

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.

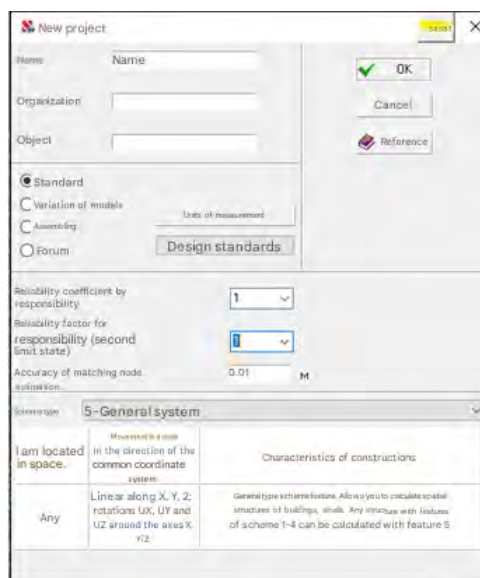


Figure 3.1 – New Project dialog box

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

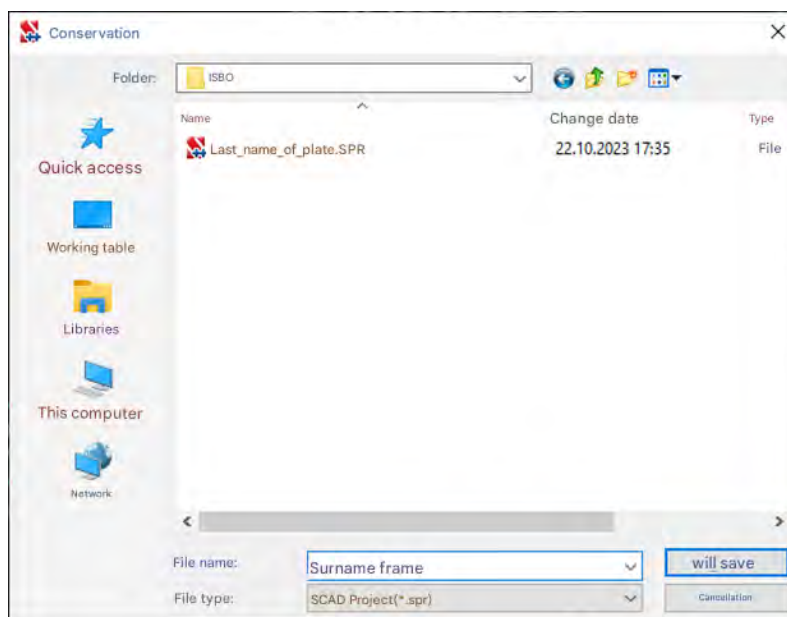


Figure 3.2 – Save dialog box

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.

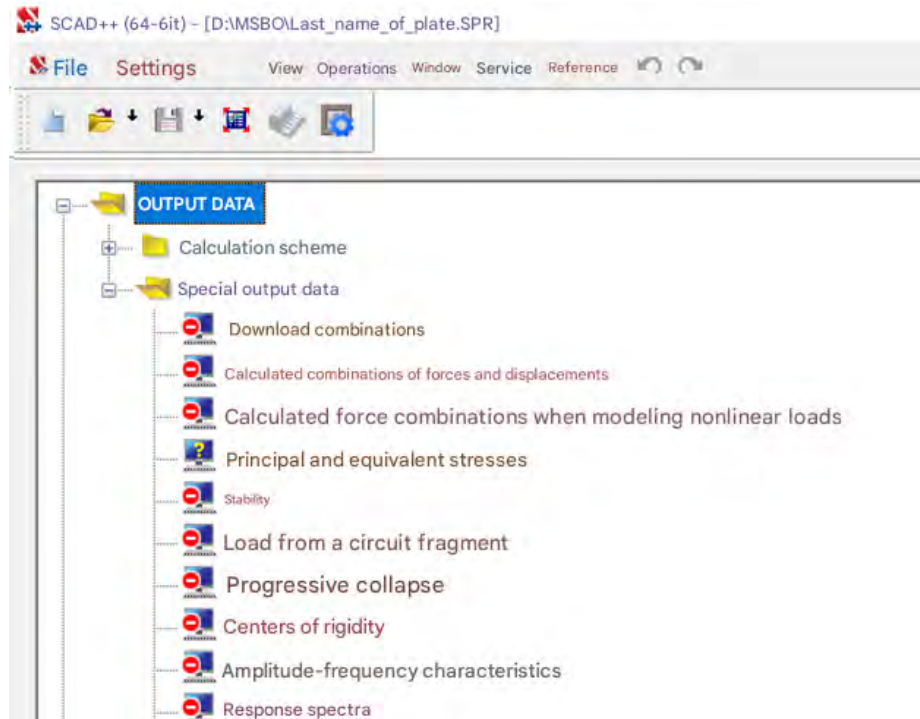


Figure 3.3 – Project tree dialog box

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.

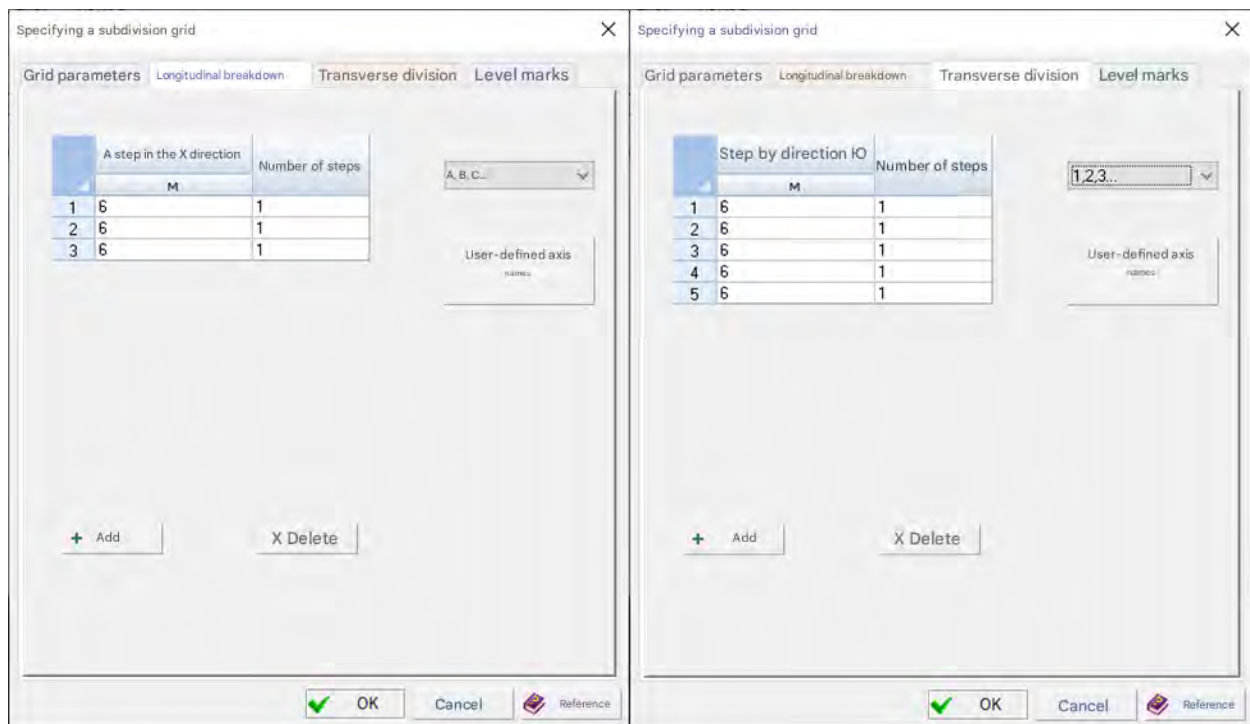


Figure 3.5 – Set Partition Grid Dialog Box

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

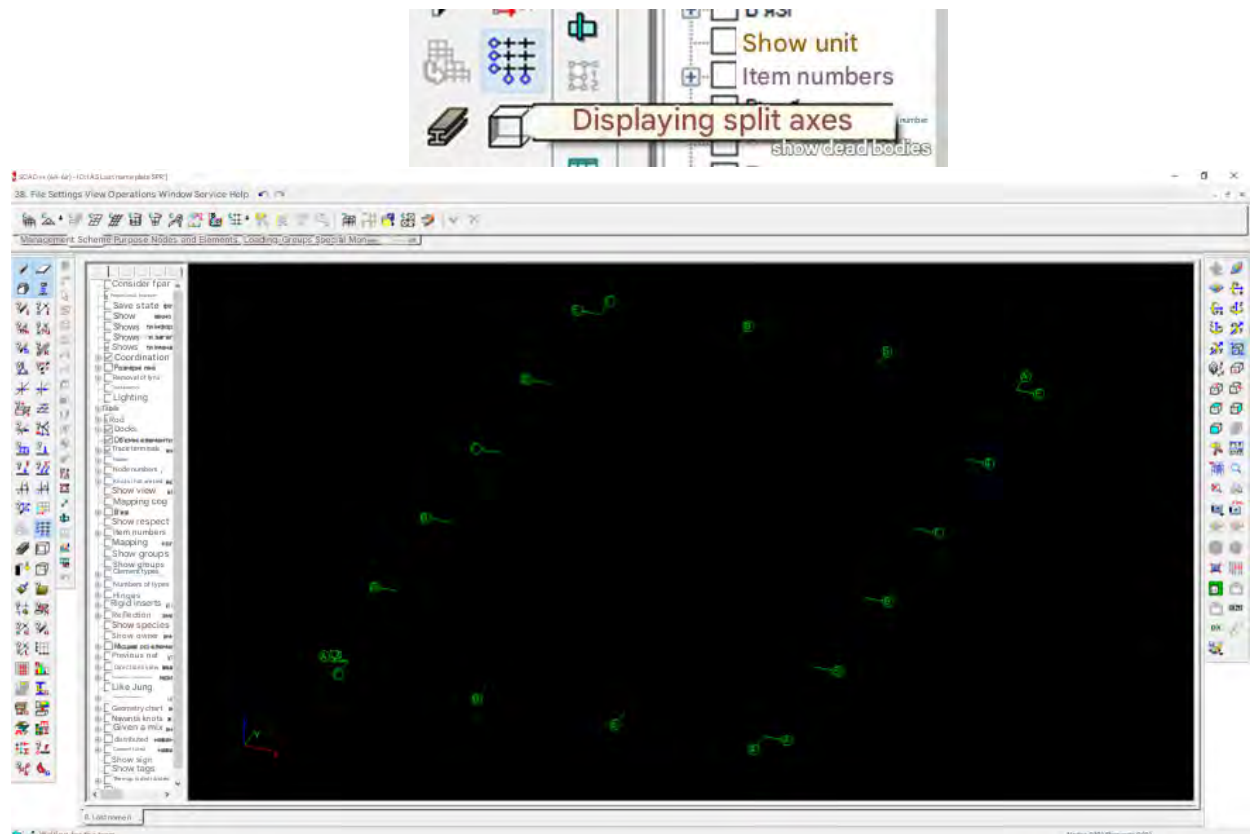


Figure 3.6 – Displaying the Layout Grid in the Application Work Field

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.

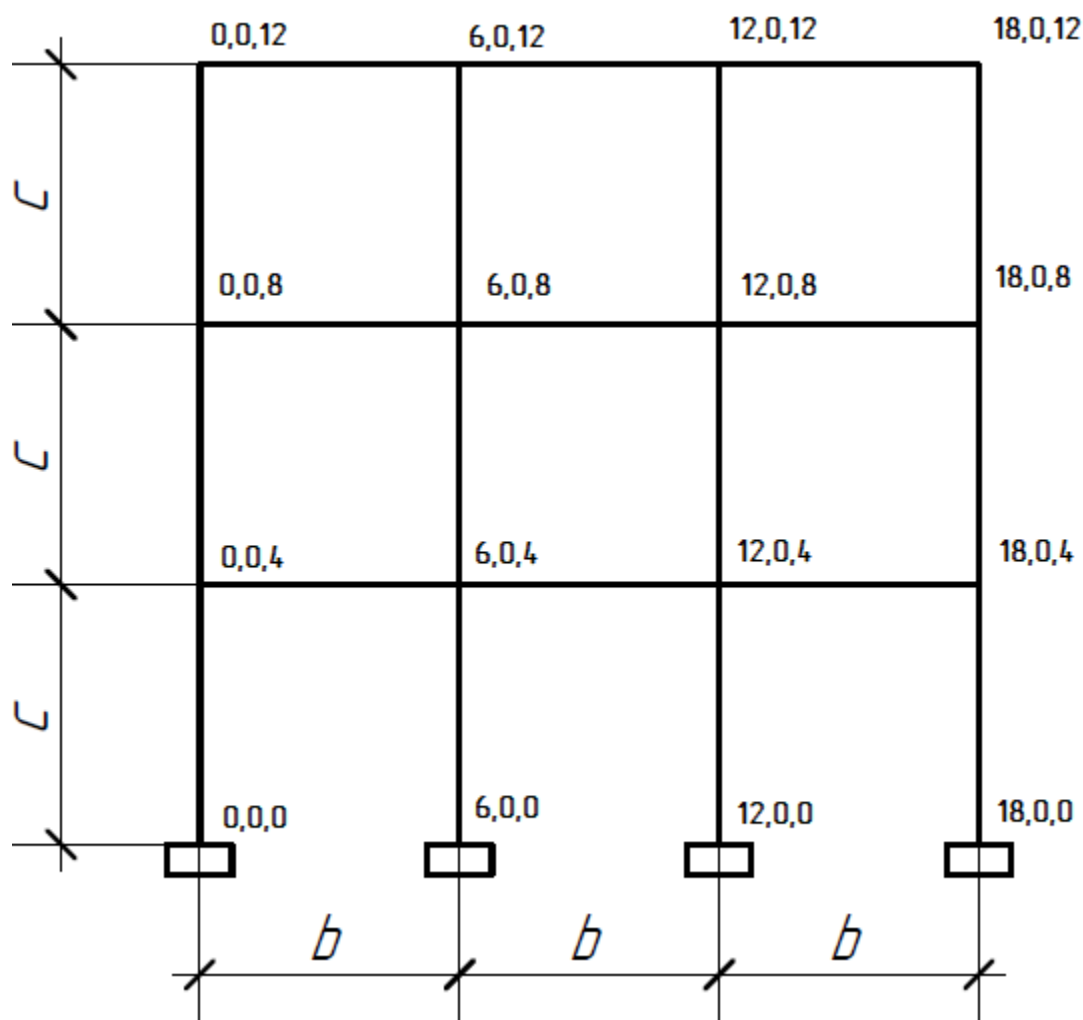


Figure 3.7 – Determination of the coordinates of the frame nodes along axis 1



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

In the "Node Input" dialog box, enter the coordinates of all nodes of the first frame frame along axis 1.

Figure 3.9 – Node Input dialog box

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

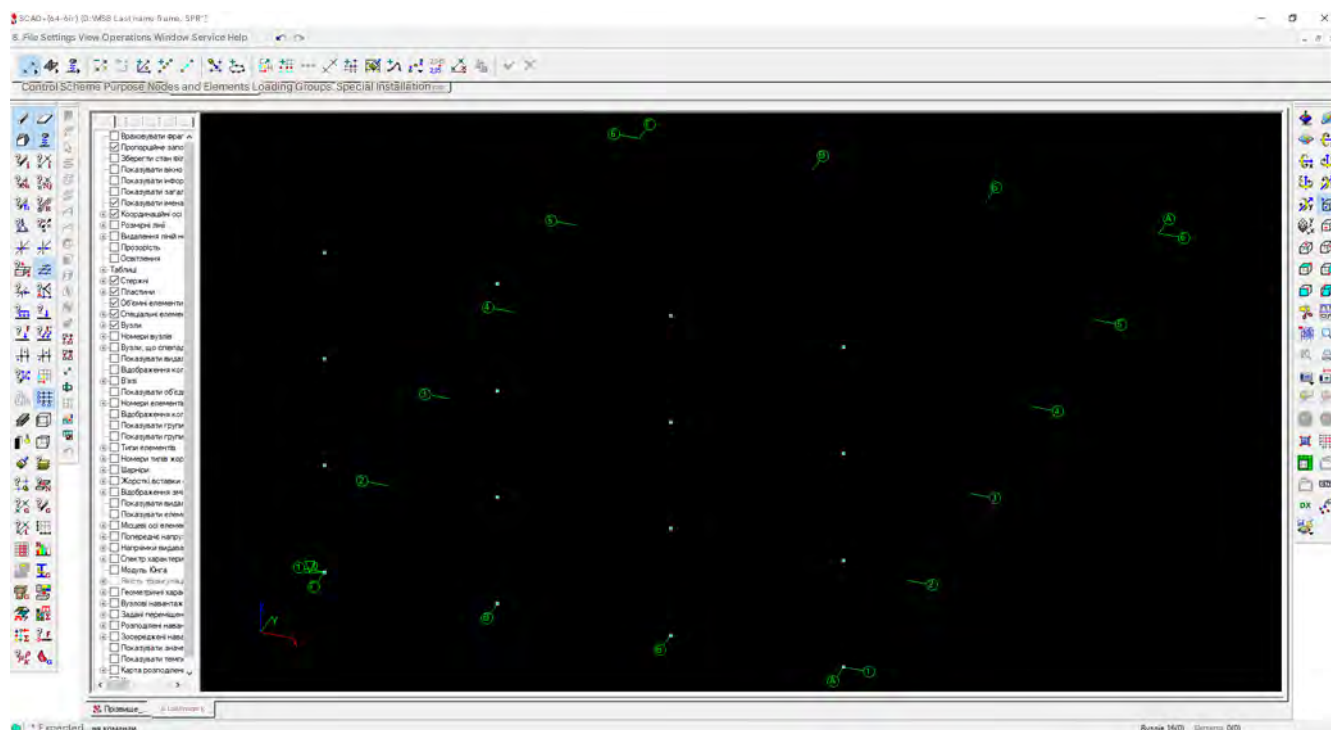


Figure 3.10 – Display of frame units in the working field of the program

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

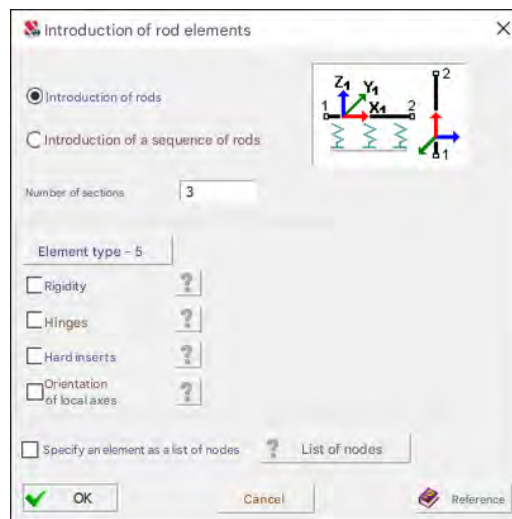


Figure 3.12 – Input of Core Elements dialog box

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"

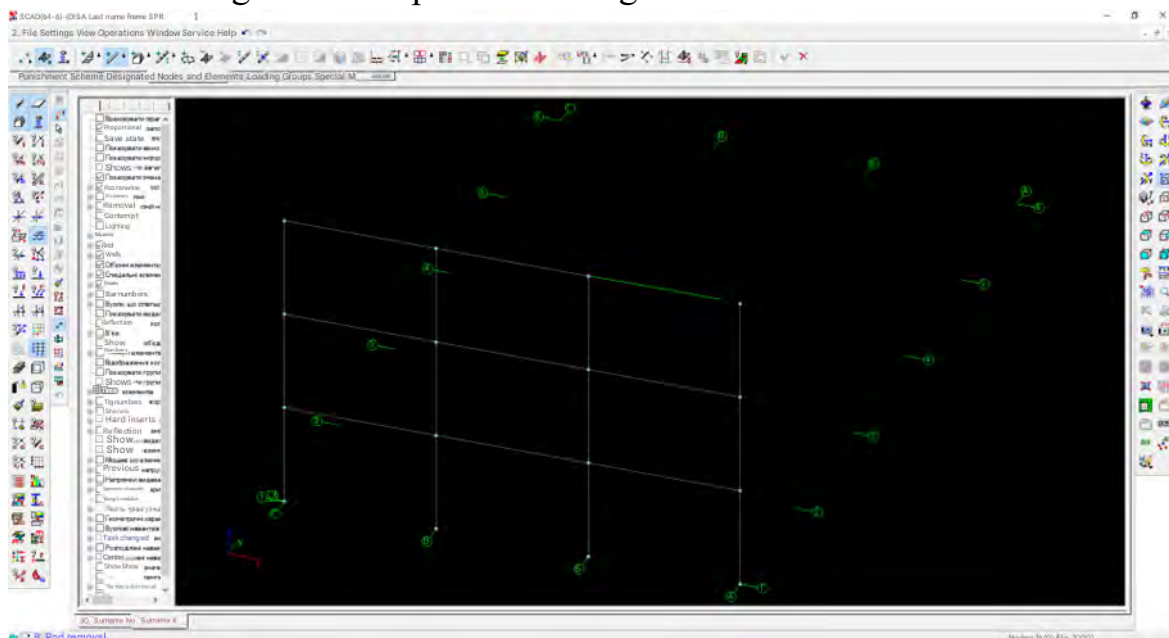


Figure 3.13 – Connection of frame assemblies with rods

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.

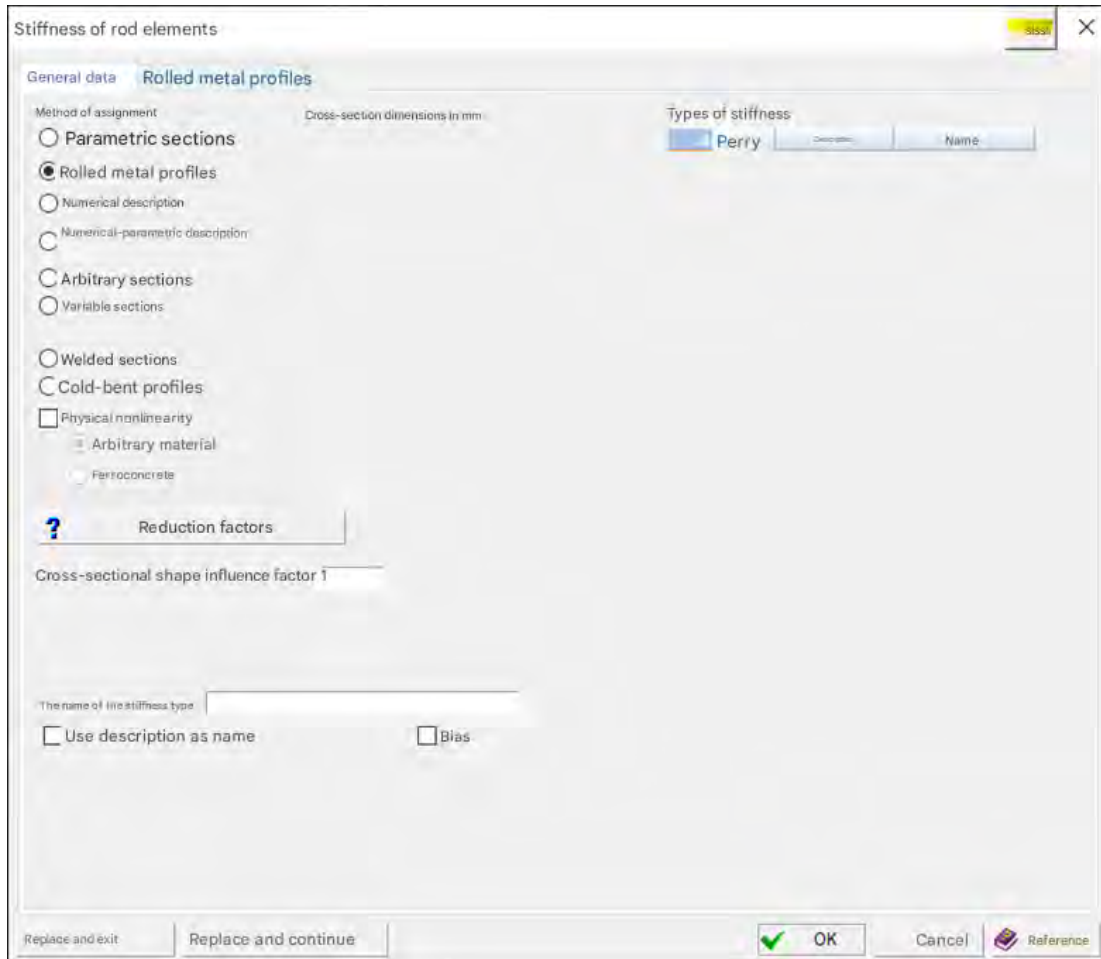


Figure 3.14 – Bar Stiffness Dialog Box

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.

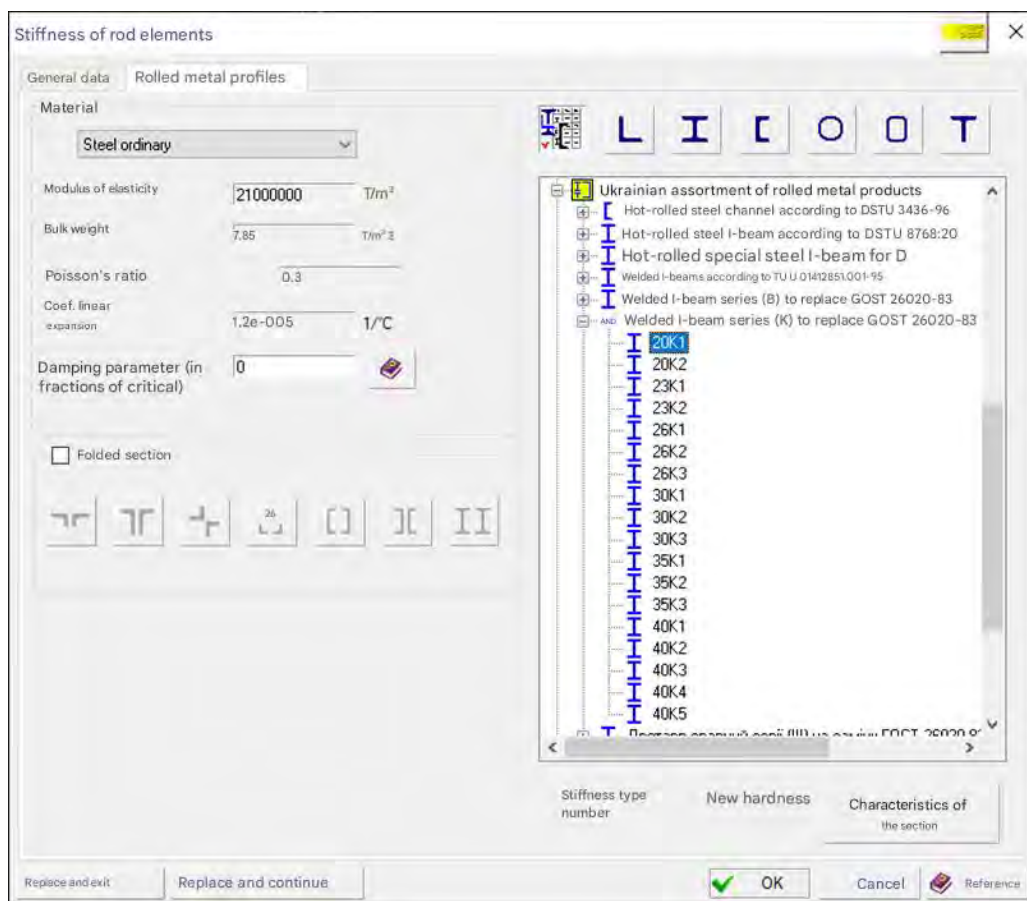


Figure 3.15 – Selection of assortment for the selection of column rigidities

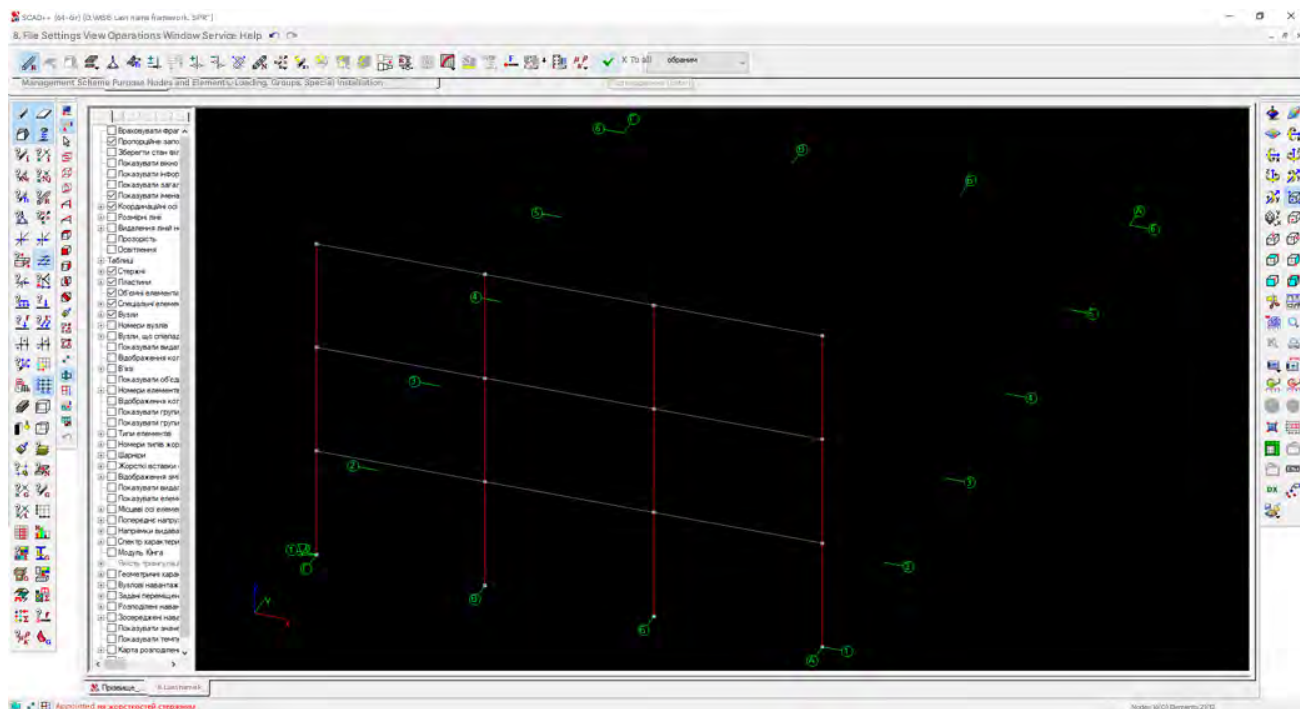


Figure 3.16 – Assignment of the selected stiffness for columns

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.

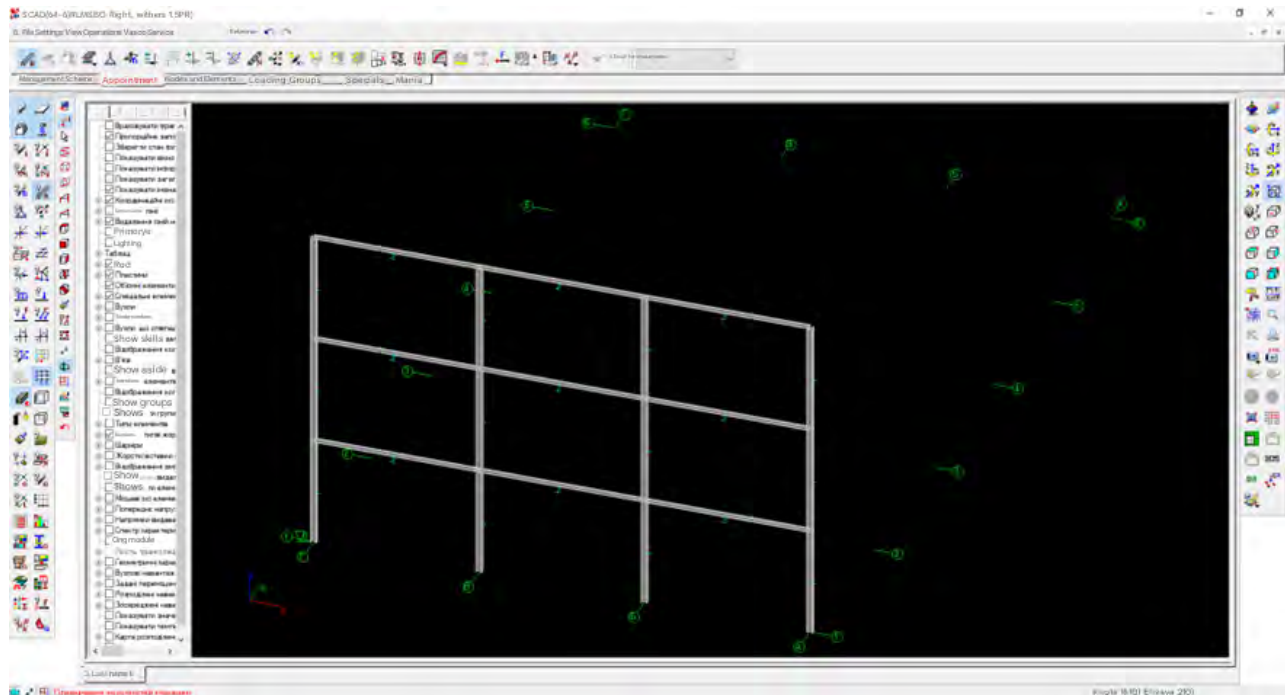


Figure 3.19 – Control of the assignment of stiffeners by enabling the corresponding functions in the filter panel

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.

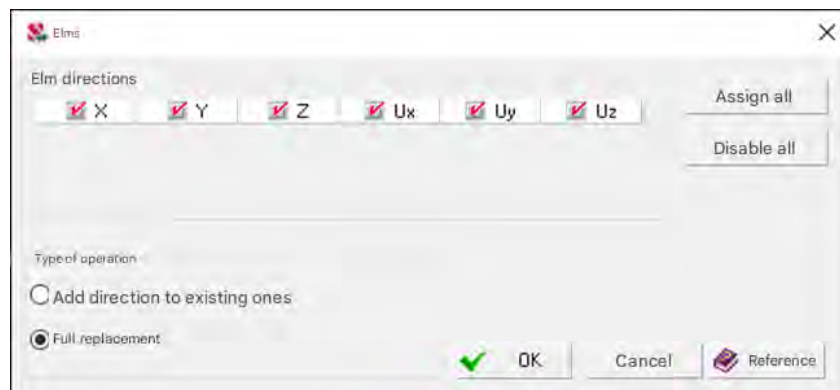


Figure 3.21 – Knitting dialog box

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

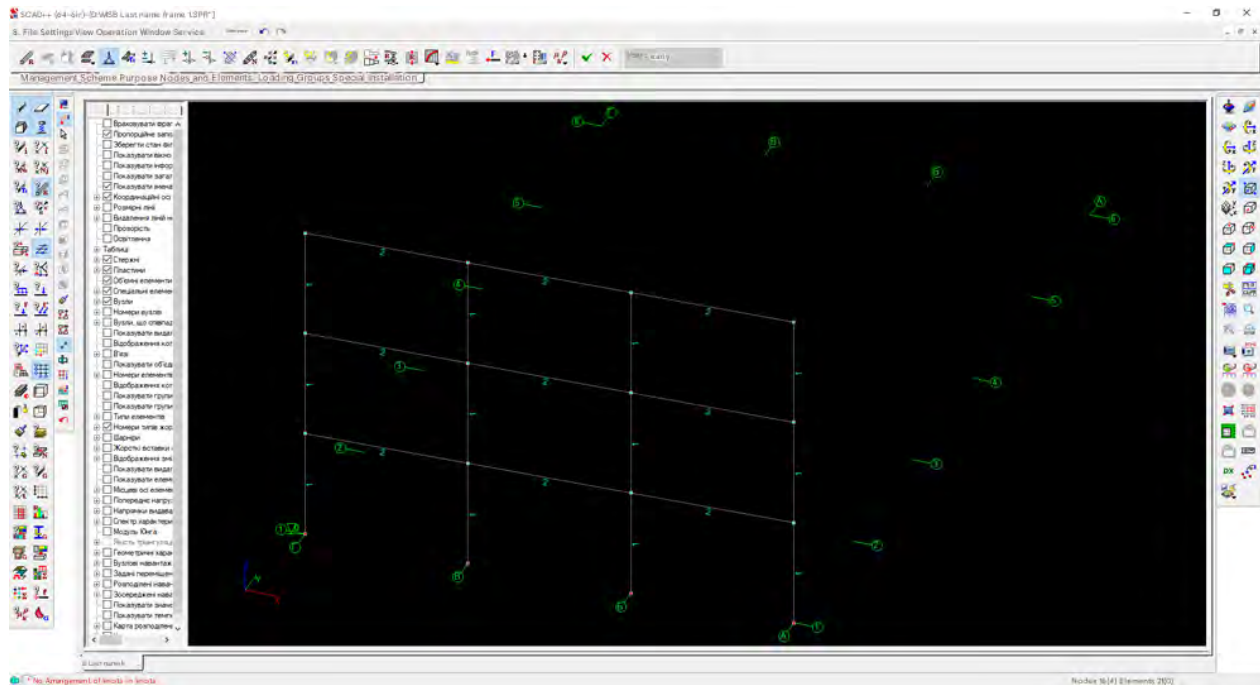


Figure 3.22 – Assignment of frame node junctions to ref. 0,000

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

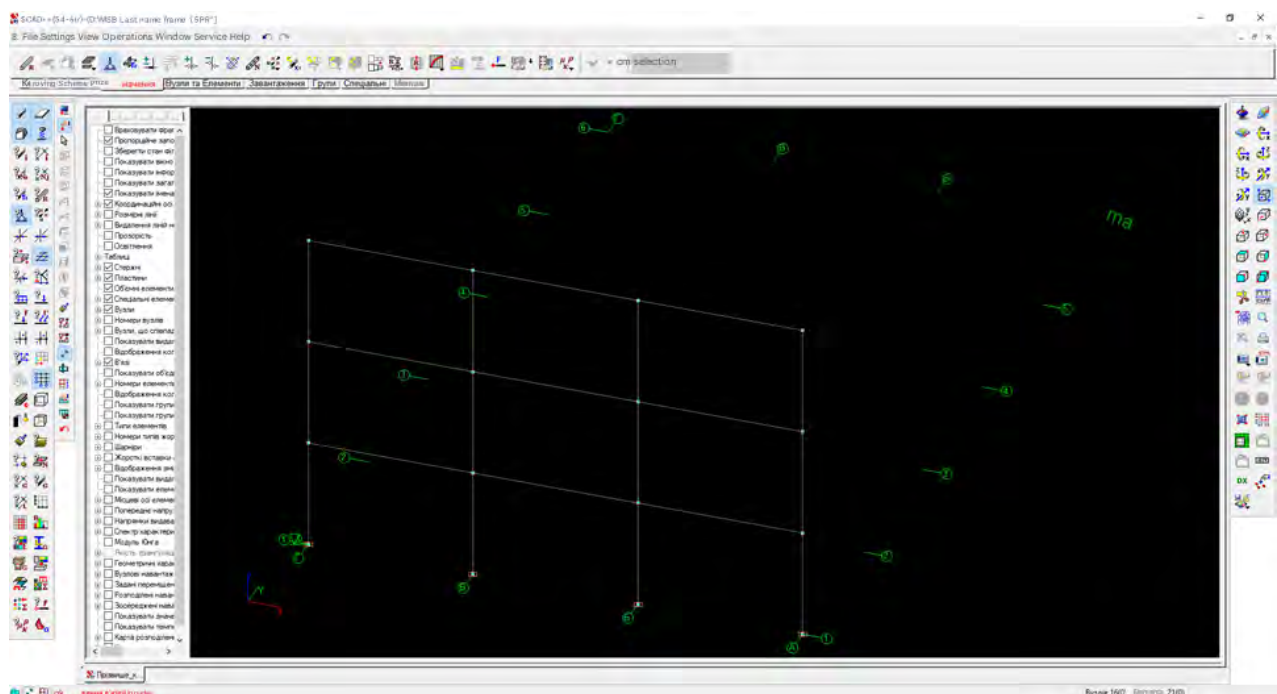


Figure 3.23 – Displaying elms in the working field of the program

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.

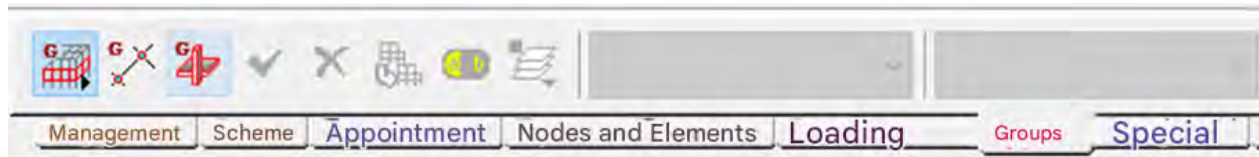


Figure 3.24 – "Mark items" button on the "Groups" tab

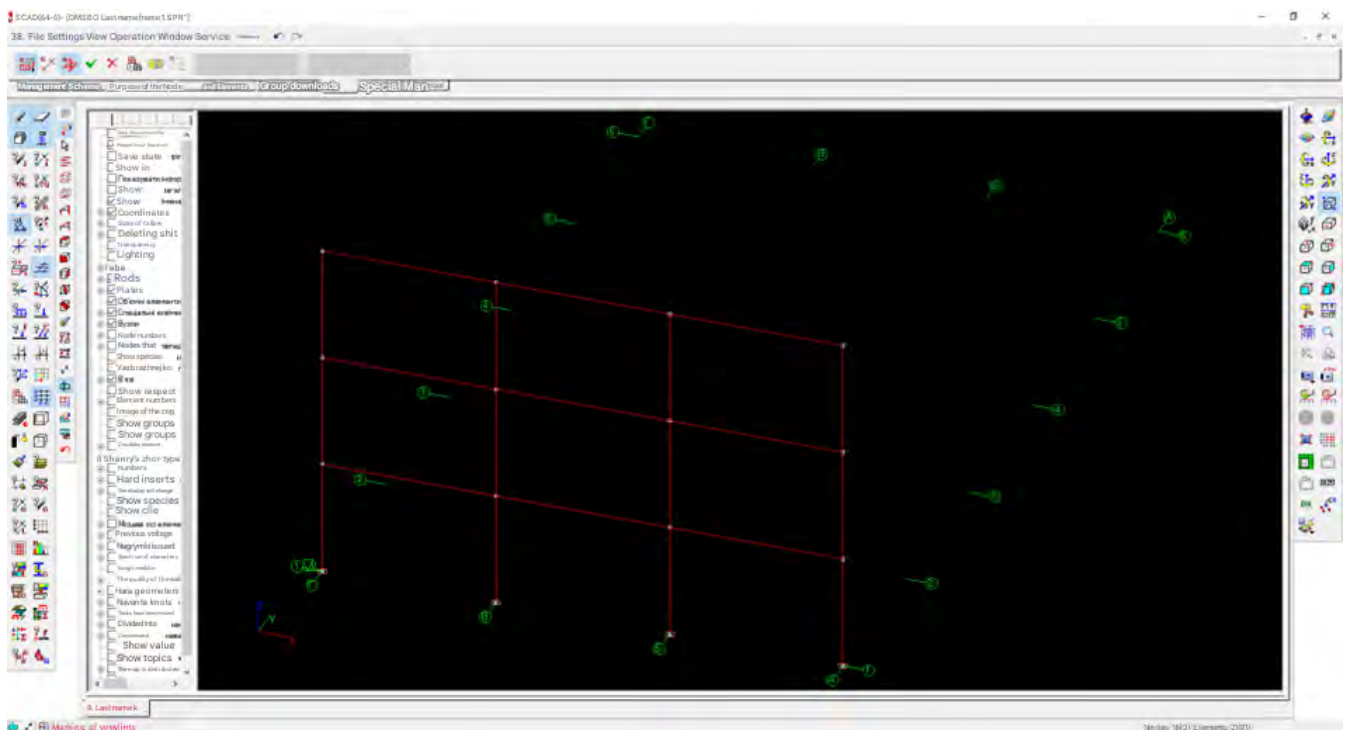


Figure 3.25 – The result of selecting frame elements in the working field of the program

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

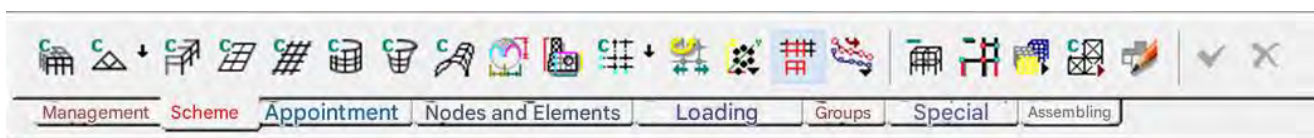


Figure 3.26 – Diagram toolbar

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.

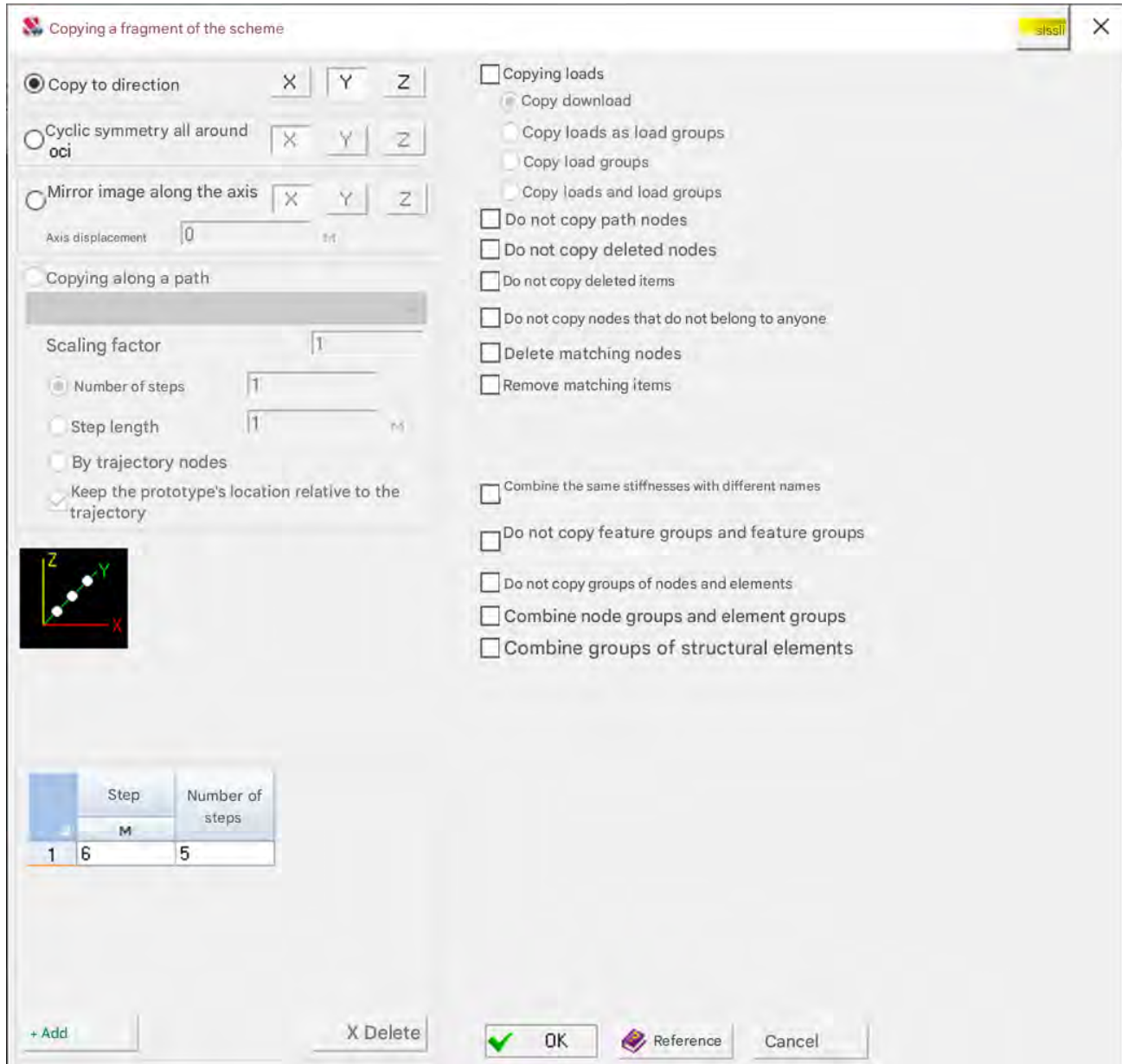


Figure 3.27 – Copy Schema Fragment Dialog Box

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

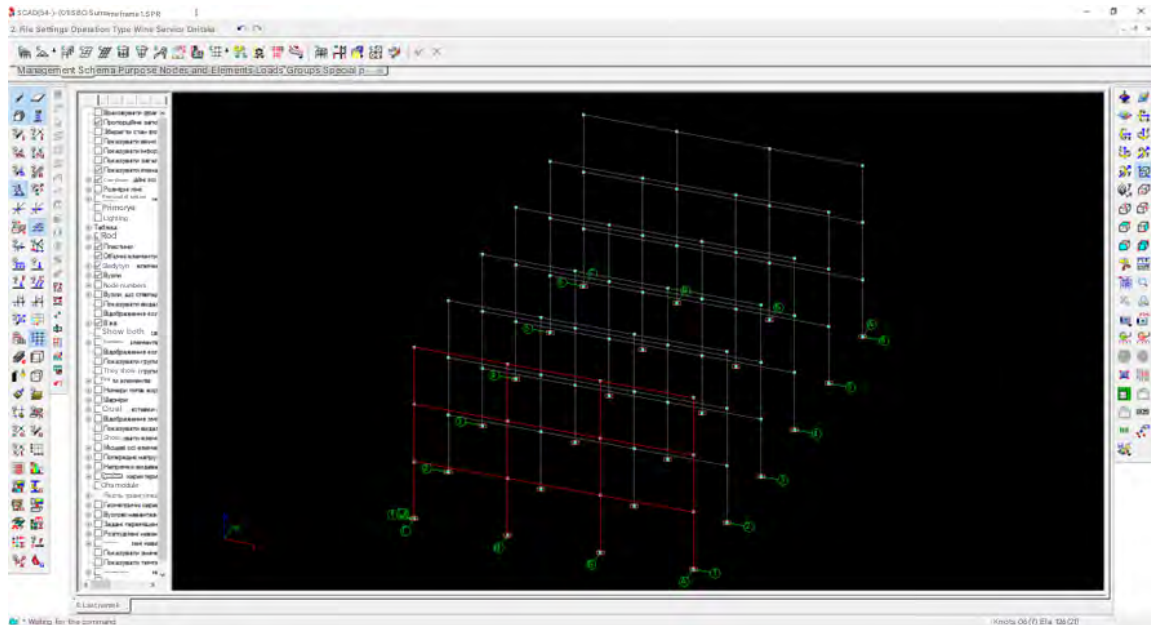


Figure 3.28 – The result of copying in the working window of the program

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.

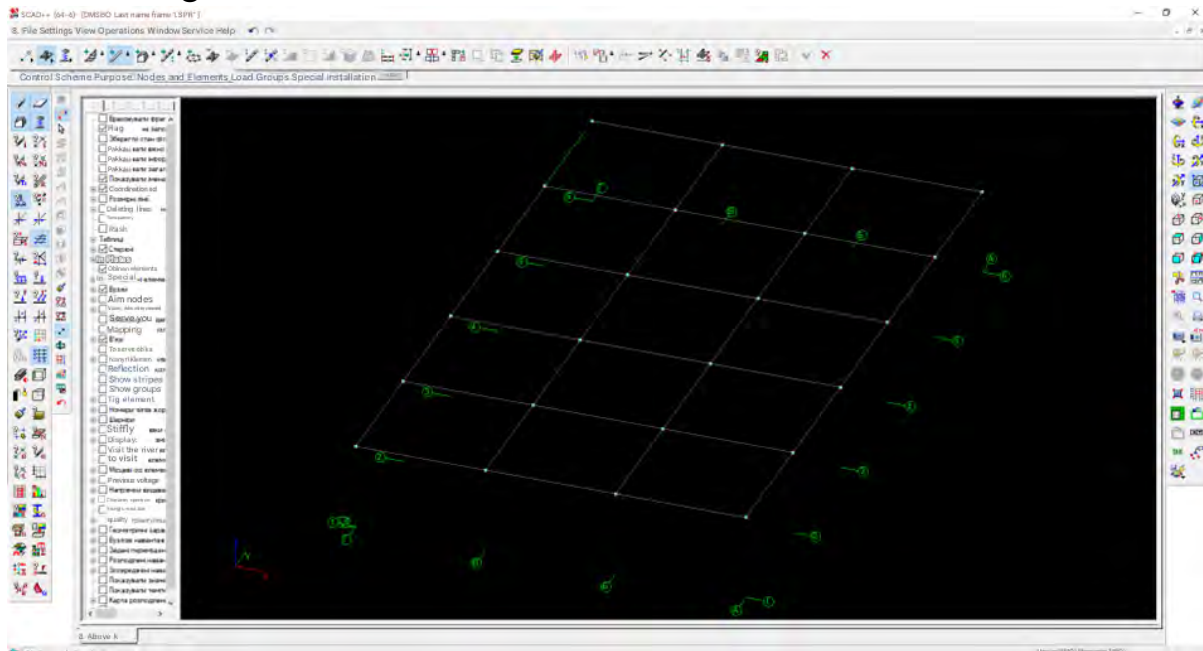


Figure 3.29 – Connecting adjacent frames with horizontal rods

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.

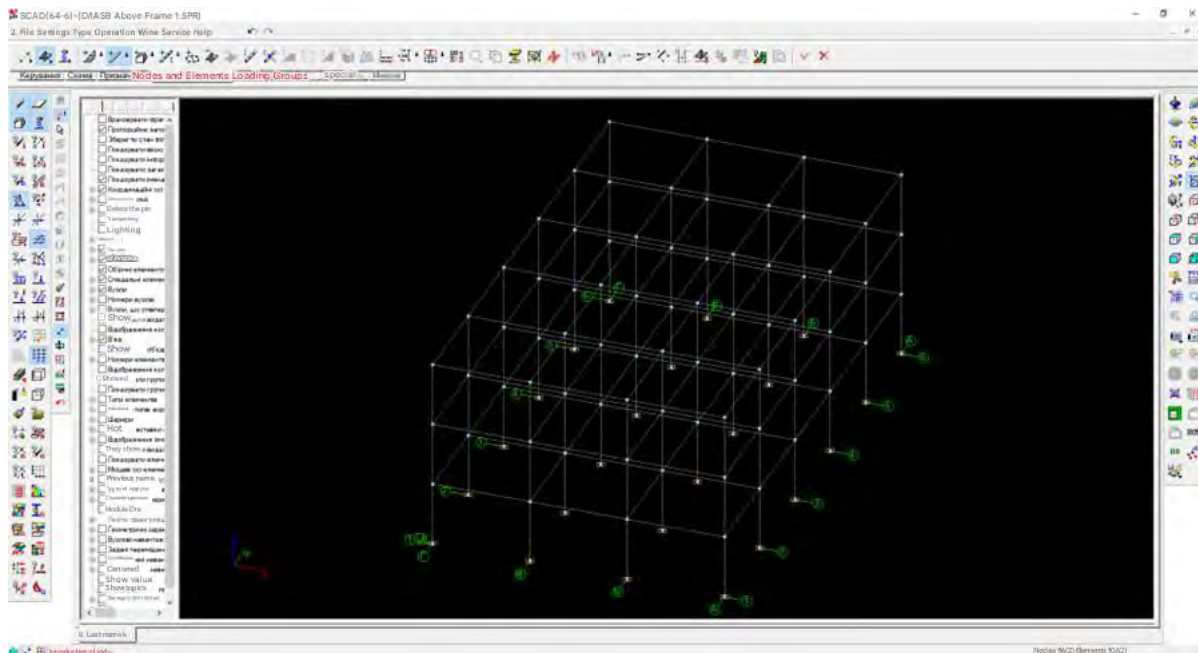


Figure 3.30 – Spatial display of the design scheme

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.

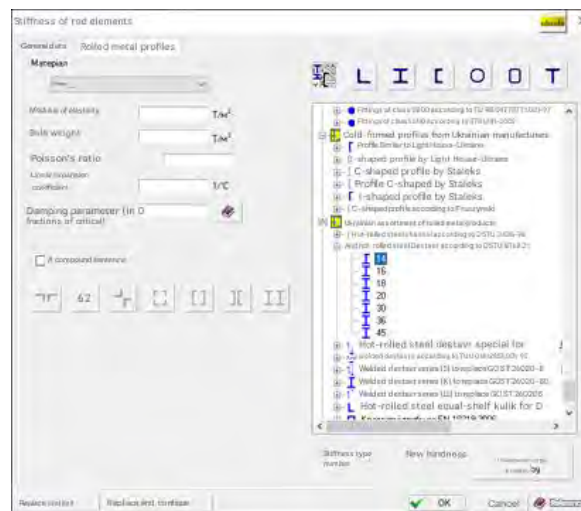


Figure 3.31 – Selection of assortment for the selection of stiffeners of longitudinal beams

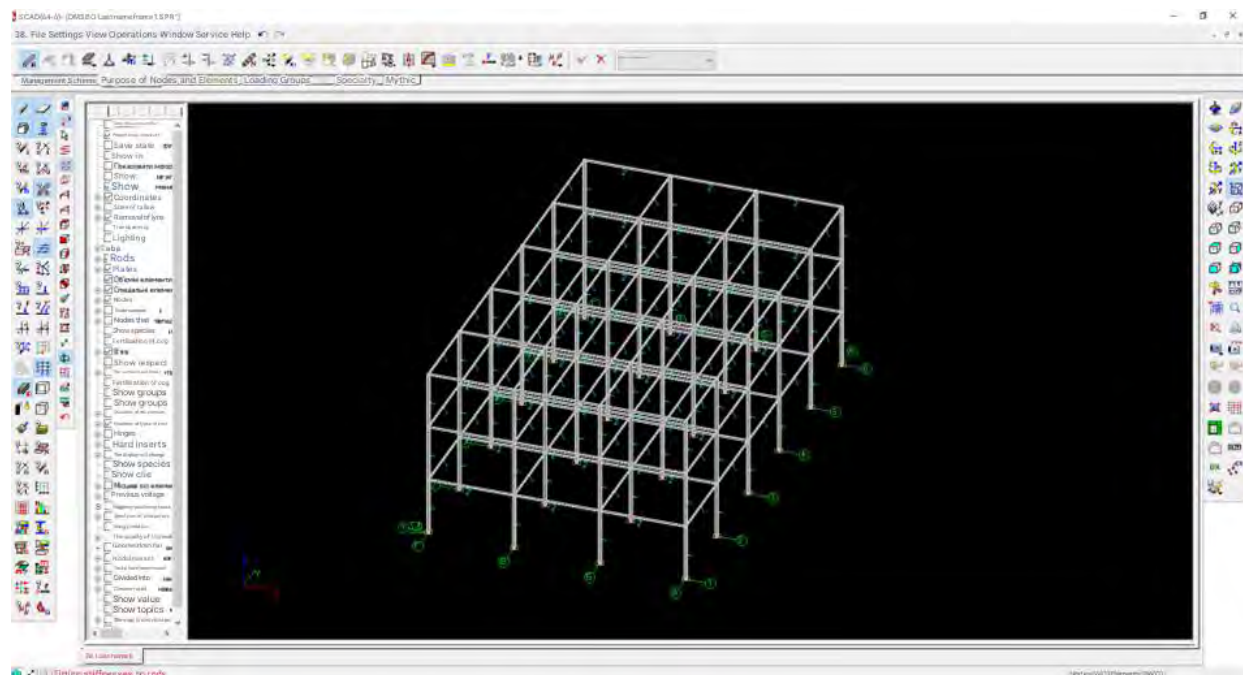


Figure 3.32 – Display of the circuit with assigned stiffeners

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".

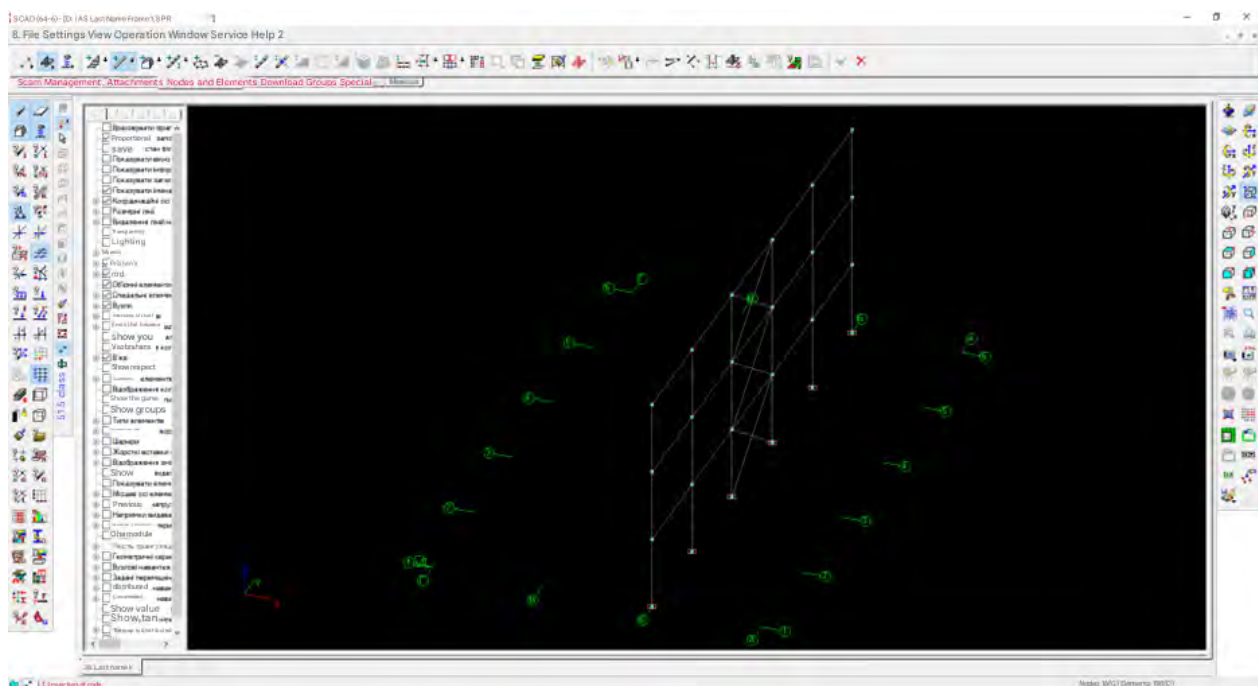


Figure 3.33 – Construction of vertical elms at the section along the B axis

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.

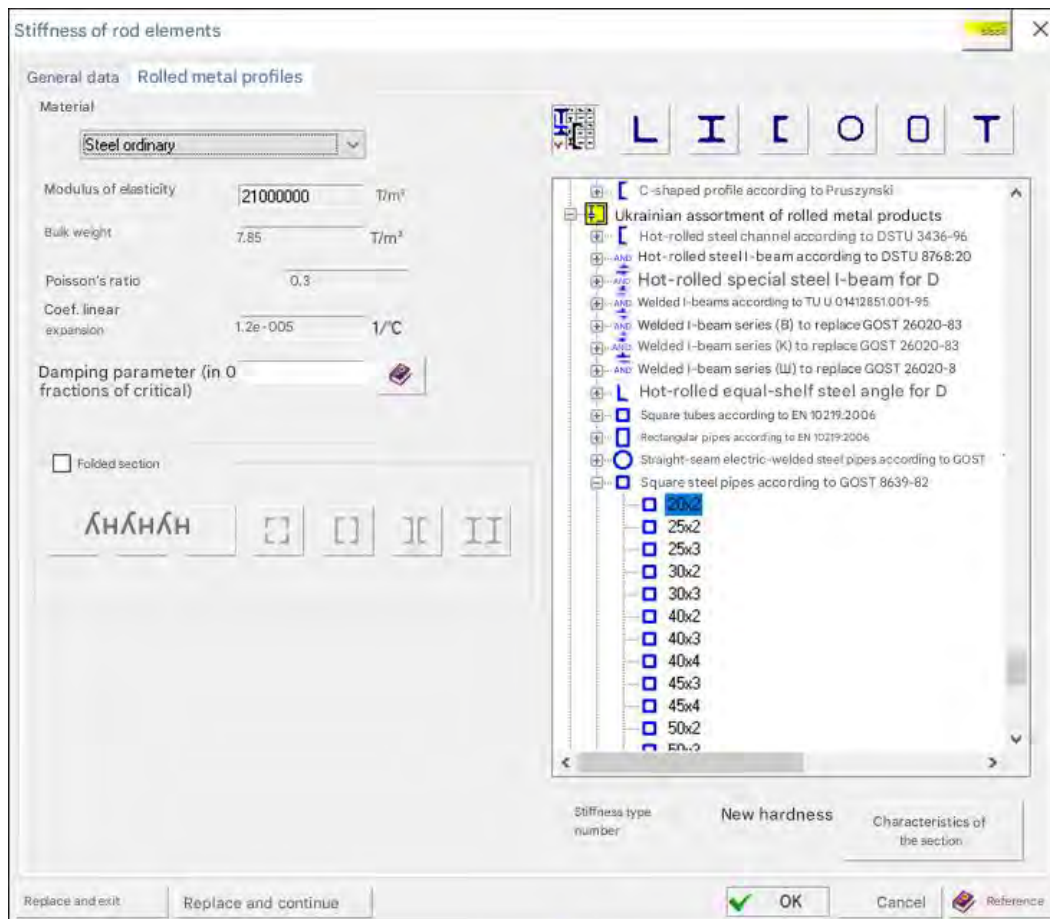


Figure 3.34 – Selection of assortment for assigning stiffness to vertical knits

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.

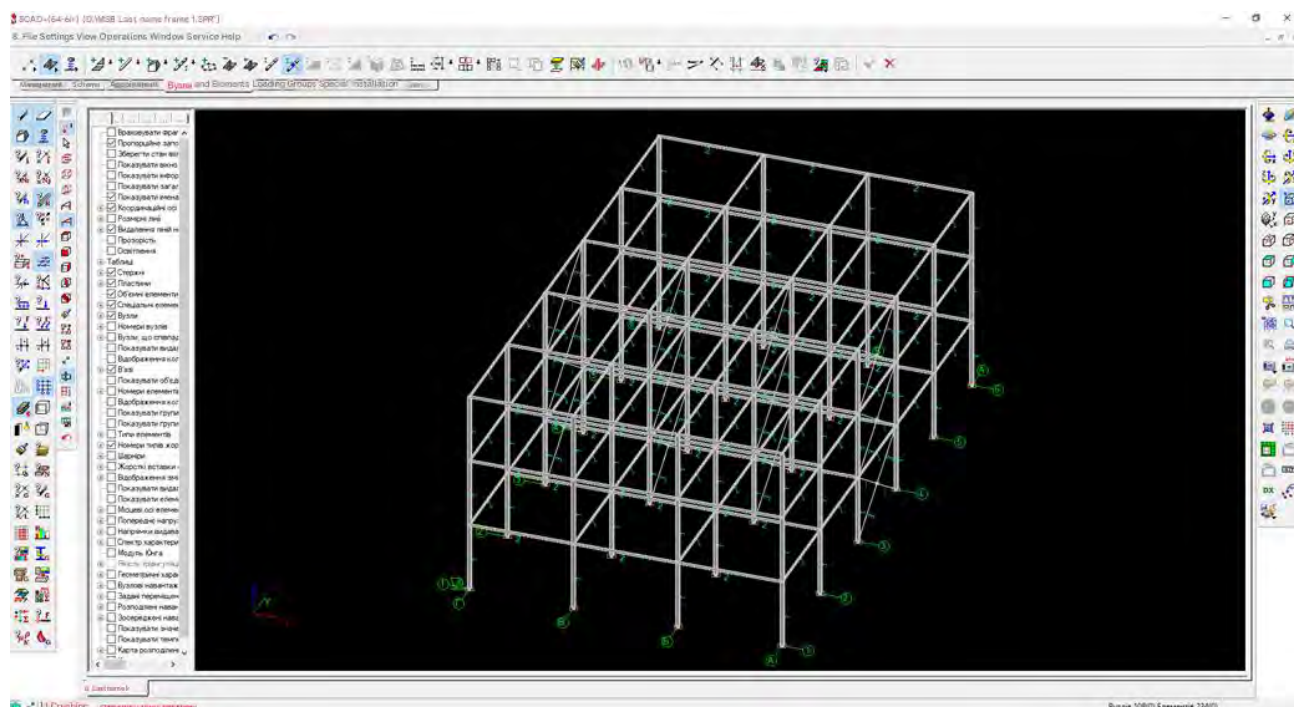
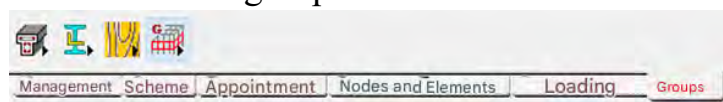


Figure 3.35 – Design diagram of the steel frame

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.

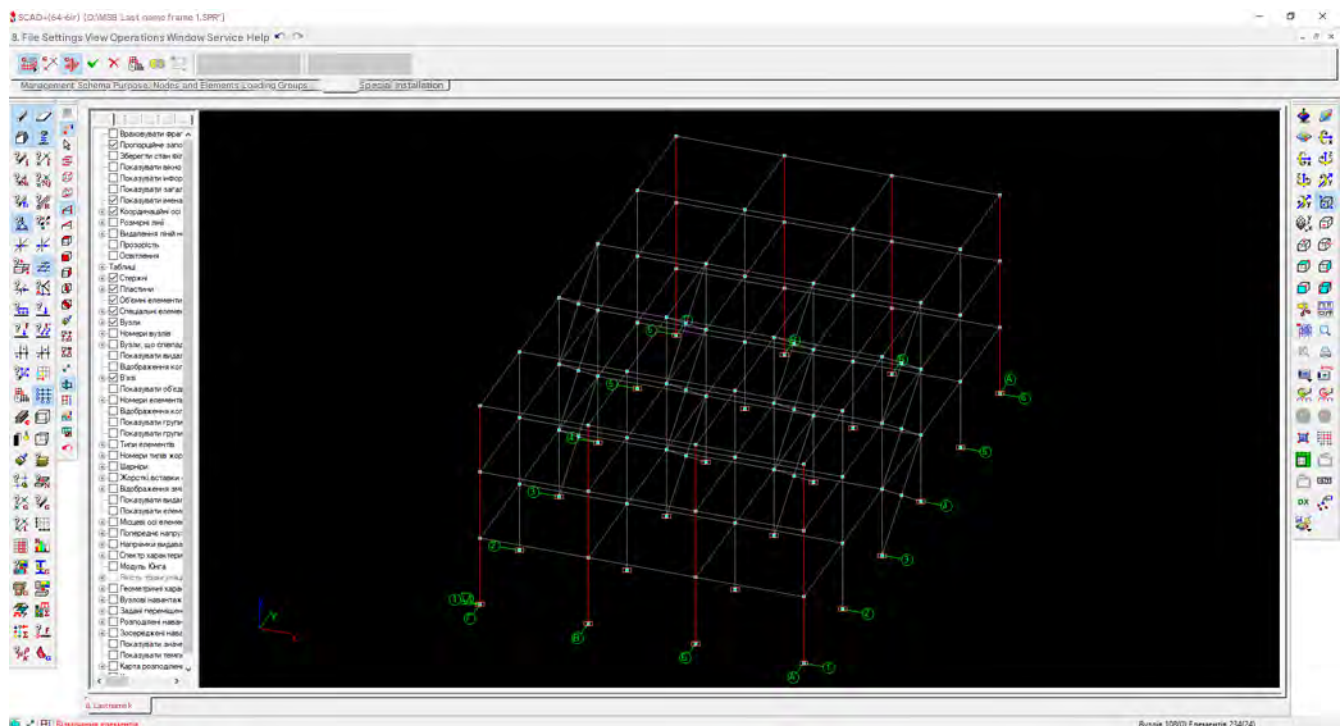


Figure 3.36 – Marking the elements of the group "columns of extreme frames"

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.

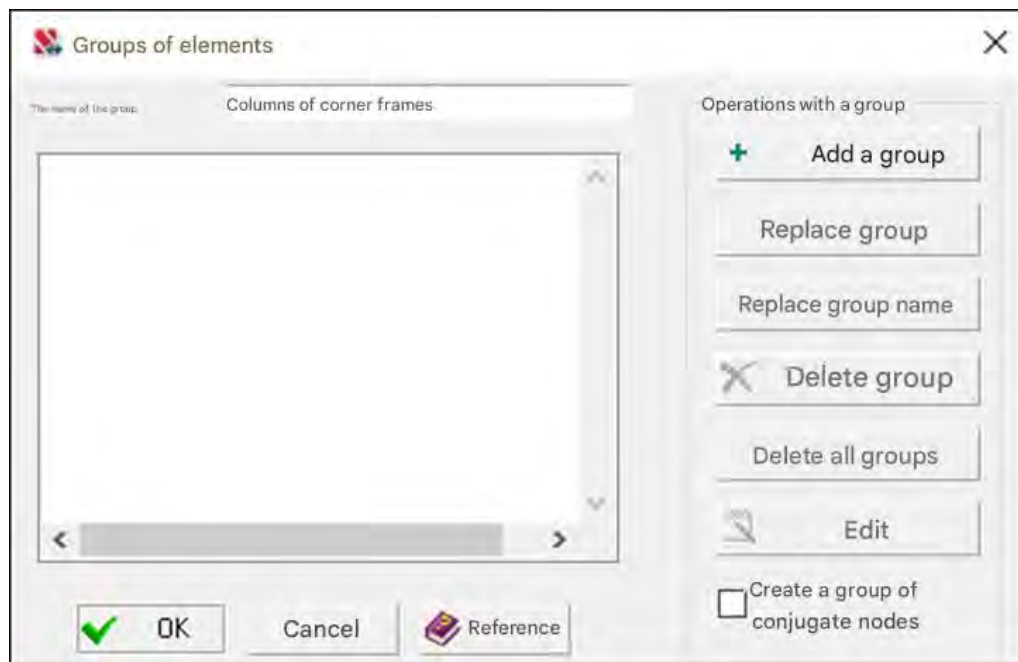


Figure 3.37 – Adding a group group "outermost frame columns" in the Item Groups dialog box

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

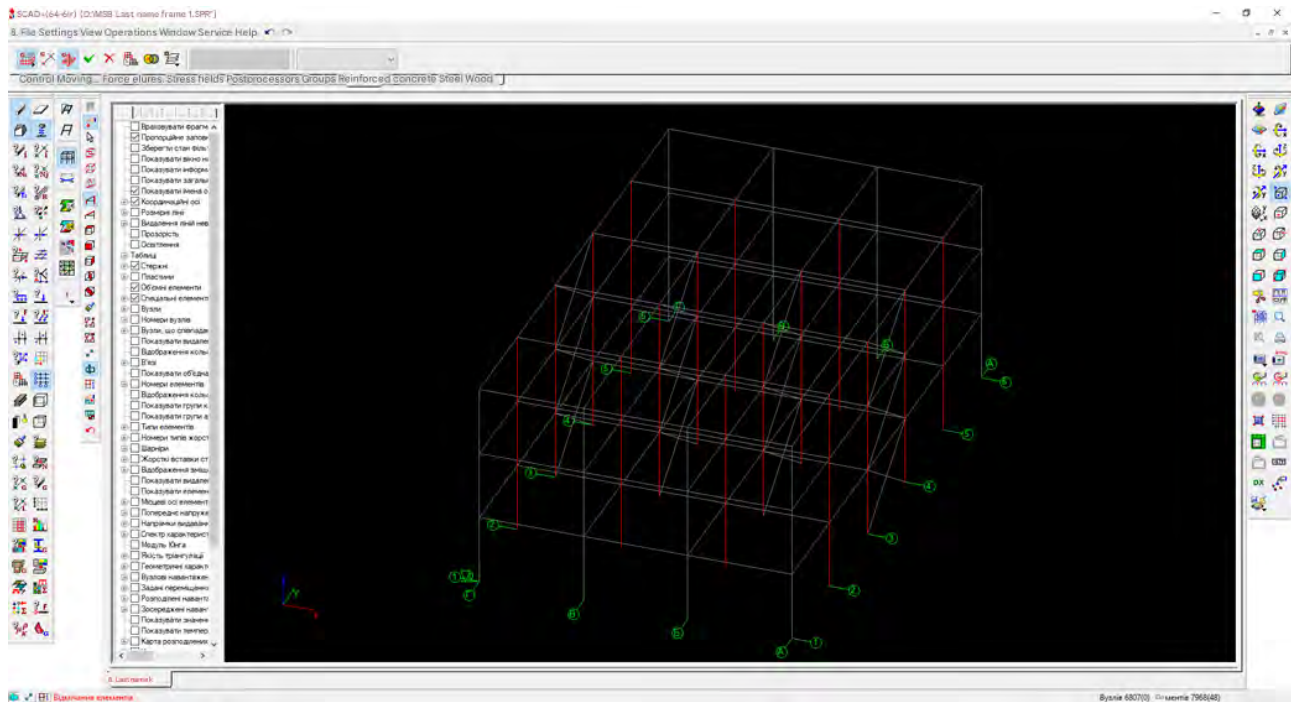


Figure 3.38 – Marking the elements of the group "columns of middle frames"

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.

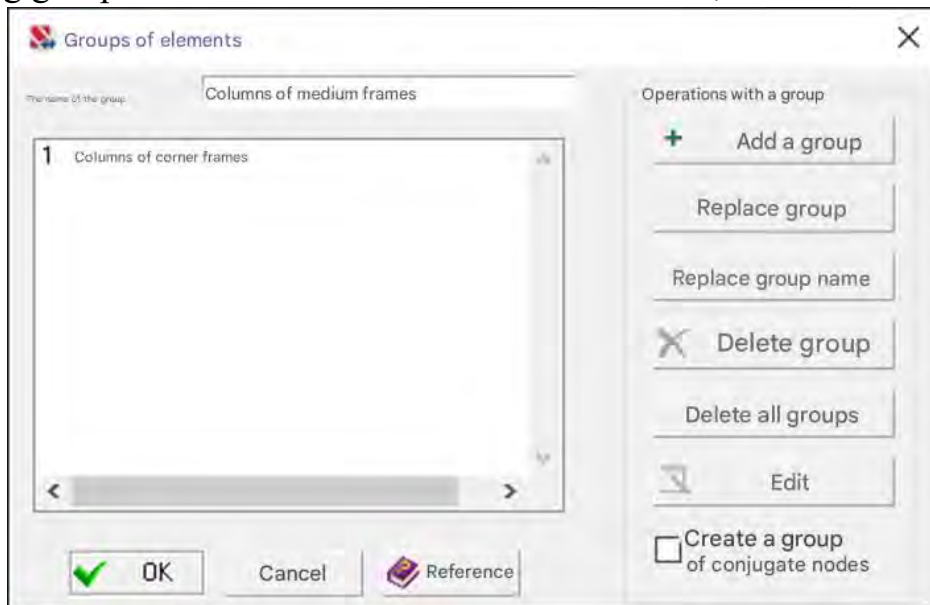


Figure 3.39 – Add a group group "columns of middle frames" in the "Groups of Elements" dialog box

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.

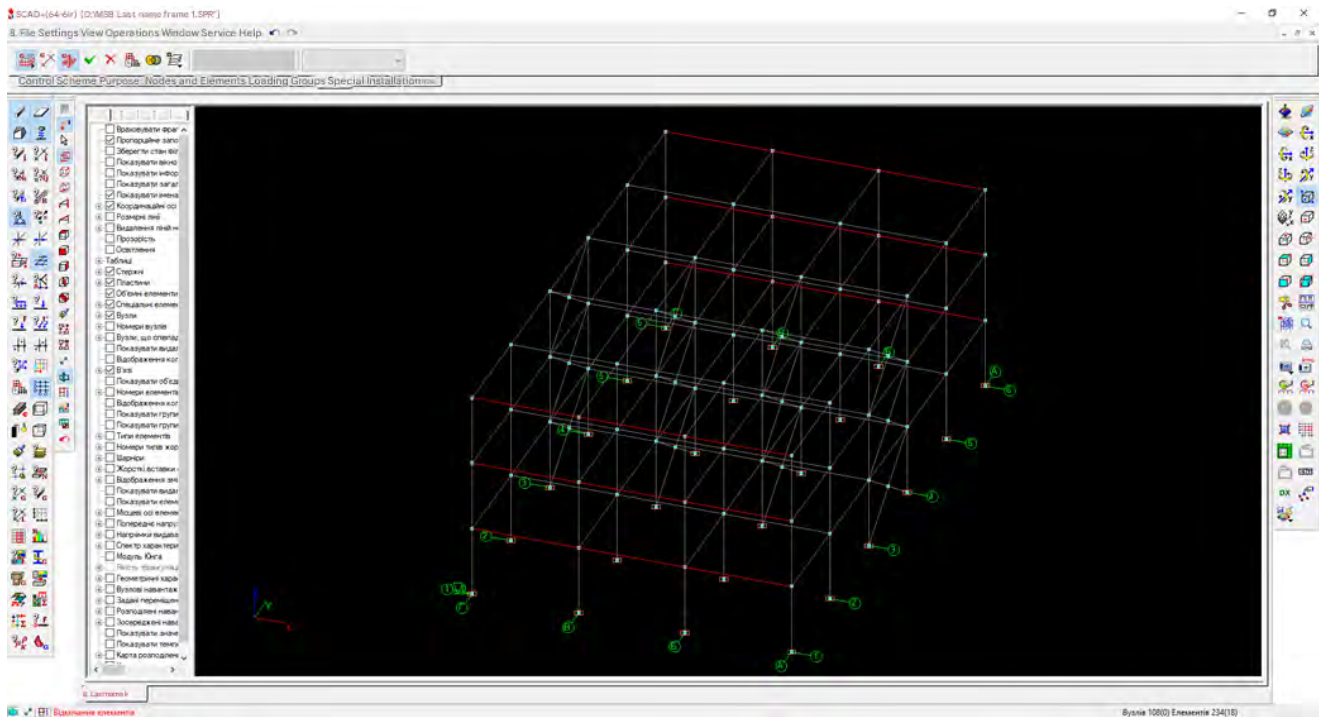


Figure 3.40 – Marking the elements of the group "crossbars of the extreme frames"

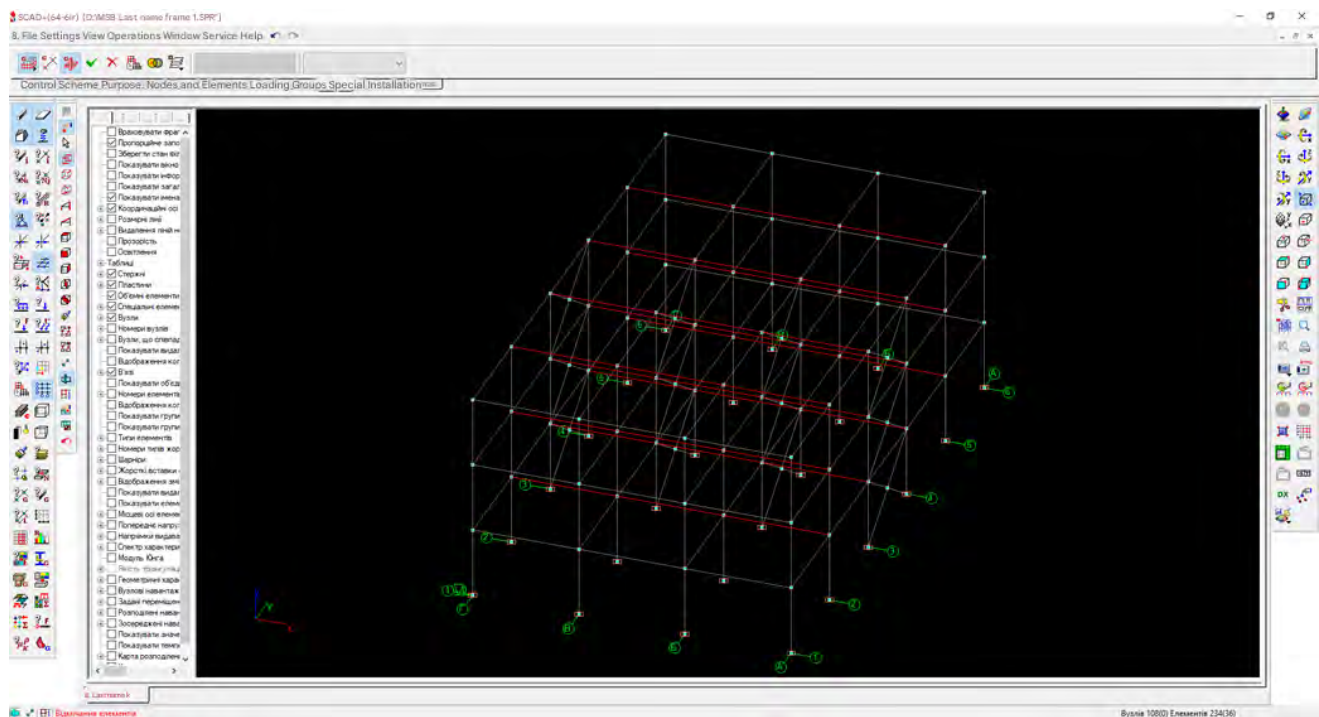


Figure 3.41 – Marking the elements of the group "crossbars of middle frames"

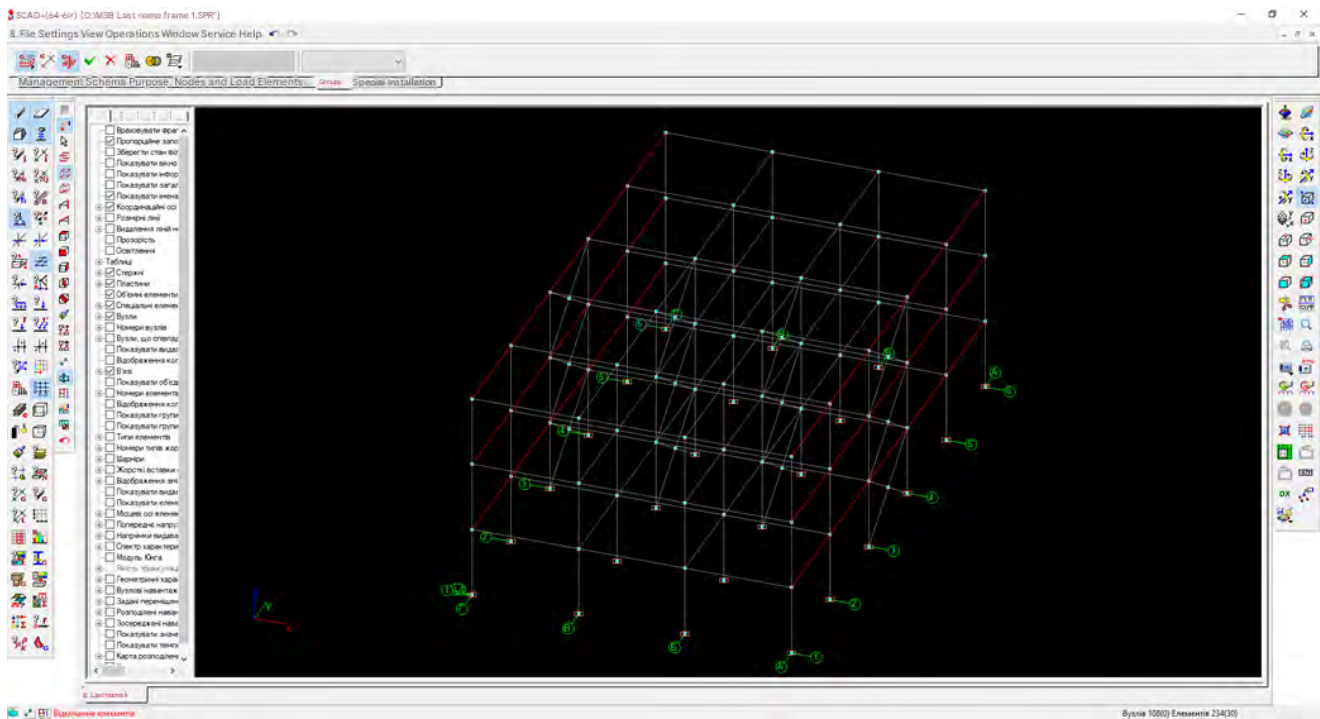


Figure 3.42 – Marking the elements of the group "longitudinal crossbars of extreme frames"

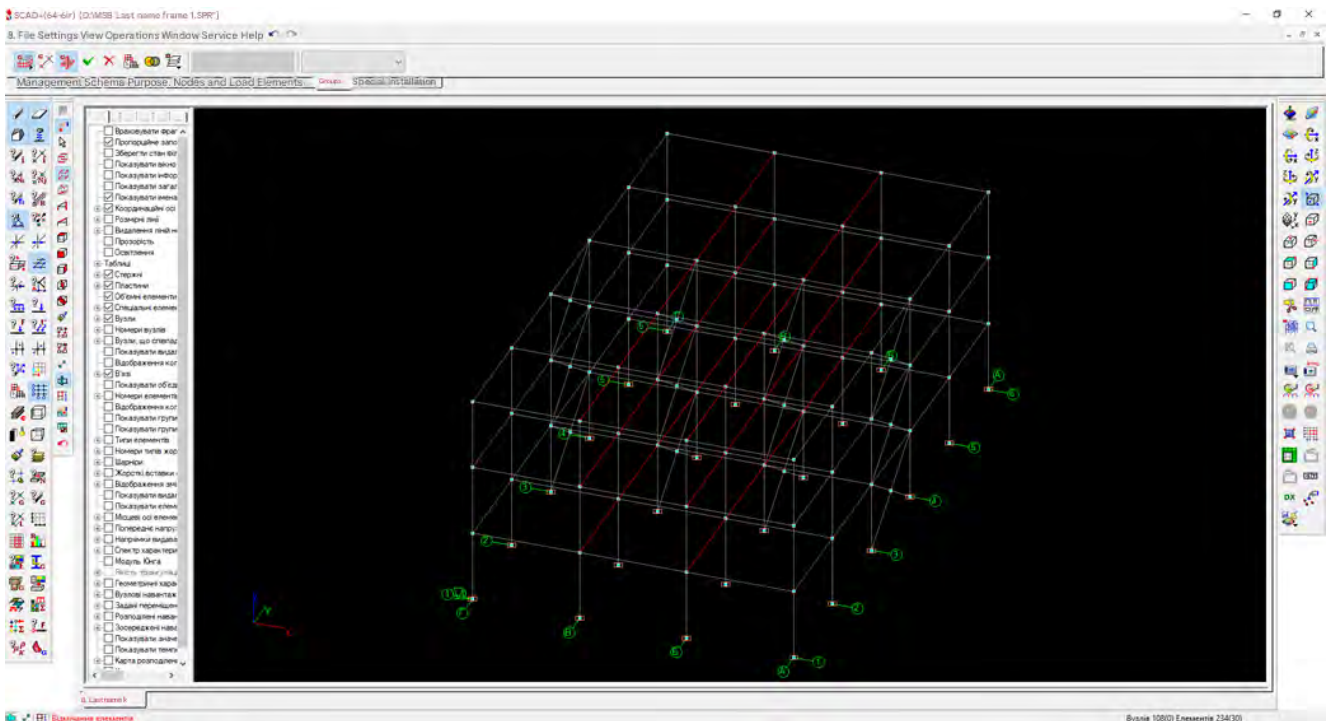


Figure 3.43 – Marking the elements of the group "longitudinal crossbars of middle frames"

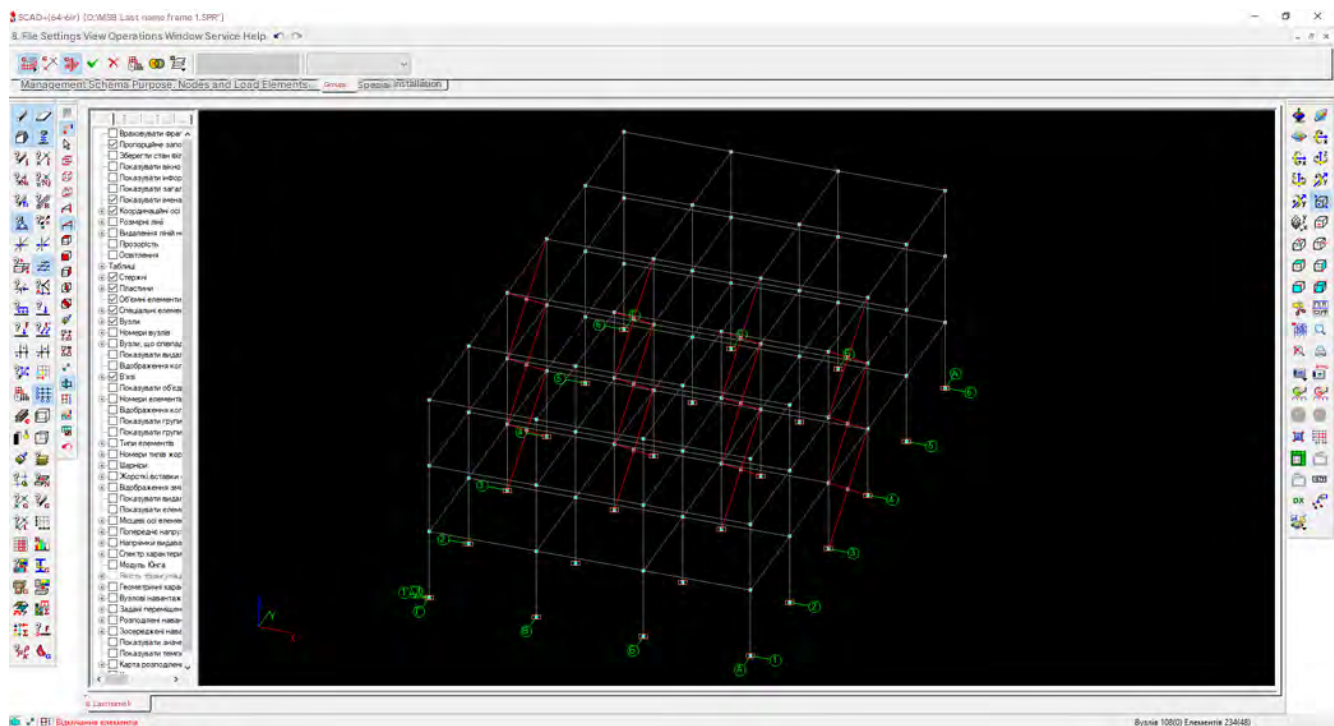


Figure 3.44 – Marking the elements of the group "vertical knits"

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".

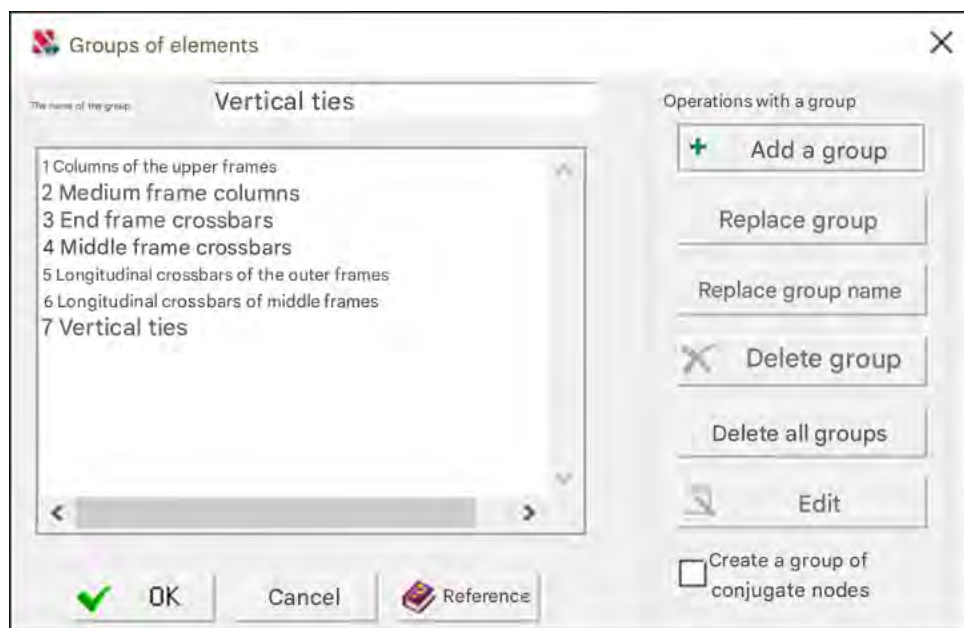


Figure 3.45 – Item Groups dialog box with created groups

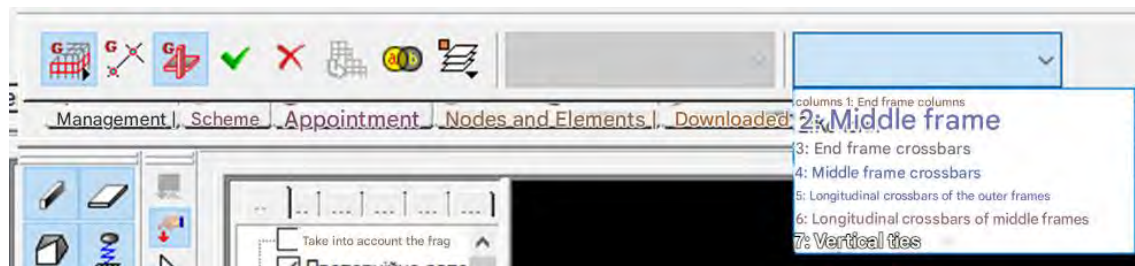


Figure 3.46 – Item Groups drop-down list

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.

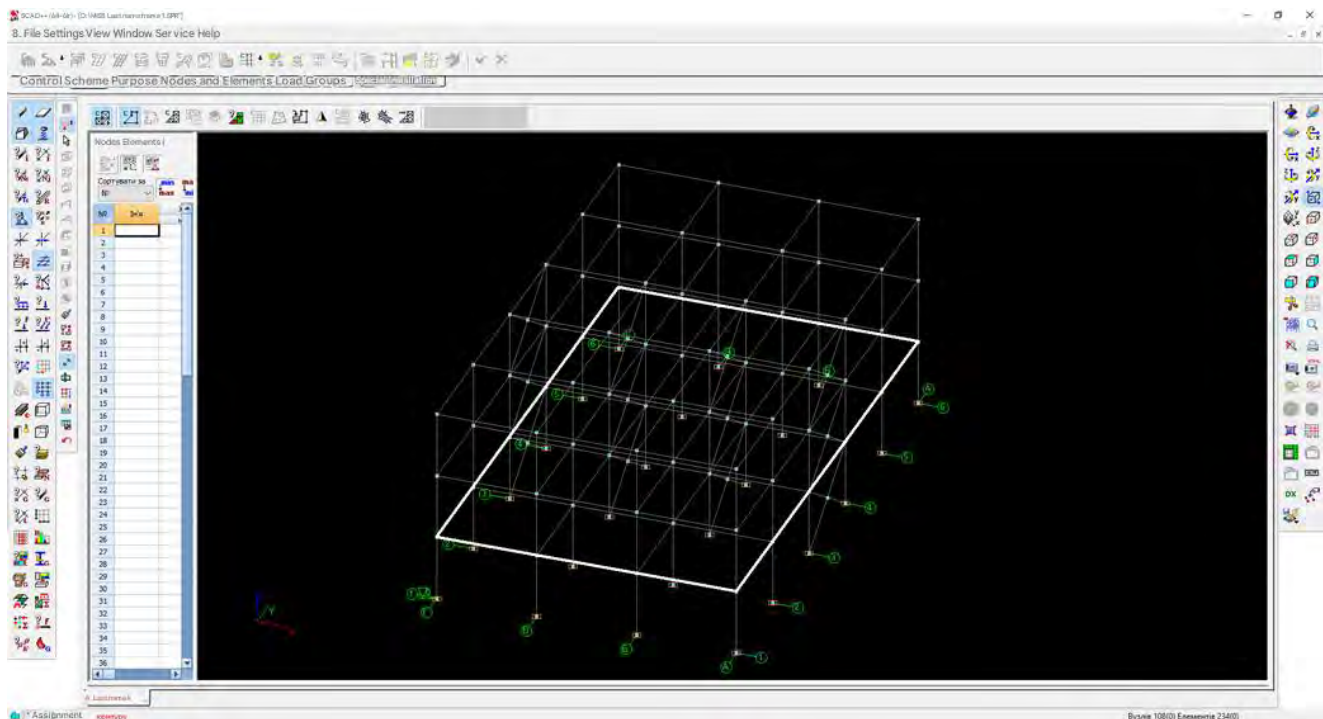


Figure 3.47 – Creating contours of floor floor triangulation on the design diagram

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

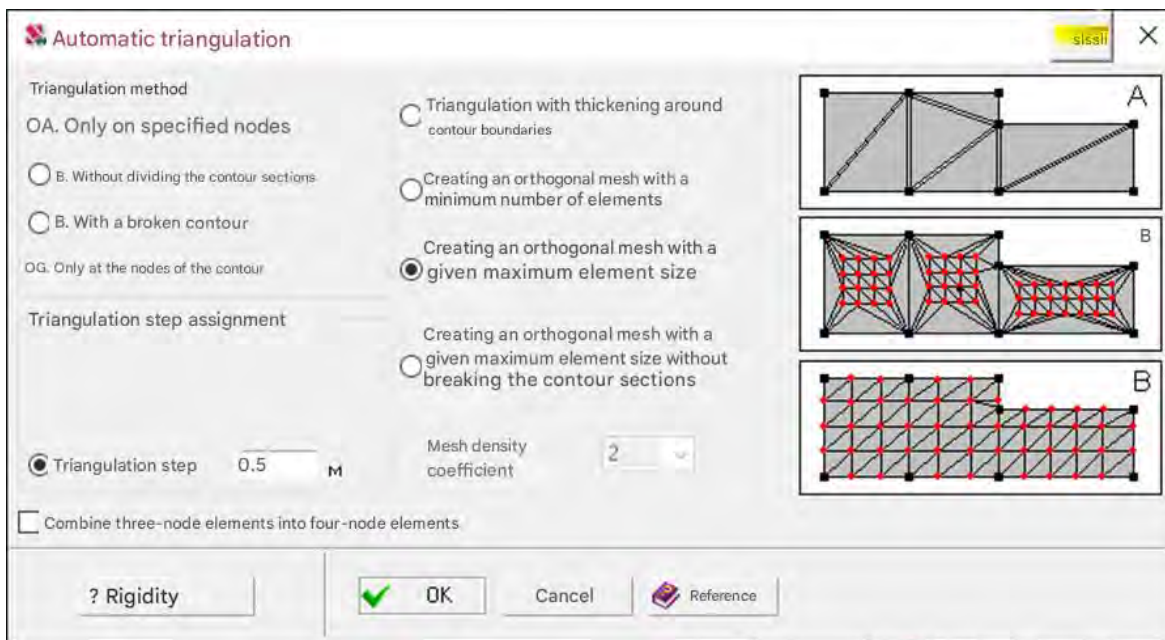


Figure 3.48 – Creating floor triangulation contours on the design diagram

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

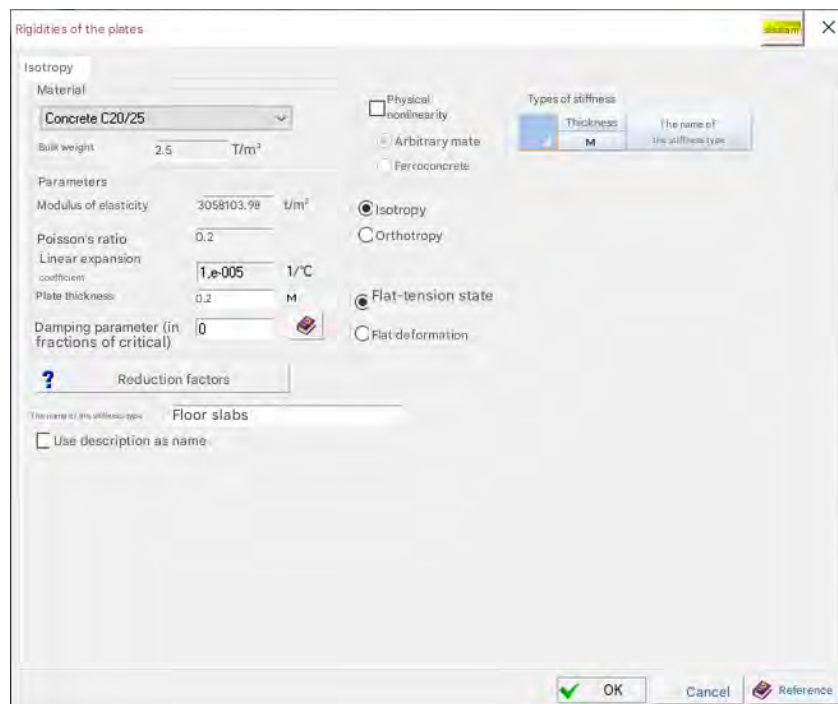


Figure 3.49 – Creation of floor triangulation contours on the design diagram

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

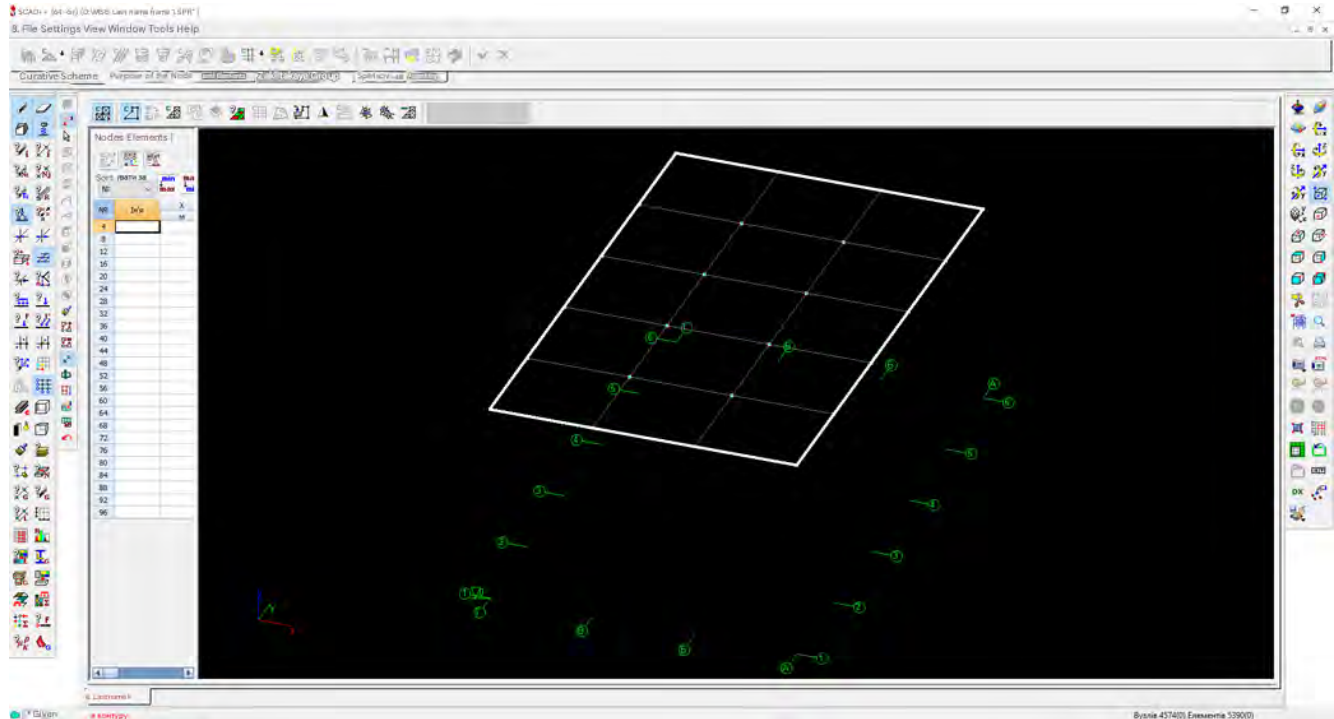


Figure 3.50 – Creating slab triangulation contours on the design diagram using the "XOY Plane Section" tool

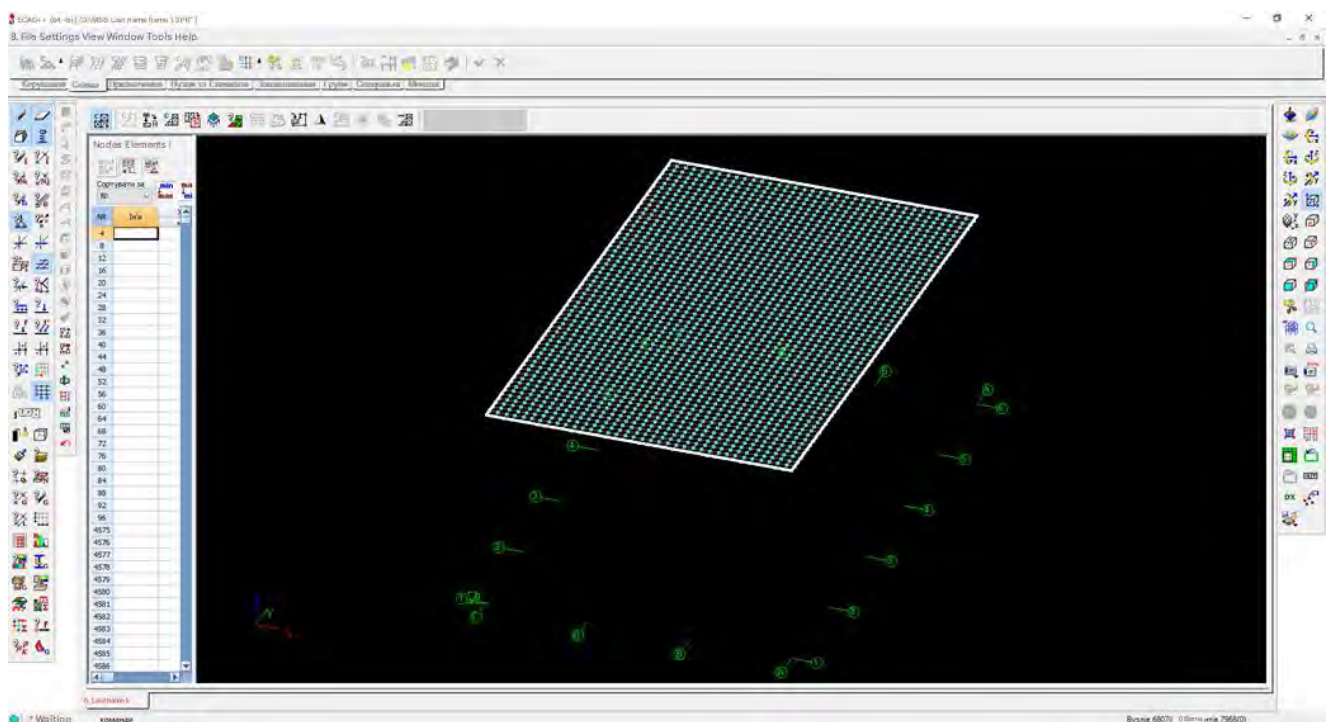


Figure 3.51 – Results of floor triangulation

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

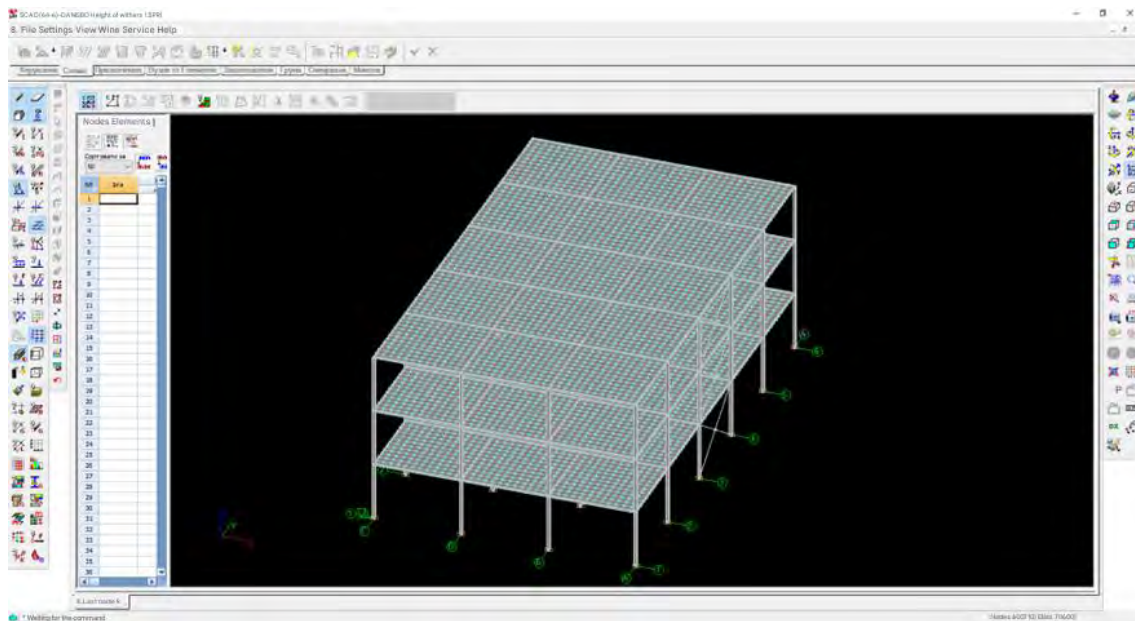


Figure 3.52 – Results of creating a design scheme with the display of the intended profiles of the steel frame and reinforced concrete floors and coatings

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.

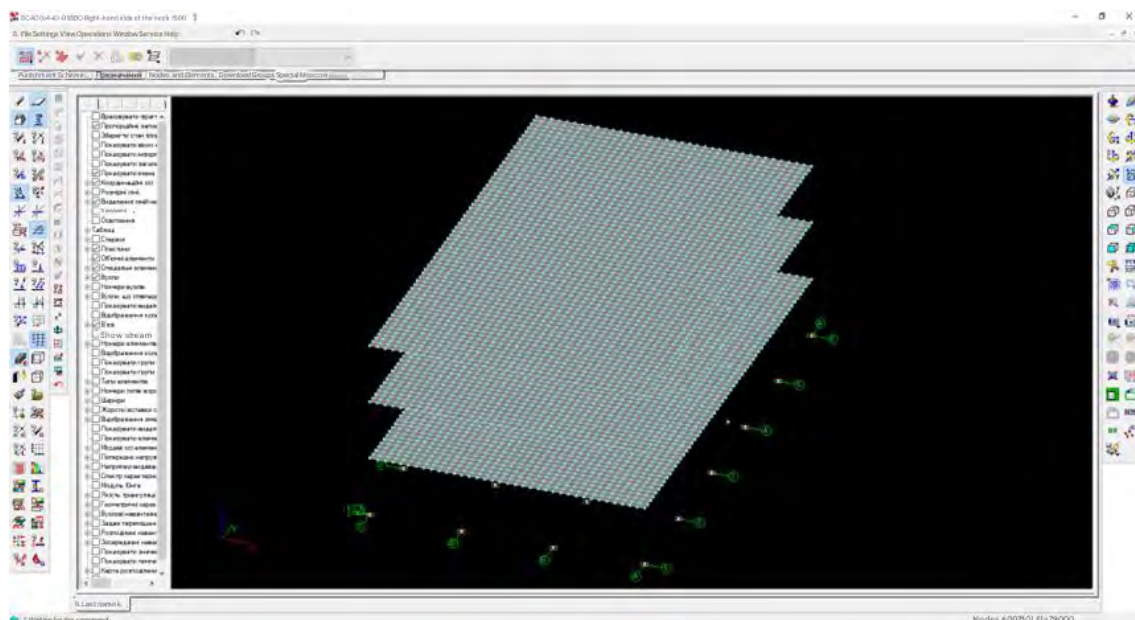


Figure 3.53 – Design scheme with switched off rod elements

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".

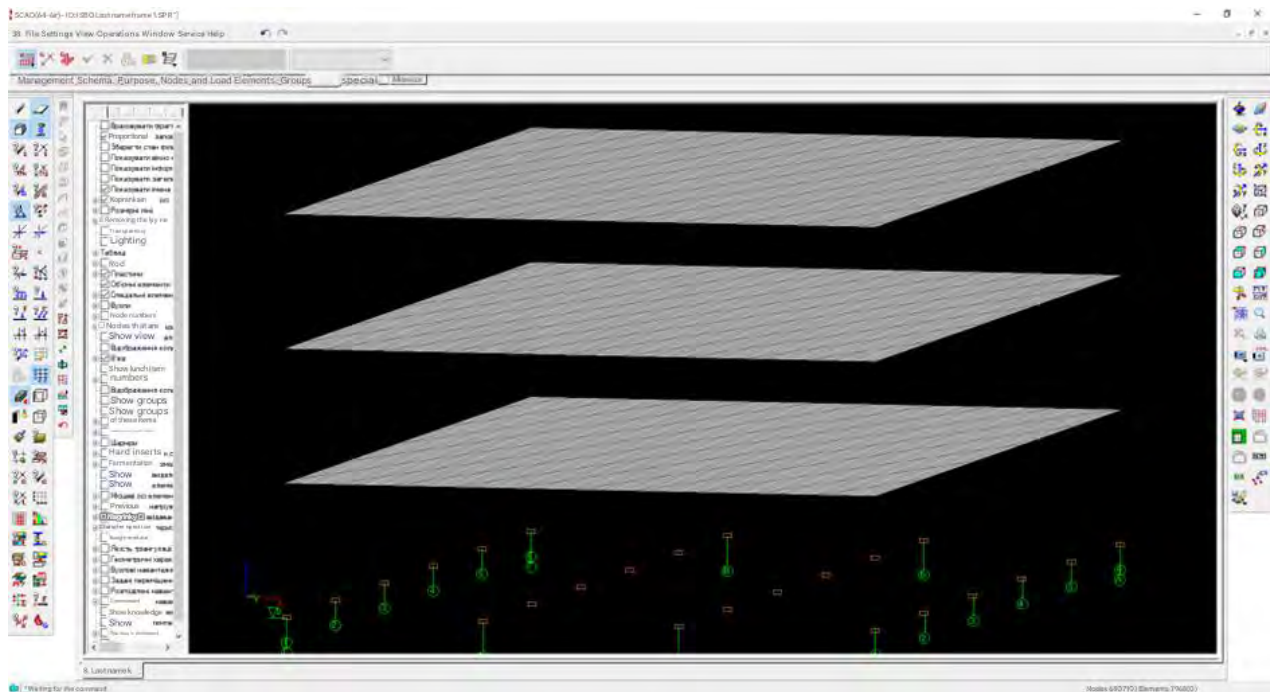


Figure 3.54 – Floor discs and coatings rotated for ease of operation

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.

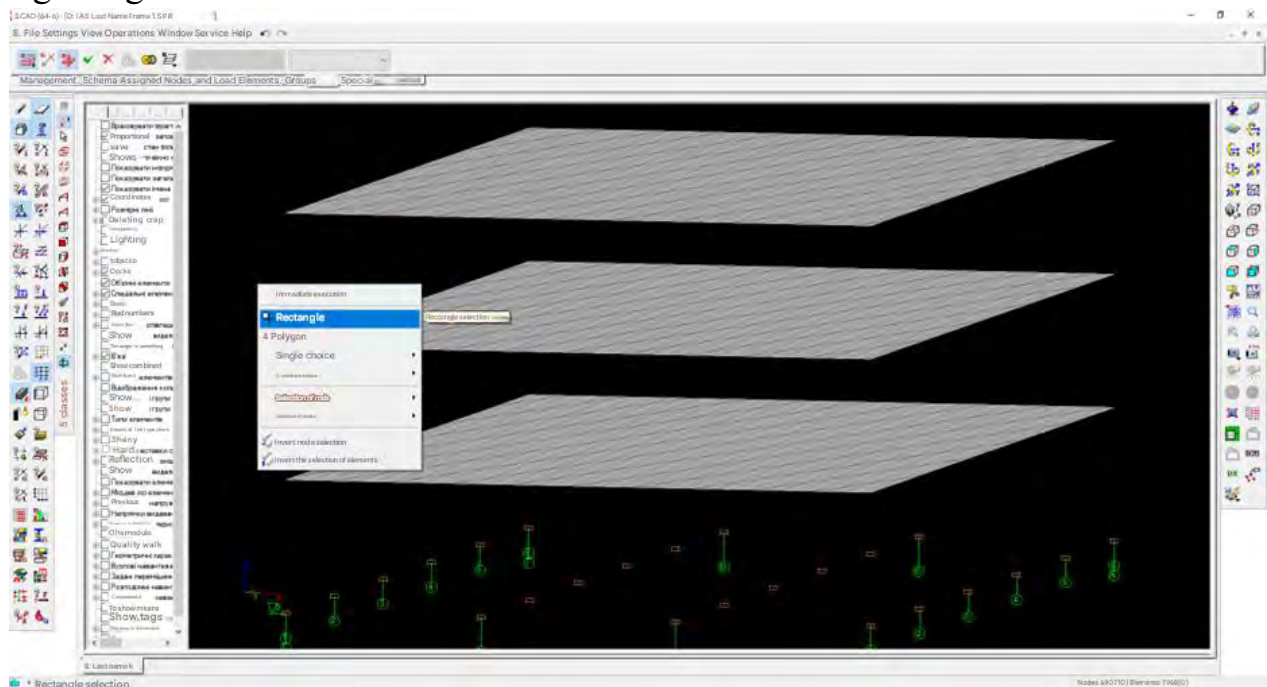


Figure 3.55 – Activating the Rectangle Select tool

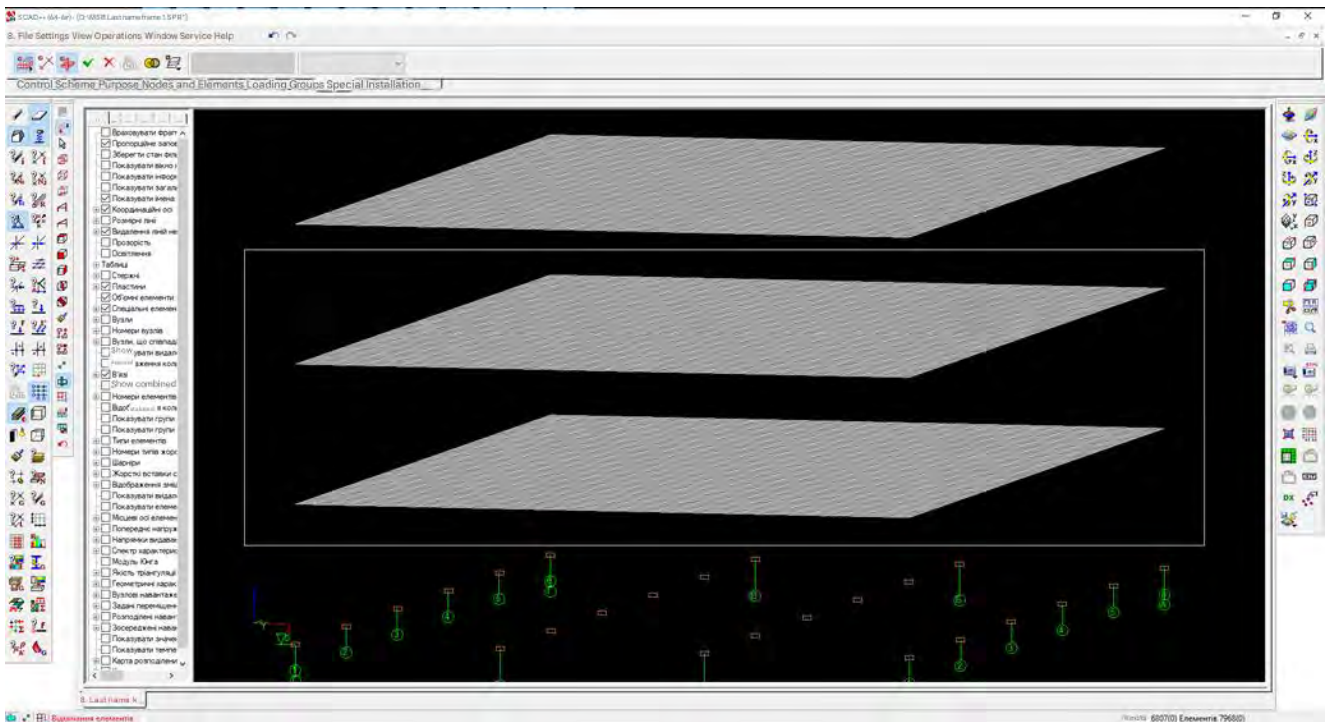


Figure 3.56 – Using the Rectangle Selection tool

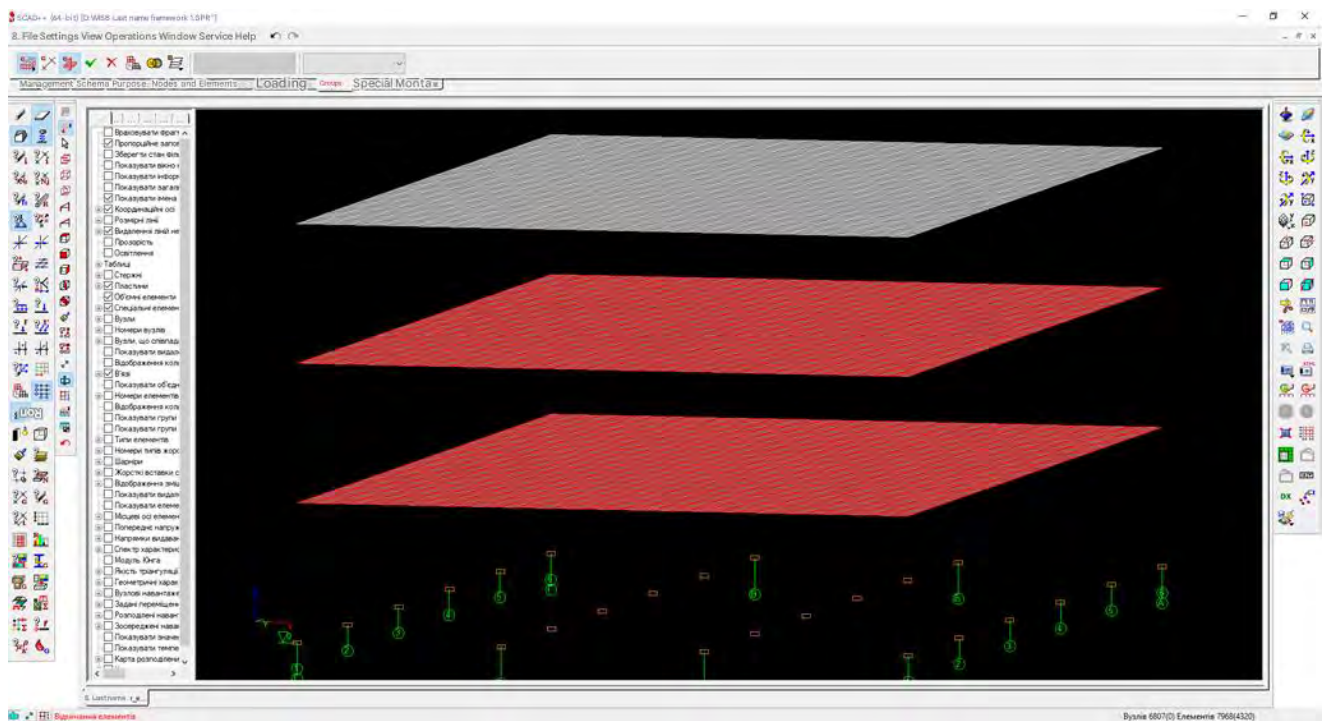


Figure 3.57 – Results of using the Rectangle Selection tool

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.

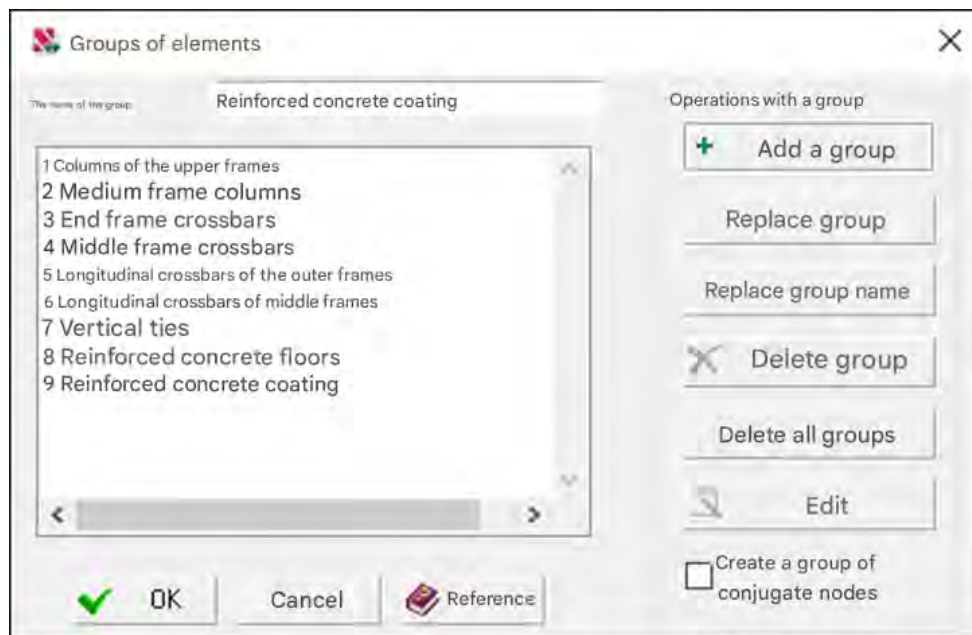


Figure 3.60 – List of created groups in the Item Groups dialog box

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".

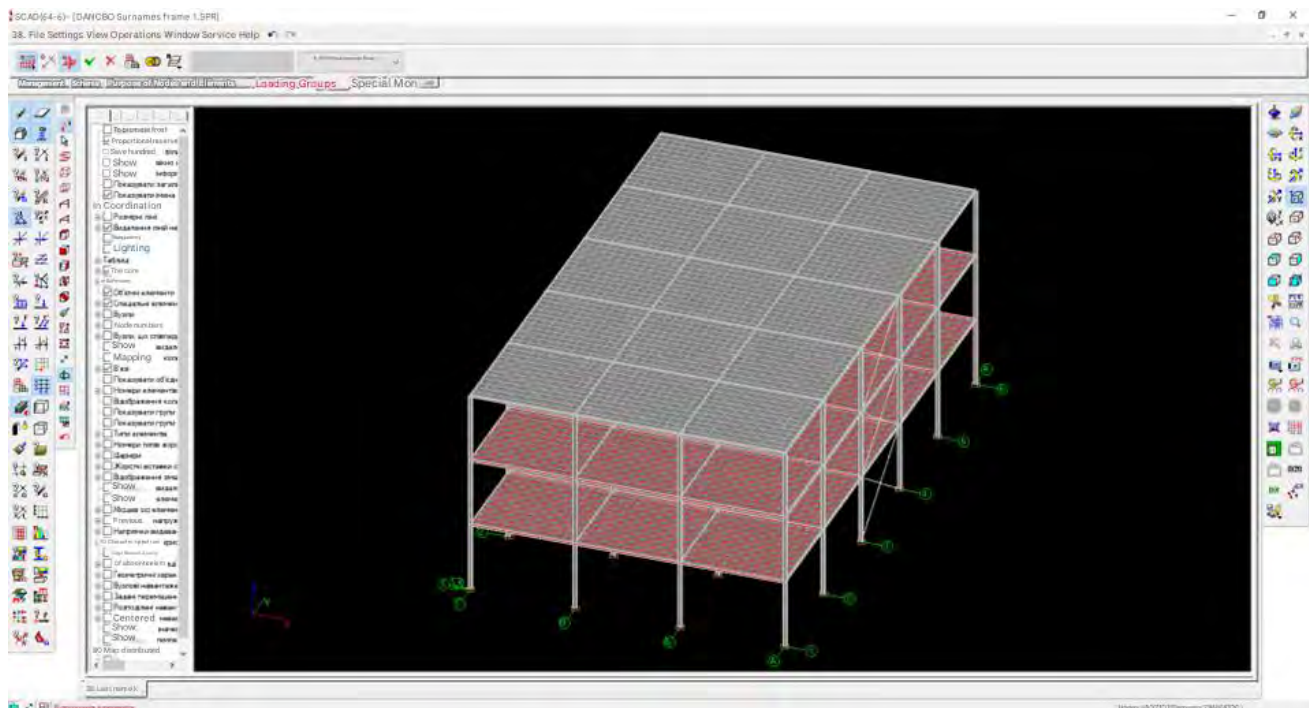


Figure 3.61 – Activated group "Reinforced concrete floors" on the design diagram

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.

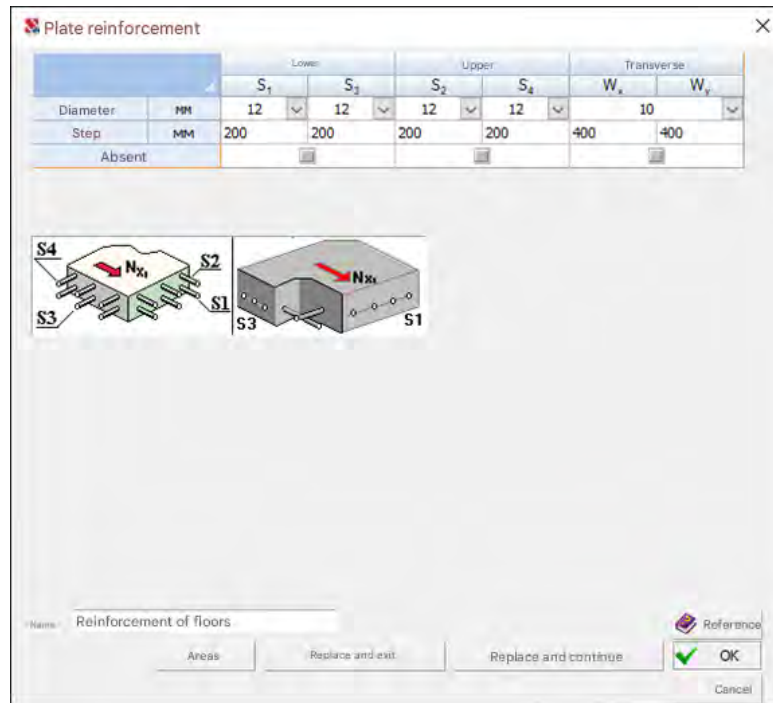


Figure 3.62 – Purpose of reinforcement for reinforced concrete floor slabs

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

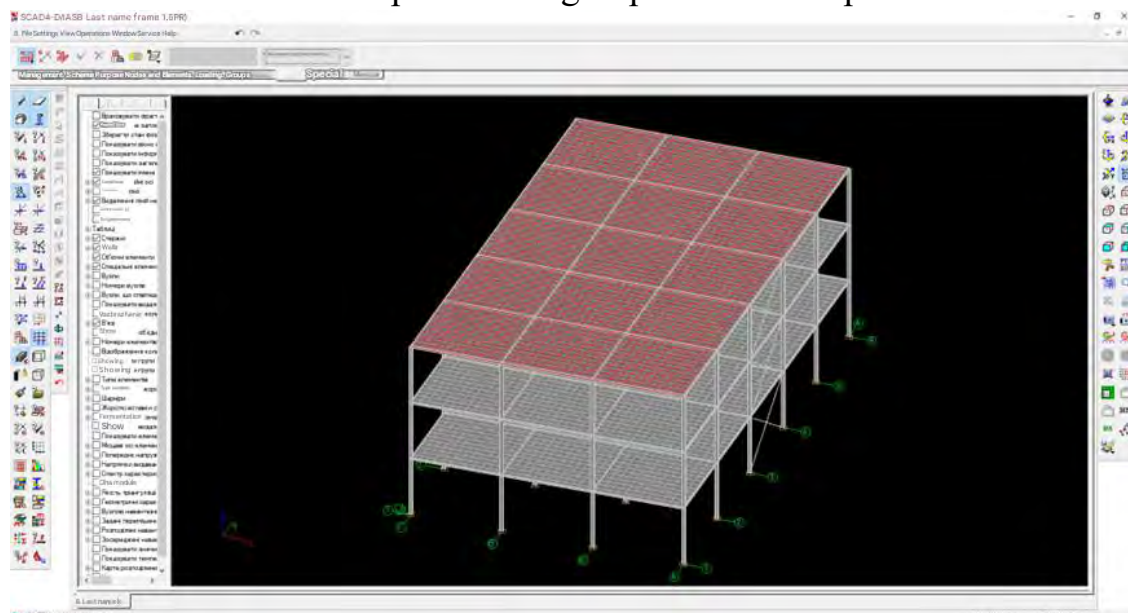


Figure 3.63 – Activated group "Reinforced concrete pavement" on the design diagram

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

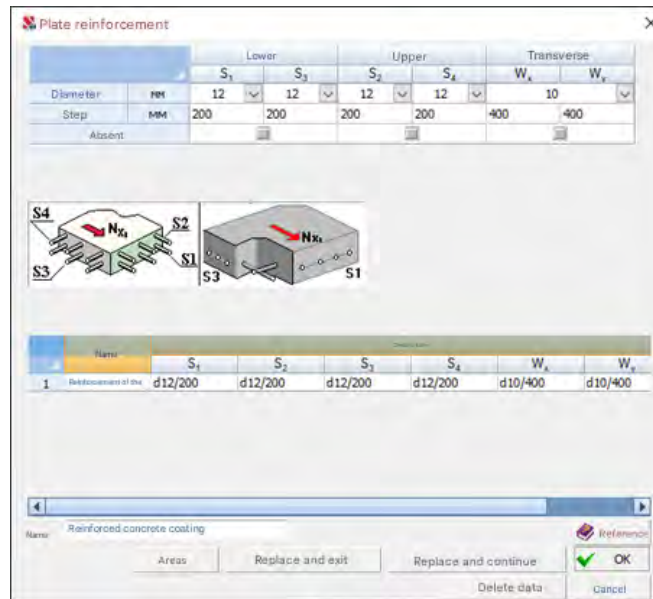


Figure 3.64 – Purpose of reinforcement for reinforced concrete coating slab

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  of the WEST program.



Figure 3.65 – Dead weight dialog box

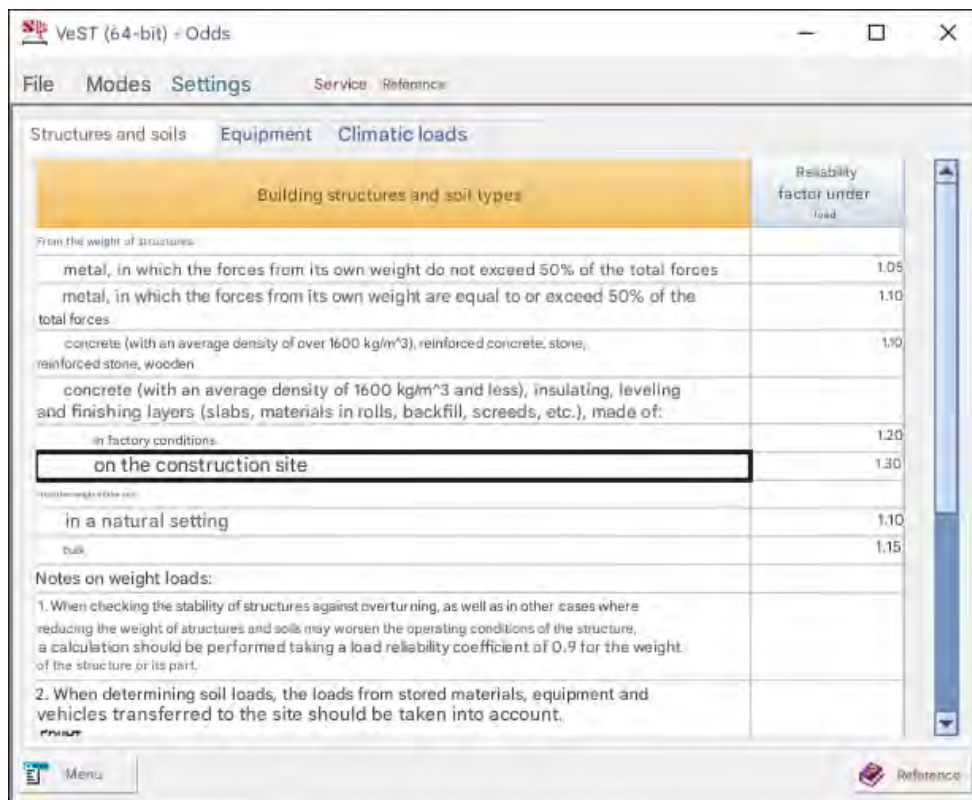


Figure 3.66 – Page coefficients of the WEST program

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

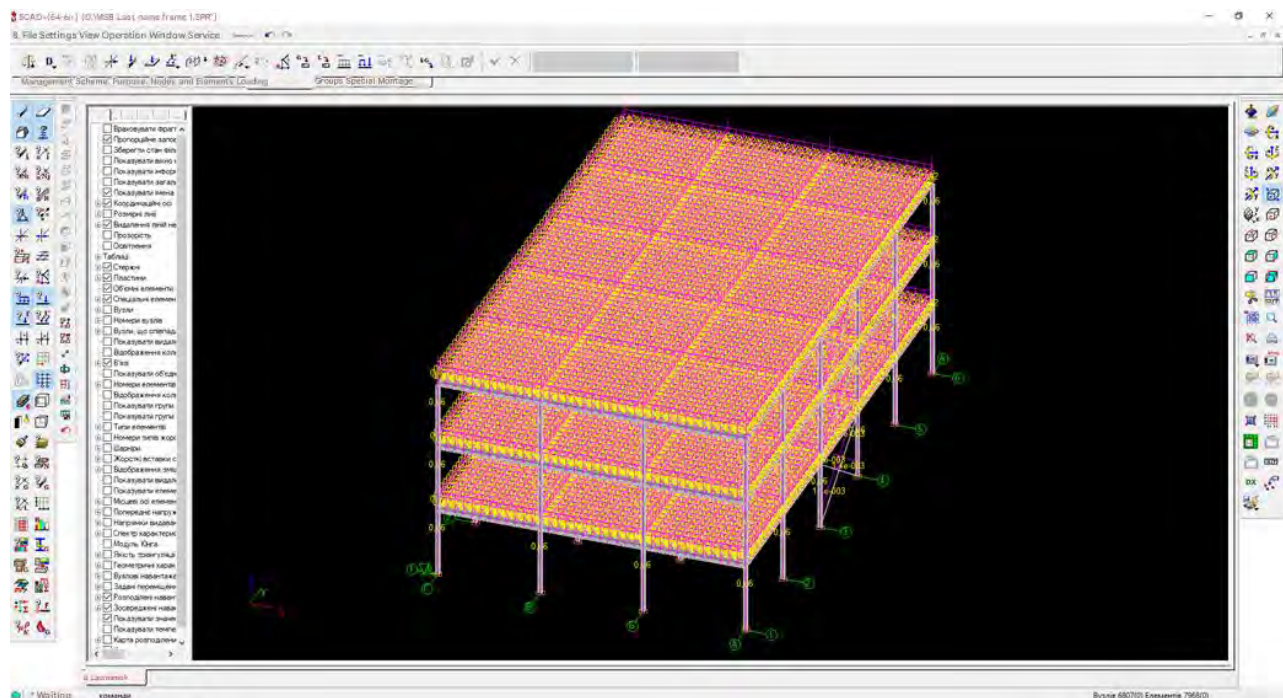


Figure 3.67 – Display of dead weight loads on the design diagram

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.

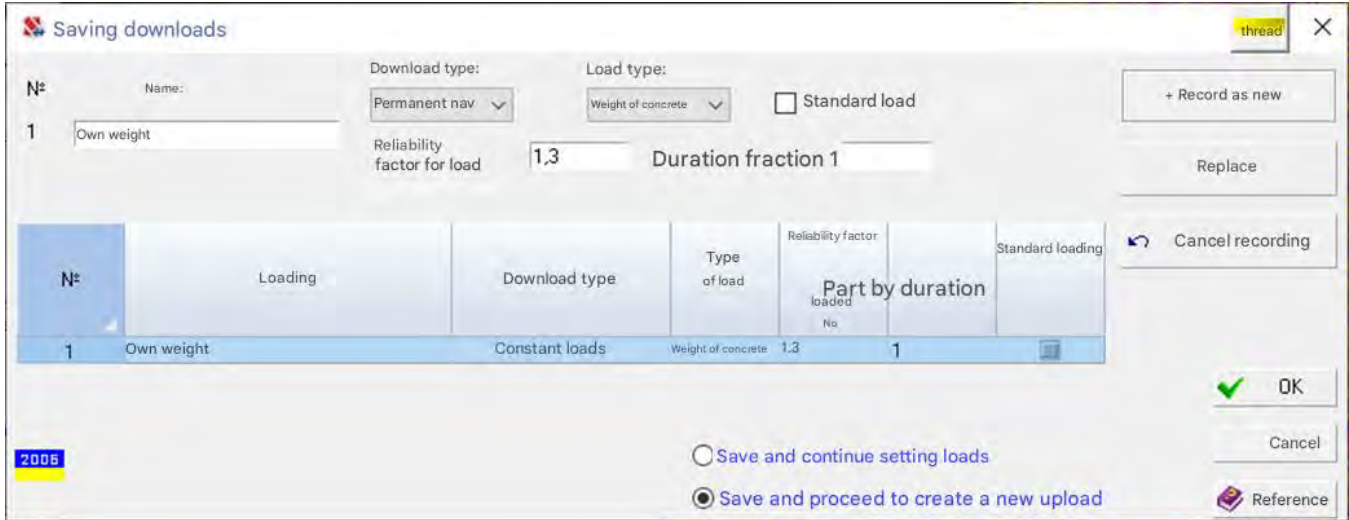


Figure 3.68 – Saving the load "Dead weight" in the "Saving loads" dialog box

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.

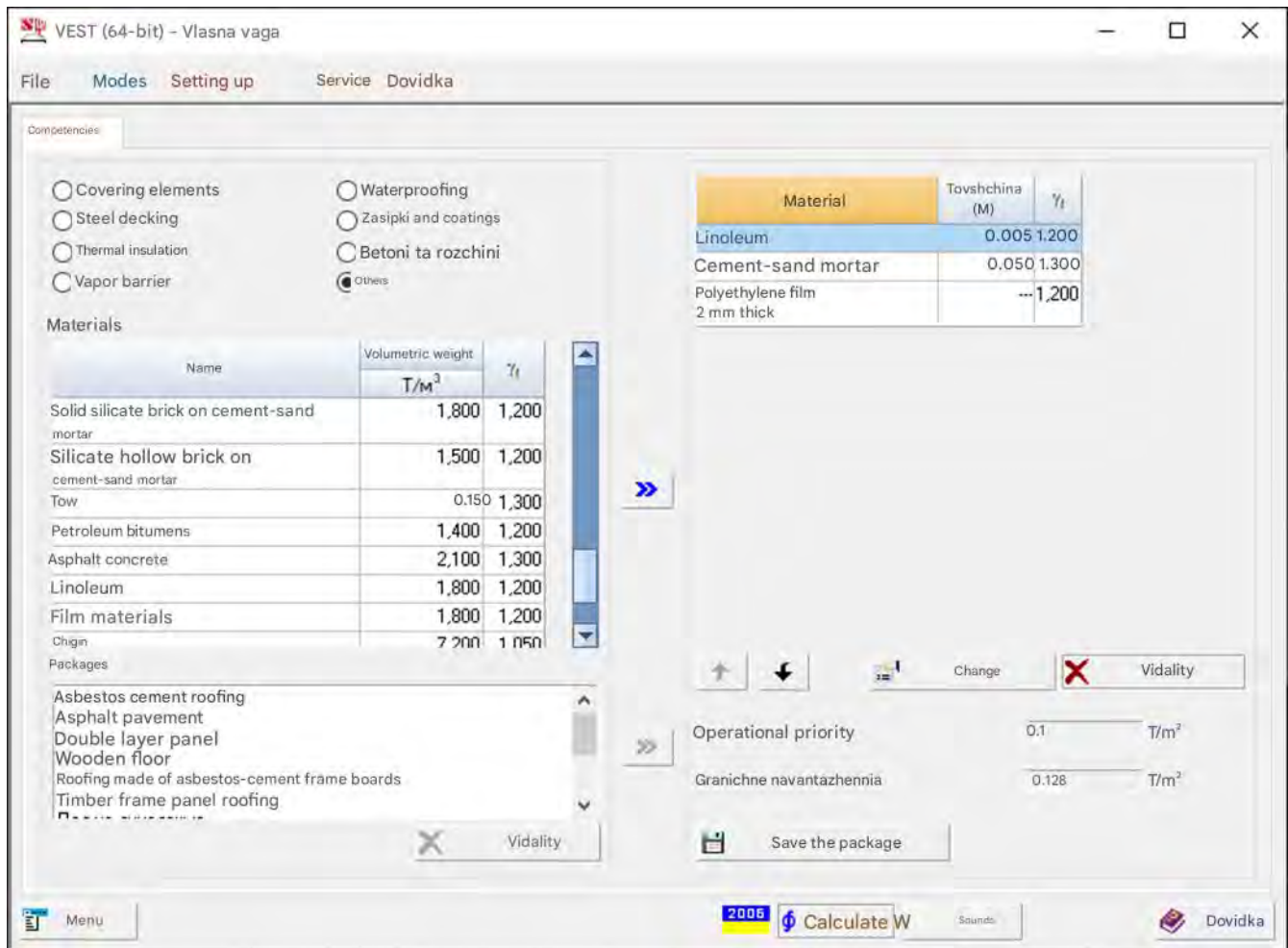


Figure 3.69 – Results of calculation of constant loads from the floor structure in the WEST program

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.

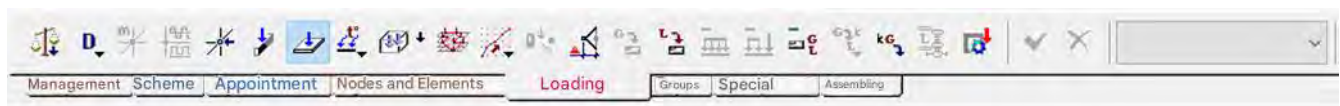


Figure 3.70 – Activating the "Plate Load" tool on the "Loading" tab

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.

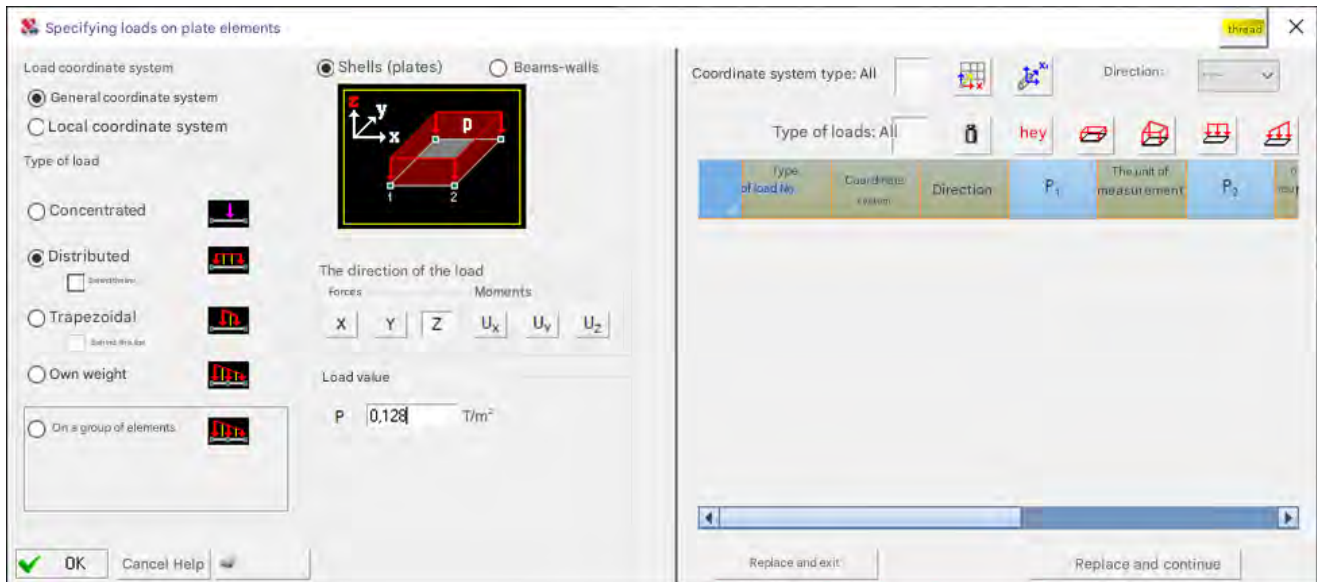


Figure 3.71 – Plate Load Dialog Box

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

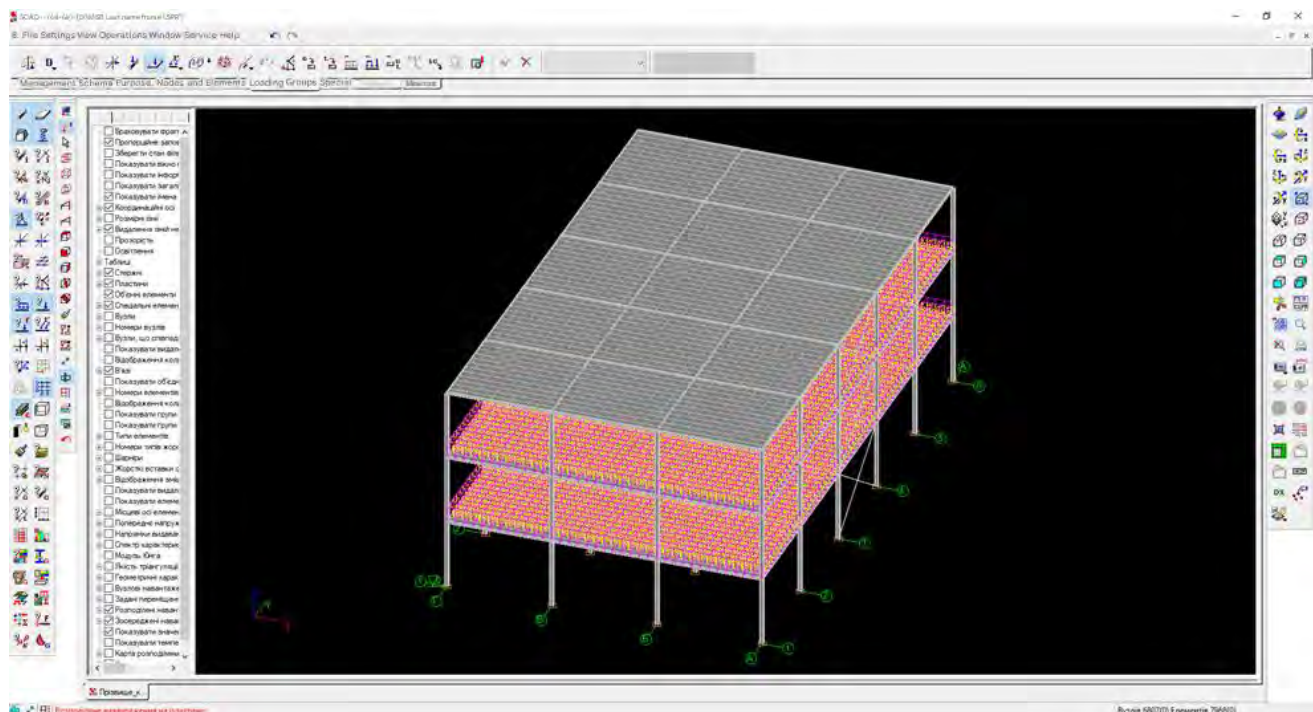


Figure 3.72 – Display of constant loads on the floor

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.

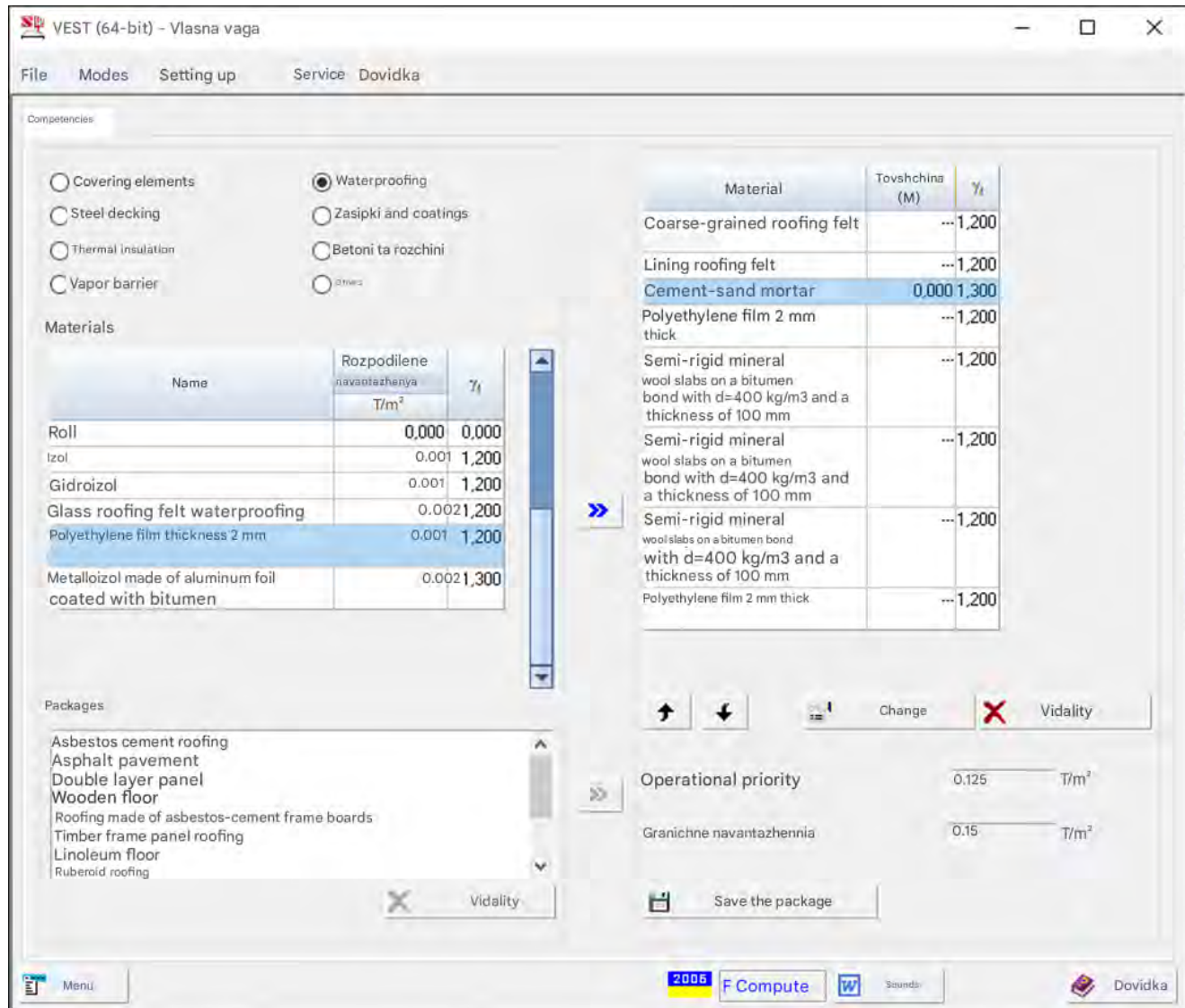


Figure 3.73 – Results of calculation of constant loads from the roof structure in the WEST program

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.

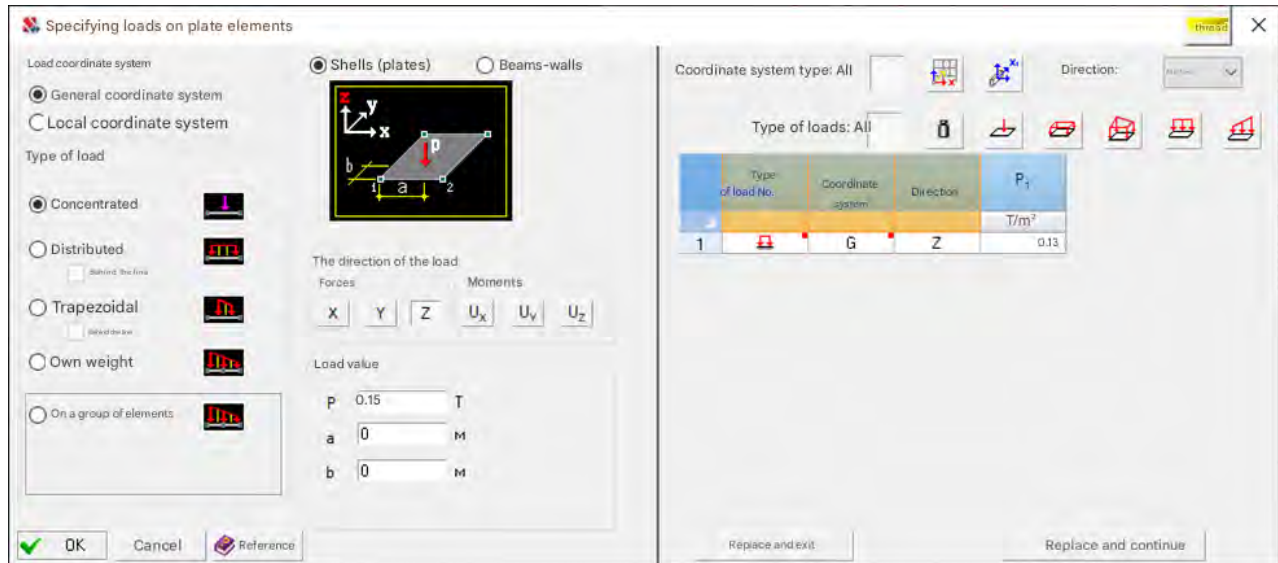


Figure 3.74 – Plate Load Dialog Box

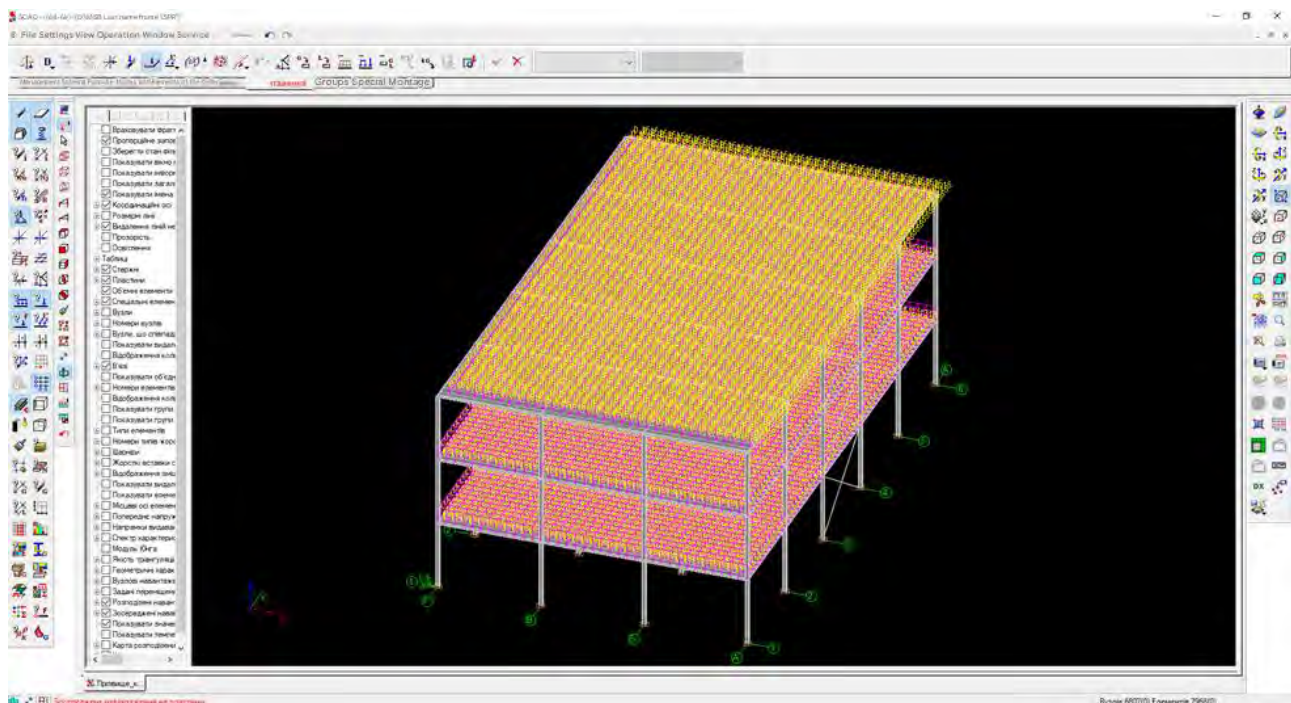


Figure 3.75 – Display of constant loads on floors and coatings

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.

Input data: Snow loads

Building height H : 12 m

Height of the construction site above sea level: 0.16 m

Possibility of melting or clearing snow: ☒

Calculate

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

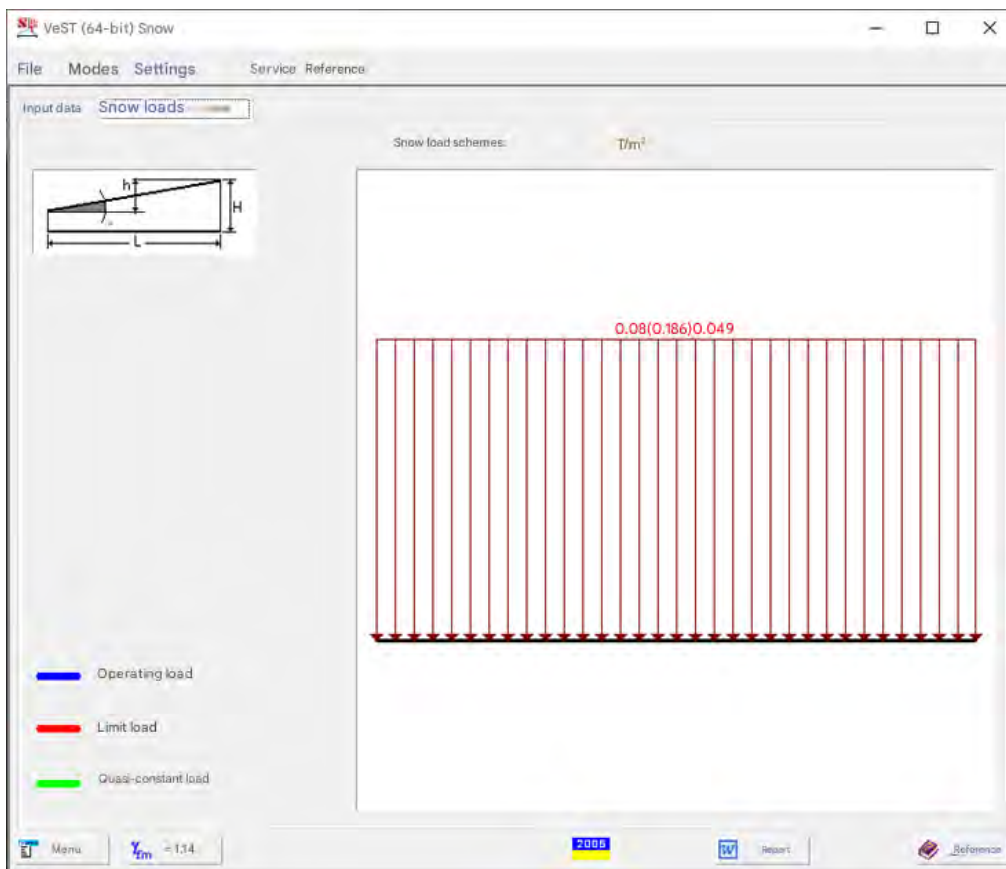


Figure 3.78 – Results of calculation of snow loads in the WEST program

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.

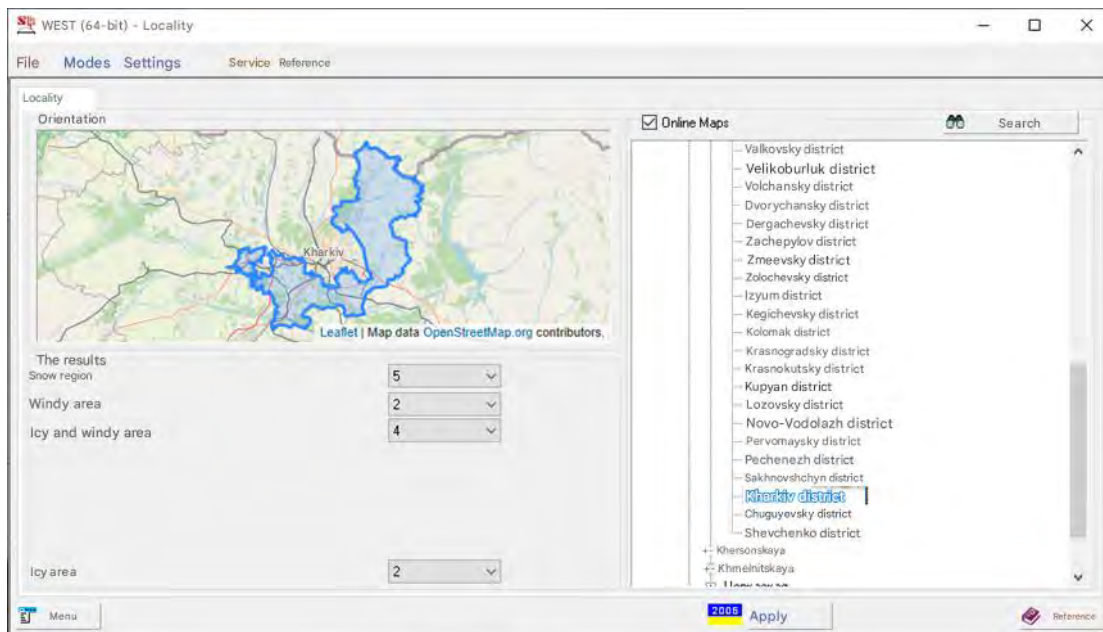


Figure 3.79 – Determining the zoning of the area using the WEST program

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.

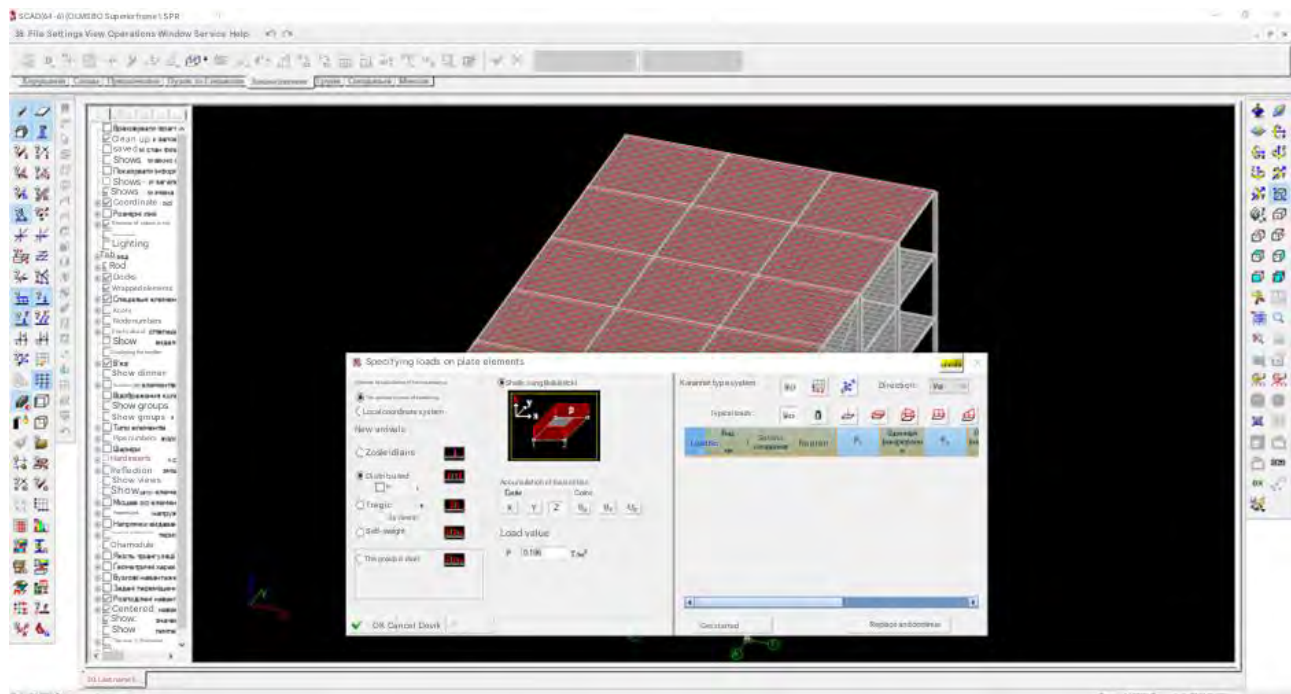


Figure 3.80 – Applying a snow load to the roof of the building

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.

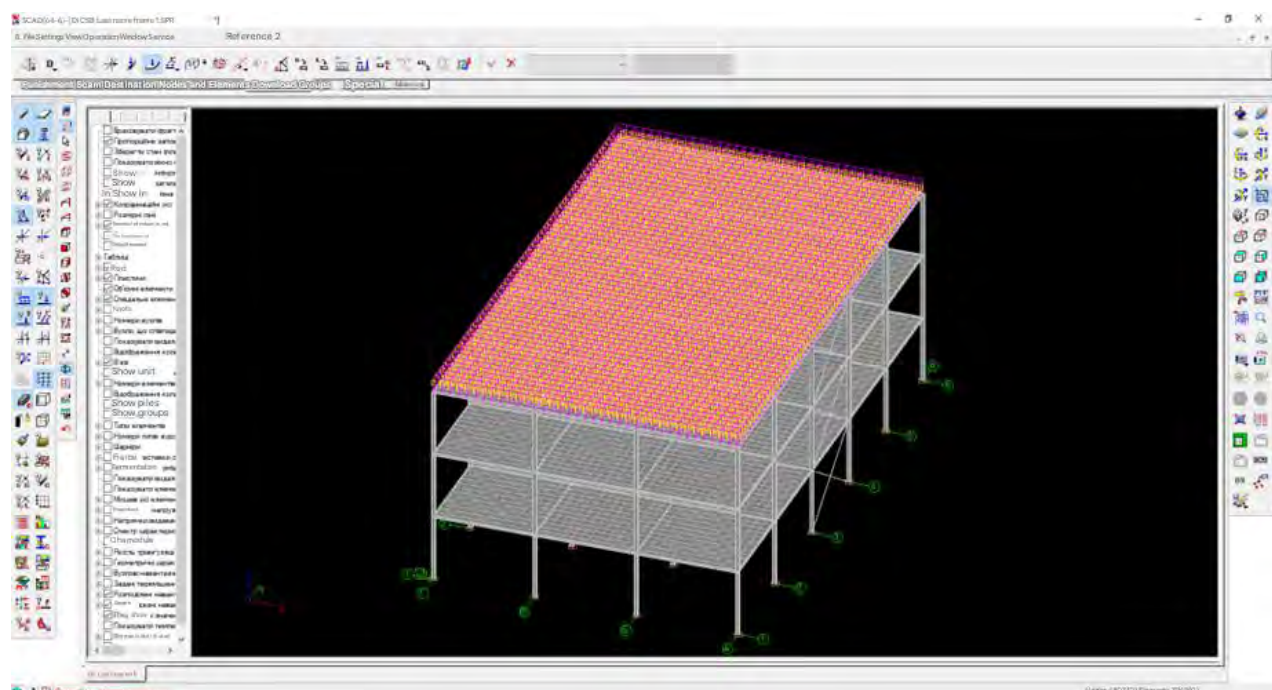


Figure 3.81 – Display of the snow load on the pavement

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.

Saving downloads

Nº: Name: Snow

Download type: Short-term Load type: Full of snow ☐ Standard load

Reliability factor for load: 1.14 Duration fraction: 0.43

Nº	Loading	Download type	Load Type	Reliability factor for load Nym	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	☐

Buttons: Record as new, Replace, Cancel recording, OK, Cancel, Save and continue setting loads, Save and proceed to create a new upload, Reference.

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.

WEST (64-bit) - Temporary loads

File Modes Settings Settings Reference

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:
- 5. Reading rooms
- 6. Lunch (in cafes, restaurants, canteens)
- 7. Gatherings and gatherings of catch and contest sports

Load: operational 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

Buttons: Menu, Calculate W, Report, Reference.

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.

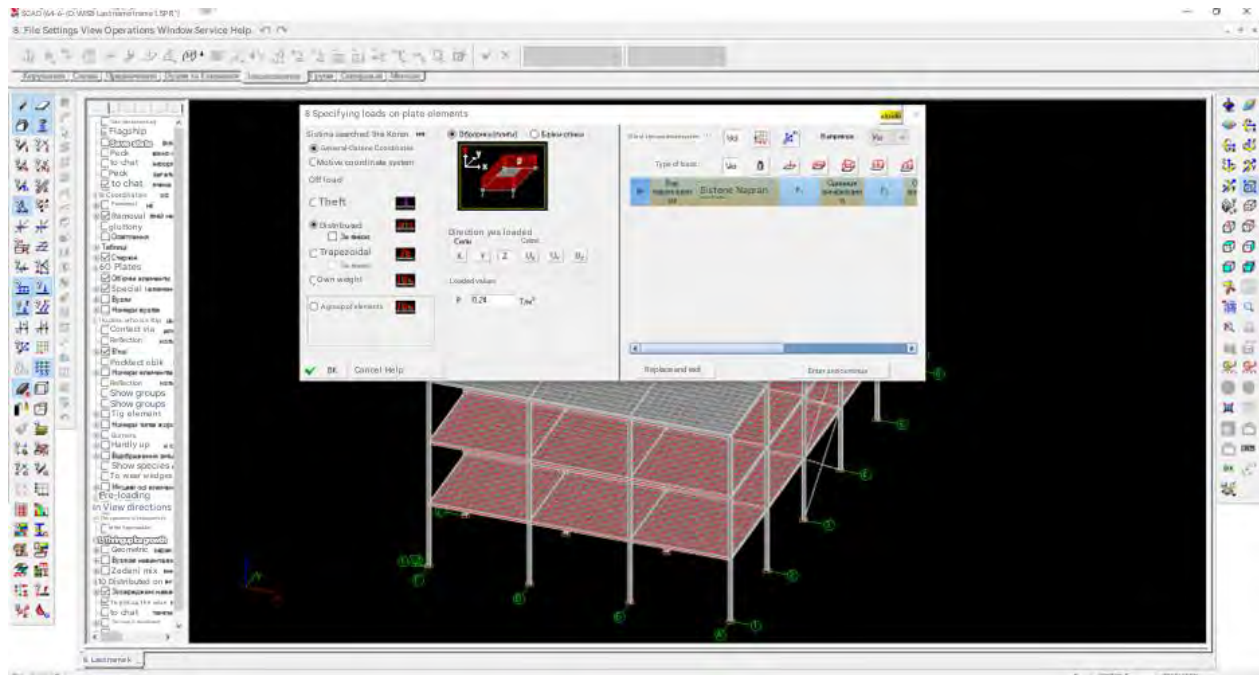


Figure 3.84 – Applying a payload to the ceilings of a building

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.

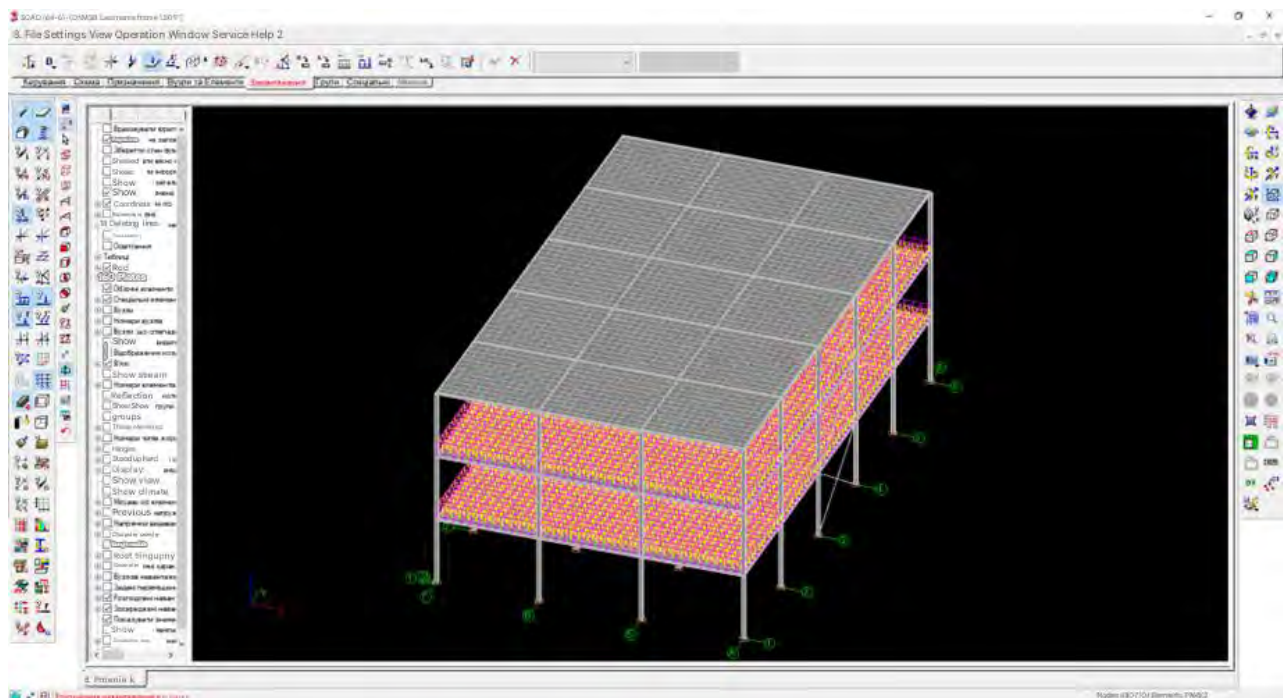


Figure 3.85 – Display of the payload on the ceiling

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.

N:	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.

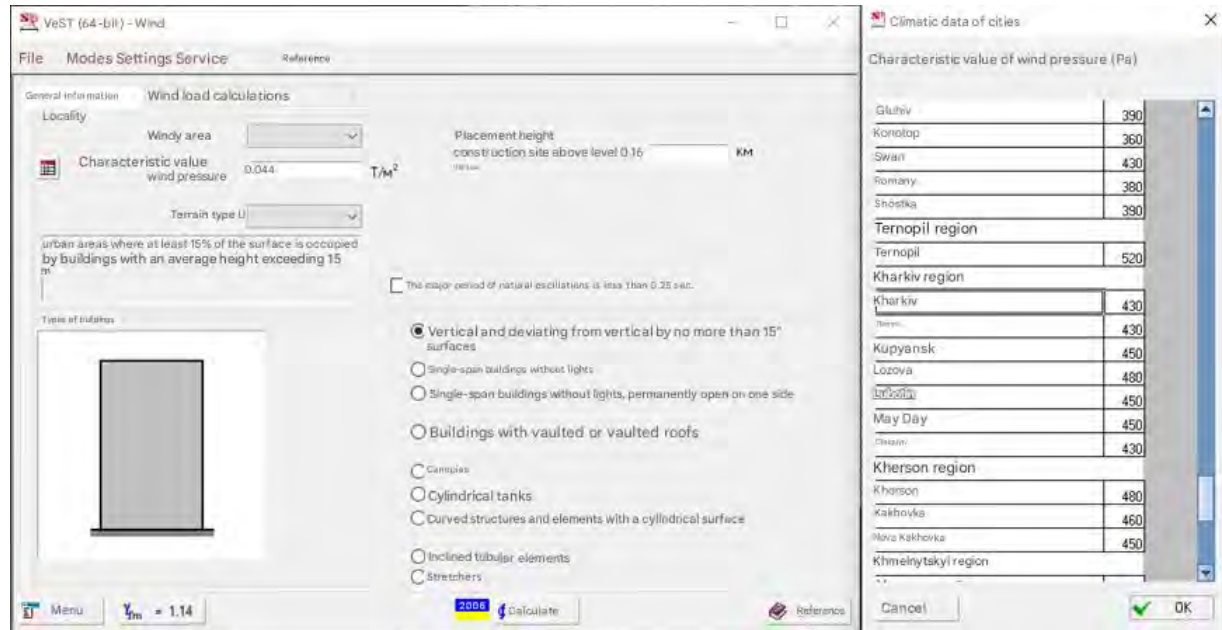


Figure 3.87 – Setting the parameters for calculating the wind load

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.

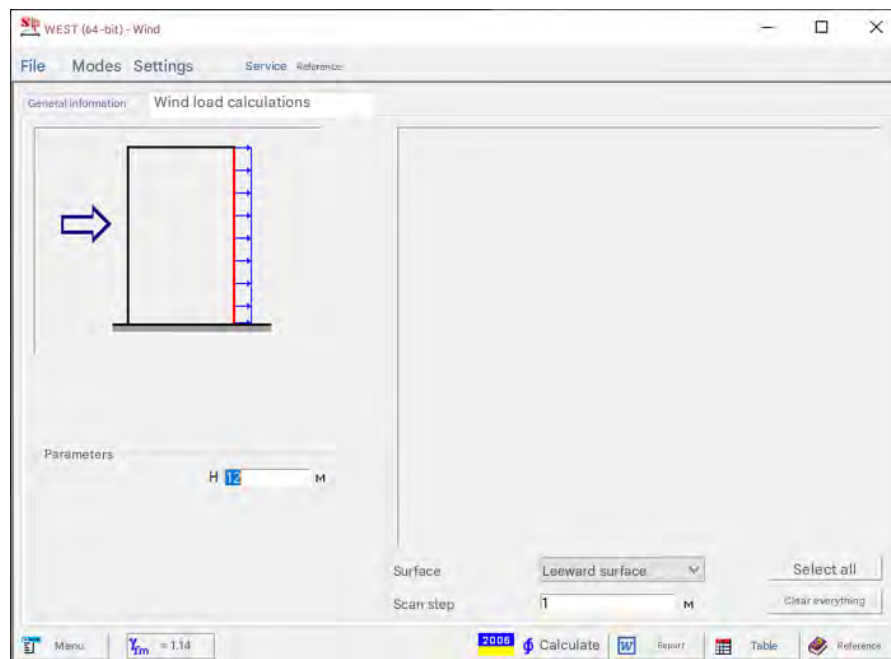


Figure 3.88 – Setting the parameters for calculating the wind load

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  .

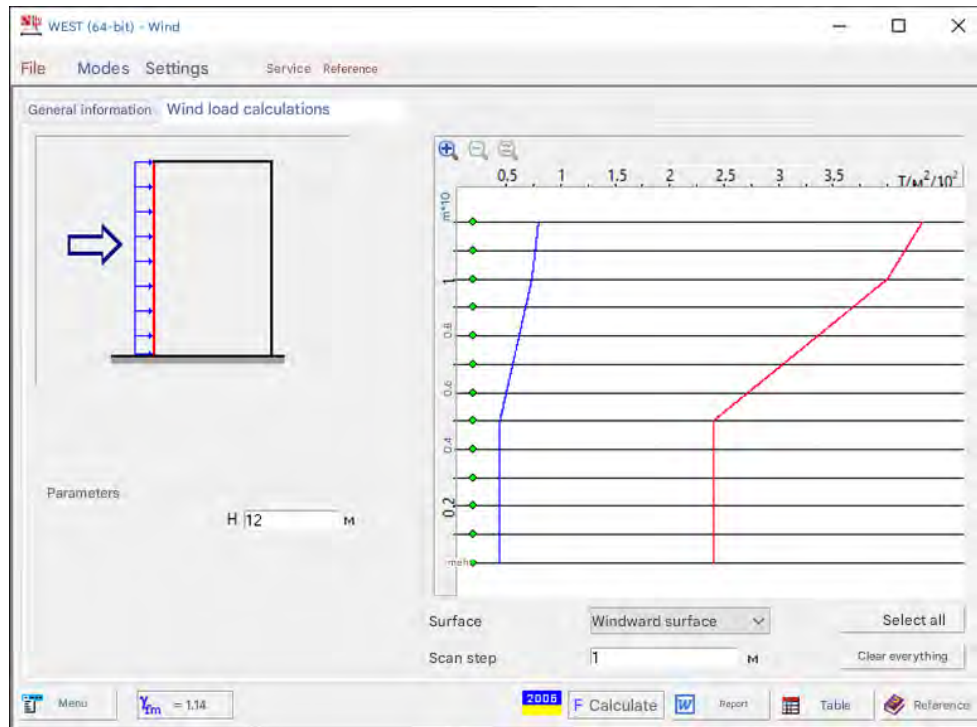


Figure 3.89 – Results of calculation of wind loads on the windward side

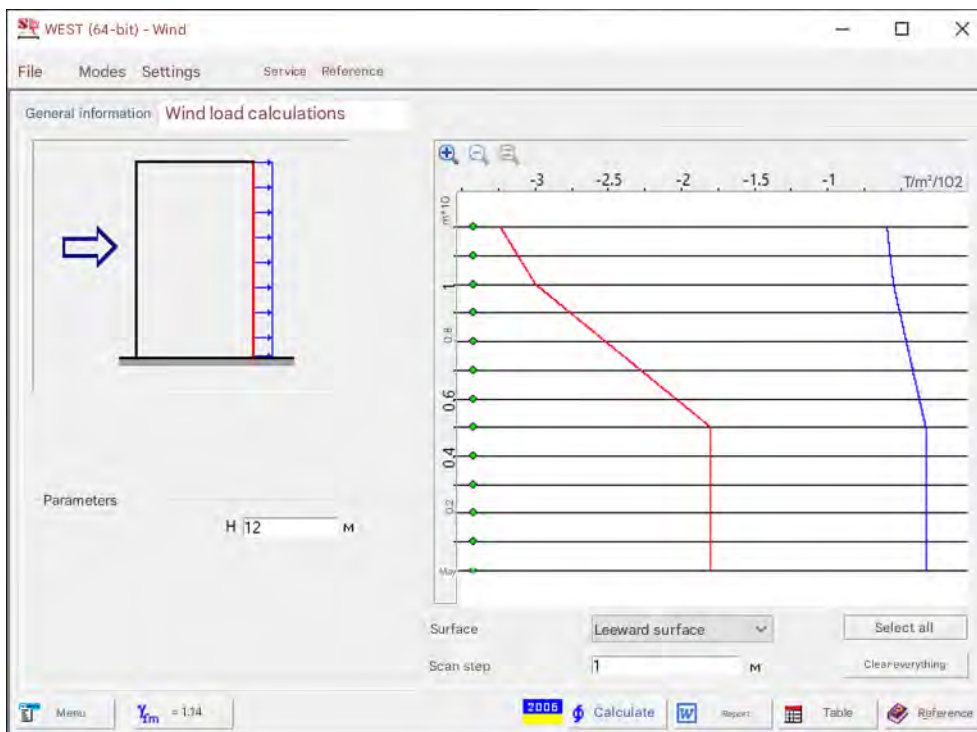


Figure 3.90 – Results of calculation of wind loads on the leeward side

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.

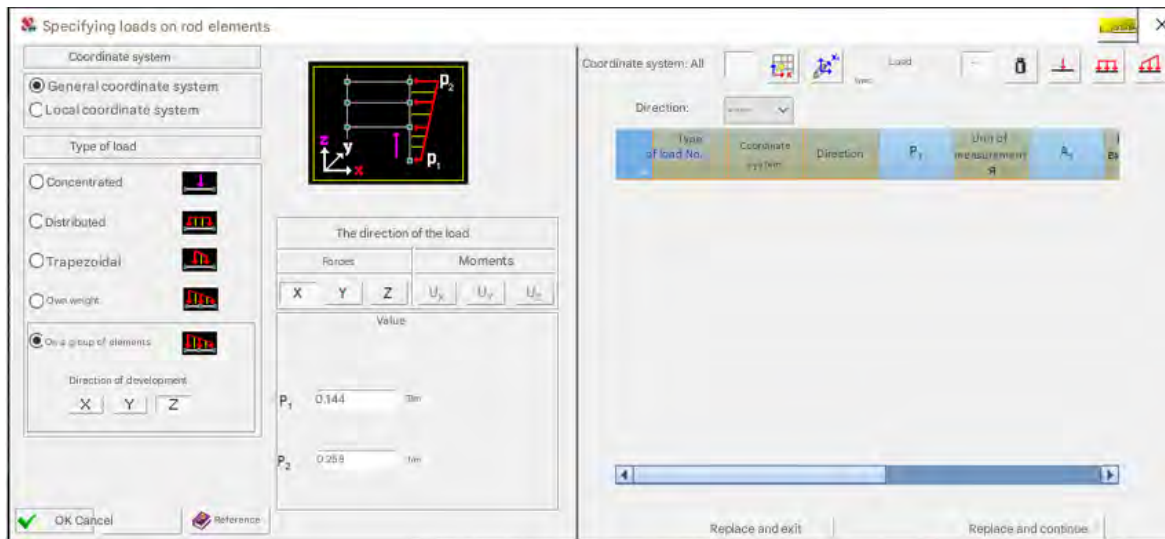


Figure 3.91 – Application of wind loads "Wind on the right" from the windward side (middle columns)

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

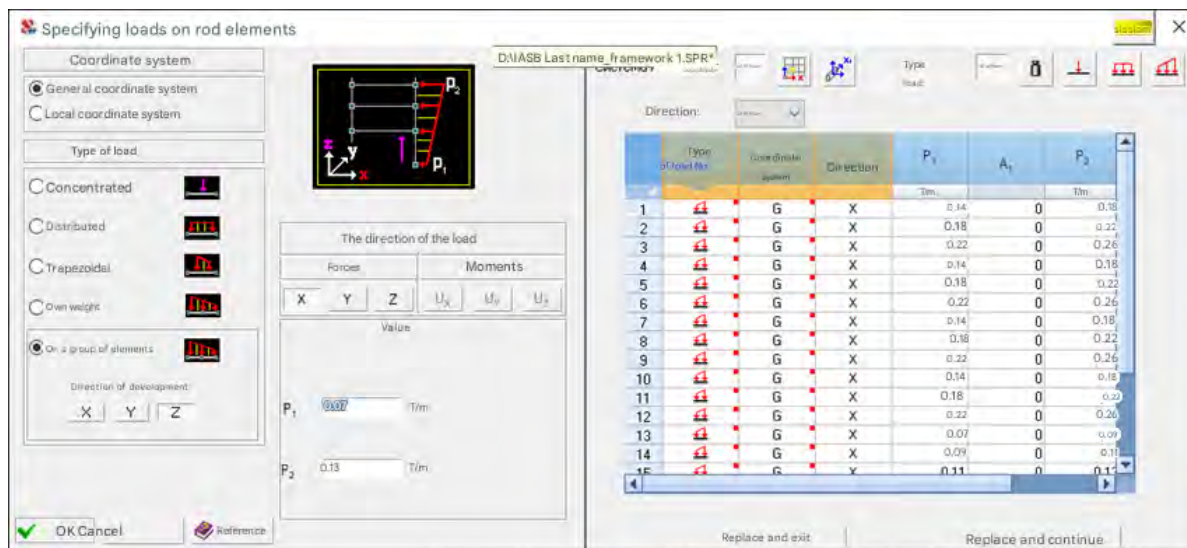


Figure 3.92 – Application of wind loads "Wind on the right" from the windward side (extreme columns)

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

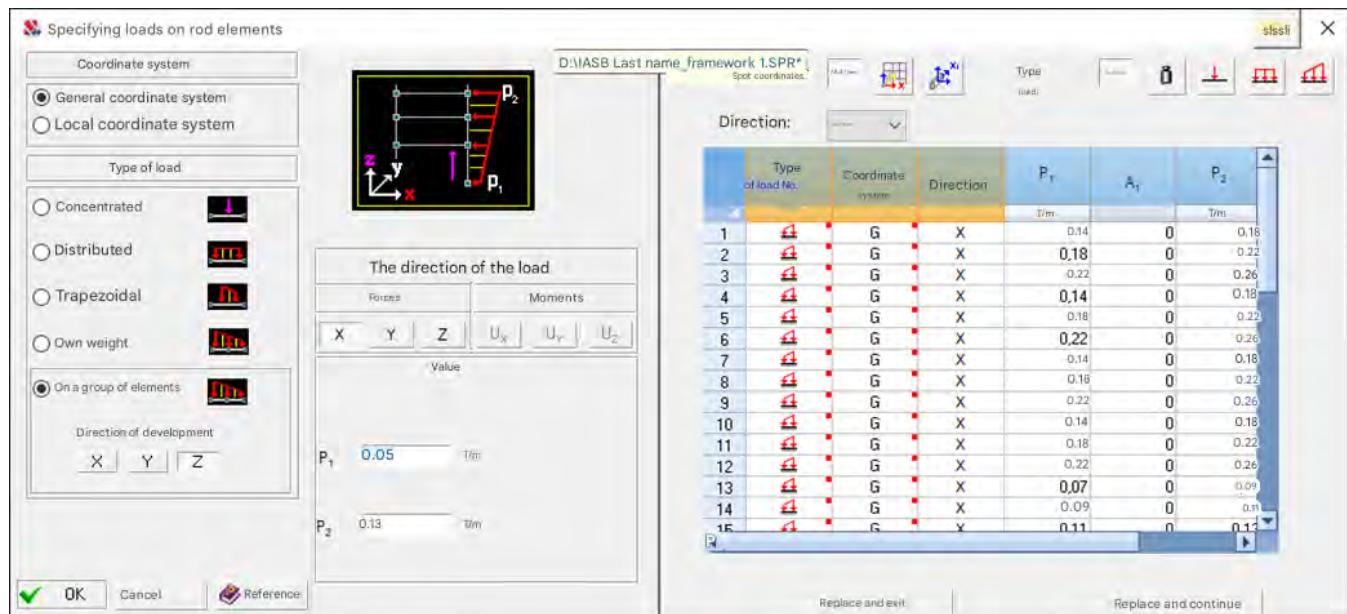


Figure 3.93 – Application of wind loads "Wind on the right" from the leeward side (middle columns)

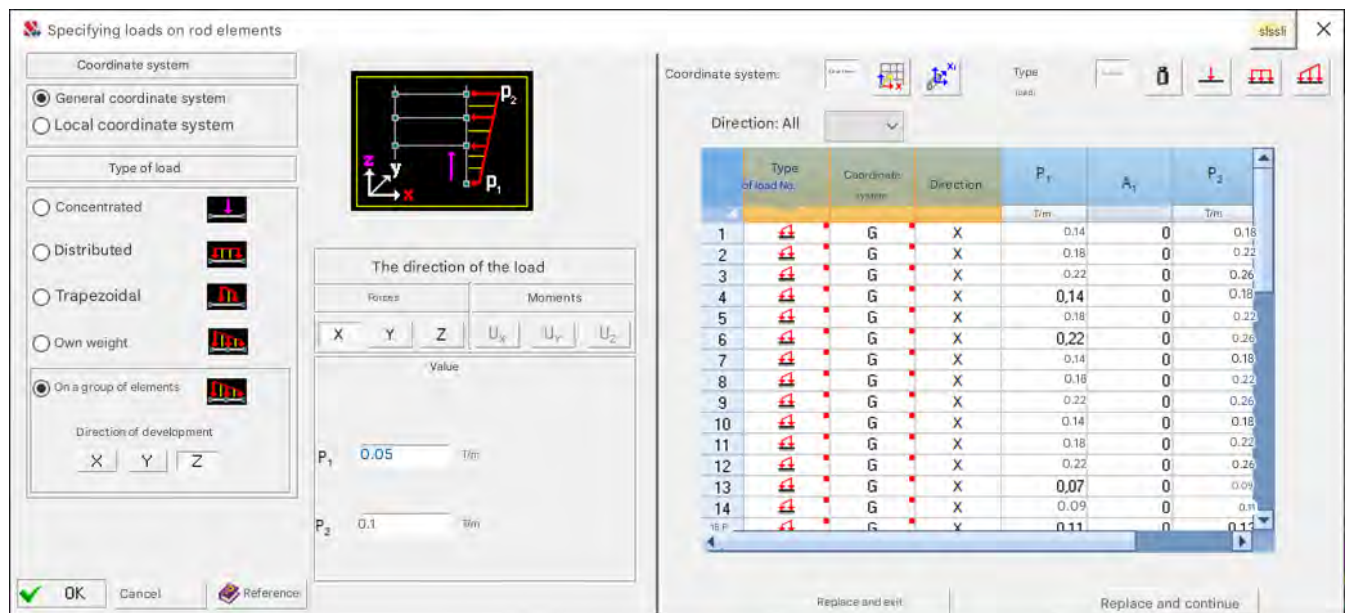


Figure 3.94 – Application of wind loads "Wind on the right" from the leeward side (extreme columns)

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

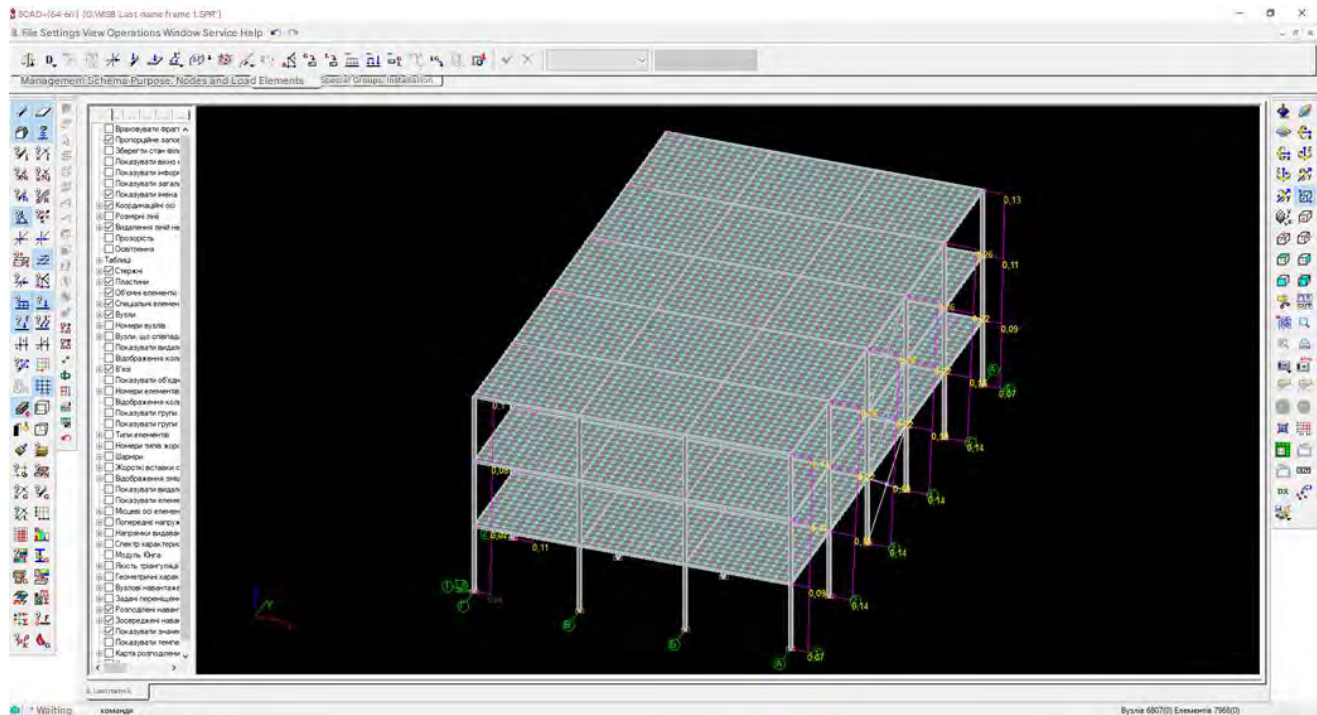


Figure 3.95 – Wind loads "Wind on the right" are applied to the design model of the building

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.

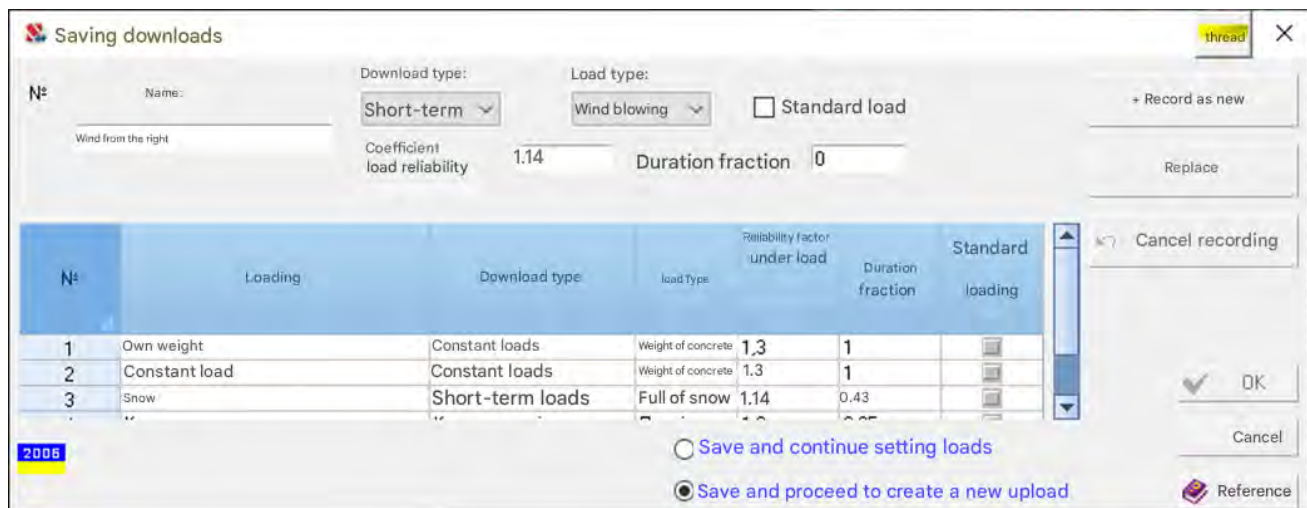


Figure 3.96 – Preservation of wind load "Wind on the right"

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.

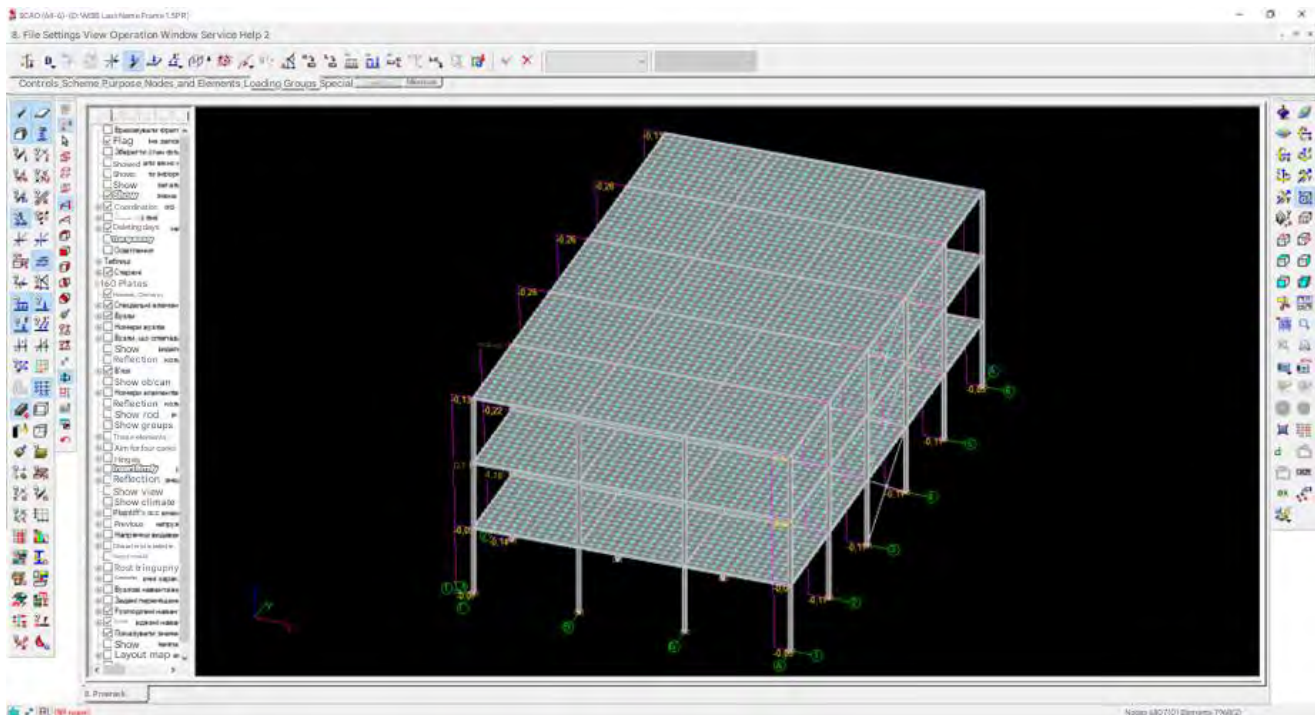


Figure 3.97 – Wind loads "Wind on the left" are applied to the design model of the building

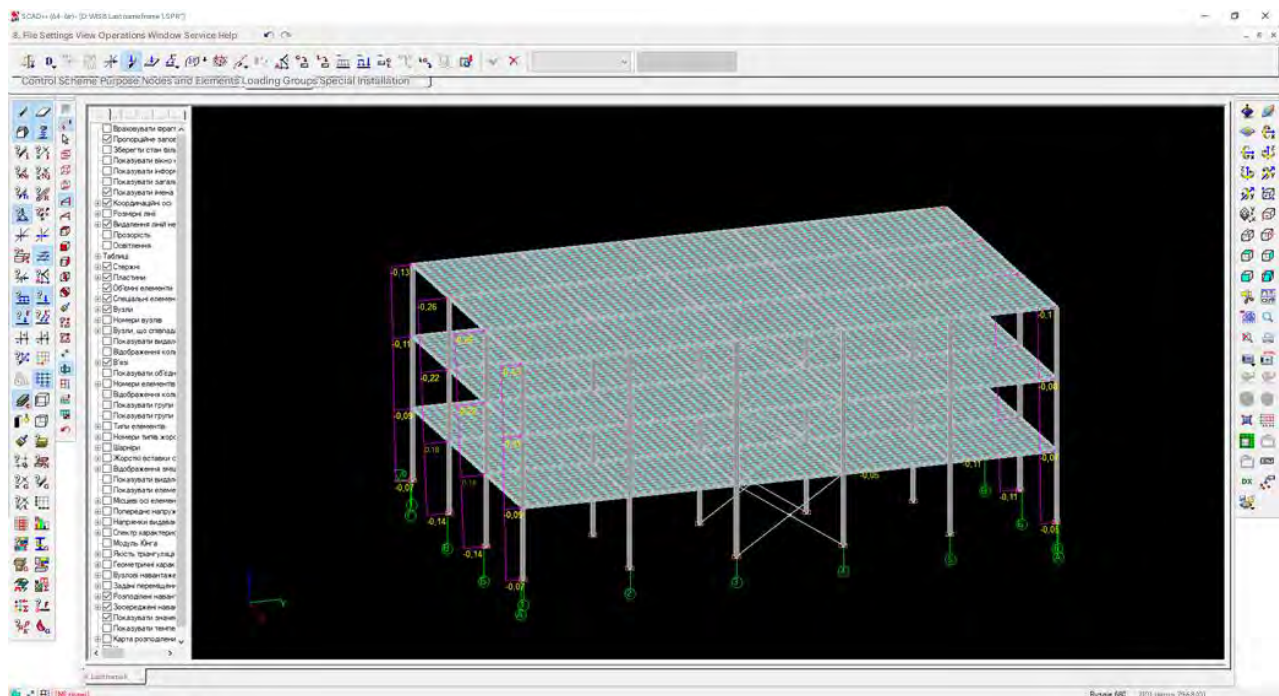


Figure 3.98 – Wind Ahead Wind Loads Applied to the Design Model of the Building

