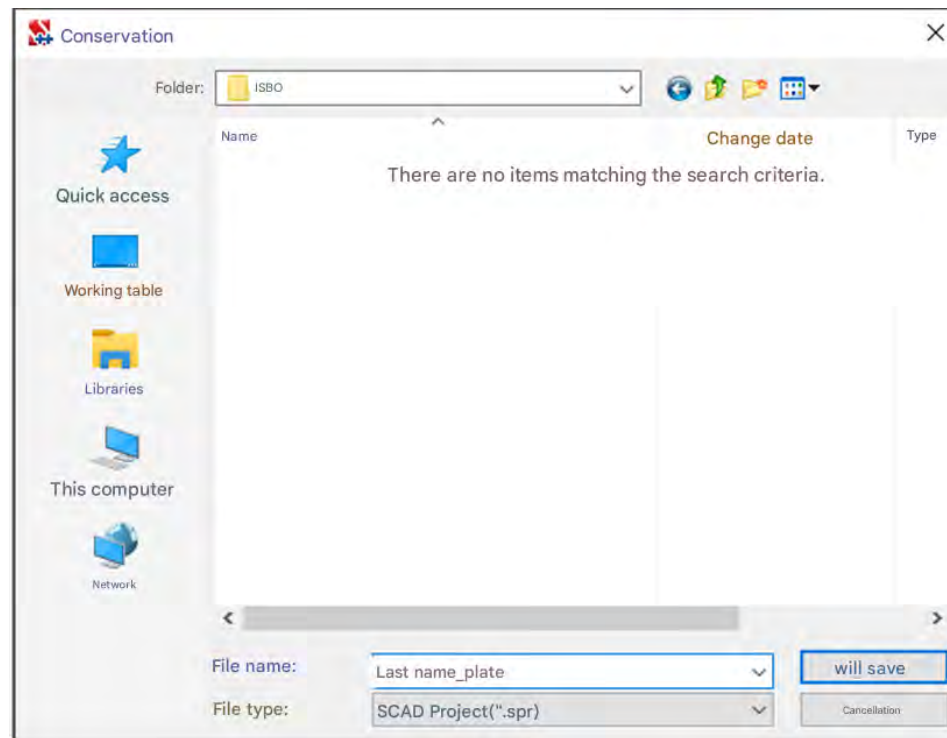


Figure 2.2 – Dialog box for choosing where to save a new project file

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

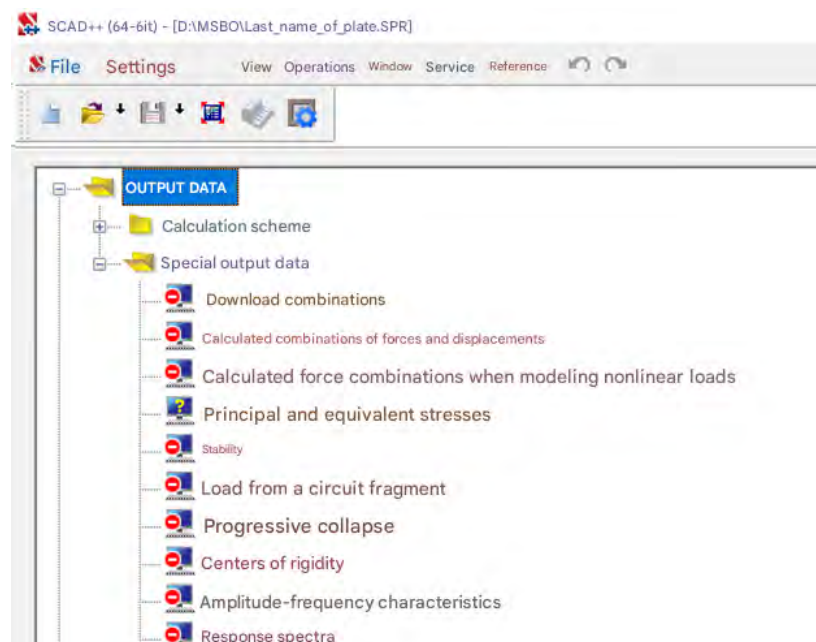


Figure 2.3 – Project tree dialog box

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.

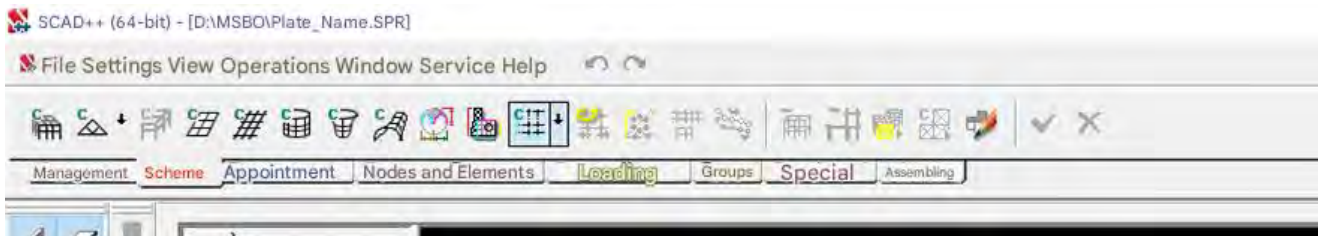


Figure 2.4 – Invoking the Set Partition Grid dialog box

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.

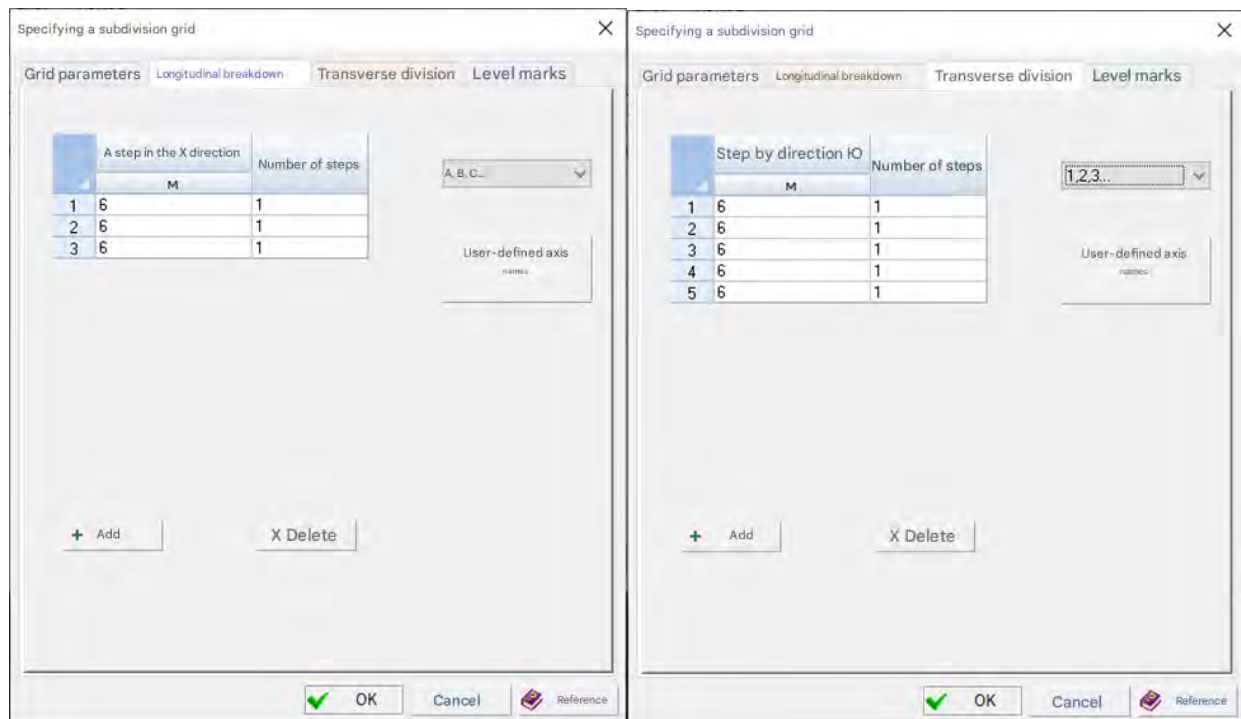


Figure 2.5 – Set Partition Grid Dialog Box

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



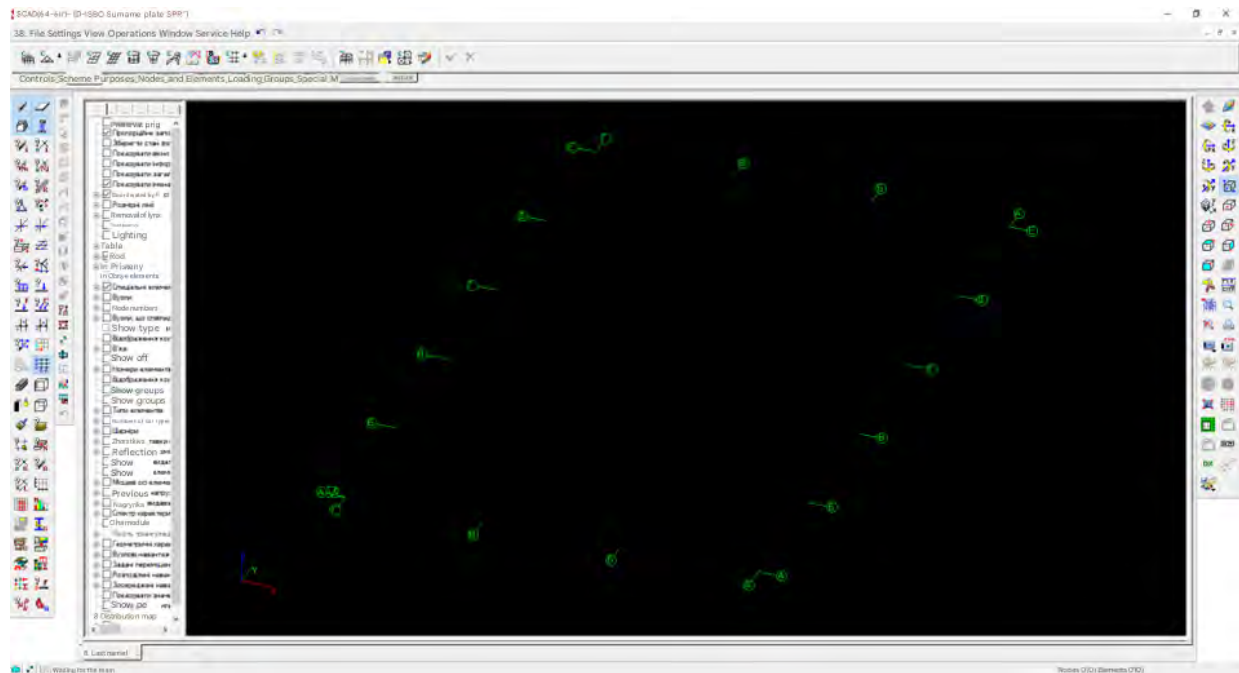


Figure 2.6 – Displaying the Layout Grid in the Program Work Field

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.

Introduction of nodes

X: -1 M ΔX: 0 M

Y: -1 M ΔY: 0 M

Z: 0 M ΔZ: 0 M

☐ Repeat N: 0

☐ Automatic transfer of the origin to the last entered node

☐ Changing the input direction

☐ Around the X axis Angle of rotation: 0

☐ Around the Y axis

☐ Around the Z axis

? Elms

+ Add close Reference

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.

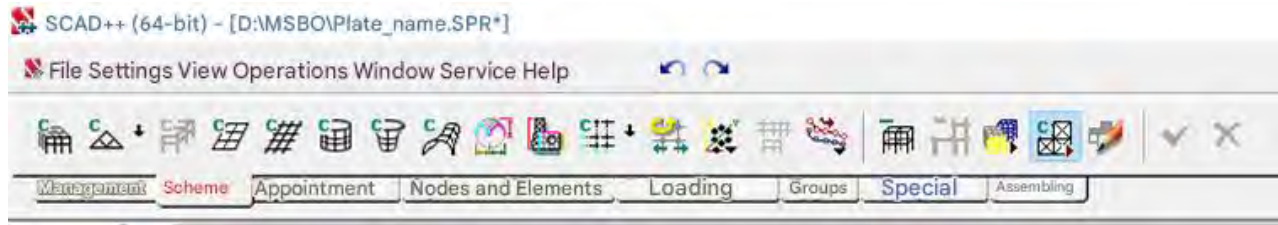


Figure 2.8 – Invoking the triangulation tool in the toolbar

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.

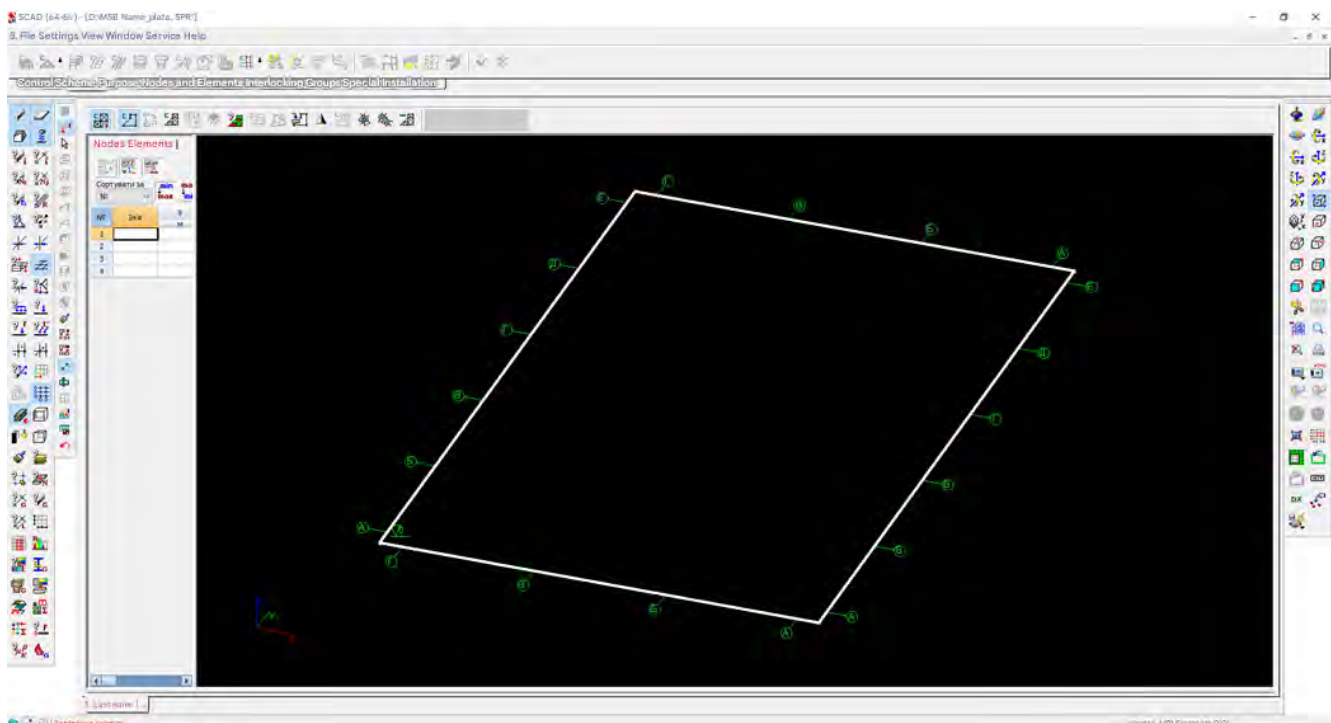


Figure 2.10 – The result of creating a path

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.

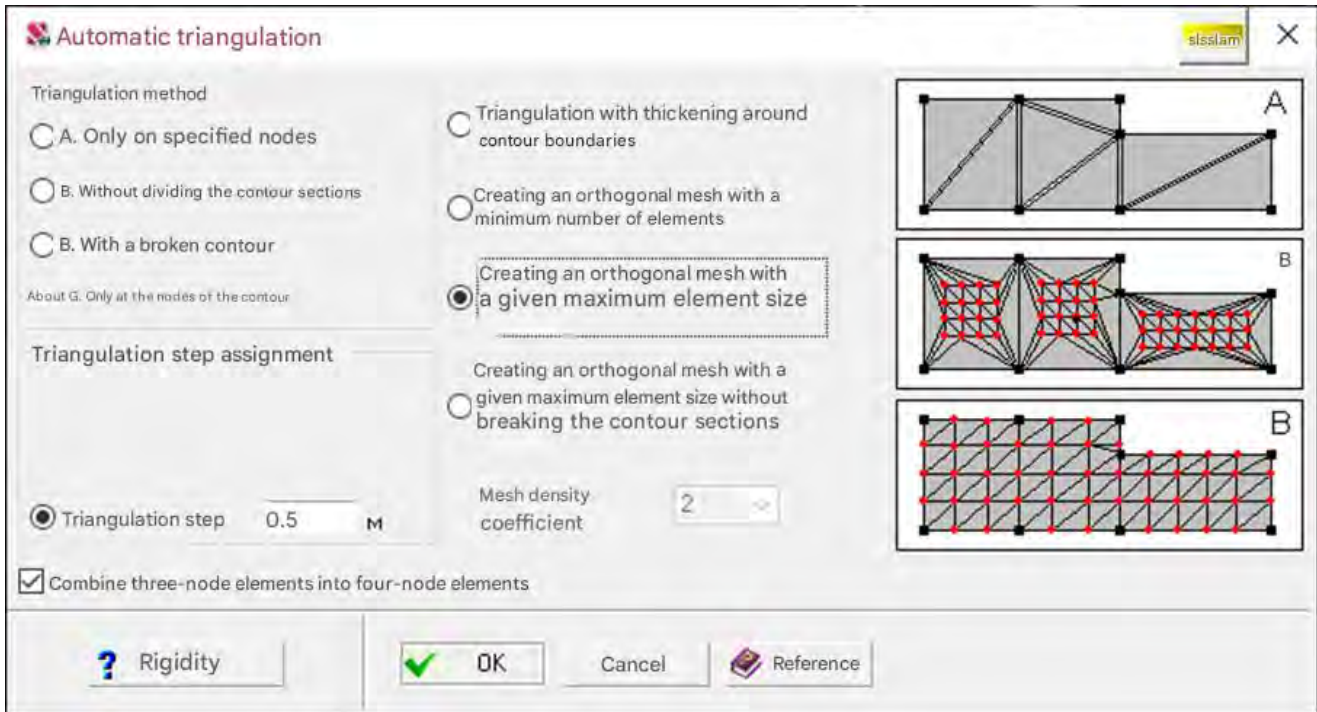


Figure 2.11 – Automatic Triangulation dialog box

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

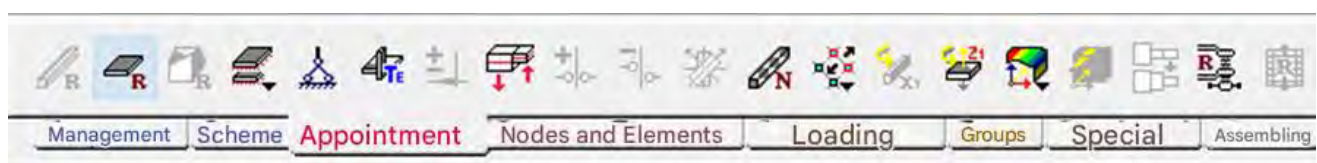


Figure 2.12 – Assigning stiffeners to plates on the assignment tab

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.

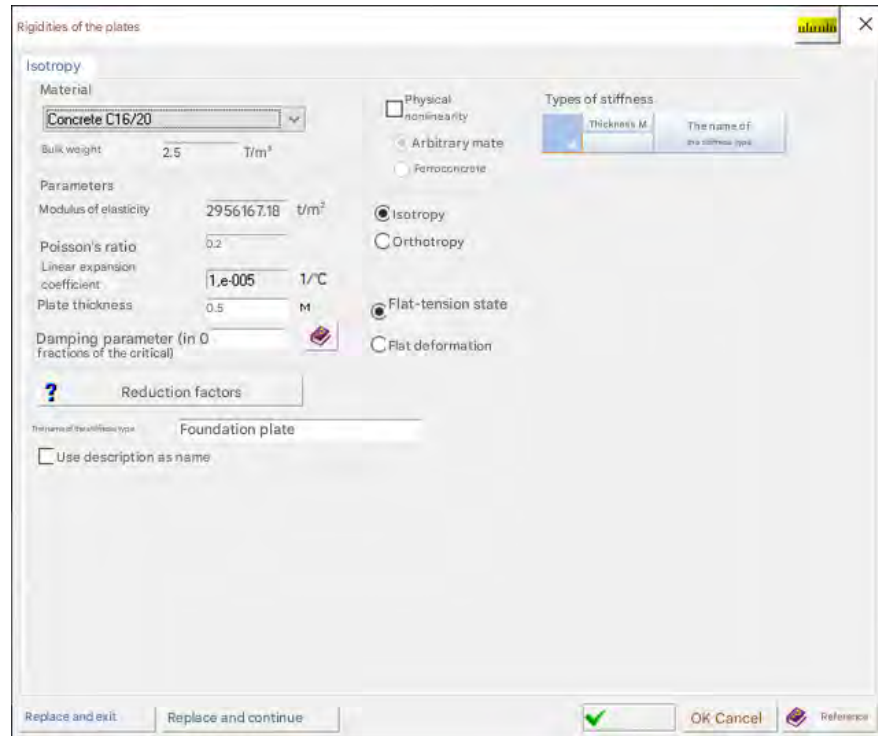


Figure 2.13 – Plate Stiffness Dialog Box

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

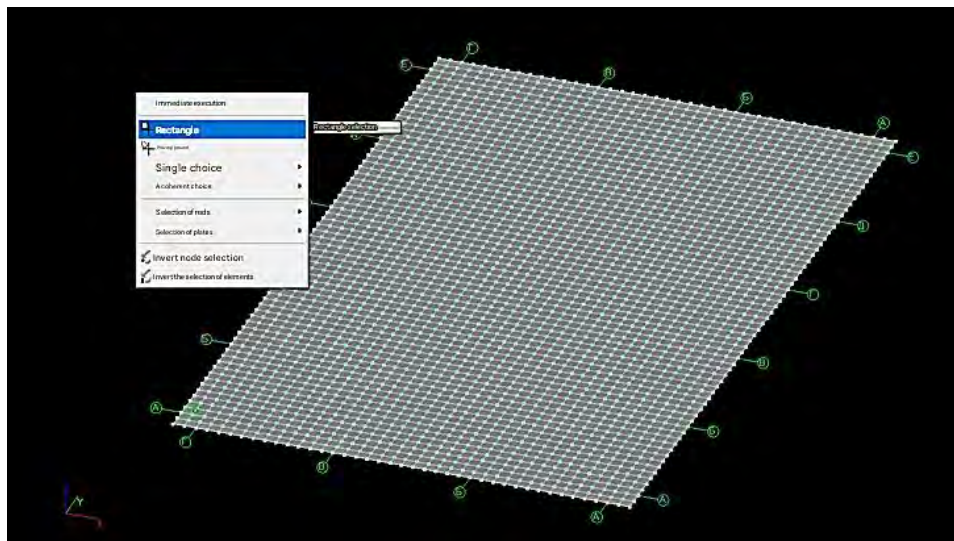


Figure 2.14 – Selection of plates for assigning stiffness in the working field of the program

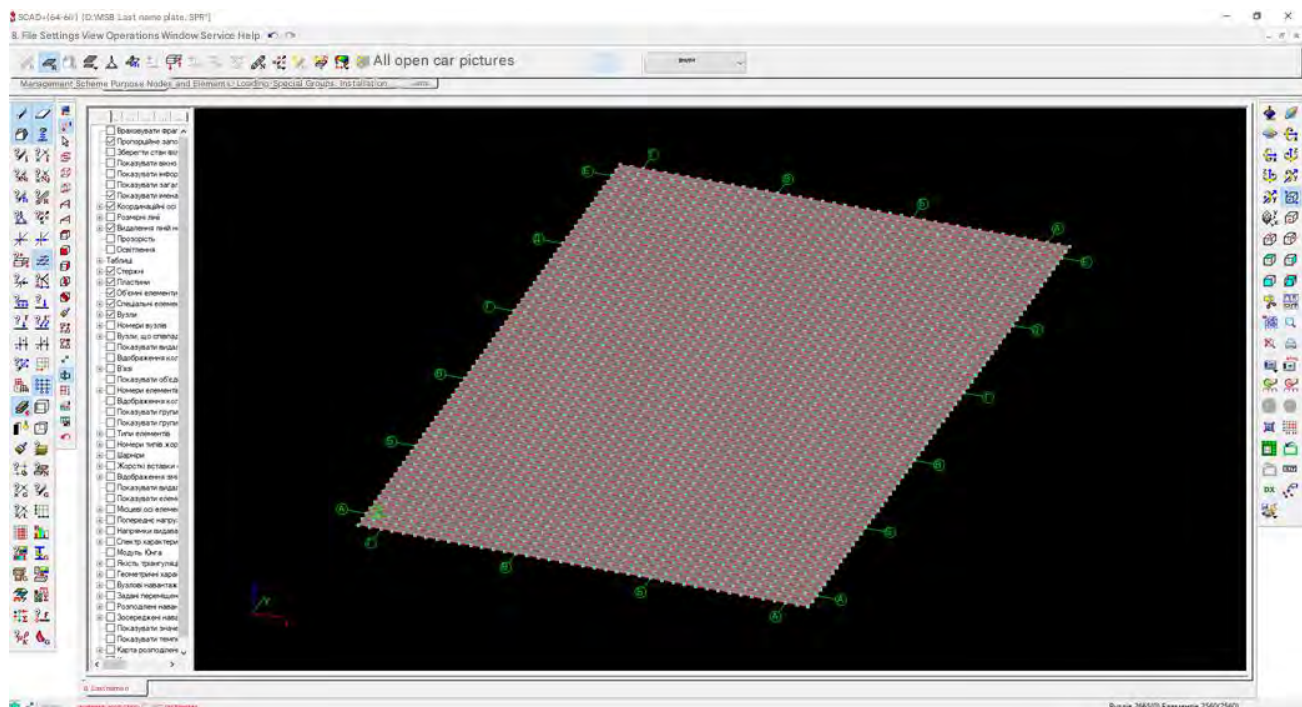


Figure 2.15 – Confirmation of the application of stiffeners to the foundation slab

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.

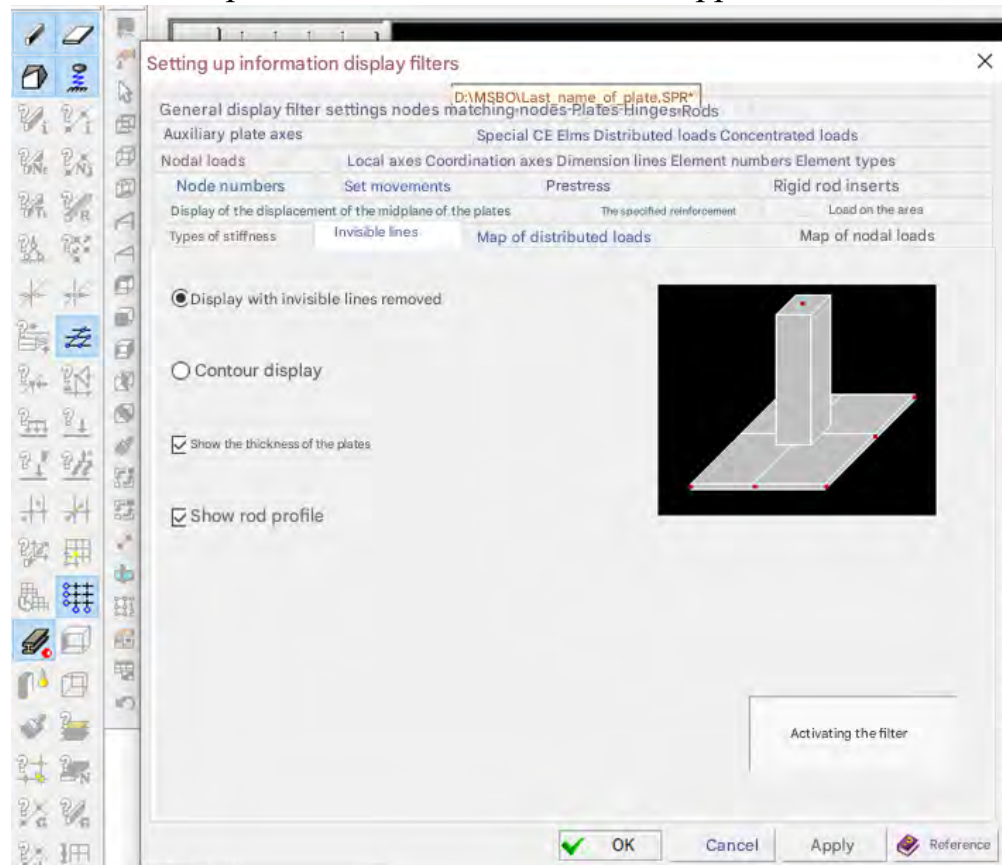


Figure 2.16 – Activating the plate thickness display filter

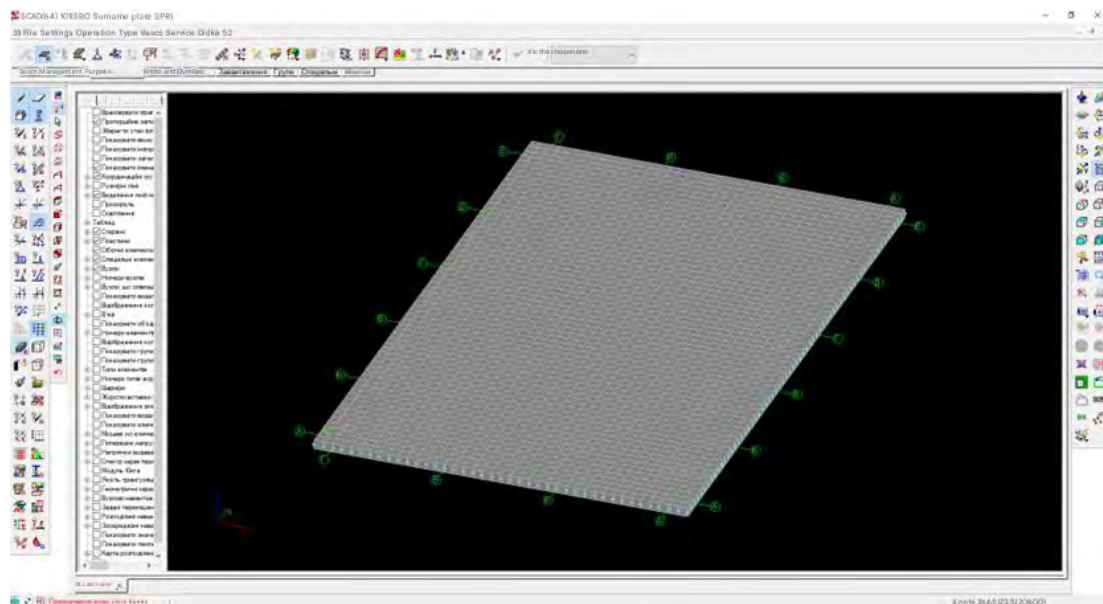


Figure 2.17 – Display of the created foundation slab with the activated filter for displaying the thickness of the plates

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.

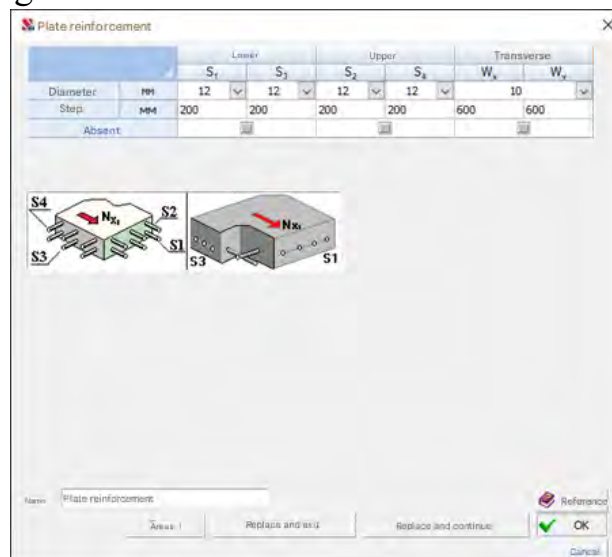


Figure 2.19 – Purpose of plate reinforcement

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

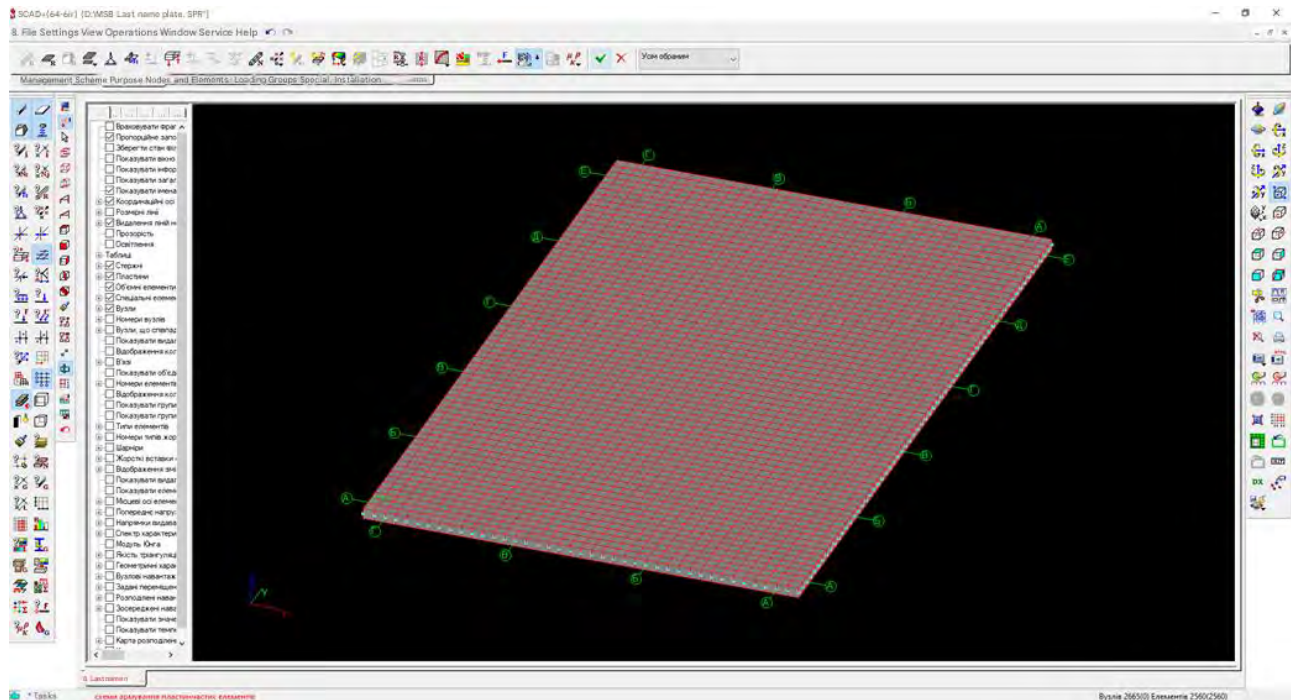


Figure 2.20 – Application of the specified reinforcement to the foundation slab model

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

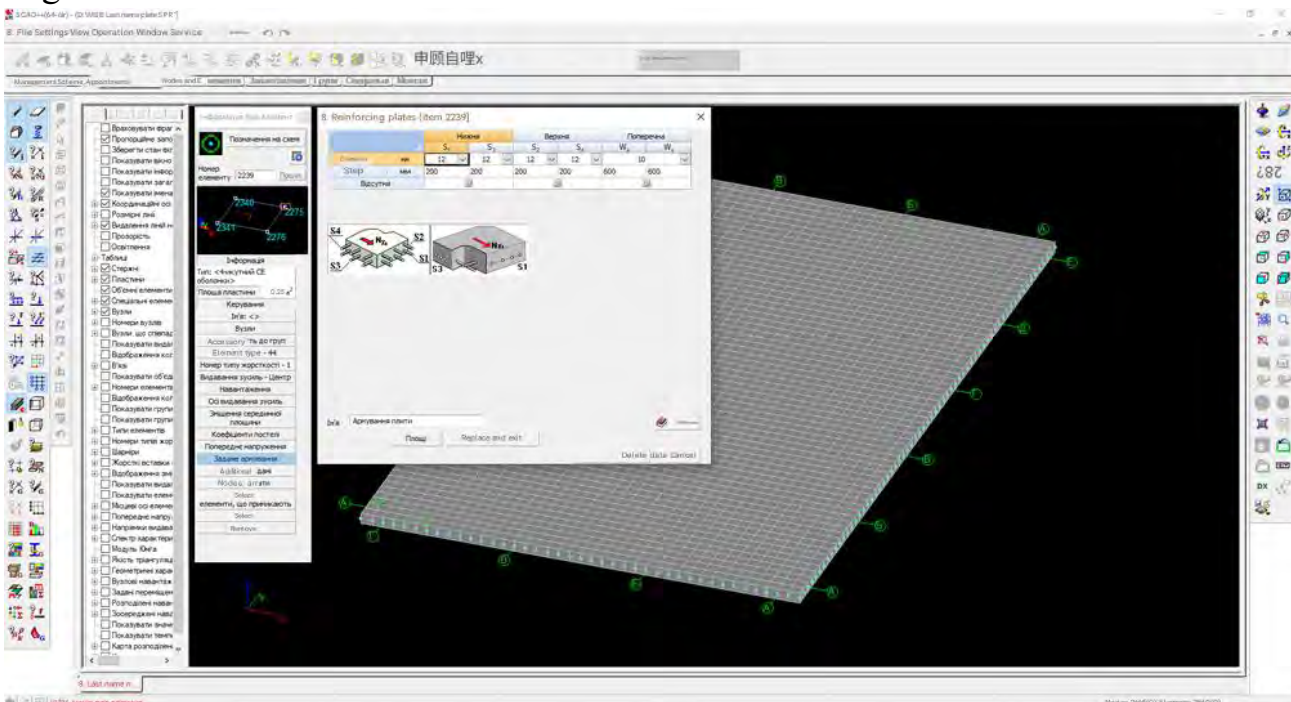


Figure 2.21 – Information about the reinforcement of the selected finite element

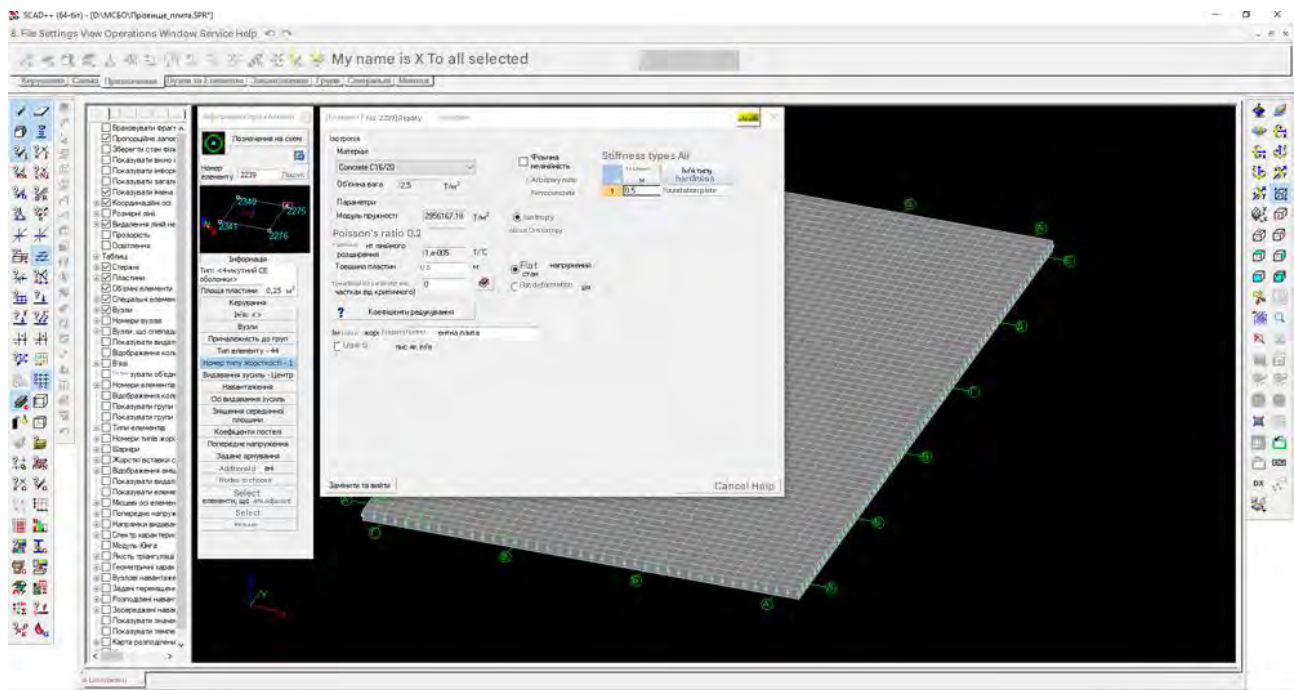


Figure 2.22 – Information about the stiffness of the selected finite element

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.

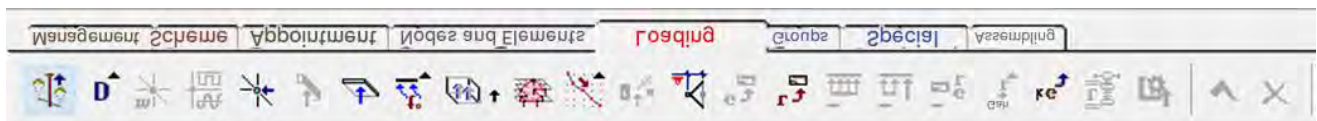


Figure 2.23 – "Dead weight" button on the "Downloads" tab

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.

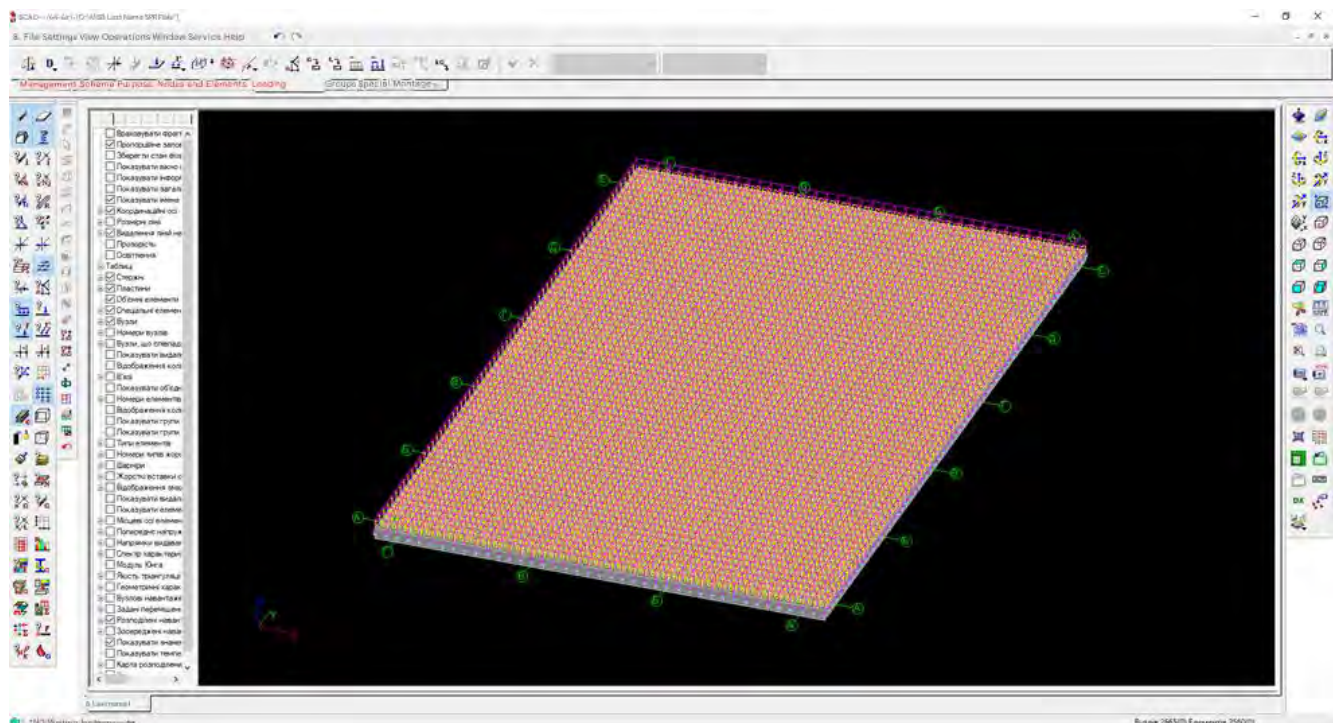


Figure 2.25 – Display of the dead weight of the foundation slab

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.

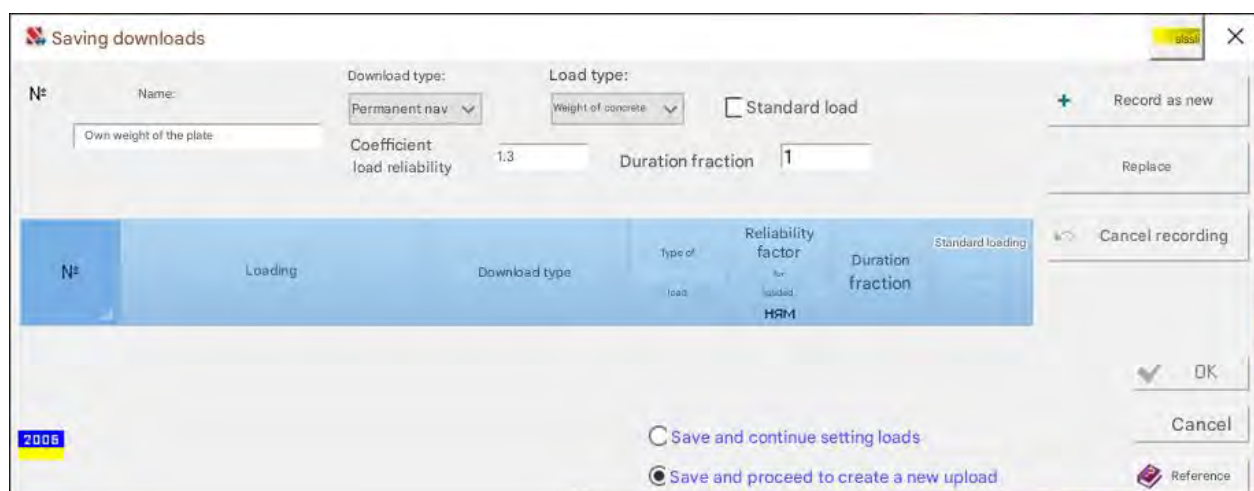


Figure 2.26 – Save Downloads dialog box

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.

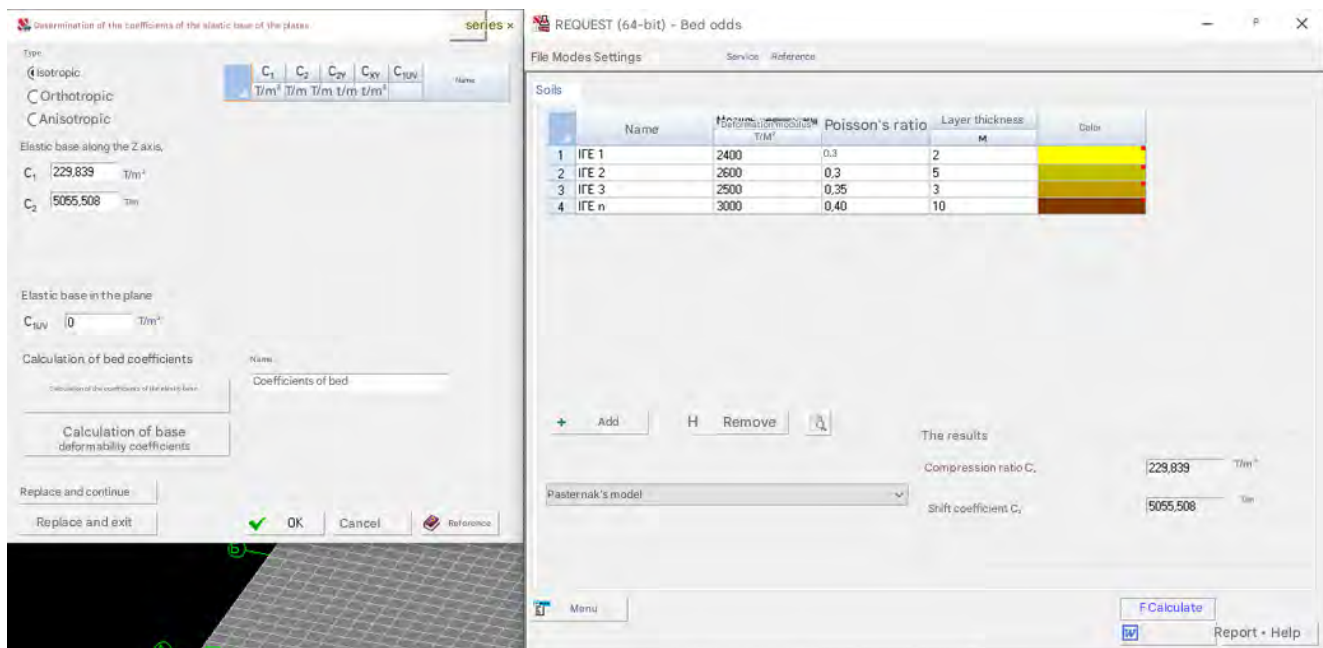
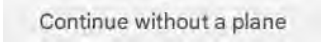


Figure 2.27 – Dialog box "Assignment of elastic base coefficients of plates" and results of calculation of previous bedding coefficients in the REQUEST program

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  "Continue without a plane". 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.

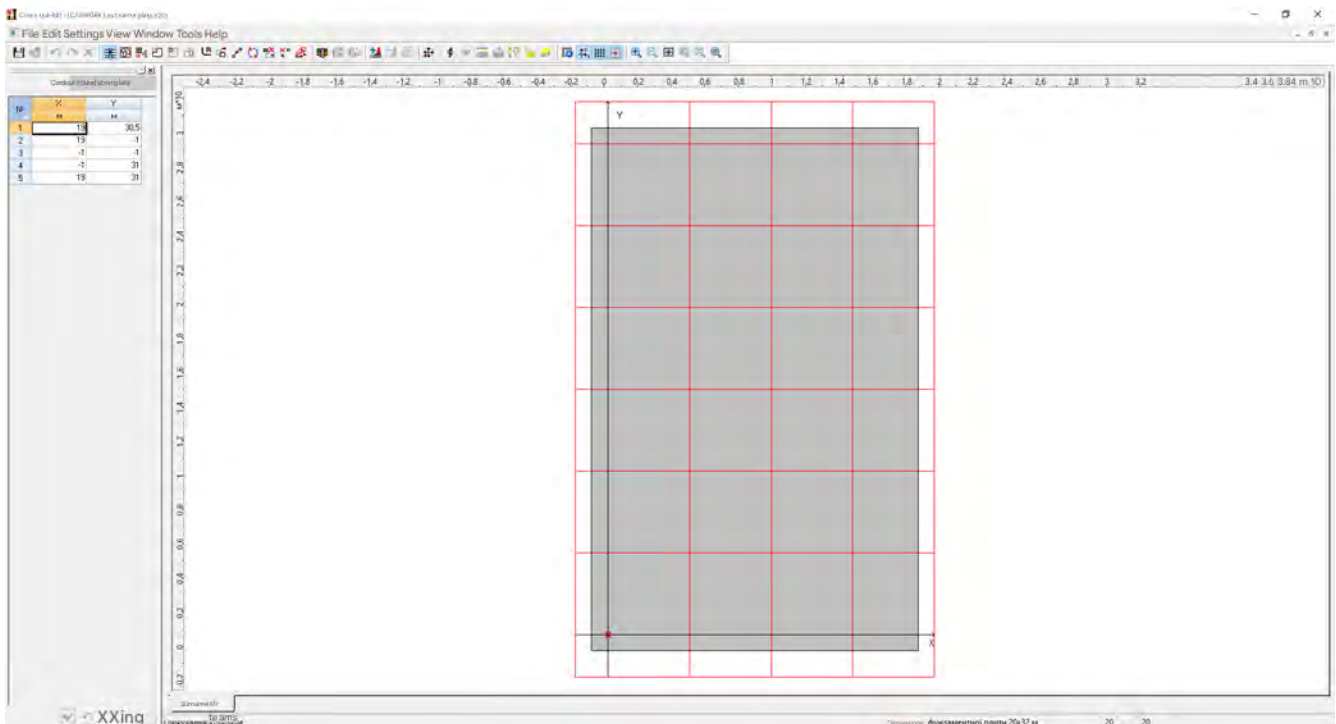


Figure 2.28 – Workspace of the CROSS program with imported foundation slab from the SCAD++ program

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

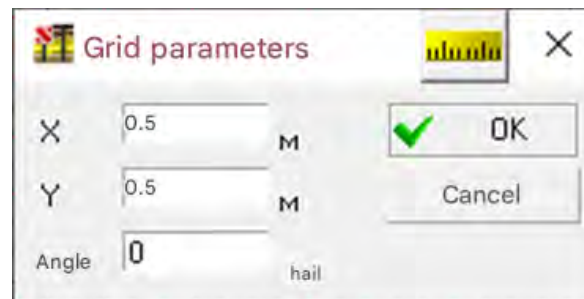


Figure 2.29 – Grid Options dialog box of CROSS

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

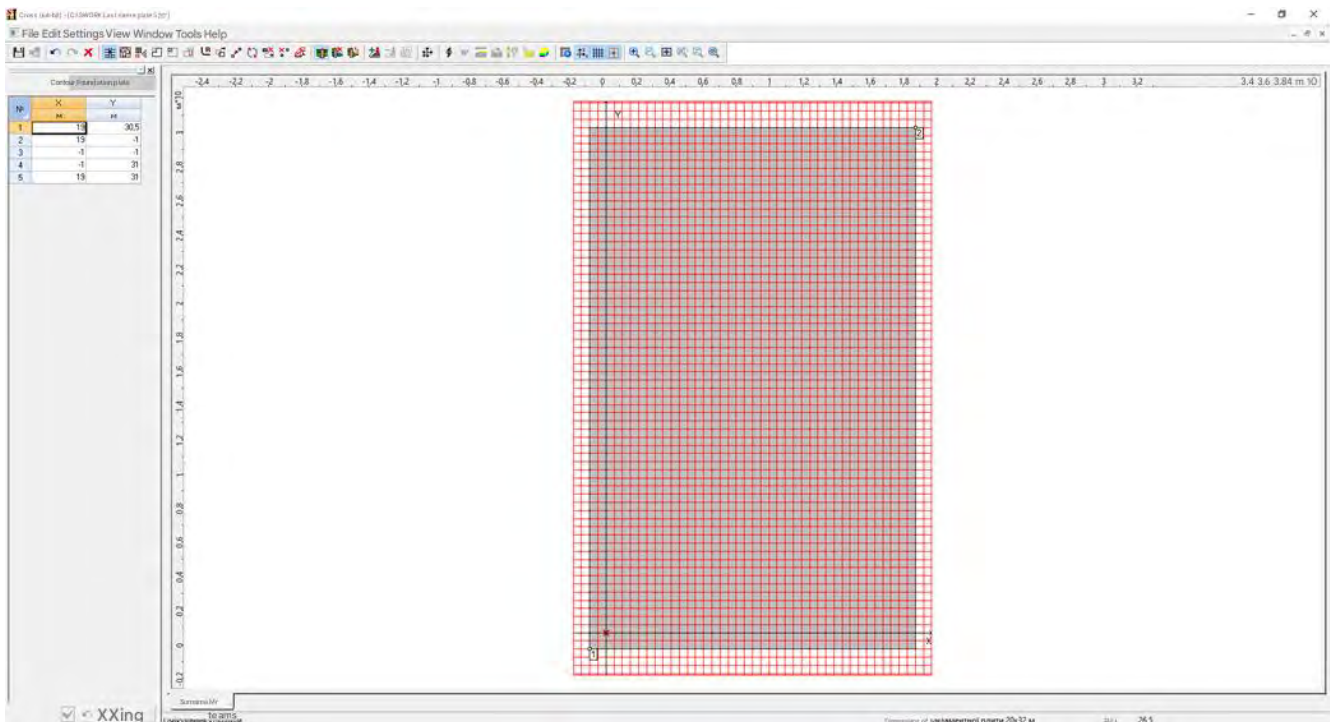


Figure 2.31 – Placement of wells on site

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

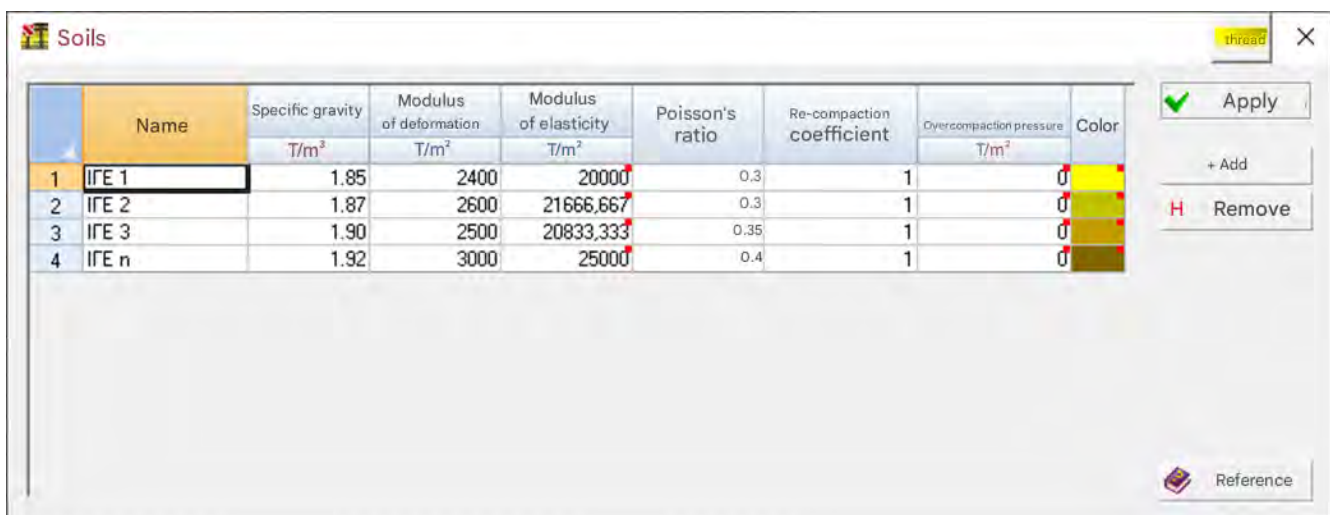


Figure 2.32 – Entering information about soil conditions on the soil tab

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.

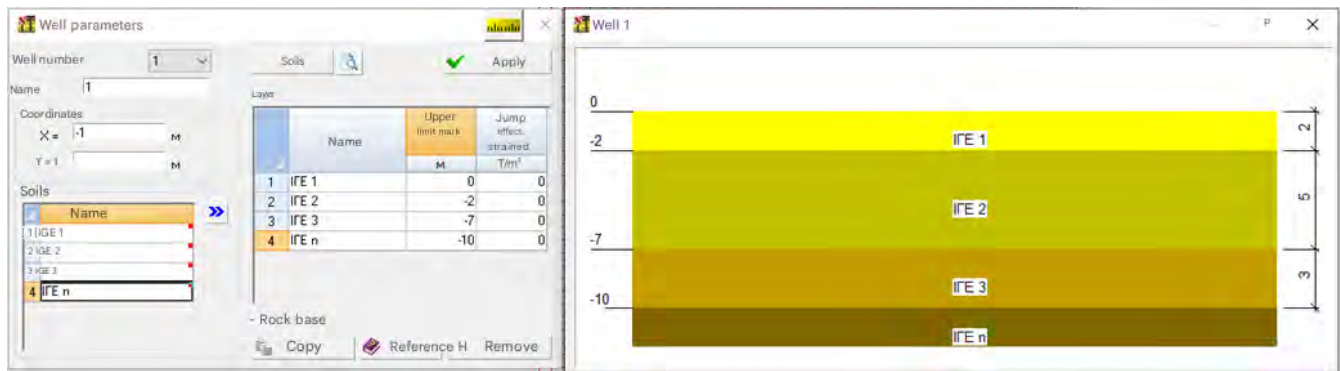


Figure 2.33 – Creation of a geological column for well 1

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.

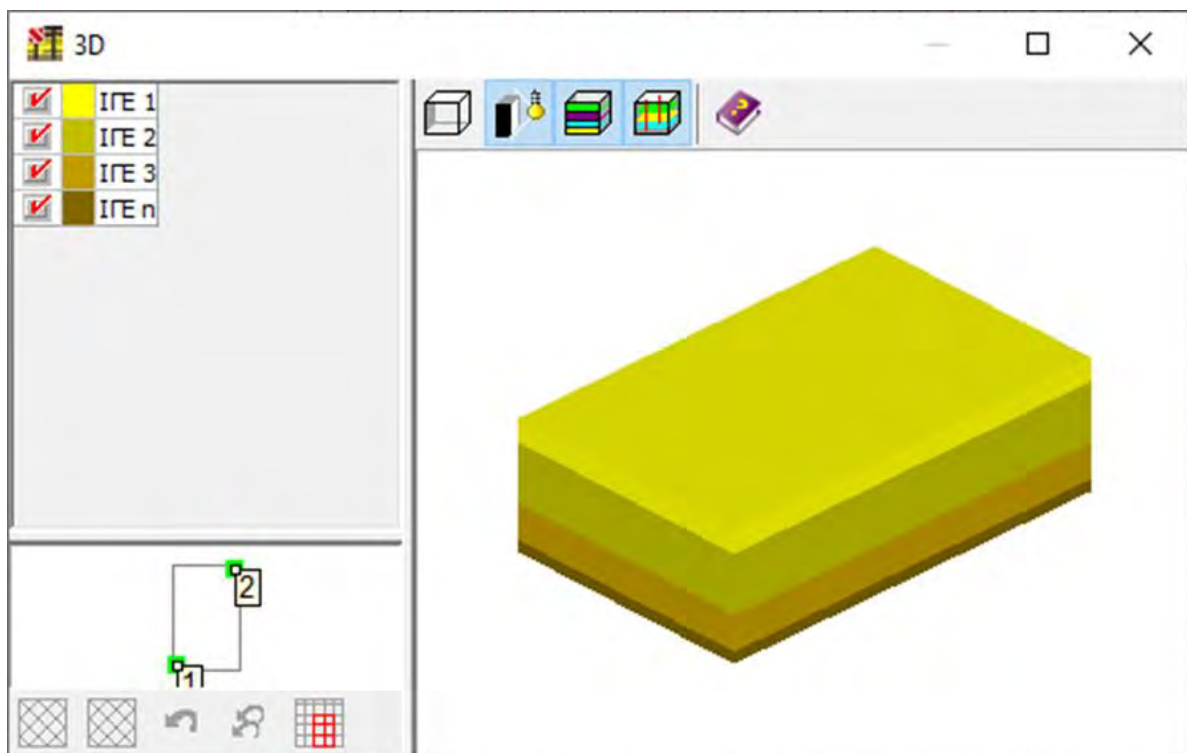


Figure 2.34 – Volumetric display of the geological structure of the site

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.

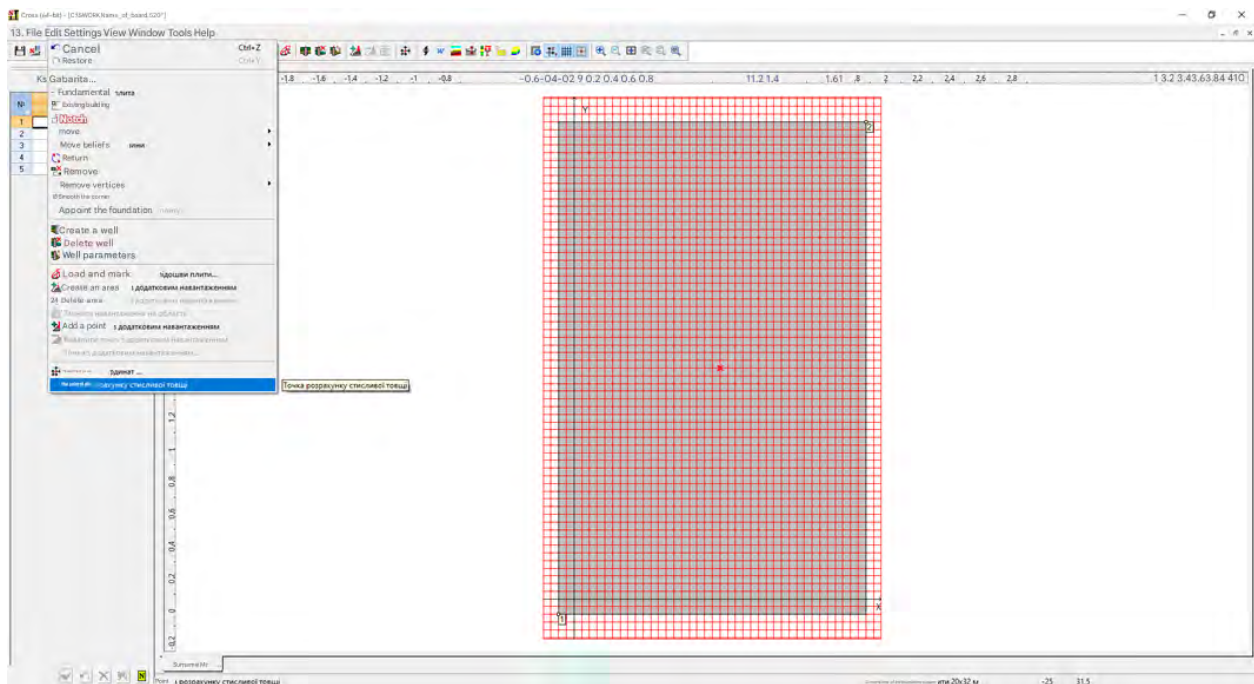


Figure 2.36 – Setting the compressible thickness calculation point

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

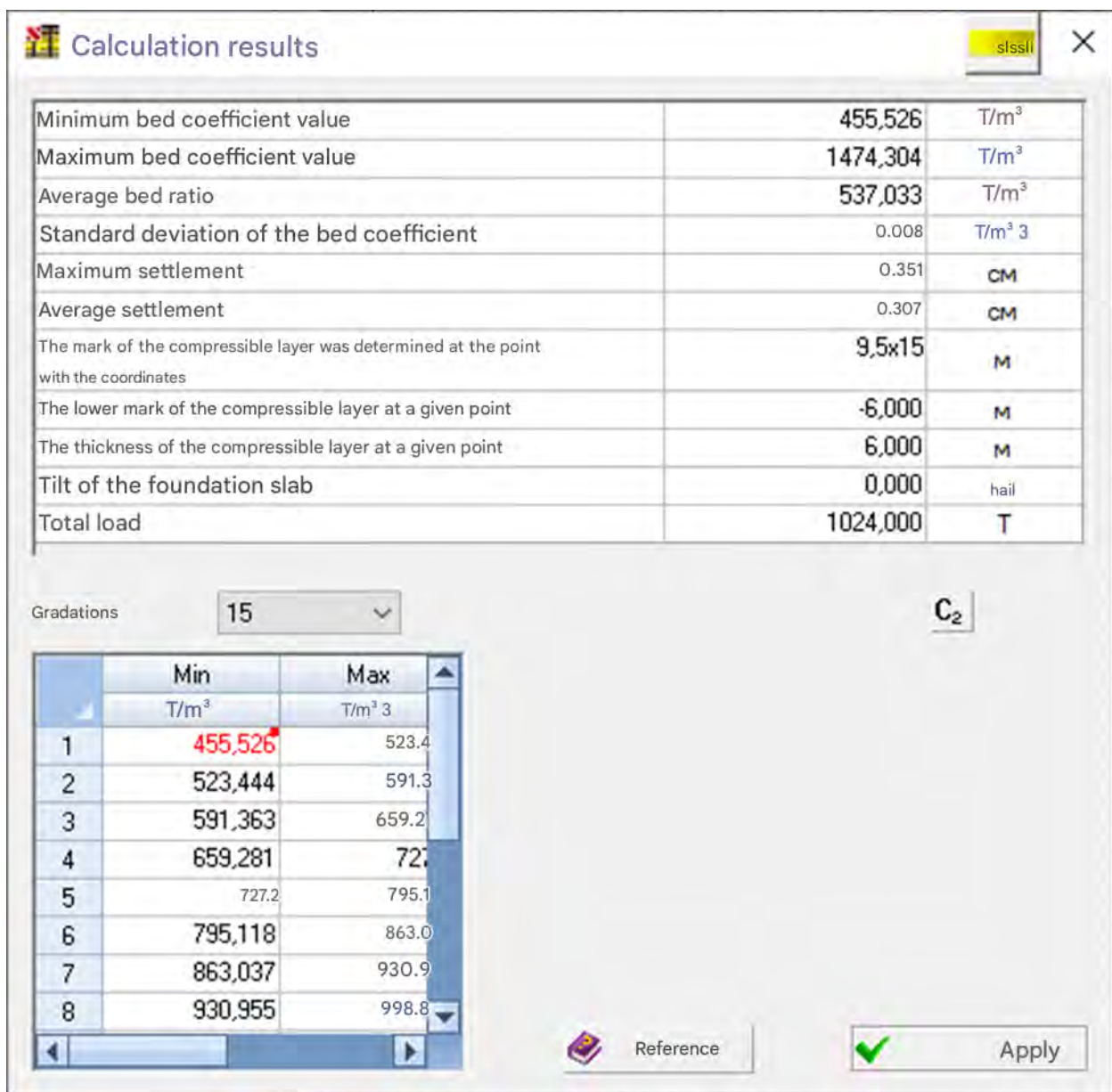


Figure 2.37 – Calculation Results Dialog Box

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.

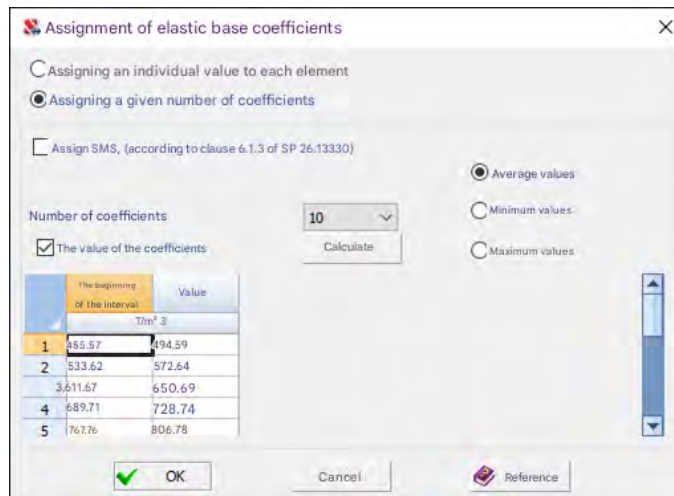


Figure 2.38 – Assignment of elastic base coefficients dialog box

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.

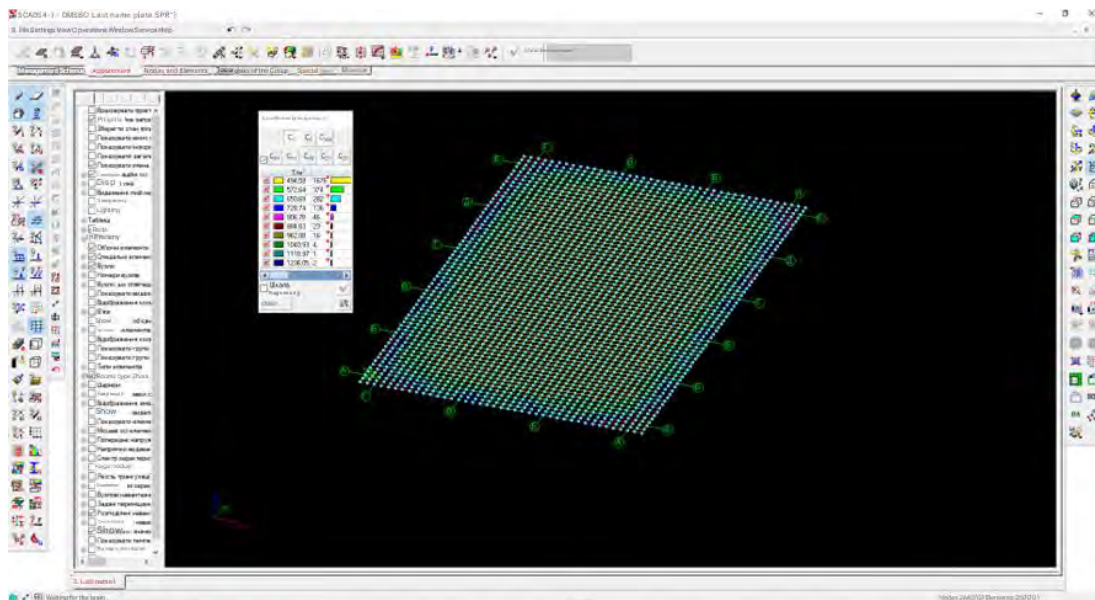


Figure 2.39 – Isopoles displaying the assigned coefficient C1 on the foundation slab

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.